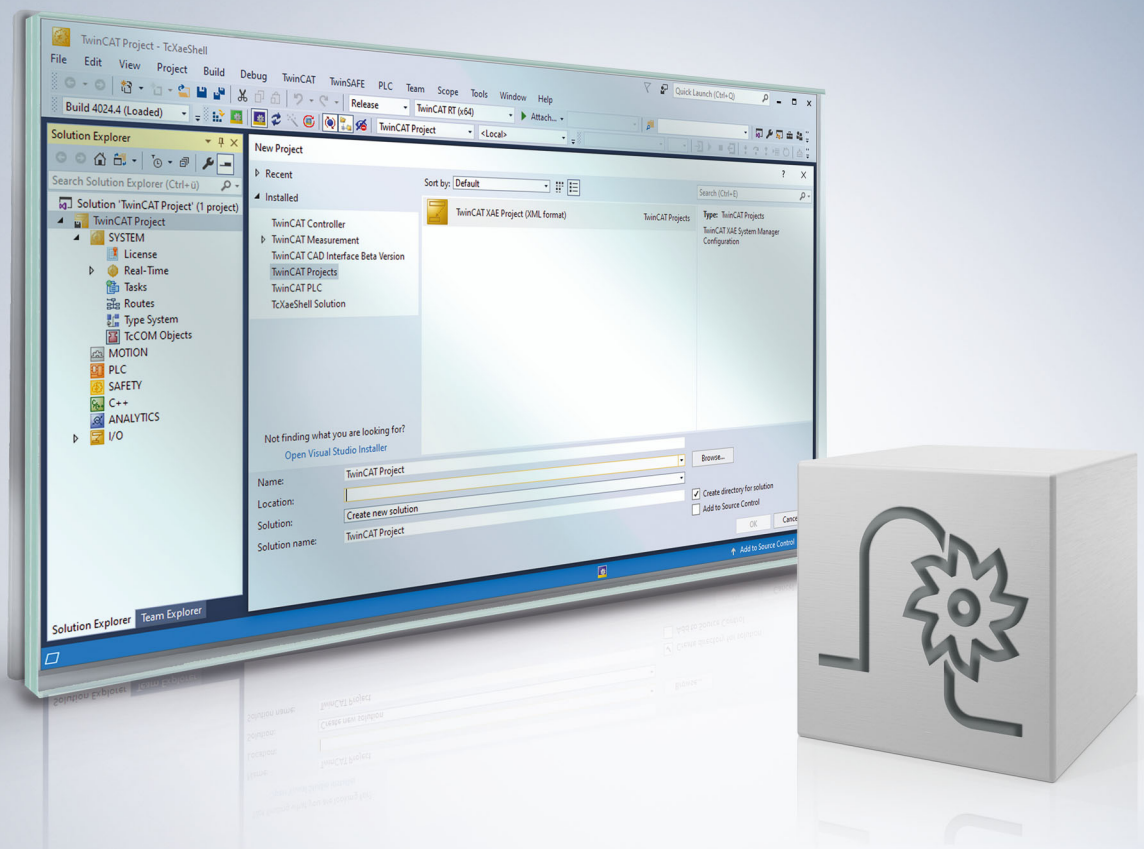
BECKHOFF New Automation Technology

Manual | EN

TF5200 | TwinCAT 3 CNC

Diagnosis manual



6/5/2025 | Version: 1.251

Notes on the documentation

This description is only intended for the use of trained specialists in control and automation engineering who are familiar with the applicable national standards.
It is essential that the documentation and the following notes and explanations are followed when installing and commissioning the components.
It is the duty of the technical personnel to use the documentation published at the respective time of each installation and commissioning.

The responsible staff must ensure that the application or use of the products described satisfy all the requirements for safety, including all the relevant laws, regulations, guidelines and standards.

Disclaimer

The documentation has been prepared with care. The products described are, however, constantly under development.
We reserve the right to revise and change the documentation at any time and without prior announcement.
No claims for the modification of products that have already been supplied may be made on the basis of the data, diagrams and descriptions in this documentation.

Trademarks

Beckhoff®, TwinCAT®, TwinCAT/BSD®, TC/BSD®, EtherCAT®, EtherCAT G®, EtherCAT G10®, EtherCAT P®, Safety over EtherCAT®, TwinSAFE®, XFC®, XTS® and XPlanar® are registered trademarks of and licensed by Beckhoff Automation GmbH.
Other designations used in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owners.

Patent Pending

The EtherCAT technology is patent protected, in particular by the following applications and patents:
EP1590927, EP1789857, EP1456722, EP2137893, DE102015105702
with corresponding applications or registrations in various other countries.



EtherCAT® is registered trademark and patented technology, licensed by Beckhoff Automation GmbH, Germany

Copyright

© Beckhoff Automation GmbH & Co. KG, Germany.
The reproduction, distribution and utilisation of this document as well as the communication of its contents to others without express authorisation are prohibited.
Offenders will be held liable for the payment of damages. All rights reserved in the event of the grant of a patent, utility model or design.

General and safety instructions

Icons used and their meanings

This documentation uses the following icons next to the safety instruction and the associated text. Please read the (safety) instructions carefully and comply with them at all times.

Icons in explanatory text

1. Indicates an action.

⇨ Indicates an action statement.

⚠ DANGER

Acute danger to life!

If you fail to comply with the safety instruction next to this icon, there is immediate danger to human life and health.

⚠ CAUTION

Personal injury and damage to machines!

If you fail to comply with the safety instruction next to this icon, it may result in personal injury or damage to machines.

NOTICE

Restriction or error

This icon describes restrictions or warns of errors.

Tips and other notes

This icon indicates information to assist in general understanding or to provide additional information.

General example

Example that clarifies the text.

NC programming example

Programming example (complete NC program or program sequence) of the described function or NC command.

Specific version information

Optional or restricted function. The availability of this function depends on the configuration and the scope of the version.

Table of contents

Notes on the documentation ... **3**

General and safety instructions ... **4**

1 Introduction ... **10**

1.1 Structure of a CNC error message ... 10
1.2 Overview of error reaction classes ... 10
1.3 Overview of troubleshooting classes ... 12
1.3.1 Overview of possible error types ... 12
1.3.2 Additional diagnostic values, overview of possible meanings ... 18
1.3.3 Dimensions identifiers ... 19
1.4 Presentation of an error message ... 20
1.4.1 Error message from NC program decoding ... 20
1.4.2 Error message from look-ahead ... 21

2 Overview of CNC error messages ... **22**

2.1 General error, start-up error (ID range 1000-12999) ... 22
2.1.1 ID-range 1000-1249 ... 22
2.1.2 ID-range 1500-1749 ... 24
2.1.3 ID-range 2000-2249 ... 29
2.1.4 ID-range 3000-3249 ... 39
2.1.5 ID-range 3250-3499 ... 41
2.1.6 ID-range 4000-4500 ... 42
2.1.7 ID-range 10000-10249 ... 49
2.1.8 ID-range 11000-11249 ... 50
2.2 Cycle errors (13000-19999) ... 54
2.2.1 ID-range 13000-13249 ... 54
2.2.2 ID-range 13250-13499 ... 78
2.2.3 ID-range 13500-13749 ... 109
2.2.4 ID-range 14000-14249 ... 117
2.2.5 ID-range 14250-14499 ... 130
2.3 Decoding errors (syntax, semantic) (ID-range 20000-29999) ... 168
2.3.1 ID-range 20000-20249 ... 168
2.3.2 ID-range 20250-20499 ... 241
2.3.3 ID-range 20500-20749 ... 282
2.3.4 ID-range 20750-20999 ... 333
2.3.5 ID-range 21000-21249 ... 386
2.3.6 ID-range 21250-21499 ... 433
2.3.7 ID-range 21500-21749 ... 511
2.3.8 ID-range 21750-21999 ... 574
2.3.9 ID-range 22000-22249 ... 631
2.3.10 ID range 22250-22499 ... 683
2.4 Configuration error (ID-range 30000-39999) ... 706
2.4.1 ID-range 30000-30249 ... 706
2.5 PLC function block error (ID-range 40000-49999) ... 707
2.5.1 ID-range 40000-40249 ... 707

2.5.2 ID-range 42000-42249 ... 738
2.5.3 ID-range 44000-44249 ... 740
2.5.4 ID-range 46000-46249 ... 762
2.6 Path interpolator error (ID-range 50000-59999) ... 777
2.6.1 ID-range 50000-50249 ... 777
2.6.2 ID-range 50250-50499 ... 796
2.6.3 ID-range 50500-50749 ... 818
2.6.4 ID-range 50750-50999 ... 864
2.6.5 ID-range 51000-51250 ... 917
2.7 Spindle interpolator error (ID-range 60000-69999) ... 948
2.7.1 ID-range 60000-60249 ... 948
2.7.2 ID-range 60250-60499 ... 987
2.8 Position controller error (ID-range 70000-79999) ... 1009
2.8.1 ID-range 70000-70249 ... 1009
2.8.2 ID-range 70250-70499 ... 1064
2.8.3 ID-range 70500-70749 ... 1150
2.9 Axis filter errors (ID-range 80000-89999) ... 1202
2.9.1 ID-range 80000-80249 ... 1202
2.10 Tool radius compensation error (ID-range 90000-99999) ... 1206
2.10.1 ID-range 90000-90249 ... 1206
2.11 Axis management errors (ID-range 110000-119999) ... 1242
2.11.1 ID-range 110000-110249 ... 1242
2.11.2 ID-range 110250-110499 ... 1294
2.11.3 ID-range 110500-110749 ... 1371
2.12 Path preparation error (ID-range 120000-129999) ... 1439
2.12.1 ID-range 120000-120249 ... 1439
2.12.2 ID-range 120250-120499 ... 1467
2.12.3 ID-range 120500-120749 ... 1495
2.12.4 ID-range 120750-120999 ... 1542
2.13 Communication error (ID-range 130000-139999) ... 1556
2.13.1 ID-range 130000-130249 ... 1556
2.14 Drive management error (ID-range 140000-149999) ... 1561
2.14.1 ID-range 140000-140249 ... 1561
2.15 Manual mode error (ID-range 150000-159999) ... 1572
2.15.1 ID-range 150000-150249 ... 1572
2.16 Utility error (as of ID area 220000) ... 1594
2.16.1 ID-range 220000-220249 ... 1594
2.16.2 ID-range 230000-230249 ... 1597
2.16.3 ID-range 240000-240249 ... 1598
2.16.4 ID-range 250000-250249 ... 1607
2.16.5 ID-range 260000-260249 ... 1611
2.16.6 ID-range 270000-270249 ... 1630
2.16.7 ID-range 280000-280249 ... 1649
2.16.8 ID-range 280250-280499 ... 1650
2.16.9 ID-range 280500-280749 ... 1651
2.16.10 ID-range 292000-292249 ... 1663

2.16.11 ID-range 293000-293249 ... 1689
2.16.12 ID-range 294000-294249 ... 1692
2.16.13 ID-range 294500-294749 ... 1709
2.16.14 ID-range 295000-295249 ... 1729
2.16.15 ID-range 296000-296249 ... 1738
2.16.16 ID-range 297000-297249 ... 1743
2.16.17 ID-range 301000-301249 ... 1744
2.16.18 ID-range 310000-399999 ... 1750
2.16.19 ID-range 400000-400249 ... 1770
2.16.20 ID-range 401000-401249 ... 1773
2.16.21 ID-range 402000-402249 ... 1774
2.16.22 ID-range 403000-403249 ... 1775
2.16.23 ID-range 404000-405005 ... 1776
2.16.24 ID-range 1000000-1000249 ... 1776
2.16.25 ID-range 1001000-1001249 ... 1790
2.16.26 ID-range 1001750-1001999 ... 1804
2.16.27 ID-range 1003000-1003249 ... 1811

3 Support and Service ... 1813

Index ... 1814

List of figures

1 Introduction

This document describes the structure of the CNC error messages and provides an overview of all possible error messages.

1.1 Structure of a CNC error message

Each CNC error message contains a unique error number and other information that describes the cause of the error as accurately as possible. Also, the error message contains identifiers of the minimum error response that the CNC will internally perform immediately after the error occurs.

For easier diagnosis, the error messages are classified with regard to the cause of the error. Each corresponding error type receives a unique identifier and a data packet, which is always provided with this error type. These data packets (error roots) are described in the chapter entitled Error Types.

They also include up to nine further parameters that contain additional information and, if applicable, provide limit values or incorrect internal control system values for display.

1.2 Overview of error reaction classes

Each CNC error message is assigned to an error response class. This error response describes the internal, prompt reaction of the CNC to the error and initiates minimal measures to achieve a safe state from the CNC's point of view.

Error reaction class	internal error reaction	Example(s)
1	**No reaction** Only possible in the event of a warning (error class 1)	Programmed feed rate not reached (BAVO).
2	**NC program execution aborted and transition to error state** If an error occurs in the NC block preparation area, the interpolator executes the NC blocks that have already been prepared. In this case, the time between the occurrence of the error and machine standstill depends on the type and number of buffered NC blocks. The BF reporting the error assumes an error state.	Syntax error in the NC program (DECODER) Geometry error in block processing (TRC) Software limit switch error in block processing (BAVO)
3	**Job execution aborted and transition to normal state** After an error message, BFs that provide services for other BFs (servers), for example AXIS ADMINISTRATION, FILE ADMINISTRATION, MANUAL MODE, etc., abort job execution and return to normal state.	Invalid axis requested in the case of BF AXIS ADMINISTRATION Unknown handwheel activated in MANUAL MODE.
4	**Motion stopped (feedhold) for the entire axis group and transition to error state**.	Measure: Probe already actuated.
5	**Abrupt axis stop for defective axis, feedhold for the other axes in the axis group and transition to an error state**	Due to the position lag of an axis, the interpolator reacts with an axis stop for the axes in its axis group (PATH AXES).
6	**Abrupt axis stop for all axes and transition to an error state** Position control assumes an error state.	Unknown state in the INTERPOLATOR or POSITION CONTROLLER

7	**Closed–loop controlled axis stop for defective axis, feedhold for the other axes in the axis group and transition to an error state**	Crossing software limit switches in the POSITION CONTROLLER
8	**Open–loop controlled axis stop for defective axis, feedhold for the other axes in the axis group and transition to an error state** The position control loop of the defective axis is opened.	Exceeding the maximum permissible position lag during axis motion in the POSITION CONTROLLER

1.3 Overview of troubleshooting classes

In a standardised form, the troubleshooting class describes the measures for remedying an error.

NOTICE

Difference between troubleshooting and error-free state

In this connection, remedying an error does not mean that the cause of the problem is remedied, but that the CNC can be set to an error-free state.

Troubleshooting class	Internal troubleshooting	
	Simulation	**Normal mode**
0	Error message acts as warning; this causes **automatically** one **internal troubleshooting**	Error message acts as warning; this causes **automatically** one **internal troubleshooting**
2	**Program processing** continued **can call an implicitly called subroutine at program end.**	Requires complete **reset** of the NC channel
5	Requires complete **reset** of the NC channel	Requires complete **reset** of the NC channel
6	Requires **restart** of the CNC kernel	Requires **restart** of the CNC kernel
7	Requires **restart** of the entire CNC kernel after switch-off	Requires **restart** of the entire CNC kernel after switch-off

NOTICE

If an error of error class 7 occurs, please contact the support of your control manufacturer.

They will need the complete error message with the error number and all further message parameters.

1.3.1 Overview of possible error types

An error type classifies the information that is provided to the operator with the error message. The more extensive this additional information is, the easier it becomes to recognise the cause of the error on the basis of the error pattern.

Error messages generated due to errors in the NC program contain the NC block number, the file names and the current offset. This makes it possible for an operator to visualise the NC line containing the error.

1.3.1.1 Type 1: Error message from NC program

Error messages generated due to errors in the NC program contain the following additional diagnostic data:

Data format	Element	Contains
UNS16	log_pfad_nr	Logical path number
PROG_NAME	prog_name	Program name
FILE_NAME	file_name	File name
UNS32	fileoffset	File offset
UNS16	satzoffset	Satzoffset
UNS16	tokenoffset	Expression offset
UNS32	satz_nr	NC block number

Practical examples:

- Syntax error in the NC program
- Violation of logical or technological limits by corresponding programming in the NC program
- Programming of incorrect values, parameters, etc.

1.3.1.2 Type 2: Error message by data transfer from parameter list to control device

An error message is generated if a parameter file cannot be completely transferred to the controller when the controller is started or when the parameters are updated.

These error messages contain the following additional diagnostic data:

Data format	Element	Contains
UNS16	listen_typ	List type
STRUKT_NAME	strukt_name	Structure element

Practical examples:

- Violation of technological or logical limits in the MDS
- Incorrect assignment of MDS_entries

1.3.1.3 Type 3: Communication error

Communication errors happen when messages cannot be delivered or timeouts occur. The recipient of the message is either unknown or busy.

These error messages contain the following additional diagnostic data:

Data format	Element	Contains
UNS16	Medium	Communication medium
UNS16	Typ	PDU code or function block type
UNS32	Partner	Sender

Practical examples:

- Invalid PDU code received
- Channel block marked as 'not processed'
- Incorrect content of a PDU block

1.3.1.4 Type 4: RAM disk (optional)

Errors of this type can occur in connection with access to the RAM disk. This includes e.g. writing, reading or deleting files (using a RAM disk is an optional extended function).

These error messages contain the following additional diagnostic data:

Data format	Element	Contains
UNS16	Medium	Communication medium
UNS16	Type	PDU code or function block type
UNS32	Partner	Sender
FILE_NAME	file_name	File name
UNS32	file offset	File offset

Practical examples:

- Invalid PDU code received
- Channel block marked as 'not processed'
- Incorrect content of a PDU block

1.3.1.5 Type 5: Error messages on access to files

Error messages are generated if errors occur in the file system when accessing files during control start-up, during program decoding, or when updating parameter lists.

These error messages contain the following additional diagnostic data:

Data format	Element	Contains
FILE_NAME	file_name	File name
UNS32	file offset	File offset

Practical examples:

File cannot be opened and, consequently, cannot be read.

1.3.1.6 Type 6: Error messages on data transfer to control device

Error messages are generated if unknown entries are detected when the controller is started or parameter lists are updated.

These error messages contain the following additional diagnostic data:

Data format	Element	Contains
FILE_NAME	file_name	File name
UNS32	file offset	File offset
UNS16	listen_typ	List type
STRUKT_NAME	strukt_name	Structure element

Practical examples:

- Incorrect entry in the parameter list of the decoder (SDA-MDS), unknown element

1.3.1.7 Type 7: Error messages on transfer of binary data to control device (optional)

Error messages are generated if unknown entries are detected when the controller is started, or parameter lists are updated in binary form.

These error messages contain the following additional diagnostic data:

Data format	Element	Contains
STRUKT_NAME	strukt_name	Structure element

Practical examples:

- Incorrect entry in the parameter list of the decoder (SDA-MDS), unknown element

1.3.1.8 Type 8: Error messages of the Job Manager (optional)

The error messages are generated during execution of Job Manager commands.

These error messages contain the following additional diagnosis data:

Data format	Element	Contains
VAR_STRING	token	Token
FILE_NAME	file_name	Filename
UNS32	interp_no	Interpreter number
UNS32	line_no	Line number

UNS32	column	Column
UNS32	Offset	Offset
SGN32	command_no	Number of command
SGN32	part_id	Line number
SGN32	step_id	Step number
SGN32	if_level	Level of if construct

1.3.1.9 Type 9: General error messages

This class contains general error messages that could not be assigned to any other class. No further specific diagnostic data is available here.

In particular, this error class is used for system messages that cannot occur during regular operation.

1.3.1.10 Type 10: Error messages of the PLC

If error messages are generated by the PLC and sent to the CNC via the HLI, they are of type 10. There is no type specific content.

1.3.1.11 Type 11: Error messages from position controller

If an error occur in the position controller, the reference to the NC program is established via the additional parameters.

These error messages contain the following additional diagnostic data:

Data format	Element	Contains
UNS16	log_pfad_nr	Logical path number
PROG_NAME	prog_name	Program name
FILE_NAME	file_name	File name
UNS32	satz_nr	NC block number

Practical examples:

- Following error
- Missing measuring signal

1.3.2 Additional diagnostic values, overview of possible meanings

Each additional value of an error message contains an identifier that describes the meaning of that value:

Flag	Description	Example
1	**Limit value**	Following error monitoring limit
2	**Current value**	Current position lag
3	**Error value**	Unknown mode
4	**Expected value**	Necessary braking distance at the current speed
5	**Corrected value**	Due to incorrect parameterisation, a parameter is corrected during controller start-up.
6	**Logical axis number**	Axis-specific error messages
7	**Drive type**	Drive type-specific error message (SERCOS, +-10V)
8	**Logical control element number**	Setting an unknown operator control
9	**State**	Unknown internal state; change to an unknown mode
10	**Transition**	Unknown transition specified
11	**Sender**	Error in internal communication
12	**Class**	Too many variables of one specific class are to be created.
13	**Instance**	Unknown gear number
14	**Identification number**	Unknown SERCOS identification number
15	**Status**	Currently not used
16	**Ring number**	Currently not used
17	**NC block number**	10 for N10
18	**Lower limit value**	Negative software limit switch
19	**Upper limit value**	Positive software limit switch
20	**Initial value**	Starting position of Z coordinate for helical interpolation
21	**Target value**	Target position of Z coordinate for helical interpolation

1.3.3 Dimensions identifiers

The following identifiers have been introduced to simplify the presentation of diagnostic values:

Flag	Description	Example
0	**Dimensionless** No dimension is available for the value provided.	Index, internal computing quantity
1	**Position** in 0,1 µm or 0,0001 °	Travel over a software limit switch
2	**Velocity** in µm / s or 0,001 ° / s	Programming of a negative speed
3	**Acceleration** in mm / s^2 or ° / s^2	Acceleration exceeds the maximum permitted axis acceleration
4	**Jerk** in mm / s^3 or ° / s^3	Exceeding of the maximum permitted axis jerk
5	**Time** in µs	Inadmissible assignment of the cycle or ramp time
6	**Percentage** as a percentage factor with in 0.1 %	Negative percentage acceleration weighting specified
7	**Address**	Inadmissible address, access to address
8	**Increments** The value is a position in the dimension increments.	Inadmissible grid of the compensation value intervals in the case of SSFK
9	**Feed rate per revolution** in 0.1 µm / rev.	Programmed feed rate per revolution low small
10	**Cutting speed** in µm / s	Programmed cutting speed too small
11	**Path resolution** in increments / 0.1 µm	Current path resolution of an axis
12	**Increments per revolution** The value is the number of increments per revolution, the measurement system is providing	Feed forward control of profibus drives
13	**Byte** The value is the minimum number of bytes required for certain functionalities.	Configurable memory for forward-/backward on path
14	**Proportional gain**	Kv-factor in the position control loop
15	**Frequency** in Hz	Filter cut off frequency is zero (nil).
16	**Motor load** in kg or kg*m^2	Invalid value for motor load.

The specified units can be used to display error messages.

1.4 Presentation of an error message

Information supplied with an error message must be displayed in a suitable way to assist the operator to trace the error cause.

1.4.1 Error message from NC program decoding

In the following example, the error information features simple formatting. It shows an error message in which an unknown G function has been programmed.

```
Version: V253_02R.001    Module: DIN_FKT1.C    (Date/Time): 0/144836
--------------------------------------------------------------------
Unknown G function.
--------------------------------------------------------------------
Error ID  : 20131       BF type: 9                   Channel ID: 1
Multiple ID: 0           Line  : 8605                Commu ID: 32
Recovery class: 2       Reaction class: 2
----------------------- Root type:  1 ---------------------------------
NC file    : log. path no. 9 -> timer1.nc
NC program: Timer1
Block number : 30          File offset: 72               Block offset: 29
-------------------------- NC block -------------------------------------
                        >     ?
N030 G91 X10 Y200 F100 G1234 G1
------------------ Output of values 1->5 -----------------------------
Wert_1: current value is 1234 [-]
--------------------------------------------------------------------
```

Besides the display of internal control information, the following elements are important for diagnosis:

- Unique error ID, in this case 20131
- remedy class and reaction class
- name of the NC program
- Block number in the NC program
- and, if applicable, file and block offsets

The incorrect location can be marked clearly by a suitable evaluation and presentation of the file and block offsets.

1.4.2 Error message from look-ahead

The following example shows an error message generated when programming a position outside the permissible travel range.

```
Version: V253_02R.001    Module: BAVO_FIN.C    (Date/Time): 0/253852
------------------------------------------------------------------------
Travel over positive software limit switch
------------------------------------------------------------------------
Error ID  : 120002       BF type: 5                  Channel ID: 1
Multiple ID: 0           Line  : 2239                Commu ID: 50
Recovery class: 5       Reaction class: 2
----------------------- Root type:  1 --------------------------------
NC file    : log. path No. 9 -> t_err.nc
NC program: Test
Block number : 20          File offset: 42             Block offset: 0
------------------------- NC block -------------------------------------
*
N020 X2345 Y15
------------------ Output of values 1->5 -----------------------------
Value_1: current value is 23450000.000000 [0.1*10^-3 mm or °]
Value_2: limit value is 15000000.000000 [0.1*10^-3 mm or °]
Value_3: logical axis number is 1
------------------------------------------------------------------------
```

Besides the display of internal control information, the following elements are important for diagnosis:

- Unique error ID, in this case 120002
- remedy class and reaction class
- name of the NC program
- program and block number in the NC program

From the values, we can tell that the problem lies in the axis with the logical axis number 1, that the limit (of the limit switches) is at the position 1.500 mm and that the target position is 2.345 mm.

2 Overview of CNC error messages

2.1 General error, start-up error (ID range 1000-12999)

2.1.1 ID-range 1000-1249

ID 1030

Index value out of permissible range.			
Description	In the start-up list, the specified axis index or channel index is greater than permissible. The maximum permissible number of channels and axes can be found in the system parameter documentation[SYSP]. Example (from start up list): sda_mds[**123**] ..\..\listen\sda_mds.lis ... achs_mds[**230**] ..\..\listen\achsmd.lis Correct: sda_mds[**11**] ..\..\listen\sda_mds.lis ... achs_mds[**63**] ..\..\listen\achsmds.lis		
Response	Class	3	The affected parameter list is not evaluated
Solution	Class	7	Correct the start up list
Parameter	%1:	Current value [-]	
		Incorrect axis or channel index	
	%2:	Limit value [-]	
Error type	-		

ID 1031 / 1032

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 1033

Unknown state in office function ascii_wr_regie().			
Description			
Response	Class	3	
Solution	Class	7	

Error type	5, Error message by access on files.

ID 1034

Unknown dimension id.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 1035

User-specific message via #MSG to the event logger.			
Description	The #MSG command generated this message using `#MSG EVENT_LOGGER[“text”]` . The text is displayed in the specified parameters		
Response	Class	1	Message output
Solution	Class	1	-
Parameter	%1:		
		Text output	
	%2:		
		Text addition	
	%3:		
		Text addition	
	%4:		
		Text addition	
	%5:		
		Text addition	
Error type	-		

ID 1036

Message #MSG to the event logger is only possible without acknowledgement.			
Description	Modes with acknowledgements are not permitted when programming a message to the Event Logger with the #MSG command. Modes with acknowledgement are ACK or SYN_ACK. The syntax is described in Programming a message to the Event Logger. For example: `#MSG EVENT_LOGGER["Text"]`		
Response	Class	1	Output of warning; the acknowledgement request is deleted.
Solution	Class	1	Check and modify the NC program; correct the programmed #MSG command mode.
Error type	1, Error message from NC program.		

2.1.2 ID-range 1500-1749

ID 1527 - 1598

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 1599

Not enough path in motion block for position controlled stop.			
Description	After activation of measurement signal via measuring probe, residual path P-CHAN-00030 defined in channel parameter list is reduced with active function edge banding. The residual path available of current active channel block at this moment must be greater than P-CHAN-00030. Depending on setting of P-CHAN-00029 the residual path is reduced by one or more blocks. Braking distance after activation of probing sensor depends on dynamic settings in axis machine data! For further information, see PROG// Section: Edge banding.		
Response	Class	6	Program abort.
Solution	Class	7	Check whether there is sufficient block residual path available in the NC program after activating the measuring probe. Increase residual path or, if possible, reduce the value of channel parameter P-CHAN-00030.
Parameter	%1:	Expected value [0.1 µm or 0.0001°]	
		Current braking distance.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current residual distance in the block.	
Error type	1, Error message from NC program.		

ID 1600 - 1623

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 1624

Inadmissible feed.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 1625

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 1626

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 1700

Velocity 0 is programmed.			
Description	The programmed velocity when commissioning a single axis via the PLC is 0. This is not permitted.		
Response	Class	0	Movement stop.
Solution	Class	0	Correction oft the programmed velocity in the commanded function block.
Parameter	%1:		
	Logical axis number of single axis P-AXIS-00016		
	%2:		
	Block number		
	%3:		
	Function block ID of PLCopen Part 1		
Error type	-		

ID 1701

Single axis is commanded but not configured in axis data list.			
Description	The commissioned axis is not configured as a single axis. To use the axis as a single axis, parameter P-AXIS-00457 must be set.		
Response	Class	0	Movement stop.
Solution	Class	0	Check and enable the axis parameter P-AXIS-00457
Parameter	%1:		
	Logical axis number of single axis P-AXIS-00016		
	%2:		
	Block number		
	%3:		
	Function block ID of PLCopen Part 1		
Error type	-		

ID 1702

Limiting of motion path of the single axis to the positive software limit switch.			
Description	The assigned path of the single axis is greater than the positive software limit switch P-AXIS-00178 of the axis. The path is limited to the value of the positive software limit switch.		
Response	Class	0	Movement stop.
Solution	Class	0	Correction of the commanded motion path or modification of the software limit switch P-AXIS-00178..
Parameter	%1:		
		Logical axis number of single axis P-AXIS-00016	
	%2:		
		Necessary braking distance	
	%3:		
		Current position of single axis	
	%4:		
		Value of the positive software limit switch P-AXIS-00178	
Error type	-		

ID 1703

Limiting of motion path of the single axis to the negative software limit switch.			
Description	The assigned path of the single axis is greater than the negative software limit switch P-AXIS-00177 of the axis. The path is limited to the value of the negative software limit switch.		
Response	Class	0	Movement stop.
Solution	Class	0	Correction of the commanded motion path or modification of the software limit switch P-AXIS-00177..
Parameter	%1:		
		Logical axis number of single axis P-AXIS-00016	
	%2:		
		Necessary braking distance	
	%3:		
		Current position of single axis	
	%4:		
		Value of the negative software limit switch P-AXIS-00177	
Error type	-		

ID 1704

Buffer allocation for single axis command failed.			
Description	Commanding a job for a single axis is not possible because no buffer is available. Possible cause is that previously many jobs were commanded but not yet processed.		
Response	Class	0	Movement stop.
Solution	Class	0	Reduce number of jobs for the single axes.

Parameter	%1:	
		Logical axis number of single axis P-AXIS-00016
	%2:	
		Block number
	%3:	
		Function block ID of PLCopen Part 1
Error type	-	

ID 1705

Buffer allocation for job acknowledge of single axis failed.			
Description	There is no buffer available for the acknowledgement of the job of a single axis. This is a system error of the CNC that the user cannot generate or rectify himself.		
Response	Class	0	Movement stop.
Solution	Class	0	Internal error of CNC – system error
Parameter	%1:		
			Logical axis number of single axis P-AXIS-00016
	%2:		
			Block number
	%3:		
			Function block ID of PLCopen Part 1
Error type	-		

ID 1706

Buffer release for single axis command failed.			
Description	The buffer location for a single axis command could not be released. This is a system error of the CNC that the user cannot generate or rectify himself.		
Response	Class	0	Movement stop.
Solution	Class	0	Internal error of CNC – system error
Parameter	%1:		
			Logical axis number of single axis P-AXIS-00016
	%2:		
			Block number
	%3:		
			Function block ID of PLCopen Part 1
Error type	-		

ID 1708

Writing into job acknowledge FIFO failed.			
Description	It is not possible to write a job acknowledgement. Possible cause is that the PLC has not read the existing acknowledgements in the corresponding FIFO.		
Response	Class	0	Movement stop.
Solution	Class	0	Check sequence in the PLC. Read the acknowledgement FIFOs

Parameter	%1:	
		Logical axis number of single axis P-AXIS-00016
	%2:	
		Job number
	%3:	
		Function block ID of PLCopen Part 1
Error type	-	

ID 1709

Single axis interpolation for active transformation axis not possible.			
Description	An active transformation axis cannot be traversed as a single-axis motion. Before commanding the motion, deselect the transformation (#CS OFF or #TRAFO OFF).		
Response	Class	2	Path motion and single-axis motion are stopped. Transition to error state.
Solution	Class	3	Check and modify the NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Block number [-]	
		Block number of single-axis job	
	%3:	Current value [-]	
	%4:	Current value [-]	
		Data item signals whether axis participates in transformation.	
Error type	1, Error message from NC program.		

2.1.3 ID-range 2000-2249

ID 2009

Index exceeds permitted range.			
Description	When a parameter was specified in a parameter list, it was specified by an array. The index specified for this array is greater than the maximum index entry permitted for this data item. Example with error in channel parameter list: `m_synch[12345]  2` The maximum permissible index for the above data item is 999.		
Response	Class	3	Abort current job.Start-up is resumed. No access can be made to the parameter with the out-of-range index.
Solution	Class	7	Check and modify the specified parameters; correct the index.
Parameter	%1:	Current value [-]	
		Specified index	
	%2:	Upper limit value [-]	
		Maximum permissible index	
Error type	6, Error message by data transfer to control device.		

ID 2023

Specified string is too long.			

Description	A specified character string is too long. For example, this may happen for a string valued parameter in a list configuration file.
	Example in a channel list (sda_mds):
	`…`
	`# Identifiers of axis groups are limited to 16 characters.`
	`# Hence, the following line causes the error 2023.`
	`gruppe[0].bezeichnung AXIS_GROUP_123456789`
	`…`		

Response	Class	3	In start-up: Start-up is continued. Otherwise: Behaviour depends on the specific situation.
Solution	Class	7	Shorten the respective string value.
Error type	6, Error message by data transfer to control device.		

ID 2024

Specified file name is empty.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 2030

Internal error of xml parser.			
Description	An internal error occurred within the xml parser.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Restart the controller.
Error type	6, Error message by data transfer to control device.		

ID 2031

XML parser out of memory.			
Description	There is not enough memory available to interpret the XML file.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Restart the controller.
Error type	6, Error message by data transfer to control device.		

ID 2032

Document start or document end tag is missing or wrong.			
Description	Document start or document end tag is missing or wrong.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Check and correct start/end tag of XML file.
Error type	6, Error message by data transfer to control device.		

ID 2033

Unknown or unsupported character encoding.			
Description	Unknown or unsupported character encoding.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Check XML file for unknown or unsupported encoded characters and correct it.
Error type	6, Error message by data transfer to control device.		

ID 2034

Unknown or wrong attribute or attribute redefined.			
Description	Unknown or wrong attribute or attribute redefined.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Check and correct attributes.
Error type	6, Error message by data transfer to control device.		

ID 2035

Error in comment.	
Description	XML file contains corrupted comment.

Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Check and correct corrupted comment.
Error type	6, Error message by data transfer to control device.		

ID 2036

XML key word used as element or attribute name.			
Description	XML key words must not be used as element or attribute names.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Correct wrong element or attribute.
Error type	6, Error message by data transfer to control device.		

ID 2037

Start and end tag do not fit.			
Description	Start and end tag must have the same name.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Check and correct start/end tag.
Error type	6, Error message by data transfer to control device.		

ID 2038

XML document is not well balanced.			
Description	XML document is not well balanced.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Check and correct XML document.
Error type	6, Error message by data transfer to control device.		

ID 2039

General error in XML document.			
Description	General error in XML document.		
Response	Class	3	Interpretation of XML file is aborted.
Solution	Class	7	Check and correct XML document.
Error type	6, Error message by data transfer to control device.		

ID 2040

Error at parameter access, parameter not found.			
Description	The specified parameter could not be found.		
Response	Class	2	Reading/writing of parameter is aborted.
Solution	Class	6	Check and correct parameter.

Parameter	%1:	Error value [-]
		Wrong parameter id.
	%2:	Current value [-]
		Parameter domain
Error type	6, Error message by data transfer to control device.	

ID 2041

Error at parameter access, unknown dimension.			
Description	The given dimension does not belong to the parameter.		
Response	Class	5	Reading/writing of parameter is aborted.
Solution	Class	6	Check and correct dimension.
Parameter	%1:	Error value [-]	
		Wrong parameter dimension	
	%2:	Current value [-]	
		Parameter ID	
Error type	6, Error message by data transfer to control device.		

ID 2042

Error when reading parameter value.			
Description	The value to the specified parameter id cannot be read.		
Response	Class	5	Reading the parameter value is aborted.
Solution	Class	6	Check the parameter and the parameter values
Parameter	%1:	Current value [-]	
		Parameter ID	
Error type	6, Error message by data transfer to control device.		

ID 2043

Error when writing parameter value.			
Description	The value to the specified parameter id cannot be written.		
Response	Class	5	Writing the parameter value is aborted.
Solution	Class	6	Check the parameter and the parameter values
Parameter	%1:	Current value [-]	
		Parameter ID	
Error type	6, Error message by data transfer to control device.		

ID 2044

Referenced parameter is obsolete.			
Description	The specified parameter id is obsolete.		
Response	Class	5	Reading/writing of parameter is aborted.
Solution	Class	3	Check documentation for newer parameter id.

Parameter	%1:	Error value [-]
		Obsolete id of the parameter
	%2:	Current value [-]
		Parameter domain
Error type	6, Error message by data transfer to control device.	

ID 2045

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 2046

Target string is too long.			
Description	The specified string is too long for the target string.		
Response	Class	1	Reading/writing of parameter is aborted.
Solution	Class	1	Correct the length of the string.
Parameter	%1:	Current value [-]	
		ID of the parameter	
	%2:	Upper limit value [-]	
		Maximum permissible length of the string	
	%3:	Error value [-]	
		Current length of the string	
Error type	6, Error message by data transfer to control device.		

ID 2047

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 2048

Parameter reference already exists.			
Description	Parameter reference cannot be created because it already exists.		
Response	Class	-	Parameter reference is not created.
Solution	Class	-	Correct parameter reference.
Error type	6, Error message by data transfer to control device.		

ID 2049

Parameter units do not match.			
Description	The unit of the parameter does not match the unit of the parameter reference.		
Response	Class	-	Reading/writing of parameter is aborted.
Solution	Class	-	Check and correct parameter unit.
Parameter	%1:	Error value [-]	
		Invalid or incorrect parameter	
	%2:	Error value [-]	
		Wrong parameter unit	
	%3:	Current value [-]	
		ID of the parameter reference	
	%4:	Expected value [-]	
		Expected unit of the parameter	
	%5:	Current value [-]	
		Parameter domain	
Error type	6, Error message by data transfer to control device.		

ID 2050

Parameter reference table is not initialized.			
Description	The table containing the paramteter references has not been initialized. The configuration files will be normally read but parameter references cannot be uses. This can result in further error messages.		
Response	Class	-	Parameter references cannot be created.
Solution	Class	-	Restart the controller.
Error type	-		

ID 2051

Reference points to itself.			
Description	The specified parameter reference points to itself.		
Response	Class	-	Reading/writing of parameter is aborted.
Solution	Class	-	Correct parameter reference.
Parameter	%1:	Error value [-]	
		Current parameter id [-].	
	%2:	Current value [-]	
		Parameter range affected	
Error type	6, Error message by data transfer to control device.		

ID 2052

Non-resolvable reference found.			
Description	The reference cannot be resolved and may possibly not exist.		
Response	Class	-	Reading/writing of parameter is aborted.
Solution	Class	-	Check the parameter reference.

Parameter	%1:	Error value [-]
		Wrong parameter reference.
	%2:	Current value [-]
		Parameter domain affected
Error type	6, Error message by data transfer to control device.	

ID 2053

No more elements available in reference table.			
Description	Cannot create further parameter references.		
Response	Class	-	Parameter reference is not created.
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Number of references	
	%2:	Limit value [-]	
		Maximum number of references.	
Error type	-		

ID 2054

Token number not valid.			
Description	Token number could not be found in token table.		
Response	Class	3	Reading/writing of parameter is aborted.
Solution	Class	7	Restart the controller.
Parameter	%1:	Current value [-]	
	%3:	Current value [-]	
Error type	6, Error message by data transfer to control device.		

ID 2055

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Reading/writing of parameter is aborted.
Solution	Class	8	Requires controller restart.

ID 2056

The parameter already has a valid value.			
Description	A value was already assigned to the parameter. Additional assignments overwrite the previous values.		
Response	Class	1	No error response
Solution	Class	1	Check and correct parameters.

Parameter	%1:	Parameter name [-]
		Name of parameter
	%2:	Parameter name[-]
		Parameter ID
	%3:	Current value [-]
		Current value of the parameter.
	%4:	Final value [-]
		Parameter target value.
	%5:	Domain parameters [-]
		Parameter domains.
Error type	-	

ID 2057

Table with parameter information is not initialised.			
Description	The table containing the parameter references has not been initialised. The configuration files will be read as normal but no parameter references can be used. This can result in further error messages.		
Response	Class	-	Parameter references cannot be created.
Solution	Class	-	Restart the controller.
Error type	-		

ID 2058

Table of parameter information has no more free elements.			
Description	Cannot create further parameter references.		
Response	Class	-	Parameter reference is not created.
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Number of references	
	%2:	Limit value [-]	
		Maximum number of references.	
Error type	-		

ID 2059

Error while decrypting parameters.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	6, Error message by data transfer to control device.		

ID 2062

Parameter reference has the wrong data type.

Description	When using parameter references, the data type of the parameter and the reference parameter must be identical.		
Response	Class	3	Abort current job. There is no use of the reference parameter.
Solution	Class	7	Check and modify the parameter used
Parameter	%1:	Expected value [-]	
		Data type of the reference parameter	
	%2:	Error value [-]	
		Data type of the parameter	
Error type	-6, Error message by data transfer to control device.		

ID 2063

Parameter reference has no assigned value.			
Description	The parameter to be referenced has no assigned value. A reference must be assigned a valid value in order to be valid.		
Response	Class	3	Abort current job. No reference created
Solution	Class	7	Check and modify the parameter to be referenced.
Parameter	%1:	Parameter name [-]	
		Name of the parameter containing the reference.	
	%2:	Current value [-]	
		ID of the parameter containing the reference.	
	%3:	Domain parameters [-]	
		Parameter domains	
Error type	6, Error message by data transfer to control device.		

ID 2065

Parameter reference does not exist.			
Description	The parameter to be referenced does not exist.		
Response	Class	3	Abort current job. No reference created
Solution	Class	7	Check and modify the parameter to be referenced.
Parameter	%1:	Error value [-]	
		Wrong parameter reference	
	%2:	Domain parameters [-]	
		Parameter domains	
Error type	6, Error message by data transfer to control device.		

ID 2100

The given diagnosis file could not be opened for writing.			
Description	The file specified with the NC command #DIAGNOSIS [PATH=…] cannot be opened for writing. Possible causes: • The file path specified is incorrect • The file is write protected		
Response	Class	2	Abort NC program processing.

Solution	Class	7	Check and modify the file path and the write authorisations for the specified file.
Error type	6, Error message by data transfer to control device.		

2.1.4 ID-range 3000-3249

ID 3200 - 3209

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 3210

Requested axis / gear / interface is not available in the system.			
Description	The message is displayed: • during controller start-up, if in channel parameter list (SDA-Parameters) at assigning axes to an axis group, a parameter P-CHAN-00035 contains a logical axis number, which isn't found as value of parameter P-AXIS-00016 in the axis parameter lists. • during NC program execution, if in an axis exchange command (see PROG // Section: Requesting axes, e.g. #CALL AX) a logical axis number is used but was not used in any of the axis lists for the parameter P-AXIS-00016. • at gear switching, if the programmed gear or by the spindle range table selected gear stage is not found in the axis data of the axis (P-AXIS-00135).		
Response	Class	3	At startup: NC channel where the wrong parametrization happened didn't get any axes. NC program processing: Abort Program.
Solution	Class	6	Use of a logical axis number which was used to parameterise axes.
Parameter	%1:	Logical axis number [-]	
		Logical axis number which was specified in the parameter P-CHAN-00035 or it was used in the axis exchange command but was not used to parameterise the axis.	
Error type	-		

ID 3213 - 3215

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 3314

There are no open axis reference requests.			
Description			
Response	Class	2	
Solution	Class	8	

Parameter	%1:	Logical axis number [-]
Error type	3, Error in the communication.	

ID 3315

The last order was no axis reference request.			
Description			
Response	Class	2	
Solution	Class	8	
Parameter	%1:	Logical axis number [-]	
Error type	3, Error in the communication.		

ID 3316

Received axis reference was not requested.			
Description			
Response	Class	2	
Solution	Class	8	
Parameter	%1:	Logical axis number [-]	
Error type	3, Error in the communication.		

2.1.5 ID-range 3250-3499

ID 3300 - 3316

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

2.1.6 ID-range 4000-4500

ID 4000 – 4118

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 4119

The tool radius or the maximum infeed in XY is less than or equal to zero..			
Description	The tool radius or the maximum infeed in XY was transferred with a value less than or equal to zero.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Select a tool with a greater radius or increase the maximum infeed in XY.
Error type	1, Error message from NC program.		

ID 4120

The available memory is not sufficient for the contour calculation.			
Description	The available contour memory is not sufficient to add contour elements or to calculate the contour path. Additional memory space is reserved by P-CHAN-00467.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check, modify or set P-CHAN-00467, increase the memory space. If the error occurs when a contour milling cycle is called, the required memory can be reduced by increasing the tool radius or increasing the maximum infeed in XY.
Parameter	%1:	Identifier [-]	
		Description value 2	
	%2:	Current value [-]	
		Currently used memory	
Error type	1, Error message from NC program.		

ID 4121

The radius for removing the residual material is incorrectly defined.			
Description	The tool radius in the previous reference cycle is too large. It was not possible to calculate a removal with it.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Possible solutions: • Use a tool with a smaller tool radius in the reference cycle. • Use a suitable contour milling cycle without residual material removal to completely re-mill the contour.

Error type	1, Error message from NC program.

ID 4122

No contours were selected for machining.			
Description	No contour was activated. Machining is therefore impossible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Activate a contour before machining with the command #CONTOUR LOAD.
Error type	1, Error message from NC program.		

ID 4123

A full circle has been defined.			
Description	During contour definition a full circle was defined or a circular element that is too short.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour definition. • If a full circle was defined, divide the full circle into several circle elements. • If the circle element is too short, increase the element length.
Parameter	%1:	Identification number [-]	
		File offset of the circle element definition.	
	%2:	Block number [-]	
		Block number of the circle element definition.	
	%3:	Current value [0.1 µm or 0.0001°]	
		Length of the circle element.	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Minimum permitted length of a circle element.	
Error type	1, Error message from NC program.		

ID 4124

The defined contour element is too short.			
Description	The contour definition contains a contour element that falls below the minimum permissible value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour, increase the length of the affected contour element.
Parameter	%1:	Identification number [-]	
		File offset of the circle element definition.	
	%2:	Block number [-]	
		Block number of the circle element definition.	
	%3:	Current value [0.1 µm or 0.0001°]	
		Length of the contour element.	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Minimum permitted length of a contour element.	

Error type	1, Error message from NC program.

ID 4125

The defined contour is too short.			
Description	The contour definition contains a contour that falls below the minimum permissible length.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour, increase the length of the contour.
Parameter	%1:	Identification number [-]	
		Identification number of the contour.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Length of the contour.	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Minimum permitted length of a contour.	
Error type	1, Error message from NC program.		

ID 4126

The defined contour is too long.			
Description	The contour definition contains a contour that exceeds the maximum permissible length.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour, reduce the length of the contour.
Parameter	%1:	Identification number [-]	
		Identification number of the contour.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Length of the contour.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Maximum permitted length of a contour.	
Error type	1, Error message from NC program.		

ID 4127

The defined island is invalid.			
Description	An island contour transferred for machining is not located within the outer boundary contour or overlaps another island contour.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the transferred island contour.
Parameter	%1:	Identifier [-]	
		Description value 2	
	%2:	Identification number [-]	
		Identification number of the invalid island contour.	
	%3:	Identifier [-]	
		Description value 3	
	%4:	Identification number [-]	
		Identification number of the overlapping island contour (in case of an overlap).	

Error type	1, Error message from NC program.

ID 4128

In spigot milling, the island must not overlap the blank contour.			
Description	A transferred island contour for spigot milling overlaps the blank contour. This is not possible with spigot milling.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the given island contour.
Parameter	%1:	Identifier [-]	
		Description value 2	
	%2:	Identification number [-]	
		ID number of the invalid island contour.	
Error type	1, Error message from NC program.		

ID 4129

The defined contour intersects itself.			
Description	The defined contour contains two intersecting contour elements.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the defined island contour.
Parameter	%1:	Identification number [-]	
		Identification number of the contour.	
	%2:	Identification number [-]	
		File offset of the first contour element involved.	
	%3:	Block number [-]	
		Block number of the first contour element involved.	
	%4:	Identification number [-]	
		File offset of the second contour element involved.	
	%5:	Block number [-]	
		Block number of the second contour element involved.	
Error type	1, Error message from NC program.		

ID 4130

The defined contour does not exist.			
Description	A contour identification number transferred for machining has no related contour.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Adapt the given contour identification number or create the required contour using #CONTOUR BEGIN / #CONTOUR END.
Parameter	%1:	Identifier [-]	
		Description value 2	
	%2:	Identification number [-]	
		Transferred contour identification number.	
Error type	1, Error message from NC program.		

ID 4131

The defined contour does not contain enough contour elements.			
Description	The number of contour elements in the defined contour is below the minimum limit.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour, increase the number of contour elements.
Parameter	%1:	Identification number [-]	
		Contour identification number.	
	%2:	Current value [-]	
		Number of contour elements.	
	%3:	Lower limit value [-]	
		Minimum permitted number of contour elements.	
Error type	1, Error message from NC program.		

ID 4132

The start and end points of the contour are not identical.			
Description	A contour transferred for machining has start and end points that are not identical. A closed contour is required for machining.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour. Start and end points may not be identical.
Parameter	%1:	Identifier [-]	
		Description value 2	
	%2:	Identification number [-]	
		Given contour identification number.	
Error type	1, Error message from NC program.		

ID 4133

The contour has been selected several times.			
Description	The same contour was transferred for machining several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the transferred contour identification numbers.
Parameter	%1:	Identifier [-]	
		Description value 2	
	%2:	Identification number [-]	
		Transferred contour identification number.	
Error type	1, Error message from NC program.		

ID 4134

The defined approach movement leads to a collision with the contour.			
Description	The transferred parameters create an approach movement leading to a collision with the contour.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the approach movement parameters.

Error type	1, Error message from NC program.		

ID 4135

The defined retract movement leads to a collision with the contour.			
Description	The transferred parameters create a retract movement leading to a collision with the contour.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the retract movement parameters.
Error type	1, Error message from NC program.		

ID 4136

No suitable approach and departure movements could be determined for the defined contour.			
Description	No suitable start and end points were determined for the transferred path milling contour. This occurs when the start and end points lie close to other contour elements and cause intersections when tool radius compensation is active,.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Possible solutions: • Modify the transferred contour. • Deactivate tool radius compensation.
Error type	1, Error message from NC program.		

ID 4137

The starting point of a contour must be defined by a linear motion.			
Description	The starting point of the contour must be defined by a linear motion (G0/G1). A definition with a circular motion (G2/G3) is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Definition of the contour starting point by a linear motion.
Parameter	%1:	Identification number [-]	
		File offset of the starting point definition.	
	%2:	Block number [-]	
		Block number of the starting point definition.	
Error type	1, Error message from NC program.		

ID 4138 – 4141

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 4142

The available memory is not sufficient to save additional patterns.			
Description	The available contour memory is not sufficient to add more pattern elements. Memory space is allocated by P-CHAN-00479.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Increase memory space with P-CHAN-00479.
Parameter	%1:	Identification number [-]	
		Description of value 2	
	%2:	Current value [-]	
		Currently used memory.	
Error type	1, Error message from NC program.		

ID 4143

The specified pattern does not exist.			
Description	The transferred pattern identification number or creation of the required pattern using #PATTERN BEGIN / #PATTERN EN		
Response	Class	3	Abort current job.
Solution	Class	2	Adapt the transferred contour identification number or create the required contour using #PATTERN BEGIN / #PATTERN END.END.
Parameter	%1:	Identification number [-]	
		Transferred pattern identification number.	
Error type	1, Error message from NC program.		

ID 4144

The defined contour does not contain enough pattern elements.			
Description	The number of pattern elements is below the minimum limit of pattern elements.		
Response	Class	3	Abort current job.
Solution	Class	2	Increase the number of pattern elements.
Parameter	%1:	Identification number [-]	
		Pattern identification number.	
	%2:	Current value [-]	
		Number of pattern elements.	
	%3:	Lower limit value [-]	
		Minimum permitted number of pattern elements.	
Error type	1, Error message from NC program.		

ID 4145

Incorrect pattern ID.			
Description	No valid identification number was transferred for the pattern. It must have an integer value greater than zero.		
Response	Class	3	Abort current job.
Solution	Class	2	Transfer a valid identification number.

Parameter	%1:	Current value [-]
		Pattern identification number.
Error type	1, Error message from NC program.	

ID 4146

The specified pattern element does not exist.			
Description	When a call pattern was read out, an attempt was made to access a pattern element that does not exist. The transferred element index exceeds the number of pattern elements.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the transferred element index. It must be less than the number of pattern elements of the corresponding pattern.
Parameter	%1:	Identifier [-]	
		Pattern ID	
	%2:	Current value [-]	
		Transferred element index	
	%3:	Upper limit value [-]	
		Number of pattern elements	
Error type	1, Error message from NC program.		

2.1.7 ID-range 10000-10249

ID 10000 - 10024

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 10030

BF order could not be acknowledged.			
Description			
Response	Class	3	
Solution	Class	3	
Error type	-		

2.1.8 ID-range 11000-11249

ID 11016

Same object ID already in use.			

Description	One cause of error can be the generation of communication objects of V.E. variables with great arrays.
	
	Extract from the V.E. List containing error that occurred
	`#`
	`var[3].name VARIABLE_4`
	`var[3].type REAL64`
	`var[3].scope GLOBAL`
	`var[3].synchronisation TRUE`
	`var[3].array_size` **`3000`**
	`var[3].access_rights READ_WRITE`
	`Error case`
	`var[3].create_hmi_interface TRUE`
	`Correction`
	`var[3].create_hmi_interface FALSE`
	
	Alternatively, the number of communication objects declared in the V.E. list may be too big.		

Response	Class	3	Abort job processing
Solution	Class	7	Reduce number of required communication objects
Parameter	%1:	Current value [-]	
		Concerning ID Nr.	
	%2:	Error value [-]	
		Name of the concerning object	
	%3:	Current value [-]	
		Name oft he object with the existing object ID	
Error type	-		

ID 11020

Allocation of memory failed.			
Description	The requested memory is larger than the available memory in system.		
Response	Class	3	Job processing aborted
Solution	Class	7	Possible solutions: Reduce number of channels or axes to reduce requested memory Extension of the main storage of the computer.
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	-		

ID 11066

Write operation is not possible, previous write operation is not complete.

Description	The object write operation cannot be executed since a previous write operation was not completed for the same object. The last assigned writing operation is aborted and discarded. This message may occur, for example, when updating parameter lists. The reason for this is that distribution of the parameters in the controller requires some time. To avoid the message, the flow should be processed sequentially.		
Response	Class	3	None
Solution	Class	1	Modification of the process flow.
Parameter	%1:	Current value [-]	
		Object ID	
	%2:	Current value [-]	
	%3:	Limit value [-]	
	%4:	Limit value [-]	
	%5:	Current value [-]	
		Name of object.	
Error type	-		

ID 11072 / 11073

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 11074

Generate HMI interface with self-defined types of variables is not possible.

Description	It is not possible to generate communication objects for user-defined variable types.
	
	Extract from the V.E. list
	#
	`struct[0].name MY_TYPE`
	`struct[0].element[0].name FirstElement`
	`struct[0].element[0].type REAL64`
	..
	#
	`var[4].name Test_VAR`
	`var[4].type MY_TYPE`
	`var[4].scope GLOBAL`
	`var[4].synchronisation TRUE`
	`var[4].access_rights READ_WRITE`
	Wrong
	`var[4].create_hmi_interface TRUE`
	Correct
	`var[4].create_hmi_interface FALSE`
	#		

Response	Class	1	None
Solution	Class	1	Deactivation of the flag create_hmi-interface in the V.E. list
Parameter	%1:	Name of the affected V.E. variable	
Error type	-		

ID 11075 - 11078

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 11203

Error in start-up of OCCI-server of task INT.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 11204

Error in start-up of OCCI-server of task RND.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 11205

Error in start-up of OCCI-server of task COM.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 11206

Declared name of OCCI-items is already used.			
Description			
Response	Class	3	
Solution	Class	3	
Error type	-		

2.2 Cycle errors (13000-19999)

2.2.1 ID-range 13000-13249

ID 13100

Safety clearance not or incorrectly defined in XY.			
Description	The safety clearance limits the starting motion during rapid traverse. This message is generated if the safety clearance is programmed with 0 or not at all.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the safety clearance and correct the input parameters.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 13101

Multiple definition of machining depth.			
Description	The machining depth must be defined either as absolute or as relative. This message is generated if both input parameters for the machining depth are assigned for absolute and relative entries.		
Response	Class	-	Warning: Cycle is continued using the absolute machining depth.
Solution	Class	-	Define whether the machining depth is to be defined as absolute or relative and correct the input parameters accordingly.
Error type	1, Error message from the NC program		

ID 13102

Programmed machining depth exceeds permissible tool length.			
Description	The maximum useful tool length can be entered in the tool parameters (see Section: Cycle-specific tool data). If a value is unequal to 0, check whether the programmed machining depth is smaller than the permissible usable tool length. If not, the message is generated.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check whether the permissible tool length is defined correctly. If necessary, use a different tool for machining with a tool length that is adequate for the machining depth.
Error type	1, Error message from the NC program		

ID 13103

Number of infeeds not defined.	
Description	With multi-step work operations up to the programmed machining depth, the number of infeeds is checked in the cycle. This message is generated if the number of infeeds is programmed as 0 or not at all.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the number of infeeds and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13104

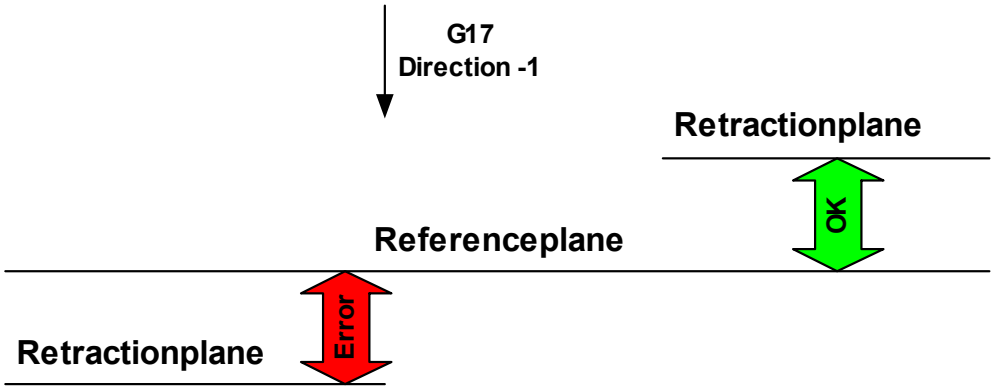
Infeed depth exceeds permissible machining stroke.			
Description	The maximum permissible machining stroke can be entered in the tool parameters (P[1], see section: Cycle-specific tool data). If a value is unequal to 0, a check is made as to whether the calculated infeed depth is less than the maximum permissible machining stroke. If not, the message is generated.		
Response	Class	-	Warning: The number of infeeds or the value of one machining stroke is adjusted internally in the cycle based on the maximum stroke and the cycle is continued.
Solution	Class	-	Accordingly, increase the number of infeeds in the input parameters or check whether the permissible machining stroke is correctly defined. If necessary, use a different tool for machining with a machining stroke that is adequate for the infeed depth.
Error type	1, Error message from the NC program		

ID 13105

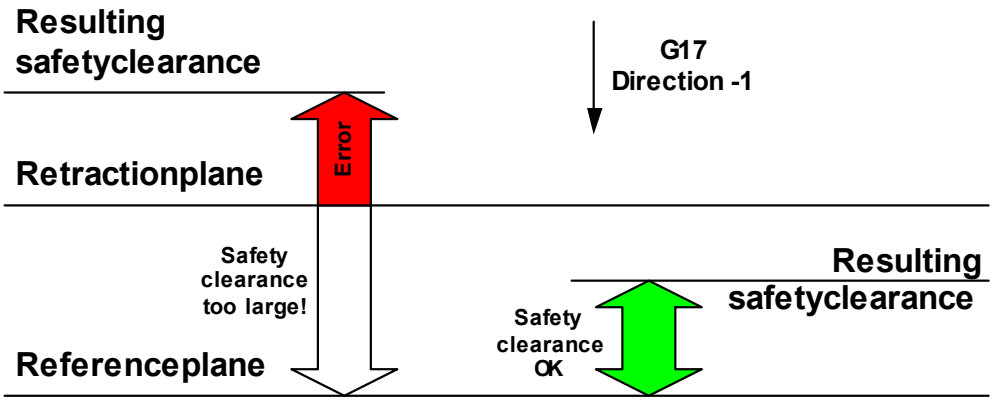
Return distance/holding distance not defined.			
Description	With step drilling, the drilling cycle requires a return distance or holding distance. This message is generated if this value is programmed as 0 or not at all.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the return or holding distance and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13106

Reference and retraction planes incorrectly defined.

Description	Depending on the direction, a check is made in the cycle whether the retraction plane comes before the reference plane. This message is generated if the distances are equal or the retraction plane lies below the reference plane.  		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the locations of the return and reference planes and correct the input parameters.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 13107

Incorrect definition of the safety clearance in Z.			
Description	Depending on the direction and based on the programmed safety clearance, a check is made in the cycle to determine whether the resulting safety plane lies below the retraction plane. This message is generated if the safety and retraction planes are equal or the safety plane lies above the retraction plane.  		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the safety clearance and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13108

No or unknown machining direction defined.

Description	The machining direction is required to define the right position and effect on the safety clearance. This message is generated if no machining direction is specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the machining direction and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13109

Unknown process defined.			
Description	Various processes are available for drilling. They are selected by specifying an identification number: 0: With one cut (only available for thread tapping! (default)), 1: Step drilling with chip breaking, 2: Step drilling with chip evacuation. This message is generated when an identification number other than 0, 1 or 2 was programmed in the corresponding data input parameters.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the identification number of the process and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13110

Machining depth not defined.			
Description	The machining depth must be defined relative to the reference plane. This message is generated if no input parameter is assigned as a relative value for the machining depth.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Program input parameters for machining depth.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 13111

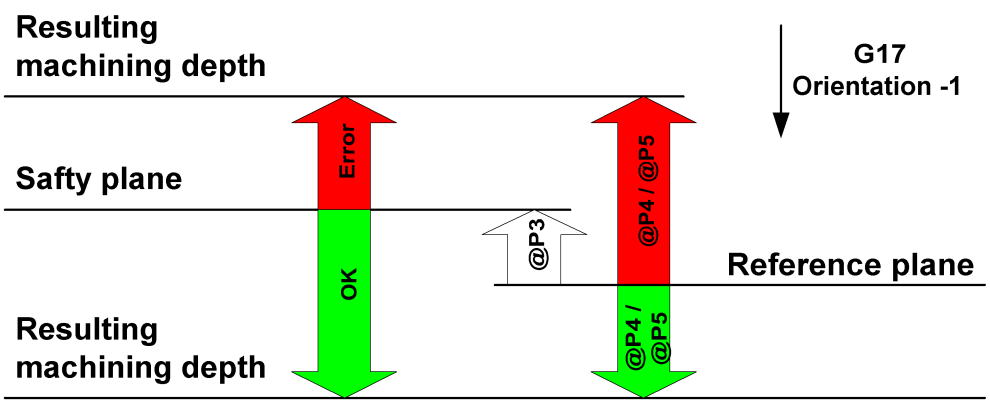
Relative machining depth is 0.			
Description	The machining depth must be defined either as absolute or as relative. This message is generated when the input parameter is assigned the value 0 while a relative machining depth is used.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	For a relative machining depth, program the input parameter with a value unequal to 0.

Error type	1, Error message from the NC program		

ID 13112

Retraction mode is unknown.			
Description	An invalid value was found in the optional parameter that defines the retraction process after the machining depth is reached. The following modes are available: 0 - Retract in rapid traverse in Z direction G00 (default), 1 - Retract at feed rate G01, not available in all cycles ->, 2 - Stop spindle at defined position M19 S.POS Retract at feed rate G01 in X and Z directions Retract at rapid traverse in Z direction G00 3 - Stop spindle M5, retract at rapid traverse in Z direction G00		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for retraction motion.
Error type	1, Error message from the NC program		

ID 13113

Incorrect definition of machining depth.			
Description	The machining depth is outside the safety plane. 		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the machining depth and the correct the input parameters.
Error type	1, Error message from the NC program		

ID 13114

Required license key is missing.			
Description	The invoked cycle requires a license key which is not on the CNC control.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Please contact your CNC control supplier.

Error type	1, Error message from the NC program

ID 13115

Use of cycle not permitted.			
Description	The invoked cycle is not enabled for use on the specified CNC control. Possible reasons for this restriction may be licensing conditions, for example.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Please contact your CNC control supplier.
Error type	1, Error message from the NC program		

2.2.1.1 ID 13116

A #CS call within the cycle is not possible with the current CNC version.			
Description	The CNC version is obsolete and does not permit a #CS call within the cycle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Update the CNC version
Error type	1, Error message from the NC program		

ID 13120

Invalid spindle position defined.			
Description	The spindle must be positioned between 0° and 360° for thread tapping. This message is generated when the spindle position was programmed outside this permissible range. Note: Optionally, the spindle position can also be pre-assigned in the tool parameters (see section: Cyclic-specific tool data).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the spindle position and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13121

Spindle speed not defined.			
Description	For thread tapping, the spindle speed must be specified to calculate the feed rate. This message is generated if the spindle speed is neither programmed in the input parameters nor pre-assigned in the tool parameters. (See section: Cyclic-specific tool data).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the spindle position and correct the input or tool parameters. If both are assigned, the spindle position from the input parameters is used.
Error type	1, Error message from the NC program		

ID 13122

Thread pitch not defined.			
Description	For thread tapping, the pitch must be specified to calculate the feed rate. This message is generated if the thread pitch is neither programmed in the input parameters nor pre-assigned in the tool parameters. (See section: Cyclic-specific tool data).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the pitch and correct the input or tool parameters. If both are assigned, the pitch from the input parameters is used.
Error type	1, Error message from the NC program		

ID 13123

Thread size not defined.			
Description	If a thread tapping drill with countersink (special tool (P[6], see Section: Cyclic-specific tool data) is used for machining, the metric thread size is needed to calculate countersink motion. This message is generated if the thread size is not assigned by default in the tool parameters. (P[7] see Section: Cyclic-specific tool data).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define thread size and correct the tool parameter P[7].
Error type	1, Error message from the NC program		

ID 13124

Thread type (@P__) for left-hand and right-hand thread is defined incorrectly.			
Description	Error in the definition of parameter @P__ for the thread type. @P__ must be -1 for left-hand thread or +1 for right-hand thread; other values are not permitted!		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct definition of thread type @P__: @P__ = -1 left-hand thread, @P__ = +1 right-hand thread (default)
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 13125

Number of infeeds for thread cutting (@P__) is incorrect, must be a positive integer.			
Description	Error in the definition of parameter @P__ for the machining mode. @P__ must be a positive integer.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the number of infeeds (@P__).
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 13126

The machining mode for threads (@P__) is defined incorrectly.			
Description	Error in the definition of parameter @P__ for the machining mode. Valid values for @P__ are: 0 - thread tapping to final drilling depth (default), 1 - chip breaking, 2 - chip evacuation		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the definition of thread type @P__.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 13127

Incorrect definition of the helical plunging movement.			
Description	The radius of the helical plunging movement was defined with a value less than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the radius of the helical plunging movement with a value greater than 0.
Error type	1, Error message from the NC program		

ID 13128

The measuring or calibration cycle must not be started with an active 0 offset.			
Description	A measuring or calibration cycle was called with an active zero offset (G54-G59). This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Deactivate the zero offset (G53) when the cycle is called. If necessary, activate the required offset by defining a machining coordinate system (CS).
Parameter	%1:	Current value	
		Active zero offset G.	
Error type	1, Error message from the NC program		

ID 13140

Diameter or thread diameter not defined.			
Description	For helical milling or thread milling, the thread diameter must be specified to calculate the helical motion. This message is generated if the diameter is not programmed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the diameter and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13141

Pitch not defined.			
Description	For helical milling and thread milling, a pitch must be specified to calculate helical motion. This message is generated if the pitch is not programmed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the pitch and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13142

Unknown machining direction defined.			
Description	The machining direction must be specified to calculate the milling motion.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the machining direction and correct the input parameters.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 13160

Pocket length not defined.			
Description	Pocket milling requires specification of the 1st pocket side (pocket length) to calculate countersinking motions. This message is generated if the pocket length is not programmed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the pocket length and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13161

Pocket width not defined.			
Description	Pocket milling requires specification of the 2nd pocket side (pocket length) to calculate countersinking motions. This message is generated if the pocket width is not programmed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the pocket width and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13162

Corner radius not defined.

Description	For pocket milling, it is necessary to specify the corner radius to calculate countersinking motions. This message is generated if the corner radius is not programmed.		
Response	Class	-	Warning: Cycle is continued by using the current (milling cutter) tool radius as the corner radius.
Solution	Class	-	Check and define the corner radius and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13163

Tool diameter too large.			
Description	A check is made whether the tool diameter of the current milling cutter is less than the programmed width or length of the pocket/slot, taking into account the finishing allowance. This message is displayed if the result is a violation of the contour.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Use a tool with a smaller diameter for machining to avoid violating the contour.
Error type	1, Error message from the NC program		

ID 13164

Finishing allowance is 0.			
Description	A finishing allowance must be specified for pocket milling with roughing operations to calculate countersinking motions. This message is generated if this value is programmed as 0 or not at all.		
Response	Class	-	Warning: Cycle is continued. Only the roughing operation is executed. A possibly commanded finishing operation is discarded.
Solution	Class	-	Check and define the finishing allowance and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13165

Unknown milling direction defined.			
Description	For pocket milling, it is necessary to specify a milling direction to calculate countersinking motions. This can be selected by specifying an identification number: 2: Clockwise milling (CW), 3: Counter-clockwise milling (CCW). This message is generated when an identification number other than 2 or 3 was programmed in the corresponding input parameters.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the milling direction and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13166

Unknown machining mode defined.			
Description	Pocket milling requires the specification of a machining mode to calculate countersinking motions. This can be selected by specifying an identification number: 1: Roughing, 2: Finishing, 3: Roughing + finishing. This message is generated when an identification number other than 1, 2 or 3 was programmed in the corresponding data input parameters.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the machining mode and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13167

Tool radius greater than corner radius.			
Description	A check is made as to whether the programmed corner radius can be manufactured with the current milling cutter. In this case, the tool radius must be less than or at maximum equal to the corner radius. If not, the message is generated.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Use a tool with a radius less than or equal to the programmed corner radius for machining.
Error type	1, Error message from the NC program		

ID 13168

Incorrect definition of the overlapping factor.			
Description	If the (optional) programmed overlapping factor is greater than 0%, a check is made whether it lies within the range up to 100%. This message is generated when the overlapping factor was programmed outside this permissible range. Note: Alternatively, the overlapping factor can also be pre-assigned in the tool parameters.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the overlapping factor and correct the input parameters. If the overlapping factor is neither programmed nor pre-assigned in the tool parameters, a default of 50% is set in the cycle.
Error type	1, Error message from the NC program		

ID 13169

Corner radius is limited because it is too large.			
Description	During roughing, a check is made whether the resulting contour degenerates to a circular pocket when using the programmed corner radius. This message is generated if this is the case.		
Response	Class	-	Warning: The corner radius is limited to a specific value so that a rectangular pocket is produced and the cycle is continued.
Solution	Class	-	If possible, reduce the corner radius and correct the input parameters.

Error type	1, Error message from the NC program

ID 13170

Unknown plunging process defined.			
Description	For pocket milling, a plunge process can be (optionally) specified to calculate the countersinking motions. If a value is unequal to 0, a check is made whether one of the following identification numbers is set: 1. plunging on an inclined path, 2. perpendicular plunging 3. helical plunging This message is generated when an identification number other than 1, 2 or 3 was programmed in the corresponding data input parameters.		
Response	Class	-	Warning: Cycle is continued using plunging process 1 (inclined path).
Solution	Class	-	Check and define the plunging process and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13171

Pocket centre point is not completely defined.			
Description	The pocket centre point is defined by the (optional) parameters. The tool then moves to this position in the cycle at machining start. This message is generated if only one of the two parameters is programmed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the two parameters or correct as required. Alternatively, the parameters can be omitted. In this case the current position in the plane is adopted as the pocket centre point in the cycle.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 13172

Tool radius is 0. Set to default value 1mm.			
Description	The radius of the exchanged tool is 0. It is set to the default value 1mm.		
Response	Class	-	Warning, cycle is continued with default value.
Solution	Class	-	Check and define the tool radius and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13173

The tool radius is less than or equal to 0.	
Description	The radius of the active tool is less than or equal to 0.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the current tool or tool radius.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 13180

Parameter (@P1-@P4) not defined for cycle.			
Description	Parameters required for calibration are not defined		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and complete the programmed parameters.
Error type	1, Error message from the NC program		

ID 13181

Invalid value @P__(radius) <= 0			
Description	The specified adjusting ring radius is <= 0		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the programmed parameter.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 13182

Invalid value @P2(number of measuring positions) <3			
Description	The number of measured values entered is less than 3		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the programmed parameter.
Error type	1, Error message from the NC program		

ID 13183

Invalid value @P3 (velocity) <= 0			
Description	The entered measurement velocity is less than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the programmed parameter.
Error type	1, Error message from the NC program		

ID 13184

Invalid tool number @P4 <= 0			
Description	The entered touch probe tool number is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the programmed parameter.
Error type	1, Error message from the NC program		

ID 13185

The data set of the probe is not enabled @P4 != D			
Description	The exchanged tool and the active tool data record are different.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the programmed parameter.
Error type	1, Error message from the NC program		

ID 13186

Not touched			
Description	The target position of the measurement run was reached but not touched.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check whether the probe tip is within the ring and whether the correct value is entered for the ring radius (@P1) and correct if necessary.
Error type	1, Error message from the NC program		

ID 13187

Measured radii for S0 and S180 are different			
Description	The measured spindle adjusting ring radii S=0 and S=180 are different and the difference is greater than the maximum permissible deviation @P5 or, if it was not parameterised, it is greater than 0.05mm. The measurement is not reliable.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the probe and ring for soiling and clean if necessary. Check ring attachment to the table. Check the probe or probe tip attachment.
Error type	1, Error message from the NC program		

ID 13188

Variable for return of results is defined wrong.

Description	Error in definition of array V.L.RESULT_V containing the return value entries. The array must be defined with at least three elements. #VAR V.L.RESULT_V[3] #ENDVAR.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13189

Reference size of 1st main axis is not defined @P84.			
Description	No reference size was defined in the 1st main axis @P84.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the reference size at cycle call with the parameter @P84
Error type	1, Error message from the NC program		

ID 13190

Reference size of 2nd main axis is not defined @P85.			
Description	No reference size was defined in the 2nd main axis @P85.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the reference size at cycle call with the parameter @P85
Error type	1, Error message from the NC program		

ID 13191

Reference size of 3rd main axis is not defined @P86.			
Description	No reference size was defined in the 3rd main axis @P86.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the reference size at cycle call with the parameter @P86
Error type	1, Error message from the NC program		

ID 13192

Diameter of sphere is not defined @P87.			
Description	No sphere diameter is defined or the value is <=0		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the sphere diameter with the parameter @P87
Error type	1, Error message from the NC program		

ID 13193

Incorrect definition of the measuring feed rate.			
Description	The measuring feed rate was not or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring feed rate with a value greater than or equal to 0 in the configuration file SysCalibConfigTouchprobe.nc.
Error type	1, Error message from the NC program		

ID 13194

Incorrect definition of the positioning feed rate.			
Description	The positioning feed rate was not defined or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the positioning feed rate with a value greater than or equal to 0 in the configuration file SysCalibConfigTouchprobe.nc.
Error type	1, Error message from the NC program		

2.2.1.2 ID 13195

Incorrect definition of spindle speed V.CYC.SysConf_Probes_spdl_feed.			
Description	The spindle speed was not defined or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spindle speed with a value greater than or equal to 0 in the configuration file SysCalibConfigTouchprobe.nc.
Error type	1, Error message from the NC program		

2.2.1.3 ID 13196

Incorrect definition of the spindle waiting time V.CYC.SysConf_Probes_spdl_wait.			
Description	The spindle waiting time was not defined or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spindle waiting time with a value greater than or equal to 0 in the configuration file SysCalibConfigTouchprobe.nc.
Error type	1, Error message from the NC program		

ID 13200

Start- or target point (@P1, @P2) of measuring not defined.			
Description	The start and target points of the measurement are not defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13201

Path distance for measuring first edge of workpiece (@P3) not defined.			
Description	The length of the path distance measured along the workpiece edge was not defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13202

Number of measuring points @P4 first edge is too small.			
Description	At least two points must be measured along the first axis. The multiple ID shows: 1. Parameter not defined 2. Parameter too small		
Response	Class	-	Warning, cycle is continued with default value.The minimum number of measuring points is @P4=2.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13203

Start point for measuring second edge (@P5) not/incorrectly defined.			
Description	The start point for measuring the second edge @P5 is not defined or defined incorrectly. @P5 must be unequal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13204

Number of measuring points (@P8) for second edge is too small.			
Description	The number of points to be measured along the second axis @P8 is either not defined or too small. One point is measured.		
Response	Class	-	Warning, cycle is continued with default value.The parameter is set to the minimum number of measuring points @P8=1.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13205

Target point for measuring second edge @P6 not defined.			
Description	The target point for measuring the second workpiece edge @P6) is not defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13206

Path distance for measuring second edge of workpiece (@P7) not defined.			
Description	No definition for measuring the path distance along the workpiece edge.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13207

Angle of workpiece (@P11) not defined, 90 degrees are assumed.			
Description	The angle of the workpiece edge (@P11) is not specified; 90° are assumed.		
Response	Class	-	Warning, cycle is continued with default value.A right angle is assumed @P11=90.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13208

Angle of the workpiece (@P11) and several measuring points are specified. Angle is measured.			
Description	Both the angle of the workpiece edge (@P11) and more than one measuring point in the Y direction (@P8) are specified. The Y direction is measured and the angle is calculated from the two measured straight lines. The specified angle is not used.		
Response	Class	-	Warning, cycle is continued with default value.The parameter@ P11 is ignored.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13209

Variable for return of results is defined wrong.

Description	Error in definition of array V.L.RESULT_V containing the return value entries. The array must be defined with at least three elements, e.g.. #VAR V.L.RESULT_V1[3] #ENDVAR The multiple ID shows which return value is defined incorrectly: 1. V.L.RESULT_V1 2. V.L.RESULT_V2 3. V.L.RESULT_V3 4. V.L.RESULT_V4 5. V.L.RESULT_V5		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct input parameters.
Error type	1, Error message from NC program.		

ID 13210

Positioning velocity @P9 is not defined or wrong.			
Description	The positioning velocity must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13211

Measurement velocity @P10 is not defined or wrong.			
Description	The defined measurement velocity must be greater than 0. It is either not defined or incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13220

Necessary parameters for friction measuring are not defined.
Description: One or more required parameters are not defined This affects the following parameters: The multiple ID shows which parameter is defined incorrectly: 1. @P1: logical axis number 2. @P2: maximum duration of measurement of a velocity 3. @P3: maximum path distance to measurement 4. @ P4: number of measurement velocities 5. @P5: minimum measurement velocity 6. @P6: maximum measurement velocity

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13221

Axis number (@P1) is wrong.			
Description	Invalid specification of the logical axis number (@P1) of the axis to be measured. The specified value is either negative or greater than 65535. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the parameter @P1 and specify a valid value.
Error type	1, Error message from the NC program		

ID 13222

Duration of measurement (@P2) is 0 or negative.			
Description	The programmed maximum duration of an individual measuring stroke at constant velocity @P2 is less than or equal to 0. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the parameter @P2 and correct the specified value.
Error type	1, Error message from the NC program		

ID 13223

Travel distance (@P3) is too short.			
Description	The maximum travel distance for a measuring stroke at constant velocity must be greater than twice the parameterised backlash @P7. Select a sufficiently large measuring distance to reach the measurement velocity. It is recommended to parameterise a measuring distance that is much greater than the backlash. The multiple IDs show which parameter is defined incorrectly: 1. ABS[@P3] = 0 The distance is parameterised as 0, 2. ABS[@P3]-2*@P7 <= 0 the travel distance less twice the parameterised backlash is less than or equal to 0		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the two parameters @P3 and @P7 and enter a sufficiently large value.
Error type	1, Error message from the NC program		

ID 13224

Negative backlash (@P7), set to default value 5% of travel distance.			
Description	The backlash @P7 was specified with a negative sign. It was set to 5% of the travel distance @P3 and the measurement is continued.		
Response	Class	-	Warning, cycle is continued with default value.

Solution	Class	-	The backlash parameter @P7 must be greater than 0. Check and correct the parameter.
Error type	1, Error message from the NC program		

ID 13225

Number of table entries (@P4) is wrong.			
Description	The parameterised number of table entries to be determined is invalid. The valid value range is 1 – 19. According to the multiple ID: 1. @P4 > 19 the number of specified measured values is too large 2. @P4 <= 0 the number of measured values is specified as 0 or negative.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the parameter @P4.
Error type	1, Error message from the NC program		

ID 13226

Failure in programmed velocities (@P5, @P6).			
Description	The measurement velocity is defined by the parameters @P5 and @P6. The multiple IDs show which parameter is defined incorrectly: 1. @P5 = 0 minimum measurement velocity equal to 0 2. @P5< 0 minimum measurement velocity is negative 3. @P6 = 0 maximum measurement velocity equal to 0 4. @P6 < 0 maximum measurement velocity is negative 5. @P5 >= @P6 minimum measurement velocity is greater than or equal to the maximum		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the two parameters @P5 and @P6.
Error type	1, Error message from the NC program		

ID 13240

Necessary parameters for probing the sphere are not defined.			
Description	One or more required parameters are not defined This affects the following parameters: The multiple ID shows which parameter is not defined: 1. @ P1: Radius of the tool 2. @ P2: Security clearance 3. @ P3: Radius of the workpiece 4. @ P4: Positioning velocity 5. @ P5: Measurement velocity 6. @ P7: Position of rotary axis		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.

Error type	1, Error message from the NC program

ID 13241

Radius of the sphere (@P1) is defined negative.			
Description	The transferred value for the measuring sphere radius is negative. The value must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the measuring sphere radius.
Parameter	%1:	Current value	
		Parameterised measuring sphere radius	
	%2:	Lower limit value	
		Minimum measuring sphere radius	
Error type	1, Error message from the NC program		

ID 13242

Safety clearance (@P2) is defined negative.			
Description	The transferred value for the safety clearance for prepositioning between the touch probe and the measuring sphere is negative.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the safety clearance.
Parameter	%1:	Current value	
		Parameterised safety clearance	
	%2:	Lower limit value	
		0 - minimum safety clearance	
Error type	1, Error message from the NC program		

ID 13243

Radius of the touch probe (@P3) is defined negative.			
Description	The transferred value for the touch probe radius is negative.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the touch probe radius.
Parameter	%1:	Current value	
		Parameterised touch probe radius	
	%2:	Lower limit value	
		0 – minimum touch probe radius	
Error type	1, Error message from the NC program		

ID 13244

Velocity for positioning (@P4) is wrong.			
Description	The positioning velocity is defined wrong: The velocity must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the positioning velocity.
Parameter	%1:	Current value	
		Parameterised positioning velocity	
	%2:	Lower limit value	
		0 – minimum positioning velocity	
Error type	1, Error message from the NC program		

ID 13245

Velocity for measuring (@P5) is wrong.			
Description	The measurement velocity is defined incorrectly: The velocity must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the measurement velocity.
Parameter	%1:	Current value	
		Parameterised measurement velocity	
	%2:	Lower limit value	
		0 – minimum measurement velocity	
Error type	1, Error message from the NC program		

ID 13246

Transferred value for scaling (@P6) is invalid.			
Description	The scaling factor must be in the interval [0, 1]. The value 0 is not a valid value; the value 1 is contained in the interval.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the scaling factor.
Parameter	%1:	Current value	
		Parameterised measuring sphere radius	
	%2:	Lower limit value	
		Minimum scaling factor	
	%3:	Upper limit value	
		Maximum scaling factor (inclusive)	
Error type	1, Error message from the NC program		

ID 13247

Not touched.

Description	The measuring sphere was not touched, no measurement result.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13248

Failure in definition of variables for return values.			
Description	The arrays of variables V.L.X[i], V.L.Y[i] or V.L.Z[i] are not defined or defined wrong.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct definitions of input parameters in form of V.L. arrays.
Error type	1, Error message from the NC program		

2.2.2 ID-range 13250-13499

ID 13260

Start angle for measurement is greater than end angle.			
Description	The parameterised start angle is greater than the end angle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13261

Unknown rotary axis shall be measured.			
Description	The parameterised rotary axis does not exist.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13262

Maximum deviation of a measuring point from the surface of the sphere is exceeded.			
Description	A measuring point is outside the specified tolerance.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Parameter	%1:	Upper limit value	
		Deviation radius to measuring point	
	%2:	Current value	
		Lower tolerance limit	
	%3:	Current value	
		Upper tolerance limit	
Error type	1, Error message from the NC program		

ID 13263

Maximum deviation of spherical shape is exceeded.			
Description	The measured values are not on a sphere.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13264

Number of spheres handed over incorrectly.			
Description	The parameterised number of spheres to be measured in wrong.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13265

Start angle is not defined.			
Description	No start angle was transferred. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13266

End angle is not defined.			
Description	No start angle was transferred. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13280

Measurement velocity @P9 is not defined.			
Description	No feed rate was transferred for the measurement. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13281

Positioning velocity @P8 is not defined.			
Description	No feed rate was transferred for positioning. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13282

Measuring distance in x @P2 is not defined.			
Description	No distance for the measurement was transferred in X. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13283

Measuring distance in z @P10 is not defined.			
Description	No distance for the measurement was transferred in Z. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13284

Safety clearance @P1 is not defined.			
Description	No distance for lifting after the measurement was transferred. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13285

Mode @P11 is not defined or wrong.			
Description	Only the following values are permitted for mode: • 0 – pocket • 1 – spigot • 2 – slot • 3 - web		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct input parameters for the mode.
Error type	1, Error message from NC program.		

ID 13286

Plunge depth z @P7 is not or incorrectly defined.	
Description	The plunge depth in Z is not defined or defined incorrectly. It must be greater than 0.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13287

Prepositioning in x @P5 is not defined.			
Description	Prepositioning in X is not defined. A path for web and spigot is absolutely necessary, it must be sufficiently large for positioning outside the spigot.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for prepositioning in X.
Error type	1, Error message from the NC program		

ID 13288

Prepositioning in y @P6 is not defined.			
Description	Prepositioning in Y is not defined. A path for spigot is absolutely necessary; it must be sufficiently large for positioning outside the spigot.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for prepositioning in Y.
Error type	1, Error message from the NC program		

ID 13289

Variable for return of results is defined wrong.			
Description	Error in definition of array V.L.RESULT_V containing the return value entries. The array must be defined with at least two elements. #VAR V.L.RESULT_V[2] #ENDVAR		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter settings and create variable for return value.
Error type	1, Error message from the NC program		

ID 13300

Length of longhole or slot is not defined or 0.			
Description	The length of slot or longhole is not defined or 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the length.

Error type	1, Error message from the NC program

ID 13301

Radius of circle is not defined or incorrect.			
Description	The radius of the circle on which the slots or longhole lie is not defined or 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the circle radius.
Error type	1, Error message from the NC program		

ID 13302

Number of longholes or slots is not defined or incorrectly defined.			
Description	The number of longholes is not defined or defined incorrectly. An integer greater than 0 must be transferred.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the number.
Error type	1, Error message from the NC program		

ID 13303

Width of slot is not defined or smaller than tool diameter.			
Description	The width of the slot is defined incorrectly. It must be greater than or equal to the tool diameter.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the slot width.
Error type	1, Error message from the NC program		

ID 13304

Contour violation between longholes or slots.			
Description	The slots or longholes touch or overlap. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and check placement of the elements.
Error type	1, Error message from the NC program		

ID 13305

Opening angle of longhole or slot is not defined or 0.

Description	The opening angle of the longhole or slot is not defined or incorrect. This is not permitted		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the opening angle.
Error type	1, Error message from the NC program		

ID 13306

Contour violation between first and last slot or longhole.			
Description	There are overlaps between the first and last slot or longholes. The results in contour violations and is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and placement and correct the input parameters for the contours.
Error type	1, Error message from the NC program		

ID 13307

Feed rate is incorrect or not defined.			
Description	No feed rate or an invalid feed rate (>=0) is defined for machining or plunging. No feed rate was programmed in the main program either.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the feed rate.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 13308

Number of helical infeeds is defined incorrectly.			
Description	The number of helical infeeds must be a positive integer.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct input parameters for the number.
Error type	1, Error message from NC program.		

2.2.2.1 ID 13309

Invalid font size computation type.			
Description	The specified calculation for the font size is not supported.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Check parameter setting and correct the input parameters to calculate the font size.
Error type	1, Error message from the NC program		

2.2.2.2 ID 13310

Invalid anchor point.			
Description	The specified anchor point is not supported.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the anchor point.
Error type	1, Error message from the NC program		

2.2.2.3 ID 13311

Invalid character coding.			
Description	The specified character coding is not supported.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for character coding.
Error type	1, Error message from the NC program		

2.2.2.4 ID 13312

Missing or invalid value for font size.			
Description	The value for the font size must be positive.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the font size.
Error type	1, Error message from the NC program		

2.2.2.5 ID 13313

Invalid value for the monospaced attribute.			
Description	The value for the monospaced attribute must be 0/FALSE or 1/TRUE.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the monospaced attribute.
Error type	1, Error message from the NC program		

2.2.2.6 ID 13314

Invalid value for the mirror attribute.			
Description	The value for the mirror attribute must be 0/FALSE or 1/TRUE.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the mirror attribute.
Error type	1, Error message from the NC program		

2.2.2.7 ID 13315

The text layout was not defined as an integer.			
Description	The text layout is an integer between 0 and 1.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the text layout.
Error type	1, Error message from the NC program		

2.2.2.8 ID 13316

The font cycle could not be found.			
Description	The font cycle must exist and be legible.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the font cycle.
Error type	1, Error message from the NC program		

2.2.2.9 ID 13317

The name of the font file is invalid.			
Description	The name of the font file must be minimum 4 characters long.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the name of the font file.
Error type	1, Error message from the NC program		

2.2.2.10 ID 13318

Invalid layout.			
Description	The specified layout is not supported.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the layout.

Error type	1, Error message from the NC program

2.2.2.11 ID 13319

Invalid output type.			
Description	The specified output type is not supported.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the output type.
Error type	1, Error message from the NC program		

ID 13320

Missing or invalid parameter for a cycle of the linalg package.			
Description	When the cycle was called, either a necessary parameter was not transferred or an invalid parameter was transferred.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the cycle parameter setting in the calling NC program/cycle.
Error type	1, Error message from the NC program		

ID 13321

An invalid number of matrix rows was transferred to a cycle of the linalg package.			
Description	When a cycle was called, an invalid number of matrix rows was transferred.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the number of matrix rows.
Parameter	%1:	Identification number	
		Index of incorrect parameter	
	%2:	Upper limit value	
		Transferred number of rows	
	%3:	Current value	
		Required minimum number of rows	
	%4:	Current value	
		Maximum highest number of rows	
Error type	1, Error message from the NC program		

ID 13322

An invalid number of matrix columns was transferred to a cycle of the linalg package.			
Description	When the cycle was called, an invalid number of matrix columns was transferred.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Check parameter setting and correct the input parameters for the number of matrix columns.
Parameter	%1:	Identification number	
		Index of incorrect parameter	
	%2:	Upper limit value	
		Transferred number of columns	
	%3:	Current value	
		Required minimum number of columns	
	%4:	Current value	
		Maximum highest number of columns	
Error type	1, Error message from the NC program		

ID 13323

There is not enough memory for the creation of a new matrix in the linalg package.			
Description	There is not enough memory for the creation of a new matrix in the linalg package.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and increase the size of the variable V.CYC.CNC_LINALG.
Parameter	%1:	Current value	
		Required memory including the new matrix	
	%2:	Upper limit value	
		Maximum available memory	
Error type	1, Error message from the NC program		

ID 13324

A required matrix id parameter was not transferred to a cycle of the linalg package.			
Description	A required matrix id parameter was not transferred to a cycle of the linalg package. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and transfer the input parameters for the matrix id.
Parameter	%1:	Identification number	
		Required memory including the new matrix	
Error type	1, Error message from the NC program		

ID 13325

An invalid matrix id parameter was transferred to a cycle of the linalg package.			
Description	An invalid matrix id parameter was transferred to a cycle of the linalg package.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the matrix id.

Parameter	%1:	Identification number
		Index of incorrect parameter
	%2:	Current value
		Transferred matrix id
	%3:	Lower limit value
		Smallest valid matrix id
	%4:	Upper limit value
		Largest valid matrix id
Error type	1, Error message from the NC program	

ID 13326

The sizes of two matrices are incompatible.			
Description	The sizes of two matrices are incompatible. This error occurs, for example, when a matrix is copied to a matrix of a different size or two matrices are multiplied, whereby the number of columns in the first matrix is not equal to the number of rows in the second matrix.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct input parameters for the matrix id in the calling program/cycle.
Parameter	%1:	Expected value	
		Expected number of lines/columns	
	%2:	Current value	
		Actual number of lines/columns	
	%3:	Expected value	
		Smallest valid matrix id (parameter is optional)	
	%4:	Current value	
		Actual number of lines/columns (parameter is optional)	
Error type	1, Error message from the NC program		

ID 13327

Invalid matrix initialisation.			
Description	When a matrix was created, an invalid initialisation variant was defined. See the documentation for a list of all available variants.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the matrices in the calling NC program/cycle.
Parameter	%1:	Identification number	
		Index of incorrect parameter	
	%2:	Current value	
		Invalid initialisation identifier	
Error type	1, Error message from the NC program		

ID 13328

Invalid type of matrix norm.			
Description	When determining the matrix norm, an invalid norm type was transferred. See the documentation for a list of all available norm types.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the norm type in the calling program/cycle.
Parameter	%1:	Identification number	
		Index of incorrect parameter	
	%2:	Error value	
		Invalid norm type	
Error type	1, Error message from the NC program		

ID 13329

Missing variable V.CYC.CNC_LINALG.			
Description	The variable V.CYC.CNC_LINALG must be created to use the linalg package. For example: #VAR V.CYC.CNC_LINALG[1000] : REAL64 #ENDVAR		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and create V.CYC.CNC_LINALG in the calling NC program/cycle.
Error type	1, Error message from the NC program		

2.2.2.12 ID 13330

The font file could not be found.			
Description	The font file must exist and be legible.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the font file.
Error type	1, Error message from the NC program		

ID 13340

Either none or an invalid kinematic id was transferred to a cycle of the calib package.			
Description	A cycle expects a valid kinematic id as parameter. No kinematic id or an invalid kinematic id was transferred.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the kinematic id.
Parameter	%1:	Current value	
		Specified kinematic id	
Error type	1, Error message from the NC program		

ID 13341

Either none or an invalid kinematic variant was transferred to a cycle of the calib package.			
Description	A cycle expects a valid kinematic variant as parameter. No kinematic variant or an invalid kinematic variant was transferred. See cnc_calib_fit_init for a list of all valid combinations of kinematics and kinematic variants.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the kinematic variant.
Parameter	%1:	Current value	
		Transferred kinematic id	
	%2:	Identification number	
		Transferred kinematic variant	
Error type	1, Error message from the NC program		

ID 13342

A cycle expects an axis value parameter, but this value was not transferred.			
Description	A cycle expects an axis value parameter, but this value was not transferred.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for axis value.
Parameter	%1:	Identification number	
		Index of the missing axis value parameter	
Error type	1, Error message from the NC program		

ID 13343

Unexpected number of measurement records.			
Description	Th number of measurement records added with cnc_calib_fit_add_record does not coincide with the number given in cnc_calib_fit_init.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and compare the number of measurement records in cnc_calib_fit_add_record and cnc_calib_fit_init.
Parameter	%1:	Identification number	
		Number of previously added measurement records	
	%2:	Expected value	
		Expected number and highest number of measurement records	
Error type	1, Error message from the NC program		

ID 13344

The specified output format is invalid.			
Description	Th number of measurement records added with cnc_calib_fit_add_record does not coincide with the number given in cnc_calib_fit_init.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Check and compare the number of measurement records in cnc_calib_fit_add_record and cnc_calib_fit_init.
Parameter	%1:	Current value	
		Number of previously added measurement records	
Error type	1, Error message from the NC program		

ID 13345

Missing or invalid number measurement records.			
Description	When initialising the calibration of a machine kinematic, the number of measurement records must be specified. If this number is less than 1 or is missing, this error is output.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the number of measurement records.
Parameter	%1:	Identification number	
		Number of previously added measurement records	
	%2:	Error value	
		(optional parameter)	
	%3:	Lower limit value	
		Minimum number of measurement records (optional parameter)	
Error type	1, Error message from the NC program		

ID 13346

Insufficient calibration precision.			
Description	When calibrating a machine kinematic inaccuracies occurred as the limit exceeds the one set by the user. This limit precision is specified when cnc_calib_fit is called. Exceeding the limit precision indicates errors that are not considered in the error model. Examples of such errors are • inaccurate measuring equipment • errors in measurement set-up • incorrect measuring strategy • machine errors not detected by the measurement set-up		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Find and correct the error source in the measurement set-up. Adapt limit precision
Parameter	%1:	Current value	
		Determined precision	
	%2:	Upper limit value	
		Maximum permissible limit precision	
Error type	1, Error message from NC program.		

ID 13360

Mode for array operations is missing or defined wrong.

Description	The mode for an array operation is missing or defined incorrectly. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the mode.
Error type	1, Error message from the NC program		

ID 13361

An element of a vector is not defined.			
Description	A required vector element is not defined; this is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the vector element.
Error type	1, Error message from the NC program		

ID 13362

An element of a vector is not defined.			
Description	A required vector element is not defined; this is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the vector element.
Error type	1, Error message from the NC program		

ID 13363

Size of matrices is not consistent.			
Description	Check the matrices concerned before a matrix operation. The matrix size of the matrices concerned does not coincide. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters. Check and correct the size of the matrices.
Error type	1, Error message from the NC program		

ID 13364

The defined matrix is not a square matrix.			
Description	Check the matrices concerned before a matrix operation. This case must be about a square matrix. This is not the case and is therefore invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting. Check and correct the size of the matrices.

Error type	1, Error message from the NC program

ID 13365

It is impossible to calculate the inverse.			
Description	Check the matrices concerned before a matrix operation. The inverse of a matrix cannot be calculated; this is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting. Check and correct the matrix concerned.
Error type	1, Error message from the NC program		

ID 13366

Size of matrix exceeds maximum.			
Description	Check the matrices concerned before a matrix operation. The matrix size of the matrices concerned is too large. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting. Check and correct the size of the matrices.
Error type	1, Error message from the NC program		

ID 13380

The pattern is defined wrong or missing.			
Description	The mode (@P14) for pattern was transferred incorrectly or not at all.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and transfer the input parameters for the pattern mode.
Parameter	%1:	Error value	
		Transferred value for the pattern.	
Error type	1, Error message from the NC program		

ID 13381

Distance between positions is not defined.			
Description	The distance of a point in the first or second main direction is not defined or defined incorrectly. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 13382

Number of positions is not defined or 0.			
Description	The number of points transferred is not or incorrectly defined. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the number of points.
Error type	1, Error message from the NC program		

ID 13383

Radius of circle is not defined.			
Description	The radius is defined incorrectly or missing.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the radius.
Error type	1, Error message from the NC program		

ID 13384

Neither incremental angle nor the end angle is defined.			
Description	Neither the incremental angle @P12 nor the end angle @P13 is defined. This is not permitted		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters @P12 @P13 for the angle.
Error type	1, Error message from the NC program		

ID 13385

The end angle is smaller than the start angle.			
Description	The end angle @P13 is smaller than the start angle @P9. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters @P13 + @P9 for the angles.
Error type	1, Error message from the NC program		

ID 13400

Measuring feed is not defined			
Description	No measuring feed is defined, or the value is <=0		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Define the measuring feed in the extra parameter [0] of the measuring tool. If the parameter V.P.Sys_Probes_in_spindle is set to 0 in the NC program SysMeasWcsConfig.nc, the value of the variable V.P.Sys_Probes_feed must be adjusted in the NC program SysMeasWcsConfig.nc.
Error type	1, Error message from the NC program		

ID 13401

Diameter of the hole is not defined @P68.			
Description	No hole diameter is defined, or the value is <= 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the drilling diameter at cycle call with parameter @P68.
Error type	1, Error message from the NC program		

ID 13402

Diameter of the spigot is not defined @P68.			
Description	No spigot diameter is defined, or the value is <= 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spigot diameter at cycle call with the parameter @P68.
Error type	1, Error message from the NC program		

ID 13403

Width of the slot is not defined @P3.			
Description	No slot width is defined, or the value is <= 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the slot width at cycle call with parameter @P3.
Error type	1, Error message from the NC program		

ID 13404

Width of the web is not defined @P3.			
Description	No web width is defined, or the value is <= 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the web width at cycle call with parameter @P3.
Error type	1, Error message from the NC program		

ID 13405

Measuring distance in 1st main axis is not defined @P4.

Description	No measuring distance was defined in the 1st main axis.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring distance at cycle call with parameter @P4.
Error type	1, Error message from the NC program		

ID 13406

Measuring distance in 2nd main axis is not defined @P5.			
Description	No measuring distance was defined in the 2nd main axis.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring distance at cycle call with parameter @P5.
Error type	1, Error message from the NC program		

ID 13407

Measuring distance in 3rd main axis is not defined @P6.			
Description	No measuring distance was defined in the 3rd main axis, or the value is 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring distance at cycle call with parameter @P6.
Error type	1, Error message from the NC program		

ID 13408

Zero offset is not defined or incorrectly defined @P7, values 1- 9 53-59 are allowed.			
Description	No zero point is defined, or a wrong value is set.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the zero point at cycle call with parameter @P7. Valid numbers are 1 to 9 and 53 to 59.
Error type	1, Error message from the NC program		

ID 13409

Angle 1 @P14, angle 2 @P15 and angle 3 @P16 must be different.			
Description	The same angle was specified at least twice.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Different definition of angles in the cycle call with parameters @P14, @P15 and @P16.
Error type	1, Error message from the NC program		

ID 13410

Positioning in 1st main axis is not defined @P17.			
Description	No positioning of the 1st main axis, or the value is 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the positioning at cycle call with parameter @P18.
Error type	1, Error message from the NC program		

ID 13411

Positioning in 2nd main axis is not defined @P18.			
Description	No positioning of the 2nd main axis, or the value is 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the positioning at cycle call with parameter @P18.
Error type	1, Error message from the NC program		

ID 13412

Positioning in 3rd main axis is not defined @P19.			
Description	No positioning of the 3rd main axis, or the value is 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the positioning at cycle call with parameter @P19.
Error type	1, Error message from the NC program		

ID 13413

No workpiece detected, please check your measuring distance.			
Description	No measuring signal was triggered within the specified measuring distance.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Either the measuring distance must be adapted or the position before the measurement must be redefined.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 13414

Collision at positioning, please check your motion range.			
Description	The measurement signal was triggered unexpectedly when positioning at the next measuring point.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	The positioning conditions must be reviewed and adjusted.

Error type	1, Error message from the NC program

ID 13415

Angle 1 is not defined @P14.			
Description	The first angle is not defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the first angle at cycle call with parameter @P14.
Error type	1, Error message from the NC program		

ID 13416

Angle 2 is not defined @P15.			
Description	The second angle is not defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the first angle at cycle call with parameter @P15.
Error type	1, Error message from the NC program		

ID 13417

Angle 3 is not defined @P16.			
Description	The third angle is not defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the first angle at cycle call with parameter @P16.
Error type	1, Error message from the NC program		

ID 13418

Length in 1st main axis of rectangle not defined @P30.			
Description	No length of the 1st main axis was defined, or the value is <= 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the length of the 1st main axis at cycle call with parameter @P30.
Error type	1, Error message from the NC program		

ID 13419

Length in 2nd main axis of rectangle not defined @P31.			
Description	No length of the 2nd main axis was defined, or the value is <= 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Define the length of the 2nd main axis at cycle call with parameter @P31.
Error type	1, Error message from the NC program		

ID 13420

Something went wrong with the cycle.			
Description	At the end of cycle something unpredictable happened.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Contact your manufactures.
Error type	1, Error message from the NC program		

ID 13421

Define either @P15 or @P30 but not both.			
Description	Only one of the parameters @P15 or @P30 may be defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Remove one of the parameters, @P15 or @P30 at cycle call
Error type	1, Error message from the NC program		

ID 13422

The depth @P22 is not defined.			
Description	No depth is defined, or the value is <=0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the depth at cycle call with the parameter @P22
Error type	1, Error message from the NC program		

ID 13423

Measurement position 1 is overwritten.			
Description	The measurement position 1 will be overwritten since no position was selected.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measurement position at cycle call with the parameter @P20
Error type	1, Error message from the NC program		

ID 13424

Diameter of the measuring plate is not defined @P21.	
Description	No measuring plate diameter is defined, or the value is <=0.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring plate diameter at cycle call with the parameter @P21.
Error type	1, Error message from the NC program		

ID 13425

Measurement position must be 0, 1, 2 or 3 @P25.			
Description	No measurement position or an invalid measurement position is defined.. Permitted values are 0, 1, 2 or 3.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measurement position at cycle call with the parameter @P25.
Error type	1, Error message from the NC program		

ID 13426

Rotation direction must be 3(M3), 4(M4) or 5(M5) @P26.			
Description	No valid value for the rotation direction is set. Permitted values are 3, 4 or 5.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a valid value for the direction of rotation at cycle call with the parameter @P26.
Error type	1, Error message from the NC program		

ID 13427

Max spindle speed for measuring is exceeded, it will be set to max.			
Description	The maximum of the defined spindle speed (V.E.Max_rpm) is exceeded. The spindle speed it set to the highest value.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Reduce the spindle speed at cycle call with the parameter @P27. Alternatively, raise the value from V.E.Max_rpm.
Error type	1, Error message from the NC program		

ID 13428

Diameter of the second hole is not defined @P29.			
Description	No diameter is defined for the second hole, or the value is <=0..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the second diameter at cycle call with parameter @P29.
Error type	1, Error message from the NC program		

ID 13429

Diameter of the spigot is not defined @P29.			
Description	No diameter is defined for the spigot, or the value is <=0..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spigot diameter at cycle call with the parameter @P29.
Error type	1, Error message from the NC program		

ID 13430

Diameter of the second spigot is not defined @P29.			
Description	No diameter is defined for the second spigot, or the value is <=0..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the second diameter at cycle call with parameter @P29.
Error type	1, Error message from the NC program		

ID 13431

For the variable V.P.Sys_Probes_in_spindle only 0 or 1 permitted.			
Description	An incorrect value is entered for the variable V.P.Sys_Probes_in_spindle; only 0 or 1 are permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the variable V.P.Sys_Probes_in_spindle in the NC program SysMeasWcsConfig.nc.
Error type	1, Error message from the NC program		

ID 13432

Positioning feed rate is not defined.			
Description	The positioning feed rate is not defined, or the value is less than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the positioning feed rate in the NC program SysMeasWcsConfig.nc.
Error type	1, Error message from the NC program		

ID 13433

Positioning feed rate must be higher than the measuring feed rate.			
Description	The positioning feed rate V.P.Sys_Probes_feed_max must be greater than or equal to the measuring feed rate V.P.Sys_Probes_feed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Change the feed values V.P.Sys_Probes_feed_max and V.P.Sys_Probes_feed in the NC program SysMeasWcsConfig.nc.

Error type	1, Error message from the NC program

ID 13434

Radius is not defined.			
Description	Radius is not defined or is incorrectly defined; the value must be higher than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	During the cycle call, a valid value must be defined.
Error type	1, Error message from the NC program		

ID 13435

The number of measurements must be an integer.			
Description	The parameter @P30 must be an integer.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	At cycle call the parameter @P30 must be changed into an integer.
Error type	1, Error message from the NC program		

ID 13436

At least 3 measuring points are required.			
Description	At least 3 measuring points are required for calculation.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	At cycle call the parameter @P30 must be set to at least 3.
Error type	1, Error message from the NC program		

ID 13437

Invalid selection of the measuring mode			
Description	Invalid selection of the measuring mode. Permitted values are 1, 2 or 3.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible measuring mode during the cycle call Correct input parameter @P64..
Error type	1, Error message from the NC program		

ID 13438

Measurement position is not available.			
Description	Only positive values are permitted for the number of measurement positions @P20. The values 1, 2 or 3 are permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Defining a permissible value for the measurement position at cycle call.
Error type	1, Error message from the NC program		

ID 13439

The speed must be greater than 0.			
Description	The transferred speed (@ P27) must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value for the speed at cycle call.
Error type	1, Error message from the NC program		

ID 13440

At least one measuring station is required.			
Description	At least one measuring station is required for the measurement process.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	The definition of the variable V.P.Sys_NumberOfPositions in the file must be > 0.
Error type	1, Error message from the NC program		

ID 13441

The plate diameter was defined incorrectly.			
Description	Sys_Plate_Diam may not be smaller than Sys_Diam_max.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a correct value for Sys_Plate_Diam in the configuration file.
Error type	1, Error message from the NC program		

ID 13442

Incorrect value in the additional parameters.			
Description	Only values between 0 and 59 are permissible for additional parameters..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define valid values in the file SysCalibConfigTouchProbe.nc.
Error type	1, Error message from the NC program		

ID 13443

Inadmissible value for specifying whether measurement is repeated @P88.

Description	The value for repeated measurement @P88 is invalid. This input defines whether the measurement is repeated . Only the values 1 or 2 are permitted. 1: Measurement can be repeated 2: no measurement repetition		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value for measurement repetition
Error type	1, Error message from NC program.		

ID 13444

2nd positioning of first main axis is not defined @P41.			
Description	No second positioning of the 1st main axis at cycle call with parameter @P41.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value of second positioning of the 1st main axis
Error type	1, Error message from the NC program		

ID 13445

2nd positioning of second main axis is not defined @P43.			
Description	No second positioning of the 2nd main axis at cycle call with parameter @P43.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value of second positioning of the 2nd main axis.
Error type	1, Error message from the NC program		

ID 13446

3rd positioning of the first main axis is not defined @P42.			
Description	No third positioning of the 1st main axis at cycle call with parameter @P42.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value of third positioning of the 1st main axis.
Error type	1, Error message from the NC program		

ID 13447

3rd positioning of the second main axis is not defined @P44.			
Description	No third positioning of the 2nd main axis at cycle call with parameter @P44.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value of third positioning of the 2nd main axis.
Error type	1, Error message from the NC program		

ID 13448

ID of coordinate system is not defined @P39.			
Description	The ID of the coordinate system @P39 is not defined or a wrong value was defined. Values 1 to 10 are permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a valid coordinate system ID @P39
Error type	1, Error message from the NC program		

ID 13449

The index of rotary axis to be measured is not defined @P89.			
Description	The index of the rotary axis to be measured was not defined. The axis index defines the location of the axis inside the axis group of the NC channel. Integers in the range [0… maximum number of axes -1] and the value -1 for manual axes are permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a valid value for the axis index with the parameter @P89.
Error type	1, Error message from the NC program		

ID 13450

Tool wear is outside the specified tolerance.			
Description	Tool wear is outside of specification after inspection tolerance.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	The tool needs to be replaced.
Error type	1, Error message from the NC program		

ID 13451

No measurement signal was triggered.			
Description	No measurement signal was triggered.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the set position of your laser.
Error type	1, Error message from the NC program		

ID 13452

Invalid selection of the measuring mode, only 0 or 1 is permissible.			
Description	Invalid selection of the measuring mode. Permitted values are 0 or 1.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible measuring mode at cycle call.

Error type	1, Error message from the NC program

ID 13453

Invalid selection of the measuring direction, only 0-3 is allowed.			
Description	Invalid selection of the measuring direction. Permitted values are 0-3.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible measuring direction at cycle call.
Error type	1, Error message from the NC program		

ID 13454

Scattering of the measurement outside the set tolerance.			
Description	Scattering of the measurement outside the set tolerance.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Repeat the measurement and check the permitted tolerance.
Error type	1, Error message from the NC program		

ID 13455

No measurement type, or an incorrect one, was specified. 0, 1 and 2 are permissible.			
Description	No measurement type, or an incorrect one, was specified. Permitted values are 0, 1 or 2.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a valid measurement type at cycle call.
Error type	1, Error message from the NC program		

ID 13456

Only positive values above 0 are permitted for the tolerance.			
Description	Only positive values above 0 are permitted for the tolerance.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a valid tolerance value at cycle call.
Error type	1, Error message from the NC program		

ID 13457

Only positive values above 0 are allowed for length offset.			
Description	Only positive values above 0 are allowed for length offset.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Define a permissible value for the length offset at cycle call.
Error type	1, Error message from the NC program		

ID 13458

Only positive values > = 0 are allowed for radius offset.			
Description	Only positive values > = 0 are allowed for radius offset.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value for the radius offset at cycle call.
Error type	1, Error message from the NC program		

ID 13459

Only positive values above 0 are allowed for repeats.			
Description	Only positive values above 0 are allowed for repeats		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value for repeats at cycle call.
Error type	1, Error message from the NC program		

ID 13460

Only positive values above 0 are allowed for tool diameter.			
Description	Only positive values above 0 are allowed for tool diameter.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value for the tool diameter at cycle call.
Error type	1, Error message from the NC program		

ID 13461

Tool breakage			
Description	The measurement detected a tool breakage.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	The tool needs to be replaced.
Error type	1, Error message from the NC program		

ID 13462

A subroutine necessary for configuration cannot be called.

Description	A subroutine required for the cycle cannot be found in the configured program paths in the start-up list. This may affect the following files. 1. SysCalibConfigLaser.nc 2. SysCalibConfigLaserOn.nc 3. SysCalibConfigLaserOff.nc 4. SysCalibConfigToolSettingProbe.nc 5. SysCalibConfigTouchprobe.nc		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check whether the subroutine exists and ensure that it is available in the configured program paths.
Parameter	%1:	Current value	
		Output value in the above list indicates what file was not found.	
Error type	1, Error message from the NC program		

2.2.3 ID-range 13500-13749

ID 13501

Max spindle speed is not defined. (Only positive values allowed).			
Description	The maximum spindle speed is not defined, or the value is <=0..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the value for the maximum spindle speed.
Error type	1, Error message from the NC program		

ID 13502

No or invalid direction of rotation defined. 2=M3+M4 3=M3 4=M4			
Description	No rotation direction or an invalid direction is defined: Permitted values: 2=M3+M4, 3=M3, 4=M4		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value for the direction of rotation.
Error type	1, Error message from the NC program		

ID 13503

No or invalid input for the full run (0=No 1=Yes)			
Description	No or invalid input for the full run (0=No 1=Yes)		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a correct selection.
Error type	1, Error message from the NC program		

ID 13504

No or invalid input for the long run (0=No 1=Yes)			
Description	The long run (0=No 1=Yes) is not specified or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a correct selection.
Error type	1, Error message from the NC program		

ID 13505

No or invalid time specification (only positive numbers permitted)			
Description	No time specification or an invalid one, only positive integers are permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permitted time specification.

Error type	1, Error message from the NC program

ID 13506

The two comparative values @P1 and @P2 were not fully defined.			
Description	Both comparison values (@P1 and @P2) were not completely sent to the cycle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Send both a value for @P1 and for @P2 to the cycle.
Error type	1, Error message from the NC program		

ID 13507

Error when transferring parameter 3 (test selection).			
Description	Parameter 3 was not sent or was incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Send a correct value for parameter 3.
Error type	1, Error message from the NC program		

ID 13508

Error when transferring parameter 4 (language selection).			
Description	Parameter 4 was incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Send a correct value for parameter 4.
Error type	1, Error message from the NC program		

ID 13509

Error when transferring parameter 5 (usage of new line).			
Description	Parameter 5 was incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Send a correct value for parameter 5.
Error type	1, Error message from the NC program		

ID 13551

Invalid value at pre-positioning @P90.

Description	The value which describes how to execute pre-positioning is defined wron in the parameter @P90. Permitted values are 0 or 1: • 0: automatic pre-positioning by NC subroutines • 1: manual pre-positioning or pre-positioning in manual mode		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	3	Define a valid value for pre-positioning at the cycle call with the parameter @P90
Error type	1, Error message from NC program.		

ID 13552

Incomplete or invalid vector of rotary axis.			
Description	A complete definition of the expected rotary axis is required with the parameters @P91, @P92 and @P93 to adapt the measurement movements to the rotation of the rotary axis to be measured. The rotary axis is defined as a vector, where @P91 is defined as the X component, @P92 the Y component and @P93 the Z component. The vector must have a length greater than 0. Here, an approximate parameter is sufficient. The values are only used to estimate the measurement movement.		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	3	Complete definition of the expected rotary axis with the parameters @P91, @P92 and @P93.
Error type	1, Error message from the NC program		

ID 13553

Incomplete or invalid vector of rotary axis when defining the inclination of probe.			
Description	A definition of the inclination by a rotary axis is required with the parameters @P94, @P95 and @P96 to adapt the inclination and measurement motions at the beginning of the cycle. The rotary axis is defined as a vector, where @P94 is defined as the X component, @P95 the Y component and @P96 the Z component. The vector must have a length greater than 0. Here, an approximate parameter is sufficient. The values are only used to estimate the measurement movement.		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	3	Complete definition of the inclination of the probe by a rotary axis with the parameters @P94, @P95 and @P96.
Error type	1, Error message from the NC program		

ID 13554

Missing angle information between measurements during manual positioning.	
Description	The manually travelled angles between measurements were not fully defined with the parameters @P100 and @P101. @P100 defines the manually travelled angle between the first and second measurements. @P101 defines the manually travelled angle between the first and third measurements. Here, an approximate parameter is sufficient. The values are only used to estimate the measurement movement.

Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	3	Complete definition of the manually travelled angle with the parameters @P100 and @P101.
Error type	1, Error message from the NC program		

ID 13555

File access not possible with automatic prepositioning.			
Description	With automatic prepositioning (@P90 = 0) the corresponding NC subroutines must be saved to the file system. The file “SysMeasRotAxPrePos2.nc” defines the prepositioning between the first and second measurements. The file “SysMeasRotAxPrePos3.nc” defines the prepositioning between the second and third measurements.		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	3	Check and restore the availability of the files “SysMeasRotAxPrePos2.nc” and “SysMeasRotAxPrePos3.nc”.
Error type	1, Error message from the NC program		

ID 13556

Retraction distance is not or incorrectly defined.			
Description	The retraction distance.is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the retraction distance at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 13557

Swivel mode is not or incorrectly defined.		
Description	The swivel mode is not defined or is invalid.	
Response	Class	- Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	- Define the swivel mode at cycle call with a valid value.
Parameter	%1:	Current value
		Transferred swivel mode
	%2:	Lower limit value
		Smallest permitted swivel mode (integer)
	%3:	Upper limit value
		Largest permitted swivel mode (integer)
Error type	1, Error message from the NC program	

ID 13558

The origin of new coordinate system is incompletely defined.

Description	Incomplete definition of the origin of the new coordinate system.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Complete definition of the origin of the new coordinate system with the corresponding input parameters.
Error type	1, Error message from the NC program		

ID 13559

The rotation mode is not or incorrectly defined.			
Description	The rotation mode is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the rotation mode at cycle call with a valid value.
Parameter	%1:	Current value	
		Transferred rotation mode	
	%2:	Lower limit value	
		Smallest permitted rotation mode (integer)	
	%3:	Upper limit value	
		Largest permitted rotation mode (integer)	
Error type	1, Error message from the NC program		

ID 13560

The rotation sequence is not or incorrectly defined.			
Description	The rotation sequence is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the rotation sequence at cycle call with a valid value.
Parameter	%1:	Current value	
		Transferred rotation sequence	
	%2:	Limit value	
		Smallest permitted rotation sequence (integer)	
	%3:	Limit value	
		Largest permitted rotation sequence (integer)	
Error type	1, Error message from the NC program		

ID 13561

The target plane is not or incorrectly defined.			
Description	Invalid definition of the plane for swivelling. This must be completely defined with all three reference points (@P6 to @P14) or all three rotation angles (@P15 to @P17).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the destination plane at cycle call with all reference points or all rotation angles.
Error type	1, Error message from the NC program		

ID 13562

The transferred rotation angles lead to an inclination greater than 90 degrees.			
Description	The rotation angles specified in the input parameters lead to an inclination of the tool exceeding 90°.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the destination plane at cycle call with valid rotation angles or reference points.
Error type	1, Error message from the NC program		

ID 13563

The transferred reference points lie on a line.			
Description	The reference points to define the destination plane specified in the input parameters lie on a straight line		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the destination plane at cycle call with valid reference points.
Error type	1, Error message from the NC program		

ID 13600

Timer-ID missing.			
Description	If the SysServiceWait cycle is called by a timeout limit in @P2, a timer ID must also be specified in @P3. This timer is then used for the timeout. The timer ID @P3 was not specified.		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify the timer ID in @P3 or dispense with a timeout mechanism.
Error type	1, Error message from the NC program		

ID 13601

Max. waiting time for service was exceeded.			
Description	The SysServiceWait cycle waits for a reply from the external service for the time specified in @P2. This error is output if the external service fails to reply within this waiting time.		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Increase the waiting time in @P2 or reduce the runtime of the external service
Parameter	%1:	Upper limit value	
		Maximum waiting time	
	%2:	Current value	
		Actual waiting time	
Error type	1, Error message from the NC program		

ID 13602

Command for Mar-Wireless cycle was not defined.			
Description	A valid value for @P1 must be transferred when the SysServiceMwl cycle is called. This is not the case. For more information, see [FCT-C31].		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify a valid value for @P1 at the cycle call of SysServiceMwl.
Error type	1, Error message from the NC program		

ID 13603

Invalid command for the Mar-Wireless cycle.			
Description	A valid value for the command in @P1 must be transferred when the SysServiceMwl cycle is called. For a list of all valid numbers for commands, see [FCT-C31].		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify a valid value of the command in @P1 at the cycle call of SysServiceMwl.
Parameter	%1:	Current value	
		Number of the command used in @P1	
	%2:	Lower limit value	
		Smallest valid number for the command	
	%3:	Upper limit value	
		Largest valid number for the command	
Error type	1, Error message from the NC program		

ID 13604

Missing device number in @P2 for Mar-Wireless cycle.			
Description	The SysServiceMwl expects a valid device number in @P2. It must be set in advance on the device concerned.		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify the device number in @P2 or check the device number.
Error type	1, Error message from the NC program		

ID 13605

Missing device type in @P3 for Mar-Wireless cycle.			
Description	The SysServiceMwl expects a valid number for the device type in @P3. For a list of all valid numbers for device types, see [FCT-C31].		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify a valid value for the device type in @P3 at the cycle call of SysServiceMwl.
Error type	1, Error message from the NC program		

ID 13606

Invalid device type for the Mar-Wireless cycle.			
Description	The SysServiceMwl cycle expects a valid number for use of the device type in @P3. For a list of all valid numbers for device types, see [FCT-C31].		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify a valid value for the device type in @P3 at the cycle call of SysServiceMwl.
Parameter	%1:	Current value	
		Number of the device type used in @P3	
	%2:	Lower limit value	
		Smallest valid number of the device type in @P3	
	%3:	Upper limit value	
		Largest number of the device type in @P3	
Error type	1, Error message from the NC program		

2.2.4 ID-range 14000-14249

ID 14000

Number of assigned positions is wrong.			
Description	The variable V.L.N containing the number of assigned positions is either not defined or the assigned number is too small.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the number of positions.
Parameter	%1:	Error value	
		Number of positions (integer)	
	%2:	Lower limit value	
		Minimum number (integer)	
Error type	1, Error message from the NC program		

ID 14001

Assigned positions are wrong.			
Description	The variables V.L.X, V.L.Y or V.L.Z containing the assigned positions are not or incompletely defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Parameter	%1:	Logical axis number	
		Logical axis number of concerned axis (integer)	
Error type	1, Error message from the NC program		

ID 14002

Failure in definition of parameter for return values.			
Description	The variables V.L.RESULT_V, V.L.RESULT_V1… or V.L.RESULT_M, V.L.RESULT_M1… representing returned vectors or matrices are defined in the program to be called. However, the defined size does not correspond to the required size. This means that the defined vectors or matrices are too small to save all the return values.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting of transferred arrays, correct the size.
Error type	1, Error message from the NC program		

ID 14020

Length of vector of direction of rotary axis or shift direction [@P4, @P5, @P6] is 0.			
Description	The length of the direction vector is 0, this is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Check parameter setting and correct the input parameters @P4, @P5, @P6.
Error type	1, Error message from the NC program		

ID 14021

Unknown mode @P20 is defined.			
Description	An unknown mode was transferred with parameter @P20. Permitted values are: 0 – rotation (default), 1 - translation		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameter @P20 for the mode.
Error type	1, Error message from the NC program		

ID 14040

Missing index of Volumetric Compensation for logging/file archiving (@P1).		
Description	The file handling index for logging/archiving/activating the Volumetric Compensation was not defined.	
Response	Class	- Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	- Check parameter setting and correct the input parameter @P1.
Parameter	%1:	Current value
	%2:	Lower limit value
	%3:	Upper limit value
Error type	1, Error message from the NC program	

ID 14041

Missing index of parameter file for Volumetric Compensation for logging/file archiving (@P2).		
Description	The index for the corresponding file of the Volumetric Compensation was not defined.	
Response	Class	- Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	- Check parameter setting and correct the input parameter @P2.
Parameter	%1:	Current value
	%2:	Lower limit value
	%3:	Upper limit value
Error type	1, Error message from the NC program	

ID 14042

Invalid string in V.G.VOLCOMP[@P1].FILE[@P2] for Volumetric Compensation for logging/file archiving.			
Description	The string for the path and name of logging and compensation files is not permitted. The files cannot be written or archived.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the transferred file paths and names.
Error type	1, Error message from the NC program		

ID 14043

Undefined mode for handling of files (@P3).			
Description	The transferred parameter @P3 defines the mode for file handling. Permitted values are 0 or 1. The value describes whether a measurement to determine parameters for Volumetric Compensation is executed (@P3=1) or a previously compensated kinematic is remeasured (@P3=0). Accordingly, the logging file is given the suffix *_log or *_control_log or the compensation file is given the suffix * or *_control.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameter @P3.
Parameter	%1:	Current value	
	%2:	Lower limit value	
	%3:	Upper limit value	
Error type	1, Error message from the NC program		

ID 14060

Kinematic is not defined or not supported.			
Description	The parameter V.L.KIN_TYP which must be defined before the calculation subroutine is called is either not defined or assigned with an unsupported kinematic.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and correct the parameter setting of the kinematic ID V.L.KIN_TYP.
Parameter	%1:	Error value	
		Transferred kinematic ID	
Error type	1, Error message from the NC program		

ID 14061

Missing measurement for calculation of compensation data.	
Description	The measurement data required to calculate the compensation table is incomplete, e.g. missing or incompletely defined vector elements.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Parameter	%1:	Current value	
		Specified array size	
	%2:	Lower limit value	
		Minimum number or required elements in the corresponding array	
Error type	1, Error message from the NC program		

ID 14062

The errors of kinematic offsets are greater than allowed maximum.			
Description	The calculated errors are outside the tolerances specified by the machine manufacturer. Request the machine manufacturer to check the state of the machine.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Contact your machine manufacturer.
Parameter	%1:	Current value	
		Deviation	
	%2:	Lower limit value	
		Maximal permitted negative deviation	
	%3:	Upper limit value	
		Maximal permitted positive deviation	
Error type	1, Error message from the NC program		

ID 14063

The sequence for measuring C0, A0 and A90 is not correct.			
Description	The sequence for measuring the position of the A and C axes and to correct the cardanic angle must be maintained since the measurements are based on the results of previous or already executed corrections. 1. The measurement C0 and the cardanic angle must first be started in the program to be called: V.P.MEASURE_C0_A0_A90 = 1 C0 must be corrected with the output value. 2. Secondly, the A0 position must be measured: V.P.MEASURE_C0_A0_A90 = 2 A0 must be corrected with the output value. 3. Finally, the measurement of A+/-90 must be started: V.P.MEASURE_C0_A0_A90 = 3. The cardanic angle must be corrected with the output value.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Contact your machine manufacturer.
Parameter	%1:	Current value	
Error type	1, Error message from the NC program		

ID 14064

Kinematic parameter HD1 has to be 0.			
Description	The parameter V.P.NUMBER_MEAS_PROC was not defined in the main program.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameter V.P.NUMBER_MEAS_PROC accordingly.
Error type	1, Error message from the NC program		

ID 14065

Position of second rotary axis is not defined.			
Description	The position of the second rotary axis was not transferred. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters.
Error type	1, Error message from the NC program		

ID 14066

The minimum width when probing the sphere is too small.			
Description	The minimum width on probing the sphere (V.P.LATITUDE) is too small to estimate correct values.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer a value between 0 and 70 for V.P.LATITUDE.
Error type	1, Error message from the NC program		

ID 14067

Error during transfer of start and end angle.			
Description	The start angle (V.P.START_ANGLE_MEAS) and end angle (V.P.END_ANGLE_MEAS) must have identical values if only one sphere is measured.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer identical values for start and end angles,
Error type	1, Error message from the NC program		

ID 14068

Incorrect value for the output format.			
Description	Only the values 0, 1 and 2 are permissible for the output format.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define valid values for V.P.LIST_FORMAT.

Error type	1, Error message from the NC program

ID 14069

Incorrect value for the distance to the sphere during deactivation of the measurement.			
Description	Only values greater than 0 are permissible for the distance to the sphere when the measurement is disabled (@P6).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define valid values for@ P6.
Parameter	%1:	Current value	
		Current active distance.	
Error type	1, Error message from the NC program		

ID 14080

One of the two main axes is missing.			
Description	One of the two main axes is missing.		
Response	Class	2	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	3	Check the configuration.
Error type	1, Error message from the NC program		

ID 14081

Main spindle not available.			
Description	Main spindle not available.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the configuration.
Error type	1, Error message from the NC program		

ID 14082

Thread depth @P22 not specified.			
Description	Thread depth @P22 not specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14083

Thread depth @P22 must be greater than 0.	
Description	Thread depth @P22 must be greater than 0

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14084

Thread pitch @P82 not specified.			
Description	Thread pitch @P82 not specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14085

Thread pitch @P82 must be greater than 0.			
Description	Thread pitch @P82 must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14086

Thread end @P85 or @P92 not specified.			
Description	Thread end @P85 or @P92 not specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14087

Specification of machining mode for inside/outside thread is invalid (1=inside thread, 2=outside thread).			
Description	Specification of machining mode for inside/outside thread is invalid (1=inside thread, 2=outside thread).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14088

Thread start @P100 or @P101 not specified.

Description	Thread start @P100 or @P101 not specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14089

Thread end given via end point @P92 and angle @P95.			
Description	Thread end specified by end point @P92 and angle @P95.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14090

Cone angle @P95 must be greater than -89° and smaller than 89°.			
Description	Cone angle @P95 must be greater than -89° and smaller than 89°.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14091

Start infeed @P51 must be greater than or equal to 0.			
Description	Start infeed @P51 must be greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14092

Number of roughing cycles @P84 must be an integer and greater than or equal to 0.			
Description	Number of roughing cycles @P84 must be an integer and greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14093

Start infeed @P51 or number of roughing cycles @P84 are missing.

Description	Start infeed @P51 or number of roughing cycles @P84 are missing.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14094

Specification of start infeed @P51 and number of roughing cycles @P84 are mutually exclusive.			
Description	Specification of start infeed @P51 and number of roughing cycles @P84 are mutually exclusive.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14095

Finishing allowance @P51 must be greater than or equal to 0.			
Description	Finishing allowance @P51 must be greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14096

Infeed angle @P14 must be greater than -89° and smaller than 89°.			
Description	Infeed angle @P14 must be greater than -89° and smaller than 89°.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14097

Freerun distance @P51 must be greater than 0.			
Description	Freerun distance @P51 must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14098

Number of threads @P90 must be an integer and greater than or equal to 0.			
Description	Number of threads @P90 must be an integer and greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14099

Number of empty cuts @P89 must be an integer and greater than or equal to 0.			
Description	Number of empty cuts @P89 must be an integer and greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14100

Infeed mode @P111 must be specified as an integer and be greater than or equal to 0.			
Description	Infeed mode @P111 must be specified as an integer and be greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14101

Lead distance @P80 must be greater than or equal to 0.			
Description	Lead distance @P80 must be greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14102

Overrun distance @P87 must be greater than or equal to 0.			
Description	Overrun distance @P87 must be greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14103

Feed rate @P5 must be greater than or equal to 0.			
Description	Feed rate @P5 must be greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14104

Specification of alternating flank infeed @P110 is invalid (0=off 1=on).			
Description	Specification of alternating flank infeed @P110 is invalid (0=off 1=on)..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14105

Specification for unit @P93 is invalid (0=default 1=mm 2=inch)..			
Description	Specification for unit @P93 is invalid (0=default 1=mm 2=inch)..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14106

Thread start angle offset @P112 must be greater than/equal to 0° and smaller than 360°.			
Description	Thread start angle offset @P112 must be greater than/equal to 0° and less than 360°.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14107

Specification of absolute/relative programming @P96 is invalid (0=absolute 1=relative)..			
Description	Specification of absolute/relative programming @P96 is invalid (0=absolute 1=relative)..		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14108

Thread change depth @P113 must be greater than or equal to 0.			
Description	Thread change depth @P113 must be greater than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14109

Specification @P114 for thread sequence is invalid (0=ascending 1=opposite).			
Description	Specification @P114 for thread sequence is invalid (0=ascending 1=opposite).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14110

Plane specification @P115 is invalid (0=default 1=G17 2=G18 3=G19).			
Description	Plane specification @P115 is invalid (0=default 1=G17 2=G18 3=G19).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14111

Cone angle @P95 not specified.			
Description	Cone angle @P95 not specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14112

Thread length too small.			
Description	Thread length too small.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14113

Thread end specified by end point @P95 or thread length @P116 is missing.			
Description	Thread end specified by end point @P95 or thread length @P116 is missing.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

ID 14114

Thread end given via end point @P95 and thread length @P116.			
Description	Thread end specified by end point @P95 and thread length @P116.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the input parameters.
Error type	1, Error message from the NC program		

2.2.5 ID-range 14250-14499

ID 14300

Illegal machining mode.			
Description	The machining mode parameter for the face milling cycle is not permitted. Permitted values are: 1. Rough milling, 2. Finishing, 3. Rough milling + finishing		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Assign a valid value to the machining mode parameter.
Error type	1, Error message from the NC program		

ID 14301

No valid starting point defined.			
Description	No valid starting point is defined for the face milling cycle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a valid starting point.
Error type	1, Error message from the NC program		

ID 14302

No valid target point defined.			
Description	No valid target point is defined for the face milling cycle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a valid target point.
Error type	1, Error message from the NC program		

ID 14303

Starting point and target point must be different.			
Description	The starting point and the target point of the cycle must be different.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the specified points.
Error type	1, Error message from the NC program		

ID 14304

Finishing allowance must be smaller than the difference between Z position start point and Z position end point.

Description	The finishing allowance is greater than the maximum possible delta. This is not permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the finishing allowance.
Error type	1, Error message from the NC program		

ID 14305

Z coordinate must be greater at starting point than at target point.			
Description	In the face milling cycle in the plane G17, the Z coordinate of the starting point must be greater than the Z coordinate of the target point. This is not the case.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the Z coordinate of the start point.
Error type	1, Error message from the NC program		

ID 14306

X coordinate must be smaller at starting point than at target point.			
Description	In the face milling cycle in the plane G19, the X coordinate of the starting point must be smaller than the X coordinate of target point. This is not the case.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the X coordinate of the starting point.
Error type	1, Error message from the NC program		

ID 14307

Y coordinate must be smaller at starting point than at target point.			
Description	In the face milling cycle in the plane G18, the Y coordinate of the starting point must be smaller than the Y coordinate of the target point. This is not the case.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the Y coordinate of the starting point.
Error type	1, Error message from the NC program		

ID 14308

Depth of spigot is not or incorrectly defined.			
Description	The spigot depth is not defined or incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spigot length in X at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14309

Length of spigot is not or incorrectly defined in X.			
Description	The spigot length in the X direction is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spigot length in X at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14310

Width of spigot in Y is not or incorrectly defined.			
Description	The spigot width in the Y direction is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spigot width in Y at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14311

Diameter of spigot is not or incorrectly defined.			
Description	The spigot diameter is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spigot diameter at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14312

Length of blank in X is not or incorrectly defined.			
Description	The length of the blank in the X direction is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the length or the blank in X at cycle call with a value greater than or equal to the spigot length or spigot diameter.
Error type	1, Error message from the NC program		

ID 14313

Length of the blank in Y is not or incorrectly defined.			
Description	The length of the blank in the Y direction is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the length of the blank in Y at cycle call with a value greater than or equal to the spigot length or spigot diameter.
Error type	1, Error message from the NC program		

ID 14314

Incomplete definition of spigot/pocket centre point.			
Description	The definition of the spigot/pocket centre point is incomplete. At least one specification for the X or Y axis is missing.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Complete specification of the definition of the spigot/pocket centre point with a value for the X and Y axis.
Error type	1, Error message from the NC program		

ID 14315

Definition of corner radius is invalid.			
Description	Invalid definition of the corner radius. It may not exceed half of the shortest side length of the spigot.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the corner radius at cycle call with a value between 0 and half the shortest side length of the spigot.
Error type	1, Error message from the NC program		

ID 14316

The maximum infeed in Z is not or incorrectly defined.			
Description	The maximum infeed in Z is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the maximum infeed in Z at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14317

The maximum infeed in X and Y is not or incorrectly defined.			
Description	The maximum infeed in X and Y is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the maximum infeed in X and Y at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14318

The finishing allowance is incorrectly defined.			
Description	The finishing allowance is not defined with a permitted value.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the finishing allowance at cycle call with a value not less than 0.

Error type	1, Error message from the NC program

ID 14319

The machining mode is not or incorrectly defined.			
Description	The machining mode is not defined or is invalid. The values 1, 2 and 3 are valid. 1 – only roughing, 2 - only finishing, 3 – roughing and finishing		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the machining mode at cycle call with an integer value of 1, 2 or 3.
Parameter	%1:	Current value	
		Transferred machining mode	
	%2:	Lower limit value	
		Smallest permitted machining mode (integer)	
	%3:	Upper limit value	
		Largest permitted machining mode (integer)	
	%4:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 14320

The number of edges is not or incorrectly defined.			
Description	The number of edges is not defined or is invalid. Integers with a value greater than 0 are valid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the number of edges at cycle call with an integer value greater than 0.
Error type	1, Error message from the NC program		

ID 14321

Edge length and width across flats are not or incorrectly defined.			
Description	The edge length or width across flats are not defined or are invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the edge length or the width across flats at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14322

The radius of the blank is not or incorrectly defined.	
Description	The blank radius is not defined or is invalid.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the blank diameter at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14323

Corner radius and chamfer width are not or incorrectly defined.			
Description	The corner radius and chamfer width are not defined or are invalid. Both values may not be transferred at the same time. If the number of corners is less than 3, no specification is permitted here.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the corner radius or chamfer width with a value permitted for geometric data.
Error type	1, Error message from the NC program		

ID 14324

Safety clearance in Z is not or incorrectly defined.			
Description	The safety clearance in the Z plane is not defined or is invalid. The value of the reference plane added to the safety clearance may not be greater than the value of the retraction plane		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the safety clearance in the Z plane at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14325

Safety clearance in X and Y is not or incorrectly defined.			
Description	The safety clearance in the X and Y planes is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the safety clearance in the X and Y planes at cycle call with a value greater than 0.
Parameter	%1:	Current value	
		Transferred safety clearance in XY.	
	%2:	Upper limit value	
		Maximum permitted safety clearance in XY.	
Error type	1, Error message from the NC program		

ID 14326

The depth to be manufactured is not or incorrectly defined.
Description: The depth to be manufactured is not defined or is invalid.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the depth to be manufactured at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14327

Pocket diameter is not or incorrectly defined.			
Description	The pocket diameter is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the pocket diameter at cycle call with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14328

Infeed mode in the XY plane is defined invalid.			
Description	Invalid definition of infeed mode in the XY plane.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the infeed mode at cycle call with a value of 1 or 2.
Error type	1, Error message from the NC program		

ID 14329

Orientation angle is not defined.			
Description	The orientation angle is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the orientation angle at cycle call with a valid value.
Error type	1, Error message from the NC program		

ID 14330

Mode of infeed is not or incorrectly defined.			
Description	The infeed mode is not defined or is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the infeed mode at cycle call with a valid value.
Parameter	%1:	Current value	
		Transferred infeed mode	
	%2:	Lower limit value	
		Smallest permitted infeed mode (integer)	
	%3:	Upper limit value	
		Largest permitted infeed mode (integer)	

Error type	1, Error message from the NC program

ID 14331

The number of available machining coordinate systems (CS) is not sufficient.			
Description	The cycle uses internal machining coordinate systems (CS). There is no longer a sufficient number of free machining coordinate systems.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Deactivate unused machining coordinate systems before the cycle call.
Parameter	%1:	Current value	
		Current number of free machining coordinate systems.	
	%2:	Lower limit value	
		Required number of free machining coordinate systems. (integer).	
Error type	1, Error message from the NC program		

ID 14332

The defined dimensions lead to overlapping of the milling grooves.			
Description	The dimensions defined in the input parameters cannot be applied since they result in overlapping of the milling grooves.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Adjust the dimensions in the input parameters.
Error type	1, Error message from the NC program		

ID 14333

Width of slot is incorrectly defined.			
Description	Invalid definition of the slot width. This may not exceed double the value of the tool diameter.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the slot width at cycle call with a valid value.
Error type	1, Error message from the NC program		

ID 14334

The defined dimensions don`t permit helical plunging.			
Description	Helical plunging was defined in the input parameters. However, the dimensions specified for plunging do not permit this.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Increase the dimension or change over to vertical plunging.
Error type	1, Error message from the NC program		

ID 14335

The tool type is not defined or not allowed.			
Description	No permissible value is defined for the tool type.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible tool type.
Parameter	%1:	Current value	
		Defined value for the tool type	
	%2:	Lower limit value	
		Smallest value for tool type	
	%3:	Upper limit value	
		Greatest value for tool type	
Error type	1, Error message from the NC program		

ID 14336

The cutting length of the insert tool is not or incorrectly defined.			
Description	No permissible value is defined for the cutting length of the insert tool.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a permissible value for the cutting length.
Error type	1, Error message from the NC program		

ID 14337

The identification number of the main contour was not defined or is incorrect.			
Description	No main contour identification number or an incorrect one was transferred. With pocket milling, the main contour describes the pocket contour; with spigot milling, it describes the blank contour; and with path milling it describes the path contour.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer a positive and integer identification number of the main contour with a value greater than 0.
Error type	1, Error message from the NC program		

ID 14338

The identification number of a contour is incorrect.			
Description	The identification number of a contour was transferred with an incorrect value.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer an identification number or an existing defined contour with an integer value greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14339

Incorrect definition of the parameter for the finishing allowance.			
Description	The finishing allowance was incorrectly defined. It must be greater than 0 but may not be greater than the maximum infeed in Z at the base.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a correct value for the finishing allowance.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14340

Incorrect definition of the parameter for the pitch of the plunging helix.			
Description	The pitch of the plunging helix was defined with a value of less than or equal to 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a pitch for the plunging helix with a value greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14341

The transferred radius of the plunging helix is not admissible.			
Description	The transferred radius of the plunging helix results in a collision with the contour.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Reduce the transferred radius of the plunging helix or select vertical plunge (predrill if necessary).
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Current value	
		Current radius	
	%3:	Upper limit value	
		Largest possible radius	
Error type	1, Error message from the NC program		

ID 14342

The transferred radius of the reference tool from the previous machining operation is not defined or is defined incorrectly.			
Description	The transferred radius of the reference tool is not or is incorrectly defined. In order to remove residual material, its radius must be greater than the current tool radius.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer a reference tool radius that is greater than the current radius.

Parameter	%1:	Error value
		Affected input parameter @P__.
Error type	1, Error message from the NC program	

ID 14343

The transferred radius for contour milling is not defined or defined incorrectly.			
Description	The transferred radius of contour milling is not defined or is incorrectly defined. The value must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer a contour milling radius that is greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14344

Collision with the contour during plunging.			
Description	The current tool radius causes a collision with the contour on plunging.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Reduce the tool radius for predrilling.
Parameter	%1:	Current value	
		Current radius	
	%2:	Upper limit value	
		Largest possible radius	
Error type	1, Error message from the NC program		

ID 14345

Feed rate is not defined or incorrect.			
Description	No or an invalid feed rate (<=0) for machining or plunging is defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the feed rate.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14346

Reference cycle is not defined or incorrect.	
Description	No reference cycle or an invalid reference cycle is defined for machining.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the reference cycle.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14347

An approach or retract movement was defined incorrectly.			
Description	A parameter to define an approach or retract movement was incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and correct the input parameters for the approach or retract movement.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 14348

The mode for tool radius compensation was defined incorrectly.			
Description	The parameter to define the mode for tool radius compensation was incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and modify the input parameters for the tool radius compensation mode.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 14349

The drilling diameter must be larger than the tool diameter.			
Description	The drilling diameter must be larger than the tool diameter.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Use a smaller tool.
Error type	1, Error message from the NC program		

2.2.5.1 ID 14350

No configuration of the measuring station exists for the given station number available.			
Description	The configuration file SysCalibConfigToolSettingProbe.nc does not contain any values for the transferred measuring station number.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the required configuration values in the configuration file as described in the documentation.
Parameter	%1:	Current value	
		Transferred station number	
Error type	1, Error message from the NC program		

ID 14351

Incorrect numbering of the measuring station.			
Description	The numbering of the measuring stations was not transferred or the number transferred is invalid.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer a valid measuring station number with an integer value greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Current value	
		Current numbering	
Error type	1, Error message from the NC program		

ID 14352

Invalid diameter of the measuring station.			
Description	The configuration file SysCalibConfigToolSettingProbe.nc contains an incorrect value for the diameter of the measuring station.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Configure the diameter of the measuring station in the configuration file SysCalibConfigToolSettingProbe.nc with a value greater than 0 as described in the documentation.
Parameter	%1:	Current value	
		Current diameter	
	%2:	Lower limit value	
		Minimum diameter	
Error type	1, Error message from the NC program		

ID 14353

Invalid feed rate definition.	
Description	The configuration file SysCalibConfigToolSettingProbe.nc contains an incorrect value for the measuring or positioning feed rate.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Configure the measuring or positioning feed rate in the configuration file SysCalibConfigToolSettingProbe.nc with a value greater than 0 as described in the documentation.
Parameter	%1:	Current value	
		Current measuring feed rate	
	%2:	Current value	
		Current positioning feed rate	
Error type	1, Error message from the NC program		

ID 14354

Invalid measurement offset in Z.			
Description	The measurement offset in Z for the lateral measurement is not defined or is incorrect.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measurement offset in Z with a value greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Current value	
		Current measurement offset in Z.	
Error type	1, Error message from the NC program		

ID 14355

Invalid safety clearance for lateral measurement.			
Description	The safety clearance for lateral measurement is not defined or is incorrect.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the safety clearance in Z with a value greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Current value	
		Current safety clearance	
Error type	1, Error message from the NC program		

ID 14357

Invalid axis selection			
Description	The transferred parameter to select the axis is defined incorrectly.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the parameter to select the axis with a value 0 (X axis) or 1 (Y axis).
Error type	1, Error message from the NC program		

ID 14358

Centring diameter missing.			
Description	After specifying the point angle of the drilling tool, a centring diameter must be specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify a centring diameter.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14359

Parameter not allowed!			
Description	The specified parameter is not required or is not allowed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Deselect the parameter.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14360

Infeed too big, residual material will remain.			
Description	Number of infeeds is too low. More infeeds are required to prevent any residual material from remaining between infeeds.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Increase the number of infeeds.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14361

Value of parameter not allowed!			
Description	The specified parameter may not be 0 or negative.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the value of the parameter.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14362

Value of parameter not allowed!			
Description	The specified parameter may only be 0 or 1.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the value of the parameter.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14363

Spindle rotation direction is incorrectly defined.			
Description	The value may be 3 for M3 (cw) and 4 for M4 (ccw).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the value.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14364

Thread pitch not defined.			
Description	Thread milling requires the specification of the thread pitch. This message is generated if the thread pitch is not specified in the input parameters or is 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the thread pitch and correct the input parameters.
Error type	1, Error message from the NC program		

ID 14365

Machining direction not allowed for centre milling.			
Description	Permitted machining directions are 1 to 4.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Deselect centre milling or change the machining direction.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14366

Diameter of the thread is smaller than the tool diameter.	
Description	The thread diameter may not be less than the tool diameter.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the value.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.2 ID 14368

The corrected safety plane exceeds the retraction plane.			
Description	In the cycle, the safety clearance specified in the input parameters is corrected slightly upwards to allow the correct thread pitch. The correction means that the specified retraction plane cannot be maintained.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Increase the retraction plane or reduce the safety clearance.
Error type	1, Error message from the NC program		

ID 14369

The maximum infeed in X and Y could leave residual material.			
Description	The maximum infeed defined in X and Y is too high and could leave residual material.		
Response	Class	-	
Solution	Class	-	Reduce the maximum infeed in X and Y at cycle call. Maximum value 85.3% or tool diameter
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14370

Unknown milling direction defined.			
Description	A milling direction is required to calculate milling motions. This can be selected by specifying an identification number: 0: Down milling, 1 Up milling. This message is generated when a different identification number other than 0 or 1 was programmed in the corresponding input parameter.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the milling direction and correct the input parameters.
Error type	1, Error message from the NC program		

ID 14371

Spindle speed not defined.	
Description	A valid spindle speed must be programmed and the spindle must be activated by M3/ M4 before the cycle is called up.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify a valid spindle speed and activate the spindle before the cycle is called.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 14372

Machining selection not set.			
Description	The selected machining type defines which cycles are processed. The selection includes: SySMillContourPreDrilling, SySMillContourPocket, SySMillContourSpigot, SySMillContourResidual and finishing the SySMillContourPocket and SySMillContourSpigot cycles.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Select a machining type
Error type	1, Error message from the NC program		

ID 14373

Tool number is not defined or defined incorrectly.			
Description	A tool number > 0 must be specified to call a tool.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the tool number.
Error type	1, Error message from the NC program		

ID 14374

The tool compensation selection is not defined or defined incorrectly.			
Description	A tool compensation selection >= 0 must be specified to call a tool.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a tool compensation selection.
Error type	1, Error message from the NC program		

ID 14375

Spindle speed not or incorrectly defined.			
Description	A speed > 0 must be specified to program the spindle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spindle speed.
Error type	1, Error message from the NC program		

ID 14376

Machining feed rate not defined or defined incorrectly.			
Description	A feed rate > 0 must be specified for machining.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the machining feed rate.
Error type	1, Error message from the NC program		

ID 14377

Spindle function not defined or incorrectly defined.			
Description	A spindle function (M3, M4) must be specified for machining to define the rotation direction of the spindle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the spindle function.
Error type	1, Error message from the NC program		

ID 14378

The additional offset is incorrectly defined.			
Description	Invalid definition of the additional offset in X and Y.		
Response	Class	-	
Solution	Class	-	Define the additional offset in X and Z at cycle call with a value greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14386

No valid reference cycle was defined.			
Description	The reference cycle was not defined or was incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a reference cycle according to the cycle documentation.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.3 ID 14388

No valid definition of the tool tip angle.			
Description	The tip angle of the tool is incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Define a tool tip angle between 0° and 180°.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

2.2.5.4 ID 14389

No valid definition of the font weight.			
Description	The font weight is incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define a font weight between 0 and the tool diameter.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.5 ID 14390

No valid entry of the 0-position of the spindle.			
Description	The 0 position of the spindle was not defined correctly.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Permitted values: 0, 90, 180, 270.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14391

Plunge angle too high or zero.			
Description	The plunge angle for the Z infeed is too high or is zero.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Reduce the value of the ramp angle. Permitted range is from -90 to +90 with the exception of zero.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14392

Unknown spindle direction of the reference cycle defined.

Description	A spindle direction is required to calculate the ramp position. This can be selected by specifying an identification number: 3: Clockwise, 4: counter-clockwise. This message is generated when a different identification number other than 3 or 4 was programmed in the corresponding input parameter.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the milling direction and correct the input parameters.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14393

Chamfer depth and chamfer width are not or incorrectly defined.			
Description	There is no definition of the chamfer width and chamfer depth or the definition is invalid. At least one of the two values must be defined. Both values may not be transferred at the same time.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the chamfer width or chamfer depth with a value permitted for geometric data.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14394

The radial tool offset to the edge is not or incorrectly defined.			
Description	The tool offset to the edge is not defined or is incorrectly defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the tool offset to the edge with a value greater than 0.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14395

The tool radius is too small for the selected geometry data.			
Description	The current tool radius is too small to maintain the transferred chamfer width (chamfer depth) or the machining depth (radial offset).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Use a wider chamfer milling cutter or reduce the chamfer width.
Error type	1, Error message from the NC program		

ID 14396

The pattern retraction plane is located below the end position of a cycle execution.			
Description	The tool is retracted to the pattern retraction plane to a position between the individual pattern positions. This error message appears if the tool is above this retraction plane after cycle execution.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Use a higher value for the retraction plane PATTERN_RETRACT.
Error type	1, Error message from the NC program		

ID 14397

There is no spindle configured.			
Description	A configured spindle is required for the called cycle to run successfully.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Configure a spindle.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

ID 14398

Machining depth and radial tool offset are defined improperly.			
Description	There is no definition of the machining depth or radial tool offset or the definition in invalid. One of the two values must be defined. Both values may not be transferred at the same time.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the machining depth or the radial tool offset with a value permitted for geometric data.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

ID 14399

The specified machining depth is too low for the selected geometry data.			
Description	The specified machining depth is too low to maintain the transferred chamfer depth (chamfer depth).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Reduce the chamfer width or increase the machining depth.
Error type	1, Error message from the NC program		

ID 14400

A rotation about the X or Y axis was defined.			
Description	A rotation about the X or Y axis is currently not available for pattern cycles.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Delete the rotation about the X or Y axis from the pattern description.
Error type	1, Error message from the NC program		

2.2.5.6 ID 14401

Invalid value of an input parameter.			
Description	An input parameter was not defined or it was defined outside the permissible value range.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the input parameter affected with a value within the permissible value range.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Current value	
		Transferred value	
	%3:	Lower limit value	
		Lower limit value	
	%4:	Upper limit value	
		Upper limit value	
Error type	1, Error message from the NC program		

2.2.5.7 ID 14402

Incomplete definition of circle centre point.			
Description	The definition of the circle centre point is incomplete. At least one specification for the X or Y axis is missing.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Complete specification of the definition of the circle centre point with a value for the X and Y axes.
Error type	1, Error message from the NC program		

2.2.5.8 ID 14403

The machining mode is not or incorrectly defined.			
Description	The machining mode is not defined or is defined incorrectly. The values 1, 2, 3, 4 and 5 are valid. 1 – only roughing, 2 - only finishing, 3 – roughing and finishing, 4 - pre-finishing, 5 - roughing, pre-finishing and finishing		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the machining mode at cycle call with an integer value of 1, 2, 3, 4 or 5.

Parameter	%1:	Current value
		Transferred machining mode
	%2:	Lower limit value
		Smallest permitted machining mode (integer)
	%3:	Upper limit value
		Largest permitted machining mode (integer)
Error type	1, Error message from the NC program	

2.2.5.9 ID 14404

Slot width not defined.			
Description	For slot milling, it is necessary to specify the slot width to calculate milled recess motions. This message is generated if the slot width is not programmed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the slot width and correct the input parameters.
Error type	1, Error message from the NC program		

2.2.5.10 ID 14405

Slot length not defined.			
Description	For slot milling, it is necessary to specify the slot length to calculate milled recess motions. This message is generated if the slot length is not programmed.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check and define the slot length and correct the input parameters.
Error type	1, Error message from the NC program		

2.2.5.11 ID 14406

Z start position in pilot hole is incorrectly defined.			
Description	The Z start position in the pilot hole must be deeper than the reference plane.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and specify the input parameters for the Z start position in the pilot hole.
Error type	1, Error message from the NC program		

2.2.5.12 ID 14407

No spindle speed defined for deep hole drilling.			
Description	A spindle speed must be specified for deep hole drilling.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and specify the spindle speed for deep hole drilling.

Error type	1, Error message from the NC program

2.2.5.13 ID 14408

Initial speed of the spindle too high (max 100 rpm).			
Description	The spindle speed for entry into the pilot hole is too high. This could lead to deformation or breakage of the tool and the initial speed of the spindle should not exceed 100 rpm.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check initial speed and adapt if necessary.
Error type	1, Error message from the NC program		

2.2.5.14 ID 14409

Missing indication of feed rate from safety clearance to Z start position in pilot hole.			
Description	The feed rate is not specified from the safety clearance to the Z start position in the pilot hole.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check parameter setting and specify the feed rate of the safety clearance to the Z start position in the pilot hole.
Error type	1, Error message from the NC program		

2.2.5.15 ID 14410

If the parameter @P11 (number of infeeds) is 1, the parameter @P10 (first infeed depth) is not necessary.			
Description	If the parameter @P11 (number of infeeds) is 1, the parameter @P10 (first infeed depth) is not necessary.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Disable parameter @P10 (first infeed depth) or, if 2 infeeds are required, set the parameter @P11 (number of infeeds) to 2.
Error type	1, Error message from the NC program		

2.2.5.16 ID 14411

Prepositioning of the probe not possible.			
Description	The configuration rile of the probe (SysCalibConfigTouchprobe.nc) defined a preposition before each measurement run in the measuring direction (V.CYC.SysConf_Spindle_orientation = 1). This is not permitted in this cycle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Disable preposition of the touch probe for this cycle-
Error type	1, Error message from the NC program		

2.2.5.17 ID 14412

System error			
Description	This error message contains information about internal states, information about the error cause and the error location.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Please contact the control manufacturer.
Parameter	%1:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

2.2.5.18 ID 14413

For the variable V.CYC.SysConf_Spindle_Pos_Dir only 0 or 1 permitted.			
Description	An incorrect value is entered for the variable V.CYC.SysConf_Spindle_Pos_Dir; only 0 or 1 are permitted.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the variable V.CYC.SysConf_Spindle_Pos_Dir in the NC program SysCalibConfigTouchprobe.nc.
Error type	1, Error message from the NC program		

2.2.5.19 ID 14414

Incorrect definition of the measuring feed rate for the second probing.			
Description	The measuring feed rate for the second probing was not or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring feed rate for the second probing with a value greater than or equal to 0 in the configuration file SysCalibConfigTouchprobe.nc.
Error type	1, Error message from the NC program		

2.2.5.20 ID 14415

Incorrect definition of the retraction distance for the second probing.			
Description	The retraction distance for the second probing was not or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the retraction distance for the second probing with a value greater than or equal to 0 in the configuration file SysCalibConfigTouchprobe.nc.
Error type	1, Error message from the NC program		

2.2.5.21 ID 14416

The calculated radial tool offset to the workpiece edge is too small.			
Description	The calculated radial tool offset to the workpiece edge for chamfer milling is less than or equal to zero. This problem occurs when the machining depth @P4 was selected too deep.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Reduce the specified machining depth.
Error type	1, Error message from the NC program		

2.2.5.22 ID 14418

The mandrel direction vector has a length of zero.			
Description	All the values of the mandrel direction vector were defined as zero.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify the length of a direction vector of the mandrel greater than zero.
Error type	1, Error message from the NC program		

2.2.5.23 ID 14419

The probe direction vector has a length of zero.			
Description	All the values of the touch probe direction vector were defined as zero.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify the length of a direction vector of the touch probe greater than zero.
Error type	1, Error message from the NC program		

2.2.5.24 ID 14420

The first specified axis compensation must already be active at the start.			
Description	The first axis compensation defined by the parameters must be selected at the start of the cycle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Select the first axis compensation before cycle start.
Error type	1, Error message from the NC program		

2.2.5.25 ID 14421

Changing an active zero offset (G54-G59) not permitted.			
Description	It is only permitted to change a zero offset (G54-G59) when the zero offset (G53) is disabled.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.

Solution	Class	-	Disable the function to describe the zero offset or disable the zero offset (G53).
Error type	1, Error message from the NC program		

2.2.5.26 ID 14422

Reducing the depth infeed only permitted with enabled @P10 and disabled @P11.			
Description	Reducing the depth infeed is only permitted when @P10 is enabled and @P11 disabled.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Disable the number of infeeds.
Error type	1, Error message from the NC program		

2.2.5.27 ID 14423

Minimum depth infeed only permitted with active reduction of depth infeed.			
Description	Minimum depth infeed only permitted when reduction of depth infeed is active.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Disable minimum depth infeed or enable the reduction of depth infeed.
Error type	1, Error message from the NC program		

2.2.5.28 ID 14424

Reduction of depth infeed requires specification of a minimum depth infeed.			
Description	A reduction of depth infeed requires that a minimum depth infeed is specified.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Specify a value for minimum depth infeed.
Error type	1, Error message from the NC program		

2.2.5.29 ID 14425

Centring diameter is larger than tool diameter.			
Description	The centring diameter may not be larger than the tool diameter.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the value.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.30 ID 14426

No centring diameter defined.			
Description	The centring diameter may not be 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the value.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.31 ID 14427

Update of the CNC version required.			
Description	The invoked subcycle uses CNC commands that are not available in the current CNC version.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Update the CNC version.
Error type	1, Error message from the NC program		

2.2.5.32 ID 14428

The transferred points do not have the required diversity.			
Description	Points were transferred to the calculation cycle and all the points lie on a straight line. This results in a singularity in the calculation.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check the transferred measuring points.
Error type	1, Error message from the NC program		

2.2.5.33 ID 14429

No feed rate for drilling through defined.			
Description	The feed rate for drilling through may not be 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Correct the value.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
	%2:	Logical axis number	
		Cycle affected.	
Error type	1, Error message from the NC program		

2.2.5.34 ID 14430

Length of measuring probe not defined.

Description	The length of the measuring probe may not be 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the length of the measuring probe.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.35 ID 14431

Positioning in at least one of the main axes is not defined.			
Description	Positioning was not defined in at least one of the main axes or the value is 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the positioning at cycle call with parameter @ P17 and/or @P18.
Error type	1, Error message from the NC program		

2.2.5.36 ID 14432

Pitch circle diameter is not defined.			
Description	No pitch circle diameter was defined or the value is <= 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the pitch circle diameter at cycle call with the parameter @P29.
Error type	1, Error message from the NC program		

2.2.5.37 ID 14433

Only one of the two parameters (@P15/@P30) may be defined.			
Description	Only one of the two parameters (@P15/@P30) may be defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define @P15 or @P30. Defining both is not permitted.
Error type	1, Error message from the NC program		

2.2.5.38 ID 14434

Error when specifying the angle between the holes.			
Description	The angle between the holes was specified in the cycle. The number of holes is calculated by dividing the total angle of the hole circle by the angle between the holes. This quantity must result in an integer.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the angle between the holes so that the total angle of the hole circle divided by the intermediate angle results in an integer.
Error type	1, Error message from the NC program		

2.2.5.39 ID 14435

At least 3 holes are required to calculate the centre of a pitch circle.			
Description	At least 3 holes are required to calculate the centre of a pitch circle.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Change the number of holes to at least 3 or the parameters required to calculate the number of holes.
Error type	1, Error message from the NC program		

2.2.5.40 ID 14436

Safety clearance too large.			
Description	The safety clearance exceeds the centre of the hole/slot.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Reduce the safety clearance or use a probe with a smaller diameter.
Error type	1, Error message from the NC program		

2.2.5.41 ID 14437

Machining direction of the infeed axis incorrectly defined.			
Description	The machining direction of the infeed axis is defined with 0 (e.g. Z0->Z1) or 1 (e.g.. Z1->Z0).		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the required machining direction of the infeed axis with 0 or 1.
Error type	1, Error message from the NC program		

2.2.5.42 ID 14438

Only one of the two parameters (@P6/@P12) may be defined.			
Description	Only one of the two parameters (@P6/@P12) may be defined.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define @P6 or @P12. Defining both is not permitted.
Error type	1, Error message from the NC program		

2.2.5.43 ID 14439

Only the specification 0 or 1 is permitted for this parameter.			
Description	The position of the chamfer before (0) or behind the contour (1) is defined with 0 or 1.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the required position of the chamfer with 0 or 1.
Error type	1, Error message from the NC program		

2.2.5.44 ID 14440

Only the specification 0 or 1 is permitted for this parameter.			
Description	Breakthrough detection active (1) or inactive (0) is defined with 0 or 1.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the breakthrough detection with 0 or 1.
Error type	1, Error message from the NC program		

2.2.5.45 ID 14441

Maximum value is 360°. Value is reduced to 360°.			
Description	The maximum value is 360°. The value is reduced to 360°.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Enter a value greater than 0 and smaller than or equal to 360.
Error type	1, Error message from the NC program		

2.2.5.46 ID 14442

Positioning in at least one of the main axes is not defined.			
Description	Positioning was not defined in at least one of the main axes.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the positioning at cycle call with parameter @P40 and @P41.
Error type	1, Error message from the NC program		

2.2.5.47 ID 14443

The parameter @P1 for the allowed position deviation has to be positive or equal to zero.			
Description	The specified position deviation for the CNC cycle for retraction of the electrode is less than zero.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Assign the parameter @P1 with a plausible permitted position deviation that is greater than or equal to zero.
Error type	1, Error message from the NC program		

2.2.5.48 ID 14444

The parameter @P2 for retraction velocity has to be positive.			
Description	The specified retraction velocity for the CNC cycle for retraction of the electrode is less than or equal to zero.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Assign the parameter @P2 with a permitted retraction velocity deviation that is greater than zero.

Error type	1, Error message from the NC program

2.2.5.49 ID 14445

The file for the given program name in @P3 cannot be found.			
Description	The specified program to enable transformation and axis settings in the ‘path erosion’ function cannot be found		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Assign the correct subroutines to the parameter @P3.
Error type	1, Error message from the NC program		

2.2.5.50 ID 14447

The file for the given program name in @P5 cannot be found.			
Description	The specified program containing the geometry for the ‘path erosion’ function cannot be found		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Assign the correct subroutines to the parameter @P5.
Error type	1, Error message from the NC program		

2.2.5.51 ID 14448

The file for the given program name in @P6 cannot be found.			
Description	The specified program to enable transformations after the ‘path erosion’ function cannot be found		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Assign the correct subroutines to the parameter @P6.
Error type	1, Error message from the NC program		

2.2.5.52 ID 14449

The CNC-cycle to retract the electrode requires the EDM license.			
Description	The CNC cycle was started without the necessary EDM license.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Activate the EDM license.
Error type	1, Error message from the NC program		

2.2.5.53 ID 14450

The parameter @P2 for retraction velocity has to be set.	
Description	The CNC cycle to retract the electrode was started without the parameter @P2 for retraction velocity. This parameter must be assigned.

Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Call the CNC cycle again with a valid value for the parameter @P2 for retraction velocity.
Error type	1, Error message from the NC program		

2.2.5.54 ID 14451

The parameter @P3 for the program name to activate the kinematic transformation has to be set.			
Description	The CNC cycle to retract the electrode was started without the parameter @P3 for the program name to enable the kinematic transformations. This parameter must be assigned.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Call the CNC cycle again with a valid value for the parameter @P3 for the program name to enable the kinematic transformations.
Error type	1, Error message from the NC program		

2.2.5.55 ID 14452

The parameter @P4 for the program name with the geometry of the ‘path erosion’ has to be set.			
Description	The CNC cycle to retract the electrode was started without the parameter @P4 for the program name containing the ‘path erosion’ function. This parameter must be assigned.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Call the CNC cycle again with a valid value for the parameter @P4 for the program name containing the geometry for the ‘path erosion’ function.
Error type	1, Error message from the NC program		

2.2.5.56 ID 14453

The parameter @P5 for the program name to activate the transformation after the ‘path erosion’ has to be set.			
Description	The CNC cycle to retract the electrode was started without the parameter @P5 for the program name to enable the transformations after the ‘path erosion’ function. This parameter must be assigned.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Call the CNC cycle again with a valid value for the parameter @P5 for the program name to enable the transformations after the ‘path erosion’ function.
Error type	1, Error message from the NC program		

2.2.5.57 ID 14461

The current position of the electrode exceeds the allowed deviation to the saved positions in the interruption point.			
Description	Before the start of the CNC cycle to retract the electrode, the electrode moved in relation to the stored positions. The distance of the current position exceeds the permitted distance specified in parameter @P1.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Check whether the electrode can be moved with a linear motion to the stored positions. If this is the case, the cycles can be recalled with a larger permitted deviation.
Error type	1, Error message from the NC program		

2.2.5.58 ID 14462

Start position is below laser position Z.			
Description	With a push measurement, the start position must be located above the laser position in Z.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Adapt the values in SysCalibConfigLaser.nc or change to pull measurement.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.59 ID 14463

Start position is above laser position Z.			
Description	With a pull measurement, the start position must be located below the laser position in Z.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Adapt the values in SysCalibConfigLaser.nc or change to push measurement.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.60 ID 14477

The measuring distance between measurements must not be 0.			
Description	The measuring distance between measurements must be greater than 0.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Enter a value for the measuring distance between measurements.
Parameter	%1:	Error value	
		Affected input parameter @P__.	

Error type	1, Error message from the NC program

2.2.5.61 ID 14478

String input parameter not transferred as string.			
Description	The input parameter of the string type could not be read as a string.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Transfer the affected parameter as a string.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.62 ID 14479

Simultaneous definition of measuring mode via the configuration file and the cycle parameters not allowed			
Description	The measuring mode may not be defined simultaneously in the configuration file and the cycle parameters.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Enter a unique definition of the measuring mode in the configuration file SysCalibConfigLaser.nc. It is recommended to execute laser orientation and calibration in the same way as measuring or a breakage check.
Parameter	%1:	Error value	
		Affected input parameter @P__.	
Error type	1, Error message from the NC program		

2.2.5.63 ID 14480

Incorrect definition of the accelerated measuring feed rate.			
Description	The accelerated measuring feed rate was not defined or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring feed rate with a value greater than or equal to 0 in the configuration file SysCalibConfigLaser.nc
Error type	1, Error message from the NC program		

2.2.5.64 ID 14481

Incorrect definition of the slowed measuring feed rate.			
Description	The slowed measuring feed rate was not defined or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring feed rate with a value greater than or equal to 0 in the configuration file SysCalibConfigLaser.nc

Error type	1, Error message from the NC program

2.2.5.65 ID 14482

Incorrect definition of the retraction distance for accelerated measurement.			
Description	The retraction distance for accelerated measurement was not or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the retraction distance for fast probing with a value greater than or equal to 0 in the configuration file SysCalibConfigLaser.nc.
Error type	1, Error message from the NC program		

2.2.5.66 ID 14483

Incorrect definition of the retraction distance for slowed measurement.			
Description	The retraction distance for slowed measurement was not or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the retraction distance for slow probing with a value greater than or equal to 0 in the configuration file SysCalibConfigLaser.nc.
Error type	1, Error message from the NC program		

2.2.5.67 ID 14484

The position of the laser focal point is not fully configured.			
Description	The position of the laser focal point was not fully defined in all three axes in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the position of the laser focal point in all three axes in the configuration file SysCalibConfigLaser.nc.
Error type	1, Error message from the NC program		

2.2.5.68 ID 14485

Incorrect definition of measuring and positioning direction.			
Description	The measuring and positioning directions were not defined or incorrectly defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the measuring and positioning directions with a value between 0 and 3 in the configuration file SysCalibConfigLaser.nc.
Error type	1, Error message from the NC program		

2.2.5.69 ID 14486

The start and end points of the measurement in Z are not fully configured.			
Description	The start and end points of the measurement in Z are not fully defined in the configuration file.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Define the start and end points of the measurement in Z in the configuration file SysCalibConfigLaser.nc.
Error type	1, Error message from the NC program		

ID 14356

Invalid start position of the tool or probe.			
Description	The start position of the tool is too far away from the expected start position.		
Response	Class	-	Error: Transition to error state. Complete reset of the NC channel necessary.
Solution	Class	-	Place the tool close to the expected start position at the start of the cycle.
Parameter	%1:		Expected value
			Expected start position in X.
	%2:		Expected value
			Expected start position in Y.
	%3:		Current value
			Current distance.
	%4:		Upper limit value
			Maximum permitted distance.
Error type	1, Error message from the NC program		

2.3 Decoding errors (syntax, semantic) (ID-range 20000-29999)

2.3.1 ID-range 20000-20249

ID 20002

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20003

G-code only sensible with programming of an axis.			

Description	At least one axis must be specified for the programmed G function, for syntax reasons or to program additional information,. Achtung: A coordinate value MUST be entered after the axis name, otherwise a new error message will be displayed. That value will be interpreted differently according to the programmed G-function.
	Example:
	Wrong:
	`N10 G00 X0 Y0 Z0`
	`N20 G160=2`
	`N1000 M30`
	
	`Correct:`
	`N10 G00 X0 Y0 Z0`
	`N20 G160=2 X1 Z1`
	N1000 M30		

Response	Class	1	Continue NC program processing.
Solution	Class	1	Programming of the additional axis informations for the G-function.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20007

Spindle speed is 0.			

Description	With thread tapping G63, the spindle speed must not be zero (nil).
	Example:
	`Wrong:`
	`N10 G63 Z10 F300 S0`
	
	`Correct:`
	`N10 G63 Z10 F300 S17`		

Response	Class	2	Abort NC program processing. .
Solution	Class	3	Match spindle speed, feed rate and selected pitch PROG//Section: Tapping.

Parameter	%1:	Logical axis number [-]
Error type	1, Error message from NC program.	

ID 20008

Feed rate in motion block is 0.			
Description	At least the first motion command with interpolation (e.g. G01, G02, G03) requires a feed value (F-word). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G01 X10 **Correct:** N10 G00 X0 Y0 Z0 N20 G01 X10 **F1000**		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Program the speed value using the F word.
Error type	1, Error message from NC program.		

ID 20010

Programmed function requires at least one more axis.			
Description	A previous axis release prevents this G function from executing. Example of circular interpolation: **Wrong:** N10 **#PUT AX[X]** N20 G02 R10 Y20 F200 **Correct:** Solution (do NOT return axis): … N20 G02 R10 Y20 F200 …		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Remove invalid axis release from NC program code.
Error type	1, Error message from NC program.		

ID 20011

Coordinate is out of data format.			
Description	The permissible value range for coordinates is exceeded. A movement command with calculated target coordinates outside this acceptable range was programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 X**94967596** N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and change coordinates in the NC program or check data involved in the calculation (e.g. offsets). For information always all possibly invalid values, involved in calculation are monitored.
Parameter	%1:	Logical axis number [-]	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
		Limit value for negative coordinates	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Limit value for positive coordinates.	
	%4:	Error value [0.1 µm or 0.0001°]	
		Programmed coordinate or calculated value in internal unit	
	%5:	Error value [0.1 µm or 0.0001°]	
		Further programmed coordinate or calculated value in internal unit (optional)	
Error type	1, Error message from NC program.		

ID 20012

Radius or bevel is 0 in G301/302.			
Description	At rounding or rounding of corners (G301/G302), an I-value as distance parameter is necessary. Example 1 (Chamfers): **Wrong:** N10 G00 X0 Y0 Z0 N20 G01 X10 F2000 N30 **G301** N40 G01 Y10 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G01 X10 F2000 N30 G301 **I5** N40 G01 Y10 N1000 M30 Example 2 (Rounding): **Wrong:** N10 G00 X0 Y0 Z0 N20 G01 X10 F2000 N30 **G302** N40 G01 Y10 N1000 M30 **Right:** N10 G00 X0 Y0 Z0 N20 G01 X10 F2000 N30 G302 **I5** N40 G01 Y10 N1000 M30		
Response	Class	2	.Abort NC program processing.
Solution	Class	3	Programming of missing I-value.
Error type	1, Error message from NC program. .		

ID 20013

Dwell time is programmed directly and via coordinate.

Description	A dwell time may only be programmed alone in the block without a motion command. It can be defined directly or in conjunction with the name of the first main axis. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G04 **10 X2** N1000 M30 **Correct: (Dwell time direct):** N10 G00 X0 Y0 Z0 N20 G04 **10** N1000 M30 **Correct: (Dwell time with axis information):** %dec0013A2.sol N10 G00 X0 Y0 Z0 N20 G04 **X2** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify dwell time according to G04.
Error type	11, Error message from NC program. .		

ID 20014

Dwell time is not programmed with the coordinate of first axis.			
Description	The dwell time was not defined as the coordinate of the first main axis, but as the coordinate of another axis. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G04 **Y2** N1000 M30 **Right:** N10 G00 X0 Y0 Z0 N20 G04 **X2** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	After G04, program the first main axis to define the dwell time.
Error type	1, Error message from NC program. .		

ID 20015

Dwell time is out of data format.			
Description	The dwell time exceeds the permissible data range.		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check the programmed dwell time and modify it according to the value range.
Parameter	%1:	Error value [µs]	
	%2:	Lower limit value [µs]	
		Lower limit value.	
	%3:	Upper limit value [µs]	
		Upper limit value.	
Error type	1, Error message from NC program. ..		

ID 20016

First axis gaping! Dwell time via coordinate not possible.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program. ..		

ID 20017

Negative dwell time programmed.			
Description	The dwell time exceeds the permissible data range.		
Response	Class	1	Abort NC program processing. .
Solution	Class	1	Check the programmed dwell time and modify it according to the value range.
Parameter	%1:	Error value [µs]	
Error type	1, Error message from NC program. .		

ID 20019

Diameter programming does not allow to mirror on the primary axis.			
Description	Activated diameter programming starts from rotationally symmetric parts, the programmed co-ordinates are interpreted as distance to the rotational (symmetry) axis. Therefore, mirroring and negative coordinates are invalid. Example: **Wrong:** N10 G90 G01 F1000 N20 G51 X80 N30 G92 X10 N40 X0 N50 G91 X50 N60 **G21 X30** N1000 M30 **Correct:** N10 G90 G01 F1000 N20 G51 X80 N30 G92 X10 N40 X0 N50 G91 X50 N60 **X30** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove NC mirroring command.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20020

Circle centre point parameter outside permissible data format.

Description	An interpolation parameter (I, J, K) used to specify the circle centre point is outside the permissible value range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G02 I**94967596** F1000 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the programmed interpolation parameters (I, J, K) and modify them according to the value range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Upper limit value [-]	
		Maximum permissible value.	
	%3:	Lower limit value [-]	
		Minimum permissible value.	
	%4:	Error value [-]	
		The incorrectly programmed value.	
Error type	1, Error message from NC program.		

ID 20022

Negative software limit switch is out of data format.			
Description	The programmed negative software limit switch (G98) exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G98 **X-94967596** N1000 M30		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Check and modify NC program. Program negative software limit switch within the permissible value range.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00177	
	%2:	Lower limit value [-]	
		Lower limit value	
	%3:	Upper limit value [-]	
		Upper limit value	
Error type	1, Error message from NC program.		

ID 20023

Positive software limit switch is out of data format.			
Description	The programmed positive software limit switch (G99) exceeds the permissible value range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G99 **X94967596** N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Program positive software limit switch within the permissible value range.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00178	
	%2:	Lower limit value [-]	
		Lower limit value	
	%3:	Upper limit value [-]	
		Upper limit value.	
Error type	1, Error message from NC program..		

ID 20024

Measured value cannot be considered as it was not requested for.			
Description	If there was no previous measurement run, there are no measured values available for calculations. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G101 X1 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 **N20 G100 X10 F100** N30 G101 X1 N1000 M30		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify NC program. Before the access on the measured values firstly execute the measuring sequence.
Error type	1, Error message from NC program..		

ID 20025

Negative ramp time weighting programmed.			
Description	The ramp time weighting with G132 was programmed with a negative value. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G132 **X-10** N1000 M30		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programme ramp time weighting with a positive value.
Parameter	%1:	Logical axis number	
	%2:	Error value [0.1%]	
	%3:	Limit value [0.1%]	
		Lower limit value	
Error type	1, Error message from NC program..		

ID 20028

Negative pre-control weighting programmed.			
Description	The precontrol weigthing with G136 has been programmed with a negative value. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G136 X**-10** N1000 M30		
Response	Class	1	Abort NC program processing. .
Solution	Class	3	Check and modify NC program. Programme precontrol weigthing with a positive value.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [0.1%]	
	%3:	Limit value [0.1%]	
Error type	1, Error message from NC program.		

ID 20029

Invalid gear stage.			
Description	The number of available gear stages per axis, and thus the allowed value range, depends on system configuration. The counting of the stages starts at 1, so that values smaller than 1 are invalid. Also values starting from 65536 and above are rejected as well. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G112 **X0** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a correct value for the gear stage.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
		Lower limit value	
	%3:	Logical axis number [-]	
Error type	1, 1, Error message from NC program..		

ID 20030

Gear stage is out of data format.

Description	The number of available gear stages per axis, and thus the allowed value range, depends on system configuration. The counting of the stages starts at 1, so that values smaller than 1 are invalid. Also values starting from 65536 and above are rejected as well. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G112 **X65536** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a correct value for the gear stage.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
	%4:	Logical axis number [-]	
Error type	1, Error message from NC program..		

ID 20032

Slave axis is programmed, but moving distance is 0.			
Description	The calculated moving distance for a programmed slave axis for tapping is zero (nil), so the tapping can not be executed. Example: **Wrong:** (Current Position: 0) N10 G63 **C0** F100 S100 **Correct:** (Current Position: 0) N10 G63 C123 F100 S100		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a correct depth for tapping.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis.	
Error type	1, Error message from NC program..		

ID 20033

Circle with programmed radius impossible.

Description	Circle construction is geometrically impossible with programmed radius. The radius is too small. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G02 X10 **R1** F1000 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G02 X10 **R7** F1000 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Adjust the radius, i.e. increase it so that the circle passes through the programmed end point.

Parameter	%1:	Error value [0.1 µm or 0.0001°]
		Invalid radius.
	%2:	Current value [0.1 µm or 0.0001°]
		Start point of 1st main axis
	%3:	Current value [0.1 µm or 0.0001°]
		Start point of 2nd main axis
	%4:	Current value [0.1 µm or 0.0001°]
		End point of 1st main axis
	%5:	Current value [0.1 µm or 0.0001°]
		End point of 2nd main axis
Error type	1, Error message from NC program.	

ID 20034

Programming of a full circle with radius specification not possible.			
Description	Specifying only the radius is insufficient data for a full circle, since this specification would allow an indefinite number of circles – the circle centre could lie on a full circle with radius R around the start/end point. Therefore, it is necessary to program full circles via the centre point instead of using the radius. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G02 X0 Y0 **R10** F1000 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G02 **I10** F1000 N1000 M30		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Always program full circles via the centre point definition.
Error type	1, Error message from NC program.		

ID 20035

Difference between programmed and calculated centre point too big.

Description	With active centre point correction (G165) based on the programmed centre point coordinates I, J, K, a centre point is calculated. This error message is generated if this calculated centre point deviates too much from the programmed centre point (P-CHAN-00059). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G02 X10 **I7** F1000 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G02 X10 **I5** F1000 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Adjust centre coordinates. Check in which measurement system (absolute/relative) the centre point is programmed (G161/G162). Possibly complete missing centre point coordinates.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
	%2:	Limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 20036

Radius of circle is 0.			
Description	During inactive centre point correction (G164 the centre point co-ordinates I, J, K are zero (nil) or still not programmed or the programmed co-ordinates of the circle end point are identical to the co-ordinates of the start point. From the calculation of the centre point results a circle radius with the value zero (nil) and this error message is displayed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G164 G02 X10 F1000 (missing centre point coordinates) N1000 M30 **Wrong:** N10 G00 X0 Y0 Z0 N20 G164 G02 X0 F1000 (circle end point = start point) N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G164 G02 **X10 I5** F1000 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correction of the invalid centre point co-ordinates I,J,K or circle end points.
Error type	1, Error message from NC program.		

ID 20037

#ACHSE is programmed without G200/G201/G202.

Description	The NC command #ACHSE[] was programmed but cannot be evaluated because G200/G201/G202 were not specified as well. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #ACHSE[X] N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G201** #ACHSE[X] N1000 M30		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify NC program. Add G200 / G201 / G202.
Error type	1, Error message from NC program.		

ID 20038

Programming G201 the definition of axes is expected.			
Description	With G21 (manual mode with parallel interpolation), at least one axis to be activated **must** be specified. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G201 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G201 **X1** ;or **#ACHSE[X]** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Complete G201 with an axis statement.
Error type	1, Error message from NC program..		

ID 20041

Motion information is required for MNE_SNS function.			
Description	M functions of type MNE_SNS are triggered by an external event, and interrupt the current motion block accordingly. Therefore, they are not meaningful without a programmed motion information at the same time. Example: **Wrong:** **M33** M30 **Correct:** G01 **X150 Y200 M33** F8 M30		
Response	Class	2	Abort of the NC program processing.
Solution	Class	3	Check and modify NC program. Add a motion information.
Error type	1, Error message from NC program. .		

ID 20042

No face turning axis available when selecting or deselecting diameter programming.			
Description	When selecting diameter programming, there is exactly one face turning axis required within the working face selected by G17, G18 or G19. Appearance of this error message as a result of G51 therefore indicates a configuration problem within the axis parameters list.		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Change the configuration. Define a facing axis P-AXIS-00015
Error type	1, Error message from NC program.		

ID 20044

Parameter in #CONTOUR MODE command is out of data format or out of range of permissible values.		
Description	The value of a keyword programmed in the command #CONTOUR MODE is outside the permissible value range.	
Response	Class	1 Continue NC program processing. .
Solution	Class	1 Invalid value is corrected and NC program processing is continued.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]
	%2:	Lower limit value [0.1 10^-3 mm or ø]
	%3:	Corrected value [0.1 10^-3 mm or ø]
	%4:	Current value [-]
		Keyword belonging to the incorrect value
Error type	1, Error message from NC program.	

ID 20045

SLOPE parameter is out of data format.			
Description	One of the programmed slope parameters exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #SET SLOPE PROFIL[**2222222222**, 0, 0] N1000 M30		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify NC program. Change the value of the invalid parameter.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20048

Centre point coordinate within linear motion block ignored.			
Description	The centre point coordinate (I,J,K) of the linear block (G01) is ignored. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G01 **I10** X10 F1000 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G01 X10 F1000 N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Remove the Centre point co-ordinates (I,J,K).
Error type	1, Error message from NC program.		

ID 20049

Measurement run is not permitted with active tool radius compensation.			
Description	As long as the TRC function (tool radius compensation) is enabled, no measurement run (G100) can be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G41** X100 F1000 (TRC activated) N30 **G100** X0 (attempts to start measurement run) N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Deselect before selecting measurement run (G40).
Error type	1, Error message from NC program.		

ID 20050

Old measured value is still considered.

Description	During a measurement run, the offset from the last measurement run is still improperly enabled. Important: Please be aware, that before any measuring movement deactivating the existing offset using G102 is as well mandatory as making sure, that the measurement probe is reached during the movement! Example: **Wrong:** N5 X0 N10 G100 X20 F1000 N20 G101 X1 N30 **G100** X10 N40 G101 X1 M30 **Correct:** N5 X0 N10 G100 X20 F1000 N20 G101 X1 N22 G1 X0.4711 F1000 N25 **G102** X1 N30 **G100** X10 F1000 N40 G101 X1 M30		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify NC program. Before selection of measuring cycle deselection of old measuring offset via G102.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20051

Homing for axes in manual operation mode invalid.			
Description	An axis with activated manual operation mode cannot perform homing. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G201 #ACHSE [X]** N30 X100 N40 **G74 X1** N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G201 # AXIS [X] N30 X100 N35 **G202** N40 G74 X1 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program sequence. Before G74 (homing), G202 must be programmed (deselect manual operation mode).
Parameter	%1:	Logical axis number [-]	
		Axis is still in manual operation mode.	
Error type	1, Error message from NC program.		

ID 20052

NC command requires programming of axes.

Description	At least one axis must be programmed in the same NC block in connection with an NC command . Example: **Wrong:** N10 G00 X0 Y0 Z0 N30 **G100** N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N30 **G100 X10** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Add the corresponding axis data to the NC command.
Error type	1, Error message from NC program. .		

ID 20054

Corrected centre point is out of data format.			
Description	The values chosen for start / end point and radius imply centre coordinates that are outside the permissible numerical range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G02 X1 R1000000 F1000 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G02 X1 R0.5 F1000 N1000 M30		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Check and modify NC program. Change the value of the radius.
Parameter	%1:	Logical axis number [-]	
		Axis with exceeded data range	
	%2:	Upper limit value [-]	
	%3:	Lower limit value [-]	
	%4:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20055

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20057

Tool compensation is out of data format.			
Description	The permissible data range is exceeded when calculating tool-related compensations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change tool compensation data. An axis may have a tool compensation data configured that is too large, or the tool change may be carried out at a position that is at the limit of the numerical range. In this case move to another position to consider the tool compensation.
Parameter	%1:	Logical axis number [-]	
	%2:	]Error value [0.1 µm or 0.0001°]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20063

Considering the tool radius the data format is violated.			
Description	When calculating the tool radius, the permissible data range is exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change tool compensation data. Adjust defined tool radius P-TOOL-00004.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20064

Face turning axis is used twice in processing plane.			
Description	Only one axis, the first or the second main axis, can be used as face turning axis. In this case, however, both axes have the property of a facing axis (P-AXIS-00015).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axes parameter settings. Only one of the main axes may be facing axis P-AXIS-00015.
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 20065

Considering the tool offsets the data format is violated.			
Description	When calculating the tool based corrections; the permissible data range is exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change tool compensation data. Adjust defined tool offsets P-TOOL-00006.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20066

When calculating the zero point offsets, the data format is violated.			
Description	When calculating the zero point offsets, the permissible data range is exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change zero offsets. Adjust defined zero shift parameters P-ZERO-00003.
Parameter	%1:	Logical axis number [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20068 - 20073

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20075

When calculating the measured value, the data format is violated.			
Description	When calculating the measurement offset based on the measured value, the permissible data range is exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Possibly check the latched measured values in the PLC.
Parameter	%1:	Logical axis number [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	

Error type	1, Error message from NC program.

ID 20076

Considering the command value the data format is violated.			
Description	When calculating the setpoint, the permissible data range is exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check plausibility of current setpoints.
Parameter	%1:	Logical axis number [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20077

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20078

Clamp position index is out of range of permissible values.			
Description	The clamp position offset index entered in the program job sequence has an invalid value.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check program assignment. Enter correct clamp position offset index P-CLMP-00001.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	-		

ID 20079

Coordinate initialization is out of data format.			
Description	At program start or RESET during initialization of axes coordinates, the permissible data range is exceeded.		
Response	Class	2	Abort NC program processing.

Solution	Class	6	Repeat program start or RESET.
Parameter	%1:	Current value [-]	
		Current axis position	
	%2:	Current value [-]	
		Programmed axis position	
	%3:	Logical axis number [-]	
	%4:	Lower limit value [-]	
	%5:	Upper limit value [-]	
Error type	-		

ID 20083

Write access to tool radius is not permitted with D word in the same NC block.			
Description	A D word in the block has read access to various tool parameters such as tool length, tool radius, etc. Therefore, the tool radius cannot be written in the same block. This also applies to access using V.G. variables. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 D1 **V.G.WZ_AKT.R=3** N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **D1** N30 **V.G.WZ_AKT.R=3** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Program D word in the following NC block after write access.
Error type	1, Error message from NC program.		

ID 20084

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20085

Too many external variables were declared.	
Description	In the list for external variables [EXTV] the number of external variables declared (global and channel-specific) was too high, or the number of existing variables in the parameter P-EXTV-00010 is wrong. The maximum permissible limit of external variables can be found in [SYSP].

Example:

```
anzahl_belegt               500
#
var[0].name                     VARIABLE_1
var[0].type                     SGN32
#
...
#
var[100].name               VARIABLE_100
var[100].type                    UNS32
```

Possible solutions:

Check if the parameter P-EXTV-00010 contains the correct number of variable declarations

If there are in fact too many variables, you can try to combine variables to variable arrays (same data type) or variable structures (different variable types):

Variable array:

```
anzahl_belegt               1
#
var[0].name                     VAR_ARRAY_100
var[0].type                     UNS32
var[0].array_elements 100
```

Variable structure:

```
anzahl_belegt 2
#
struct[0].name STRUCT_DEF
struct[0].element[0].name VARIABLE_1
struct[0].element[0].type SGN32
struct[0].element[1].name VARIABLE_2
struct[0].element[1].type UNS32
#
var[0].name                     VAR_STRUCT
var[0].type                     STRUCT_DEF
#
var[1].name                     VAR_STRUCT_ARRAY
var[1].type                     STRUCT_DEF
var[1].array_elements 50
```

Response	Class	2	Not all configured variables are generated.
Solution	Class	3	Check whether the value in the parameter P-EXTV-00010 is correct. Merge the external variable declarations in the list [EXTV].
Parameter	%1:	Error value [-]	
		P-EXTV-00010: Number of configured external variables	
	%2:	Upper limit value [-]	
		Maximum number of configured external variables	
Error type	1, Error message from NC program.		

ID 20087

Maximum number of external variables in one NC block exceeded.			
Description	External variables can be assigned values or be read in the NC program synchronously with machining. The maximum permissible number per NC block is limited. If this maximum permissible number is exceeded, this error message is issued. The characteristic "Synchronous read/write access" is defined in the list for external variables [EXTV] via the element "var[...].synchronisation".		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Distribute the synchronous read/write access to the V.E. variables over several NC blocks. **Maximum number of external variables in one NC block exceeded.** Non-synchronous read/write access to V.E. variables per NC block is unlimited!

Parameter	%1:	Limit value [-]
		Maximum permissible number of synchronised ext. variables per NC block
Error type	1, Error message from NC program.	

ID 20088

Data type of external variable unknown.			
Description	During start-up it is detected, that the data type of one of the external variables is unknown. The data type is defined in the list of external variables [EXTV] via the element "var[...].type".		
Response	Class	2	Start-up of the control is continued. The external variables which were read in after the invalid data type are not stored inside the control, so they are not available after start-up.
Solution	Class	3	Change the invalid data type in the list of external variables [EXTV] and repeat start-up.
Parameter	%1:	Current value [-]	
		Index of the external variable with invalid data type	
	%2:	Current value [-]	
	%3:	Current value [-]	
Error type	-		

ID 20092

Variable access on an unknown axis.			
Description	Attempting to access axes that do not exist in the channel with axis-specific variables (V.A.) generates this error message. Occurs especially after returning an axis to axis management. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **#PUT AX[X]** N30 P1=**V.A.PROG.X** (Error: no X axis anymore in the channel) N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=**V.A.PROG.X** N30 **#PUT AX[X]** N1000 M30 **or** N10 G00 X0 Y0 Z0 N20 #PUT AX[X] N25 $IF **EXIST[V.A.LOG_AX_NR.X]** N30 P1=V.A.PROG.X N35 $ENDIF N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and adjust NC program sequence. Program access to variables before issuing the axis or check first with the command EXIST[V.A.LOG_AX_NR.xx] [PROG// Section: Arithmetic expressions] whether the axis is available in the channel.
Error type	1, Error message from NC program.		

ID 20095 / 20096

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20097

Read access to variable denied.			
Description	The attempt to read from CNC an external variable which has only access rights for a write access, causes this error. The write/read access rights are defined in the list of external variables [EXTV] via the element "var[...].access_rights".		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Adjust the corresponding variable for the access rights in the list of external variables [EXTV] and repeat start-up.
Error type	1, Error message from NC program.		

ID 20098

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20100

Write access to variable denied.

Description	A write access is made to a variable of type 'READ_ONLY' for the CNC. It may either be a system-defined variable or an external variable (V.E.) which might be declared as ‘READ_ONLY’, ‘WRITE_ONLY’ and ‘READ_WRITE’ in the list definition [EXTV].
	Example 1:
	`%example1`
	`N10 G00 X0 Y0 Z0`
	`N20 V.A.MENT.X=100`
	`N1000 M30`
	Example 2:
	`%example2`
	`N10 G00 X0 Y0 Z0`
	`N20 V.A.MODE[4]=0`
	`N1000 M30`
	Example 3:
	`%example3`
	`N10 G00 X0 Y0 Z0`
	`N20 V.A.MODULO_VALUE[4]=0`
	`N1000 M30`
	Example 4:
	(Requires the following variable definition)
	`var[0].name MYREADONLY`
	`var[0].index 8`
	`var[0].type SGN32`
	`var[00].scope CHANNEL`
	`var[0].synchronisation TRUE`
	`var[0].access_rights READ_ONLY`
	`var[0].array_size 0`
	`var[0].size 4`
	`var[0].create_hmi_interface 0`
	`%example4`
	`N10 G00 X0 Y0 Z0`
	`N20 V.P.DEMO = V.E.MYREADONLY`
	`N30 V.E.MYREADONLY = 4711`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	In the list of external variables [EXTV], adjust the access rights for the corresponding variable using P-EXTV-00006 and repeat the start-up if this error occurs with an external variable. For all other variable types, only a read access may be programmed.
Error type	1, Error message from NC program.		

ID 20101

Too many variables programmed in NC block for report of changes.			
Description	This error message is not in present use.		
Response	Class	1	Warning
Solution	Class	1	For your attention
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20103

Too many variables programmed in NC block for report of changes.			
Description	This error message is not in present use.		
Response	Class	1	Warning
Solution	Class	1	For your attention
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20104

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20105

Double-programmed block number N.			
Description	In the same NC block more than one block number with the N word was programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 **N20** X10 **N30** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the unnecessary NC block number.
Error type	1, Error message from NC program.		

ID 20106

NC block number is out of data format.			
Description	The block number programmed with the N word is outside the permissible limits of the data format.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a permissible value for the invalid block number.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20107

Number of G function is out of valid data format.			
Description	The numerical value programmed with the G word is outside the permissible limits of the data format. Remark: If an unassigned G function is programmed, error message 20131 is output: Unknown G function. [▶ 201]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the incorrect G function with a value that addresses a permissible G function.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20108

Double-programmed block mode.			
Description	In the same NC block, more than one G-function from the group of moving conditions (G00, G01, G02, G03 etc.) was programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G01** X10 **G00** Y20 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20109

Double-programmed prescription of feedrate.			
Description	More than one G function from the group of acceleration conditions (G08/G193/ G293) has been programmed in the same NC block. Example: **Wrong:** N10 G01 X500 F1000 N20 **G193** X900 F400 **G293** N30 X1000 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20110

Double-programmed adaptation of the feed rate.			
Description	In the same NC block, more than one G function has been programmed from the feedrate adaptation group (G10/G11/G92 R). Example: **Wrong:** N10 G00 X0 Y0 Z0 G42 N20 G01 **G11** X100 **G10** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20111

Double-programmed plane.			
Description	More than one G function from the plane selection group (G17/G18/G19) has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G18** Y20 Z4 **G19** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20112

Double-programmed mirror function.			
Description	More than one G function from the group of mirroring (G20/G21/G22/G23/G351) has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G20** X10 Y20 **G21** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20113

Double programmed transition block mode of tool radius compensation.			
Description	More than one G function from the group of TRC transition types (G25/G26) was programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 G42 N20 **G25 G25** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the invalid G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20114

Double programmed selection of tool radius compensation.			
Description	In the same NC block, more than one G function was programmed from the group of TRC commands for selecting and deselecting (G40/G41/G42). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G41** X10 Y20 **G42** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20115

Double-programmed diameter selection.			
Description	More than one G function from the diameter programming group (G51/G52) has been programmed in the same NC block. Example: **Wrong:** N10 G90 G01 F1000 N20 **G51** X80 **G51** : M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20116

Double-programmed zero point offsets.			
Description	More than one G function from the group of zero offsets (G53-G59/G159) has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G53** X10 Y10 **G54** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20117

Double programmed block transition.			
Description	More than one G function from the group of block transition definitions (G60/G359/ G360/G61/G260/G261) was programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G360** X100 **G60** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20118

Double programmed unit declarations.			
Description	In the same NC block, more than one G function was programmed from the group of units (G70/G71). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G70** X100 **G71** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20119

Double-programmed working cycles.			
Description	Error message is not in use.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20120

Double programmed dimension declarations.			
Description	More than one G function from the dimension data group (G90/G91) was programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G90** X100 **G91** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20121

Double-programmed feed rate declarations.			
Description	In the same NC block, more than one G function has been programmed from the group of feed agreements (G93/G94/G95/G194). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G93** F500 X100 **G94** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20122

Double-programmed spindle speed definitions.

Description	More than one G function from the group of spindle speed agreements (G96/G97/G196) has been programmed in the same NC block. Example: **Wrong:** N10 M03 S1000 G01 F1000 X100 N20 **G96** S63 **G97** : M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20123

Double-programmed measuring functions.			
Description	More than one G function from the group of measuring functions (G100-G108) has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G100** X100 **G102** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20124

Double programmed path preparation commands.			
Description	In the same NC block, more than one G function was programmed from the look-ahead function group (G115/G116/G117). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G115**=2 **G116** X1 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20125

Double-programmed feedforward control function.			
Description	In the same NC block, more than one G function was programmed from the group of feedforward control functions (G135/G136/G137). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G135** X100 **G137** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the invalid G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20126

Double programmed selection mode of tool radius compensation.			
Description	In the same NC block, more than one G function was programmed from the group of TRC selection modes. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G138** G41 **G139** : N1000 M30 Exceptions: Programming G05 in combination with another TRC selection type is permissible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the second incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 20127

Double-programmed centre point selection.			
Description	In the same NC block, more than one G function from the group of centre point selection types (G161/G162) has been programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G02 **G162** I10 X20 **G162** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.

Parameter	%1:	Error value [-]
		Number of inadmissible programmed second G function
Error type	1, Error message from NC program.	

ID 20128

Double-programmed centre point correction.			
Description	In the same NC block, more than one G function has been programmed from the group of centre point offset commands (G164/G165). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G165** G02 I10 X20.05 **G164** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20129

Double-programmed manual mode selection/deselection.			
Description	In the same NC block, more than one G function has been programmed from the group of manual operation selection/deselection commands (G200/G201/G202). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G201** #ACHSE[X] **G200** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20130

Double-programmed chamfer/phase selection.	
Description	More than one G function from the group of chamfers and radius commands (G301/G302) has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G01 X20 F1000 N30 **G301** I10 **G302** N40 Y50 : N1000 M30

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 20131

Unknown G function.			
Description	A non-existing G function was programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G88** X100 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the unknown G function.
Parameter	%1:	Error value [-]	
		Number of the unknown programmed G function	
Error type	1, Error message from NC program.		

ID 20133

Plane selection not allowed during active tool radius compensation.			
Description	As long as the tool radius compensation is enabled, it is not permitted to program a plane selection via G17, G18 or G19. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G41** N30 **G19** N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G41 : N25 **G40** N30 **G19** N35 G41 : N999 G40 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program sequence. Deselect the TRC with G40 before programming plane selection.
Parameter	%1:	Error value [-]	
		Invalid G function	
Error type	1, Error message from NC program.		

ID 20136

Selection of tool radius compensation not possible with missing main axis in active plane.

Description	The tool radius compensation requires the first 2 main axes in the current plane. TRC is not possible, if one of this main axes is not available in the NC channel. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **#PUT AX[X]** N30 **G41** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure that both main axes of the current plane are available in the NC channel when TRC is selected.
Error type	1, Error message from NC program.		

ID 20137

G74 not allowed during active synchronous operation.			
Description	It is not possible to execute homing (G74) while an axis coupling group is enabled. Also axes not involved in the current couplings must not be referenced. Example: **Wrong:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=Y] N40 **#ENABLE AX LINK[1]** N50 **G74** X1 N1000 M30 **Correct:** (temporarily disable coupling): N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=Y] N40 #ENABLE AX LINK[1] : N45 **#DISABLE AX LINK[1]** N50 **G74** X1 N55 #ENABLE AX LINK[1] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program sequence. Deselect all active coupling groups before homing.
Parameter	%1:	Error value [-]	
		Invalid G function	
Error type	1, Error message from NC program.		

ID 20138

G100 is not allowed with current measuring type.			
Description	A measurement with G100 cannot be executed because the current valid measuring type is not permissible for a G100 measurement run.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Make sure that a permissible measuring type P-CHAN-00057 is entered in the channel parameter list, or first switch to a permissible measuring type in the NC program before G100 with #MEAS MODE [...] [PROG// Section: Settings for measurement].
Error type	1, Error message from NC program.		

ID 20147

After this G-function a '=' is expected.			
Description	The called G-function expects an argument, assigned by “=”. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G115** N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G115=14** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the missing assignment.
Parameter	%1:	Current value [-]	
		Number of the G function	
Error type	1, Error message from NC program.		

ID 20149

Value following G159 is out of range of permissible values.			
Description	The numerical value programmed with G159=<expr> represents an index (data record) in the zero offset table. The size of this table depends on the respective application. Here the programmed index exceeds the maximum permissible index. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G159=500** N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G159=7** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the index present in the zero offset table P-ZERO-00003.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20151

Unknown path preparation mode at G115.

Description	The value specified with the look-ahead function G115=<*value*> is out of the permissible value range. Integers from 0 to 14 are permissible for <*value*> [PROG// Section: Influence on the look-ahead functionality. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G115=20** N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G115=14** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program correct path preparation mode.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20154

Value following 'M' is out of data format.			
Description	The programmed number of the M function exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **M150000** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of M function must be within the permissible value range. Make sure that the M function is configured in the channel parameters P-CHAN-00041.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
	%3:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20155

Double programmed M function for spindles.			
Description	A spindle M function (M3/M4/M5/M19) in the NC block is programmed multiple times in spindle-specific syntax. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 X10 S[**M3** S1000 **M4**] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the programming of the M functions in the NC block. Remove surplus spindle-specific M function.

Parameter	%1:	Error value [-]
		Number of the double programmed spindle M function
Error type	1, Error message from NC program.	

ID 20156

Two M-functions with synchronisation mode MNE_SNS programmed.			
Description	Two M functions with the synchronization type MNE_SNS cannot be programmed in the same NC block. For further information see [[CHAN// Synchronisation method MNE_SNS] and [FCT-C1// Example with MNE_SNS].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Distribute the M-function on two NC-blocks.
Parameter	%1:	Current value [-]	
		Number of the second M function with synchronisation mode MNE_SNS.	
Error type	1, Error message from NC program.		

ID 20157

Unknown M-function, since not defined in the channel parameters.			
Description	The programmed M function is not configured in the channel parameter list P-CHAN-00041 and therefore not known in the NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Define M function in the channel parameter list P-CHAN-00041.
Parameter	%1:	Error value [-]	
		Number of the unknown M function	
Error type	1, Error message from NC program.		

ID 20158

Double-programmed M02/M30 or M17/M29.

Description	Within the same NC block, the subroutine-terminating M functions M2, M17, M29 or M30 are programmed several times. Remark: Depending on the sequence of programming of M2, M17, M29 M30 in this context, the error message 20376 "Unexpected M17 or M29" might occur. Example 1: **Wrong:** %UP1 N10 X10 Y10 Z10 N20 **M17 M30** %HP N10 G00 X0 Y0 Z0 N20 LL UP1 N30 M30 **Correct:** %L UP1 N10 X10 Y10 Z10 N20 **M17** %HP1 N10 G00 X0 Y0 Z0 N20 LL UP1 N30 **M30**		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove surplus M function.
Parameter	%1:	Current value [-]	
		Number of the multiple programmed M function	
Error type	1, Error message from NC program.		

ID 20160

Unknown internal M-function.			
Description	The programmed M function is above the permissible limit for self-defined M functions due to its number (P-CHAN-00041) and is therefore initially treated as a so-called internal M function (e.g. M30). However, since it is also not known as an internal M function, the present error is output. Note: Internal M functions are M00, M01, M02, M10, M11, M17, M29, M30 Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **M1001** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure that the M function is defined in the channel parameters P-CHAN-00041 and that the number of the M function is within the permissible value range. At the moment numbers higher than 1000 are not allowed.
Parameter	%1:	Error value [-]	
		Number of the unknown M function	
Error type	-		

ID 20161

Value following 'H' is out of data format.			
Description	The programmed number of the H function exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **H150000** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of H function must be within the permissible value range. Make sure that the H function is configured in the channel parameters P-CHAN-00027.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
	%3:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20162

Unknown H-function, since not defined in the channel parameters.			
Description	The programmed H function is not configured in the channel parameter list P-CHAN-00027 or exceeds a maximum value and is therefore not known in the NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Define H function in the channel parameter list P-CHAN-00027.
Parameter	%1:	Error value [-]	
		Number of the unknown H function	
Error type	1, Error message from NC program.		

ID 20163

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20165

Double-programmed T-function.

Description	In the same NC block, more than one T word was programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **T1 T2** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove invalid T word.
Error type	1, Error message from NC program.		

ID 20166

Value following 'T' is out of data format.			
Description	The programmed number of the T function exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **T2147483648** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of T function must be within the permissible value range.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20167

Unknown T-function.			
Description	The programmed T function addresses a tool location that does not exist in the list of tool parameters. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **T999** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of T function must dress a permissible tool place.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20168

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20169

Tool change not allowed during active synchronous operation.			
Description	As long as an axis coupling group is enabled (#ENABLE AX LINK), it is not possible to enable a different set of parameters for tool geometry compensation with the D word or #TOOL DATA. Example: **Wrong:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=C] N40 **#ENABLE AX LINK[1]** : N100 **D2** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=C] N40 #ENABLE AX LINK[1] : N90 **#DISABLE AX LINK[1]** N100 **D2** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program sequence. Synchronous operation with #DISABLE AX LINK must be deselected before the D word or #TOOL DATA.
Error type	1, Error message from NC program.		

ID 20170

Double-programmed D-function.

Description	More than one D word has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **D1 D2** : N1000 M30 When the channel parameter P-CHAN-00014 is active, this error occurs when the following line occurs in NC program: N010 T1 D1 The cause is that, with the activated channel parameter P-CHAN-00014, the programming of “T1” corresponds to the programming of “T1 D1”.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove invalid D word. When programming T Word and D word within a NC block, check and modify the channel parameter P-CHAN-00014.
Error type	1, Error message from NC program.		

ID 20171

Value following 'D' is out of data format.			
Description	The programmed number of the D word is outside the permissible value range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **D2**32** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of D word must be within the permissible value range.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20172

Unknown D-function.

Description	The programmed D word addresses a tool compensation data record which does not exist in the tool parameter list. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **D999** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of the D word must address a permissible tool compensation data record.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20175

Double-programmed F-word.			
Description	More than one F word has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **F100** G01 X100 **F200** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove invalid F word.
Error type	1, Error message from NC program.		

ID 20176

Programmed feedrate is negative or 0.			
Description	The feed rate programmed with the F word is negative or zero (nil). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **F[-1000]** G01 X100 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **F1000** G01 X100 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the feedrate (F word) with a meaningful value greater than zero (nil).
Parameter	%1:	Error value [-]	

Error type	1, Error message from NC program.

ID 20177

Programming an e-feedrate is not permissible.			
Description	The programming of speeds using an E word is not implemented and therefore generally not permitted. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G01 X100 **E2000** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G01 X100 **F2000** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the feedrate with the F word.
Error type	1, Error message from NC program.		

ID 20178

Programmed E feed is not allowed.			
Description	The E feedrate was programed with 0 or a negative value.		
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
		The incorrect E feedrate.	
Error type	1, Error message from NC program.		

ID 20180

Unknown spindle name or equal sign is missing.

Description	Either the spindle name programmed in NC block is unknown in NC channel (P-CHAN-00007, P-CHAN-00053) or, if the spindle name has more than one character, the equals sign is missing before the speed value . Example (assuming: Spindle name is „SPINDLE"): **Wrong:** N10 G00 X0 Y0 Z0 N20 **SPDL=1000** M03 : N1000 M30 **or** N10 G00 X0 Y0 Z0 N20 **SPINDLE1000** M03 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **SPINDLE=1000** M03 : N1000 M30		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify NC program. Correct spindle name according to the configured name P-CHAN-00007 **. ****Hint:** Only the main spindle name P-CHAN-00053 may be used for the main spindle in the NC program.
Error type	1, Error message from NC program.		

ID 20181

Double-programmed spindle.			
Description	In NC block the same spindle was programmed several times in DIN syntax. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **S1000** M03 X100 G01 F2000 **S1500** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible spindle programming.
Error type	1, Error message from NC program.		

ID 20182

No assignment after name of parameter.

Description	There was a parameter declaration programmed, but the assignment of a value is missing. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **P2** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **P2=47.11** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the parameter declaration.
Error type	1, Error message from NC program.		

ID 20183

Parameter index is out of data format or out of range of permissible values.			
Description	The programmed parameter index exceeds the permissible value range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **P0**=10 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **P1=10** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed parameter index must be within the permissible value range.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20186

Syntax error after #GET CMDPOS or #SET IPO SOLLLPOS.

Description	The syntax of the NC command #GET CMDPOS[...] or #SET IPO SOLLPOS[...] is violated after the opening bracket (comma, equals sign, closing bracket). Syntax example: **Wrong:** #GET CMDPOS [P1=X P2=Y] or #GET CMDPOS [V.L.POS1 X] or #GET CMDPOS [P1=X, P2=Y, P3=Z **Correct:** #GET CMDPOS [P1=X**,** P2=Y]or #GET CMDPOS [V.L.POS1**=**X] or #GET CMDPOS [P1=X, P2=Y, P3=Z**]**		
Response	Class	3	Abort NC program processing.
Solution	Class	3	In the NC program, check and correct the command syntax of #GET CMDPOS[...] or #SET IPO SOLLPOS[...] regarding comma setting, equal sign, closing square brackets.
Error type	1, Error message from NC program.		

ID 20187

Radius index is out of data format.			
Description	The value used as radius**index** is outside the permissible data range. Note: For indexed radius programming, the only permissible **index is 1**, otherwise error message 20188 [▶ 216] "Illegal radius index" is output. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **R-1**=10 N50 G02 F1000 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **R1**=10 N50 G02 F1000 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed radius index must be within the permissible value range.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20188

Illegal radius index.			
Description	For indexed radius programming, the only allowed index is 1; all other values cause either this error message 20188 or 20187 [▶ 215] "Radius index is outside allowed number range". Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **R2**=10 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **R1**=10 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Radius index must be programmed with value 1.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20191

Value following 'O' is out of data format.			
Description	Programming the O function is not implemented and therefore generally not allowed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the O function.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20192

Unknown O-function.			
Description	Programming O function is not implemented and therefore generally not allowed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the O function.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20193

Percent sign not allowed within main routine part.			
Description	Within a main routine a percent character (%) is programmed, although the main routine is already in execution. A percentage sign can only be used to mark main or local subroutine names, but not within the actual main program itself. Further information: [PROG//Section Subroutine techniques]. Example: **Wrong:** %MAIN N10 G00 X0 Y0 Z0 N20 **%L SUB1** : N1000 M30 **Correct:** **%L SUB1** : %MAIN N10 G00 X0 Y0 Z0 : N1000 M30 The execution of a main program is also started, if the first character found in the file outside comments is neither a space nor a "%". In this case, this character is evaluated as the first character of an unnamed program. It also means that no block numbers, variable declarations etc. may be programmed before the first “%”.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program sequence. Remove the invalid percent sign or move the definitions of the local sub programs before the beginning of the main program part. Ensure that no NC block numbers, variable declarations etc. were programmed before the first %-character (e.g. remove block numbers at the beginning of comment lines!).
Error type	1, Error message from NC program.		

ID 20194

Assignment operation missing or unknown.			
Description	In the NC program, an assignment is expected for a variable (V.A., V.G., V.P., V.L., V.S., V.E.). Either here the programmed assignment operator is unknown or the complete assignment of a value is missing. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **V.G.T_AKT** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **V.G.T_AKT =5** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the invalid or missing assignment.
Error type	1, Error message from NC program.		

ID 20195

Overflow of variable stack.			
Description	The programmed variable operation causes an overflow of an internal system resource, e.g. when using too many indirection operations (nesting levels). Example: **Wrong:** N10 #VAR N20 V.L.INDEX[10] = [1, 2, 3, 4, 5, 6, 7, 8, 9, 10] N30 **V.P.MY_ARRAY[V.L.INDEX[V.L.INDEX[..[..[..[3]]]]]]** N40 #ENDVAR N1000 M30 **Correct:** N10 #VAR N20 V.L.INDEX[10] = [1, 2, 3, 4, 5, 6, 7, 8, 9, 10] N30 **V.P.MY_ARRAY[3]** N40 #ENDVAR N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the dimension of the array variable or nesting depth or generally avoid the nesting of array variables.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20196

Unknown variable.			
Description	When linking V.G.NP[...].ALL variables, an invalid operation is programmed on the right-hand side of an assignment. A linkage with axis-specific constants or variables is not permitted. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 V.G.NP[1].ALL = V.G.NP[2].ALL **+ 100** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 V.G.NP[1].ALL = V.G.NP[2].ALL N30 **V.G.NP[1].V.X += 100** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Apply linking rules correctly to ".ALL" variables, [PROG//Section: Adding and subtracting offsets].
Error type	1, Error message from NC program.		

ID 20197

Index of variable is out of data format.

Description	The programmed index of a variable exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=**V.G.WZ[-1].L** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=**V.G.WZ[65].L** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed variable index must be within the permissible value range.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20198

Missing ‘]’ after variable.			
Description	The closing square bracket ‘]’ is missing in a variable expression with index. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=V.G.WZ**[V.G.WZ[1].L** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=V.G.WZ**[V.G.WZ[1].L]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the variable expression for the missing closing square bracket ’]’.
Error type	1, Error message from NC program.		

ID 20200

NC-command or name of axis expected.

Description	An axis unknown in the channel is programmed in the NC block. The programmed axis name cannot be assigned to an axis. This applies to the following cases: • slave axes of gantry or axis couplings • axes that are temporarily not available in the channel because of axis exchange commands • axes or axis names that are not configured in the NC channel • axis names which do not start with the permitted axis letters X, Y, Z, A, B, C, U, V, W or Q. • axes that are tracked at this time with #CAXTRACK. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK [1, W=X] N30 #ENABLE AX LINK [1] N40 X10 **W**20 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK [1, W=X] N30 #ENABLE AX LINK [1] N40 X10 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only use axes names, which are available in NC channel.
Error type	1, Error message from NC program.		

ID 20203

Double-programmed axis.			
Description	In the same NC block, an axis was programmed several times. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **X10** Y20 **X10** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant axis programming.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20205

Double-programmed centre point coordinate.

Description	In the same NC block, a centre point coordinate was programmed several times. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G02 **I10 I20** J30 F2000 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant centre point coordinate.
Error type	1, Error message from NC program.		

ID 20206

$ command must be programmed exclusively in NC block.			
Description	A control block sequence starting with the dollar character $ is programmed with other NC commands in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **$FOR** P10=0, 20, 1 **G01 X10 F100** N30 G91 X5 N40 $ENDFOR : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N15 **G01 X10 F100** N20 **$FOR** P10=0, 20, 1 N30 G91 X5 N40 $ENDFOR : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the control block sequence and the other NC commands in separate NC blocks.
Error type	1, Error message from NC program.		

ID 20207

# command must be programmed exclusively in NC block.			
Description	A plain text command starting with the hash character # is programmed with other NC commands in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G01 X10 F100 #TIME 8.15** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G01 X10 F100** N30 **#TIME 8.15** : N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Program the plain text command and the other NC commands in separate NC blocks. **# command must be programmed exclusively in NC block.** The special plain text commands #ADD and #SUPPRESS OFFSETS can also be programmed together with other NC commands in the same NC block.
Error type	1, Error message from NC program.		

ID 20209

Unknown NC-command.			
Description	A # command or keywords contained in it or axis names and other NC commands cannot be identified. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **#UNKNOWN** ;… N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the syntax of the NC commands or axis names correctly.
Error type	1, Error message from NC program.		

ID 20210

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20211

Control block depth not sufficient. Control block nesting too big.			
Description	In NC program too many control block commands are nested. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=1 N30 $IF N40 $IF... **Nxx $IF...** **: etc.** **Nxx $ENDIF** Nxx $ENDIF Nxx $ENDIF : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce nesting depth, modify structure of control block statements.

Parameter	%1:	Limit value [-]
		Maximum nesting depth
Error type	1, Error message from NC program.	

ID 20212

After 'IF' only the condition is permissible.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $IF instruction, only the corresponding condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=0 N20 $IF P1==1 **X100 G01 F1000** N30 Y200 N40 $ENDIF : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=0 N20 **$IF P1==1** N25 **X100 G01 F1000** N30 Y200 N40 $ENDIF : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them. **Exception:** A $IF instruction may be combined with a $GOTO command in the same block. In this case, no $ENDIF is required any longer.
Error type	1, Error message from NC program.		

ID 20213

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20214

Unexpected ELSE. Does not match the current control block.

Description	A control block statement was programmed in incomplete syntax. A $ELSE can only be programmed in combination with $IF/$ENDIF. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **$ELSE** : N1000 M30 **Correct:** N01 P1=0 N05 **$IF** P1==1 N10 G00 X0 Y0 Z0 N20 **$ELSE** N25 G01 X100 Y0 Z0 F1000 N30 **$ENDIF** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $IF/$ENDIF instruction or delete the $ELSE.
Error type	1, Error message from NC program.		

ID 20215

'ELSE' must be programmed exclusively in NC-block.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $ELSE no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=0 N20 $IF P1 N30 G01 X100 F100 N40 **$ELSE G01 X200 F500** N50 $ENDIF : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=0 N20 $IF P1 N30 G01 X100 F100 N40 **$ELSE** **N45 G01 X200 F500** N50 $ENDIF : N1000 M30 For further information see [PROG// Section: The IF - ELSE branch]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $ELSE to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20216

Unexpected ELSEIF. Does not match the current control block.

Description	A control block statement was programmed in incomplete syntax. A $ELSEIF can only be programmed in combination with $IF/$ENDIF. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **$ELSEIF** : N1000 M30 **Correct:** N01 P1=0 N05 **$IF** P1==1 N10 G00 X0 Y0 Z0 N20 **$ELSEIF** P1==2 N30 G00 X100 Y0 Z0 N40 **$ENDIF** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $IF/$ENDIF instruction or delete the $ELSEIF.
Error type	1, Error message from NC program.		

ID 20217

After 'ELSEIF' only the condition is permissible.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $ELSEIF instruction, only the corresponding condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N251 P2=0 N252 $IF P2 : N253 **$ELSEIF P2==100 G01 X200 F500** : N254 $ENDIF : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N251 P2=0 N252 $IF P2 : N253 **$ELSEIF P2==100** N254 **G01 X200 F500** : N254 $ENDIF : N1000 M30 For further information see [PROG// Section: The IF - ELSE branch]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20218

Unexpected ENDIF. Does not match the current control block.

Description	A control block statement was programmed in incomplete syntax. A $ENDIF can only be programmed in combination with a previous $IF. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=0 N20 G01 X100 F10000 N30 **$ENDIF** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=0 N15 **$IF** P1==0 N20 G01 X100 F10000 N30 **$ENDIF** : N1000 M30 For further information see [PROG// Section: The IF - ELSE branch]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $IF instruction or delete the $ENDIF.
Error type	1, Error message from NC program.		

ID 20219

'ENDIF' must be programmed exclusively in NC-block.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $ENDIF no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=0 N20 $IF P1 N30 **$ENDIF G01 X100 F10000** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=0 N20 $IF P1 N30 **$ENDIF** N40 **G01 X100 F10000** : N1000 M30 For further information see [PROG// Section: The IF - ELSE branch]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $ENDIF to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20220

After 'SWITCH' only the condition is permissible.

Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $SWITCH instruction, only the corresponding condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=5 N50 **$SWITCH P1 G01 X100 F10000** : N65 $ENDSWITCH : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=5 N25 **G01 X100 F10000** N50 **$SWITCH P1** : N65 $ENDSWITCH : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20221

Unexpected CASE. Does not match the current control block.			
Description	A control block statement was programmed in incomplete syntax. A $CASE can only be programmed together with $SWITCH/$ENDSWITCH. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=47 N20 **$CASE P1** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=47 N20 **$SWITCH P1** N20 **$CASE 1** : N30 **$ENDSWITCH** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $SWITCH/ $ENDSWITCH instruction or delete the $CASE.
Error type	1, Error message from NC program.		

ID 20222

After 'CASE' only the condition is permissible.

Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $CASE instruction, only the corresponding condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=5 N30 $SWITCH P1 N40 **$CASE 5 G01 X100 F100** N50 $BREAK : N90 $ENDSWITCH : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=5 N30 $SWITCH P1 N40 **$CASE 5** N45 **G01 X100 F100** N50 $BREAK : N90 $ENDSWITCH : N1000 M30 For further information see [PROG// Section: Switch branching]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20223

Unexpected DEFAULT. Does not match the current control block.			
Description	A control block statement was programmed in incomplete syntax. A $DEFAULT can only be programmed together with $SWITCH/$ENDSWITCH. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **$DEFAULT** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=0 N15 **$SWITCH P1** : N20 $CASE : N30 **$DEFAULT** : N40 **$ENDSWITCH** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $SWITCH/ $ENDSWITCH instruction or delete the $DEFAULT.
Error type	1, Error message from NC program.		

ID 20224

'DEFAULT' must be programmed exclusively in NC-block.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $DEFAULT no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=5 N50 $SWITCH P1 : N62 **$DEFAULT G01 X100 F10000** : N65 $ENDSWITCH : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=5 N50 $SWITCH P1 : N62 **$DEFAULT** N63 **G01 X100 F10000** : N65 $ENDSWITCH : N1000 M30 For further information see [PROG// Section: Switch branching]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $DEFAULT to other NC blocks or delete them.
Error type	Abort NC program processing.		

ID 20225

Unexpected ENDSWITCH. Does not match the current control block.			
Description	A control block statement was programmed in incomplete syntax. A $ENDSWITCH can only be programmed in combination with $SWITCH. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=5 N30 **$ENDSWITCH** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=5 N30 **$SWITCH P1** : N40 **$ENDSWITCH** : N1000 M30 For further information see [PROG// Section: Switch branching]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $SWITCH instruction or delete the $ENDSWITCH.
Error type	Abort NC program processing.		

ID 20226

'ENDSWITCH' must be programmed exclusively in NC-block.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $ENDSWITCH no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=5 N50 $SWITCH P1 : N62 $DEFAULT : N65 **$ENDSWITCH G01 X100 F10000** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=5 N50 $SWITCH P1 : N62 $DEFAULT : N65 **$ENDSWITCH** N66 **G01 X100 F10000** : N1000 M30 For further information see [PROG// Section: Switch branching]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $ENDSWITCH to other NC blocks or delete them.
Error type	Abort NC program processing.		

ID 20227

Wrong counting variable after 'FOR'. Name has to be P, V or R.			
Description	The count variable in a FOR loop was programmed in invalid syntax. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 $FOR **COUNT**=1,10,1 N30 $ENDFOR : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 $FOR **V.P.COUNT**=1,10,1 N30 $ENDFOR : N1000 M30 For further information see [PROG//Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The count variable in a FOR loop must have a valid variable name: Any parameter: starts with character ‘P’ (or ‘R’, if permitted) Any user-defined variable: starts with V.P., V.L., V.S. Predefined variables or variables created in the system configuration: start with V.A., V.G., V.E. (Caution: Not all variables allow write access!!!)

Error type	Abort NC program processing.

ID 20228

FOR loop: After counting variable an equal sign '=' is expected.			
Description	On initialisation of a FOR loop the starting value of the counting variable has to be assigned directly. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **V.P.EIGENDEF = 0** N30 $FOR **V.P.EIGENDEF**,100,10 : N150 $ENDFOR : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N30 $FOR **V.P.EIGENDEF = 0**,100,10 : N150 $ENDFOR : N1000 M30 For further information see [PROG//Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Initialise the counting variable correctly in the FOR loop.
Error type	Abort NC program processing.		

ID 20229

FOR loop: During initialisation of counting variable a comma ',' is expected.			
Description	Start value, increment and end value of a FOR loop must be separated by a comma ‘,’. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 $FOR P1 = **1** 10,1 : N150 $ENDFOR : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 $FOR P1 = **1**,10,1 : N150 $ENDFOR : N1000 M30 For further information see [PROG//Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Add missing commas in the initialisation of the FOR loop.
Error type	Abort NC program processing.		

ID 20230

After 'FOR' only the condition is permissible.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $FOR instruction, only the corresponding loop condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 $FOR P2=1,10,1 **G01 X100 F1000** : N30 $ENDFOR : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 $FOR P2=1,10,1 N25 **G01 X100 F1000** : N30 $ENDFOR : N1000 M30 For further information see [PROG// Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20231

'FOR' condition consists of too many characters. Cache overflow.			
Description	The string length of the loop condition exceeds the permissible limit. For further information see [PROG// Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the string length of the loop condition.
Parameter	%1:	Limit value [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20232

Unexpected ENDFOR. Does not match the current control block.

Description	A control block statement was programmed in incomplete syntax. A $ENDFOR can only be programmed in combination with a previous $FOR. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=1 N20 **$ENDFOR** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=1 N15 **$FOR P1=100, 200, 300** : N20 **$ENDFOR** : N1000 M30 For further information see [PROG// Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $FOR instruction or delete the $ENDFOR.
Error type	1, Error message from NC program.		

ID 20233

ENDFOR must be programmed exclusively in the block.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $ENDFOR no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 $FOR P1=1, 10, 2 : N30 $ENDFOR **G01 X100 F10000** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 $FOR P1=1, 10, 2 : N30 **$ENDFOR** N40 **G01 X100 F10000** : N1000 M30 For further information see [PROG// Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $ENDFOR to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20234

Continuation of parameter syntax is wrong.

Description	The count variable in a FOR loop may only begin with the letters P, V, and R. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 $FOR **COUNT**=1,10,1 N30 $ENDFOR : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 $FOR **V.P.COUNT**=1,10,1 N30 $ENDFOR : N1000 M30 For further information see [PROG// Section: Counting loop]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The count variable in a FOR loop must have a valid variable name: Any parameter: starts with character ‘P’ (or ‘R’, if permitted) Any user-defined variable: starts with V.P., V.L., V.S. Predefined variables or variables created in the system configuration: start with V.A., V.G., V.E. (Caution: not all variables allow write access!!!)
Error type	1, Error message from NC program.		

ID 20235

After 'WHILE' only the condition is permissible.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $WHILE instruction, only the corresponding condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=1 N30 $WHILE P1>0 G01 X100 F1000 : N40 $ENDWHILE : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=1 N30 $WHILE P1>0 N35 G01 X100 F1000 N40 $ENDWHILE : N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20236

'WHILE' condition consists of too many characters. Cache overflow.			
Description	The string length of the loop condition exceeds the permissible limit. For further information see [PROG// Section: Verification of running condition at loop start]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the string length of the loop condition.
Parameter	%1:	Limit value [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20237

Unexpected ENDWHILE. Does not match the current control block.			
Description	A control block statement was programmed in incomplete syntax. A $ENDWHILE can only be programmed in combination with a previous $WHILE. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=1 N20 **$ENDWHILE** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=1 N15 **$WHILE P1>0** : N20 **$ENDWHILE** : N1000 M30 For further information see [PROG// Section: Verification of running condition at loop start]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $WHILE instruction or delete the $ENDWHILE.
Error type	11, Error message from NC program.		

ID 20238

'ENDWHILE' must be programmed exclusively in NC-block.

Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $ENDWHILE no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=1 N30 $WHILE P1 : N40 **$ENDWHILE G01 X200 F500** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=1 N30 $WHILE P1 : N40 **$ENDWHILE** N50 **G01 X200 F500** : N1000 M30 For further information see [PROG// Section: Verification of running condition at loop start]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $ENDWHILE to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20239

'DO' must be programmed exclusively in NC-block.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $DO no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=0 N20 **$DO P1=P1+1** : N50 $ENDDO P1<2 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=0 N20 **$DO** **N30 P1=P1+1** : N50 $ENDDO P1<2 : N1000 M30 For further information see [PROG// Section: Verification of running condition at loop end]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $DO to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20240

Unexpected ENDDO. Does not match the current control block.			
Description	A control block statement was programmed in incomplete syntax. A $ENDDO can only be programmed in combination with a previous $DO. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=1 N20 **$ENDDO P1<2** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=1 N15 **$DO** : N20 **$ENDDO P1<2** : N1000 M30 For further information see [PROG// Section: Verification of running condition at loop end]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $DO instruction or delete the $ENDDO.
Error type	1, Error message from NC program.		

ID 20241

After 'ENDDO' only the condition is permissible.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $ENDDO instruction, only the corresponding condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P2=10 N30 $DO : N40 **$ENDDO P2 >= 10 G01 X100 F1000** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P2=10 N30 $DO : N40 **$ENDDO P2 >= 10** N45 **G01 X100 F1000** : N1000 M30 For further information see [PROG// Section: Verification of running condition at loop end]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20242

'BREAK' must be programmed exclusively in NC-block.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $BREAK no further NC commands may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=5 N30 $SWITCH P1 N60 $CASE 5 N70 G01 X100 F10000 N80 **$BREAK G01 Y200 F10000** N90 $DEFAULT N100 P1=0 N110 $BREAK N120 $ENDSWITCH : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P1=5 N30 $SWITCH P1 N60 $CASE 5 N70 G01 X100 F10000 N75 **G01 Y200 F10000** N80 **$BREAK** N90 $DEFAULT N100 P1=0 N110 $BREAK N120 $ENDSWITCH : N1000 M30 For further information see [PROG// Section: Switch branching]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $BREAK to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 20243

Unexpected 'BREAK'. No valid/open control blocks.			
Description	A control block statement was programmed in incomplete syntax. A $BREAK can only be programmed within a loop or $SWITCH statement. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **$BREAK** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=1 N20 $SWITCH P1 N30 $CASE 2 N35 G01 X100 F10000 N40 **$BREAK** : N100 $ENDSWITCH : N1000 M30 For further information see [PROG//Section: Switch branching]		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify the program flow sequence. Insert a loop or $SWITCH statement or delete the $BREAK.
Error type	1, Error message from NC program.		

ID 20244

Unexpected *CONTINUE'. No valid/open control blocks.			
Description	A control block statement was programmed in incomplete syntax. A $CONTINUE can only be programmed inside any loop or $SWITCH statement. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **$CONTINUE** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=1 N20 **$DO** N30 P1=P1+1 N40 $IF P1 == 2 N50 **$CONTINUE** N60 $ENDIF : N100 **$ENDDO P1 < 10** : N1000 M30 For further information see • [PROG// Section: Switch branching] • [PROG// Section: Counting loops]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert a loop or $SWITCH statement or remove the $CONTINUE command.
Error type	1, Error message from NC program.		

ID 20245

'CONTINUE' must be programmed exclusively in NC-block.

<table>
<tr><td>Description</td><td colspan="3">In the same NC block, a control block instruction was programmed with further NC commands. After a $CONTINUE no further NC commands may be programmed.
Example:
Wrong:
N10 G00 X0 Y0 Z0
N20 $FOR P1=1, 10, 2
N30 $IF P1 == 8
N40 $CONTINUE G01 X100 F10000
N50 $ENDIF
:
N100 $ENDFOR
:
N1000 M30

Correct:
N10 G00 X0 Y0 Z0
N20 $FOR P1=1, 10, 2
N30 $IF P1 == 8
N35 G01 X100 F10000
N40 $CONTINUE
N50 $ENDIF
:
N100 $ENDFOR
:
N1000 M30

For further information see [PROG// Section: Counting loops]</td></tr>
<tr><td>Response</td><td>Class</td><td>2</td><td>Abort NC program processing.</td></tr>
<tr><td>Solution</td><td>Class</td><td>3</td><td>Check and modify NC program. Move the not permissible NC commands after $CONTINUE to other NC blocks or delete them.</td></tr>
<tr><td>Error type</td><td colspan="3">1, Error message from NC program.</td></tr>
<tr><td></td><td colspan="3"></td></tr>
</table>

2.3.2 ID-range 20250-20499

ID 20251 - 20259

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20260

Writing from external device caused error due to parameter number less than or equal to 0.			
Description	When writing a P parameter from the user interface, a negative parameter number is used.		
Response	Class	3	Continue NC program processing.
Solution	Class	1	Restart the NC program. Use a positive parameter number when writing the P parameter from the user interface.
Parameter	%1:	Error value [-]	
		Invalid parameter number	
Error type	3, Error in the communication.		

ID 20262

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20263

Reading from external device caused error due to parameter number less than or equal to 0.			
Description	When writing a P parameter from the user interface, a negative parameter number is used.		
Response	Class	3	Continue NC program processing.
Solution	Class	1	Restart the NC program. Use a positive parameter number when writing the P parameter from the user interface.
Parameter	%1:	Error value [-]	
		Invalid parameter number	
Error type	3, Error in the communication.		

ID 20264

External request of unknown parameter value.	
Description	An unknown P parameter shall be read in the NC channel from the user interface.

Response	Class	3	Continue NC program processing.
Solution	Class	1	Restart the NC program. The read access from the user interface only is possible for known P parameters in the NC channel.
Parameter	%1:	Error value [-]	
		Number of the unknown parameter.	
Error type	3, Error in the communication.		

ID 20265

Reading of an external device caused error due to invalid parameter type.			
Description	The read order for a P parameter from the user interface contains an invalid type definition for this P parameter.		
Response	Class	3	Continue NC program processing.
Solution	Class	1	Check the order generation of the user interface. The following types are defined for the P parameter: long double (REAL64) or long int (SGN32)
Parameter	%1:	Current value [-]	
		Value of the parameter to be read	
	3, Error in the communication.		

ID 20266

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20267

The received manual block is too long.			
Description	The manual block (MDI), assigned over the user interface, exceeds the permissible length, which can be processed control internally.		
Response	Class	-	Abort NC program processing.
Solution	Class	-	Limit the manual set to the permissible length. Possibly divide the invalid manual block into several manual blocks (additive manual block processing).
Parameter	%1:	Limit value [-]	
		Maximal length of manual block	
Error type	-		

ID 20268 - 20281

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20282

List interpreter cannot evaluate the predefined list.			
Description	During the controller start-up, the list interpreter cannot evaluate a file path or list name entered in the start-up list. Possibly some of these data are wrong.		
Response	Class	1	Start-up of the control is aborted.
Solution	Class	7	Check the way of writing of the corresponding file path or list file name, and correct it if necessary.
Error type	-		

ID 20283

Loading the zero point data to the DECODER failed.			
Description	During controller start-up or updating, the list interpreter cannot evaluate the zero offset list due to invalid or unknown data.		
Response	Class	3	Controller start-up or data update is aborted.
Solution	Class	7	Check the entries in the zero offset list and correct if necessary.
Error type	-		

ID 20284 - 20287

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20288

Loading the tool data to the DECODER failed.			
Description	During controller start-up or update, the list interpreter cannot evaluate the tool data list due to invalid or unknown information.		
Response	Class	3	Controller start-up or data update is aborted.
Solution	Class	7	Check the entries in the tool data list and correct if necessary.
Error type	-		

ID 20289

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	

Solution	Class	8	Requires controller restart.

ID 20290

Loading the channel parameter data to the DECODER failed.			
Description	During controller start-up or update, the list interpreter cannot evaluate the channel parameter list due to invalid or unknown information.		
Response	Class	3	Controller start-up or data update is aborted.
Solution	Class	7	Check entries in the channel parameter list and correct if necessary.
Error type	-		

ID 20291 - 20295

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20296

Invalid tool was sent from tool management.			
Description	The tool received from the external tool management is invalid. There are various reasons: The T, D number used is unknown in the tool management. Tool management error.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the data of the tool in the external tool management.
Parameter	%1:	Current value [-]	
		Number of the requested but not valid tool	
	%2:	Current value [-]	
	%3:	Current value [-]	
Error type	3, Error in the communication.		

ID 20297

Unexpected tool was sent from tool management.			
Description	The tool received from the external tool management is different to the requested tool.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify processing and supply of tool requests in the external tool management.
Parameter	%1:	Error value [-]	
		Number of the received tool	
	%2:	Expected value [-]	
		Number of the requested tool	

Error type	3, Error in the communication.

ID 20298 - 20311

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20312

The requested logical axis is unknown.			
Description	In an axes exchange command the programmed logical axis number of an axis is unknown in the system.		
Response	Class	-	Abort NC program processing.
Solution	Class	-	Check and modify NC program. Only axes that are available in the system through configuration (P-CHAN-00035) can also be programmed in the axis exchange command.
Error type	-		

ID 20317

Axis offset is out of data format.			
Description	When exchanging an axis in the channel, offsets are accepted or deducted depending on the programming. The system detects that one of the assigned axis specific offsets exceeds the permissible data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the values of all offsets of the exchanged axis values in the lists or in the NC program whether one of the offsets was assigned an invalid value.
Parameter	%1:	Logical axis number [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	3, Error in the communication.		

ID 20324 - 20338

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20339

Overflow of output-FIFO for reporting of changes. Display data is lost.			
Description	An internal system resource of the active change logging function is full.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Reduce the number of enabled display data [CHAN// Section: Change logging function].
Error type	1, Error message from NC program.		

ID 20341 - 20346

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20347

Overflow of channel block buffer.			
Description	No contouring block (chamfer or rounding) can be inserted between two motion blocks when contouring (G301/G302) because a necessary system resource is not available.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	The NC program is continued without contouring (G301/G302). As soon as the system resource is available again, also the contouring blocks are inserted again. This behaviour ensures that the NC program is completely executed in each case.
Error type	-		

ID 20348 - 20350

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20351

No primary motion block for chamfer/rounding contouring available.

Description	Chamfers (G301) or radii (G302) can only be inserted **between** motion blocks of group G00, G01, G02, G03. Therefore, a preceding motion block must be programmed before G301/G302. Example: **Wrong:** N10 **G301 I20** N20 G01 X100 Y100 F10000 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G01 X100 F10000** N30 **G301 I20** N40 **G01 Y100 F10000** : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. G301/G302 must always be programmed between 2 motion blocks.
Error type	1, Error message from NC program.		

ID 20352

Internal buffers should be cleared although chamfer/rounding contouring is active.			
Description	Certain NC commands (e.g. #FLUSH) cause a so-called “flushing” of the NC channel, i.e. internal buffers of the NC channel are enabled. This results in a conflict if the contouring G301/G302 function is enabled at this time since this is exactly where the buffer contents are required. In this case no additional motion block is inserted. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G01 X50 F10000 N30 **G301 I20** N40 **#FLUSH** N50 X100 Y100 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G01 X50 F10000 N30 **G301 I20** N40 X100 Y100 N50 **#FLUSH** : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. NC commands that shorten a so-called flushing of the NC channel may not be programmed within a contouring sequence, only before or behind it.
Error type	1, Error message from NC program.		

ID 20353

Missing plane axis for chamfer/rounding contouring.

Description	One of the axes of the plane in which contouring mode was programmed with G301/ G302 is not available. Possibly a plane axis was released by an axis exchange command. In this case no additional motion block is inserted. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **#PUT AX[Y]** N30 G01 X200 F10000 N40 **G302 I10** N50 Y200 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G01 X200 F10000 N40 G302 I10 N50 Y200 : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Make sure that no plane main axes necessary for the contouring sequence are missing due to axis exchange commands.
Error type	1, Error message from NC program.		

ID 20354

Chamfer/rounding contouring impossible.			
Description	No contouring block (chamfer or rounding) can be inserted between two motion blocks with contouring (G301/G302) because of the programmed geometry data. Example: **Wrong:** N10 G00 X0 Y0 Z0 N180 G03 X80 I40 F1000 N190 **G302 I10** (I too large) N200 G03 X95 Y-40 I20 J-15 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N180 G03 X80 I40 F1000 N190 **G302 I2** N200 G03 X95 Y-40 I20 J-15 : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. If possible adapt the geometry data of the contouring block (chamfer or rounding).
Error type	1, Error message from NC program.		

ID 20355 - 20359

System error [▶ 12]	
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.

Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20360

Stack overflow for global sub-routines.			
Description	The maximum call depth for nested global subroutines is exceeded. This can occur, for example, when programming a recursion without termination criteria, e.g. a global subroutine calls itself continuously.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the call depth of nested global subroutines or modify the program flow. Avoid recursions.
Parameter	%1:	Limit value [-]	
		Maximum nesting depth for global subroutines.	
Error type	1, Error message from NC program.		

ID 20361

Stack overflow for local sub-routines.			
Description	The system resource for the management of subroutines is exceeded. This can occur, for example, if several local subroutines are defined and then nested global subroutines are also called.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. All in all, reduce the number of subroutines or the nesting depth of global subroutines, or change the program flow. Avoid recursions.
Parameter	%1:	Limit value [-]	
		Maximum size of the system resource.	
Error type	1, Error message from NC program.		

ID 20362

File name is too long.			
Description	At program start, start of block search or when a global subroutine is called, a file name of excessive length is used.		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Check the file names of the NC programs and use shorter names as necessary.
Parameter	%1:	Limit value [-]	
Error type	5, Error message by access on files.		

ID 20364

No file access possible.			
Description	The NC program to be opened or a log file cannot be found in the program paths configured in the start-up list.		
Response	Class	-	Abort NC program processing.

Solution	Class	-	Check whether file exists and ensure that the file is available in the configured program paths. Check NC program file names or log files for correct notation and restart.
Parameter	%1:	Current value	
		Id for the invalid file access: "Main prg not found": Main program "Init prg not found": Implicit subroutine at program start (P-CHAN-00119) "Sub prg not found": Global subroutine "Invalid MsgSavePrgName": #MSG SAVE..	
	%2:	Block number	
		Only with global subroutine L.. and #MSG SAVE: NC block number of program call.	
	%3:	Error value	
		Only with #MSG SAVE display of the log file that cannot be opened.	
Error type	5, Error message by access on files.		

ID 20365

Overflow of sub-routine stack.			
Description	All in all too many local sub-routines were defined in the NC main program or in combination with global subroutines. The number is determined by the sum of all %L definitions in the current "program stack" (main program UP1-UP11-UP111…). The sum may not exceed the specified maximum number The version-dependent start-up parameter "configuration.channel[i].decoder.max_local_subroutine_definitions" can be used to modify the maximum counter. This error message has nothing to do with the nesting depth of actual subroutine calls!		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the number of local subroutines. For example convert local subroutines to global subroutines.
Parameter	%1:	Limit value [-]	
		Maximum number of local subroutines	
Error type	1, Error message from NC program.		

ID 20367

Name of main program too long.			
Description	The name of the main program in the NC program (%Main_program_name...) is too long.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the length of the main program name.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20368

Main program not found, syntax error at definition of program.			
Description	The called file does not contain a main program definition (%main program name), but only local subroutine definitions, for example. Example: **Wrong:** %L sub1 N10 G92 X10 Y20 Z30 : N100 M17 %L sub2 N10 G92 X-10 Y-20 Z-30 : N100 M17 **N10 X0 Y0 Z0** : N1000 M30 **Correct:** %L sub1 N10 G92 X10 Y20 Z30 : N100 M17 %L sub2 N10 G92 X-10 Y-20 Z-30 : N100 M17 **%Main_program_name** **N10 X0 Y0 Z0** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify NC program. Before the beginning of the main program sequence insert the main program name. Note: The main program definition (*%Main program name*) can only be omitted if no local sub-routines exist.
Error type	1, Error message from NC program.		

ID 20369

Global subroutine not found, syntax error at definition of program.

Description	The called file of the global subroutine does not contain a corresponding program definition (%Glob_Program name), but only local subroutine definitions, for example. Example: **Wrong:** (Global subroutine, separate file): %L sub1 N10 G92 X10 Y20 Z30 : N100 M17 **N10 X0 Y0 Z0** : N1000 M17 **Correct:** %L sub1 N10 G92 X10 Y20 Z30 : N100 M17 **%Glob_program_name** **N10 X0 Y0 Z0** : N1000 M17		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Insert the name of the global subroutine before the start of the corresponding program sequence. Note: The program definition (*%Glob_program_name*) can only be omitted if no local sub-routines exist.
Error type	1, Error message from NC program.		

ID 20370

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20371

Overflow of program stack, too many local sub-routines opened.			
Description	The maximum call depth for nested local subroutines was exceeded. This can occur, for example, when programming a recursion without termination criteria, e.g. a local subroutine calls itself continuously. Example: %L OVERFLOW.nc ;N10 **Overflow of program stack, too many local sub-routines opened.** N20 M17 %main.nc N10 G00 X0 Y0 Z0 N20 LL OVERFLOW.nc N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Reduce the call depth of nested local subroutines or modify the program flow. Avoid recursions.
Parameter	%1:	Limit value [-]	
		Maximum nesting depth for local subroutines.	
Error type	1, Error message from NC program.		

ID 20372

Name of local sub-routine too long.			
Description	The name of the local subroutine in the NC program (%L Sub_routine_name...) is too long.		
Response	Class	2	`Abort NC program processing.`
Solution	Class	3	Check and modify NC program. Shorten subroutine name.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20373

Local subroutine not found.			

Description	The local subroutine called does not exist.
	Example:
	`%main`
	`N10 G00 X0 Y0 Z0`
	`N20 LL sub1`
	`:`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Define the local subroutine before the main program (%L subroutine name...). Only local subroutines may be called by "LL". Global sub-routines only call by "L"!
Error type	1, Error message from NC program.		

ID 20374

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20375

There are valid open control blocks at program end.

Description	A valid control block statement (for example, $IF-$ENDIF) was not closed in a main program or subroutine, i.e. an assigned $ENDIF is missing. Valid here means that the programmed condition is true and the NC blocks in this branch are executed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P1=1 P2=2 N30 **$IF P1 == 1** N40 Y100 N50 $IF P2==4 N60 Z200 N70 $ENDIF : N100 X100 : N1000 M30 ← $ENDIF of $IF P1.. missing. **Correct:** N10 G00 X0 Y0 Z0 N20 P1=1 P2=2 N30 **$IF P1 == 1** N40 Y100 N50 $IF P2==4 N60 Z200 N70 $ENDIF N80 **$ENDIF** : N100 X100 : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Close the control block statement correctly. Complete the missing $ENDIF/$ENDSWITCH... .
Parameter	%1:	Error value [-]	
		Number of open control blocks	
	%2:	Current value [-]	
		Initial keyword of incomplete control block statement (e.g. $IF)	
	%3:	Expected value [-]	
		Closing keyword of control block statement (e.g. $ ENDIF)	
Error type	1, Error message from NC program.		

ID 20376

Unexpected M17 or M29.			
Description	In the NC main program, the end of a subroutine (M17 or M29) is read although no subroutine is open. Example: %main N10 G00 X0 Y0 Z0 : N100 **M17** : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Remove the redundant M17/M29.
Error type	1, Error message from NC program.		

ID 20377

Unexpected M30 or M02.			
Description	In a subroutine the end of the main program (M02/M30) is detected. Example: **Wrong:** %L sub1 N10 G91 X10 Y20 Z30 : N100 **M30** %main N10 G00 X0 Y0 Z0 N20 LL sub1 : N1000 M30 **Correct:** %L sub1 N10 G91 X10 Y20 Z30 : N100 **M17** %main N10 G00 X0 Y0 Z0 N20 LL sub1 : N1000 **M30**		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Replace M02/M30 in subroutines with M17/M29. Finish main program with M02/M30.
Error type	1, Error message from NC program.		

ID 20378

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20381

Unknown axis designation in #-command.			
Description	In NC channel, the axis name programmed in the #-command is unknown. Only the axis names configured in the channel parameter list (P-CHAN-00006) or the axis names defined via axis exchange commands can be used in the NC program.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only use in the #-commands the configured or defined axes names of the NC channel.
Error type	1, Error message from NC program.		

ID 20382 - 20384

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20385

Coordinate is out of data format.			
Description	When the axis positions are calculated after deselection of synchronous operation, or after deselection of a coordinate transformation, it was found that an axis position exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the machine axis positions before deselecting synchronous operation or coordinate transformation. Move to a smaller axis position in the NC program before deselecting this option. Check whether the correct kinematic type is entered in the channel parameters P-CHAN-00032 or in the NC program (#KIN ID).
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20386

Overflow of arithmetic stack.			
Description	Too many nested bracketed terms are programmed when a mathematical expression is calculated or resolved.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Simplify mathematical expression or, if possible, divide it into several NC blocks.
Parameter	%1:	Limit value [-]	
		Maximum nesting depth	
	%2:	Error value [-]	
		Currently reached nesting depth	
Error type	1, Error message from NC program.		

ID 20387

After function call an opening square bracket is expected.

Description	Before the argument of a function call, no opening **square** bracket is programmed Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P2=**SIN45]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P2=**SIN[45]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the bracket programming of the function call.
Error type	1, Error message from NC program.		

ID 20392

Unknown term in mathematical expression.			
Description	The programmed mathematical expression is incomplete or includes unknown or invalid terms and functions.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use correct syntax to program mathematical expression.
Error type	1, Error message from NC program.		

ID 20394

Variable access caused syntax error, variable can not be identified.			
Description	A variable (e.g. V.A., V.G., V.E., V.S., V.P., V.L., etc.) used in NC program is unknown.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. When programming, keep to or ensure the correct spelling of the variables and that all the variables used are defined.
Error type	1, Error message from NC program.		

ID 20396

Division by 0.			
Description	In NC program within a mathematical expression, a division by zero (nil) is detected. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **P2=10/COS[90]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use correct syntax to program mathematical expression.

Parameter	%1:	Error value [-]
Error type	1, Error message from NC program.	

ID 20397

After the argument of the function call a closing square bracket is missing.			
Description	After the argument of a function call or a #-command, no closing **square** bracket is programmed Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P2=**[SIN45]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P2=**SIN[45]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the bracket programming of the function call or #-command.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20398

Negative root.			
Description	The square root is to be calculated from a negative number. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P2=SQRT[5-6] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The radical of the square root must be ≥ zero.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20399

Logarithm less than/equal 0.

Description	The logarithm is to be calculated from a negative number or zero (nil). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **P2=LOG[0]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The argument of the logarithm must be > zero (nil).
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20400

P parameter number is 0, negative or greater than maximum limit.			
Description	The programmed number of the P parameter exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **P0=10** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of P parameter must be within the permissible value range.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20401

R parameter number is negative or greater than maximum limit.			
Description	The programmed number of the R parameter exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **R-5=10** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed number of R parameter must be within the permissible value range.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20402

Unknown radius syntax.			
Description	For circle programming, an invalid radius syntax is used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. For radius programming, only R or R1 (indexed radius programming) is permitted.
Error type	1, Error message from NC program.		

ID 20403

Variable number is negative or greater than maximum limit.			
Description	The programmed number of the V. variable exceeds the permissible data range. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P10 = **V.G.NP[-1].V[0]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P10 = **V.G.NP[1].V[0]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed index of V. variable must be within the permissible value range.

Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]

Error type	1, Error message from NC program.

ID 20407

Zero point data contain errors.			
Description	During the interpretation at start-up or update of modified data, the plausibility check of the zero point data list detects an error. (parameter unknown or incorrectly written, invalid values). This error message usually is output as a subsequent error message. Check, if a previous error message has been created, which described the error cause more detailed.		
Response	Class	3	Abort job processing.
Solution	Class	6	Check and change the list of zero offsets.

Parameter	%1:	Current value [-]

Error type	-

ID 20408

Tool data contain errors.			
Description	When interpreting the binary tool data list, the plausibility check of the binary channel parameters list detects an error during the start-up . (for example, invalid value). This error message usually is output as a subsequent error message. Check, if a previous error message has been created, which described the error cause more detailed.		
Response	Class	3	Abort job processing.
Solution	Class	6	Check and modify binary list of tool data.
Parameter	%1:	Current value [-]	
		Start address of binary list.	
Error type	-		

ID 20409

Channel parameters contain errors.			
Description	When interpreting the binary channel parameters list, the plausibility check of the binary channel parameters list detects an error during the start-up . (for example, invalid value). This error message usually is output as a subsequent error message. Check, if a previous error message has been created, which described the error cause more detailed.		
Response	Class	3	Abort job processing.
Solution	Class	6	Check and modify binary list of channel parameters.
Parameter	%1:	Current value [-]	
		Start address of binary list.	
Error type	-		

ID 20410

During the calculation of the actual coordinates the data format is violated.			
Description	When the current axes coordinates are calculated, the permissible value range is exceeded at start-up.		
Response	Class	2	Abort job processing.
Solution	Class	6	Check and modify binary list of axis coordinates.
Parameter	%1:	Current value [-]	
		Start address of binary axis coordinates.	
Error type	-		

ID 20411

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20418

Channel parameters: Number of spindles and number of configured spindle axes do not coincide.			
Description	At start-up, the channel parameter check determines that the actual number of configured spindles (spindle data) does not coincide with the total number of spindles P-CHAN-00082.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In case of conflict, the total number of spindles at start-up is set to the real number of configured spindles and the start-up is continued.
Parameter	%1:	Current value [-]	
		Total number of spindles P-CHAN-00082	
	%2:	Expected value [-]	
		Real number of configured spindles (spindle data)	
	%3:	Corrected value [-]	
		Automatically corrected total number of spindles	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20419

Channel parameters: Name of spindle is used several times.			
Description	During start-up, the channel parameter check determines that two spindles have the same name P-CHAN-00007.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the names of the spindles P-CHAN-00007 in the channel parameter list.
Parameter	%1:	Logical axis number [-]	
		Axis number of the first spindle with the same name	
	%2:	Logical axis number [-]	
		Axis number of the second spindle with the same name	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20420

Channel parameters: The same axis number has been assigned to two spindles.			
Description	During start-up, the check of the channel parameter detect that two spindles have the same logical axes number P-CHAN-00036.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the logical axis numbers of the spindles P-CHAN-00036 in the channel parameter list.
Parameter	%1:	Logical axis number [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20423

Channel parameters: Configured path axes uses several times an axis number.	
Description	During start-up, the check of the channel parameter detects that two path axes have the same logical axis number P-CHAN-00035.

Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the logical axis numbers of the path axes P-CHAN-00035 in the channel parameter list.
Parameter	%1:	Logical axis number [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20424

Channel parameters: Configured path axes have used axis name several times.			
Description	During start-up, the channel parameter check determines that two path axes have the same name P-CHAN-00006.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the names of the path axes P-CHAN-00006 in the channel parameter list.
Parameter	%1:	Logical axis number [-]	
		Axis number of the first path axis with the same name	
	%2:	Logical axis number [-]	
		Axis number of the second path axis with the same name	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20425

Channel parameters: Name of axis too long or starts with illegal character.			
Description	During start-up, the channel parameter check determines that the name of a path axis P-CHAN-00006 or the default name of a spindle in the coordinated motion P-CHAN-00010 begins with an incorrect letter, or that the axis name is too long. For an axis name the following first characters are permitted: X, Y, Z, A, B, C, U, V, W, Q.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	During start-up in case of conflict the axis name is corrected in a useful way and the start-up is continued. - For a wrong default name of a spindle the path compound P-CHAN-00010 the axis designation "C1" is generated. - For a wrong name of a path axis P-CHAN-00006, an axis designation is generated from the combination of "X" and the associated logical axis number.
Parameter	%1:	Current value [-]	
		Invalid first character of axis name	
	%2:	Limit value[-]	
		Maximum length of axis name	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20426

Channel parameters: Axis is configured both as spindle and as path axis.

Description	During start-up, the check of the channel parameter detects that a path axis and a spindle axis have the same logical axis number P-CHAN-00035 (P-CHAN-00035, P-CHAN-00036).		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and change the logical axis numbers of the path axes P-CHAN-00035 and spindle axes P-CHAN-00036 in the channel parameter list. In the system, a logical axis number only may be assigned to one single axis.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value[-]	
		Index of spindle has same logical axis number as path axis	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20427

Channel parameters: Number of axes of a group is out of range of permissible values.			
Description	At start-up, a check of the channel parameters detected that the real number of configured path axes P-CHAN-00007 in the axes group exceeds the permissible value range.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In case of conflict, the total number of path axes at start-up is set to the maximum permissible limit value and the start-up is continued. Note: The complete number of path axes configured in the channel parameter list may then not be available in the channel after start-up!
Parameter	%1:	Error value [-]	
	%2:	Limit value[-]	
	%3:	Corrected value[-]	
		Total number of path axes automatically corrected	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20428

Channel parameters: Number of axes and number of configured path axes do not coincide.			
Description	At start-up, the channel parameter check determines that the actual number of configured path axes does not match the specified total number of path axes P-CHAN-00003.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In case of conflict, the total number of path axes at start-up is set to the real number of configured path axes and the start-up is continued.
Parameter	%1:	Current value[-]	
		Total number of path axes P-CHAN-00003	
	%2:	Expected value [-]	
		Real number of configured path axes	
	%3:	Corrected value [-]	
		Total number of path axes automatically corrected	

Error type	2, Error message by data transfer from parameter list into control device.

ID 20429

Channel parameters: Incorrect setting of number of groups.			
Description	During start-up, the channel parameter check determines that the actual number of configured axis groups does not correspond to the specified total number of axis groups P-CHAN-00023.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	During start-up in case of conflict the total number of axes groups is set to the smaller number of both values and the start-up is continued. Note: The complete number of configured axis groups in the channel parameter list may then not be available in the channel after start-up! At the moment only one axis group (index 0) is permissible in the NC channel.
Parameter	%1:	Current value [-]	
		Real number of configured axes groups	
	%2:	Expected value [-]	
		Total number of axis groups P-CHAN-00023	
	%3:	Corrected value [-]	
		Total number of axis groups automatically corrected	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20430

Channel parameters: Number of groups exceeds maximum limit.			
Description	At start-up, a check of the channel parameters detects that the total number of axis groups P-CHAN-00023 exceeds the permissible value range.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In case of conflict, the total number of axis groups at start-up is set to the maximum permissible limit value and the start-up is continued. Note: The complete number of configured axis groups in the channel parameter list may then not be available in the channel after start-up! At the moment only one axis group (index 0) is permissible in the NC channel.
Parameter	%1:	Current value [-]	
		Total number of axis groups P-CHAN-00023	
	%2:	Limit value [-]	
		Maximum permissible number of axis groups	
	%3:	Corrected value [-]	
		Total number of axis groups automatically corrected	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20434

Channel parameters: The characteristic of P-parameters is not configured.

Description	During start-up, the channel parameter check determines that the effectiveness of P parameters P-CHAN-00067 is not defined.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	In the event of a conflict, the effectiveness of P parameters P-CHAN-00067 is set to 0 (FALSE) during the start-up and the start-up is continued. P-CHAN-00067 must be assigned 0 (FALSE) or 1 (TRUE).
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20435

Channel parameters: Value for the absolute centre point offset was not defined.			
Description	A check of the channel parameters at start-up detects that the absolute centre point offset P-CHAN-00059 is set to zero (nil).		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In the event of a conflict, the absolute centre point offset P-CHAN-00059 is set to 20000.0 at start-up, and the start-up is continued. P-CHAN-00059 must be set to a value unequal to zero (nil).
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20436

Channel parameters: Value for the relative centre point offset was not defined.			
Description	At start-up, a check of the channel parameters detects that the relative centre point offset P-CHAN-00060 is set to zero (nil).		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In the event of a conflict, the relative centre point offset P-CHAN-00060 is set to 50 at start-up, and the start-up is continued. P-CHAN-00060 shall be set to a value unequal to zero (nil).
Parameter	%1:	Error value[-]	
	%2:	Corrected value[-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20437

Channel parameters: Measuring type is illegal or not defined.
Description: During start-up, the channel parameter check determines that the standard measurement type P-CHAN-00057 is not correctly defined.

Response	Class	1	Start-up of the control is continued.
Solution	Class	1	In the event of a conflict, the default measurement type P-CHAN-00057 is set to 1 (measurement type 1) during the start-up, and thMe start-up is continued. P-CHAN-00057 must be assigned a permissible measurement type (1 to 6).
Parameter	%1:	Error value[-]	
	%2:	Corrected value[-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20438

Channel parameters: Axis number equal to 0 within master/slave coupling.			
Description	During start-up, the check of the channel parameters detects that in the default coupling group 0 (zero) a master or slave axis (P-CHAN-00037, P-CHAN-00038) with the logical axis number 0 (zero) is defined.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	During start-up in case of conflict the master-/slave axes pair with the wrong logical axis number is skipped and the start-up is continued. P-CHAN-00037 or P-CHAN-00038 must be specified with permissible logical axis numbers (> 0).
Parameter	%1:	Current value [-]	
		Index of the invalid master-/slave axes pair	
	%2:	Logical axis number [-]	
		Axis number of master axis	
	%3:	Logical axis number [-]	
		Axis number of slave axis.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20439

Channel parameters: Number of master and slave axes is different.			
Description	At start-up, a check of the channel parameters detects that a different number of master and slave axes (P-CHAN-00037, P-CHAN-00038) are defined in the default coupling group 0 (zero).		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In case of conflict, the total number of master and slave axes at start-up is set to the smaller number of both values and the start-up is continued. Note: The complete number of master/slave axis pairs configured in the channel parameter list may then not be available after start-up in the channel.
Parameter	%1:	Current value [-]	
		Number of master axes	
	%2:	Corrected value[-]	
		Number of slave axes	
	%3:	[-]	
		Total number of master and slave axes automatically corrected	

Error type	2, Error message by data transfer from parameter list into control device.

ID 20440

Channel parameters: Slave axis can not be its own master axis.			
Description	During start-up, the check of the channel parameters detects that in the default coupling group 0 (zero), the master and slave axis (P-CHAN-00037, P-CHAN-00038) of a coupling pair, have the same logical axis number.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	During start-up in case of conflict the master-/slave axes pair with the identical logical axis number is skipped and the start-up is continued. P-CHAN-00037 or P-CHAN-00038 must be specified with different permissible logical axis numbers (> 0). **Note**: Possibly not the complete number of configured master-/slave axes pairs of the channel parameter list is available after start-up in the channel!
Parameter	%1:	Current value [-]	
		Index of the invalid master-/slave axes pair	
	%2:	Logical axis number [-]	
		Redundant logical axis number	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20441

Channel parameters: Slave axis can not be configured as slave axis several times.			
Description	During start-up, the check of the channel parameters detects that, in the default coupling group 0 (zero), an axis is defined as slave axis P-CHAN-00038 in several coupling pairs.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	During start-up in case of conflict the master-/slave axes pair with the redundant slave axis is skipped and the start-up is continued. A slave axis P-CHAN-00038 may only be assigned to one master axis within the coupling group. Note: Possibly not the complete number of configured master-/slave axes pairs of the channel parameter list is available after start-up in the channel!
Parameter	%1:	Current value [-]	
		Index of the first master-/slave axes pair	
	%2:	Current value [-]	
		Index of the second master-/slave axes pair	
	%3:	Logical axis number [-]	
		Axis number of the redundant slave axis	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20442

Channel parameters: Recursive master-slave assignment is not permitted.

Description	At start-up, a check of the channel parameters detects that an axis in the default coupling group 0 (zero) is defined both as a master axis P-CHAN-00037 and as a slave axis P-CHAN-00038 in different coupling pairs.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In case of conflict, the master/slave axis pair with the recursive master/slave axes linking is skipped at start-up and the start-up is continued. A master axis P-CHAN-00037 must always remain the master axis within a coupling group in different coupling pairs. Note: The complete number of master/slave axis pairs configured in the channel parameter list may then not be available after start-up in the channel.
Parameter	%1:	Current value [-]	
		Index of the first master/slave axes pair	
	%2:	Current value [-]	
		Index of the second master/slave axes pair	
	%3:	Logical axis number [-]	
		Axis number of the invalid master axis configured	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20444

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20450

In NC block #-function has to be programmed exclusively.			
Description	After # command <*#COMMAND ...*>, no further NC commands are permitted in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #MSG ["PROGRAMMSTART"] **G74 X1 Y2 Z3** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #MSG ["PROGRAMMSTART"] **N30 G74 X1 Y2 Z3** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Move the invalid NC commands to the following NC block. Note: Other NC commands in the same NC block are only permissible with #ACHSE, #PSET and #PRESET.

Parameter	%1:	Error value [-]
	%2:	Current value [-]
Error type	1, Error message from NC program.	

ID 20457

De-/selection of synchronous operation not allowed during active TRC.			
Description	During active tool radius compensation (TRC, G41/G42), the selection/ deselection of synchronous operation mode of axes is not permissible. Example: **Wrong:** N10 #SET AX LINK [1, C=X] N20 G41 N30 **#ENABLE AX LINK [1]** N30 X100 N40 G40 N50 #DISABLE AX LINK [1] N60 X0 **Correct:** N10 #SET AX LINK [1, C=X] N30 **#ENABLE AX LINK [1]** N20 **G41** N30 X100 N40 **G40** N50 #DISABLE AX LINK [1] N60 X0		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Activate synchronous operation before selecting the radius compensation (G41/G42) or deactivate tool radius compensation before deselecting synchronous operation (G40).
Error type	1, Error message from NC program.		

ID 20461

Number of the coupling group is out of range of permissible values.			
Description	The value of the coupling group number programmed with an axis coupling command (definition, selection, deselection) exceeds the permissible data range. For further information see [PROG//Section: Synchronous operation]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible coupling group number.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20464

No coupling condition for selected coupling group.			
Description	The number of the coupling group programmed with an axis coupling command (selection, deselection) is not defined. For further information see [PROG// Section: Synchronous operation]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before selection or deselection, the coupling group with the missing number must be defined.
Parameter	%1:	Error value [-]	
		Unknown number of the coupling group	
Error type	1, Error message from NC program.		

ID 20465

Master axis not available as path axis.			
Description	With selection or deselection of synchronous operation, it is detected that one of the defined master axis is not available in the NC channel. The test is also executed during the automatic restoration of axle couplings (P-CHAN-00104, P-CHAN-00105) and during the deselection of transformations. For further information see [PROG// Section: Synchronous operation]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure that the master axis is available in the NC channel before selecting or deselecting synchronous operation. Possibly check axes exchange commands used before.
Parameter	%1:	Logical axis number [-]	
		Axis number of the missing master axis	
Error type	1, Error message from NC program.		

ID 20466

Slave axis not available as path axis.			
Description	With selection or deselection of synchronous operation, it is detected that one of the defined slave axis is not available in the NC channel. The test is also executed during the automatic restoration of axle couplings (P-CHAN-00104, P-CHAN-00105) and during the deselection of transformations. For further information see [PROG// Section: Synchronous operation]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure that the slave axis is available in the NC channel before selecting or deselecting synchronous operation. Possibly check axes exchange commands used before.
Parameter	%1:	Logical axis number [-]	
		Axis number of the missing slave axis	
Error type	1, Error message from NC program.		

ID 20467

Slave axis is not allowed to be a main axis.

Description	When selecting synchronous operation, it is detected that one of the defined slave axes is a main axis, this means that it has the axis index 0, 1 or 2. For further information see [PROG// Section: Synchronous operation]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Make sure that the slave axes are not main axes when synchronous operation is selected. Check the defined coupling groups and axes exchange commands possibly used before.
Parameter	%1:	Logical axis number [-]	
		Axis number of the invalid slave axis	
Error type	1, Error message from NC program.		

ID 20468

Master and slave axis are of different axis type.			
Description	When selecting synchronous operation, it is detected that master and slave axes of one coupling pair have different axis types P-AXIS-00018. For further information see [PROG// Section: Synchronous operation]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only axes of the same axis type may be coupled. An attempt may have been made to couple linear axes and rotator axes. Check that the axes are correctly configured P-AXIS-00018.
Parameter	%1:	Logical axis number [-]	
		Axis number of master axis	
	%2:	Logical axis number [-]	
		Axis number of slave axis.	
	%3:	Current value [-]	
		Number of the coupling group with the invalid axes pair	
Error type	1, Error message from NC program.		

ID 20469

Master and slave axis are of different axis mode.			
Description	When selecting synchronous operation, it is detected that master and slave axes of one coupling pair do have different axis modes P-AXIS-.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only axes from the same axis mode may be coupled. Check that the axes are correctly configured P-AXIS-00015.
Parameter	%1:	Logical axis number [-]	
		Axis number of master axis	
	%2:	Logical axis number [-]	
		Axis number of slave axis.	
	%3:	Current value [-]	
		Number of the coupling group with the invalid axes pair	
Error type	1, Error message from NC program.		

ID 20470

G201 may not be enabled for slave axis selecting coupling mode.			
Description	When selecting synchronous operation, the system detects that one of the defined slave axes still is in active manual operation mode G201 (in parallel with interpolation). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G201 #ACHSE [**B**] : N100 #SET AX LINK [**1, B=X**] N110 #ENABLE AX LINK[**1**] : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G201 #AXIS [B] : N100 #SET AX LINK [1, B=X] **N105 G202 #AXIS [B]** N110 #ENABLE AX LINK[1] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Make sure that the manual operation mode for the slave axes of the activated coupling group was deselected with G202.
Parameter	%1:	Logical axis number [-]	
		Axis number of the invalid slave axis	
Error type	1, Error message from NC program.		

ID 20472

Coupling description may not be changed.			
Description	When defining a coupling group, the system detects that this coupling group is still currently active. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK[1,C=X] N30 #ENABLE AX LINK[1] : N100 **#SET AX LINK[1,C=Y]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK[1,C=X] N30 #ENABLE AX LINK[1] : **N90 #DISABLE AX LINK[1]** N100 #SET AX LINK[1,C=Y] : N1000 M30 For further information see [PROG// Section: Programming axis couplings]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The active access group must first be deactivated before a new definition can be made.
Parameter	%1:	Error value [-]	
		Number of the coupling group	

Error type	1, Error message from NC program.

ID 20475

Master axis can not be its own slave axis.			
Description	With the definition of a coupling group it is detected, that one of the master axis also is its own slave axis. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK[1,**X=X**] : N1000 M30 For further information see [PROG// Section: Programming axis couplings]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Master axis and slave axis must not be identical. Observe the restrictions regarding master/slave axes dependence[PROG//Section: Synchronous mode].
Parameter	%1:	Logical axis number [-]	
		Axis number of the invalid axis	
	%2:	Current value [-]	
		Index of the coupling pair	
	%3:	Current value [-]	
		Number of the coupling group	
Error type	1, Error message from NC program.		

ID 20476

A slave axis is not allowed to be programmed as slave axis repeatedly.			
Description	When defining a coupling group, it is detected that a slave axis is already programmed as slave axis in another coupling pair. This inter coupling pair relation is not permissible, because a slave axis may only be assigned to one master axis. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK[1,**C=X, C=Y, B=Y**] : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK[1,**C=X, B=Y**] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. A slave axis may only be assigned to one master axis. All additional coupling pairs with that slave axis have to be removed from coupling group. For further information see [PROG// Section: Synchronous mode].
Parameter	%1:	Logical axis number [-]	
		Logical number of slave axis	
Error type	1, Error message from NC program.		

ID 20477

Master axis can not be slave axis in different assignment and vice versa.			
Description	During the definition of a coupling group it is detected, that in different coupling pairs one of the axis is defined both as master axis and as slave axis. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK[1,**B**=X, C=**B**] : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #SET AX LINK[1,**B**=X, **C**=X] : N1000 M30 For further information see [PROG// Section: Programming axis couplings]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the same coupling group a master axis can not be defined as slave axis at the same time. Assign the "linked" slave axes to the common master axis.
Parameter	%1:	Logical axis number [-]	
		Axis number of the invalid axis	
Error type	1, Error message from NC program.		

ID 20478

Two identical axes names after #ACHSE.			
Description	When programming the manual mode axes with NC command #ACHSE[...], identical axes names are used several times. Syntax example: **Wrong:** N10 G200 #ACHSE[**X**,Y,**X**] **Correct:** N10 G200 #ACHSE[**X**,Y]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant axis name.
Parameter	%1:	Logical axis number [-]	
		Axis number of the multiple programmed axis	
Error type	1, Error message from NC program.		

ID 20480

Empty square brackets not allowed.			
Description	When programming some specific NC commands (e.g. #SET TIP), no NC command data were defined inside the following square brackets. Syntax example: **Wrong:** N10 #SET TIP**[ ]** X **Correct:** N10 #SET TIP**[1.0,1.5,2.0]** X		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Add the missing NC command data in the square brackets.
Error type	1, Error message from NC program.		

ID 20481

Too many parameters after #SET HR.			
Description	When programming the NC command #SET HR[…], only a limited number of parameters may be defined inside the square brackets. For further information see [PROG// Section: Handwheel mode] Syntax example: **Wrong:** N10 #SET HR**[0.1,0.2,0.5,2.0,2.5]** X **Correct:** N10 #SET HR**[0.1,0.2,0.5]** X		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program only the maximum permissible number of parameters inside the square brackets.
Parameter	%1:	Limit value [-]	
		Maximum number of handwheel parameters	
Error type	1, Error message from NC program.		

ID 20482

Hand wheel resolution has to be positive.			
Description	When programming the NC command #HANDWHEEL[...] or #SET HR[...], a negative resolution is defined in the square brackets. For further information see [PROG//Section:: Handwheel operating mode] Syntax example: **Wrong:** N10 #HANDWHEEL[AX=X RES1=0.1 **RES2=-0.2** RES3=0.5] **Correct:** N10 #HANDWHEEL[AX=X RES1=0.1 **RES2=0.2** RES3=0.5]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program only positive resolutions in the square brackets.
Parameter	%1:	Error value [-]	
		Negative resolution	
Error type	1, Error message from NC program.		

ID 20483

Comma in front of closing brackets not allowed.

Description	When programming the NC command #SET HR[…], no further resolution after a comma is defined inside the square brackets. For further information see [PROG// Section: Handwheel operating mode] Syntax example: **Wrong:** N10 #SET HR[0.1,0.2,] X **Correct:** N10 #SET HR[0.1,0.2,**0.5]** X or N10 #SET HR[0.1,0.2] X		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the missing resolution value or remove the redundant comma inside the square brackets.
Error type	1, Error message from NC program.		

ID 20485

Jog step size has to be positive.			
Description	When programming of the NC command #JOG INCR[...] or #SET JOG[.[...].], a negative distance is defined inside square brackets. For further information see [PROG// Section: Incremental jog operating mode] Syntax example: **Wrong:** N10 #JOG INCR[AX=X **DIST1=-0.1** FEED=1.0] **Correct:** N10 #JOG INCR[AX=X **DIST1=0.1** FEED=1.0]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program only positive step widths inside the square brackets.
Parameter	%1:	Error value [-]	
		Negative step width	
Error type	1, Error message from NC program.		

ID 20486

Incremental jog velocity must be programmed positive.			
Description	When programming of the NC command #JOG INCR[...] or #SET JOG[.[...].] a negative jog velocity is defined inside square brackets . For further information see [PROG// Section: Incremental jog operating mode] Syntax example: **Wrong:** N10 #JOG INCR[AX=X DIST1=0.1 **FEED=-1.0**] **Correct:** N10 #JOG INCR[AX=X DIST1=0.1 **FEED=1.0**]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only positive jog speeds may be programmed inside the square brackets.
Parameter	%1:	Error value [-]	
		Negative jog velocity	
Error type	1, Error message from NC program.		

ID 20487

Manual mode parameter is out of data format.			
Description	One of the manual operation parameters (increment or speed) programmed with the NC commands #JOG CONT[...], JOG INCR[...] or #SET TIP[...] or #SET JOG[.[...].] is out of value range. For further information, see: • [PROG//Section: Handwheel mode] • [PROG//Section: Jogging mode]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a permissible value for the incremental widths or speeds.
Parameter	%1:	Error value [-]	
		Incremental width or speed	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20488

Too many parameters after #SET JOG.			
Description	When programming the NC command #SET JOG[.[...].], only a limited number of parameters may be defined inside the square brackets . For further information see [PROG// Section: Jogging mode] Syntax example: **Wrong:** N10 #SET JOG[[0.1,1.0] [0.2,1.5] [0.3,2] [1,4]] X **Correct:** N10 #SET JOG[[0.1,1.0] [0.2,1.5] [0.3,2]] X		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program only the maximum permissible number of parameters inside the square brackets.
Parameter	%1:	Limit value [-]	
		Maximum number of jog parameters	
Error type	1, Error message from NC program.		

ID 20490

Too many parameters after #SET TIP.			
Description	When programming the NC command #SET TIP[…], only a limited number of parameters may be defined inside the square brackets. For further information see [PROG// Section: Continuous jog operating mode] Syntax example: **Wrong:** N10 #SET TIP[1.0,1.5,2.0,5.0,10.0] X **Correct:** N10 #SET TIP[1.0,1.5,2.0] X		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Program only the maximum permissible number of parameters inside the square brackets.
Parameter	%1:	Limit value [-]	
		Maximum number of continuous jog parameters	
Error type	1, Error message from NC program.		

ID 20491

Continuous jog velocity must be programmed positive.			
Description	When programming of the NC command #JOG CONT[...] or #SET TIP[…], a negative continuous jog velocity is defined inside square brackets . For further information see [PROG// Section: Continuous jog operating mode] Syntax example: **Wrong:** N10 #JOG CONT[AX=X FEED1=1.0 **FEED2=-1.5** FEED3=2.0] **Correct:** N10 #JOG CONT[AX=X FEED1=1.0 **FEED2=1.5** FEED3=2.0]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program only positive feed rates inside the square brackets.
Parameter	%1:	Error value [-]	
		Negative continuous jog velocity	
Error type	1, Error message from NC program.		

ID 20493

Negative manual mode offset must not be positive.			
Description	With the programming of the NC command #MANUAL LIMITS[...] or #SET OFFSET[…] the negative offset limit is defined inside square brackets with a positive value. For further information see [PROG// Section: Manual operation mode – Presetting offset limits] Syntax example: **Wrong:** N10 #MANUAL LIMITS [AX=X **NEGATIVE=5.0** POSITIVE=5.0] **Correct:** N10 #MANUAL LIMITS [AX=X **NEGATIVE=-5.0** POSITIVE=5.0]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The negative offset limit must be programmed with a negative value inside the square brackets.
Parameter	%1:	Error value [-]	
		Value of the negative offset limit	
Error type	1, Error message from NC program.		

ID 20494

Negative offset value is out of data format.

Description	When programming the NC command #MANUAL LIMITS[...] or #SET OFFSET[...], the negative offset limit exceeds the permissible data range. For further information see [PROG//Section: Manual operation mode – Specify offset limits]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a permissible value for the negative offset limit.
Parameter	%1:	Error value [-]	
		Value of the negative offset limit	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20495

Positive manual mode offset must not be negative.			
Description	When programming the NC command #MANUAL LIMITS[...] or #SET OFFSET[…], the positive offset limit is defined with a negative value inside the square brackets. For further information see [PROG// Section: Manual operation mode – Presetting offset limits] Syntax example: **Wrong:** N10 #MANUAL LIMITS [AX=X NEGATIVE=-5.0 **POSITIVE=-5.0**] **Correct:** N10 #MANUAL LIMITS [AX=X NEGATIVE=-5.0 **POSITIVE=5.0**]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The positive offset limit must be programmed with a positive value inside the square the square brackets.
Parameter	%1:	Error value [-]	
		Value of the positive offset limit	
Error type	1, Error message from NC program.		

ID 20496

Positive offset value is out of data format.			
Description	When programming the NC command #MANUAL LIMITS[...] or #SET OFFSET[...], the positive offset limit exceeds the permissible value range. For further information see [PROG//Section: Manual operation mode – Specify offset limits]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a permissible value for the positive offset limit.
Parameter	%1:	Error value [-]	
		Value of the positive offset limit	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	

Error type	1, Error message from NC program.

ID 20498

Positive and negative offset limits equal to 0.			
Description	When programming the NC command #SET OFFSET[…], the square brackets indicate the positive and the negative offset limits with zero (nil) value. For this reason no movement of the axis in manual operation mode is possible. For further information see [PROG// Section: Manual operation mode – Presetting offset limits] Syntax example: **Wrong:** N10 #SET OFFSET[**0.0, 0.0**] X **Correct:** N10 #SET OFFSET[**-5.0**, 0.0] X		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. At least one of the two offset limits has to be set to a permissible value unequal to zero (nil).
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20499

Syntax error in command for position request.			
Description	When programming the NC command #GET CMDPOS[..] or #SET IPO SOLLPOS[..], a syntax error (commas, equal signs, closing square brackets) was detected after the open square bracket. Syntax example: **Wrong:** #GET CMDPOS [P1=X P2=Y] or #GET CMDPOS [V.L.POS1 X] or #GET CMDPOS [P1=X, P2=Y, P3=Z **Correct:** #GET CMDPOS [P1=X, P2=Y]or #GET CMDPOS [V.L.POS1**=**X] or #GET CMDPOS [P1=X, P2=Y, P3=Z**]**		
Response	Class	2	Abort NC program processing.
Solution	Class	3	In the NC program, check and correct the command syntax of #GET CMDPOS[...] or #SET IPO SOLLPOS[...] regarding comma setting, equal sign, closing square brackets, etc. For further information see [PROG// Section: Requesting offset and setpoint values]
Error type	1, Error message from NC program.		

2.3.3 ID-range 20500-20749

ID 20500

Too many parameters in command for position request.			
Description	When programming the NC command #GET CMDPOS[...] or #SET IPO SOLLPOS[...], the maximum number of parameter assignments is exceeded inside the square brackets . For further information see [PROG// Section: Requesting offset and command values]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	In the NC program reduce the parameter assignments on the maximum number.
Parameter	%1:	Upper limit value [-]	
		Maximum number of parameter assignments	
Error type	1, Error message from NC program.		

ID 20506

The string to be defined is too long.			
Description			
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Limit value [-]	
	%2:	Current value [-]	
Error type	-		

ID 20509

Unknown NC command after #.			
Description	The programmed # command was not identified,because the syntax of the command contains a spelling mistake or the command is completely unknown or the corresponding (optional) function is generally not available in the system.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the syntax of the # command regarding the spelling and availability in the NC program.
Parameter	%1:	State [-]	
Error type	1, Error message from NC program.		

ID 20521

Programmed offset mode unknown for this axis.

Description	In the axis exchange command, an unknown offset mode is programmed for the axis in the system. For further information see [PROG//Section: Axis exchange commands]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command. Program a permissible shift mode.
Parameter	%1:	Logical axis number [-]	
		Number of the invalid axis	
Error type	1, Error message from NC program.		

ID 20524

Name of axis too long or starts with illegal character.			
Description	The check of an axis name programmed in a #-command detects that the name starts with a wrong character or that the axis name is too long. For an axis name the following first characters are permitted: X, Y, Z, A, B, C, U, V, W, Q.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only use in the #-command the permissible first characters respectively reduce the length of the name.
Error type	1, Error message from NC program.		

ID 20535

No axis requests during active tapping allowed.			
Description	During active tapping (G63), no axis exchange commands may be programmed. Example: **Wrong:** N10 G63 Z10 F1000 S100 **N20 #SET AX[X, 1, 0]** : N1000 M30 **Correct:** N10 G63 Z10 F1000 S100 **N20 G01 G90 X100 Y100 (Deselect G63)** N30 #SET AX[X, 1, 0] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Before the axis exchange command, tapping (G63) must be deselected.
Error type	1, Error message from NC program.		

ID 20545

An axis involved in synchronous operation can not be released.

Description	An axis programmed in the axis exchange command is currently used in synchronous operation mode and must not be released. Example: **Wrong:** N30 #SET AX LINK[1, B=**X**] N40 #ENABLE AX LINK[1] : N70 #PUT AX[**X**] : N80 #DISABLE AX LINK[1] : N1000 M30 **Correct:** N30 #SET AX LINK[1, B=**X**] N40 #ENABLE AX LINK[1] : **N70 #DISABLE AX LINK[1]** : N80 #PUT AX[**X**] : N1000 M30 For further information see [PROG// Section: Synchronous operation]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Before the axis exchange command, the synchronous operation must be deselected.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20546

Release of face turning axis not allowed during active diameter programming.			
Description	The axis programmed in the axis exchange command is currently used in diameter programming (G51) as facing axis P-AXIS-00015 and cannot be issued. Example: **Wrong:** N10 G51 X80 **N20 #PUT AX[X]** : N100 G52 : N1000 M30 **Correct:** N10 G51 X80 : **N100 G52** **N110 #PUT AX[X]** : N1000 M30		
Response	Class	1	Abort NC program processing.
Solution	Class	1	Check and modify the program flow sequence. Before the axis exchange command, the diameter programming (G51) must be deselected.
Parameter	%1:	Logical axis number [-]	

Error type	1, Error message from NC program.

ID 20547

An axis used in manual mode can not be released.			
Description	The axis programmed in the axis exchange command is currently used in manual operation mode and must not be released. Example: **Wrong:** N10 G201 #AXIS[X] **N20 #PUT AX[X]** : N100 G202 : N1000 M30 **Correct:** N10 G201 #AXIS[X] : **N100 G202** **N110 #PUT AX[X]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Before specifying the axis exchange command, manual operation mode must be deselected.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20551

Name of axis too long or starts with illegal character.			
Description	The check of an axis name programmed in a #-command detects that the name starts with a wrong character or that the axis name is too long. For an axis name the following first characters are permitted: X, Y, Z, A, B, C, U, V, W, Q.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only use in the #-command the permissible first characters respectively reduce the length of the name.
Error type	1, Error message from NC program.		

ID 20566

In NC block #COMMENT BEGIN has to be programmed exclusively.

Description	After the #COMMENT BEGIN command, no further NC commands are permitted in the same NC block. Example: **Wrong:** N10 #COMMENT BEGIN **X100 F1000 G01 F100** : N30 #COMMENT END : N1000 M30 **Correct:** **N10 #COMMENT BEGIN** N20 **X100 F1000 G01 F100** : N30 #COMMENT END : N1000 M30 For further information see [PROG// Section: Cross-block comments]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Move the invalid NC commands to the following NC block. Note: Other NC commands in the same NC block are only permissible with #ACHSE, #PSET and #PRESET.
Error type	1, Error message from NC program.		

ID 20567

#COMMENT END needs an accompanied #COMMENT BEGIN.			
Description	A #COMMENT END was found in the NC program without a corresponding #COMMENT BEGIN. Example: **Wrong:** N20 X100 F1000 G01 N30 **#COMMENT END** : N1000 M30 **Correct:** N10 **#COMMENT BEGIN** N20 X100 F1000 G01 N30 **#COMMENT END** : N1000 M30 For further information see [PROG// Section: Cross-block comments]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Add missing #COMMENT BEGIN or remove redundant #COMMENT END. Note: Even with nested comments, both #COMMENT BEGIN / #COMMENT END must always be programmed! N10 **#COMMENT BEGIN** : N100 **#COMMENT BEGIN** : N200 **#COMMENT END** N500 **#COMMENT END** : N1000 M30

Error type	1, Error message from NC program.

ID 20568

Comment block contains end of file.			
Description	In the comment block after #COMMENT BEGIN, the end of the file is reached when searching for #COMMENT END. **Wrong:** N20 X100 F1000 G01 N50 **#COMMENT BEGIN** : : N1000 M30 **<End of File>** **Correct:** N20 X100 F1000 G01 N50 #COMMENT BEGIN : **N500 #COMMENT END** : N1000 M30 **<End of File>** For further information see [PROG// Section: Cross-block comments]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Add missing #COMMENT END in an NC block before M30 or remove redundant #COMMENT BEGIN.
Error type	1, Error message from NC program.		

ID 20569

In NC block #COMMENT END has to be programmed exclusively.			
Description	After the #COMMENT END command, no further NC commands are permitted in the same NC block. Example: **Wrong:** N10 #COMMENT BEGIN : N30 #COMMENT END **X100 F1000 G01 F100** : N1000 M30 **Correct:** N10 #COMMENT BEGIN : **N30 #COMMENT END** **N40 X100 F1000 G01 F100** : N1000 M30 For further information see [PROG// Section: Cross-block comments]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Move the invalid NC commands to the following NC block. Note: Further NC commands are only permitted in the same NC block with #ACHSE, #PSET and #PRESET.
Error type	1, Error message from NC program.		

ID 20570

PDU received from tool management contains wrong tool number.			
Description	An external tool management provides the corresponding tool data in segmented data packages (PDU's) based on the request from the NC program (D word, #TOOL DATA). On this occasion the plausibility check in the NC channel detects, that the tool numbers of this requested data and the received data are not identical.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify processing and supply of requested data in external tool management.
Parameter	%1:	Error value [-]	
		Number of the received tool	
	%2:	Expected value [-]	
		Number of the requested tool	
Error type	1, Error message from NC program.		

ID 20571

PDU received from tool management contains invalid parameter index.			
Description	An external tool management provides the corresponding tool data in segmented data packages (PDUs) based on the request from the NC program (D word, #TOOL DATA). On this occasion the plausibility check of the received data in the NC channel detects, that the segmentation index of the tool parameters is not correct.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify processing and supply of requested data in external tool management.
Parameter	%1:	Error value [-]	
		Inadmissible parameter index	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20573

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20574

Division by 0.

Description	The calculation of a mathematical expression with the combined assignment operator /= detects, that the divisor is zero (nil). Example: **Wrong:** N10 **P1=0** : N500 P2**/=P1** : N1000 M30 **Correct:** N10 P1=0 : **N495 $IF P1 != 0** N500 P2**/=P1** N505 $ENDIF : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure by a $IF-construct, that the division is only executed for divisor values unequal to zero (nil).
Parameter	%1:	Error value [-]	
		Value of divisor	
Error type	1, Error message from NC program.		

ID 20576

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20577

No longitudinal turning axis configured for cutter radius compensation.			
Description	When the tool offsets are accepted with active cutter radius compensation (CRC) for turning tools, the system detects that neither of the plane axes is configured as face turning axis P-AXIS-00015.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	For lathes in the axis parameter list, one of the plane axes (first or second main axis) must be configured as a face turning axis P-AXIS-00015.
Parameter	%1:	Current value [-]	
		Axis mode of the axis	
	%2:	Expected value [-]	
		Number of the "face turning axis"	
	%3:	Logical axis number [-]	
	%4:	Logical axis number[-]	
Error type	1, Error message from NC program.		

ID 20578

No longitudinal turning axis configured for cutter radius compensation.			
Description	When tool offsets are accepted with active cutter radius compensation (CRC) for turning tools, the system detects that neither of the plane axes is configured as a longitudinal turning axis P-AXIS-00015.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	For lathes in the axis parameter list, one of the plane axes (first or second main axis) in the axis parameter list must be configured as a longitudinal turning axis P-AXIS-00015
Parameter	%1:	Current value [-]	
		Axis mode of the axis	
	%2:	Expected value [-]	
		Number of the "longitudinal turning axis"	
	%3:	Logical axis number [-]	
	%4:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20581

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20582

Value following G159 is out of data format.			
Description	The numerical value programmed with G159=<expr> (zero offset group) is outside the permitted value range. [PROG//Section: Extended zero offset (G159)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a zero offset group with a valid value.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20583

Value following G160 is out of data format.

Description	The numerical value programmed with G160=<expr> (zero offset group) is outside the permitted value range. [PROG//Section – Enable/disable zero offsets axis-specific (G160)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a zero offset group with a valid value.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20584

Value following G160 is out of range of permissible values.			
Description	The numerical value programmed with G160=<expr> represents an index (data record) in the zero offset table. The size of this table depends on the respective application. Here the programmed index exceeds the maximum permissible index. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G160=500** X1 Y1 Z0 N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G160=7** X1 Y1 Z0 N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the index present in the zero offset table P-ZERO-00003.
Parameter	%1:	Limit value [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20585

Default index is out of range of permissible values.			
Description	During the interpretation at start-up or update of modified data, the plausibility check of the zero point data list detects that the default index P-ZERO-00002 exceeds the maximum permissible index value.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and change the list of zero offsets.
Parameter	%1:	Limit value [-]	
	%2:	Error value [-]	

Error type	-

ID 20587

This assignment or linking in not allowed with .ALL variables.			
Description	When zero point offsets via V.G.NP[..]ALL are linked, do not use multiplication or division operations. Syntax example: **Wrong:** N10 V.G.NP[0].ALL ***=** V.G.NP[1].ALL N10 V.G.NP[0].ALL = V.G.NP[0].ALL**/6**		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. See also [PROG//Section: Adding and subtracting offsets]
Error type	1, Error message from NC program.		

ID 20588 / 20589

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20590

Nesting degree of conditional instructions too high.

Description	When programming nested conditional instructions (#IF/ #ELSE/ #ENDIF), the maximum permissible nesting depth is exceeded. Syntax example: **Wrong:** **#IF** P1==10 N10 G01 X10 Y10 F100 : **#IF** : **#IF** : #ELSE : **#IF** : #ELSE : #ENDIF #ENDIF : : #ELSE : #ENDIF : #ENDIF :		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the nesting depth of the conditional instructions (#IF/#ELSE/#ENDIF). Simplify NC program sequence.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20591

Missing corresponding #IF for current #ELSE.

Description	A conditional interpreter instruction was programmed with incomplete syntax. A #ELSE can only be programmed in combination with #IF/#ENDIF. Example: **Wrong:** N10 G00 X0 Y0 Z0 **#ELSE** : N1000 M30 **Correct:** N01 P1=0 **#IF** P1==1 N10 G00 X0 Y0 Z0 **#ELSE** N25 G01 X100 Y0 Z0 F1000 **#ENDIF** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the #IF/#ENDIF instruction or delete the #ELSE.
Error type	1, Error message from NC program.		

ID 20592

Unexpected #ENDIF. No conditional instruction open.			
Description	A conditional interpreter instruction was programmed with incomplete syntax. A #ENDIF can only be programmed in combination with a previous #IF. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=0 N20 G01 X100 F10000 **#ENDIF** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=0 **#IF** P1==0 N20 G01 X100 F10000 **#ENDIF** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the #IF instruction or delete the #ENDIF.
Error type	1, Error message from NC program.		

ID 20593

Conditional instruction has to be the first command in NC-block.

Description	The commands #IF/#ELSE/#ENDIF have to be programmed without a block numberdirectly at the beginning of a NC block.
	
	Syntax example:
	`#IF P1 == TRUE`
	`N10 X100`
	`#ELSE`
	`N20 X200 Y200 Z100`
	`#ENDIF`
	`:`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Ensure, that #IF/#ELSE/#ENDIF are programmed at the beginning of a NC block.
Error type	1, Error message from NC program.		

ID 20595

The block count for block search is inside invalid control block branch.			
Description	The block search is assigned a block count which is positioned in an invalid part of a control block branch in the NC program (e.g. an invalid $ELSE branch).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Assign a block count that is positioned in a valid part of a control block structure.
Parameter	%1:	Current value [-]	
		NC block number at the assigned block count	
	%2:	Current value [-]	
		Assigned block count	
Error type	1, Error message from NC program.		

ID 20596 - 20600

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20601

Received PDU not permissible at active state of block search operation.			
Description	The operator commanded a block search (SV) with an action that is not permissible in the current state since no block search is active. The assigned actions can be: Continue motion at block search position Abort program execution Continue block search		
Response	Class	1	Block search processing is continued.
Solution	Class	1	Only execute the actions above if the block search already is enabled.
Error type	3, Error in the communication.		

ID 20602

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20603

Received unknown PDU from user interface during active block search.			
Description	The block search has been instructed by the operator with an undefined action, which can not be executed.		
Response	Class	2	Block search processing is continued.
Solution	Class	1	Check,why the operator interface has assigned the unknown action.
Parameter	%1:	Current value [-]	
		Identification number of the unknown action	
Error type	3, Error in the communication.		

ID 20604

Current value is out of data format.			
Description	The current value programmed with the #PSET command for an axis exceeds the permissible numerical range. Syntax example: `N10 #PSET X<curr_value>`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the actual value for the corresponding axis.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20605

When calculating the actual value offset, the data format is violated.			
Description	The current value programmed for an axis with the #PSET command causes an exceeding of the permissible data format during the control internal processing. Syntax example: `N10 #PSET X<acr_value>`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the actual value for the corresponding axis.

Parameter	%1:	Logical axis number [-]
	%2:	Error value [-]
	%3:	Lower limit value [-]
	%4:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 20606

Excluding the actual value offset the data format is violated.			
Description	The current value programmed for an axis with the #PSET command causes an exceeding of the permissible data format during the control internal processing. Syntax example: `N10 #PSET X<acr_value>`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the actual value for the corresponding axis.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20607

#PSET/PRESET can not be used in combination with path preparatory function.			
Description	The NC commands #PSET and #PRESET may not be used in the same NC block with NC commands from the group of so-called path conditions since they also switch over the meaning of the programmed axis coordinates. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G01** F1000 **#PSET X200** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 **N20 G01 X200 F1000** **N30 #PSET X200** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Move the NC command from the path preparatory functions group into a separate NC block.

Error type	1, Error message from NC program.

ID 20608

Setting of current value not allowed during active tool radius compensation.			
Description	As long as the TRC function (tool radius compensation) is enabled, no current value setting ((#PSET, #PRESET) may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G41** X100 F1000 **N50 #PSET X200** : N100 G40 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G41 X100 F1000 : **N100 G40** **N110 #PSET X200** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Deselect TRC (G40) before setting the current value.
Error type	1, Error message from NC program.		

ID 20610

Setting of actual value not allowed during active diameter programming.			
Description	While diameter programming is enabled (G51), no current value setting (#PSET, #PRESET) can be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G51** X100 **N50 #PSET X200** : N100 G52 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G51 X100 : **N100 G52** **N110 #PSET X200** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before position preset, deselect diameter programming (G52).
Error type	1, Error message from NC program.		

ID 20612

Setting of actual value not allowed during active mirroring.			
Description	As long as Mirroring is enabled (G21 – G23), no current value setting (#PSET, #PRESET) can be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G21** **N50 #PSET X200** : N100 G20 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G21 : **N100 G20** **N110 #PSET X200** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before position preset, deselect an active mirroring (G20).
Error type	1, Error message from NC program.		

ID 20613

Missing quotation mark after macro name.			
Description	When calling a macro, after the macro name the closing quotation mark is missing. Syntax example: **Wrong:** N10 **"MACRONAME** **Correct:** N10 **"MACRONAME"** For further information see [PROG// Section: Macros]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Macronames must always be enclosed in inverted commas.
Error type	1, Error message from NC program.		

ID 20614

Macro name too long.			
Description	When defining or calling a macro, it is detected that the macro name exceeds the maximum number of permissible characters. Syntax example: **Wrong:** N10 **"THIS_MACRONAME_IS_TOO_LONG"** = "X100 G01 F1000" For further information see [PROG// Section: Macros]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the length of the macro name.

Parameter	%1:	Limit value [-]
Error type	1, Error message from NC program.	

ID 20615

Missing quotation mark after macro NC-code.			

Description	When defining a macro, the closing quotation mark is missing after the macro NC code.
	`Syntax example:`
	`Wrong:`
	`N10 "MACRONAME" = "X100 G01 F1000`
	
	`Correct:`
	`N10 "MACRONAME" = "X100 G01 F1000"`
	
	For further information see [PROG// Section: Macros]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Macro NC code must always be enclosed in inverted commas.
Error type	1, Error message from NC program.		

ID 20616

Nesting depth not completed at macro definition.			

Description	`The nesting of macros in a macro definition is programmed in an incomplete or incorrect syntax.`
	Syntax example:
	`Wrong:`
	`N10 "MACRO1" = "X100"`
	`N20 "MACRO2" = "Y200"`
	`N30 "MACRO3" = "G01 F1000 \"MACRO1\" "MACRO2\"`
	`:`
	
	`Correct:`
	`N10 "MACRO1" = "X100"`
	`N20 "MACRO2" = "Y200"`
	`N30 "MACRO3" = "G01 F1000 \"MACRO1\" \"MACRO2\" "`
	`:`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Each nested macro name has to be included in \" (Backslash – quotation mark). Ensure that the quotation mark after the macro NC code is programmed! [PROG// Section: Nesting macros]
Error type	1, Error message from NC program.		

ID 20617

Macro NC-code is too long.

Description	When defining a macro, it is detected that the macro NC code exceeds the maximum number of permissible characters.
	Syntax example:
	`Wrong:`
	`N10 "MACRO1" = "X100 G01 F1000…(MACRO_NC_CODE_TOO_LONG)"`
	For further information see [PROG// Section: Macros]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Shorten macro NC code.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20618

Overflow of string store.			
Description	The memory capacity for macro definitions in the NC program is reached. The maximum total number of macros is formed from the sum of reserved macros in the channel parameter lists and the macros defined in the NC program. For further information see [PROG// Section: Macros]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove macro definitions in the NC program or, if possible, transfer them as predefined macros to the channel parameter list P-CHAN-00085, if there is still some free capacity.
Parameter	%1:	Limit value [-]	
		Maximum number of macros in NC channel	
	%2:	Current value [-]	
		Number of macros, which can be defined in NC program	
	%3:	Current value [-]	
		Number of macros, which can be defined in channel parameter list	
Error type	1, Error message from NC program.		

ID 20619

Unknown macro name. Macro is not defined.			
Description	The programmed macro is unknown. It was neither defined in the NC program nor in the channel parameter list.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before the call, the macro must have been defined either in the NC program according to [PROG//Section: Macros] or in the channel parameter list (P-CHAN-00085, P-CHAN-00062).
Error type	1, Error message from NC program.		

ID 20620

Missing quotation mark before macro NC-code.

Description	When defining a macro, the opening quotation mark is missing before the macro NC code . Syntax example: **Wrong:** N10 "MACRONAME" = **X100 G01 F1000"** **Correct:** N10 "MACRONAME" = **"X100 G01 F1000"** For further information see [PROG// Section: Macros]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Macro NC code must always be enclosed in inverted commas.
Error type	1, Error message from NC program.		

ID 20621

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20622

Macro NC-code between quotation marks is missing.			
Description	In NC block an "empty" macro is programmed, this means between the quotation marks on the right side of the assignment no NC code is programmed. Example: N10 G00 X0 Y0 Z0 : **N100 "MACRO_1" = ""** : N1000 M30 For further information see [PROG// Section: Macros]		
Response	Class	1	Continue NC program processing. Warning is only used for diagnosis.
Solution	Class	1	Check and modify NC program. Modify macro definition.
Error type	1, Error message from NC program.		

ID 20624

Channel parameters: No NC-code was assigned to the macro name.			
Description	During start-up, the channel parameter check determines that an empty string has been assigned to a macro name P-CHAN-00085 as NC code P-CHAN-00062. For further information see [PROG// Section: Macros]		
Response	Class	2	Start-up of the control is continued.
Solution	Class	1	During start-up in case of conflict the empty string is stored in the channel data and the start-up is continued. Complete the invalid macro definition or remove it and repeat start-up.

Parameter	%1:	Current value [-]
		Index of the incomplete macro definition in the channel parameter list
Error type	2, Error message by data transfer from parameter list into control device.	

ID 20625

Mathematical expression not correctly terminated within macro NC-code.			
Description	During the resolution of mathematical terms within macros, an inadmissible character has been detected at the end of a term. Syntax example: **Wrong:** N10 "MACRO1"="5*12]" : N100 X"MACRO1" (error output in this line!) : For further information see [PROG// Section: Macros]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the mathematical terms of the used macros for inadmissible characters and remove this characters.
Error type	1, Error message from NC program.		

ID 20626

Overwriting of macro NC code is not allowed.			
Description	An already defined macro is to be written again in the NC program. This option is not allowed because of the version-specific configuration. Example: **Wrong:** **N10 "MACRO_1" = "** G00 X0 Y0 Z0" **N15 "MACRO_2" = "** G00 X10 Y20 Z30" **N20 "MACRO_1" = "** G01 F2000 X0 Y0 Z0" : N1000 M30 **Correct:** **N10 "MACRO_1" = "** G00 X0 Y0 Z0" **N15 "MACRO_2" = "** G00 X10 Y20 Z30" **N20 "MACRO_3" = "** G01 F2000 X0 Y0 Z0" : N1000 M30 For further information see [PROG// Section: Macros]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove all additional macros with the same name or create the macro with a new name.
Error type	1, Error message from NC program.		

ID 20629

Missing quotation mark after macro name at definition.

Description	When defining a macro, the closing quotation mark is missing after the macro name.
	Syntax example:
	`Wrong:`
	`N10 "MACRONAME = "X100 G01 F1000"`
	`Correct:`
	`N10 "MACRONAME" = "X100 G01 F1000"`
	For further information see [PROG// Section: Macros]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Macronames must always be enclosed in inverted commas.
Error type	1, Error message from NC program.		

ID 20630

Double-programmed actual value.			
Description	The NC command #PSET/ #PRESET is programmed several times in the NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the actual value programming in the NC block. Remove excess #PSET/ #PRESET.
Error type	1, Error message from NC program.		

ID 20633

Position of face turning axis too small with G96: Spindle speed is limited.			
Description	The calculated starting speed of the spindle is not permissible, because the distance between the tool tip and the turning centre point at G96 (constant cutting speed) is too close and the maximum spindle speed predefined by G196 or P-AXIS-00212 is exceeded.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	The starting speed is limited to the maximum spindle speed defined by G196 or P-AXIS-00212. To avoid this warning, at selection of G96, the tool tip should not be positioned too close to the centre point of turning.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [0.1 µm or 0.0001°]	
		Distance of the tool tip to the turning centre point	
Error type	1, Error message from NC program.		

ID 20634

No face turning axis is configured in the actual plane at G96.			
Description	With the selection of constant cutting speed (G96) it is detected, that not any of the plane axes is configured as face turning axis P-AXIS-00015.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	For lathes in the axis parameter list, one of the plane axes (first or second main axis) must be configured as face turning axis P-AXIS-00015.

Error type	1, Error message from NC program.		

ID 20635

Tapping not allowed during active feedrate revolution.			
Description	If tapping is selected with G63), it is detected that feedrate per revolution (G95) is still enabled.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change NC program. Before selecting thread tapping with G63, explicitly deselect the active feedrate per revolution (G95) with G94.
Error type	1, Error message from NC program.		

ID 20636

Tapping not allowed during active constant cutting speed.			
Description	If tapping is selected with G63, it is detected that constant cutting speed (G96) is still enabled.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change NC program. Before selecting thread tapping with G63, explicitly deselect the constant cutting speed (G96) with G97.
Error type	1, Error message from NC program.		

ID 20637

Selection of tapping and G196 in the same NC-block not allowed.			
Description	If tapping is selected with G63, it is detected that in the same NC block G196 is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove G196 from the NC block by selecting tapping (G63).
Error type	1, Error message from NC program.		

ID 20638

Gear switching not permitted during active constant cutting speed.			
Description	When selecting a new gear stage with G112 or M40 - M45, it is detected that constant cutting speed (G96) is still active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change NC program. Before selecting a new gear speed with G112or M40 – M45, explicitly deselect the constant cutting speed (G96) with G97.
Error type	1, Error message from NC program.		

ID 20639

Spindle stop not allowed during active constant cutting speed.			
Description	When stopping the spindle with M05, the constant cutting speed (G96) is still enabled.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change NC program. Before selecting spindle stop with M05, explicitly deselect the constant cutting speed (G96) with G97.
Error type	1, Error message from NC program.		

ID 20640

Manual mode not allowed during active constant cutting speed.			
Description	When selecting manual operation with G200 or G201, it is detected that constant cutting speed (G96) is still enabled.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before selection of manual mode with G200 or G201, deselect explicitly the active constant cutting speed (G96) with G97.
Error type	1, Error message from NC program.		

ID 20641

Release of face turning axis not allowed during active constant cutting speed.			
Description	An axis programmed in an axis exchange command is currently used with the active constant cutting speed enabled (G96) as a face turning axis P-AXIS-00015 and cannot be released. Example: **Wrong:** N10 G96 S63 **N20 #PUT AX[X]** : N100 G97 : N1000 M30 **Correct:** N10 G96 S63 : **N100 G97** **N110 #PUT AX[X]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Before the axis exchange command, deselect active constant cutting speed (G96).
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20642

M19 not allowed during active constant cutting speed

Description	With the selection of spindle positioning (M19, it is detected that constant cutting speed (G96) is still active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change NC program. Before selecting spindle positioning with M19, explicitly deselect the constant cutting speed (G96) with G97.
Error type	1, Error message from NC program.		

ID 20643

M19 and G196 in the same NC-block not allowed.			
Description	When selecting the spindle positioning (M19), it is detected that in the same NC block G196 is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove selection of spindle positioning (M19) from the NC block with G196 .
Error type	1, Error message from NC program..		

ID 20644

The macro contains an exclusive NC command.			

Description	If a macro contains an exclusive NC command (e.g. # command), no further NC commands may be programmed in the same NC block after the macro call.
	Example:
	Wrong:
	`%dummy`
	`N10 "MACRO"="#FLUSH WAIT" (macro with excl. command)`
	`N20 "MACRO" G01 X10 F1000`
	`:`
	`M30`
	Correct:
	`%kont`
	`N10 "MACRO" = "#FLUSH WAIT" (macro with excl. command)`
	`N20 "MACRO"`
	`N30 G01 X10 F1000`
	`:`
	`M30`
	For further information see [PROG// Section: Macros]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Move the NC commands after the macro call to a separate NC block.
Error type	1, Error message from NC program.		

ID 20648

Axis specific acceleration weighting is programmed negative.

Description	One of the axis-specific weight values programmed with G130 is negative. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G130 X70 **Y-60** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G130 X70 **Y60** : N1000 M30		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programming weighting value with a meaningful value greater than or equal to zero (nil).
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
	%3:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20650

Global acceleration weighting is programmed negative.			
Description	The global acceleration weighting value programmed with G131 is negative. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G131 = **-60** (alternatively: G131 -60) : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G131 = **60** (alternatively: G131 60) : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	In the event of a conflict, the acceleration weighting is automatically set to 100% and NC program processing is continued. Before the next program start, a meaningful value greater than zero (nil) should be programmed.
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
		Automatically corrected value of global acceleration weighting	
Error type	1, Error message from NC program.		

ID 20651

NC command is not allowed during active TRC.

Description	With active tool radius compensation (G41/G42) no function selection or deselection is possible which leads to flushing of CNC channel. These are, for example, the following CNC commands / functions: • #FLUSH, #FLUSH WAIT • Check the programming of the D and/or T words with set F_AW_D in P-CHAN-00271, as well as P-CHAN-00014 and P-CHAN-00106. • #CHANNEL INIT, #SET DEC LR SOLL • #GET CMDPOS, #SET IPO SOLLPOS • #CS ON[], CS OFF • #TRAFO ON/OFF • #OTC OFF [FCT-C20//section: Command syntax] • G200 • Read synchronous V.E. variables • Read V.A. variables that require the NC channel to be flushed • Functionality of measurement The Exception list of commands with active TRC/SRK lists the commands that are not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Deselect tool radius compensation with G40 before using NC commands that result in flushing of the CNC channel.
Error type	1, Error message from NC program.		

ID 20652

Double-programmed pitch.			
Description	In the same NC block, the pitch was programmed several times during thread cutting (G33). Example: **Wrong:** N10 G74 S1 N20 T1 M03 S400 N30 G00 X72 Z105 N40 G33 Z48 **K1 I2** : N1000 M30 **Correct:** N10 G74 S1 N20 T1 M03 S400 N30 G00 X72 Z105 N40 G33 Z48 **K1** (Assumption: Z is longitudinal axis) : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant pitch.
Error type	1, Error message from NC program.		

ID 20653

Pitch is zero or not programmed.

Description	With thread cutting (G33) the pitch is programmed with zero (nil) or missing. Example: **Wrong:** N10 G74 S1 N20 T1 M03 S400 N30 G00 X72 Z105 N40 G33 Z48 **K0** : N1000 M30 **Correct:** N10 G74 S1 N20 T1 M03 S400 N30 G00 X72 Z105 N40 G33 Z48 **K1** (Assumption: Z is longitudinal axis) : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the missing or invalid pitch with a correct value.
Error type	1, Error message from NC program.		

ID 20654

Pitch is out of data format.			
Description	When thread cutting (G33), the pitch converted into an internal unit exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Programmed pitch must be within the permissible value range.
Parameter	%1:	Current value [-]	
		Programmed pitch	
	%2:	Error value [1µm/s or 0.001°/s]	
		Converted pitch	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20657

Kinematic type unknown.			
Description	When using NC commands from the field of 5-axis processing (ECS, CAX, TOOL ORI CS…), the system detected that the currently used kinematic transformation is unknown. The kinematic transformation to be used is defined as the default value in the channel parameters (P-CHAN-00032) or can also be defined with the command #KIN ID[...] in the NC program [PROG].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Define the kinematic transformation (KIN-ID) required for the machine kinematic used and select it as default value (P-CHAN-00032) or in the NC program using #KIN ID[...].

Parameter	%1:	Error value [-]
		ID of the selected kinematic
	%2:	Error value [-]
		Type of the selected kinematic
Error type	1, Error message from NC program.	

ID 20658

Not enough axes for RTCP/TLC.			
Description	Depending on the specific machine kinematic in NC channel, a minimum number of axes is necessary at selection of kinematic transformation (#TRAFO ON). This axes have to be in a fixed order without any gap.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust the order of the channel axes. Make sure that no wrong kinematic ID (#KIN ID[...]) has been selected, and that no axes required for the kinematic transformation have been removed from the channel by axis exchange commands.
Parameter	%1:	Error value [-]	
		Number of axes in NC channel up to the first gap	
	%2:	Current value [-]	
		Number of necessary axes for the enabled machine kinematics without a gap	
Error type	1, Error message from NC program.		

ID 20659

TLC offset is out of data format.			
Description	When programming the NC command #TLC ON/OFF (tool length compensation), the change into the machine coordinate system via #MCS... or during the supply of new axis positions in the NC channel, the TLC offset used during the internal calculation exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	In the NC program, execute the TLC offset calculation only for axis positions which are well within the permissible numerical range.
Parameter	%1:	Error value [-]	
		Calculated TLC offset	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20660

The first three primary axes are necessary for the Cartesian transformation.			
Description	When programming a Cartesian transformation with #ACS or #CS the system detects that not all main axes (first three axes) are completely present in the NC channel.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and adjust the order of the main axes in the NC channel. Make sure that no axes required for the Cartesian transformation were sent out of the channel by axis exchange commands or that the main axes are correctly configured (P-CHAN-00006, P-CHAN-00035, P-CHAN-00003).
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20661

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20662

TOOL ORI CS needs a rotary axis.			
Description	With the NC command #TOOL ORI CS (tool orientation perpendicular to the rotated machining plane), the system detects that a rotary axis necessary for the tool orientation is not available in NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the order of the main axes and rotary axes in the NC channel and adjust them according to the basic kinematic. As a rule at least two rotary axes are necessary for automatic tool orientation. For further information see [PROG//Section – Orienting tool] Ensure that no axes required for alignment were sent out of the channel by axis exchange commands or are incorrectly arranged in the axis order (e.g. gaps).
Parameter	%1:	Current value [-]	
		Place or index in axis order where a rotary axis is expected.	
Error type	1, Error message from NC program.		

ID 20663

G-function not allowed during active (A)CS.

Description	Some specific G functions may not be programmed with an active Cartesian transformation. Example: **Wrong:** N10 G00 X0 Y0 Z0 **N20 #CS ON[10,20,0,0,0,45]** **N30 G98 X-500 Y-800** : N100 #CS OFF : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #CS ON[10,20,0,0,0,45] : **N100 #CS OFF** **N110 G98 X-500 Y-800** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the concerned G functions only when the Cartesian transformation has been deselected (#CS OFF).
Parameter	%1:	Error value [-]	
		Number of invalid G function	
Error type	Abort NC program processing.		

ID 20664

G-function not allowed during active RTCP/TLC.			
Description	Some specific G functions may not be programmed with an active kinematic transformation. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] **N30 #TRAFO ON** **N40 G98 X-500 Y-800** : N100 #TRAFO OFF : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] N30 #TRAFO ON : **N100 #TRAFO OFF** **N110 G98 X-500 Y-800** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the concerned G functions only when the kinematic transformation has been deselected (#TRAFO OFF).
Parameter	%1:	Error value [-]	
		Number of the invalid G function	
Error type	Abort NC program processing.		

ID 20665

Coordinate from RTCP forward transformation is out of data format.			
Description	When a kinematic transformation is selected (#TRAFO ON), the system detects that, after the transformation, an axis position exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the machine axis positions before selection of the coordinate transformation. In the NC program, move to a smaller axis position before selecting this option. Check whether the correct kinematic type is entered in the channel parameters P-CHAN-00032 or in the NC program (#KIN ID).
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
	%4:	Logical axis number [-]	
Error type	1Abort NC program processing.		

ID 20666

Active TLC can only be deselected with #TLC OFF.			
Description	If active tool length compensation (TLC) is already active, #TLC ON[...] is programmed again. Example: **Wrong:** N10 G00 X0 Y0 Z0 **N20 #TLC ON[100]** : **N100 #TLC ON[-50]** : N990 #TLC OFF N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #TLC ON[100] : **N90 #TLC OFF** : N100 #TLC ON[-50] : N990 #TLC OFF N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In order to change tool length compensation, first deselect the active TLC using #TLC OFF.
Error type	Abort NC program processing.		

ID 20668

Deselection with #TLC OFF has no effect.

Description	The NC command #TLC OFF is programmed in the NC block, although the tool length compensation (TLC) is not enabled. Example: **Wrong:** N10 G00 X0 Y0 Z0 : **N990 #TLC OFF** N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Remove NC command #TLC OFF.
Error type	Abort NC program processing.		

ID 20669

Active RTCP can only be deselected with #RTCP OFF.			
Description	With active kinematic transformation (#TRAFO ON), #TLC ON[...] or #TLC OFF are programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] N30 #TRAFO ON **N20 #TLC ON[100]** : **N100 #TLC OFF** : N990 #TRAFO OFF N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] **N30 #TRAFO ON** : **N990 #TRAFO OFF** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Kinematic transformation and tool length compensation are mutually exclusive. #TLC ON[...] or #TLC OFF shall be removed from the active kinematic transformation area. Use of the TLC function see also[FCT-C2].
Error type	Abort NC program processing.		

ID 20671

Selecting TLC is not permitted during active tool radius compensation.

Description	If active tool radius compensation (TRC) is already active, tool length compensation (TLC) is selected with #TLC ON[...].
	Example:
	Wrong:
	`N10 G41`
	`N20 #TLC ON [25]`
	`:`
	`N100 #TLC OFF`
	`N110 G40`
	`:`
	`N1000 M30`
	Correct:
	`N10 #TLC ON [25]`
	`N20 G41`
	`:`
	`N100 G40`
	`N110 #TLC OFF`
	`:`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program sequence. Deselect the TRC before selecting tool length compensation (#TLC ON[.]) or select only afterwards (G41/G42).
Error type	1, Error message from NC program.		

ID 20675

Syntax error at #(A)CS [...].			

Description	When programming the NC command #ACS ON[...] or #CS ON[...], a syntax error (commas, number of parameters) was detected after the open square bracket .
	Syntax examples:
	`Wrong:`
	`#CS ON [10,20,0,0,0 45]or`
	`#CS ON [10,20,0,0,0]or`
	`Correct:`
	`#CS ON [10,20,0,0,0,45]or`
	`#CS ON [10,20,0,0,0.45]or`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the command syntax of #ACS ON[...] or #CS ON[...] in the NC program with regard to decimal point setting, number of parameters, closing square brackets, etc. For further information see [PROG// Section: Defining a machining coordinate system]
Error type	1, Error message from NC program.		

ID 20676

Invalid angle.

Description	When programming the NC command #CS ON[...], an inadmissible rotation angle was programmed inside the square brackets. Syntax example: **Wrong:** #CS ON [10,20,0,45,0,**450**] **Correct:** #CS ON [10,20,0,45,0,**90**]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the NC command #CS ON[...] the rotation angles may only be programmed between -360° and +360°. For further information see [PROG// Section: Defining a machining coordinate system]
Parameter	%1:	Error value[0.1 µm or 0.0001°]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20679

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20680

Coordinate from forward transformation is out of data format.			
Description	When a Cartesian transformation (#CS ON[...] or #ACS ON[...]) is selected, the program detects that an axis position exceeds the permissible value range after transformation.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the machine axis positions before selection of the coordinate transformation. In the NC program, move to a smaller axis position before selecting this option.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
	%4:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20681

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20683

Axis exchange not allowed during active RTCP/TLC.			
Description	During active kinematic transformation (#TRAFO ON) no axis exchange commands may be used in the NC program. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] **N30 #TRAFO ON** **N40 #PUT AX[X]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] N30 #TRAFO ON : **N100 #TRAFO OFF** **N110 #PUT AX[X]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before using the axis exchange commands in the NC program, deselect the kinematic transformation (#TRAFO OFF).
Error type	1, Error message from NC program.		

ID 20684

Axis exchange not allowed during active (A)CS.			
Description	While a Cartesian transformation (#CS ON or #ACS ON) is active, no axis exchange commands may be used in the NC program. Example: **Wrong:** N10 G00 X0 Y0 Z0 **N20 #CS ON[10,20,0,0,0,45]** **N30 #PUT AX[X]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 **N20 #CS ON[10,20,0,0,0,45]** : **N100 #CS OFF** **N110 #PUT AX[X]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before you use axis exchange commands in the NC program, deselect the Cartesian transformation (#CS OFF or #ACS OFF).

Error type	1, Error message from NC program..

ID 20685

Dwell time programmed with additional axes.			
Description	When dwell time is programmed with G04, more than the first main axis in the NC block is used to define the dwell time. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G04 X2 **Y3** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G04 X2** : N1000 M30 Alternatively, dwell time is programmable using the command #TIME.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. With G04, only the first main axis may be used to assign the dwell time. Remove redundant axis programming.
Parameter	%1:	Logical axis number [-]	
		Number of the redundant axis	
Error type	1, Error message from NC program.		

ID 20686

Kinematic ID is out of data format.			
Description	When programming the command #KIN ID[...] the number of the kinematic identifier exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Use a permissible kinematic ID.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20687

Kinematic change not allowed during active RTCP/TLC.

Description	A kinematic change with #KIN ID[...] must not be programmed while the kinematic transformation is active. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] **N30 #TRAFO ON** **N40 #KIN ID[2]** : N100 #TRAFO OFF : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #KIN ID[1] N30 #TRAFO ON : **N100 #TRAFO OFF** **N110 #KIN ID[2]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. #KIN ID[...] shall be programmed only when the kinematic transformation has been deselected (#TRAFO OFF).
Error type	1, Error message from NC program.		

ID 20688

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 20689

Error while opening file for returning to contour.			
Description	After block search, the manual block or an NC program for returning to contour [FCT-C6] cannot be executed because the file does not exist at all or the program name is incorrect.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Use the correct NC program name or ensure that an access to the file is possible.
Error type	-		

ID 20690

Missing program name for returning to contour after block search.			
Description	The NC program for restarting to the contour [FCT-C6] after block search cannot be executed because it was not transmitted with the command.		
Response	Class	1	Block search processing is continued.
Solution	Class	1	NC program for restarting to the contour transferred with the command.
Error type	-		

ID 20691

Data set of programmed tool is not enabled.			
Description	The tool data requested with the D command from the internal tool list are not available because the corresponding data set is locked via the validity identifier P-TOOL-00005.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Release the data set in the tool parameter list by setting the validity identifier P-TOOL-00005. Update the tool list in NC channel and restart NC program. Note: In NC program the status of the valid flag can be checked with the global variable V.G.WZ[.].OK or V.G.WZ_AKT.OK For further information see [PROG// Section: Global variables]
Parameter	%1:	Current value [-]	
		Index of the disabled data set	
Error type	1, Error message from NC program.		

ID 20692

Selection of tool radius compensation during additive manual block mode. Channel buffers blocks.			
Description	In additive manual block mode, select the tool radius compensation with G41 or G42. This buffers all the other motion blocks commanded in manual block mode in the TRC and they are executed with delay.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	If possible, avoid selecting TRC in active additive manual block mode.
Error type	-		

ID 20693

In actual state a manual block is expected.			
Description	During active additive current interface it is tried to start a NC program.		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Before starting the NC program, terminate the additive current interface with M30 or RESET.
Error type	-		

ID 20695

Action is not allowed within actual state of block search.			
Description	The assignment from the user interface to continue the block search is only allowed, if the block search is waiting for this action.		
Response	Class	1	Block search processing is continued.
Solution	Class	1	Assign the block search with a permissible action.
Error type	-		

ID 20697 / 20698

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20699

Number of parameter R is 0, negative or greater than maximum limit.			
Description	The index of the R parameter is outside the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Use a permissible value for R parameter.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20701

Double programmed G301/G302 without motion block.			

Description	A motion block must always be programmed between two NC commands G301/ G302 so that a chamfer or rounding can be inserted. Without this motion block these is not possible.
	Example:
	Wrong:
	`%kont`
	`N10 X0 Y0`
	`N20 X100`
	`N30 G301 I20`
	`N40 G301`
	`N50 Y100`
	`N60 X200`
	`N99 M30`
	Correct:
	`%kont`
	`N10 X0 Y0`
	`N20 X100`
	`N30 G301 I20`
	`N40 Y100`
	`N50 G301`
	`N60 X200`
	`N99 M30`		

Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove superfluous G301/G302 or insert motion blocks.
Error type	1, Error message from NC program.		

ID 20702

End of program reached with active contouring mode.			

Description	The NC program end is reached when G301/G302 is active. This means the necessary second contour element for inserting the chamfer or rounding is not programmed.
	Example:
	Wrong:
	`%kont`
	`N10 X0 Y0`
	`N20 X100`
	`N30 G301 I20`
	`N99 M30`
	Correct:
	`%kont`
	`N10 X0 Y0`
	`N20 X100`
	`N30 G301 I20`
	`N40 Y100`
	`N99 M30`		

Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove G301/G302 or insert a second contour element.
Error type	1, Error message from NC program.		

ID 20703

Machine axis position calculated by backward transformation is out of data format.			
Description	The necessary calculation of machine axis positions during kinematic transformations supplies results that exceed the permissible data range. Possible causes: • Invalid setting of the kinematic tool head offsets in the channel parameters P-CHAN-00094 • Missing homing process • Invalid axis positions at program start		
Response	Class	2	Abort NC program processing.
Solution	Class	3	If necessary check and eliminate the possible causes mentioned above.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20704

Detected end of program without reaching block search breakpoint.			
Description	The assigned block leader mark could not be found in the NC program until the program end (M02, M30).		
Response	Class	1	NC program is finished without any action.
Solution	Class	1	Restart block search mode with an existing block search break point.

Error type	1, Error message from NC program.

ID 20705

Unknown type of block search received with program assignment.			
Description	A block search type is selected, and that is not permitted. The error is detected during the assignment of the NC program.		
Response	Class	3	Abort NC program processing.
Solution	Class	6	Check and modify the block search assignment.
Parameter	%1:	Error value [-]	
Error type	-		

ID 20706

Unknown type of program path.			

Description	A program path with an unknown type P-STUP-00020 is configured in the start-up list.
	
	Example with error:
	`pfad[0].prg[0] v:\ref_test\nc_prg\init`
	`pfad[0].log_nr[0] 1`
	`pfad[0].typ[0] 0x06`
	`pfad[0].prioritaet[0] 1`
	`#`
	`pfad[0].prg[1] v:\ref_test\nc_prg\dec`
	`pfad[0].log_nr[1] 1`
	`pfad[0].typ[1] 0x03`
	`pfad[0].prioritaet[1] 2`		

Response	Class	2	Start-up of the control is aborted.
Solution	Class	6	Check and modify the settings of the types P-STUP-00020 for the program path in the start-up list. Repeat the NC start-up.
Parameter	%1:	Current value [-]	
		Invalid type of the program path	
Error type	-		

ID 20707

Program paths of the same type are not allowed to have the same priority.			

Description	The start-up list contains configured program paths that may be of the same type as P-STUP-00020 but may not have the same priority asP-STUP-00021.
	
	Example with error:
	`pfad[0].prg[0] v:\ref_test\nc_prg\init`
	`pfad[0].log_nr[0] 1`
	`pfad[0].typ[0] 0x03`
	`pfad[0].prioritaet[0] 1`
	`#`
	`pfad[0].prg[1] v:\ref_test\nc_prg\dec`
	`pfad[0].log_nr[1] 1`
	`pfad[0].typ[1] 0x03`
	`pfad[0].prioritaet[1] 1`		

Response	Class	2	Controller start-up is aborted.

Solution	Class	6	Check and modify the priority settings for program paths in the start-up list. Repeat the NC start-up.
Parameter	%1:	Error value [-]	
		Invalid priority P-STUP-00021	
	%2:	Current value [-]	
		Program path type P-STUP-00020	
Error type	-		

ID 20708 / 20709

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20711

Current value offset is out of data format.			
Description	The current value programmed with the #PSET command for an axis results in a current value offset that exceeds the permissible numerical range. Syntax example: `N10 #PSET X<curr_value>`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the actual value for the corresponding axis.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20712

Value assigned to G-function is out of data format.			
Description	The value programmed with the Look-Ahead function G115 exceeds the permissible numerical range. Syntax example: `N10 G115= <single_function>` For further information see [PROG// Section: Influence on the look-ahead functionality]		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify the value.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20713

Receiver ID of the message is out of data format.			
Description	In the #MSG command, the message receiver is specified by its communication device ID (ISG_DIAG_BED, HMI; PLC). However, the receiver ID can also be programmed directly by specifying a numerical value. This value here is outside the permissible data range. For further information see [PROG//Section: Messages from the NC program]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the #MSG command and correct the receiver ID.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20715

The message string must be enclosed in quotation marks.			

Description	In the #MSG command, a quotation mark is expected after the square brackets before the start and after the end of the message text.
	
	Syntax example:
	`N10 #MSG ["MESSAGE_TEXT"]`
	`:`
	For further information see [PROG// Section: Messages from the NC program]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the #MSG command and enter the missing quotation marks.
Error type	1, Error message from NC program.		

ID 20716

Close bracket or comma is expected.

Description	In the #MSG command the next character after the closing inverted commas of the message text and with a single output of a text is expected to be a square closing bracket or, if the format elements (%s, %S, %d, %D, %f, %F) are used, a comma .
	
	Syntax example:
	`N10 #MSG ["SIMPLE TEXT"]`
	`:`
	`N20 #MSG ["TEXT_MIT_WERT=%D ", 10]`
	For further information see [PROG// Section: Messages from the NC program]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	#MSGcommand must be checked, and missing syntax added.
Error type	1, Error message from NC program.		

ID 20718

Unknown receiver ID during #MSG...			
Description	The receiver ID programmed with #MSG is not known in the NC channel or in the system. This is why the message is not output.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Specify a permissible receiver ID with #MSG. See also [PROG//Section: Programming a message]
Parameter	%1:	Error value [-]	
		Programmed value of receiver ID	
	%2:	Expected value [-]	
		Permissible value of receiver ID of CNC diagnosis interface.	
	%3:	Expected value[-]	
		Permissible value of receiver ID of PLC	
	%4:	Expected value[-]	
		Permissible value of receiver ID of HMI	
Error type	1, Error message from NC program.		

ID 20721

Number of parameters and the number of format specifiers do not match.			

Description	In the #MSG command, the number of programmed format elements (%s, %S, %d, %D, %f, %F) must be associated with all the corresponding values.
	
	Syntax example (3 format elements, 3 values):
	`N10 #MSG ["WERT1=%D  WERT2=%D  WERT3=%D "Number of parameters and the number of format specifiers do not match.]`
	`:`
	For further information see [PROG// Section: Messages from the NC program]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the #MSG command and enter any missing values.
Parameter	%1:	Current value [-]	
		Number of the programmed format elements (%s, %S, %d, %D, %f, %F)	
Error type	1, Error message from NC program.		

ID 20722

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20723

The message string is too long.			
Description	When the #MSG command replaces the format elements (%s, %S, %d, %D, %f, %F) with the corresponding values, it detects that the resulting message text becomes too long. For further information see [PROG// Section: Messages from the NC program]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	#MSG command must be checked, and the length of the message string reduced.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20724

Too many parameters in the message string.			
Description	The command #MSG is programmed with too many format elements (%s, %S, %d, %D, %f, %F), i.e. too many values must be output. For further information see [PROG// Section: Messages from the NC program]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the #MSG command and reduce the number of format elements to correspond with the limit value.
Parameter	%1:	Limit value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20726

After a comma a mathematical expression is expected.

Description	In the #MSG command, the corresponding value must also be programmed for a format element (%s, %S, %d, %D, %f, %F). This value is specified after the message string in the comma-separated area. If there are programmed several format elements, also the values have to be programmed in the corresponding order, separated by commas.
	
	Syntax example:
	`N10 #MSG ["WERT=%D", 10]`
	`:`
	`N20 #MSG ["WERT1=%D  WERT2=%D", 10, 20]`
	For further information see [PROG// Section: Messages from the NC program]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	#MSG command must be checked, and missing value(s) added.
Error type	1, Error message from NC program.		

ID 20727 - 20733

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20734

Overflow of ASCII-pointer stack.			
Description	If the nesting depth in NC commands such as #MSG or macrodefinitions ("macro") is exceeded when programming, the command cannot be further processed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and reduce nesting depth in the affected NC commands.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20735 / 20736

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20737

Unable to open logfile for manual blocks.			
Description	The programmed manual blocks, especially in additive manual mode, can be logged in a file with the name "dec00.hs". This name is configured in the controller.		
Response	Class	1	Continue NC program processing.

Solution	Class	1	Check the system rights (write access, user rights, etc.).
Error type	-		

ID 20738

Write access to tool length is not allowed with D code in the same block.			
Description	In the same NC block in which a D word is programmed, an attempt is also made to modify the tool length of the current tool by means of write access using the variable V.G.WZ_AKT.L. Syntax example: `N10 V.G.WZ_AKT.L=25 X10 Y20 D1`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC block and remove either the write access to the tool length or the D word.
Error type	1, Error message from NC program.		

ID 20739

Write access to tool offset is not allowed with D code in the same block.			
Description	In the same NC block in which a D word is programmed, an attempt is also made to modify a tool offset of the current tool by means of write access using the variable V.G.WZ_AKT.Vxx. Syntax example: `N10 V.G.WZ_AKT.V[1]=40 X10 Y20 D1`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC block and remove either the write access to the tool offset or the D word.
Error type	1, Error message from NC program.		

ID 20740

Only one write access to tool radius is allowed per NC block.			
Description	An attempt is made to modify the tool radius of the current tool several times in the same NC block by means of write access using the variable V.G.WZ_AKT.R. Syntax example: `N10 V.G.WZ_AKT.R=25 X10 Y20 V.G.WZ_AKT.R=30` For further information see [PROG// Section: Global variables (V.G.)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC block and remove all write accesses to the tool radius except for one.
Error type	1, Error message from NC program.		

ID 20741

Only one write access to tool length is allowed per NC block.			
Description	An attempt is made to modify the tool length of the current tool several times in the same NC block by means of write access using the variable V.G.WZ_AKT.L. Syntax example: `N10 V.G.WZ_AKT.L=50 X10 Y20 V.G.WZ_AKT.L=70` For further information see [PROG// Section: Global variables (V.G.)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC block and remove all write accesses to the tool length except for one.
Error type	1, Error message from NC program.		

ID 20743

After the #-command an opening bracket is expected.			
Description	A # command is programmed in the NC block, which should be followed by a opening square bracket "[" and further syntax elements. This square bracket is not programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and complete the syntax of the # command.
Error type	Abort NC program processing.		

ID 20744

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20745

Too many parameters with #-command.			
Description	A # command is programmed in the NC block but it contains more parameters than specified by the syntax.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust the number of parameters within the #-command
Parameter	%1:	Limit value [-]	
		Number of permissible parameters in the respective NC command	
Error type	Abort NC program processing.		

ID 20746

SPLINE parameter is out of data format.

Description	When programming the commands #AKIMA TRANS [..] / #AKIMA STARTVECTOR/ #AKIMA ENDVECTOR or #SET ASPLINE MODE [...] or #SET ASPLINE STARTTANG/ZIELTANG, there is a spline parameter outside the permissible data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the spline parameter.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	Abort NC program processing.		

ID 20747

Calling of subroutine from manual block fails due to too many opened files.			
Description	In a deeply nested NC program, another subroutine is to be called within a subroutine in manual operation mode. However, this exceeds the memory system resources.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and simplify the NC program structure. Reduce nesting depth of the subroutines.
Parameter	%1:	Limit value [-]	
		Minimum number of necessary cache elements which is undershot.	
Error type	Abort NC program processing.		

ID 20748

While tool radius compensation is active information of additive manual block is buffered.			
Description	The TRC retains motion blocks to calculate the equidistant contour, . Thus, additive manual blocks may be retained and not executed immediately.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Before selecting additive manual mode (G201), deselect active TRC (G40).
Error type	5, Error message by access on files.		

ID 20749

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

2.3.4 ID-range 20750-20999

ID 20750

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20751

Dwell time is out of range of permissible values.			
Description	When programming the dwell time immediately after the #TIME command, the value of the dwell time exceeds the permissible data range. Syntax example: `N10 #TIME <dwell_time>`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value of the dwell time.
Parameter	%1:	Error value [µs]	
	%2:	Lower limit value [µs]	
	%3:	Upper limit value [µs]	
Error type	1, Error message from NC program.		

ID 20752

Dwell time is out of range of permissible values.			

Description	When programming the dwell time immediately after the G04 command or in combination with the F word in a configuration-specific syntax, the value of the dwell time exceeds the permissible data range.
	
	Syntax example:
	`N10 G04 <dwell_time>`
	
	`or`
	
	`N10 G04 Dwell time is out of range of permissible values.<dwell_time> (configuration-specific syntax)`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value of the dwell time.

Parameter	%1:	Error value [µs]
	%2:	Lower limit value [µs]
	%3:	Upper limit value [µs]
Error type	1, Error message from NC program.	

ID 20753 / 20754

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20756

When calculating the tool coordinate, the data format is violated.			
Description	When calculating tool offsets during active 5-axis transformation, it is detected that a new calculated coordinate is outside the permissible data format.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check sign and value of the tool offsets and if necessary correct it.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Transformed axis-specific tool offset	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20757

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20758

Programming centre point coordinate of missing axis.	
Description	A centre point coordinate I, J or K has been programmed for an axis that does not currently exist in the NC channel.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Make sure that the main axes are present in the corresponding planes in the NC channel or, in the case of chamfers and radii, that G301 or G302 have been programmed previously.
Error type	1, Error message from NC program.		

ID 20759

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20760

Channel parameters: Process time has to be 0 for synchronization mode MOS.			
Description	During start-up, the check of the channel parameters determines that, for technology functions (M, H, S, T) defined with synchronization mode MOS (see P-CHAN-00041), the assigned process times are set to a value unequal to zero (nil) (see e.g. P-CHAN-00040).		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	During start-up in case of conflict the process time is set to 0 and the start-up is continued.
Parameter	%1:	Current value [-]	
		Index of the invalid technology function (M, H, S, T)	
	%2:	Error value [-]	
		Invalid process time	
	%3:	Corrected value [-]	
		Automatically corrected process time	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20762

Maximum number of M/H functions per NC block reached.			
Description	The maximum number of M/H technology functions programmed in the NC block was exceeded. The sum of M and H functions is always considered here.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Split and distribute the M/H technology functions over several NC blocks.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20764

Channel parameters: Unknown synchronization mode.

Description	During start-up, the channel parameters are checked to determine that techno functions (M, H, S, T) are assigned unknown synchronization modes.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	In the event of a conflict, the techno function is assigned the synchronization mode MVS_SVS (see P-CHAN-00041) during the start-up, and the start-up is continued.
Parameter	%1:	Current value [-]	
		Index of the unknown technology function (M, H, S, T)	
	%2:	Error value [-]	
		Unknown synchronization mode	
	%3:	Corrected value [-]	
		Automatically corrected synchronization mode (MVS_SVS)	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20765

Double-programmed M-function.			

Description	In the NC block, an M function (techno function) with the same number is programmed several times channel-specifically, axis-specifically or spindle-specifically.
	
	Example for inadmissible double programming:
	`N10 G00 X10 M10 M10 (channel-specific technofct.)`
	`N10 G00 X10 X[M10] X[M10]] (axis-specific technofct.)`
	`N10 G00 X10 S[M10 M10] (spindle-specific technofct.)`
	`:`
	Example for allowed programming:
	`N10 G00 X10 M10 X[M10] S[M10] (combined technofct.)`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the programming of the M functions in the NC block. Remove surplus channel specific, axis specific or spindle specific M functions.
Parameter	%1:	Current value [-]	
		Number of the multiple programmed M function	
Error type	1, Error message from NC program.		

ID 20766

Double-programmed H-function.			

Description	In the NC block, an H function (techno function) with the same number is programmed several times channel-specifically, axis-specifically or spindle-specifically.
	
	Example for inadmissible double programming:
	`N10 G00 X10 H10 H10 (channel-specific technofct.)`
	`N10 G00 X10 X[H10] X[H10]] (axis-specific technofct.)`
	`N10 G00 X10 S[H10 H10] (spindle-specific technofct.)`
	`:`
	Example for allowed programming:
	`N10 G00 X10 H10 X[H10] S[H10] (combined technofct.)`		

Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify the programming of the H functions in the NC block. Remove surplus channel specific, axis specific or spindle specific H functions.
Parameter	%1:	Current value [-]	
		Number of the multiple programmed H function	
Error type	1, Error message from NC program.		

ID 20770

Invalid combination of operating modes selected.			
Description	A invalid combination of operating mode settings is selected. Examples of settings of operation modes: • Command contour visualisation • Syntax check • Online machining time calculation The error is detected when the NC program is commanded.		
Response	Class	3	Abort NC program processing.
Solution	Class	6	Check and modify the settings of operation modes.
Parameter	%1:	Error value [-]	
Error type	-		

ID 20771

Multiple use of same program path number.			

Description	In the start-up list, program paths with the same program path number P-STUP-00019 are configured in the start-up list. This is not permitted.
	
	Example with error:
	`pfad[0].prg[0] v:\ref_test\nc_prg\init`
	`pfad[0].log_nr[0] 1`
	`pfad[0].typ[0] 0x03`
	`pfad[0].prioritaet[0] 1`
	`#`
	`pfad[0].prg[1] v:\ref_test\nc_prg\dec`
	`pfad[0].log_nr[1] 1`
	`pfad[0].typ[1] 0x03`
	`pfad[0].prioritaet[1] 2`		

Response	Class	2	Start-up of the control is aborted.
Solution	Class	6	Check and modify the settings of the program path numbers in the start-up list. Repeat the NC start-up.
Parameter	%1:	Error value [-]	
		Invalid program path number P-STUP-00019	
Error type	-		

ID 20772

G-function with current measuring type not allowed.

Description	The programmed G function is not permissible in connection with the current measuring type. [PROG// Section: Measuring with motion through to target point (G100/G106)(Type 3)] [PROG// Section: Calculating measuring offsets (G101/G102		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the measuring type or do not use the G function.
Parameter	%1:	Current value [-]	
		Number of the programmed G function	
	%2:	Current value [-]	
		Used measuring type	
Error type	1, Error message from NC program.		

ID 20773

Measurement type is out of data format.			
Description	When programming the #MEAS MODE[...] or #MEAS [TYPE...] command, the value of the measurement type is outside the permissible data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value of measurement type.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20774

Effector coordinate system may not be selected if (A)CS is already active.			

Description	An effector coordinate system cannot be selected (#ECS ON) while another coordinate system is already enabled (e.g. #CS ON).
	
	Example with error:
	`N40 #CS ON[1,2,3,4,5,6]`
	`N50 G01 X100`
	`N60 #ECS ON`
	`N70 G01 Z50`
	`N80 #ECS OFF`
	`N90 #CS OFF`
	
	Corrected example:
	`N40 #CS ON[1,2,3,4,5,6]`
	`N50 G01 X100`
	`N60 #CS OFF`
	`N70 #ECS ON`
	`N80 G01 Z50`
	`N90 #ECS OFF`		

Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify the program flow sequence. Deselect the active coordinate system before selecting the effector coordinate system.
Error type	1, Error message from NC program.		

ID 20775

A rotary axis is missing for effector coordinate system.			
Description	Rotary axes are required for the orientation or alignment of an effector coordinate system. With selected with #ECS ON, the system detects that one of these rotary axes is not available in the current axis configuration.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the axis configuration in the NC channel. Ensure that all axes in the valid kinematic are present in the NC channel.
Parameter	%1:	Current value [-]	
		Index of the missing axis	
Error type	1, Error message from NC program.		

ID 20776

Effector coordinate system cannot be selected via parameters.			
Description	The #ECS ON command cannot be used with parameters.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove the parameter after the #ECS ON command.
Error type	1, Error message from NC program.		

ID 20777

Using tool radius compensation not possible, it does not exist in the system.			
Description	Tool radius compensation cannot be selected or deselected because it is not available in the NC channel. The tool radius compensation (TRC) is enabled via the channel parameter P-CHAN-00092.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the channel parameter P-CHAN-00092. Then update the channel parameters list.
Parameter	%1:	Error value [-]	
		Used G function which requires an existing tool radius compensation	
Error type	1, Error message from NC program.		

ID 20778 / 20779

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20780

Tool management was not able to prepare tool change.			
Description	The order to prepare a tool change (T word) was acknowledged incorrectly by the external tool management.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify external tool management. If the parameters 1 and 2 are identical, the tool management has acknowledged negative. If the parameters 1 and 2 are not identical, the tool management has acknowledged a tool, which was not instructed.
Parameter	%1:	Current value [-]	
		Number of the received tool	
	%2:	Expected value [-]	
		Number of the requested tool	
Error type	3, Error in the communication.		

ID 20781

Cancellation of tool request was not possible.			
Description	The command to cancel a tool request was acknowledged incorrectly by the external tool management. A tool cancellation is executed if a tool request is pending during NC reset.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify external tool management. If the parameters 1 and 2 are identical, the tool management has acknowledged negative. If the parameters 1 and 2 are not identical, the tool management has cancelled a tool that was not commanded.
Parameter	%1:	Current value [-]	
		Number of the received tool	
	%2:	Expected value [-]	
		Number of the requested tool	
Error type	3, Error in the communication.		

ID 20782

Reset of external tool management was not possible.			
Description	When an NC reset is commanded, the external tool management system incorrectly confirms the reset request. The reset is then no longer guaranteed as correct.		
Response	Class	3	Abort NC reset.
Solution	Class	6	Repeat NC reset. Check external tool management.
Error type	3, Error in the communication.		

ID 20783

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20786

Transformed tool offset is out of data format.			
Description	During the change to a different coordinate system (#CS, #ACS, #MCS), the offsets of fixed tools are also transformed. The result of a transformed tool offset then exceeds the permissible data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and reduce the appropriate tool offsets which are entered by V.G.WZ_AKT.V[i] or which are defined by P-TOOL-00006 in the tool list.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20787

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20788

Inadmissible character within name of external variable.			
Description	When external variables are initialised, an invalid character was detected in one of the variables. Access to the incorrect variable is not possible in the NC program. [EXTV// Section: Syntax]		
Response	Class	2	Controller start-up is continued.
Solution	Class	3	Check and change the name of the external variable in the list of external variables before the next controller start-up.
Parameter	%1:	Current value [-]	
		Index of the external variable	
	%2:	Current value [-]	
		Position number of the invalid character in the variable name	
	%3:	Current value [-]	

Error type	-

ID 20789

In front of the string a quotation mark is missing.			
Description	When using external variables of the type string within the NC program, there's the quotation mark missing before the assigned text string. Example: **Wrong:** N10 V.E.TYPESTRING = TEXTSTRING" N20 G01 X10 **Right:** N10 V.E.TYPESTRING = "TEXTSTRING" N20 G01 X10		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and insert the quotation mark before the text string.
Error type	1, Error message from NC program.		

ID 20790

After the string a quotation mark is missing.			
Description	When using external variables of the type string within the NC program, there's the quotation mark missing after the assigned text string. Example: **Wrong:** N10 V.E.TYPESTRING = "TEXTSTRING N20 G01 X10 **Right:** N10 V.E.TYPESTRING = "TEXTSTRING" N20 G01 X10		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and insert the quotation mark after the string.
Error type	1, Error message from NC program.		

ID 20791

Read string is too long.			
Description	The identifier (name) of a variable is too long. The type of variable can be self-defined variable (V.S., V.P. V.L.) or external variables (V.E.).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the identifier of the variable.
Parameter	%1:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20792

Double programming of spline interpolation

Description	It is not permitted to program repeatedly commands for selecting and deselecting spline interpolation (G150/G151) within an NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove the multiple programming within the NC block.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20794

Double programming of corner deceleration.			
Description	It is not permitted to program the selection and deselection of corner deceleration (G12/G13) several times within a NC block. [PROG// Section: Selecting/deselecting corner deceleration (G12/G13]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove the multiple programming within the NC block.
Parameter	%1:	Error value [-]	
		Number of the invalid G function	
Error type	1, Error message from NC program.		

ID 20795

Selection of corner deceleration without parameter not permitted.			
Description	The corner deceleration cannot be activated without previously specifying parameters with the #CORNER PARAM[...] command. [PROG// Section: Selecting and deselecting corner deceleration (G12/G13]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Insert the command #CORNER PARAM[...] in the NC program before the NC block with the activation of the corner deceleration (G13).
Parameter	%1:	Current value [-]	
		Number of the invalid G function.	
Error type	1, Error message from NC program.		

ID 20796

CORNER parameter is out of data format.			
Description	One of the parameters programmed with the #CORNER PARAM[...] command exceeds the permitted data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value of parameters within the command.
Parameter	%1:	Error value [-]	
	%2:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20797

Unknown axis designation respectively character not allowed.			
Description	In commands in which axis names are programmed, an unknown designation of an axis was read in or an unauthorized character was found in place of an axis name.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis designations or the syntax of the appropriate command.
Error type	1, Error message from NC program.		

ID 20798

Spindle value is out of range of permissible values.			
Description	The value programmed with the S word exceeds the permissible numerical range. Take especially into account the spindle speed, the feed rate per revolution, the constant cutting speed or the spindle position.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the values programmed with the S words.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20799

Feed rate is out of range of permissible values.			
Description	The F word was not yet programmed or the feed rate programmed with the F word is outside the permissible value range. The feed rate value is influenced by the setting of the unit (P-CHAN-00108) or by the definition of the default feed rate (P-CHAN-00099).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program or check F word and set feed rate within the value range.
Parameter	%1:	Lower limit value [-]	
		Minimum limit value	
	%2:	Upper limit value [-]	
		Maximum limit value	
	%3:	Error value [-]	
		Feed value	
Error type	1, Error message from NC program.		

ID 20800

Chamfer value is out of data format.			
Description	The chamfer value programmed with the G301 or G302 command is out of range. [PROG// Section: Inserting chamfers and rounding (G301/G302)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the programmed contouring value.

Parameter	%1:	Lower limit value [-]
	%2:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 20801

Residual path for edge banding is out of data format or out of range of permissible values.			
Description	The programmed residual path for edge banding that can be entered in the NC program using the variable V.G.RW or which can be defined by the channel parameter P-CHAN-00030 exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the channel parameter P-CHAN-00030 or the programmed variable V.G.RW in the NC program.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20802

Coordinate system transition during active tool radius compensation not allowed.			

Description	During the transition of coordinate systems with the commands#CS, #ACS and/or #MCS, the tool radius compensation may not be enabled. This restriction applies both to the selection and deselection of the respective function.
	
	Example with error:
	`G237`
	`G41`
	`#MCS ON`
	`X100`
	`Y100`
	`G40`
	`X200`
	`#MCS OFF`
	
	Corrected example:
	`G237`
	`#MCS ON`
	`G41`
	`X100`
	`Y100`
	`G40`
	`X200`
	`#MCS OFF`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence.
Error type	1, Error message from NC program.		

ID 20803

Temporary transition into machine coordinate system is already active.			
Description	Multiple selection of the temporary transition to the machine coordinate system with the command #MCS ON has no effect since the temporary transition is already active.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove the repeated programming of the #MCS ON command.
Error type	1, Error message from NC program.		

ID 20804

Temporary transition into machine coordinate system is inactive.			
Description	Deselection of the temporary transition to machine coordinate system with the command #MCS OFF has no effect because the temporary transition is not active.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove the #MCS OFF command.
Error type	1, Error message from NC program.		

ID 20805

Programmed function in temporary machine coordinate system not allowed.			
Description	The programmed function is not permitted when the temporary machine coordinate system (#MCS ON) is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deselect the temporary machine coordinate system with #MCS OFF before programming the function concerned.
Error type	1, Error message from NC program.		

ID 20806

While changing kinematic transformation (RTCP) tool radius compensation may not be active.			
Description	It is not possible to select or change the kinematic transformation while tool radius compensation (TRC) is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Deselect TRC before changing or select transformation.
Error type	1, Error message from NC program.		

ID 20809

Slave axis must not participate at kinematic transformation.			
Description	An axis that is involved in an active kinematic transformation is to be operated simultaneously as a slave axis in an axis coupling. This is not possible. [PROG// Section: Programming axis couplings]		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Deselect active kinematic transformation before selection of coupling or remove the master-slave linkage of the concerned axis from the coupling definition.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of slave axis	
Error type	1, Error message from NC program.		

ID 20810

Double programming of contour masking mode.			
Description	It is not possible to program the selection type for contour masking more than once within an NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and remove double programming.
Parameter	%1:	Error value [-]	
		Double programmed G-function	
Error type	1, Error message from NC program.		

ID 20811

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20812

No file name was entered, therefore processing will not start.			
Description	At program start, no name of a NC program was specified. Therefore, the program cannot be processed.		
Response	Class	1	NC program can be restarted.
Solution	Class	1	Program start after entering the program name.
Error type	5, Error message by access on files.		

ID 20813

Too many actions in the NC block. Job buffer capacity insufficient.			
Description	The NC block contains too much NC commands, which strain the internal system resources.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Split the NC block into several NC blocks.
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 20814

Measuring offset was not included into the programmed axis.			
Description	It is not possible to calculate the measuring offset with command G102 in the programmed axis because no measuring offset has been calculated. [PROG// Section: Calculating measuring offsets (G101/G102		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the programming of offsets (G101).
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016)	
Error type	1, Error message from NC program.		

ID 20816

Coordinate for homing is out of data format.			
Description	When using the axis referencing command G74, the value of a coordinate is out of the permissible numerical range. The individual coordinate values of the axes specify the order of the homing axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value of the coordinate.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016)	
	%2:	Error value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20817 / 20818

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20819

Channel parameters: G or M function is not permitted as default value.			
Description	A defined default value for G functions (P-CHAN-00063) or M functions (P-CHAN-00064) in the channel parameter list is out of the permissible data range.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In the channel parameters, check and change the default value of the corresponding G or M function before the next controller start-up. In case of conflict, set the invalid entry to a permitted value for this function at start-up.

Parameter	%1:	Error value [-]
	%2:	Corrected value [-]
	%3:	Current value [-]
		Group number of the invalid default entry
Error type	2, Error message by data transfer from parameter list into control device.	

ID 20820

Programmed CS-ID not allowed.			
Description	The programmed (A)CS-ID value is not permitted. In builds up to V300, the value may be between minimum and maximum ID value. In builds after V300, the ID may be programmed with any value greater than zero, where maximum 5 CS or 5 ACS definitiosn are permitted in any case.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the programmed (A)CS-ID.
Parameter	%1:	Error value [-]	
		Invalid (A)CS-ID	
	%2:	Lower limit value [-]	
		Minimum permissible ID value	
	%3:	Upper limit value [-]	
		Maximum value or number of CS IDs	
	%4:	Upper limit value [-]	
		Maximum value or number of ACS IDs (display of parameter 4 only up to V300)	
Error type	1, Error message from NC program.		

ID 20821

(A)CS not defined.			
Description	Activating a coordinate systems with #(A)CS ON without any parameters is only permitted if these parameters are already defined before with #(A)CS DEF or #(A)CS ON [...]. [PROG// Section: Defining a machining coordinate system (CS)] [PROG// Section: Defining/activating a coordinate system for fixture adaptation (ACS)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and insert the complete definition of the coordinate system.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20822

Parameters must be programmed for the definition of an (A)CS.

Description	When defining a machining coordinate systems (#CS DEF) or a coordinate system for fixture adaptation (#ACS DEF), it is essential to program the parameters for translation and rotation. [PROG// Section: Defining a machining coordinate system (CS)] [PROG// Section: Defining/activating a coordinate system for fixture adaptation (ACS)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the command #(A)CS DEF, complete the parameters for translation rotation.
Error type	1, Error message from NC program.		

ID 20823

An ECS must not be defined with DEF.			
Description	It is not possible to define an effector coordinate system (ECS) in the NC program with the syntax element "DEF". The definition of the ECS-axes is based implicitly on the orientation of the tool axis. [PROG// Section: Effector Coordinate System (ECS)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the command #ECS.
Error type	1, Error message from NC program.		

ID 20824

Deselecting (A)CS has no effect.			
Description	Deselection of coordinate system with the command #CS OFF and/or the command #ACS OFF has no effect because there's no coordinate system enabled.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove the command of deselection.
Error type	1, Error message from NC program.		

ID 20825

Definition of (A)CS may not be changed if (A)CS is active.			
Description	The definition of a coordinate system (CS/ACS) may not be changed while it is enabled. [PROG// Section: Coordinate systems]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before redefinition, deselect the corresponding coordinate system.
Error type	1, Error message from NC program.		

ID 20826

Overflow of defined (A)CS-stack.

Description	The number of concatenated links of self-defined coordinate systems exceeds the permissible upper limit.. The error occurs when exceeding the links both with #CS ON as and with #ACS ON. [PROG// Section: Linkage of coordinate systems]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and reduce the number of concatenated links of CS and/or ACS-coordinate systems.
Parameter	%1:	Upper limit value [-]	
		Maximum number of links with #CS ON	
	%2:	Upper limit value [-]	
		Maximum number of links with #ACS ON	
Error type	1, Error message from NC program..		

ID 20827 / 20828

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20829

Number of string labels too big for optimised program execution.			
Description	The maximum number of memory locations for string labels for fast execution of a jump call was reached. It's not possible to save any more string labels with the corresponding program position. Optimised program execution means that the program position of the string label (jump target) is stored and that this jump target is available at once without a search run. If the upper limit is exceeded, every new string label must be searched again in case of a jump call. This lengthens the program runtime.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and reduce the number of string labels used.
Parameter	%1:	Limit value [-]	
		Maximum possible number of string labels stored	
Error type	1, Error message from NC program.		

ID 20830

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20831

Number of expression labels too big for optimised program execution.			
Description	The maximum number of memory locations for expression labels for fast execution of a jump call was reached. It's not possible to save any more expression labels with the corresponding program position. Optimised program execution means that the program position of the expression label (jump target) is stored and can be jumped to immediately when this jump target is called without a search run. If the upper limit is exceeded, every new expression label must be searched again in case of a jump call. This lengthens the program runtime.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and reduce the number of used expression labels.
Parameter	%1:	Upper limit value [-]	
		Maximum possible number of stored expression labels	
Error type	1, Error message from NC program.		

ID 20832

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20833

Label has too many characters.			
Description	The number of characters in the string label exceeds the allowed upper limit. The string label length is checked during reading in, searching and during a jump call ($GOTO).		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Check and modify the name of the string label.
Parameter	%1:	Limit value [-]	
		Maximum limit of characters	
Error type	1, Error message from NC program.		

ID 20834

Value is out of data format.			
Description	The value of the block number used as label is outside the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value of the block number.

Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 20836

Multiple defined label			
Description	Within an NC program it is not permitted to define multiple identical labels in the same program level. This restriction is valid for main and subroutines. [PROG// Section: The $GOTO-instruction]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the labels within the same program level. **Note:** For string labels multiple ID 1 is indicated. For expression labels, multiple ID 2 is indicated.
Error type	1, Error message from NC program.		

ID 20837

Label has too many characters.			
Description	The number of characters in the label exceeds the maximum limit.		
Response	Class	-	Abort NC program processing.
Solution	Class	-	Check and modify the length of the string label.
Parameter	%1:		
		Maximum limit of characters	
Error type	-		

ID 20838

Expression-Label is out of data format.			
Description	The value of the expression label programmed with the command $GOTO exceeds the permitted data range. [PROG// Section: The $GOTO statement]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the expression label.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	

Error type	1, Error message from NC program.

ID 20839

The character after $GOTO is not permitted.			
Description	The programmed character after the jump statement $GOTO is not permitted. [PROG// Section: The $GOTO statement]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the sequence of characters after the jump call. Permissible characters are: "[" for the call of a string label "N" for the call of an expression label "V.E.. to call an external variable of the <string> type
Error type	1, Error message from NC program.		

ID 20840

Label not found			
Description	When using the jump call with the command $GOTO, the programmed label in NC program cannot be found. [PROG// Section: The $GOTO-instruction]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Supplement the missing jump label in the NC program or use another existing jump label for the jump call with the command $GOTO. Block numbers are only jump labels if the are labelled with a colon Nx:
Error type	1, Error message from NC program.		

ID 20841

Double-programmed block number N.			

Description	When defining an expression label, the block number has been programmed several times, that's not permitted.
	
	Example with error:
	`N10 $GOTO N100`
	`N20 X200`
	`:`
	`N100 N100: X.. Y..`
	
	Corrected example:
	`N10 $GOTO N100`
	`N20 X200`
	`:`
	`N100: X.. Y.. (Assignment of the label)`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove the block number.
Error type	1, Error message from NC program.		

ID 20842

Value is out of data format.			
Description	An assigned value is out of range. Example: The block number that is read before a string label ([<string>]) during label search is outside the permitted value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value in the NC program.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20844

Double programming of dwell time.			
Description	In the same NC block, the dwell time was programmed several times in conjunction with the F word. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G04 F10** G41 G17 **G04 F20** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant dwell time programming.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20845

Parameter F is missing for dwell time.			
Description	In this configuration specific programming of G04, the Dwell time in combination with the F word is missing. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G04 10 (or only G04)** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G04 F10** : N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Complete the missing F word for the definition of the dwell time.
Error type	1, Error message from NC program.		

ID 20846

Double programming of override influence.			
Description	It is not possible to program command G166 more than once in a block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove the multiple programmed commands G166.
Parameter	%1:	Error value [-]	
		Number of the G function	
Error type	1, Error message from NC program.		

ID 20847

Value of variable is out of data format.			
Description	When writing to a variable, the value exceeds the permitted range of the corresponding variable type used. Only with external variables it is possible to change the type and, thus, the value range of the variable P-EXTV-00003.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the assigned value of variable; with external variables a change of type and thus data range is possible in the list of external variables [EXTV].
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20848 - 20850

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20851

Double programming of spindle position.			
Description	The position of a spindle has been programmed several times, this is not allowed with the spindle-specific (POS) or DIN syntax (S.POS).		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and remove the multiple programmed spindle position.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 20852

Endless FOR-loop is programmed.			
Description	The increment 0 is programmed in a FOR loop. It is therefore not possible to leave the loop, it is an endless loop [PROG//Section: Counting loops].		
Response	Class	2	Continue NC program processing.
Solution	Class	1	Check and modify the increment of the loop. The execution of the NC program must be aborted by the operator.
Error type	1, Error message from NC program.		

ID 20856

G function has no effect because no tool radius compensation in the system.			
Description	G functions affecting the tool radius compensation have no effect because the TRC in the channel is deactivated by parameter P-CHAN-00092.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify the parameter P-CHAN-00092.
Parameter	%1:	Current value [-]	
		G function used	
Error type	-		

ID 20857

Channel parameters: Enabling of external tool management not permitted, because no external tool management is available.			
Description	Since the current system configuration basically does not allow external tool management, but the tool data is loaded exclusively via the tool list [TOOL], it does not make any sense to assign P-CHAN-00016 with 1.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	Parameter is set implicitly to default value 0. Or please contact the CNC manufacturer to enable the use of an external tool management.
Parameter	%1:	Error value [-]	
		Channel parameter P-CHAN-00016 incorrectly assigned	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20860

Selection or deselection of mirroring not allowed.

Description	With axis-specific mirroring, selection or deselection is determined via the coordinate value of an axis. Permissible values for deselection of mirroring are 1 or +1 and for selection, the value is –1. The programmed value differs from the permissible values. [PROG// Section: Mirroring with axis specification (G351)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the coordinate values.
Parameter	%1:	Error value [-]	
		Invalid coordinate value. Only +1, 1 or –1 are allowed.	
Error type	1, Error message from NC program.		

ID 20861

HUEMNOS tool only allowed with the corresponding tool management.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 20862

Factor to calculate tool life is out of range of permissible values.			
Description	The programmed factor for the calculation of tool life data calculation exceeds the permissible range [PROG//Section: Weighting factors for tool life and tool life distance].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the tool life factor.
Parameter	%1:	Error value [-]	
		Invalid tool life factor **Factor to calculate tool life is out of range of permissible values.** For distance factor, multiple ID 1 is displayed. For time factor, multiple ID 2 is displayed.	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20863

TOOL ID is out of data format.			
Description	When using one of the commands #TOOL LIFE READ, #TOOL DATA or #TOOL PREP, one of the three possible TOOL IDs (Basic, Sister, Variant) within the brackets is outside the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the TOOL IDs.

Parameter	%1:	Lower limit value [-]
	%2:	Upper limit value [-]
	%3:	Error value [-]
		Invalid TOOL ID (Basic, Sister, Variant)
Error type	1, Error message from NC program.	

ID 20864

After axis designation an orientation operator is expected.			
Description	With axis-specific tool length compensation (#TLAX [...]), an orientation operator is expected after the axis name. Permissible orientation operators are “+” and “-“. The currently programmed operator is either not permissible or is missing [PROG//Kapitel: Tool length compensation(TLC)].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command. Programming of a correct orientation operator.
Error type	1, Error message from NC program.		

ID 20865

With active tool radius compensation 1st or 2nd main axis cannot be exchanged.			
Description	When tool radius compensation is active, it is not possible to exchange axes with participation of the first and second main axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Deselect TRC before axis exchange with 1st or 2nd main axis.
Error type	1, Error message from NC program.		

ID 20868

Invalid symbol after EXIST.			
Description	An invalid character is programmed after the EXIST command. The cause can be the absence of the opening parenthesis immediately after the command, or an invalid character after this square parenthesis. Permissible signs directly after the opening square bracket are 'P' for parameter and 'V' for variable. A further possible cause of the error is the absence of the closing square bracket if a variable or parameter is unknown.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the square brackets and the spelling of the variable or parameter.
Error type	1, Error message from NC program.		

ID 20870

Unknown string during decoding of variable.

Description	When decoding a V.STR.xx variable, unknown character string elements in the designation were found.		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify the spelling of the V.STR.xx variables. The permissible variable names are defined in the machine data list (str_d*.lis).
Error type	1, Error message from NC program.		

ID 20872

CLAMP-Offset: Coordinate is out of data format.			
Description	The transfer of clamp position offset data takes place during the commissioning of an NC program. This determines that the data set to be transferred contains clamping offsets that lie outside the permitted data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the clamp position offset data (P-CLMP-00001)
Parameter	%1:	Logical axis number [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	-		

ID 20873

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20875

Unknown axis designation.			

Description	When using the commands #FACE and #CYL, an axis identifier has been programmed, which is either not present in NC channel or does not correspond to the permissible syntax.
	
	Examples fur incorrect handling of the commands:
	`N110 #FACE[X,J,Z]  (axis identifier J dies not exist)`
	`N110 #FACE[$,J,Z]  (" $ " is not a valid identifier)`
	`N110 #CYL[X,J]  (axis identifier J does not exist)`
	`N110 #CYL[$,C]  (" $ " is not a valid identifier)`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the designation of axis
Error type	1, Error message from NC program.		

ID 20876

Additional text behind axis designation is not allowed.			

Description	When using the commands #FACE, #CYL and #CAX, an invalid character is programmed after an axis name, or a syntax-relevant character is missing.
	
	Invalid examples:
	`N110 #CAX[SPDL,C`
	`N110 #FACE[X  C]`
	`N110 #FACE[X, C`
	`N110 #CYL [Z  C, X60]`
	`N110 #CYL [Z, C  X60]`		

Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify NC command. Either a comma or a closing square bracket is expected after the designation of the axis.
Error type	1, Error message from NC program.		

ID 20877

At the end of #CYL-command a ']' must be programmed.			
Description	When using the #CYL command, the closing square bracket is missing.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check NC command and complete the missing closing square bracket.
Error type	1, Error message from NC program..		

ID 20879

Axis name is already used.			
Description	When executing the command #CAX, it is detected that the default name of the C-axis is P-CHAN-00010 identical to the name of a already existing channel axis name. [PROG// Section: C axis machining mode 1...]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Renaming the default identifier P-CHAN-00010 for the C axis.
Error type	1, Error message from NC program.		

ID 20880

A comma must follow.			

Description	When programming the command #CAX, the separating comma is missing.
	
	Example with error:
	`#CAX[S C_axis]`
	
	Corrected example:
	`#CAX[S,C_axis]`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Insert a comma.
Error type	1, Error message from NC program.		

ID 20881

No free axis index available.			
Description	The axis configuration of the NC channel can not be extended by an additional new axis, because the maximum permissible number of axes in the NC channel has been reached.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Possibly release not needed axes to create free place for the new axes in NC channel.
Error type	1, Error message from NC program.		

ID 20882 / 20883

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20885

Both axes have axis mode C-axis.			
Description	When using the #FACE command, both programmed axis have the operating mode of C-axis. This is not permitted. Only one of the programmed axes may have this mode.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the two modes of the axes, change an axis mode P-AXIS-00015.
Error type	1, Error message from NC program.		

ID 20886

None of the programmed axes has axis mode C-axis.			
Description	When face or lateral surface machining with the command #FACE or #CYL, the axes programmed in the command are not configured for C axis mode. In this case, the cause may be an incorrectly configured axis type or an incorrect axis mode in the associated axis parameter lists.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check the configured axes in the command for their axis type P-AXIS-00018 and axis mode P-AXIS-00015 .
			Note: A C axis must be configured as follows:
			`kenngr.achs_typ 0x0002 or 0x0004`
			`kenngr.achs_mode 0x0204`

Error type	1, Error message from NC program.		

ID 20887

Unknown Face-ID was entered in channel parameters list.

Description	Using the #FACE command, the invalid entry P-CHAN-00008 was detected in the channel parameters.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value of the Face-ID P-wertCHAN-00008 in the channel parameters.
Parameter	%1:	Error value [-]	
		Invalid Face-ID	
Error type	1, Error message from NC program.		

ID 20888

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20889

Channel parameters: Face-ID is out of range of permissible values.			
Description	During start-up, the entry for the facing ID is checked for its valid value range. The value of the Face ID P-CHAN-00008 in the channel parameters list is out of range.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	Check and modify the value of the Face ID P-CHAN-00008 in the channel parameters before the next controller start-up. In case of conflict, the value of Face D is set to the maximum limit at start-up.
Parameter	%1:	Error value [-]	
		Invalid value of Face ID in channel parameter list	
	%2:	Upper limit value [-]	
		Maximal value of Face ID	
	%3:	Corrected value [-]	
		Value of Face ID automatically corrected	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20890

Selection has no effect.			
Description	Selecting the C-axis function has no effect, since the functionality has already been selected with identical parameters.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove the repeatedly programmed selection of the C axis function from the NC program.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20891

Deselection has no effect.			
Description	Deselection of the C-axis has no effect, because it's not enabled.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Delete the #CAX OFFcommand.
Error type	1, Error message from NC program.		

ID 20893

#CAX OFF not allowed during active transformation.			

Description	It is not permitted to deselect the C axis with the command #CAX OFF when the transformation is active.
	
	Example with error:
	`N100 #CAX`
	`N110 #CYL [Z, C, X60]`
	`N120 G00 G90 Z0 C0`
	`N130 G01 C100 F500`
	`N140 G02 Z100 R50`
	`N150 #CAX OFF`
	`N160 G01 C0`
	`N170 Z0`
	`N180 #CYL OFF`
	
	Corrected example:
	`N100 #CAX`
	`N110  #CYL [Z, C, X60]`
	`N110  G00 G90 Z0 C0`
	`N130 G01 C100 F500`
	`N140 G02 Z100 R50`
	`N150 G01 C0`
	`N160 Z0`
	`N170 #CYL OFF`
	`N180 #CAX OFF`
	
	[PROG// Section: Exchange spindles in coordinated motion]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Before deselecting the C axis (#CAX OFF), the active transformation (#CYL OFF, #FACE OFF) must be deselected in the NC program.
Error type	1, Error message from NC program.		

ID 20894

#FACE OFF has no effect.			
Description	Deselecting face machining has no effect because it is not enabled.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove the #FACE OFF command.
Error type	1, Error message from NC program.		

ID 20895

Type of kinematic not allowed at #FACE OFF.

Description	When using the #FACE OFF command, there's a type of kinematic active which is not permitted. Only the kinematics type is allowed for face machining in combination with #FACE OFF command. Example for causing the error: Selection of lateral surface machining with the command #CYL[…] and deselection with the #FACE OFF.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program. Remove change of kinematic type.
Parameter	%1:	Current value [-]	
		Actual active type of kinematic	
Error type	1, Error message from NC program.		

ID 20897

Plane change not allowed while machining on lateral surface.			
Description	A plane change is not allowed during lateral surface machining.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. Program a plane change after deselecting lateral surface machining.
Error type	1, Error message from NC program.		

ID 20899

Axis position cannot be calculated during geometry function is active.			
Description	The axis position cannot be determined with active geometry function (G61, G261, G302 or G301) in the turning centre.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program, leave the tool centre point before actualisation of the tool data.
Error type	1, Error message from NC program.		

ID 20900

#CYL OFF has no effect.			
Description	Deselecting lateral surface machining has no effect because it is not enabled.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove the command #CYL OFF.
Error type	1, Error message from NC program.		

ID 20901

Type of kinematic not allowed at #CYL OFF.

Description	When using the #CYL OFF command, there's a type of kinematic active which is not permitted. Only the kinematics type for lateral surface machining in combination with the #CYL OFF command is permitted. Example for causing the error: Selection of the facing mode with the command #FACE[…] and deselection with the #CYL OFF.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program. Remove change of kinematic type.
Parameter	%1:	Current value [-]	
		Actual active type of kinematic	
Error type	1, Error message from NC program.		

ID 20902

Invalid cylinder radius.			
Description	When using the #CYL command, a cylinder radius less than or equal to 0 is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the cylinder radius.
Parameter	%1:	Error value [-]	
		Programmed cylinder radius	
Error type	1, Error message from NC program.		

ID 20903

Negative axis position when starting #FACE.			
Description	With selection of face machining, the tool centre point is at a negative axis position. Negative axis position means that the tool tip is behind the centre of rotation. The front face machining is set via the channel parameter P-CHAN-00008. In milling machines, the entry should be "2" and in lathe machines "1". When the error occurs with milling machines the entry is to be examined in the channel parameter list. If an entry with the value 1 is selected on purpose for milling machines, correct the position of the tool tip in the NC program before transformation is selected so that it is positioned in front of the rotation centre. [FCT-S1//Section: C-axis machining (Facing and lateral surface machining)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the entry P-CHAN-00008 in the channel parameter list.
Parameter	%1:	Current value [-]	
		Adjusted mode of facing	
Error type	1, Error message from NC program.		

ID 20904

Invalid character in variable/ parameter declaration.			

Description	An invalid character is programmed in the declaration of variables or parameters.
	
	Example with error:
	`#VAR`
	`  V.L.Test[2]=[1;2]`
	`#ENDVAR`
	
	Corrected example:
	`#VAR`
	`  V.L.Test[2]=[1,2]`
	`#ENDVAR`
	
	The error may also result from an incorrect variable and/or parameter identifier when the drive parameters are read.		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the invalid syntax in the NC program.
Error type	1, Error message from NC program.		

ID 20905

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20906

Invalid index value.			
Description	An invalid index is used in the programming of variable V.xx.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. The programmed index must be inside the valid range.
Parameter	%1:	Error value [-]	
		Incorrectly programmed index.	
	%2:		
		Maximum permissible index	
Error type	1, Error message from NC program.		

ID 20907

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	

Solution	Class	8	Requires controller restart.

ID 20908

Programmed variable type must not be declared in NC-program.			
Description	The programmed variable type must not be declared in the NC program, only self-defined variables (V.L., V.S., V.P.) can be declared. [PROG// Section: Variables and calculation of variables]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 20909

Variable already exists.			
Description	It's not permitted to define a variable several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program. Remove superfluous variable definitions.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20910

No more free nodes available.			
Description	The maximum number of free nodes was exceeded. Free nodes are used for the declaration of variables.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. Reduce the number of self-defined variables.
Error type	1, Error message from NC program.		

ID 20912

Too many values during array initialization.			
Description	The number of registered values during initialization of the array is larger than the size of the array.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program. Adapt the number of values to the array size.
Error type	1, Error message from NC program.		

ID 20913

Error during read-access to a variable.	
Description	Error reading the variable. Error while reading the variable, caused by an simultaneous access to the variable for example by the PLC.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of the process flow. Avoid the simultaneous access to variables.
Error type	1, Error message from NC program.		

ID 20915

Deleting of non-user defined variables not allowed.			
Description	Deleting non-user-specific variables with the #DELETE command in the NC program is not permitted. Non user-specific variables include external (V.E.), global (V.G.) and axis-specific (V.A.) variables [PROG//Section: Variables and calculation of variables].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 20916

Invalid variable or parameter designation.			
Description	Then the function SIZEOF was used, an invalid identifier was programmed for a variable or a parameter within the square brackets. [PROG// Section: Arithmetic expressions.]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the SIZEOF command. The designation of a variable must start with a "V". The designation of a parameter must start with a "P".
Error type	1, Error message from NC program.		

ID 20917

Dimension is out of data format.			
Description	The value for the size of the array exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the array size.
Parameter	%1:	Error value [-]	
		Invalid size of array	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20918

Dimension does not exist.

Description	The dimension programmed in the SIZEOFfunction is not present in the corresponding parameter or variable array.
	Example:
	Wrong:
	:
	#VAR
	P20[3][4] = [40,41,42,43, 50,51,52,53, 60,61,62,63]
	#ENDVAR
	
	N10 P1 = SIZEOF [P20, **3**] -> **Dimension 3 does not exist!**
	N20 #MSG ["%d", P1]
	:
	N1000 M30
	
	Correct:
	:
	#VAR
	P20[3][4] = [40,41,42,43, 50,51,52,53, 60,61,62,63]
	#ENDVAR
	
	N10 P1 = SIZEOF [P20, **1**]
	N20 #MSG ["%d", P1] -> **Output value is 3**
	:
	N1000 M30		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Correct SIZEOF-access.
Parameter	%1:	Current value [-]	
		Unknown programmed dimension	
Error type	1, Error message from NC program.		

ID 20919

Parameter already exists.			
Description	A parameter or parameter array that is to be created already exists. It is not permitted to create multiple entries with the same name.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the declaration of the parameters.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 20920

Unable to create new parameter.			
Description	The parameter was not generated since the maximum number permissible is reached.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the number of used parameter in the NC program (#DELETE) or use the already existing parameters.
Parameter	%1:	Upper limit value [-]	
		Maximum number of parameters	
Error type	1, Error message from NC program.		

ID 20921

Parameter array was not declared.			
Description	The write access on an array parameter failed, because the associated parameter array was not ye defined. [PROG// Section: Parameters and parameter calculation]		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Before write access declare the parameter array via #VAR...#ENDVAR.
			
			Example:
			`#VAR`
			`  P5[20][20]  Declaration`
			`#ENDVAR`
			`...`
			`N10 P5[10][10] = 15  Write access`
			`...`

Error type	1, Error message from NC program.		

ID 20924

Too many indices programmed within parameter array.			
Description	The maximum number of permissible dimensions for the parameter array has been exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the number of dimensions of the parameter array.
Parameter	%1:	Current value [-]	
		Number of programmed dimensions of the parameter	
	%2:	Upper limit value [-]	
		Maximum number of dimensions	
Error type	1, Error message from NC program.		

ID 20925

Unknown parameter.			
Description	The read access to a parameter or parameter array failed. The cause for it is that this parameter generally does not exist or that the indexing of the programmed array parameter is different to the indexing of the defined array.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Declare parameter or parameter array respectively consider the correct indexing of the programmed parameter array.
Parameter	%1:	Error value [-]	
		Invalid parameter number	
	%2:	Error value [-]	
		Incorrect number of programmed dimensions (indices); value is only output with parameter arrays.	
	%3:	Current value [-]	
		Correct number of declared dimensions (indices); value is only output with parameter arrays.	
Error type	1, Error message from NC program.		

ID 20926

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20927

Parameter stack overflow.			
Description	The maximum nesting depth for parameters within the mathematical expression was exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the nesting level.
Parameter	%1:	Upper limit value [-]	
		Maximum nesting level	
Error type	1, Error message from NC program.		

ID 20930

Invalid character during deletion of variables/parameters.			
Description	An invalid character is programmed after the command #DELETE. [PROG// Section: Self-defined variables]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 20931

Programmed type of variable may not be deleted in NC program.			
Description	The programmed variable type may not be deleted in the NC program with the #DELETE command. These variable types include external (V.E.) and all static variables (V.G., V.A. ..) [PROG// Section: Variables and calculation of variables]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 20932

Variable does not exist.			
Description	The variable (V.L., V.P., V.S.) to be deleted with the #DELETE command does not exist.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	First define the variable in NC program.

Parameter	%1:	Current value [-]
Error type	1, Error message from NC program.	

ID 20933

Variable was not declared in the current subroutine level.			

Description	The V.L. Variable or V.CYC. variable to be deleted was not declared in the subroutine. It can only be deleted if it was declared in the subroutine.
	
	Invalid subroutine:
	`%L sub (subroutine)`
	`N500 F2000`
	`N510 #DELETE V.L.LOC_VAR`
	`N520 M17`
	
	Corrected subroutine:
	`%L sub (subroutine)`
	`N410 #VAR`
	`N420 V.L.LOC_VAR (Deklaration der Variablen)`
	`N430 #ENDVAR`
	`N500 F2000`
	`N510 #DELETE V.L.LOC_VAR`
	`N520 M17`
	
	Main program:
	`%test.nc`
	`N10 #VAR`
	`N20 V.L.LOC_VAR`
	`N30 #ENDVAR`
	`…`
	`N100 LL sub (call subroutine)`
	`N1010 G0 X0 Y0`
	`N1099 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of the subroutine in the NC program.
Error type	1, Error message from NC program.		

ID 20934

Parameter does not exist.			
Description	A P parameter that is to be read or deleted with the #DELETE command does not exist.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	First define the P-Parameter in NC program.
Error type	1, Error message from NC program.		

ID 20935 / 20938

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20939

Tool change for this block mode not possible.			
Description	A tool change with active C axis mode and active tool radius compensation is only possible with a preceding linear or circular block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 20940

Programmed spindle speed is out of speed ranges.			
Description	In the channel parameters, spindle gear changes are enabled (P-CHAN-00052). The programmed spindle speed cannot be covered by any of the speed ranges or gear stages defined in the channel parameters for this spindle (P-CHAN-00058, P-CHAN-00055).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Adapt the programmed spindle speed in the way, that it is within the defined speed ranges of the spindle. Alternatively, if it is (mechanically) possible to redefine the speed ranges in the channel parameters (P-CHAN-00058, P-CHAN-00055) so that the programmed spindle speeds can be executed.
Parameter	%1:	Current value [-]	
		Programmed spindle speed	
Error type	1, Error message from NC program.		

ID 20941

Wrong or undefined gear range programmed.			
Description	In the channel parameters, spindle gear change is activated (P-CHAN-00052). The programmed gear stage (M40-M45) does not match the spindle speed or the corresponding speed range is not defined in the channel parameter list (P-CHAN-00058, P-CHAN-00055) for this spindle.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. For the spindle speed the correct gear range according to the speed ranges in the channel parameter list (@@P-CHAN-00058, @@P-CHAN-00055) has to be programmed.
Parameter	%1:	Current value [-]	
		Programmed gear range	
Error type	1, Error message from NC program.		

ID 20942

Gear switching not permitted during active homing.			
Description	In the channel parameters, spindle gear change is activated (P-CHAN-00052). When selecting a new gear stage with G112 or M40 - M45, it is detected that homing (G74) is enabled in the same NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before selecting a new gear speed with G112 or M40 – M45, homing must be programmed and ended in a separate NC block.
Error type	1, Error message from NC program.		

ID 20943

Wrong or missing gear range.			
Description	In the channel parameters, spindle gear change is activated (P-CHAN-00052). The spindle speed was programmed in the NC block without a gear stage (M40-M45). So this spindle speed does not match to the currently active gear stage and also the automatic gear stage determination is disabled.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	The following remedial actions are possible: • the programmed spindle speed must match the gear stage already active, or • the automatic gear stage determination P-CHAN-00004 must be activated in the channel parameter list, or • the corresponding gear stage (P-CHAN-00058, P-CHAN-00055) must be programmed for the spindle speed.
Parameter	%1:	Current value [-]	
		Programmed spindle speed	
Error type	1, Error message from NC program.		

ID 20944

G-function not allowed during active C-axis machining.			
Description	A G function has been programmed that is not allowed while C axis machining is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Parameter	%1:	Error value [-]	
		Number of the invalid G function	
Error type	1, Error message from NC program.		

ID 20948 / 20949

System error [▶ 12]	
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.

Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 20953

Double programming of interruptible block.			
Description	Multiple programming of the interruptible block (G310) is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Parameter	%1:	Error value [-]	
		Number of the invalid G-command	
Error type	1, Error message from NC program.		

ID 20954

Circular movement not permitted for measuring.			
Description	Circular block (G02/G03) are not permitted as measuring motion blocks. Only linear blocks (G00/G01) are allowed as measuring motion blocks. [PROG// Section: Measuring functions]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 20955

NC command not permitted together with interruptible block.			
Description	The interruptible block (G310) contains NC commands that cannot be programmed at the moment.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 20956

Interruptible block not permitted for selected measuring type.			
Description	The function interruptible block (G310) with jump ($GOTO) after measurement run is not possible with the adjusted measuring type [PROG//Section: Measuring functions].		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Modification of NC program. Use the appropriate measurement type. Select the correct measuring type e.g. with…
			
			`Nxx #MEAS MODE[5]  (use measurement type 5)`
			`Nxx G310 X100 $GOTO Nxx  (measurement motion block)`
			`Nxx ...`
			
			...or configure the correct default measurement type in the channel parameters (P-CHAN-00057).

Parameter	%1:	Current value [-]
		Current active measuring type
Error type	1, Error message from NC program.	

ID 20958

Negative traverse path during relative modulo programming not permitted.			
Description	With relative programming and explicit indication of the direction of rotation, the use of negative coordinates is not permissible with modulo axes. [PROG// Section: Programming modulo axes]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Error type	1, Error message from NC program.		

ID 20959

Selected synchronisation mode for this M-function not permitted.			
Description	In connection with axis clamping, the synchronisation type currently assigned in the channel parameters (P-CHAN-00041) for the M function M10 or M11 [PROG] is not permitted. For further information see [FCT-C1]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	In the channel parameters (P-CHAN-00041) for M10 or M11, enter the type of synchronization required for axis clamping. • **M10** (select axis clamping) must be of the **MNS_SNS** type. • **M11** (deselect axis clamping) must be of the **MVS_SVS** type. Then a controller start-up of the control or a reload of the channel parameters is necessary.
Parameter	%1:	Current value [-]	
		Number of the M function with invalid synchronisation type	
Error type	1, Error message from NC program.		

ID 20961

Interpolation parameter was programmed for a tracking axis.			
Description	During circular programming one of the interpolation parameters I, J, K was assigned to a tracking axis.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check axis configuration or previous axis exchange commands in the NC program.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 20962

Minimum radius is out of data format.			
Description	The value of the programmed minimum radius in the NC command for adapting the tangential feed #TANGFEED[...] exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify value of the minimum radius in NC command.
Parameter	%1:	Error value [-]	
		Invalid minimum radius	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 20963

Negative contour radius not permitted.			
Description	Programming a negative minimum value for the contour radius in the command for tangential feedrate adjustment #TANGFEED[...] is not permissible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Parameter	%1:	Current value [-]	
		Invalid value of radius	
Error type	1, Error message from NC program.		

ID 20965

Channel parameters: Synchronisation mode not permitted.			
Description	The used synchronisation mode is not permitted for the M-function.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the synchronization mode of the M function P-CHAN-00041 in the channel parameters before the next controller start-up. During start-up in case of conflict the invalid synchronisation mode is set to "MVS_SVS" and the start-up is continued.
Parameter	%1:	Current value [-]	
		Index of the invalid M- and/or spindle function	
	%2:	Current value [-]	
		Invalid synchronisation mode	
	%3:	Corrected value [-]	
		Corrected synchronisation mode	
Error type	2, Error message by data transfer from parameter list into control device. .		

ID 20966

Dimension values less than 1 not allowed.			
Description	It is not possible to declare an array variable with a dimension value smaller than 1.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Correct the dimension value.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 20967

Memory for variables limited, allocation of new variable impossible.			
Description	The new variable cannot be created because no memory is available for this variable.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce number of variables or size of arrays.
Parameter	%1:	Current value [-]	
		Number of used places for variables of this class	
	%2:	Upper limit value [-]	
		Permissible number of variables of this class	
	%3:	Class [-]	
		Identification of the variable class: 1 -> V.P. 2 -> V.S. 3 -> V.E. 4 -> V.L.	
	%4:	Current value [-]	
		Minimum number of necessary free spaces for the new variable of this class (relevant for array variables only)	
Error type	1, Error message from NC program.		

ID 20971

Channel parameters: Spindle designation does not start with an S.			
Description	The spindle designation in the channel parameters does not start with the letter “s”. This is not permitted since all spindle designations in the channel parameters must start with the letter “s”.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the spindle designations P-CHAN-00007 in the channel parameters before the next controller start-up. During start-up in case of conflict the invalid spindle name is set to "S" and the start-up is continued.
Parameter	%1:	Error value [-]	
		Invalid spindle name	
	%2:	Corrected value [-]	
		Corrected spindle name	
	%3:	Logical axis number [-]	
		Logical axis number of the invalid spindle	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 20972

Variable arrays must be declared within a #VAR ... #ENDVAR block.			

Description	The declaration of a variable array must be made in a #VAR...#ENDVAR block.
	
	Incorrect part of the program:
	`V.P.NEW_VAR_1[6]=[1,2,3,4,5,6]`
	
	Corrected part of the program:
	`#VAR`
	`  V.P.NEW_VAR_1[6]=[1,2,3,4,5,6]`
	`#ENDVAR`
	
	[PROG//Section: Self-defined variables]		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 20973

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 20974

Too many indices in variable array programmed.			
Description	The programmed variable array exceeds the maximum limit of indices/ dimensions.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Parameter	%1:	Current value [-]	
		Number of used indices	
	%2:	Upper limit value [-]	
		Number of maximal limit of indices	
Error type	1, Error message from NC program.		

ID 20977

Double programming of deceleration adjustment.			
Description	The deceleration adjustment was programmed several times in the same NC block. It is not permissible to program commands G09, G900 or G901 combined in one NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Parameter	%1:	Error value [-]	

Error type	1, Error message from NC program.

ID 20978

Term in logical operation is out of data format.			
Description	The partial result of a term or the value of an individual operand exceeds the permissible numerical range when checked.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. Verify and correct the term or operand. With larger terms, it is recommended to split them into smaller terms and program them in several NC blocks.
Parameter	%1:	Error value[-]	
		Invalid value of the term	
	%2:	Lower limit value [-]	
		Minimum value of the term	
	%3:	Upper limit value [-]	
		Maximum value of the term	
Error type	1, Error message from NC program..		

ID 20980

Indirect parameter access not possible.			
Description	Access via an indirect parameter "Pxx" cannot be executed because this parameter is not initialised in the value table.. [PROG// Section: Indirect parameters].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The parameter table must be previously initialised with the R word.
Error type	1, Error message from NC program.		

ID 20981

Arrays are not available for R-parameters.			
Description	When programming indirect parameter, the definition and use of parameter arrays are not possible. Example: **Wrong:** : #VAR **R20[3][4] = [40,41,42,43, 50,51,52,53, 60,61,62,63]** #ENDVAR : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Note the restrictions when using indirect parameters.
Error type	1, Error message from NC program.		

ID 20984

Double programming of gear switch commands.			
Description	In the channel parameters, spindle gear change is activated (P-CHAN-00052). G112 and a spindle speed with one gear stage (M40-M45) are programmed simultaneously in the NC block. Example with error: `Nxx G112 S1000 M42`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program gear switching of the spindle either with G112 or M40-M45.
Error type	1, Error message from NC program.		

ID 20985

Function is only available in absolute programming.			
Description	The #SUPPRESS OFFSETS command can only be used with absolute programming (G90) [PROG//Section: Suppressing offsets]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Parameter	%1:	Logical axis number [-]	
		Logical number of the axis, which is in relative mode (G90).	
Error type	1, Error message from NC program.		

ID 20986

Function not allowed for clampable axes.			
Description	One of the axes involved in synchronous operation (master or slave) is a so-called "clampable axis". This property is not permissible for synchronous axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	The property "Clampable axis - ACHSMODE_CLAMPABLE" must be disabled in the corresponding axis list in parameter P-AXIS-00015 (achs_mode). Then a controller restart or a reload of the axis parameters is necessary.
Parameter	%1:	Logical axis number [-]	
		Axis number of the master axis	
	%2:	Logical axis number [-]	
		Axis number of slave axis.	
Error type	1, Error message from NC program.		

ID 20990

Kinematics change for new tool with inactive kinematic transformation only.			
Description	A change of the machine kinematics with the command #KIN ID is not possible while kinematic transformation is active [PROG//Section: Machine kinematics (#KIN ID)].		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Modify the NC program. Solution proposed: Deselect the kinematic transformation before changing the machine kinematic.
Parameter	%1:	Current value [-]	
		Currently active KIN-ID	
	%2:	Current value [-]	
		KIN-ID of the new tool	
Error type	1, Error message from NC program.		

ID 20991

Negative spindle speed not permitted.			
Description	The spindle speed in spindle-specific programming S[...] is negative. This is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correction of the invalid spindle speed.
Parameter	%1:	Error value [1µm/s or 0.001°/s]	
		Invalid spindle speed	
Error type	1, Error message from NC program.		

ID 20994

Syntax error in #SIGNAL- or #WAIT-command.			
Description	There's a syntax error in the command #SIGNAL and/or #WAIT, e.g. not allowed character, unknown keyword… [PROG// Section: Sending signals], [PROG// Section: Waiting for signals]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of the NC program.
Error type	1, Error message from NC program.		

ID 20995

Signal number is out of data format.			
Description	The signal number (ID) within the #SIGNAL or #WAIT command is outside the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the incorrect signal number in the NC program.
Parameter	%1:	Current value[-]	
		Incorrect signal number	
	%2:	Lower limit value[-]	
		Minimum value of the signal number	
	%3:	Upper limit value[-]	
		Maximum value of the signal number	
Error type	1, Error message from NC program.		

ID 20996

Channel number is out of data format.			
Description	A channel number (CH) in the #SIGNAL or #WAIT command is outside the permitted value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the incorrect channel number in the NC program.
Parameter	%1:	Current value [-]	
		Incorrect channel number	
	%2:	Lower limit value [-]	
		Minimum value of the channel number	
	%3:	Upper limit value [-]	
		Maximum value of the channel number	
Error type	1, Error message from NC program.		

ID 20997

Too many channel numbers programmed.			

Description	The number of permissible channel numbers within the square brackets was exceeded when using the #SIGNAL or #WAIT command .
	
	Suggested solutions:
	Reduction of channel numbers within the square brackets or splitting the channel numbers in several commands
	
	Example with error:
	`#SIGNAL [ID44 CH2 CH3 CH4 CH5 CH6 CH7 CH8 CH9 CH10 CH11 CH12 CH13 CH14]`
	
	Corrected example:
	`#SIGNAL [ID44 CH2 CH3 CH4 CH5 CH6 CH7 CH8]`
	`#SIGNAL [ID44 CH9 CH10 CH11 CH12 CH13 CH14]`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Parameter	%1:	Upper limit value [-]	
		Number of programmed channel numbers	
	%2:	Upper limit value[-]	
		Limit of channel numbers which can be programmed	
Error type	11, Error message from NC program.		

ID 20998

After #SIGNAL or #WAIT the keyword SYN or an open bracket must follow.			
Description	After the commands #SIGNAL and/or #WAIT there must be either the keyword SYN or an open square bracket. [PROG// Section: Sending signals] [PROG// Section: Waiting for signals]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the #SIGNAL or #WAIT command.

Parameter	%1:	State [-]
Error type	1, Error message from NC program.	

ID 20999

Invalid weighting of ramp time.			
Description	The ramp time weighting programmed with G133/G134 is negative. [PROG// Section: Ramp time weighting (G132/G133/G134)] Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G133 = **-60** (alternatively: G133 -60) : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G133 = **60** (alternatively: G133 60) : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	In the event of a conflict, ramp time weighting is automatically set to 100%, and NC program processing is continued. Before the next program start, a meaningful value greater than zero (nil) should be programmed.
Parameter	%1:	Error value [µs]	
		Incorrect weighting of ramp time	
	%2:	Corrected value [-]	
		Automatically corrected value of weighting of ramp time	
Error type	1, Error message from NC program.		

2.3.5 ID-range 21000-21249

ID 21000

Thread tapping in combination with spindle movement not allowed.			
Description	It is not possible to use a spindle M function (M3, M4, M5, M19, S.POS) during tapping [PROG//Section: Tapping (G63)]. Since thread tapping (G63) is modal, another modal G-function (G0, G1, …) must be programmed before the next spindle M-function.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program..		

ID 21001 - 21019

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21020

The axis, received in the EF decoder, is a GANTRY slave axis.			
Description	The slave axis of a gantry combination may not be configured as an axis in a channel.		
Response	Class	3	Abort current job. Start-up is aborted.
Solution	Class	7	Remove the slave axis from the path compound.
Parameter	%1:	Logical axis number [-]	
		Logical number of slave axis	
	%2:	Current value [-]	
		Optional parameter, slave axis attributes	
Error type	-		

ID 21022

Writing of a variable by active kinematic transformation not allowed.			
Description	It’s not possible to change the kinematic parameters of the current tool with active kinematic transformation via V.G.WZ_AKT.KIN_PARAM[..]. [PROG// Section: Global variables (V.G.)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Change the machining sequence in the NC program. Only change kinematic tool data after deselecting a kin. transformation.
Error type	1, Error message from NC program.		

ID 21023

SIZEOF access not permitted for this variable.			
Description	The SIZEOF access to indexed global variables (V.G., e.g. V.G.WZ_AKT.V[i] or V.G.WZ[j].[P[i] is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program. With SIZEOF, only self-defined variable arrays (V.L., V.P., V.S.) may be accessed.
Error type	1, Error message from NC program.		

ID 21024

Circle centre point and radius are not permitted at the same time.			

Description	Circle programming is done either by defining the circle centre point or by programming the radius.
	
	Example with error:
	`%err_21024.err`
	`N10 G00 X0 Y0 Z0 G17`
	`N20 G01 X10 Y100 F10000`
	`N30 X100Y200`
	`N40 X200`
	`N50 G02 X200 I50 J50 R50`
	`N60 G01 X400 Y0`
	`N99 M30`
	
	Corrected example:
	`%err_21024.kor`
	`N10 G00 X0 Y0 Z0 G17`
	`N20 G01 X10 Y100 F10000`
	`N30 X100Y200`
	`N40 X200`
	`N50 G02 X200 I50 J50`
	`N60 G01 X400 Y0`
	`N99 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 21025 / 21026

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21027

3D tool geometry compensation is already active. Renewed call not allowed.

Description	With active 3D tool geometry compensation in NC program, #TGC ON is programmed again. Example: **Wrong:** N10 #TGC ON : **N.. #TGC ON** : N.. #TGC OFF : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove redundant #TGC ON.
Error type	1, Error message from NC program.		

ID 21028

Tool change is not permitted if 3D tool geometry compensation is active.			
Description	With active 3D tool geometry compensation (TGC) in NC program, a tool change is programmed. Example: **Wrong:** N10 D1 **N20 #TGC ON** **N.. D2** N40 #TGC OFF **Correct:** N10 D1 N20 #TGC ON : **N.. #TGC OFF** **N.. D2** N.. #TGC ON : N.. #TGC OFF : M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Tool change may only be executed during deselected 3D tool geometry compensation (#TGC OFF).
Error type	1, Error message from NC program.		

ID 21029

Write access to tool data is not allowed during 3D-WGK is active.			
Description	With active 3D tool geometry compensation (TGC) in NC program, a write access via V.G.WZ_AKT.* on current tool data (length, radius, axes offsets) is programmed. Example: **Wrong:** N10 #TGC ON : **N.. V.G.WZ_AKT.R = 50** : N.. #TGC OFF : N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Write access on current tool data is only permissible during deselected 3D tool geometry compensation (#TGC OFF).
Error type	1, Error message from NC program.		

ID 21035

Unknown NC-command in spindle sequence.			
Description	A programmed command is invalid in conjunction with spindle-specific programming [PROG//Section: Spindle specific programming].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 21036

Double programming of the spindle speed.			
Description	It is not possible to assign a spindle speed more than once within a block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Parameter	%1:	Current value [-]	
		Last programmed spindle speed	
Error type	1, Error message from NC program.		

ID 21037

Channel parameters: Invalid log. axes number of the main spindle.			
Description	A spindle axis with the logical axis number of the main spindle is not available in the NC channel. The error message can appear during start-up.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the logical axis number of the main spindle P-CHAN-00051 before next controller start-up in the channel parameters. During start-up, in case of conflict, the logical axis number of the first spindle P-CHAN-00036 is assigned to the main spindle, and the start-up continues.
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21038

Tool dynamic data for an unknown spindle.			
Description	A spindle that does not exist in the channel is assigned to the dynamic data of the requested tool.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Modify tool data (P-TOOL-00012)
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the assigned spindle	
Error type	1, Error message from NC program.		

ID 21039

NC-command in this context not permitted.			
Description	In combination with the used NC command a programmed keyword is not allowed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Parameter	%1:	State [-]	
Error type	1, Error message from NC program.		

ID 21040

Double programming in #HSC-command.			

Description	In the command #HSC, invalid combinations of keywords were programmed.
	Example with error:
	`#HSC ON[CONTERROR 1, OPMODE 2, OPMODE 1]`
	or
	`#HSC ON[CONTERROR 1, OPMODE 2, CONTERROR 0.1]`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of #HSC command in the NC program.
Error type	1, Error message from NC program.		

ID 21041

Invalid HSC mode.			
Description	In the #HSC […] command the keyword OPMODE has an invalid value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. In the #HSC command, assign a valid value to the machining mode.
Parameter	%1:	Error value [-]	
		Invalid HSC mode	
Error type	1, Error message from NC program.		

ID 21042

Logical axis number is out of data format.			
Description	The spindle programming is programmed with a logical spindle axis number which exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command.

Parameter	%1:	Error value [-]
		Incorrect logical axis number of the spindle. P-AXIS-00016,
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21043

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 21044

DIN-syntax or programming only for main spindle.			
Description	The DIN syntax is only permitted for the main spindle. The spindle used is not the main spindle [PROG//Section: Spindle programming].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. 1st possibility : Change the main spindle with #MAIN SPINDLE 2nd possibility: Use spindle-specific syntax for the spindle to be programmed.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the programmed spindle	
	%2:	Logical axis number [-]	
		Logical axis number of the main spindle	
Error type	1, Error message from NC program.		

ID 21045

Channel parameters: Main spindle name used several times.

Description	The designation of a spindle P-CHAN-00007 is identical to the name of the main spindle P-CHAN-00053. This is not permitted.
	Example with error in channel parameter list:
	`# Spindle data`
	`spdl_anzahl 2`
	`#`
	`main_spindle_ax_nr 6`
	`main_spindle_name B`
	`...`
	`spindel[0].bezeichnung S`
	`spindel[0].log_achs_nr 6`
	`...`
	`spindel[1].bezeichnung S2`
	`spindel[1].log_achs_nr 11`
	Corrected example:
	`# Spindle data`
	`spdl_anzahl 2`
	`#`
	`main_spindle_ax_nr 6`
	`main_spindle_name`
	`...`
	`spindel[0].bezeichnung S1`
	`spindel[0].log_achs_nr 6`
	`...`
	`spindel[1].bezeichnung S2`
	`spindel[1].log_achs_nr 11`		

Response	Class	1	Start-up of the control is aborted.
Solution	Class	1	Modification of channel parameter list.
Parameter	%1:	Current value [-]	
		Designation of the main spindle	
	%2:	Logical axis number [-]	
		Logical axis number of the spindle with the same name	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21046

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21047

Unknown spindle name or close bracket missing..			
Description	While changing the main spindle with the command #MAIN SPINDLE, either the identifier of the spindle is unknown or the closing square bracket is missing.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Error type	1, Error message from NC program.		

ID 21050

Changing the main spindle not allowed during active turning functions.

Description	A change in the main spindle (#MAIN SPINDLE) is programmed while turning functions are active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the sequence of the NC program, for example deselect the turning function before changing the main spindle.
Error type	1, Error message from NC program.		

ID 21056

Double programming of signal number.			
Description	In the commands #SIGNAL or #WAIT, the signal ID has been programmed several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 21057

Too many parameters programmed in signal command.		
Description	The maximum number of permissible parameters when waiting for a signal (#WAIT) or sending a signal (#SIGNAL) was exceeded.	
Response	Class	2 Abort NC program processing.
Solution	Class	3 Modify the NC program.
Parameter	%1:	Limit value [-]
		Maximum permissible number of parameters
Error type	1, Error message from NC program.	

ID 21058

No signal number was programmed.			
Description	In the #SIGNAL or #WAIT commands, the signal ID is missing in the square brackets.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 21059

Parameter is already used in this command.			
Description	It is not possible to use a parameter several times within the #SIGNAL command. Example with error: `#SIGNAL[ID44 P[0]=200 P[0]=944]`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Parameter	%1:	Current value [-]	

Error type	1, Error message from NC program.

ID 21060

No signal parameter available.			
Description	The number of the expected parameters in the #WAIT command does not match the number of the sent parameters in #SIGNAL .		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the parameters in the commands #WAIT and #SIGNAL.
Error type	1, Error message from NC program.		

ID 21061

HSC parameter is out of data format.		
Description	The programming of additional HSC parameters is out of data range.	
Response	Class 2	Abort NC program processing.
Solution	Class 3	Modify the NC program.
Parameter	%1:	Error value [-]
		Value of the additional HSC parameter
	%2:	Lower limit value [-]
		Minimum limit of the parameter
	%3:	Upper limit value [-]
		Maximum limit of the parameter
Error type	1, Error message from NC program.	

ID 21067

Redundant information programmed.			
Description	In axis-specific programming, redundant information is programmed. Example with error: "POS" shows such redundant information: `N10 X10 Y11 Z[INDP_SYN POS50 POS40 G01 FEED100 G90]`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 21068

Syntax error when programming an independent axis.

Description	When programming independent axes, a syntax error occurred when using the G command within the brackets. The G command used does not exist.
	
	Example with error:
	`N10 X10 Y11 Z[INDP_SYN G03 FEED100 G90]`
	
	Corrected example:
	`N10 X10 Y11 Z[INDP_SYN G01 FEED100 G90]`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 21071

Independent axes during active transformation not allowed.			
Description	While a kinematic or Cartesian transformation is active, an independent axis motion is programmed for a transformation axis.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the sequence of the NC program, for example deselect the transformation before programming the independent axis. The motion of the independent axis may not influence the transformation. **Independent axes during active transformation not allowed.** Multiple ID 1 enables a kinematic transformation. Multiple ID 2 enables a Cartesian transformation.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21072

No independent movement of circle axes in the circle block.			
Description	The first two main axes may not be programmed as independent in a circular block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 21076

GANTRY limit is out of data format.			
Description	The value of a programmed monitoring limit for gantry mode is out of range. [PROG// Section: Extended programming of axis couplings]		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Correct the monitoring limit. **GANTRY limit is out of data format.** For first monitoring limit, multiple ID 1 is indicated. For second monitoring limit, multiple ID 2 is displayed.
Parameter	%1:	Error value [-]	
		Invalid monitoring limit	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21077

Scaling factor is 0.			
Description	In the axis coupling command, the numerator (<numerator>) required for defining a coupling factor (scale factor) is assigned the value 0. [PROG// Section: Extended programming of axes couplings]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check axis coupling command and assign the <numerator> a meaningful value.
Parameter	%1:	Current value [-]	
		Invalid value of numerator	
Error type	1, Error message from NC program.		

ID 21078

File to write the active axis couplings was not generated.			
Description	The log file (P-CHAN-00355) was not generated since it already exists and is open [PROG//command #SET AX LINK].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Close the open log file.
Error type	1, Error message from NC program.		

ID 21079

No synchronous operation active. Deselection has no effect.			
Description	The deselection of an axis couplings has no effect, since no active coupling operation is present.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Remove command #DISABLE AX LINK.
Error type	1, Error message from NC program.		

ID 21080

No file name defined in the channel parameters for backup.

Description	In the channel parameter list in P-CHAN-00355, there is no file name defined for saving the data.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Definition of backup file name in the channel parameter list.
Error type	1, Error message from NC program.		

ID 21081

Coupling command is too long for saving in file.			
Description	The number of characters used in the coupling command is too large to save the log file.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the number of characters.
Parameter	%1:	Upper limit value [-]	
		Number of characters used in the coupling command	
	%2:		
		Upper limit of characters	
Error type	Abort NC program processing.		

ID 21083

In current state no independent axes may be programmed.			
Description	Programming an independent axis is not possible at this time because spline interpolation or a turning function is enabled [PROG//Section: Positioning axes]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Parameter	%1:	Current value [-]	
Error type	Abort NC program processing.		

ID 21085

Clampable axes are not permitted to move independently.			
Description	An axis moved as a positioning axis (e.g. independent axis, oscillating axis) is a so-called "clampable axis". This property is not permissible for positioning axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	The property "Clampable axis - ACHSMODE_CLAMPABLE" must be deactivated in the corresponding axis list [AXIS] in parameter P-AXIS-00015 (achs_mode). Then a controller restart or a reload of the axis parameters is necessary.
Error type	1, Error message from NC program.		

ID 21086

Maximum number of M and/or H functions per command exceeded.

Description	The permissible number of M and H functions with axis- or spindle-specific programming (e.g. X[.....]) was exceeded. , S[.....]). What is decisive is the sum of M and H functions.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the M and H functions of the axis.
Parameter	%1:	Upper limit value [-]	
		Maximum possible number of permitted M and H functions	
Error type	Abort NC program processing.		

ID 21087

Value is out of data format.			
Description	The value of the data item is outside the permitted range of the corresponding data format.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the value.
Parameter	%1:	Error value [-]	
		Actual value	
	%2:	Lower limit value [-]	
		Lower limit value	
	%3:	Upper limit value [-]	
		Upper limit value	
Error type	Abort NC program processing.		

ID 21088

Declaration of decimal places is out of data format.			
Description	When programming SERCOS parameter with #IDENT, the parameter for the number of places after the decimal point “DEC” exceeds the limit of the permissible data format.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the parameter for the number of places after the decimal point.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	Abort NC program processing.		

ID 21089

Double programming in #IDENT-command.			
Description	In the command #IDENT, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of the #IDENT command in the NC program.

Error type	1, Error message from NC program.

ID 21090

Error in ident number.			
Description	The specified ID number of the SERCOS drive in the #IDENT command is not correct. Examine the ID number. Further diagnosis can be carried out with the aid of the documentation provided by the drive manufacturer.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Examination of the ident number.
Error type	1, Error message from NC program.		

ID 21091 - 21093

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21094

Fatal SERCOS error.			
Description	In the SERCOS drive arose an error which can not be assigned.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Consult the documentation of the drive manufacturer.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
Error type	Abort NC program processing.		

ID 21095

SERCOS-Parameter: Unknown SERCOS ID.			
Description	The SERCOS ID used in the #IDENT or #COMMAND command is unknown.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Examination and correction of the SERCOS-ID.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		Wrong ident number	
Error type	1, Error message from NC program.		

ID 21096

SERCOS parameter: Data size too short.			
Description	The parameter specified for the data length in the #IDENT command is too small. The entry is after the keyword “TYP”.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Correct the incorrect parameter.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21097

SERCOS parameter: Data size too long.			
Description	The parameter specified for the data length in the #IDENT command is too large. The entry is after the keyword “TYP”.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Correct the incorrect parameter.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21098

SERCOS parameter: Data item cannot be changed.			
Description	The data item cannot be changed. Possibly the parameter is only changeable in a certain phase during start-up of the SERCOS drive. For more detailed information, refer to the documentation of the drive manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Consult the documentation of the drive manufacturer.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21099

SERCOS parameter: Data item currently write-protected.	
Description	This data item is write-protected. To change this data item, first cancel write protection. For more detailed information, refer to the documentation of the drive manufacturer.

Response	Class	3	Abort NC program processing.
Solution	Class	3	Consult the documentation of the drive manufacturer.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21100

SERCOS-Parameter: Date shorter than min. value.			
Description	SERCOS drive data blocks are transmitted during the cyclic data communication of the . These contain names, attribute, unit, maximum and minimum input value and the date. When this error occurs, the input value of date is smaller than the permissible minimum value of date. More detailed informations you can get from the documentation of the drive manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Consult the documentation of the drive manufacturer.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21101

SERCOS parameter: Data item greater than maximum value.			
Description	SERCOS drive data blocks are transferred during the cyclic data communication of the SERCOS drive. These contain names, attribute, unit, maximum and minimum input value and the data item. When this error occurs, the input value of the data item is greater than the permissible maximum value of data item. For more detailed information, refer to the documentation of the drive manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Consult the documentation of the drive manufacturer.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21102

SERCOS-Parameter: Date incorrect.			
Description	SERCOS drive data blocks are transmitted during the cyclic data communication of the . These contain names, attribute, unit, maximum and minimum input value and the date. When this error occurs, the input value of date is incorrect. More detailed informations you can get from the documentation of the drive manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Consult the documentation of the drive manufacturer.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21103

Other errors during transfer in service channel.			
Description	Collective error message for general SERCOS errors.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Consult drive manufacturer's documentation.
Parameter	%1:	Current value [-]	
		Error message of the service channel	
	%2:	Identification number [-]	
		SERCOS ident number	
Error type	1, Error message from NC program.		

ID 21104

NC-command is not complete.			
Description	The NC command is incomplete; the necessary keywords or values were not programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check NC command and supplement the missing keywords or values.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21105

Unknown drive type.			
Description	An unknown drive type was used in the NC command #IDENT or #COMMAND.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	At the moment, the NC command may be used only with SERCOS drives. I.e. only the keyword SERC is permitted [PROG].
Error type	1, Error message from NC program.		

ID 21106

Axis identifier or NC command not permissible.			
Description	An impermissible axis identifier or an impermissible control word was used in the NC command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC program.
Error type	1, Error message from NC program.		

ID 21107

Syntax error during axis programming.			
Description	A syntax error was found in the command sequence for axis-specific programming (e.g. X[.....]).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axes specific command sequence. Syntax errors for example are: Double programming Excluding keywords Missing syntax elements; open brackets, close brackets
Error type	1, Error message from NC program.		

ID 21110

Denominator of the scaling factor is missing.			
Description	In the axis coupling command, the denominator (<denominator>) required to define a coupling factor (scale factor) is not programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check axis coupling command and assign missing <denominator>.
Parameter	%1:	Current value [-]	
		Value of assigned numerator.	
	%2:	Error value [-]	
		Default value of the missing denominator.	
Error type	1, Error message from NC program.		

ID 21111

Invalid scaling factor.			
Description	The coupling factor (scale factor) defined in the axis coupling command by <numerator><denominator> results in an invalid value.		
Response	Class	1	Continue NC program processing.

Solution	Class	1	The invalid coupling factor is implicitly set to 1. The coupling is handled like a standard coupling. At the moment, factors that produce pure scaling or scaling with simultaneous mirroring (-1<factor <0 or factor > -1) are not permissible. Permitted coupling factors resulting from <numerator><denominator> are: -1 Mirror coupling 1 Standard coupling
Parameter	%1:	Current value [-]	
		Value of numerator	
	%2:	Current value [-]	
		Value of denominator	
	%3:	Error value [-]	
		Invalid coupling factor	
Error type	1, Error message from NC program.		

ID 21120

Double programming of counter parameter.			
Description	The count parameter (COUNT) is programmed several times in the #SIGNAL command for sending signals.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the signal command and remove the redundant COUNT.
Error type	1, Error message from NC program.		

ID 21121

Syntax errors: Programming of this ID range not permitted.			
Description	No ID range (IDMIN, IDMAX) may be programmed in the #SIGNAL REMOVE command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check signal command, remove IDMIN, IDMAX and replace with a single ID... The ID range is only specific-application available, and only useful if the COUNT keyword in the command #SIGNAL is also allowed. Please inform manufacturer of the control.
Error type	1, Error message from NC program.		

ID 21122

Syntax errors: Double programming of ID range.			
Description	In the command to delete signals #SIGNAL REMOVE the upper limit of the ID range (IDMAX) is programmed several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the signal command and remove the redundant IDMAX.
Error type	1, Error message from NC program.		

ID 21123

Syntax error: Counter parameter not permitted.			
Description	No counting parameter (COUNT) may be used in the #SIGNAL command for sending signals.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check signal command and remove COUNT. The COUNT count parameter is only available for specific applications. Please inform controller manufacturer.
Error type	1, Error message from NC program.		

ID 21126

At current axis position kinematic transformation is undefined.			
Description	Some kinematic transformations depend on very exact input coordinates for the calculation. The check is used especially for machines with very high safety requirements, for example robots used in medical systems.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Check the input variables of the kinematic transformation. Increase degree of accuracy.
Parameter	%1:	Current value [-]	
		Kinematic ID of the active transformation	
	%2:	Logical axis number [-]	
		Logical number of the axis whose permitted deviation between the forward and backward transformation is too large.	
	%3:	Current value [-]	
		Index of the axis whose permitted deviation between the forward and backward transformation is too large.	
	%4:	Expected value [0.1 µm or 0.0001°]	
		Expected position value of the axis	
	%5:	Current value [0.1 µm or 0.0001°]	
		Calculated incorrect position value of the axis	
Error type	1, Error message from NC program.		

ID 21128

Channel parameters: Channel specific override exceeds limit.			
Description	During start-up, a check of the channel parameters detected that P-CHAN-00056 exceeds the permissible limit values.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In the event of a conflict, P-CHAN-00056 is assigned a limit value during start-up and the start-up is continued.

Parameter	%1:	Error value [-]
		Incorrect parameter value
	%2:	Upper limit value [-]
		Permissible limit value **Note:** With multiple ID 1 the maximum limit is indicated. With multiple ID 2 the minimum limit is indicated.
	%3:	Corrected value [-]
		Parameter value automatically corrected **Channel parameters: Channel specific override exceeds limit.** With multiple ID 1 the parameter is set on the maximum limit. With multiple ID 2 the parameter is set on the minimum limit.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 21129

M02/M30 is missing.			
Description	At the end of the main program, no M02 or M30 is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program M02 or M30 at the end of the main program.
Error type	1, Error message from NC program.		

ID 21130

Missing M17/M29.			
Description	At the end of a global subroutine, no M17 or M29 is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program M17 or M29 at the end of the global subroutine.
Error type	1, Error message from NC program.		

ID 21131

Axis or spindle for axis specific M/H-function unknown.			
Description	The axis or spindle of an M/H function assigned by programming (e.g. X[M20], S[M25]) or by channel-specific parameterization P-CHAN-00039, P-CHAN-00025 is missing in the channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify axis specific M/H programming and/or channel-specific default parametrization (P-CHAN-00039, P-CHAN-00025). During program execution, the corresponding axes may not have left the NC channel by axis exchange commands (#PUT AX).
Parameter	%1:	Current value [-]	
		Number of the axis specific M/H-function	
Error type	1, Error message from NC program.		

ID 21132

Double programming of spindle override.			
Description	Error message occurs when programming the spindle override (G167). The override is programmed several times in the NC block for one spindle.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the spindle programming within NC-block. Remove redundant G167 commands.
Error type	1, Error message from NC program.		

ID 21134

Syntax error when programming tracking of an axis.			
Description	A syntax error (e.g. multiple programming, unknown keyword etc.) was detected in the command for axis tracking (#CAXTRACK).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the command for axis tracking.
Error type	1, Error message from NC program.		

ID 21135

Parameter of axis tracking is out of data format.		
Description	In the #CAXTRACK command, one of the programmed parameters (ANGLIMIT, OFFSET, ANGPOS) exceeds the permitted numerical range.	
Response	Class	2 Abort NC program processing.
Solution	Class	3 Check and modify the command for axis tracking.
Parameter	%1:	Error value [-]
	%2:	Error value [-]
	%3:	Lower limit value [-]
	%4:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21136

Unknown tracking axis.			
Description	When axis tracking (#CAXTRACK ON) was selected, the system detected that the required tracking axis is not available in the NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	A tracking axis P-CHAN-00095 must be defined in the channel parameter list. Ensure that the axis is present in the NC channel when axis tracking mode is selected.
Parameter	%1:	Logical axis number [-]	

Error type	1, Error message from NC program.

ID 21137

Axis exchange not allowed during axis tracking.			
Description	No axis exchange commands may be used in the NC program when axis tracking mode (#CAXTRACK ON) is active, .		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before using axis exchange commands in the NC program, deselect the axis tracking mode (#CAXTRACK OFF).
Error type	1, Error message from NC program.		

ID 21138

Missing definition of default C axis name.			
Description	When C axis mode (#CAX) is active, the system detects that no default C axis name is configured in the NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	A default C axis name P-CHAN-00010 must be defined in the channel parameter list.
Error type	1, Error message from NC program.		

ID 21139

Channel parameters: C-axis activity requires a spindle.			
Description	With the selection of C axis processing (#CAX), it is detected that the required spindle is not configured in the NC-channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	At least one spindle has to be configured in the channel parameter list. P-CHAN-00010, P-CHAN-00082, [CHAN//Section Definition of a main spindle], P-CHAN-00051
Parameter	%1:	Limit value [-]	
		Invalid number of configured spindles in NC-channel.	
Error type	11, Error message from NC program.		

ID 21141

Invalid NC command in manual block.			
Description	NC commands that are only useful in a NC program were used in the manual block, .		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Do not use the corresponding commands in manual block. **Invalid NC command in manual block.** With #IF / #ELSE / #ENDIF (old syntax), multiple ID 1 is displayed. With #COMMENT BEGIN / END, multiple ID 2 is displayed.
Error type	1, Error message from NC program.		

ID 21142

Contour rotation not possible, if 1st or 2nd main axis is not available.			
Description	When contour rotation (#ROTATION) was selected, the system detects that the first or second main axis is missing in the NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis configuration of the NC channel. The error cause may be e.g. an incomplete basic configuration in the channel parameter list or the first or second main axis from the channel was released when axis exchange commands are used in the NC program.
Parameter	%1:	Incorrect value [-]	
		String acts as reference for which main axis is missing. FIRST: first main axis SECOND: second main axis	
Error type	1, Error message from NC program.		

ID 21143

Deselection of contour rotation at plane change.			
Description	With active contour rotation #ROTATION ON [...] a plane change (G17, G18, G19, G20) is programmed. Since this results in a modified order of the main axes, the contour rotation is implicitly deselected.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Modify the NC program. Deselect contour rotation before plane change with #ROTATION OFF.
Error type	1, Error message from NC program.		

ID 21144

Loading the V.E.-Variables data to the DECODER failed.			
Description	When external variables [EXTV] were initialised, a non-correctable error occurred. External variables not adopted in the controller.		
Response	Class	3	Controller start-up is continued and/or no update of external variables.
Solution	Class	7	Check the list of external variables and/or previous error messages.
Error type	-		

ID 21145

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21146

Syntax error within programming of contour rotation.			
Description	A syntax error has been found in the contour rotation command (#ROTATION).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour rotation command. Syntax errors for example are: Double programming Excluding keywords Missing syntax elements; open brackets, close brackets
Error type	1, Error message from NC program.		

ID 21147

Programmed rotation centre is out of data format.			
Description	In the contour rotation command (#ROTATION), one of the programmed coordinates of the rotation centre (CENTERxx) exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour rotation command. **Programmed rotation centre is out of data format.** For CENTER1, multiple ID 1 is indicated. For CENTER2, multiple ID 2 is indicated.
Error type	1, Error message from NC program.		

ID 21151

Invalid combination of spindle commands.			
Description	NC commands that mutually cancel each other were used in spindle programming. For example, M3 and M4 are programmed in the same NC block. [PROG//Section: Spindle programming]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify spindle programming.
Error type	1, Error message from NC program.		

ID 21152

Reset timeout for external tool data management.			
Description	When an NC reset was commanded, the reset request for the channel was not acknowledged by the external tool data management [FCT-C10] within the required time limit.		
Response	Class	3	Abort NC reset
Solution	Class	6	Repeat NC reset. Check external tool management.
Parameter	%1:	Error value [µs]	
		Actual exceeded time	
	%2:	Limit value [µs]	
		Maximum time period for reset acknowledge after reset request.	

Error type	-

ID 21153 - 21155

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21157

SYN not permitted for this command.			
Description	The SYN keyword is programmed in an invalid combination with the #IDENT or #COMMAND commands.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the #IDENT and/or #COMMAND commands and remove SYN. Synchronisation may only be programmed in combination with #IDENT WR, #COMMAND WR or #COMMAND WAIT [PROG].
Parameter	%1:	Error value [-]	
		Invalid keyword	
Error type	1, Error message from NC program.		

ID 21158

Read procedure command not possible (write only).			
Description	The #COMMAND command should be executed with a read request (RD). However, only a write request (WR) is permitted here.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command. Only a write request (WR) may be programmed.
Parameter	%1:	Error value [-]	
		Invalid keyword	
Error type	1, Error message from NC program.		

ID 21159

Machine data item could not be taken over.			
Description	The modification of an axis parameter programmed with the #MACHINE DATA command could not be executed.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Move the command #MACHINE DATA within the program sequence. If necessary check and modify the syntax or the value of the parameter. Some parameters cannot be changed while the NC program is active.
Error type	1, Error message from NC program.		

ID 21160

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21164

Missing absolute or relative programming.			
Description	In the command sequence of an independent axis, the absolute/relative programming is missing (G90/G91).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the command sequence of the independent axis and complete the absolute-/ relative programming.
Error type	1, Error message from NC program.		

ID 21165

Missing feed- or time programming.			
Description	The command sequence of an independent axis does not specify the feedrate or the motion time (FEED/TIME).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the command sequence of the independent axis and complete the feed or moving time.
Error type	1, Error message from NC program.		

ID 21167

TIMER ID is out of range of permissible values.			
Description	The counter value (ID…) programmed in the NC command #TIMER is outside the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program a permissible counter value (valid values: 0...127, see also [PROG//description of NC command #TIMER]).
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21168

Missing the programming of linear interpolation type.

Description	The interpolation type (G00/G01) is missing from the command sequence of an independent axis.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the command sequence of the independent axis and complete the type of interpolation.
Error type	1, Error message from NC program.		

ID 21169

Given path exceeds the max. internal length limit.			
Description	A program path defined in the start-up list exceeds the maximum permissible controller-internal length of the complete program call.		
Response	Class	2	Abort NC program processing.
Solution	Class	7	Shorten the length of the program paths defined in the start-up list (P-STUP-00018) to such an extent that the maximum permissible controller-internal length of the complete program call is not exceeded, even in combination with file names.
Parameter	%1:	Limit value [-]	
		Actual length of program path	
	%2:		
		Maximum permissible internal length of program path + filename	
Error type	5, Error message by access on files.		

ID 21170

Given filename exceeds the max. internal length limit.			
Description	The combination of program path and filename exceeds the permissible controller-internal length of the complete program call.		
Response	Class	2	Abort NC program processing.
Solution	Class	7	Shorten the length of the complete program call. There are 2 possibilities: Abbreviate the filename accordingly. Modify (shorten) the program paths defined in the start-up list (P-STUP-00018) so that the long file names can be retained.
Parameter	%1:	Limit value [-]	
		Actual length of program path + filename	
	%2:		
		Maximum permissible internal length of program path + filename	
Error type	5, Error message by access on files.		

ID 21171

ERROR-ID is out of range of permissible values.			
Description	The error number (ID...) programmed in the #ERROR command exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the error number (ID…).

Parameter	%1:	Error value [-]
		Specified error number
	%2:	Lower limit value [-]
		Lower limit for range of user-specific error messages
	%3:	Upper limit value [-]
		Upper limit for range of user-specific error messages
	%4:	Lower limit value [-]
		Lower limit for range of CNC cycles
	%5:	Upper limit value [-]
		Upper limit for range of CNC cycles
Error type	1, Error message from NC program.	

ID 21173

Unknown error class in #ERROR command.			
Description	The error class (RC...) programmed in the #ERROR command has an invalid value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Programming of a permissible error class (valid values are 0, 2 or 7, see also [PROG//Description of command #ERROR]).
Parameter	%1:	Error value [-]	
		Invalid value of the error class.	
	%2:	[-]	
	%3:	[-]	
	%4:	[-]	
	%5:	[-]	
Error type	1, Error message from NC program.		

ID 21174

The list contains an unknown element.			
Description	During interpretation of the lists, an unknown list element is detected.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Remove or modify the unknown list element in the corresponding list.
Error type	-		

ID 21175

NC-command #CAX only with main spindle allowed.			
Description	C axis machining (#CAX [...]) must be selected for a spindle which is currently not the main spindle.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Modify the NC program. There are 2 possibilities: Programming of C-axis processing for the current main spindle. First use #MAIN SPINDLE [...] to change to the appropriate main spindle. Then select C axis machining with the new main spindle.
Error type	1, Error message from NC program.		

ID 21176

Facing during active ACS or CS not allowed.			
Description	When a Cartesian transformation (#CS ON or #ACS ON) is active, face machining (#FACE) is selected.		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Deselect the Cartesian transformation (# CS OFF or # ACS OFF) before selecting face machining.
Error type	1, Error message from NC program.		

ID 21177

Lateral surface processing during active ACS or CS not allowed.			
Description	When a Cartesian transformation (#CS ON or #ACS ON) is active, lateral surface machining (#CYL) is selected.		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Deselect the Cartesian transformation (# CS OFF and/or # ACS OFF) before selecting lateral surface machining.
Error type	1, Error message from NC program.		

ID 21178

The NC command ACS or CS cannot be used during active CAX transformation.			
Description	When C axis machining (#CYL or #FACE) is active, a Cartesian transformation (#CS ON or #ACS ON) is selected.		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Deselect C axis processing (# CYL OFF and/or # FACE OFF) before selecting a Cartesian transformation.
Error type	1, Error message from NC program.		

ID 21179

Axis of a predefined M/H function is also programmed as independent axis in the same NC block.			
Description	The axis assigned to an M function (P-CHAN-00039) in the channel parameters is also programmed in the same NC block (see also [PROG//Section: Independent axes]).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program the M function and independent axis in separate NC blocks.

Parameter	%1:	Logical axis number [-]
	%2:	Error value [-]
Error type	1, Error message from NC program.	

ID 21180

Unknown format token within the message string.			
Description	An unknown format string (%...) is programmed in the #MSG command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the #MSG command. Permissible format strings are: %s , %S , %d , %D , %f , %F
Parameter	%1:	Current value [-]	
		Incorrect format string.	
Error type	1, Error message from NC program.		

ID 21182

The name of the external variable is too long.			
Description	The name of the external variable consists of too many characters.		
Response	Class	2	Start-up of the control is continued.
Solution	Class	3	The name of the external variable is implicitly shortened to the maximum permissible length. To avoid any error message, change the name in the [EXTV] list accordingly.
Parameter	%1:	Current value [-]	
		Maximum number of allowed characters inclusive the final character.	
	%2:	Current value [-]	
		Wrong number of characters.	
	%3:	Limit value [-]	
	%4:	Error value [-]	
	%5:	Corrected value [-]	
Error type	-		

ID 21183

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21184

CAX spindle is not in axis mode CAX.			
Description	The spindle axis received by the NC-channel is a CAX-spindle axis, but it has not the correct axis mode.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify axis mode in axis parameter list (P-AXIS-00015).
Parameter	%1:	Logical axis number [-]	
		Logical axis number of CAX-spindle. P-AXIS-00016, P-CHAN-00036, P-CHAN-00051	
Error type	1, Error message from NC program.		

ID 21185

Logical axis number is out of data format.			
Description	The logical axis number programmed in the drive parameter command exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the drive parameter command.
Parameter	%1:	Error value [-]	
		Incorrect logical axis number of the axis in the command. P-AXIS-00016, P-CHAN-00035	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21187

Programmed NC-commands before label definition.			
Description	Other NC commands are already programmed in the NC block before a string label definition [STRINGLABEL].		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Programming of the string label definition [STRINGLABEL] at the beginning of the NC block or directly behind the block number.
			
			Example:
			
			Wrong:
			`N10 X100 [STRINGLABEL] Y200`
			
			Correct:
			`N10 [STRINGLABEL] X100 Y200`
			or
			`[STRINGLABEL] N10 X100 Y200`

Error type	1, Error message from NC program.		

ID 21188

SIGNAL for open WAIT from given channel not expected.			
Description	The received signal comes from a channel not listed in the WAIT .		
Response	Class	3	Abort NC program processing.
Solution	Class	7	Either send the signal from the correct channel or extend the WAIT by the unexpected channel.
Parameter	%1:	Current value [-]	
		Number of the signal.	
	%2:	Error value [-]	
		Number of the channel, the signal comes from.	
Error type	1, Error message from NC program.		

ID 21189

Axis not found in command administration data.			
Description	No command request could be issued for the axis programmed in the SERCOS command (#COMMAND..).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Axis must be a SERCOS axis.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 21190

No more space in command administration data.			
Description	Too many SERCOS commands (#COMMAND..) have been programmed for an axis.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce number of SERCOS commands for this axis.
Parameter	%1:	Logical axis number [-]	
	%2:	Identification number [-]	
	%3:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 21191

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21192

Channel parameters: Axis specific use of internal M-function not allowed.			
Description	An internal M function is used in an axis-specific context. The M function is either preset axis-specific in the channel parameters (P-CHAN-00039) or programmed in the NC program in an axis-specific command sequence [PROG]. In the NC-control internal M-functions have a firm meaning and are not freely available. This also includes: M0, M1, M2, M3, M4, M5, M17, M19, M29, M30, M40 – M45 **Exception:** M0 and M1 may also be used axis specifically. Dependent on P-CHAN-00098, also M3, M4, M5 and M19 can be used for specific axis.		
Response	Class	2	Either the start-up of the control is continued and the wrong setting is removed or the NC program is aborted.
Solution	Class	3	Check and modify the settings of the channel parameters (P-CHAN-00039) or remove the M function from the axis-specific command sequence in the NC program.
Parameter	%1:	Error value[-]	
Error type	1, Error message from NC program.		

ID 21193

ECS: Tool vector has length 0 (KIN ID 0: see wz_kopf_versatz[0],[1],[2]).			
Description	The tool vector especially necessary during 2.5 D processing for the tool retraction on inclined planes in a ECS system is not defined in the channel parameter list in the corresponding kinematic parameters (ID 0).		
Response	Class	3	Abort NC program processing.

Solution	Class	3	Check and modify the corresponding channel parameter. A non-standard tool vector (0,0,1) can be defined under kinematic ID 0 via the elements of P-CHAN-00094.
			
			Example:
			Tool vector for a plane, rotated 45° around Y:
			`kinematik[0].param[0] 0,707`
			`kinematik[0].param[1] 0`
			`kinematik[0].param[2] 0,707`

Parameter	%1:	Current value[-]	
		Active kinematic ID	
Error type	-		

ID 21195

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	

Solution	Class	8	Requires controller restart.

ID 21196

Scope of validity (channel / global) of external variable not configured.			
Description	In an external variable list, an invalid identification is entered for the scope of validity (*scope* element).		
Response	Class	2	Controller start-up is continued. The incorrect identifier is implicitly replaced by the identifier VE_CHANNEL. the external address of this variable is ALSO calculated but not checked for plausibility.
Solution	Class	7	Initialise the external variable correctly in the list [EXTV].
Parameter	%1:	Error value[-]	
		Name of the concerned variable P-EXTV-00001	
	%2:	Corrected value [-]	
		Invalid scope of validity	
	%3:	Corrected value[-]	
		Corrected scope of validity	
Error type	-		

ID 21197

Too many external variables, no more internal memory.			
Description	The memory made available by the control is not sufficient to read in and to store all external variables during start-up.		
Response	Class	2	Start-up of the control is continued. The external variables which were read in after memory overflow are not stored inside the control, so they are not available after start-up.
Solution	Class	7	In the external variable list [EXTV], too large arrays are defined. Reduce number of array variables or reduce array sizes.
Parameter	%1:	Current value [-]	
	%2:	Upper limit value [-]	
Error type	-		

ID 21198

Feed link factor is out of data format.			
Description	A coupling factor (FACT or CORR) outside the permitted numerical range is programmed in the command for coupling the spindle speed to the path feed (FEED_LINK...).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command. Program a value within the permissible data range.

Parameter	%1:	Error value [-]
		Incorrect coupling factor **Feed link factor is out of data format.** For FACT, multiple ID 1 is indicated. For CORR, multiple ID 2 is indicated.
	%2:	Upper limit value[-]
Error type	1, Error message from NC program.	

ID 21199

Compare operator '==' expected.			
Description	Within a $IF control block, the result of an EXIST query is checked for TRUE or FALSE. For this check the assignment operator "=" is used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC block. For the check of TRUE or FALSE the comparison operator "==" has to be used.
Error type	1, Error message from NC program.		

ID 21200

Do not wait for release at block search. Processing start is ignored.			
Description	The block search (SV) has been instructed by the operator to start processing, which is not permissible at the current state, because the block search position does not wait for release at the SV marker.		
Response	Class	1	
Solution	Class	1	Check,why the user interface has assigned the unexpected block search action.
Error type	-		

ID 21201

Double programming of spindle feed forward control.			
Description	Error message is output when programming the spindle feedforward control (G135, G136, G137). Either feedforward control commands are programmed which are mutually exclusive (G135,G137) or they occur several times [PROG//Section: Spindle feedforward control].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command. Remove redundant or exclusive commands.
Error type	1, Error message from NC program.		

ID 21202

For a PLC-spindle this command is not available.

Description	NC commands were programmed in (spindle) axis-specific programming but this is not permitted with a so-called PLC spindle (P-CHAN-00069). Only the standard functions M3, M4, M5, M19, REV and POS may be used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove all commands from (spindle) axis-specific programming with the exception of those listed above or deselect any corresponding predefined functions (P-CHAN-00039, P-CHAN-00025).
Parameter	%1:	Error value[-]	
		Number of the incorrect M/H/G functions	
Error type	1, Error message from NC program.		

ID 21203

Channel parameters: Synchronisation mode MET_SVS needs pre output time greater than or equal to NC cycle time of control.			
Description	During start-up of M/H functions with synchronisation mode MET_SVS, a check of the assigned pre-output time P-CHAN-00070 / P-CHAN-00107 detected that this time is smaller than the cycle time.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In the channel parameters, check and modify the pre-output time P-CHAN-00070 / P-CHAN-00107for the corresponding M/H function before the next controller start-up. In case of conflict, the pre-output time is set to the cycle time at start-up and start-up is continued.
Parameter	%1:	Current value	
		Number of the M/H function **Channel parameters: Synchronisation mode MET_SVS needs pre output time greater than or equal to NC cycle time of control.** For M functions multiple ID 1 is indicated. For H functions multiple ID 2 is indicated.	
	%2:	Error value[-]	
		Incorrect pre-output time.	
	%3:	Corrected value [-]	
		Pre-output time (= cycle time) automatically corrected	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21204

Maximum number of Look-ahead M/H functions per NC block exceeded.			
Description	The maximum number of M and H functions programmed in the NC block with synchronisation types MET_SVS / MEP_SVS (seeP-CHAN-00070) was exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	If possible, split the M/H functions concerned into several motion blocks or reduce the number of functions in the block.
Parameter	%1:	Limit value [-]	
	%2:	Error value [-]	
		Number of the look ahead M/H functions programmed in the NC block.	
Error type	1, Error message from NC program.		

ID 21205

Velocity limit on the path is negative or 0. Channel parameter may not be set.			
Description	The velocity limit on the path programmed by #VECTOR LIMIT ... [...] is negative or zero.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the command. Programmed velocity limit must have a value greater than zero (nil).
Parameter	%1:	Error value .][1µm/s or 0.001°/s]	
Error type	1, Error message from NC program.		

ID 21206

Acceleration limit on the path is negative or 0. Channel parameter may not be set.			
Description	The acceleration limit on the path programmed by #VECTOR LIMIT ... [...] is negative or zero.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the command. Programmed acceleration limit must have a value greater than zero (nil).
Parameter	%1:	Error value [1mm/s^2 or 1°/s^2]	
Error type	1, Error message from NC program.		

ID 21207

Channel parameters: Change in external tool management, new start of system is necessary.			
Description	A change in P-CHAN-00016 is to be updated by reloading the channel parameter list with the controller active.		
Response	Class	2	Controller start-up is aborted.
Solution	Class	7	A change in P-CHAN-00016 can only be adopted by restarting the controller.
Parameter	%1:	Current value [-]	
		Name of channel parameter.	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21208

Filename not defined.			
Description	When a subroutine is called, the name of a local subroutine is missing in case of LL and the filename of a global subroutine is missing in case of L.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	After LL, complete the name of a local subroutine and after L, complete the file name of the global subroutine [PROG].
Error type	1, Error message from NC program.		

ID 21209

No feed axes defined. Main axes are set to feed axes.			
Description	If no axes are configured as feed axes (P-CHAN-00011) in the channel parameters, and P-CHAN-00096 is also assigned 0, all main axes are automatically defined as feed axes in the #FGROUP command.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Define axes as feed axes in the #FGROUP command or in the channel parameters (P-CHAN-00011, P-CHAN-00096).
Error type	1, Error message from NC program.		

ID 21210

Too many hardware numbers programmed.			
Description	In the command #SIGNAL the keyword HW is present several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command. Remove redundant keyword from #SIGNAL command.
Parameter	%1:	Current value[-]	
		Programmed redundant hardware number.	
Error type	1, Error message from NC program.		

ID 21211

Hardware ID is out of data format.			
Description	The hardware identifier (HW) programmed in the #SIGNAL command is outside the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command. The permissible hardware IDs are defined in the start-up list [STUP].
Parameter	%1:	Error value [-]	
		Incorrect hardware ID.	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21212

Programmed HW-ID is not known (s. parameter list of NC).			
Description	The value of the hardware identifier (HW) wert programmed in the NC command #SIGNAL is unknown.		
Response	Class	1	Abort NC program processing.
Solution	Class	1	Check and modify NC command. If the hardware ID is programmed correctly, then ensure that the hardware ID is also entered correctly in the start-up list [STUP].
Parameter	%1:	Error value [-]	
		Incorrect hardware-ID.	

Error type	1, Error message from NC program.

ID 21213

Double programming of AHEAD.			
Description	In the #WAIT command, the AHEAD password exists multiple times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Error type	1, Error message from NC program.		

ID 21215

Invalid ESC parameter configured.			
Description	When the #ECS command is used. the program detects that the required channel parameters are incorrectly configured.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the setting of the ECS parameters in the channel parameter list: kind_of_2nd_ecs_ax[0] (P-CHAN-00031) mach_plane_of_2nd_ecs_ax[0] (P-CHAN-00050) Re-read the channel parameter list after modification.
Parameter	%1:	Error value [-]	
		Incorrect channel parameters as string.	
Error type	1, Error message from NC program.		

ID 21216

Variable is an array, index not programmed.			
Description	An array**V**.xx variable is missing the index.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the variables within the NC block.
Error type	1, Error message from NC program.		

ID 21217

Closing bracket of index is missing.			
Description	The closing bracket "]" is missing is missing behind the index in a **V**.xx variable.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the variables in the NC block.
Error type	1, Error message from NC program.		

ID 21218

Type of variable is not available.			
Description	Variables of type V.L. are used in the NC program. This option is not allowed because of the version-specific configuration.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Replace V.L. variables by V.S. or V.G. variables or P parameters. Or please contact the CNC manufacturer to enable the use of V.L. variables in this version .
Error type	1, Error message from NC program.		

ID 21219

Double programming in #MACHINE DATA-command.			
Description	The #MACHINE DATA command contains multiple syntax elements.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 21220

Unknown or incomplete token in NC command.			
Description	A character string programmed in the NC command cannot be decoded by the controller. The cause can be a write error or an invalid or incomplete element or keyword in connection with the command itself.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Parameter	%1:	State [-]	
Error type	1, Error message from NC program.		

ID 21224- 21227

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21230

Excursion of oscillation is 0.			
Description	Within the oscillating axis command, the excursion (EXCUR) is too small or zero (nil).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the oscillating axis command.

Parameter	%1:	Error value [-]
		Incorrect excursion.
Error type	1, Error message from NC program.	

ID 21231

Missing zero position programming.			
Description	No zero position (ZERO_POS) is programmed in the oscillating axis command when specifying the path motion via the excursion.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the oscillating axis command.
Error type	1, Error message from NC program.		

ID 21232

Unknown format string.			
Description	An unknown format string (%...) is programmed in the #MSG command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the #MSG command. Permissible format strings are: %s , %S , %d , %D , %f , %F
Parameter	%1:	Current value [-]	
		Incorrect format string.	
Error type	1, Error message from NC program.		

ID 21233

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21234

Number of permissible (A)CS definitions exceeded.			
Description	The number of (A)CS coordinate systems defined in the NC program exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the number of (A)CS definitions or modify the NC program so that (A)CS definitions no longer required in the following program sequence can be used for new definitions.

Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21235

Double programming in #WCS/MCS-command.			
Description	In the NC commands #WCS TO MCS [...] or #MCS TO WCS [...] syntax elements are programmed several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove redundant or exclusive syntax elements from the NC commands.
Error type	1, Error message from NC program.		

ID 21236

NC-command during active transformation not allowed.			
Description	The commands #WCS TO MCS or #MCS TO WCS may not be programmed during active cartesian ((#(A)CS ON) or kinematic transformation (#TRAFO ON).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before using the #WCS TO MCS or #MCS TO WCS commands, deselect all active transformations (#TRAFO OFF, #(A)CS OFF).
Error type	1, Error message from NC program.		

ID 21237

Number of oscillations is out of data format.			
Description	In the oscillating axis command, the programmed number of oscillations (NBR_OSC) exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the oscillating axis command.
Parameter	%1:	Error value [-]	
		Incorrect number of oscillations.	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21238

Path axis configured in channel parameters cannot be a SAI-axis or spindle.

Description	A path axis configured in the channel parameter list is a single axis (SAI, PLCOpen) or a spindle axis.		
Response	Class	3	Start-up of the control is aborted.
Solution	Class	7	Either the axis must not be configured as a path axis in the channel parameter list (P-CHAN-00035), or the SAI property must be switched off in the axis parameter list (P-AXIS-00250).
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Index of the axis within the axis group [CHAN-chapter Axis structure]	
Error type	-		

ID 21239

Spindle command is disabled.			
Description	Wrong setting for a synchronisation mode of a spindle function.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the synchronisation modes of the spindle functions in the channel parameter list: REV (P-CHAN-00081) M3 (P-CHAN-00045) M4 (P-CHAN-00047) M5 (P-CHAN-00049) M19 (P-CHAN-00043) Reinterpret channel parameter list after modification.
Parameter	%1:	Error value [-]	
		Incorrect spindle function	
	%2:	Logical axis number [-]	
		Logical axis number of the concerned spindle axis P-CHAN-00036	
Error type	-		

ID 21240

G-command has no effect, because precontrol is disabled in one axis.			
Description	In an axis, the feedforward cannot be switched on because it is disabled by P-AXIS-00256.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	P-AXIS-00256 must be set to 0 in the concerned axis .
Parameter	%1:	Current value [-]	
		Number of the G command	
	%2:	Logical axis number [-]	
	%3:	Error value [-]	
		Value of P-AXIS-00256	
Error type	1, Error message from NC program.		

ID 21241

Value exceeds definition range of trigonometric function.			
Description	The value programmed with the trigonometric function is outside the definition range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify trigonometric function.
Parameter	%1:	Error value [-]	
	%2:	Current value [-]	
		Incorrect trigonometric function.	
Error type	1, Error message from NC program.		

ID 21242

Value programmed with D is out of data format.			
Description	The value programmed with the D word exceeds the permissible data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and change D word. Program a value within the permissible data range.
Parameter	%1:	Error value [-]	
		Incorrect value after D.	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21243

D command not allowed in spindle-specific syntax for main spindle.			
Description	The D word is programmed inside the spindle-specific command sequence delimited by parentheses for a main spindle "S". Example: **Wrong:** N10 S[REV1000 M3 **D5**] **Correct:** N10 **D5** S[REV1000 M3]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program D word outside the spindle-specific command sequence of the main spindle.
Error type	1, Error message from NC program.		

ID 21244

Channel parameters: T with implicit FLUSH without active "t_info_to_wzv" is not useful.			
Description	Setting P-CHAN-00106 to 1 is only meaningful if P-CHAN-00087 is also set to 1.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Parameter is set implicitly to default value 0.

Parameter	%1:	Error value [-]
		Channel parameter P-CHAN-00106 incorrectly assigned
	%2:	Corrected value [-]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 21247

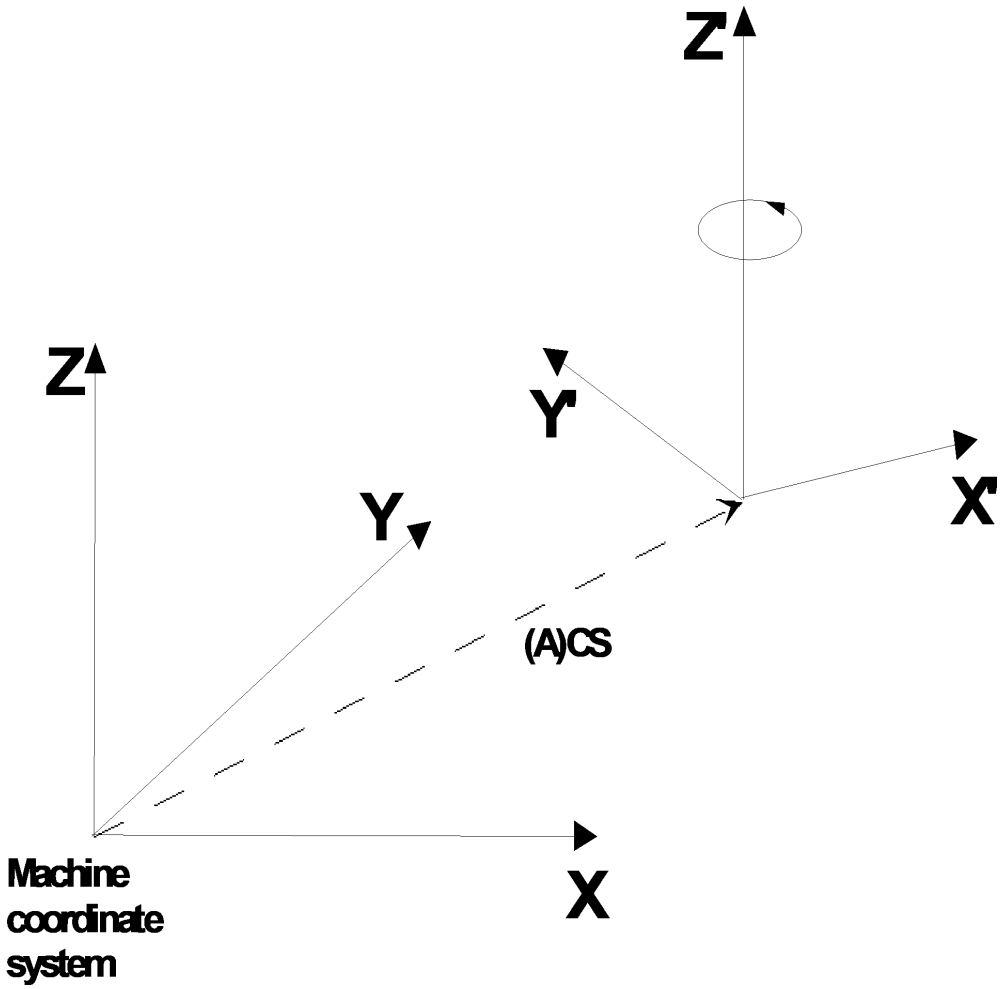
Cartesian transformations are still enabled at program end.			
Description	At program end (M02, M30), coordinate systems (transformations) selected by the command #CS ON / #ACS ON are still active.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Add NC program. Enter CS OFF / #ACS OFF or #CS OFF ALL / #ACS OFF ALL before program end.
Error type	1, Error message from NC program.		

ID 21248

Deselection of axis tracking has no effect.			
Description	Axis tracking is deselected by #CAXTRACK OFF although no axis tracking is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	1	Delete #CAXTRACK OFF from NC program.
Error type	1, Error message from NC program.		

ID 21249

Axis tracking during active (A)CS only for rotation about Z axis allowed.

Description	An attempt is made to select axis tracking in a coordinate system defined by #CS or #ACS that was rotated in several axes. Axis tracking in a coordinate system defined by #CS or #ACS is only permitted if the new coordinate system planes are shifted in parallel in X/Y and rotated about Z only.  		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Define a suitable coordinate system with #CS or #ACS or reclamp the workpiece in a position that allows the definition of a suitable coordinate system for axis tracking.
Error type	1, Error message from NC program.		

2.3.6 ID-range 21250-21499

ID 21423

Help channel without license, move only with link axis(see. P-AXIS-00101).			
Description	For the programmed NC function this channel requires a licence.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Please contact your distribution partner for a channel licence or just use link axes.
Parameter	%1:	Logical axis number [-]	
		Number of the unlinked axis	
Error type	1, Error message from NC program.		

ID 21250

Active axis tracking at program end.			
Description	At program end (M02, M30), axis tracking selected by the #CAXTRACK ON command is still enabled.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Supplement NC program. Enter #CAXTRACK OFF before program end.
Error type	1, Error message from NC program.		

ID 21251

Missing comma between axis name and log. axis number.			
Description	Within the axis exchange command, a comma must be programmed between the axis name and the logical axis number.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command.
Error type	1, Error message from NC program.		

ID 21252

Missing comma between log. axis number and axis index.			
Description	Within the axis exchange command, a comma must be programmed between the logical axis number and the axis index.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command.
Error type	1, Error message from NC program.		

ID 21253

Logical axis number is out of data format.			
Description	The axis exchange command is programmed with a logical axis number which exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify the axis exchange command.
Parameter	%1:	Error value [-]	
		Incorrect logical axis number of the axis in the axis exchange command. P-AXIS-00016, P-CHAN-00035	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21254

Axis index is out of data format.			
Description	The axis exchange command is programmed with an axis index which exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command.
Parameter	%1:	Error value [-]	
		Wrong index of the axis in the axis exchange command. [CHAN//Section: Configuration of path axes]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
		Maximum permissible axis index in channel. Version-specific setting [SYSP].	
	%4:	Logical axis number [-]	
		Logical axis number of the axis with wrong index. P-AXIS-00016, P-CHAN-00035	
Error type	1, Error message from NC program.		

ID 21255

Missing comma between axis index and offset mode.			
Description	Within the axis exchange command, a comma must be programmed when specifying an offset mode after the axis index.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command.
Error type	1, Error message from NC program.		

ID 21256

Multiple use of the same axis name.			
Description	Within the axis exchange command, an attempt is made to exchange different axes with the same name into the channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command. Assign different names to the corresponding axes.

Error type	1, Error message from NC program.

ID 21257

Multiple use of the same axis index.			
Description	Within the axis exchange command, an attempt is made to exchange different axes for the same index (space) in the channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command. Assign different free indices (places) of the channel to the corresponding axes.
Parameter	%1:	Error value [-]	
		Index of the axes in the axis exchange command. [CHAN//section: Configuration of the path axes]	
Error type	1, Error message from NC program.		

ID 21258

Multiple use of the same log. axis number.			
Description	Within the axis exchange command, an attempt is made to exchange the same axis (clearly identified by the logical axis number) several times into the NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command. The axis with the corresponding logical axis number may be programmed only once within the axis exchange command.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis, which is used several times within the axis exchange command. P-AXIS-00016, P-CHAN-00035	
Error type	1, Error message from NC program.		

ID 21259

Too many axes requested.			
Description	More axes are requested for the channel than are permissible at all in the channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command. Request fewer axes or rethink program concept. There is no way to exchanged more axes into a NC channel, than the system limits allow.
Parameter	%1:	Error value [-]	
		Current number of requested axes.	
	%2:	Upper limit value [-]	
		Maximum permissible number of axes in the channel. Version-specific setting [SYSP].	
Error type	1, Error message from NC program.		

ID 21260

Axis index is already used in channel.

Description	The index (position) of a requested new axis is already occupied by an existing channel axis.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command . Assign to the affected axis an index (place), which is not used yet in channel.
Parameter	%1:	Error value [-]	
		Index of the axis in the axis exchange command. [CHAN//section: Configuration of the path axes]	
Error type	1, Error message from NC program.		

ID 21261

Axis name is already used in channel.			
Description	The name of a new requested axis corresponds to the name of an axis that already exists in the channel. P-CHAN-00006		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the axis exchange command . Request concerned axis with a name that is not yet used in channel.
Error type	1, Error message from NC program.		

ID 21262

Channel parameters: Default feedrate factor negative or 0.			
Description	The default feed rate factor P-CHAN-00108 defined in the channel parameters has a negative value or is zero (nil).		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and change the default feedrate factor P-CHAN-00108 in the channel parameters before the next controller start-up. In the event of a conflict, the incorrect default feedrate factor P-CHAN-00108 is set to -1 (not defined) during the start-up, and the start-up is continued.
Parameter	%1:	Error value [-]	
		Invalid feed rate factor	
	%2:	Corrected value [-]	
		Corrected default value	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21263 - 21272

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21273

Double programming in #SAVE/LOAD-command.

Description	In the commands #SAVE CONFIG or #LOAD CONFIG, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the commands #SAVE CONFIG or #LOAD CONFIG in the NC program.
Error type	1, Error message from NC program.		

ID 21274

No recoverable axes configuration found.			
Description	The #LOAD CONFIG command can not be executed, because no recoverable axis configuration is stored.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before using #LOAD CONFIG[AX] for the first time, an axis configuration must be saved in the NC program with #SAVE CONFIG[AX].
Parameter	%1:	State [-]	
Error type	1, Error message from NC program.		

ID 21275

No recoverable axes couplings found.			
Description	The #LOAD CONFIG command can not be executed, because no recoverable axes couplings (synchronous operation) are stored.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before using #LOAD CONFIG[AXLINK] for the first time, the active axis couplings must be saved in the NC program with #SAVE CONFIG[AXLINK].
Parameter	%1:	State [-]	
Error type	1, Error message from NC program.		

ID 21276

Axis names have to be separated by commas or terminated by ']'.			
Description	In the #PUT AX or #AX RELEASE command, commas ',' are missing as separators for specifying axes or the closing square bracket ']' is missing after the last axis names.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the commands #PUT AX or #AX RELEASE in the NC program.
Error type	1, Error message from NC program.		

ID 21277

#LOAD CONFIG not allowed during active synchronous operation.

Description	The command #LOAD CONFIG is programmed in the NC program while synchronous operation ((#ENABLE AX LINK or #AX LINK ON) is active. This is not permitted. Example: **Wrong:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=X] **N40 #ENABLE AX LINK[1]** : **N100 #LOAD CONFIG [AX]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=X] N40 #ENABLE AX LINK[1] : **N90 #DISABLE AX LINK[1]** **N100 #LOAD CONFIG [AX]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deselect the active synchronous operation with #DISABLE AX LINK or #AX LINK OFF (ALL) before using the command #LOAD CONFIG.
Error type	1, Error message from NC program.		

ID 21278

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21279

Maximum number of oscillation axes exceeds limit.			
Description	Too many oscillating axes (<axis_name>[OSC ON...]) are programmed in the NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the number of programmed oscillation axes in the invalid NC block to the maximum permissible number.
Parameter	%1:	Limit value [-]	
		Maximum permissible number of oscillation axes	
Error type	1, Error message from NC program.		

ID 21280

With #SAVE CONFIG, too many coupling pairs are to be saved.

Description	When using the #SAVE CONFIG[AXLINK] command, the total number of coupled axis pairs of all enabled axis coupling groups to be saved is too large. The total number of coupling pairs to be backed up is limited because #LOAD CONFIG[AXLINK] implicitly assigns coupling group 1 to the restore, which can also only hold a limited number of coupling pairs.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before using #SAVE CONFIG, if possible reduce the number of active axis coupling groups with #DISABLE AX LINK or #AX LINK OFF.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 21281

Maximum number of coupling pairs exceeds limit.			
Description	In the #SET AX LINK or #AX LINK commands, the maximum permissible number of coupling pairs is exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the number of coupling pairs to the maximum permissible number. Divide if possible the coupling pairs on several coupling groups.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 21282

The slave axis to be blocked, is unknown.			
Description	When activating of a coupling group it is detected that one of the slave axis is not present in channel, or that the axis name is unknown. Especially this error can occur during the so called implicit axes exchange of gantry axes, if the gantry coupling shall be restored in the new channel.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Before axis exchange, deactivate gantry couplings if they are active and explicitly exchange the axes involved using #PUT AX/ #AX RELEASE and #CALL AX/ #AX REQUEST between the channels.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21283

A slave axis is already programmed in an active coupling group as slave axis.			
Description	When activating of a coupling group, it is detected that a slave axis is already programmed as slave axis in another active coupling group. This inter coupling group relation is not permissible, because a slave axis may only be assigned to one master axis.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Deactivate the active coupling group where the slave axis causes a conflict (#DISABLE AX LINK / #AX LINK OFF).
Parameter	%1:	Logical axis number [-]	
		Logical number of the invalid coupling axis	
	%2:	Current value [-]	
		Number of the already active coupling group	
Error type	1, Error message from NC program.		

ID 21284

A slave axis is already programmed in an active coupling group as master axis.			
Description	When activating of a coupling group, it is detected that a slave axis is already programmed as master axis in another active coupling group. This inter coupling group relation is not permissible, because a slave axis cannot be master axis at the same time.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Deactivate the active coupling group in which the master axis causes a conflict beforehand (#DISABLE AX LINK / #AX LINK OFF).
Parameter	%1:	Logical axis number [-]	
		Logical number of the invalid coupling axis	
	%2:	Current value [-]	
		Number of the already active coupling group	
Error type	1, Error message from NC program.		

ID 21285

A master axis is already programmed in an active coupling group as slave axis.			
Description	When activating of a coupling group, it is detected that a master axis is already programmed as slave axis in another active coupling group. This inter coupling group relation is not permissible, because a master axis cannot be slave axis at the same time.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Deactivate the active coupling group where the slave axis causes a conflict (#DISABLE AX LINK / #AX LINK OFF).
Parameter	%1:	Logical axis number [-]	
		Logical number of the invalid coupling axis	
	%2:	Current value [-]	
		Number of the already active coupling group	
Error type	1, Error message from NC program.		

ID 21286

Control area ID is out of data format.			
Description	When the command #CONTROL AREA BEGIN[...] is programmed the area number identifier exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible area number.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21287

Definition of a previous control area has not been finished.			

Description	It is noted that the definition of a new control area with #CONTROL AREA BEGIN[...] is still programmed within the NC program sequence of the previous control area. But the nested definition of control areas is not allowed.
	Syntax example:
	`:`
	`N10 #CONTROL AREA BEGIN...`
	`N20 G01 X10 Y10`
	`N30 G01 X20 Y20`
	`N40 #CONTROL AREA BEGIN... -> ERROR`
	`:`
	`Nxx #CONTROL AREA END`
	`:`		

Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. The definition of a new control area only is permissible, if the definition of a previous control area is finished with #CONTROL AREA END.
			Syntax example:
			`:`
			`N10 #CONTROL AREA BEGIN...`
			`N20 G01 X10 Y10`
			`N30 G01 X20 Y20`
			`:`
			`Nxx #CONTROL AREA END`
			`Nxx #CONTROL AREA BEGIN...`

Error type	1, Error message from NC program.		

ID 21288

No associated definition of a control area. Command has no effect.

Description	For the command #CONTROL AREA END, the corresponding start of the definition of the control area with #CONTROL AREA BEGIN[...] is missing. Therefore, #CONTROL AREA END is ignored. Syntax example: **WRONG:** N20 G01 X10 Y10 N30 G01 X20 Y20 : **Nxx #CONTROL AREA END -> FEHLER** : **RIGHT:** **N10 #CONTROL AREA BEGIN...** N20 G01 X10 Y10 N30 G01 X20 Y20 : **Nxx #CONTROL AREA END** :		
Response	Class	1	Abort NC program processing.
Solution	Class	1	Check and modify NC program. The definition of control areas must always begin with #CONTROL AREA BEGIN[...] and end with #CONTROL AREA END.
Error type	1, Error message from NC program.		

ID 21289

Missing programming of control area mode.			
Description	The command #CONTROL AREA BEGIN[...] does not specify the keywords WORK or PROT.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. . Complete the definition of the control area with WORK or PROT.
Error type	1, Error message from NC program.		

ID 21290

Missing programming of control area type.			
Description	The command #CONTROL AREA BEGIN[...] does not specify the keywords POLY or CIRC.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the definition of the control area with POLY or CIRC.
Error type	1, Error message from NC program.		

ID 21291

Control area has no excursion in the third dimension.			
Description	In the command #CONTROL AREA BEGIN[...] neither a minimum extension MIN_EXCUR nor a maximum extension MAX_EXCUR is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the definition of the control area with the definition of a minimum and/or maximum excursion (MIN_EXCUR / MAX_EXCUR).

Parameter	%1:	Error value [-]
	%2:	Error value [-]
Error type	1, Error message from NC program.	

ID 21292

Invalid Control-Area-ID programmed.			
Description	The area number identifier in the command #CONTROL AREA BEGIN[...] is assigned the value zero (nil).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible area number.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21293

Missing programming of Control-Area-ID.			
Description	The command #CONTROL AREA BEGIN[...] does not specify the keyword ID with an area number.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.. Complete the definition of the control area with an area number ID.
Error type	11, Error message from NC program.		

ID 21294

Double programming in #CONTROL AREA-command.			
Description	In the command #CONTROL AREA BEGIN[...] mutually exclusive keywords or keywords are programmed several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove redundant or mutually excluding keywords in the #CONTROL AREA BEGIN[...] commands.
Error type	1, Error message from NC program.		

ID 21295

Definition of a control area during active thread cutting or tapping is not allowed.			
Description	The definition of a control area with #CONTROL AREA BEGIN[...] is programmed while active thread cutting (G33) or tapping (G63).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before defining a control area, deselect tapping (G33) or thread cutting (G63) with a motion block (G00, G01, G02, G03).
Parameter	%1:	Error value [-]	

Error type	1, Error message from NC program.

ID 21296

Keyword value is out of data format.			
Description	A # command is programmed with a keyword that has a value which exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program invalid keyword in the # command with a permissible value.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21297

Requested data of a data base group are invalid.			
Description	The database manager in the PLC acknowledged the data of a specific group requested by the command #DB INFO[...] as invalid. A check for validity of the received data and output of an error message is only executed in the NC channel if the command #DB INFO[...] was programmed with the logic switch CHV. Data cannot be checked for validity if there is no logic switch CHV.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	In the PLC, check and validate the data of the corresponding group.
Parameter	%1:	Error value [-]	
		Identification string of the invalid database group	
Error type	1, Error message from NC program.		

ID 21298

Number of received data exceeds number of data requested by #DB INFO-command.			
Description	The data base manager in PLC has send more data groups than requested by the command #DB INFO[…].		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the data base manager in PLC.
Error type	1, Error message from NC program.		

ID 21299

Double programming in #DB INFO-command.			
Description	In the command #DB INFO[...] redundant keywords are programmed several times.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. In the command #DB INFO[...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21300

Axis not referenced! Programmed movement is not allowed.			
Description	An unreferenced axis must be moved although P-AXIS-00277 is assigned 1. Execute the check in automatic mode for programmed motions (path axes, Independent Axes, oscillation axes) using interpolation types G00/G01/G02/G03 Spindles and so-called single axes (SAI axes/PLCopen) are not checked.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Homing of the axis before moving.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the unreferenced axis. P-AXIS-00016, P-CHAN-00035	
Error type	1, Error message from NC program.		

ID 21301

Starting keyword invalid.			
Description	In the axis specific programming X[...], an invalid keyword is programmed after 'opening square bracket'. At the first position, the programming of the function specific keyword (identification) is necessary.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure that the following identifiers are used first in axis-specific programming: Independent axes: **INDP_SYN** or **INDP_ASYN** -> e.g. X[INDP_SYN...] Oscillating axes: **OSC** -> e.g. X[OSC...] Sampled spindles: **DIST_CTRL** -> e.g. X[DIST_CTRL...] Spindle feed link: **FEED_LINK** -> e.g. S[FEED_LINK…] Techno informations: **M or H** -> e.g. X[M10…]
Error type	1, Error message from NC program.		

ID 21302

Maximum number of distance controlled axes exceeded.

Description	Too many "sampled spindles" (distance axes: X[DIST_CTRL...]) are programmed in the NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure that only the maximum permissible number of distance controlled axes are programmed in the NC block.
Parameter	%1:	Error value [-]	
		Invalid number of programmed distance controlled axes in NC block	
	%2:	Limit value [-]	
		Permissible number of programmed distance controlled axes in NC block	
Error type	1, Error message from NC program.		

ID 21303

Automatic determination of feed coupling factor without selection not allowed.			
Description	When programming Spindle feed link, the keyword AUTO was programmed without ON or OFF. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 S[FEED_LINK **AUTO**] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The keyword AUTO only may be programmed in combination with ON or OFF. Example: **Correct:** N10 G00 X0 Y0 Z0 N20 S[FEED_LINK **ON** **AUTO**] : N1000 M30
Error type	1, Error message from NC program.		

ID 21304

Channel parameters: SLOPE parameter is out of range of permissible values.			
Description	One of the default slope parameters specified in the channel parameters (P-CHAN-00071, P-CHAN-00073, P-CHAN-00001) exceeds the permissible numerical range.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	Check and modify the corresponding default slope parameter in the channel parameters before the next controller start-up. In case of conflict, the invalid default slope parameter is set to zero (nil) at start-up and the start-up is continued.

Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
	%4:	Corrected value [-]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 21305

Channel parameters: Default feed rate 'F' is negative.			
Description	The default feedrate defined in the channel parameters P-CHAN-00099 is assigned a negative value.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and modify the default feedrate P-CHAN-00099 in the channel parameters before the next controller start-up. During start-up, in case of conflict, the default feedrate P-CHAN-00099 is set to zero (nil), and the start-up is continued.
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21306

Invalid log. axes number of the main spindle.			
Description	A spindle axis with the logical axis number of the main spindle is not available in the NC channel. The error message can occur in the active NC program when using the #MAIN SPINDLE command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the command #MAIN SPINDLE and program the number of a spindle, which is configured in the channel.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21307

Axis tracking must be deactivated before enable/disable of this function.	
Description	When evaluating one of the following functions #TRAFO OFF #CS OFF #MCS ON the system detected that tangential tracking (#CAXTRACK ON/OFF[...]) is still enabled.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Before selecting or deselecting the above functions, the tangential tracking function (#CAXTRACK OFF) must be programmed as deselected.
Error type	1, Error message from NC program.		

ID 21308

Double-programmed feedforward control adaptation.			
Description	More than one G function was programmed in the same NC block from the group of adaptive feedforward control (G335/G337). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G335 G337** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the invalid G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 21309

Syntax check in channel must always be instructed with MACHINE_LOCK.			
Description	An NC program start is commissioned in machining mode Syntaxcheck. It is detected that the MACHINE_LOCK bit is not set.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	The missing bit MACHINE_LOCK is set implicitly and NC program start is continued. With the generation of a NC-program instruction in MMI or PLC in syntax check machining mode both, the bit for syntax check and the bit MACHINE_LOCK has to be set.
Error type	1, Error message from NC program.		

ID 21310

Maximum number of non-acknowledged SIGNAL/WAIT exceeded. SIGNAL could not be saved any more.			
Description	In the Event Manager, the internal system resource for storing open SIGNAL/WAIT is used up. The last SIGNAL sent cannot be processed any more.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Reduce the number of SIGNAL/ WAIT synchronisations in the NC programs of the channels and optimise the program sequences.
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	

Error type	1, Error message from NC program.

ID 21311

Maximum number of non-acknowledged SIGNAL/WAIT exceeded. WAIT could not be saved any more.			
Description	In the Event Manager, the internal system resource for storing open SIGNAL/WAIT is used up. The last WAIT sent cannot be processed any more.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Reduce the number of SIGNAL/ WAIT synchronisations in the NC programs of the channels and optimise the program sequences.
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 21312

Axis involved in synchronous operation cannot not be rearranged.

Description

An axis that is currently active in a coupling group as master or slave axis in Synchronous operation, may be programmed with #SET AX... or #AX DEF... must not be re-sorted.

Example:

```
Wrong:
N10   G00 X0 Y0 Z0
N30   #SET AX LINK[1, C=Y]
N40   #ENABLE AX LINK[1]
N50   #SET AX [X,1,0] [Y,2,1] [Z,3,2] [C,4,6]
:
N1000 M30

Correct:
N10   G00 X0 Y0 Z0
N30   #SET AX LINK[1, C=Y]
N40   #ENABLE AX LINK[1]
:
N500  #DISABLE AX LINK[1]
N510  #SET AX [X,1,0] [Y,2,1] [Z,3,2] [C,4,6]
:
N1000 M30
```

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program sequence. Before #SET AX... or #AX DEF... deselect the active coupling groups, were the axes to be rearranged are used.
Parameter	%1:	Logical axis number [-]	
		Number of the involved axis	
	%2:	Current value [-]	
		Number of the coupling group	
	%3:	Current value [-]	
		Number of the coupling pair within the coupling group	
Error type	1, Error message from NC program.		

ID 21313

Percent sign not allowed within global subroutine part.			
Description	In a global subroutine (GUP), a percentage sign (%) is read, although the global subroutine is already executed. The execution of a global subroutine is started, if either a % character followed by a program name (without "L"!) was found in the file, or if the first character found in the file outside comments is one that is neither a space nor a "%". In this case, this character is evaluated as the first character of an unnamed program. It also means that no block numbers, variable declarations etc. may be programmed before the first "%". [PROG//Chapter – Subroutine techniques]		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify global NC subroutine. Ensure that no NC block numbers, variable declarations etc. were programmed before the first %-character (e.g. remove block numbers at the beginning of comment lines!).
			`Nxx (Comment) (GUP-Start here)`
			`:`
			`%L UP1 (<- Error 21313 here!)`
			`Nxx`
			`:`
			`%GUP`
			`Nxx`
			`:`
			or
			`Nxx (Comment) (GUP-Start here)`
			`:`
			`%GUP (<-Error 21313 here!)`
			`:`
			or
			`(Comment)`
			`:`
			`%L UP1`
			`Nxx`
			`:`
			`%GUP (GUP-Start here)`
			`Nxx`
			`:`
			`%STRING (<-Error 21313 here!)`
			`:`
			or
			`Nxx (GUP-Start here)`
			`:`
			`%STRING (<-Error 21313 here!)`
			`:`

Error type	1, Error message from NC program.		

ID 21314

Singular position in kinematic transformation, no valid result.			
Description	The kinematic backward transformation #WCS TO MCS cannot be calculated because the values defined with the axis-specific variables V.A.WCS... describe a singular (ambiguous) position. This delivers no clear result for the machine coordinate system.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	The kinematic backward transformation must be executed for another position to produce a unique position in the machine coordinate system.
Error type	1, Error message from NC program.		

ID 21315

Removing of signals failed.			
Description	The Event Manager cannot completely execute the deletion of open signals programmed with the NC command #SIGNAL REMOVE[...].		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Check the NC command #SIGNAL REMOVE[...]. Remove or modify any programmed unknown signal IDs, signal areas or channel numbers.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21316

Tangential selection or deselection during active inner corner selection not allowed.			
Description	When the inner corner selection of tool radius compensation (TRC) is active, the tangential selection or deselection with G05 is not permissible. The command G05 used is ignored. Example: **Wrong:** N10 G238 : N100 G41 **G05** N110 G01 X100 F1000 : N1000 M30 **Correct:** N10 G238 : **N100 G41** N110 G01 X100 F1000 : N1000 M30		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify the NC program. Remove G05.
Parameter	%1:	Current value [-]	
		G function used	
Error type	-		

ID 21317

Inner corner selection during active tool radius compensation not allowed.			
Description	When tool radius compensation (TRC) is active, a change to inner corner selection with G238 is not permissible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Program the selection of inner corner (G238) before selecting TRC (G41/G42).
Parameter	%1:	Error value [-]	
		Invalid G command	
Error type	1, Error message from NC program.		

ID 21318

Changing selection mode during active tool radius compensation not allowed.			
Description	When inner corner selection G238 of tool radius compensation (TRC) is active, it is not permitted to change selection mode with G138, G139 or G237.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Remove incorrect G function.
Parameter	%1:	Error value [-]	
		Invalid G command	
Error type	1, Error message from NC program.		

ID 21319

Full circle with radius greater than 214 m is not permissible.			
Description	A full circle with a radius greater than 214 m is programmed in the NC program. This exceeds the maximum permissible motion range of the axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Approach and reproduce the circle contour using linear blocks. Arcs of a circle with a radius greater than 214 m can only be programmed if their starting and end points are located inside the maximum permissible motion range of the axes.
Error type	1, Error message from NC program.		

ID 21320

Six axes are necessary for complete Cartesian transformation.			
Description	In the complete Cartesian transformation, The system detected that at least one axis of the first six axes is missing (P-CHAN-00006, P-CHAN-00035, P-CHAN-00003).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Channel configuration or Check and modify NC program. The first six axes must be present in the channel without gaps for a complete Cartesian transformation. White the coordinate system is active with #CS, the axis configuration of these six axes may not be changed.
Error type	1, Error message from NC program.		

ID 21321

Kinematic transformation must be selected for the effector coordinate system.

Description	For non-Cartesian machine kinematics, such as hexapod kinematics, selecting the effector coordinate system (#ECS ON) detects that the kinematic transformation is not enabled. Example: **Wrong:** N10 #KIN ID[26] N20 #TRAFO ON N30 G01 G90 X10 Y10 Z100 A15 B15 C0 F5000 N40 G01 G90 X-10 Y-10 Z100 A15 B15 C0 N50 **#TRAFO OFF** **N60 #ECS ON** N70 G01 G90 X25 Y25 Z0 A-15 B-15 C0 : N1000 M30 **Correct:** N10 #KIN ID[26] N20 #TRAFO ON N30 G01 G90 X10 Y10 Z100 A15 B15 C0 F5000 N40 G01 G90 X-10 Y-10 Z100 A15 B15 C0 **N50 #ECS ON** N70 G01 G90 X25 Y25 Z0 A-15 B-15 C0 N80 **#TRAFO OFF** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before activating the effector coordinate system (#ECS ON) with non Cartesian machine kinematics, the kinematic transformation must be active (#TRAFO ON). The channel parameters P-CHAN-00031, P-CHAN-00050 have no influence on the #ECS function for these kinematic structures and specification of the tool orientation via three spatial rotation angles.
Error type	1, Error message from NC program.		

ID 21322

Immediate exiting of tool offsets with this block mode not allowed.			
Description	The D word is programmed together with a G function in the same NC block. This changes the block mode and also the meaning of the programmed axis positions (e.g. G04, G63, G74, G92, G98, G99, G100...). This causes a conflict If the tool offsets (P-CHAN-00100) are configured for immediate exiting. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G92** X10 Y20 Z10 **D1** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 **G92** X10 Y20 Z10 N30 **D1** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program D word in a separate NC block.
Error type	1, Error message from NC program.		

ID 21323

For #TOOL ORI CS function active kinematic transformation is required.

Description	For non-Cartesian machine kinematics, such as hexapod kinematics, selecting the automatic alignment function (#TOOL ORI CS) detects that the kinematic transformation is not enabled. Example: **Wrong:** N10 #KIN ID[26] N20 #TRAFO ON N30 G01 G90 X10 Y10 Z100 A15 B15 C0 F5000 N40 G01 G90 X-10 Y-10 Z100 A15 B15 C0 N50 **#TRAFO OFF** **N60 #TOOL ORI CS** N70 G01 G90 X25 Y25 Z0 A-15 B-15 C0 : N1000 M30 **Correct:** N10 #KIN ID[26] N20 #TRAFO ON N30 G01 G90 X10 Y10 Z100 A15 B15 C0 F5000 N40 G01 G90 X-10 Y-10 Z100 A15 B15 C0 **N50 #TOOL ORI CS** N70 G01 G90 X25 Y25 Z0 A-15 B-15 C0 N80 **#TRAFO OFF** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	With non-Cartesian machine kinematics, kinematic transformation must be enabled (#TRAFO ON) before selecting the automatic alignment function.
Error type	1, Error message from NC program.		

ID 21326

TRANS_ID is out of data format.			
Description	When the command #COLL DEF[…] is programmed, the translation identifier TRANS_ ID exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use the permissible translation identifier.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21327

Programmed TRANS_ID not permissible.			
Description	In the #COLL DEF[...] command, the translation identifier TRANS_ID is assigned an incorrect value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use translation identifier which is within the permissible value range.

Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21328

ROT_ID is out of data format.			
Description	When the command #COLL DEF[…] is programmed, the identifier for rotary orientation ROT_ID exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use the permissible identifier for rotary orientation.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21329

Programmed ROT_ID not allowed.			
Description	In the command #COLL DEF[...] the programmed identification number for the rotary orientation ROT_ID has an invalid value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use an identification number for the rotary orientation, which is within the range of permissible values.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21330

Programmed rotation coefficient not permissible.			
Description	In the command #COLL DEF[...] or #COLL POS[...] one of the programmed rotation coefficients (XX - ZZ) has an invalid value.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Use rotation coefficients, which are within the range of permissible values.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21332

Invalid programming of translatory shifts.			
Description	When programming the command # COLL DEF [TRANS_ID.. ...] the translatory shifts (X.. Y.. Z..) were not completely programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The translatory shifts (X.. Y.. Z..) must be completely programmed for the command #COLL DEF [TRANS_ID.. ...].
Error type	1, Error message from NC program.		

ID 21333

Incorrect programming of rotary orientation.			
Description	When programming the command # COLL DEF [ROT_ID.. ...] the rotary coefficients (XX - ZZ) were not completely programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The rotary coefficients (XX - ZZ) must be completely programmed with the command #COLL DEF [ROT_ID..... ...].
Error type	1, Error message from NC program.		

ID 21335

Radius too small, circle is skipped .			
Description	A circular block was programmed with too small a radius. So no useful calculation of a centre point is possible.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	The circular block is skipped and transformed to a linear block with moving distance 0.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
		Circle radius in µm	
	%2:	Lower limit value [I[increments / revolution]	
		Smallest permissible circle radius	
Error type	1, Error message from NC program.		

ID 21336

Element positioning incorrectly programmed.			
Description	In the command #COLL POS [...] the position respectively the orientation of an element is not correctly programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The position must either be programmed by defining translatory shifts (X.. Y.. Z..) or by a predefined shift (TRANS_ID). The orientation has to be programmed by the complete definition of the rotation coefficients (XX - ZZ) or by a predefined orientation ID (ROT_ID). If no orientation is programmed, the default values are valid (XX=1, YY=1, ZZ=1, all other coefficients = 0).
Error type	1, Error message from NC program.		

ID 21337

No group name or pair name programmed.			
Description	When programming the command #COLL ON / OFF / CLEAR [...], no group name GROUP or element name ELEM is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Complete the command #COLL ON / OFF / CLEAR [...] with a permissible group name GROUP or element name ELEM.
Error type	1, Error message from NC program.		

ID 21338

Double programming in #COLL command.			
Description	In the command #COLL ... [...] redundant keywords are programmed several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #COLL ... [...].
Error type	1, Error message from NC program.		

ID 21339

Invalid programming of #COLL DEF command.			
Description	When the command #COLL DEF[…] is programmed, neither an identifier for a translation TRANS_ID nor for a rotary orientation ROT_ID is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Supplement missing identifier TRANS_ID or ROT_ID in the command.
Error type	1, Error message from NC program.		

ID 21340

Double programming in #TRC command.

Description	In the command #TRC keywords are programmed several times or in invalid combinations. Example with error: `#TRC [KERF_MASKING=15, CONV_CIR_TO_LIN=1, KERF_MASKING=10]`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of the #TRC command in the NC program [PROG].
Error type	1, Error message from NC program.		

ID 21341

Cancellation of request without axis registration.			
Description	This does not occur in normal operation. The message is only useful for diagnosis of resulting operation states.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Message is used for diagnosis of axis exchange problems.
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 21342

Memory for external variables is not configured.			
Description	The CNC was instructed by the PLC or another process (e.g. Object Browser) to provide the description of the memory allocation for the external variables. During this operation, the system detects that there is no memory allocated for using external variables.		
Response	Class	2	The description of the memory allocation for the external variables is not provided.
Solution	Class	7	The parameter P-STUP-00037 must be assigned a value > 0.
Error type	-		

ID 21343

Memory for external variables too small. Could not display layout of external variables.			
Description	The CNC received a request from the PLC or another process (e.g. ISG Object Browser) to provide the description of the memory allocation for external variables.		
Response	Class	3	Only part of the description of the memory provided for external variables fits in the existing memory.
Solution	Class	7	The parameter P-STUP-00037 must be assigned a value that is greater than the current value.
Parameter	%1:	Current value [-]	
		This is the index in the declaration (e.g.: var[**Memory for external variables too small. Could not display layout of external variables**]....) of an external variable from the list in which the external variables are defined.	
	%2:	Current value [-]	
		These are the names of the external variables as of which the information about the memory occupied by THE external variables could no longer be provided.	

Error type	-

ID 21344

Synchronous WAIT or SIGNAL READ with parameters not allowed.			
Description	Signal parameters can only be evaluated at decoder level; synchronised waiting with transfer of parameters #WAIT SYN[... P[0] = ...] is not allowed. See also [PROG// Section: Channel synchronisation – Waiting for signals]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of #WAIT command in the NC program. Modify the program run so that waiting is possible without synchronisation #WAIT[... P[0] = ...].
Error type	1, Error message from NC program.		

ID 21345

No license to use spline.			
Description	The license key installed with the CNC software package does not permit spline programming.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	To use the spline, the corresponding license key must be installed. Please contact the control manufacturer.
Parameter	%1:	Current value [-]	
		TRUE: Spline type REDBSPLINE programmed FALSE: Spline types AKIMA or BSPLINE programmed	
Error type	1, Error message from NC program.		

ID 21346

No license to use transformation.			
Description	The license key installed with the CNC software packet does not permit the programming of transformations (Cartesian/kinematic).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	To use transformations, the corresponding license key must be installed. Please contact the controller manufacturer.
Parameter	%1:	Error value [-]	
		ID of the selected kinematic	
	%2:	Error value [-]	
		Type of the selected kinematic	
Error type	1, Error message from NC program.		

ID 21347

The slave axis must not be a main spindle.	
Description	When a coupling group is selected (#ENABLE AX LINK [..] or #AX LINK ON [..]), the program detects that the slave axis of a coupling pair of spindle axes is the main spindle. This is not allowed.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Reverse definition of spindle coupling so that the master axis is the main spindle or, before the affected coupling group is selected, define the spindle master axis as the main spindle using #MAIN SPINDLE.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 21348

Helical interpolation with slope not permissible without 3rd dimension.			
Description	Helical interpolation is the superposition of a circular interpolation (plane of 1st and 2nd main axis) and a linear motion in the 3rd main axis. The message is generated if the position specified in the 3rd main axis (“Z axis”) is missing when helical motion is programmed. Example: **Wrong:** N10 G17 G90 X-10 Y0 Z0 F500 G161 N20 G03 I0 J0 K10 : N1000 M30 **Correct:** N10 G17 G90 X-10 Y0 Z0 F500 G161 N20 G03 I0 J0 **Z50** K10 (helical movement with 5 windings) : N1000 M30		
Response	Class	1	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Supplement the position in the 3rd main axis (“Z axis”) for helical motion.
Parameter	%1:	Initial value [0.1 µm or 0.0001°]	
		Start position of the 3rd main axis	
	%2:	Final value [0.1 µm or 0.0001°]	
		End position of the 3rd main axis	
Error type	1, Error message from NC program.		

ID 21349

Double programming in #TGC command.			
Description	In the command #TGC keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of the #TGC command in the NC program.
Parameter	%1:	State [-]	
Error type	1, Error message from NC program.		

ID 21350

Programmed minimum transition velocity is negative.

Description	The minimum block transition velocity on the path programmed with #TRANSVELMIN ON is negative.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the command. Value of programmed minimum block transition velocity must be greater than or equal to zero (nil).
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21351

Relative programmed path distance not permissible with this variable.			
Description	The use of the variables V.A.ACT_POS.** (current actual position, [PROG]) to program of a relative linear movement of the same axis is not allowed. This reaction is version-specific. Example: **Wrong:** N10 G90 X10 Y20 Z30 F500 G01 N20 **G91 XV.A.ACT_POS.X** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Change relative programming (G91) into absolute programming (G90).
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 21352

Invalid programming of #SCENE command.			
Description	When the command #SCENE ... [...] is programmed, either an incorrect index value for the keywords KEY or VALUE is programmed or necessary keywords are missing and the command is incomplete.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Correct the invalid value or complete the missing keywords according to [PROG] in the NC command.
Error type	1, Error message from NC program.		

ID 21353

Double programming in #SCENE_command.			
Description	In the command #SCENE ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #SCENE ... [...].
Error type	1, Error message from NC program.		

ID 21354

Too many link points programmed.			
Description	The command #SCENE DELETE [LINKPOINT<*name*> ...] contains too many programmed linkpoints that are to be deleted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the maximum number of linkpoints command to the permissible limit. If necessary, use step-by-step additional #SCENE DELETE [...] for deleting of further linkpoints.
Parameter	%1:	Error value [-]	
	%2:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21355

Too many graphic objects programmed.			
Description	Too many graphic objects are programmed in the commands #COLL DEF [...], #SCENE DELETE [...] or #GROUP ADD/ DELETE[...].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Reduce the maximum number of graphical objects in the command to the permissible limit. If necessary, use step-by-step additional commands to handle additional graphic objects.
Parameter	%1:	Error value [-]	
	%2:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21356

Too many groups programmed.			
Description	Too many groups are programmed in the commands #COLL DEF [...] or #SCENE DELETE [...].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Reduce the maximum number of groups in the command to the permissible limit. If necessary, treat other groups step by step in additional commands.
Parameter	%1:	Error value [-]	
	%2:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21357

Invalid programming of #GROUP command.

Description	When the command #GROUP ... [...] was programmed, some necessary keywords are missing and the command is incomplete.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Supplement missing keywords in command as described in [PROG].
Error type	1, Error message from NC program.		

ID 21358

Double programming in #GROUP-command.			
Description	In the command #GROUP ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #GROUP ... [...].
Error type	1, Error message from NC program.		

ID 21359

Value of the parameter set is out of data format.			
Description	The parameter set number (PARAM_SET) programmed in the #DRIVE command is outside the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible parameter set number.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21360

Motor number is out of data format.			
Description	The drive number (MOTOR) programmed in the #DRIVE command is outside the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible motor number.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21362

Redundant programming in #DRIVE command.			
Description	In the command #DRIVE, keywords are programmed several times or in invalid combinations. Example with error: `#DRIVE WR SYN[AXNR=1 MOTOR=4 PARAM_SET=2 ID=2 AX=X VAL=233]`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command #DRIVE remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21363

Axis to define control range not permissible.			

Description	Only the first 2 main axes, that form the working plane (G17, G18, G19) may be programmed for the definition of the contour of a control area. The third main axis and all other coupled axes shall not be programmed during the definition of a control area contour.
	Example:
	Wrong:
	`N10 #CONTROL AREA START [ID1 WORK POLY …]`
	`N20 G01 F1000 G90 X-150 Y75 (Starting point)`
	`N30 X-50 Y150 Z200`
	`:`
	`N130 X-150 Y75`
	`N140 #CONTROL AREA END`
	`:`
	`N1000 M30`
	
	Correct:
	`N10 #CONTROL AREA START [ID1 WORK POLY …]`
	`N20 G01 F1000 G90 X-150 Y75 (Starting point)`
	`N30 X-50 Y150 Z200`
	`:`
	`N130 X-150 Y75`
	`N140 #CONTROL AREA END`
	`:`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Ensure, that only the first 2 main axes are used for the definition of the contour of a control area. See also [PROG// Chapter - Definition of a control area]
Parameter	%1:	Logical axis number	
		Logical axis number of impermissible programmed axis	
Error type	1, Error message from NC program.		

ID 21364

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21365

Negative distance not permissible.			
Description	The values of the monitoring distances programmed with the keywords ERROR or WARNING of the command #COLL DEF [...] are negative.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	The invalid negative value is multiplied by –1 and the NC program is continued. Check and modify command #COLL DEF [...] before next program start. Assign the programmed monitoring distances with values greater than or equal to zero (nil).
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
Error type	1, Error message from NC program.		

ID 21366 - 21368

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21369

PDU received from tool management with invalid parameter index.			
Description	A transparent read access with V.G.WZ[j].GOBJECT[i].* to certain tool object data in an external tool database results in the NC channel receiving an object data record with an invalid index. The highest index may have a maximum value of 4 with a maximum of 5 permissible object data records.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check internal processes in the external tool database (e.g. PLC) during transparent read access. The maximum index in the object data set to be provided must be in the range 0 - 4.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21370

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21371

NC command only permissible with SYN.			
Description	The command #DRIVE may be programmed only with synchronisation (SYN). Example: `#DRIVE WR SYN[AX=X MOTOR=4 PARAM_SET=2 ID=2 VAL=233]`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Supplement keyword SYN in #DRIVE command.
Error type	1, Error message from NC program.		

ID 21372

Unknown or invalid logical axis number programmed.		
Description	The logical axis number programmed in a NC command (e.g. #IDENT , #DRIVE, S[MC_GearIn...], #SCENE, axis exchange commands) could not be assigned to a configured channel axis or an other available axis of the system.	
Response	Class	2 Abort NC program processing.
Solution	Class	3 Check and modify NC program. Only available and known axes of the system may be programmed by their logical axis numbers.
Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21373

Redundant programming in SAI command sequence.			
Description	In a SAI-specific command sequence (e.g. S[MC_GearIn...]) keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the SAI specific command sequence remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21374

Programmed SAI value is out of data format.

Description	The value of a keyword programmed in an SAI-specific command sequence (e.g. S[MC_GearIn...]) is outside the permitted value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Check the values of the keywords programmed for plausibility and adjust them if necessary.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21375

Invalid SAI value programmed.			
Description	The value of a keyword programmed in an SAI-specific command sequence (e.g. S[MC_GearIn...]) is not permitted. Example: `S[MC_GearIn Master=1 Invalid SAI value programmed. RatioDenominator=3 Mode=1]`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Check the values of the keywords programmed for plausibility and adjust them if necessary.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21376

No master axis programmed.			
Description	No master axis (MASTER) has been programmed in the SAI-specific command sequence for the gear coupling of axes S[MC_GearIn...].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command. Add the missing specification of the master axis (MASTER=<*expr*>) to the command sequence.
Error type	1, Error message from NC program.		

ID 21377

Invalid programming of #COLL PRINT command.			

Description	The keywords REL or LOG in the #COLL PRINT command may only be programmed if a log file FILE is specified.
	Examples:
	`Nxx #COLL PRINT [REL] -> Error`
	`Nxx #COLL PRINT [LOG] -> Error`
	`Nxx #COLL PRINT [FILE=prot1.log REL] ->OK`
	`Nxx #COLL PRINT [FILE=prot1.log LOG] ->OK`		

Response	Class	2	

Solution	Class	3	Check and modify NC program. If the #COLL PRINT command is programmed without specifying the FILE log file, either remove the programmed REL or LOG keywords or add a FILE log file to the command. [PROG// Description #COLL PRINT]
Error type	1, Error message from NC program.		

ID 21379

Redundant programming in CYCLE call.			
Description	In the cycle-specific command sequence (L CYCLE[NAME=...]) keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove redundant keywords in cycle-specific command sequence.
Error type	1, Error message from NC program.		

ID 21380

Cycle call not permissible if thread cutting or tapping is active.			
Description	A cycle call L CYCLE[NAME=...] is not permitted with active tapping/drilling (G33/ G63). Example: **Wrong:** N05 X0 Y0 Z0 N10 G91 N20 M03 S100 X100 M19 S.POS180 N30 G63 Z-100 F300 N40 Z100 S-200 **N45 L CYCLE [NAME=…]** N50 G01 X200 F3000 N60 G63 Z-70 F300 S200 N70 Z70 S-200 N80 M05 G01 X300 F1000 N90 M30 **Correct:** N05 X0 Y0 Z0 N10 G91 N20 M03 S100 X100 M19 S.POS180 N30 G63 Z-100 F300 N40 Z100 S-200 N50 G01 X200 F3000 **N55 L CYCLE [NAME=…]** N60 G63 Z-70 F300 S200 N70 Z70 S-200 N80 M05 G01 X300 F1000 N90 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before calling the cycle thread cutting, tapping must have been deselected by selecting another modal block type (e.g. G01).
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21381

Cycle call not permissible if modal cycle is active.			
Description	An attempt is made to call another cycle although a modal cycle (L CYCLE [NAME=... MODAL ..]) is still enabled. Example: **Wrong:** N05 X0 Y0 Z0 G00 **N10 L CYCLE [NAME=dummy1.cyc MODAL @P1=…]** N20 G01 X200 F3000 N30 G01 X300 F1000 **N40 L CYCLE [NAME=dummy2.cyc …]** N50 G01 Y300 F1000 N60 G01 Z300 F2000 N90 M30 **Correct:** N05 X0 Y0 Z0 G00 **N10 L CYCLE [NAME=dummy1.cyc MODAL @P1=…]** N20 G01 X200 F3000 N30 G01 X300 F1000 **N35 #DISABLE MODAL CYCLE** **N40 L CYCLE [NAME=dummy2.cyc …]** N50 G01 Y300 F1000 N60 G01 Z300 F2000 N90 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The modal cycle must first be deselected (#DISABLE MODAL CYCLE) before calling another cycle.
Error type	1, Error message from NC program.		

ID 21382

Impermissible write/read access to cycle parameters.			
Description	This error output is caused by a write/read access on a cycle parameter within a main program or by a direct call of a cycle as a stand-alone program. A write/read access to cycle parameters (@Pxx=...) may only be carried out in the cycle program itself (e.g. @P1=@P5+@P6) or in cycle call L CYCLE [NAME=... @Pxx=..]. Example: **Wrong:** %main N05 X0 Y0 Z0 G00 N20 G01 X200 F3000 **N25 @P1=10** N40 L CYCLE [NAME=dummy1.cyc …] N50 G01 Y300 F1000 N60 G01 Z300 F2000 N90 M30 **Correct:** %main N05 X0 Y0 Z0 G00 N20 G01 X200 F3000 **N25 V.L.VAR1=10** **N40 L CYCLE [NAME=dummy1.cyc @P1=V.L.VAR1…]** N50 G01 Y300 F1000 N60 G01 Z300 F2000 N90 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove write/read accesses in main or normal subroutines, or move them into the cycle. Start cycle only by a global sub program call within the main program (L CYCLE [...]).

Parameter	%1:	Current value [-]
	%2:	Error value [-]
Error type	1, Error message from NC program.	

ID 21383

Impermissible use of a cycle axis name.			

Description	This error output is caused by using a cycle axis name (@X..) in the main program, or by a direct call of a cycle as a stand-alone program.
	A cycle axis name can only be used in the cycle program itself (e.g. @X=@P5/2).
	
	Example with error:
	`N05 X0 Y0 Z0 G00`
	`N20 G01 X200 F3000`
	`N25 @X=10`
	`N40 L CYCLE [NAME=dummy1.cyc …]`
	`N50 G01 Y300 F1000`
	`N60 G01 Z300 F2000`
	`N90 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Only use cycle axis name in the cycle. Start cycle only by a global sub program call within the main program (L CYCLE [...]).
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21384

Impermissible use of a cycle centre point coordinate.			

Description	This error output is caused when using a cycle center point coordinate (@I..) in the main program, or by a direct call of a cycle as a stand-alone program.
	A cycle centre point coordinate may only be used in the cycle program itself (e.g. @I= @P6)
	
	Example with error:
	`N05 X0 Y0 Z0 G00`
	`N20 G01 @I=100 X200 F500`
	`N40 L CYCLE [NAME=dummy1.cyc …]`
	`N50 G01 Y300 F1000`
	`N60 G01 Z300 F2000`
	`N90 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. . Use cycle centre point coordinates only in the cycle. Start cycle only by a global sub program call within the main program (L CYCLE [...]).
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21385

Impermissible use of a cycle spindle name.			

Description	This error output is caused by using a cycle spindle name (@X..) in the main program, or by a direct call of a cycle as a stand-alone program.
	A cycle spindle name may only be used in the cycle program itself (e.g. @S=1000).
	
	Example with error:
	`N05 X0 Y0 Z0 G00`
	`N20 G01 X200 F500`
	`N25 @S=1000 M3`
	`N40 L CYCLE [NAME=dummy1.cyc …]`
	`N50 G01 Y300 F1000`
	`N60 G01 Z300 F2000`
	`N90 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. . Only use cycle spindle name in the cycle. Start cycle only by a global sub program call within the main program (L CYCLE [...]).
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21386

Wrong sequence of channel-specific signal/wait requests.			
Description	The ID of a #WAIT function programmed in the NC channel does not match the incoming signal ID.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Normal channel-specific signals (not broadcast) must be used by #WAIT in the order that they were generated by #SIGNAL. This means the IDs of the #SIGNAL/#WAIT must be processed in the correct order in NC channels.
Parameter	%1:	Current value [-]	
		Value of unexpected ID	
Error type	1, Error message from NC program.		

ID 21387

Conditional interpretation: #IF without corresponding #ENDIF programmed.

Description	A “Conditional interpretation” was programmed in incomplete syntax. An #IF must be terminated by an #ENDIF. Example: **Wrong:** N01 P1=0 **#IF** P1==1 N10 G00 X0 Y0 Z0 : N25 G01 X100 Y0 Z0 F1000 : N1000 M30 **Correct:** N01 P1=0 **#IF** P1==1 N10 G00 X0 Y0 Z0 : N25 G01 X100 Y0 Z0 F1000 **#ENDIF** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Complete the #IF/ #ENDIF instruction or delete the #IF.
Error type	1, Error message from NC program.		

ID 21388

Double programming in #OTC-command.			
Description	In the command #OTC ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #OTC ... [...].
Error type	1, Error message from NC program.		

ID 21389

Parameterization not permitted when OTC is active.

Description	The parameterization of the OTC function (Online Tool Compensation) may only take place before or simultaneously with the selection #OTC ON [...]. If OTC is already active, do not change the parameter settings any more. Example: **Wrong:** N05 G01 X100 **N10 #OTC ON** **N15 #OTC [LENGTH CONT WEAR_CONST_TOOL]** **or** **N10 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** **N15 #OTC [LENGTH AUTO WEAR_CONST_TOOL]** N20 G01 X200 : N900 #OTC OFF N1000 M30 **Correct:** N05 G01 X100 : **N10 #OTC [LENGTH CONT WEAR_CONST_TOOL]** **N15 #OTC ON** **or** **N10 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** : N20 G01 X200 : N900 #OTC OFF N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Adapt the sequence of parameterisation and selection of OTC.
Error type	1, Error message from NC program.		

ID 21390

OTC is already active. Renewed call not allowed.			
Description	With already active OTC function (Online Tool Compensation), the OTC must be selected again. Example: **Wrong:** N05 G01 X100 **N10 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** N20 G01 X200 : **Nxx #OTC ON** : N900 #OTC OFF N1000 M30 **Correct:** N05 G01 X100 **N10 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** N20 G01 X200 : N900 #OTC OFF N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant selection of OTC.

Error type	1, Error message from NC program.

ID 21391

Invalid tool type for OTC operation.			
Description	When the OTC function (Online Tool Compensation) is selected, it must be ensured that the tool used is a grinding tool. That is why the tool type entered in the tool data is checked. This error message is output if the active tool is not a grinding tool, i.e. the corresponding parameter P-TOOL-00001 is not correctly assigned in the tool data.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Enter tool tool type 2 for grinding tool (P-TOOL-00001) in the corresponding tool data, update lists and restart NC program.
Parameter	%1:	Error value [-]	
		Current tool type entered	
	%2:	Expected value [-]	
		Tool type for grinding tools	
Error type	1, Error message from NC program.		

ID 21392

OTC is not enabled. Deselection not permitted.			
Description	The programming of #OTC OFF is only allowed if the OTC function (Online Tool Compensation) was previously selected. Example: **Wrong:** N05 G01 X100 N20 G01 X200 : **N900 #OTC OFF** N1000 M30 **Correct:** N05 G01 X100 **N10 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** N20 G01 X200 : **N900 #OTC OFF** N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the unnecessary deselection of OTC (#OTC OFF) or select OTC before.
Error type	1, Error message from NC program.		

ID 21393

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21394

Enabled OTC at program end.			
Description	At program end (M02, M30) the online tool compensation selected by the command #OTC ON […] is still enabled.		
Response	Class	1	Abort NC program processing.
Solution	Class	1	Add NC program. Enter #OTC OFF before program end.
Error type	1, Error message from NC program.		

ID 21395

Plane selection during active OTC not allowed.			
Description	While OTC function (Online Tool Compensation) is enabled, it is not permitted to make a selection of planes with G17, G18 or G19. Example: **Wrong:** N10 G00 X0 Y0 Z0 **N20 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** **N100 G19** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #OTC ON [LENGTH CONT WEAR_CONST_TOOL] : **N90 #OTC OFF** **N100 G19** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program sequence. Deselect the OTC function with #OTC OFF before programming of plane selection.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21396

Changing kinematic transformation (RTCP) during active OTC not allowed.			
Description	It is not possible to select or change the kinematic transformation when the OTC function (Online Tool Compensation) is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Deselection of OTC function (#OTC OFF) before changing or selection of kinematic transformation.
Error type	1, Error message from NC program.		

ID 21397

Coordinate system transition during active OTC not allowed.

Description	During the transition of coordinate systems with the commands #CS, #ACS and/or #MCS, the OTC function (Online Tool Compensation) may not be enabled. This restriction applies both to the selection and deselection of the respective function. Example: **Wrong:** N10 G00 X0 Y0 Z0 **N20 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** **N30 #CS ON [10,20,30,0,0,45]** : N900 #OTC OFF N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 **N20 #CS ON [10,20,30,0,0,45]** **N30 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** : N900 #OTC OFF N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Select the ORC function only after the coordinate system transition (#OTC ON [...]).
Error type	1, Error message from NC program.		

ID 21398

Tool change during active OTC not allowed.			
Description	while the OTC function (Online Tool Compensation) is enabled, it is not possible to enable a different set of parameters for the tool geometry correction with the D word or #TOOL DATA. Example: **Wrong:** N10 G00 X0 Y0 Z0 **N20 #OTC ON [LENGTH CONT WEAR_CONST_TOOL]** : **N90 D2** N100 #OTC OFF : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 #OTC ON [LENGTH CONT WEAR_CONST_TOOL] : **N90 #OTC OFF** **N100 D2** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Deselect OTC function with #OTC OFF before programming of the D word or #TOOL DATA[…].
Error type	1, Error message from NC program.		

ID 21399

The continuation position of block search is located in a locked program area.			
Description	Block search is to continue at a position in the NC program that is located within an area locked to block search by #BLOCKSEARCH LOCKED/ RELEASED.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Select a start position, which is outside the locked area for block search or redefine the forbidden areas in NC program.
Error type	1, Error message from NC program.		

ID 21400

Global feed weighting is programmed negative.			
Description	The global feed weighting value programmed with G129 for rapid velocity is negative. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G129 = **-60** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 G129 = **60** : N1000 M30		
Response	Class	1	NC program processingContinue NC program processing. is continued.
Solution	Class	1	In the event of a conflict, the feed weighting is automatically set to 100%, and NC program processing is continued. Before the next program start, a meaningful value greater than zero (nil) should be programmed.
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
		Automatically corrected value of feed weighting	
Error type	1, Error message from NC program.		

ID 21401

Programmed PATH_DEV not allowed.			
Description	If, during the extended HSC programming (#HSC ON/OFF [BSPLINE...]), the keyword MERGE is set on 1 (TRUE), the deviation from path contour PATH_DEV must also be programmed with a value unequal to zero (nil).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Program the deviation from path contour PATH_DEV with a value unequal to zero or completely remove it from the command (in this case the related default value is valid, see [PROG]).
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21402

System error [▶ 12]	
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.

Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21403

The compensation settings of an axis involved in synchronous operation may not be modified.			
Description	The NC command <axis>[COMP ...] is used to change a compensation setting of an axis that is defined as master or slave axis in the currently active synchronous operation. Example: **Wrong:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=**X**] N40 #ENABLE AX LINK[1] : **N100 X[COMP ON CROSS]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=X] N40 #ENABLE AX LINK[1] : N90 **#DISABLE AX LINK[1]** **N100 X[COMP ON CROSS]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before using the <axis>[COMP ...] command, disable active synchronous operation with #DISABLE AX LINK or #AX LINK OFF (ALL).
Parameter	%1:	Logical axis number [-]	
		Number of the axis that conflicts with an active master or slave axis.	
	%2:	Current value [-]	
		Number of the active coupling group	
	%3:	Current value[-]	
		Number of the axes coupling pair	
Error type	1, Error message from NC program.		

ID 21404

Channel parameters: Multiple axes programming without active "multi_dimension_in_block" is not useful.					
Description	During start-up, the channel parameter check determines that it only makes sense to set P-CHAN-00148 to 1 if P-CHAN-00116 is also set to 1.				
Response	Class	1	Start-up of the control is continued.		
Solution	Class	1	During start-up in case of conflict the multiple axes programming is set to 0 (FALSE) and the start-up is continued. Before the next start-up, set the channel parameters P-CHAN-00116 and P-CHAN-00148 correctly:		
			P-CHAN-00116	P-CHAN-00148	Check result
			0	0	OK
			1	0	OK
			1	1	OK
			0	**1**	**NOT OK**

Parameter	%1:	Error value [-]
		Channel parameter P-CHAN-00148 incorrectly assigned
	%2:	Corrected value [-]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 21405

Double programming in #FGROUP-command.			
Description	In the command #FGROUP ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #FGROUP ... [...].
Error type	1, Error message from NC program.		

ID 21406

No license available for extended HSC programming.			
Description	The license key installed with the CNC software package does not permit extended HSC programming.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	To use the extended HSC programming, the corresponding license key must be installed. Please contact the control manufacturer.
Error type	1, Error message from NC program.		

ID 21407

WCS target value is out of data format.			
Description	When the motion limits in the WCS system are calculated with the command #GET WCS POSLIMIT, the resulting target position of one of the main axes exceeds the permissible numerical range. The value of the multiple ID shows the associated axis whose target position was calculated incorrectly: Multiple ID 1 → 1st main axis Multiple ID 2 → 2nd main axis Multiple ID 3 → 3rd main axis		
Response	Class	2	Abort NC program processing.
Solution	Class	3	This error can only occur due to incorrect parameterisation of the kinematic / Cartesian transformation. Check and correct the settings as necessary.
Parameter	%1:	Error value [-]	
		Calculated distance to the motion limit	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	

Error type	1, Error message from NC program.

ID 21408

#GET WCS POSLIMIT not allowed during active synchronous operation.			
Description	During active synchronous operation (#ENABLE AX LINK or #AX LINK ON) the command #GET WCS POSLIMIT is programmed in the NC program. This is not permitted. Example: **Wrong:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=X] **N40 #ENABLE AX LINK[1]** : **N100 #GET WCS POSLIMIT [VC1=1 VC2=0 VC3=0]** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N30 #SET AX LINK[1, B=X] N40 #ENABLE AX LINK[1] : **N90 #DISABLE AX LINK[1]** **N100 #GET WCS POSLIMIT [VC1=1 VC2=0 VC3=0]** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Deselect the active synchronous operation with #DISABLE AX LINK or #AX LINK OFF (ALL) before using the command #GET WCS POSLIMIT.
Error type	1, Error message from NC program.		

ID 21409

Current MCS position outside software limits.			
Description	When the motion limits in the WCS system are calculated with the command #GET WCS POSLIMIT, the system detects that at least the start position of one of the main axes in the machine coordinate system exceeds the corresponding software limits. As a result no motion limit can be determined with the programmed direction vector.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Before this function is selected, ensure that all the axes involved in the transformation are within the motion range defined by the software limit switches.
Parameter	%1:	Logical axis number [-]	
		Axis exceeds the software limit switches	
	%2:	Current value [0.1 µm or 0.0001°]	
		Axis position in the MCS	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Positive software limit switch	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Negative software limit switch	
Error type	1, Error message from NC program.		

ID 21410

Movement direction vector is 0.			
Description	The direction vector, programmed in the command #GET WCS POSLIMIT [...] is zero (nil) or too short. The calculation of a motion limit is not possible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The components of the direction vector must be programmed with a useful value.
Parameter	%1:	Current value [-]	
		First component of the direction vector (VC1)	
	%2:	Current value [-]	
		Second component of the direction vector (VC2)	
	%3:	Current value [-]	
		Third component of the direction vector (VC3)	
Error type	1, Error message from NC program.		

ID 21411

Double programming in #FILTER command.			
Description	In the command #FILTER ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #FILTER ... [...].
Error type	1, Error message from NC program.		

ID 21412

Second main axis not allowed.			
Description	In connection with the command #ROTATION and the function "rotation of several axes", the additional axes may only be linked to the second main axis (master axis) of the current plane. In case of error an inadmissible master axis is programmed as second main axis in the definition of the axes pairs. Example: **Wrong:** G00 X0 X1=0 X2=0 X3=0 Y0 Z0 G01 F1000 X1=0 X2=260 X3=520 X4=780 Y=-10 #ROTATION ON[ANGLE=1 AXPAIR[X1,Y] AXPAIR**[X2,Z]** AXPAIR[X3,Y]] G91 Y1=-1200 G91 Y1=1200 : N1000 M30 **Correct:** G00 X0 X1=0 X2=0 X3=0 Y0 Z0 G01 F1000 X1=0 X2=260 X3=520 X4=780 Y=-10 #ROTATION ON[ANGLE=1 AXPAIR[X1,Y] AXPAIR**[X2,Y]** AXPAIR[X3,Y]] G91 Y1=-1200 G91 Y1=1200 : N1000 M30		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Correct the definition of the axes pairs in the command #ROTATION
Error type	1, Error message from NC program.		

ID 21413

WAIT without SYN is not allowed.			
Description	The WAIT keyword may only be used in the #MACHINE DATA command if the axis parameter (SYN) is set to synchronised . Example: Set a software limit switch and wait in the channel: **Wrong:** G00 X0 Y0 Z0 : #MACHINE DATA [AX=X AXPARAM="kenngr.swe_pos 15000000" **WAIT**] : N1000 M30 **Correct:** G00 X0 Y0 Z0 : #MACHINE DATA **SYN** [AX=X AXPARAM="kenngr.swe_pos 15000000" **WAIT**] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Add the SYN keyword to the #MACHINE DATA command or remove WAIT. See also [PROG// Chapter Writing of machine data]
Error type	1, Error message from NC program.		

ID 21414

Spindle axis is not allowed.			
Description	A spindle axis was programmed with keywords AX or AXNR in # commands. But the programming is only allowed for path axes available in the NC channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.The # command may only be used for path axes in the NC channel.
Error type	1, Error message from NC program.		

ID 21415

Axis not homed! Automatic trafo selection not possible.			
Description	A kinematic transformation may only be selected if all axes involved are homed. If automatic transformation selection is set in the channel parameter P-CHAN-00151 (auto_enable_kin_trafo), an implicit check is made whether all transformation axes are homed when manual mode is selected (G200, G201) or in the first motion block in the NC program. If this is not the case, this warning message is output for simulation drives.		
Response	Class	1	Continue NC program processing. The transformation is not active.
Solution	Class	1	Check and modify NC program. When the channel parameter P-CHAN-00151 is active (auto_enable_kin_trafo), all transformation axes must be homed before the first motion (G74). Alternatively, P-AXIS-00014 can be set for axes with absolute encoders.

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of the non-homed axis
Error type	Abort NC program processing.	

ID 21416

New operation mode is not permitted if 3D tool geometry compensation is active.			
Description	With active 3D tool geometry compensation (TGC) in NC program a new operation mode (OPMODE) is programmed. This is not allowed. Example: **Wrong:** N10 #TGC ON [OPMODE=2] : **N.. #TGC [OPMODE=1]** : N.. #TGC OFF : N1000 M30 **Correct:** N10 #TGC ON [OPMODE=2] : **N.. #TGC OFF** **N.. #TGC ON [OPMODE=1]** **... or** **N.. #TGC [OPMODE=1]** **N.. #TGC ON** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The setting of a new operation mode (OPMODE) is only allowed either in combination with a repeated selection of 3D-TGC or during inactive 3D-TGC.
Parameter	%1:	Current value [-]	
Error type	Abort NC program processing.		

ID 21417

Invalid TGC mode.			
Description	In the #TGC […] command the keyword OPMODE has an invalid value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the #TGC command, assign a valid value to the machining mode.
Parameter	%1:	Current value [-]	
		Keyword	
	%2:	Error value [-]	
		Invalid TGC mode	
Error type	Abort NC program processing.		

ID 21418

Double programming in #MEAS command.

Description	In the command #MEAS [...], keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command #MEAS [...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21419

Override factor is out of range of permissible value.			
Description	A programmed path or axis-specific override factor (e.g. FEED_FACT…) exceeds the permissible numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program a permissible override factor (valid values: 0.1% ... 200%, see also [PROG//Description of command #OVERRIDE or Axis-specific override programming]).
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	Abort NC program processing.		

ID 21420

Double programming in #OVERRIDE command.			
Description	In the command #OVERRIDE [...], keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command #OVERRIDE [...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21422

Double programming in OVERRIDE command sequence.			
Description	In the spindle specific override command S[OVERRIDE...], keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command sequence S[OVERRIDE...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21424

Channel parameters: Invalid parameter setting of edge banding.

Description	When the edge banding parameters are checked during ramp-up, the system determines that it only makes sense to set P-CHAN-00029 to 1 if P-CHAN-00149 and P-CHAN-00153 are set to 0.					
Response	Class	1	Controller start-up is continued.			
Solution	Class	1	In the event of a conflict, block global edge banding P-CHAN-00029 is assigned 0 (FALSE) during start-up and start-up is continued. Before the next start-up, set the channel parameters P-CHAN-00029, P-CHAN-00149 and P-CHAN-00153 correctly:			
			P-CHAN-00029	P-CHAN-00149	P-CHAN-00153	Check the result
			0	1	0	OK
			0	0	1	OK
			0	1	1	OK
			1	**0**	**0**	**OK**
			1	**1**	**0**	**NOT OK**
			1	**0**	**1**	**NOT OK**
			1	**1**	**1**	**Channel parameters: Invalid parameter setting of edge banding.**
Parameter	%1:	Error value [-]				
		Channel parameter P-CHAN-00029 incorrectly assigned				
	%2:	Corrected value [-]				
Error type	2, Error message by data transfer from parameter list into control device.					

ID 21425

Axis not homed! Automatic trafo selection not possible.			
Description	A kinematic transformation may only be selected if all axes involved are homed. If automatic transformation selection is set in the channel parameter P-CHAN-00151 (auto_enable_kin_trafo), an implicit check is made whether all transformation axes are homed when manual mode is selected (G200, G201) or in the first motion block in the NC program. If this is not the case, this warning message is output for real drives.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. When the channel parameter P-CHAN-00151 is active (auto_enable_kin_trafo), all transformation axes must be homed before the first motion (G74). Alternatively, P-AXIS-00014 can be set for axes with absolute encoders.
Parameter	%1:	Logical axis number [-]	
		Number of the non-homed axis	
Error type	1, Error message from NC program.		

ID 21426

Tool length is smaller than minimum limit value.	
Description	When the tool data accepts the data, the system detects that the tool length is smaller than the permissible minimum length defined by the channel parameter P-CHAN-00156.

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check whether a sister tool is available with sufficient length. Then execute NC program with that tool. Alternatively: Set P-CHAN-00156 to 0; the check for minimum length is then not enabled. Restart controller.
Parameter	%1:	Error value [-]	
		Length of requested tool	
	%2:	Lower limit value [-]	
		Configured minimum length P-CHAN-00156	
Error type	1, Error message from NC program.		

ID 21427

Tool radius is smaller than minimum limit value.			
Description	When the tool data accepts the data, it is detected that the tool radius is smaller than the permissible minimum radius, defined by the channel parameter P-CHAN-00157 .		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check whether a alternative tool with a sufficient tool radius is available. Then execute NC program with that tool. Alternatively: Set P-CHAN-00157 to 0; the check for minimum length is then not enabled. Restart controller.
Parameter	%1:	Error value [-]	
		Radius of the requested tool.	
	%2:	Lower limit value [-]	
		Configured minimum radius P-CHAN-00157	
Error type	1, Error message from NC program.		

ID 21429

Name of external variable structure too long.			
Description	User-defined variable types can be defined for the external variables, e.g. for variable structures (see [EXTV]). However, the default name for the variable type (struct[i].name) is too long.		
Response	Class	2	Name of the variable structure is shortened.
Solution	Class	3	Correct the name of the variable structure.
Parameter	%1:	Current value [-]	
		Incorrect name	
	%2:	Current value [-]	
		Index i of the variable type definition	
	%3:	Error value [-]	
		Length of given name	
	%4:	Limit value [-]	
		Maximum permissible length of name	
	%5:	Corrected value [-]	
		Corrected name for variable type	

Error type	-

ID 21430

Invalid character in the name of the external variable structure.			
Description	User-defined variable types can be defined for external variables, e.g. for variable structures (see [EXTV]). However, the default name for the variable type (struct[i].name) contains at least one invalid character. Permitted characters are letters, the numbers 0-9 and an underline.		
Response	Class	2	The variable type is deleted.
Solution	Class	3	Correct the name of the variable type.
Parameter	%1:	Error value [-]	
		Incorrect name	
	%2:	Current value [-]	
		Index i of the variable type definition.	
	%3:	Current value [-]	
		Position of the invalid character in the name	
Error type	-		

ID 21431

Name of external variable structure corresponds elementary data type.			
Description	User-defined variable types can be defined for the external variables, e.g. for variable structures (see [EXTV]). However, the default name for the variable type (struct[i].name) corresponds to an elementary data type of the control controller. Default variable types of the numerical control are: BOOLEAN SGN08, SGN16, SGN32 UNS08, UNS16, UNS32 REAL64 STRING		
Response	Class	2	The variable type is deleted.
Solution	Class	3	Change name of the variable type.
Parameter	%1:	Error value [-]	
		Incorrect name	
	%2:	Current value [-]	
		Index i of the variable type definition.	
Error type	-		

ID 21432

Name of the structure element too long.	
Description	User-defined variable types can be defined for the external variables, e.g. for variable structures (see [EXTV]). However, the default name for the structure element (struct[i].element[j].name) is too long.

Response	Class	2	Name of the structure element is shortend.
Solution	Class	3	Correct the name of the structure element.
Parameter	%1:	Current value [-]	
		Name of affected variable type definition	
	%2:	Current value [-]	
		Incorrect element name	
	%3:	Error value [-]	
		Length of given name	
	%4:	Limit value [-]	
		Maximum permissible length of name	
	%5:	Current value [-]	
		Corrected name for variable type	
Error type	-		

ID 21433

Invalid character in the name of the structure element.			
Description	User-defined variable types can be defined for external variables, e.g. for variable structures (see [EXTV]). However, the default name for the structure element (struct[i].element[j].name) contains at least one invalid character. Permitted characters are letters, the numbers 0-9 and an underline.		
Response	Class	2	The structure element is deleted.
Solution	Class	3	Correct the name of the structure element.
Parameter	%1:	Current value [-]	
		Incorrect name	
	%2:	Current value [-]	
		Index i of the variable type definition.	
	%3:	Current value [-]	
		Index j of the structure element	
	%4:	Error value [-]	
		Incorrect name	
	%5:	Current value [-]	
		Position of the invalid character in the name	
Error type	-		

ID 21434

Variable type identifier of structure element too long.			
Description	User-defined variable types can be defined for the external variables, e.g. for variable structures (see [EXTV]). However, the default variable type of the structure element (struct[i].element[j].type) is too long.		
Response	Class	2	Variable type is shortened.
Solution	Class	3	Correct the variable type of the structure element.

Parameter	%1:	Current value [-]
		Name of affected variable type definition
	%2:	Current value [-]
		Name of affected structure element
	%3:	Current value [-]
		Given variable type
	%4:	Error value [-]
		Length of given variable type
	%5:	Limit value [-]
		Maximum permissible length of name
Error type	-	

ID 21435

Invalid character in the variable type of the structure element.			
Description	User-defined variable types can be defined for external variables, e.g. for variable structures (see [EXTV]). However, the default name for the structure element (struct[i].element[j].name) contains at least one invalid character. Permitted characters are letters, the numbers 0-9 and an underline.		
Response	Class	2	The variable structure is deleted.
Solution	Class	3	Correct the variable type of the structure element.
Parameter	%1:	Current value [-]	
		Name of affected variable structure	
	%2:	Current value [-]	
		Index j of affected structure element	
	%3:	Current value [-]	
		Name of affected structure element	
	%4:	Error value [-]	
		Incorrect variable type name	
	%5:	Current value [-]	
		Position of the invalid character	
Error type	-		

ID 21436

No elements in external variable structure.			
Description	User-defined variable types can be defined for external variables, e.g. for variable structures (see [EXTV]). However, the specified variable type has no elements (struct[i].element[j]).		
Response	Class	2	The variable structure is deleted.
Solution	Class	3	Define elements for the variable structure.
Parameter	%1:	Current value [-]	
		Name of affected variable structure	
	%2:	Current value [-]	
		Index i of affected variable structure	
Error type	-		

ID 21437

Variable type of external variable too long.			
Description	The identifier for the variable type of the external variable is too long (see [EXTV]).		
Response	Class	2	Variable type is shortened.
Solution	Class	3	Correct the variable type of the external variable.
Parameter	%1:	Current value [-]	
		Name of the effected variable	
	%2:	Current value [-]	
		Specified variable type	
	%3:	Error value [-]	
		Length of given variable type	
	%4:	Limit value [-]	
		Maximum permissible length of name	
Error type	-		

ID 21438

Invalid character in the variable type of the external variable.			
Description	The variable type identifier of the external variable contains at least one invalid character (see [EXTV]). Permitted characters are letters, the numbers 0-9 and an underline.		
Response	Class	2	External variable is deleted
Solution	Class	3	Correct the variable type identifier
Parameter	%1:	Current value [-]	
		Name of the affected external variable	
	%2:	Current value [-]	
		Index of the affected external variable	
	%3:	Error value [-]	
		Incorrect variable type	
	%4:	Current value [-]	
		Position of the invalid character	
Error type	-		

ID 21439

Memory index of the external variable exceeds valid range.			
Description	The external variables are stored in the memory in 24-byte blocks (see [EXTV]). The memory block of an external variable is addressed with the memory index var[i].index . However, the value found is too high.		
Response	Class	2	The external variable is deleted.
Solution	Class	3	Correct the memory index of the variable.

Parameter	%1:	Current value [-]
		Name of the affected external variable
	%2:	Current value [-]
		Index i of the affected external variable
	%3:	Error value [-]
		Incorrect value for memory index
	%4:	Upper limit value [-]
		Maximum permissible value for memory index
Error type	-	

ID 21440

Maximum nesting depth in external variable structure exceeded.			
Description	User-defined variable types can be defined for external variables, e.g. for variable structures (see [EXTV]). The elements of a variable type definition (struct[i].element[j]) can also be a variable structure. However, the possible nesting depth is limited.		
Response	Class	2	The external variable is deleted.
Solution	Class	3	Reduce the nesting depth of the structure definitions.
Parameter	%1:	Current value [-]	
		Name of the affected external variable	
	%2:	Current value [-]	
		Index of the affected variable	
	%3:	Current value [-]	
		Current nesting depth	
	%4:	Upper limit value	
		Maximum permissible nesting depth	
Error type	-		

ID 21441

Variable type in external variable structure unknown.			
Description	User-defined variable types can be defined for external variables, e.g. for variable structures (see [EXTV]). However, the variable type of a structure element (type[I].element[j].type) is unknown. It must be a default variable type of the controller or a self-defined type.		
Response	Class	2	The external variable is deleted.
Solution	Class	3	Correct the variable type of the structure element.
Parameter	%1:	Current value [-]	
		Name of the variable structure	
	%2:	Current value [-]	
		Name of the affected external variable	
	%3:	Error value [-]	
		Name of the unknown variable type	
Error type	1, Error message from NC program.		

ID 21442

No variable type for structure element specified.			
Description	User-defined variable types can be defined for external variables, e.g. for variable structures (see [EXTV]). No variable type (struct[i].element[j].type) was specified for an element of the variable structure.		
Response	Class	2	Variable structure definition is deleted.
Solution	Class	3	Specify a variable type for the structure element.
Parameter	%1:	Current value [-]	
		Name of affected variable structure	
	%2:	Current value [-]	
		Name of affected structure element	
	%3:	Error value [-]	
		Incorrect variable type	
Error type	-		

ID 21443

Incompatible variable types. Assignment operation not possible.			
Description	The assignment operation cannot be performed because the variable types on the left and right side are not compatible (see [EXTV]), e.g. if an SGN32 variable is assigned to an external structure variable: var[1].name var_1 var[1].type STRUCT_1 var[2].name var_2 var[2].type SGN32 Incorrect assignment in the NC program: V.E.var_1 = V.E.var_2		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Error type	1, Error message from NC program.		

ID 21444

Structure copy only possible with equal structure types.

Description	Copying entire variable structures (structure copy) in the NC program is only permitted between variables with identical variable types (var[i].type, see [EXTV]). Example: var[1].name var_1 var[1].type STRUCT_1 var[2].name var2_1 var[2].type STRUCT_2 var[3].name var2_2 var[3].type STRUCT_2 NC program: Error: V.E.var_1 = V.E.var_2 Correct: V.E		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program or change the variable type.
Parameter	%1:	Error value [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21445

Different sizes of variable structures or variable arrays. Copying not possible.			
Description	Copying entire variable structures (structure copy) in the NC program is only permitted between variables with identical variable type var[i].type and same size var[i].size (see [EXTV]).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Parameter	%1:	Error value [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21446

Assignment operation not permitted for variable structures or arrays.			
Description	For external variables that are a variable structure (see [EXTV]), only the assignment operator "=" is allowed (structure copy). The assignment operators "+=", "-=", "*=" and "/=" may not be used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Error type	1, Error message from NC program.		

ID 21447

Structure copy only possible with two external variables.			
Description	Copying a whole variable structure (structure copy) is only possible between two external variables (see [EXTV]). They must both have the same variable type. Example: var[1].name var1 var[1].type STRUCT_1 var[2].name var2 var[2].type STRUCT_1 Assignment in the NC program: Error: V.E.var1 = 123 Correct: V.E.var1 = V.E.VAR2		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Error type	1, Error message from NC program.		

ID 21448 - 21450

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21451

Too many tools programmed per tool unit.			
Description	Too many tools T=... are grouped into a tool unit (UNIT) in the #TOOL DEF [...] command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In #TOOL DEF [...], do not exceed the maximum permissible number of tools per unit.
Parameter	%1:	Error value [-]	
		Decoded number of tools up to now	
	%2:	Limit value [-]	
		Maximum number of tools per unit	
Error type	1, Error message from NC program.		

ID 21452

Invalid UNIT value programmed.	
Description	The tool unit (UNIT) programmed in the NC command #TOOL DEF [...] can no longer be stored in the internal working data.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the number of tool units defined or if possible replace redundant tool units with new unit definitions.
Parameter	%1:	Error value [-]	
		ID number of the programmed UNIT	
	%2:	Lower limit value [-]	
		Maximum number of permissible UNIT definitions	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21453

Double programming in #TOOL DEF command.			
Description	In the command #TOOL DEF [...] keywords are programmed several times or in wrong combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command #TOOL DEF [...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21454

For the T function in NC channel still no complete tool data set is available.			
Description	No data is yet available for the tools grouped in the programmed tool unit (UNIT). This is required in particular in order to consider tool offsets.		
Response	Class	2	Abort NC program processing.
Solution	Class	1	1st option: Set channel parameter P-CHAN-00014 to 1 (TRUE). This implicitly completes the tool data when programming the T word. 2nd option: Check and modify the NC program. Program the corresponding D word before the T word. This first completes the tool data and is then available for the T word.
Parameter	%1:	Current value [-]	
		Programmed T number (UNIT)	
Error type	Abort NC program processing.		

ID 21455

Combined string exceeds maximum permissible string length.			
Description	Two strings are merged using the "+" operator. However, the resulting string is too long.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.

Parameter	%1:	Error value [-]
		Length of the combined string
	%2:	Upper limit value [-]
		Maximum permissible string length
Error type	Abort NC program processing.	

ID 21456

Assignment operation not permitted for string variables.			
Description	Only the assignment operators "=" and "+=" can be used for string variables. The operators "-=", "*=" and "/=" are not permitted. Example of external variables: var[1].name str1 var[1].type STRING Error: V.E.str1 *= “Hello world!” Correct: V.E.str1 = “Hello” V.E.str1 += “world!”		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Error type	1, Error message from NC program.		

ID 21457

Incompatible data types in mathematical calculation or string operation.			
Description	The found expression contains both mathematical and string expressions. It cannot be evaluated. Example: Error: V.E.string = „Peter“ + 23 Correct: V.E.string = „Peter“ + „23“ V.E.string = „Peter„ + INT_TO_STR[23]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct NC program. Use converting functions.
Parameter	%1:	Current value [-]	
Error type	Abort NC program processing.		

ID 21458

Operator not permitted for string expressions.

Description	For strings, only the "+" operator for joining two strings or the comparison operators "==" and "!=" are allowed. All other operators such as "-" or "*“ are not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Parameter	%1:	Error value [-]	
Error type	Abort NC program processing.		

ID 21459

Invalid argument type for function call.			
Description	The type of argument for the function is invalid e.g. a string expression was transferred to a mathematical function. Example: Error: SIN[“Peter”] STR_TO_INT[123] Correct: SIN[1.23] STR_TO_INT[“123”]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	Abort NC program processing.		

ID 21460

String expression expected.			
Description	The result of the given expression is expected to be a character string e.g. since it is assigned to a string variable. Example: Error: V.E.string = 100 + 10 Correct: V.E.string = „1“ + „10“ V.E.string = INT_TO_STR[100 + 10]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct NC program. Use conversion functions.
Error type	1, Error message from NC program.		

ID 21461

Mathematical term expected.			
Description	The result of the given expression is expected to be a numerical value e.g. since it is assigned to an integer or REAL64 variable. Example: Error: V.E.sgn32 = „1" + „10“ Correct: V.E.sgn32 = 100 + 10 V.E.sgn32 = STR_TO_INT[„1“ + „10“]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct NC program. Use conversion functions.
Error type	1, Error message from NC program.		

ID 21462

String for variable too long.			
Description	A string is assigned to a variable of type string but the string is too long for the variable.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Parameter	%1:	Error value [-]	
		Length of string	
	%2:	Upper limit value [-]	
		Maximum string length of variable	
Error type	1, Error message from NC program. .		

ID 21463

Argument of integer string conversion exceeds valid range.			
Description	The function INT_TO_STR can be used to convert an integer number into a character string. However, the specified numerical value for the function exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program.
Parameter	%1:	Error value [-]	
		Specified value for conversion	
	%2:	Lower limit value [-]	
		Minimum permissible value	
	%3:	Upper limit value [-]	
		Maximum permissible value	
Error type	1, Error message from NC program. .		

ID 21464

Numeric character for string to number conversion expected.			
Description	The STR_TO_INT or STR_TO_REAL functions can be used to convert character strings to integer or floating point numbers. The digits 0 ... are used in the character string for the conversion. 9.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct NC program.
Error type	1, Error message from NC program.		

ID 21465

String too long when converting from string to numerical value.		
Description	The STR_TO_INT or STR_TO_REAL functions can be used to convert character strings into integer or floating point numbers. However, the string contains too many numeric characters.	
Response	Class	2 Abort NC program processing.
Solution	Class	3 Correct the NC program.
Parameter	%1:	Upper limit value [-]
		Maximum permissible length of name
Error type	1, Error message from NC program.	

ID 21466

Result of conversion from string to number exceeds valid range.		
Description	The STR_TO_INT function can be used to convert a character string into an integer value. However, the calculated value exceeds the valid range.	
Response	Class	2 Abort NC program processing.
Solution	Class	3 Correct the NC program.
Parameter	%1:	Error value [-]
		Converted value
	%2:	Lower limit value [-]
		Minimum permissible value
	%3:	Upper limit value [-]
		Maximum permissible value
Error type	1, Error message from NC program.	

ID 21467

Exponent of string to floating point number conversion exceeds valid range.		
Description	The STR_TO_REAL function can be used to convert a character string into a floating point number. However, the calculated exponent during conversion exceeds the valid range.	
Response	Class	2 Abort NC program processing.
Solution	Class	3 Correct the NC program.
Parameter	%1:	Upper limit value [-]
		Maximum permissible exponent

Error type	11, Error message from NC program.

ID 21468

'REPEAT' must be programmed exclusively in NC-block.			

Description	In the same NC block, a control block instruction was programmed with further NC commands. No further NC command may be programmed after a $REPEAT.
	Example:
	`Wrong:`
	N10 G00 X0 Y0 Z0 P1=0
	N20 **$REPEAT P1=P1+1**
	:
	N50 $UNTIL P1>2
	:
	N1000 M30
	
	`Correct:`
	N10 G00 X0 Y0 Z0 P1=0
	N20 **$REPEAT**
	N30 P1=P1+1
	:
	N50 $UNTIL P1>2
	:
	N1000 M30		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands after $REPEAT to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 21469

Unexpected UNTIL. Does not match the current control block.

Description	A control block statement was programmed in incomplete syntax. A $UNTIL can only be programmed in combination with a previous $REPEAT. Example: **Wrong:** N10 G00 X0 Y0 Z0 P1=1 N20 **$UNTIL P1>2** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 P1=1 N15 **$REPEAT** : N20 **$UNTIL P1>2** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the program flow sequence. Insert the $REPEAT instruction or delete the $UNTIL.
Error type	1, Error message from NC program.		

ID 21470

After 'UNTIL' only the condition is permissible.			
Description	In the same NC block, a control block instruction was programmed with further NC commands. After a $UNTIL instruction, only the corresponding condition may be programmed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 P2=10 N30 $REPEAT : N40 **$UNTIL P2 > 20 G01 X100 F1000** : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N20 P2=10 N30 $REPEAT : N40 **$UNTIL P2 > 20** N45 **G01 X100 F1000** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the not permissible NC commands to other NC blocks or delete them.
Error type	1, Error message from NC program.		

ID 21471

Not enough parameters for function call.

Description	Too few parameters were transferred to the programmed function. Example: N10 V.E.string = LEFT[“Hello world!”] Correct: N10 V.E.string = LEFT[“Hello world!“, 5]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
		Name of programmed function	
	%3:	Error value [-]	
		Number of specified parameters	
	%4:	Expected value [-]	
		Expected number of parameters	
Error type	1, Error message from NC program.		

ID 21472

Parameter value for number of characters in string operation invalid.			
Description	The number of desired characters must be specified for the string operation. However, the value of the parameters is smaller or greater than permissible. Example: N10 V.E.string = LEFT["Hello world!”, 100000] Correct: N10 V.E.string = LEFT[“Hello world!”, 5]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program
Parameter	%1:	Current value	
		Name of programmed function	
	%2:	Current value [-]	
		Number of incorrect parameter	
	%3:	Error value [-]	
		Incorrect value	
	%4:	Lower limit value [-]	
		Minimum permissible value	
	%5:	Upper limit value[-]	
		Maximum permissible value	
Error type	1, Error message from NC program.		

ID 21473

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	

Solution	Class	8	Requires controller restart.

ID 21474

Too many parameters for function call.			
Description	Too many parameters were passed to the programmed function. Example: N10 V.E.string = LEFT[“Hello world!“, 5, 6] Correct: N10 V.E.string = LEFT[“Hello world!“, 5]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21475

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21476

Streaming data has no correct end-of-line mark.			
Description	The data package read in over the streaming interface exceeds the permissible internal cache length.		
Response	Class	-	Abort NC program processing.
Solution	Class	-	Check whether the NC program contains an overlength NC row. If possible, split it into several lines. Make sure that end-of-line markers are correct in the NC program as per Windows (carriage return + line feed).
Error type	-		

ID 21477

Backward jump during streaming not allowed.			
Description	In streaming mode, a backward jump with $GOTO is to be executed in the NC program. This is not allowed.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Linearise the program flow or design it so that no backward jumps with $GOTO are required. Note: Forward jumps with $GOTO in streaming mode are allowed!
Error type	1, Error message from NC program.		

ID 21478

Loop during streaming not allowed.			
Description	In streaming mode, a loop ($FOR, $WHILE, $DO, $REPEAT...) is to be executed in the NC program. This is not allowed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Linearise or redesign the program sequence so that loops are no longer necessary.
Error type	1, Error message from NC program.		

ID 21479

Call of local subroutine during streaming not allowed.			
Description	In streaming mode, a local subroutine (LL...) is to be called in the NC program. This is not allowed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Linearise the program flow or design it so that no local subroutines are required. Replace any local subroutines with global subroutines (L...).
Error type	1, Error message from NC program.		

ID 21480

Jump inside a loop is not allowed.			
Description	Within a loop ($FOR, $WHILE, $DO, $REPEAT...) a jump shall be executed with $GOTO. This is not allowed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Linearise or redesign the program sequence within the loop in a way, that jumps will not be necessary any longer. **Note:** A loop always can be left completely.
Parameter	%1:	Error value [-]	
		Name of the loop with the programmed jump	
Error type	1, Error message from NC program.		

ID 21481

Active synchronous axes are not permitted to move independently.	
Description	When synchronous operation is enabled (#SET AX LINK[...]) an independent axis motion is programmed for a selected master axis.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the sequence of the NC program, for example deselect synchronous operation before programming the "master" axis as an independent axis.
Parameter	%1:	Logical axis number [-]	
		Number of the axis which is already moved as master axis	
	%2:	Current value [-]	
		Number of the selected coupling group	
Error type	1, Error message from NC program.		

ID 21482

Channel parameters: Invalid parameter setting of synchronisation tolerance.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21483

Channel parameters: Invalid parameter setting for velocity factor on hold limits.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21484

Channel parameters: Invalid parameter setting of hold limit tolerance.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21485

Name of the external variable is defined twice.			
Description	During the initialization of the external variables, it is detected that a variable name is already used in another definition. Access to the incorrect variable is not possible in the NC program. The name of the external variable is assigned /0 [EXTV].		
Response	Class	2	Start-up of the control is continued.
Solution	Class	3	Check and change the name of the external variable before the next controller start-up in the list of external variables. To avoid error messages, enter unique names in the list of external variables.
Parameter	%1:	Current value [-]	
		Name of the multiple defined variable names	
	%2:	Current value [-]	
		List index of the first variable of the same name	
	%3:	Error value [-]	
		List index of the second variable of the same name	
Error type	-		

ID 21486

Double programming in #MIRROR-command.			
Description	In the command #MIRROR [...] keywords are programmed several times or in wrong combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command #MIRROR [...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21487

Linkage of (A)CS during active #MIRROR not allowed.			
Description	Further coordinate systems should be activated after selection of #MIRROR [...].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Coordinate systems must be linked before #MIRROR [...] is selected.
Parameter	%1:	Current value [-]	
		Type of the error causing coordinate system.	
Error type	1, Error message from NC program.		

ID 21488

Value for the maximum radius difference exceeds limit.			
Description	During start-up, a check of the channel parameters found that the absolute radius difference P-CHAN-00171 is greater than the permissible value of 100mm.		
Response	Class	1	Controller start-up is continued.

Solution	Class	1	In case of conflict, the absolute radius difference P-CHAN-00171 is set to 1000000 [0.1µm] at start-up and start-up is continued.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21489

Double programming in #ORI MODE-command.			
Description	In the command #ORI MODE [...] keywords are programmed several times or in wrong combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command #ORI MODE [...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21490

Invalid number of transformation axes at ORI MODE selection.			
Description	The #ORI MODE[] function can only be used or selected in conjunction with complete transformations with orientation axes [KITRA].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the NC program.
Parameter	%1:	Current value [-]	
		Kinematic ID	
	%2:	Error value [-]	
		Number of transformation axes of active kinematic transformation.	
	%3:	Lower limit value [-]	
		Minimum number of necessary transformation axes.	
Error type	1, Error message from NC program.		

ID 21491

Invalid kinematic type for ORI MODE selection.			
Description	The #ORI MODE[] function can only be used or selected in conjunction with complete transformations with orientation [KITRA], The kinematic ID can be defined in the tool or the channel parameters or can be selected in the NC program via #KIN ID[] command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the NC program and if necessary the tool parameter or channel parameters.
Parameter	%1:	Current value [-]	
		Active kinematic ID.	
Error type	1, Error message from NC program.		

ID 21492

Error in calculation angle from vector components.			
Description	The #ORI MODE[] function is enabled. Rotation about the tool axis causes no change in tool orientation if the rotation is the last in the rotation sequence. There is a change in tool coordinate system orientation. The tool coordinate system is defined by kinematic parameters defined by channel or tool parameters.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct parameters.
Parameter	%1:	Current value [-]	
		First vector component	
	%2:	Current value [-]	
		Second vector component	
	%3:	Current value [-]	
		Third vector component	
	%4:	Current value [-]	
		Axis index of rotation	
Error type	1, Error message from NC program.		

ID 21493

Value for the maximum relative radius difference exceeds limit.			
Description	At start-up, a check of the channel parameters found that the relative radius difference P-CHAN-00172 is greater than the permissible value of 500.		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In case of conflict, the absolute radius difference P-CHAN-00172 at start-up is set to 500 [0.1%] and start-up is continued.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21494

Additive 'M' value is out of data format.			
Description	The additional value programmed with the M word (M<expr>=<Additive_expr>) is outside the permitted numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible additional M value (negative or positive integer). [PROG//Section: M/H functions with optional additional information]
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	

Error type	1, Error message from NC program.

ID 21495

Additive 'H' value is out of data format.			
Description	The additional value programmed with the H word (M<expr>=<Additive_expr>) is outside the permitted numerical range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible additional H value (negative or positive integer). [PROG//Section: M/H functions with optional additional information]
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21496

Coupling group is already active. Selection is ignored.			
Description	The selection of a coupling group with #ENABLE AX LINK [...] or #AX LINK ON [...] is not executed because this coupling group is already enabled.		
Response	Class	1	Continue NC program processing. Re-selection is ignored.
Solution	Class	1	Check and modify NC program. Remove redundant selection of coupling group.
Parameter	%1:	Current value [-]	
		Number of the already active coupling group	
Error type	1, Error message from NC program.		

ID 21497

Channel parameters: Number of channel axes exceeds permissible limit of the CNC export version.			
Description	The CNC export version only permits a limited number of path axes per NC channel. At start-up, the channel parameter check found that the configured total number of path axes P-CHAN-00003 in the axis group was exceeded.		
Response	Class	3	Controller start-up is aborted.
Solution	Class	7	Include the permissible number of path axes in the configuration.
Parameter	%1:	Error value [-]	
		Configured number of path axes	
	%2:	Limit value [-]	
		Permissible number of path axes	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21498

Contouring mode in combination with master axis not permitted.			
Description	The parameterisation with #CONTOUR MODE [DIST_SOFT...] cannot be executed because at least one channel axis is configured as master axis.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program or configuration. Either select a different contouring mode or configure all channel axes with an axis mode according to P-AXIS-00015.
Parameter	%1:	Current value [-]	
		Identifier of programmed contouring process	
	%2:	Logical axis number [-]	
		Number of the axis which is defined as master axis	
Error type	1, Error message from NC program.		

ID 21499

Channel parameters: Restoring of default axes configuration after reset with active "reset_no_axis_to_axv" not allowed.			
Description	During start-up, the channel parameter check determines that it only makes sense to set P-CHAN-00179 to 1 if P-CHAN-00075 is set to 0.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	In the event of a conflict, P-CHAN-00075 is assigned 0 (FALSE) during the start-up, and the start-up is continued. Before the next start-up, set the channel parameters P-CHAN-00179 and P-CHAN-00075 correctly.
Parameter	%1:	Error value [-]	
		Channel parameter P-CHAN-00075 incorrectly assigned	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

2.3.7 ID-range 21500-21749

ID 21500

There are pending invalid control blocks at local subroutine end.			
Description	In a local subroutine, an invalid control block statement (e.g. $IF-$ENDIF) was not closed, i.e. an assigned $ENDIF is missing. Invalid in this context means that the programmed condition is false and the NC blocks in this branch are not executed. Example: **Wrong:** **%L sub1** N10 G00 X0 Y0 Z0 N20 P1=0 N30 **$IF P1 == 1** N40 Y100 N100 X100 N110 M29 ← $ENDIF of $IF P1.. is missing. %main N010 X0 Y0 Z0 N020 LL sub1 : N1000 M30 **Correct:** **%L sub1** N10 G00 X0 Y0 Z0 N20 P1=0 N30 **$IF P1 == 1** N40 Y100 N100 X100 N105 **$ENDIF** ← Invalid $IF is terminated N110 M29 % main N010 X0 Y0 Z0 N020 LL sub1 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Close the control block statement correctly. Complete the missing $ENDIF/$ENDSWITCH... .
Parameter	%1:	Error value [-]	
		Number of open control blocks	
	%2:	Current value [-]	
		Initial keyword of incomplete control block statement (e.g. $IF)	
	%3:	Expected value [-]	
		Closing keyword of control block statement (e.g. $ ENDIF)	
Error type	1, Error message from NC program.		

ID 21501

There are pending invalid control blocks at main program end.

Description	In a main program, an invalid control block statement (e.g. $IF-$ENDIF) was not closed, i.e. an assigned $ENDIF is missing. Invalid in this context means that the programmed condition is false and the NC blocks in this branch are not executed. Example: **Wrong:** **% main** N10 G00 X0 Y0 Z0 N20 P1=0 N30 **$IF P1 == 1** N40 Y100 N100 X100 N150 M30 ← $ENDIF of $IF P1.. is missing. **Correct:** **% main** N10 G00 X0 Y0 Z0 N20 P1=0 N30 **$IF P1 == 1** N40 Y100 N100 X100 N105 **$ENDIF** ← Invalid $IF is ended N150 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Close the control block statement correctly. Complete the missing $ENDIF/$ENDSWITCH... .
Parameter	%1:	Error value [-]	
		Number of open control blocks	
	%2:	Current value [-]	
		Initial keyword of incomplete control block statement (e.g. $IF)	
	%3:	Expected value [-]	
		Closing keyword of control block sequence (e.g. $ ENDIF)	
Error type	1, Error message from NC program.		

ID 21502

There are pending invalid control blocks at global subroutine end.	
Description	In a global subroutine, an invalid control block statement (for example, $IF-$ENDIF) was not closed, that is, an assigned $ENDIF is missing. Invalid in this context means that the programmed condition is false and the NC blocks in this branch are not executed. Example: **Wrong:** **% glob_sub** N10 G00 X0 Y0 Z0 N20 P1=0 N30 **$IF P1 == 1** N40 Y100 N100 X100 N150 M29 ← $ENDIF of $IF P1.. is missing. **Correct:** **% glob_sub** N10 G00 X0 Y0 Z0 N20 P1=0 N30 **$IF P1 == 1** N40 Y100 N100 X100 N105 **$ENDIF** ← Invalid $IF is ended N150 M29

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Close the control block statement correctly. Complete the missing $ENDIF/$ENDSWITCH... .
Parameter	%1:	Error value [-]	
		Number of open control blocks	
	%2:	Current value [-]	
		Initial keyword of incomplete control block statement (e.g. $IF)	
	%3:	Expected value [-]	
		Closing keyword of control block sequence (e.g. $ ENDIF)	
Error type	1, Error message from NC program.		

ID 21503

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21504

Invalid index for fixed rotary axis.			
Description	The #ORI MODE[] function was programmed. Three vector components which define tool orientation result in two rotary angles; the third rotary angle can be assigned 0 or another fixed value. A parameter can be used to specify the index for the fixed rotary axis when converting vector components into rotary angles. This index was programmed incorrectly or the channel parameter is invalid. The index can also be preset by the channel parameter P-CHAN-00178.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the NC program.
Parameter	%1:	Error value [-]	
	%2:	Expected value [-]	
Error type	1, Error message from NC program.		

ID 21505

Invalid mode for rotation sequence.			
Description	The #ORI MODE[] function was programmed. The function supports different rotation sequences of orientation axes, e.g. typically Y(aw) P(itch) R(oll). The rotation sequence is a property of the selected kinematic transformation. The rotation sequence can be set via the channel parameter P-CHAN-00112. The active kinematic transformation [KITRA] does not support the required rotation sequence.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the NC program.

Parameter	%1:	Current value [-]
Error type	1, Error message from NC program.	

ID 21506

Maximum number of programmed channel axes exceeds limit.			
Description	The #CHANNEL INIT [ACTPOS...] command has too many axes programmed (AX, AXNR) to request current values.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the number of programmed axes.
Parameter	%1:	Error value [-]	
		Programmed number of axes	
	%2:	Limit value [-]	
		Permissible maximum number of axes	
Error type	1, Error message from NC program.		

ID 21507

Double programming in #CHANNEL INIT-command.			
Description	In the command #CHANNEL INIT [ACTPOS...], keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. In the command #CHANNEL INIT [ACTPOS...] remove the redundant keywords.
Error type	1, Error message from NC program.		

ID 21508

Programming of channel axes in combination with CMDPOS not allowed.			
Description	The #CHANNEL INIT [CMDPOS] command for initializing nominal positions has axes programmed (AX, AXNR). Because command positions always are initialized for all channel axes, the programming of specific axes is not allowed.		
Response	Class	2	Abort NC program processing..
Solution	Class	3	Check and modify NC program. In the command #CHANNEL INIT [CMDPOS] remove the programmed axes.
Error type	1, Error message from NC program.		

ID 21509

In NC block #ADD must be the last NC command.			
Description	Other NC commands (e.g. G functions, axis positions, variables, etc.) can be programmed in the NC block before the command #ADD [...] but not after it.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program.. Change the sequence of commands in the NC block so that #ADD [...] is always the last command in the sequence. Example: **Wrong:** % main N10 G00 X0 Y0 Z0 N20 X100 **#ADD["ID123"] Y200** N30 Z100 : N1000 M30 **Correct:** % main N10 G00 X0 Y0 Z0 N20 X100 **Y200 #ADD["ID123]** N30 Z100 : N1000 M30
Error type	1, Error message from NC program.		

ID 21510

Read access on distance value in combination with active unit "time" not allowed.			
Description	The corner distance of the velocity look-ahead parameters should be read in mm or inch with V.G.SPEED_LIMIT.DIST_TO_CORNER or V.G.SPEED_LIMIT.DIST_FROM_CORNER . This read access is not possible, because the corresponding parameters currently are defined as time values in seconds.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The read access on the time based corner distances have to be programmed with V.G.SPEED_LIMIT.TIME_TO_CORNER or V.G.SPEED_LIMIT.TIME_FROM_CORNER **Note:** The active unit can be read with the variable V.G.SPEED_LIMIT.TIME (0: distance, 1: time).
Parameter	%1:		Error value [-]
			Identifier of the programmed variable
	%2:		Current value [-]
			Value of P-CHAN-00018
Error type	1, Error message from NC program.		

ID 21511

Read access on time value in combination with active unit "distance" not allowed.

Description	The time-based corner distance of the velocity look-ahead parameters should be read in seconds with V.G.SPEED_LIMIT.TIME_TO_CORNER or V.G.SPEED_LIMIT.TIME_FROM_CORNER . This read access is not possible, because the corresponding parameters currently are defined as distance values in mm or inch.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The read access on the time based corner distances have to be programmed with V.G.SPEED_LIMIT.DIST_TO_CORNER or V.G.SPEED_LIMIT.DIST_FROM_CORNER Hint: **Note:** The active unit can be read with the variable V.G.SPEED_LIMIT.TIME (0: distance, 1: time).
Parameter	%1:	Error value [-]	
		Identifier of the programmed variable	
	%2:	Current value [-]	
		Value of P-CHAN-00018	
Error type	1, Error message from NC program.		

ID 21512

Read access on pre output value "path" in combination with synchronization mode not allowed.			
Description	There is a programmed read access with V.G.M_FCT[<M_Nummer>].PRE_OUTP_PATH or V.G.H_FCT[<H_Nummer>].PRE_OUTP_PATH on the early output that can be read as lead distance P-CHAN-00070 in mm or inch. This read access is not permitted since the corresponding synchronization mode of this M/H function is not of type MEP_SVS (P-CHAN-00041).		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Set synchronization mode MEP_SVS in the corresponding M/H function and define path advance with P-CHAN-00070. Then, in NC program, a read access with V.G.M_FCT[<M_Nummer>].PRE_OUTP_PATH or V.G.H_FCT[<H_Nummer>].PRE_OUTP_PATH is possible. **Note:** The current synchronisation mode can be read with the variables V.G.M_FCT[<M_Nummer>].SYNCH or V.G.H_FCT[<H_Nummer>].SYNCH Hint:

Parameter	%1:	Error value [-]
		Identifier of the programmed variable
	%2:	Current value [-]
		Identifier of the techno function
	%3:	Current value [-]
		Current synchronization mode of the M/H function
	%4:	Current value [-]
		Number of the M/H function
Error type	1, Error message from NC program.	

ID 21513

Read access on pre output value "time" in combination with synchronization mode not allowed.			
Description	There is a programmed read access with V.G.M_FCT[<M_Nummer>].PRE_OUTP_TIME or V.G.H_FCT[<H_Nummer>].PRE_OUTP_TIME on the early output as time advance P-CHAN-00070 in seconds. This read access is not permitted, since the corresponding synchronization mode of this M/H function is not of type MET_SVS (P-CHAN-00041).		
Response	Class	2	Abort NC program processing.

Solution	Class	3	In the corresponding M/H function, set synchronization mode MET_SVS and define path advance with P-CHAN-00070. Then, in NC program, a read access with V.G.M_FCT[<M_Nummer>].PRE_OUTP_TIME or V.G.H_FCT[<H_Nummer>].PRE_OUTP_TIME is possible. **Note:** The current synchronisation mode can be read with the variables V.G.M_FCT[<M_Nummer>].SYNCH or V.G.H_FCT[<H_Nummer>].SYNCH Hint:

Parameter	%1:	Error value [-]
		Identifier of the programmed variable
	%2:	Current value [-]
		Identifier of the techno function
	%3:	Current value [-]
		Current synchronization mode of the M/H function
	%4:	Current value [-]
		Number of the M/H function
Error type	1, Error message from NC program.	

ID 21514

Invalid programmed orientation vector .			
Description	The #ORI MODE[...] function was programmed. The direction vector need not be normalised, but the vector value must be greater than 0 [KITRA].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the NC program.

Parameter	%1:	Current value [-]
		U-component of orientation vector
	%2:	Current value [-]
		V-component of orientation vector
	%3:	Current value [-]
		W-component of orientation vector
Error type	1, Error message from NC program.	

ID 21515

Invalid plane mode for tool coordinate system axes.			
Description	The #ORI MODE[...] function was programmed. The plane mode programmed with the keyword TOOL_AX_IN_PLANE is unknown.		
Response	Class	1	Continue NC program processing.

Solution	Class	1	Before the next program start, assign a permitted identification number to the plane mode,as in [PROG// Programming and configuration of 6-axis kinematics (robot) (#ORI MODE)].
Parameter	%1:	Error value [-]	
		Unknown plane mode.	
	%2:	Corrected value [-]	
		Corrected plane mode	
Error type	1, Error message from NC program. [-]		

ID 21516

INPUT value is out of data format.			
Description	The number of probing input on the drive programmed in the #MEAS command (INPUT) is outside the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible input number.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21517

Invalid combination of keywords in # command.			
Description	The plausibility check of the programmed # command detected forbidden combinations of keywords.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Error type	1, Error message from NC program.		

ID 21518

Channel-specific external variable exceeds valid address range. Variable not created!

Description	When the address is calculated, the system detects that the channel-specific external variable exceeds the valid address range. At start-up the external variable is skipped and as a result it cannot be used. Possible solutions: In the configuration list [EXTV] for external variables, check the memory indices "index" or byte offsets "byte_offset" and, if necessary, reduce them if they were explicitly specified. In the configuration list [EXTV] for array variables, check and reduce the element "array_elements". If the configuration of the global variables is correct, the memory for the external variables can be increased. The number of available 24-byte memory blocks is specified in the start-up list (see [STUP]) in the parameter P-STUP-00037. The required value is reported in the follow-up error message P-ERR-21520 [▶ 520].		
Response	Class	2	Controller start-up is continued, external variable is skipped!
Solution	Class	7	Correct configuration or increase memory for external variables
Parameter	%1:	Error value [-]	
		Name of the variables affected P-EXTV-00001	
	%2:	Current value [-]	
		Index of the variables affected	
	%3:	Error value[Byte]	
		Memory offset of the variables affected	
	%4:	Error value[Byte]	
		Memory available	
	%5:	Current value [-]	
		Memory required	
Error type	-		

ID 21519

Global external variable exceed valid address range. Variable not created!			
Description	When the address is calculated, the system detects that the global external variable (cross-channel scope) exceeds the permitted address range. At start-up the external variable is skipped and as a result it cannot be used. Possible solutions: In the configuration list [EXTV] for external variables, check the memory indices "index" or byte offsets "byte_offset" and, if necessary, reduce them if they were explicitly specified. In the configuration list [EXTV] for array variables, check and reduce the element "array_elements". If the configuration of the global variables is correct, the memory for the external variables can be increased. The number of available 24-byte memory blocks is specified in the start-up list (see [STUP]) in the parameter P-STUP-00037. The required value is reported in the follow-up error message P-ERR-21520 [▶ 521].		
Response	Class	2	Controller start-up is continued, external variable is skipped!
Solution	Class	7	Correct configuration or increase memory for external variables

Parameter	%1:	Error value [-]
		Name of the variable affected P-EXTV-00001
	%2:	Current value [-]
		Index of the affected variable
	%3:	Error value[Byte]
		Memory offset of the variable affected
	%4:	Error value [Byte]
		Memory available
	%5:	Current value [-]
		Memory required
Error type	-	

ID 21520

Memory for external variables too small! Not all external variables available!			
Description	This is a follow-up error message from ID 21518 [▶ 519] or ID 21519 [▶ 520]. When the address of one (or several) global or channel-specific external variables is addressed, the system detected that the calculated address exceeds the permissible memory range. Parameter 2 of this error message reads the required number of 24-byte memory blocks (see P-STUP-00037) which are required to create all external variables. The error message may also be displayed when V.CH variables are used. The memory space for V.CH variables is specified in P-CHAN-00424.		
Response	Class	2	Controller start-up is continued but not all configured external variables are available.
Solution	Class	6	Reduce the number of variables or increase the memory for the V.E. or V.CH variables.

Parameter	%1:	Error value [-]
		Current configured number of 24-byte memory blocks P-STUP-00037
	%2:	Expected value [-]
		Required number of 24-byte memory blocks P-STUP-00037
	%3:	Expected value [-]
		Optional parameter: Required memory space
	%4:	Current value [-]
		Optional parameter: Identifier of variable type ”E” for V.E. variables ”CH” for V.CH variables
Error type	-	

ID 21521

Negative FEED_CONT not allowed.			
Description	The programming of a negative velocity (FEED_CONT) for synchronisation with a conveyor belt in the NC command #SYNC IN / OUT [...] is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.

Parameter	%1:	Error value [-]
		Invalid value of velocity

Error type	1, Error message from NC program.

ID 21522

Negative CONV_VEL not allowed.			
Description	The programming of a negative conveyor velocity (CONV_VEL) in the NC command #SYNC IN / OUT [...] is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC command.
Parameter	%1:	Error value [-]	
		Invalid value of conveyor velocity	
Error type	1, Error message from NC program.		

ID 21523

Trafo PTP programming is already enabled.			
Description	If the PTP motion control is already enabled, the NC command #PTP ON is programmed again.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Remove the redundant PTP command.
Error type	1, Error message from NC program.		

ID 21524

Trafo PTP programming is already inactive.			
Description	If the PTP motion guide has already been deselect, the NC motion control #PTP OFF is programmed again.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify NC program. Remove the redundant PTP command.
Error type	1, Error message from NC program.		

ID 21525

Trafo PTP programming during active tool radius compensation not allowed.

Description	When tool radius compensation (WRK, G41/G42) is active, selection/deselection of PTP motion control with #PTP ON / OFF is not permitted. Example: **Wrong:** N10 X50 Y50 N20 **G41** N30 X100 … Nxx **#PTP ON** … Nxx M30 **Correct:** N10 X50 Y50 N20 G41 N30 X100 Nxx **G40** … Nxx **#PTP ON** … Nxx M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Only activate PTP motion control after deselecting tool radius compensation (G40).
Error type	1, Error message from NC program.		

ID 21526

Trafo PTP programming during active OTC not allowed.			
Description	With active online tool compensation (OTC, #OTC ON), the selection/deselection of PTP motion control with #PTP ON / OFF is not allowed. Example: **Wrong:** N10 X50 Y50 N20 **#OTC ON** N30 X100 … Nxx **#PTP ON** … Nxx M30 **Correct:** N10 X50 Y50 N20 #OTC ON N30 X100 Nxx **#OTC OFF** … Nxx **#PTP ON** … Nxx M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Do not activate PTP motion control until online tool compensation has been deselected (#OTC OFF).
Error type	1, Error message from NC program.		

ID 21527

Trafo PTP programming is allowed during active kinematic transformation only.

Description	The programming of PTP motion control with #PTP ON / OFF is only allowed during active kinematic transformation (#TRAFO ON) Example: **Wrong:** N10 X50 Y50 N20 X100 … Nxx **#PTP ON** … Nxx #**PTP OFF** Nxx M30 **Correct:** N10 X50 Y50 N20 X100 N30 **#TRAFO ON** … Nxx **#PTP ON** … Nxx **#PTP OFF** … Nxx **#TRAFO OFF** Nxx M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Activate PTP motion control only after selection of the kinematic transformation (#TRAFO ON).
Error type	1, Error message from NC program.		

ID 21528

Deselection of kinematic transformation during active trafo PTP programming not allowed.			
Description	The deselection of kinematic transformation with #TRAFO OFF is not permitted with active PTP motion control (#PTP ON). Example: **Wrong:** N10 X50 Y50 N20 X100 N30 #TRAFO ON … Nxx **#PTP ON** … Nxx **#TRAFO OFF** Nxx #PTP OFF … Nxx M30 **Correct:** N10 X50 Y50 N20 X100 N30 #TRAFO ON … Nxx #PTP ON … Nxx **#PTP OFF** … Nxx **#TRAFO OFF** Nxx M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust NC program sequence. Do not deselect kinematic transformation until PTP motion control has been deselected (#PTP OFF).

Error type	1, Error message from NC program.

ID 21529

Deselection of Cartesian transformation during active trafo PTP programming not allowed.			
Description	The deselection of cartesian transformation with #CS OFF or #ACS OFF is not permitted with active PTP motion control (#PTP ON). Example: **Wrong:** N10 X50 Y50 N20 X100 N30 #CS ON … Nxx **#PTP ON** … Nxx **#CS OFF** Nxx #PTP OFF … Nxx M30 **Correct:** N10 X50 Y50 N20 X100 N30 #CS ON … Nxx #PTP ON … Nxx **#PTP OFF** … Nxx **#CS OFF** Nxx M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Do not deselect Cartesian transformation until PTP motion control has been deselected (#PTP OFF).
Error type	1, Error message from NC program.		

ID 21530

Circular interpolation during active trafo PTP programming not allowed.			
Description	The circle programming (G02/G03) is not allowed when PTP motion control (#PTP ON) is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.. Remove the circular programming or shift it behind #PTP OFF .
Error type	1, Error message from NC program.		

ID 21531

Coordinate system transition during active PTP programming not allowed.			
Description	A transition to the machine coordinate system with #MCS ON is not allowed when PTP motion control ((#PTP ON) is active.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Remove the transition to the machine coordinate system (#MCS ON) or shift it behind #PTP OFF .
Error type	1, Error message from NC program.		

ID 21532

Commanded start block number not found.			
Description	When ordering a NC program start, a block number (Nxx) unequal to zero (nil) can be declared, which defines the real point of program execution. The corresponding block number is also searched for in subroutines. The message (warning) is output, if the declared block number has not been found until main program end M30.		
Response	Class	2	No NC program processing.
Solution	Class	1	Check the NC program for the commanded block number and complete it if applicable. Note: NC blocks without block number are always executed during the search. So if you use this function, please attach to all relevant NC blocks a unique block number.
Parameter	%1:	Current value [-]	
		Commanded start block number.	
Error type	1, Error message from NC program.		

ID 21533

Commanded abort block number not found.			
Description	When ordering a NC program start, a block number (Nxx) unequal to zero (nil) can be declared, which defines the real point of program abortion. The corresponding block number is also searched for in subroutines. The message (warning) is output, if the declared block number has not been found until main program end M30.		
Response	Class	1	NC program processing until M30.
Solution	Class	1	Check the NC program for the commanded block number and complete it if applicable. Note: NC blocks without block number are always executed during the search. So if you use this function, please attach to all relevant NC blocks a unique block number.
Parameter	%1:	Current value [-]	
		Commanded abort block number.	
Error type	1, Error message from NC program.		

ID 21534

Double programming in #EXPORT-command.			
Description	In the command #EXPORT ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #EXPORT ... [...].

Error type	1, Error message from NC program.

ID 21535

Error during writing of export file.			
Description	An error occurred when writing the export file with the command #EXPORT ... [... [...].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check NC command. None of the data to be written may be available or the file system is write-protected.
Parameter	%1:	Current value [-]	
		String ID for the export type	
Error type	1, Error message from NC program.		

ID 21536

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21537

Double programming in #TIMER-command.			
Description	In the command #TIMER ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant keywords from the command #TIMER ... [...].
Error type	1, Error message from NC program.		

ID 21538

Channel parameters: Table of path-dependent dynamic weightings contains an active line with invalid parameter value.			
Description	At start-up, the channel parameter check detects that the table of dynamic weights P-CHAN-00191 - P-CHAN-00194 contains an impermissible minimum limit value. The smallest permissible limit value is 1% (10 per mille).		
Response	Class	1	Controller start-up is continued.
Solution	Class	1	In the event of a conflict, P-CHAN-00191 - P-CHAN-00194 is assigned a default value at start-up, and start-up is continued.

Parameter	%1:	Current value [-]
		Table index (line) with the invalid value
	%2:	Error value [0.1%]
		Weighting factor for rapid feed velocity
	%3:	Error value [0.1%]
		Weighting factor for rapid feed acceleration
	%4:	Error value [0.1%]
		Weighting factor for rapid feed ramp time
	%5:	Corrected value [0.1%]
		Value of the corrected weighting factor
Error type	2, Error message by data transfer from parameter list into control device.	

ID 21539

Spline or polynomial contouring during relative orientation programming not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21540

Double programming of a thread offset angle.			
Description	When programming multi-start threads with G33, the thread offset angle (S.OFFSET) is declared several times in NC block.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the redundant thread offset angle.
Error type	1, Error message from NC program.		

ID 21541

Programmed thread offset angle is out of data format.			
Description	The thread offset angle (S.OFFSET) when multi-start threads are programmed with G33 exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program a permissible thread offset angle.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 21542

No further NC commands allowed after subroutine or cycle call.			
Description	After a subroutine call with L, LL or L CYCLE, no further NC commands may be programmed in the same NC block. Example: **Wrong:** N10 G91 X100 L GLOBALUP.NC **Y200 C60** N20 G90 X100 LL LOCALUP.NC **Z300** N20 G01 X100 L CYCLE […] **Y200 F2000** **Correct:** N10 G91 X100 **Y200 C60** L GLOBALUP.NC N20 G90 X100 **Z300** LL LOCALUP.NC N20 G01 X100 **Y200 F2000** L CYCLE […]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Move the wrong positioned NC commands before the sub program call or if possible to the following NC block.
Error type	1, Error message from NC program.		

ID 21543

Tool dynamic data for PLC spindle not effective.			
Description	The assigned spindle is configured in the channel parameters as a spindle controlled directly by the PLC (P-CHAN-00069). No tool-specific dynamic data can be assigned to this kind of spindles .		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check spindle configuration and modify if necessary. Either the spindle must be created as a controlled spindle in the channel (see P-CHAN-00069) or the spindle assignment of the dynamic data must be removed from the tool data (P-TOOL-00012).
Parameter	%1:		Error value [-]
			ID of tool parameter
	%2:		Error value [-]
			Value of tool parameter
	%3:		Expected value [-]
			Correct value of tool parameter
	%4:		Current value [-]
			Tool number
	%5:		Current value [-]
			ID of channel parameter with value
Error type	1, Error message from NC program.		

ID 21544

Invalid write/read access on cycle parameter data.			
Description	The variable V.G.@P[i].VALID can check in the whether a certain transfer parameter (@P<> i) was programmed in the cycle call L CYCLE [...] and is therefore valid. This error message is output if the variable V.G.@P[i].VALID is programmed outside of a cycle.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Use the variable V.G.@P[i].VALID inside cycles only.
Parameter	%1:	Current value [-]	
		General ID of a cycle parameter	
	%2:	Error value [-]	
		Number of the programmed cycle parameter in V.G.@P[i].VALID	
Error type	1, Error message from NC program.		

ID 21545

Invalid position in character string for string operation.			
Description	A start position in the string must be specified for the string operation. However, the value of the parameter is smaller or greater than allowed. Example: N10 V.E.string = DELETE[“Hello world!”, 7, **100**] Correct: N10 V.E.string = DELETE[“Hello world!“, 7, **5**]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program. Set the start position within the string.
Parameter	%1:	Current value [-]	
		Name of programmed function	
	%2:	Current value [-]	
		Number of incorrect parameter	
	%3:	Error value [-]	
		Incorrect value	
	%4:	Lower limit value [-]	
		Minimum permissible value	
	%5:	Upper limit value [-]	
		Maximum permissible value	
Error type	1, Error message from NC program.		

ID 21546

Deceleration limit on the path is negative or 0. Channel parameter may not be set.			
Description	The velocity limit on the path programmed by #VECTOR LIMIT ... [...] is negative or zero.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the command. Programmed deceleration limit must have a value greater than zero (nil).
Parameter	%1:	Error value [mm/s² or °/s²]	
Error type	1, Error message from NC program.		

ID 21547

Double programming in #VECTOR LIMIT-command.

Description	In the command #VECTOR LIMIT ... [...] keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove redundant keywords from the command #VECTOR ... [...].
Error type	1, Error message from NC program.		

ID 21548

M function is not freely available or requires a configured spindle.			
Description	A programmed M3, M4, M5, M19 or M40 - M45 cannot be executed because either there is no spindle in the NC channel or the the M function is not enabled for general use.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	In the channel parameters, correct the parametrisation of the use of M3, M4, M5, M19 and M40 - M45: Either at least one spindle must be completely configured in the NC channel (P-CHAN-00082 et seq.)... ...or the spindle-specific link of the M function must be disabled: - P-CHAN-00098 for M3, M4, M5, M19 - P-CHAN-00052 et seq. for M40 - M45
Parameter	%1:	Error value [-]	
		Number of spindles configured in NC channel (P-CHAN-00082)	
	%2:	Error value [-]	
		Value of release parameter (P-CHAN-00098, P-CHAN-00052)	
	%3:	Current value [-]	
		Number of M function	
Error type	1, Error message from NC program.		

ID 21549

Assignment from left to the right requires a parameter or variable.			
Description	A transferred signal parameter (P[<i>]) in the #WAIT command cannot be adopted in the programmed "destination variable".		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Here for syntactical reasons, it has to take into consideration that the assignment is executed from left to the right. The value of the transferred signal parameter *P[<>]* is therefore assigned to the "destination variable" on the right-hand side of the equals sign. This "destination variable" has to be a P parameter or a V. variable (self-defined variable, external variable, pre-defined variable with write access). Example: N10 #SIGNAL [ID1 CH1 **P[0]=5**] Nxx… **:** N90 #WAIT [ID1 CH1 **P[0]=V.L.WAIT**] -> Value 5 is stored in V.L.WAIT

Error type	1, Error message from NC program.

ID 21550

Selection of Cartesian transformation during active trafo PTP programming not allowed.			
Description	The selection of cartesian transformation with #CS ON or #ACS ON is not allowed during active PTP motion control (#PTP ON). Example: **Wrong:** N10 X50 Y50 N20 X100 … Nxx **#PTP ON** … Nxx **#CS ON** … Nxx #PTP OFF … Nxx #CS OFF Nxx M30 **Correct:** N10 X50 Y50 N20 X100 N30 **#CS ON** … Nxx **#PTP ON** … Nxx #PTP OFF … Nxx #CS OFF Nxx M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Select the cartesian transformation before selecting the PTP motion control (#PTP ON) or after deselecting (#PTP OFF).
Error type	1, Error message from NC program.		

ID 21551

Axes involved in contour rotation may not be moved independently.			
Description	In the current plane, an independent motion (independent INDP_SYN/ASYN, oscillation OSC or lifting/lowering LIFT) is programmed for an axis involved in contour rotation with #ROTATION.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. Program independent motion only for axes which are not involved in a contour rotation in plane.
Parameter	%1:	Logical axis number [-]	
		Axis number of the invalid programmed independent axis.	
Error type	1, Error message from NC program.		

ID 21552

Selection of kinematic transformation during active trafo PTP programming not allowed.

Description	The selection of kinematic transformation with #TRAFO ON is not allowed during active PTP motion control (#PTP ON). Example: **Wrong:** N10 X50 Y50 N20 X100 … Nxx **#PTP ON** … Nxx **#TRAFO ON** … Nxx #PTP OFF … Nxx #TRAFO OFF Nxx M30 **Correct:** N10 X50 Y50 N20 X100 N30 **#TRAFO ON** … Nxx **#PTP ON** … Nxx #PTP OFF … Nxx #TRAFO OFF Nxx M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Enable kinematic transformation before selecting PTP motion control (#PTP ON) or after deselecting (#PTP OFF).
Error type	1, Error message from NC program.		

ID 21553

The requested gear data are not available.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21555

Tool radius compensation selection when 1st or 2nd main axis is a modulo axis is not allowed.			
Description	The main axes of the corresponding plane (G17/G18/G19) may not be modulo axes. A modulo axis is identified in the axis mode (P-AXIS-00015). The only exception is when TRC is used with an active #CYL[…].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the axis mode. (P-AXIS-00015).
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	1, Error message from NC program.		

ID 21556

During coordinate system transition no (A)CS axis may be a modulo axis.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 21557

Maximum lift height POS_LIMIT not specified.			
Description	The maximum lift height was not specified when the lifting motion was programmed. See Programming - Cross-block lifting/lowering		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the lifting motion. Supplement POS_LIMIT.
Error type	1, Error message from NC program.		

ID 21558

LIFT command already activated. LIFT_END is expected.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21559

LIFT command already deactivated. LIFT_START is expected.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21560

#Invalid programming of #CACHE command.			
Description	The programming of the #CACHE command in the NC program is incorrect.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; correct the #CACHE command.
Error type	1, Error message from NC program.		

ID 21561

Double programming in #CACHE-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21562

Maximum number of files to be stored in local cache exceeded.			
Description	The maximum permissible number of files to be stored in the local cache was exceeded. The maximum number of files is specified by P-CHAN-00506 (alternatively P-STUP-00051).		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the assignment of P-CHAN-00506 or reduce the number of files which can be loaded in the cache.
Parameter	%1:	Limit value [-]	
		Maximum possible number of files which can be loaded in the local cache.	
Error type	5, Error message by access on files.		

ID 21563

File to be stored in local cache not found.			
Description	Individual files can be loaded into the cache using the NC command #CACHE. `#CACHE LOAD [ FILE<name>]` The specified file was not found.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the filename entered. Check the specified NC program paths (path[i].*) of each channel.
Error type	5, Error message by access on files.		

ID 21564

Indicated file exceeds the maximum permissible size of local cache.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Limit value [-]	
	%2:	Current value [-]	
Error type	5, Error message by access on files.		

ID 21565

Channel parameters: Feed of edge machining falls below minimum permissible value.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value[1µm/s or 0.001°/s]	
	%2:	Corrected value [1µm/s or 0.001°/s]	
	%3:	[-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21566

Channel parameters: Limit angle exceeds maximum permissible value.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21567

Logical axis number is out of data format.			
Description	The programmed axis number is outside the valid value range.		
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
		The incorrectly programmed axis number.	
	%2:	Lower limit value [-]	
		The smallest possible axis number.	
	%3:	Upper limit value [-]	
		The biggest possible axis number.	
Error type	1, Error message from NC program.		

ID 21568

Double programming in #-manual mode command.			
Description	In the #-manual mode command, keywords are programmed several times or in invalid combinations.		
Response	Class	2	
Solution	Class	3	

Error type	1, Error message from NC program.

ID 21569

Double programming in #SLOPE command.			
Description	In the command #SLOPE, keywords were programmed in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 21570

Double programming in #AKIMA command.			
Description	In the command #AKIMA, keywords were programmed in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 21571

Double programming of axes during manual mode selection.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21572

Position request during active OTC not allowed.	
Description	Activation of a position request during active OTC is not allowed. Example: **Wrong:** N10 X50 Y50 N20 X100 … Nxx **#OTC ON** … Nxx **#CHANNEL INIT [CMDPOS]** … Nxx #OTC OFF Nxx M30 **Correct:** N10 X50 Y50 N20 X100 … Nxx **#CHANNEL INIT [CMDPOS]** … Nxx **#OTC ON** … Nxx #OTC OFF Nxx M30

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Activate the position request before activation of OTC (#OTC ON) or after deactivation of OTC (#OTC OFF).
Error type	1, Error message from NC program.		

ID 21573

Channel parameters: Table of radius-dependent dynamic weightings contains an active line with invalid parameter value.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Error value [0.1%]	
	%3:	Error value [0.1%]	
	%4:	Error value [0.1%]	
	%5:	Corrected value [0.1%]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21574

Channel parameters: Parallel activation of multiple decoder ahead operations is not allowed.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Current value [µs]	
	%3:	Corrected value [-]	
	%4:	Corrected value [µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21575

Parallel activation of multiple decoder ahead operations is not allowed.			
Description			
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Current value [µs]
Error type	1, Error message from NC program.	

ID 21576

Double programming in #CYL-command.			
Description	In the function tube profile machining #CYL[...], key words were used several times for parameter identification.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct programming
Error type	1, Error message from NC program.		

ID 21577

Invalid edge number in profile description.			
Description	When the tube profile machining #CYL[] function is active, an invalid number of edges was programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct programming
Parameter	%1:	Current value [-]	
		Number of profile edges	
	%2:	Lower limit value	
		Minimum number of profile edges	
	%3:	Upper limit value	
		Maximum number of profile edges	
Error type	1, Error message from NC program.		

ID 21578

Double programming of H-function by tube profile parameter.			
Description	The parameter setting of active function tube profile machining #CYL[] contains double setting of M/H number. Assignment in section h_synch[<H-Number>] or m_synch[<M-Number>] and tube_profile.techno_nr_rnd_on, tube_profile.techno_nr_rnd_off. Invalid parameter setting leads to deactivation of specific M/H-Code output for #CYL[] function.		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Check and correct parameter settings of channel machine data in tube_profile.techno_nr_rnd_on, tube_profile.techno_nr_rnd_off.

Parameter	%1:	Error value [-]
		Current M/H number with double setting from h_synch[] or m_synch[]
	%2:	Corrected value [-]
		Assigned M/H value off tube_profile.techno_nr_rnd_on, tube_profile.techno_nr_rnd_off
	%3:	
		Corrected value off tube_profile.techno_nr_rnd_on, tube_profile.techno_nr_rnd_off
Error type	2, Error message by data transfer from parameter list into control device.	

ID 21579

Invalid setting of H function by tube profile parameter.			
Description	The parameter setting of the tube profile processing #CYL[] function in the machine data record contains incorrect M/H code numbers. Either the M/H code exceeds maximum permissible value or an internally assigned M code number was used (e.g. M30 -> program end code). Invalid parameter setting leads to deactivation of specific M/H code output.		
Response	Class	1	Abort NC program processing.
Solution	Class	1	Check and correct parameter settings of channel machine data tube_profile.techno_nr_rnd_on, tube_profile.techno_nr_rnd_off
Parameter	%1:	Current value [-]	
		Current invalid M/H number	
	%2:	Limit value [-]	
		Maximum permissible M/H code number	
	%3:	Corrected value [-]	
		Corrected M/H code number	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21580

Double programming in #EDM-command.			
Description	In the command #EDM, keywords are programmed several times or in invalid combinations.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21581

Feed coupling on main axes without selection not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21582

NC command G26 will be ignored.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 21585

Number of rotary axis pair is out of data format.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21586

Unknown rotational axis pair. No axes pair defined.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Error value [-]
	%2:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21587

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21588

Incorrect measurement type for independent measurement run.

Description	A measurement run in an independent axis is not possible with the set measurement type P-CHAN-00057. Valid measurement types for independent measurement runs are: Measurement type P-CHAN-00057 Meaning 1 Measurement run with at least one axis, Measurement feed programmable by F word. 2 Measurement run with exactly one axis. Measurement feed is specified in the axis data list. 7 Measurement run (G100) by moving to a fixed stop with at least one axis, Measurement feed programmable by F word. The measurement type can either be changed in the channel parameter list or in the NC program with the #MEAS or #MEAS MODE commands.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Use a different measurement type P-CHAN-00057
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Incorrect measurement type P-CHAN-00057	
Error type	1, Error message from NC program.		

ID 21589

Only axes of the active rotary axis pair may be programmed.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 21590

Invalid type setting of M/H function by tube profile parameter.			
Description			
Response	Class	1	

Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21591

This axis cannot be programmed in this path.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21592

G function is not allowed during active 2 path programming.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21593

Path specific programming of G function not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21594

NC command is not allowed in the same block with path specific programming.
Description

Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21595

# command is not allowed during active 2-path programming.			
Description	The # command used is not allowed in combination with a 2-path configuration and the associated programming.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC program and the configuration used.
Error type	1, Error message from NC program.		

ID 21596

Axis for mirroring not available.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Current value [-]
Error type	1, Error message from NC program.	

ID 21597

Distance value is out of data format.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21598

CS name too long.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Limit value [-]

Error type	1, Error message from NC program.

ID 21599

CS name is already in use.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21600

CS stack is full. CS cannot be saved.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 21601

CS name not found.			
Description	When #CS.. or #TRANSFORM .. was programmed, a coordinate system name that is unknown or not configured was used.		
Response	Class	2	Abort NC program processing. Check and modify NC program. Check CS name for spelling errors or create CS with this name.
Solution	Class	3	
Parameter	%1:	Error value	
		Name of the unknown CS	
	%2:	Upper limit value	
		Maximum number of defined CS	
Error type	1, Error message from NC program.		

ID 21602

Variable exceeds maximum permissible data size.			
Description	The size of the variable value does not fit in the memory location assigned to it.		
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
		The variable size required.	
	%2:	Upper limit value [-]	
		The maximum size of the variable available.	

Error type	1, Error message from NC program.

ID 21603

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21604

#MSG SAVE and #MSG INFO cannot be used with binary data.			
Description	SAVE and INFO tokens cannot be combined with binary data.		
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
		The incorrectly programmed token.	
Error type	1, Error message from NC program.		

ID 21605

Sending binary data only possible to receiver tool server.			
Description	The binary data receiver can only be the tool server.		
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Expected value [-]	
		Expected receiver.	
Error type	1, Error message from NC program.		

ID 21606 / 21607

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21608

Modal edge banding not active. TRIGGER has no effect.			
Description			
Response	Class	2	
Solution	Class	3	

Error type	1, Error message from NC program.

ID 21609

Intersection calculation during Cartesian forward transformation impossible.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21610

Intersection calculation during Cartesian backward transformation impossible.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21611

Invalid circular mode.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Error value [-]
Error type	1, Error message from NC program.	

ID 21612

Double programming in #SEGMENTATION command.			
Description	In the command #SEGMENTATION, keywords are programmed several times or in invalid combinations.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21614

Kinematic ID not programmed.			
Description	A programmed kinematic ID is expected.		
Response	Class	2	
Solution	Class	3	

Error type	1, Error message from NC program.

ID 21617

Can't overwrite the file.			
Description	It was not possible to overwrite the file since the user has no file overwrite rights or the file is still open by another program or process. Another possible reason is the file may have write protection		
Response	Class	-	
Solution	Class	-	Make sure that the file is not used by another process or program and that the user has the necessary rights to modify the file. As required, remove the file write protection.
Error type	-		

ID 21618

The target file already exists.			
Description	The target file already exists and the OVRMODE parameter is 0.		
Response	Class	-	
Solution	Class	-	Delete existing file or set OVRMODE to 1 to overwrite the existing file when called.
Error type	-		

ID 21620

Source file does not exist or there is no access to it.			
Description	Source file does not exist or the user has no read access to the file.		
Response	Class	-	
Solution	Class	-	Make sure that the source file is at the specified location and the current user has access rights to the file.
Error type	-		

ID 21621

Could not rename file due to write protection.			
Description	The file is write-protected so it cannot be renamed.		
Response	Class	-	
Solution	Class	-	Remove the file write protection manually.
Error type	-		

ID 21622

Directory is write-protected or does not exist.			
Description	The directory in which the file is to be written does not exist or is write-protected.		
Response	Class	-	

Solution	Class	-	Create the required directory or remove the write protection from the directory.
Error type	-		

ID 21623

Double programming in #FILE command.			
Description	A redundant parameter was programmed in the #FILE command.		
Response	Class	2	
Solution	Class	3	Remove the redundant programmed parameter from the command.
Error type	1, Error message from NC program.		

ID 21624

OVRMODE can only have values between 0 and 1.			
Description	A value outside 1 or 0 was assigned to the OVRMODE parameter		
Response	Class	2	
Solution	Class	3	Check the assigned value and make sure it is 0 or 1.
Error type	1, Error message from NC program.		

ID 21625

File does not exist.			
Description	Could not find the file specified.		
Response	Class	-	
Solution	Class	-	Check file path and make sure the file is located in the correct directory.
Error type	-		

ID 21627

File could not be accessed.			
Description	Source file does not exist or the user has no read access to the file		
Response	Class	-	
Solution	Class	-	Make sure the source file is located at the specified location.
Error type	-		

ID 21628

Unknown SLOPE type.			
Description			
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Error value[-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21629

Unknown identifier for SLOPE type.			
Description	The acceleration profile type is defined in the NC command #SLOPE. #SLOPE[TYPE=<ident>] The specified identifier does not exist.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the identifier in the NC program.
Parameter	%1:	Error value[-]	
		Programmed identifier	
Error type	1, Error message from NC program.		

ID 21630

Unknown kinematic ID. Is not defined in channel parameters.			

Description	A kinematic ID or the ID of a PCS transformation that is not available in the channel parameters was selected in the NC program.
	Example:
	The line below was programmed in the NC program.
	`N20 #KIN ID [28] (Error ID 21630)`
	When a PCS transformation is used:
	Kinematics (P-CHAN-00262) are defined in the corresponding channel as follows:
	`# Assignment in channel parameter list`
	`#==========================================`
	`trafo[0].id 9`
	`trafo[0].param[0] 100`
	`…`
	Addition to kinematic ID 28
	`trafo[1].id 28`
	`trafo[1].param[0] 456 ( HD1`
	`trafo[1].param[1] 123 ( HD2`
	`…`
	When a PCS transformation is used:
	`N20 #TRAFO PCS ID [28] (Error ID 21630)`
	Addition for PCS transformation
	`trafo_pcs[0].id 28`
	`trafo_pcs[0].type 212`
	`trafo_pcs[0].param[0] 1`
	`trafo_pcs[0].param[1] 2`		

Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify the corresponding list of channel parameters. Supplement the missing kinematic ID (P-CHAN-00262) with the parameters (P-CHAN-00263).
Parameter	%1:	Error value[-]	
		Kinematic ID programmed in the NC program.	
Error type	1, Error message from NC program.		

ID 21631

Invalid internal state in #LOCK command.			
Description	An impermissible internal state was reached.		
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value[-]	
		The incorrect value.	
Error type	1, Error message from NC program.		

ID 21633

Job manager command contains multiple open brackets.			
Description	The Job Manager command contains multiple open brackets. This is not permitted.		
Response	Class	3	Abort NC program processing.
Solution	Class		Check and correct the Job Manager command.
Error type	8, Error message from Job Manager		

ID 21634

The combination of PLC and CTM parameters is not permissible.			
Description	PLC and CTM parameters mutually exclude each other.		
Response	Class	2	
Solution	Class	0	
Error type	1, Error message from NC program.		

ID 21635

Double programming of PLC parameter.			
Description	PLC parameter was programmed several times.		
Response	Class	2	
Solution	Class	0	
Error type	1, Error message from NC program.		

ID 21636

Redundant programming of CTM parameter.

Description	CTM parameter was programmed several times.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21637

Redundant programming of CHANNEL parameter.			
Description	CHANNEL parameter was programmed several times.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21638

Invalid error class in error message detected.			
Description	The decoder error function was called with an incorrect remedy class.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21640

Job manager start-command contains multiple CMD ID Tokens.			
Description	The CMD ID token was programmed several times.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21641

Missing assignment of an external variable to a memory block.			
Description	No memory was assigned to the variable.		
Response	Class	2	
Solution	Class	7	
Error type	-		

ID 21642

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21643

The address is outside the permissible range.			
Description	The calculated address is outside the permissible address range.		
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Current value [-]	
	%2:	Error value[-]	
	%3:	Current value [-]	
	%4:	Upper limit value [-]	
	%5:	Current value [-]	
Error type	-		

ID 21644

A sub tree could not be found in the variable tree of job manager.			
Description	Could not find a sub-tree of the GCM variable.		
Response	Class	3	
Solution	Class	3	
Error type	-		

ID 21645

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21646

Effective distance is negative or zero (nil).			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [mm/s² or °/s²]	
Error type	1, Error message from NC program.		

ID 21647

Double programming in #VECTOR OFFSET command.

Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21648

Programmed negative software limit switch is out of permissible value range.			
Description	Use G98 to change negative (lower) software limit switches. A negative software limit switch can only be increased. This means a reduction in range. Use the parameter P-CHAN-00498 to influence the behaviour of programmed software limit switches.		
Response	Class	1	Output the warning and limit the negative software limit switch
Solution	Class	1	Check and modify the value for G98 in the NC program. Increase the range by using G98 to set the parameter P-CHAN-00498 to 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Error value [0.1 µm or 0.0001°]	
		Programmed value for the software limit switch	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Limiting value	
Error type	1, Error message from NC program.		

ID 21649

Negative software limit switch greater than positive software limit switch.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [0.1 µm or 0.0001°]	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 21650

Programmed positive software limit switch is out of permissible value range.

Description	Use G99 to change positive (upper) software limit switches. A positive software limit switch can only be reduced. This means a reduction in range. Use the parameter P-CHAN-00498 to influence the behaviour of programmed software limit switches		
Response	Class	1	Output the warning and limit the positive software limit switch
Solution	Class	1	Check and modify the value for G99 in the NC program. Increase the range by using G99 to set the parameter P-CHAN-00498 to 0.

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of affected axis
	%2:	Error value [0.1 µm or 0.0001°]
		Programmed value for the software limit switch
	%3:	Upper limit value [0.1 µm or 0.0001°]
		Limiting value
Error type	1, Error message from NC program.	

ID 21651

Online TRC mode out of permissible value range.			
Description			
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Current value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21655

Skip level is out of permissible value range.			
Description			
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21656

Only possible to copy several elements between array variables.			
Description	With external variables (see [EXTV]) complete field variables (array)can be assigned to each other. The condition is that a VE.Array variable of the same dimension and type is placed on the left and right sides. Example: Extract from the V.E. configuration list: var[0].name V_ARRAY_1 var[0].type UNS32 var[0].array_elements 10 var[1].name V_ARRAY_2 var[1].type UNS32 var[1].array_elements 10 var[2].name V_UNS32 var[2].type UNS32 var[2].array_elements 0 Excerpt from the NC program: V.E.V_ARRAY_1 = V.E.V_UNS32 (Error 21656, V.E.V_UNS32 is not an array variable) V.E.V_ARRAY_1 = V.E.V_ARRAY_2 -> (Correct, all 10 UNS32 elements of V_ARRAY_2 are copied)		
Response	Class	2	Error message output and NC program abort
Solution	Class	3	Correct NC program
Error type	1, Error message from NC program.		

ID 21657

Different number of array elements. Array copy not possible.

Description	With external variables (see [EXTV]) complete field variables (arrays) can be assigned to each other. This depends on whether the array variables on the left and right hand sides are of the same dimension and type. Example: Extract from the V.E. configuration list: var[0].name V_ARRAY_1 var[0].type UNS32 var[0].array_elements 10 var[1].name V_ARRAY_2 var[1].type UNS32 var[1].array_elements 10 var[2].name V_ARRAY_3 var[2].type UNS32 var[2].array_elements 20 Excerpt from the NC program: V.E.V_ARRAY_1 = V.E. V_ARRAY_3 (Error 21657, The array_elements dimension is different in V.E.V_ARRAY_2 and V.E.V_ARRAY_3) V.E.V_ARRAY_1 = V.E.V_ARRAY_2 -> (Correct, all 10 UNS32 elements of V_ARRAY_2 are copied)		
Response	Class	2	Output error message and abort NC program
Solution	Class	3	Correct the NC program
Parameter	%1:	Error value [-]	
		The array_elements dimension of the variable on the left hand side	
	%2:	Error value [-]	
		The array_elements dimension of the variable on the right hand side	
Error type	1, Error message from NC program.		

ID 21658

Copy of complete variable arrays only possible with two external variables.			
Description	Only entire variable arrays can be copied between two external variables (see [EXTV]). Example: V.E.V_ARRAY = 1234 (Error 21658, left-hand side is an array variable) V.E.V_ARRAY = V.P.ARRAY (Error 21658, right-hand side is not an external variable) V.E.V_ARRAY_1 = V.E.V_ARRAY_2 (Correct, all elements of V.E.VARRAY_2 are assigned to V.E.VARRAY_1)		
Response	Class	2	Output error message and abort NC program
Solution	Class	3	Correct the NC program
Error type	1, Error message from NC program.		

ID 21659

Copy of complete arrays or structures only possible between V.E array or structure variables.			
Description	Only entire variable arrays can be copied between two external variables (see [EXTV]). Example: V.E.V_UNS32 = V.E.V_ARRAY (Error 21659, left-hand side is not an array variable) V.P.V_ARRAY = V.E.V_ARRAY (Error 21659, left-hand side is not an external variable) V.E.V_ARRAY_1 = V.E.V_ARRAY_2 (Correct, all elements of V.E.VARRAY_2 are assigned to V.E.VARRAY_1)		
Response	Class	2	Output error message and abort NC program
Solution	Class	3	Correct the NC program
Error type	1, Error message from NC program.		

ID 21660

Maximum size of external variables in one NC block exceeded.			
Description	The specified synchronous external variables exceed the possible total size of one NC block. Possible solutions are: If several synchronous V.E. variables are programmed in this block, split it into several NC lines. If synchronous V.E structure or array variables are written in this block, split the assignment into their single elements. Convert the external variable to asynchronous and program an explicit synchronisation #FLUSH WAIT in the NC program before every read and write access.		
Response	Class	2	Output error message and abort NC program
Solution	Class	3	Correct the NC program
Parameter	%1:	Current value [-]	
		Number of synchronous V.E variables in current block	
	%2:	Error value [-]	
		Current data size	
	%3:	Upper limit value [-]	
		Maximum permissible data size per NC block	
Error type	1, Error message from NC program.		

ID 21661 / 21662

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21663

Unknown type for job manager variable list.			
Description	This type of job manager variable list is unknown.		
Response	Class	2	
Solution	Class	3	
Error type	-		

ID 21664

Double programming of START parameter.			
Description	The START parameter was programmed several times.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21665

Double programming of END parameter.			
Description	The END parameter was programmed several times.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21666

The combination of START and END parameters is not permissible.			
Description	START and END parameters must not be programmed at the same time.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21667

Double programming of ID parameter.			
Description	The ID parameter was programmed several times.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21668

The ID parameter must be programmed.			
Description	The ID parameter must be programmed in the command.		
Response	Class	2	

Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21669

Invalid combination of parameters or missing parameters at the WAIT command.			
Description	There are parameters programmed that may only be programmed exclusively or parameters are missing.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21670

Brackets are not permissible without parameters.			
Description	Brackets without a parameters are not permissible as an argument.		
Response	Class	2	
Solution	Class	0	
Error type	1, Error message from NC program.		

ID 21673

Impermissible modulo range for tube profile machining.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Error value [0.1 µm or 0.0001°]
	%2:	Error value [0.1 µm or 0.0001°]
	%3:	Error value [0.1 µm or 0.0001°]
	%4:	Expected value [0.1 µm or 0.0001°]
	%5:	Logical axis number [-]
Error type	1, Error message from NC program.	

ID 21674

Errors in calculated modulo parameters.			
Description			
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Current value [0.1 µm or 0.0001°]
	%2:	Current value [0.1 µm or 0.0001°]
	%3:	Current value [0.1 µm or 0.0001°]
Error type	1, Error message from NC program.	

ID 21675

Only CTM tokens of the string type are permissible in the startup list.			
Description			
Response	Class	2	
Solution	Class	7	
Error type	6, Error message by data transfer to control device.		

ID 21676

An unknown type of a CTM token has been found.			
Description	The CTM token type is unknown.		
Response	Class	2	
Solution	Class	7	
Error type	6, Error message by data transfer to control device.		

ID 21678

Job manager variables are not allowed on the right side of an assignment.			
Description	Job manager variables themselves cannot be placed on the right-hand side of an equation.		
Response	Class	3	
Solution	Class	3	
Error type	-		

ID 21680

Double programming in #LEAD command.			
Description	In the command #LEAD, keywords are programmed several times or in invalid combinations.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21683

The encryption password was not set.

Description	Every file extension of an encrypted NC program is assigned a specific key (password). It must be saved in the controller before program start. No matching key was found for the current encrypted NC program. (See:FCT-C12// Encryption groups and configuration)		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Reload the key belonging to the file extension using the corresponding CNC object “mc_encryption_key_*” and restart the NC program. If necessary, also check whether the file extension matches the key used.
Parameter	%1:	Current value [-]	
		Name of the encrypted NC program	
Error type	5, Error message by access on files.		

ID 21684

The maximum length of the extension for encryption files has been exceeded.			
Description	The file extension of an encrypted NC program is limited to three characters. The file extension of the current encrypted NC program has more than 3 characters.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check file extension and adapt. The file extension must match the key used. If necessary, re-encrypt the NC program with one of the permissible keys and use the corresponding file extension.
Error type	5, Error message by access on files.		

ID 21686

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21687

Factor programming not allowed during active scaling function.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21688

Axis exchange not allowed during active scaling.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21689

Scaling factor is negative or zero.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21690

Scaling for modulo or spindle axes not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21691

Update of channel parameters during active NC program not possible.			
Description	For safety reasons, it is not permissible to update channel parameters while the NC program is active. The new parameters are not effective. This check can be disabled by setting the channel parameter P-CHAN-00146 to the value 1.		
Response	Class	1	Warning: Parameter changes are discarded
Solution	Class	7	New update when CNC channel is inactive, e.g. in Selected state.
Parameter	%1:	Current value [-]	
		TRUE = program decoding enabled	
	%2:	Current value [-]	
		Current decoder status	
	%3:	Current value [-]	
		Value of parameter P-CHAN-00146	
Error type	-		

ID 21692

Redundant token bracket found in NC command.			
Description	The ‘Open bracket’ token was programmed several times in the command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check syntax. Delete redundant ‘Open bracket’ token.
Error type	1, Error message from NC program.		

ID 21693

Redundant Close bracket token found in NC command.			
Description	The ‘Close bracket’ token was programmed several times in the command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check syntax. Delete redundant ‘Close bracket’ token.
Error type	1, Error message from NC program.		

ID 21694

Redundant PATH token found in NC command.			
Description	The PATH token was programmed several times in the command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check syntax. Delete redundant PATH token.
Error type	1, Error message from NC program.		

ID 21695

Size-of operator for static variables not allowed.		
Description	The size-of operator cannot be used for static variables.	
Response	Class	2 Abort NC program processing.
Solution	Class	3 Check syntax. Adapt program run.
Parameter	%1:	Error value [-]
		The type of static variables.
Error type	1, Error message from NC program.	

ID 21696

Double programming in L SEQUENCE command.			
Description	In the command #L SEQUENCE, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax.
Error type	-		

ID 21697

Sequence end not found.			
Description	During sequence execution, the programmed end number of the sequence command was not found up to program end.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check programmed and existing end numbers and adapt if necessary.
Parameter	%1:	Error value [-]	
		Programmed end number.	
Error type	1, Error message from NC program.		

ID 21698

Sequence start not found.			
Description	The sequence was not executed because the programmed start number of the sequence command was not found up to program end.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check programmed and existing start numbers and adapt if necessary.
Parameter	%1:	Error value [-]	
		Programmed start number.	
Error type	1, Error message from NC program.		

ID 21701

Double programming in polynomial definition.			
Description	In the polynomial definition, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax.
Error type	1, Error message from NC program.		

ID 21702

Number of programmed axis specific polynomials not allowed.			
Description	Too many axis-specific polynomials were programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the number of programmed axis-specific polynomials in the NC block to match the limit value. If necessary, split the programmed polynomials into several NC blocks.
Parameter	%1:	Error value [-]	
		The number of axis-specific programmed polynomials.	
	%2:	Limit value [-]	
		The maximum number of possible axis-specific polynomials.	
Error type	1, Error message from NC program..		

ID 21703

L parameter of axis specific polynomial has to be greater than zero.			
Description	The L parameter must be greater than zero in order to calculate the polynomial.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax.
Error type	1, Error message from NC program.		

ID 21705

Polynomial parameter A0 has to be programmed in the polynomial.	
Description	A0 must be programmed in order to calculate the polynomial.

Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21706

Not allowed parameter setting for machine-type.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21709

Invalid combination of parameters or missing parameter in #MSG command.			
Description	Invalid combinations of keywords were used in the In the #MSG command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax.
Error type	1, Error message from NC program.		

ID 21710

Found double programmed token in MSG command.			
Description	The command #MSG was programmed with identical keywords several times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check syntax. Delete redundant keywords.
Error type	1, Error message from NC program.		

ID 21711 / 21712

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21713

Error while decoding a number.	
Description	An error occurred when a number was converted from ASCII to FLOAT.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the format or size of the programmed number and adapt if necessary.
Error type	1, Error message from NC program.		

ID 21714

Double programming of e-word.			
Description	More than one E word has been programmed in the same NC block. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **E100** G01 X100 **E200** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundant E word.
Error type	1, Error message from NC program.		

ID 21715 / 21716

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21717

E feed rate is out of permissible value range.			
Description	The E word is not yet programmed or the feed rate programmed with the E word is outside the permissible value range. The feed rate value is influenced by the unit setting (P-CHAN-00108) or by the definition of the default feed rate (P-CHAN-00099).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Program and check the E word or set the feed rate within the value range.
Parameter	%1:	Lower limit value [-]	
		The minimal programmable feed rate.	
	%2:	Upper limit value [-]	
		The maximal programmable feed rate.	
	%3:	Error value [-]	
		The incorrect E feed rate.	
Error type	1, Error message from NC program.		

ID 21718

Selection of chamfers/roundings not allowed, since disabled with channel parameter "disable_chamfers_roundings".			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21719

#OPTIONAL EXECUTION has to be set to OFF before program end.			
Description	The end identifier #OPTIONAL EXECUTION OFF is not set for a sequence to be masked in forward/backward motion up to the end of the main program or subroutine.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	#OPTIONAL EXECUTION must be set to OFF before the end of the main program or subroutine.
Error type	1, Error message from NC program.		

ID 21720

#OPTIONAL EXECUTION was not set to ON on this current program level.			
Description	If #OPTIONAL EXECUTION OFF is programmed, the program detects that no corresponding #OPTIONAL EXECUTION ON was set on the same program level.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the programming of #OPTIONAL EXECUTION ON/OFF. Paired selection/deselection may only be set on the same program level.
Error type	1, Error message from NC program.		

ID 21721

Double programming in #FRICTION-command.			
Description	In the command #FRICTION, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21723

Double programming in #AX LOCK-command.			
Description	In the command #AX LOCK, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21724

Double programming in #AX LOCK-command.			
Description	In the command #AX UNLOCK, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21725

#AX LOCK must be programmed after #PTP ON.			
Description	An axis lock is only permissible if #PTP ON is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. Before selecting #AX LOCK, #PTP ON must be programmed.
Error type	1, Error message from NC program.		

ID 21726

##AX UNLOCK must be programmed before #PTP OFF.			
Description	It is not possible to deselect #PTP OFF if an axis lock is active. First unlock the axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. Before deselecting #PTP OFF, #AX UNLOCK ALL must be programmed.
Error type	1, Error message from NC program.		

ID 21727

Unknown TRC MULTIPATH MODE.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21728

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	

Solution	Class	8	Requires controller restart.

ID 21729

Negative acceleration weighting programmed.			
Description	The spindle-specific acceleration weighting G130 was programmed with a negative value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Positive programming of spindle-specific acceleration weighting in the NC program.
Parameter	%1:	Error value [-]	
		Current, incorrect value of the acceleration weighting.	
Error type	1, Error message from NC program.		

ID 21730

Double programming in #PUNCH or #NIBBLE-command.			
Description	Invalid keyword combinations were used in the #PUNCH or #NIBBLE command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21731

Definition of a previous STROKE sequence has not been finished.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21732

No related definition of a STROKE sequence. Command has no effect.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 21734

Thread tapping in combination with spindle specific programming not allowed.			
Description			
Response	Class	2	
Solution	Class	3	

Error type	1, Error message from NC program.

ID 21735

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21736

Axis commands exclude each other.			
Description	A combination of mutually exclusive spindle-specific axis commands is programmed. For example, feedforward control and acceleration weighting are both programmed in one NC block: **Wrong:** N10 S[M03 S1000 **G136=80 G130=60**] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Split the programming of weightings into 2 NC blocks.
Error type	1, Error message from NC program.		

ID 21737

Double programming in acceleration weighting.			
Description	The spindle-specific acceleration weighting G130 was programmed several times for the corresponding spindles in the NC block. Example: **Wrong:** N10 S[**G130=80** M03 S1000 **G130=60**] : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21738

Double programming feedfoward weighting.			
Description	The spindle-specific feedforward control weighting G136 was programmed several times for the corresponding spindles in the NC block. Example: **Wrong:** N10 S[**G136=90** M03 S1000 **G136=0**] : N1000 M30		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21739

Mathematical function leads to an overflow/underflow operation.			
Description	The mathematical function was not executed since it would trigger an underflow/ overflow of the REAL data format.		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Check and adjust syntax. Modify the calculation used so that the real data format does not cause an overflow/underflow.
Error type	1, Error message from NC program.		

ID 21743

Double programming in #SINGLE STEP-command.			
Description	In the command #SINGLE STEP, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21744

Invalid token combination in #ADD command.			
Description	Several tokens in the #ADD command are mutually exclusive.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21745

Double programming in #DCC-command.			
Description	In the command #DCC, keywords are programmed several times or in invalid combinations.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21748

Double programming in #VOLCOMP-command.	
Description	In the command #VOLCOMP, keywords are programmed several times or in invalid combinations.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Remove redundancy.
Error type	1, Error message from NC program.		

ID 21749

VOLCOMP-ID is out of permissible value range.			
Description	The programmed ID of the Volumetric Compensation is outside the value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and adjust syntax. Program a VOLCOMP ID within the limit values.
Parameter	%1:	Limit value [-]	
		The incorrectly programmed ID.	
	%2:	Limit value [-]	
		The lowest possible ID of the volumetric compensation.	
	%3:	Error value [-]	
		The highest possible ID of volumetric compensation.	
Error type	1, Error message from NC program.		

2.3.8 ID-range 21750-21999

ID 21750

Invalid combination of keywords in #TRC command.			
Description	An invalid combination of keywords was used in the #TRC command in the NC program. For example, the combination of `#TRC[KERF_MASKING=1 REVERSE=1]` is not permissible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove one of the options in the #TRC command.
Parameter	%1:	Current value [-]	
		Option 1	
	%2:	Current value [-]	
		Option 2	
Error type	1, Error message from NC program.		

ID 21751

Invalid definition for 3D circles.			
Description	The starting, intermediate and end points are required to obtain the unambiguous definition of a 3D circle. These 3 points must meet specific conditions in order to calculate a 3D circle. For example, the points may not • be identical or • lie on a straight line		
Response	Class	1	Abort NC program processing.
Solution	Class	3	With error value 1, program the points (not identical, not colinear) so that a 3D circle can be described taking the calculation tolerance into consideration.
Parameter	%1:	Current value [-]	
		Value for the cause of error: 1: The starting, intermediate and end points are identical or lie on a straight line (colinear). An internal calculation tolerance is used to determine colinearity. Therefore the 3 points may not be anywhere approximating a straight line. 2: Internal calculation error (negative root). Please forward the complete error message to the CNC manufacturer	
Error type	1, Error message from NC program.		

ID 21752

Double programming in #EDGE MACHINING-command.			
Description			
Response	Class	2	
Solution	Class	3	

Error type	1, Error message from NC program.

ID 21753 - 21755

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21756

Cycle axis under given index in current plane not available.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Error value [-]
	%2:	Current value [-]
Error type	1, Error message from NC program.	

ID 21757

Double programming in #DEBUG-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21758

Maximum difference between radii exceeded.			
Description	When circles G02, G03 are programmed with inactive circle centre point compensation (G164), it may occur that the calculated radii of the circle starting point and the circle end point are different. The resulting radius difference between the circle start point and the circle end point should not exceed a limit value. The limit value for the radius difference can be set with the parameters P-CHAN-00171 and P-CHAN-00172.		
Response	Class	2	Program abort.
Solution	Class	3	Check and correct programmed circle end point in CNC program.

Parameter	%1:	Error value [0.1 µm or 0.0001°]
		Current radius difference between circle start point and circle end point.
	%2:	Upper limit value [0.1 µm or 0.0001°]
		Permissible radius difference
	%3:	Current value [0.1 µm or 0.0001°]
		Circle start point radius
	%4:	Current value [0.1 µm or 0.0001°]
		Circle end point radius
Error type	1, Error message from NC program.	

ID 21760 - 21764

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21766

Missing or wrong job manager token.			
Description	A job manager token is missing or a wrong one was programmed.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Modify the job manager tokens.
Parameter	%1:	Expected value [-]	
		First required value.	
	%2:	Expected value [-]	
		Second required value.	
	%3:	Expected value [-]	
		Third required value.	
Error type	1, Error message from NC program.		

ID 21767

Parameter update not allowed during active volumetric compensation.			
Description	A list update was triggered when Volumetric Compensation was active. Deactivate Volumetric Compensation before you update.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21768

Variable is no array, but index is programmed.	
Description	Variable was programmed with index, although it is not an array. Check if you programmed the right variable or define the variable as an array.

Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21769 / 21770

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21771

V.I. array can't be initialised.			
Description	A V.I. array cannot be initialised in a #VAR / #ENDVAR block. The values must be initialised individually.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21772

V.I.-Array is already defined with different dimensions.		
Description	The V.I.-Array is already defined with other dimensions.	
Response	Class	2
Solution	Class	3
Parameter	%1:	Identification number [-]
		The V.I. Arrayname.
	%2:	Expected value [-]
		The expected array index.
	%3:	Error value [-]
		The incorrect programmed value.
Error type	1, Error message from NC program..	

ID 21773

Name of V.I. array cannot be resolved.			
Description	The name of array could not be resolved.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Change the name of the array.
Error type	1, Error message from NC program.		

ID 21774

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21775

The job manager has sent an unknown response type as answer.			
Description	The job manager received a message of an unknown type.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	
Parameter	%1:	Current value [-]	
		Response type sent.	
Error type	-		

ID 21776

Filename in #START command is missing.			
Description	The #START command requires a filename.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Program a filename for the #START-command.
Error type	-		

ID 21777

Key word index is out of data format.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21778

Key word index out of permissible value range.			
Description			
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Current value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21779

Double programming in #GEO FEED ADAPT-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21780

Double programming in #DYNAMIC WEIGHT-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21781

Default values have higher priority. Programmed modulo limits are skipped.			
Description			
Response	Class	2	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 21782

V.I. array dimension is out of valid range.			
Description	The programmed dimension of the array is outside the limits.		
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Lower limit value [-]
		The minimum index of the array.
	%2:	Upper limit value [-]
		The maximum index of the array.
	%3:	Error value [-]
		The programmed value.
Error type	1, Error message from NC program.	

ID 21783

V.I.-Array Dimension does not exist.		
Description	The programmed dimension was not defined for the array.	
Response	Class	2
Solution	Class	3
Parameter	%1:	Error value [-]
		The programmed array dimension.
	1, Error message from NC program.	

ID 21784

The programmed variable may not be used for the currently active kinematic or axis coupling.			
Description	When a kinematic or axis coupling is active, the variable V.A.ACS.ABS may not be programmed for a read access.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Relocate the read access in V.A.ACS.ABS to a program section which has no active kinematic or axis coupling. The slave axes of an active coupling may not be programmed.
Parameter	%1:	Current value	
		Active kinematic ID or the logical axis number of the slave axis with an active axis coupling	
	%2:	Axis name	
		Optional, the name of the slave axis with an active slave axis	
Error type	1, Error message from NC program.		

ID 21785

Multiple programming of dwell time.		
Description	The dwell time was programmed several times. Make sure that the dwell time was programmed only once.	
Response	Class	2
Solution	Class	3
Error type	1, Error message from NC program.	

ID 21786

G04 has to be programmed with a value.			
Description	G04 was programmed without an associated value. The corresponding value must also be programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program: Program G04 with value
Error type	1, Error message from NC program.		

ID 21788

The given size of the variable with type VSTRING is too big.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Error value [-]	
	%4:	Upper limit value [-]	
Error type	-		

ID 21789

Double programming in #STOP REVERSIBLE-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21790

The given index for the compensation is out of permissible value range.			
Description	The index for the compensation was programmed outside the value range.		
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Error value [-]	
		The incorrectly programmed value.	
	%2:	Limit value [-]	
		The minimum programmable value.	
	%3:	Limit value [-]	
		The maximum programmable value.	
Error type	1, Error message from NC program.		

ID 21791

The given index of the file for the compensation is out of permissible value range.			
Description	The programmed index is not the permissible value for a compensation file.		
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Error value [-]	
		The incorrectly programmed value.	
	%2:	Limit value [-]	
		Minimum value that index can assume.	
	%3:	Limit value [-]	
		Maximum value that index can assume.	
Error type	1, Error message from NC program.		

ID 21792

Wrong edge definition for open profile.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21793

G function during control area definition not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21795

No axis of type rotator for C-axis found.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21796

No file name for V.G.VOL_COMP[i].FILE[j].			
Description	No name was defined for V.G.VOL_COMP[i].FILE[j]. Please specify a name.		
Response	Class	3	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21797

Too many axes when defining the control area. Maximum two permitted.			

Description	A maximum of two axes may be used for the definition of a workspace or protection area ([FCT-C14]).
	
	Example:
	`#CONTROL AREA START [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X0 Y0`
	`X0 Y100  Z100 (Error 21797)`
	`X100 Y100`
	`X100 Y0`
	`X0  Y0`
	`#CONTROL AREA END`
	
	Correct:
	`#CONTROL AREA START [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X0 Y0`
	`X0 Y100`
	`X100 Y100`
	`X100 Y0`
	`X0  Y0`
	`#CONTROL AREA END`		

Response	Class	2	Error message output and NC program abort
Solution	Class	3	Correct NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the excessive axis specified	
	%2:	Upper limit value [-]	
		Maximum permissible number of axis in control area definition	
Error type	1, Error message from NC program.		

ID 21798

Byte Order Mark for UTF detected, please save the file as ASCII-file.			
Description	The file is UTF encoded and can therefore not be processed. Encode the file as ASCII-Format.		
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Current value [-]	
		The found format of the Byte Order Mark.	
Error type	-		

ID 21799

The configured number of job manager nodes is less than zero.			
Description	The number of job manager nodes configured in the list is less than zero.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Increase the value of the job manager node count so that it is greater than zero.
Parameter	%1:	Error value [-]	
		Current incorrect value.	
Error type	1, Error message from NC program.		

ID 21800

Invalid variable type. Only V.E. variables are allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21801

V.E.-variable structure for surface scanning has no write access.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21802

V.E.-counter variable for surface scanning has invalid data type.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21803

V.E.-measuring point variable for surface scanning is no structure.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21804

V.E.-measuring point variable for surface scanning is no array.
Description

Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21805

An element of V.E.-measuring point variable for surface scanning has invalid data type.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21806

V.E. measuring point variable for surface scanning has too many elements.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21807

V.E.-measuring point variable for surface scanning has too less elements.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21808

No V.E.-measuring point variable for surface scanning configured.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21809

No V.E. counter variable for surface scanning configured.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 21810

Unknown axis identifier in kinematic chain of Volumetric Compensation.

Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21811

Channel number is out of data format.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21812

Channel number is out of permissible value range.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21813

Double programming in #TRACK CS-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21814

Double programming in #CHANNEL INTERFACE-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21815

Double programming in #CLAMP-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program..		

ID 21816

Maximum number of tools reached; number is set to Max Value.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Error value [-]
	%2:	Corrected value [-]
Error type	-	

ID 21818

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21819

Factor for axis resolution may not be negative.			
Description	The parameter P-CHAN-00315 "resolution_factor" represents the scaling factor between the programmed positions and the internal CNC format. The factor must be greater zero.		
Response	Class	1	Warning output and correct the parameter P-CHAN-00315 to default value
Solution	Class	1	Correct the channel parameter P-CHAN-00315

Parameter	%1:	Error value [-]
		Incorrect value for P-CHAN-00315
	%2:	Lower limit value [-]
		Smallest permissible value for P-CHAN-00315
	%3:	Corrected value [-]
		Corrected value for P-CHAN-00315
Error type	2, Error message by data transfer from parameter list into control device. .	

ID 21820

The read floating point number of the external variable is infinite.			
Description	External variables (V.E. see[EXTV]) can also be assigned values outside of the CNC, e.g. in the PLC. Therefore, when reading a variable of type REAL64, the CNC checks if the variable value is a valid number. This error message is generated when the found number is infinite.		
Response	Class	2	Error message and abortion of NC program
Solution	Class	3	Correct the value assignment of the external variable (e.g. in the PLC)
Parameter	%1:	Current value [-]	
		Name of the effected variable	
	%2:	Error value [-]	
		Incorrect variable value	
Error type	1, Error message from NC program.		

ID 21821

The read floating point value of the external variable is not a number (NaN).			
Description	External variables (V.E. see[EXTV]) can also be assigned values outside of the CNC, e.g. in the PLC. Therefore, when reading a variable of type REAL64, the CNC checks if the variable value is a valid number. This error message is generated when the found number is “not a number” (NaN).		
Response	Class	2	Error message and abortion of NC program
Solution	Class	3	Correct the value assignment of the external variable (e.g. in the PLC)
Parameter	%1:		
		Name of the effected variable	
	%2:	Error value [-]	
		Incorrect variable value	
Error type	1, Error message from NC program.		

ID 21822

Double programming in #CHANNEL SET-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program..		

ID 21823

Double programming in #FILE NAME-command.			
Description	A token at the #FILE NAME-command was programmed multiple times.		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Remove the multiple token.
Parameter	%1:	[-]	
		Token, that was programmed multiple times.	
Error type	1, Error message from NC program.		

ID 21826

Negative programmed feedrate factor not allowed.			
Description	The programmed feedrate factor is negative.		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Program the feedrate factor positive.
Parameter	%1:	Limit value [-]	
		Minimal feedrate factor.	
	%2:	Error value [-]	
		Current, incorrect value for the feedrate factor.	
Error type	1, Error message from NC program.		

ID 21827

Axis exchange while tracking of coordinate system (#TRACK CS) not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21828

Option while tracking of coordinate system (#TRACK CS) out of data format.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21829

Filter while tracking of coordinate system (#TRACK CS) out of data format.	
Description	

Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21830 / 21831

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21832

Axis factor is out of data format.			
Description	The axis factor is too big or too small.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Set the axis factor to the correct data range.
Parameter	%1:	Error value [-]	
		Current incorrect value.	
	%2:	Upper limit value [-]	
		Maximum value of the axis factor.	
Error type	1, Error message from NC program.		

ID 21833

Programmed value exceeds valid data range in #FILE NAME cmd.			
Description	The value of the programmed token in #FILE NAME is incorrect.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Set the value to a value within the correct limits.
Parameter	%1:	[-]	
		Identifier of the incorrect token.	
	%2:	Error value [-]	
		Current incorrect value of the token.	
	%3:	Lower limit value [-]	
		Minimum value of the token.	
	%4:	Upper limit value [-]	
		Maximum value of the token.	
Error type	1, Error message from NC program.		

ID 21834

The programmed BACKUP_COUNT_MAX is too big.			
Description	The programmed value for BACKUP_COUNT_MAX is too large. The value is corrected to the largest possible value.		
Response	Class	2	Reset the NC channel.
Solution	Class	1	Set BACKUP_COUNT_MAX to a value within the limits.
Parameter	%1:	[-]	
		The identifier.	
	%2:	Error value [-]	
		Current incorrect value.	
	%3:	Corrected value [-]	
		New corrected value.	
Error type	1, Error message from NC program.		

ID 21836

Error while creating the backup.			
Description	An error occurred while writing the backup file. This can happen because of missing access rights or an externally deleted file.		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Make sure you have correct access rights and the file path is correct.
Parameter	%1:	Status [-]	
		Error status trying to write the file.	
Error type	5, Error message by access on files.		

ID 21837

No licence available for option CuttingPlus.			
Description	The NC command used or the synchronisation type of the M or H function is part of the CuttingPlus license packet.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Contact your controller supplies for a “CuttingPlus” license.
Error type	1, Error message from NC program.		

ID 21838 / 21839

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21840

External variable could not be initialised.

Description	An error occurred during the initialisation of the external variable, as the evaluation of the initialisation string after the init token is not possible (see p-extv-00013). The cause of the error can be found in the previous error message. The variable is set to zero.		
Response	Class	2	The variable is initialized with zero
Solution	Class	3	Correct the initialisation string in the external variable list
Parameter	%1:	Error value [-]	
		Variable which could not be initialized	
	%2:	Current value [-]	
		Line number in the V.E. configuration file where the error occurred.	
	%3:	Current value [-]	
		File access	
Error type	6, Error message by data transfer to control device.		

ID 21841

Variable for initialisation of external variables not permitted.			
Description	Only other external variables (V.E.) may be used in the initialisation string init of the external variables (see P-EXTV-00013). All other variables like V.A or V.G are not allowed.		
Response	Class	2	The external variable is initialised with 0
Solution	Class	3	Correct the initialisation string in the external variable list
Parameter	%1:	Error value [-]	
		Invalid variable used	
Error type	1, Error message from NC program.		

ID 21842

Configuration list of external variables could not be opened. Initialisation of the variables not possible.			
Description	The initialisation of the external variables (see P-EXTV-00013) failed because the configuration list of external variables could not be opened.		
Response	Class	-	The external variables are initialised with 0
Solution	Class	-	Check the configuration file of the external variables
Parameter	%1:		
		Name of the external variable configuration file	
Error type	-		

ID 21843

Could not read from the configuration file of the external variables. Initialisation of the variables not possible.			
Description	An error occurred during the initialisation of the external variables (see P-EXTV-00013). It was not possible to read from the configuration list of the external variables. The following variables are set to 0.		
Response	Class	-	The following external variables are initialised with 0
Solution	Class	-	Check the configuration file of the external variables

Parameter	%1:	
		Name of the external variable configuration file
Error type	-	

ID 21844

Start address and byte offset/index cannot be specified at the same time.			
Description	When the external variable is created, the CNC normally assigns the start addresses automatically. However, the start address can also be specified explicitly by specifying a byte offset (see P-EXTV-00002) or the 24-byte index or by the variable name of a preceding variable (see P-EXTV-00048). However, only one of these two address methods may be used at the same time.		
Response	Class	2	Current start address default in P-EXTV-00048 is deleted
Solution	Class	3	Check the start address specified for the external variable, see P-EXTV-00002, P-EXTV-00048
Parameter	%1:	Current value [-]	
		Name of the variable affected P-EXTV-00001	
	%2:	Current value [-]	
		Index var[i] of the variable affected in the configuration file	
	%3:	Current value [-]	
		Byte offset P-EXTV-00002 or 24-byte index of the variable	
	%4:	Error value [-]	
		Start address of the variable via variable name P-EXTV-00048	
	%5:	Corrected value [-]	
		Corrected value for the start address P-EXTV-00048	
Error type	-		

ID 21845

The definition of the start address requires a V.E. variable.			
Description	The start address of external variables can be specified by the name of a preceding external variable (see P-EXTV-00048), e.g. to define variables with overlapping memory areas. However, the value found is not an external variable.		
Response	Class	2	External variable is deleted.
Solution	Class	7	Check start address specified for external variable P-EXTV-00048
Parameter	%1:	Current value [-]	
		Name of the variable affected P-EXTV-00001	
	%2:	Current value [-]	
		Index var[i] of the variable affected in the configuration file	
	%3:	Error value [-]	
		Incorrect start address specified P-EXTV-00048	
Error type	-		

ID 21846

The given V.E. variable as start address could not be found.

Description	The start address of external variables can be specified by the name of a preceding external variable (see P-EXTV-00048), e.g. to define variables with overlapping memory areas. However, the specified variable could not be found.		
Response	Class	2	External variable is deleted.
Solution	Class	7	Check start address specified for external variable P-EXTV-00048
Parameter	%1:	Current value [-]	
		Name of the variable affected P-EXTV-00001	
	%2:	Current value [-]	
		Index var[i] of the variable affected in the configuration file	
	%3:	Error value [-]	
		Incorrect start address specified P-EXTV-00048	
Error type	-		

ID 21847

The start address is not within the given variable scope (GLOBAL/CHANNEL).			
Description	The start address of external variables can be specified by the name of a preceding external variable (see P-EXTV-00048), e.g. to define variables with overlapping memory areas. However, the specified variable is not in the same memory area than the current variable (see P-EXTV-00004).		
Response	Class	2	The validity of the variable P-EXTV-00004 is corrected.
Solution	Class	7	Check start address specified P-EXTV-00048 or validity range P-EXTV-00004 of the external variables.
Parameter	%1:	Current value [-]	
		Name of the variable affected P-EXTV-00001	
	%2:	Current value [-]	
		Index var[i] of the variable affected in the configuration file	
	%3:	Error value [-]	
		Validity range P-EXTV-00004 of the current variable	
	%4:	Expected value [-]	
		Validity range P-EXTV-00004 of the start address variable	
	%5:	Corrected value [-]	
		Corrected validity range P-EXTV-00004 for the current variable	
Error type	-		

ID 21848

Overlapping memory area of external variable. Variable will be deleted!			
Description	You can use the parameter check_overlapping_variables P-EXTV-00011 to check the memory layout when creating the external variables. If the variable currently to be created overlaps the memory area of another variable, this error message is output and the variable is deleted. Variables that were created explicitly by specifying a start address using another variable (see P-EXTV-00048) are excluded from this check.		
Response	Class	2	Current variable is deleted.
Solution	Class	7	Check Byte Offset (see P-EXTV-00002) or 24-byte indices of the variables and sizes P-EXTV-00008, P-EXTV-00007

Parameter	%1:	Error value [-]
		Name of the variable affected P-EXTV-00001
	%2:	Current value [-]
		Name of the variables P-EXTV-00001 whose memory area is violated.
	%3:	Current value [-]
		Index var[i] of the variables affected in the configuration file
	%4:	Error value [Byte]
		Memory offset of the variables affected
	%5:	Error value [Byte]
		Size of the variables affected
Error type	-	

ID 21849

Deselection of tracking axis while active edge cutting correction not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program..		

ID 21850

Missing axis may not be part of rotation or translation.			
Description	The channel parameter P-CHAN-00213 can also be used for Cartesian transformations #CS (see [PROG//Definition of a machining coordinate system]) if not all main axes are present (i.e. the first three axes of an NC channel). However, the following limitations exist: Programming a translatory shift for a missing axis is not allowed The rotation must be in the plane of the two available axes meaning only a rotation around the missing axis is permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program or exchange the main axes in the CNC channel (see [PROG//Requesting of axes])
Parameter	%1:	Error value [-]	
		Index of the missing axis (in the active working plane)	
Error type	1, Error message from NC program.		

ID 21851

Programming of closing bracket expected in #CS command.			
Description	After programming a #CS command, terminate the definition with a closing bracket.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Add a closing bracket to the #CS command.
Error type	1, Error message from NC program..		

ID 21852

Programming of opening bracket expected in #CS command.			
Description	An opening bracket is expected at the start of a #CS command.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Insert an opening bracket at the start of the #CS command.
Error type	1, Error message from NC program..		

ID 21853

Too many positions programmed in #CS TRACK command.			
Description	Too many values were programmed in the #CS TRACK command.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Adapt the number of values in the #CS command.
Parameter	%1:	Upper limit value [-]	
		Maximum index of a value.	
	%2:	Error value [-]	
		Current incorrect index of the value.	
Error type	1, Error message from NC program.		

ID 21854

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21855

Unexpected token in #CS TRACK-command.			
Description	An unexpected token was found at the #CS TRACK-command.		
Response	Class	3	Reset of the NC channel.
Solution	Class	3	Correct the #CS TRACK-command to the correct syntax.
Parameter	%1:	Error value [-]	
		Current, incorrect token.	
Error type	-		

ID 21856

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21857

Unexpected token in #CS SHIFT-command.			
Description	An unexpected token was found at the #CS TRACK-command.		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Correct the #CS SHIFT-command to the correct syntax.
Parameter	%1:	Error value [-]	
		Current, incorrect index of the value.	
	%2:	Expected value [-]	
		The expected value.	
Error type	1, Error message from NC program.		

ID 21858

Distance between MCS reference and secondary plane is zero (nil).			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 21859

Implicit Kinematic-ID cannot be selected directly.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21860

Double programming in #DIST2GO-command.			
Description	A token at the #DIST2GO-command was programmed multiple times.		
Response	Class	3	Reset of the NC channel.
Solution	Class	3	Remove the multiple token.
Parameter	%1:	Error value [-]	
		The token, that was programmed multiple times.	
Error type	-		

ID 21861

Missing parameter in #DIST2GO-command.	
Description	The #DIST2GO needs additional parameters.

Response	Class	2	Reset of the NC channel.
Solution	Class	3	Add the missing parameters.
Parameter	%1:	Expected value [-]	
		The missing parameter.	
Error type	1, Error message from NC program.		

ID 21862

Unexpected token in #DIST2GO-command.			
Description	An unexpected token was found at the #DIST2GO-command		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Correct the #DIST2GO-command to the correct syntax.
Error type	1, Error message from NC program.		

ID 21863

The program was externally changed during the runtime.			
Description	The program was changed while it was being processed.		
Response	Class	3	Reset the NC channel.
Solution	Class	3	Make sure the program is not changed while the NC is processing it.
Parameter	%1:	File name [-]	
		Filename of the current file.	
	%2:	Error value [-]	
		Current incorrect file position.	
	%3:	Expected value [-]	
		Expected file position.	
Error type	-		

ID 21864

HSC Slope not allowed with G95/G96.			
Description	The combination of HSC Slope and G95/G96.		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Remove the HSC Slope or the G95/G96.
Parameter	%1:	[-]	
		The identifier of the Slope-Type.	
	%2:		
		The used Slope-Type.	
Error type	1, Error message from NC program.		

ID 21865

Double programming in #GET MOVING FRAME-command.			
Description			
Response	Class	2	

Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21866

Double programming of cycle synchronisation on block end.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21867

A unknown was found during programming spindle axis.			
Description	At programming of spindle axis an unknown token was programmed.		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Remove the incorrect token.
Parameter	%1:	[-]	
		Identifier of the incorrect token.	
	%2:	Error value [-]	
		Current, incorrect token.	
Error type	-		

ID 21868

Variable could not be created, there is not enough memory.			
Description	The local variable cannot be created because too little memory space was defined. Memory space is allocated by P-CHAN-00418 for V.CYC. variables or by P-CHAN-00481 for @P variables. The memory space cannot be increased for V.L. variables.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the NC program. or channel parameters • Correct P-CHAN-00418 for V.CYC. variables. • P-CHAN-00481 for @P variables • Modify the programming accordingly for all other variables.
Parameter	%1:	Identifier [-]	
		Variable type (V.L, V.CYC., @P,…)	
	%2:	Limit value [-]	
		Free memory in bytes	
	%3:	Error value [-]	
		Required memory in bytes	
Error type	1, Error message from NC program.		

ID 21869

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21870

String exceeds the maximum length.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Upper limit value [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21871

For #CYL 2ROT the second rotary axis is missing.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 21872

Double programming in #FACE-command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21873

For #FACE 2ROT the second rotary axis is missing.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	

Error type	1, Error message from NC program.

ID 21874

Machine coordinate system is already active. Re-selection because of local offsets not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21875

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21876

The axis does not exist in the NC channel.			
Description	The requested axis was not found in this channel.		
Response	Class	3	Reset the NC channel.
Solution	Class	3	Check whether if the axis is properly configured in this channel.
Error type	-		

ID 21877

Double programming in #TIME TO DISTANCE command.			
Description	A token was programmed multiple times at the #TIME TO DISTANCE ON-command.		
Response	Class	3	Reset of the NC channel.
Solution	Class	3	Remove the multiple tokens.
Error type	-		

ID 21878

Positive software limit switch smaller than negative software limit switch.	
Description	The positive software limit switch was modified with G99. The new value of the positive software limit switch is smaller than the negative software limit switch. This is not permitted.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the positive software limit switch.
Parameter	%1:	Logical axis number 1, Error message from NC program.	
	%2:	[-][0.1 µm or 0.0001°]	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 21879

STROKE-sequence definition must not contain asynchronous SIGNAL/WAIT commands.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21880

The result of the addition is too large to calculate.			
Description	The addition of the two values will exceed the numerical range of the calculation.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Make sure the result of your calculation is small enough for the permitted numerical range.
Parameter	%1:	Current value 1, Error message from NC program.	
		The first operand.	
	%2:	Current value 1, Error message from NC program.	
		The second operand.	
Error type	1, Error message from NC program.		

ID 21881

The result of the subtraction is too large to calculate.			
Description	The subtraction of the two values will exceed the numerical range of the calculation.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Make sure the result of your calculation is small enough for the permitted numerical range.
Parameter	%1:	Current value 1, Error message from NC program.	
		The first operand.	
	%2:	Current value 1, Error message from NC program.	
		The second operand.	
Error type	1, Error message from NC program.		

ID 21882

The result of the multiplication is too large to calculate.			
Description	The multiplication of the two values will exceed the numerical range of the calculation.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Make sure the result of your calculation is small enough for the permitted numerical range.
Parameter	%1:	Current value 1, Error message from NC program.	
		The first operand.	
	%2:	Current value 1, Error message from NC program.	
		The second operand.	
Error type	1, Error message from NC program.		

ID 21883

The result of the division is too large to calculate.			
Description	The division of the two values will exceed the numerical range of the calculation.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Make sure the result of your calculation is small enough for the permitted numerical range.
Parameter	%1:	Current value 1, Error message from NC program.	
		The first operand.	
	%2:	Current value 1, Error message from NC program.	
		The second operand.	
Error type	1, Error message from NC program.		

ID 21884

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21885

The channel id defined at SCALE is unknown.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21886

The channel id defined at ESCAPE is unknown.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21887

Double programming in #-Command.			
Description	A token was programmed multiple times in the #-Command.		
Response	Class	2	Reset of the NC channel.
Solution	Class	3	Remove the token that are programmed multiple times.
Error type	1, Error message from NC program.		

ID 21889

Feed rate is out of permissible value range.			
Description	The programmed feed rate is outside the permitted value range.		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Correct the feed rate to a valid value.
Parameter	%1:	Lower limit value [-]	
		Minimum value that the feed can assume.	
	%2:	Upper limit value [-]	
		Maximum value that the feed can assume.	
Error type	1, Error message from NC program.		

ID 21891

Double programmed keywords in the same NC Block.			
Description	A keyword was programmed multiple times in the same NC block		
Response	Class	2	Reset the NC channel.
Solution	Class	3	Remove the keywords that were programmed multiple times.
Parameter	%1:	[-]	
		Keyword that was programmed incorrectly/multiple times.	
	%2:		
		Keyword that was programmed incorrectly/multiple times.	
Error type	1, Error message from NC program.		

ID 21892

Identical angles in contour line are not allowed.

Description	A contour line consisting of 2 blocks was programmed. The angles are identical or opposed by 180 degrees. Incorrect example with identical angles: N240 G18 G1 Z15 X10 N250 **#ANG** 60 N260 **#ANG** 60 Z40 X30 (identical angles) Incorrect example with opposed angles: N240 G18 G1 Z15 X10 N250 **#ANG** 60 N260 **#ANG** 240 Z-40 X-30 (opposed angles) This is not permitted. The straight lines are parallel to the specified target point.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour line in NC program.
Error type	1, Error message from NC program.		

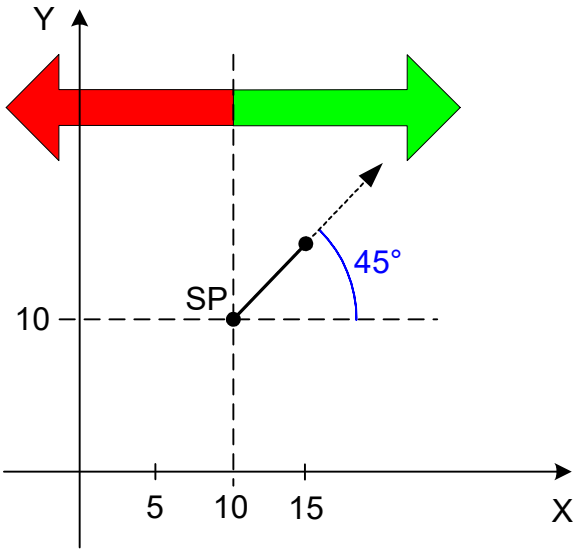
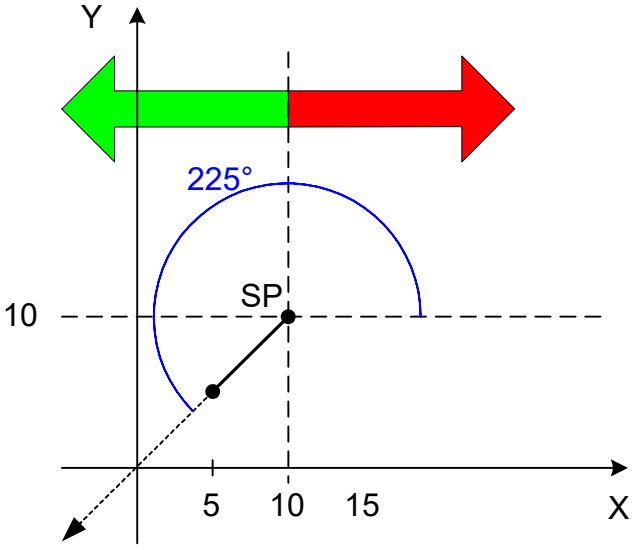
ID 21893

Plane change not allowed contour path programming.			
Description	A contour line consisting of 2 blocks was programmed. The motion blocks are programmed in different planes. This is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour line in NC program. Program the contour path in the same plane.
Error type	1, Error message from NC program.		

ID 21894

Programmed angle with given coordinate not possible.

<table>
<tr><td>Description</td><td>The programmed end position cannot be achieved with the given angle and coordinate.
Example with error:
N100 G17 X10 Y10 (SP - start position)
N110 #ANG=45 X5

Corrected example while maintaining the specified angle:
N100 G17 X10 Y10
N110 #ANG=45 X15 (position corrected)
 
</td></tr>
<tr><td></td><td>Alternatively, the programmed angle can be changed to reach the position:
N100 G17 X10 Y10
N110 Programmed angle with given coordinate not possible.=225 X5 X5 (angle corrected)
 
</td></tr>
<tr><td>Response</td><td>Class 2 Abort NC program processing.</td></tr>
</table>

Solution	Class	3	Check and modify the contour line in NC program. • Correct • Angle Angle
Parameter	%1:	Initial value [0.1 µm or 0.0001°]	
		Start position of movement	
	%2:	Current value [0.1 µm or 0.0001°]	
		Programmed angle	
	%3:	Current value [0.1 µm or 0.0001°]	
		Programmed axis coordinate	
Error type	1, Error message from NC program.		

ID 21895

Programming angle with two coordinates not allowed.			
Description	The two main axes of the active level were programmed during angle programming. This is not permitted. Remove one of the programmed axis positions. Example with error: N100 G17 X10 Y10 (start position) N110 **#ANG**=45 **X20 Y25** Corrected example: N100 G17 X10 Y10 (start position) N110 **#ANG**=45 **X20**		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour line in NC program. Remove one of the programmed main axis positions
Error type	1, Error message from NC program.		

ID 21896

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21897

Axis position is out of permissible value range.			
Description	The position is outside the motion range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour line in NC program.

Parameter	%1:	Current value [-]
		Axis index
	%2:	Error value [0.1 µm or 0.0001°]
	%3:	Lower limit value [0.1 µm or 0.0001°]
		Minimum permissible value
	%4:	Upper limit value [0.1 µm or 0.0001°]
		Maximum permissible value
Error type	1, Error message from NC program.	

ID 21898

Movement block without angle programmed.			
Description	A contour path consisting of 2 blocks is to be programmed. The second motion block was programmed without an angle. Programming this angle is necessary. Example with error: N240 G17 G01 X10 Y5 N250 **#RND**=3 N260 **#ANG**=30 **#RND**=4 N270 G01 X30 Y25 Corrected example: N240 G17 G01 X10 Y5 N250 **#RND**=3 N260 **#ANG**=30 **#RND**=4 N270 **#ANG**=80 G01 X30 Y25		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour path in NC program. Add the necessary angle.
Error type	1, Error message from NC program.		

ID 21899 - 21902

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21903

Circular block in contour path not allowed.			
Description	Circular blocks are not permitted while programming contour paths.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour path in NC program.

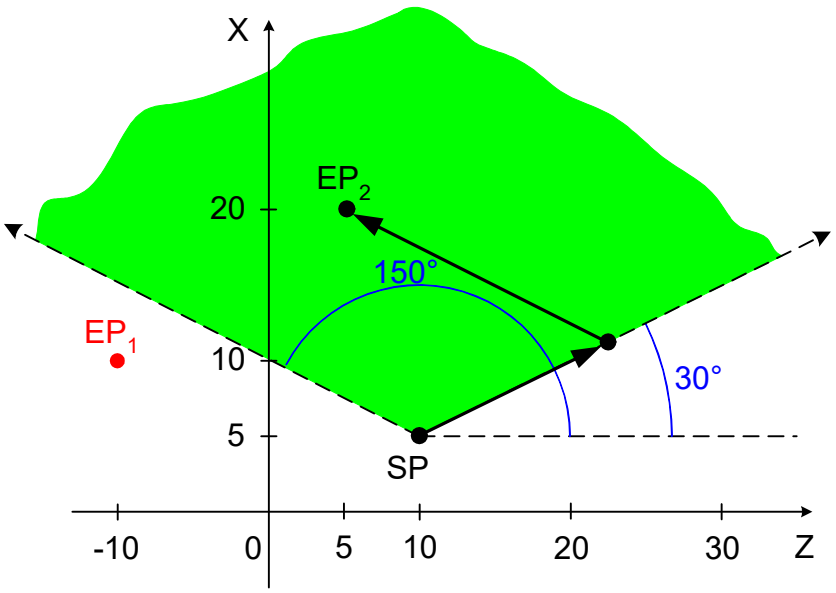
Parameter	%1:	Block number [-]
Error type	1, Error message from NC program.	

ID 21904

Contour line is not completely defined.			
Description	The contour line is not completely defined. Possible causes: an axis position is missing for a contour line consisting of one block in a contour line consisting of 2 blocks, the angle specification of the 2nd block is missing Example with error: N250 G18 G01 Z15 X2 N260 **#ANG**=30 **#RND**=3 N270 G01 Z40 X30 Corrected example for contour line of one block: N250 G18 G01 Z15 X2 N260 **#ANG**=30 **#RND**=3 **Z25** (added position) N270 G01 Z40 X30 Corrected example for contour line consisting of 2 blocks: N250 G18 G01 Z15 X2 N260 **#ANG**=30 **#RND**=3 N270 **#ANG**=80 G01 Z40 X30 (added angle specified)		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour line in NC program.
Error type	1, Error message from NC program.		

ID 21905

Contour path with given angles and end position not possible.

Description	Contour path of 2 blocks is programmed. The programmed end position of contour path is unfit with the given angles. Example with error: N250 G18 G01 Z10 X5 (SP - Starting point) N260 **#ANG**=30 N270 **#ANG**=120 G01 Z-10 X10 (EP_1 - End position) Corrected example: N250 G18 G01 Z10 X5 (SP - Starting point) N260 **#ANG**=30 N270 **#ANG**=120 G01 Z5 X20 (EP_2) In the following figure the permitted area of the end position with the given angles is shown: 

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour path in NC program. Correct the end position of contour path or Correct the angle of contour path

Parameter	%1:	Initial value [0.1 µm or 0,0001°]
		Startpoint of contour path.
	%2:	Current value [0.1 µm or 0,0001°]
		Programmed angle of first block
	%3:	Current value [0.1 µm or 0,0001°]
		Programmed angle of second block.
	%4:	End value [0.1 µm or 0,0001°]
		Incorrect end position of contour path
Error type	1, Error message from NC program.	

ID 21906

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21907

At least one main axis is missing for the function.			
Description	At least one main axis is missing in order to use the function. Example with error: N240 G18 G1 Z15 X2 N250 **#PUT AX** [Z] N260 **#ANG**=30 X25 To use the #ANG command, both main axes of the active plane are required. In the given example, the Z axis is missing.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 21908

Incomplete contour line cannot be given out.			
Description	A contour line consisting of 2 blocks was programmed. Too many control and/or technology blocks were programmed between the two blocks of the contour line so that the controller is forced to output blocks. Since the contour line is only complete when the second block is presented, it can neither be determined nor output. Solution proposed: Reduce the number of blocks between the relevant blocks of contour line.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 21909

Flush channel with incomplete contour path not possible.

Description	Contour path of 2 blocks is programmed. The NC channel requires flushing between the two blocks of the contour path, or an NC command is programmed that requires flushing the NC channel, e.g. #CHANNEL INIT[CMDPOS] or reading a synchronous V.E. variable. This is not permitted. Example with error: N250 G18 G1 Z10 X5 N260 **#ANG**=30 N270 **#FLUSH CONTINUE** N280 **#ANG**=120 X25 Corrected example: N250 G18 G1 Z10 X5 N260 **#FLUSH CONTINUE** N270 **#ANG**=30 N280 **#ANG**=120 X25		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Move the channel flush.
Error type	1, Error message from NC program.		

ID 21910

The second block of contour line with 2 blocks may not be relative.

Description	A contour line consisting of 2 blocks was programmed. The second block of the contour line is programmed as relative. This is not permitted. **Example with error:** %Test.nc N210 G18 G90 G0 X0 Y0 Z0 N220 F1000 N250 G91 G18 G1 Z10 X5 (relative programming) N260 **#ANG** 30 N270 **#ANG** 80 Z70 X95 (relative programming - active) N310 G90 G0 X130 N320 G0 Z0 M30 **Corrected example:** %Test.nc N210 G18 G90 G0 X0 Y0 Z0 N220 F1000 N250 G91 G18 G1 Z10 X5 (relative programming) N260 **#ANG** 30 N270 **G90 #ANG** 80 Z80 X100 (absolute programming) N310 G0 X130 N320 G0 Z0 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the contour line in NC program. Program the second block of the contour line as absolute.
Error type	1, Error message from NC program.		

ID 21912

Invalid combination of keywords in L SEQUENCE command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21913

Invalid label identifier in L SEQUENCE command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21914

Identical tool number specified multiple times, tool will be deleted.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
	%3:	Corrected value [-]	
	%4:	Current value [-]	
	%5:	Current value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21915

Unknown tool id.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21916

Channel parameters: Parameter can't be changed, value is kept.			
Description	The channel parameter P-CHAN-00418 was assigned a new value. It is not possible to change the parameter while the controller is running. A change in parameter requires a restart with the new parameter value.		
Response	Class	1	NC program processing is continued
Solution	Class	1	A parameter change requires a controller restart. This must be executed if a parameter change is required.
Parameter	%1:	Identifier [-]	
		Name of parameter	
	%2:	Error value [-]	
		Changed new parameter value	
	%3:	Corrected value [-]	
		Previous parameter value	
Error type	1, Error message from NC program.		

ID 21917

Unknown ESCAPE geometry.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21918

The angle of ESCAPE is out of permissible value range.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 21919

Inserting the programmed chamfer not possible.			
Description	The programmed chamfer cannot be inserted. Causes: • the programmed chamfer is too long or • At least one of the adjacent motion blocks between which the chamfer is programmed is too short.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify the NC program: • Reduce length of chamfer • Extend the motion blocks adjacent to the chamfer
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
		Programmed length of chamfer	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
		Maximum possible chamfer length	
Error type	1, Error message from NC program.		

ID 21920

Array initialisation has to begin with '[' .

Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21921

Orientation programming with different dimensions (G90/G91) not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21922 - 21924

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21925

Keywords are incompatible with the measurement type used.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21927

At the #SHIFT ON-command a modus has to be programmed.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21928

Array index is out of data range.

Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Lower limit value [-]	
	%2:	Upper limit value [-]	
	%3:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21929

An unknown Shift-Mode was programmed.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 21931

Kinematics ID was configured multiple times.			
Description	A kinematic ID was configured more than once in the channel parameter list. This is not permitted.		
Response	Class	1	Output of warning
Solution	Class	1	Check and modify the parameters of the kinematics in the channel parameter list.
Parameter	%1:	Current value [-]	
		Kinematic ID specified more than once	
	%2:	Current value [-]	
		Index of the kinematic step	
	%3:	Current value [-]	
		Index of the kinematic ID entered	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 21932

Angle programming, selection of chamfers/roundings not allowed, since disabled with channel parameter "disable_chamfers_roundings".			
Description	Programming the commands #ANG, #CHR, #CHF and #RND (Contour path programming) is not possible if the channel parameter disable_chamfers_roundings (P-CHAN-00273) is set. It is also not possible to use the commands G301 and G302.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Modification of NC program or disable the channel parameter P-CHAN-00273.
Error type	1, Error message from NC program.		

ID 21935

Missing identifier in additional axis command.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21936

Axis has no SAI characteristic.			
Description	In order to use PLCopen programming in an axis, the parameter P-AXIS-00457 must be set for the axis in question.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct P-AXIS-00457
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	[-]	
		Parameter	
Error type	1, Error message from NC program.		

ID 21939

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21941

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 21944

Function not available -> configure in start-up list.	
Description	

Response	Class	2	
Solution	Class	3	
Parameter	%1:	[-]	
Error type	1, Error message from NC program.		

ID 21945

Radial acceleration limit on the path is negative or zero (nil).			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [mm/s² or °/s²]	
Error type	1, Error message from NC program.		

ID 21946

Expected one of the first two main axes.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Axis name [-]	
	%3:	Axis name [-]	
Error type	1, Error message from NC program.		

ID 21947

Unknown axis identifier found.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21949

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		

Solution	Class	8	Requires controller restart.

ID 21950

It is not possible to combine decoder and interpolator context.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21952

P parameter are not allowed to be assigned while in real time context.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21955

Loading of NC program data not possible any more. File size exceeds limit.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	5, Error message by access on files.		

ID 21956

Names of orientation axes must start with A, B or C.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Axis name [-]	
	%2:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 21957 - 21958

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 21964

Actual position request for axis moving to fixed stop not possible.			

Description	While an axis is standing at the fixed stop (see [FCT-M8]), no actual position request may be made for this axis.
	
	Example with error:
	`G01 X100 F1000 X[FIXED_STOP ON]`
	`#CHANNEL INIT [ACTPOS] (* Error message 21964 *)`
	`G01 X0 X[FIXED_STOP OFF]`
	
	Possible solution: no position request for the axis in question.
	`G01 X100 F1000 X[FIXED_STOP ON]`
	`#CHANNEL INIT [ACTPOS AX=Y AX=Z]`
	`G01 X0 X[FIXED_STOP OFF]`		

Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	3	Check and modify the NC program. Ignore the axis in question in case of an actual position request.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	1, Error message from NC program.		

ID 21965

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 21966

Activation of movement towards fixed stop requires movement information.			

Description	When the Move to fixed stop function is activated (see [FCT-M8]), the axis must be moved towards the fixed stop.
	Example with error:
	`N10 X[FIXED_STOP ON] (* Error 21966 *)`
	
	Possible solution:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`		

Response	Class	2	Abort NC program processing. Transition to error state.

Solution	Class	3	Check and modify the NC program. Program an axis movement for the affected axis when the Move to fixed stop function is activated.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2	Current value [-]	
		Bitmask of programmed axes	
Error type	1, Error message from NC program.		

ID 21967

Axis moving to fixed stop can not be programmed again.			

Description	An axis moving to fixed stop (see [FCT-M8]) may not be programmed again.
	
	Example with error:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`N20 G01 X200 (* Error 21967 *)`
	`N30 G01 X0 X[FIXED_STOP OFF]`
	
	Possible solution:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`N30 G01 X0 X[FIXED_STOP OFF]`		

Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	3	Check and modify NC program.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
Error type	1, Error message from NC program.		

ID 21968

Start-/end value of torque limit monitoring for movement to fixed stop is invalid.			

Description	During the Move to fixed stop function (see [FCT-M8]), the monitoring of the fixed stop can be started with a delay in the approach movement or ended earlier at the end in order to avoid false detection when the axis starts moving or decelerates. This is specified as a percentage in the START and END tokens referred to the distance travelled of the approach block (see [FCT-M8//Programming]).
	
	Example with error:
	`N10 G01 X100 F1000 X[FIXED_STOP ON START=10 END=200] (Error 21968)`
	
	Possible solution:
	`N10 G01 X100 F1000 X[FIXED_STOP ON START=10 END=90]`		

Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	3	Check and modify the NC program. Correct the NC program for START and END.

Parameter	%1:	Error value [0.1%]
		Incorrect value
	%2:	Lower limit value [0.1%]
		Lower limit value
	%3:	Upper limit value [0.1%]
		Upper limit value
Error type	1, Error message from NC program.	

ID 21969

Moving to fixed stop already active.			

Description	When the Move to fixed stop function is activated (see [FCT-M8]), the function is already active on the axis.
	
	Example with error:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`…`
	`N50 G01 X200 F1000 X[FIXED_STOP ON]`
	`(Error 21969)`
	
	Possible solution:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`…`
	`N40 G01 X0 X[FIXED_STOP OFF]`
	`N50 G01 X200 F1000 X[FIXED_STOP ON]`		

Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	3	Check and modify the NC program. Deselect the Move to fixed stop function to prevent reactivation.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
Error type	1, Error message from NC program.		

ID 21970

An axis moving to fixed stop may not be released.			

Description	An axis that is active for the Move to fixed stop function (see [FCT-M8]) cannot be released. The function must be deactivated before release.
	
	Example with error:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`N30 #PUT AX [X] (Error 21970)`
	
	Possible solution:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`N20 G01 X0 X[FIXED_STOP OFF]`
	`N30 #PUT AX [X]`		

Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	3	Check and modify NC program. Deselect the Move to fixed stop function before releasing the axis.

Parameter	%1	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2	Error value [-]
		State of the Move to fixed stop function
Error type	1, Error message from NC program.	

ID 21972

Double programming of an NC command.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Identifier	
	%2:		
	%3		
	%4		
	%5		
Error type	1, Error message from NC program.		

ID 21974

Programming of #-cmd at this position of the NC-program is not allowed.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21977 - 21978

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 21978

Address type for variable not supported.			
Description			
Response	Class	2	

Solution	Class	8	
Error type	1, Error message from NC program.		

ID 21979

Variables of this type are not allowed in this context.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21980

Main spindle exchanged to channel. Feed per revolution without main spindle not possible.			
Description	No main spindle exists with an active feed rate per revolution G95 since it is replaced by a path axis in the CNC channel. After the main spindle is replaced in the NC channel by #CAX or #PUT/#CALL AX, deselect the feed rate per revolution (G94) or alternatively select another main spindle (#MAIN SPINDLE): **Example with error:** N010 G95 N020 #CAX[S] N030 G01 X100 F1.5 **Corrected example:** N010 G95 F1.5 N020 #CAX[S] N030 **G94** G01 X100 F1000 **or** N010 G95 F1.5 N020 #CAX[S] N030 #MAIN SPINDLE[S2] N030 G01 X100 F1.5		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the NC program
Parameter	%1:		
		Logical axis number P-CHAN-00051 of current main spindle	
Error type	1, Error message from NC program.		

ID 21981

Variables with the type STRING are not supported for RT-Cycles.

Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21982

RT cycle functionality is not activated.			
Description	An RT cycle is programmed in the NC program. In order to use RT cycles, the parameters P-CHAN-00406 and P-CHAN-00407 must be parameterised accordingly. P-CHAN-00406 activates RT cycles and the required memory is provided for the function by P-CHAN-00407.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the parameters P-CHAN-00406/ P-CHAN-00407
Error type	1, Error message from NC program.		

ID 21984

The copying of structures is not allowed within an RT cycle.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21985-21986

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 21987

Feed factor is out of permissible value range.			
Description	An external path feed can be commanded via the PLC. In the NC program, this can be additionally weighted by a feed factor with #FF in [%] . The programmed feed factor here exceeds the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	7	Check and modify NC program. Program a feed factor within the permissible data range.

Parameter	%1:	Error value [%]
	%2:	Lower limit value [%]
	%3	Upper limit value [%]
	%4	
	%5	
Error type	1, Error message from NC program.	

ID 21988

Variable exceeds valid address range. Variable not created!			
Description	The channel-specific V.CH variable cannot be created since it does not fit in the available V.CH memory. The memory space can be increased in the parameter P-CHAN-00424. A controller reboot is then necessary.		
Response	Class	2	V.CH variable is not created and is not available in the NC program.
Solution	Class	3	Check and modify P-CHAN-00424. Increase memory space or reduce the number of V.CH variables.
Parameter	%1:	Current value [-]	
		Variable name of the affected variable	
	%2:	Current value [-]	
		Start address of the variable	
	%3:	Current value [-]	
		Size of the variable	
	%4:	Current value [-]	
		End of memory space for V.CH variables	
Error type	1, Error message from NC program.		

ID 21989

Non-positive parameters not permitted.			
Description	A non-positive filter parameter was programmed in the NC program.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the non-positive filter parameter.
Parameter	%1:	Error value [-]	
		Invalid parameter value.	
Error type	1, Error message from NC program.		

ID 21990

ORDER_TIME or TIME_CONSTANT smaller than cycle time. Filter not active.	
Description	In the NC program, an ORDER_TIME or TIME_CONSTANT which is smaller than the cycle time was programmed in the #FILTER command. This is not allowed.

Response	Class	1	The filter is deactivated and Continue NC program processing. NC program processing is continued.
Solution	Class	1	In order to use the filter, an order greater than the cycle time must be programmed.
Error type	1, Error message from NC program.		

ID 21991

Order of filter too high. Maximum possible order is used.			
Description	In the NC program, an order that exceeds the permitted maximum value was programmed in the #FILTER command. This is not allowed.		
Response	Class	1	The filter order is set to the maximum possible value and Continue NC program processing. NC program processing is continued.
Solution	Class	1	In order to use the filter correctly, program an order that does not exceed the permitted maximum value.
Error type	1, Error message from NC program.		

ID 21992

Memory for channel variables (V.CH) could not be allocated.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Error value [Byte]	
	%2:	Expected value [Byte]	
	%3	Current value [-]	
Error type	7, Error message by overflow of NC block.		

ID 21993

Mathematic operation is not alllowed for RT-Variable.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 21994

Path from start-up list will be ignored.	
Description	The program paths of NC channels can either be configured in the channel lists (P-CHAN-00401 to P-CHAN-00404) or in the start-up list (P-STUP-00018 to P-STUP-00021). Then the program paths in the channel lists are given priority. The transfer of program paths from the start-up list is discarded, The controller start-up is continued.

Response	Class	1	Warning, no reaction.
Solution	Class	6	Delete the configuration of the program paths for the channel from one of the two lists before the next start-up.
Parameter	%1:	Error value [-]	
		Path from start-up list	
	%2:	Error value [-]	
		Logical path number	
	%3	Error value [-]	
		Path priority	
	%4	Error value [-]	
		Path type	
Error type			

ID 21995

Function is not available. Has to be configured in startup parameters.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Identifier [-]	
Error type	1, Error message from NC program.		

ID 21996

Too many ESCAPE channels defined.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Current value [-]	
	%2:	Error value [-]	
	%3	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 21997

MAX_SCALE must be defined smaller than MAX_EQUID.			
Description			
Response	Class	2	
Solution	Class	3	

Parameter	%1:	Error value [-]
	%2:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 21998

MAX_EQUID must be defined greater than MAX_SCALE.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
Error type	1, Error message from NC program.		

ID 21999

At least MAX_SCALE or MAX_EQUID must be defined.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

2.3.9 ID-range 22000-22249

ID 22000

In ESCAPE no EXTEND_PARAM-channel was defined.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22001

The initialisation of structures is not allowed within an RT cycle.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22002

The used functionality is not usable with RT CYCLE.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22003

RT CYCLE DEF always has to be programmed with an ID.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22004

Token is not valid for this #-command in RT-Cycle-Context.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1	Error value [-]	
Error type	1, Error message from NC program.		

ID 22005

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 22006

Only synchronous programming is allowed for #TIMER in RT cycles.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22007

Programmed stretch factor is out of permissible value range.			
Description	The value for the STRETCH_FACTOR #TRC option is outside the permitted range.		
Response	Class	1	NC program is continued with the corrected value.
Solution	Class	1	Correct STRETCH_FACTOR #TRC option in the NC program.
Parameter	%1:	Error value [-]	
		Programmed value	
	%2:	Lower limit value [-]	
		Minimum value to specify the factor	
	%3:	Corrected value [-]	
	%4:	Upper limit value [-]	
		Maximum value to specify the factor	
Error type	1, Error message from NC program.		

ID 22008

Multiple implicit calls of subroutines or actions at NC block end.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 22009

It is not allowed to write asynchronous variables in a real-time cycle.

Description	A write access to asynchronous variables is not permissible in the real-time cycle.
	Only synchronous variables may be written within a real-time cycle.
	`#RT CYCLE [ID=4711 SCOPE = GLOBAL]`
	`V.E.Count = 2  ( asynchronous V.E. variable)`
	`#RT CYCLE END`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify P-EXTV-00005 of the affected variable in the list of external variables.
Error type	1, Error message from NC program.		

ID 22010

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 22012

Number of programmed indices in #MACHINE DATA does not match the number of indices in the parameter.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Expected value [-]	
	%2:	Expected value [-]	
Error type	1, Error message from NC program.		

ID 22013

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 22014

Missing meas signal in trafo axis. Calculation of meas values not possible.			
Description	Measurement with an active Cartesian or kinematic transformation supplies no measurement values although a measuring signal was latched in the programmed measurement axes.		
Response	Class	1	NC programming is continued without adoption of measurement values.

Solution	Class	1	A value is latched in the PLC for each axis involved in the transformation, not only for the programmed measurement axes. Only if a value is available for all trafo axes can the PCS measurement values be calculated,.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the trafo axis which is missing a measuring signal or latched value.	
Error type	1, Error message from NC program.		

ID 22015

The nc prog version number was programmed in a wrong scheme.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22016

If P-CHAN-00253 is active, A1 to A32 may only be configured as aliases.			
Description	A configured channel axis has the same name P-CHAN-00006 as the configured alias name of this axis P-AXIS-00297 in the axis list. This is not permitted when P-CHAN-00253 is active.		
Response	Class	1	Warning and correct the parameter P-CHAN-00253 to 0. Start-up is continued.
Solution	Class	1	Either correct P-CHAN-00253 to 0 or change the configured name of the channel axis P-CHAN-00006 to avoid any name collision with A1 to A31.
Parameter	%1:	Error value [-]	
		Configured axis name in the channel list	
	%2:	Corrected value [-]	
		Necessary change in the configuration	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22018

Release of C axis not allowed while active C axis processing.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 22019

Deselection of face machining during active contour rotation not permitted.	
Description	

Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22020

Deselection of surface machining during active contour rotation not permitted.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22021

In RT cycle, $ELSEIF is not allowed to have a qualifier.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22022

In RT cycle, an $IF with qualifier is not allowed to have a $ELSEIF / $ELSE.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 22023

Missing license for RT-Cycles.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 22024

Parameter of the tool has an invalid value.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 22025

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 22026

Error while decrypting data of the nc program.			
Description	An error occurred when decrypting a string in the NC program. [PROG// Encryption function]		
Response	Class	3	CNC reset
Solution	Class	2	Check whether the key matches the encrypted string.
Error type	1, Error message from NC program.		

ID 22027

Channel parameters: Parallel definition of the fixed rotary axis by "fixed_axis_idx" and "tool_ax_in_plane" is not allowed.			
Description	The channel parameters *ori.fixed_axis_index* (P-CHAN-00178) and *ori.tool_ax_in_plane* (P-CHAN-00436) may not have the identical setting.		
Response	Class	1	Warning is output and correct the parameter.
Solution	Class	1	Check and modify the parameters P-CHAN-00178 and P-CHAN-00436.
Parameter	%1:	Current value [-]	
		Parameterised value for *ori.fixed_axis_index* (P-CHAN-00178)	
	%2:	Current value [-]	
		Parameterised value for *ori.tool_ax_in_plane* (P-CHAN-00436)	
	%3	Corrected value [-]	
		New value for P-CHAN-00178	
	%4	Corrected value [-]	
		New value for P-CHAN-00436	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22028

Invalid angle programming mode with tube kinematic active.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Current value [-]	
	%2:	Parameter name [-]	
	%3	Error value [-]	

Error type	1, Error message from NC program.

ID 22029

No key for encryption of a string was defined.			
Description	A string is to be encrypted. This is not possible without this key. [PROG// Encryption function]		
Response	Class	3	CNC reset
Solution	Class	2	Enter an encryption key
Error type	1, Error message from NC program.		

ID 22030

No key for decryption of a string was defined.			
Description	A string is to be decrypted. This is not possible without the matching key. [PROG// Encryption function]		
Response	Class	3	CNC reset
Solution	Class	2	Enter the matching decryption key.
Error type	1, Error message from NC program.		

ID 22031

No license for use of calibration cycles.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 22038 / 22039

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 22043

Invalid realignment detected, check CAD data generation.

Description	Solution changes may occur in the alignment representation of the robot hand in the CAD/CAM system. The CNC treats these solution changes correctly by default, but the robot hand may execute undesirable rotary movements in the case of multiple solution changes. This is suppressed to avoid damage to the robot. The upper position limit represented in parameter 2 was exceeded in the present case.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the generation of the robot hand motion control; create an alternative motion control.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Upper limit value [0.1 10^-3 mm or ø]	
		Upper position limit	
	%3:	Current value[0.1 10^-3 mm or ø]	
		Motion path of the axis	
	%4:	Current value[0.1 10^-3 mm or ø]	
		Current position of the orientation axes	
	%5:	Current value[0.1 10^-3 mm or ø]	
		Previous position of the orientation axes	
Error type	1, Error message from NC program.		

ID 22053

Couple kinematics: Kinematics name specified in tool data cannot be found in channel parameter list.			
Description	The kinematic name specified in the tool data (P-TOOL-00148) must be identical with a kinematic name in the kinematic chain (P-CHAN-00449) of the active couple kinematic. Alternatively, P-TOOL-00148 may also contain an empty string. In this case, the tool is switched to the kinematic with the highest index in the first group.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the kinematic name specified in the tool data.
Parameter	%1:	Current value [-]	
		Kinematic ID configured in the tool data	
	%2:	Error value [-]	
		Kinematic name configured in the tool data	
Error type	1, Error message from NC program.		

ID 22060

#FILTER and #VIB GUARD cannot be combined.			
Description	The NC program was programmed with the NC commands for the Vibration Guard function (#VIB GUARD) and with the NC commands for the FIR filter function (#FILTER). This is not allowed. It is not permitted to combine Vibration Guard and Filter functions.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	It is only allowed to program either Virbation Guard or filter functions.

Error type	1, Error message from NC program.

ID 22062

Invalid Vibration Guard mode programmed.			
Description	An invalid Vibration Guard Mode was programmed in the NC program in the #VIB GUARD command. Refer to the corresponding axis parameter description (P-AXIS-00571) for the permissible values.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible value for MODE in the #VIB GUARD command.
Parameter	%1:	Current value [-]	
		Programmed mode value	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 22066

Invalid damping value programmed.			
Description	An invalid damping value (DAMPING) was programmed in the NC program in the #VIB GUARD command. Refer to the corresponding axis parameter description (P-AXIS-00568) for the permissible values.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible value for DAMPING in the #VIB GUARD command.
Parameter	%1:	Current value [-]	
		Programmed value for damping	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 22067

Invalid value for filter share factor programmed.			
Description	An invalid value for the filter share factor (SHARE) was programmed in the NC program in the #FILTER command. Refer to the corresponding axis parameter description (P-AXIS-00590) for the permissible values.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible value for SHARE in the #FILTER command.

Parameter	%1:	Current value [-]
		Programmed value for the filter share factor
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 22068

Invalid mode programmed.			
Description	An invalid value for mode (MODE) was programmed in the NC program in the #FILTER command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a permissible value for MODE in the #FILTER command.
Parameter	%1:	Current value [-]	
		Programmed mode value.	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 22073

NC command is not allowed inside of real-time loop			
Description	An NC command which is not allowed for this application was programmed in the real-time loop.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Check and modify the real-time loop in the NC program.
Parameter	%1:	Current value [-]	
		Programmed block type.	
Error type	1, Error message from NC program.		

ID 22077

Value of QUALITY filter parameter is out of permissible value range.			
Description	A filter quality of less than 0 was programmed in the NC command #FILTER (QUALITY). A filter quality of less than 0 is not defined or not possible. For more information on FIR filers:[FCT-C37//Overview]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Program a permissible value for filter quality.
Parameter	%1:	Current value [-]	
		Current value of the QUALITY programming parameter.	

Error type	1, Error message from NC program.

ID 22079

Not enough memory for creating V.CH variable.			
Description	The channel-specific V.CH variable cannot be created because there is not enough memory left. The memory space can be increased in the parameter P-CHAN-00424. A controller reboot is then necessary.		
Response	Class	2	V.CH variable is not created and is not available in the NC program.
Solution	Class	3	Check and modify P-CHAN-00424. Increase memory space or reduce the number of V.CH variables.
Parameter	%1:	Expected value [-]	
		Required memory space	
	%2:	Current value [-]	
		Available memory space	
Error type	1, Error message from NC program.		

ID 22080

Maximal amount of V.GCM. variables exceeded.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:		
	%2:		
	%3:		
	%4		
	%5:		
Error type	1, Error message from NC program.		

ID 22081

Circular programming with G303 during active TRC not permitted.

Description	When TRC is active (G41/G42), it is not allowed to program an arc in space (G303)
	
	Example with error:
	`N10 G42`
	`; ..`
	`N50 G303 I50 J50 K50 X100 Y0 Z100`
	`;..`
	`N100 G40`
	
	Corrected example:
	`N10 G42`
	`; ..;..`
	`N100 G40 X2 Y20`
	`N110 G303 I50 J50 K50 X100 Y0 Z100`
	`;…`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Before programming the circle in space, deselect the TRC.
Error type	1, Error message from NC program.		

ID 22083

Programming independent axis movements in combination with contour lines not allowed.			
Description	In contour line programming, it is not allowed to program an independent axis movement in one of the two main axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Separate the independent axis movement and contour line programming in the NC program.
Parameter	%1:	Block number [-]	
Error type	1, Error message from NC program.		

ID 22084

Invalid Kp factor for distance control programmed.			
Description	When programming distance control, the weighting value of the distance control exceeds the permissible range. The value is analogous to P-AXIS-00759.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the Kp factor in NC program.
Parameter	%1:	Error value [-]	
		Programmed value for Kp factor	
	%2:	Lower limit value [-]	
		Minimum value for Kp factor	
	%3:	Upper limit value [-]	
		Maximum value for Kp factor	
Error type	1, Error message from NC program.		

ID 22088

Axis index of programmed axis is out of range of permissible values..			
Description	Too many member kinematics are configured for a couple kinematic. The number of member kinematics results from the sum of member kinematics per group.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Check whether the axis index is within the defined limits in the programmed axis identifiers or logical axis numbers.
Parameter	%1:	Error value [-]	
		Invalid axis index	
	%2:	Current value [-]	
		Related logical axis number	
	%3:	Current value [-]	
		Related axis identifier	
	%4:	Lower limit value [-]	
		Minimum axis index	
	%5:	Upper limit value [-]	
		Maximum axis index	
Error type	1, Error message from NC program.		

ID 22089

The name of the kinematics is not allowed.			
Description	The configured kinematics name (P-CHAN-00443) may not be used. A list of names that are not allowed is in the parameter description of P-CHAN-00443.		
Response	Class	1	Output of warning
Solution	Class	1	Check and modify the kinematics name in P-CHAN-00443
Parameter	%1:	Error value [-]	
		Invalid kinematics name	
	%2:	Current value [-]	
		Related kinematics ID.	
	%3:	Current value [-]	
		Index of transformation step	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22090

The name of a kinematics contains invalid characters.			
Description	The configured kinematics name (P-CHAN-00443) contains characters that may not be used. As overview of names that are not allowed is in the parameter description of P-CHAN-00443.		
Response	Class	1	Warning output.
Solution	Class	1	Check and modify the kinematics name in factor in P-CHAN-00443

Parameter	%1:	Error value [-]
		Invalid kinematics name
	%2:	Error value [-]
		Invalid character
	%3:	Current value [-]
		List of permitted characters
Error type	2, Error message by data transfer from parameter list into control device.	

ID 22091

Homing during active couple kinematics and programmed TCP-axes is not allowed.			
Description	When a transformation and couple kinematics are active, it is not allowed to execute homing with TCP axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove the TCP axes from the homing function.
Parameter	%1:	Error value [-]	
		Axis identifier of the programmed TCP axis	
	%2:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%3:	Current value [-]	
		ID of the active kinematics	
Error type	1, Error message from NC program.		

ID 22092

Manual mode with parallel interpolation and active couple kinematics is not allowed.			
Description	Manual mode may not be activated when a transformation and couple kinematics are active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Manual mode may only be selected after couple kinematics is deselected.
Parameter	%1:	Error value [-]	
		ID of the active couple kinematics	
	%2:	Current value [-]	
		Index of transformation step	
Error type	1, Error message from NC program.		

ID 22093

The couple kinematics cannot be part of a multistep transformation.

Description	It is not allowed to use the couple kinematics as part of a multistep transformation. Not allowed: kin_step[1].trafo[1].id 210 Allowed: kin_step[**0**].trafo[1].id 210		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the programming of the kinematics.
Parameter	%1:	Error value [-]	
		ID of the active kinematics	
	%2:	Error value [-]	
		Type of active couple kinematics	
	%3:	Current value [-]	
		Index of transformation step	
Error type	1, Error message from NC program.		

ID 22094

2-path programming during active couple kinematics is not allowed.			
Description	It is not allowed to use multipath programming when couple kinematics is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Parameter	%1:	Error value [-]	
		ID of the active couple kinematics	
	%2:	Current value [-]	
		Index of transformation step	
Error type	1, Error message from NC program.		

ID 22095

5**Double programming in #TURN command.**			
Description	Double programming is in the #TURN command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the double programming.
Error type	1, Error message from NC program.		

ID 22096

Invalid Tn factor for distance control programmed.			
Description	An integral action time Tn which is not allowed for distance control is programmed in the NC program. Refer to the corresponding axis parameter description P-AXIS-00764 for the permissible values.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify NC program. Use a valid value for the integral action time Tn for distance control.
Parameter	%1:	Error value [-]	
		Incorrect value of the programmed integral action time Tn.	
	%2:	Lower limit value [-]	
		Lower limit value.	
	%3:	Upper limit value [-]	
		Upper limit value	
Error type	1, Error message from NC program.		

ID 22097

Invalid Tv factor for distance control programmed.			
Description	A derivative action time Tv which is not allowed for distance control is programmed in the NC program. Refer to the corresponding axis parameter description P-AXIS-00765 for the permissible values.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a valid value for the derivative action time Tv for distance control.
Parameter	%1:	Error value [-]	
		Incorrect value of the programmed derivative action time Tv.	
	%2:	Lower limit value [-]	
		Lower limit value	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 22099

Start position of contour definition in plane is not complete.			
Description	The first and also the second main axis must be programmed for the definition of a contour starting point (target point of the first motion block in #CONTOUR BEGIN / #CONTOUR END environment). For more information, go to Definition of a contour in NC program code		
Response	Class		Abort NC program processing.
Solution	Class		Complete definition of the contour starting point via the first two main axes.
Parameter	%1		Axis name [-]
	%2		Axis name [-]
Error type	1, Error message from NC program.		

ID 22100

Error during contour milling assignment.	
Description	It was impossible to executed the contour milling assignment successfully. Please refer to the additional information in the utility error message.

Response	Class	2	Abort NC program processing.
Solution	Class	2	Include the additional information in the utility error message.
Error type	1, Error message from NC program.		

ID 22101

NC command inside contour definition not allowed.			
Description	An NC command which is not allowed was programmed within the contour definition (#CONTOUR BEGIN / #CONTOUR END).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Remove the NC command which is not allowed from the contour definition.
Error type	1, Error message from NC program.		

ID 22102

Double programming in #MILLCONT command.			
Description	An input parameter was programmed multiple times within the #MILLCONT CALC command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Remove the double programming from the #MILLCONT command.
Error type	-		

ID 22103

Radius of rounding fall below the permitted limit.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
	%2:	Lower limit value [0.1 10^-3 mm or ø]	
Error type	1, Error message from NC program.		

ID 22104

Error while reading the calculated contour path data.			
Description	It was not possible to successfully read the contour path data. Please refer to the additional information in the utility error message.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Include the additional information in the utility error message.
Error type	1, Error message from NC program.		

ID 22105

Invalid encoding of string..			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:		
	%2:		
Error type	-		

ID 22106

Double programming in #LIMIT command.			
Description	Part of the #LIMIT command was programmed twice.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the NC program.
Parameter	%1:	Current value [-]	
		Number of references	
	%2:	Limit value [-]	
		Maximum number of references.	
Error type	1, Error message from NC program.		

ID 22107

Kinematic ID of the limit functionalities is unknown.			
Description	The ID of the kinematic specified in the channel parameter list by P-CHAN-00830 which is intended for use as the velocity limit is not defined in the channel.		
Response	Class	1	The limit was deactivated.
Solution	Class	1	Check and correct parameter P-CHAN-00830.
Parameter	%1:	Current value [-]	
		Given kinematic ID	
	%2:	Current value [-]	
		Index i from limit.kin[i].id of the invalid value	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22108

Kinematic ID of the limit functionalities is invalid.			
Description	The P-CHAN-00830 entry for the kinematic ID may not be 0.		
Response	Class	1	TCP monitoring is disabled.
Solution	Class	1	Check and correct parameter (P-CHAN-00830) for limit.kin[].
Parameter	%1:	Current value [-]	
		Given kinematic ID	
	%2:	Current value [-]	
		Index i from limit.kin[i] of the invalid value	

Error type	2, Error message by data transfer from parameter list into control device.

ID 22109

The specified velocity for the kinematic velocity limits is too low.			
Description	The specified velocity (P-CHAN-00466) for the velocity limits is too low		
Response	Class	1	Velocity limiting is deactivated.
Solution	Class	1	Check and correct parameter P-CHAN-00466 for limit.kin[].
Parameter	%1:	Current value [-]	
		Specified velocity for P-CHAN-00466	
	%2:	Lower limit value [-]	
		Minimum permissible velocity	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22110

The weighting factor for the velocity in the kinematic velocity limit is too low.			
Description	The weighting factor P-CHAN-00478 for the velocity component of the interpolation using G201 with the kinematic velocity limit is too low.		
Response	Class	1	Value is corrected
Solution	Class	1	Check and correct parameter P-CHAN-00478 for limit.kin[].
Parameter	%1:	Current value [-]	
		Specified factor P-CHAN-00478	
	%2:	Lower limit value [-]	
		Minimum permissible weighting	
	%3:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22111

The weighting factor for the velocity in the kinematic velocity limit is too high.			
Description	The weighting factor P-CHAN-00478 for the velocity component of the interpolation using G201 with the kinematic velocity limit is too high.		
Response	Class	1	Value is corrected
Solution	Class	1	Check and correct parameter P-CHAN-00478 for limit.kin[].
Parameter	%1:	Current value [-]	
		Specified factor P-CHAN-00478	
	%2:	Lower limit value [-]	
		Maximum permissible weighting	
	%3:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22112

Error while reading from file.			
Description	An unexpected error occurred while reading a file.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Contract your controller manufacturer.
Parameter	%1:	Expected value [-]	
		Expected number of read characters	
	%2:	Current value [-]	
		Number of read characters	
Error type	5, Error message by access on files.		

ID 22114

Error in commissioning pattern processing.			
Description	It was impossible to execute the pattern assignment successfully. Please refer to the additional information in the utility error message.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Include the additional information in the utility error message.
Error type	1, Error message from NC program.		

ID 22115

NC command inside pattern definition not allowed.			
Description	An invalid NC command was programmed in the pattern definition (#PATTERN BEGIN / #PATTERN END).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the pattern definition. Delete the invalid NC command.
Error type	1, Error message from NC program.		

ID 22116

Double programming in #PATTERN command.			
Description	An input parameter was programmed multiple times in the #PATTERN command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Delete the double programming from the #PATTERN command.
Error type	1, Error message from NC program.		

ID 22117

NC command outside pattern definition not allowed.			
Description	The #AT command was programmed outside the permitted range. It may only be called in the pattern definition (#PATTERN BEGIN / #PATTERN END).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Delete the invalid NC command.

Error type	1, Error message from NC program.		

ID 22118

Error while reading the programmed pattern data.			
Description	It was not possible to successfully read the pattern data. Please refer to the additional information in the utility error message.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Include the additional information in the utility error message.
Error type	1, Error message from NC program.		

ID 22121

Trafo PTP programming with the active kinematics not allowed.			
Description	It is not allowed to program the command #PTP with the kinematics used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; remove the PTP programming.
Parameter	%1:	Error value [-]	
		ID of the active kinematics	
	%2:	Error value [-]	
		Name of the active kinematics	
	%3:	Current value [-]	
		Index of transformation step	
Error type	1, Error message from NC program.		

ID 22122

Contouring mode PTP during active kinematics not allowed.			
Description	It is not allowed to use the PTP contouring mode (#CONTOUR MODE [PTP …]) with the kinematic used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; use a different contouring mode.
Parameter	%1:	Error value [-]	
		ID of the active kinematics	
	%2:	Current value [-]	
		Index of transformation step	
Error type	1, Error message from NC program.		

ID 22123

Bisector Escape parameter D1 must be positive.			
Description	A negative parameter D1 was specified when the bisector escape strategy was used. This is not permitted The value for D1 must be greater than zero.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Check and modify the NC program. Specify a value greater than or equal to zero for parameter D1.
Parameter	%1:	Error value [0.1 µm]	
		Parameterised value for distance D1	
Error type	1, Error message from NC program.		

ID 22124

Bisector Escape parameter D2 must be positive.			
Description	A negative parameter D2 was specified when the bisector escape strategy was used. This is not permitted The value for D2 must be greater than zero.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Specify a value greater than or equal to zero for parameter D2.
Parameter	%1:	Error value [0.1 µm]	
		Parameterised value for distance D2	
Error type	1, Error message from NC program.		

ID 22125

TRC options ONLINE and INVERSE are only permissible when combined.			

Description	Only one of the TRC options ONLINE or INVERSE is activated. This is not permitted.
	Either both options must be activated or deactivated before TRC is activated by G41/G42.
	Activate both options:
	• #TRC[ONLINE=1,2,3] and #TRC[INVERSE = 1]
	Deactivate both options:
	• #TRC[ONLINE=0] and #TRC[INVERSE = 0]
	
	Option switching may not take place in a program line but before TRC is activated.
	
	`N30 #TRC[ONLINE=1]`
	`…`
	`N60 #TRC[INVERSE = 1]`
	`…`
	`N100 G01 G41 X10 Y10`		

Response	Class	3	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. For selecting TRC, either: • Activate both options • Or deactivate them
Error type	1, Error message from NC program.		

ID 22126

Production factor is out of range of permissible value.	
Description	The production factor was specified by PROD_FACT in the NC command #LAH. The assigned value is outside the permissible range.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the specified value for the production factor.
Parameter	%1:	Error value [-]	
		Specified value for the production factor.	
	%2:	Lower limit value [-]	
		Minimum permissible production factor	
	%3:	Upper limit value [-]	
		Maximum permissible production factor	
Error type	1, Error message from NC program.		

ID 22127

Double programming in #LAH command.			
Description	The NC command #LAH[…] was programmed with a keyword multiple times.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 22128

Spline is active at start of #RT WHILE/ENDWHILE sequence.			

Description	No spline mechanism may be active before an #RT WHILE is programmed.
	For example, a spline mechanism can be activated with the NC commands #SPLINE ON, #HSC ON, G151 or G261.
	The corresponding NC commands to deselect the function in each case is #SPLINE OFF, #HSC OFF, G150 or G260.
	
	Example with error:
	`N100 #SPLINE ON`
	`N110 #RT WHILE`
	`N120 G01 X10`
	`N130 G01 Y10`
	`N140 G01 X0`
	`N150 G01 Y0`
	`N160 #SPLINE OFF`
	`N170 #RT ENDWHILE`
	
	Corrected example:
	`N100 #RT WHILE`
	`N110 #SPLINE ON`
	`N120 G01 X10`
	`N130 G01 Y10`
	`N140 G01 X0`
	`N150 G01 Y0`
	`N160 #SPLINE OFF`
	`N170 #RT ENDWHILE`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Possible solutions: • Deselect the spline with the corresponding NC command paired with the selection before #RT WHILE • Select the splined after #RT WHILE (see corrected example)

Parameter	%1:	Block number [-]
		Block number of the #RT WHILE command where a spline mechanism is active.
Error type	1, Error message from NC program.	

ID 22129

Spline is active at end of #RT WHILE/ENDWHILE sequence.			

Description	A spline mechanism was activated within a #RT WHILE/ ENDWHILE area. This must be deactivated before #RT ENDWHILE.
	For example, a spline mechanism can be activated with the NC commands #SPLINE ON, #HSC ON or G151. The deactivating NC commands are #SPLINE OFF, #HSC OFF and G150.
	
	Example with error:
	`N100 #RT WHILE`
	`N110 #SPLINE ON`
	`N120 G01 X10`
	`N130 G01 Y10`
	`N140 G01 X0`
	`N150 G01 Y0`
	`N160 #RT ENDWHILE`
	`N170 #SPLINE OFF`
	
	Corrected example:
	`N100 #RT WHILE`
	`N110 #SPLINE ON`
	`N120 G01 X10`
	`N130 G01 Y10`
	`N140 G01 X0`
	`N150 G01 Y0`
	`N160 #SPLINE OFF`
	`N170 #RT ENDWHILE`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deselect the spline mechanism before #RT ENDWHILE
Parameter	%1:	Block number [-]	
		Block number of the #RT WHILE command where a spline mechanism is active.	
Error type	1, Error message from NC program.		

ID 22130

Double programming for distance interpretation in #INSERT CMD command.			
Description	Several keywords for distance interpretation were specified in the programming in the #INSERT CMD command. This is not permitted. Keywords for distance interpretation: • REL -relative repetitive • ABS -absolute • REL_ONCE – relative once		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Parameter	%1:	Block number [-]	
		Block number of the incorrect #INSERT CMD command.	

Error type	1, Error message from NC program.

ID 22131

Double programming in #INSERT CMD command.			
Description	Both ON and OFF were programmed in the NC command #INSERT CMD. This is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Parameter	%1:	Block number [-]	
		Block number of the incorrect #INSERT CMD command.	
Error type	1, Error message from NC program.		

ID 22132

Invalid combination of keywords words in #OPTIONAL EXECUTION command.			
Description	In the NC command #OPTIONAL EXECUTION[] the combination of APPROACH and SIMULATE or SIMULATE MASK was used. These combinations are not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 22134

Number of measured values of distance control is out of data format.			
Description	When the sensor values for distance control were filtered, the number of measured values used for filtering can be specified by the keyword “DISTC_N_CYCLES”. The number exceeds the permissible range. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Check and modify the value for “DISTC_N_CYCLES” in the NC program.
Parameter	%1:	Error value [-]	
		Incorrectly programmed value for “DISTC_N_CYCLES”	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 22135

An invalid number of measured values was programmed for distance control.

Description	When the sensor values for distance control were filtered, the number of measured values used for filtering can be specified by the keyword “DISTC_N_CYCLES”. The number exceeds the permissible range. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Check and modify the value for “DISTC_N_CYCLES” in the NC program.
Parameter	%1:	Error value [-]	
		Incorrectly programmed value for “DISTC_N_CYCLES”	
	%2:	Lower limit value [-]	
		Lower limit for “DISTC_N_CYCLES” specification	
	%3:	Upper limit value [-]	
		Upper limit for “DISTC_N_CYCLES” specification	
Error type	1, Error message from NC program.		

ID 22136

Invalid filter type for distance control programmed.			
Description	When the sensor values for distance control were filtered, an invalid filter type was programmed in the NC program. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the filter type in the NC program.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 22137

An invalid value for KALMAN_SIGMA was programmed for distance control.			
Description	When the sensor values for distance control are filtered, the uncertainty of the measured values can be specified by the keyword "KALMAN_SIGMA". The value is outside the specified limits. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing. .
Solution	Class	3	Check and modify the value for KALMAN_SIGMA in the NC program.
Parameter	%1:	Error value [-]	
		Programmed value for KALMAN_SIGMA	
	%2:	Lower limit value [-]	
		Minimum value for KALMAN_SIGMA	
	%3:	Upper limit value [-]	
		Maximum value for KALMAN_SIGMA	
Error type	1, Error message from NC program.		

ID 22138

Invalid smoothing factor for distance control programmed.

Description	When the sensor values for distance control are filtered, the smoothing factor for the exponential average filter can be specified by the keyword "SMOOTH_FACT". The value is outside the specified limits. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value for “SMOOTH_FACT” in the NC
Parameter	%1:	Error value [-]	
		Incorrectly programmed value for “SMOOTH_FACT”	
	%2:	Lower limit value [-]	
		Minimum value for “SMOOTH_FACT”	
	%3:	Upper limit value [-]	
		Maximum value for “SMOOTH_FACT”	
Error type	1, Error message from NC program.		

ID 22139

Invalid maximum distance between two measuring points of the distance control programmed.			
Description	When axis-specific distance control was used, the maximum distance between two measuring points was specified. The specified value is invalid.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		Programmed value.	
	%2:	Lower limit value [0.1 10^-3 mm or ø]	
		Minimum value for the distance.	
Error type	1, Error message from NC program.		

ID 22140

Invalid cut-off frequency of distance control programmed.			
Description	When the sensor values for distance control are filtered, the limit frequency for the low-pass filter can be specified by the keyword "DISTC_FG_F0". The value is outside the permissible limits. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value for “DISTC_FG_F0” in the NC program.
Parameter	%1:	Error value [Hz]	
		Incorrectly programmed value for “DISTC_FG_F0”	
	%2:	Lower limit value [Hz]	
		Lower limit value for “DISTC_FG_F0"	
	%3:	Upper limit value [Hz]	
		Upper limit value for “DISTC_FG_F0”	
Error type	1, Error message from NC program.		

ID 22141

Invalid order of the low-pass filter for the distance control programmed.			
Description	When the sensor values for distance control are filtered, the order for the low-pass filter can be specified by the keyword "DISTC_ORDER". The value is outside the permissible limits. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value for “DISTC_ORDER” in the NC program.
Parameter	%1:	Error value [-]	
		Incorrectly programmed value for “DISTC_ORDER”	
	%2:	Lower limit value [-]	
		Lower limit value for “DISTC_ORDER"	
	%3:	Upper limit value [-]	
		Upper limit value for “DISTC_ORDER”	
Error type	1, Error message from NC program.		

ID 22142

Order of the low-pass filter of distance control is out of data format.			
Description	When the sensor values for distance control are filtered, the order for the low-pass filter can be specified by the keyword "DISTC_ORDER". The number exceeds the permissible data format. Further information on programming distance control in [FCT-M3// Programming]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value for “DISTC_ORDER” in the NC program.
Parameter	%1:	Error value [-]	
		Incorrectly programmed value for “DISTC_ORDER”	
	%2:	Lower limit value [-]	
		Lower limit value for “DISTC_ORDER"	
	%3:	Upper limit value [-]	
		Upper limit value for “DISTC_ORDER”	
Error type	1, Error message from NC program.		

ID 22143

Incorrect programming in #OPTIONAL EXECUTION command.			
Description	At least 2 of the keywords ON or OFF were used in the same command when programming the NC command #OPTIONAL EXECUTION. This is not permitted. Example: `#OPTIONAL EXECUTION ON OFF`		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.Delete the superfluous keywords
Error type	1, Error message from NC program.		

ID 22144

#CHANNEL SET: option EXT_FEEDRATE_WAIT can only be used with EXT_FEEDRATE_RESOLUTION.			
Description	It is not possible to use the option EXT_FEEDRATE_WAIT in the #CHANNEL SET command without the option EXT_FEEDRATE_RESOLUTION. See Synchronisation with an external velocity interface		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 22145

Double programming in #DIAGNOSIS command.			
Description			
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Error value [-]	
Error type	-		

ID 22146

Area #DIST TO GO BEGIN has to be closed with #DIST TO GO END.			

Description	The range for the Distance to go display in a program section in an NC program must be specified between
	`#DIST TO GO BEGIN`
	and
	`#DIST TO GO END`
	. The NC command #DIST TO GO END was not programmed.		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 22147

#DIST TO GO END without #DIST TO GO BEGIN not allowed.			

Description	The range for the Distance to go display in a program section in an NC program must be specified between
	`#DIST TO GO BEGIN`
	and
	`#DIST TO GO END`
	. The NC command #DIST TO GO BEGIN was not programmed.		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 22148

Not enough memory to inherit @P parameters.			
Description	The INHERIT parameter in a cycle call transfers the higher-level @P parameters in the called cycle. This error message is output if the available memory is not sufficient. Parameter 2 is used to specify the additional memory required. The additional memory required must be specified by P-CHAN-00481 (*cycle_stack_memory*), additively if required.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the parameter P-CHAN-00481
Parameter	%1:	Current value [-]	
		Memory currently available.	
	%2:	Expected value [-]	
		Memory requirement.	
Error type	1, Error message from NC program.		

ID 22149

Contouring parameter is out of data format.			
Description	The value of a parameter in the command #CONTOUR MODE is outside the permissible data format.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the value in the NC program.
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 22150

Unknown keyword in #DIAGNOSIS command.			

Description	An invalid keyword was used to program the diagnosis upload from the NC program.
	Example with error:
	`#DIAGNOSIS [ULOAD=TO_FILE FILEPATH=D:\temp\test.txt]`
	Corrected example:
	`#DIAGNOSIS [UPLOAD PATH =D:\temp FILE=test.txt]`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 22151

Invalid character in cycle call.

Description	An implicit subroutine via a G function was parameterised.
	This is possible using P-CHAN-00160 for G80 up to P-CHAN-00169 for G89.
	When the G function is called, additional parameters can be transferred to the subroutine. This takes place as follows:
	`G80 [1,2,3]`
	The parameters transferred contain an invalid character, e.g.
	`G80 [12 !]`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Delete the invalid character.
Error type	1, Error message from NC program.		

ID 22153

Missing lead axis with DIST MASTER contouring mode.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 22155

Double programming at selection/ deselection of contour rotation.			

Description	In the same NC block, more than one G function was programmed from the contour rotation group G68/G69.
	Example:
	`Wrong:`
	`N10 G00 X0 Y0 Z0`
	`N20 G68 R30 X150 Y200 G69`
	`:`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the invalid G function.
Parameter	%1:	Error value [-]	
		Number of the incorrectly programmed G function	
Error type	1, Error message from NC program.		

ID 22156

When selecting contour rotation third main axis may not be programmed.

Description	The third main axis is programmed in an NC block with G68. However, only main axes one and two of the current plane may specify the offset for a contour rotation (e.g. X and Y with G17). Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 G68 R30 X150 Y200 **Z100** : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Delete the third main axis programmed (e.g. Z axis with G17) from the NC block or move it to the next NC block.
Parameter	%1:	Error value [-]	
		Logical axis number of the third main axis	
Error type	1, Error message from NC program.		

ID 22157

Selection of contour rotation and plane in same NC block not allowed.			
Description	A plane selection (G17, G18 or G19) is programmed in an NC block with G68. However, this must occur before a new NC block with G68 is executed. Example: **Wrong:** N10 G00 X0 Y0 Z0 N20 **G17** G68 R30 X150 Y200 : N1000 M30 **Correct:** N10 G00 X0 Y0 Z0 N15 **G17** N20 G68 R30 X150 Y200 : N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Delete the plane selection (G17, G18 or G19) from the NC block and move it to a previous NC block.
Parameter	%1:	Error value [-]	
		G number of the plane	
	%2:	Current value [-]	
		G number of the contour rotation selection	
Error type	1, Error message from NC program.		

ID 22158

#RT WHILE/ ENDWHILE while active tool radius compensation not allowed.			
Description	If tool radius compensation is activated with G41 or G42, it is not possible to program a real-time loop with #RT WHILE/ ENDWHILE.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deselect tool radius compensation with G40 before the real-time loop.
Error type	1, Error message from NC program.		

ID 22159

Double programming of radius definition.			
Description	When a circle is programmed (G01/G03) in the NC block, the radius is simultaneously defined indirectly by specifying the circle centre point I, J, K and directly using the R word or G163. As a result, circle programming is no longer unique.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Delete one type of radius specification from the NC block.
Error type	1, Error message from NC program.		

ID 22162

Only strings and numbers can be used within a FSTRING.			
Description	When FSTRING is used, only strings or numerical values may be used in the programmed NC lines.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program.
Error type	1, Error message from NC program.		

ID 22163

The same CS ID has been defined several times.		
Description	P-CHAN-00490 was assigned multiple times to the same string.	
Response	Class	1	NC program is continued with the corrected value.
Solution	Class	1	The second definition is deleted.
Parameter	%1:	Current value [-]
		Index of the deleted CS definition
	%2:	Error value [-]
		The specified CS ID.
Error type	-	

ID 22164

Name of transformation stack already defined.			
Description	P-CHAN-00752 was assigned multiple times to the same string..		
Response	Class	1	NC program is continued with the corrected value.
Solution	Class	1	The transformation stack definition is deleted.
Parameter	%1:	Current value [-]	
		Index of the deleted transformation stack	
	%2:	Error value [-]	
		The specified stack name.	
Error type	-		

ID 22165

The specified kinematic ID does not exist.			
Description	The ID of a non-existent kinematic was specified in P-CHAN-00831.		
Response	Class	1	NC program is continued with the corrected value.
Solution	Class	1	The transformation stack definition is deleted.
Parameter	%1:	Current value [-]	
		Index of the deleted transformation stack	
	%2:	Error value [-]	
		The kinematic ID Step 0.	
	%3:	Error value [-]	
		The kinematic ID Step 1.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22166

CS ID does not exist.			
Description	An unknown CS ID was assigned to a transformation stack. The CS ID must be defined by P-CHAN-00490. For example, `coordinate_system.def[0].id  CS_ID_10`		
Response	Class	1	NC program is continued with the corrected value.
Solution	Class	1	The CS ID is deleted from the transformation stack.
Parameter	%1:	Current value [-]	
		Index of the transformation stack containing the unknown CS ID.	
	%2:	Error value [-]	
		The specified CS ID.	
Error type	-		

ID 22167

Name of transformation stack does not exist.			
Description	An unknown stack name was assigned to P-CHAN-00757.		
Response	Class	1	NC program is continued with the corrected value.
Solution	Class	1	Parameter is deleted.
Parameter	%1:	Error value [-]	
		The specified stack name.	
Error type	-		

ID 22168

Maximum number of permissible CS translations within one stack exceeded.			
Description	Too many coordinate transformations were assigned to a transformation stack.		
Response	Class	1	NC program is continued with the corrected value.
Solution	Class	1	The CS ID is deleted from the stack

Parameter	%1:	Current value [-]
		Index of the transformation stack containing the unknown CS ID.
	%2:	Error value [-]
		The specified CS ID.
	%3:	Limit value [-]
		The maximum number of coordinate transformations within a stack.
Error type	-	

ID 22169

Transformation stack does not exist.			
Description	An unknown stack name was specified in the NC command #TRAFO STACK ON.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct the name in the NC program.
Parameter	%1:	Error value [-]	
		The specified name.	
Error type	1, Error message from NC program.		

ID 22170

Incomplete instruction inserting a translation into the transformation stack.			
Description	An error occurred in the definition of a transformation stack in the NC program when specifying a coordinate transformation. A maximum of 10 CS IDs may be specified and a maximum of 5 CS IDs per group. An assignment always consists of the GRP and ID.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct command in NC program.
Parameter	%1:	Error value [-]	
		The index of the last group	
	%2:	Error value [-]	
		Index within the group	
	%3:	Error value [-]	
		The last CS ID	
Error type	1, Error message from NC program.		

ID 22172

Missing name in definition of a transformation stack.			
Description	A name must be specified for the stack in the definition of a transformation stack in the NC program.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct command in NC program.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 22173

The given CS ID may not be added to the stack.			
Description	Only CS IDs defined in the channel parameter list P-CHAN-00490 may be assigned to a transformation stack.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct command in NC program.
Parameter	%1:	Current value [-]	
		Index of the group	
	%2:	Error value [-]	
		The specified CS ID.	
	%3:	Limit value[-]	
		The maximum number of coordinate transformations within a stack.	
	%4:	Upper limit value [-]	
		Maximum value to specify the factor	
Error type	1, Error message from NC program.		

ID 22174

Kinematic ID does not exist.			
Description	The kinematic ID specified in the NC command #TRAFO STACK does not exist.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct command in NC program.
Parameter	%1:	Current value [-]	
		ID of kinematic Step 0.	
	%2:	Current value [-]	
		ID of kinematic Step 1.	
Error type	1, Error message from NC program.		

ID 22175

Maximum quantity of transformation stacks exceeded.			
Description	This new definition exceeds the maximum number of transformation stacks.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Correct command in NC program.
Parameter	%1:	Current value [-]	
		Specified transformation stack name.	
	%2:	Limit value [-]	
		Maximum number of transformation stacks	
Error type	1, Error message from NC program.		

ID 22178

Missing PATTERN_RETRACT in pattern call.

Description	When a cycle is called as a call pattern, the corresponding call pattern retraction level PATTERN_RETRACT, to which the tool is retracted before each new positioning, must be transferred to the cycle in addition to the PATTERN_ID. Call example: L CYCLE [NAME=... @P.. = .. \ … PATTERN_ID = .. PATTERN_RETRACT = ..] **The PATTERN_RETRACT parameter is missing**.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the cycle call. Transfer the parameters PATTERN_ID and PATTERN_RETRACT at cycle call.
Error type	1, Error message from NC program.		

ID 22179

Double programming of acceleration weighting.			
Description	More than one G function from the group of acceleration weightings (G130/G131/ G231/G333/G334) was programmed in the same NC block. Example with error: N10 G130 X70 **G131** = 100 ;… N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the illegal programmed G function	
Error type	1, Error message from NC program.		

ID 22180

Double programming of ramp time weighting.			
Description	More than one G function from the group of ramp time weightings (G132/ G133/ G134/G233/G338/G339) was programmed in the same NC block. Example with error: N10 G132 X70 **G133** = 100 ;… N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the illegal programmed G function	
Error type	1, Error message from NC program.		

ID 22181

#RT ENDWHILE without previous #RT WHILE is not allowed.			
Description	Real-time cycles [FCT-C42// Real-time loops] are enclosed between the NC commands #RT WHILE and #RT ENDWHILE. It is impossible to nest real-time cycles.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. End the real-time loop with #RT ENDWHILE.
Error type	1, Error message from NC program.		

ID 22182

Double programming of maximum velocity weighting.			
Description	More than one G function from the group of maximum velocity weightings (G127/ G128) was programmed in the same NC block. Example with error: N10 G127 X70 **G128** = 100 ;… N1000 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Error value [-]	
		Number of the illegal programmed G function	
Error type	1, Error message from NC program.		

ID 22183

Programmed distance is negative or zero.			

Description	When feed splitting was used to select or deselect the tool radius compensation, DIST_SEG1 or DIST_SEG2 was assigned a value less than or equal to 0.
	Example with error:
	`N10 #TRC [SPLIT_PATH COMBINED FEED_SEG1=1111 \`
	`  **DIST_SEG1=0** FEED_SEG2=500 DIST_SEG2=20]`
	This is not permitted.
	For more information on feed splitting, see TRC option SPLIT.		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check NC program and use path parameter greater than 0.
Parameter	%1:	Error value [-]	
		Incorrectly programmed path parameter	
Error type	1, Error message from NC program.		

ID 22189

Invalid rotation sequence programmed.

Description	The rotation sequence of coordinate systems pre-assigned by P-CHAN-00394 can be changed in the NC program by the command #CS MODE ON [ROTATION_SEQUENCE=..]. In this case, an invalid rotation sequence was specified.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Specify a valid rotation sequence in the NC Command. The following combinations are permitted: XYZ XYX XZY XZX YXZ YXY YZX YZY ZXY ZXZ ZYX ZYZ
Parameter	%1:	Error value [-]	
		Invalid programmed rotation sequence	
	%2:		
	%3:		
	%4:		
	%5:		
Error type	1, Error message from NC program.		

ID 22190

Invalid rotation sequence configured.			
Description	At start-up, the channel parameter check determined that the rotation sequence pre-assigned in P-CHAN-00394 contains an invalid rotation sequence.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	At start-up and in case of conflict, assign the rotation sequence of P-CHAN-00394 with the default convention of ZYX and the start-up can continue.
Parameter	%1:	Error value [-]	
		Invalid configured rotation sequence	
	%2:	Corrected value [-]	
		Default sequence	
	%3:		
	%4:		
	%5:		
Error type	-		

ID 22191

Invalid pose status programmed.			
Description	The programmed pose status (IS) is not permitted for the active kinematic with #PTP. See also [PROG// Status & Turn (IS, IT)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program; program a permitted pose status.

Error type	1, Error message from NC program.

ID 22192

Tool life distance is negative.			
Description	The tool life distance programmed in the NC command #TOOL LIFE DEF [..] with the keyword DIST was programmed with a negative value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the tool life distance DIST with a value >= 0.
Parameter	%1:	Error value [0.1 10^-7 mm or ø]	
		Negative programmed tool life distance	
	%2:		
	%3:		
	%4:		
	%5:		
Error type	1, Error message from NC program.		

ID 22193

Tool life time is negative.			
Description	The tool life time programmed in the NC command #TOOL LIFE DEF [..] with the keyword TIME was programmed with a negative value.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Program the tool life time TIME with a value >= 0.
Parameter	%1:	Error value [µs]	
		Negative programmed tool life time	
	%2:		
	%3:		
	%4:		
	%5:		
Error type	1, Error message from NC program.		

ID 22194

Message to receiver via #MSG command is configured as warning.

Description	The channel parameters (P-CHAN-00518, P-CHAN-00519, P-CHAN-00520) can configure the behaviour of messages to each of the receivers via the #MSG command.. When this message occurs, the messages to the receiver are disabled in parameter 1 and are indicated as a warning.		
Response	Class	1	No reaction
Solution	Class	1	Possible solutions are: • Change channel parameter • Delete the #MSG command
Parameter	%1:	Current value [-]	
		Current receiver of the #MSG command.	
Error type	1, Error message from NC program.		

ID 22195

Message to receiver via #MSG command is configured as error.			
Description	The channel parameters (P-CHAN-00518, P-CHAN-00519, P-CHAN-00520) can configure the behaviour of messages to each of the receivers via the #MSG command.. When this message occurs, the messages to the receiver are disabled in parameter 1 and are acknowledged as error.		
Response	Class	2	NC program execution aborted and transition to error state
Solution	Class	6	Possible solutions are: • Change channel parameter • Delete the #MSG command
Parameter	%1:	Current value [-]	
		Current receiver of the #MSG command	
Error type	1, Error message from NC program.		

ID 22200

Number of filter cycles for tapping with actual position is out of range of permissible value.			
Description	The actual spindle positions may be noisy under certain circumstances. Values are smoothed by using this filter. The number of cycles specified in the NC program #TURN[TAPPING_N_CYCLES=..] and used for filtering is outside of the permissible value range. See PROG//Settings for turning functions (# TURN)		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Correct the value for the number of filter cycles.
Parameter	%1:	Current value [-]	
		Programmed value	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
		Maximum permissible number of filter cycles	

Error type	1, Error message from NC program.

ID 22201

Number of filter cycles for tapping with actual position is out of range of permissible value.			
Description	The actual spindle positions may be noisy under certain circumstances. Values are smoothed by using this filter. The number of cycles under review and specified in the parameter P-CHAN-00762 is outside the specified value range. The value is set to the upper limit value.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify the parameter settings in P-CHAN-00762.
Parameter	%1:	Current value [-]	
		Parameterised value of P-CHAN-00762	
	%2:	Upper limit value [-]	
		Maximum permissible number of filter cycles	
	%3:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22203

Invalid programming order of #ESCAPE PATH commands.			

Description	There is a fixed sequence for defining an #ESCAPE PATH. This sequence was not complied with.
	#ESCAPE PATH Command structure:
	1. #ESCAPE PATH DEF BEGIN
	2. #ESCAPE PATH BACKWARD STOP
	3. #ESCAPE PATH POST SEQUENCE
	4. #ESCAPE PATH DEF END
	
	Example:
	`N010 #ESCAPE PATH DEF BEGIN`
	`N020 #CS ON [APP] [@PL4,@PL5,@PL6,@PL7,@PL8,@PL9]`
	`N030 #ESCAPE PATH BACKWARD STOP`
	`N040 G01 Z7.07 F30`
	`N050 G01 Y0`
	`N060 G01 Z0`
	`N070 #ESCAPE PATH POST SEQUENCE`
	`N080 #CS DEL ALL`
	`N090 #ESCAPE PATH DEF END`
	`N100 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 22204

Double programming of a #ESCAPE PATH command.

Description	A command was programmed twice within the #ESCAPE PATH command structure. #ESCAPE PATH Command structure: 1. #ESCAPE PATH DEF BEGIN 2. #ESCAPE PATH BACKWARD STOP 3. #ESCAPE PATH POST SEQUENCE 4. #ESCAPE PATH DEF END		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 22205

Active #ESCAPE PATH on program end.			
Description	#ESCAPE PATH DEF BEGIN was not terminated before program end. #ESCAPE PATH Command structure: 1. #ESCAPE PATH DEF BEGIN 2. #ESCAPE PATH BACKWARD STOP 3. #ESCAPE PATH POST SEQUENCE 4. #ESCAPE PATH DEF END		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Add #ESCAPE PATH DEF END.
Error type	1, Error message from NC program.		

ID 22206

External tool management sent an unexpected response.			
Description	An unexpected acknowledgement was sent to the CNC by the external tool management.		
Response	Class	1	Continue NC program processing. The unexpected acknowledgement is ignored.
Solution	Class	1	Check the communication in the external tool management.
Error type	3, Error in the communication.		

ID 22207

Communication error with external tool manager.			
Description	A message containing an error code was received in a communication with the external tool manager.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check the CNC-external communication device.
Parameter	%1:	Status [-] Error code of received message.	
Error type	3, Error in the communication.		

ID 22209

Load definition without id.			

Description	A load was defined in the NC program without specifying the ID.
	Example with error:
	`#LIMIT LOAD DEF [X=0 Y=0 Z=0 RX=0 RY=0 RZ=0 IX=0.1 IY=0.1 IZ=0.025916 MASS=7.9]`
	Corrected example:
	`#LIMIT LOAD DEF [X=0 Y=0 Z=0 RX=0 RY=0 RZ=0 IX=0.1 IY=0.1 IZ=0.025916 MASS=7.9 ID=1]`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; specify the ID in the #LIMIT LOAD command.
Error type	1, Error message from NC program.		

ID 22210

Load model not possible without active TCP velocity monitoring.			

Description	Use of the load model is coupled to the velocity limit. The velocity limit is not activated for this kinematic.
	The velocity limit is activated in P-CHAN-00464.
	Incorrect configuration
	`limit.kin[0].active 0   (P-CHAN-00464)`
	Corrected configuration
	`The load model requires an active limit functionality.`
	`limit.kin[0].id     100 (P-CHAN-00830)`
	Further information:
	• Velocity limit,see FCT-C47
	• Load model,see FCT-C48		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the configuration and activate P-CHAN-00464
Parameter	%1:	Error value [-]	
		State of P-CHAN-00464	
Error type	1, Error message from NC program.		

ID 22213

Kinematic name was configured multiple times.			
Description	When the kinematics were parameterised, a kinematic name was configured more than once. This is not permissible since there is then no unique access to the kinematic.		
Response	Class	1	Output of warning; the kinematic cannot be selected in the NC program.
Solution	Class	1	Correct the kinematic name so that the names of all configured kinematics are unique.

Parameter	%1:	Current value [-]
		Name of the kinematic
	%2:	Current value [-]
		Kinematic step in which the error occurred.
	%3:	Current value [-]
		Index of incorrectly configured kinematic
Error type	2, Error message by data transfer from parameter list into control device. -	

ID 22214

Variable parameter of #LINSOLVE command may not be indicated.			

Description	When the #LINSOLVE command is used, no elements of a multi-dimensional variable may be transferred to the parameters A, B or X.
	Not allowed:
	`#LINSOLVE [A=V.L.ARRAY[2][3], …`
	Allowed:
	`#LINSOLVE [A=V.L.ARRAY, …`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the parameters for the #LINSOLVE command
Error type	1, Error message from NC program.		

ID 22215

Variable in #LINSOLVE command is not a matrix.			
Description	When the #LINSOLVE command was used, a variable that is not a matrix was transferred,. This is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the parameters for the #LINSOLVE command
Error type	1, Error message from NC program.		

ID 22216

Not enough memory for mathematical computations.			
Description	Some cycles and NC commands perform complex mathematical calculations. This error occurs when the memory provided for the calculation is not sufficient. Example of possible error handling using the output of this message:; • Expected value 1 = required memory = 85 • Expected value 2 = required memory = 13 • The parameter P-CHAN-00521 must be increased by at least 85 - 13 = 72		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the channel parameter P-CHAN-00521 A change in the parameter requires a controller restart.

Parameter	%1:	Expected value
		Required memory for the calculation
	%2:	Current value [-]
		Existing memory for the calculation
		Parameter name [-]
		Parameter to be modified
Error type	1, Error message from NC program.	

ID 22217

Invalid data type of variable in #LINSOLVE command.			
Description	When the #LINSOLVE command is used, only data types of the REAL64 type may be transferred.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the parameter data types for the #LINSOLVE command
Parameter	%1:	Error value [-]	
		Data type used	
	%2:	Expected value [-]	
		Expected data type	
Error type	1, Error message from NC program.		

ID 22218

Invalid entry in #LINSOLVE command.			
Description	When the #LINSOLVE command was used, a matrix was used with an correct form. Example: Matrix A must be square with METHOD=GEM.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; use a different solution method in the #LINSOLVE command (METHOD).
Parameter	%1:	Error value [-]	
		Description of the incorrect property	
Error type	Error message from NC cycle.		

ID 22219

Invalid variable type in #LINSOLVE command.			
Description	When the #LINSOLVE command is used, only variables of the V.CYC, V.S, V.P, V.L types may be used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the data types for the #LINSOLVE command in the NC program.
Parameter	%1:	Current value [-]	
		Variable type used	
Error type	1, Error message from NC program.		

ID 22224

Too many parameters in #KIN TCP command programmed.			
Description	Too many axis coordinates were programmed for the TCP position in the #KIN TCP DEF […] command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the number of coordinates programmed. If necessary, check the kinematics in the channel parameter list. The kinematic with ID 211 must be available.
Parameter	%1:	Current value [-]	
		Number of programmed coordinates in the command.	
	%2:	Upper limit value	
		Maximum number of programmable coordinates.	
Error type	1, Error message from NC program.		

ID 22225

#KIN TCP can only be programmed when transformation is inactive.			
Description	The #KIN TCP DEF […] command was programmed with an active transformation, i.e. after #TRAFO ON..		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Change the order of the #KIN TCP programming. Program #KIN TCP DEF [..] before #TRAFO ON.
Error type	1, Error message from NC program.		

ID 22226

The activated transformation is not supported by #KIN TCP.			
Description	The currently selected kinematic is not suitable for using #KIN TCP .. . Th #KIN TCP .. command may only be used in connection with the select double gantry kinematic (#KIN ID [211]).		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. When programming #KIN TCP DEF [..], make sure the double portal kinematic (KIN-ID 211) is selected by programming or configuration.
Error type	1, Error message from NC program.		

ID 22227

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 22229

No license for use of edm functions.			
Description	The EDM license and a function it contains is missing. The following functions are part of the license pack: • #TRANSFORM • 2 different tool radii for 2-path programming • #CHANNEL INTERFACE [ESCAPE...]		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Possible solutions are: • Change the NC programming • Add the EDM license.
Error type	1, Error message from NC program.		

ID 22230

No license for use of extended interpolation.			
Description	The license for extended interpolation is missing. The Taperlink functionality is part of the license pack.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Possible solutions are: • Change the NC programming • Add the license for extended interpolation
Error type	1, Error message from NC program.		

ID 22231

No license for use of online adaptation			
Description	The license for online adaptation is missing. The TcCOM objects TRC, GFA and DCC are part of the license pack.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Possible solutions are: • Change the NC programming • Add the license for online adaptation
Error type	1, Error message from NC program.		

ID 22233

Invalid pose status programmed.			
Description	The programmed pose status (IS) is not permitted for the active kinematic with #WCS TO MCS. See also [PROG// Status & Turn (IS, IT)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program; program a permitted pose status.

Parameter	%1:	Error value [-]
		Active kinematic ID
	%2:	Error value [-]
		Programmed pose status
Error type	1, Error message from NC program.	

ID 22234

No pose status programmed.			
Description	The active kinematic requires a pose status but none was programmed. See also [PROG// Status & Turn (IS, IT)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program; program a permitted pose status.
Parameter	%1:	Error value [-]	
		Active kinematic ID	
Error type	1, Error message from NC program.		

ID 22235

Couple kinematics: Member kinematics with programmed kinematics ID not found.			
Description	The programmed kinematic ID is not member of the active couple kinematic. For more information on couple kinematics, see [FCT-C35]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Program an existing kinematic ID
Parameter	%1:	Error value [-]	
		Programmed kinematic ID	
Error type	1, Error message from NC program.		

ID 22236

IS programming for active kinematics ID not allowed.			
Description	The active kinematic ID has no multiple solutions and can therefore not be programmed with IS. See also [PROG// Status & Turn (IS, IT)]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; remove the IS expression or enable a different kinematic ID.
Parameter	%1:	Error value [-]	
		Active kinematic ID.	
Error type	1, Error message from NC program.		

ID 22237

Multiple IS programming only for kinematics type 210 allowed.

Description	Multiple IS programming is only allowed for couple kinematics (kinematics of type 210). For more information on couple kinematics, see [FCT-C35]		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Remove the multiple IS expression or enable the couple kinematic.
Parameter	%1:	Error value [-]	
		Kinematic type	
Error type	1, Error message from NC program.		

ID 22241

Command only allowed during active transformation.			
Description	A #SINGULARITY AVOIDANCE command was programmed although no kinematic transformation is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Activate a transformation with #TRAFO ON [..] before using #SINGULARITY AVOIDANCE commands.
Error type	1, Error message from NC program.		

ID 22242

Programmed and active kinematic IDs are not identical.			
Description	With simple standard kinematics, the kinematic ID can be optionally specified when singularity monitoring is selected or deselected with #SINGULARITY AVOIDANCE ON/OFF. In this case, the kinematic ID must be identical with the ID of the currently active transformation. The kinematic ID need only be specified for #SINGULARITY AVOIDANCE ON/OFF with transformations that consist of couple kinematics. Here, the programmed ID addresses the ID of a subordinate member kinematic of the active transformation.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. There are 2 options for this: • Use the identical kinematic ID in #SINGULARITY AVOIDANCE ON/OFF as for the currently active transformation • Remove the kinematic ID specified. The ID of the currently active transformation is implicitly adopted (not allowed for couple kinematics).
Parameter	%1:	Current value [-]	
		Programmed kinematic ID	
	%2:	Current value [-]	
		Kinematic ID of the currently active transformation	
Error type	1, Error message from NC program.		

ID 22243

Start-up parameter P-STUP-00029 only useful with active external tool management P-CHAN-00016.	
Description	Only activate the start-up parameter P-STUP-00029 (common_d_t_request) with an active external tool management system (P-CHAN-00016- ext_wzv_vorhanden).

Response	Class	1	Warning and correction of parameter. P-CHAN-00016 is set to 1.
Solution	Class	1	Check and correct P-CHAN-00016
Parameter	%1:	Error value [-]	
		Value of P-CHAN-00016	
	%2:	Corrected value [-]	
		New value for P-CHAN-00016	
Error type	-1, Error message from NC program.		

ID 22244

Memory for V.I. variables not configured.			
Description	In order to use V.I. variables, the memory and the number of variables must be parameterised. The memory is parameterised using P-STUP-00183 and the number of variables is specified in P-STUP-00184.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and correct the start-up list. P-STUP-00183 and P-STUP-00184 must be configured.
Error type	-		

ID 22245

Number of filter cycles for thread cutting with actual speed is out of range of permissible value.			
Description	The actual spindle speed may be noisy under certain circumstances. Values are smoothed by using this filter. The number of cycles that are filtered can be defined in the NC program using the NC command `#TURN[THREAD_CUT_N_CYCLES=..]` . The specified number of cycles is outside of the permissible value range.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Correct the value for the number of filter cycles.
Parameter	%1:	Error value [-]	
		Programmed value for the number.	
	%2:	Lower limit value [-]	
		Minimum value for the number.	
	%3:	Error value [-]	
		Maximum value for the number.	
Error type	1, Error message from NC program.		

ID 22246

Number of filter cycles for thread cutting with actual speed is out of range of permissible value.

Description	The actual spindle speed may be noisy under certain circumstances. Values are smoothed by using this filter. The number of cycles under review is defined in P-CHAN-00835. The parameterised value for P-CHAN-00835 is greater than the permissible maximum value. The value is limited to the maximum.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify the parameters in P-CHAN-00835
Parameter	%1:	Error value [-]	
		Parameterised value for P-CHAN-00835	
	%2:	Upper limit value [-]	
		Upper limit for P-CHAN-00835	
	%3:	Corrected value [-]	
		Corrected value for P-CHAN-00835	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22247

P-STUP-00029 is active but an incompatible external tool management is used.			
Description	The start-up parameter P-STUP-00029 (common_d_t_request) is active and an external tool management system that is too old and non compatible is used. The system detects this by the invalid response to the initial handshake between the CNC and the external tool management system.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify processing and supply of requested data in external tool management. Or deactivate P-STUP-00029.
Error type	1, Error message from NC program.		

ID 22248

Incompatible interface version received.			
Description	During the initial handshake between the CNC and the external tool management system, the interface version of the two participants is checked. The requested interface version of the external tool management system does not match the expected interface version of the CNC.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify processing and supply of requested data in external tool management. Update the external tool management system.
Parameter	%1:	Expected value[-]	
		Own and expected interface version	
	%2:	Error value[-]	
		Received interface version of the external tool management system	
Error type	1, Error message from NC program.		

2.3.10 ID range 22250-22499

ID 22250

Number of programmed microjoints in NC block is greater than permissible maximum.			
Description	The number of microjoints that may be programmed in an NC block is limited. The permitted number was exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Reduce the number of microjoints per line.
Parameter	%1:	Current value [-]	
		Programmed number of microjoints in NC block	
	%2:	Upper limit value [-]	
		Maximum permitted number of microjoints in NC block	
	%3:	Block number [-]	
		Block number of affected NC line	
Error type	1, Error message from NC program.		

ID 22253

#CANG cannot be programmed with an explicit endpoint.			

Description	A circle may be either programmed by an angle and the centre point or by a centre point and the circle end point. The combination of an angle and a circle end point is not allowed.
	
	Examples of impermissible programmings:
	`N10 G00 X2 Y20 F500`
	`N20 I0 J-20 #CANG cannot be programmed with an explicit endpoint.`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Either program by specifying an angle #CANG or change to specifying a circle end point.
Error type	1, Error message from NC program.		

ID 22254

Invalid symbol after MACRO_LENGTH.			
Description	The parameter of the MACRO_LENGTH function must be the identifier of a macro. Accordingly, an inverted comma (“) must be programmed directly after the opening square bracket. (see Arithmetic expressions <expr>)		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the notation of the macro name in the square brackets. It must be in inverted commas (“). Then restart the NC program.

Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Error value [-]
	%4:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 22255

Slave axis does not exist for the PCS axis coupling in the channel.			
Description	The axis with the logical axis number specified in the transformation parameter **trafo_pcs[i].trafo[i].param[1]** for the PCS axis coupling does not exist in the channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program or the configuration. Request the required axis in the NC program, or modify the transformation parameters.
Error type	1, Error message from NC program.		

ID 22256

Master axis does not exist for the PCS axis coupling in the channel.			
Description	The axis with the logical axis number specified in the transformation parameter **trafo_pcs[i].trafo[i].param[0]** for the PCS axis coupling does not exist in the channel.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program or the configuration. Request the required axis in the NC program, or modify the transformation parameters.
Error type	1, Error message from NC program.		

ID 22257

Axis exchange not allowed during active PCS transformation.

Description	While a PCS transformation (#TRAFO PCS ON) is active, no axis exchange commands may be used in the NC program.
	Example with error:
	`N10 G00 X0 Y0 Z0`
	`N20 #TRAFO PCS ON`
	`N30 #PUT AX[X]`
	`; …`
	`N100 #TRAFO PCS OFF`
	`; …`
	`N1000 M30`
	Corrected example:
	`N10 G00 X0 Y0 Z0`
	`N20 #TRAFO PCS ON`
	`:`
	`N100 #TRAFO PCS OFF`
	`N110 #PUT AX[X]`
	`:`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program; change the command sequence.
Error type	1, Error message from NC program.		

ID 22258

Tool dynamic data: Maximum acceleration is zero.			
Description	The received tool data contain relevant dynamic limits for velocity and acceleration. When this data was adopted, the system detected that the value zero is zero for acceleration P-TOOL-00015.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the assignment of the corresponding tool data., Enter a reasonable value for acceleration.
Parameter	%1:	Error value [mm/s² or °/s²]	
Error type	1, Error message from NC program.		

ID 22261

Couple kinematics: Tool not found.			
Description	When the transformation is selected with #TRAFO ON the tool belonging to the couple kinematic was not found. Possible cause: missing or incomplete definition of the couple kinematic groups (cf. P-CHAN-00447)		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the couple kinematic in the channel parameter list
Parameter	%1:	Current value [-]	
		Kinematic ID	
	%2:	Current value [-]	
		Kinematic type	
	%3:	Current value [-]	

Error type	1, Error message from NC program.

ID 22262

#SYNC IN with selected coordinate system is not allowed.			

Description	When the Conveyor tracking [FCT-M4] function was used, synchronisation was programmed using the #SYNC IN command with an active coordinate system. This is not permitted.
	Example with error:
	`N010 #CS DEF [1][1,15,5,20,30,45] (CS definition)`
	`N020 #CS ON[1]  (enable the CS with ID 1)`
	`N030 #SYNC IN`
	`; …`
	`N100 #CS OFF  (deselect the CS )`
	`N110 #SYNC OFF`
	`; …`
	Corrected example:
	`N010 #CS DEF [1][1,15,5,20,30,45] (CS definition)`
	`N020 #SYNC IN`
	`N030 #CS ON[1]  (enable the CS with ID 1)`
	`; …`
	`N100 #CS OFF  (deselect the CS )`
	`N110 #SYNC OFF`
	`; …`
	Alternative corrected example:
	`N010 #CS DEF [1][1,15,5,20,30,45] (CS definition)`
	`N020 #CS ON[1]  (enable the CS with ID 1)`
	`; …`
	`N030 #CS OFF  (deselect the CS )`
	`N040 #SYNC IN`
	`;…`
	`N110 #SYNC OFF`
	`; …`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; change the command sequence.
Error type	1, Error message from NC program.		

ID 22263

#SYNC OUT with selected coordinate system is not allowed.

Description	When the Conveyor tracking [FCT-M4] function was used, desynchronisation was programmed using the #SYNC OUT command with an active coordinate system. This is not permitted.
	Example with error:
	`N010 #CS DEF [1][1,15,5,20,30,45] (CS definition)`
	`N020 #SYNC IN`
	`N030 #CS ON[1]  (enable the CS with ID 1)`
	`; …`
	`N090 #SYNC OFF`
	`N100 #CS OFF` `(deselect the CS )`
	`; …`
	
	Corrected example:
	`N010 #CS DEF [1][1,15,5,20,30,45] (CS definition)`
	`N020 #SYNC IN`
	`N030 #CS ON[1]  (enable the CS with ID 1)`
	`; …`
	`N090 #CS OFF` `(deselect the CS )`
	`N100 #SYNC OFF`
	`; …`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; change the command sequence.
Error type	1, Error message from NC program.		

ID 22264

The selection of a kinematic transformation is not permitted when conveyor synchronisation is active.			

Description	While the conveyor tracking function is active, a transformation may neither be selected nor deselected between selection with #SYNC IN and deselection with #SYNC OUT.
	Program example with error with deselection of transformation:
	`N010 #KIN ID[37]`
	`N020 G0 G90 X34.9485 Y-18.0290 Z34.0535`
	`N020 #TRAFO ON`
	`;…`
	`N070 #SYNC IN [CONVEYOR …`
	`;…`
	`N110 #TRAFO OFF (Error ID 22264)`
	`N100 #SYNC OUT [FEED_CONT=5]`
	`;...`
	
	Corrected example:
	`N010 #KIN ID[37]`
	`N020 G0 G90 X34.9485 Y-18.0290 Z34.0535`
	`N020 #TRAFO ON`
	`;…`
	`N070 #SYNC IN [CONVEYOR …`
	`;…`
	`N100 #SYNC OUT [FEED_CONT=5]`
	`N110 #TRAFO OFF`
	`;…`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; relocate the selection or deselection of the transformation in the program.
Error type	1, Error message from NC program.		

ID 22265

Conveyor synchronisation is already active. Renewed call not allowed.			

Description	The ‘Conveyor Tracking’ [FCT-M4] function was already activated with the #SYNC IN command.
	It is not allowed to program #SYNC IN again without a previous #SYNC OUT.
	Example with error:
	`N020 #SYNC IN[…]`
	`; …`
	`N40 #SYNC IN[… ] (Error ID 22265)`
	`; …`
	`N090 #SYNC OFF`
	Corrected example:
	`N020 #SYNC IN`
	`; …`
	`N090 #SYNC OFF`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; remove one of the #SYNC IN commands
Error type	1, Error message from NC program.		

ID 22266

Conveyor synchronisation is not active. Deselection not permitted.			
Description	The ‘Conveyor Tracking’ [FCT-M4] function is not selected. It is not allowed to program #SYNC OUT without previous synchronisation with #SYNC IN. It is not possible to deselect using the #SYNC OUT command.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; remove the #SYNC OUT command.
Error type	1, Error message from NC program.		

ID 22267

A change in slope type is not permitted with an active ‘Conveyor Tracking’ function.			
Description	When the ‘Conveyor Tracking’ [FCT-M4] function is active, it is not allowed to change the slope type using the #SLOPE NC command. It is not possible to program the #SLOPE NC command between #SYNC IN and #SYNC OUT.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and modify the NC program; remove the # SLOPE command.
Error type	1, Error message from NC program.		

ID 22269

Couple kinematics: Group has too many redundant degrees of freedom.			
Description	A group of couple kinematics has too many redundant degrees of freedom for a single TCP coordinate.		
Response	Class	2	Abort NC program processing.

Solution	Class	3	Adapt the kinematic configuration
Parameter	%1:	Error value [-]	
		Index of the incorrect group	
	%2:	Error value [-]	
		Axis index of the associated TCP coordinate	
	%3:	Upper limit value [-]	
		Maximum number of redundant degrees of freedom for a group	
Error type	1, Error message from NC program.		

ID 22272

Timeout during drive request.			
Description	The permissible response time was exceeded for a read or write access to the drive via (#IDENT WR/ #IDENT RD). The permitted duration is 10 seconds.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check whether the drive has a problem.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of drive affected	
	%2:	Identification number [-]	
		Number of the object that received no response. (decimal notation)	
	%3:	Current value [-]	
		Return value 1 of the service channel	
	%4:	Current value [-]	
		Return value 2 of the service channel	
Error type	1, Error message from NC program.		

ID 22273

Queue for pending drive requests is already full.			
Description	The maximum number of possible drive requests is dependent on the configuration. This number was exceeded. Possible causes: • The drive requires longer than expected to send the responses. • Too many drive requests were programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Reduce the number of drive requests
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of drive affected	
	%2:	Identification number [-]	
		Number of object. (decimal notation)	
	%3:	Current value [-]	
		Return value 1 of the service channel	
	%4:	Current value [-]	
		Return value 2 of the service channel	
Error type	1, Error message from NC program.		

ID 22274

Invalid NC block with active axis coupling.			

Description	There are restrictions in the scope of the NC blocks when a transformation is active with an axis coupling (#TRAFO PCS [212]):
	This includes motions in the slave axis or changes to the coordinates, e.g. via #CS or #TRAFO ON.
	Example with error:
	`N10 G00 X0 Y0 Z0`
	`N20 #TRAFO PCS [212]`
	`…`
	`N100 #CS…/ #TRAFO ON`
	`:…`
	`N1000 M30`
	Corrected example:
	`N10 G00 X0 Y0 Z0`
	`N20 #TRAFO PCS ON`
	`:`
	`N100 #TRAFO PCS OFF`
	`N110 #CS…/ #TRAFO ON …`
	`:`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the flow in the NC program
Error type	1, Error message from NC program.		

ID 22275

Invalid symbol in MACRO_CONTENT.			
Description	An invalid character was programmed in the mathematical function MACRO_CONTENT		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. The parameter of the MACRO_CONTENT function must be the macro identifier. Accordingly, an inverted comma (“) must be programmed directly after the opening square bracket.
Error type	1, Error message from NC program.		

ID 22276

Could not resolve macro in MACRO_CONTENT.			
Description	The macro in the mathematical function MACRO_CONTENT could not be resolved. This may have several reasons: • The macro does not exist. • The macro recursion depth is too large. • The resulting string is too long.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check macro for existence, recursion depth and string length and change as necessary.
Error type	1, Error message from NC program.		

ID 22277

The specified oscillation positions are not within the modulo range.			
Description	If the keyword SHORT is assigned OSC in the programming of an oscillating motion, both oscillation positions must be located within the modulo range. This is not the case.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the positions 1ST_POS and 2ND_POS in the NC command.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Currently specified incorrect position.	
	%2:	Upper limit value[0.1 10^-3 mm or ø]	
		Upper modulo limit	
	%3:	Lower limit value [0.1 10^-3 mm or ø]	
		Lower modulo limit	
Error type	1, Error message from NC program.		

ID 22278

The specified key for the blank definition does not exist.			

Description	The #SCENE command defined a blank.
	The BLANK_VALUE_OF_KEY[..] function reads the related value using the specified key.
	Example:
	`;Initial situation:`
	`SCENE ADD [GOBJECT=test FILE=test.wrl \`
	`LINKPOINT=GROUND \`
	`KEY[0]="TYPE" VAL[0]="CUBE" \`
	`KEY[1]="LENGTH" VAL[1]=10 \`
	`X=0 Y=0 Z=770 BLANK]`
	OK case:
	`V.L.BLANK_TYPE = BLANK_VALUE_OF_KEY["TYPE"]`
	`;-> Function returns "CUBE"`
	Error case:
	`V.L.BLANK_TYPE = BLANK_VALUE_OF_KEY["UNDEFINED"]`
	`;-> Error 22278`
	The specified key does not exist.		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the created blank and its data. Correct the NC program.
Parameter	%1:	Error value [-]	
		Incorrectly specified key	
Error type	1, Error message from NC program.		

ID 22279

Robot has negative mass.	
Description	The mass of the robot was defined in P-CHAN-00774 when the load model was parameterised. A negative mass was specified. When mass 0 is specified, it means that the load model is not calculated.

Response	Class	1	Warning and correction of parameter. No influence of the robot on the load model
Solution	Class	1	Check and modify P-CHAN-00774.
Parameter	%1:	Error value [kg]	
		Parameterised value of P-CHAN-00774	
	%2:	Lower limit value [kg]	
		The robot mass is corrected to the lower limit.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22280

Negative mass of linear mover.			
Description	The mass of the linear mover was defined in P-CHAN-00773 when the load model was parameterised. A negative mass was specified. When mass 0 is specified, it means that the load model is not calculated.		
Response	Class	1	Warning and correction of parameter. No influence of the linear mover on the load model.
Solution	Class	1	Check and modify P-CHAN-00773.
Parameter	%1:	Error value [kg]	
		Parameterised value of P-CHAN-00773	
	%2:	Lower limit value [kg]	
		The linear mover mass is corrected to the lower limit.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22281

Axis for pcs angle projection not present in channel.			
Description	The PCS angle projection (kinematic type 214) specifies the logical axis number for the projection in the transformation parameter trafo_pcs[i].trafo[i].param[1]. This axis does not exist in the NC channel.		
Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	3	Possible solutions: • Change the NC program; request the corresponding axis • Change the parameters in the PCS transformation
Parameter	%1:	Error value [-]	
		Parameterised value for the logical axis number	
Error type	1, Error message from NC program.		

ID 22282

Change of PCS ID not allowed during active PCS transformation.

Description	A change of PCS ID using #TRAFO PCS ID[...] may not be programmed while the PCS transformation is active.
	
	Example with error:
	`N10 G00 X0 Y0 Z0`
	`N20 #TRAFO PCS ID[1]`
	`N30 #TRAFO PCS ON`
	`N40 #TRAFO PCS ID [2]`
	`;…`
	`N100 #TRAFO PCS OFF`
	`;…`
	`N1000 M30`
	
	Example:
	`N10 G00 X0 Y0 Z0`
	`N20 #TRAFO PCS ID[1]`
	`N30 #TRAFO PCS ON`
	`;…`
	`N100 #TRAFO PCS OFF`
	`N110 #TRAFO PCS ID[2]`
	`;…`
	`N1000 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. #TRAFO PCS ID[...] may only be programmed when the kinematic transformation is deselected (#TRAFO PCS OFF).
Error type	1, Error message from NC program.		

ID 22283

Set dynamic model id could not be found.			
Description	The dynamic model ID (P-CHAN-00868) specified in the kinematic data does not exist. The dynamic model cannot be calculated without parameters.		
Response	Class	1	Dynamic model is deactivated.
Solution	Class	1	Check and correct the entered dynamic model ID (P-CHAN-00868).
Parameter	%1:	Current value [-]	
		Parameterised kinematic ID in the dynamic model	
	%2:	Current value [-]	
		Incorrect dynamic model ID (P-CHAN-00868)	
Error type	1, Error message from NC program.		

ID 22284

The set dynamic model is not fully parameterised.			
Description	A kinematic was specified with a reference to a dynamic model ID (P-CHAN-00868). The specified dynamic model is not completely parameterised.		
Response	Class	1	Warning output. Dynamic model is not active.
Solution	Class	1	Check and modify the parameters of the dynamic model. A mass entry with 0 is allowed.
Parameter	%1:	Current value [-]	
		Parameterised kinematic ID in the dynamic model	
	%2:	Current value [-]	
		Dynamic model ID of the kinematic P-CHAN-00868	

Error type	1, Error message from NC program.

ID 22285

The ID of the dynamic model is not unique.			
Description	Each parameter block of the dynamic model requires a unique ID (P-CHAN-00868) to link it to a kinematic. Multiple models were specified with this ID.		
Response	Class	1	Dynamic model is deactivated.
Solution	Class	1	Check and correct the entered ID (P-CHAN-00868).
Parameter	%1:	Current value [-]	
		Parameterised kinematic ID in the dynamic model	
	%2:	Current value [-]	
		Incorrect dynamic model ID (P-CHAN-00868)	
Error type	1, Error message from NC program.		

ID 22286

The dynamic model can only be used with one couple kinematic group at once.			
Description	The dynamic model can only be calculated if the member kinematics in a group (trafo[i].group[j]) are each part of the dynamic model and have a dynamic model ID (P-CHAN-00868) specified in the transformation data. For more information on couple kinematics, see [FCT-C35]		
Response	Class	1	Dynamic model is deactivated.
Solution	Class	1	Check the parameterisation. The dynamic model can only be calculated for a couple kinematic group.
Parameter	%1:	Current value [-]	
		Kinematic ID of the dynamic model	
	%2:	Current value [-]	
		Group ID of the second couple kinematic group	
Error type	1, Error message from NC program.		

ID 22287

Kinematic type not allowed on PCS level.			
Description	An invalid kinematic type was specified when the PCS transformation was used. The PCS transformation type is defined in the channel parameter P-CHAN-00829. The PCS transformation is selected by the NC command #TRAFO PCS ID[<id>]. The ID is assigned by the channel parameter P-CHAN-00262. Permitted types include McCOM transformations [500; 999] or configured PCS transformations.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and correct the parameters.
Parameter	%1:	Error value [-]	
		Specified kinematic type on PCS level.	
Error type	1, Error message from NC program.		

ID 22288

Dynamic model is set but no kinematic is set.			
Description	The dynamic model was activated (P-CHAN-00867) but no kinematic ID (P-CHAN-00871) is specified.		
Response	Class	1	Dynamic model is deactivated.
Solution	Class	1	Check and modify the parameterisation. Specify the kinematic ID (P-CHAN-00871) which is to be used to calculate the dynamic model.
Parameter	%1:	Error value [-]	
		Kinematic ID of the dynamic model	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22303

Kinematic does not support programming of IS without IT.			
Description	The currently active kinematic was programmed with Status (IS) without the use of Turn (IT). However, when Status&Turn is used, the kinematic must be programmed with the two values.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Specify the associated Turn value (IT) or program without Status&Turn
Error type	1, Error message from NC program.		

ID 22306

Index of V.A.RETRACT_ACS_POS[i] is out of permissible value range.			
Description	The axis variables V.A.RETRACT_ACS_POS are only valid for the axes on the first 6 axis indices. The programmed axis is on an axis index above the maximum permitted index.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; use a permitted axis index.
Parameter	%1:	Error value [-]	
		Axis index of programmed axis	
	%2:	Upper limit value [-]	
		Maximum permitted axis index	
Error type	1, Error message from NC program.		

ID 22307

3D distance control function is not configured.	
Description	The 3D distance control function was programmed but it was not configured in the channel parameter list using P-CHAN-00500. `configuration.decoder.function  FCT_3D_DIST_CTRL` The 3D distance control function is programmed using the NC command #DIST CTRL. See also Programming 3D distance control with #DIST CTRL command.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Parameterise 3D distance control using P-CHAN-00500.
Parameter	%1:	Current value [-]	
		Name of programmed function to be configured	
Error type	1, Error message from NC program.		

ID 22310

Given kinematic ID does not exist.			
Description	The kinematic ID (P-CHAN-00262) specified in the #LIMIT command is not defined in the channel parameters or is specified incorrectly.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the specified kinematic ID
Error type	1, Error message from NC program.		

ID 22311

The specified couple kinematic has too many member kinematics.			
Description	The couple kinematic [FCT-C35] specified in the #LIMIT command has more than 3 member kinematics and therefore not permissible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the kinematic configuration.
Error type	1, Error message from NC program.		

ID 22312

A member kinematic of the set couple kinematic does not exist.			
Description	A couple kinematic [FCT-C35] was specified in the #LIMIT command.. A member kinematic of this couple kinematic does not exist.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the kinematic configuration.
Error type	1, Error message from NC program.		

ID 22313

Negative mass of linear mover.			
Description	The load model that is to be activated with #LIMIT is not correctly parameterised. The mass of the linear unit must be positive.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the configuration of the load model and correct the mass of the linear unit.
Error type	1, Error message from NC program.		

ID 22314

Robot mass is negative.			
Description	The load model that is to be activated with #LIMIT is not correctly parameterised. The mass of the robot must be positive.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the configuration of the load model and correct the mass of the robot.
Error type	1, Error message from NC program.		

ID 22315

No kinematic ID is specified.			
Description	When the #LIMIT on command is used, a kinematic ID must be specified. This affects • load model • dynamic model • and also TCP velocity monitoring		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC program. and correct the #LIMIT command.
Error type	1, Error message from NC program.		

ID 22316

Given dynamic model ID could not be found.			
Description	When the #LIMIT ON command was used for the kinematic, a valid dynamic model (P-CHAN-00868) must be specified for the kinematic.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the dynamic model and the kinematic.
Error type	1, Error message from NC program.		

ID 22317

The specified dynamic model is not fully parameterised.			
Description	The dynamic model that is to be activated with #LIMIT ON is not complete.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the dynamic model.
Error type	1, Error message from NC program.		

ID 22318

The ID of the dynamic model is not unique.	
Description	When the dynamic model was activated with #LIMIT ON, a kinematic ID was specified that refers to a dynamic model. The ID of the dynamic model (P-CHAN-00868) is not unique. Multiple parameter sets of dynamic models are assigned the same ID.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the dynamic model.
Error type	1, Error message from NC program.		

ID 22319

The dynamic model can only be used with one couple kinematic group at once.			
Description	The #LIMIT command specifies a couple kinematic [FCT-C35] that is incorrectly parameterised for the dynamic model. A dynamic model may only be specified within a group of couple kinematics.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the couple kinematic.
Error type	1, Error message from NC program.		

ID 22320

A member kinematic of the couple kinematic set for the dynamic model does not exist.			
Description	The #LIMIT command specifies a couple kinematic [FCT-C35] that is incorrectly parameterised. A member kinematic that does not exist was specified.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the couple kinematic.
Error type	1, Error message from NC program.		

ID 22321

The referenced kinematic is not supported by the dynamic model.			
Description	No dynamic model exists for the kinematic specified in #LIMIT.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the configuration and the NC program.
Error type	1, Error message from NC program.		

ID 22322

The number of axes in the kinematic referenced for the dynamic model exceeds the permissible maximum.			
Description	The number of possible axes for the dynamic model is limited to 8 axes. The number of axes of the referenced kinematic is too large.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the configuration and the NC program.
Error type	1, Error message from NC program.		

ID 22323

The specified velocity for the kinematic velocity limit is too low.

Description	The maximum velocity of the TCP velocity limit [FCT-C47] activated by #LIMIT ON is set too low.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the TCP velocity limit.
Error type	1, Error message from NC program.		

ID 22324

The specified velocity weighting of the tcp velocity limit is too low.			
Description	The velocity weighting of the TCP velocity limit [FCT-C47] activated by #LIMIT ON is set too low.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the TCP velocity limit.
Error type	1, Error message from NC program.		

ID 22325

The specified velocity weighting of the tcp velocity limit is too high.			
Description	The velocity weighting of the TCP velocity limit [FCT-C47] activated by #LIMIT ON is set too high..		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the TCP velocity limit.
Error type	1, Error message from NC program.		

ID 22326

Not enough axes in channel available for the calculation of the dynamic model.			
Description	The NC channel has not enough axes to enable the dynamic model at program start.		
Response	Class	1	Output of warning; the dynamic model is deactivated.
Solution	Class	1	Check the configuration of the NC program.
Error type	1, Error message from NC program.		

ID 22327

Not enough axes in channel available for the calculation of the dynamic model.			
Description	The NC channel has not enough axes to activate the dynamic model at program start with #LIMIT ON.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC program and the configuration of the NC channel.
Error type	1, Error message from NC program.		

ID 22328

Axis exchange not allowed while dynamic model is active.

Description	It is not allowed to use NC commands for axis exchange while the dynamic model is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deactivate the dynamic model before axis exchange.
Error type	1, Error message from NC program.		

ID 22329

Axis exchange not allowed while load model is active.			
Description	It is not allowed to use NC commands for axis exchange while the load model is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deactivate the dynamic model before axis exchange.
Error type	1, Error message from NC program.		

ID 22330

Axis exchange not allowed while tcp velocity limit is active.			
Description	It is not allowed to use NC commands for axis exchange while the tcp velocity limit is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deactivate the TCP velocity limit before axis exchange.
Error type	1, Error message from NC program.		

ID 22331

Load and dynamic model can only be used exclusively.			
Description	All the load model functions are available even when the dynamic model is used. Combined use is therefore unnecessary and at the same time impossible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deactivate the load model so that only the dynamic model can be used.
Error type	1, Error message from NC program.		

ID 22332

The number of the interpolation points is out of permissible value range.			
Description	The number of the parameterised interpolation points for the counter balancing system is specified in P-CHAN-00872. The specified value is out of the permissible value range.		
Response	Class	1	Warning output. The dynamic model is deactivated.
Solution	Class	1	Check and modify P-CHAN-00872

Parameter	%1:	Current value [-]
		Incorrect number of interpolation points
	%2:	Lower limit value [-]
		Minimum number of permitted interpolation points
	%3:	Upper limit value [-]
		Maximum number of permitted interpolation points
Error type	2, Error message by data transfer from parameter list into control device.	

ID 22333

The number of interpolation points is zero.			
Description	The number of interpolation points for the counter balancing system is zero but interpolation points are defined.		
Response	Class	1	Warning output. Dynamic model is deactivated.
Solution	Class	1	Check and modify P-CHAN-00872
Parameter	%1:	Current value [-]	
		Incorrect number of interpolation points	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22334

The dynamic model is deactivated.			
Description	The dynamic model was deactivated since the anonymous parameters of the counter balancing system (P-CHAN-00875) were changed.		
Response	Class	1	Warning output. The dynamic model is deactivated.
Solution	Class	1	Check the assignment of P-CHAN-00875.
Parameter	%1:	Current value [-]	
		Index of the incorrect anonymous parameter	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22335

The specified axis positions of the counter balancing system are not unique.			
Description	The parameterised axis positions (P-CHAN-00876) for the counter balancing system are not unique.		
Response	Class	1	Warning output. The dynamic model is deactivated.
Solution	Class	1	Check and modify the entries in P-CHAN-00876
Parameter	%1:	Current value [-]	
		Index of the incorrect axis position	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22336

The number of the interpolation points is out of permissible value range.

Description	The counter balancing system parameterised in the dynamic model is incorrect. A maximum of 4 points can be specified in P-CHAN-00872 to parameterise the counter balancing system.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the configuration of the dynamic model; correct the parameters in P-CHAN-00872.
Parameter	%1:	Current value [-]	
		Specified value for	
	%2:	Lower limit value [-]	
		Minimum number of points	
	%3:	Upper limit value [-]	
		Maximum number of points	
Error type	1, Error message from NC program.		

ID 22337

The number of interpolation points is zero.			
Description	The counter balancing system parameterised in the dynamic model is incorrect. A maximum of 4 points can be specified in P-CHAN-00872 to parameterise the counter balancing system.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the configuration of the dynamic model; correct the parameters in P-CHAN-00872.
Parameter	%1:	Current value [-]	
		Current value of P-CHAN-00872.	
Error type	1, Error message from NC program.		

ID 22338

The dynamic model is not valid.			
Description	The parameters of the dynamic model are incorrect.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the dynamic model.
Error type	1, Error message from NC program.		

ID 22339

The specified axis positions of the counter balancing system are not unique.			
Description	The counter balancing system parameterised in the dynamic model is incorrect. The specified axis positions (P-CHAN-00876) must be unique.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the dynamic model; correct the values for P-CHAN-00876.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 22340

Double programming for dynamic collision monitoring.			
Description	A parameter was programmed multiple times when programming dynamic collision monitoring with the NC command #COLL MON ON[…].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program; remove the multiple entry.
Error type	1, Error message from NC program.		

ID 22341

#ESCAPE PATH definition has to be programmed before #TRACK CHAN ON[ESCAPE...].			
Description	Before the channel interface is activated in the escape channel with #TRACK CHAN ON [ESCAPE …], the definition of path erosion must be programmed with #ESCAPE PATH DEF BEGIN and #ESCAPE PATH DEF END.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program for the escape channel.
Error type	1, Error message from NC program.		

ID 22342

At start of #ESCAPE PATH definition coordinate system must be inactive.			
Description	When defining the path erosion, no coordinate system may be programmed in the escape channel before the NC command #ESCAPE PATH DEF BEGIN. In order to use a coordinate system in the path erosion definition, it can be programmed directly after #ESCAPE PATH DEF BEGIN.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program so that no coordinate system is active.
Error type	1, Error message from NC program.		

ID 22343

#ESCAPE PATH definition has to be programmed before #TRACK CHAN ON[ESCAPE...].			
Description	When defining the path erosion, no coordinate system may be active in the escape channel at the end of the definition before the NC command #ESCAPE PATH DEF END. The coordinate system previously selected in the path erosion definition is stored in the escape channel and is reselected online when the escape request is active. A coordinate system that is still active would result in an incorrect coordinate system stack.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Deselect the coordinate system, e.g. using #CS DEL ALL before #ESCAPE PATH DEF END in the escape channel.
Error type	1, Error message from NC program.		

ID 22344

The channel number of the programmed V.I. server is unknown.			
Description	The logical channel number specified for the channel parameter in P-CHAN-00525 was not found. The value is corrected to the first channel.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-CHAN-00525.
Parameter	%1:	Error value [-]	
		Unknown channel number	
	%2:	Corrected value [-]	
		Value of first channel.	
Error type	-		

ID 22345

Only one default receiver can be configured for #MSG.			
Description	The parameter in P-CHAN-00878 defines the default receiver for the #MSG command. It is not allowed to link several receivers in the parameterisation.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-CHAN-00878.
Parameter	%1:	Corrected value [-]	
		Corrected receiver	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 22346

#SCENE has no effect without active P-STUP-00138 (enable_scene).			
Description	The use of #SCENE NC commands has no effect while P-STUP-00138 is inactive..		
Response	Class	1	Warning output; #SCENE commands have no effect
Solution	Class	1	Check and activate P-STUP-00138.
Error type	1, Error message from NC program.		

ID 22347

The active dynamic model is specified with a kinematic type that cannot be used with a dress pack.			
Description	The dynamic model is parameterised with a kinematic that does not allow a dress pack.		
Response	Class	1	Output of warning; the dynamic model is deactivated.
Solution	Class	1	Check and modify the parameterisation.

Parameter	%1:	Current value [-]
		Kinematic ID of the dynamic model (P-CHAN-00871). The value refers to the kinematic used by the dynamic model.
	%2:	Current value [-]
		ID of the dynamic model (P-CHAN-00866). This value refers to the kinematic of the dynamic model.
	%3:	Current value [-]
		Kinematic type (P-CHAN-00829); type of the kinematic selected by the dynamic model.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 22348

The active dynamic model is specified with a kinematic type that cannot be used with a counter balancing system.			
Description	The dynamic model is parameterised with a kinematic that does not allow a counter balancing system.		
Response	Class	1	Output of warning; the dynamic model is deactivated.
Solution	Class	1	Check and modify the parameterisation.
Parameter	%1:	Current value [-]	
		Kinematic ID of the dynamic model (P-CHAN-00871). The value refers to the kinematic used by the dynamic model.	
	%2:	Current value [-]	
		ID of the dynamic model (P-CHAN-00866). This value refers to the kinematic of the dynamic model.	
	%3:	Current value [-]	
		Kinematic type (P-CHAN-00829); type of the kinematic selected by the dynamic model.	
Error type	2, Error message by data transfer from parameter list into control device.		

2.4 Configuration error (ID-range 30000-39999)

2.4.1 ID-range 30000-30249

ID 30000 - 30013

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 30014

Heap for BF data engaged.			
Description	The requested memory is larger than the available memory in system.		
Response	Class	3	Job processing aborted
Solution	Class	7	Possible solutions: • Reduce number of channels or axes to reduce requested memory • Increase the computer's main memory.
Parameter	%1:	Current value [-]	
		Number of free bytes on the heap.	
	%2:	Expected value [-]	
		Expected number of free bytes on the heap.	
Error type	-		

ID 30015 - 30074

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

2.5 PLC function block error (ID-range 40000-49999)

2.5.1 ID-range 40000-40249

ID 40001

Axis specific MC error.			
Description	Due to an axis-specific error in the motion controller, the axis is in the ErrorStop state according to the state diagram defined in the PLCopen specification for axes [MCP-P1//PLCopenaxis state model]. Each instance of a FB with this axis assigned at the input “Axis” as axis reference, now displays this error number at its output pin “ErrorID”.		
Response	Class	-	The reaction of the motion controller depends on the concrete occurred error.
Solution	Class	-	To diagnose which specific axis error has occurred in the Motion Controller, the "AxisErrorID" output of an FB of type MC_ReadAxisError must be evaluated, which has this axis assigned at the "Axis" input as axis reference. The error can be reset by executing an instance of the MC_Reset for this axis.
Error type	-		

ID 40002

Axes group specific MC error.			
Description	Due to an error in the Motion Controller, the axis group is in the state **GroupErrorStop**, according to the state diagram defined in the PLCopen specification for axis groups [MCP-P4//Chapter: PLCopen axis group state model]. The FB whose job resulted in the error in the Motion Controller and applied this axis group as a reference to its “AxesGroup” input displays this error code at its “ErrorID” output pin and sets the “Error” output to TRUE.		
Response	Class	-	The reaction of the motion controller depends on the concrete occurred error.
Solution	Class	-	To diagnose which specific axis group error has occurred in the Motion Controller, the output "GroupErrorID" of an FB of type MC_GrpReadError must be evaluated, which has this axis group assigned at the input "AxesGroup" as axis group reference. The error can be reset by executing an instance of the MC_GrpReset for this group of axes.
Error type	-		

ID 40003

Platform specific MC error.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 40004

Copying of version string from HLI into PLC's string variable failed.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkHLIVer is called within the FB MCV_PlatformBase to check whether the version identifiers of the HLI interface of the PLC and motion controller match. The utility FB displays this error message if the version identifier of the motion controller cannot be copied to the PLC. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets sets its “Error” output to TRUE and the “Done” output to FALSE.
Solution	Class	-	Please contact the controller supplier and quote the error number.
Error type	-		

ID 40005

Different size of axis or channel specific HLI section of PLC and MC.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkHLIVer is called within the FB MCV_PlatformBase to check whether the size of the axis- and channel-specific areas on the HLI interface on the PLC side and the motion controller side match. The utility FB notifies the error message if the axis or channel area on the HLI or both are not the same size. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets sets its “Error” output to TRUE and the “Done” output to FALSE.
Solution	Class	-	Please contact the controller supplier and quote the error number.
Error type	-		

ID 40006

Different HLI version string of PLC and MC.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkHLIVer is called within the FB MCV_PlatformBase to check whether the version identifiers of the HLI interface on the PLC side and those of the motion controller match. The utility FB notifies the error message if the version identifiers of the PLC and the motion controller are different. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets sets its “Error” output to TRUE and the “Done” output to FALSE.

Solution	Class	-	In every PLC application that sends jobs to the motion controller via the HLI interface, there must be an instance of the FB MCV_HliInterface. At the outputs “ExpectedHli” and “ActualHli” of this instance, the version identifiers of the HLI interface can be checked on the PLC and the Motion Controller side. The output “ExpectedHli” indicates the version identifier of the PLC’s HLI interface and output “ActualHli” indicates the version identifier of the motion controller HLI interface. Then check whether a motion controller with the wrong version identifier was started or whether the PLC application included an HLI library with the wrong version identifier. In the first case please start the motion controller with the correct version identifier; in the second case please include the HLI library with the correct version identifier in the PLC application. If none of the above causes exists, contact the controller supplier and specify the version identifier at the outputs "ExpectedHli" and "ActualHli".
Error type	-		

Only the first 12 characters of the version identifier are checked and if they are equal that’s sufficient consistence of the HLI interface at PLC and motion controller to interact together.

ID 40007

Checking of HLI version string impossible, because interface is occupied.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkLCsHLIVer is called within the FB MCV_PlatformBase, with which the version ID of the HLI interface on the PLC side is transferred to the Motion Controller for checking. The utility FB notifies the error message if the PLC can not command a check of version identifier by the motion controller because at least one of the elements command_semaphor_rw or command_w of the control unit of type MC_CHECK_PLC_VERSION is already set to TRUE. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets its output “Error” to TRUE and the output “Done” to FALSE.
Solution	Class	-	Reset and logout the PLC application. Retransmit and start the PLC application during PLC runtime. If error occurs again, please check if the data of control unit of type MC_CHECK_PLC_VERSION is overwritten on the HLI by the PLC application or another PLC application accesses the HLI. If this fails to start the Motion Control Platform successfully, please contact the motion controller supplier.
Error type	-		

ID 40008

Error occurred while copying PLC version string to HLI.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkLCsHLIVer is called within the FB MCV_PlatformBase, with which the version ID of the HLI interface on the PLC side is transferred to the Motion Controller for checking. The utility FB notifies the error message if providing the version identifier to the Motion controller failed, because copy version identifier onto HLI returns with an error. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets sets its “Error” output to TRUE and the “Done” output to FALSE.
Solution	Class	-	Reset and logout the PLC application. Retransmit and start the PLC application during PLC runtime. If the error occurs again, contact the control supplier.
Error type	-		

ID 40009

Can't acknowledge check of PLC HLI version string.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkLCsHLIVer is called within the FB MCV_PlatformBase, with which the version ID of the HLI interface on the PLC side is transferred to the Motion Controller for checking. The utility FB notifies the error message if the check by the motion controller was successful, then a different result was obtained when the test results were requested again. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets its output “Error” to TRUE and the output “Done” to FALSE.
Solution	Class	-	Reset and logout the PLC application. Retransmit and start the PLC application during PLC runtime. If the error occurs again, please check whether the data on the control unit of type MC_CHECK_PLC_VERSION is overwritten on the HLI by the PLC application. If this fails to start the Motion Control Platform successfully, please contact the motion controller supplier.
Error type	-		

ID 40010

Not defined state of the state machine of an FB.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkLCsHLIVer is called within the FB MCV_PlatformBase. This FB is in a non implemented state. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets sets its “Error” output to TRUE and the “Done” output to FALSE.
Solution	Class	-	Please contact your controller supplier and quote the error number.
Error type	-		

ID 40011

Control unit mc_check_plc_version: state variable is FALSE.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_ChkLCsHLIVer is called within the FB MCV_PlatformBase, with which the version ID of the HLI interface on the PLC side is transferred to the Motion Controller for checking. The Utility FB supplies this error message if the version identifier check was commanded but the PLC and Motion Controller strings are different. At the output "ErrorID" of the FB MCV_PlatformBase this error message is made available for the PLC application.		
Response	Class	-	The FB MCV_PlatformBase sets sets its “Error” output to TRUE and the “Done” output to FALSE.
Solution	Class	-	In every PLC application that sends jobs to the motion controller via the HLI interface, there must be an instance of the FB MCV_HliInterface. At the outputs “ExpectedHli” and “ActualHli” of this instance, the version identifiers of the HLI interface can be checked on the PLC and the Motion Controller side. The output “ExpectedHli” indicates the version identifier of the PLC’s HLI interface and output “ActualHli” indicates the version identifier of the motion controller HLI interface. Then check whether a motion controller with the wrong version identifier was started or whether the PLC application included an HLI library with the wrong version identifier. In the first case please start the motion controller with the correct version identifier; in the second case please include the HLI library with the correct version identifier in the PLC application. If none of the above causes exists, contact the controller supplier and specify the version identifier at the outputs "ExpectedHli" and "ActualHli".
Error type	-		

Only the first 12 characters of the version identifier are checked and if they are equal that's sufficient consistence of the HLI interface at PLC and motion controller to interact together.

ID 40014

Pointer to platform specific HLI section is 0.			
Description	When the HLI interface is activated by an instance of the FB MCV_HliInterface, global pointer variables are initialised. The implemented FBs can then access the various HLI areas. To ensure the pointers are valid they are checked. The check detects that the global pointer variable gpPform is invalid because its value is 0.		
Response	Class	-	FB MCV_HliInterface sets his "Error" output to TRUE, waits a fixed amount of cycles and tries again to communicate via the HLI with the motion controller. Occurs the error at output "Error" of an utility FB the calculation of that FB is interrupted immediately and output "Error" is set to TRUE.
Solution	Class	-	If the error occurs at an instance of the FB MCV_HliInterface, check whether the PLC application overwrites the variable gpPform with the value 0 after a valid pointer already existed. For all other FBs, you have to check if they are calculated before a FB of type MCV_HliInterface, signals the successful connection to the HLI by setting its output "Initialized" to TRUE. The FBs than have to be moved to an area of the PLC application, where they are calculated only if connection to the HLI is established successful.
Error type	-		

ID 40015

Version string of HLI of MC and of PLC are different.			
Description	A PLC application that communicates with the motion controller via the HLI interface must instantiate and call exactly one instance of the MCV_HliInterface function block. In the initialization phase of the Motion Control Platform, the FB MCV_HliInterface checks, among other things, the version identification of the HLI interface between the PLC and motion controller. The error message is then output by the FB MCV_HliInterface if the check of the version identifier results in different version identifiers of the PLC and Motion Controller.		
Response	Class	-	FB MCV_HliInterface sets its "Error" output to TRUE.

Solution	Class	-	At the outputs “ExpectedHli” and “ActualHli” at the instance of type MCV_HliInterface, the version identifiers of the HLI interface can be checked on the PLC and Motion Controller side. Please look for the version identifiers and consider that output “ExpectedHli” is the version identifier of the PLC’s HLI interface and output “ActualHli” is that of the motion controllers HLI interface. Now check if the motion controller is that with the wrong version identifier or if the PLC application included a HLI library with the wrong version. In first case please start the correct motion controller, in the second case please include the correct HLI-library in the PLC application. If none of the above causes is present, contact the control supplier and specify the version identifier at the outputs "ExpectedHli" and "ActualHli".
Error type	-		

Only the first 12 characters of the version identifier are checked and if they are equal that’s sufficient consistence of the HLI interface at PLC and motion controller to interact together.

ID 40016

HLI of MC and PLC has different characteristics.			
Description	When the HLI interface is activated by an instance of the FB MCV_HliInterface, the system checks the maximum number of commandable elements such as channels, axes, etc. on the PLC and motion controller sides. The error message is notified if the number of commandable channels or axes, etc. is different on the PLC and motion controller sides.		
Response	Class	-	The FB MCV_HliInterface sets its “Error” output to TRUE, waits for a fixed number of cycles and tries to communicate again with the motion controller via the HLI.
Solution	Class	-	At the outputs “ExpectedHli” and “ActualHli” of the FB instance of type MCV_HliInterface, the maximum number of channels, axes, etc. can be checked by the variable …Max. The output “ExpectedHli” displays the maximum number of the PLC’s HLI interface and the output “ActualHli” display the maximum number of the motion controller’s HLI interface. If there are different values for 2 variables with the same name, check the version identifier of the HLI interface on the PLC and motion controller sides by comparing the variable versionString of both outputs. Then check whether a motion controller with the wrong version identifier was started or whether the PLC application included an HLI library with the wrong version identifier. In the first case please start the motion controller with the correct version identifier; in the second case please include the HLI library with the correct version identifier in the PLC application. If none of the above causes exists, contact the controller supplier and specify the version identifier at the outputs "ExpectedHli" and "ActualHli".

Error type	-

Only the first 12 characters of the version identifier are checked and if they are equal that's sufficient consistence of the HLI interface at PLC and motion controller to interact together.

ID 40017

Axis specific HLI section of MC and PLC differ concerning the amount of bytes.			
Description	When the HLI interface is activated by an instance of the FB MCV_HliInterface, it is checked whether the axis-specific areas of the HLI on the PLC side and on the Motion Controller side have the same size. The error message is notified if area size differ about amount of bytes.		
Response	Class	-	The FB MCV_HliInterface sets its "Error" output to TRUE, waits for a fixed number of cycles and tries to communicate again with the motion controller via the HLI.
Solution	Class	-	At the outputs "ExpectedHli" and "ActualHli" at the FB instance of type MCV_HliInterface, the version identifiers of the HLI interface can be checked on the PLC and Motion Controller side. The output "ExpectedHli" indicates the version identifier of the PLC's HLI interface and output "ActualHli" indicates the version identifier of the motion controller HLI interface. Then check whether a motion controller with the wrong version identifier was started or whether the PLC application included an HLI library with the wrong version identifier. In the first case please start the motion controller with the correct version identifier; in the second case please include the HLI library with the correct version identifier in the PLC application. If none of the above causes exists, contact the controller supplier and specify the version identifier at the outputs "ExpectedHli" and "ActualHli".
Error type	-		

Only the first 12 characters of the version identifier are checked and if they are equal that's sufficient consistence of the HLI interface at PLC and motion controller to interact together.

ID 40018

Channel specific HLI section of MC and PLC differ concerning the amount of bytes.			
Description	When the HLI interface is activated by an instance of the FB MCV_HliInterface, it is checked whether the channel-specific areas of the HLI on the PLC side and on the Motion Controller side have the same size. The error message is notified if area size differ about amount of bytes.		
Response	Class	-	The FB MCV_HliInterface sets its "Error" output to TRUE, waits for a fixed number of cycles and tries to communicate again with the motion controller via the HLI.

Solution	Class	-	At the outputs “ExpectedHli” and “ActualHli” at the FB instance of type MCV_HliInterface, the version identifiers of the HLI interface can be checked on the PLC and Motion Controller side. The output “ExpectedHli” indicates the version identifier of the PLC’s HLI interface and output “ActualHli” indicates the version identifier of the motion controller HLI interface. Then check whether a motion controller with the wrong version identifier was started or whether the PLC application included an HLI library with the wrong version identifier. In the first case please start the motion controller with the correct version identifier; in the second case please include the HLI library with the correct version identifier in the PLC application. If none of the above causes exists, contact the controller supplier and specify the version identifier at the outputs "ExpectedHli" and "ActualHli".
Error type	-		

Only the first 12 characters of the version identifier are checked and if they are equal that’s sufficient consistence of the HLI interface at PLC and motion controller to interact together.

ID 40019

Pointer to structure containing information about the HLI specification is 0.			
Description	A PLC application that communicates with the motion controller via the HLI interface has to call one instance of the function block MCV_HliInterface. This FB checks whether the definition of the HLI interface is the same on the PLC side as the HLI definition on the motion controller side. Therefore, this FB requests information from the motion controller via the HLI interface implemented. If the FB receives no information after the request, the FB MCV_HliInterface issues then provides this error message at its output "ErrorID" for the PLC application.		
Response	Class	-	The FB MCV_HliInterface sets its “Error” output to TRUE and the output “Initialized” to FALSE.
Solution	Class	-	Please contact the controller supplier and quote the error number.
Error type	-		

ID 40020

Number of detected axes differ from configured axes.			
Description	A PLC application that communicates with the motion controller via interface HLI has to call one instance of function block MCV_HliInterface. This FB checks for each axis that is configured at the motion controller if there is a valid axis specific interface available at side of the PLC. When the number of valid interfaces for axes differs from the number of axes configured in the Motion Controller, the FB MCV_HliInterface issues this error message to the PLC application at its "ErrorID" output.		
Response	Class	-	FB MCV_HliInterface sets his “Error” output to TRUE, and “Initialized” output to FALSE.

Solution	Class	-	Please contact the motion control supplier and indicate the error number.
Error type	-		

ID 40021

Number of detected channels differ from configured channels.			
Description	A PLC application that communicates with the motion controller via the HLI interface has to call one instance of the function block MCV_HliInterface. This FB checks each channel that is configured on the motion controller whether there is a valid channel-specific interface available on the PLC side. If the number of valid interfaces for channels differs from the number of channels configured on the Motion Controller, the FB MCV_HliInterface makes provides this error message to its "ErrorID" output for the PLC application.		
Response	Class	-	The FB MCV_HliInterface sets sets its “Error” output to TRUE and the output “Initialized” to FALSE.
Solution	Class	-	Please contact the controller supplier and quote the error number.
Error type	-		

ID 40022

Platform specific section of PLC and MC differ concerning the amount of bytes.			
Description	When the HLI interface is activated by an instance of the FB MCV_HliInterface, a check is made whether the platform-specific areas of the HLI on the PLC side and on the Motion Controller side have the same size. The error message is notified if area size differ about amount of bytes.		
Response	Class	-	The FB MCV_HliInterface sets its “Error” output to TRUE, waits for a fixed number of cycles, and tries to communicate again with the motion controller via the HLI .
Solution	Class	-	At the outputs “ExpectedHli” and “ActualHli” at the FB instance of type MCV_HliInterface, the version identifiers of the HLI interface can be checked on the PLC and Motion Controller side. Please look for the version identifiers and consider that output “ExpectedHli” is the version identifier of the PLC’s HLI interface and output “ActualHli” is that of the motion controllers HLI interface. Now check if the motion controller is that with the wrong version identifier or if the PLC application included a HLI library with the wrong version. In first case please start the correct motion controller, in the second case please include the correct HLI-library in the PLC application. If none of the above causes is present, contact the control supplier and specify the version identifier at the outputs "ExpectedHli" and "ActualHli".
Error type	-		

ID 40023

Pointer to global external variables HLI section is 0.

Description	When the HLI interface is activated by an instance of the FB MCV_HliInterface, global pointer variables are initialised. The implemented FBs can then access the various HLI areas. To ensure the pointers are valid they are checked. The check detects that the global pointer variable gpVEGlobal is invalid because its value is 0.		
Response	Class	-	FB MCV_HliInterface sets its “Error” output to TRUE, waits for a fixed number of cycles, and tries to communicate again with the motion controller via the HLI . Occurs the error at output “Error“ of an utility FB the calculation of that FB is interrupted immediately and output “Error“ is set to TRUE.
Solution	Class	-	Occurs the error at an instance of FB MCV_HliInterface you have to check if the PLC application overwrites the variable gpVEGlobal with 0 already after a valid pointer was available. For all other FB you have to check if they are calculated before a FB of type MCV_HliInterface signalled the successful connection to the HLI by setting its output “Initialized” to TRUE. The FBs than have to be moved to an area of the PLC application, where they are calculated only if connection to the HLI is established successful.
Error type	-		

ID 40024

Pointer to channel specific external variables HLI section is 0.			
Description	When the HLI interface is activated by an instance of the FB MCV_HliInterface, global pointer variables are initialised. The implemented FBs can then access the various HLI areas. To ensure the pointers are valid they are checked. The check detects that at least one global pointer variable gpVECh[*Idx*] (*idx* = [0, gNrCh - 1]), that points to channel specific external variables is invalid because its value is 0.		
Response	Class	-	FB MCV_HliInterface sets its “Error” output to TRUE, waits for a fixed number of cycles, and tries to communicate again with the motion controller via the HLI . Occurs the error at output “Error“ of an utility FB the calculation of that FB is interrupted immediately and output “Error“ is set to TRUE.
Solution	Class	-	If the error occurs at an instance of the FB MCV_HliInterface, check whether the PLC application overwrites the variable gpVECh[*idx*] with the value 0 after a valid pointer already existed. For all other FBs, you have to check if they are calculated before a FB of type MCV_HliInterface, signals the successful connection to the HLI by setting its output “Initialized” to TRUE. The FBs than have to be moved to an area of the PLC application, where they are calculated only if connection to the HLI is established successful.
Error type	-		

ID 40025

Invalid handle to HLI during set-up of HLI.

Description	The PLC application requests the control system for a handle to access the HLI interface. This is the interface between the PLC and the motion controller. The error is output because the returned handle is invalid.		
Response	Class	-	The PLC application has no access to the HLI interface and is therefore unable to send jobs.
Solution	Class	-	Check whether the control system is active. If this is the case and the error is still displayed, contact your controller manufacturer.
Error type			

ID 40026

Pointer to HLI is 0.			
Description	The PLC application requests the control system for a pointer to access the HLI interface. This is the interface between the PLC and the motion controller. The error is output because the returned pointer is invalid.		
Response	Class	-	The PLC application has no access to the HLI interface and is therefore unable to send jobs.
Solution	Class	-	Check whether the control system is active. If this is the case and the error is still displayed, contact your controller manufacturer.
Error type			

ID 40027

Invalid handle to description of HLI configuration.			
Description	The PLC application requests the control system for a handle to access the configuration description of the HLI interface. This is the interface between the PLC and the motion controller. The error is output because the returned handle is invalid.		
Response	Class	-	The PLC application has no access to the configuration description of the HLI interface. It is therefore impossible to set up a connection to the motion controller.
Solution	Class	-	Check whether the control system is active. If this is the case and the error is still displayed, contact your controller manufacturer.
Error type			

ID 40028

Error occurred during setup of HLI. HLI not accessible.			
Description			
Response	Class	-	
Solution	Class	-	
Error type			

ID 40032

Axis is not configured to be used as motion axis.			
Description	During the initialisation phase of the Motion Control Platform, it is checked whether an axis reference refers to an axis that can be assigned "single axis" and "multiple axes" functions by function blocks. The check happens in the initialization phase of the FB MCV_P1_Platform and is executed by the utility FB MCV_AxisInit. If this is not the case, this error is entered in the axis reference. Every FB that commands an axis by means of this reference detects the error and displays it at its error output.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Please check the following options: Was the correct axis reference used? Otherwise connect another axis reference to the input “Axis”. If yes, check the axis configuration in the axis configuration list. See P-AXIS-00015, P-AXIS-00018, P-AXIS-00250.
Error type	-		

ID 40033

Registering at HLI control units failed.			
Description	During the initialisation phase, the PLC logs on to certain control units that are required for the function of the motion libraries. This occurs in all axis-specific HLI areas that can be addressed using a valid axis reference. The check happens during the initialization phase of the FB MCV_P1_Platform and is executed by the utility FB Ax_HLI_Init. If this does not succeed, this incorrect value is stored in the respective axis reference. Each FB that wants to transmit a job to the Motion Controller with this axis reference detects this error in the axis reference, and displays it at its error output.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Please contact your motion controller supplier.
Error type	-		

ID 40034

MC is not yet in cyclic operating state.			
Description	A PLC application that uses PLC libraries containing function blocks according to the PLCopen specifications to solve function blocks must call exactly one instance of the MCV_PlatformBase function block. During the initialisation phase of the Motion Control Platform, the utility FB MCV_InitPlatform is called within the utility FB MCV_PlatformBase. This checks whether the motion controller reports that it is running and if this is fulfilled, whether the motion controller performs its calculations cyclically. If it cannot be determined within a defined number of PLC cycles that the Motion Controller itself is in cyclic operation, this error message is transmitted to the calling FB MCV_PlatformBase, which makes it available to the PLC application at its "ErrorID" output.		
Response	Class	-	The FB MCV_PlatformBase sets its output “Error” to TRUE and the output “Done” to FALSE.
Solution	Class	-	Please contact the controller supplier and quote the error number.
Error type	-		

ID 40035

MC hasn't finished the start-up phase already.			
Description	When the Motion Controller reports to the HLI interface that it has completed its startup phase and is ready to receive jobs and move axes, it reports this via the HLI interface by setting gpPform^.hochlauf_quit_r to TRUE in the platform-specific area of the HLI. No job can be sent via the HLI to the Motion Controller if this data item is FALSE.		
Response	Class	-	No jobs are sent to the motion controller.
Solution	Class	-	Is the function block that displays that error message a MCV_PlatformBase, this must be called further until the error message is no longer displayed. If that doesn't happen please contact the supplier of the control system. other function block, the job has to be sent again, because the former sent job wasn't sent to the motion controller.
Error type	-		

ID 40037

Pointer to platform specific HLI section is 0.			
Description	A function block, that exchanges data and jobs with the motion controller via the platform-specific area on the HLI interface, checks whether the global pointer gpPform contains a valid address before accessing this area. If the global pointer gpPform contains an invalid address, such a function block displays this error message.		
Response	Class	-	No data or jobs are exchanged via the interface HLI.
Solution	Class	-	Check whether the function module was commanded before the instance of the function block MCV_HliInterface signalled that the interface HLI was successfully checked and is now ready to exchange data and jobs. If this is the case, implement the PLC application so that the function block is not called until the function block MCV_HliInterface at output "Initialized" shows TRUE.
Error type	-		

ID 40038

Axis reference: addresses a HLI section that exceeds HLI definition.			
Description	A function block displays this error code at its output "ErrorID" if the axis reference that is assigned to an input of type AXIS_REF, i.e. inputs with the name "Axis", "Master" or "Slave", refers to an axis-specific HLI area that lies outside the specification of the motion controller. The error code is not only displayed if the FB is triggered at its input “Execute but also when the FB is called.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller. The output “Error” is set to TRUE.

Solution	Class	-	There are 2 possible reasons why the error code is displayed: The PLC application overwrites the value specified in the axis reference during the initialisation phase of the motion environment, which is the reference to an axis-specific area of the HLI. Therefore, it must therefore be checked whether the structure element hli_axis_idx is overwritten in the global variable g_array_axis_ref[*idx*], which is assigned to an input of the FB of type AXIS_REF. The value of hli_axis_idx must be in the range [0 .. HLI_SYS_AX_MAXIDX]. The FB is called without checking if the instance of the FB MCV_PlatformBase displays the value TRUE at the output "Done". Then move the FB to a PLC application area that is only processed when the instance of the FB type MCV_PlatformBase indicates that the motion platform was successfully initialised. If none of the two cases described apply, please contact your motion controller supplier and quote the error code.
Error type	-		

ID 40039

Axis reference: addressed HLI section doesn't exist.			
Description	A function block notifies this error code at its output “ErrorID”, if an axis reference that is assigned to an input of type AXIS_REF, means at an input named by identifier “Axis”, “Master” or “Slave”points to an axis specific area that haven’t an axis assigned by the start-up list of the motion controller. The error code isn’t displayed only if the FB is triggered at his input „Execute“, but also at the moment the FB is calculated.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller. The output “Error” is set to TRUE.
Solution	Class	-	There are 2 matters possible why the error code is notified: In fact, the controller startup list contains no entry for the axis. In the start-up list for the axis, make an entry for P-STUP-00015 and then adjust P-STUP-00014 accordingly. The order of axes in the startup list reflects the order of axis-specific HLI areas. Check whether the axis list specified in parameter P-STUP-00015 exists. If this is not the case, create the axle list. The PLC application overwrites the value specified in the axis reference during the initialization phase of the motion environment, which is the reference to an axis-specific area of the HLI. Therefore, it must therefore be checked whether the structure element hli_axis_idx is overwritten in the global variable g_array_axis_ref[*idx*], which is assigned to an input of the FB of type AXIS_REF. The value of hli_axis_idx has to be in range [0 .. gNrAx (number of all configured axes in the motion system)].
Error type	-		

ID 40040

Axis reference: addresses a HLI section that exceeds HLI definition.

Description	A function block displays this error code at its output "ErrorID" if the axis group reference that is assigned at an input of type AXES_GROUP_REF with the name "AxisGroup" refers to a channel-specific HLI area that lies outside the specification of the motion controller. The error code is not only displayed if the FB is triggered at its input “Execute but also when the FB is called.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller. The output “Error” is set to TRUE.
Solution	Class	-	There are 2 possible reasons why the error code is displayed: The PLC application overwrites the value specified in the axis group reference during the initialization phase of the motion environment, which is the reference to a channel-specific area of the HLI. Therefore, it must be checked whether the structure element HliIfcIdx is overwritten in the global variable gAxesGroupRef[*idx*] which is assigned to an input of the FB of type AXES_GROUP_REF. The value of hli_axis_idx must be in the range [0 .. HLI_SYS_CH_MAXIDX]. The FB is called without checking if the instance of the FB MCV_PlatformBase displays the value TRUE at the output "Done". Then move the FB to a PLC application area that is only processed when the instance of the FB type MCV_PlatformBase indicates that the motion platform was successfully initialised. If none of the two cases described apply, please contact your motion controller supplier and quote the error code.
Error type	-		

ID 40041

Axis group reference: addressed HLI section doesn't exist.			
Description	A function block displays this error code at its output "ErrorID", if the axis group reference that is assigned at an input of type AXES_GROUP_REF with the name "AxesGroup", refers to a channel-specific HLI area to which no channel has been assigned in the motion controller start-up list. The error code isn’t displayed only if the FB is triggered at his input „Execute“, but also at the moment the FB is calculated.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller. The output “Error” is set to TRUE.
Solution	Class	-	There are 2 matters possible why the error code is notified: 1. In fact, the controller startup list contains no entry for the channel. In the start-up list for the channel, make an entry for P-STUP-00009 and then adjust P-STUP-00001 and P-STUP-00007 accordingly. The order of channels in the startup list reflects the order of channel-specific HLI areas. Check whether the channel list specified for parameter P-STUP-00009 exists. If this is not the case, create the channel list. 2. The PLC application overwrites the value specified in the axis group reference during the initialization phase of the motion environment, which is the reference to a channel-specific area of the HLI. Therefore, it must be checked whether the structure element HliIfcIdx is overwritten in the global variable gAxesGroupRef[*idx*], which is assigned to an input of the FB of type AXES_GROUP_REF. The value of hli_axis_idx must be in range [0 .. gNrCh (number of all configured channels in the motion system)].

Error type	-

ID 40042

Pointer to table manager memory is 0.			
Description	Before accessing this area, a function block that accesses the memory with cam tables checks whether the gpTabMgr pointer to this memory contains a valid address. If the global pointer gpTabMgr contains an invalid address, such a function block displays this error message.		
Response	Class	-	Function block doesn't access the cam table memory.
Solution	Class	-	Please contact the motion control supplier and indicate the error number.
Error type	-		

ID 40044

FB error: MCV_TRACE. Got new trace command, but trace runs already.			
Description	The MCV_TRACE function block can be used to command the Motion Controller to record internal data, to stop recording or to write the recorded data to a file. These jobs may not be interrupted at any point of time. The function block displays this error message if it is commanded to trace a new set of data but did not finish the former trace or is still writing the previously traced data to a file.		
Response	Class	-	Job to trace a new set of data is discarded.
Solution	Class	-	Start job to trace a new set of data only if output "StateString" displays the states 'IDLE' or 'TRACE_STOPPED'.
Error type	-		

ID 40045

FB error: MCV_TRACE. Writing to file is impossible.			
Description	The function block MCV_TRACE is to send a command to the motion controller to write the traced data to a file. This command can only be sent successfully to the motion controller if output "StateString" displays the state 'TRACE_STOPPED'.		
Response	Class	-	Job to write traced data to a file will be dismissed.
Solution	Class	-	Start job to write traced data to file only if output "StateString" displays the states 'TRACE_STOPPED'.
Error type	-		

ID 40046

FB error: axis techno unit of that number isn't available.

Description	Input "No" identifies the technology function that is to be handled with the respective function block. If this error message occurs, an identification number is used that is outside the range specified by the implementation of the function blocks. This error message is output by function modules that handle technology functions that are output on an axis-specific interface on the HLI. The permissible identification numbers are dependent on the function block type. The upper limits for this are defined in the technology library [MCP-TECH] and listed in the table below.		
	Function block data type	**FB error: axis techno unit of that number isn't available.**	
	MCV_HFctAxis	0 <= No <= MCV_HFCT_MAXIDX	
	MCV_MFctAxis	0 <= No <= MCV_MFCT_MAXIDX	
	MCV_SFctAxis	0 <= No <= MCV_SFCT_MAXIDX	
	MCV_TFctAxis	0 <= No <= MCV_TFCT_MAXIDX	
Response	Class	-	No interaction takes place with the motion controller.
Solution	Class	-	Choose an identification number for the technology function that is within the allowed range.
Error type	-		

ID 40047

FB error: axis group specific technology function with this number isn't available.			
Description	Input "Nr" identifies the technology function that is to be handled with the respective function block. If this error message occurs, an identification number is used that is outside the range specified by the implementation of the function blocks. This error message is output by function modules that handle technology functions that are output on an axis group-specific interface on the HLI. The permissible identification numbers are dependent on the function block type. The upper limits for this are defined in the technology library [MCP-TECH] and listed in the table below.		
	Function block data type	**FB error: axis group specific technology function with this number isn't available.**	
	MCV_HFctChannel	0 <= Nr <= MCV_HFCT_MAXIDX	
	MCV_MFctChannel	0 <= Nr <= MCV_MFCT_MAXIDX	
	MCV_SFctChannel	0 <= Nr <= MCV_SFCT_MAXIDX	
	MCV_TFctChannel	0 <= Nr <= MCV_TFCT_MAXIDX	
Response	Class	-	No interaction takes place with the motion controller.
Solution	Class	-	Choose an identification number for the technology function that is within the allowed range.
Error type			

ID 40048

HLI HMI: addresses a HLI section that exceeds HLI definition.

Description	In the HLI interface, there are areas for various user interface elements (HMI) through which the properties of these elements can be influenced. Each of these areas is created as an array and there is one for each element type. The elements include keys (Key), parameters (Param) or strings (String). If one of the function blocks listed in the table below accesses the corresponding array with an invalid index ("Idx" input), the error is displayed at its "ErrorID" output. The permissible index values are dependent on the function block type and are listed in the table below. The upper limit values are defined in the HLI library [HLI].		
	Function block data type	**HLI HMI: addresses a HLI section that exceeds HLI definition.**	
	MCV_AxPlcKey	0 <= Idx <= HLI_HPK_AXIS_MAXIDX	
	MCV_AxPlcParam	0 <= Idx <= HLI_HPP_AXIS_MAXIDX	
	MCV_AxPlcString	0 <= Idx <= HLI_HPS_AXIS_MAXIDX	
	MCV_ChPlcKey	0 <= Idx <= HLI_HPK_CHAN_MAXIDX	
	MCV_ChPlcParam	0 <= Idx <= HLI_HPP_CHAN_MAXIDX	
	MCV_ChPlcString	0 <= Idx <= HLI_HPS_CHAN_MAXIDX	
	MCV_GlobPlcKey	0 <= Idx <= HLI_HPK_GLOB_MAXIDX	
	MCV_GlobPlcParam	0 <= Idx <= HLI_HPP_GLOB_MAXIDX	
	MCV_GlobPlcString	0 <= Idx <= HLI_HPS_GLOB_MAXIDX	
Response	Class	-	No interaction takes place with the motion controller.
Solution	Class	-	Select the index for the element type so that it lies within the permissible numerical range.
Error type	-		

ID 40049

HLI HMI: addressed HLI section doesn't exist.			
Description	Function blocks that can change the properties of user interface elements using the HLI interface require access to the corresponding area on the HLI. This access takes place using the global pointer gpHmiPlc which is initialised by the function block MCV_HliInterface during the PLC start-up phase. The error is displayed by such function blocks, if the pointer addresses an invalid address.		
Response	Class	-	No interaction takes place with the motion controller.
Solution	Class	-	Check whether exactly one instance of the function block MCV_HliInterface is called in the PLC application. Check whether the function block that displays the error is not called until the function block MCV_HliInterface displays TRUE at its "Initialized" output.
Error type	-		

ID 40051

No drive with SERCOS interface within all axis specific interfaces.			
Description	If this error message is displayed checking the drive type of all axes of the control system finished but there was no axis with drive type SERCOS found.		
Response	Class	-	
Solution	Class	-	Check the axis parameter P-AXIS-00020 of the axes.

Error type	-

ID 40052

Value of SERCOS phase exceeds range 0..4.			
Description	The value at the "Phase" input or output of a function block that allows access to the phase of a SERCOS ring does not correspond to the defined values for the phase of the SERCOS ring.		
Response	Class	-	The job to read/write the phase of the SERCOS ring is aborted.
Solution	Class	-	After correcting the value at input "Phase", restart the job for when the value should be written. If the value of the phase of the SERCOS ring should be read, try to do it with another software tool.
Error type	-		

ID 40053

Error occurred while setting or commanding SERCOS phase.			
Description	Reading or writing the phase of a SERCOS ring is not possible. The service is finished with an error.		
Response	Class	-	Occurs error during writing the phase of the SERCOS ring, the job finishes, and there is no effect to the current phase of the SERCOS ring. When reading, the output "Phase" of the function block is set to value 0.
Solution	Class	-	Check whether SERCOS devices are configured in the control system or whether the controller start-up was completed without an error message indicating problems with the SERCOS fieldbus.
Error type	-		

ID 40055

FB error: MCV_PartTracker. Maximum number of tracked parts exceeds number of managed parts.			
Description	The function block manages a defined number (MAX_PARTS) of parts. However, the number of parts detected by the measuring probe exceeds this number. This is why the error is output at the "ErrorID" output.		
Response	Class	-	The function block goes to error state.
Solution	Class	-	Check if more parts really entered the probing area than exited it. Remove the error. When the function block is again commanded, the FB will exit its error state.
Error type	-		

ID 40056

FB error: MCV_PartTracker. Position of next part is requested, but there is no part left in the administration.	
Description	The function block receives over the input "GetNextPartPos" the job to request the the position of the next part from the internal administration. But there is no position for the next part available.

Response	Class	-	The function block goes to error state.
Solution	Class	-	Check why the positions of the parts were not recorded.
Error type	-		

ID 40059

Error occurred while setting or getting SERCOS ident.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 40066

FB error: MCV_AddCmdValues. Value at input "Mode" is invalid.			
Description	Valid values that can be assigned to the input "Mode" are:		
	Constant	**Value**	**FB error: MCV_AddCmdValues. Value at input "Mode" is invalid.**
	ADD_POS_VALUE	1	Value at input "Value" is a position value.
	ADD_SPEED_VALUE	2	Value at input "Value" is a velocity value.
Response	Class	-	There is no interaction with the Motion Controller.
Solution	Class	-	Correct value at input "Value" to correspond to one of the values in the table above.
Error type	-		

ID 40067

FB error: MCV_RedSpeedZone. Value at input "ZoneNr" is invalid.			
Description	Zones can be defined for an axis in the respective parameter list (P-AXIS-00085, P-AXIS-00097) in which the motion is carried out at reduced speed after a corresponding job. The job specifies which of the zones configured in the parameter list (1 or 2) is to be activated. The error message is displayed if the value at the input "ZoneNr" points to an undefined zone.		
Response	Class	-	There is no interaction with the Motion Controller.
Solution	Class	-	Correct value at input "ZoneNr" so that value points to a defined zone.
Error type	-		

ID 40068

FB error: MCV_DriveWord. Value at input "DriveWordNr" is invalid.			
Description			
Response	Class	-	
Solution	Class	-	

Error type	-

ID 40069

FB error: MCV_Mode. Value at input "ModeNr" is invalid.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 40070

FB error: MCV_GrpRedSpeedZone. Value at input "ZoneNr" is invalid.			
Description	Zones can be defined for an axis group in the respective parameter list (P-CHAN-00085, P-CHAN-00093), in which the path movement is executed at reduced speed after a corresponding job. The job specifies which of the zones configured in the parameter list (1 or 2) is to be activated. The error message is displayed if the value at the input "ZoneNr" points to an undefined zone.		
Response	Class	-	There is no interaction with the Motion Controller.
Solution	Class	-	Correct value at input "ZoneNr" so that value points to a defined zone.
Error type	-		

ID 40071

FB error: error id 1 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 1. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40072

FB error: error id 2 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 2. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40073

FB error: error id 3 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 3. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40074

FB error: error id 4 of FB STR_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 4. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40075

FB error: error id 5 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 5. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40076

FB error: error id 6 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 6. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40077

FB error: error id 7 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 7. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40078

FB error: error id 8 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 8. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40079

FB error: error id 9 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 9. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40080

FB error: error id 10 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 10. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40081

FB error: error id 11 of FB STRING_TO_BUF.			
Description	Function block calls the PLC runtime specific function block STRING_TO_BUF. This specific function block displays error code 11. For more information we point to help for *_TO_BUF function blocks and the specific help of function block STRING_TO_BUF of the PLC runtime supplier.		
Response	Class	-	Character string at input "InputString" isn't copied to the byte array at output "ByteArray".
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40084

FB error: STRING_TO_BUF. Returns unknown error id.			
Description	The function block calls the PLC runtime-specific function block STRING_TO_BUF. This specific function block displays an invalid error code.		
Response	Class	-	The character string at the input "InputString" is not copied to the byte array at the output "ByteArray".
Solution	Class	-	Please contact your motion controller supplier and specify the invalid error number displayed at the output "ErrorID".
Error type	-		

ID 40085

FB error: MCV_GetParamType. Assigned empty string to input "Param".			
Description			
Response	Class	-	Output "ParamType" displays an empty character string. Value at output "ParamNr" isn't changed and may display the value of a former successful finished trial to get the parameter type.
Solution	Class	-	Assign a character string to input "Param" that is the identifier of a parameter of a fieldbus (actually only SERCOS idents are processed).
Error type	-		

ID 40086

FB error: Platform specific control unit can't take order.			
Description	A function block that uses a platform specific control unit to command jobs to the motion controller can't send a job because the control unit is occupied. The error is displayed after function block has made further attempts unsuccessfully to send the job. The number of attempts is defined by global constant MAX_RETRIALS. MAX_RETRIALS = 0. Therefore, this error usually occurs when more than one function block sends an order per PLC cycle, and these are to be sent to the motion controller via the same platform-specific control unit.		
Response	Class	-	The job isn't executed.

Solution	Class	-	Check whether more than one function block sends jobs to the Motion Controller during a PLC cycle via the same platform-specific Control Unit. If this is the case, change the PLC application accordingly so that this no longer occurs.
Error type	-		

ID 40087

FB error: Axis specific control unit can't take order.			
Description	A function block that uses an axis specific control unit to command jobs to the motion controller can't send a job because the control unit is occupied. The error is displayed after function block has made further attempts unsuccessfully to send the job. The number of attempts is defined by global constant MAX_RETRIALS. MAX_RETRIALS = 0. Therefore, this error usually occurs when more than one function block sends an order per PLC cycle, and these are to be sent to the motion controller via the same axis specific control unit.		
Response	Class	-	The job isn't executed.
Solution	Class	-	Check whether more than one function block sends jobs to the Motion Controller during a PLC cycle via the same axis-specific Control Unit. If this is the case, change the PLC application accordingly so that this no longer occurs.
Error type	-		

ID 40088

FB error: Channel specific control unit can't take order.			
Description	A function block that uses a channel specific control unit to command jobs to the motion controller can't send a job because the control unit is occupied. The error is displayed after function block has made further attempts unsuccessfully to send the job. The number of attempts is defined by global constant MAX_RETRIALS. MAX_RETRIALS = 0. Therefore, this error usually occurs when more than one function block sends an order per PLC cycle, and these are to be sent to the motion controller via the same channel specific control unit.		
Response	Class	-	The job isn't executed.
Solution	Class	-	Check whether more than one function block sends jobs to the Motion Controller during a PLC cycle via the same channel-specific Control Unit. If this is the case, change the PLC application accordingly so that this no longer occurs.
Error type	-		

ID 40089

FB error: Value that classifies the axis type is invalid.

Description	The value representing the axis type does not correspond to the allowed values for the axis type. Valid values for the axis type are:		
	Global constant		**FB error: Value that classifies the axis type is invalid.**
	HLI_AXIS_TYPE_TRANSLATOR		1
	HLI_AXIS_TYPE_ROTATOR		2
	HLI_AXIS_TYPE_SPINDLE		4
Response	Class	-	The job is not executed and the output values of the function block dependent on the axis type are reset.
Solution	Class	-	Check the value at the "AxisType" input if a function block reports this error or the value of the corresponding variable for a function, and correct it.
Error type	-		

ID 40090

FB error: Value that classifies the type of value is invalid.			
Description	A function block, or a function whose flow depends on the type of the transferred value, displays this error message if the transferred type does not correspond to any of the defined types. Permissible types for values are:		
	Global constant		**FB error: Value that classifies the type of value is invalid.**
	MCV_PositionValue		0
	MCV_VelocityValue		1
	MCV_AccelerationValue		2
	MCV_JerkValue		3
Response	Class	-	The job is not executed. The function block outputs are reset depending on their type.
Solution	Class	-	Check the value at the “ValueType” input and correct if necessary.
Error type	-		

ID 40091

FB error: Searched axis at HLI by logical axis number, but didn't find such an axis.			
Description	A job is sent via a function block, and the function block then shows TRUE at output "Busy". The FB receives a response for that job from de motion controller, and part of the response is a reference to an axis. The FB checks this reference by searching for this axis in the axis-specific HLI areas. The reference in this case is the logical number of the axis. Because the FB doesn’t find an axis specific HLI area containing this reference, output “ErrorID” notifies this error code and output “Error” is set to TRUE.		
Response	Class	-	The error code is displayed. The FB is able to take further orders. Falling edge at input “Execute” or “Enable” sets output “Error” to FALSE and resets output “ErrorID”.

Solution	Class	-	Is the commanded FB of type MC_GrpReadCfg you have to check, if the content of the variable assigned to input “IdentInGroup” points to an axis that is applied in the configuration of the machine. If that’s not the matter you • Either the content of the variable at the “IdentInGroup” input must be corrected • or create the axis in the machine configuration.
Error type	-		

ID 40092

FB error: Searched axis within axis references by logical number, but didn't find such an axis.			
Description	A job is sent via a function block, and the function block then shows TRUE at output "Busy". The FB receives a response for that job from de motion controller, and part of the response is a reference to an axis. The FB checks whether an axis reference (g_array_axis_ref[*idx*]) exists for this reference representing this axis. The reference in this case is the logical number of the axis. However, no axis representing an axis with this logical axis number is found. Because the FB doesn't find an axis reference that also points to that axis, “ErrorID” output notifies this error code and output “Error” is set to TRUE.		
Response	Class	-	The error code is displayed. The FB is able to take further orders. Falling edge at input “Execute” or “Enable” sets output “Error” to FALSE and resets output “ErrorID”.
Solution	Class	-	Is the commanded FB of type MC_GrpReadCfg you have to check, if the content of the variable assigned to input “IdentInGroup” points to an axis that is applied in the configuration of the machine. If that’s not the matter you • Either the content of the variable at the “IdentInGroup” input must be corrected • or create the axis in the machine configuration.
Error type	-		

ID 40093

FB error: Configured drive type doesn't support the configured drive interface.			
Description	Some drive types (P-AXIS-00020) have additional interfaces implemented (e.g. exchange of specified drive data via a PLC interface) in addition to the default interface via which drive data is directly exchanged with the motion controller. Specific function blocks are implemented for each type of interface (P-AXIS-00572) in order to read or write drive data. Before access to this type of interface, each function block checks whether the interface for which it is implemented was configured in the axis lists. If the interface which the function block accesses is not configured depending on its implementation, the function block indicates an error.		
Response	Class	-	No access is executed to an interface with drive data.

Solution	Class	-	Check the configured values of P-AXIS-00020 and P-AXIS-00572 in the axis list. Possible combinations of the parameters values are: **Interface for drive data** | **Drive type** HLI_DRIVE_IFC_0 1 | PROFIdrive: 0x0003 CANopen: 0x0008
Error type	-		

ID 40094

FB error: Configured drive type doesn't support a plc drive interface.			
Description	The applied function block cannot be used to read or write drive data because an interface (P-AXIS-00572) which this function block cannot operate according to its implementation was configured for this axis.		
Response	Class	-	No access is executed to an interface with drive data.
Solution	Class	-	Check the configured values of P-AXIS-00020 and P-AXIS-00572 in the axis list. Possible combinations of the parameters values are: **Value for P-AXIS-00572** | **Value** | **Drive type** Parameter does not exist in the axis parameter list | - | Any configurable drive type DRIVE_IFC_DEFAULT (same effect as line above) | 0 | Any configurable drive type HLI_DRIVE_IFC_0 | 1 | PROFIdrive: 0x0003 CANopen: 0x0008
Error type			

ID 40095

FB error: Bus state notifies that bus communication is invalid.			
Description	The function block no longer reads or writes data from/to the interface for drive data, because the fieldbus status reports that the transmission of data via the fieldbus is faulty.		
Response	Class	-	No access to interface with drive data is executed.
Solution	Class	-	Check fieldbus and eliminate disruption.
Error type	-		

ID 40097

System error: More configured spindles notified than axes configured.	
Description	During the start-up phase of the PLC application a check is made how many spindles were configured. They are axes in which the axis type is defined by ACHSTYP_SPINDEL in the axis parameter list (P-AXIS-00008) of the motion controller. If this error message appears, more configured spindles were signalled than there are axes. However, the number of spindles is always less than or equal to the number of axes.

Response	Class	-	No link is made to the motion controller.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40098

System error: Number of configured channels and available HLI channel interfaces differs.			
Description	A check is made in the start-up phase of the PLC application which and how many interfaces are sent by the motion controller. This error message indicates that the number of configured axis groups reported by the control system differs from the number of PLC interfaces to axis groups.		
Response	Class	-	No link is made to the motion controller.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40099

System error: Number of configured axes and available HLI axis interfaces differs.			
Description	A check is made in the start-up phase of the PLC application which and how many interfaces are sent by the motion controller. This error message indicates that the number of configured axes reported by the control system differs from the number of PLC interfaces to axes.		
Response	Class	-	No link is made to the motion controller.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40100

System error: Number of configured spindles and available HLI axis interfaces for spindles differs.			
Description	A check is made in the start-up phase of the PLC application which and how many interfaces are sent by the motion controller. This error message indicates that the number of configured spindles reported by the control system differs from the number of PLC interfaces to spindles.		
Response	Class	-	No link is made to the motion controller.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 40101

FB error: Only one instance of the function block is allowed in PLC application.	
Description	There are function blocks that may only have one instance occurring in a PLC application in order to ensure data consistency. If several instances of this function block were created and called in a PLC application, the first instance called is not indicated with this error message. However, all the other instances occurring in the application sequences are.

Response	Class	-	The instance of the function block indicating this error message is only called but not exited immediately. It neither acts on the PLC application nor on the motion controller.
Solution	Class	-	Please contact your motion controller supplier.
Error type	-		

ID 40102

FB error: logical channel number is invalid..			
Description	If a POE displays this error message, either an invalid identifier was transferred for an axis group or detected during the POE call. The identifier then had a value of < 0.		
Response	Class	-	The POE displays the error and cannot execute the implemented job.
Solution	Class	-	Check the identifier for an axis group transferred to the POE.
Error type	-		

ID 40103

FB error: index of axis within axis group is invalid.			
Description	The error is displayed by a POE if the calculated or transferred index for sorting an axis in an axis group exceeds the maximum permissible index. See also the definition of the global constant HLI_CH_AX_MAXIDX.		
Response	Class	-	The POE displays the error and cannot execute the implemented job.
Solution	Class	-	Check the index transferred to the POE for sorting the axis in an axis group.
Error type	-		

2.5.2 ID-range 42000-42249

ID 42000

FB error: Value at pin "BufferMode" is invalid.				
Description	Invalid value assigned to input "BufferMode" at the function block that displays this error.			
Response	Class	-	The job is not sent to the motion controller. The function block can send the job again in the next but one PLC cycle.	
Solution	Class	-	Valid values for input "BufferMode" are:	
			Global constant	**FB error: Value at pin "BufferMode" is invalid.**
			mcAborting	0
			mcBuffered	1
			mcBlendingLow	2
			mcBlendingPrevious	3
			mcBlendingNext	4
			mcBlendingHigh	5
Error type	-			

ID 42001 - 42011

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 42012

FB error: platform specific PLCopen interface couldn't adopt a new request. Request FIFO full.			
Description	Jobs of platform-specific PLCopen function blocks are managed by a FIFO. The size of this FIFO is fixed (HLI_PLCO_PF_REQUEST_NUM). If more jobs are written to the FIFO in a PLC cycle than the space available, this error code is output to the function block whose job can no longer be sent to the motion controller.		
Response	Class	-	The job is discarded by the function block and not transferred to the motion controller.
Solution	Class	-	Check if it is necessary for more jobs to be sent to the motion controller within a PLC cycle than places are available in the FIFO to solve the motion task.
Error type	-		

ID 42014

FB error: value at input "VelFactor" exceeds allowed value range.	
Description	Due to the implementation of the motion platform, the value at the "VelFactor" input of the FB MC_GrpSetOverride must be in the interval [0.0, 65535.0].

Response	Class	-	The FB sets the “Error” output to TRUE and the “Enabled” output to FALSE. The value that lies outside the limits is ignored. The last correctly calculated override value is still active.
Solution	Class	-	Please assign a value within in the limits of the interval [0.0, 65535.0] to the input “VelFactor”.
Error type	-		

ID 42015

FB error: Not able to push a job to local request job stack.			
Description	An job is passed through several components of the control system. Every component which passes on a job writes its identifier to a stack belonging to the job. In this case, a copy of the received stack is generated locally when the job is commanded. The job writes its own identifier to the stack and the content of the local copy is passed on to the subsequent component. Is the stack full and the identifier can't be copied to the stack this error message is displayed.		
Response	Class	-	The error is displayed at the output "Error" and "ErrorID" of the function block.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 42016

FB error: value at input "TimeFactor" exceeds allowed value range.			
Description	Due to the implementation of the motion platform, the value at input “TimeFactor” of FB MC_GrpSetTimeOverride is limited to the interval [0.0, 65535.0].		
Response	Class	-	The FB sets the “Error” output to TRUE and the “Enabled” output to FALSE. The value that lies outside the limits is ignored. The last correctly calculated override value is still active.
Solution	Class	-	Please assign a value within the limits of the interval [0.0, 65535.0] to the input “TimeFactor”.
Error type	-		

2.5.3 ID-range 44000-44249

ID 44000

FB error: axis-specific PLCopen interface couldn't adopt a new request. Request FIFO full. >			
Description	Jobs of axis-specific PLCopen function blocks are managed by a FIFO. The size of this FIFO is fixed (HLI_PLCO_AXIS_REQUEST_NUM). If more jobs are written to the FIFO in a PLC cycle than the space available, this error code is output to the function block whose job can no longer be sent to the motion controller.		
Response	Class	-	The job is discarded by the function block and not transferred to the motion controller.
Solution	Class	-	Check if it is necessary for more jobs to be sent to the motion controller within a PLC cycle than places are available in the FIFO to solve the motion task.
Error type	-		

ID 44001

FB error: axis reference changed, but former order isn't acknowledged.			
Description	A job is to be sent via an FB to the motion controller. The commanded FB also sets the “Error” output to TRUE in the same PLC cycle and displays this error message at the “ErrorID” output since the FB instance already sent a job for another axis that is still active. Therefore, a different variable of type AXIS_REF was assigned to the “Axis” input and a rising edge was applied to the FB “Execute” input. This is not permissible, otherwise the FB does not receive the job acknowledgement for the previously commanded axis and continues to be enabled.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	In general there are 3 possibilities to respond to this error code: 1. wait until the ongoing job is completed and then start the new job for the required axis. 2. interrupt the ongoing job by an instance of MC_Halt or MC_Stop. When the axis is then in the state of **FB error: axis reference changed, but former order isn't acknowledged.**, the job for the required axis can then be sent via the FB. 3. the implementation is designed so that an axis is always assigned a definite FB instance. This means that changes do not occur at the “Axis” inputs of the FB.
Error type	-		

ID 44002

FB error: order isn't acknowledged within a defined number of PLC cycles.			
Description	After a defined number (MAX_RESET_WAIT_CYCLES) of PLC cycles, the job is still active and the function block did not receive the acknowledge information for that job.		
Response	Class	-	The "Error" and "ErrorID" outputs of the function block indicate the error situation. Function block switches to its idle state and is able to command a new job in the next PLC cycle.

Solution	Class	-	Check whether the motion controller is still running and calls its tasks cyclically. If the motion controller fails to run, restart the control system, otherwise contact your control system supplier.
Error type	-		

ID 44003

FB error: position exceeds allowed range.			
Description	With the FBs MC_Home, MC_MoveAbsolute the value at the input “Position” is checked before the command is sent to the motion controller. This error code is output if the value is outside the interval of [-2147483648.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “Position” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44004

FB error: velocity exceeds allowed range.			
Description	With FBs MC_MoveAbsolute, MC_MoveAdditive, MC_MoveRelative, MC_MoveSuperImposed, MC_MoveVelocity and MC_Phasing, the value at the input "Velocity” or “VelocityDiff” is checked before a command is sent to the motion controller. This error message is output If the value exceeds the interval [1.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “Velocity” or VelocityDiff” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44005

FB error: acceleration exceeds allowed range.			
Description	With FBs MC_MoveAbsolute, MC_MoveAdditive, MC_MoveRelative, MC_MoveSuperImposed, MC_MoveVelocity, MC_GearIn and MC_Phasing the value at the input variable “Acceleration” is checked before a command is sent to the motion controller. This error message is output If the value exceeds the interval [1.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “Acceleration” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44006

FB error: deceleration exceeds allowed range.

Description	With FBs MC_MoveAbsolute, MC_MoveAdditive, MC_MoveRelative, MC_Stop, MC_MoveSuperImposed, MC_MoveVelocity, MC_GearIn and MC_Phasing the value at the input “Deceleration” is checked before a command is sent to the motion controller. This error message is output If the value exceeds the interval [1.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “Deceleration” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44007

FB error: MC_Home. Homing is still active, but is commanded again.			
Description	The function block outputting this error message is to send a job to the motion controller to home an axis. However, this function block has already sent such a job to the motion controller and this job is still enabled as the motion controller has not yet sent an acknowledgement for this job.		
Response	Class	-	The function block displays the error situation at his outputs "Error" and "ErrorID" and then waits for the acknowledge for the active job that is still active.
Solution	Class	-	Prevent the command for homing as long as homing is still active.
Error type	-		

ID 44008

FB error: MC_Stop. Stop command is still active, but is commanded again.			
Description	This is an error of an instance of the MC_Stop. The FB was commanded and sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because this FB instance already commanded a stop of the axis, but the motion controller hasn't finished up to now this stopping operation.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	If the error code is displayed, it is necessary to wait until the previous commanded stopping operation is acknowledged before placing a new job. In general, it must be prevented that a stop operation is ordered via an instance of the MC_Stop as long as a previous stopping order is still active.
Error type	-		

ID 44009

FB error: MC_TouchProbe. Touch probing is still active, but commanded again.			
Description	This is an error of an instance of the MC_TouchProbe The FB was commanded and sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the FB is still waiting for acknowledgement of a previously commanded measurement job and then is commanded again to execute a measurement job again.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	Commanding a probing job must be prevented before a previously commanded probing job is acknowledged by the FB.

Error type	-

ID 44010

FB error: MC_AbortTrigger. Aborting touch probe is still active, but commanded again.			
Description	This is an error of an instance of the MC_AbortTrigger. The FB was commanded and sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the FB is still waiting for acknowledgement of a previously commanded abort of a measurement job and then is commanded again to execute an abort.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	A command to abort a measurement process must be prevented before receipt of the acknowledgement of a measurement process that was already aborted.
Error type	-		

ID 44011

FB error: MC_TouchProbe. Touch probe channel changed, before the active request was acknowledged.			
Description	A job for measuring the position was already sent to the Motion Controller via the MC_TouchProbe function block. This job is still enabled because the function block has not yet received an acknowledgement for the job. Now, another probing job is to be sent to the motion controller but the reference to the probing signal source was changed. This information is part of the structure which is assigned at the input "TriggerInput" and is stored in the variable **FB error: MC_TouchProbe. Touch probe channel changed, before the active request was acknowledged.**. This is not permitted as long as a job is active.		
Response	Class	-	The new job is discarded and the error situation is displayed at the "Error" and "ErrorID" outputs of the function block. The function block is still waiting for the acknowledge signal of the measuring job previously sent to the motion controller.
Solution	Class	-	As long as a job is enabled, neither the reference to the probing signal source nor the trigger mode of the measurement may be changed.
Error type	-		

ID 44012

FB error: MC_TouchProbe. Touch probe mode changed, before the active request was acknowledged.			
Description	A job for measuring the position was already sent to the Motion Controller via the MC_TouchProbe function block. This job is still enabled because the function block has not yet received an acknowledgement for the job. Now, another measurement job is to be sent to the motion controller, but the trigger condition for the measurement process was changed. This information is part of the structure which is assigned at the input "TriggerInput" and is stored in the variable **FB error: MC_TouchProbe. Touch probe channel changed, before the active request was acknowledged.**. This is not permitted as long as a job is active.		
Response	Class	-	The new job is discarded and the error situation is displayed at the "Error" and "ErrorID" outputs of the function block. The function block is still waiting for the acknowledge signal of the measuring job previously sent to the motion controller.

Solution	Class	-	As long as a job is enabled, neither the reference to the probing signal source nor the trigger mode of the measurement may be changed.
Error type	-		

ID 44013

FB error: order not allowed, because an initialisation error occurred.			
Description	This value is output if the axis state diagram of an axis is in the state 0 (INIT_STATE) when a FB is called. The cause is that the initialization phase of axis specific HLI areas isn't finished, but the PLC application sends an order by commanding a FB that communicates with the motion controller by using this already not initialized interface.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	Command an axis not until the state of the axis is unequal INIT_STATE.
Error type	-		

ID 44014

FB error: order not allowed, because axis is in state STANDSTILL.			
Description	An order should be send via an FB to the motion controller. The commanded FB sets output "Error" to TRUE and displays this error code at the output "ErrorID" in the same PLC cycle, because the axis is in state **Standstill** . This is the case if the ordered axis is at a standstill at this time and is not a slave axis of a synchronized movement. Therefore, it was not previously assigned by an instance of an MC_CamIn or MC_GearIn. If this had been the case, the axis would have been both in the state Standstill and the state **Synchronised Motion**.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	Axes that are at a standstill and are not slave axes of a synchronized movement at the time of commissioning must not be commissioned by an instance of a MC_CamOut, MC_GearOut or MC_Phasing.
Error type	-		

ID 44015

FB error: order not allowed, because axis is in state HOMING.			
Description	A job is to be sent via an FB to the motion controller. The commanded FB also sets the "Error" output to TRUE and displays this error code at the output "ErrorID" in the same PLC cycle, because the axis already works off an order from an MC_Home instance and therefore is in state **Homing** .		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	In general, there are two possibilities to handle the situation: 1. wait until the job is finished, then transfer the axis into the state **Standstill** . 2. homing is aborted by a job to a MC_STOP. Afterwards, it's possible to reassign the FB that notified the error code.
Error type	-		

ID 44016

FB error: order not allowed, because the axis is in state STOPPING.			
Description	An order should be send via an FB to the motion controller. The commanded FB sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the axis is in state **Stopping** . In the state **Stopping** an axis is located after a job by an instance of the MC_Stop and also when it is at a standstill and the "Execute" input of the MC_Stop has the value TRUE.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	An order by a MC_Stop instance can’t be aborted by another order. Therefore, you have to wait until the axis stands still. Before a new assignment by another FB, however, it must also be ensured that the "Execute" input has the value FALSE, because otherwise the axis remains in the state **Stopping** . Only on the falling edge at the “Execute” input sets the axis to the state **Standstill**. Afterwards, it’s possible to reassign the FB that notified the error code.
Error type	-		

ID 44017

FB error: order not allowed, MC notifies already an axis error.			
Description	An order should be send via an FB to the motion controller. The commanded FB sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the axis is in state **Errorstop** since the motion controller has already signalled an error for this axis.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	For the affected axis, a job must be sent via an instance of the FB MC_Reset. This acknowledges the reported error and sets the axis to the state **Standstill** . Now the axis is able to execute new orders from FB.
Error type	-		

ID 44018

FB error: order not allowed, because axis is in state DISCRETE_MOTION.			
Description	An order should be send via an FB to the motion controller. The commanded FB also sets the “Error” output to TRUE in the same PLC cycle and displays this error message at the “ErrorID” output because a motion command, triggered by an instance of an FB MC_MoveAbsolute, MC_MoveAdditive, MC_MoveRelative or MC_MoveSuperImposed is still active for this axis. This order had not yet been acknowledged with “Done” = TRUE.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller.

Solution	Class	-	General there exist three possibilities to handle the situation: 1. wait until the job is finished, then set the axis to the state **Discrete Motion** . 2. a new job is sent for the axis concerned which cancels the job that led to the state **Discrete Motion** . The job must therefore be commanded in the state **Discrete Motion** but may not cause the state **Discrete Motion** of the axis in active state. These are, for example, requests to instances of the FB types MC_Halt and MC_Stop. If the axis is to be converted to another state, please refer to the PLCopen axis state diagram (see [MCP-P1//PLCopen axis state model]). 3. The commanded FB is an instance of MC_Reset. FB of this type can only send an order successfully if the axis is in state **Errorstop** . Afterwards, it's possible to reassign the FB that notified the error code.
Error type	-		

ID 44019

FB error: order not allowed, because axis is in state CONTINUOUS_MOTION.			
Description	An order should be send via an FB to the motion controller. The commanded FB also sets the "Error" output to TRUE in the same PLC cycle and displays this error message at the "ErrorID" output because a motion command, triggered by an instance of an FB MC_MoveVelocity, MC_CamOut or MC_GearOut is still active for this axis. This order had not yet been acknowledged with "Done" = TRUE.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	In general, there are two possibilities to handle the situation: 1. a new job is placed for the axis concerned, which leads to a change in the axis state. The job must therefore be commanded in the state **Continuous Motion** . These are, for example, jobs from instances of the FB types MC_Halt and MC_Stop. If the axis is to be converted to another state, please refer to the PLCopen axis state diagram (see [MCP-P1//PLCopen axis state model). 2. The commanded FB is an instance of MC_Reset. FB of this type can only send an order successfully if the axis is in state **Errorstop** . Afterwards, it's possible to reassign the FB that notified the error code.
Error type	-		

ID 44020

FB error: order not allowed, because axis is in the state SYNCHRONIZED_MOTION.			
Description	A job is to be sent via an FB to the motion controller. The commanded FB sets output "Error" to TRUE and displays this error code at the output "ErrorID" in the same PLC cycle, because the axis is in state **Synchronised Motion** . The state **Synchronised Motion** results from assigning the axis via MC_CamIn or MC_GearIn.		
Response	Class	-	The order is dismissed by the FB and not issued to the motion controller.

Solution	Class	-	In general, there are two possibilities to handle the situation: 1. A new job is placed for the axis concerned, which leads to a change in the axis status. The job must therefore be commanded in the state **Synchronised Motion** . These are, for example, jobs from instances of the FB types MC_Halt and MC_Stop. If the axis is to be converted to another state, please refer to the PLCopen axis state diagram (see [MCP-P1//PLCopen axis state model]). 2. The commanded FB is an instance of MC_Reset. FB of this type can only send an order successfully if the axis is in state **Errorstop** . Afterwards, it is possible to reassign the FB that notified the error code.
Error type	-		

ID 44021

FB error: order not allowed, because axis is in unallowed state.			
Description	Before a job for an axis is sent to the motion controller, it is checked if the job is allowed. If this error message is output, the axis is in a state that is unknown and therefore not allowed. Note: The PLCopen specification defines an axis state graph that defines states for an axis controlled by PLCopen function blocks. Furthermore the diagram there defines which PLCopen commands change the actual axis state and which PLCopen commands are allowed for the actual state of the axis.		
Response	Class	-	The "Error" and "ErrorID" outputs of the function block indicate the error situation. Function block is able to command a new job in the next PLC cycle.
Solution	Class	-	Please contact your motion control supplier and mention the error number.
Error type	-		

ID 44022

FB error: order not allowed, because axis state is invalid.			
Description	A job should be send via an FB to the motion controller. The commanded FB sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the commanded axis is in a state that isn't covered by the implementation.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	Check whether the value of the element axsd_state in variable of type AXIS_REF that is applied at input “Axis” is changed by the PLC application, although there is no job enabled for this axis. If that is not the case, please contact the controller supplier and quote the error number.
Error type	-		

ID 44024

FB error: order not allowed, because touch probing is still active.

Description	A job is to be sent to the motion controller via an instance of the FB MC_TouchProbe. The commanded FB sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the measurement channel that is applied at input “TriggerInput” is occupied by a touch probing job that was sent before.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	In general, there are two possibilities to handle the situation: 1. the previous touch probing job is aborted by an instance of the MC_AbortTrigger. 2. wait until the previous touch probing job is finished. Afterwards, it’s possible to reassign the FB that notified the error code.
Error type	-		

ID 44025

FB error: order not allowed, because aborting touch probing is still active.			
Description	This is an error of an instance of the MC_TouchProbe or MC_AbortTrigger. The commanded FB also sets the “Error” output to TRUE in the same PLC cycle and displays this error message at the “ErrorID” output because a touch probing order is still active and not acknowledged until now.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	If the error code is displayed, wait until the previous commanded probing abort is acknowledged before placing a new job. In general, a probing process must be prevented from starting or the abort of a probing process must be prevented as long as an order to abort a probing process is still active.
Error type	-		

ID 44026

FB error: order not allowed, because MC_TouchProbe is in an undefined state.			
Description	A job is to be sent to the motion controller via an instance of an MC_TouchProbe or MC_AbortTrigger. The commanded FB sets output “Error” to TRUE and displays this error code at output “ErrorID” in the same PLC cycle because the measurement channel that is applied to input “TriggerRef” is in a state that is not covered by the implementation.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Check whether the value of the element tp_state.tpsd_state is changed by the PLC application in the variable of type AXIS_REF applied to the "Axis" input, even though there is no active job for probing positions or aborting a probing job. If that is not the case, please contact the controller supplier and quote the error number.
Error type	-		

ID 44027

FB error: order not allowed, because input "Execute" of any MC_STOP instance is TRUE.			
Description	A job is to be sent via an FB to the motion controller. The commanded FB sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the input “Execute” is currently set to TRUE for one or more instances of the FB MC_Stop affecting this axis.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Check at which instance of a FB of type MC_Stop the input "Execute" is set to TRUE and the axis reference is the same as at the FB that displays this error message. Then set the value of the input "Execute" of this instance of the MC_Stop to FALSE. Afterwards, the FB that returned this error code can be reassigned.
Error type	-		

ID 44028

FB error: order not allowed, because axis is in state DISABLED			
Description	A job is to be sent via an FB to the motion controller. The commanded FB sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the axis is in state **Disabled** .		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	Send a job to the affected axis via an instance of the FB MC_Power.
Error type	-		

ID 44029

FB error: job number 0 is not allowed.			
Description	The value 0 was assigned to the input "JobNr” of the function block. However, this is an invalid value for the job number.		
Response	Class	-	The job triggered on the rising edge at the "Execute" input is not forwarded to the motion controller.
Solution	Class	-	Assign a job number > 0 at input "JobNr” and execute the function block again.
Error type	-		

ID 44030

FB error: job nr at HLI for acknowledge differs from job nr of active job.			
Description	The function block has the task of confirming the confirmation for an order because this blocks the confirmation interim storage area. To do this, the order number of the order to be acknowledged is transferred to the function block at input "JobNr". The error messages is displayed, because the acknowledge interface contains a response with the job identification number of another job.		
Response	Class	-	The response present in the acknowledge interface won't be acknowledged.
Solution	Class	-	Check the job number of the response that blocks the acknowledgement interface, and assign this number to "JobNr" input. Then execute the function block again.

Error type	-

ID 44031

FB error: want acknowledge active job, but at HLI no job is notified for acknowledge.			
Description	The function block has the task of confirming the confirmation for an order because this blocks the confirmation interim storage area. During execution of the job it's detected that there doesn't exist a not acknowledged response in the job acknowledge interface.		
Response	Class	-	Display error without having an effect on the motion controller.
Solution	Class	-	Only command function block if job acknowledge interface is blocked.
Error type	-		

ID 44032

FB error: MCV_ForceJobAck: For this purpose the value assigned to that input should be > 0.			
Description	The function block should check if the acknowledge interface is occupied by one and the same response of a job for a number of PLC cycles in which the number of cycles is assigned to the function blocks input "NrChkCycles". The function block requires a value > 0 at input "NrChkCycles" for this purpose. But in case the error is displayed the value is 0.		
Response	Class	-	The error is displayed and in the following PLC cycle the output "JobBlocks" is set TRUE.
Solution	Class	-	Assign a value > 0 to input "NrChkCycles".
Error type	-		

ID 44033

FB error: value at input "Distance" is out of range.			
Description	With the FBs MC_MoveAdditive, MC_MoveRelative, MC_MoveSuperImposed the value at the input “Distance” is checked before a command is sent to the motion controller. This error code is output if the value exceeds the interval [-2147483648.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “Distance” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44034

FB error: value at input "Jerk" is out of range.			
Description	With the FBs MC_MoveAbsolute, MC_MoveAdditive, MC_MoveRelative, MC_Stop, MC_MoveSuperImposed, MC_MoveVelocity, MC_GearIn and MC_Phasing the value at the input “Jerk” is checked before a command is sent to the motion controller. This error code is output if the value is outside the interval of [1.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “Jerk” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.

Error type	-

ID 44035

FB error: value at input "MasterOffset" is out of range.			
Description	Specific error message of the FB MC_CamIn. The value at the input is checked before the command is sent to the motion controller. This error code is output if the value is outside the interval [-2147483648.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “MasterOffset” so that it lies within the limits of the permitted interval. Afterwards, the FB can be commanded again.
Error type	-		

ID 44036

FB error: value at input "SlaveOffset" is out of range.			
Description	Specific error message of the FB MC_CamIn. The value at input "SlaveOffset" is checked before the Motion Controller is commanded. This error code is output if the value of “SlaveOffset” input exceeds interval [-2147483648.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “SlaveOffset” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44037

FB error: value at input “MasterScaling” is out of range.			
Description	Specific error message of the FB MC_CamIn. The value at the “MasterScaling” input is checked before the command is sent to the motion controller. This error code is output if the value is outside the interval [-2147483648.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “MasterScaling” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44038

FB error: value at input "SlaveScaling" is out of range.	
Description	Specific error message of the FB MC_CamIn. The value at the "SlaveScaling" input is checked before the Motion Controller is commanded. This error code is output if the value is outside the interval [-2147483648.0, 2147483647.0].

Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “SlaveScaling” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44039

FB error: value at input "PhaseShift" is out of range.			
Description	Specific error message of the FB MC_Phasing. The value at the “PhaseShift” input is checked before the command is sent to the motion controller. This error code is output if the value is outside the interval [-2147483648.0, 2147483647.0].		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “PhaseShift” so that it lies within the limits of the permitted interval. Afterwards, the FB can be reassigned.
Error type	-		

ID 44040

FB error: value at input "RatioDenominator" is 0.			
Description	With this function block, the value at input "RatioDenominator" is checked at rising edge at input "Execute" before the command is sent to the motion controller. Because the value is the denominator of a gear ratio the value cannot not be 0, otherwise this error code is output.		
Response	Class	-	The order is dismissed by the function block and not issued to the motion controller.
Solution	Class	-	Adjust the value at input “RatioDenominator” so that it is not equal to 0. Afterwards, it’s possible to reassign the function block.
Error type	-		

ID 44043

FB error: value 0 at input "GearNr" is not a valid identification number for a gear.			
Description	This function block checks the value at input "GearIn" on the rising edge at input "Execute" before the command is sent to the motion controller. This value is the identification number of a gear stage. According to the definition, this identification number is an integer and > 0.		
Response	Class	-	The job is discarded by the function block and not transferred to the motion controller.
Solution	Class	-	Adjust the value at input “GearIn” so that it is > 0 and an integer. Afterwards, the function block can be reassigned.
Error type	-		

ID 44044

FB error: MC_GearShift. Gear shift command is still active, but is commanded again.

Description	This is an error of an instance of the MCV_ShiftGear. The FB was commanded and sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because this FB instance already commanded shift of the gear stage of an axis, but the motion controller hasn't finished up to now the operation of shifting the gear.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	If the error code is displayed, wait until the previous commanded gear stage operation is acknowledged before placing a new job. In general, it must be prevented that an instance of the MCV_ShiftGear is used to order a gear change as long as a previous gear shift order sent via this FB instance is still active.
Error type	-		

ID 44045

FB error: Logical axis number less than 1 is invalid.			
Description	The function block checks the value that should be a logical axis number. Valid logical axis numbers in the environment of the motion controller are integer values greater than 0 according to the definition. The error code is displayed because the value doesn't match the criteria.		
Response	Class	-	The function block displays the error. No further evaluation will be executed and no command will be sent to the motion controller.
Solution	Class	-	It's a function block • that gets the value for the logical axis number by an input of the function block, the value has to be changed according to the definition for logical axis numbers (>0, integer). • that doesn't get the value by the PLC application, but internal accesses the value, please contact your control system supplier and tell the function block type displaying this error.
Error type	-		

ID 44046

FB error: MC error occurred while reading/writing of configuration parameters.			
Description	Specific error of FB that handle reading/writing of parameters.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44047

FB error: MC warning occurred while reading/writing of configuration parameters.			
Description	The message is issued by function blocks that can read/write to parameters in the configuration lists because the motion controller has issued a warning when accessing parameters in the configuration lists.		
Response	Class	-	The message code is displayed at output "ErrorID" oft he function block. The outputs "Error" and "NCWarning" are set to TRUE. The warning code generated in the motion controller is displayed at output "NCErrorCode" Output "Valid" displays FALSE.

Solution	Class	-	Determine the problem that caused the warning of the motion controller by checking the code at output "NCErrorCode" and fix the problem. Afterwards on rising edge at input "Execute" the function block can send further commands to the motion controller.
Error type	-		

ID 44049

FB error: MCV_SetDriveMode. Value at input "DriveMode" is invalid.			
Description	Function block MCV_SetDriveMode has an input "DriveMode". The operation mode in which the drive is to be operated is then transferred to the operator. The value transferred on the rising edge at the "Execute" input is invalid for this operation mode.		
Response	Class	-	The job triggered with the rising edge at the "Execute" input is not forwarded to the motion controller.
Solution	Class	-	Assign input "DriveMode" a valid value for the drive mode and reassign the function block again. Valid values for the drive operation modes are:

Global constant	**Value**	**Meaning**
HLI_LR_DRV_POS_CTRL	0	Position control
HLI_LR_DRV_VEL_CTRL	1	Velocity control
HLI_LR_DRV_TORQ_CTRL	2	Torque control
HLI_LR_DRV_POS_TORQ_CTRL	3	Position control with torque pilot control

Error type	

ID 44050

FB error: order not allowed, because axis is in state TORQUE_CONTROL.			
Description	A job is to be sent via an FB to the motion controller. The commanded FB sets output “Error” to TRUE and displays this error code at the output “ErrorID” in the same PLC cycle, because the axis is in state **Torque Control** . The state **Torque Control** results from the assignment of the axis via MCV_SetDriveMode.		
Response	Class	-	The job is dismissed by the FB and not issued to the motion controller.
Solution	Class	-	The state **Torque Control** can only be changed by a job sent by an instance of MCV_SetDriveMode, whereas a value not equal to 2 (HLI_LR_DRV_TORQ_CTRL) must be applied at input “DriveMode”. After this job is received, the axis is in state **Standstill**. Afterwards, it is possible to reassign the FB that notified the error code.
Error type	-		

ID 44052

FB error: Axis reference has to point to a SAI axis or reference.

Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44053

FB error: Axis reference has to point to a SAI axis, a axes group axis or an IO station.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44054

FB error: Axis reference has to point to a SAI axis or an encoder axis.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44055

FB error: Axis reference has to point to a SAI axis or an axes group axis.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44056

FB error: Axis reference points to an axis that isn't configured as spindle.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44057

FB error: MC_Power. Power can't be switched on for a CANopen drive, because a drive bus error occurred.	
Description	The error is output because the function block is to send a job to the motion controller that leads to data being exchanged via the fieldbus using the CANopen protocol. However, the fieldbus is not ready for cyclic data transmission.

Response	Class	-	As long as the error is present, the function block sets the "Error" output to TRUE and outputs the error code at the "ErrorID" output. It makes a cyclic check whether the fieldbus is ready to communicate. If it then detects that the fieldbus is ready for cyclic data transmission, its subsequent reaction depends on the type of the function block. If the type is MC_Power, the reaction depends on the values assigned to inputs "Enable", "Enable_Positive" and "Enable_Negative". If these inputs are still assigned TRUE, the drive is enabled as long as no other error occurs.
Solution	Class	-	If the function block is of type MC_Power, nothing else needs to be done since the error is only output if the inputs "Enable", "Enable_Positive" and "Enable_Negative" are set to TRUE. You can still wait and check when the state of output "Status" displays TRUE as soon as the drive is switched on and under control.
Error type	-		

ID 44058

FB error: MC_Power. CANopen drive isn’t ready to switch power on.			
Description	The error is output because the function block MC_Power commands the motion controller to bring the drive of an axis under control, but via its status data (2#0000000 = GC_CO_NOT_READY_TO_SWITCH_ON) the drive signals that the controller is still performing self-tests and the CAN communication is not yet working.		
Response	Class	-	As long as the drive sends this status information, the function block sets the output "Error" TRUE and displays the error code at output "ErrorID". Its subsequent reaction depends on the values assigned to inputs "Enable", "Enable_Positive" and "Enable_Negative" and the status that the drive sends to the motion controller.
Solution	Class	-	Nothing else needs to be done since the error is only output if the inputs "Enable", "Enable_Positive" and "Enable_Negative" are set to TRUE. You can still wait and check when the state of output "Status" displays TRUE as soon as the drive is switched on and under control.
Error type	-		

ID 44059

FB error: MC_Power. Power for a CANopen drive can't be switched on, because drive release isn't set by the hardware.			
Description	The error is output because the function block MC_Power commands the motion controller to bring the drive of an axis under control, but via its status data (2#0100000 = GC_CO_NO_DRIVE_RELEASE) the drive signals that the drive has not yet been enabled by the controller.		
Response	Class	-	As long as the drive sends this status information, the function block sets the output "Error" TRUE and displays the error code at output "ErrorID". Its subsequent reaction depends on the values assigned to inputs "Enable", "Enable_Positive" and "Enable_Negative" and the status that the drive sends to the motion controller.

Solution	Class	-	Nothing else needs to be done since the error is only output if the inputs "Enable", "Enable_Positive" and "Enable_Negative" are set to TRUE. You can still wait and check when the state of output "Status" displays TRUE as soon as the drive is switched on and under control.
Error type	-		

ID 44060

FB error: MC_Power. State machine implemented within CANopen drive doesn't react as expected.			
Description	When a CANopen drive is switched on, the drive controller passes through several states. The correct sequence of states is monitored in the function block MC_Power. The error is output because the combination of previous and current states may not occur during the power-on sequence. The sequence of states during drive power-on is: 1. GC_CO_SWITCH_ON_DISABLED_1 or GC_CO_SWITCH_ON_DISABLED_2 2. GC_CO_READY_TO_SWITCH_ON 3. GC_CO_SWITCHED_ON 4. GC_CO_OPERATION_ENABLED.		
Response	Class	-	As long as the drive sends this status information, the function block sets the output "Error" TRUE and displays the error code at output "ErrorID". Its subsequent response depends on the values assigned to inputs "Enable", "Enable_Positive" and "Enable_Negative" and the status data sent by the drive.
Solution	Class	-	Check why the power-on sequence was interrupted. Once the problem has been solved, you can command the MC_Power again.
Error type	-		

ID 44062

FB error: MC_GearIn A value of the data type MC_GearInMode must be transferred to the input pin “mode”.			
Description	The function block MC_GearIn has a "Mode" input to determine how the slave axis and master axis are coupled. The error message is output because an invalid value is assigned to input "Mode".		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Assign a valid value to the "Mode" input. Please refer to the description of the function block MC_GearIn [MCP-P1//MC_GearIn] for permissible values. A rising edge at the "Execute" input sends a new job to the motion controller.
Error type	-		

ID 44064

FB error: Functionality is not implemented.			
Description	A function block displays this error message if it has no functionality.		
Response	Class	-	Output this error message.

Solution	Class	-	Do not use this function block in the current PLC application and PLC environment.
Error type	-		

ID 44065

FB error: Axis reference of slave axis has to point to a SAI axis.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44066

FB error: Axis reference of master axis has to point to a SAI axis.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44069

FB error: Repeated command of FB discarded since previous job sent was not acknowledged.			
Description	A job is to be sent via an FB to the motion controller. The commanded FB sets the “Error” output to TRUE in the same PLC cycle and displays this error message at the “ErrorID” output because the FB is supposed to send a new job but is still busy with a job sent earlier.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Either wait until the previous job sent is acknowledged by the FB or an FB was commanded to abort the job previously sent. Afterwards, the FB can be commanded again. Generally, this case can be prevented if you ensure that a new job is not sent via an FB before a previous job already sent is acknowledged.
Error type	-		

ID 44070

FB error: value at input of type LREAL exceeds allowed value range of -2147483648.0 .. 2147483647.0.			
Description	The function block outputting this error message has at least one input of type LREAL. A value that is outside the interval [-2147483648.0, 2147483647.0] was assigned to one of these inputs. The value range of this input is limited because the value is assigned internally to a REAL variable.		
Response	Class	-	The job is discarded by the FB and not transferred to the motion controller.
Solution	Class	-	Assign a valid value to the corresponding input. A rising edge at the "Execute" input sends a new job to the motion controller.

Error type	-

ID 44071

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 44075

FB error: Axis specific PLCopen interface can't take order, because it's occupied by another order.			
Description	A function block that uses an axis specific PLCopen control unit to command jobs to the motion controller can't send a job because the control unit is occupied. The error is displayed after function block has made further attempts unsuccessfully to send the job. The number of attempts is defined by global constant MAX_RETRIALS. By default MAX_RETRIALS = 0. Therefore, this error usually occurs when more than one function block places an order per PLC cycle and these are to be sent to the motion controller via the same axis specific PLCopen control unit.		
Response	Class	-	The job isn't executed.
Solution	Class	-	Check whether more than one function block sends jobs to the Motion Controller during a PLC cycle via the same axis-specific PLCopen Control Unit. If this is the case, change the PLC application accordingly so that this no longer occurs.
Error type	-		

ID 44076

FB error: MCV_ConveyorControl; Value at input "ControlMode" is invalid.			
Description	There is an invalid value assigned to input "ControlMode" of function block MCV_ConveyorControl. Allowed values at input "ControlMode" are:		
	Global constant		**FB error: MCV_ConveyorControl; Value at input "ControlMode" is invalid.**
	HLI_CONVEYOR_SPEED_CONST		0
	HLI_CONVEYOR_ACCELERATION		1
	HLI_CONVEYOR_DECELERATION		2
	HLI_CONVEYOR_SPEED_ZERO		3
	HLI_CONVEYOR_MACH_WITH_SPEED_ZERO		4
Response	Class	-	The invalid value is not sent to the motion controller via the corresponding control unit.
Solution	Class	-	Correct the assigned value. The corrected value is sent immediately to the motion controller if input "Enable" is TRUE.

Error type	-

ID 44077

FB error: The motor is decoupled already.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44078

FB error: The motor is coupled already.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44079

FB error: For coupling or decoupling, MC_Power must be disabled.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 44082

FB error: MC_Power. Invalid value at input PowerDefaultState, a valid value is used.			
Description	The drive was switched ready for operation using MC_Power. To switch off the drive, at least one Enable_... input of the MC_Power was set to FALSE. When the MC_Power function checks the shutdown job, it detects that the value at the PowerDefaultState input determining the target state at shutdown is invalid. This would prevent the drive from being switched off.		
Response	Class	-	The FB corrects the invalid value at the PowerDefaultState input internally to the valid value **mcvPowerStateReadyForPower**. By applying this value, the drive can be shut down so that it does not react to command values. If a valid value is applied to input PowerDefaultState, the FN can be commanded again. The output Error displays FALSE and ErrorID displays no error.
Solution	Class	-	The values at input PowerDefaultState must be of type **FB error: MC_Power. Illegal value at input PowerDefaultState, a valid value is used.**. Please apply one of these values. If the applied value was calculated by the application, check why the value exceeds the allowed range.
Error type	-		

ID 44083

FB error: MC_Power. Illegal value at input PowerDefaultState with drive ready for operation.			
Description	The drive was set to operational state using a MC_Power function block. TRUE is still assigned to all Enable... inputs and the drive is still ready for operation. It is now detected that an invalid value was applied to the PowerDefaultState input of the function block.		
Response	Class	-	No response, only information. If a valid value is applied to the input PowerDefaultState, the output Error displays FALSE and ErrorID displays no error.
Solution	Class	-	The values at input PowerDefaultState must be of type **FB error: MC_Power. Illegal value at input PowerDefaultState with drive ready for operation.** Please apply one of these values. If the applied value was calculated by the application, check why the value exceeds the allowed range.
Error type	-		

ID 44084

FB error: MC_Power. Illegal value at input PowerDefaultState with the drive not ready for operation.			
Description	The drive is to be switched to operational state using a MC_Power function block. At least one of the Enable... inputs is still assigned FALSE and the drive is not ready for operation. It is now detected that an invalid value was applied to the PowerDefaultState input of the function block.		
Response	Class	-	No response, only information as long as at least one Enable_... input has the value FALSE. If all Enable_... inputs are set to TRUE, the drive is not switched ready for operation. If a permitted value is applied to the PowerDefaultState input, the Error output displays FALSE and ErrorID displays no error detection. If TRUE is assigned to all Enable_... inputs, the drive is switched to operational state at this moment in time.
Solution	Class	-	The values at input PowerDefaultState must be of type **FB error: MC_Power. Illegal value at input PowerDefaultState with drive not ready for operation.** Please apply one of these values. If the applied value was calculated by the application, check why the value exceeds the allowed range.
Error type	-		

ID 44085 / 44086

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	No job is executed by the motion controller.
Solution	Class	8	Requires controller restart.

2.5.4 ID-range 46000-46249

ID 46000

FB error: axis group specific PLCopen interface couldn't adopt a new request. Request FIFO full.			
Description	The jobs of the PLCopen function blocks are managed via a FIFO. The size of this FIFO is fixed. If more jobs are written to the FIFO in a PLC cycle than the space available, this error code is output to the function block whose job can no longer be sent to the motion controller.		
Response	Class	-	The job is discarded by the function block and not transferred to the motion controller.
Solution	Class	-	Check if it is necessary for more jobs to be sent to the motion controller within a PLC cycle than places are available in the FIFO to solve the motion task.
Error type	-		

ID 46001

System error: An unknown PLCopen Part4 function block type was requested.			
Description	Each job transmitted to the Motion Controller contains the type of function block responsible for that job. This error message is output if a job contains a function block type that is unknown to the motion controller.		
Response	Class	-	The error is displayed at the outputs "Error" and "ErrorID" of the function block. The motion controller does not react to the unknown function block type. The motion controller can accept other jobs and execute them.
Solution	Class	-	Please contact your motion controller supplier.
Error type	-		

ID 46002

FB error: order not allowed, because axis group is not in state GroupDisabled.			
Description	A job was sent via the MC_GrpEnable. According to the axis group state diagram defined in the PLCopen Part 4 specification, a job for this function block is only permissible if the axis group is in the GroupDisabled state. The axis group is therefore in a different axis group state when this error is output.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	Only command the function block MC_GrpEnable if the axis group is in GroupDisabled state. The state of the axis group is displayed by the function block MC_GrpReadStatus.
Error type	-		

ID 46003

FB error: order not allowed, because axis group is in state GroupDisabled or GroupErrorStop.			
Description	A function block that can send jobs for axis groups to the motion controller has sent a job, but the axis group is either in GroupDisabled or GroupErrorStop state. Therefore the job is not executed.		
Response	Class	-	The job results in no response from the motion controller.

Solution	Class	-	Only command the function block if the axis group is either in GroupDisabled or GroupErrorStop state. The state of the axis group is displayed by the function block MC_GrpReadStatus.
Error type	-		

ID 46004

FB error: order not allowed, because axis group is not in state GroupDisabled.			
Description	A function block that can send jobs for axis groups to the motion controller has sent a job, but the axis group is in GroupStandby state. Therefore the job is not executed.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	Only command the function block only if axis group is in GroupStandby state. The state of the axis group is displayed by the function block MC_GrpReadStatus.
Error type	-		

ID 46005

FB error: order not allowed, because axis group is neither in state GroupStandby nor in state GroupMoving.			
Description	A function block that can send jobs for axis groups to the motion controller has sent a job, but the axis group is neither in GroupStandby state nor in GroupMoving state. Therefore the job is not executed.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	Only command the function block if the axis group is either in GroupStandby or GroupMoving state. The state of the axis group is displayed by the function block MC_GrpReadStatus.
Error type	-		

ID 46006

FB error: order not allowed, because axis group is in state GroupErrorStop.			
Description	A function block that can send jobs for axis groups to the motion controller has sent a job, but the axis group is in the GroupErrorStop state. Therefore the job is not executed.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	Only command the function block if the axis group is not in GroupErrorStop state. The state of the axis group is displayed by the function block MC_GrpReadStatus.
Error type	-		

ID 46007

FB error: order not allowed, because axis group is not in state GroupErrorStop.			
Description	A function block of type MC_GrpReset, has sent a job to the motion controller, but the axis group is not in the GroupErrorStop state. Therefore the job is not executed.		
Response	Class	-	The job results in no response from the motion controller.

Solution	Class	-	Only command the function block if the axis group is in GroupErrorStop state. The state of the axis group is displayed by the function block MC_GrpReadStatus.
Error type	-		

ID 46008

FB error: order not allowed, because axis group is not in state GroupStopping.			
Description	A function block that can send jobs for axis groups to the motion controller has sent a job, but the axis group is not in the GroupStopping state. Therefore the job is not executed.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	Only command the function block only if the axis group is in GroupStopping state. The state of the axis group is displayed by the function block MC_GrpReadStatus.
Error type	-		

ID 46009

FB error: order not allowed, because given axis index is out of range.			
Description	The function block outputting this error message is processing information concerning the arrangement of an axis within an axis group. The system detects that the value defining the position of the axis in the axis group is outside the permissible limits. This value is specified as the placement index and is usually stored in a structure of type IDENT_IN_GROUP_REF in the variable "ChAxIdx". The structure is assigned to input "IdentInGroup" of the function block.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	The value stored in variable "ChAxIdx" that is member of a structure of type IDENT_IN_GROUP_REF must meet the following condition: ChAxIdx ϵ [0, HLI_CH_AX_MAXIDX]. The global constant HLI_CH_AX_MAXIDX is defined in the HLI library.
Error type	-		

ID 46010

FB error: order not allowed, because such a function isn't available for an axis group.			
Description	A function block displays this error message if it is intended for assigning axis groups, but cannot or may not execute its implemented function in the PLC runtime environment used.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	The function block may not be implemented in the PLC application.
Error type	-		

ID 46011

System error [▶ 12]	
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.

Response	Class	6	The FB is not initialized, and is left immediately. No job can be sent to the Motion Controller via this FB instance. The output “Error” is set to TRUE.
Solution	Class	8	Requires controller restart.

ID 46016

FB error: There is no axis with this name in axis group.			
Description	A function block that displays this error message contains axis-specific information that is part of the job for an axis group, and this information is transmitted to the motion controller. The motion controller detects that there is no axis with this name in the assigned axis group. This name is stored as a string in a structure of type IDENT_IN_GROUP_REF in the variable "Name". This structure is transferred to the function block at the "IdentInGroup" input.		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	Correct the string stored in the "Name" variable. Afterwards, the character string must correspond to a name of an axis in the commanded axis group.
Error type	-		

ID 46017

System error: Size of channel-specific requested data exceeds size of the definition at HLI.			
Description	At motion controller start-up, a check is made whether the size of the data structure at the HLI that contains axis group specific jobs transferred by the PLC to the motion controller is the same size as the data structure in the motion controller that adopts this data. The error message is displayed if the data structures differ in size.		
Response	Class	-	No job is executed by the motion controller.
Solution	Class	-	Please contact your motion controller supplier.
Error type	-		

ID 46018

System error: Size of channel-specific responded data exceeds size of the definition at HLI.			
Description	At controller start-up, a check is made whether the size of the data structure at the HLI that contains axis group specific job acknowledgements transferred by the PLC to the motion controller is the same as the size of the data structure in the motion controller that adopts this data. The error message is displayed if the data structures differ in size.		
Response	Class	-	No job is executed by the motion controller.
Solution	Class	-	Please contact your motion controller supplier.
Error type	-		

ID 46019

System error: An unknown function block type was send; platform specific interface.

Description	Each job transmitted to the Motion Controller contains the type of function block responsible for that job. This error message is output if a job contains a function block type that is unknown to the motion controller. In this case the job was sent via the platform specific area of the interface HLI to the motion controller.		
Response	Class	-	The error is displayed at the output "Error" and "ErrorID" of the function block. The motion controller doesn't react to the unknown function block type. The motion controller can accept other jobs and execute them.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 46020

FB error: the axis group is not in GroupStandby or GroupDisabled state.			
Description	A function block that can send jobs for axis groups to the Motion Controller has sent a job, but the axis group is neither in GroupStandby state nor in GroupDisabled state. Therefore the job won't be executed.		
Response	Class	-	The motion controller doesn't execute the job.
Solution	Class	-	Command the function block only if axes group is either in GroupStandby or GroupDisabled states. The state of the axis group is displayed at function block MC_GrpReadStatus.
Error type	-		

ID 46021

FB error: counter of axis group errors is negative.			
Description	The function block MCV_GrpReadErrorHistory is used to display errors that occurred since the last reset of the motion controller. The errors are displayed in an array, whereby the oldest error has the index 0 and the most current error always has the highest index, even if the array size is not large enough to display all the errors that occur. The errors that occur must be manage for this procedure. If the counter for the errors that occurred becomes negative,there was a system error.		
Response	Class	-	FB sets output “Error“ to TRUE and output “Valid“ to FALSE.
Solution	Class	-	Please contact the controller supplier and quote the error number.
Error type	-		

ID 46022

FB error: index of axis within axis group at IdentInGroup reference exceeds the limits.			
Description	The function block outputting this error has an input of type IDENT_IN_GROUP_REF. The content of this data structure is used to determine an axis in the axis group. This data structure contains, among other things, the classification of the axis in the axis group. This classification is determined by an index number. This error is displayed because it is determined, during execution of the function block, that the index number is outside the valid range of [0..HLI_SYS_AX_MAXIDX]. The global constant HLI_SYS_AX_MAXIDX is defined in the HLI library [HLI].		
Response	Class	-	The job results in no response from the motion controller.
Solution	Class	-	Choose the index number that defines the place of the axis within the axis group so that it is within the range of [0..HLI_SYS_AX_MAXIDX].

Error type	-

ID 46023 / 46024

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 46025

Number of jobs in PLCopen-job-stack exceeds maximum.			
Description	A PLCopen job is routed through several components of the PLC environment and the motion controller. Each of these components records is in the transmitted job. If a component can no longer enter its ID, the error message is output. It is an indication that more components are involved in mapping the job than were planned.		
Response	Class	-	The error is displayed at the output "Error" and "ErrorID" of the function block and the motion controller dismisses the job.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 46026 / 46027

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 46028

PLCopen Part4 : The specified type of group configuration is not known.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 46029

PLCopen Part4 : The specified data type of group configuration is wrong.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 46030

FB error: reading axis group specific job state list.			
Description	Function block shall read the content of a field containing data concerning the active jobs. If this isn't possible in the implemented number of motion controller cycles the error message is displayed.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. Additionally output "Valid" ist set to FALSE to point that the values of other outputs are outdated.
Solution	Class	-	Implement function block in a task that runs with less PLC cycle time and/or higher priority. If the error is displayed permanently, please contact your motion control supplier.
Error type	-		

ID 46031

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 46032

PLCopen Part4 : Still waiting for acknowledgement of dispatcher.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 46033

Invalid pointer to axis group specific control unit.			
Description	The function block is to send a job for an axis group. To do this, it requires a pointer for access to a control unit of an axis group. However, the pointer is invalid.		
Response	Class	-	The function block sets its output "Error" to TRUE and displays the error number at output "ErrorID". Access is prevented to the control unit and the function block does not send an axis group job to the motion controller.
Solution	Class	-	Please contact your motion controller supplier.
Error type	-		

ID 46034

Error writing CNC object: MCV_OBJ_CMD_BS_TYPE_R.	
Description	The function block is to have read access to the CNC object containing the active block search method. Reading this information results in an error.

Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Valid” output is also set to FALSE indicating that the data displayed at the outputs is invalid
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46035

Error writing CNC object: MCV_OBJ_CMD_BS_TYPE_W.			
Description	The function block is to have write access to the CNC object containing the block search method to be activated. Writing this information results in an error.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46036

Error writing CNC object: MCV_OBJ_CMD_BS_COVERED_DISTANCE_W.			
Description	The value of the “CoveredDistance” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46037

Error writing CNC object: MCV_OBJ_CMD_BS_DISTANCE_PROG_START_W.			
Description	The value of the “DistProgStart” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46038

Error writing CNC object: MCV_OBJ_CMD_BS_AUTO_RETURN_W.			
Description	The value of the “AutoReturn” function block input must be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46039

Error writing CNC object: MCV_OBJ_CMD_BS_NO_HOLD_AT_RETURN_W.			
Description	The value of the “NoHoldAtRestart” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46040

Error writing CNC object: MCV_OBJ_CMD_BS_DEVIATION_MAX_W.			
Description	The value of the “Deviation” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46041

Error writing CNC object: MCV_OBJ_CMD_BS_BREAKPOINT_POSITION_W.			
Description	The value of the “BreakPointPosition” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.

Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46042

Error writing CNC object: MCV_OBJ_CMD_BS_INTERN_BLOCK_COUNT_W.			
Description	The value of the “BlockCount” function block input must be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46043

Error writing CNC object: MCV_OBJ_CMD_BS_BLOCK_NR_W.			
Description	The value of the “BlockNumber” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46044

Error writing CNC object: MCV_OBJ_CMD_BS_BLOCK_NR_PASS_W.			
Description	The value of the “PassCounter” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46045

Error writing CNC object: MCV_OBJ_CMD_BS_START_FILE_OFFSET_W.			
Description	The value of the “StartFileOffset” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46046

Error writing CNC object: MCV_OBJ_CMD_BS_START_FILE_NAME_W.			
Description	The value of the “StartFileName” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46047

Error writing CNC object: MCV_OBJ_CMD_BS_START_PATH_TYPE_W.			
Description	The value of the “StartPathType” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46048

Error writing CNC object: MCV_OBJ_CMD_BS_START_PASS_COUNTER_W.			
Description	The value of the “StartPathCounter” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.

Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46049

Error writing CNC object: MCV_OBJ_CMD_BS_END_FILE_OFFSET_W.			
Description	The value of the “EndFileOffset” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46050

Error writing CNC object: MCV_OBJ_CMD_BS_END_FILE_NAME_W.			
Description	The value of the “EndFileName” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46051

Error writing CNC object: MCV_OBJ_CMD_BS_END_PATH_TYPE_W.			
Description	The value of the “EndPathType” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46052

Error writing CNC object: MCV_OBJ_CMD_BS_END_PASS_COUNTER_W.			
Description	The value of the “EndPassCounter” function block input is to be sent to the motion controller. This occurs via a communication system using the CNC object specified in the error message. An error occurred on transfer.		
Response	Class	-	Function block sets his output "Error" to TRUE and displays at output "ErrorID" the error number. The “Done” output is also set to FALSE indicating that the job was not executed.
Solution	Class	-	Check the error number at the “ErrorID” output. This error number is a specific communication system error number which is used by the controller.
Error type	-		

ID 46053

While processing the START command the operation mode changed.			
Description	When processing the rising edge at the "Start" input of the MCV_GrpOperationMode function block, the system detects that the current operation mode is none of the operation modes • Automatic mode • Manual mode • Homing although one of these operation modes was detected as the current mode.		
Response	Class	-	No job is sent to the motion controller.
Solution	Class	-	Please contact your motion controller supplier.
Error type	-		

ID 46054

Operation mode of axis group doesn't allow the Start command.			
Description			
Response	Class	-	
Solution	Class	-	
Error type			

ID 46055

Current operation mode of axis group is not defined.			
Description	The current operation mode of an axis group which the motion controller sends to the PLC fails to correspond to any of the defined and permissible operation modes.		
Response	Class	-	No job is sent to the motion controller.
Solution	Class	-	Please contact your motion control supplier.
Error type	-		

ID 46056

An operation mode change is discarded because in same plc cycle more than one operation mode was commanded.			
Description	The MCV_GrpOperationMode function block is used to change the operation mode (Standby, Automatic, MDI, Homing, Manual Mode) of an axis group and then command the required state (Start, Stop, Reset) in this operation mode. The function block is implemented so that there is an input for each operation mode and state. The error message is always output if no job is sent to the motion controller. There are 2 reasons for this: 1. No unique command for the motion controller can be derived from the signals at the inputs for operation mode and state. This is possible if several inputs for switching the operation modes or states receive a rising edge in a PLC cycle. 2. It is not permissible to switch to a different operation mode from every constellation of operation mode and state. The permissible constellations are listed below. ◦ The NC program, manual block or homing is finished or interrupted by a stop. ◦ The axis group is in manual mode or in standby mode.		
Response	Class	-	No job is sent to the motion controller.
Solution	Class	-	Check whether one of the two reasons listed in the description applies to why the operation mode change is not executed. Change the PLC application appropriately to prevent this from reoccurring.
Error type	-		

ID 46057

No filename or filepath was assigned to input ParamAuto.			
Description	An axis group is in operation mode **Automatic mode**. To start an NC program, a rising edge was applied to the "Start" input of the MCV_GrpOperationMode function block. To start the NC program, the file containing the NC program must be defined via the “ParamAuto” input at the time of the rising edge at the “Start” input. Assign the filename or absolute filepath for this file to the “ParamAuto” input as a character string. An error is output since only an empty character string is applied to the “ParamMdi” input at the time of the rising edge at the “Start” input.		
Response	Class	-	No job is sent to the motion controller.
Solution	Class	-	At the time of the rising flank at the “Start” input of the MCV_GrpOperationMode function block, assign a valid filename or absolute filepath to a file containing an NC program to the “ParamAuto” input.
Error type	-		

ID 46058

No NC block was assigned to input ParamMdi.

Description	An axis group is in operation mode **MDI (**Manual block**)**. To start NC commands, a rising edge was applied to the "Start" input of the MCV_GrpOperationMode function block. NC commands can be started by defining them at the time of the rising edge at the "Start" input via the "ParamMdi" input of the function block. Therefore, assign the NC command sequence as a character string to the "ParamMdi" input. An error is output since only an empty character string is applied to the “ParamMdi” input at the time of the rising edge at the “Start” input.		
Response	Class	-	No job is sent to the motion controller.
Solution	Class	-	A character string containing valid NC commands must be assigned to the “ParamMdi” input at the time of the rising edge at the “Start” input of the MCV_GrpOperationMode function block.
Error type	-		

2.6 Path interpolator error (ID-range 50000-59999)

2.6.1 ID-range 50000-50249

ID 50000 - 50009

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50010

Motion path of the measurement block is too small.			
Description	To use edge bending function G108 , the probe hardware has to be initialized before entered the first movement block and after programmed MNE_SNS M code. The travelling distance is too small for initialisation of the probe hardware. For detailed information on measurement function G108, see [PROG// Section: Edge bending].		
Response	Class	6	Movement stop.
Solution	Class	6	Check path distance in measurement block; short blocks may occur due to contour-changing functions such as TRC, polynomial smoothing etc.
Error type	1, Error message from NC program.		

ID 50011

Measurement function is not initialised.			
Description	To use edge bending function G108 , the probe hardware has to be initialized before entered the first movement block and after programmed MNE_SNS M code. The probe hardware is not initialised. For detailed information on measurement function G108, see [PROG// Section: Edge bending].		
Response	Class	6	Movement stop.
Solution	Class	6	Check measurement hardware and delay cycles in chain of signal.
Error type	1, Error message from NC program.		

ID 50012 - 50023

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50025

No interpolation for axis in tracking mode.			
Description	The path axis is moving or is to be moved although the axis is in tracking mode (command position = actual position). There are different reasons why an axis is in tracking mode: 1. The user commands axis tracking mode via the High Level Interface by setting the HLI signal of the control unit lr_mc_control.follow_up.command to TRUE. (For detailed information on CNC real-time control signals, see [HLI// Control commands of an axis]. 2. The axis reverts automatically to tracking mode (internal tracking mode). Possible reasons for this include: ◦ This status indicates that the drive is not ready for operation (not yet initialized). ◦ There is no valid process data received from the drive. ◦ The drive is in error state. ◦ The axis is parked (Profidrive drive).		
Response	Class	4	Movement stop.
Solution	Class	6	Reset the lr_mc_control.folllow_up.command signal or eliminate the cause for internal tracking mode.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
Error type	1, Error message from NC program.		

ID 50026 - 50038

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50041

Positive software limit switch tripped.			
Description	When manual operation mode is enabled, the movement is monitored to offset limits and software limit switches. In standard operation movement stops without error message on offset limits. The axis parameter P-AXIS-00554 can influence the behaviour of software limit switches.		
Response	Class	4	Movement stop.
Solution	Class	6	Move axis into valid range. • Limit the permissible movement via manual mode offset limits (P-AXIS-00137, P-AXIS-00138) • Reduce the velocity of the manual mode motion (P-AXIS-00213).

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis affected P-AXIS-00016
	%2:	Current value [0.1 µm or 0.0001°]
		Current active command position of manual mode.
	%3:	Upper limit value [0.1 µm or 0.0001°]
		Positive software limit switch P-AXIS-00178.
	%4:	Current value [0.1 µm or 0.0001°]
		Current manual mode position of the axis
	%5:	Current value [0.1 µm or 0.0001°]
		Relative manual mode increments in the current cycle
Error type	1, Error message from NC program.	

ID 50042

Negative software limit switch tripped.			
Description	When manual operation mode is enabled, the movement is monitored to offset limits and software limit switches. In standard operation movement stops without error message on offset limits. The axis parameter P-AXIS-00554 can influence the behaviour of software limit switches.		
Response	Class	4	Movement stop.
Solution	Class	6	Move axis into valid range. • Limit the permissible movement via manual mode offset limits (P-AXIS-00137, P-AXIS-00138) • Reduce the velocity of the manual mode motion (P-AXIS-00213).
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current active command position of manual mode.	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Negative software limit switch P-AXIS-00177.	
	%4:	Current value [0.1 µm or 0.0001°]	
		Current manual mode position of the axis	
	%5:	Current value [0.1 µm or 0.0001°]	
		Relative manual mode increments in the current cycle	
Error type	1, Error message from NC program.		

ID 50043

No axes programmed in manual mode.			
Description	No axes were selected when manual mode was selected. At least one axis must be selected.		
Response	Class	1	None.
Solution	Class	1	NC program or or PLC project.
Error type	1, Error message from NC program.		

ID 50044

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50045

Setting the manual mode parameters not possible.			
Description	It is not possible to parameterise the axis for manual mode. For additional information on manual operation mode, see [PROG// Section: Manual mode], [HL// Section: Manual mode].		
Response	Class	4	Movement stop.
Solution	Class	6	Check if the axis or axes is/are available in the NC channel. Check the parameterisation of the manual mode axes in the NC program or PLC program.
Error type	1, Error message from NC program.		

ID 50046 - 50049

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50050

Probing cycle not initialized.			
Description	When using the measurement function G100, different measurement types can be completely contoured in the measurement block.. For preparation of probe hardware the measurement process in CNC is activated in previous movement block. The initialization of probe hardware has to be finished before moving in measure movement block. For detailed information on measurement function G100, FCT-C4], [PROG// Section: Measuring functions].		
Response	Class	4	Movement stop.
Solution	Class	6	Either use measurement type with motion stop or reduce delay time before initialization of probe hardware.
Parameter	%1:	Block number [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50051

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50052

Probe already actuated.			
Description	Before starting measurement motion with Measuring functions G100, check whether the measuring probe is currently active. The signal may not be enabled. For more information on Measuring functions G100 see [FCT-C4], [PROG// Section: Measuring functions].		
Response	Class	4	Movement stop.
Solution	Class	6	Check measurement signal and signal level.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Block number [-]	
	%3:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50053

Received probe before reaching the measuring motion block.			
Description	When using the measurement function G100, different measurement types can be completely contoured in the measurement block.. For preparation of probe hardware the measurement process in CNC is activated in previous movement block. In this case the probe sensor can be activated before moving in measure movement block. For detailed information on measurement function G100, see[FCT-C4], [PPROG// Section: Measuring functions].		
Response	Class	4	Movement stop.
Solution	Class	6	Either use measurement type with motion stop or program special MOS M/H function in the measurement motion block and check the measuring sensor logic in the PLC before enable.
Parameter	%1:	Upper limit value [-]	
	%2:	Upper limit value [-]	
	%3:	Block number [-]	
	%4:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 50054

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50057

Measuring not executed.			
Description	The measuring signal was not enabled in one axis when using the measurement function G100. For detailed information on measuring functions G100, [FCT-C4], [PROG// Section: Measuring functions].		
Response	Class	6	Movement stop.
Solution	Class	6	Check measurement signal in affected axis. Set measurement signal in all measurement axes with measurement interrupt.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	1, Error message from NC program.		

ID 50058

Measurement function is already selected (edge banding).			
Description	It is not allowed to measure with G100 while edge bending G108 is still enabled. For detailed information on measuring functions G100 and G108, see [PROG// Section: Measuring functions] [PROG// Section: Edge banding (G108)]		
Response	Class	4	Movement stop.
Solution	Class	6	Check CNC program and measuring sequence; G100 and G108 are mutually exclusive.
Error type	1, Error message from NC program.		

ID 50059

Unable to measure with specified axis.			
Description	If the measurement function G100 is used, axes involved in the measurement run must be marked with the MDS parameter (P-AXIS-00118). For detailed information on measurement function G100, [FCT-C4], [PROG// Section: Measuring functions].		
Response	Class	4	Movement stop.
Solution	Class	6	P-AXIS-00118 of the concerned axis must be set in the MDS.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
Error type	-		

ID 50060

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50061

Number of programmed axes exceeds range within measuring block.			
Description	The measuring function G100 makes several measurement types available. These types can be selected via the MDS or the CNC program. The maximum number of programmed measurement axes depends on measurement type selected. For detailed information on G100, see[FCT-C4], [PROG// Section: Measuring functions].		
Response	Class	4	Movement stop.
Solution	Class	6	Only program the maximum permitted number of axes for the measurement type selected or change the measurement type.
Parameter	%1:	Current value [-]	
		Number of programmed axes.	
	%2:	Upper limit value [-]	
		Maximum permissible number of programmed axes of selected measurement type.	
	%3:	Current value [-]	
		Currently selected measurement type.	
Error type	1, Error message from NC program.		

ID 50063

Measurement is not effected.			
Description	With the edge banding function G108 the measurement signal must be received within a motion block or by the time the edge banding function is deactivated. For more information on edge bending see [PROG// Section: Edge bending (G108)].		
Response	Class	1	Movement stop.
Solution	Class	1	Check probe and signal, Measurement signal during active edge bending movement
Error type	1, Error message from NC program.		

ID 50064

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50065

Remaining interpolation path greater than remaining braking distance.			
Description	In the Edge banding function, the residual path P-CHAN-00030 defined in channel parameter list is reduced is reduced after activating the measurement signal using the measuring probe. The braking distance at speed 0 required at this point must be smaller than the residual path. Note: The axis parameters P-AXIS-00001, P-AXIS-00002 and P-AXIS-00011, P-AXIS-00012 influence the braking distance. For more information on edge banding see [PROG// Section: Edge banding (G108)].		
Response	Class	1	Movement stop.
Solution	Class	1	Either increase P-CHAN-00030, or reduce the programmed feed rate in the edge banding motion blocks, or if possible increase the axis acceleration.
Parameter	%1:	Current value[0.1 µm or 0.0001°]	
		Required braking distance to velocity 0.	
	%2:	Expected value [0.1 µm or 0.0001°]	
		Residual path depending on P-CHAN-00030.	
	%3:	Current value[0.1 µm or 0.0001°]	
		Path distance of measurement actual positions and target positions.	
	%4:	Current value [1µm/s or 0.001°/s]	
	%5:	Current value[0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 50066

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50067

No motion information available.

Description	With the measurement function G100, a relevant motion path must be programmed. Wrong: %mess N10 G00 X100 Y0 Z100 N20 G100 X100 F2000 N30 M30 Correct: %mess N10 G00 X100 Y0 Z0 N20 G100 X200 F2000 N30 M30 For detailed information on G100, see[FCT-C4], [PROG// Section: Measuring functions].		
Response	Class	4	Movement stop.
Solution	Class	6	Measurement movement block with path > 0
Error type	1, Error message from NC program.		

ID 50069 - 50089

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 50092

Detected error within position control.			
Description	The position controller module of the NC Kernel has reported an error. Additional to this error message another error message with information on error cause and error removal is emitted.		
Response	Class	2	Movement stop.
Solution	Class	6	Removal of error in position controller and NC reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
Error type	-		

ID 50095 - 50110

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 50116

Emergency stop actuated.

Description	The command for a channel-specific emergency stop was set on the High Level Interface. For detailed information on CNC real-time control signals, see [HLI//Control commands of a channel].		
Response	Class	6	Movement stop.
Solution	Class	6	• The command for channel-specific emergency stop *bahn_mc_control.emergency_stop.command_w* must be set to FALSE. • Conduct an NC reset. • Restart program.
Parameter	%1:	Current value [-]	
		Control bit mask of motion influence (Emergency stop = 0x2)	
Error type	-		

ID 50118

Maximum permissible axis velocity exceeded.			
Description	The maximum axis velocity set in the axis parameter list (P-AXIS-00212) was exceeded. Dynamic look-ahead also considers the error threshold defined by the parameter P-AXIS-00440.		
Response	Class	4	Movement stop.
Solution	Class	6	Diagnosis
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current command velocity of the axis.	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum permissible velocity of the axis (P-AXIS-00212).	
	%4:	Current value [-]	
		Ratio of current velocity to maximum permissible axis velocity (absolute value).	
	%5:	Upper limit value [-]	
		Factor for permissible excess of velocity (P-AXIS-00440).	
Error type	1, Error message from NC program.		

ID 50119

Warning level of axis velocity exceeded.			
Description	The maximum axis velocity set in the axis parameter list (P-AXIS-00212) was exceeded. Dynamic look-ahead also considers the warning threshold defined by the parameter P-AXIS-00439.		
Response	Class	1	None
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of axis affected.
	%2:	Current value [1µm/s or 0.001°/s]
		Current nominal velocity of the axis.
	%3:	Upper limit value [1µm/s or 0.001°/s]
		Maximum permissible velocity of the axis (P-AXIS-00212).
	%4:	Current value [-]
		Ratio of current velocity to maximum permissible axis velocity (absolute value).
	%5:	Upper limit value [-]
		Factor for permissible excess of velocity (P-AXIS-00439).
Error type	1, Error message from NC program.	

ID 50120

Maximum permissible axis acceleration exceeded.			
Description	The maximum axis acceleration set in the axis parameter list (P-AXIS-00008) was exceeded above the permissible limit. Dynamic look-ahead also considers the error threshold defined by the parameter P-AXIS-00442.		
Response	Class	6	Movement stop.
Solution	Class	6	Diagnosis
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Current command acceleration of axis.	
	%3:	Limit value [-]	
		Maximum permissible acceleration of the axis (P-AXIS-00008).	
	%4:	Current value [mm/s² or °/s²]	
		Ratio of current acceleration to maximum permissible acceleration (absolute value).	
	%5:	Current value [0.1 µm or 0.0001°]	
		Factor for permissible excess of acceleration (P-AXIS-00442).	
Error type	1, Error message from NC program.		

ID 50121

Warning level of axis velocity exceeded.			
Description	The maximum axis acceleration set in the axis parameter list (P-AXIS-00008) was exceeded. Dynamic look-ahead also considers the warning threshold defined by the parameter P-AXIS-00441.		
Response	Class	1	None
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of axis affected.
	%2:	Current value[mm/s² or °/s²]
		Current nominal acceleration of axis.
	%3:	Upper limit value [mm/s² or °/s²]
		Maximum permissible acceleration of the axis (P-AXIS-00008).
	%4:	Current value [-]
		Ratio of current acceleration to maximum permissible acceleration (absolute value).
	%5:	Upper limit value [-]
		Factor for permissible excess of acceleration (P-AXIS-00441).
Error type	1, Error message from NC program.	

ID 50122

Axis jerk exceeds maximum limit.			
Description	The maximum axis jerk set in the axis parameter list (P-AXIS-00008, P-AXIS-00201) was exceeded. Dynamic look-ahead also considers the error threshold defined by the parameter P-AXIS-00444.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Generate the diagnosis file and make this file available to examine the error response.
Parameter	%1:	Logical axis number[-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value[mm/s^3 or °/s^3]	
		Current command jerk of axis.	
	%3:	Upper limit value[mm/s^3 or °/s^3]	
		Maximum permissible jerk of the axis (P-AXIS-00008, P-AXIS-00201).	
	%4:	[-] Current value	
		Ratio of current jerk to maximum permissible jerk (absolute value).	
	%5:	Upper limit value[-]	
		Factor for permissible excess of jerk (P-AXIS-00444).	
Error type	1, Error message from NC program.		

ID 50123

Warning level of axis jerk exceeded.			
Description	The maximum axis jerk set in the axis parameter list (P-AXIS-00008, P-AXIS-00201) was exceeded. Dynamic look-ahead also considers the warning threshold defined by the parameter P-AXIS-00443.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Generate the diagnosis file and make this file available to possibly examine the error response.

Parameter	%1:	Logical axis number[-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value[mm/s^3 or °/s^3]
		Current nominal jerk of axis.
	%3:	Upper limit value[mm/s^3 or °/s^3]
		Maximum permissible jerk of the axis (P-AXIS-00008, P-AXIS-00201).
	%4:	[-] Current value
		Ratio of current jerk to maximum permissible jerk (absolute value).
	%5:	Upper limit value[-]
		Factor for permissible excess of jerk (P-AXIS-00443).
Error type	1, Error message from NC program.	

ID 50124 / 50129

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 50130

Corrected dynamical parameter of axis.			
Description	The dynamic parameters for acceleration and ramp time can be changed by weighting in the NC program or directly by PLC command application. If the resulting value exceeds the permissible maximum value corresponding to the dynamic characteristics in the axis machine data record, this is limited to the permissible maximum value. Further information on weighting function: [PROG// Section: Acceleration weighting (G130/G131/G231)] [PROG// Section: Weighting ramp times (G132/G133/G134/G233)]		
Response	Class	1	None.
Solution	Class	1	No weighting or commanding of values greater than maximum limits.
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 50137 - 50157

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50168

During block search homing will not be executed.			
Description	When block search mode is active, some functions or functionalities, such as manual mode or homing, are not supported.		
Response	Class	1	None.
Solution	Class	1	Remove function selection for reported block number from the NC program.
Error type	1, Error message from NC program.		

ID 50169

During block search not allowed interpolator control data received.			
Description	When the block search function is active, some functions or functionalities, such as manual mode or homing, are not supported. For further information see FCT-C6.		
Response	Class	1	None.
Solution	Class	1	Remove function selection for reported block number from the NC program.
Error type	1, Error message from NC program.		

ID 50170

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 50172

Measurement during block search is not permitted.			
Description	When block search mode is active block search mode, some functions or functionalities, such as manual mode or homing, are not supported. [FCT-C6]		
Response	Class	2	None.
Solution	Class	1	Remove function selection for reported block number from the NC program.
Error type	1, Error message from NC program.		

ID 50180

Setted spindle speed not reached.

Description	Before path motion starts, the Constant Cutting Speed rotation function (G96) checks whether the spindle has reached the required rotation speed. If not, the real cutting speed deviates from the required cutting speed. For more information on turning functions see [PROG// Section: Turning functions]. For information on M-H functions, see [FCT-C1//Section: Spindle M functions]		
Response	Class	1	None
Solution	Class	1	Execute speed synchronisation using the PLC.
Parameter	%1:	Current value [1µm/s or 0.001°/s]	
		Current spindle speed.	
	%2:	Expected value [1µm/s or 0.001°/s]	
		Nominal spindle speed for G96.	
Error type	-		

ID 50181

Programmed revolution feed rate too small.			
Description	With turning function feedrate per revolution (G95), the programmed feedrate must be greater than a minimum value. For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	4	Movement stop.
Solution	Class	6	Programming of revolutional feed > 0.
Parameter	%1:	Current value [µm / revolution]	
		Current programmed feedrate per revolution.	
	%2:	Lower limit value	
		Lower limit for revolutional feed.	
Error type	1, Error message from NC program.		

ID 50182

Cutting speed is 0.			
Description	With turning function constant cutting speed (G96) the programmed cutting speed should be > 0. For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	6	Movement stop.
Solution	Class	6	Cutting speed > 0 in NC program
Parameter	%1:	Current value [µs]	
		Current programmed cutting speed.	
Error type	1, Error message from NC program.		

ID 50183

Maximum spindle speed is 0.	
Description	In combination with constant cutting speed turning function (G96), the maximum turning speed of spindle can be limited in the NC program (G196). This value should be not 0. For more information on turning functions see [PROG// Section: Turning functions].

Response	Class	6	Movement stop.
Solution	Class	6	Set maximum spindle speed value > 0 in NC program.
Parameter	%1:	Current value[1µm/s or 0.001°/s]	
		Current upper speed limit G196.	
Error type	1, Error message from NC program.		

ID 50184

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50185

Missing face turning axis.			
Description	In turning function constant cutting speed (G96) no face turning axis is available. For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	6	Movement stop.
Solution	Class	6	Check axis configuration, parameters and NC program. The facing axis must be marked in axis parameter P-AXIS-00015. The face turning axis must be enabled in channel.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of used face turning axis.	
Error type	-		

ID 50186 - 50189

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 50193

Illegal type of SLOPE within turning.			
Description	Linear slope type (step-shaped) is required for turning functions. For more information on turning functions and profile selection see [PROG// Section: Turning functions].		
Response	Class	4	Movement stop.
Solution	Class	6	In MDS linear (step-shaped) slope type P-CHAN-00071 or in the NC program, select the linear slope type (step-shaped) before activating the turning functions.

Parameter	%1:	Current value [-]
		Currently active slope type.
	%2:	Expected value [-]
		Required slope type.
Error type	1, Error message from NC program.	

ID 50195 - 50204

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50206

Spindle distance per cycle is 0.			
Description	With turning function thread cutting (G33), current value of path per cycle is smaller than minimum value. For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	4	Movement stop.
Solution	Class	6	Check current speed spindle, value should be > 0. Turning speed of spindle > 0 in NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned spindle axis (P-AXIS-00016).	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current spindle speed.	
	%3:	Current value [0.1 µm or 0.0001°]	
		Path per cycle.	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Lower limit value for minimum path per cycle.	
Error type	1, Error message from NC program.		

ID 50207 - 50213

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50214

Positive software limit switch tripped with active RTCP.	
Description	An additional check of axis positions is executed with active five-axis function. The command values of transformation axes must be within valid range of software limits.

Response	Class	4	Movement stop.
Solution	Class	6	Move axis back to within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Positive software limit switch P-AXIS-00178.	
Error type	1, Error message from NC program.		

ID 50215

Negative software limit switch tripped with active RTCP.			
Description	An additional check of axis positions is executed with active five-axis function. The command values of transformation axes must be within valid range of software limits.		
Response	Class	4	Movement stop.
Solution	Class	6	Move axis back to within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Negative software limit switch P-AXIS-00177.	
Error type	1, Error message from NC program.		

ID 50216 - 50222

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50233

Functionality is not supported by machining simulation.			
Description	When simulation mode is active, specific functions or functionalities, such as manual mode, cannot be activated.		
Response	Class	2	Movement stop.
Solution	Class	6	Remove function selection for reported block number from the NC program.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50246

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

2.6.2 ID-range 50250-50499

ID 50250 / 50251

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50253

Limit velocity for polynomial calculation smaller than permissible value.			
Description	The permitted minimum value of parameter P-AXIS-00130 has fallen below its lower limit.		
Response	Class	6	Stop axis immediately.
Solution	Class	7	Change P-AXIS-00130.
Parameter	%1:		
	Logical axis number of spindle axis		
	%2:		
	Parameterised value of P-AXIS-00130.		
	%3:		
	Minimum permissible value for P-AXIS-00130.		
Error type	-		

ID 50254

Unknown acceleration type.			
Description	An invalid value for parameter P-AXIS-00202 was entered when setting up curve controlled acceleration. See also @@[FCT-S1//chapter characteristic curve-controlled acceleration].		
Response	Class	6	Immediate stop of the axis.
Solution	Class	7	P-AXIS-00202 must have a valid value assigned.
Error type	-		

ID 50255 - 50259

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50260

Non linear SLOPE while homing not allowed.

Description	Wrong slope type selected for homing to a fixed stop. This homing type is only possible with slope type 0. For detailed information about profile types, see [PROG// Section: Selecting operating mode), [FCT-M1//Section: Moving to fixed stop]		
Response	Class	6	Movement stop.
Solution	Class	6	Change over slope type to linear type with channel parameter P-CHAN-00071or NC command #SLOPE[TYPE=STEP].
Parameter	%1:	Current value [-]	
		Selected slope profile.	
Error type	1, Error message from NC program.		

ID 50262

During measuring virtual and other axes must not be mixed.			
Description	When using measuring function G100, virtual axes and axes of another drive type (P-AXIS-00020) are mixed in measurement axes. This is not permitted. For more information on the G100 measuring function, see [PROG// Section: Measuring functions] [FCT-C4].		
Response	Class	4	Movement stop.
Solution	Class	6	For the drive type (P-AXIS-00020), use either only virtual axes or no virtual axes for the axes involved in the measurement run
Parameter	%1:	Current value [-]	
		Number of virtual measurement axes in measurement axis group.	
	%2:	Current value [-]	
		Programmed axes of measurement axis group.	
Error type	1, Error message from NC program.		

ID 50263

Moved main axis not flagged for edge banding.			
Description	Before the start of the edge banding process, the edge banding function checks whether all main axes moved in the channel are flagged as edge banding axes. The individual axes are flagged using P-AXIS-00098. Only main axes can be flagged as edge banding axes! For more information on edge banding see [PROG// Section: Edge banding (G108)].		
Response	Class	4	Movement stop.
Solution	Class	6	Check and set P-AXIS-00098 to the axis concerned
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis concerned	
Error type	1, Error message from NC program.		

ID 50264 - 50277

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50278

Selection of manual mode in the turning centre.			
Description	With the C axis functionality, the Cartesian programming coordinate system is transformed into cylinder machine coordinate system. If milling head is located in centre point of this coordinate system so activation of manual mode is not possible.		
Response	Class	4	Movement stop.
Solution	Class	6	Moving linear machine axis which lays in XY plane out of turning centre position with inactive transformation.
Error type	1, Error message from NC program.		

ID 50279

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50280

Velocity for manual mode cannot be reached in given cycles.			
Description	With the C axis functionality, the Cartesian programming coordinate system is mapped in a cylinder machine coordinate system. High dynamic stresses can occur next to the turning centre point. When manual mode is active, it is not always possible to operate at the maximum velocity in the MDS P-AXIS-00213.		
Response	Class	1	None.
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [1µm/s or 0.001°/s]	
	%3:	Limit value [1µm/s or 0.001°/s]	
Error type	1, Error message from NC program.		

ID 50282 - 50346

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50350

Time for movement too short.			
Description	The feed for independent axes can be defined either as a feed rate or as a motion time by the keyword “FEED” or “TIME”. The time for motion value may not be less than a minimum value. For detailed information on independent axes see [PROG// Section: Independent axes].		
Response	Class	6	Movement stop.
Solution	Class	7	Program a motion time greater than the lower limit value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected	
	%2:	Current value [1µm/s or 0.001°/s]	
		Resulting programmed feed rate	
	%3:	Lower limit value [1µm/s or 0.001°/s]	
		Lower limit value for motion time	
Error type	1, Error message from NC program.		

ID 50351

Fatal SERCOS-error.			
Description	In case of synchronous read or write of a Sercos identifier, the service channel reports an unknown error. The id could not be read or write.		
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Identification number [-]	
	%3:	Current value [-]	
	%4:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50352

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50354

SERCOS parameters: Unknown SERCOS id.			
Description	Read or write command for SERCOS parameters in the NC program via #IDENT WR[], #IDENT RD[] failed because of unknown or invalid SERCOS ID. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	Check SERCOS ID in the NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1.	
	%4:	Current value [-]	
		SERCOS return value 2.	
Error type	1, Error message from NC program.		

ID 50355

SERCOS parameters: Data size too short.			
Description	The programmed data length of the read or write command for SERCOS parameters in the NC program via #IDENT WR[], #IDENT RD[] fails to match the length of the SERCOS ID. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	Check data type and length of SERCOS parameter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1	
	%4:	Current value [-]	
		SERCOS return value 2	
Error type	1, Error message from NC program.		

ID 50356

SERCOS parameters: Data size too long.

Description	The programmed data length of the read or write command for SERCOS parameters in the NC program via #IDENT WR[], #IDENT RD[] fails to match the length of the SERCOS ID. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	Check data type and length of SERCOS parameter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1	
	%4:	Current value [-]	
		SERCOS return value 2	
Error type	1, Error message from NC program.		

ID 50357

SERCOS parameters: Data item cannot be changed.			
Description	Writing SERCOS parameters in the NC program via #IDENT WR[] failed because the SERCOS parameter cannot be changed. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	SERCOS id can only be read via #IDENT RD[].
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1.	
	%4:	Current value [-]	
		SERCOS return value 2.	
Error type	1, Error message from NC program.		

ID 50358

SERCOS parameters: Data item currently write-protected.			
Description	Writing SERCOS parameters in the NC program via #IDENT WR[] failed because the parameter is currently write-protected. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	SERCOS id can only be read via #IDENT RD[].

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Identification number [-]
		Programmed SERCOS id number.
	%3:	Current value [-]
		SERCOS return value 1.
	%4:	Current value [-]
		SERCOS return value 2.
Error type	1, Error message from NC program.	

ID 50359

SERCOS parameters: Data item less than minimum value.			
Description	Writing SERCOS parameters in the NC program via #IDENT WR[[] failed because the parameter is less than the permissible minimum value. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	Program parameter greater than minimum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1.	
	%4:	Current value [-]	
		SERCOS return value 2.	
Error type	1, Error message from NC program.		

ID 50360

SERCOS parameters: Data item greater than maximum value.			
Description	Writing SERCOS parameters in the NC program via #IDENT WR[] failed because the parameter is greater than the permissible maximum value. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	Program the parameter value less than the maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1.	
	%4:	Current value [-]	
		SERCOS return value 2.	

Error type	1, Error message from NC program.

ID 50361

SERCOS parameters: Data item incorrect.			
Description	Wrong data item used for writing or reading SERCOS parameter. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct SERCOS parameter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1.	
	%4:	Current value [-]	
		SERCOS return value 2.	
Error type	1, Error message from NC program.		

ID 50362

Other error by data transfer in service channel.			
Description	An error occurred on data transfer of SERCOS parameters or commands. For more information on this function, see [PROG// Section: Writing and reading SERCOS parameters and commands].		
Response	Class	4	Movement stop.
Solution	Class	6	Check drive and communication.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Identification number [-]	
		Programmed SERCOS id number.	
	%3:	Current value [-]	
		SERCOS return value 1.	
	%4:	Current value [-]	
		SERCOS return value 2.	
Error type	1, Error message from NC program.		

ID 50363 - 50381

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50384

Inadmissible number of late sync function reached.			
Description	The total number of simultaneously active M functions with late synchronisation (Late sync, types MVS_SLM, MVS_SLP) is limited. After triggering the active M functions with G01 motion block or explicit triggering via NC command, the M functions are enabled internally until synchronisation with the PLC. For more detailed information about M/H functions, see [FCT-C1// Section: Reaction of “late” synchronisation]. [FCT-C1// Section: Parameterising the synchronisation methods]		
Response	Class	1	Warning. If the following additional M functions occur and no M functions with later synchronisation are acknowledged, the path motion of the CNC is stopped even without triggering Late SyncM functions after a defined number of M functions (internal buffer effect).
Solution	Class	1	Reduce the M functions programmed per NC block and until triggering with late synchronisation.
Parameter	%1:	Current value [-]	
		Number of currently active late synchronisation M functions.	
	%2:	Upper limit value [-]	
		Maximum number of simultaneously active M functions with late synchronisation.	
Error type	-		

ID 50385 - 50387

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50388

Inadmissible number of late sync functions in the channel.			
Description	The number of channel-specific M functions programmed in an NC line with late synchronisation (Late sync, types MVS_SLM, MVS_SLP) is limited. After triggering the active M functions with G01 motion block or explicit triggering via NC command, the M functions are enabled internally until synchronisation with the PLC. For more detailed information about M/H functions, see [FCT-C1// Section: Reaction of “late” synchronisation]. [FCT-C1// Section: Parameterising the synchronisation methods]		
Response	Class	4	Movement stop.
Solution	Class	6	Reduce the number of channel-specific M functions programmed per NC line with later synchronisation .

Parameter	%1:	Current value [-]
		Block number in the NC program.
	%2:	Limit value [-]
		Number of axis-specific M functions currently programmed in a NC line with later synchronisation.
	%3:	
		Maximum number of permissible M functions with late synchronisation per axis and NC line.
Error type	1, Error message from NC program.	

ID 50389

Inadmissible number of late sync function in axes.			
Description	The number of axis-specific M functions programmed in an NC line with late synchronisation (Late sync, types MVS_SLM, MVS_SLP) is limited. After triggering the active M functions with G01 motion block or explicit triggering via NC command, the M functions are enabled internally until synchronisation with the PLC. For more detailed information about M/H functions, see [FCT-C1// Section: Reaction of “late” synchronisation]. [FCT-C1// Section: Parameterising the synchronisation methods].		
Response	Class	4	Movement stop.
Solution	Class	6	Reduce the of axis-specific M functions programmed per NC line with late synchronisation.

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis affected P-AXIS-00016.
	%2:	Block number [-]
		Block number in the NC program.
	%3:	Current value [-]
		Number of M functions currently programmed in the axis and NC line with late synchronisation.
	%4:	Limit value [-]
		Maximum number of permissible M functions with late synchronisation per axis and NC line.
Error type	-	

ID 50391 - 50397

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50398

No edge banding axis marked in MDS.

Description	Before the edge banding process starts, the Edge banding function checks whether machine axes are defined as edge banding axes in the MDS channel P-AXIS-00098. NB: At least one main axis must be marked as an edge banding axis. For more information on edge banding, see [PROG// Section: Edge banding (G108)].		
Response	Class	4	Movement stop.
Solution	Class	6	Set P-AXIS-00098 to 1 for at least one axis.
Error type	1, Error message from NC program.		

ID 50399 - 50401

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50403

Probe already actuated.			
Description	The edge banding function checks whether the measuring probe is already enabled before starting the edge banding motion. The signal may not be enabled. For more information on edge banding, see [PROG// Section: Edge banding (G108)].		
Response	Class	4	Motion stop, abort edge banding.
Solution	Class	6	Check measurement signal and signal level.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
Error type	1, Error message from NC program.		

ID 50404

Initialisation error in kinematic transformation.			
Description	An initialisation or parameterisation error occurred in the kinematic transformation in the Block search and Restart to the contour functions. For more information on kinematic transformation, see [KITRA] [PROG// Section: 5-axis function].		
Response	Class	6	Movement stop.
Solution	Class	6	Check and correct kinematic transformation parameters in MDS or tool management.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50405 - 50410

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50411

Timeout by PLC reset channel.			
Description	Request for NC reset of channel was not acknowledged by PLC within required time limit.		
Response	Class	4	Movement stop.
Solution	Class	6	Execute NC reset. Check PLC task and PLC project.
Parameter	%1:	Upper limit value [µs]	
		Maximum time period for reset acknowledge after reset request.	
Error type	-		

ID 50412

Timeout by PLC reset axes.			
Description	Request for CNC reset of axis was not acknowledged by PLC within required time limit.		
Response	Class	4	Movement stop.
Solution	Class	6	Execute NC reset. Check PLC task and PLC project.
Parameter	%1:	Current value [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Upper limit value [µs]	
		Maximum time period for reset acknowledge after reset request.	
Error type	-		

ID 50417

SERCOS command cannot be executed.			
Description	The SERCOS command requested by the NC program with #COMMAND WR[], #COMMAND WR SYN[] cannot be executed in the drive. For more information on this function, see [PROG// Section: SERCOS commands (COMMAND)].		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct SERCOS command.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Identification number [-]
		Programmed SERCOS id number.
	%3:	Current value [-]
		Data state
	%4:	Current value [-]
		SERCOS return value 1.
	%5:	Current value [-]
		SERCOS return value 2.
Error type	1, Error message from NC program.	

ID 50418

Machine data item could not be taken over.			
Description	The programmed machine data item could not be adopted by the controller and the NC command has no effect. Example: #MACHINE DATA [AX X “getriebe[0].kv 1000”] For further information see [PROG// Section: Writing machine data]		
Response	Class	3	
Solution	Class	6	Check command for correct syntax.
Error type	1, Error message from NC program.		

ID 50419 - 50427

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50429

No drive-controlled homing for GANTRY axis permissible.			
Description	After the command from the CNC controller, intelligent digital drives, e.g. SERCOS, execute homing independently. With gantry axis groups, no drive-controlled homing is possible because no synchronous homing motion is possible without additional measures. For more information on gantry axes, see [FCT-C11// Section: Homing]		
Response	Class	6	Movement stop.
Solution	Class	6	Check correct drive parameter settings for gantry homing. Set axis parameter P-AXIS-00253 to 1.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [-]
		Axis mode of the axis P-AXIS-00015
	%3:	Current value [-]
		Drive type of the axis P-AXIS-00018
Error type	1, Error message from NC program..	

ID 50430

Activation of manual mode failed for at least one axis.			
Description	It is not possible to select at least one axis for manual mode. For additional information on manual mode, see: [PROG// Section: Manual mode] [HLI// Section: Manual mode].		
Response	Class	4	Movement stop.
Solution	Class	6	Check if the axis or axes is/are available in the NC channel. Check selection of manual mode axes in the NC program or check the PLC program whether axes are already selected.
Error type	1, Error message from NC program.		

ID 50431

Deselection of manual mode failed for at least one axis.			
Description	It is not possible to deselect at least one axis for manual mode. For additional information on manual mode, see [PROG// Section: Manual mode] [HLI// Section: Manual mode].		
Response	Class	4	Movement stop.
Solution	Class	6	Check if the axis or axes is/are available in the NC channel. Check deselection of manual mode axes in the NC program or check the PLC program whether axes are already deselected for manual mode.
Error type	1, Error message from NC program.		

ID 50432 - 50434

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50438

Measuring probe was not actuated.

Description	The residual path defined in the channel parameter list for the edge banding function is reduced after the measurement signal is activated. Depending on the channel parameter P-CHAN-00029, the measurement signal must be received either within one motion block or by the time the edge banding function is deactivated. For more information on edge banding, see [PROG// Section: Edge banding (G108)].		
Response	Class	4	Movement stop.
Solution	Class	6	Check probe and signal, Activate probe during edge banding motion
Error type	1, Error message from NC program.		

ID 50439 - 50442

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50443

Residual path for edge banding too small.			
Description	When the edge banding function is active, the residual path P-CHAN-00030 defined in the channel parameter list is reduced after the measurement signal is activated by the leading measuring probe. The active path difference in the space between the actual measuring positions and the command positions may not be greater than P-CHAN-00030. For more information on edge banding, see [PROG// Section: Edge banding (G108)].		
Response	Class	1	Movement stop.
Solution	Class	1	Increase P-CHAN-00030.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current residual edge banding path P-CHAN-00030.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Path difference in the space between actual measuring positions and command positions.	
	%3:	Current value [0.1 µm or 0.0001°]	
	%4:	Current value [0.1 µm or 0.0001°]	
	%5:	Current value [0.1 µm or 0.0001°]	
Error type	-		

ID 50444 - 50448

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	

Solution	Class	8	Requires controller restart.

ID 50449

Start of storage reached during backward movement, no further backward movement possible.			
Description	With the backward motion on the path function, the maximum number of backward motion blocks depends on available memory size P-STUP-00033. With very large NC programs, it may occur that the memory is unable to save all forward direction motion blocks. For more information on backward motion, see [FCT-C7// Section: Forward/ backward motion on the path] The first parameter in the error message indicates whether it is no longer possible to continue backward motion due to axis positions that were changed outside the NC program. Axis positions which are changed by real-time influences cannot be considered in backward motion since they are not a component of the planned path. Real-time influences that can prevent backward motion include: 1. Tracking mode (tracking mode) 2. JogOfPath (see [FCT-C15]) 3. Additive command additional interface in the PLC		
Response	Class	1	None.
Solution	Class	1	Increase size of backward motion memory P-STUP-00033.
Parameter	%1:	Current value [-]	
		TRUE if further backward motion is not possible due to axis positions changed outside the NC program.	
Error type	1, Error message from NC program.		

ID 50450

Memory for backward motion is smaller than minimum limit.			
Description	With the backward motion on the path function, the maximum number of backward motion blocks depends on the available memory size P-STUP-00033. The memory must have a minimum size. For more information on backward motion, see [FCT-C7// Section: Forward/ backward motion on the path]		
Response	Class	-	Movement stop.
Solution	Class	-	Increase the size of backward motion memory P-STUP-00033.
Parameter	%1:	Limit value [-]	
		Current memory size.	
	%2:	Corrected value [-]	
		Required memory size.	
Error type	-		

ID 50451

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	

Solution	Class	8	Requires controller restart.

ID 50452

Axis was moved during skipping of NC lines.			
Description	With the backward motion on the path function, special marked blocks can be skipped under certain conditions for backward and forward motion with the memory (#OPTIONAL EXECUTION). One essential condition is that all axis positions must be identical before and after the marked program sequence. For more information on backward motion, see [FCT-C7// Section: Forward/ backward motion on the path]		
Response	Class	2	Movement stop.
Solution	Class	7	Modify the CNC program to ensure that the same axes positions occur at the start and end points of the marked program sequence.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [0.1 µm or 0.0001°]	
		Axis position at the end of marked program sequence.	
	%3:	Expected value [0.1 µm or 0.0001°]	
		Required axis position at the end of marked program sequence.	
Error type	1, Error message from NC program.		

ID 50453

Switching motion direction impossible, because backward storage is off.			

Description	The backward motion on the path function sets the signal
	bahn_mc_control.backward_motion.command.
	The parameter P-STUP-00033 of the corresponding channel must be assigned for the backward motion function.
	Extract from the start-up list:
	`fb_storage_size[0]  0x200000 # channel 1 [byte]`
	`fb_storage_size[1]  0x100000 # channel 2`
	For more information on backward motion, see [FCT-C7// Section: Forward/ backward motion on the path]		

Response	Class	1	None, backward motion is not possible.
Solution	Class	1	P-STUP-00033 must be set in order to use the backward motion function.
Error type	-		

ID 50454

Clearing of backward motion memory not possible during backward motion.	
Description	In the backward motion function on the path, the signal in the bahn_mc_control.backward_storage_off.command_w control unit was activated to clear the memory for backward motion. This is not possible when backward motion is active. The backward motion memory is not cleared. For more information on backward motion [FCT-C7// Section : Forward/backward motion on the path]

Response	Class	1	None.
Solution	Class	1	Do not set the PLC signal bahn_mc_control.backward_storage_off.command_w when path motion is active.
Error type	-		

ID 50455

Switching backward motion memory on/off not allowed while NC program is active.			
Description	With the backward motion on the path function, the signal bahn_mc_control.backward_storage_off.command_w was activated. When the NC program is enabled, it is not possible to switch the backward motion memory on/off. For more information on backward motion [FCT-C7// Section: Forward/backward motion on the path]		
Response	Class	1	None.
Solution	Class	1	Do not set the PLC signal bahn_mc_control.backward_storage_off.command_w when the NC program is enabled.
Error type	-		

ID 50456 - 50461

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50462

Logical interpolator axis cannot be connect with the physical position-controller axis.			
Description	Axis connection between drive and channel interpolator, based on parameters P-AXIS-00101, P-AXIS-00040 with multiple axis instantiation, was not successful.		
Response	Class	2	Movement stop.
Solution	Class	7	In MDS, check multiple instantiation of the axis using the axis configuration and parameters P-AXIS-00101, P-AXIS-00040.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned channel interpolator axis	
	%2:	Error value [-]	
		Acknowledged logical axis number P-AXIS-00016 of linked axis.	
	%3:	Expected value [-]	
		Required logical axis number P-AXIS-00016 of linked axis.	
Error type	1, Error message from NC program.		

ID 50463

Platform inclination angle exceeds limit.

Description	The permissible inclination angle if a workpiece platform was exceeded when a kinematic transformation was used. For more information on kinematic transformation, see [KITRA].		
Response	Class	4	Movement stop.
Solution	Class	6	Check the programmed position or orientation and correct to smaller values if necessary.
Error type	1, Error message from NC program.		

ID 50464

Not all axes known for automatic restart to contour after block search.			
Description	On restart after block search, the number of axes changed compared to the axis configuration at program start. This is not permitted. [FCT-C6// Section: Restart to the contour after block search]		
Response	Class	3	Movement stop.
Solution	Class	6	Check the NC program in the block search sequence for the exchange of additional axes with #CALL AX.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50465

Block search parameter: distance covered outside the valid range.			
Description	The “distance covered” parameter with active block search function is outside valid range. [FCT-C6//Section: Type 3: Continuation position defined by file offset]		
Response	Class	3	The value is corrected automatically.
Solution	Class	1	Use correct parameter setting.
Parameter	%1:	Current value [-]	
		Command value “distance covered“.	
	%2:	Corrected value [-]	
		Corrected value “distance covered“.	
	%3:	Upper limit value [-]	
		Upper limit value “distance covered“.	
	%4:	Lower limit value [-]	
		Lower limit value “distance covered““.	
Error type	1, Error message from NC program.		

ID 50466 - 50472

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50473

Switch on the backward motion memory without memory impossible.			
Description	Sufficient memory capacity must be available to activate the backward motion memory for backward motion. P-STUP-00033. For more information on backward motion, see [FCT-C7// Section: Forward/backward motion on the path]		
Response	Class	1	None.
Solution	Class	1	Release memory for backward motion P-STUP-00033.
Error type	-		

ID 50474

Deviation from contour exceeds limit after manual restart to contour during block search.			
Description	Restart to contour was performed manually but the path difference to start position in the NC program exceeds permissible limit. The user can change the permissible difference using the HMI or CNC objects. For more information on block search, see [FCT-C6// Section: Restart to the contour after block search]		
Response	Class	3	Movement stop.
Solution	Class	6	Check target position with manual restart and modify depending on start position in NC program.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current difference before restart.	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Permissible difference for restart.	
Error type	1, Error message from NC program.		

ID 50475

Block search position contents too many blocks without movement information.			
Description	The NC program contains too many irrelevant blocks up to the block search position. Non-relevant blocks are all CNC program blocks that contain no motion. In this case, the block search restart position does not exist in the internal memory.		
Response	Class	3	Movement stop.
Solution	Class	7	Check the NC program, skip irrelevant blocks if possible or move the block search position.
Parameter	%1:	Current value [-]	
		Current number of saved blocks.	
	%2:		
		Upper limit number of saved blocks.	
Error type	1, Error message from NC program.		

ID 50482 - 50486

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50487

Maximum permissible axis velocity is limiting.			
Description	For the oscillation axes function, depending on profile type, limiting effects may occur with relatively high frequency or small period time, that lead to deviation from the required programmed frequency or period time. For more information on oscillation axes see [FCT-A8// Section: Oscillating motion dynamics].		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Either reduce programmed frequency and period time respectively or if at all possible increase permissible axis velocity.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Required desired feed.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Current permissible axis velocity.	
Error type	1, Error message from NC program.		

ID 50489 - 50493

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50494

Axis acceleration is limiting			
Description	For the oscillation axes function, depending on profile type, limiting effects may occur with relatively high frequency or small period time, that lead to deviation from the required programmed frequency or period time. For more information on oscillation axes see [FCT-A8// Section: Oscillating motion dynamics].		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Either reduce programmed frequency and period time respectively or if at all possible increase permissible axis acceleration.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
	%2:	Current value [mm/s² or °/s²]	
		Current permissible axis acceleration.	

Error type	1, Error message from NC program.1, Error message from NC program.

ID 50496 / 50497

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50498

The available motion path is less than the defined remaining distance.			
Description	With the Edge banding function, the residual path P-CHAN-00030 defined in the channel parameter list is reduced after the measurement signal is activated by the leading touch probe. The block motion path available at this time must be greater than P-CHAN-00030. For more information on edge banding, see [PROG// Section: Edge banding (G108)].		
Response	Class	4	Movement stop.
Solution	Class	0	Either reduce P-CHAN-00030 or increase the programmed block motion path so that a residual path of at least P-CHAN-00030 is available after the response of the measurement signal.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Residual path available in the programmed measuring block.	
	%2:	Expected value [0.1 µm or 0.0001°]	
		Residual path depending on P-CHAN-00030	
	%3:	Current value [0.1 µm or 0.0001°]	
		Path difference of measurement actual positions and target positions.	
	%4:	Current value [0.1 µm or 0.0001°]	
		Channel block path.	
Error type	1, Error message from NC program.		

ID 50499

Spindle speed is 0.			
Description	With turning functions revolution feedrate G95 and thread cutting G33, he path feedrate is derived from the programmed spindle speed. For this reason, the spindle speed programmed via the S word must not be 0 for the active turning functions mentioned above. For further information see [PROG// Section: Turning functions].		
Response	Class	6	Movement stop.
Solution	Class	6	Program spindle speed > 0!
Error type	1, Error message from NC program.		

2.6.3 ID-range 50500-50749

ID 50500

Maximum permissible number of PLC spindle couplings exceeded.			
Description	The number of PLC spindles active and coupled simultaneously in synchronous mode is limited (#SET AX LINK[], #ENABLE AX LINK[]). Further information to synchronous mode see [PROG// Section: Synchronous mode].		
Response	Class	6	Movement stop.
Solution	Class	6	Check parameterization of synchronous operating axes in channel machine data (P-CHAN-00037, P-CHAN-00038) or reduce number of spindle couplings in #SET AX LINK command.
Parameter	%1:	Limit value [-]	
		Maximum number of links of PLC spindles.	
	%2:	Current value [-]	
		Current defined number of PLC spindle couplings.	
	%3:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the master spindle axis affected.	
	%4:	Logical axis number [-]	
		Logical axis number (@@P-AXIS-00016) of slave axis coupled to the master axis and which leads to exceeding of maximum coupling number.	
Error type	1, Error message from NC program.		

ID 50501 / 50502

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50503

Before deactivation of trafo tool geometry compensation has to be disabled.			
Description	Before deselecting the transformation with #TRAFO OFF when tool geometry compensation (TGC) is active, TGC must be deselected with #TGC OFF.		
Response	Class	6	
Solution	Class	6	Modify the NC program.
Error type	1, Error message from NC program.		

ID 50504

Before deactivation of transformation.manual mode has to be disabled.

Description	In inclusive manual operation (G201) with active transformation, before deactivating transformations (#CS ON[], #TRAFO ON), manual operation must be deactivated via G202. Programming example: Wrong: N10 #TRAFO ON N20 G201 N30 G02 I200 F2000 N40 M30 #TRAFO OFF Correct: N10 #TRAFO ON N20 G201 N30 G02 I200 F2000 N40 G202 N50 M30 #TRAFO OFF		
Response	Class	6	Movement stop.
Solution	Class	6	Deactivate manual operation mode before #CS OFF, #TRAFO OFF via G202.
Error type	1, Error message from NC program.		

ID 50505

Deviation of orthogonal ori vector to path tangent vector exceeds limit..			
Description	The deviation of the programmed orientation vector of the tool from the orthogonal to the path tangent vector exceeds limit.		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Modification of the orientation vector of tool in the NC program.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
		Deviation of the orientation vector to the orthogonal to path tangent.	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
		Maximal permitted deviation.	
Error type	1, Error message from NC program.		

ID 50506

Invalid orientation vector.			
Description	The programmed orientation vector is invalid. Error example: u=0 v=0 w=0 Corrected example: u=0 v=0 w=1		
Response	Class	6	
Solution	Class	6	Modify orientation vector in NC program.
Error type	1, Error message from NC program.		

ID 50507

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50508

Homing to fixed stop: Reference position not detected.			
Description	Homing G74 can be done with different variants parameterized via machine data setting. The torque homing process was terminated because of missing following error limit signal or torque limit signal.		
Response	Class	6	Movement stop.
Solution	Class	6	Check homing process, if necessary adjust parameter 2
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Current value [-]	
		Name of parameter	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Path for torque homing	
Error type	1, Error message from NC program.		

ID 50509 / 50511

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50512

Transition impossible, because of active position offset.			
Description	When backward motion is enabled, no position offset caused by additive manual mode (G201) or block search are permitted. Further information: Backward motion, see [FCT-C7// Section: Forward/backward motion on the path] Block search, see [FCT-C6// Section: Restart to the contour after block search].		
Response	Class	4	Program abort.
Solution	Class	6	No position offset should be enabled before backward motion is enabled. This is initialised by NC reset or a program restart.
Error type	-		

ID 50513

Position offset is still enabled.

Description	After activating the backward motion function, consider the restrictions regarding a position offset caused by additive manual mode (G201) or block search. If a position offset caused by additive manual mode is generated during backward motion and backward motion continues to program start or kinematic or Cartesian transformations are selected or deselected by the commands #CS ON[], #TRAFO ON, the backward motion function is terminated with an error. Further information: Backward motion, see [FCT-C7// Section: Forward/backward motion on the path] Block search, see [FCT-C6// Section: Restart to the contour after block search] For transformation see [PROG// Section: Tool centre point rotation] [KITRA].		
Response	Class	6	Program execution stop.
Solution	Class	6	No position offset should be enabled before activation of backward movement and no position offset should be generated while backward movement. Position offset is cleared by NC reset or a new program start.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
	%2:	Error value [0.1 µm or 0.0001°]	
		Current active position offset caused by addictive manual operation mode.	
	%3:	Error value [0.1 µm or 0.0001°]	
		Current PCS position offset caused by block search.	
	%4:	Limit value [0.1 µm or 0.0001°]	
	%5:	End value [0.1 µm or 0,0001°]	
Error type	1, Error message from NC program.		

ID 50515

Maximum number of unacknowledged signals exceeds limit. #SIGNAL could not be saved.			
Description	The maximum number of unacknowledged signals was exceeded. No new signals can be saved.		
Response	Class	4	Movement stop.
Solution	Class	6	Reduce the signals to be saved, e.g. by acknowledging signals.
Parameter	%1:	Current value [-]	
		Signal ID that could not be saved.	
	%2:	Limit value [-]	
		Maximum possible number of memory locations for signals	
Error type	1, Error message from NC program..		

ID 50516

Maximum number of unacknowledged signals exceeds limit. #WAIT could not be saved.			
Description	The number of unacknowledged #SIGNAL, #WAIT synchronisations is limited. For detailed information on cross-channel synchronisation types, see [PROG// Section: Synchronisation scenarios]		
Response	Class	4	Movement stop.
Solution	Class	6	Check the NC program and reduce the number of #SIGNAL, #WAIT active simultaneously.

Parameter	%1:	Current value [-]
		Number of the signal.
	%2:	Limit value [-]
		Maximum number of active signals
Error type	1, Error message from NC program..	

ID 50517

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50519

Cancellation of request without axis registration.			
Description	This error message is only output for diagnostic purposes. The error message appears in diagnosis protocol.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Diagnosis
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
Error type	3, Error in the communication. .		

ID 50520 - 50528

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50529

Incorrect parameter setting for measurement run to fixed stop.			
Description	In conjunction with the measurement function G100, several measurement types are available. These types can be selected via the MDS or the CNC program. An impermissible parameter setting is active during a measurement run to fixed stop. For detailed information on G100, see • [FCT-C4], • [PROG// Section: Measuring functions] • [PROG// Section: Settings for measurement]		
Response	Class	4	Movement stop.
Solution	Class	6	Change measurement type or parameter value 2

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis
	%2:	Current value [-]
		Name of machine data record parameter
	%3:	Current value [-]
		Machine parameter value
	%4:	Current value [-]
		Selected measurement type
	%5:	Current value [-]
		Measurement axes with motion
Error type	1, Error message from NC program..	

ID 50533

Maximum permissible axis acceleration exceeded.			
Description	This error message is only output when the environment variable is set for diagnosing the Slope function. The error message appears in the diagnostic log.		
Response	Class	1	None
Solution	Class	1	None
Parameter	%1:	Current value [-]	
		Current jerk excess factor.	
	%2:	Limit value [-]	
		Factor for permissible jerk excess.	
Error type	1, Error message from NC program..		

ID 50534

Invalid axis type for selected kinematic transformation.			
Description	When the kinematic transformation was activated, the program detected the incorrect axis type. For example, this may occur during axis exchange when a rotary modulo axis is exchanged to a transformation axis unit that only allows translatory axes (e.g. Cartesian machine with translatory axes X, Y, Z). For more information on transformation, see [PROG// Section: Rotation tool centre point (RTCP)] [KITRA].		
Response	Class	6	Program abort.
Solution	Class	7	Set the correct axis type.
Parameter	%1:	Current value [-]	
		String variable of axis type in axis parameter list	
	%2:	Current value [-]	
		Current axis type setting P-AXIS-00018.	
	%3:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%4:	Current value [-]	
		Active kinematic ID	
Error type	1, Error message from NC program.		

ID 50535 - 50539

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50540

Tilt angle of grinding disc exceeds limit.			
Description	The tilt angle of the grinding disc is outside the permitted range. The grinding disc tilt angle can be adjusted via the parameter grinding_disc_tilt_angle (P-TOOL-00138) in the tool list for the appropriate tool.		
Response	Class	3	Abort current job.
Solution	Class	6	Correct the grinding_disc_tilt_angle parameter in the tool list for the corresponding tool.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		Grinding disc tilt angle.	
	%2:	Upper limit value [0.1 10^-3 mm or ø]	
		Maximum tilt angle.	
	%3:	Lower limit value [0.1 10^-3 mm or ø]	
		Minimum tilt angle.	
Error type	-		

ID 50541

Maximum permissible discrete infeed exceeds limit.		
Description	The discrete infeed of an axis via PLC exceeds the permitted value range. The maximum infeed is defined in parameter P-TOOL-00031.	
Response	Class	1 Infeed is limited.
Solution	Class	1 Reduce the discrete infeed of the axis via the PLC.
Parameter	%1:	Logical axis number [0.1 µm or 0.0001°]
	%2:	Error value [0.1 µm or 0.0001°]
	%3:	Lower limit value [0.1 µm or 0.0001°]
	%4:	Upper limit value [0.1 µm or 0.0001°]
	%5:	Corrected value [0.1 µm or 0.0001°]
Error type	1, Error message from NC program.	

ID 50542

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50543

Maximal permissible discrete radius infeed exceeds limit.			
Description	The discrete radius infeed via PLC exceeds the permissible value range. The maximum infeed is defined via parameter P-TOOL-00031.		
Response	Class	1	Infeed is limited.
Solution	Class	1	Reduce the discrete radius infeed
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Maximum infeed parameterised P-TOOL-00031	
	%4:	Corrected value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 50544

Axis velocity exceeds maximum limit.			
Description	This error message is only output when the environment variable is set for diagnostic purposes of the slope function. The error message appears in the diagnostic log.		
Response	Class	4	Movement stop.
Solution	Class	1	Diagnosis
Parameter	%1:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible path velocity.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current path velocity.	
Error type	1, Error message from NC program.		

ID 50545

Axis acceleration exceeds maximum limit.			
Description	This error message is only output when the environment variable is set for diagnostic purposes of the slope function. The error message appears in the diagnostic log.		
Response	Class	4	Movement stop.
Solution	Class	1	Diagnosis.

Parameter	%1:	Limit value [mm/s² or °/s²]
		Lower limit of permitted path acceleration
	%2:	Limit value [mm/s² or °/s²]
		Upper limit of permitted path acceleration
	%3:	Current value [mm/s² or °/s²]
		Current path acceleration
Error type	1, Error message from NC program.	

ID 50546 - 50556

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50557

Characteristic filter frequency greater than half the sampling rate.			
Description	According to Shannon's sampling theorem, the maximum cut-off frequency is (½*T_A= cycle time of the NC controller). The cut-off frequency of the PT1 filter is calculated from the time constant as follows: fg = 1 / (2 * PI * P-AXIS-00357). The cut-off frequency of all other filters is specified directly in the parameter P-AXIS-00067. Possible solutions: Reduce cut-off frequency P-AXIS-00067. Increase the time constant P-AXIS-00357 for PT1 filters. Reduce the cycle time of the NC controller See [FCT-A7// Section: PT1 filters]		
Response	Class	3	The filter is deactivated.
Solution	Class	7	Reduce the cut-off frequency P-AXIS-00067 or increase the time constant P-AXIS-00357
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:		
		Number of filter affected	
	%3:		
		Filter type P-AXIS-00204	
	%4:		
		Current filter cut-off frequency P-AXIS-00067	
	%5:		
		Sampling frequency of the controller (1 / T_A)	
Error type	-		

ID 50558

Filter design with given parameters not possible.			
Description	It is not possible to find valid filter parameters for all filter parameters, for example, due to the limited calculation accuracy. Possible solutions are: Reduce order P-AXIS-00140 of the filter Reduce the reciprocal bandwidth P-AXIS-00080 of band-pass or band-stop filters Choose a different filter prototype P-AXIS-00153 Increase the ratio of cut-off frequency to sampling rate		
Response	Class	3	Filter is deactivated
Solution	Class	7	Change filter parameters
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:		
		Number of filter affected	
	%3:		
		Filter type P-AXIS-00204	
	%4:		
		Filter order P-AXIS-00140	
	%5:		
		Filter cut-off frequency P-AXIS-00067 or time constant P-AXIS-00357 of PT1 filter/PT2 filter	
Error type	-		

ID 50559 - 50561

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50562

Drive command could not be written.			
Description	Too many #DRIVE commands were commanded for one axis.		
Response	Class	6	Movement stop.
Solution	Class	6	Reduce number of #DRIVE commands in NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
Error type	1, Error message from NC program.		

ID 50563

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50564

Stability reserve of designed filter not sufficient.			
Description	It is not possible to design a stable filter for all filter parameters, e.g. due to the limited calculation accuracy. Possible solutions are: • Reduce order P-AXIS-00140 of the filter • Reduce the reciprocal bandwidth P-AXIS-00080 of band-pass or band-stop filters • Choose a different filter prototype P-AXIS-00153 • Increase the ratio of cut-off frequency to sampling rate For further information see [FCT-A7// Section: Filter types]		
Response	Class	3	Abort current job. Filter is deactivated
Solution	Class	6	Change filter parameters
Parameter	%1:	Current value [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Number of filter affected	
	%3:	Current value [-]	
		Filter type P-AXIS-00204	
	%4:	Current value [-]	
		Filter order P-AXIS-00140	
	%5:	Current value [-]	
		Filter cut-off frequency P-AXIS-00067 or time constant P-AXIS-00357 from PT1 filter or PT2 filter	
Error type	-		

ID 50565 / 50566

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 50567

Timeout: Axis filter not within tolerance window.

Description	On reset, axis position request or Exact stop, the CNC waits for the axis filter to reach the tolerance window P-AXIS-00351. If this fails to occur within a certain period of time (e.g. due to a strongly oscillating filter), the axis filters for the affected axis are temporarily switched off. They are reactivated after a CNC reset. Possible solutions are: Zoom tolerance window P-AXIS-00351 for filters Reduce order P-AXIS-00140 of the filter Reduce the reciprocal bandwidth P-AXIS-00080 of band-pass or band-stop filters Choose a different filter prototype P-AXIS-00153 Increase the ratio of cut-off frequency to sampling rate		
Response	Class	6	Filter is temporarily deactivated and reactivated after reset
Solution	Class	7	Change filter parameters or increase tolerance window
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		True, if axis filters have reached position window	
	%3:	Current value [0.1 µm or 0.0001°]	
		Unfiltered command value	
	%4:	Current value [0.1 µm or 0.0001°]	
		Filtered command value	
	%5:	Upper limit value [0.1 µm or 0.0001°]	
		Tolerance window P-AXIS-00351 for axis filter	
Error type	-		

ID 50568

WCS transformation of display data impossible.			
Description	The display of W0 coordinates is selected P-CHAN-00145. The kinematic transformation is not possible with the current selected kinematic structure. The reason could be an invalid kinematic identifier or wrong parameter setting of the kinematic. For more information on transformation parameters and W0 display data see [KITRA].		
Response	Class	1	Deactivate W0 display function.
Solution	Class	1	Only use valid kinematic identifier and correct transformation parameters.
Parameter	%1:	Current value [-]	
		Currently selected kinematic ID.	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50569

WCS transformation with current axis configuration impossible.

Description	The display of W0 coordinates is selected P-CHAN-00145. The kinematic transformation is not possible with the current selected kinematic structure. An invalid axis configuration with too few or missing axes is active. For more information on kinematic transformations and W0 display data, see [KITRA]		
Response	Class	1	Deactivate W0 display function.
Solution	Class	1	Use correct axis configuration setting.
Parameter	%1:	Current value [-]	
		Currently selected kinematic ID.	
	%2:	Current value [-]	
		Axis index of missing axis.	
Error type	1, Error message from NC program.		

ID 50570

Frames per second not a multiple integer of CNC cycle time.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
	%3:	Error value [µs]	
Error type	-		

ID 50571

Frames per second cannot be smaller than CNC cycle time.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
	%3:	Error value [µs]	
Error type	-		

ID 50572

The given neg. offset limit is greater than the positive offset limit.	
Description	The CNC objects describable via the PLC for absolute offset limits of manual mode "manual act. abs. limit-" and "manual act. abs. limit+" were set to invalid values. In this case, the lower limit is greater than the upper limit.

Response	Class	1	Offset limits are set to the current axis position. This means that the axis can no longer be moved in manual mode.
Solution	Class	1	Select the negative offset limit value so that it is less than the positive offset limit.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
		Last commanded absolute negative offset limit	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Last commanded absolute negative offset limit	
	%4:	Current value [0.1 µm or 0.0001°]	
		Current position of the axis in the programming coordinate system PCS	
Error type	1, Error message from NC program.		

ID 50573

Modal active measuring function has not been deactivated.			
Description	The channel parameter P-CHAN-00153 at edge banding controls whether the end of the measuring cycle or the SERCOS command S-0-170 is implicit or controlled via G107. In the event of an error, deactivation via G107 is missing at program end. For more information on edge banding, see [PROG// Section: Edge banding].		
Response	Class	4	Movement stop.
Solution	Class	6	Before program end or M30 execute, correct deselecting with G107
Error type	1, Error message from NC program.		

ID 50574 - 50576

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50578 - 50586

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50591

Inadmissible motion block type for the OUT synchronisation.

Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50593

Oscillation feed is zero.			
Description	The programmed oscillation axes function feedrate is too small. For more information on oscillation axes see [FCT-A8// Section: Oscillating motion dynamics].		
Response	Class	5	Movement stop.
Solution	Class	6	Check feed programming, value should be > 0
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Programmed feed.	
Error type	1, Error message from NC program.		

ID 50594

Transformation of actual positions not possible.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50595

Trafo for conveyor sync. with axis config. and kin. structure not possible.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	

Error type	1, Error message from NC program.

ID 50597

Belt speed Conveyor velocity [CONV_VEL] for #SYNC IN not programmed or equal to 0.		
Description	In the first #SYNC IN command the velocity of the conveyor belt has to be specified	
Response	Class	4
Solution	Class	6
Parameter	%1:	Current value [1µm/s or 0.001°/s]
	%2:	Lower limit value [1µm/s or 0.001°/s]
Error type	1, Error message from NC program.	

ID 50599

Enabling/disabling kin. transformation not permissible with active belt sync.		
Description		
Response	Class	6
Solution	Class	6
Error type	1, Error message from NC program.	

ID 50600

License: too many axes are to move at the same time.		
Description		
Response	Class	1
Solution	Class	7
Parameter	%1:	Limit value [-]
Error type	-	

ID 50601

Inadmissible command after Sync. OUT.		
Description	The target position after #SYNC OUT is programmed as an independent axis which can be interrupted by the following #SYNC IN command: <axis_name>[INDP_SYN … POS<value> INTERRUPTIBLE].	
Response	Class	6
Solution	Class	6

Parameter	%1:	Current value [-]
	%2:	Expected value [-]
	%3:	Current value [-]
	%4:	Block number [-]
Error type	1, Error message from NC program.	

ID 50603

Inadmissible command for independent axes.			
Description	An invalid option was programmed to command an independent axis. Some options are only allowed in combination with synchronisation to conveyor movement. Example: Wrong: X[INDP_ASYN POS=6000 G90 G00 **Inadmissible command for independent axes.**]. Correct: X[INDP_ASYN POS=6000 G90 G00]. For further information, see [PROG// Section: Independent axes		
Response	Class	4	Movement stop.
Solution	Class	6	Correct the NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016	
	%2:	Current value [-]	
		Internal ID of invalid option	
Error type	1, Error message from NC program.		

ID 50604

The order of the position filter is greater than the acceptable maximum.			
Description	The parameter setting of the order P-AXIS-00622 of the position filter is greater than the maximum value.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Correct P-AXIS-00622.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 50605

The order of the position filter is odd numbered.			
Description	The order (P-AXIS-00622) of the position filter is assigned an odd number; the value will be corrected.1		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Parameterise P-AXIS-00622 with an even number.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 50606

Inadmissible type of position filter.			
Description	The type (P-AXIS-00621) of the position filter is assigned an invalid value.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	P-AXIS-00621 must have a valid value assigned.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 50607

The order of the velocity filter is greater than the acceptable maximum.			
Description	The order of the velocity actual value filter (P-AXIS-00623) is assigned a value which is greater than the maximum permitted.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	P-AXIS-00623 must have a valid value assigned.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 50608

The order of the position filter is odd numbered.			
Description	The order of the velocity actual value filter (P-AXIS-00623) is assigned an odd value; this will be corrected.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Parameterise P-AXIS-00623 with an even number.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 50609

End of motion range in direction of positive software limit switch reached.			
Description	The positive motion range limit of the X axis was reached. With throughfeed machining, the X axis co-moving with the conveyor decelerates in order to stop in front of the positive software limit switch. The error may only occur when the conveyor moving direction is positive. There are different possible reasons: • The conveyor velocity is too high. • The feed rates in the NC program are too small. • Too many contour elements, or description is too extensive (path of Y axis)		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct conveyor velocity and NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Positive limit of travel	
	%3:	Current value [0.1 µm or 0.0001°]	
		Current position including braking distance	
	%4:	Current value [0.1 µm or 0.0001°]	
		PCS 0 position	
	%5:	Current value [0.1 µm or 0.0001°]	
		Braking distance	
Error type	1, Error message from NC program.		

ID 50610

End of motion range in direction of negative software limit switch reached.

Description	The negative motion range limit of the X axis was reached. With throughfeed machining, the X axis co-moving with the conveyor decelerates in order to stop in front of the negative software limit switch. This error should only occur if the conveyor moving direction has a negative sign. There are different possible reasons: • The conveyor velocity is too high. • The programmed feed rates in the NC program are too small • Too many contour elements, or description is too extensive (path of Y axis)		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct conveyor velocity and NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Negative limit of travel	
	%3:	Current value [0.1 µm or 0.0001°]	
		Current position including braking distance	
	%4:	Current value [0.1 µm or 0.0001°]	
		PCS 0 position	
	%5:	Current value [0.1 µm or 0.0001°]	
		Braking distance	
Error type	1, Error message from NC program.		

ID 50611

The number of look-ahead blocks is too small.			

Description	The number of blocks for look-ahead feed preparation (look-ahead) can be set.
	The value configured in the parameter P-CHAN-00653 (or P-STUP-00071) is too small.
	`configuration.interpolator.number_blocks_lah`
	
	A requirement for the effect of the parameter is that P-CHAN-00650 (or P-STUP-00070) must be together with FCT_LOOK_AHEAD_CUSTOM.
	Entry in the channel parameter list for P-CHAN-00650:
	`configuration.interpolator.function FCT_LOOK_AHEAD_CUSTOM`
	
	In older CNC versions, the value was defined by the following parameter (P-STUP-00071)
	configuration.channel[<channel_indexr>].*interpolator.parameter*
	.		

Response	Class	3	Controller corrects the value
Solution	Class	1	Check and correct the configuration parameter P-CHAN-00653 (or P-STUP-00071)
Parameter	%1:	Current value [-]	
		Parameterised value of the configuration parameter	
	%2:	Lower limit value [-]	
		Minimum value of the configuration parameter	
	%3:	Corrected value [-]	
		Corrected value for the configuration parameter	

Error type	-

ID 50612

The number of look-ahead blocks is too small.			

Description	The number of blocks for look-ahead feed preparation (look-ahead) can be set.
	The value configured in the parameter P-CHAN-00653 (or P-STUP-00071) is too large.
	`configuration.interpolator.number_blocks_lah`
	
	A requirement for the effect of the parameter is that P-CHAN-00650 (or P-STUP-00070) must be together with FCT_LOOK_AHEAD_CUSTOM.
	Entry in the channel parameter list for P-CHAN-00650:
	`configuration.interpolator.function FCT_LOOK_AHEAD_CUSTOM`
	
	In older CNC versions, the value was defined by the following parameter (P-STUP-00071)
	configuration.channel[<channel:indexr>].*interpolator.parameter*
	.		

Response	Class	3	Controller corrects the value
Solution	Class	1	Change P-STUP-00071 parameter in the start-up list.
Parameter	%1:	Current value [-]	
		Parameterised value of the configuration parameter	
	%2:	Upper limit value [-]	
		Maximum value of the configuration parameter	
	%3:	Corrected value [-]	
		Corrected value for the configuration parameter	
Error type	-		

ID 50613

The workpiece leaves the working area before it is synchronised.			
Description	The workpiece cannot be reached,;the axis parallel to the conveyor belt may be set too slow. The interval [a,b] of the working space of the belt axis is defined according to the belt direction 0/1: • Positive direction: [wait position, upper software limit switch] • Negative direction: [lower software limit switch, wait position]. The moving direction of the conveyor belt and the waiting position of the belt axis are defined in the channel parameters *conveyor_sync.move_direction* = 0/1 and *conveyor_sync.pos_limit*.		
Response	Class	6	Movement stop.
Solution	Class	-	

Parameter	%1:	Logical axis number [-]
		Logical number of the axis parallel to the conveyor belt
	%2:	Current value [0.1 10^-3 mm or ø]
		Origin of the workpiece in the axis coordinate system (ACS)
	%3:	Lower limit value [0.1 10^-3 mm or ø]
		Lower limit (a) of the workspace area in the ACS
	%4:	Upper limit value [0.1 10^-3 mm or ø]
		Upper limit (b) of the workspace area in the ACS
Error type	-	

ID 50614 / 50615

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50616

The velocity of the conveyor belt is greater than the rapid feed in direction of belt.			
Description	The velocity of the conveyor belt CONV_VEL specified in the #SYNC IN command exceeds the rapid traverse velocity of the axis parallel to the conveyor belt.		
Response	Class	6	Movement stop.
Solution	Class	-	
Parameter	%1:	Logical axis number [-]	
		Number of the axis in the conveyor belt direction	
	%2:	Current value [µm/s]	
		Programmed conveyor belt velocity	
	%3:	Upper limit value [0.1 µm/s]	
		Maximum velocity of the belt axis in rapid traverse	
Error type	-		

ID 50618

Axis exceeds its range during Sync-Out or Sync-Stop.			
Description	The workspace is violated when moving back to the rest position specified after #SYNC OUT or when stopping. The interval [a,b] of the working space of the belt axis is defined according to the belt direction 0/1: Positive direction: [wait position, upper software limit switch] Negative direction: [lower software limit switch, wait position]. The moving direction of the conveyor belt and the waiting position of the belt axis are defined in the channel parameters *conveyor_sync.move_direction* = 0/1 and *conveyor_sync.pos_limit*.		
Response	Class	6	Movement stop.
Solution	Class	-	

Parameter	%1:	Logical axis number [-]
		Logical number of the axis parallel to the conveyor belt
	%2:	Lower limit value [0.1 µm]
		Lower limit for the calculated belt axis motion in the ACS
	%3:	Upper limit value [0.1 µm]
		Upper limit of the calculated belt axis motion in the ACS
	%4:	Lower limit value [0.1 µm]
		Lower limit (a) of the workspace in the ACS
	%5:	Upper limit value [0.1 µm]
		Upper limit (a) of the workspace in the ACS
Error type	-	

ID 50619

The velocity of the conveyor belt is greater than the maximum feed rate in direction of belt.			
Description	The velocity CONV_VEL specified in the #SYNC IN command exceeds the maximum velocity of the axis parallel to the conveyor belt.		
Response	Class	6	Movement stop.
Solution	Class	-	
Parameter	%1:	Logical axis number [-]	
		Number of the axis in the conveyor belt direction	
	%2:	Current value [0.1 µm/s]	
		Programmed conveyor belt velocity	
	%3:	Upper limit value [0.1 µm/s]	
		Maximum velocity of the belt axis in G1 blocks	
Error type	-		

ID 50620 - 50624

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50625

Axis exchange during active suppression of program start initialisation not allowed.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Current value [-]	

Error type	1, Error message from NC program.

ID 50626

Gear change during active suppression of program start initialisation not allowed.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Expected value [-]	
Error type	1, Error message from NC program.		

ID 50627

Coupling on a conveyor is enabled at program end.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50629

Axis is controlled by single axis IPO.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50630

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50631

Moved axis is not at the position before the retraction.
Description

Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [0.1 µm or 0.0001°]	
	%3:	Expected value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 50633

Sync IN/OUT is interrupted by single axis movement.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 50634

Sync IN/OUT commanded during error.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50635

Forced error during SYNC IN.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50636

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50637

Changing measuring parameters only allowed for measurement axes.		
Description		
Response	Class	4
Solution	Class	6
Parameter	%1:	Logical axis number [-]
	%2:	Error value [-]
	%3:	Current value [-]
	%4:	Current value [-]
	%5:	Current value [-]
Error type	1, Error message from NC program.	

ID 50638

Drive type does not support selected probing signal input.		
Description		
Response	Class	4
Solution	Class	6
Parameter	%1:	Logical axis number [-]
	%2:	Drive type [-]
	%3:	Error value [-]
Error type	1, Error message from NC program.	

ID 50639

Selected probing input of drive is invalid.		
Description		
Response	Class	4
Solution	Class	6

Parameter	%1:	Logical axis number [-]
	%2:	Drive type [-]
	%3:	Error value [-]
	%4:	Upper limit value [-]
Error type	1, Error message from NC program..	

ID 50640

Edge banding cannot be activated since function not selected in axis parameters.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
	%3:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50641

Adoption of new measuring parameters not possible during active measurement.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50642

Maximum number of timer for conveyor synchronisation exceeded.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 50643

Positive software limit exceeded with automatic repositioning in block search.

Description	An additional check of axis positions is carried out with block search repositioning movement. It may happen that the approach movement results in software limits being exceeded, especially in combination with active kinematic transformation because of necessary compensation motions.		
Response	Class	6	Movement stop.
Solution	Class	6	Move axis back to within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the affected axis.	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current axis position.	
	%3:	Upper limit value [0.1 10^-3 mm or ø]	
		Positive software limit switch P-AXIS-00178	
Error type	1, Error message from NC program..		

ID 50644

Negative software limit switch exceeded with automatic repositioning in block search.			
Description	An additional check of axis positions is carried out with block search repositioning movement. It may happen that the approach movement results in software limits being exceeded, especially in combination with active kinematic transformation because of necessary compensation motions.		
Response	Class	6	Movement stop. .
Solution	Class	6	Move axis back to within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis effected.	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current axis position.	
	%3:	Lower limit value [0.1 10^-3 mm or ø]	
		Negative software limit switch P-AXIS-00177	
Error type	1, Error message from NC program..		

ID 50645 - 50648

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50649

Single axis movement interrupts the LIFT-function.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	

Error type	-

ID 50650 - 50652

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires controller restart.

ID 50653

Current velocity of band is greater than the programmed velocity of band.			
Description	The conveyor belt moves faster than the value CONV_VEL specified in the command #SYNC IN[… CONV_VEL=<value>].		
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 50654

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50655

Not enough block path available for path edge band path.			
Description	The “cross-block edge banding” function (G108) checks whether the edge banding residual path can be reduced when deselected by G107. Depending on the channel parameter P-CHAN-00227, the preliminary check of the block motion path is suppressed. In this case, an insufficiently long block motion path was programmed before deselection by G107. For more information on edge banding, see [PROG// Section: Edge banding (G108)].		
Response	Class	4	Movement stop. .
Solution	Class	6	Check programmed motion before G107
Error type	1, Error message from NC program.		

ID 50656 - 50659

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50660

Feedforward parameter could not be written to position controller.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50662

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50663

Given filter time constant is invalid.			
Description	The specified filter time constant P-AXIS-00357 is smaller or larger than permissible. The filter coefficients can therefore not be calculated. Permissible values are (where T_{Ab} = cycle time of the NC controller): For PT1 and PT2 filters: $T_{Ab} \leq$ P-AXIS-00357 $\leq$ MAX(UNS32) For time delay filters: $0 \leq$ P-AXIS-00357 $< 6 * T_{Ab}$		
Response	Class	3	The filter is deactivated.
Solution	Class	7	Correct parameter P-AXIS-00357

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Current value [-]
		Number of filter affected
	%3:	Current value [-]
		Filter type P-AXIS-00204
	%4:	Current value [µs]
		Invalid time constant P-AXIS-00357
	%5:	Upper limit value [µs]
		Maximum permissible time constant P-AXIS-00357
Error type	-	

ID 50664

Measuring in independent axis not initialised.			
Description	The measuring hardware was not activated when the measurement function G100 is used in an independent axis. For detailed information on measurement function G100, see [FCT-C4] [PROG// Section: Independent axes].		
Response	Class	4	Movement stop. .
Solution	Class	6	Reduce the delay time for initialising the measurement hardware.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	1, Error message from NC program.		

ID 50665

Maximum number of synchronous axes exceeded (license).			
Description	The number of simultaneously used axes is exceeded. The limit is caused by the available license.		
Response	Class	6	Movement stop.
Solution	Class	7	Contact your controller manufacturer to procure the required license or reduce the number of axes required.
Error type	-		

ID 50666

Interpolation type is not supported for npath IPO.			
Description			
Response	Class	3	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50667

Main axis is missing for circular interpolation in npath IPO.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value[-]	
	%2:	Current value[-]	
Error type	1, Error message from NC program.		

ID 50668 - 50671

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50673

CS can not be stored because stack is full.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%3:	Limit value [-]	
Error type	1, Error message from NC program..		

ID 50674

CS can not be stored because the CS name is not known.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50675

Synchronisation: synchronisation axis in error! Motion is stopped.
Description

Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50676

Intersection calculation during Cartesian forward transformation impossible.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50677

Intersection calculation during Cartesian backward transformation impossible.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50678

Virtual lead axis not available.			
Description	The function "spindle as lead axis" requires a virtual lead axis configured as path axis which ensures coupling of spindle movement to path movement.		
Response	Class	6	Program execution stop.
Solution	Class	6	Check and correct configuration.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50679 / 50680

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50681

Virtual lead axis not moved.			
Description	The virtual lead axis should be programmed and moved in function "spindle as lead axis“. In current case the virtual lead axis does not move.		
Response	Class	6	Program execution stop.
Solution	Class	6	Check and correct program.
Error type	1, Error message from NC program.		

ID 50682 / 50683

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50684

Warning: Synchronisation movement inaccurate.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 50685

Synchronisation: error: acceleration needed for synchronisation axis too high.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value[0.1 µm or 0.0001°]
	%2:	Limit value [0.1 µm or 0.0001°]
	%3:	Limit value[1µm/s or 0.001°/s]
	%4:	Identification number [-]
Error type	-	

ID 50686

Homing of axis not possible since homing is disabled in axis parameters.			
Description	The axis cannot be homed since homing is deactivated (see axis parameter homing_type P-AXIS-00299).		
Response	Class	6	Error message output, abort NC program

Solution	Class	6	Correct the NC program. If the axis does not have an absolute measurement system, change parameter P-AXIS-00299.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis	
	%3:	Current value [-]	
		Current value of the parameter P-AXIS-00299	
Error type	1, Error message from NC program.		

ID 50687

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50688

Axis not at standstill! Update of filter parameters not possible! Changes are discarded!			
Description	A list update or the #MACHINE DATA command changed parameters of the axis filter ([FCT-A7//Section: Parameter overview]). However, this is only permitted when the axis is stationary.		
Response	Class	3	Error messages; new parameters are not accepted
Solution	Class	7	Repeat the parameter update when the axis is at standstill
Parameter	%1:	Current value [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Index of the filter affected	
	%3:	Current value [-]	
		TRUE : Axis filter has reached the target position	
	%4:	Current value [-]	
		TRUE : Command values are output to the axis, i.e. the axis moves	
Error type	-		

NOTICE

The changed filter parameters have no effect in the controller. The changes are discarded.

ID 50689

Calculation of the online tool radius compensation reports an error.			
Description			
Response	Class	4	
Solution	Class	6	

Parameter	%1:	Current value [-]
Error type	1, Error message from NC program.	

ID 50690 / 50691

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Movement stop of axes.
Solution	Class	8	Requires controller restart.

ID 50692

Online tool radius compensation returns error during activation.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program..		

ID 50693

Error during calculation of normal to contour tangent.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50694

Command position of axis exceeds maximum value range.		
Description		
Response	Class	4
Solution	Class	7
Parameter	%1:	Logical axis number [-]
	%3:	Lower limit value [0.1 µm or 0.0001°]
	%4:	Upper limit value [0.1 µm or 0.0001°]

Error type	1, Error message from NC program..

ID 50695

Programmed function not available in this configuration.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Block number [-]	
	%2:	Logical axis number [-]	
	%3:	Expected value [-]	
Error type	1, Error message from NC program.		

ID 50696

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50697

The commanded positive/negative ACS limitation exceeds software limit switch.			
Description	When manual mode is enabled, a CNC object was commanded to change the motion range. There are 2 possible causes for the output of this warning: 1. The commanded CNC object “manual acs absolute limit –” is greater than the positive software limit switch (P-AXIS-00178), 2. The commanded CNC object “manual acs absolute limit +” is smaller than the positive software limit switch (P-AXIS-00177). (See also absolute manual mode movement limits P-AXIS-00492, P-AXIS-00493 in the MDS). The commanded value is ignored.		
Response	Class	1	None.
Solution	Class	1	Set commanded motion limits correctly, negative motion limit must be smaller than the positive motion limit or the commanded positive motion limit must be larger than the negative motion limit.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
		Current value of negative movement limit.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Current value for positive movement limit.	

Error type	1, Error message from NC program.

ID 50698

The commanded negative ACS limitation is above positive ACS limit.			
Description	When manual mode is enabled, a CNC object was commanded to change the motion range. The value of the CNC object “manual acs absolute limit +” is smaller than the value of the CNC object “manual acs absolute limit –”. (See also absolute manual mode movement limits P-AXIS-00492, P-AXIS-00493 in the MDS). The commanded value is ignored.		
Response	Class	1	None.
Solution	Class	1	Correct setting of movement limits inside soft limit range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
		Current value of negative movement limit.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Current value for positive movement limit.	
Error type	1, Error message from NC program.		

ID 50700 - 50705

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50706

End of movement range reached, measuring not executed.			
Description	No measurement signal was enabled in one or more axes with measurement function G100 and the option of automated motion path limiting during measurement run (see P-CHAN-00268). The movement stops before reaching the movement range limits defined by the soft limit switch. For detailed information on measurement function G100, see [FCT-C4], [PROG// Section: Measuring functions].		
Response	Class	6	Movement stop.
Solution	Class	6	Check measuring signal of the measurement axis/axes affected. Set measurement signal in all measurement axes with measurement interrupt.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of an affected measurement axis.	

Error type	1, Error message from NC program.

ID 50707

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50708

Axis polynomial programmed without main axis motion.			
Description	An axis polynomial was programmed (see [PROG// Section: Programming axis polynomials]), however, no motion of another axis was programmed for the motion block. Since the axis polynomial is de-interpreted synchronously with the other axes programmed in this NC block, it is not possible to evaluate the polynomial in this case.		
Response	Class	4	Program execution stop.
Solution	Class	6	Program a movement of another axis in the faulty NC line.
Error type	1, Error message from NC program.		

ID 50712

Forward/backward transformation decoder/interpolator inconsistent. Check the transformation.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%3:	Limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 50713

Between SYNC OUT and SYNC IN must be programmed an independent movement.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50715 / 50716

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50717

Homing (G74) not allowed during block search.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program..		

ID 50718

Measurement function (G100) not allowed during block search.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50719

SYNC_IN time marker passed. Target position not reached.			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [0.1 µm or 0.0001°]	
	%3:	Expected value [0.1 µm or 0.0001°]	
	%4:	Current value [0.1 µm or 0.0001°]	
	%5:	Current value [µs]	
Error type	1, Error message from NC program.		

ID 50720

Manual mode PCS movement limit reached.

Description	The maximum moving range with manual operation mode is defined by axis parameters P-AXIS-00137, P-AXIS-00138 and P-AXIS-00492, P-AXIS-00493. P-AXIS-00137, P-AXIS-00138 are PCS relative values relative to the position at the start of manual mode operation, P-AXIS-00492, P-AXIS-00493 are ACS absolute values similar to software limit switch parameters. When approaching these motion range limits, no error message is output by default, and the axis movement stops. The output of this warning can be activated by the manual MDS parameter P-MANU-00014. In this case one of the PCS limits is reached. The error message can be displayed in the user interface so that the operator can see the reason for the movement stop.		
Response	Class	1	Warning output, stop the axis movement at limit of motion range.
Solution	Class	1	Move axis back to permissible motion range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [0.1 µm or 0.0001°]	
		PCS command value	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Negative PCS motion limit P-AXIS-00137	
	%4:	Upper limit value [0.1 µm or 0.0001°]	
		Positive PCS motion limit P-AXIS-00138	
Error type	1, Error message from NC program.		

ID 50721

Manual mode ACS movement limit reached.			
Description	The maximum moving range with manual operation mode is defined by axis parameters P-AXIS-00137, P-AXIS-00138 und P-AXIS-00492, P-AXIS-00493. P-AXIS-00137, P-AXIS-00138 are PCS relative values relative to the position at the start of manual mode operation, P-AXIS-00492, P-AXIS-00493 are ACS absolute values similar to software limit switch parameters. When moving to this moving ranges limits, no error message is output by default, and the axis movement stops. The output of this warning can be activated via the Hand MDS parameter P-MANU-00014. In current case one of the ACS limits is reached. The error message can be displayed in the user interface so that the operator can see the reason of the movement stop.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [0.1 µm or 0.0001°]	
		ACS command value	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Negative ACS motion limit P-AXIS-00492	
	%4:	Upper limit value [0.1 µm or 0.0001°]	
		Positive ACS motion limit P-AXIS-00493	
Error type	1, Error message from NC program.		

ID 50722 / 50723

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50724

Maximum permissible velocity is exceeded by the sum of synchronous and superimposed path motions.		
Description		
Response	Class	4
Solution	Class	6
Parameter	%1:	Logical axis number [-]
	%2:	Current value [1µm/s or 0.001°/s]
	%3:	Upper limit value [1µm/s or 0.001°/s]
	%4:	Current value [-]
	%5:	Upper limit value [-]
Error type	1, Error message from NC program.	

ID 50725

Independent movement of synchronous axis after #SYNC OUT not programmed.		
Description		
Response	Class	2
Solution	Class	6
Error type	1, Error message from NC program.	

ID 50726

The communication of the Volumetric Compensation does not work, because the channel contains no axis.			
Description	This error occurs if a **#VOLCOMP** NC command is executed in a channel that does not contain an axis. For more information on volumetric compensation, see [FCT-C26]		
Response	Class	2	Stopping the axis, and aborting the NC program.
Solution	Class	3	Before executing the NC command **The communication of the Volumetric Compensation does not work, because the channel contains no axis.** add an axis to the axes in the channel and reset the CNC.

Parameter	%1:	Logical channel number [-]
		Channel number
	%2:	Identification number [-]
		ID of the Volumetric Compensation
Error type	1, Error message from NC program.	

ID 50727

Specified axis number in drive command not found.			
Description	The axis number specified in a #DRIVE command was not found.		
Response	Class	6	Program abort.
Solution	Class	6	Correct the NC program. The axis number programmed must be known in the NC channel. When the function 'Inserting NC programs' [FCT-C15]' is used, the channel parameter P-CHAN-00282 can be used to specify whether the axis numbers of the axis in the clone channel or the axis numbers of the axis in the master channel are to be used.
Parameter	%1:	Logical axis number [-]	
		Number of axis not found.	
	%2:	Current value [-]	
		Value of the channel parameter P-CHAN-00282.	
Error type	1, Error message from NC program.		

ID 50728

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50729

Interpolation direction not invertible during delete distance to go.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 50730

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	Error message output, abort NC program.
Solution	Class	8	Requires controller restart.

ID 50731

Inadmissible trafo selection in level 0.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50732

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 50733

Dynamic contour control returns error during activation.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50734

The option for dynamic contour control is not included in the current software version.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	Abort NC program processing.		

ID 50735

CalculateOnNewGeometry() of dynamic contour control returned an error.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50736

CalculateOrientation() of dynamic contour control returned an error.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50737

CalculateOffset() of dynamic contour control returned an error.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50738

An error occurred in a volumetric compensation.			
Description	This is a following error to an error in volumetric compensation.		
Response	Class	4	See preceding error
Solution	Class	6	See preceding error
Error type	1, Error message from NC program.		

ID 50745

Block search with covered distance and distance from program start double defined.			
Description			
Response	Class	2	
Solution	Class	7	

Parameter	%1:	Current value [-]
	%2:	Error value [-]
Error type	1, Error message from NC program.	

ID 50746

Restart position of block search not found within program.			
Description			
Response	Class	2	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50747

Block search to given distance from program start not possible, distance too low.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Lower limit value [-]	
	%3:	Corrected value [-]	
Error type	1, Error message from NC program.		

ID 50748

#FLUSH WAIT not allowed during block search.			
Description			
Response	Class	2	
Solution	Class	7	
Error type	1, Error message from NC program.		

2.6.4 ID-range 50750-50999

ID 50750

Scene database is not present.			
Description	An attempt was made to set the frame rate for 3D visualisation although no scene database is present exists in the CNC kernel.		
Response	Class	1	Warning output, frame rate is not set.
Solution	Class	1	Set parameter "enable_scene" in start-up list.
Error type	-		

ID 50752

Geometric feed adaptation returns error during activation.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50753 - 50760

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50761

Deviation of Master Slave offset exceeds restart limit.			
Description			
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program..		

ID 50762

Block search with per mille block segmentation to position request not possible.

Description	The use of block search with per mille block segmentation cannot be placed in NC program lines together with NC commands that are used for NC channel synchronisation. Example for these commands: • #CHANNEL INIT[…] • #OTC OFF		
Response	Class	4	Movement stop.
Solution	Class	6	Check and modify the jump point for block search in the NC program.
Parameter	%1:	Error value [-]	
		Specified per mille value	
Error type	1, Error message from NC program.		

ID 50763

Switch of IPO/LR axis interfaces cannot be done during measurement.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	-		

ID 50765

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart..

ID 50766

Axis movement not possible because single axis interpolation is enabled.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [0.1 µm or 0.0001°]	
	%3:	Current value [0.1 µm or 0.0001°]	
	%4:	Current value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program..		

ID 50767

Volumetric compensation file requested without configured axis.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 50768

The option for geometric feed adaptation is not contained in the current software version.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	Abort NC program processing.		

ID 50769 / 50770

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart..

ID 50771

Delete distance to go impossible, because backward storage is off.			
Description	The Delete distance to go function requires a parameter assignment in the backward memory. The backward memory is deactivated and can be assigned using P-STUP-00033.		
Response	Class	4	Movement stop.
Solution	Class	6	Activate the backward memory by assigning P-STUP-00033.
Error type	-		

ID 50772

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart..

ID 50773

Area not saved / maximum number areas exceeded.

Description	The maximum number of stored areas is limited. This limit was exceeded. The limit value is defined by a constant in the CNC. If you need to have a greater number of areas stored, contact your controller manufacturer. For this reason, the assigned area could not be saved. See also [FCT-C14].		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the NC program; do not delete required areas.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Limit value [-]	
		Maximum number of areas	
	%3:	Current value [-]	
		Current number of areas	
Error type	1, Error message from NC program.		

ID 50774

Area not saved / active area cannot be overwritten.			
Description	An attempt was made to overwrite an active area. Overwrite areas means to define an area with the same existing ID. See also [PROG// Defining a control area] and [FCT-C14// Description]. An area already exists with the commanded ID and it is currently activated. To overwrite an existing area, it should be inactive. **Example:** %areaoverwrite N10 #CONTROL AREA ON [**ID2**] N20 #CONTROL AREA BEGIN [**ID2** WORK …… (< error **Solution:** %areaoverwrite N10 #CONTROL AREA ON [**ID2**] N15 #CONTROL AREA OFF [**ID2**] N20 #CONTROL AREA BEGIN [**ID2** WORK.. (< overwrite		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify NC program; deactivate the area to be overwritten.
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

ID 50775 - 50777

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 50778

Z plane definition incorrect / minimum greater than maximum..			
Description	The description of the commanded areas is incorrect. The minimum value of the Z planes is greater than the maximum value. Constant limit values for the Z direction can be specified as limit values in the Z plane. See also [PROG// Workspace and protection space monitoring] and [FCT-C14// Description]. **Example:** %zmin_zmax N10 #CONTROL AREA BEGIN…. MIN_EXCUR=55 MAX_EXCUR=-20] (> error **Solution:** %zmin_zmax N10 #CONTROL AREA BEGIN…. MIN_EXCUR=-20 MAX_EXCUR=55] (> correct		
Response	Class	6	Movement stop.
Solution	Class	6	Correct the limit values for the Z plane.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Lower limit value [0.1 10^-3 mm or ø]	
		Lower limit of the Z plane	
	%3:	Upper limit value [0.1 10^-3 mm or ø]	
		Upper limit of the Z plane	
Error type	1, Error message from NC program.		

ID 50779

Start and endpoint must be the same for defining a polygonal area.			
Description	When defining polygon-shaped areas, the last dot of the polygon with the first dot of the polygon must coincide. A continuous polygon can be thereby guaranteed. See [PROG// Monitoring the work and protection area]. **Example:** %start not end N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X70 Y70 G01 F10000 G90 N30 X70 Y0 N40 X0 Y0 N50 X0 Y70 **N60 X75 Y75** N70 #CONTROL AREA END (> error **Solution:** %start not end N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X70 Y70 G01 F10000 G90 N30 X70 Y0 N40 X0 Y0 N50 X0 Y70 **N60 X70 Y70** N70 #CONTROL AREA END (> correct		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program.

Parameter	%1:	Identification number [-]
		Area ID
	%2:	Expected value [0.1 10^-3 mm or ø]
		Startingpoint of area definition.
	%3:	Current value [0.1 10^-3 mm or ø]
		Endpoint of area definition.
Error type	1, Error message from NC program.	

ID 50780

Definition of polygon is invalid. No intersections permitted.			
Description	The polygonal area must be a convex envelope. This means that individual polygon lines may not intersect each other. **Example:** %poly intersects N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X65 Y0 N30 X-65 Y0 **N40 X65 Y65** **N50 X-65 Y65** N60 X65 Y0 N100 #CONTROL AREA END (> error **Solution:** %poly intersects N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X65 Y0 N30 X-65 Y0 **N40 X-65 Y65** **N50 X65 Y65** N60 X65 Y0 N100 #CONTROL AREA END (> error		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Error value [0.1 10^-3 mm or ø]	
		Intersection of polygon lines	
Error type	1, Error message from NC program.		

ID 50781

The second block to define a circular area must be circular.

Description	The second motion block after starting a definition of a circular area must be a circular block for the circular description of the area. A circular area definition consists of two motion blocks, one linear at the starting point, and one circular to describe the circle. See also [PROG// Workspace and protection space monitoring] and [FCT-C14// Description]. **Example:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 **N30 G01 X150 Y200 F10000** N40 #CONTROL AREA END N50 M30 (> error **Solution:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 (start point) **N30 G02 G162 I70 J150** N40 #CONTROL AREA END N50 M30 (> correct)		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

ID 50782

No area with this ID available.			
Description	You programmed an area command with an invalid area ID. The programmed area ID does not exist.		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program. Correct the area ID.
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

ID 50783

Active area could not be cleared.			
Description	An active workspace or protection area cannot be cleared with the "CLEAR" command. This should prevent the deleting of currently monitored areas. **Example:** %areaclear N10 #CONTROL AREA ON [**ID2**] N20 #CONTROL AREA CLEAR[**ID2**] …… (< error **Solution:** %areaclear N10 #CONTROL AREA ON [**ID2**] N15 #CONTROL AREA OFF [**ID2**] N20 #CONTROL AREA CLEAR [**ID2**]< clear		
Response	Class	6	Movement stop.

Solution	Class	6	Deactivate the area using the command #CONTROL AREA OFF [ID xx] before deleting it
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

ID 50784

Movement limitation because of work area.			
Description	The work area in the IMCS (Intermediate Coordinate System) when a kinematic transformation is active can be restricted in manual mode by using corresponding limits. The limits are defined by parameters in the work area definition. The work area definition is defined by the NC command **#CONTROL AREA BEGIN [ID 1 WORK POLY MONITOR_LVL = IMCS…...]** When manual mode is active, motion jobs outside of these limits result in a motion stop within the permitted motion limits. A reset is not required; in most cases the axis can continue to be moved by a suitable backward motion. The output of this warning is dependent on the parameter P-MANU-00014. The following parameters can additionally restrict motion limits in manual mode: • P-AXIS-00137 / P-AXIS-00138 • P-AXIS-00177 / P-AXIS-00178		
Response	Class	1	Output of warning; stop axes
Solution	Class	1	Move the affected axis backwards from the motion limit in the work area.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the affected MCS/ACS axis.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Negative motion limit	
	%4:	Upper limit value [0.1 µm or 0.0001°]	
		Positive motion limit	
Error type	1, Error message from NC program.		

ID 50785

Only linear motion blocks G00/G01 can be used to define polygonal control areas.

Description	An invalid motion block was used in the definition of a polygonal control area of the workspace monitoring function (see [FCT-C14// Description]). Only linear motion blocks G00 or G01 are permitted for defining polygonal control areas.
	
	Example:
	`N010 G17 F10000`
	`N020 #CONTROL AREA BEGIN`
	`[ID=1 PROT POLY MIN_EXCUR = 0 AX_EXCUR = 200]`
	`N030 G01 X0 Y0`
	`N040 G01 X100 Y0`
	`N050 G02 X100 Y100 J+50 (error 120691)`
	`N060 G01 X100 Y0`
	`N070 G01 X0 Y0`
	`N080 #CONTROL AREA END`
	
	Correct:
	`N010 G17 F10000`
	`N020 #CONTROL AREA BEGIN`
	`[ID=1 PROT POLY MIN_EXCUR = 0 AX_EXCUR = 200]`
	`N030 G01 X0 Y0`
	`N040 G01 X100 Y0`
	`N050 G01 X100 Y100`
	`N060 G01 X100 Y0`
	`N070 G01 X0 Y0`
	N080 #CONTROL AREA END		

Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the definition of the control area in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID of affected control area	
	%2:	Error value [-]	
		Incorrect type of motion block	
Error type	1, Error message from NC program.		

ID 50786

Maximum number of points allowed for polygon exceeded.			
Description	The number of points to define a polygonal area is restricted to one limit value. This limit was exceeded. A polygonal area consists of a number of points representing a closed polygon. Thereby the starting point must the same as the target point. See also [FCT-C14// Polygonal control areas].		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program. Reduce the number of points in the area definition
Parameter	%1:	Identification number [-]	
		Affected area ID	
	%3:	Limit value [-]	
		Maximum number of points for a polygonal area.	
	%3:	Error value [-]	
		Current number of points for a polygonal area.	
Error type	1, Error message from NC program.		

ID 50787

Plane change in work or protection area definition not permitted.			
Description	The machining plane may not be changed during the definition of a work or protection area (see [FCT-C14// Definition of work and protection areas]). **Example with error:** #CONTROL AREA START [ID1 PROT CIRC MIN_EXCUR=-50 MAX_EXCUR=50] G01 Y100 Z100 F1000 **G19** G02 G162 Y100 Z100 J-100 F1000 #CONTROL AREA END **Corrected example:** **G19** #CONTROL AREA START [ID1 PROT CIRC MIN_EXCUR=-50 MAX_EXCUR=50] G01 Y100 Z100 F1000 G02 G162 Y100 Z100 J-100 F1000 #CONTROL AREA END		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Identification number [-]	
		Identifier of affected work or protection area	
Error type	1, Error message from NC program.		

ID 50788

Axis of the third control area direction (EXCUR) may not be used for the definition of the base plane.	

Description	A workspace or protection area in the workspace monitoring function (see [FCT-C14// Defining workspace and protection areas]) is defined by a base plane (polygon or circle) and an excursion EXCUR in the 3rd dimension. The axis defining the third excursion direction of the control area may not be part of the base plane definition:
	
	Example with error:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360 EXCUR_AX=X]`
	`G01 F1000 G90`
	`X0 Y0 (error 120734)`
	`X0 Y100`
	`X100 Y100`
	`X100 Y0`
	`X0 Y0`
	`#CONTROL AREA END`
	
	Corrected example:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360 EXCUR_AX=Z]`
	`G01 F1000 G90`
	`X0 Y0`
	`X0 Y100`
	`X100 Y100`
	`X100 Y0`
	`X0 Y0`
	#CONTROL AREA END

Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Current value [-]	
		ID of affected control area (area ID)	
	%2:	Current value [-]	
		Axes of the base plane	
	%3:	Error value [-]	
		3rd dimension axis	
Error type	1, Error message from NC program.		

ID 50789

The definition of the control area base plane requires the specification of two axes.			

Description	Normally, the control areas of the work space monitoring function (see [FCT-C14// Description]) refer to the first three Cartesian main axes. If a workspace or protection area is defined with tracking axes, both axes must be specified to define the base plane.
	See [PROG// Defining a control area]
	
	Example with error:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X2 = 0 (error 120736)`
	`X2 = 0 Y2=100`
	`X2 = 100 Y2=100`
	`X2 = 100 Y2=0`
	`X2 = 0 Y2=0`
	`#CONTROL AREA END`
	
	Corrected example:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X2 = 0 Y2=0`
	`X2 = 0 Y2=100`
	`X2 = 100 Y2=100`
	`X2 = 100 Y2=0`
	`X2 = 0 Y2=0`
	#CONTROL AREA END		

Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Current value [-]	
		ID of affected workspace or protection area	
	%2:	Error value [-]	
		Given axes in the area definition	
Error type	1, Error message from NC program.		

ID 50790

Too many control area groups.			
Description			
Response	Class	6	
Solution	Class	6	

Parameter	%1:	
	%2:	
	%3:	
	%4:	
Error type	1, Error message from NC program.	

ID 50791

Control area group not found.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50792

One linear and one circular block needed for circular area.			
Description	A circular area consists of a linear block as starting point and a circular block as circular description. The second motion block after starting a definition of a circular area must be followed by a circular block as circle description. See also [PROG// Monitoring the workspace and protection space] **Example with error:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 **N30 G01 X150 Y200 F10000** N40 #CONTROL AREA END N50 M30 **Corrected example:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 (start point) **N30 G02 G162 I70 J150** N40 #CONTROL AREA END N50 M30		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program

Parameter	%1:	Identification number [-]
		Area ID
	%2:	Error value [-]
		Intersection of polygon lines
Error type	1, Error message from NC program.	

ID 50793

A full circle without target point needed for circular area.			

Description	A circular area consists of a linear block as starting point and a circular block as circular description.
	A circular block must be a programmed full circle since areas can only consist of a full circle.
	See also [PROG// Workspace and protection space monitoring]
	
	Example with error:
	`%circular area`
	`N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU…`
	`N20 G01 X150 Y200 F10000`
	`N30 G03 G162 I50 J0 X250 Y200 F10000`
	`N40 #CONTROL AREA END`
	`N50 M30`
	
	`Corrected example:`
	`%circular area`
	`N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU…`
	`N20 G01 X150 Y200 F10000 (start point)`
	`N30 G03 G162 I70 J150`
	`N40 #CONTROL AREA END`
	`N50 M30`		

Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:		Identification number [-]
			Area ID
	%2:		Initial value [0.1 10^-3 mm or ø]
			Starting point for the circle description.
	%3:		Final value [0.1 10^-3 mm or ø]
			Target point for the circle description.
Error type	1, Error message from NC program.		

ID 50794

The first block at the start of a circular area definition must be linear.

Description	The first motion block after starting a definition of a circular area should be linear. A circular area definition consists of a linear block as starting point and a circular block as circle description. See also [PROG// Section: Workspace and protection space monitoring] and [FCT-C14// Section: Description]. **Example with error:** %circular area N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU… N20 **G02 G162 I70 J150** N30 G01 X150 Y200 F10000 N40 #CONTROL AREA END N50 M30 **Corrected example:** %circular area N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU… N20 **G01 X150 Y200 F10000** (start point) N30 G02 G162 I70 J150 N40 #CONTROL AREA END N50 M30		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Expected value [-]	
		Intersection of polygon lines	
	%3:	Error value [-]	
Error type	1, Error message from NC program.		

ID 50795

Axis for definition of cylindrical work or protection area not permitted.			
Description	When defining cylindrical workspace or protection area, only main axes may be used to specify the base plane (see [FCT-C14// Cylindrical control areas]). See also [PROG// Defining a control area] **Example with error:** G17 G0 G90 #CONTROL AREA BEGIN [ID1 PROT CIRC MIN_EXCUR=0 MAX_EXCUR=360] **G0 X0 X2=0 (error 120743)** G02 X0 X2=0 I+100 #CONTROL AREA END **Corrected example:** G17 G0 G90 #CONTROL AREA BEGIN [ID1 PROT CIRC MIN_EXCUR=0 MAX_EXCUR=360] G0 X0 Y0 G02 X0 Y0 I+100 #CONTROL AREA END		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the area definition in the NC program.

Parameter	%1:	Identification number [-]
		ID of affected workspace or protection area
	%2:	Error value [-]
		Given axes
	%3:	Expected value [-]
		Permitted axes for definition of a cylindrical control area
Error type	1, Error message from NC program.	

ID 50796

Error in workspace / protection space monitoring.			
Description			
Response	Class	6	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50797

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50798

Invalid number of active MCS / IMCS areas.			
Description	The number of simultaneously active MCS / IMCS was exceeded while workspace monitoring was active		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Check and modify NC program.
Parameter	%1:	Current value [-]	
		Number of active MCS / IMCS	
	%2:	Limit value [-]	
		Maximum number of MCS / IMCS that may be active at the same time	
Error type	1, Error message from NC program.		

ID 50799

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart. .

ID 50806

Programmed channel ID of #TRACK CS ON is out of valid range.			
Description			
Response	Class	6	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 50807

No interface known for programmed channel-ID of #TRACK CS ON.			
Description			
Response	Class	6	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 50808

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50810

Program end reached without detection of end of delete distance to go.			
Description	With Delete distance to go with activated jump mark , the end mark was not found in the NC program. A list of these commands is contained in [FCT-C28// Extension: Delete distance to go up to an explicit end marker]		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Set an END mark in the NC program.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 50812

Maximum number of M/H functions in Time-To-Distance functionality exceeded.			
Description	The number of M/H function covered by the TTD functionality is limited. This error occurs when the maximum number is exceeded.		
Response	Class	4	TTD cannot be calculated correctly.
Solution	Class	6	Reduce the number of M/H functions
Parameter	%1:	Upper limit value [-]	

Error type	-

ID 50819

Time-To-Distance calculation cannot be switched on because there is no TCP path axis in the channel.			
Description	The channel must contain an auxiliary axis which is marked by the ACHSMODE bit 0x02000000 (ACHSMODE_PATH_LENGTH_TCP), see P-AXIS-00015.		
Response	Class	4	TTD functionality not available
Solution	Class	6	Provide an auxiliary axis
Error type	1, Error message from NC program.		

ID 50820

Invalid combination of the frequency and damping ratio parameters.			
Description	No Vibration Guards can be applied to the defined frequency and damping ratio parameters.		
Response	Class	4	Movement stop.
Solution	Class	6	If the machining process allows, it may be helpful to use a smaller cycle time. Alternatively, reduce the damping ratio or increase the frequency.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50821

No interface known for programmed channel ID of ESCAPE ON.			
Description	In order to activate the ESCAPE channel, there must be an interface between the DR and ESCAPE channels. This interface is opened by the DR channel with the command #CHANNEL INTERFACE ON [ESCAPE …] and coupled by the ESCAPE channel with the command #TRACK CHAN ON= "ESCAPE channel number" START_POS="DR channel number" …]. The specified "ESCAPE/DR channel number" does not match the ESCAPE/DR channel or no interface was opened in the DR channel with #CHANNEL INTERFACE ON [ESCAPE …].		
Response	Class	6	Movement stop.
Solution	Class	7	Program #CHANNEL INTERFACE ON [ESCAPE …] in the DR channel before #TRACK CHAN ON [ESCAPE= "channel number" START_POS="DR channel number" …] in the ESCAPE channel or enter the correct entry of the channel number in #TRACK CHAN ON [ESCAPE= "channel number" START_POS="DR channel number" …]
Error type	1, Error message from NC program.		

ID 50822

#EXTEND ON: No interface for programmed channel number.

Description	In order to activate the ORBIT channel, there must be an interface between the DR and ORBIT channels. This interface is opened by the DR channel with the command #CHANNEL INTERFACE ON [EXTEND …] and coupled by the ORBIT channel with the command #TRACK CHAN ON [EXTEND = "DR channel number" …]. The specified "DR channel number" does not match the DR channel or no interface was opened in the DR channel with #CHANNEL INTERFACE ON [EXTEND …].		
Response	Class	6	Movement stop.
Solution	Class	7	Program #CHANNEL INTERFACE ON [EXTEND …] in the DR channel before #TRACK CHAN ON [EXTEND= "DR channel number" …] in the ORBIT channel or enter the correct specification of the channel number in #TRACK CHAN ON [EXTEND= "DR channel number" …]
Error type	1, Error message from NC program.		

ID 50824

Currently, the FCT_CALC_TIME and FCT_CALC_STATE_AT_T functions are exclusive. Please check startup list.			
Description			
Response	Class	3	
Solution	Class	5	
Error type	-		

ID 50825 - 50832

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50833

Measurement function is already active.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50834 - 50836

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50837

Time-To-Distance calculation cannot be switched on because there is no contour path axis in the channel.			
Description	The channel must contain an auxiliary axis which is marked by the ACHSMODE bit 0x04000000 (ACHSMODE_PATH_LENGTH_CONTOUR), see P-AXIS-00015.		
Response	Class	4	TTD functionality not available.
Solution	Class	6	Provide an auxiliary axis.
Error type	1, Error message from NC program.		

ID 50839

Axes must not be moved after APPROACH definition.			
Description	After the APPROACH definition in the ESCAPE channel with #ESCAPE PATH DEF BEGIN and #ESCAPE PATH DEF END , no further motion block may be programmed in the ESCAPE channel.		
Response	Class	6	Movement stop.
Solution	Class	7	Delete all motion blocks after #ESCAPE PATH DEF END in the ESCAPE channel.
Parameter	%1:	Current value [-]	
		Program line number	
	%2:	Error value [-]	
		Program line number of the motion block	
Error type	1, Error message from NC program.		

ID 50841

Could not determine actual edge joint.			
Description	The actual edge joint could not be determined. Possible reasons are: • The range between #MEAS RECORD BEGIN and #MEAS RECORD END was selected too small. More motion blocks can be placed in this range by reprogramming these two commands. • The touch probe geometry was incorrectly configured in the #MEAS command.		
Response	Class	4	Could not determine the measurement difference between command edge and actual edge.
Solution	Class	6	Increase the #MEAS RECORD range; check touch probe configuration
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Roll radius	
	%2:	Current value [0.1 µm or 0.0001°]	
		Touch probe length	
	%3:	Current value [0.1 µm or 0.0001°]	
		Measurement value X	
	%4:	Current value [0.1 µm or 0.0001°]	
		Measurement value Y	
	%5:	Current value [0.1 µm or 0.0001°]	
		Number of blocks available.	
Error type	1, Error message from NC program..		

ID 50842

No contour path axis in channel, function not enabled.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50843

The measurement parameters DISTANCE and/or OFFSET are invalid.			
Description	The #MEAS parameters DISTANCE and/or OFFSET are invalid. The following must apply: 1. DISTANCE >= 0 2. OFFSET – DISTANCE >= 0		
Response	Class	4	Could not determine the measurement difference.
Solution	Class	6	Correct the parameters
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		DISTANCE	
	%2:	Current value [0.1 µm or 0.0001°]	
		OFFSET	
Error type	1, Error message from NC program.		

ID 50844

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50845-50848

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 50849

SET_POSITION for interpolated axis programmed.			
Description	A [SET_POSITION ...] command has been programmed for an axis that is still being interpolated. This error message is for example output when the axis is executing an independent movement and a set position command is programmed.		
Response	Class	4	Movement stop.

Solution	Class	6	Removal of error in position controller and NC reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
	%2:	Error value [-]	
		Indication that the axis is executing a independent movement.	
Error type	1, Error message from NC program.		

ID 50852

Maximum shift value exceeded.			
Description	In connection with the #SHIFT function, a shift value greater than the predefined maximum shift value has resulted from the current conditions due to the programmed geometry, the technology parameters and the measurement. The CNC has insufficient motion information to execute the shift operation. Execute the definition of the maximum shift value taking into consideration the programmed NC block lengths and the parameters of the motion look-ahead in the interpolator. This includes any active geometric smoothing/block segmentation functions for motion blocks. The maximum shift value should be set as small as possible. The channel-specific look-ahead function is parameterised in The channel-specific look-ahead function is parameterised in P-CHAN-00650 and P-CHAN-00653, (alternatively P-STUP-00070 and P-STUP-00071). FCT_SHIFT_NCBL must be active for P-CHAN-00650 in order to use the function: `configuration.interpolator.function FCT_IPO_DEFAULT | FCT_SHIFT_NCBL`		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct the parameter setting of the look-ahead:P-CHAN-00650 / P-CHAN-00653, (alternatively P-STUP-00070 / P-STUP-00071). Correct the maximum shift value or the programmed geometry
Parameter	%1:	Block number [-]	
		Block number of affected NC block	
	%2:	Line number in file [-]	
		Line number of affected NC block	
	%3:	Current value [-]	
		NC block type (e.g. circular block, linear block)	
	%4:	Current value [0.1 µm or 0.0001°]	
		Required shift distance length	
	%5:	Limit value [0.1 µm or 0.0001°]	
		Maximum shift distance length #SHIFT BEGIN[MAX_DIST = …])	
Error type	1, Error message from NC program.		

ID 50854

Too many execution steps in the real-time cycle.			
Description			
Response	Class		
Solution	Class		

Error type	-

ID 50855

Not enough memory for the addition of a new realtime cycle.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50856

A realtime cycle was not initialised correctly.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50857

Not enough memory for the real-time cycle.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50858

The given ID is not a valid ID of a real-time cycle.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50859-50860

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50861

A mathematical expression in a realtime cycle is too complex.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50862

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50863

Invalid access to variable data of the real-time cycle.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50864-50869

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50870

Shift of control block not done.			
Description	In connection with the #SHIFT functionality, the interpolator must hold back motion blocks until the shift operation is completed. In the present case, it was detected that the required shift operation was not executed when one of the technology functions to be shifted was crossed.		
Response	Class	4	Movement stop.
Solution	Class	6	Please contact the control manufacturer.
Error type	1, Error message from NC program.		

ID 50871

Path not available for Shift.

Description	In connection with the #SHIFT function, an excessively large shift value compared to the existing motion blocks has resulted from the current conditions due to the programmed geometry, the technology parameters and the measurement. Execute the definition of the maximum shift value taking into consideration the NC block length and the parameters of the contour look-ahead in the interpolator. This includes any active geometric smoothing/block segmentation functions for motion blocks. The shift distance is related to the geometry. The maximum shift value should be set as small as possible. The channel-specific look-ahead function is parameterised in P-CHAN-00650 and P-CHAN-00653, (alternatively P-STUP-00070 and P-STUP-00071). FCT_SHIFT_NCBL must be active for P-CHAN-00650 in order to use the function: `configuration.interpolator.function FCT_IPO_DEFAULT | Path not available for Shift.`		
Response	Class	4	Movement stop.
Solution	Class	6	Check the block length of motion blocks, the parameter setting of the look-ahead function and the geometry.
Parameter	%1:	Current value [-]	
		Block number of the current motion block	
	%2:	Current value [-]	
		Block number of the technology block to be shifted	
	%3:	Current value[0.1 µm or 0.0001°]	
		Current shift value	
	%4:	Current value [0.1 µm or 0.0001°]	
		Required shift value	
Error type	1, Error message from NC program.		

ID 50872

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50873

Kind of orientation cannot be changed while dynamic CS is active.			
Description			
Response	Class	4	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 50874

An error occurred during the execution of a realtime cycle.

Description			
Response	Class	4	
Solution	Class	0	
Error type	1, Error message from NC program.		

ID 50875

The given ID for a real-time cycle is invalid.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50876

The functionality for real-time cycles is not enabled..			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50877

The maximum number of realtime cycles was reached.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50878

Function 'path-controlled offset of technology functions' is not configured.			
Description	This must be parameterised in order to use the function ‘Path-controlled offset of technology functions’. The function is enabled in the parameter P-CHAN-00650. `configuration.interpolator.function FCT_IPO_DEFAULT | FCT_SHIFT_NCBL` (Alternatively, the function can be enabled using P-STUP-00070)		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	6	Check and modify P-CHAN-00650.
Parameter	%1:	Identification number [-]	
		Name of the missing identifier	
Error type	1, Error message from NC program.		

ID 50879

An intermediate result of a computation in a real-time cycle is the invalid value 0.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50880

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50881

Moving to fixed stop with deactivated backward memory not possible.			
Description	The backward memory may not be completely disabled for the Move to fixed stop function [FCT-M8]. Increase the memory size in the parameter P-STUP-00033: Example for 1st channel in the start-up list: `fb_storage_size[0] 0x100000`		
Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	7	Check and modify the parameter P-STUP-00033.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2	Error value [-]	
		Maximum available backward memory	
Error type	1, Error message from NC program.		

ID 50882

Delete distance to go during Move to fixed stop not possible.			
Description	During the Move to fixed stop function [FCT-M8] the Delete distance to go function cannot be used (see [FCT-C28]). First deselect the Move to fixed stop function.		
Response	Class	6	Abort NC program processing. Transition to error state.
Solution	Class	7	Check and modify the sequences in the NC program and in your PLC.
Error type	-		

ID 50883

System error [▶ 12]	
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.

Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50884

Actual position request for axis moving to fixed stop not possible.			

Description	While an axis is located at the fixed stop [FCT-M8], no actual position request may be made for this axis. The Move to fixed stop function was not deactivated during a preceding CNC reset since the axis is still located at the fixed stop.
	
	Example with error:
	`#CHANNEL INIT [ACTPOS] (* Error message 50884 *)`
	
	Possible solution:
	`G01 X0 F1000 X[FIXED_STOP OFF]`
	`#CHANNEL INIT [ACTPOS]`		

Response	Class	4	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program. Ignore the affected axis in the actual position request.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		State of the Move to fixed stop function	
	%3:	Current value [-]	
		State of the fixed stop detection function	
Error type	1, Error message from NC program.		

ID 50885

Moving to fixed stop already active.			

Description	When the Move to fixed stop function is activated [FCT-M8], the function is already active on the axis. The Move to fixed stop function was not deactivated during a preceding CNC reset since the axis is still located at the fixed stop.
	
	Example:
	`N20 G01 X200 F1000 X[FIXED_STOP On](error 50885)`
	
	Possible solution:
	`N10 G01 X0 F1000 X[FIXED_STOP OFF]`
	`N20 G01 X200 X[FIXED_STOP ON]`		

Response	Class	4	Controlled halt of the axis, the control loop is closed. Transition to error state.
Solution	Class	6	Check and modify the NC program. Deactivate the Move to fixed stop function before reselecting.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2	Error value [-]	
		State of the Move to fixed stop function	

Error type	1, Error message from NC program.

ID 50886

Fixed stop not detected.			
Description	The fixed stop was not detected during the Move to fixed stop function [FCT-M8]. Possible causes: 1. Approach block path too short. 2. The fixed stop moved. 3. Position lag limit too large (see P-AXIS-00712) 4. Too many wait cycles (see P-AXIS-00714) 5. Torque limit too small; the axis did not move 6. Torque limit too large; the fixed stop slipped 7. Monitoring area in the approach block was too small, see START and END tokens in [FCT-M8//Programming]		
Response	Class	4	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the fixed stop and the parameterisation.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [0.1%]	
		Start the fixed stop monitoring function in the approach block	
	%3	Current value [0.1%]	
		End the fixed stop monitoring function in the approach block	
Error type	1, Error message from NC program.		

ID 50887-50888

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50889

During Move to fixed stop, axis may not be reprogrammed.			
Description			
Response	Class	4	
Solution	Class	6	

Parameter	%1:	Logical axis number [-]
	%2:	Current value [-]
	%3:	Error value [-]
	%4:	Error value [-]
	%5:	Block number [-]
Error type	1, Error message from NC program.	

ID 50890

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50891

The probe for edge joint has been misconfigured or has delivered wrong measurement values.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1	Current value [0.1 10^-3 mm or ø]	
	%2	Current value [0.1 10^-3 mm or ø]	
Error type	1, Error message from NC program. .		

ID 50892

The value of an external variable is not in the range of the variables data type.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50893

The floating point value of an external variable is not a number (NaN or infinite).			
Description			
Response	Class		

Solution	Class		
Error type	-		

ID 50898

The given axis index is invalid.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50899

The given axis is not in the channel.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50894

Axis must not be released when programmed axis coupling is active.			
Description	An attempt is made to release an axis by axis exchange command when a coupling activated by #GEAR LINK [] is active. This is not permissible.		
Response	Class	6	
Solution	Class	4	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
		State of coupling via #GEAR LINK[]	
Error type	1, Error message from NC program.		

ID 50895

An invalid array index was used during access to an external variable.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50896

TRC can here only be selected with perpendicular tool.
Description

Response	Class	5	
Solution	Class	6	
Parameter	%1	Current value [0.1 10^-3 mm or ø]	
Error type	1, Error message from NC program.		

ID 50900

The given ID is already in use for another real-time cycle.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50901

The Time-To-Distance computation has to be based on the active slope profile. P-CHAN-00209 must have value 1.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50902

The Time-To-Distance computation has to be based on the TCP path, not the contour path. P-CHAN-00340 must have value 0.			
Description			
Response	Class	4	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 50903

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires controller restart.

ID 50905

Invalid parametrisation for computation of shift parameter, tool radius must not be 0.			
Description			
Response	Class	4	

Solution	Class	6	
Parameter	%1	Block number [-]	
Error type	1, Error message from NC program.		

ID 50906

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 50909

Error in calculation / preparation of shift value.			
Description	In connection with the #SHIFT functionality (*) the iterative calculation algorithm was unable to calculate a valid shift value with the current conditions due to the programmed geometry, the technology parameters and the measurement.		
Response	Class	4	Movement stop.
Solution	Class	6	Please contact the control manufacturer.
Parameter	%1:	Block number [-]	
		Block number of the current block described	
	%2:	Line number in file [-]	
		Line number of the current block described	
	%3:	Current value [-]	
		Block type of the current block described	
Error type	1, Error message from NC program.		

ID 50910

Warning: order_time or time_constant smaller than cycle time. Filter not active.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	1, Error message from NC program.		

ID 50911

Warning: Order of filter too high. Maximum possible order is used.			
Description			
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
Error type	1, Error message from NC program.	

ID 50913

Frequency value for Vibration Guard is too high. Please check limit value.			
Description	The defined frequency range is too high and cannot be used by the Vibration Guard.		
Response	Class	4	Movement stop.
Solution	Class	6	Increase damping ratio.
Parameter	%1:	Limit value [Hz]	
	%2:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 50914

Online extension: inconsistent circle parameter.			
Description	In die sinking technology with active equidistant calculation, a block parameter inconsistency occurred in a circular block. The equidistant calculation is defined by #TRACK CHAN ON[EXTEND=**1** MAX_SCALE=**6** MAX_EQUID=**8**]		
Response	Class	4	Movement stop.
Solution	Class	7	Please contact the control manufacturer.
Error type	1, Error message from NC program.		

ID 50915

Online extension: inconsistent linear parameter.			
Description	In die sinking technology with active equidistant calculation, a block parameter inconsistency occurred in a linear block. The equidistant calculation is defined by #TRACK CHAN ON[EXTEND=**1** MAX_SCALE=**6** MAX_EQUID=**8**]		
Response	Class	4	Movement stop.
Solution	Class	7	Please contact the control manufacturer.
Error type	1, Error message from NC program.		

ID 50916

.Position of X-axis at DownShapeStart/ESCAPE-start must be ZERO.	
Description	The DR and SCAPE channels move only in the Y-Z plane after the APPROACH. The 3-dimensional contours results from superimposing the orbit channel. The error message means that an ESCAPE request was reported by the DR channel where the start position has a PCS position unequal to 0. This is not permitted since the DR and ESCAPE channels move in the Y-Z plane only.

Response	Class	6	Movement stop.
Solution	Class	7	No programming of the X axis in the DR channel after the APPROACH
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		X PCS position reported with ESCAPE request.	
	%2:	Current value [-]	
		ESCAPE start request ID	
	%3	Current value [-]	
		ESCAPE restart request ID	
Error type	1, Error message from NC program.		

ID 50917

Invalid parameter setting, r_max_equidist , p_admin->param.r_max_scale.			
Description	When the ORBIT channel is activated, the parameters for the maximum scaling and equidistant radii are defined with the command #TRACK CHAN ON [EXTEND="DR channel number" MAX_SCALE=<val> MAX_EQUID=<val>]. The error is output when the value for the maximum scaling radius is less than 0 or the value for the maximum equidistant radius is less than the maximum scaling radius.		
Response	Class	4	Movement stop.
Solution	Class	7	Check and correct the programming of the values for MAX_SCALE and MAX_EQUID in #TRACK CHAN ON [EXTEND="DR channel number" MAX_SCALE=<val> MAX_EQUID=<val>] in the ORBIT channel.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Value for MAX_EQUID in #TRACK CHAN ON [EXTEND="DR channel number" MAX_SCALE=<val> MAX_EQUID=<val>].	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Value for MAX_SCALE in #TRACK CHAN ON [EXTEND="DR channel number" MAX_SCALE=<val> MAX_EQUID=<val>].	
	%3	Current value [0.1 10^-3 mm or ø]	
		Difference between MAX_EQUID and MAX_SCALE.	
	%4	Limit value [0.1 10^-3 mm or ø]	
		Limit for difference.	
Error type	1, Error message from NC program.		

ID 50918

Invalid non convex geometry.			
Description	In die sinking technology, only convex contours may be programmed with the active equidistant calculation (*) to define orbit geometry. The geometry element attributes are also dependent on the direction of rotation of the orbit (forward/backward motion). In the present case, an inconsistency occurred in the direction of rotation and the required correction side of the equidistant. (*) #TRACK CHAN ON[EXTEND**=1** MAX_SCALE**=6** MAX_EQUID**=8**]		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct orbit program for correct convex geometry.

Parameter	%1:	Current value [-]
		Correction side equidistant
	%2:	Current value [-]
		Direction of rotation of orbit
	%3	Current value [-]
		Control information / direction of circular rotation
Error type	1, Error message from NC program.	

ID 50920/50921

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50922

Radius not zero on start move.			
Description	When the ORBIT channel is activated, the radius must be 0 (corresponds to the Y PCS position in the DR channel). This is not the case.		
Response	Class	4	Movement stop.
Solution	Class	7	Before the ORBIT channel is activated, the radius (Y axis) must be programmed to 0 in the DR channel.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Current radius (Y PCS position in the DR channel)	
	%2:	Limit value [0.1 10^-3 mm or ø]	
		Maximum permitted radius when the ORBIT channel is activated	
Error type	1, Error message from NC program.		

ID 50924

Too many blocks within programmed #CONTOUR LOOKAHEAD area.			
Description	NC program sequences can be marked with the command #CONTOUR LOOKAHEAD LOG ON/OFF so that the CNC can supply the HLI with motion blocks in the contour look-ahead area. The maximum number of motion blocks in the area can be configured using P-CHAN-00658 (or P-STUP-00076). This error occurs if more motion blocks are programmed in the contour look-ahead area. The contour look-ahead function must be activated by the parameter P-CHAN-00650 (or P-STUP-00070). (FCT_CONTOUR_LAH) If the function is deactivated, this error is also output when the command is programmed.		
Response	Class	2	Abort NC program processing.
Solution	Class	7	Increase P-CHAN-00658 (or P-STUP-00076) or reduce the contour look-ahead area and check P-CHAN-00650 (or P-STUP-00070)

Parameter	%1:	Block number [-]
		Current motion block which exceeds the maximum number.
	%2:	Upper limit value [-]
		Maximum number of motion blocks in the contour look-ahead area (P-CHAN-00658 or P-STUP-00076)
Error type	1, Error message from NC program.	

ID 50925

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50926

ESCAPE restart position not found.			
Description	In sliding or balancing with sink erosion, the motions of DR and ESCAPE channels overlap. The ESCAPE channel must then touch down on the new position of the DR channel after every DR channel movement. From there, the ESCAPE channel plans the new ESCAPE motion online and slices this movement from the last cycle using the stored distance. This error occurs when the distance of the new planned ESCAPE movement is smaller than the last distance stored. This can only occur if the distance of the DR in a cycle is greater than the distance of the ESCAPE channel to the APPROACH start.		
Response	Class	6	Movement stop.
Solution	Class	7	Possible solutions: • Reduce the sliding distance • Allow balancing only at a greater distance from the APPROACH start.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Current distance of the ESCAPE channel to the new ESCAPE start position	
	%2:	Expected value [0.1 10^-3 mm or ø]	
	%3:	Identification number [-]	
		ESCAPE start request ID	
	&4:	Current value[-]	
		ESCAPE restart request ID	
Error type	1, Error message from NC program.		

ID 50927

DownShape position changed during JUMP execution.

Description	A jump operation commanded by the HLI (Control Unit) moves the electrode as fast as possible away from the workpiece and generates a flushing movement. The flushing movement evacuates removed particles and prevents short circuits from causing and excessive load on the electrode. During a jump, the DR channel may not move until the jump is completed. Before a jump is commanded, the PLC must ensure that both the DR and the ORBIT channels are at standstill and do not move either during the jump. For example, this can be ensured with a feedhold in the DR and ORBIT channels. This error is output if a movement takes place.		
Response	Class	6	Movement stop.
Solution	Class	6	Possible solution: Set feedhold in DR and ORBIT channels
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Current PCS positions of the DR channel	
	%2:	Expected value [0.1 10^-3 mm or ø]	
		PCS positions of the DR channel at jump start.	
	%3:	Current value [0.1 10^-3 mm or ø]	
		Distance between the current PCS position and the PCS position at jump start in the DR channel.	
Error type	1, Error message from NC program.		

ID 50929

The given array index is invalid.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50930

The execution of the real-time cycles took too long.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50931

Position of X-axis at ESCAPE-End must be zero.			
Description	On receipt of a request, the ESCAPE channel calculates online to the currently reported position an ESCAPE path up to the APPROACH END, which is also the ESCAPE end, on receipt of a request online At this position, the X axis position must be 0. This is not the case.		
Response	Class	6	Movement stop.
Solution	Class	7	Correct the programming of the APPROACH path; X position must be 0 at the end.

Parameter	%1:	Error value [0.1 10^-3 mm or ø]
		X PCS position at ESCAPE end.
	%2:	Error value [0.1 10^-3 mm or ø]
		Y PCS position at ESCAPE end.
	%3:	Identification number [-]
		ESCAPE start request ID
	%4:	Identification number [-]
		ESCAPE restart request ID
Error type	1, Error message from NC program.	

ID 50932

Position of Y-axis at ESCAPE-End must be zero.			
Description	On receipt of a request, the ESCAPE channel calculates online to the currently signalled position an ESCAPE path up to the APPROACH END, which is also the ESCAPE end, on receipt of a request online At this position, the Y axis position must be 0. This is not the case.		
Response	Class	6	Movement stop.
Solution	Class	7	Correct the programming of the APPROACH path; Y position must be 0 at the end.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		X PCS position at ESCAPE end.	
	%2:	Error value [0.1 10^-3 mm or ø]	
		Y PCS position at ESCAPE end.	
	%3:	Identification number [-]	
		ESCAPE start request ID	
	%4:	Identification number [-]	
		ESCAPE restart request ID	
Error type	1, Error message from NC program.		

ID 50933

ESCAPE activation must be done before ORBIT activation for optional mark of APPROACH path.			
Description	The die sinking configuration consists of the three channels DR, ORBIT and ESCAPE. Before machining with an orbiting movement, it is possible to define an APPROACH geometry in 3D space. In this geometry, with the DR channel moves the electrode forwards or backwards at negative velocities or jumps of the ESCAPE channel. This APPROACH geometry is implicitly detected between activation of the ESCAPE channel and activation of teh ORBIT channel. For this reason, the ESCAPE channel must be defined first followed by the ORBIT channel, even if an APPROACH geometry is not required.		
Response	Class	6	Movement stop.
Solution	Class	7	Activation of the ESCAPE channel before activation of the ORBIT channel in the NC program.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		Invalid channel number	

Error type	1, Error message from NC program.

ID 50934

Given APPROACH start position not within defined APPROACH geometry.			
Description	If there is an ESCAPE request in the APPROACH; the reported position of the DR channel is searched in the APPROACH geometry stored in the ESCAPE channel. The error indicates that the reported position was not found in the APPROACH geometry stored in the ESCAPE channel. One cause may be different APPROACH geometries programmed in the DR and ESCAPE channels. Another cause may be an escape request from the DR channel after the APPROACH geometry, ifthe ORBIT channel was not activated at the APPROACH end, since this activation is implicitly defined as the APROACH end.		
Response	Class	6	Movement stop.
Solution	Class	7	Program the identical APPROACH geometry in the DR and ESCAPE channels. Check whether the APPROACH end is correctly programmed with the activation of the ORBIT channel.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		Reported distance in the APPROACH path of the DR channel.	
	%2:	Upper limit value [0.1 10^-3 mm or ø]	
		Total distance of the APPROACH path stored in thee ESCAPE channel.	
	%3:	Identification number [-]	
		ESCAPE start request ID	
	%4:	Current value [-]	
		ESCAPE restart request ID	
Error type	1, Error message from NC program.		

ID 50935-50937

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50938

A realtime cycle attempts to emit an unconfigured M function.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50939

A realtime cycle attempts to emit a reserved M function.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50940

A realtime cycle attempts to emit an unconfigured H function.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50941

Superimposed offsets from #TRACK CH ON[] are active during position reinitialisation from NC command.			
Description			
Response	Class	6	
Solution	Class	7	
Parameter	%1:	Current value[0.1 10^-3 mm or ø]	
	%2:	Error value [0.1 10^-3 mm or ø]	
	%3:	Logical axis number [-] Movement stop.	
Error type	1, Error message from NC program.		

ID 50942

At least one realtime techno function could not be emitted.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50943

.Too many realtime techno functions in the current CNC cycle.			
Description			
Response	Class		
Solution	Class		

Error type	-

ID 50947

No coordinate system may be changed within the APPROACH area.			
Description	Only one coordinate system for the entire APPROACH is permitted in the APPROACH definition in the ESCAPE channel. A change of coordinate system is not permitted between the NC commands #ESCAPE PATH DEF BEGIN and #ESCAPE PATH DEF END. **Example with error:** (Start of definition of approach N3050 #ESCAPE PATH DEF BEGIN (Approach CS N3060 #CS ON [APP] [@PL4,@PL5,@PL6,@PL7,@PL8,@PL9] (No backward motion in the CS N3070 #ESCAPE PATH BACKWARD STOP N3080 G01 X100 N3090 #CS ON [ERR] [10, 20, 30 , 0, 0, 0] (**Error 50947**) (Deselect the Approach CS N3100 #CS DEL ALL (End of definition of approach N3110 #ESCAPE PATH DEF END **Corrected example:** (Start of definition of approach N3050 #ESCAPE PATH DEF BEGIN (Approach CS N3060 #CS ON [APP] [@PL4,@PL5,@PL6,@PL7,@PL8,@PL9] (No backward motion in the CS N3070 #ESCAPE PATH BACKWARD STOP ;... Approach geometry (Deselect the Approach CS N3100 #CS DEL ALL (End of definition of approach N3110 #ESCAPE PATH DEF END		
Response	Class	7	Movement stop.
Solution	Class	2	Check and correct the programming
Parameter	%1:	Current value [-]	
		Stack number of the active coordinate system.	
Error type	Abort NC program processing.		

ID 50948

Single axis interpolation for active transformation axis not possible.			
Description	An active transformation axis cannot be traversed as a single-axis motion. Before commanding the motion, deselect the transformation (#CS OFF or #TRAFO OFF).		
Response	Class	4	Path motion and single-axis motion are stopped. Transition to error state
Solution	Class	6	Check and modify the NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Block number [-]	
		Block number of single-axis job	
	%3	Current value [-]	
		Job number of single-axis job	
Error type	-		

ID 50950

APPROACH-definition in DownShape and Escape-channel different.			
Description	The APPROACH definitions in the DR and ESCAPE channels must be identical. The APPROACH start in the DR channel is defined on activation of the ESCAPE channel and the APPROACH end is defined on activation of the ORBIT channel. The APPROACH start in the ESCAPE channel is defined with the command #ESCAPE PATH DEF BEGIN and the APPROACH end is defined with the command #ESCAPE PATH DEF END. This error occurs when the programmed APPROACH geometries in the channels are different after the ORBIT channel was activated.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Program the identical APPROACH geometry in the DR and ESCAPE channels.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Distance of the entire APPROACH geometry in the DR channel.	
	%2:	Expected value [0.1 10^-3 mm or ø]	
		Distance of the entire APPROACH geometry in the ESCAPE channel.	
	%3:	Current value [-]	
		Difference between the programmed distances.	
	%4:	Identification number [-]	
		ESCAPE start request ID	
	%5:	Identification number [-]	
		ESCAPE restart request ID	
Error type	1, Error message from NC program.		

ID 50951

APPROACH position at given path distance different from reported ESCAPE start position.			
Description	The reported start position of the DR channel does not match the position in the ESCAPE channel in the case of an ESCAPE request within the APPROACH area. This error occurs when there are different APPROACH definitions in the DR and ESCAPE channels.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Program the identical APPROACH geometry in the DR and ESCAPE channels.

Parameter	%1:	Current value [0.1 10^-3 mm or ø]
		Distance reported in the APPROACH geometry reported by the DR channel.
	%2:	Current value [0.1 10^-3 mm or ø]
		Position reported in the APPROACH geometry reported by the DR channel.
	%3:	Error value [0.1 10^-3 mm or ø]
		Position calculated in teh ESCAPE channel.
	%4:	Error value [0.1 10^-3 mm or ø]
		Deviation between reported and calculated positions
Error type	1, Error message from NC program.	

ID 50952

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50953

A new APPROACH backward request was received while already moving backwards in APPROACH.			
Description	A simultaneous overlapping of DR and ESCAPE channels is not permitted in the APPROACH geometry. This error indicates that an ESCAPE request was received by the DR channel although the ESCAPE channel is already in backward motion.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	After an ESCAPE request in the APPROACH geometry, the DR channels must wait with another request until the previous one ends. This can be checked with the bit bahn_state.approach_active_r in the PLC.
	%1:	Identification number [-]	
		ESCAPE start request ID	
	%2:	Identification number [-]	
		ESCAPE restart request ID	
Error type	1, Error message from NC program.		

ID 50954

Too many or too few parameters were programmed for the spindle.			
Description			
Response	Class		
Solution	Class		
Error type	-		

ID 50955

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50956

DownShape position changed during backward movement in APPROACH.			
Description	A simultaneous overlapping of DR and ESCAPE channels is not permitted in the APPROACH geometry. This error indicates that a movement of the DR channel was detected during an active ESCAPE request.		
Response	Class	7	Movement stop.
Solution	Class	6	After an ESCAPE request in the APPROACH geometry, the DR channels must wait with another request until the previous one ends. This can be checked with the bit bahn_state.approach_active_r in the PLC.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Expected and stored start position of the DR channel with the currently active ESCAPE request.	
	%2:	Expected value [0.1 10^-3 mm or ø]	
		Expected and stored start position of the DR channel with the currently active ESCAPE request.	
	%3:	Current value[0.1 10^-3 mm or ø]	
		Distance between the current DR and the stored DR positions.	
Error type	1, Error message from NC program.		

ID 50960

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 50961

Start position not in workspace..			
Description	The start position in manual mode is outside the workspace when workspace and protection area monitoring is active [FCT-C14// Special features in manual mode]. This is not possible with the selected setting of P-CHAN-00442.		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Check the workspace, check P-CHAN-00442. Set the Suppress error output from workspace monitoring in manual mode control unit to move backward in the workspace in manual mode.

Parameter	%1:	Identification number [-]
		ID of the workspace affected
	%2:	Current value [-]
		Bit identifier of the affected axes
	%3	Error value [0.1 10^-3 mm or ø]
		3-dimensional output of positions
Error type	1, Error message from NC program.	

ID 50962

Start position in protection area..			
Description	The start position in manual mode is outside the protection area when workspace and protection area monitoring is active [FCT-C14// Special features in manual mode]. This is not possible with the selected setting of P-CHAN-00442.		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Check the protection area, check P-CHAN-00442. Set the Suppress error output from workspace monitoring in manual mode control unit to move backward in the protection area in manual mode.
Parameter	%1:	Identification number [-]	
		ID of the affected protection area	
	%2:	Current value [-]	
		Bit identifier of the affected axes	
	%3	Error value [0.1 10^-3 mm or ø]	
		3-dimensional output of positions	
Error type	1, Error message from NC program.		

ID 50963

Axis for workspace monitoring not present.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:	Identification number [-]	
	%2:		
	%3	Current value [-]	
	%4		
	%5		
Error type	1, Error message from NC program.		

ID 50964

Movement limitation because of work area/protection area.			
Description	The work area in the MCS can be defined in manual operation mode by corresponding values in the MC program. The work area definition is defined by the NC command **#CONTROL AREA BEGIN [ID 1 WORK POLY...]** When manual mode is active, motion jobs outside of these limits result in a motion stop within the permitted motion limits. A reset is not required; in most cases the axis can continue to be moved by a suitable backward motion. The output of this warning is dependent on the parameter P-MANU-00014. The following parameters can additionally restrict motion limits in manual mode: • P-AXIS-00137/P-AXIS-00138 • P-AXIS-00177/P-AXIS-00178		
Response	Class	1	Warning output. Axes stopped.
Solution	Class	1	Move the affected axis backwards from the motion limit in the work area.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the affected MCS/ACS axis.	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current axis position	
	%3	Current value [-]	
		Axes with motion stop (bit-coded).	
	%4	Identification number [-]	
		Affected area of the work area	
	%5	Current value [-]	
		Type of work area element (circle, polygon)	
Error type	-		

ID 50965

Invalid G programming of single axis in real-time cycle.			
Description	An inadmissible G command was used when a single axis is programmed within a real-time cycle.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution		6	Check and modify the programming of the single axis in the RT cycle.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 50966

Axis has no SAI characteristic.			
Description	In order to use the single axis in real-time cycle function, the parameter P-AXIS-00457 for SAI characteristics must be set.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and correct P-AXIS-00457.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Parameter name [-]
		Name of parameter
Error type	1, Error message from NC program.	

ID 50967

Invalid programming of SAI axis.			
Description	When the single axis in real-time cycles is programmed, a combination of invalid commands was used.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the programming of the single axis in the RT cycle.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	1, Error message from NC program.		

ID 50968

With FAST_VEL_IN_CENTER different axes than x, y, and z are programmed.			
Description	After fast positioning is activated in the ORBIT channel, only Cartesian axes X, Y or Z may be programmed.		
Response	Class	6	Movement stop.
Solution	Class	4	Check and correct the programming after the command #CHANNEL SET [FAST_FORWARD_IN_CENTER=ON].
Parameter	%1:	Current value [-]	
		Value of the bitmask of the programmed axes.	
	%2:	Upper limit value [-]	
		Maximum value of the bitmask of the programmed axes for programming of X, Y and Z.	
Error type	-		

ID 50973

Velocity weighting for moved axes in Jump not equal.			
Description	The ESCAPE path for a jump is planned with 100% axis dynamic since only the ESCAPE channel may be active in a jump. The dynamic weighing programmed by default as reduced to 50% is therefore increased. This error message indicates that the same dynamic weighting was not programmed for all axes. In this case, the dynamic cannot be increased for a jump.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and correct the programming of the dynamic weighting with G128/G130 in the ESCAPE channel

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis with different dynamic weighting
	%2:	Error value [%]
		Dynamic weighting of the invalid axis.
	%3:	Expected value [%]
		Dynamic weighting of the previous axes.
Error type	-	

ID 50974

Command position outside valid range.			
Description	The command position that the position controller is to transfer to the decoder when calling an axis position is outside the valid motion range.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the axis position.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [0.1 10^-3 mm or ø]	
		Position of the axis	
	%3:	Upper limit value [0.1 10^-3 mm or ø]	
		Maximum permissible position	
	%4:	Lower limit value [0.1 10^-3 mm or ø]	
		Minimum permissible position	
Error type	1, Error message from NC program.		

ID 50975

Partner channel of channel coupling reports an error (#TRACK CHAN).			
Description	With coupled channels across the channel interface, all coupled channels are switched to error state. This error indicates that an error was received from a coupled channel.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	4	Check and remove the original error from the coupled channel.
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 50976

Timeout: measurement not executed or incorrect			
Description	A timeout error occurred in connection with the #SHIFT functionality. In the present case, the previously executed measurement was not or incorrectly executed.		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct the measuring sequence.

Parameter	%1:	Block number [-]
		Block number of the current block
	%2:	Line number in file [-]
		Line number of the current block
	%3:	Current value [0.1 10^-3 mm or ø]
		Block type of the current block
Error type	1, Error message from NC program.	

ID 50977

Timeout: waiting for sufficient number of motion blocks.			
Description	In connection with the #SHIFT functionality (+), a timeout error occurred while waiting for the calculation/supply of the shift value. In the present case, there is an insufficient number of motion blocks to determine the correct shift value.		
Response	Class	4	Movement stop.
Solution	Class	6	Check and correct the geometry in the shift area of the NC program.
Parameter	%1:	Block number [-]	
		Block number of the current block	
	%2:	Line number in file [-]	
		Line number of the current block	
	%3:	Current value [0.1 10^-3 mm or ø]	
		Block type of the current block	
Error type	1, Error message from NC program.		

ID 50978

Timeout: program mark #MARK[SHIFT_POS] not found.			
Description	In connection with the #SHIFT functionality, a timeout error occurred while waiting for the calculation/supply of the shift value. In the present case, the program mark #MARK SHIFT POS was not found. Execute the definition of the possible shift values taking into consideration the programmed NC block lengths and the parameters of the look-ahead motion in the interpolator. This includes any active geometric smoothing/block segmentation functions for motion blocks. The channel-specific look-ahead function is parameterised in P-CHAN-00650 and P-CHAN-00653, (alternatively P-STUP-00070 and P-STUP-00071). FCT_SHIFT_NCBL must be active for P-CHAN-00650 in order to use the function: `configuration.interpolator.function FCT_IPO_DEFAULT | Timeout: program mark #MARK[SHIFT_POS] not found.`		
Response	Class	4	Movement stop.
Solution	Class	6	Check the NC program for existing program mark #MARK. Check the number of marks used., Check the parameter setting of the look-ahead: P-CHAN-00650/ P-CHAN-00653, (alternatively P-STUP-00070 / P-STUP-00071).

Parameter	%1:	Block number [-]
		Block number of the current block
	%2:	Line number in file [-]
		Line number of the current block
	%3:	Current value [0.1 10^-3 mm or ø]
		Block type of the current block
Error type	1, Error message from NC program.	

ID 50979-50986

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 50987

#FILTER cannot be used with Vibration Guard. Vibration Guard is deactivated.			
Description	A #FILTER command was programmed in the NC program although the Vibration Guard was activated by the P-AXIS-00588 of an axis. No filter commands may be programmed while the Vibration Guard is active.		
Response	Class	1	Active Vibration Guards are deactivated and NC program processing is continued; the #FILTER command has no effect,
Solution	Class	1	In order to use filter commands correctly, no Vibration Guard may be activated in the axis lists.
Parameter	%1:	Logical axis number [-]	
		Logical axis number at which P-AXIS-00588 is set.	
Error type	1, Error message from NC program.		

ID 50988

#VIB GUARD cannot be used with FIR-filters. Filter not active.			
Description	A #VB GUARD command was programmed in the NC program although FIR filters are configured in the axis lists. Vibration Guard commands cannot be used at the same time as FIR filters are configured in the axes.		
Response	Class	1	Active filters are deactivated, NC program processing is continued and the Vibration Guard is activated.
Solution	Class	1	In order to use Vibration Guard commands correctly, no filters may be activated in the axis lists.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis where FIR filters are configured.	
Error type	1, Error message from NC program.		

ID 50989

Filter is not active, because no filter order is set.			
Description	No filter order (P-AXIS-00587) was defined for the filter in the required axis. The filter in this axis was therefore deactivated. To activate the filter, at least the filter type and a filter order must be specified for this filter.		
Response	Class	2	Warning.
Solution	Class	3	Check and modify the parameterisation. Specify an order for the filter in the appropriate axis (order of axis-specific filter P-AXIS-00587 or alternative order in time P-AXIS-00591 or across all axes using #FILTER [ORDER=…]).
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis concerned. (Parameter is optional)	
Error type	1, Error message from NC program.		

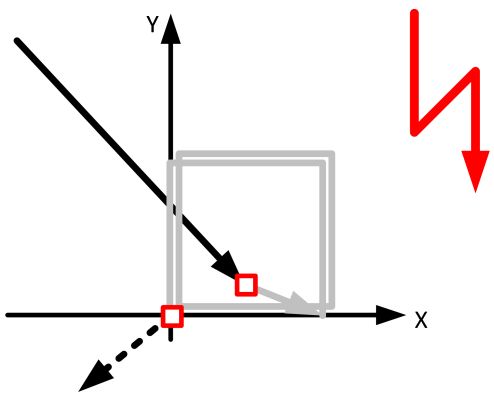
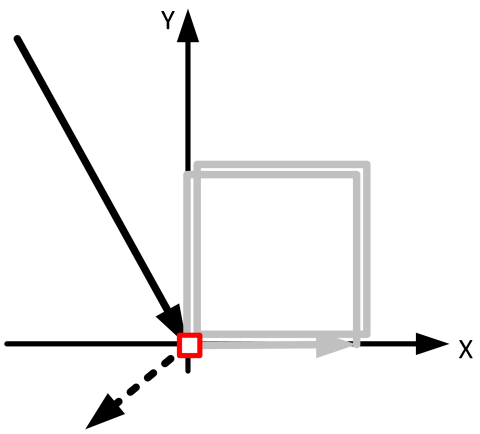
ID 50990

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 50991

Start and end positions of real-time loop are not identical.		
Description	The positions on entry to the real-time loop must be identical with the position at the end of the loop. The real-time loop is programmed with #RT WHILE / #RT ENDWHILE.	
	Example with error:	Corrected example:

	`N0130 G1 X50 Y10 (Start position)`
	`N0140 #RT WHILE`
	`N0150 X100`
	`N0160 Y100`
	`N0170 X0`
	`N0180 X0 Y0 (End position)`
	`N0190 #RT ENDWHILE`
		`N0130 G1 X0 Y0 (Start position)`
		`N0140 #RT WHILE`
		`N0150 X100`
		`N0160 Y100`
		`N0170 X0`
		`N0180 X0 Y0 (End position)`
		`N0190 #RT ENDWHILE`

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the positions at the start and end of the real-time loop.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current position of the affected axis	
	%3:	Expected value [0.1 10^-3 mm or ø]	
		Expected position of the affected axis	
Error type	-		

ID 50992-50994

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 50995

Given DR APPROACH start position not within defined APPROACH geometry.			
Description	If there is an ESCAPE request in the APPROACH; the reported position of the DR channel is searched in the APPROACH geometry stored in the ESCAPE channel. The error indicates that the current position is before the APPROACH definition.		
Response	Class	6	Movement stop.
Solution	Class	7	Check whether the DR channel after the APPRAOCH start definition moved backwards before the APPROACH start definition.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Distance from the APPROACH start position at new ESCAOT request	
	%2:	Error value [0.1 10^-3 mm or ø]	
		Current distance from program start in the DR channel.	
	%3:	Limit value [0.1 10^-3 mm or ø]	
		Distance from program start in the DR channel at start of APPROACH definition.	
Error type	1, Error message from NC program.		

ID 50996

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 50997

No manual movement allowed while active MDS update.			
Description	The axis was moved in manual mode while the axis list was updated. This is not allowed if critical parameters are also changed. The parameters include, for example, path resolution (P-AXIS-00234/P-AXIS-00233)		
Response	Class	6	Controlled halt of the axis, the control loop is closed., the axis list is being updated.
Solution	Class	6	Carry out a CNC reset; the updated lists were adopted. Carry out movement of the axis in manual mode after the lists are updated.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [0.1 µm/tick]	
		Commanded command position of the affected axis per cycle in manual mode	
Error type	1, Error message from NC program.		

ID 50998

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 50999

Invalid value for parameter m_function_nr in Control Unit Insert_Command.			
Description	The specified value for the m_function_nr parameter in the “Insert stop marks” control unit is outside the valid value range. The valid value range for this parameter correspond to the valid value range for M functions in the NC program. See CNC system parameters		
Response	Class	4	Movement stop.
Solution	Class	6	Check and modify the value for m_function_nr. A reset is required.
Parameter	%1:	Error value [-]	
		Incorrect value of commanded value for m_function_nr in the control unit	
	%2:	Lower limit value [-]	
		Lower limit for value of the M function	
	%3:	Upper limit value [-]	
		Upper limit for value of the M function	
Error type	1, Error message from NC program.		

2.6.5 ID-range 51000-51250

ID 51000

Value of axis_nr in Control unit is above the maximum number of axes per channel.			
Description	The specified value for the axis_nr parameter of the “Insert stop marks” control unit exceeds the maximum permissible value. A maximum of 32 axes can be configured in each channel.		
Response	Class	4	Movement stop.
Solution	Class	6	Check and modify the commanded value for the axis_nr parameter. A reset is required.
Parameter	Error value [-]		
%1:	Incorrect value of commanded value for axis_nr in the “Insert stop marks” control unit““		
Error type	1, Error message from NC program.		

ID 51001

Invalid value for parameter rel_abs_mode in Insert_Command CU.			
Description	Only values in the interval [0,2] are permitted for the rel_abs_mode parameter in the “Insert stop marks” control unit. 0 = repetitive stops with relative specified dimension 1 = single stop with absolute specified dimension 2 = single stop with relative specified dimension.		
Response	Class	4	Movement stop.
Solution	Class	6	Check and modify the commanded value for the rel_abs_mode parameter of the “Insert stop marks” control unit. A reset is required.
Parameter	%1:	Error value [-]	
		Incorrect value of commanded value for rel_abs_mode.	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 51002

Commanded axis_nr of Control Unit Insert_Cmd is not configured in channel.			
Description	The logical axis number specified in the axis_nr parameter of the “Insert stop marks” control unit is not configured in the channel. In order to insert a motion stop at a commanded axis position, the correct logical axis number must be specified for this axis (POSITION mode). Enter the value 0 for a stop with a commanded path distance (DISTANCE mode).		
Response	Class	4	Movement stop.
Solution	Class	6	Specify a logical axis number which is configured in the channel. A CNC reset is required.

Parameter	%1:	Error value [-]
		Incorrect value of commanded value for axis_nr in the “Insert stop marks” control unit““
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
		Upper limit for logical axis number value.
Error type	-	

ID 51004

The functionality "Insert stop command" can only be used with configuration of P-STUP-00033.			
Description	With the “Insert motion stops during machining” function, the memory fb_storage_size (P-STUP-00033) must be parameterised. It was not parameterised.		
Response	Class	4	Movement stop.
Solution	Class	6	Check and modify P-STUP-00033 of the corresponding channel. A CNC reset is required.
Error type	1, Error message from NC program.		

ID 51005

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51006

The Y coordinate of the specified point at POINT-ESCAPE may not be smaller than zero.			
Description	When POINT-ESCAPE is used, the specified value of the Y coordinate of the point is less than zero. This is not permitted.		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the point in NC program. Select a Y coordinate value of the point greater than or equal to zero.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		Incorrect position of the Y coordinate.	
	%2:	Identification number [-]	
		ESCAPE start request ID	
	%3:	Identification number [-]	
		ESCAPE restart request ID	
Error type	1, Error message from NC program.		

ID 51007

The Z coordinate of the specified point at POINT-ESCAPE must be below of DownShapeStart or ESCAPE end.			
Description	The value of the Z coordinate is greater than the Z coordinate of the DownShape start. This is not permitted.		
Response	Class	6	Movement stop.
Solution	Class	6	Check and modify the NC program. Select a Z coordinate value of the point in the NC program that is smaller than the Z coordinate of the DownShape start.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		Incorrect position of the Z coordinate.	
	%2:	Identification number [-]	
		ESCAPE start request ID	
	%3:	Identification number [-]	
		ESCAPE restart1, Error message from NC program. Request ID	
Error type			

ID 51009

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51010

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51011

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51012

When turning off the orbit interface, the radius has to be zero.			
Description	It is only permitted to deactivate the orbit interface if the current orbit radius is zero.		
Response	Class	4	Movement stop.

Solution	Class	6	Modify the NC program. The orbit radius must be zero when commanded by #CHANNEL INTERFACE OFF [EXTEND].
Parameter	%1:	Error value [0.1 µm]	
		Current orbit radius when channel interface is deactivated.	
	%2:	Expected value [0.1 µm]	
		Expected value of the orbit	
Error type	1, Error message from NC program.		

ID 51013

Deactivation of channel interface must be done before deactivation of Orbit interface.			
Description	In order to decouple the orbit channel, #CHANNEL INTERFACE OFF [EXTEND] must first be commanded in the NC program before #TRACK CHAN OFF [DYN_CS].		
Response	Class	4	Movement stop.
Solution	Class	6	Modify the NC program. First program #CHANNEL INTERFACE OFF [EXTEND] before #TRACK CHAN OFF [DYN_CS].
Error type	1, Error message from NC program.		

ID 51014

Deactivation of Orbit is not allowed while Escape offset is active.			
Description	In order to decouple the escape channel, #CHANNEL INTERFACE OFF [ESCAPE] must first be commanded in the NC program before #TRACK CHAN OFF[DYN_CS SUPERIMPOSE].		
Response	Class	4	Movement stop.
Solution	Class	6	Modify the NC program. First program #CHANNEL INTERFACE OFF [ESCAPE] before ##TRACK CHAN OFF[DYN_CS SUPERIMPOSE].
Error type	1, Error message from NC program.		

ID 51015

Deactivation of Orbit is not allowed while Escape offset is active.			
Description	There may be no position shift due to an active escape (or jump) when the orbit is deactivated.		
Response	Class	4	Movement stop.
Solution	Class	6	Modify the NC program or the PLC so that no escape or jump is executed when the orbit is deactivated.
Error type	1, Error message from NC program.		

ID 51016

Invalid NC syntax while searching end mark of delete distance to go.	
Description	During a search for the end mark for Delete distance to go with an activated jump mark, an invalid NC command or function is to be skipped. Invalid NC commands may include coordinate system or transformation changes since such a change can only take place a exactly the programmed position.

Response	Class	4	Movement stop.
Solution	Class	6	Modify the NC program. Place jump mark before the invalid NC command.
Error type	1, Error message from NC program.		

ID 51018

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51019

Orbit radius may be changed after the complete Orbit geometry has been received.			
Description	In an orbit configuration with an active equidistant calculation, a closed orbit geometry must first exist before the radius can be changed in the DR channel. The “orbit_active_r” status signal is displayed to indicate the existence of a closed orbit geometry.		
Response	Class	4	Movement stop.
Solution	Class	6	Check and modify the process. Only change the radius in the DR channel after the orbit geometry is complete.
Error type	-		

ID 51020

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51021

Coupled master channel is moving during initialisation of slave channel.			
Description	While a slave channel is executing a channel initialisation, e.g. with #CHANNEL INIT, the master channel may not move when the channel couplings are active with #CHANNEL INTERFACE and #TRACK CHAN.		
Response	Class	4	Movement stop.
Solution	Class	6	Motion stop of the master channel, e.g. by setting feedhold
Parameter	%1:	Logical channel number [-]	
		Logical channel ID of the moving master channel.	
Error type	1, Error message from NC program.		

ID 51023

NC command can not be executed in backward direction.			
Description	There are NC commands which are only possible in the forward direction in the “Forward/backward motion on the path” function. For example: • #CHANNEL INTERFACE ON • #TRACK CHAN ON If one of these commands is to be executed in the backward direction, the channel remains standing and it cannot move any further backward. The program can continue in the forward direction by commanding a positive velocity.		
Response	Class	1	Warning, no reaction. Stop the movement.
Solution	Class	1	Check and modify the process. The program can continue in the forward direction by commanding a positive velocity.
Parameter	%1:		Current value [-]
Error type	1, Error message from NC program.		

ID 51024

Due to a movement of the DR channel the Escape channel is shifted into the APPROACH.			
Description	When an escape shift is active, the escape channel can be “forced” into the approach by a movement of the DR channel in the direction of the approach end. However, a superposition of the DR channel is not permitted when escape in the approach is active.		
Response	Class	4	Movement stop.
Solution	Class	6	Prevent a DR movement in the approach direction by the PLC when escape is active in the area of the approach end.
Parameter	%1:		Current value [0.1 10^-3 mm or ø]
			Current distance of the active escape channel.
	%2:		Limit value [0.1 10^-3 mm or ø]
			Current distance up to the approach end.
	%3:		State [0.1 10^-3 mm or ø]
			Total distance of the approach geometry.
	%4:		Upper limit value [0.1 10^-3 mm or ø]
			Total distance of the escape geometry (distance to the approach end + approach distance)
Error type	1, Error message from NC program.		

ID 51025

Initialization of the kinematic of the TCP velocity limit failed.			
Description	The kinematic parameters for the kinematics for the TCP velocity limits are incorrectly parameterised.		
Response	Class	4	Movement stop.
Solution	Class	6	Check, correct and update the kinematic parameters.

Parameter	%1:	Current value [-]
Error type	1, Error message from NC program.	

ID 51026

The relative start value for fixed stop search is too big.			
Description	A value of > 1000 was specified for the P-AXIS-00772 parameter. The P-AXIS-00772 parameter specifies the activation position of the fixed stop search in 0.1% referred to the length of the measurement block. A value > 1000 is not possible.		
Response	Class	1	Warning output
Solution	Class	1	P-AXIS-00777 must be assigned a value of less than 1000. If P-AXIS-00777 has a value less than zero, check P-AXIS-00772.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Error value [0.1%]	
		Incorrect value of P-AXIS-00777	
	%3:	Upper limit value [0.1%]	
		Upper limit value	
Error type	1, Error message from NC program.		

ID 51027

Storage value P-STUP-00033 is too small to save a contiguous program sequence.			
Description	Functions such as real-time loops must first have a fully contiguous program sequence in the interpolator before processing can continue. With this error, the interpolator cannot store a contiguous program sequence since the memory configured in P-STUP-00033 is too small. There is no more memory space available after the current NC block.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Check and increase P-STUP-00033 or reduce the number of NC blocks in the program sequence.
Parameter	%1:	Block number [-]	
		Block number of the NC block after which the memory space is insufficient.	
	%2:	Current value [Byte]	
		Memory space required by the current NC block.	
	%3:	Limit value [Byte]	
		Free memory space still available.	
	%4:	Upper limit value [Byte]	
		Currently configured total memory space.	
	%5:	Current value [-]	
		Number of NC blocks saved so far in the sequence.	
Error type	1, Error message from NC program.		

ID 51028

No buffer available for realtime cycles.

Description	The number of possible actions within a real-time cycle (single-axis movement, spindle command, etc.) was exceeded. This number is defined in the parameter P-CHAN-00480.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the actions within the real-time cycle. Alternatively, increase P-CHAN-00480.
Parameter	%1:	Identification number [-]	
		ID of the real-time cycle what triggered the action.	
Error type	-		

ID 51029

Not enough memory for realtime cycles.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Lower limit value [-]	
	%2:	Current value [-]	
Error type	-		

ID 51030

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51031

With Bisector Escape Strategy the DR geometry moved more than 180 degrees.			
Description	The tangent directions are checked before and after exceeding the horizontal in the geometry of the DR channel above the height $Z_{Bisector}$ + D1 in the Bisector Escape Strategy during program runtime. These orientations must match. This is not the case.		
Response	Class	6	Movement stop.
Solution	Class	7	Select the parameter D1 in the Bisector Escape Strategy so that the escape geometry corresponds to a flat escape strategy for problem elements or else change the escape strategy used. Alternatively, you can also change the DR geometry.
Error type	1, Error message from NC program.		

ID 51032

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51033

During activation of #TRACK CHAN ON [ESCAPE], no commanded geometry from the master channel was found.			
Description	No commanded escape geometry was found by the master channel when #TRACK CHAN ON [ESCAPE] was activated. Therefore #CHANNEL INTERFACE ON[ESCAPE GEOMETRY= ..] must be programmed in the master channel before this command.		
Response	Class	6	Movement stop.
Solution	Class	3	Check and modify the master program. Program #CHANNEL INTERFACE ON[ESCAPE GEOMETRY= ..] before activating #TRACK CHAN ON [ESCAPE] in the escape channel
Error type	1, Error message from NC program.		

ID 51034

During activation of #CHANNEL INTERFACE ON [ESCAPE] no Escape geometry is available.			
Description	When an escape is first commanded, the escape strategy must also be transferred since there is no preceding escape strategy. It is not permitted to program an escape command without previously specifying a corresponding strategy.		
Response	Class	6	Movement stop.
Solution	Class	3	Check and modify the NC program. Specify the escape strategy before #CHANNEL INTERFACE ON [ESCAPE].
Error type	1, Error message from NC program.		

ID 51035

Command #CHANNEL INTERFACE SET [ESCAPE GEOMETRY ...] cannot be executed because no Escape is activated.			
Description	The command #CHANNEL INTERFACE SET [ESCAPE GEOMETRY ...] cannot be executed if no escape was activated.		
Response	Class	6	Movement stop.
Solution	Class	3	Check and modify the NC program. Activate the escape channel with V#TRACK CHAN ON [ESCAPE] before the command #CHANNEL INTERFACE SET.
Error type	1, Error message from NC program.		

ID 51036

Delete distance to go is aborted before end mark is reached because NC command can not be skipped.

Description	The “Delete distance to go” function (see FCT-C28) interrupts the current path motion and generated a linear path motion to the target position of the next motion block of the selected end identifier. Certain block types, e.g. a measuring motion G100, may not be changed or omitted. In this case, the controller generates this warning and produces a linear path motion to the start point of the block that cannot be changed or omitted. The “Delete distance to go” is then deselected.		
Response	Class	1	Generate a direct path motion to the NC block that cannot be changed or omitted.
Solution	Class	1	Check and modify the PLC command “Delete distance to go”
Parameter	%1:	Block number [-]	
		Block number before which the “Delete distance to go” function was implicitly deselected.	
Error type	1, Error message from NC program.		

ID 51037

Position request during active APPROACH is not permitted.			
Description	No command value requests are permitted in commands #ESCAPE PATH DEF BEGIN and #ESCAPE PATH DEF END. For example, they were generated by activating a coordinate system with an active transformation with position request. … N30 #TRAFO ON N40 #ESCAPE PATH DEF BEGIN … N60 CS ON[..] **(Position request during active APPROACH is not permitted.** !)		
Response	Class	6	Movement stop.
Solution	Class	3	Check and modify the programming of the NC command #ESCAPE PATH ...
Error type	1, Error message from NC program.		

ID 51038 / 51039

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51040

Function ESA not configured. Set configuration parameter FCT_CALC_STATE_AT_T.

Description	The estimation function is to be used.
	The NC command
	`#CHANNEL SET[ESA_TIME1=2]`
	a measuring point can be defined for the “Estimation of future data” function.
	In order to use this function, it must be activated by the parameter P-CHAN-00655 (or P-STUP-00070).
	
	Activate in the channel parameter list using P-CHAN-00650:
	`configuration.interpolator.function FCT_DEFAULT | FCT_CALC_STATE_AT_T`
	Alternatively in the start-up list for the first channel using P-STUP-00070:
	`configuration.channel[0].interpolator.function FCT_DEFAULT | FCT_CALC_STATE_AT_T`		

Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	5	Check P-CHAN-00650 (or P-STUP-00070) and activate the function.
Error type	-		

ID 51041

Changing condition of realtime loop during active delete distance to go not permitted.			
Description	While the Delete distance to go function is active, do not change the condition for existing a real-time loop. The CNC stops with feedhold.		
Response	Class	1	Output the warning and stop with feedhold
Solution	Class	1	Check and modify the change for the loop condition of the real-time loop
Error type	-		

ID 51042

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51043

Filter is not active, because no filter type is set.			
Description	No filter type (P-AXIS-00586) was defined for the filter in the required axis. The filter in this axis was therefore deactivated. To activate the filter, at least the filter type and a filter order must be specified for this filter.		
Response	Class	2	Warning.
Solution	Class	3	Check and modify the filter type (P-AXIS-00586) for the filter of the axis concerned
Error type	1, Error message from NC program.		

ID 51044

Invalid combination: NC command #FILTER or #VIB GUARD has no effect.			
Description	The FIR filter and the Vibration Guard cannot be used together or combined. The two functions can be influenced both via the axis parameters and via the NC program. • FIR-Filter - #FILTER[…] • Vibration Guard - #VIB GUARD[…] If combinations occur due to an NC command programmed at a later date, this command has not effect.		
Response	Class	1	Warning.
Solution	Class	1	When the FIR filter or the Vibration Guard is activated via the axis parameters, the NC commands used must always match.
Error type	1, Error message from NC program.		

ID 51045

Warning: Functionality Vibration Guard activated with NC command #VIB GUARD in all axes.			
Description	With the Vibration Guard function, the NC command #VIB GUARD always acts on all axes. If a Vibration Guard was not activated in all axes, this NC command now activates the Vibration Guard in the axes where the Vibration Guard was previously not activated.		
Response	Class	1	Warning.
Solution	Class	1	If the Vibration Guard should only act in some axes, this cannot be controlled by the NC command. Axis specific use can only be achieved by parameterisation and activation in the axis lists (see P-AXIS-00588, P-AXIS-00571, P-AXIS-00589, P-AXIS-00568).
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis.	
Error type	1, Error message from NC program.		

ID 51046

The measured position is before the trigger position for probing in the movement block.			
Description	When measuring to a fixed stop, a measured value was acquired before the configured enable position for acquiring the measured value (P-AXIS-00776, P-AXIS-00777). The output of this error message can be suppressed by the channel parameter P-CHAN-00312. In this case, it is necessary to check whether a correct measurement was performed by checking the variable V.A.MERF.*.		
Response	Class	6	Error message output abort program.
Solution	Class	4	Check parameterisation for measuring travel to fixed stop. P-AXIS-00774, P-AXIS-00775, P-AXIS-00776, P-AXIS-00777, P-AXIS-00778

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of axis concerned.
	%2:	Error value [0.1 10^-7 mm or ø]
		Command position in the current cycle
	%3:	Current value [0.1 10^-7 mm or ø]
		Actual value in the current cycle
	%4:	Current value [%]
		Distance covered in movement block
	%5:	Current value [%]
		Configured minimum path (P-AXIS-00776, P-AXIS-00777)
Error type	1, Error message from NC program.	

ID 51047

During active "delete distance to go" an invalid value of dist_to_geom_end_r is monitored.			
Description	The originally programmed contour was modified by the “Delete distance to go” function. However, the readout dist_to_geom_end_r refers to the original programmed contour, which is why a "Delete distance to go" in a sequence flagged with #DIST TO GO BEGIN/END results in an incorrect readout.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Avoid “Delete distance to go” in the program section.
Error type	-		

ID 51048

Too small P-STUP-00033 leads to delayed display of the HLI variable dist_to_geom_end_r.			
Description	The NC command #DIST TO GO BEGIN flags the start of a sequence for which the axis path is to be calculated up to the end of the sequence with #DIST TO GO END. The entire sequence must be available in order to determine the axis path up to the end of the sequence. This is not always possible if P-STUP-00033 is parameterised too small. If the memory is too small, the axis travel path cannot be calculated up to the #DIST TO GO END flag and the display via the HLI datum dist_to_geom_end_r remains invalid until the sequence is fully available.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and correct the parameter P-STUP-00033
Parameter	%1:	Block number [-]	
		Block number of the last NC block that is still within the sequence. Acts as a reference point for increasing P-STUP-00033.	
Error type	-		

ID 51049

Monitoring levels in activated areas has to be identical.

Description	Starting position: Several areas are active.
	However, the activated workspace and protection areas do not have the identical monitoring planes.
	Example with error:
	`%area`
	`%Lsub_area_def_imcs`
	`#CONTROL AREA BEGIN [ID=1 WORK MONITOR_LVL = "IMCS" POLY`
	`MIN_EXCUR=0MAX_EXCUR=100]`
	`N20G01F1000G90X50Y50`
	`N30X-50`
	`N50Y-50`
	`N60X50`
	`N70X50 Y50`
	`N80 #CONTROL AREA END`
	`M29`
	
	`%Lsub_area_def_mcs`
	`#CONTROL AREA BEGIN [ID=2 PROT MONITOR_LVL = "MCS" POLY`
	`MIN_EXCUR=0MAX_EXCUR=100]`
	`N20G01F1000G90X50Y50`
	`N30X55`
	`N50Y55`
	`N60X50`
	`N70X50 Y50`
	`N80 #CONTROL AREA END`
	`M29`
	
	`%main`
	`LL sub_area_def_imcs`
	`LL sub_area_def_mcs`
	`#CONTROL AREA ON ALL`
	`; …`
	`M30`
	
	The two lines must be corrected, either both must contain
	`MONITOR_LVL = "IMCS" or MONITOR_LVL = "MCS"`
	.		

Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Check NC program
Parameter	%1:	Current value [-]	
		Identification number of the control area	
	%2:	Expected value [-]	
		Active monitoring level	
	%3:	Current value [-]	
		Monitoring level of the areas	
Error type	1, Error message from NC program.		

ID 51050

Exception in CalculateOrientation() of dynamic contour control.			
Description	When a TcCOM object of the Dynamic Contour Control type was used, an exception occurred in the CalculateOrientation() function.		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	4	Check and modify the calculations in the CalculateOrientation() function implemented.
Error type	1, Error message from NC program.		

ID 51051

Exception in calculation of the online tool radius compensation.			
Description	When a TcCOM object of the Online Tool Radius Compensation type was used, an exception occurred in the CalculateTrcPath() function.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify the calculations in the CalculateTrcPath() function implemented.
Parameter	%1:	Current value [-]	
		Mode used for online tool radius compensation	
Error type	1, Error message from NC program.		

ID 51052

Exception in CalculateOnNewGeometry() of dynamic contour control.			
Description	When a TcCOM object of the Dynamic Contour Control type was used, an exception occurred in the CalculateOnNewGeometry() function.		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	4	Check and modify the calculations in the CalculateOnNewGeometry() function implemented.
Error type	1, Error message from NC program.		

ID 51053

Exception in CalculateOffset() of dynamic contour control.			
Description	When a TcCOM object of the Dynamic Contour Control type was used, an exception occurred in the CalculateOffset() function.		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	4	Check and modify the calculations in the CalculateOffset() function implemented.
Error type	1, Error message from NC program.		

ID 51054

Exception in method CalculateFeedAdaption() of geometric feed adaptation.			
Description	When a TcCOM object of the Geometric Feed Adaptation type was used, an exception occurred in the CalculateFeedAdaption() function.		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	4	Check and modify the calculations in the CalculateFeedAdaption() function implemented.
Error type	1, Error message from NC program.		

ID 51055

Error in method CalculateFeedAdaption() of geometric feed adaptation.	
Description	When a TcCOM object of the Geometric Feed Adaptation type was used, an error occurred in the CalculateFeedAdaption() function.

Response	Class	6	Controlled stop for the axis group.´
Solution	Class	4	Check and modify the calculations in the CalculateFeedAdaption() function implemented.
Parameter	%1:	Current value [-]	
		Return value of the function	
Error type	1, Error message from NC program.		

ID 51056

‘Conveyor Tracking’ function is not configured.			

Description	The NC commands #SYNC IN/OUT are only available if the configuration parameter FCT_DLM is set P-CHAN-00650 (alternatively P-STUP-00070).
	`configuration.interpolator.function FCT_IPO_DEFAULT |`
	The ‘Conveyor Tracking’ function is not configured.
	`configuration.interpolator.function FCT_IPO_DEFAULT|FCT_DLM`		

Response	Class	4	Abort NC program processing.
Solution	Class	6	Check and modify the configuration. Set configuration parameter FCT_DLM and reboot the CNC.
Error type	1, Error message from NC program.		

ID 51057

NC command inside the APPROACH geometry not allowed.			
Description	An NC command that is not allowed is programmed within the approach definition marked by #ESCAPE PATH DEF BEGIN.		
Response	Class	3	Abort NC program processing.
Solution	Class	2	Check and modify the NC program. Remove the faulty NC command from the approach definition
Error type	1, Error message from NC program.		

ID 51058

The parameter ERR_NO_SIGNAL is ignored in the measurement movement because of delete distance to go.			
Description	An active measurement run was aborted by the Delete distance to go function. No error is output if no measuring event occurs up to this point, even if the ERR_NO_SIGNAL parameter was set in the NC command #MEAS.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	If the abort is not required, the program must wait for the correct point in time to use Delete distance to go.
Error type	1, Error message from NC program.		

ID 51059- 51061

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51063

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51065

Invalid combination of moving axes in manual mode.			
Description	The combination of commanded axes in manual mode is not permissible. The motion of the invalid axis is discarded. The motion of the other axes is continued.		
Response	Class	1	Output a warning; invalid axis motion is discarded.
Solution	Class	1	Stop the invalid axes.
Parameter	%1:	Error value [-]	
		Axes moved by user in manual mode (bit array)	
	%2:	Error value[-]	
		Axes whose motions are discarded (bit array)	
	%3:	Corrected value [-]	
		Axes that actually move (bit array)	
Error type	-		

ID 51066

#GEAR LINK ON without coupling definition programmed.			
Description	The NC command #GEAR LINK ON [TARGET=…] was programmed for an axis without programming coupling definitions in a previous or in the same NC block.		
Response	Class	2	Error message output, program abort
Solution	Class	3	Before activating the axis coupling, specify the coupling definitions using #GEAR LINK … [TARGET= AX1= … NUM1= … DENOM1= … …]
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	-		

ID 51068

Movement too close to kinematic singularity. Axis velocity exceeds maximum limit.			
Description	The maximum axis velocity (P-AXIS-00212) set in the axis parameter list was exceeded near a singularity. Dynamic look-ahead also considers the error threshold defined by P-AXIS-00440.		
Response	Class	4	Controlled stop for the axis group.´

Solution	Class	6	Create diagnosis data.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current command velocity of the axis.	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum permissible velocity of the axis (P-AXIS-00212).	
	%4:	Current value[-]	
		Ratio of current velocity to maximum permissible axis velocity (absolute value).	
	%5:	Upper limit value[-]	
		Factor for exceeding the maximum permissible velocity (P-AXIS-00440)	
Error type	1, Error message from NC program.		

ID 51069

Movement too close to kinematic singularity. Axis acceleration exceeds maximum limit.			
Description	The maximum axis acceleration (P-AXIS-00008) set in the axis parameter list was exceeded above the permissible limit. Dynamic look-ahead also considers the error threshold defined by P-AXIS-00442.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Create diagnosis data.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [mm/s² or °/s²]	
		Current command acceleration of axis.	
	%3:	Upper limit value [mm/s² or °/s²]	
		Maximum permissible acceleration of the axis (P-AXIS-00008).	
	%4:	Current value[-]	
		Ratio of current acceleration to maximum permissible acceleration (absolute value).	
	%5:	Upper limit value[-]	
		Factor for permissible excess of acceleration (P-AXIS-00442).	
Error type	1, Error message from NC program.		

ID 51070

Movement too close to kinematic singularity. Axis jerk exceeds maximum limit.			
Description	The maximum axis jerk set in the axis parameter list (P-AXIS-00008, P-AXIS-00201) was exceeded. Dynamic look-ahead also considers the error threshold defined by P-AXIS-00444.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Create diagnosis data.

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of axis affected.
	%2:	Current value [mm/s^3 or °/s^3]
		Current command jerk of axis.
	%3:	Upper limit value [mm/s^3 or °/s^3]
		Maximum permissible jerk of the axis (P-AXIS-00008, P-AXIS-00201).
	%4:	Current value[-]
		Ratio of current jerk to maximum permissible jerk (absolute value).
	%5:	Upper limit value[-]
		Factor for permissible excess of jerk (P-AXIS-00444).
Error type	1, Error message from NC program.	

ID 51071

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51072

Motion blocks are not allowed after #ESCAPE PATH POST SEQUENCE.			

Description	Programming a motion block is not permitted after processing #ESCAPE PATH POST SEQUENCE.
	#ESCAPE PATH Command structure:
	1. #ESCAPE PATH DEF BEGIN
	2. #ESCAPE PATH BACKWARD STOP
	3. #ESCAPE PATH POST SEQUENCE
	4. #ESCAPE PATH DEF END
	
	Incorrect example (N75)
	`N010 #ESCAPE PATH DEF BEGIN`
	`N020 #CS ON [APP] [@PL4,@PL5,@PL6,@PL7,@PL8,@PL9]`
	`N030 #ESCAPE PATH BACKWARD STOP`
	`N040 G01 Z7.07 F30`
	`N050 G01 Y0`
	`N060 G01 Z0`
	`N070 #ESCAPE PATH POST SEQUENCE`
	`N075 G01 Y0.1 (Incorrectly placed motion block)`
	`N080 #CS DEL ALL`
	`N090 #ESCAPE PATH DEF END`
	N100 M30		

Response	Class	2	NC program execution aborted and transition to error state
Solution	Class	3	Check and modify the NC program
Parameter	%1:	Current value [-]	
		Block number of #ESCAPE PATH POST SEQUENCE	
Error type	1, Error message from NC program.		

ID 51073

There are too many NC control commands programmed between #ESCAPE PATH BACKWARD STOP and #ESCAPE PATH DEF END.			
Description	Only a limited number of control commands is permitted between the NC commands #ESCAPE PATH BACKWARD STOP and #ESCAPE PATH DEF END. An unlimited number of movement blocks can be programmed between the commands. #ESCAPE PATH Command structure: 1. #ESCAPE PATH DEF BEGIN 2. #ESCAPE PATH BACKWARD STOP 3. #ESCAPE PATH POST SEQUENCE 4. #ESCAPE PATH DEF END		
Response	Class	2	NC program execution aborted and transition to error state
Solution	Class	3	Check and modify the NC program; reduce the control commands after #ESCAPE PATH BACKWARD STOP.
Parameter	%1:	Current value [-]	
		Block number of control command that exceeds the permitted number.	
Error type	1, Error message from NC program.		

ID 51076 - 51078

System error [▶ 12]			
Description	This error message contains information about internal states, information about the error cause and the error location. Please pass on the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51079

Missing license for real-time loops.			
Description	The EDM license pack includes the programming of only one real-time loop. The corresponding license must be added to perform additional real-time loops.		
Response	Class	6	Abrupt axis stop for all axes and transition to an error state
Solution	Class	6	Possible solutions are: • Change the NC programming • Add the license for real-time loops
Error type	1, Error message from NC program.		

ID 51081

Channel synchronisation within #OPTIONAL EXECUTION sequence not possible.

Description	When the Skipping program sequences (#OPTIONAL EXECUTION) function is used, the programming of commands that trigger a synchronisation between the decoder and the interpolator, e.g. #FLUSH WAIT, is not allowed when this control unit is active..
	The following commands are not allowed within the following programming functions:
	`#OPTIONAL EXECUTION ON`
	`…`
	`#OPTIONAL EXECUTION OFF`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 51082

Non-homed axis is to be moved although prog_move_req_homing is set.			
Description	A non-homed axis is to be interpolated although the parameter P-AXIS-00277 (kenngr.prog_move_requires_homing) is set for this axis.		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Conduct a rest; home the axis before interpolating it. Or do not set P-AXIS-00277.
Parameter	%1:	Logical axis number	
		Logical axis number P-AXIS-00016 of the axis affected	
Error type	-		

ID 51083

Timeout during drive request.			
Description	The permissible response time was exceeded for a read or write access to the drive via (#IDENT WR/ #IDENT RD). The permitted duration is 10 seconds.		
Response	Class	6	Controlled stop of the axis
Solution	Class	6	Check whether the drive has a problem.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of drive affected	
	%2:	Identification number [-]	
		Number of the object that received no response. (decimal notation)	
	%3:	Current value [-]	
		Return value 1 of the service channel	
	%4:	Current value [-]	
		Return value 2 of the service channel	
Error type	-		

ID 51084

Queue for pending drive requests is already full.

Description	The maximum number of possible drive requests is dependent on the configuration. This number was exceeded. Possible causes: • The drive requires longer than expected to send the responses. • Too many drive requests were programmed.		
Response	Class	6	Controlled stop of the axis.
Solution	Class	6	Reduce the number of drive requests
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of drive affected	
	%2:	Identification number [-]	
		Number of object. (decimal notation)	
	%3:	Current value [-]	
		Return value 1 of the service channel	
	%4:	Current value [-]	
		Return value 2 of the service channel	
Error type	-		

ID 51085

Negative acknowledgement of a drive request.			
Description	The drive returned a negative acknowledgement for an NC command #IDENT WR SYN.		
Response	Class	6	Controlled stop of the axis.
Solution	Class	6	Check why the drive returned a negative acknowledgement.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of drive affected	
	%2:	Identification number [-]	
		Number of object. (decimal notation)	
	%3:	Current value [-]	
		Return value 1 of the service channel	
	%4:	Current value [-]	
		Return value 2 of the service channel	
Error type	-		

ID 51086

Invalid parameters for axis coupling in escape geometry.			
Description	In the Defining escape geometry (#ESCAPE PATH DEF BEGIN - #ESCAPE PATH DEF END) function, it is only possible to activate the PCS transformation with axis coupling (#TRAFO PCS [212]) if the transformation data are correctly parameterised. The logical axis number of the master axis must match the second axis index. The logical axis number of the slave axis must match the third axis index.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Possible solutions: • Change the programming • Change the configuration

Parameter	%1:	Current value [-]
		Axis index of the master axis
	%2:	Expected value [-]
		Expected axis index for the master axis
	%3:	Current value [-]
		Axis index of he slave axis
	%4:	Expected value [-]
		Expected axis index for the slave axis
Error type	1, Error message from NC program.	

ID 51087

During active orbit extension NC commands with position initialisation are not permitted.			
Description	As soon as the command #TRACK CHAN ON [EXTEND …] was evaluated in a channel configured as an orbit channel, the orbiting process becomes active. Afterwards, no NC commands requiring an initialisation of the NC channel with axis positions may be used if the radius in the down channel is > 0. NC commands requiring a channel initialisation include, for example, #CS, #TRAFO and #CHANNEL INIT.		
Response	Class	4	Abort NC program processing.
Solution	Class	6	Check and modify the NC program in the orbit channel.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Current radius in the orbit channel.	
Error type	-		

ID 51088

In the escape channel the command #CHANNEL SET [EXT_FEEDRATE_WAIT=1] is missing.			
Description	In the escape channel, electrode escape is controlled either by the escape motion control unit or by the external commanded velocity control unit. To ensure that the escape is executed at the correct velocity, the command #CHANNEL SET [EXT_FEEDRATE_WAIT = 1] must be programmed in the NC program of the escape channel. The command can also be used to change the command velocity unit, if required. If the command is not programmed, this warning is output and a switchover takes place in the CNC so that it only responds to the external velocity.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and modify the NC program.
Error type	-		

ID 51089

The end positions of path erosion and escape geometry are not identical.

Description	To ensure a continuous escape geometry can be calculated in the escape channel for the electrode, the origin of the coordinate system in the down channel must be located at the path erosion end point when the escape channel is activated the first time. If this error is output, the origin is not located at the path erosion end point, i.e. the positions are not identical. For more information, see Coordinate system for orbiting		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program; change the definition of the coordinate system in the down channel before the first time the escape channel is activated.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		The end point of the geometry calculated online in the path erosion coordinate system.	
	%2:	Expected value [0.1 10^-3 mm or ø]	
		The end point of the path erosion geometry in the path erosion coordinate system.	
Error type	-		

ID 51090

Preparation of a measurement was aborted.			
Description	An axis can be prepared for a measurement run by the NC command #MEAS PREPARE The preparation was aborted. Possible causes for abortion: • Program end was reached without execution of a measurement run. • An axis prepared for a measurement run was removed from the channel; measurement preparation is aborted for all axes. • Homing was programmed for a prepared axis; measurement preparation is aborted for all axes.		
Response	Class	1	Output of the warning; preparation for the measurement run aborted.
Solution	Class	1	Check and modify the flow in the NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	-		

ID 51091

A position initialisation in the channel is not allowed while orbiting motion is active.			
Description	When the interface between an orbit channel and a down channel is active and when a radius is > 0, the orbit channel transforms the supplied radius to the orbit geometry. Due to this asynchronous superimposition, it is not possible to perform position synchronisation in the down channel. If position synchronisation is required while the interface is active, the radius in the down channel must first be moved to 0.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program. The NC block specified in the error results in position synchronisation.
Error type	-		

ID 51092

Angle projection only possible while orbit is in scaled area.			
Description	A combination of PCS transformation with angle projection (#TRAFO PCS ID [214]) and an active channel interface #TRACK CHAN ON[EXTEND MAX_SCALE=<> MAX_EQUID=<>] is only possible in the scaled area.		
Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	3	Possible solutions: • Modify the NC program • Change the configuration
Parameter	%1:	Current value [-]	
		Maximum value of the scaled area	
	%2:	Current value [-]	
		Maximum value of the equidistant area	
Error type	1, Error message from NC program.		

ID 51093

The moving direction of the virtual conveyor axis is wrong programmed.			

Description	When the “Machining on moved workpiece” functionality is used with a virtual conveyor axis, the virtual conveyor axis was programmed in the wrong direction. A virtual conveyor axis assumes that the machine cannot move in the same or opposite conveying direction. This means that the virtual axis can only move in the opposite conveying direction.
	
	Incorrect program example where X is the virtual conveyor axis and conveyor_sync.move_direction = 0 (P-CHAN-00363):
	`N10 #SYNC IN`
	`N20 G0 G90 X-50 Y10 F100`
	`N30 G01 X-25 Y10`
	`;…`
	
	Incorrect program example where X is the virtual conveyor axis and conveyor_sync.move_direction = 0:
	`N10 #SYNC IN`
	`N20 G0 G90 X50 Y10 F100`
	`N30 G01 X25 Y10`
	`;…`
	For further information, see Synchronisation without a parallel axis to the conveying direction		

Response	Class	4	Controlled stop for the axis group.´
Solution	Class	2	Check and modify the NC program. Change the conveying direction of the virtual conveyor axis in the NC program.

Parameter	%1:	Error value [-]
		Currently programmed conveying direction of the virtual conveyor axis in the NC program: 1: positive direction -1: negative direction
	%2:	Current value [-]
		Configured conveying direction of the conveyor belt in P-CHAN-00363 conveyor_sync.move_direction: 1: positive conveyor direction 0: negative conveyor direction
	%3:	Current value [0.1 µm]
		Start position of the virtual conveyor axis in the current motion block.
	%4:	Current value [0.1 µm]
		End position of the virtual conveyor axis in the current motion block.
Error type	1, Error message from NC program.	

ID 51098

Thread cutting exceeds maximum permissible path velocity.			
Description	During thread cutting G33 the path velocity results from the specified thread pitch and the current spindle speed. However, the resulting velocity is greater than the maximum permissible path velocity in the thread cutting block, i.e. the maximum velocity of at least one of the axes participating in the motion is exceeded (see P-AXIS-00212). One remedy is to reduced the speed S of the spindle		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Reduce spindle speed S
Parameter	%1:	Error value [1µm/s or 0.001°/s]	
		Resulting path velocity (thread pitch * spindle speed)	
	%2:	Upper limit value[1µm/s or 0.001°/s]	
		Maximum permissible axis velocity	
	%3:	Current value[1µm/s or 0.001°/s]	
		Programmed thread pitch velocity	
	%4:	Current value[1µm/s or 0.001°/s]	
		Velocity scaling factor	
Error type	1, Error message from NC program.		

2.6.5.1 ID 51099

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51101

Axis overload in thread chain with the given spindle speed or axis movement in thread block too short.

Description	During thread cutting G33 the path velocity results from the specified thread pitch and the current spindle speed. When transitions are not tangential between two thread cutting blocks G33, the axis movements must be contoured However, the existing block length is not sufficient for the resulting path velocity. One remedy is to reduce the speed S of the spindle.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Reduce spindle speed S
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Block number [-]	
		Block number of first thread cutting block	
	%3:	Error value [0.1 10^-3 mm or ø]	
		Axis movement in the first thread cutting block	
	%4:	Block number [-]	
		Block number of second thread cutting block	
	%5:	Current value [0.1 10^-3 mm or ø]	
		Axis movement in the second thread cutting block	
Error type	1, Error message from NC program.		

ID 51102

Too many block transitions in thread chain, no segment memory left.			
Description	When block transitions are not tangential in the thread chain (series of consecutive G33 thread cutting blocks), contouring must take place. The maximum possible number of non-tangential axis transitions is limited in a thread chain. The maximum value is contained in the third parameter.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Number of current calculated axis transitions	
	%3:	Upper limit value [-]	
		Maximum number of possible non-tangential axis transitions per thread chain	
Error type	1, Error message from NC program.		

ID 51103

Axis overload in thread chain with the given spindle speed or axis movement in thread block too short.			
Description	During thread cutting G33 the path velocity results from the specified thread pitch and the current spindle speed. When transitions in a thread chain are non-tangential between two thread cutting blocks G33, the axis movements must be contoured. However, the existing block length is not sufficient for the resulting path velocity. One remedy is to reduce the speed S of the spindle.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Reduce spindle speed S

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Error value [µs]
		Parameter for internal error analysis
	%3:	Upper limit value [µs]
		Parameter for internal error analysis
	%4:	Current value [µs]
		Parameter for internal error analysis
	%5:	Current value [µs]
		Parameter for internal error analysis
Error type	1, Error message from NC program.	

2.6.5.2 ID 51104 - 51108

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51109

Feedhold or override zero in spindle during thread chain.			
Description	When a thread chain (series of consecutive G33 thread cutting blocks) is processed, 0 feedhold or override was set on the spindle axis. It is not permitted to stop while a spindle chain is processed.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check the flow. The spindle axis may not be stopped while the thread chain is processed.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		TRUE: Spindle was stopped with feedhold or override 0.	
Error type	1, Error message from NC program.		

2.6.5.3 ID 51110

System error [▶ 12]			
Description	This error message contains information about internal states, information about the error cause and the error location. Please pass on the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51111

No data for an electrode retraction were saved.

Description	When a V.A.RETRACT_ACS_POS variable was accessed, it was found that no ACS positions had yet been saved. Before variables can be used, positions must be saved either implicitly by a CNC error or explicitly using the bahn_mc_control.save_retract_data control unit. The function is activated by P-CHAN-00865.		
Response	Class	4	Controlled stop for the axis group.´ Program abort
Solution	Class	3	Only use variables for an electrode retraction if positions were previously saved.
Error type	1, Error message from NC program.		

ID 51112

The programmed NC command is only allowed with active functionality retract_after_error.			
Description	The electrode retraction function must be activated using P-CHAN-00865 (retract_after_error). This error occurs because an NC command of this function was used but the parameter was not set.		
Response	Class	4	Controlled stop for the axis group.´ Program abort
Solution	Class	3	Reboot the CNC with the parameter activated and restart the NC program.
Error type	1, Error message from NC program.		

2.6.5.4 ID 51113

System error [▶ 12]			
Description	This error message contains information about internal states, information about the error cause and the error location. Please pass on the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 51114

3D distance control activated in mode ECS without active kinematic 98.			
Description	The “ECS” mode in the 3D distance control function compensates for height differences in the tool direction. This requires activation of kinematic type 98 and a subordinate kinematic as a kinematic chain.		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	6	Activate transformation, #TRAFO ON
Parameter	%1:	Current value [-]	
		Channel number	
	%2:	Current value [-]	
		Selected transformation type	
	%3:	Current value [-]	
		Kinematic ID of the first step of the kinematic chain	
	%4:	Error value [-]	
		Kinematic ID of the second step of the kinematic chain; kinematic type 98 must be configured in this step.	

Error type	1, Error message from NC program.

ID 51115

3D distance control in mode ECS activated without active transformation.			
Description	The “ECS” mode in the 3D distance control function compensates for height differences in the tool direction. This requires activation of kinematic type 98 and a subordinate kinematic as a kinematic chain.. Kinematic type 98 was not configured or not yet activated. See also documentation Multistep kinematic transformations.		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	6	Activate or configure kinematic type 98 in step 2 of the kinematic chain
Parameter	%1:	Current value [-]	
		channel number	
	%2:	Error value [-]	
		Selected transformation type: 1: RTCP active (required) 3: Transformation not active	
	%3:	Current value [-]	
		Kinematic ID of the first step of the kinematic chain	
	%4:	Current value [-]	
		Kinematic ID of the second step of the kinematic chain	
Error type	1, Error message from NC program.		

ID 51116

Active PCS transformation at escape request could not be found in escape definition.			
Description	If a PCS transformation is active in the down channel, the system checks whether the same transformation type is also programmed in the escape channel in the event of an escape request. This error occurs if different transformation types are programmed.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify the PCS transformation types in the down and escape channel programs.
Parameter	%1:	Expected value [-]	
		Active PCS transformation type in the down channel	
Error type	1, Error message from NC program.		

ID 51117

For electrode retraction the parameter P-STUP-00033 has to be configured.			
Description	An attempt was made to execute an electrode retraction in the die sinking system after an error. One condition for this is a configured memory with the parameter P-STUP-00033 (fb_storage_size).		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Reboot the CNC with a configured memory using the memory in P-STUP-00033.

Error type	1, Error message from NC program.

ID 51118

No retraction data saved, but retraction cycle was called.			
Description	The CNC cycle for electrode retraction uses stored system data at interruption time to calculate the retraction. System data is saved either by a CNC error of the save_retract_data control unit. Only then can the CNC cycle be called to retract the electrode.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check whether the data is correctly saved by the control unit.
Error type	1, Error message from NC program.		

ID 51119

Other kinematic types are active than at interruption time.			
Description	An attempt was made to execute an electrode extraction via the CNC cycle. The stored data for the retraction contains different kinematic transformation types than were active at interruption time. A program name must be transferred to the electrode retraction to activate the kinematic transformation. If this error is output, the kinematic types selected in the transferred program do not coincide with the stored kinematic types.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and correct the program to select the kinematic transformations.
Parameter	%1:	Current value [-]	
		Index of the incorrect kinematic transformation	
	%2:	Current value [-]	
		Stored transformation type	
	%3:	Error value [-]	
		Transformation type selected by the electrode cycle.	
Error type	1, Error message from NC program.		

2.6.5.5 ID 51120

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

2.7 Spindle interpolator error (ID-range 60000-69999)

2.7.1 ID-range 60000-60249

ID 60004

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 60006

Emergency stop actuated.			
Description	On the High Level Interface (HLI) the command ipo_mc_control.emergency_stop.command_w was set for an axis-specific emergency stop. The affected axis is configured as spindle or SAI axis.		
Response	Class	2	The axis is stopped at the deceleration defined in the axis parameters by P-AXIS-00003 and the NC program is aborted.
Solution	Class	6	• The command ipo_mc_control.emergency_stop.command_w of the axis-specific emergency stop must be set to FALSE. • Conduct an NC reset. • Restart NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	-		

ID 60007 - 60017

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60018

Feed rate is 0.			
Description	For the spindle a movement with revolution zero (nil) is commanded.		
Response	Class	6	Spindle is stopped and program is aborted.
Solution	Class	6	Reset of CNC New command to spindle speed bigger than zero (nil).

Parameter	%1:	
		Logical axis number (P-AXIS-00016) of the spindle
Error type	1, Error message from NC program.	

ID 60019

Programmed feed rate greater than maximum limit.			
Description	In the tool parameters, dynamic limits can be defined for a spindle (see also [TOOL// Section: Dynamic and gear data]). These limit values are accepted and, if necessary, the feed rate is adapted to these limit values if a D word (e.g. D17) is programmed in the NC program. After adopting the tool dynamic parameters, a check is carried out of the spindle feed rate against the maximum spindle feed rate. This error message is output if the minimum spindle speed in the tool data (P-TOOL-00013) is greater than the maximum permissible axis velocity (P-AXIS-00212).		
Response	Class	1	Spindle feed rate is limited to parameterised maximum feed rate and the program continues to run.
Solution	Class	1	Set the minimum spindle speed in the tool data (P-TOOL-00013) lower than the maximum axis velocity (P-AXIS-00212).
Parameter	%1:		
			Logical axis number (P-AXIS-00016) of the spindle
	%2:		Current value [1µm/s or 0.001°/s]
			Programmed feed rate.
	%3:		Limit value [1µm/s or 0.001°/s]
			Maximum feed rate.
Error type	1, Error message from NC program.		

ID 60020

Interpolation of spindle impossible with tracking mode active.			
Description	Spindle shall be moved although the axis is in tracking mode (command position = actual position). There are different reasons, why an axis is in tracking mode: • The user commands axis tracking mode via the High Level Interface by using lr_mc_control.follow_up For detailed information on CNC realtime control signals see [HLI// Control commands of a channel]). • The axis reverts automatically to tracking mode (internal tracking mode). Possible reasons for this include: ◦ The drive displays, with this status, information that it is not ready for operation (not yet initialized). ◦ There is no valid process data received from the drive. ◦ The drive is in error state. ◦ The axis is parked (Profidrive-drive).		
Response	Class	6	The axis is stopped and the NC program is aborted.

Solution	Class	6	• Reset the signal lr_mc_control.follow_up.command_w. • Or remedy the cause for the internal tracking mode. • Do NC reset. • Restart program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60027 / 60031

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60032

Target position exceeds range of modulo.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 60034

Direction of rotation programmed towards default direction.			
Description	The spindle is to be moved against its preferred direction of rotation defined in the axis parameter list (P-AXIS-00224, P-AXIS-00031).		
Response	Class	6	Stopping the axis, and aborting the NC program.
Solution	Class	6	Change movement direction of spindle according to default direction in axis parameter list.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [-]	
		Programmed direction: 0: negative direction of rotation 1: positive direction of rotation	
	%3:	Limit value [-]	
		Preferred direction of spindle rotation (P-AXIS-00031).	
Error type	1, Error message from NC program.		

ID 60040

Indexing position not reached for spindle.			
Description	The gear is to be shifted for the spindle although it is not at the gear switch position parameterized in the axis parameter list (P-AXIS-00078).		
Response	Class	6	Stopping the axis, and aborting the NC program.
Solution	Class	6	Positioning of spindle to gear switch position before switching gears.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [-]	
		Actual nominal position of spindle.	
	%3:	Limit value [-]	
		Gear change/switch position (P-AXIS-00078)	
Error type	1, Error message from NC program.		

ID 60041 / 60049

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 60053

An error has already occurred in position controller.			
Description	The position controller module of the NC Kernel has reported an error. Another error message with information on error cause and error removal is output in addition to this error message.		
Response	Class	2	Stop the axis and abort the program.
Solution	Class	6	Removal of error in position controller and NC reset.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	-		

ID 60054

The axis must be of type spindle.			
Description	An attempt was made with an axis exchange command such as S[CALLAX] to exchange an axis into the spindle which is not configured as a spindle (see P-AXIS-00018) or as a single axis interpolator (SAI, see P-AXIS-00250). See [PROG// Section: Releasing/requesting spindle axes]		
Response	Class	2	Abort NC program.
Solution	Class	6	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Configured axis type (P-AXIS-00018)	

Error type	-

ID 60061 - 60084

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 60085

Maximum spindle speed is 0.			
Description	When turning at constant cutting speed (G96), the value 0 was programmed as the maximum spindle speed (G196). See also [[PROG// Section: Constant cutting speed].		
Response	Class	6	Stopping the axis, and aborting the NC program.
Solution	Class	6	Change NC-program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Programmed maximum cutting speed.	
Error type	1, Error message from NC program.		

ID 60086 - 60093

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60100

Homing not possible for this type of drive.			
Description	Homing was programmed for an axis whose drive type (P-AXIS-00020) does not support homing.		
Response	Class	5	Abort NC program.
Solution	Class	6	Change NC or PLC program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Configured drive type (P-AXIS-00020).	
Error type	1, Error message from NC program.		

ID 60101

Maximum permissible axis velocity was exceeded.			
Description	The maximum axis velocity set in the axis parameter list (P-AXIS-00212) was exceeded by at least 9 percent. This error message is used for diagnosis of the dynamic look-ahead and is only output if the NC kernel is parameterised accordingly.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current command velocity of the axis.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible velocity of the axis (P-AXIS-00212).	
	%4:	Current value [-]	
		Ratio of current velocity to maximum permissible axis velocity (absolute value).	
	%5:	Limit value [-]	
		Maximum permissible ratio of actual velocity to maximum permissible velocity.	
Error type	1, Error message from NC program.		

ID 60102

Warning level of axis velocity exceeded.			
Description	The maximum axis velocity set in the axis parameter list (P-AXIS-00212) was exceeded by at least 5 percent. This error message is used for diagnosis of the dynamic look-ahead and is only output if the NC kernel is parameterised accordingly.		
Response	Class	1	None
Solution	Class	1	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current command velocity of the axis.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible velocity of the axis (P-AXIS-00212).	
	%4:	Current value [-]	
		Ratio of current velocity to maximum permissible axis velocity (absolute value).	
	%5:	Limit value[-]	
		Maximum tolerable ratio of actual velocity to maximum permissible velocity.	
Error type	1, Error message from NC program.		

ID 60103

Maximum permissible axis acceleration exceeded.

Description	The maximum axis acceleration set in the axis parameter list (P-AXIS-00008) was exceeded by at least a factor of two. This error message is used for diagnosis of the dynamic look-ahead and is only output if the NC kernel is parameterised accordingly.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [mm/s² or °/s²]	
		Current command acceleration of axis.	
	%3:	Limit value [mm/s² or °/s²]	
		Maximum permissible acceleration of the axis (P-AXIS-00008).	
	%4:	Current value[-]	
		Ratio of current acceleration to maximum permissible acceleration (absolute value).	
	%5:	Limit value [-]	
		Maximum permissible ratio of actual to maximum acceleration.	
Error type	1, Error message from NC program.		

ID 60104

Warning level of axis acceleration exceeded.			
Description	The maximum axis acceleration set in the axis parameter list (P-AXIS-00008) was exceeded by at least 30%. This error message is used for diagnosis of the dynamic look-ahead and is only output if the NC kernel is parameterised accordingly.		
Response	Class	1	None
Solution	Class	1	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [mm/s² or °/s²]	
		Current command acceleration of axis.	
	%3:	Limit value [mm/s² or °/s²]	
		Maximum permissible axis acceleration	
	%4:	Current value[-]	
		Ratio of actual acceleration to maximum permissible acceleration (absolute value).	
	%5:	Limit value [-]	
		Maximum tolerable ratio of actual to maximum acceleration.	
Error type	1, Error message from NC program.		

ID 60105

Maximum permissible axis jerk exceeded.

Description	The maximum permissible axis jerk of the spindle axis was exceeded by at least a factor of 100. It is defined in the axis parameter list via the maximum permissible axis acceleration P-AXIS-00008 and minimum ramp time P-AXIS-00201: Permissible axis jerk = maximum axis acceleration / minimum ramp time This error message is used for diagnosis of the dynamic look-ahead and is only output if the NC kernel is parameterised accordingly.		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Generate the diagnosis file and make this file available to examine the error response.

Parameter	%1:	Logical axis number[-]
		Logical axis number P-AXIS-00016 of the concerned spindle axis.
	%2:	Current value[mm/s^3 or °/s^3]
		Current command jerk of axis.
	%3:	Upper limit value[mm/s^3 or °/s^3]
		Maximum permissible jerk of the axis (P-AXIS-00008, P-AXIS-00201).
	%4:	[-] Current value
		Ratio of current jerk to maximum permissible jerk (absolute value).
	%5:	Upper limit value[-]
		Maximum permissible exceeding factor
Error type	1, Error message from NC program.	

ID 60106

Warning level of axis jerk exceeded.			
Description	The maximum permissible axis jerk of the spindle axis was exceeded by at least a factor of 10. It is defined in the axis parameter list via the maximum permissible axis acceleration P-AXIS-00008 and minimum ramp time P-AXIS-00201: Permissible axis jerk = maximum axis acceleration / minimum ramp time This error message is used for diagnosis of the dynamic look-ahead and is only output if the NC kernel is parameterised accordingly.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Generate the diagnosis file and make this file available to possibly examine the error response.

Parameter	%1:	Logical axis number[-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value[mm/s^3 or °/s^3]
		Current command jerk of axis.
	%3:	Upper limit value[mm/s^3 or °/s^3]
		Maximum permissible jerk of the axis (P-AXIS-00008, P-AXIS-00201).
	%4:	[-] Current value
		Ratio of current jerk to maximum permissible jerk (absolute value).
	%5:	Upper limit value[-]
		Maximum permissible exceeding factor

Error type	1, Error message from NC program.

ID 60107

Function feedforward control is not available in this version.			
Description	In the axis parameter list the feedforward control was activated (P-AXIS-00223) although the feedforward control cannot be activated with the version of the NC kernel used.		
Response	Class	2	
Solution	Class	6	Set parameter (P-AXIS-00223) in the axis parameter list to zero.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	-		

ID 60109 - 60130

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60131

Selection of feedrate in millimetre per revolution by stationary spindle.			
Description	In the NC program, with the spindle stopped, the command G96 (feedrate per revolution) has been programmed.		
Response	Class	6	Abort NC program.
Solution	Class	6	Before programming G96, program M3/M4 in the NC program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program..		

ID 60132

Feedrate in millimetre per revolution and programmed spindle speed is 0.			
Description	Spindle speed zero (nil) was programmed while feedrate per revolution was enabled.		
Response	Class	6	Abort NC program.
Solution	Class	6	Deselection of feedrate in millimetre per revolution before programming spindle speed zero (nil) in the NC-program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60134 - 60136

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 60137

Axis request/release during endless turning.			
Description	An axis exchange command (S[PUTAX]/S[CALLAX].) was programmed while an M3/M4 command was active for the spindle. For further information, see [[PROG// Section: Releasing/requesting spindle axes]		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Stop the spindle before programming axis exchange commands.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60138

Technology function for spindle not permitted, if spindle not present.			

Description	An attempt was made to output a technology function (M/H/S function) on the spindle axis although the spindle axis was previously released due to an axis exchange command.
	
	Example:
	
	`..`
	`N1000 S[PUTAX]`
	`N1100 S100 (Error ! )`
	`N1200 X1000`
	`..`
	
	In the above example program, an error message is issued in line N1100 because the spindle axis was issued by the S[PUTAX] command in line N1000 at the time the S function was programmed.
	See also [PROG// Section: Releasing/requesting spindle axes]
	[FCT-C1// Section: Spindle M functions].		

Response	Class	6	Movement stop., Abort the NC program.Movement stop.
Solution	Class	6	Change the NC program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60139

Timeout from spindle at plc reset.

Description	The PLC has the option of requesting information about a CNC reset via the HLI element ipo_lc_control.plc_reset . The process response is as follows: If the PLC sets the signal enable_w in the ipo_lc_control.plc_reset control unit, the NC kernel sets the signal command_r when there is a reset. As soon as the PLC completes its own reset actions, it must set the state_wr signal to indicate that the PLC reset is completed. If the acknowledgement signal is not set within 20 seconds, this error message is output.		
Response	Class	6	CNC channel changes into error state.
Solution	Class	6	Clear the cause for the delayed PLC acknowledgement and restart the NC kernel.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	- -		

ID 60141

Spindle is configured in too many channels.			

Description	A spindle can only be configured in a certain number of channels at the same time. A spindle is configured in a channel by inserting a section
	
	`spindel[].bezeichnung ...`
	`spindel[].log_achs_nr ...`
	`spindel[].plc_control ...`
	`spindel[].s_synch ...`
	`spindel[].m3_synch ...`
	`spindel[].m4_synch ...`
	`spindel[].m5_synch ...`
	`spindel[].m19_synch ...`
	`spindel[].s_prozess_zeit ...`
	`spindel[].m3_prozess_zeit ...`
	`spindel[].m4_prozess_zeit ...`
	`spindel[].m5_prozess_zeit ...`
	`spindel[].m19_prozess_zeit ...`
	
	in the channel parameter list.
	See also [CHAN//Section: Configuration of spindles].		

Response	Class	2	Spindle changes to error state.

Solution	Class	7	Delete the entries or comments
			
			`spindel[].bezeichnung ...`
			`spindel[].log_achs_nr ...`
			`spindel[].plc_control ...`
			`spindel[].s_synch ...`
			`spindel[].m3_synch ...`
			`spindel[].m4_synch ...`
			`spindel[].m5_synch ...`
			`spindel[].m19_synch ...`
			`spindel[].s_prozess_zeit ...`
			`spindel[].m3_prozess_zeit ...`
			`spindel[].m4_prozess_zeit ...`
			`spindel[].m5_prozess_zeit ...`
			`spindel[].m19_prozess_zeit ...`
			
			in at least one channel parameter list.

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	Current value [-]
		Number of channels in which the spindle is configured.
	%3:	Limit value [-]
		Maximum number of channels the spindle can be configured in.
Error type	-	

ID 60142

Position with unreferenced spindle not allowed.			
Description	For the spindle a movement to an absolute position was programmed although it is not homed.		
Response	Class	6	Axis is stopped and NC-program is aborted.
Solution	Class	6	Homing of spindle before programming an absolute movement.
Parameter	%1:		
			Logical axis number (P-AXIS-00016) of the spindle
Error type	1, Error message from NC program.		

ID 60146

Maximum number of open late sync functions exceeded.			
Description	The maximum permissible number of open technology functions with cross-block synchronisation is exceeded. See also FCT-C1// Sections: Synchronisation types] and [FCT-C1// Section: Example of MVS_SLM]		
Response	Class	7	Stop the axis and abort the program.
Solution	Class	6	Change NC program, reduce number of simultaneously open (not acknowledged) technology functions.
Parameter	%1:		
			Logical axis number (P-AXIS-00016) of the spindle
	%2:		Current value [-]
			Number of open technology functions.
	%3:		Limit value [-]
			Maximum number of simultaneously open technology functions.
Error type	1, Error message from NC program.		

ID 60147

Inadmissible commanding for controlled spindle.

Description	A spindle was configured as externally controlled, see (P-AXIS-00015). In this operation mode the spindle axis is interpolated by the PLC; motion commands from the NC kernel are transferred via the High Level Interface to the PLC and executed by the PLC. See also [FCT-S1// Section: Controlled spindle]. Since the spindle axis is interpolated by the PLC, this spindle type cannot execute any NC commands that require a coordinated motion between the spindle axis and the interpolated axes. The following commands are not permitted: • Axis exchange (S[PUTAX]). • Tapping (G63) • Homing (G74) • C axis machining (#CAX, #FACE, #CYL) • Constant cutting speed (G96) • Spindle path couplings (S[FEED_LINK ...])		
Response	Class	3	Stop the NC program.
Solution	Class	6	Change the NC program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [-]	
		Number of channel that commanded the spindle.	
Error type	1, Error message from NC program.		

ID 60148 - 60151

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60154

Selection of thread cutting with stopped spindle.			
Description	Thread cutting (G33) was programmed while the spindle is stopped. See also [[PROG// Section: Thread cutting].		
Response	Class	6	Abort NC program.
Solution	Class	6	Before programming G33 endless turning, program the spindle (M3/M4).
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60155

Thread cutting and spindle speed 0 programmed.

Description	The spindle has been stopped with a M5 or S0 command while thread cutting (G33) was enabled. See also [PROG//Section: Thread cutting with endlessly rotating spindle].		
Response	Class	6	Stop of NC-program.
Solution	Class	6	Deselect thread cutting before spindle stop. The function can be deselected by programming a motion block before M5.
Parameter	%1:	Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60156

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60157

Job acknowledge FIFO can not be written.			
Description	Commands to the spindle which are transferred by the HLI interface are acknowledged by the spindle after execution. A FIFO memory is used as interface to the PLC. This error message is output if the spindle cannot write a job acknowledge to the PLC in this acknowledgement FIFO because the FIFO is full.		
Response	Class	2	Immediate axis stop.
Solution	Class	6	This error message indicates a PLC problem. Check why the PLC does not read acknowledgements from the FIFO.
Parameter	%1:	Logical axis number [-] Logical axis number of the spindle	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 60158

Positive software limit switch exceeded.			
Description	The positive software limit switch is crossed. Software limit switch monitoring performs a check based on the current position and velocity of the axis, taking into account the braking distance of the software limit switches. This error message is output if the axis crosses the software limit switch. The position of the software limit switch is defined by parameterP-AXIS-00178 and can be changed in the NC program by G99. See also [PROG// Section: Set the positive software limit switch (G99)] [FCT-A2// Section: Programming a positive software limit switch].		
Response	Class	7	Stop the axis, abort the NC program.

Solution	Class	6	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Position of software limit switch	
	%4:	Current value [0.1 µm or 0.0001°]	
		Current axis position + braking distance (=stop position)	
Error type	-		

ID 60159

Negative software limit switch exceeded.			
Description	The negative software limit switch is crossed. Software limit switch monitoring performs a check based on the current position and velocity of the axis, taking into account the braking distance of the software limit switches. This error message is output if the axis crosses the software limit switch. The position of the software limit switch is defined by parameterP-AXIS-00177 and can be changed in the NC program by G98. See also [PROG// Section: Set a negative software limit switch (98)] [FCT-A2// Section: Programming a negative software limit switch].		
Response	Class	7	Stop the axis, abort the NC program.
Solution	Class	6	
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Position of software limit switch	
	%4:	Current value [0.1 µm or 0.0001°]	
		Current axis position + braking distance (=stop position)	
Error type	-		

ID 60160 / 60161

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60162

Auto homing impossible, homing direction not identical with moving direction.

Description	Spindles have the option of so-called automatic homing. Here, as soon as a motion to an absolute position is programmed for the spindle, the spindle is first automatically homed without it being programmed in the NC program by a G74 command. See also [FCT-M1]. The direction of homing is defined by parameter P-AXIS-00158. This error message is output if the direction of spindle movement does not correspond to the one indicated in parameter P-AXIS-00158 at the start of automatic homing For SERCOS drives automatic homing is not possible. This error message is shown in this case, too.		
Response	Class	6	Program abort
Solution	Class	6	Change the NC program so that the spindle moves in the direction defined in P-AXIS-00158 at the start of the homing direction. Automatic homing can be suppressed by setting the bit NO_AUTO_RPF (value 0x0100) in parameter P-AXIS-00015. In this case, the spindle must be explicitly homed by a G74 command before moving to an absolute position. See also [PROG// Section: Homing (G74)].
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 60163

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 60164

Error in table manager function.			
Description	An error occurred while using camming tables.		
Response	Class	5	Abort command/NC program.
Solution	Class	6	Read parameter No.3 and contact service personal.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Identification number [-]	
		ID of CAM table that you want to access.	
	%3:	Current value [-]	
		Error identifier.	
Error type	-		

ID 60165 / 60166

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60167

Commanded order only permitted in the main interpolator, not in SuperImposed interpolator.			
Description	The second interpolator for superimposed motions was commanded with an impermissible NC block.		
Response	Class	3	Abort command/NC program.
Solution	Class	6	Check NC block command.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [-]	
		CNC channel number	
	%3:	Current value [-]	
		NC block type	
Error type	-		

ID 60168 - 60171

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 60172

CamIn: table type does not fit to the selected Cam Interpolation type			
Description	A curve synchronization (block MC_CamIn) was commanded via the PLC, whereby the interpolation rule passed in input table_fkt_type does not match the type of the curve table selected with input CamTableID.		
Response	Class	5	Abort NC program or PLC module.
Solution	Class	6	Changing the interpolation rule in the PLC module MC_CamIn.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Table type	
	%3:	Current value [-]	
		Commanded interpolation type.	
Error type	-		

ID 60173

Phasing not possible because synchronised motion is not enabled.			
Description	Phasing was commanded for an axis (PLC module MC_Phasing) although the slave axis is not currently executing a synchronous motion with another axis.		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Synchronise the affected axis with a master axis before executing the phase shift (PLC blocks MC_CamIn, MC_GearIn).
Parameter	%1:		
	Logical axis number (P-AXIS-00016) of the spindle.		
Error type	1, Error message from NC program.		

ID 60174

Master Phasing not possible because synchronisation is not reached.			
Description	Phasing was commanded for the master axis of a synchronous group although a slave axis of this group has not yet synchronised on the master axis.		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Do not perform a phase shift for the master axis until the synchronising slave axis sets the output InGear or InSync.
Parameter	%1:		
	Logical axis number (P-AXIS-00016) of the spindle.		
Error type	1, Error message from NC program.		

ID 60175

Slave Phasing not possible because synchronisation is not reached.			
Description	A phase shift (PLC module MC_Phasing) was commanded for the slave axis of a synchronous group although the axis was still in the process of synchronising with its master axis.		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Only command phasing after the slave axis has synchronised with the master axis (output InGear or InSync of the slave axis is TRUE).
Parameter	%1:		
	Logical axis number (P-AXIS-00016) of the spindle.		
Error type	1, Error message from NC program.		

ID 60176

Slave slave phasing active, superimposed not possible.			
Description			
Response	Class	5	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 60177

Master phasing enabled, superimposed not possible.			
Description	A superimposed motion (PLC module MC_MoveSuperImposed) was programmed for the master axis of a synchronous group while a phase shift (PLC module MC_Phasing) is executed for this axis.		
Response	Class	5	Abort NC program / PLC block reports an error.
Solution	Class	6	Start the superimposed motion after phasing has finished.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
Error type	1, Error message from NC program.		

ID 60179 / 60180

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60181

Camming is only allowed for modulo axis.			
Description	Camming (PLC block MC_CamIn) was commanded for an axis that is not configured as a modulo axis (see P-AXIS-00015). Camming is only possible for modulo axes.		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Configure the axis for which camming is to be used as the modulo axis (see P-AXIS-00015).
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Configured axis mode (P-AXIS-00015).	
Error type	1, Error message from NC program.		

ID 60182

Gearing is only allowed for modulo axis.			
Description	Gearing (PLC module MC_GearIn) is assigned for an axis that is not configured as a modulo axis (see P-AXIS-00015).. Gearing is only possible with modulo axes.		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Configure the axis for which gearing is to be used as the modulo axis (see P-AXIS-00015).
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Configured axis mode (P-AXIS-00015).	
Error type	1, Error message from NC program.		

ID 60183

CAM/GEAR: the selected master axis is not configured as a master			
Description	The master axis specified in the command of a camming / gearing movement (PLC modules MC_CamIn, MC_GearIn) is not configured as a master axis. The parameter P-AXIS-00288 must be assigned the value 1 to configure an axis as master axis.		
Response	Class	5	Abort NC program / PLC block reports an error.
Solution	Class	7	Set parameter P-AXIS-00288 of the master axis specified in the command (parameter 2 of the error message) to 1.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Logical axis number [-]	
		Axis number (P-AXIS-00016) of the master axis specified in the ordering.	
Error type	-		

ID 60184

Homing during synchronised motion not allowed.			
Description	Homing was commanded for the slave axis of a synchronous motion. Homing can only be commanded if the axis does not participate in a synchronous motion (PLC blocks MC_CamIn/ MC_GearIn).		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Perform homing of slave axis before or after synchronous motion.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
Error type	1, Error message from NC program.		

ID 60185

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 60186

Slave could not follow the master velocity.			
Description	At a cam in /gear in (PLC module MC_CamIn, MC_GearIn), the synchronisation between master and slave was not reached within the time set up in P-AXIS-00291.		
Response	Class	3	PLC module reports an error.
Solution	Class	7	Check the parameters for velocity synchronisation. They are P-AXIS-00289, P-AXIS-00290, P-AXIS-00291. Also check whether the maximum axis velocity P-AXIS-00212 is exceeded.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Maximum time for synchronization (P-AXIS-00291).	

Error type	-

ID 60187

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60188

Acceleration ramp 1 is below limit.			
Description	A movement was commissioned in which the acceleration level 1 is less than 1 mm/s2. The acceleration level 1 value is automatically corrected to 1 mm/s2. The cause for this warning is indicated by the parameters 4 and 5 of this warning: If parameter 4 equals 1, the movement was commissioned by the PLC and the invalid value was transferred during the commissioning. Otherwise either P-AXIS-00011 or P-AXIS-00005 are less than 1 mm/s2. Parameter 5 indicates which of the two values is incorrect.		
Response	Class	1	Warning output, value is corrected to 1 mm/s2.
Solution	Class	1	Change erroneous value in axis parameter list or when commanding via HLI.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [mm/s² or °/s²]	
		Current value.	
	%3:	Corrected value [mm/s² or °/s²]	
		Corrected value	
	%4:	Current value[-]	
		This value indicates if the erroneous value is passed over via the HLI. 1: The value was transferred via the HLI when the order was placed. 0: The invalid value comes from a parameter list. Parameter 5 of the error message indicates the erroneous value.	
	%5:	Current value[-]	
		If parameter 4 is zero (nil), this value indicates which parameter triggered the warning: 0: P-AXIS-00011 is faulty. 1: P-AXIS-00005 is faulty.	
Error type	1, Error message from NC program.		

ID 60189

Acceleration ramp 1 is greater than maximum acceleration.

Description	A motion was commanded where acceleration step 1 is greater than the maximum permissible axis acceleration P-AXIS-00008. The acceleration step 1 value is automatically corrected to P-AXIS-00008 . The warning is triggered by the following parameters depending on the type of command: If parameter 4 equals 1, the motion was commanded by the PLC and the invalid value was transferred with the command. Otherwise either P-AXIS-00011 or P-AXIS-00005 is greater than P-AXIS-00008. Parameter 5 indicates which of the two values is incorrect.		
Response	Class	1	Warning output, value is corrected to P-AXIS-00008.
Solution	Class	1	Change invalid value in axis parameter list or with the command via the HLI.

Parameter	%1:	
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	
		Current value.
	%3:	
		Corrected value
	%4:	
		This value indicates if the invalid value was transferred with the command via the the HLI. 1: The value was transferred with the command via the HLI. 0: The invalid value comes from a parameter list. Parameter 5 indicates the incorrect list parameter.
	%5:	
		If parameter 4 is zero, this value indicates which parameter triggered the warning: 0: P-AXIS-00011 is faulty. 1: P-AXIS-00005 is faulty.
Error type	1, Error message from NC program.	

ID 60190

Acceleration step 2 is below limit.			
Description	A movement was commissioned in which the acceleration step 2 is less than 1 mm/s^2. The acceleration step 1 value is automatically corrected to 1 mm/s^2. The warning is triggered by the following parameters depending on the type of command: If parameter 4 equals 1, the motion was commanded by the PLC and the invalid value was transferred with the command. Otherwise either P-AXIS-00012 or P-AXIS-00006 are less than 1 mm/s^2. In this case parameter 5 indicates which value is erroneous.		
Response	Class	1	Warning output, value is corrected to 1 mm/s^2.
Solution	Class	1	Change invalid value in axis parameter list or with the command via the HLI.

Parameter	%1:	Logical axis number [mm/s² or °/s²]
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	Current value [mm/s² or °/s²]
		Current value.
	%3:	Corrected value [mm/s² or °/s²]
		Corrected value
	%4:	Current value[-]
		This value indicates if the invalid value was transferred with the command via the HLI. 1: The value was transferred with the command via the HLI. 0: The invalid value comes from a parameter list. Parameter 5 indicates the incorrect list parameter.
	%5:	Current value[-]
		This value specifies which parameter triggered the warning: 0: P-AXIS-00012 is faulty. 1: P-AXIS-00006 is faulty.
Error type	1, Error message from NC program.	

ID 60191

Acceleration step 2 is greater than maximum acceleration.			
Description	A motion was commanded where acceleration step 2 is greater than the maximum permissible axis acceleration P-AXIS-00008. The acceleration step 2 value is automatically corrected to P-AXIS-00008 . The warning is triggered by the following parameters depending on the type of command: If parameter 4 equals 1, the motion was commanded by the PLC and the invalid value was transferred with the command. Otherwise either the acceleration P-AXIS-00012 or the current acceleration limit P-AXIS-00006 is greater than the maximum allowable axis acceleration P-AXIS-00008. In this case parameter 5 indicates which value is erroneous.		
Response	Class	1	Warning output, value is corrected to P-AXIS-00008.
Solution	Class	1	Change invalid value in axis parameter list or with the command via the HLI.

Parameter	%1:	
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	Current value [mm/s² or °/s²]
	%3:	Corrected value [mm/s² or °/s²]
	%4:	Current value[-]
	%5:	Current value[-]
		Value indicates whether current limit acceleration causes the warning: 0: Axis acceleration P-AXIS-00012 is faulty 1: Current limit acceleration P-AXIS-00006 is faulty

Error type	1, Error message from NC program.

ID 60192

Deceleration step 1 is below limit.			
Description	A motion was commanded where the deceleration step 1 is less than 1 mm/s^2. The acceleration step 1 value is automatically corrected to 1 mm/s^2 . The warning is triggered by the following parameters depending on the type of command: If parameter 4 equals 1, the motion was commanded by the PLC and the invalid value was transferred with the command. Otherwise either P-AXIS-00282 or P-AXIS-00280 are less than 1 mm/s^2. Parameter 5 indicates which of the two values is incorrect.		
Response	Class	1	Warning output, value is corrected to 1 mm/s^2.
Solution	Class	1	Change invalid value in axis parameter list or with the command via the HLI.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [mm/s² or °/s²]	
		Current value.	
	%3:	Limit value [mm/s² or °/s²]	
		Corrected value	
	%4:	Corrected value [mm/s² or °/s²]	
		This value indicates if the invalid value was transferred with the command via the the HLI. 0: The invalid value comes from a parameter list. Parameter 5 of the error message indicates the invalid value. 1: The value was transferred with the command via the HLI.	
	%5:	Current value[-]	
		If parameter 4 is zero, this value indicates which parameter triggered the warning: 0: P-AXIS-00282 is faulty. 1: P-AXIS-00280 is faulty.	
Error type	1, Error message from NC program.		

ID 60193

Deceleration step 1 is greater than maximum acceleration.			
Description	A motion was commanded where deceleration step 1 is greater than the maximum permissible axis acceleration P-AXIS-00008. The acceleration step 1 value is automatically corrected to P-AXIS-00008 . The warning is triggered by the following parameters depending on the type of command: If parameter 4 equals 1, the motion was commanded by the PLC and the invalid value was transferred with the command. Otherwise either P-AXIS-00282 or P-AXIS-00280 is greater than P-AXIS-00008. Parameter 5 indicates which of the two values is incorrect.		
Response	Class	1	Warning output, value is corrected to P-AXIS-00008.

Solution	Class	1	Change invalid value in axis parameter list or with the command via the HLI.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [mm/s² or °/s²]	
		Current value.	
	%3:	Corrected value [mm/s² or °/s²]	
		Corrected value	
	%4:	Current value[-]	
		This value indicates if the invalid value was transferred with the command via the the HLI. 0: The invalid value comes from a parameter list. Parameter 5 of the error message indicates the invalid value. 1: The value was transferred with the command via the HLI.	
	%5:	Current value[-]	
		If parameter 4 is zero, this value indicates which parameter triggered the warning: 0: P-AXIS-00282 is faulty. 1: P-AXIS-00280 is faulty.	
Error type	1, Error message from NC program.		

ID 60194

Deceleration step 2 is below limit.			
Description	A motion was commanded where the deceleration step 2 is less than 1 mm/s^2. The acceleration level 1 value is automatically corrected to 1 mm/s^2. The warning is triggered by the following parameters depending on the type of command: If parameter 4 equals 1, the motion was commanded by the PLC and the invalid value was transferred with the command. Otherwise either P-AXIS-00283 or P-AXIS-00281 are less than 1 mm/s^2. Parameter 5 indicates which of the two values is incorrect.		
Response	Class	1	Warning output, value is corrected to 1 mm/s^2.
Solution	Class	1	Change invalid value in axis parameter list or with the command via the HLI.

Parameter	%1:	
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	Current value [mm/s² or °/s²]
		Current value.
	%3:	Corrected value [mm/s² or °/s²]
		Corrected value
	%4:	Current value[-]
		This value indicates if the invalid value was transferred with the command via the the HLI. 0: The invalid value comes from a parameter list. Parameter 5 indicates the incorrect list parameter. 1: The value was transferred with the command via the HLI.
	%5:	Current value[-]
		This value specifies which parameter triggered the warning: 0: P-AXIS-00283 is faulty. 1: P-AXIS-00281 is faulty.
Error type	1, Error message from NC program.	

ID 60195

Deceleration step 2 is greater than maximum acceleration.			
Description	A motion was commanded where deceleration step 2 is greater than the maximum permissible axis acceleration P-AXIS-00008. The acceleration step 1 value is automatically corrected to P-AXIS-00008 . The warning is triggered by the following parameters depending on the type of command: If parameter 4 equals 1, the motion was commanded by the PLC and the invalid value was transferred with the command. Otherwise either P-AXIS-00281 or P-AXIS-00283 are greater than the maximum allowable axis acceleration P-AXIS-00008. Parameter 5 indicates which of the two values is incorrect.		
Response	Class	1	Warning output, value is corrected to P-AXIS-00008.
Solution	Class	1	Change invalid value in axis parameter list or with the command via the HLI.

Parameter	%1:	
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	Current value [mm/s² or °/s²]
	%3:	Corrected value [mm/s² or °/s²]
	%4:	Current value[-]
	%5:	Current value[-]
		Logical axis number (P-AXIS-00016) of the spindle
Error type	1, Error message from NC program.	

ID 60196

Changeover velocity greater than maximum feed rate.			
Description	The changeover velocity between acceleration steps 1 and 2 (P-AXIS-00221 or P-AXIS-00211) parameterised in the axis parameter list is greater than the permissible maximum velocity of the axis P-AXIS-00212.		
Response	Class	1	Warning output and value is corrected on P-AXIS-00212.
Solution	Class	1	Change axis parameter list, set P-AXIS-00221 or. P-AXIS-00211 smaller than P-AXIS-00212.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current parametrised value either P-AXIS-00211 or P-AXIS-00221.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Limit value (P-AXIS-00212)	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		Corrected value	
	%5:	Current value [-]	
		This value specifies which parameter triggered the warning: 0: P-AXIS-00221 is faulty. 1: P-AXIS-00211 is faulty.	
Error type	1, Error message from NC program.		

ID 60197

Automatic calculation of coupling factor not possible.			
Description	When activating the spindle interpolator coupling with automatic calculation of the coupling factor (NC command S[FEED_LINK ON ..]), the current path velocity is less than 1*10-12 um/s. See also [PROG// Section: Spindle feed link (FEED_LINK)] This may be caused by the path feed programmed in the NC program or by setting an override/feedhold via the HLI.		
Response	Class	6	Program abort.
Solution	Class	6	Override or change program; at the time of activation of the spindle-path coupling, the path velocity must be higher.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current path velocity.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Permissible minimum value.	
Error type	1, Error message from NC program.		

ID 60198

Spindle speed too small for auto. link factor calculation.

Description	When the spindle-path coupling is activated with automatic calculation of the coupling factor (NC command S[FEED_LINK ON ..]), the current spindle speed is less than $1*10^{-12}$ um/s. See also [PROG// Section: Spindle feed link (FEED_LINK)] This may be caused by the spindle speed programmed in the NC program or by an override or feedhold via the HLI.		
Response	Class	6	Program abort.
Solution	Class	6	Check override or change program; at the time of activation of the spindle-path coupling, the spindle speed must be higher.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current spindle speed.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Permissible minimum value.	
Error type	1, Error message from NC program.		

ID 60199

CAM/GEAR: Interface to master axis not available.			
Description	When MC_GearIn or MC_CamIn was assigned, an interface to the master axis could not be established with the given logical axis number of the master axis.		
Response	Class	5	Abort NC program / PLC block reports an error.
Solution	Class	6	Reset, check commanded logical axis number for the master axis.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Type of channel block.	
	%3:	Logical axis number [-]	
		Logical axis number of the master axis.	
Error type	-		

ID 60200

SuperImposed interpolator is not configured.			
Description	For the axis, MC_MoveSuperImposed, MC_CamIn, or MC_GearIn was assigned a superimposed movement, but the second interpolator required for SuperImposed motions is not configured in the single-axis interpolator.		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Enable the second interpolator by setting P-AXIS-00287 to 0.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
Error type	1, Error message from NC program.		

ID 60201

Axis is not homed yet, axis velocity is limited.

Description	For axes that are not homed, the permissible axis velocity is limited to P-AXIS-00268. And here, the axis was assigned a motion whose axis velocity is greater than P-AXIS-00268.		
Response	Class	1	Warning output and limit the axis velocity to P-AXIS-00268.
Solution	Class	1	Homing the axis before requesting a motion or moving at a velocity less than P-AXIS-00268.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Programmed velocity vb_prog.	
	%3:	Corrected value [-]	
		Limited velocity vb_not_referenced (P-AXIS-00268).	
Error type	1, Error message from NC program.		

ID 60202 - 60205

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60206

Parameters for measure abort can not be written to LR.			
Description	The request FIFO between interpolator and BF LR is full, so the parameter to abort the measurement cannot be transferred.		
Response	Class	6	Abort the job or the NC program.
Solution	Class	6	No solution available because of system internal error.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
Error type	1, Error message from NC program.		

ID 60207

Measure not allowed when already enabled or abort enabled.			
Description	A measuring process (PLC-block MC_TouchProbe) was commanded for this axis although a measuring process or the abort of a measuring process is already active.		
Response	Class	6	PLC module reports an error.
Solution	Class	6	Restart measuring only if preceding command is finished respectively aborted.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Commanded measuring channel.	
Error type	-		

ID 60208

Measure abort can not be retriggered.			
Description	The abort of measuring process (PLC-block MC_AbortTrigger) was commanded while an abort of a measuring process was already active.		
Response	Class	6	PLC module reports an error.
Solution	Class	6	Command abort only when previous abort is complete.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
	%2:	Current value [-]	
		Measuring channel for which abortion was commanded.	
Error type	-		

ID 60209

Axis exchange not allowed while measure is active.			
Description	An axis exchange was programmed while a measuring process is active. See also [[PROG// Section: Releasing/requesting spindle axes (PUTAX/CALLAX)]		
Response	Class	6	Abort NC program.
Solution	Class	6	Abort measuring before axis exchange or start axis exchange after measuring is finished.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle.	
Error type	1, Error message from NC program.		

ID 60210

Accelerations and decelerations are out of limits.			
Description	The parameters P-AXIS-00011 and P-AXIS-00012 or P-AXIS-00005 and P-AXIS-00006 are greater than the limit value P-AXIS-00008.		
Response	Class	1	Warning output, value is corrected to P-AXIS-00008.
Solution	Class	1	Change incorrect value in axis parameter list.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [mm/s² or °/s²]	
		Current parameterised value.	
	%3:	Corrected value [mm/s² or °/s²]	
		Limit value	
	%4:	Current value[-]	
		Corrected value	
	%5:	Current value[-]	
		This value specifies which parameter triggered the warning: 0: P-AXIS-00011/P-AXIS-00012 are faulty. 1: P-AXIS-00005/P-AXIS-00006 are faulty.	
Error type	1, Error message from NC program.		

ID 60211

Resulting programmed velocity exceeds maximum.			
Description	The resulting programmed axis velocity exceeds the maximum axis velocity defined in parameter P-AXIS-00212 Since more than one independent motion can be programmed for a spindle, for example by programming a superimposed motion via the PLC, the resulting velocity may exceed the maximum permissible velocity even if each separate motion is within the maximum velocity.		
Response	Class	1	Warning output and correct the programmed velocity so that the resulting velocity does not exceed the maximum permissible velocity.
Solution	Class	1	Change NC or PLC program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Programmed velocity	
	%3:	Error value [1µm/s or 0.001°/s]	
		Resulting velocity	
	%4:	Error value [1µm/s or 0.001°/s]	
		Maximum permissible axis velocity (P-AXIS-00212)	
	%5:	Limit value [1µm/s or 0.001°/s]	
		Corrected programmed velocity.	
Error type	1, Error message from NC program.		

ID 60212

Velocity for the movement is 0.			
Description	The programmed velocity for the movement is less than the minimal permissible velocity.		
Response	Class	6	Program is aborted
Solution	Class	6	Modify the NC program.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Programmed velocity	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Permissible minimum velocity	
	%4:	Current value [-]	
Error type	1, Error message from NC program.		

ID 60213

Spindle is already in error state when commanding.			
Description	The spindle was commanded although it is already in error state.		
Response	Class	6	Program execution stop
Solution	Class	6	Reset of NC-control before again commanding spindle.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	

Error type	1, Error message from NC program.		

ID 60214

Homing during motion not allowed.			
Description	Homing was commanded for an axis while the axis was moving.		
Response	Class	6	Abort NC program / PLC block reports an error.
Solution	Class	6	Stop axis before homing.
Parameter	%1:		
	Logical axis number (P-AXIS-00016) of the spindle.		
Error type	1, Error message from NC program.		

ID 60215 - 60223

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Warning output.
Solution	Class	8	Requires controller restart.

ID 60224

Filter parameterisation error.			
Description	Parametrisation of filter is not valid.		
Response	Class	3	Warning output.
Solution	Class	7	Correct parameters for filter.
Error type	-		

ID 60225

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Warning output.
Solution	Class	8	Requires controller restart.

ID 60226

Invalid SLOPE type.			
Description	An invalid slope type was commanded for the spindle axis according to the machine data item P-AXIS-00270. The value is corrected automatically to default type. For more information on spindle programming, see [PROG// Section: Spindle programming]		
Response	Class	1	None.
Solution	Class	1	Command the slope type according to the machine parameter P-AXIS-00270 within the permitted range.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value[-]
		Commanded slope type.
	%3:	Corrected value [mm/s² or °/s²]
		Corrected slope type.
Error type	1, Error message from NC program.	

ID 60227

Acceleration is out of limits.			
Description	The commanded acceleration for a non-linear slope (jerk limited) according to parameter P-AXIS-00001 was commanded outside the permissible range. The acceleration is corrected to within the valid range.		
Response	Class	1	None.
Solution	Class	1	Command acceleration in valid range.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of concerned axis.
	%2:		Current value [mm/s² or °/s²]
			Commanded acceleration.
	%3:		Lower limit value [mm/s² or °/s²]
			Minimum permissible acceleration.
	%4:		Upper limit value [mm/s² or °/s²]
			Maximum acceleration P-AXIS-00008.
	%5:		Corrected value [mm/s² or °/s²]
			Corrected acceleration.
Error type	1, Error message from NC program.		

ID 60228

Deceleration is out of limits.			
Description	The deceleration of the non-linear slope (jerk limited) according to parameter P-AXIS-00002 was commanded outside the permitted range. The deceleration is corrected to within the valid range.		
Response	Class	1	None.
Solution	Class	1	Command deceleration in valid range.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of concerned axis.
	%2:		Current value [mm/s² or °/s²]
			Commanded deceleration.
	%3:		Lower limit value [mm/s² or °/s²]
			Minimum permissible deceleration.
	%4:		Upper limit value [mm/s² or °/s²]
			Maximum acceleration/deceleration P-AXIS-00008.
	%5:		Corrected value [mm/s² or °/s²]
			Corrected deceleration.
Error type	1, Error message from NC program.		

ID 60229

Ramp time acceleration up-gradation is out of limits.			
Description	The ramp time for acceleration up-gradation of the non linear slope (jerk limited) according to parameter P-AXIS-00196 was commanded smaller than the minimum value. The ramp time is corrected to minimum value.		
Response	Class	1	None.
Solution	Class	1	Command ramp time greater than minimum value.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [mm/s² or °/s²]	
		Commanded ramp time.	
	%3:	Lower limit value [mm/s² or °/s²]	
		Minimum ramp time value.	
	%4:	Corrected value [µs]	
		Corrected ramp time.	
Error type	1, Error message from NC program.		

ID 60230

Ramp time acceleration down-gradation is out of limits.			
Description	The ramp time for the acceleration down-gradation of the non-linear slope (jerk limited) according to the parameter P-AXIS-00195 was commanded smaller than the minimum value. The ramp time is corrected to minimum value.		
Response	Class	1	None.
Solution	Class	1	Command ramp time greater than minimum value.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [mm/s² or °/s²]	
		Commanded ramp time.	
	%3:	Lower limit value [mm/s² or °/s²]	
		Minimum ramp time value.	
	%4:	Corrected value [µs]	
		Corrected ramp time.	
Error type	1, Error message from NC program..		

ID 60231

Ramp time deceleration up-gradation is out of limits.			
Description	The ramp time for the deceleration up-gradation of the non-linear slope (jerk limited) according to the parameter P-AXIS-00198 was commanded smaller than the minimum value. The ramp time is corrected to minimum value.		
Response	Class	1	None.
Solution	Class	1	Command ramp time greater than minimum value.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [mm/s² or °/s²]
		Commanded ramp time.
	%3:	Lower limit value [mm/s² or °/s²]
		Minimum ramp time value.
	%4:	Corrected value [µs]
		Corrected ramp time.
Error type	1, Error message from NC program.	

ID 60232

Ramp time deceleration down-gradation is out of limits.			
Description	The ramp time for deceleration down-gradation of the non-linear slope (jerk limited) according to the parameter P-AXIS-00197 was commanded smaller than the minimum value. The ramp time is corrected to minimum value.		
Response	Class	1	None.
Solution	Class	1	Command ramp time greater than minimum value.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of axis affected.
	%2:		Current value [mm/s² or °/s²]
			Commanded ramp time.
	%3:		Lower limit value [mm/s² or °/s²]
			Minimum ramp time value.
	%4:		Corrected value [µs]
			Corrected ramp time.
Error type	1, Error message from NC program.		

ID 60234

Not allowed spindle-instruction by active feed coupling.			
Description	A movement (for example M3, M4; M19) was commanded for the spindle although feed coupling is enabled (S[FEED_LINK ...]). See also [[PROG// Section: Spindle feed link (FEED_LINK)].		
Response	Class	1	Program execution stop
Solution	Class	1	Deselect feed coupling before motion command.
Parameter	%1:		
			Logical axis number (P-AXIS-00016) of the spindle
Error type	1, Error message from NC program. .		

ID 60235

Homing to fixed stop: Reference position not detected.			
Description	The homing block was not detected inside the distance, which is defined in the axis specific parameters.		
Response	Class	6	Immediate stop of the axis with emergency deceleration.
Solution	Class	6	Check parameter in axis parameter list.

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	Limit value [0.1 µm or 0.0001°]
		Maximum path to detect homing position.
Error type	1, Error message from NC program.	

ID 60236

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 60237

Sampling time for cut-off frequency/centre frequency too high, cut-off frequency too low.			
Description	According to Shannon's sampling theorem, the maximum cut-off frequency is ($½*T_A$).		
Response	Class	3	No reaction
Solution	Class	7	-
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	-		

ID 60238

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60239

Overflow of data format in motion path.			
Description	The value of the motion path is outside the permissible numerical range. The value must be less than 214748367 (maximum signed 4 byte number).		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Correct motion path and perform NC reset.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
	%2:	Current value of the motion path.	
Error type	1, Error message from NC program.		

ID 60240

Illegal bit combination commanded in CamIn start mode.			
Description	When the PLC commanded a CamIn MC_CamIn), an invalid value for parameter ‘startmode’ was transferred to the PLC block.		
Response	Class	5	PLC block reports an error.
Solution	Class	6	Change the ‘startmode’ call parameter of PLC block MC_CamIn.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Error value [-]	
		Invalid value of ‘startmode’ parameter	
	%3:	Logical axis number [-]	
		Logical axis number of the master axis (P-AXIS-00016)	
Error type	-		

ID 60241

Programmed velocity exceeds speed monitoring limit.			
Description	The programmed velocity exceeds the maximum permissible actual velocity P-AXIS-00311 with velocity monitoring switched on (P-AXIS-00312). If P-AXIS-00311 is zero or is not specified, the limit value for velocity monitoring is set to P-AXIS-00268.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Set the programmed velocity to less than the maximum permissible actual velocity P-AXIS-00311
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value of the programmed velocity.	
	%3:	Error value [1µm/s or 0.001°/s]	
		Incorrect value of the programmed velocity.	
	%4:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible value of the actual velocity P-AXIS-00311.	
Error type	1, Error message from NC program.		

ID 60242

Command acceleration while synchronised in camming or gearing exceeded.			
Description	The permissible command acceleration P-AXIS-00304 during camming or gearing was exceeded.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Adapt command acceleration.
Parameter	%1:		
	%2:	Current value [mm/s² or °/s²]	
		Current value of the command acceleration.	
	%3:	Limit value [mm/s² or °/s²]	
		Limit value of command acceleration P-AXIS-00304.	

Error type	-

ID 60243

Command velocity exceeded during synchronisation with velocity.			
Description	The maximum permissible commanded velocity P-AXIS-00303 was exceeded during synchronisation with velocity.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Adapt command velocity.
Parameter	%1:		
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value of the command velocity.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Limit value of command velocity P-AXIS-00303.	
Error type	-		

ID 60244

Command velocity exceeded while camming.			
Description	The maximum permissible command velocity P-AXIS-00303 was exceeded during camming.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Adapt command velocity.
Parameter	%1:		
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value of the command velocity.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Limit value of command velocity P-AXIS-00303.	
Error type	-		

ID 60245

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 60246

Overflow of data format in virtual master position.			
Description	The virtual master position exceeds permissible numerical range. The position value must be smaller than 9.223.372.036.854.775.807 (maximum signed 8-byte number).		
Response	Class	5	Abrupt axis stop, feedhold for the other axes.

Solution	Class	6	Correct virtual master position and perform NC reset.
Error type	-		

ID 60247

Overflow of data format in virtual master position with gear factor.			
Description	The virtual master position with gear factor compensation is outside the permissible numerical range. The position value must be smaller than 9.223.372.036.854.775.807 (maximum signed 8-byte number).		
Response	Class	5	Abrupt axis stop, feedhold for the other axes.
Solution	Class	6	Correct virtual master position and perform NC reset.
Error type	-		

ID 60248

Overflow of data format in delay time compensated virtual master position.			
Description	The virtual master position with delay time compensation is outside the permissible numerical range. The position value must be smaller than 9.223.372.036.854.775.807 (maximum signed 8-byte number).		
Response	Class	5	Abrupt axis stop, feedhold for the other axes.
Solution	Class	6	Correct virtual master position and perform NC reset.
Error type	-		

ID 60249

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

2.7.2 ID-range 60250-60499

ID 60250

Non-homed axis exceeds maximum permissible velocity.			
Description	When the axis is not homed, the maximum permissible velocity is limited to P-AXIS-00268.		
Response	Class	6	Stopping the axis, and aborting the NC program.
Solution	Class	6	Generally there are 2 possibilities to avoid this message: increase the value of P-AXIS-00268 so that the command velocity for the axis is less than P-AXIS-00268. or command a velocity value for this axis that is smaller than P-AXIS-00268. With both methods, it is essential to know the source of the command velocity value in order to move a non-homed axis. If the message occurs because homing is triggered, refer to the function description [FCT-M1// Homing] which describes which of the parameters P-AXIS-00218 or P-AXIS-00219 is used for the selected homing method. For the different homing modes, either one of the two or both parameters is taken into account and are therefore decisive for the comparison with P-AXIS-00268. If a motion command is triggered, i.e. by NC program code or by a job from the PLC, the feeds specified there are decisive for comparison with -AXIS-00268.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the spindle	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value of the velocity.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible velocity P-AXIS-00268.	
Error type	1, Error message from NC program.		

ID 60251

Camming is only allowed with a referenced slave axis.			
Description	A coupling with a camming table is only possible for a homed slave axis.		
Response	Class	6	Immediate stop of the axis with emergency deceleration
Solution	Class	6	Homing of slave axis
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	-		

ID 60252

Feed enable in slave axis not set.	
Description	Feed enable for slave axis is not set.

Response	Class	4	Immediate stop of the axis with emergency deceleration
Solution	Class	6	Set feed enable
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60253

Positioning of spindle not possible, no actual position value configured.			
Description	A positioning job was commanded for the spindle (M19, homing) although no actual position value was configured for the spindle. Without actual position value, the spindle cannot execute any positioning commands.		
Response	Class	6	Error message output, program abort.
Solution	Class	6	Change NC program or configure a position sensor for the spindle.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
Error type	1, Error message from NC program.		

ID 60254

Direction of move is against to preferential direction.			
Description	The axis is to be moved against its preferred direction P-AXIS-00031 defined in the axis parameter list (P-AXIS-00224).		
Response	Class	6	Stopping the axis, and aborting the NC program.
Solution	Class	6	Change direction of movement according to the preferential direction in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the spindle	
	%2:	Current value [-]	
		Programmed direction: 0: negative direction of rotation 1: positive direction of rotation	
	%3:	Limit value [-]	
		Preferred direction of rotation of the axis (P-AXIS-00031).	
Error type	1, Error message from NC program.		

ID 60255

It is impossible to interpolate an axis, if it is in torque control.			
Description	When the drive is in operation state torque control, the axis can not be interpolated.		
Response	Class	6	Immediate stop of the axis with emergency deceleration
Solution	Class	6	No motion command during torque control
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Current value [-]	
		Current bit coded operation state	

Error type	1, Error message from NC program.

ID 60256

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60257

Jerk is 0, ramp time set to default values from axis parameter list.			
Description	If the jerk is zero, the ramp time is set to default value from MDS.		
Response	Class	1	Warning output. Ramp time is automatically set to the default value in MDS.
Solution	Class	1	-
Parameter	%1:		
	%2:	Current value [µs]	
	Current value for ramp time.		
Error type			

ID 60259

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 60260

Interval length in camming table smaller than permitted.			
Description	A motion segment in a POLY5-LINE or LINE-POLY5 camming table is smaller than 0.1 mm or 0.1°. See also [MCP-P1// Section: Definition of the motion pattern]		
Response	Class	5	Immediate stop of the axis with emergency deceleration
Solution	Class	6	Define a camming table with permitted length of motion segments

Parameter	%1:	
		Logical axis number (P-AXIS-00016) of the spindle
	%2:	Current value [0.1 µm or 0.0001°]
		Absolute position of left interval limit
	%3:	Current value [0.1 µm or 0.0001°]
		Absolute position of right interval limit
	%4:	Limit value [0.1 µm or 0.0001°]
		Minimum of interval length
Error type	-	

ID 60261

Number of lines in camming table not permitted.			
Description	Invalid number of lines in the camming table of type LINE-POLY5 or POLY5-LINE. The table must contain a minimum of 2 lines and a maximum of 360 lines with motion segments. See also MCP-P1// Section: Definition of the motion pattern]		
Response	Class	5	Immediate stop of the axis with emergency deceleration
Solution	Class	6	Definition of a camming table with permitted number of lines
Parameter	%1:		
			Logical axis number (P-AXIS-00016) of the spindle
	%2:		Current value [-]
			Number of lines
	%3:		Limit value [-]
			Minimum number of lines
	%4:		Limit value [-]
			Maximum number of lines
Error type	-		

ID 60262

Spindle axis is exchanged. Action cannot be executed.			
Description	The job sent to the spindle cannot be executed since the axis was replaced from the spindle interpolator. First replace the spindle axis. See also [PROG// Section: Releasing/requesting spindle axes].		
Response	Class	6	Stop axis immediately
Solution	Class	6	If the command was transferred by the PLC to the spindle, this error indicates that the command in the PLC was incorrect or the command was sent to the spindle at the wrong time. Possible reasons are: • Programming error in the PLC. See also [HLI// Section: External spindle command]. • Wrong version of the PLC library used to command the job. • No synchronisation between PLC and CNC.
Parameter	%1:		
			Logical axis number of the spindle axis.
Error type	1, Error message from NC program..		

ID 60263

Camming is only allowed for closed interpolation.			
Description	For a linear axis, one of the invalid start modes HLI_CI_ENDLESS_POSITIVE (0x20) or HLI_CI_ENDLESS_NEGATIVE (0x40) was assigned to MC_CamIn.		
Response	Class	5	Axis is stopped and the command is aborted.
Solution	Class	6	Change start mode of MC_CamIn.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of axis concerned	
	%2:	Current value [-]	
		Axis mode	
	%3:	Current value [-]	
		Start mode of MC_CamIn	
Error type	-		

ID 60264

SLOPE type not allowed for gearing slave.			
Description	For a GearIn coupling (see [MCP-P1// Section: Mc_GearIn]), the linear acceleration profile must be selected in the slave axis (see P-AXIS-00270).		
Response	Class	5	Stop the axis and abort the NC program.
Solution	Class	6	Correct parameters P-AXIS-00270
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Selected slope profile P-AXIS-00270	
	%3:	Expected value [-]	
		Expected slope profile P-AXIS-00270	
Error type	1, Error message from NC program.		

ID 60265

CAM table does not exist.			
Description	The camming table cannot be found in the table management.		
Response	Class	5	Abort command/NC-Program.
Solution	Class	6	Check commanded table ID. Check, if the requested table is entered in the table loader script “tab_ldr.lis”.
Parameter	%1:		
		Logical axis number (P-AXIS-00016) of the spindle	
	%2:	Identification number [-]	
		ID of CAM table that you want to access.	
Error type	-		

ID 60266

CAM table cannot be used in the moment.

Description	The table shall be used while it is accessed by the table manager.		
Response	Class	5	Abort command/NC-Program.
Solution	Class	6	Stop access of table manager to table before usage.
Parameter	%1:		
	Logical axis number (P-AXIS-00016) of the spindle		
	%2:		
	ID of CAM table that you want to access.		
Error type	-		

ID 60267

Axis exchange not allowed while active turning function with block search.			
Description	The spindle is in coupled operation with a path interpolator due to the selected turning functions (e.g. G96, G95). With active block search, these turning functions must be deselected before axis exchange command.		
Response	Class	6	.Motion stop
Solution	Class	6	Turning functions are disabled in the NC program before an axis exchange.
Error type	1, Error message from NC program.		

ID 60268

Spindle in machine lock was commanded from a channel which is not in machine lock.			
Description	A spindle which is currently in machine lock mode was commanded from a channel which is not in machine lock mode.		
Response	Class	3	Abrupt stop of the axis.
Solution	Class	6	If a spindle is commanded by a channel in machine lock mode, it can be only commanded by other channels which are not in machine lock mode if the channel in machine lock mode reaches its program end. Either change the mode of the commanding channel to machine lock mode or run the programs sequentially.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the spindle affected	
	%2:	Current value [-]	
		Channel number of the channel which is not in machine lock mode and which commanded the spindle.	
	%3:	Current value [-]	
		Channel number of the channel which commanded the spindle in machine lock mode.	
Error type	1, Error message from NC program.		

ID 60269

An active spindle not in operating mode machine lock was commanded by a channel in machine lock.	
Description	A spindle which is not in machine lock mode was commanded by a channel which is in machine lock mode. While a spindle is executing a job, it cannot be interrupted by another job which was commanded by a channel in machine lock mode.

Response	Class	3	Abrupt stop of the axis.
Solution	Class	6	Carry out a reset or stop the spindle in order to permit a change in operation mode.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the spindle	
	%2:	Current value [-]	
		Channel number of the channel which is not in machine lock mode and which commanded the spindle.	
Error type	1, Error message from NC program.		

ID 60270

Touch probe cannot be aborted because of error in SAI.			
Description	The command for aborting an active touch probe can not be executed by SAI, because the axis is already in error state.		
Response	Class	6	Axis is stopped immediately.
Solution	Class	6	Reset and repeat of command for aborting an active touch probe.
Error type	1, Error message from NC program.		

ID 60271 / 60272

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 60273

Stability reserve of designed filter not sufficient.			
Description	It is not possible to design a stable filter for all filter parameters, e.g. due to the limited calculation accuracy. Possible solutions are: • Reduce order P-AXIS-00140 of the filter • Reduce the reciprocal bandwidth P-AXIS-00080 of band-pass or band-stop filters • Choose a different filter prototype P-AXIS-00153 • Increase the ratio of cut-off frequency to sampling rate		
Response	Class	3	Filter is deactivated
Solution	Class	7	Change filter parameters

Parameter	%1:	
		Logical axis number P-AXIS-00016 of axis affected
	%2:	
		Number of filter affected
	%3:	
		Filter type P-AXIS-00204
	%4:	
		Filter order P-AXIS-00140
	%5:	
		Filter cut-off frequency P-AXIS-00067 or time constant P-AXIS-00357 of PT1 filter/PT2 filter
Error type	-	

ID 60274

Timeout: Axis filter not within tolerance window.			
Description	During reset, axis position request or Exact stop , the CNC waits for the axis filter to reach the tolerance window P-AXIS-00351. If this fails to occur within a certain period of time (e.g. due to a strongly oscillating filter), the axis filters for the affected axis are temporarily switched off. They are reactivated after a CNC reset. Possible solutions are: • Zoom tolerance window P-AXIS-00351 for filters • Reduce order P-AXIS-00140 of the filter • Reduce the reciprocal bandwidth P-AXIS-00080 of band-pass or band-stop filters • Choose a different filter prototype P-AXIS-00153 • Increase the ratio of cut-off frequency to sampling rate		
Response	Class	6	Filter is temporarily deactivated and reactivated after reset
Solution	Class	7	Change filter parameters or increase tolerance window

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Error value
		True, if axis filters have reached position window
	%3:	Current value [0.1 µm or 0.0001°]
		Unfiltered command value
	%4:	Current value [0.1 µm or 0.0001°]
		Filtered command value
	%5:	Upper limit value [0.1 µm or 0.0001°]
		Tolerance window P-AXIS-00351 for axis filter
Error type	-	

ID 60275

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 60284

Invalid state for request of axis positions.			
Description	The position of the spindle axis normally changes when tracking mode is activated (see [HLI//Section: Control commands of an axis]. Therefore, the spindle interpolator must request the currently valid position from the position controller at the end of tracking operation. During this request, the spindle must not be positioned (M19).		
Response	Class	7	Axis is stopped, the CNC program is aborted
Solution	Class	6	Wait for a few cycles after disabling tracking mode before positioning the spindle axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Current state of spindle interpolator	
	%3:	Current value [-]	
Error type	-		

ID 60285

Gear change during movement is not allowed.			
Description	While the spindle is moving (e.g. endlessly rotating M03/M04), no gear G112 may be changed.		
Response	Class	6	Stop the axis and abort the CNC program.
Solution	Class	6	Stop spindle axis before changing gear (M05) or use spindle-specific gear change functionality via M functions M40-M45 (see [FCT-S1// Section: Gear change])
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 60286

Measuring interface between spindle and interpolator is not available.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:		
	%2:	Current value [-]	
Error type	-		

ID 60287

Measuring interface for the interpolator is already occupied with an old measured value.

Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:		
	%2:	Current value [-]	
Error type	-		

ID 60288 / 60289

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 60290

Tool change with changing gear ratio only permitted at standstill.			
Description	If a tool with a gear stage is replaced, a gear change takes place in the position controller. Therefore, changing a tool with a gear stage is only permitted at standstill due to safety reasons.		
Response	Class	6	Stop axis and abort NC program.
Solution	Class	6	Stop the spindle before changing the tool.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the spindle affected	
	%2:	Current value [-]	
		Numerator gear ratio P-TOOL-00016 of tool	
	%3:	Current value [-]	
		Denominator gear ratio P-TOOL-00017 of tool	
	%4:	Current value [-]	
		Reversal of rotation direction P-TOOL-00018 by tool	
	%5:	Current value [-]	
		Optional parameter for diagnostic purposes	
Error type	1, Error message from NC program.		

ID 60291

Gear change not possible since axis was released.			
Description	The position controller must execute a gear change since a tool with a gear stage is replaced. However, this is not possible since the axis does not exist in the spindle interpolator.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Replace spindle axis before changing tool, see [PROG// Section: Requesting spindle axes.]

Parameter	%1:	
		Logical axis number P-AXIS-00016 of the spindle affected
	%2:	Current value [-]
		Numerator gear ratio P-TOOL-00016 of tool
	%3:	Current value [-]
		Denominator gear ratio P-TOOL-00017 of tool
	%4:	Current value [-]
		Reversal of rotation direction P-TOOL-00018 by tool
Error type	1, Error message from NC program.	

ID 60292

Position controller could not execute gear change request.			
Description	This error message is a follow-up error. The position controller must execute a gear change since a tool with a gear stage is replaced. However, due to an error in the position controller, the gear change could not be executed.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Correct error cause in position controller.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of the spindle affected
	%2:		Current value [-]
			Gear switching performed in the position controller (TRUE / FALSE)
	%3:		Current value [-]
			Numerator gear ratio P-TOOL-00016 of tool
	%4:		Current value [-]
			Denominator gear ratio P-TOOL-00017 of tool
	%5:		Current value [-]
			Reversal of rotation direction P-TOOL-00018 by tool
Error type	1, Error message from NC program.		

ID 60293

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60294

Encoder velocity limit is less than axis maximum velocity.			
Description	The specified encoder velocity limit P-AXIS-00220 is lower than the maximum permissible speed in spindle revolution monitoring.		
Response	Class	1	Axis stop and CNC-program abort
Solution	Class	1	Increase the encoder velocity limit P-AXIS-00220 (if supported by the encoder) or decrease the maximum spindle speed.

Parameter	%1:	
		Logical axis number (P-AXIS-00016) of the concerned spindle axis.
	%2:	Error value [1µm/s or 0.001°/s]
		Invalid encoder velocity limit P-AXIS-00220
	%3:	Lower limit value [1µm/s or 0.001°/s]
		Maximum permissible spindle speed
Error type	1, Error message from NC program.	

ID 60296

Invalid dynamic parameters for spindle.			
Description	Rotating tools can be provided with dynamic limit data in the tool parameters. A tool change (D, T) and the CNC command S[GET_DYNAMIC_DATA]ur*(*)* for activating the tool dynamics limit data define the maximum spindle speed P-TOOL-00014 and maximum acceleration P-TOOL-00015 of the tool in the spindle interpolator. Parameter setting of tool dynamic data is only necessary and effective if the logical axis number of assigned spindle is not 0. In this case, either the dynamic data are incorrectly defined or the values are smaller than permitted minimum value or a command was issued to activate invalid dynamic data. (*)Example for spindle named S		
Response	Class	6	Motion stop of CNC, spindle stop
Solution	Class	6	Correctly assign dynamic data in the tool
Parameter	%1:		
			Logical axis number (P-AXIS-00016) of the spindle axis affected.
	%2:		Current value [-]
			Maximum value for tool speed
	%3:		Lower limit value [-]
			Maximum value for tool acceleration
	%4:		Current value [-]
	%5:		Lower limit value [-]
Error type	1, Error message from NC program.		

ID 60297

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60298

Tool gear change requires standstill because direction of drive rotation changes.

Description	A tool was inserted which changes the direction of rotation (see P-TOOL-00018) and delays the effectiveness of the Reversal of rotation direction (see P-TOOL-00019). Afterwards, a reversal of the direction of spindle rotation must be programmed (M03->M04or M04->M03). Otherwise, the tool gear changes the direction of drive and spindle rotation must be stopped.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Program spindle stop (M05) after tool change.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of the spindle affected	
	%2:	Current value [-]	
		Numerator gear ratio P-TOOL-00016 of new tool	
	%3:	Current value [-]	
		Denominator gear ratio P-TOOL-00017 of new tool	
	%4:	Current value [-]	
		Reversal of rotation direction P-TOOL-00018 of the new tool	
	%5:	Current value [-]	
		Reversal of rotation direction P-TOOL-00018 of the active tool	
Error type	1, Error message from NC program.		

ID 60299

Motion command due to reversal of direction of rotation by tool gear not permitted.			
Description	A tool was inserted which changes the direction of rotation (see P-TOOL-00018) and delays the effectiveness of the Reversal of rotation direction (see P-TOOL-00019). The reversal of rotation of direction can only be considered for endless motions (M03/M04). The spindle must be stopped before other motion jobs, such as spindle positioning M19, are performed so that the gear can be changed in the position controller.		
Response	Class	6	Stop the axis and abort the NC program.
Solution	Class	6	Program spindle stop before motion job (M05).
Parameter	%1:		
		Logical axis number P-AXIS-00016 of the spindle affected	
	%2:	Current value [-]	
		Numerator gear ratio P-TOOL-00016 of tool	
	%3:	Current value [-]	
		Denominator gear ratio P-TOOL-00017 of tool	
	%4:	Current value [-]	
		Reversal of rotation direction P-TOOL-00018 of the tool	
	%5:	Current value [-]	
		Incorrect motion job of spindle	
Error type	1, Error message from NC program.		

ID 60300

Motor or parameter set changeover during endless motion not allowed.			
Description	An attempt was made to carry out a motor or parameter set changeover with the #DRIVE command during an active endless motion of a spindle. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	6	Error message output, abort program.

Solution	Class	6	While endless motion is active, it is not possible to change over a parameter set or the motor; change the NC Program.
Parameter	%1:		
	%2:	Error value [-]	
		Motor number programmed in the #DRIVEcommand. A value of -1 means that the motor number was not programmed in the #DRIVE command.	
	%3:	Error value [-]	
		Parameter set number programmed in the #DRIVE command. A value of –1 indicates that the parameter set number was not programmed in the #DRIVE command.	
Error type	1, Error message from NC program.		

ID 60301

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60302

Drive power off during positioning.			
Description	Drive releases were reset during spindle positioning (M19). As a result, the spindle cannot reach the position window (see P-AXIS-00236).		
Response	Class	2	Abort NC program.
Solution	Class	6	Only remove a drive release after the spindle has reached the position window (see "Axis in position" in [HLI// Section: Status information of an axis])
Parameter	%1:		
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Block number [-]	
		Number of NC block in which the error occurred.	
Error type	-		

ID 60303

Angle offset for thread cutting is outside modulo area.			
Description	For thread cutting G33, an angle offset S.OFFSET was specified greater than the spindle modulo range. The angle offset is only required for multi-start threads. See [PROG// Section: Thread cutting with endlessly rotating spindle] Example: G33 Z48 K1.5 **S.OFFSET=500** Correct: G33 Z48 K1.5 **S.OFFSET=180**		
Response	Class	7	Stop the axis and abort the NC program.

Solution	Class	6	Correct CNC program
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [0.1 µm or 0.0001°]	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Lower modulo limit P-AXIS-00127 of the axis	
	%4:	Limit value [0.1 µm or 0.0001°]	
		Upper modulo limit P-AXIS-00126 of the axis	
Error type	1, Error message from NC program.		

ID 60304

Permitted number of correction cycles for master position exceeded.			
Description	In a gear coupling based on current values (see [MCP-P1/Section: MC_GearIn]), large acceleration jumps may occur on the slave axis, e.g. due to bus errors on the master axis. Therefore, if the acceleration limit a_max_curr is exceeded, the current position of the master axis is extrapolated. This warning is displayed if the extrapolation requires several CNC cycles in sequence.		
Response	Class	1	Warning output, actual position of master axis is no longer extrapolated.
Solution	Class	1	Check the fieldbus.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [mm/s² or °/s²]	
		Calculated acceleration from master actual values.	
	%3:	Upper limit value [mm/s² or °/s²]	
		Limit a_max_corr as of which the actual position of the master axis is extrapolated.	
	%4:	Error value [-]	
		Number of CNC cycles in sequence with correction of master axis position	
	%5:	Upper limit value [-]	
		Maximum permissible CNC cycles	
Error type	-		

ID 60305

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 60306

Denominator equal to zero is not allowed in the gearing resolution of gearing.			
Description	In a gear coupling (see [MCP-P1//Section: MC_GearIn]), the denominator of the gear ratio was specified as 0 (RatioDenominator = 0).		
Response	Class	6	Stop the axis and abort the NC program.

Solution	Class	6	Correct the gear coupling factor
Parameter	%1:		
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Incorrect denominator of the gear ratio.	
Error type	1, Error message from NC program.		

ID 60307 - 60310

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 60311

Given filter time constant is invalid.			
Description	The specified filter time constant P-AXIS-00357 is smaller or larger than permissible. The filter coefficients can therefore not be calculated. Permissible values are (where Tab_{Ab} = cycle time of the NC controller): • For PT1 and PT2 filters: $T_{Ab} \leq$ P-AXIS-00357 ≤ MAX(UNS32) • For time delay filters: 0 ≤ P-AXIS-00357 < 6 * T_{Ab}		
Response	Class	3	The filter is deactivated.
Solution	Class	7	Correct parameter P-AXIS-00357
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:		
		Number of filter affected	
	%3:		
		Filter type P-AXIS-00204	
	%4:		
		Invalid time constant P-AXIS-00357	
	%5:		
		Maximum permissible time constant P-AXIS-00357	
Error type	-		

ID 60312

Programmed spindle speed is outside the permissible speed range of the tool.	
Description	When axis parameter P-AXIS-00474 is set, it checks during spindle movements whether the speed planned for the movement is within the range defined by tool parameters P-TOOL-00013 and P-TOOL-00014. If the spindle speed lies outside the valid range, the spindle movement is not executed and this error message is output. When external tool management is used (P-CHAN-00016 =1), the minimum and maximum speeds for the tool must be supplied by the external tool manager.

Response	Class	6	Error message output, NC program abort
Solution	Class	6	Change the spindle speed or change P-TOOL-00013/P-TOOL-00014.
Parameter	%1:		
		Axis number of the spindle.	
	%2:	Lower limit value [1µm/s or 0.001°/s]	
		Permissible minimum speed	
	%3:	Current value [1µm/s or 0.001°/s]	
		Commanded spindle speed	
	%4:	Upper limit value [1µm/s or 0.001°/s]	
		Permissible maximum speed	
Error type	1, Error message from NC program.		

ID 60313

Homing of axis not possible since homing is disabled in axis parameters.			
Description	The axis cannot be homed since homing is disabled (see axis parameter homing_type (P-AXIS-00299).		
Response	Class	5	Error message output, abort NC program
Solution	Class	6	Correct the NC program. If the axis does not have an absolute measurement system, change parameter P-AXIS-00299
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis	
	%3:	Current value [-]	
		Current value of the parameter P-AXIS-00299	
Error type	1, Error message from NC program.		

ID 60314

Axis not at standstill! Updating filter parameters not possible. Changes are discarded!			
Description	A list update or the #MACHINE DATA command changed parameters of the axis filter (see [FCT-A7// Section: Parameter]). However, this is only permitted when the axis is stationary.		
Response	Class	3	New parameters are not adopted
Solution	Class	7	Repeat the parameter update when the axis is at standstill
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Index of the filter affected	
	%3:	Current value [-]	
		TRUE : Axis filter has reached the target position	
	%4:	Current value [-]	
		TRUE : Command values are output to the axis, i.e. the axis moves	
Error type	-		

NOTICE
The changed filter parameters have no effect in the controller. The changes are discarded.

ID 60315

Selection of default dynamic data not allowed while axis is moved.			
Description	The S[DEFAULT_DYNAMIC_DATA] function was programmed for the spindle although the spindle is moving. Switching back to the default dynamic data is only possible at standstill. See [PROG// Section: Adopting dynamic tool data]		
Response	Class	6	Movement stop.
Solution	Class	6	Change program flow. Program command S[DEFAULT_DYNAMIC_DATA] when spindle is at standstill.
Parameter	%1:		
		Log. Axis number of spindle	
Error type	1, Error message from NC program.		

ID 60316

NC-command not allowed while axis is moved.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:		
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 60317

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60319

Master axis reports error during active coupling.			
Description	This error message was preceded by an error on the master spindle of an active MC_GearIn or MC_CamIn coupling. The axis parameter P-AXIS-00565 determines whether an error occurs on the master axis during active MC_GearIn or MC_CamIn coupling, the slave axes may also be stopped.		
Response	Class	6	Movement stop.
Solution	Class	6	Check the error cause of the master axis.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of slave spindle.
	%2:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of the master spindle that caused the error
Error type	1, Error message from NC program.	

ID 60320

Slave axis reports error during active coupling.			
Description	This error message was preceded by an error on the slave spindle of an active MC_GearIn or MC_CamIn coupling. The axis parameter P-AXIS-00564 determines whether an error occurs on a slave axis during active MC_GearIn or MC_CamIn coupling, the master axes can also be stopped.		
Response	Class	6	Movement stop.
Solution	Class	6	Check the error cause of the slave spindle.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of the master spindle.
	%2:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of the slave spindle that caused the error
Error type	1, Error message from NC program.		

ID 60321

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires controller restart.

ID 60322

Invalid minimum speed in dynamic data for spindle.			
Description			
Response	Class	6	
Solution	Class	6	
Parameter	%1:		
	%2:		Current value [-]
Error type	1, Error message from NC program.		

ID 60323

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 60324

Activation of G63 only possible at spindle standstill or synchronous motion.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:		
	%2:	Error value [-]	
	%3	Current value [-]	
Error type	-		

ID 60325 - 60327

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 60328

Invalid gear resolution for tool change.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:		
	%2:	Current value [1µm/s or 0.001°/s]	
	%3	Current value [-]	
Error type	1, Error message from NC program.		

ID 60329 - 60331

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 60332

Invalid minimum speed in dynamic data for spindle.			
Description	Rotating tools can be provided with dynamic limit data in the tool parameters. A tool change (D, T) and the CNC command S[GET_DYNAMIC_DATA]ur*(*)* for activating the tool dynamics limit data define the maximum spindle speed P-TOOL-00014 and maximum acceleration P-TOOL-00015 of the tool in the spindle interpolator. The minimum spindle speed P-TOOL-00013 is additionally set for endless turning. Parameter setting of tool dynamic data is only necessary and effective if the logical axis number of assigned spindle is not 0. In the present case, the permitted minimum value of P-TOOL-00013 is negative. This is not permitted. (*)Example for spindle named S		
Response	Class	1	Warning is output and value is corrected.
Solution	Class	1	Correctly assign P-TOOL-00013 in the tool
Parameter	%1:	Logical axis number	
		Logical axis number (P-AXIS-00016) of the spindle axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Value set for P-TOOL-00013	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		P-TOOL-00014	
	%4:	Lower limit value [1µm/s or 0.001°/s]	
	%5:	Corrected value [1µm/s or 0.001°/s]	
Error type	1, Error message from NC program.		

ID 60333

Invalid maximum speed in dynamic data for spindle.			
Description	Rotating tools can be provided with dynamic limit data in the tool parameters. A tool change (D, T) and the CNC command S[GET_DYNAMIC_DATA]ur*(*)* for activating the tool dynamics limit data define the maximum spindle speed P-TOOL-00014 and maximum acceleration P-TOOL-00015 of the tool in the spindle interpolator. In the present case, the value setting for P-TOOL-00014 is too large. (*)Example for spindle named S		
Response	Class	1	Warning is output and value is corrected.
Solution	Class	1	Correctly assign P-TOOL-00014 in the tool

Parameter	%1:	Logical axis number
		Logical axis number (P-AXIS-00016) of the spindle axis affected.
	%2:	Current value [1µm/s or 0.001°/s]
		Value set for
	%3:	Upper limit value [1µm/s or 0.001°/s]
		P-TOOL-00014
	%4:	Corrected value[1µm/s or 0.001°/s]
Error type	1, Error message from NC program.	

ID 60334

Invalid maximum acceleration in dynamic data for spindle.			
Description	Rotating tools can be provided with dynamic limit data in the tool parameters. A tool change (D, T) and the CNC command S[GET_DYNAMIC_DATA]ur*(*)* for activating the tool dynamics limit data define the maximum spindle speed P-TOOL-00014 and maximum acceleration P-TOOL-00015 of the tool in the spindle interpolator. In the present case, the value setting for P-TOOL-00015 is too large (*)Example for spindle named S		
Response	Class	1	Warning is output and value is corrected.
Solution	Class	1	Correctly assign P-TOOL-00015 in the tool
Parameter	%1:	Logical axis number	
		Logical axis number (P-AXIS-00016) of the spindle axis affected.	
	%2:	Current value [mm/s² or °/s²]	
		Set value for P-TOOL-00015	
	%3:	Upper limit value [mm/s² or °/s²]	
	%4:	Corrected value[mm/s² or °/s²]	
Error type	1, Error message from NC program.		

ID 60335

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 60336

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

2.8 Position controller error (ID-range 70000-79999)

2.8.1 ID-range 70000-70249

ID 70000

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 70001

Unknown drive type.			
Description	The specified type of drive is unknown.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct drive type, see P-AXIS-00020
Parameter	%1:	Current value [-]	
		Unknown drive type see P-AXIS-00020	
	%2:	Current value [-]	
		Logical axis number of affected axis see P-AXIS-00043	
Error type	-		

ID 70002 / 70007

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70008

Number of axis group exceeds range or is 0.			
Description	The specified axis group number is out of valid value range.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct the specified axis group number.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-CHAN-00035	
	%2:	Current value [-]	
		Invalid axis group number	
	%3:	Upper limit value [-]	
		Maximum number of axis groups, see P-CHAN-00023	

Error type	-

ID 70009

Axis number exists already.			
Description	The given logical axis number is already used for a different axis or spindle.		
Response	Class	3	Abort job processing
Solution	Class	6	Select a different logical axis number.
Parameter	%1:	Current value [-]	
		Already existing logical axis number, see P-CHAN-00035	
Error type	-		

ID 70010

Number of axis in group is 0 or greater than permissible maximum.			
Description	The specified axis number exceeds the number of existing axes ANZ_ACHSEN_IN _GR in the axis group or is equal to 0.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct axis number
Parameter	%1:	Current value [-]	
		Invalid axis number	
	%2:	Upper limit value [-]	
		Number of axes in axis group see P-CHAN-00003	
Error type	-		

ID 70011

Axis index already used within group.

Description	Two or more axes with the same index exist in the axis group. Example (extract from channel parameters list): gruppe[0].achse[0].bezeichnung X Axis1 ... gruppe[0].achse[0].bezeichnung A ... gruppe[0].achse[1].bezeichnung W1 Solution: gruppe[0].achse[0].bezeichnung X Axis1 ... gruppe[0].achse[1].bezeichnung A ... gruppe[0].achse[2].bezeichnung W1		
Response	Class	3	Abort job processing
Solution	Class	6	Indexing the axes in the axis group.
Parameter	%1:	Current value [-]	
		Index of the axis	
	%2:	Current value [-]	
		Number of the axis group	
	%3:	Current value [-]	
		Logical axis number of other axis see P-CHAN-00035	
Error type	-		

ID 70012

No axis specified in machine data.			
Description	No axes were specified, see P-STUP-00014 and P-STUP-00015.		
Response	Class	3	Abort job processing
Solution	Class	6	Configure axes.
Error type	-		

ID 70013

Number of axis groups not coherent.			
Description	The specified numbers of axis groups are not coherent.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct the numbers of axis groups.
Parameter	%1:	Current value [-]	
		Expected axis group number	
Error type	-		

ID 70014

Number of axis not coherent within group.			
Description	The specified axis numbers are not contiguous within the axis group. Example (extract from channel parameters list): gruppe[0].achse[0].bezeichnung X AXIS1 ... gruppe[0].achse[2].bezeichnung A ... gruppe[0].achse[3].bezeichnung W1 ... Solution: gruppe[0].achse[0].bezeichnung X AXIS1 ... gruppe[0].achse[1].bezeichnung A ... gruppe[0].achse[2].bezeichnung W1 ...		
Response	Class	3	Abort job processing
Solution	Class	6	Correct the given axis numbers.
Parameter	%1:	Current value [-]	
		Number of affected axis group	
	%2:	Expected value [-]	
		Expected axis number	
Error type	-		

ID 70018

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70020

Dynamic position lag exceeds maximum.

Description	When the axis was moved, the position lag exceeded a permissible limit for the current motion velocity. Dynamic tracking/following error monitoring ([FCT-A1// Section: Description]) calculates the permissible position lag as a function of motion velocity when an axis is moved. It is configured individually in the lists for axis parameters (P-AXIS-00167, P-AXIS-00168, P-AXIS-00169, P-AXIS-00170, P-AXIS-00172, P-AXIS-00176 and P-AXIS-00191). The NC controller provides two different methods to calculate the maximum permissible position lag: Type 1: Standard methods – Use a parameterisable filter Type 2: Linear method – is based on a theoretical position lag Possible causes of an excessive position lag may be, for example: • Accelerations specified too large for the axis • The occurrence of friction in the actuating system • The axis is blocked and cannot move freely • Poorly set position controller (for gain factor see P-AXIS-00099) • Drive type SERCOS: If different proportional gain values are used in the drives and the axis parameters. Possible solutions: • Reduce the command accelerations • Ensure free movement of the axis • Check setting (gain factor Kv, see P-AXIS-00099) of position controller • If necessary, change the parameterisation of the dynamic tracking/position lag monitoring ([FCT-A1// Section: Axis parameters])		
Response	Class	8	Controlled halt of the axis, the control loop is open.
Solution	Class	6	Check position controller, freedom of movement; minor accelerations
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [0.1 µm or 0.0001°]	
		Current position lag	
	%4:	Upper limit value[0.1 µm or 0.0001°]	
		Maximum allowed position lag	
	%5:	Block number [-]	
		Block number of the NC program where the error occurred	
Error type	11, Error message from position controller.		

ID 70021

Positive software limit switch actuated.

Description	The current actual position of the axis is greater than the position of the positive software limit switch. See also FCT-A2// Section: Description] The positions of the software limit switches are used to monitor the command and actual position of an axis. They are specified for each axis in the axis parameters (see P-AXIS-00177, P-AXIS-00178). In order to monitor actual positions, an additional tolerance (see P-AXIS-00179) can be specified to prevent a slight overshoot of an axis from triggering an error message. In the NC program, the positions of the software limit switches can be set or changed by the NC commands G98/G99. Possible causes: • Incorrect NC program. • Overshoot of axis is too great. Possible solutions: • Correct the NC program • Check the proportional gain factor (see P-AXIS-00099) of the position controller and reduce it if necessary. • Increase tolerance for position controller (see P-AXIS-00179)		
Response	Class	-	Closed-loop controlled axis stop.
Solution	Class	-	Correct the NC program, reduce overshoot of the axis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [0.1 10^-3 mm or ø]	
		Current position of the axis	
	%4:	Lower limit value [0.1 10^-3 mm or ø]	
		Position of the positive software limit switch; see P-AXIS-00178	
	%5:	Block number [-]	
		Block number of the NC program where the error occurred.	
Error type	-		

ID 70022

Negative software limit switch actuated.

Description	The current actual position of the axis is smaller than the position of the negative software limit switch. See also FCT-A2// Section: Description] The positions of the software limit switches are used to monitor the command and actual position of an axis. They are specified for each axis in the axis parameters (see P-AXIS-00177, P-AXIS-00178). In order to monitor actual positions, an additional tolerance (see P-AXIS-00179) can be specified to prevent a slight overshoot of an axis from triggering an error message. In the NC program, the positions of the software limit switches can be set or changed by the NC commands G98/G99. Possible causes: • An incorrect NC program • Overshoot of axis is too great. Possible solutions: • Correct the NC program • Check the proportional gain factor (see P-AXIS-00099) of the position controller and reduce it if necessary. • Increase tolerance for position controller (see P-AXIS-00179)		
Response	Class	-	Closed-loop controlled axis stop.
Solution	Class	-	Correct the NC program, reduce overshoot of the axis
Parameter	%1:		
		Logical axis number of affected axis see P-AXIS-00016	
	%2:		
		Drive type of concerned axis, see P-AXIS-00020	
	%3:		
		Current position of the axis	
	%4:		
		Position of the negative software limit switch, see P-AXIS-00178	
	%5:		
		Block number of the NC program where the error occurred.	
Error type	-		

ID 70023

Sign error: trend monitoring actuated.

Description	Trend monitoring detected a sign error. It is used for monitoring the sign of the control and encoder signals and can be activated for each axis via the axis parameters (see P-AXIS-00189). Possible error causes: • The positive or negative motion directions of the command and actual values do not match. • The manipulated variable has the wrong sign. The axis moves in the wrong direction when the NC commands G0/G1 are used. Possible solutions: • If the motion directions of the command and actual variables do not match, change the sign in the measurement system or in the drive amplifier. Alternatively, it is possible to specify a sign reversal of the actual position in the axis parameters (see P-AXIS-00230) • If the manipulated variable has the wrong sign, change it either in the drive or in the axis parameters (see P-AXIS-00231).		
Response	Class	5	Abrupt axis stop for defective axis, feedhold for other axes
Solution	Class	6	Check the motion directions
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		Number of occurred plausibility errors	
	%4:	Upper limit value [-]	
		Number of plausibility errors allowed	
	%5:	Block number [-]	
		Block number of the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70024

Error in SERCOS drive. Check S-0-0011 and S-0-0095.			
Description	An error has occurred in the SERCOS drive. Check the ident numbers S-0-0011 and S-0-0095 to obtain information about the error that occurred, and the drive status. Consult the drive manufacturer documentation for further error diagnosis.		
Response	Class	7	Controlled or abrupt stop of the axis
Solution	Class	6	Consult drive manufacturer documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [-]	
		SERCOS drive status	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred.	
	%5:	Error code of drive [-]	

Error type	11, Error message from position controller.

ID 70026 - 70029

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70030

Error during homing of SERCOS drive.			
Description	During homing (see [FCT-M1]), an error occurred with a SERCOS drive. Possible cause: Some SERCOS drives require a minimum distance between the position marker pulse and the position marker in order to complete drive-controlled homing successfully. Otherwise the position marker pulse cannot be detected precisely. Solution: • Consult the documentation of the drive manufacturer for further error diagnosis. • Check the ID number S-0-0298 whether the position marker pulse must be moved and by what distance.		
Response	Class	5	Abrupt stop for the defective axis, feedhold for the other axes.
Solution	Class	6	Consult drive manufacturer documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		SERCOS drive status	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70031

Error occurred during homing of SERCOS drive.			
Description	An error not originating in the drive occurred when homing SERCOS drives. For example, the cause may be an error in the SERCOS driver or in the drive bus.		
Response	Class	5	Abrupt axis stop
Solution	Class	6	

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	Block number [-]
		Block number of the NC program where the error occurred.
Error type	11, Error message from position controller.	

ID 70032

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70033

Error in the SERCOS drive during initialisation of the measured traverse.			
Description	An error in the SERCOS drive occurred during the initialisation of the measurement run. Consult the documentation of the drive manufacturer for a further error diagnosis.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Consult drive manufacturer's documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [-]	
	%4:	Block number [-]	
		Block number of the NC program, in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70034

No continued motion possible without homing.

Description	Unable to continue motion without homing. Cause: The limit (see P-AXIS-00220 at which the measurement system delivers faulty signals can be exceeded at higher speeds, especially with spindles. The position controller must therefore be switched to controlled operation. Therefore, start a new homing process to obtain valid actual positions again after a velocity reduction in the position controller. Solution: Start a new homing process for the axis		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Start a homing process
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70035

Error occurred in SERCOS drive during measurement run.			
Description	An error has occurred in the SERCOS drive during a measurement run. Consult the drive manufacturer documentation for further error diagnosis.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Consult drive manufacturer documentation
Parameter	%1:	Current value	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Block number [-]	
		Block number of the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70036 / 70038

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70039

Error occurred resetting the SERCOS drive.			
Description	An error occurred during reset of a SERCOS drive, the cause of which is, in all probability, the drive. Consult the documentation of the drive manufacturer for a further error diagnosis.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Consult drive manufacturer's documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Block number [-]	
		Block number of the NC program, in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70040

Error within drive could not be debugged by reset.			
Description	The error in the drive could not be rectified by a reset. Consult the documentation of the drive manufacturer for a further error diagnosis.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Consult drive manufacturer's documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Block number [-]	
		Block number of the NC program, in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70041 - 70048

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70049

Missing controller release.			
Description	The axis is to be moved although controller release is not set.		
Response	Class	5	Abrupt or controlled stop of the axis affected
Solution	Class	6	Set the controller release
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	

Error type	11, Error message from position controller.

ID 70050 - 70080

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70081

Position lag at standstill too great.			
Description	While the axis is at standstill, the position lag is exceeded. The position lag monitoring function (see [FCT-A1// Section: Description]) monitors whether the current position lag of an axis exceeds a limit. It is configured individually in the lists for axis parameters (P-AXIS-00167, P-AXIS-00168, P-AXIS-00169, P-AXIS-00170, P-AXIS-00172, P-AXIS-00176 and P-AXIS-00191). There are two different methods to monitor position lag during axis standstill: Type 1: Standard methods – Use a parameterisable filter Type 2: Linear method – is based on a theoretical position lag Possible causes of an excessive position lag may be, for example: • External force acting to move the axis from the stationary position. • If the gain factor set in the position controller is too small (see P-AXIS-00099), the permanent position lag is too large. Therefore, the axis fails to reach the specified command position.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Possible solutions: • Remove the influence of the external force • Increase the proportional gain factor Kv (see P-AXIS-00099) of position controller • If necessary, change the parameter setting of the position lag monitoring function ([FCT-A1// Section: Axis parameters])

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	Current value [0.1 µm or 0.0001°]
		Current position lag
	%4:	Upper limit value [0.1 µm or 0.0001°]
		Maximum allowed position lag
	%5:	Block number [-]
		Block number of the NC program where the error occurred
Error type	11, Error message from position controller.	

ID 70082

Axis not within position window at end of settling time.			
Description	The axis is not within the tolerance window at the end of the settling time. The position lag monitoring function (see [FCT-A3// Section: Description]) monitors whether the actual position reaches an exact stop window (see P-AXIS-00236) within a certain period of time (see P-AXIS-00532) when approaching a target position. If the interpolator no longer supplies any command values, the command position of the axis has reached its programmed target position and time monitoring is started. It is deactivated when the actual position of the axis is within the exact stop window. Possible error causes: • An incorrectly adjusted position controller, e.g. excessive permanent control deviation or excessive overshoot. • Excessive friction in the mechanical system. • The axis is blocked and cannot move freely		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Possible solutions: • Increase the proportional gain factor of the position controller if the control deviation is excessive. • Reduce the gain factor of the position controller if the axis oscillation is excessive (see P-AXIS-00099). • Reduce friction in the mechanical system • If necessary, increase the exact stop window (see P-AXIS-00236) or the settling time (see P-AXIS-00532)
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Upper limit value [µs]	
		Maximum permitted settling time see P-AXIS-00532	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred	
	%5:	Current value [-]	
		Amount of the actual position lag in increments	
Error type	11, Error message from position controller.		

ID 70083

Missing table for leadscrew error compensation.			
Description	Missing reference of table for spindle pitch error correction, although its use is required in the axis parameters (see P-AXIS-00175).		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	A reference to the corresponding compensation value list must be made in the start-up list (P-STUP-00016, P-STUP-00017, P-STUP-00036).
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 70084

Plausibility error in leadscrew error compensation on direction reversal.			
Description	Plausibility error on direction reversal in leadscrew error compensation. See also [FCT-C5// Section: Leadscrew error compensation]		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Block number [-]	
		Block number of the NC program where the error occurred	
	%4:	Current value [-]	
		Command value	
	%5:	Current value [-]	
Error type	11, Error message from position controller.		

ID 70085

Fatal error in CAN drive.			
Description	A fatal error occurred in the CAN drive.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [-]	
		Status of the CAN drive	
	%4:	Block number [-]	
		Block number of the NC program, in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70086

Axis activation not set by PLC.			
Description	The PLC controller has not set the control bit "Axis on". Solution: Check in the PLC program why the flag ‘axis on’ was not set.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Set the control bit “Axis on”
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	

Error type	11, Error message from position controller.

ID 70087

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70088

Range of set point exceeded.			
Description	The value for the manipulated variable currently calculated by the position controller exceeds the permissible value range. Possible causes: • Excessive gain factor in the position controller • Incorrect normalisation of the manipulated variable • Position lag of the axis is too large Possible solutions: • Check manipulated variable scaling (see P-AXIS-00128, P-AXIS-00129) • Reduce gain factor (see P-AXIS-00099) of position controller (or increase gain in drive)		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020[-]
	%3:	Current value [-][-]
		Current value of the manipulated variable
	%4:	Upper limit value [-]
		Maximum permissible value of the manipulated variable
	%5:	Block number [-][-][-][-][-]
		Block number of the NC program where the error occurred
Error type	11, Error message from position controller.	

ID 70090

Position deviation between master and slave axis exceeds permissible range.

Description	An error has occurred in gantry mode. The position lag between the master and slave axis is greater than the permissible path difference which can still be corrected by resetting the controller. • Possible causes: • High friction in the mechanical system • The axis is blocked and cannot move freely Possible solutions: • Reduce friction in the mechanical system • Remove the obstacle • • See also FCT-C11// Section: Overview]		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Reset the numerical control after solving the problem.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current position deviation	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
		For maximum permissible position lag, see P-AXIS-00072	
	%3:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%4:	Current value [0.1 µm or 0.0001°]	
		Actual position of slave axis	
	%5:	Current value [-]	
		Actual position of master axis	
Error type	11, Error message from position controller.		

ID 70091

Position deviation after tracking operation too big.			
Description	After switching off tracking mode, the position deviation occurring is greater than the limit specified in the axis parameters (see P-AXIS-00056). When tracking mode is active, the axis can be moved out of its command position. This leads to a difference between the original command position and the current position. If tracking mode is activated while a program is executed or manual control is active, the axis moves back to the original position after tracking modeends, provided the position lag occurring is smaller than the value specified in the axis parameters (see P-AXIS-00056). Otherwise, this error message is displayed.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	Block number [-]
		Block number of the NC program where the error occurred
	%4:	Current value [0.1 µm or 0.0001°]
		Current position deviation
	%5:	Upper limit value [0.1 µm or 0.0001°]
		Maximum permissible position deviation
Error type	11, Error message from position controller.	

ID 70092

Collision of two axes detected.			
Description	Collision monitoring function (see [FCT-C3]) detected that the distance between two axes is less than the specified minimum distance (see P-AXIS-00045). The axes were stopped in order to prevent a collision. Possible solutions: • Check the parameterisation of the axes • Correct NC program		
Response	Class	7	Controlled stop of concerned axes
Solution	Class	6	Correct the NC program, check axis parameterisation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of first axis see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Logical axis number of second axis P-AXIS-00016	
	%3:	Current value [0.1 µm or 0.0001°]	
		Current position distance of the axes	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Minimum permissible position distance, see P-AXIS-00045	
	%5:	Current value [0.1 µm or 0.0001°]	
		Command position of the second axis in the coordinate system of the first axis	
Error type	11, Error message from position controller.		

ID 70093

No collision partner found.			
Description	The second axis (collision partner) specified for this axis in the parameter setting of collision monitoring (see [FCT-C3]) does not exist. Solution: • Correct the logical axis number (see P-AXIS-00043) in the axis parameters or • parameterise the missing axis.		
Response	Class	5	Abrupt axis stop of affected axis
Solution	Class	6	Correct the given logical axis number

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis see P-AXIS-00016
	%2:	Expected value [-]
		Specified logical axis number of collision partner see P-AXIS-00043
Error type	-	

ID 70094 - 70098

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70099

Difference between command and feedback positions after controller release too big.			
Description	The difference between the new command position value and the old actual value is greater than the permissible limit value when releasing the drive controller (see P-AXIS-00108). When the controller release is set, the axis is given a new command value. If this command value differs from the actual value, an abrupt stimulation of the drive occurs. In order to limit the resulting axis motion, the control deviation with respect to a limit value (see P-AXIS-00108) is monitored. If the current difference between command and actual positions is bigger than the specified value when releasing the drive, the position difference is not extended.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	
Parameter	%1:	Upper limit value [-]	
		Maximum permissible position difference, see P-AXIS-00108	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current position difference.	
	%3:	Current value [0.1 µm or 0.0001°]	
		Command position	
	%4:	Current value [0.1 µm or 0.0001°]	
		Actual position	
	%5:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
Error type	11, Error message from position controller.		

ID 70100

Difference between positions of master and slave axis too big. No reset possible.

Description	An error has occurred in gantry mode. The position lag between the master and slave axes is greater than the permissible path difference which can only be removed by resetting the controller. Possible causes: • High friction in the mechanical system • The axis is blocked and cannot move freely Possible solutions: • Reduce friction in the mechanical system • Remove the obstacle • • See also FCT-C11// Section: Overview]		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current position deviation	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
		For maximum permissible position lag, see P-AXIS-00071	
	%3:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%4:	Current value [0.1 µm or 0.0001°]	
		Actual position of slave axis	
	%5:	Current value [0.1 µm or 0.0001°]	
		Actual position of master axis	
Error type	11, Error message from position controller.		

ID 70101

Drive shall be moved, but it is not enabled yet.			
Description	The drive must be moved, although the drive status indicates that the drive is not ready for interpolation. Possible reasons are: • -Missing enable signal on drive controller • -Power for drive not switched on. • -Actuated hardware limit switches or other lock mechanisms.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Set necessary conditions for drive to move.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [-]	
		Drive status	
	%4:	Block number [-]	
		Block number of the NC program, in which the error occurred	

Error type	11, Error message from position controller.

ID 70103

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	8	Requires restart of controller.Requires controller restart.

ID 70104

Error in PROFIDRIVE		
Description	An error has occurred in the PROFIDRIVE drive. A further error diagnosis can be done with the diagnosis tools of the drive manufacturer and the displayed information at the drive amplifier.	
Response	Class	7 Controlled stop of concerned axis
Solution	Class	6 Perform an error diagnosis at the drive
Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of concerned axis, see P-AXIS-00020
	%3:	Current value [-]
		Status of the PROFIDRIVE (Status word 1)
	%4:	Block number [-]
		Block number of the NC program, in which the error occurred.
Error type	11, Error message from position controller.	

ID 70105

Error in position sensor of the PROFIDRIVE drive.		
Description	An encoder error has occurred in a PROVIDRIVE drive. Further error diagnosis can be carried out with the aid of the diagnostic tools of the drive manufacturer. Check the functionality of the drive bus.	
Response	Class	5 Abrupt axis stop
Solution	Class	6 Perform an error diagnosis at the sensor
Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	Current value [-]
		Reported error code
	%4:	Block number [-]
		Block number of the NC program where the error occurred.
	%5:	Current value [-]

Error type	11, Error message from position controller.

ID 70106

Life sign from PROFIBUS slave missing			
Description	The number of permissible life sign failures of the PROFIBUS slave was exceeded. A further error diagnosis can be done with the diagnosis tools of the drive manufacturer and the displayed information at the drive amplifier. Possible causes: • Slave failure • BUS failure • Error in drive amplifier		
Response	Class	8	Abrupt axis stop
Solution	Class	6	Perform an error diagnosis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [-]	
		Current value of the counter for the diagnosis of missing PROFIBUS slave life signs	
	%4:	Upper limit value [-]	
		Maximum permissible counter value	
Error type	11, Error message from position controller.		

ID 70107 / 70108

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70111

Illegal device name for counter interface in axis machine data.			
Description	The name of the device for the counter interface is unknown.		
Response	Class	3	Job processing aborted.
Solution	Class	7	Check configuration setting in axis machine data record
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 70112

Illegal device name for D/A interface in axis machine data.			
Description	The name of the device for the digital/analogue interface is unknown.		
Response	Class	3	Job processing aborted.
Solution	Class	7	Check configuration setting in axis machine data record
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 70113

Illegal device name for cam in axis machine data.			
Description	The name of the device for the CAM interface (e.g. reference cam) is unknown.		
Response	Class	3	Job processing aborted.
Solution	Class	7	Check configuration setting P-AXIS-00037 in axis machine data record
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 70114

Axis machine data for counter not permissible.			
Description	Configuration data to access the counter interface invalid.		
Response	Class	3	Abort job processing.
Solution	Class	7	Check configuration settings in axis machine data record.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 70115

Axis machine data for D/A interface not permissible.			
Description	Configuration data to access the digital/analogue interface invalid.		
Response	Class	3	Job processing aborted.
Solution	Class	7	Check configuration settings in axis machine data record.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 70116

Axis machine data for cam not permissible.			
Description	Configuration data to access the cam interface (e.g. reference cam) invalid.		
Response	Class	3	Job processing aborted.
Solution	Class	7	Check configuration settings in axis machine data record.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
Error type	-	

ID 70118 / 70119

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70120

Collision partner reports axis error.			
Description	The affected axis was stopped, since its collision partner reports an error in the SERCOS-drive. Both axes are monitored as collision pair by the collision monitoring system. If an error occurs in the SERCOS drive of an axis whose cause is, for example, in the measuring system, the associated collision axis can also be stopped in the other NC channel to ensure that the error does not lead to an axis collision. This function is activated via parameter P-AXIS-00044 in the axis parameters. See also [FCT-C3// Section: Overview]		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Correct the drive error if the axis is faulty
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Logical axis number of incorrect collision partner see P-AXIS-00016	
Error type	11, Error message from position controller.		

ID 70121

Timeout while resetting an axis.			
Description	A timeout occurred when resetting the axis since the confirmation from the drive is still outstanding for a command abort executed. The cause of the error may be in the drive, the driver or the bus. Possible solutions: • Re-attempt an axis reset • Restart the controller		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Re-attempt or restart the NC

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	State [-]
	%4:	Current value [-]
Error type	11, Error message from position controller.	

ID 70123

No slave axis for gantry master configured.			
Description	No slave axis is configured in the axis parameters for the specified gantry master axis (P-AXIS-00070, P-AXIS-00015). See also FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	7	Configure a gantry slave axis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
Error type	-		

ID 70124

Defined master axis not configured as gantry master.			
Description	The master axis for gantry mode specified in the axis parameters is not configured as a master axis. Solution: Parameterise axis as gantry master with logical axis number (P-AXIS-00015) See also FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	7	Set the correct operation mode for the master axis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of gantry slave axis, see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Logical axis number of specified master axis, see P-AXIS-00016	
Error type	-		

ID 70125

The configured master axis for the gantry slave axis was not found.			
Description	The master axis specified for the gantry slave was not found since the logical axis number (P-AXIS-00070) specified in the axis parameters does not exist for the master axis. See also FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	7	Select a different master axis or configure it

Parameter	%1:	Logical axis number [-]
		Logical axis number of the slave axis affected P-AXIS-00016
	%2:	Logical axis number [-]
		Specified logical axis number of the master axis P-AXIS-00016
Error type	-	

ID 70126

Axis is similarly configured as gantry master and slave.			
Description	Both Gantry Master and Gantry Slave were selected for the axis as the operation mode (see P-AXIS-00015) in the axis parameters. See also FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	7	Correct the operation mode of the axis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
Error type	-		

ID 70127

SERCOS drive shall be moved, but communication phase is less than 4.			
Description	The SERCOS drive is to be moved although the communication phase is less than four. Possible causes: • A manual phase switch was executed • Short interruption of the data transmission e.g. kink in the optical fibre • A drive error e.g. no supply voltage Possible solutions: Wait until communication phase 4 is reached.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Wait until communication phase 4 is reached
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		Current communication phase	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred	
	%5:	Current value [* 10^-4 Incr]	
		Relative axis command value in high resolution (parameter is not included in all versions).	
Error type	11, Error message from position controller.		

ID 70151

No controller release for moving out the gantry difference.			
Description	The controller release by the PLC controller is not set for this axis. Therefore, the path difference between the master axis and this axis cannot be moved out for gantry operation. See also FCT-C11// Section: Overview].		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Set controller release
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the slave axis affected, see P-AXIS-00016	
	%2:	Current value[0.1 µm or 0.0001°]	
	%3:	Current value[0.1 µm or 0.0001°]	
	%4:	Current value[0.1 µm or 0.0001°]	
	%5:	Current value [-]	
Error type	-		

ID 70152

'Axis on' command to move out gantry difference not set.			
Description	"Axis on" for this axis is not set by the PLC controller. Therefore, the path difference between the master axis and this axis cannot be moved out for gantry operation. See also FCT-C11// Section: Overview].		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Set 'Axis on'
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the slave axis affected, see P-AXIS-00016	
	%2:	Current value[0.1 µm or 0.0001°]	
	%3:	Current value[0.1 µm or 0.0001°]	
	%4:	Current value[0.1 µm or 0.0001°]	
	%5:	Current value [-]	
Error type	-		

ID 70158

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 70159

Error in digital drive cannot set back by reset.			
Description	The error in the drive could not be rectified by a reset. Consult the documentation of the drive manufacturer for a further error diagnosis.		
Response	Class	-	Abrupt axis stop for defective axis
Solution	Class	-	Consult drive manufacturer's documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
Error type	-		

ID 70160

Configured SERCOS-ID not in use by the CNC.			
Description	The reference to a SERCOS ID number used in the axis parameters is unknown to the CNC controller. A list of all references used in the controller for the SERCOS ID numbers can be found in (see P-AXIS-00131). Example: antr.sercos.at[0].ident_nr 51 antr.sercos.at[0].ident_len 4 antr.sercos.at[0].nc_ref LAGEIST_W**ER**T Correct: antr.sercos.at[0].ident_nr 51 antr.sercos.at[0].ident_len 4 antr.sercos.at[0].nc_ref LAGEIST_W**ER**T		
Response	Class	1	Warning
Solution	Class	1	Specify a valid reference
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Error value [-]	
		Unknown SERCOS reference see P-AXIS-00131	
	%3:	Current value [-]	
		Index of unknown reference	
Error type	-		

ID 70161

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70162

Collision offsets different.	
Description	In the axis parameters, two axes are configured mutually as collision partners. Different minimum permissible position distances (see P-AXIS-00045) for collision detection are specified for both axes. In this case the collision monitoring function (s. @@[FCT-C3]) uses the larger distance. Example (from axis parameters): 1. Axis: kopf.achs_nr 1 achs_mode 0x8001 kenngr.coll_check_ax_nr 2 **kenngr.coll_offset 200000** 2. Axis: kopf.achs_nr 2 achs_mode 0x8001 kenngr.coll_check_ax_nr 1 **kenngr.coll_offset 300000** Correct: 1. Axis: kopf.achs_nr 1 achs_mode 0x8001 kenngr.coll_check_ax_nr 2 **kenngr.coll_offset 300000** 2. Axis: kopf.achs_nr 2 achs_mode 0x8001 kenngr.coll_check_ax_nr 1 **kenngr.coll_offset 300000** See also [FCT-C3// Section: Overview]

Response	Class	1	Warning
Solution	Class	1	Correct specified position distances see P-AXIS-00045

Parameter	%1:	Logical axis number [-]
		Logical axis number of first axis see P-AXIS-00016
	%2:	Current value [-]
		Minimum permissible position distance see P-AXIS-00045
	%3:	Logical axis number [-]
		Logical axis number of second axis see P-AXIS-00016
	%4:	Current value [-]
		Minimum permissible position distance see P-AXIS-00045
	%5:	Corrected value [-]
		Minimum permitted position distance used in collision monitoring function
Error type	-	

ID 70164

Own axis number is defined as collision axis.			

Description	Specify the current axis number (P-AXIS-00043) as the collision partner of this axis in the axis parameters.
	
	Example (from axis parameters):
	`kopf.achs_nr 1`
	`kenngr.coll_check_ax_nr 1`
	
	Correct::
	`kopf.achs_nr 1`
	
	kenngr.coll_check_ax_nr 2
	
	See also [FCT-C3// Section: Overview]		

Response	Class	5	Abrupt axis stop
Solution	Class	6	Specify a different collision axis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
Error type	-		

ID 70165

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70166

The axis-specific collision monitoring was switched on/off.

Description	The collision monitoring function was either switched on or off by changing the operation mode of the axis (see P-AXIS-00015), e.g. by updating the axis parameters. Turning the collision monitoring function on or off requires a controller restart. See also FCT-C3// Section: Overview]		
Response	Class	5	Abrupt axis stop
Solution	Class	6	Restart the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Current axis operation mode, see P-AXIS-00015	
	%3:	Current value [-]	
		Previous axis operation mode, see P-AXIS-00015	
Error type	-		

ID 70167

Number of collision axis changed.			
Description	For the collision monitoring function, a different axis was selected since the parameter P-AXIS-00043 for the logical axis number of the collision partner was changed in the axis parameters. Changes in the configuration of the collision monitoring function require a controller restart. See also FCT-C3// Section: Overview]		
Response	Class	5	Abrupt axis stop
Solution	Class	6	Restart the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Current logical axis number of collision partner see P-AXIS-00043	
	%3:	Current value [-]	
		Previous logical axis number of collision partner, see P-AXIS-00043	
Error type	-		

ID 70168

Error in Lightbus drive.			
Description	An error has occurred in the Lightbus drive. Further error diagnosis can be carried out with the aid of the diagnostic tools of the drive manufacturer.		
Response	Class	7	Controlled stop of concerned axis
Solution	Class	6	Perform an error diagnosis at the drive
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [-]	
		State of drive (state byte 3)	
Error type	-		

ID 70169

No release for the drive amplifier.			
Description	A Lightbus drive is to be be moved although release of the drive amplifier is not set.		
Response	Class	8	Open-loop controlled stop of axis affected
Solution	Class	6	Activate the switch for the drive amplifier release
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		State of drive (state byte 3)	
Error type	11, Error message from position controller.		

ID 70170

Drive is not initialized.			
Description	The drive must be moved, although it is not initialised. Probably an error in the drive occurred.		
Response	Class	8	Open-loop controlled stop of concerned axis
Solution	Class	6	Perform an error diagnosis at the drive
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [-]	
		State of drive (state byte 3)	
Error type	11, Error message from position controller.		

ID 70171

Axis is to be moved although feed release is missing.			
Description	The interpolation of the axis is to be started although feed release is not set by the PLC.		
Response	Class	5	Abrupt axis stop for defective axis
Solution	Class	6	Set ‘feed release’
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
Error type	11, Error message from position controller.		

ID 70172

Logical interpolator axis cannot be connect with the physical position-controller axis.			
Description	Multiple instancing of an axis allows the same drive to be commanded from different NC channels (one after the other) without axis exchange. Parameter P-AXIS-00101 can be used to assign a physical position controller axis to a logical interpolator axis. However, the axis in the position controller could not be found since the given logical axis number in this parameter does not exist.		
Response	Class	7	Open-loop controlled stop of concerned axis
Solution	Class	6	Correct parameter P-AXIS-00101

Parameter	%1:	Logical axis number [-]
		Logical axis number of interpolator axis see P-AXIS-00016
	%2:	Expected value [-]
		Logical axis number of physical axis in position controller that was not found see P-AXIS-00016
Error type	11, Error message from position controller.	

ID 70174 - 70176

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70177

Axis number for recovery function not available.			
Description	With the help of the recovery function, the current state of the controller (e.g. axis positions) can be stored on a permanent data carrier before the end so that the state can be reloaded when the controller is restarted. However, the data that was read in could not be assigned to an axis since the axis no longer exists in the current controller configuration.		
Response	Class	7	Open-loop controlled stop of axis affected
Solution	Class	6	Discard recovery data or restore old axis configuration
Parameter	%1:	Logical axis number [-]	
		Non existing logical axis number, see P-AXIS-00016	
Error type	-		

ID 70178

Recovery position greater than positive limit switch.			
Description	With the help of the recovery function, the current state of the controller (e.g. axis positions) can be stored on a permanent data carrier before the end so that the state can be reloaded when the controller is restarted. However, the current position of the axis that was read in is greater than the positive software limit switch (see P-AXIS-00178) since its position was changed.		
Response	Class	7	Open-loop controlled stop of axis affected
Solution	Class	6	Discard recovery data or change the position of the positive software limit switch (see P-AXIS-00178)
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [0.1 µm or 0.0001°]	
		Current position	
	%4:	Upper limit value [0.1 µm or 0.0001°]	
		Position of the positive software limit switch, see P-AXIS-00178	

Error type	-

ID 70179

Recovery position smaller than negative limit switch.			
Description	With the help of the recovery function, the current status of the controller (e.g. the axis positions) can be stored on a permanent data carrier before it is closed so that it can be reloaded when it is restarted. However, the current position of the axis read in is smaller than the negative software limit switch (see P-AXIS-00177) since its position was changed.		
Response	Class	7	Open-loop controlled stop of axis affected
Solution	Class	6	Discard recovery data or change the position of the negative software limit switch (see P-AXIS-00177)
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [0.1 µm or 0.0001°]	
		Current position	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Position of the negative software limit switch, see P-AXIS-00177	
Error type	-		

ID 70180 / 70181

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70182

Invalid logical axis number for plane compensation.			
Description	The logical axis number of the first or second master axis for plane compensation is invalid, since its value is specified as zero or does not exist. Axes whose actual positions influence the compensation value are named master axes. The function is switched off for the specified axis (see P-AXIS-00174). See also [FCT-C5// Section: Plane compensation]		
Response	Class	7	Closed-loop controlled axis stop. Plane compensation is turned off for the affected axis.
Solution	Class	7	Correct logical axis numbers of master axes, see P-COMP-00014 and P-COMP-00015

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Logical axis number [-]
		Logical axis number of first master axis, see P-COMP-00014
	%3:	Logical axis number [-]
		Logical axis number of second master axis, see P-COMP-00015
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70183

Compensation table does not exist for plane compensation.			
Description	No compensation value list was specified for plane compensation for the axis. The function is switched off for the specified axis (see P-AXIS-00174). See also [FCT-C5// Section: Plane compensation]		
Response	Class	3	Closed-loop controlled axis stop. Plane compensation is turned off for the affected axis.
Solution	Class	7	Specify compensation value list, see [COMP//Section: Place compensation], P-COMP-00016
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
Error type	-		

ID 70184

Invalid master axis type for plane compensation.			
Description	The specified master axis for plane compensation has an invalid axis type (see P-AXIS-00018). Axes whose actual positions influence the compensation value are named master axes. Master and slave axes for plane compensation must be linear axes. See also [FCT-C5// Section: Plane compensation]		
Response	Class	7	Closed-loop controlled axis stop. Plane compensation is turned off for the affected axis.
Solution	Class	7	Axis type of master axis P-AXIS-00018. See also [COMP//Section: Plane compensation]
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected master axis, see P-AXIS-00016, P-COMP-00014, P-COMP-00015	
	%2:	Expected value [-]	
		Expected axis type of master axis, see P-AXIS-00018	
	%3:	Current value [-]	
		Specified axis type of master axis, see P-AXIS-00018	
Error type	11, Error message from position controller.		

ID 70185

Invalid slave axis type for plane compensation.			
Description	The specified slave axis for plane compensation has an invalid axis type (see P-AXIS-00018). The axis that which is compensated is called a slave axis. Master- and slave axes for plane compensation must be translatory axes. See also FCT-C5// Section: Plane compensation]		
Response	Class	7	Closed-loop controlled axis stop. Plane compensation is turned off for the affected axis.
Solution	Class	7	Correct axis type of the master axis, see P-AXIS-00018 See also [COMP// Section: Plane compensation]
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the affected slave axis, see P-AXIS-00016	
	%2:	Expected value [-]	
		Expected axis type of slave axis, see P-AXIS-00018	
	%3:	Current value [-]	
		Specified axis type of slave axis, see P-AXIS-00018	
Error type	11, Error message from position controller.		

ID 70186

External command speed without axis in tracking mode.			
Description	An external command velocity is specified for the axis even though it is not in tracking mode.		
Response	Class	1	Warning
Solution	Class	1	Set tracking mode for the axis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Block number [-]	
		Block number of the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70187

External command speed for SERCOS-drives not implemented.			
Description	The numerical control is presently not supporting an external command velocity presetting for SERCOS-drives.		
Response	Class	1	Warning
Solution	Class	1	
Parameter	%1:	Current value [1µm/s or 0.001°/s]	
		External command velocity in µm/s	
	%2:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%3:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%4:	Block number [-]	
		Block number of the NC program, in which the error occurred.	

Error type	11, Error message from position controller.

ID 70190

Error in DSE-drive while evaluation of drive state.			
Description	Error within DSE-driver while evaluating the drive state.		
Response	Class	7	Immediate stop of the axis.
Solution	Class	6	Check drive status. Do reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Drive type [-]	
		Drive type of the axis P-AXIS-00020.	
	%3:	Block number [-]	
		NC block number or job number.	
Error type	11, Error message from position controller.		

ID 70191

Axes have been moved. Recovery not possible.			
Description	With the help of the recovery function, the current status of the controller (e.g. the axis positions) can be stored on a permanent data carrier before it is closed so that it can be reloaded when it is restarted. Since the axes were moved in the meantime, the recovery data can no longer be used. For example, the stored positions differ from the actual axes positions.		
Response	Class	7	Controlled stop of axes affected
Solution	Class	6	Discard recovery data or do not move the axes
Error type	-		

ID 70192

SLOPE error at interpolation to selection position after end of tracking mode.			
Description	During tracking mode, it is possible to move the axis manually. To continue an interrupted NC program without position offset, the axis must be moved back to its initial position (see P-AXIS-00258). Therefore, the controller moves the axis at the specified velocity (see P-AXIS-00208) back to the old position if the difference between the actual positions at the beginning and the end of tracking mode is smaller than the specified limit (see P-AXIS-00056). However, an error occurred during interpolation of the command value curve. Possible cause is a very small position difference, for example.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Abort the program and reset the controller

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Block number [-]
		Block number of the NC program where the error occurred.
	%3:	Current value [0.1 µm or 0.0001°]
	%4:	Current value [0.1 µm or 0.0001°]
	%5:	Current value [0.1 µm or 0.0001°]
Error type	11, Error message from position controller.	

ID 70193

Encoder terminal EL/KL5001 reports error.			
Description	An SSI encoder terminal EL/KL5001 reports an error. The terminal indicates an error in its status bits. Assign the error bits to the error type as follows:		
	Bit number	Bitmask	Meaning
	0	0x01	Data error (error at SSI input)
	1	0x02	Frame error (invalid data frame received by the sensor)
	2	0x04	Power failure (encoder-specific error bit is set)
	For more information, refer to the terminal and encoder documentation.		
Response	Class	2	Error message output abort program.
Solution	Class	3	Possible reasons for this error message are: • Terminal has parameter setting error • Wiring error • Error in encoder connected to the terminal Solution: Change the terminal parameter settings; correct wiring error or error in encoder.

Parameter	%1:	Current value [-]
		Read-in value of the status byte
	%2:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of affected axis
Error type	11, Error message from position controller.	

ID 70194

Setting reference position while active NC program not possible.			
Description	The reference position cannot be adopted for the axis because either an NC program is executed or the axis is in manual mode.		
Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Abort the execution of the NC program or deselect manual mode before setting reference position.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
Error type	11, Error message from position controller.	

ID 70195

Distance to software limit switch too small for current command position.			
Description	The distance to the software limit switch is smaller than the braking distance required for the specified command velocity. Therefore the axis was stopped in order to prevent the software limit switch from being crossed.		
Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Reduce command velocity
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value[-]	
		Current command velocity	
	%3:	Limit value [-]	
		Current command position	
	%4:	Current value[-]	
		Latest possible braking point	
	%5:	Current value [0.1 µm or 0.0001°]	
		Position of positive or negative software limit switch, see P-AXIS-00178, P-AXIS-00177	
Error type	11, Error message from position controller.		

ID 70196

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70197

Unknown coupling mode for axis coupling.	
Description	An unknown coupling mode was specified for an axis coupling. Valid coupling modes are: 0: HLI_AXIS_COUPLING_INACTIVE: Coupling is not active. 1: HLI_AXIS_COUPLING_ZERO: The coupling factor is zero. 2: HLI_AXIS_COUPLING_DIRECT: Coupling factor is 1. 3: HLI_AXIS_COUPLING_MIRROR: Coupling factor is -1. 4: HLI_AXIS_COUPLING_FRACT: Coupling factor is a fraction. [FCT-A9// Section: Overview] , [HLI// Section: Commanding axis couplings]

Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Correct coupling mode
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Unknown coupling mode	
	%3:	Error value [-]	
		Index of invalid coupling specifications on the HLI starting with 1.	
Error type	-		

ID 70198

Target axis for axis coupling not found.			
Description	Could not find the target axis of the axis coupling since the specified logical axis number does not exist. [FCT-A9// Section: Overview] , [HLI// Section: Commanding axis couplings]		
Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Correct the specified logical axis number
Parameter	%1:	Logical axis number [-]	
		Specified logical axis number of target axis, see P-AXIS-00016	
Error type	-		

ID 70199

Source axis for axis coupling not found.			
Description	Could not find the source axis of the axis coupling since the specified logical axis number does not exist. [FCT-A9// Section: Overview] , [HLI// Section: Commanding axis couplings]		
Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Correct the specified logical axis number of the source axis
Parameter	%1:	Logical axis number [-]	
		Logical axis number of target axis, see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Specified logical axis number of source axis, see P-AXIS-00016	
	%3:	Logical axis number [-]	
		Number of axis coupling	
Error type	-		

ID 70200

Activation of axis coupling for moved axes not possible.

Description	The axis coupling cannot be activated since at least one axis has moved. [FCT-A9// Section: Overview] , [HLI// Section: Commanding axis couplings]		
Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Stop axes before activating the axis coupling
Parameter	%1:	Logical axis number [-]	
		Logical axis number of target axis, see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Logical axis number of moved axis, see P-AXIS-00016	
	%3:	Current value[0.1 10^-7 mm or ø]	
		Relative path of the affected axis still to be covered.	
	%4:	Upper limit value [0.1 10^-7 mm or ø]	
		Maximum permitted relative axis path.	
Error type	-		

ID 70201

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70202

Modulo settings of master and slave axis not identical in gantry mode.			
Description	The modulo settings in the axis parameters are not identical for the gantry master and the slave axis. See also FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	7	Use the same modulo limits for both axes (see P-AXIS-00126 for upper and P-AXIS-00127 for lower modulo limit)
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the slave axis, see P-AXIS-00016	
	%2:	Current value [0.1 µm or 0.0001°]	
		Upper modulo limit of slave axis, see P-AXIS-00126	
	%3:	Logical axis number [-]	
		Logical axis number of the master axis, see P-AXIS-00016	
	%4:	Current value [0.1 µm or 0.0001°]	
		Upper modulo limit of master axis, see P-AXIS-00126	
Error type	-		

ID 70204

Reset is locked since maximum permissible master/slave path distance was previously exceeded.

Description	A controller reset is not possible because the maximum permissible path difference (see P-AXIS-00071) between the master and slave axis was exceeded previously in gantry mode. Resetting the control cannot solve this error.		
Response	Class	5	
Solution	Class	6	Restart the controller
Parameter	%1:	Current value [1µm/s or 0.001°/s]	
		Current path difference between master and slave	
	%2:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible path difference, see P-AXIS-00071	
	%3:	Logical axis number [-]	
		Logical axis number of the slave axis, see P-AXIS-00016	
	%4:	Current value [1µm/s or 0.001°/s]	
		Current position of the slave axis	
	%5:	Current value [1µm/s or 0.001°/s]	
		Current position of the master axis	
Error type	-		

ID 70205

Maximum acceleration (a_max) of master-/slave axis is different.			
Description	The given maximum allowable accelerations for the gantry master and slave axis are not identical. [FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	1	Use the same maximum acceleration (see P-AXIS-00008) for both axes.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the slave axis P-AXIS-00016	
	%2:	Error value [mm/s² or °/s²]	
		Permissible maximum acceleration of slave axis see P-AXIS-00008	
	%3:	Logical axis number [-]	
		Logical axis number of the master axis see P-AXIS-00016	
	%4:	Current value [mm/s² or °/s²]	
		Permissible maximum acceleration of master axis see P-AXIS-00008	
	%5:	Corrected value [mm/s² or °/s²]	
		Correct value for the maximum axis acceleration	
Error type	-		

ID 70206

Emergency acceleration (a_emergency) of master-/slave axis is different.			
Description	The given decelerations for an emergency stop for the gantry master and slave axis are not identical. See also [FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	1	Use the same deceleration for an emergency stop (s. @@P-AXIS-00003) for both axes.

Parameter	%1:	Logical axis number [-]
		Logical axis number of the slave axis P-AXIS-00016
	%2:	Error value [mm/s² or °/s²]
		Deceleration for emergency stop of slave axis see P-AXIS-00003
	%3:	Logical axis number [-]
		Logical axis number of the master axis see P-AXIS-00016
	%4:	Current value [mm/s² or °/s²]
		Deceleration for emergency stop of master axis see P-AXIS-00003
	%5:	Corrected value [mm/s² or °/s²]
		Correct value for the deceleration
Error type	-	

ID 70207

Error reaction (cnc_controlled_stop_after_error) of master/ slave axis is different.			
Description	The axis parameters contain different error reactions for the gantry master and slave axis. It is possible to choose between drive-internal and CNC-controlled error reaction (see P-AXIS-00254). For gantry mode, the same setting must be used for the master and slave axes. See also FCT-C11// Section: Overview]		
Response	Class	1	Warning
Solution	Class	1	Use the same error reaction (see P-AXIS-00254) for both axes.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the slave axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Error reaction of the slave axis, see P-AXIS-00254	
	%3:	Corrected value [-]	
		Expected value for error reaction, see P-AXIS-00254	
Error type	-		

ID 70208

Maximum acceleration too small (smaller than 1 increment per cycle).			
Description	The maximum permissible axis acceleration is set too low in the axis parameters (see P-AXIS-00008). The position controller uses 1 increment per cycle time^2“ as the dimension for the maximum permissible axis acceleration (1 increment is equivalent to 0.1 µm for translatory and 0.0001° for rotary axes). The smallest representable acceleration value in the position controller is therefore “0.1 µm per cycle time^2” or “0.0001° per cycle time^2”. The acceleration specified in the axis parameters (dimension mm/s^2 or °/s^2) is converted into the position controller internal unit format. The acceleration value cannot be represented in the position controller if it is too small.		
Response	Class	1	Warning output and correct the value
Solution	Class	1	Specify larger value for maximum axis acceleration see P-AXIS-00008 or increase controller cycle time.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [1µm/s or 0.001°/s]
		Specified maximum permissible axis acceleration see P-AXIS-00008
	%3:	Corrected value [1µm/s or 0.001°/s]
		Smallest permissible value for maximum permissible axis acceleration
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70209

Emergency deceleration too small (less than 1 increment per cycle).			
Description	The deceleration for an emergency stop is set too small in the axis parameters (see P-AXIS-00003). The position controller uses “1 increment per (cycle time)^2” as dimension of the emergency deceleration (1 increment is equivalent to 0.1 µm for translatory and 0.0001° for rotary axes). The smallest representable deceleration value in the position controller is therefore “0.1 µm per cycle time^2” or “0.0001° per cycle time^2”. The delay specified for the deceleration (dimension mm/s^2 or °/s^2) is converted into the position controller internal unit format. The acceleration value cannot be represented in the position controller if it is too small.		
Response	Class	1	Warning output and correct the value
Solution	Class	1	Solution: • Specify larger value for emergency stop deceleration, see P-AXIS-00003 • Increase the cycle time of the controller

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [1µm/s or 0.001°/s]
		Specified deceleration for an emergency stop, see P-AXIS-00003
	%3:	Corrected value [1µm/s or 0.001°/s]
		Minimum value allowed for deceleration
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70210

Terminal KL2521 reports an error. (state in wert_1).			
Description	Terminal KL2521 reports an error. A further error diagnoses can be done with the documentation of the manufacturer.		
Response	Class	8	Open-loop controlled stop of concerned axis
Solution	Class	6	Perform an error diagnosis

Parameter	%1:	Current value [-]
		Status byte of encoder
	%2:	Current value [-]
Error type	11, Error message from position controller.	

ID 70211

Master axis for cross compensation not present.			
Description	The specified master axis for cross compensation does not exist. See also [FCT-C5// Section: Cross compensation]		
Response	Class	7	Controlled stop of axis affected
Solution	Class	7	Specify a valid logical axis number, see P-COMP-00005
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Specified logical axis number of the master axis which does not exist. (seeP-COMP-00005)	
	%3:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70212

Value of 'field_bus_allows_optimized_schedule' is not identical for all axes.			
Description	The setting “field_bus_allows_optimized_schedule” in the axis parameters must be identical for all axes. This option is switched off by default. In this case, new manipulated variables are first calculated in the position controller using the command values of the interpolator from the last cycle. Only then are new command values requested by the interpolator. This reduces the time needed for the calculation of the set command values (e.g. reducing the temporal jitter). If this option is switched on, new actual values are requested by the interpolator at the beginning of each position controller cycle and new manipulated variables are only calculated afterwards. This eliminates the cycle offset (one cycle) of the generated command values.		
Response	Class	1	Warning. The function is turned off.
Solution	Class	1	Use the identical setting for all axes
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		Setting for current axis	
	%4:	Error value [-]	
		Global setting	
Error type	11, Error message from position controller.		

ID 70215

Output interface to CANopen axis is not available.			
Description	There is no output interface configured for CANopen in the axis machine data.		
Response	Class	1	Warning output.

Solution	Class	1	Configure an interface for the CANopen axis interface for the axis.
Error type	-		

ID 70216

Interface from CANopen axis not available.			
Description	There is no input interface configured for CANopen in the axis machine data.		
Response	Class	1	Warning output.
Solution	Class	1	Configure an CANopen input interface for the axis.
Error type	-		

ID 70217

Torque limit prematurely exceeded.			
Description	The torque limit while homing to fixed stop is exceeded before the minimum distance kenngr.homing.torq_min_distance was reached.		
Response	Class	5	Abrupt axis stop, feedhold for the other axes.
Solution	Class	6	Check minimum distance for homing to fixed stop and perform NC reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [0.1%]	
		Current torque actual value.	
	%3:	Limit value [0.1%]	
		Maximum torque value.	
	%4:	Current value [0.1 µm or 0.0001°]	
		Current distance covered.	
	%5:	Limit value[0.1 µm or 0.0001°]	
		Minimum distance kenngr.homing.torq_min_distance for homing to fixed stop.	
Error type	11, Error message from position controller.		

ID 70218

HLI activates torque limit while homing to fixed stop.			
Description	During homing to fixed stop, the PLC wants to activate a torque limitation.		
Response	Class	5	Immediate axis stop for defective axis, feedhold for other axes.
Solution	Class	6	Perform a reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	11, Error message from position controller.		

ID 70220

During active torque limit maximum speed was exceeded.			
Description	The maximum permissible actual velocity P-AXIS-00314 was exceeded while a torque limit is active.		
Response	Class	5	Abrupt axis stop, feedhold for the other axes.

Solution	Class	6	Check actual speed or maximum permissible actual speed P-AXIS-00314 and perform NC reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current speed value.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible actual speed P-AXIS-00314.	
	%4:	Limit value [-]	
Error type	11, Error message from position controller.		

ID 70221

Drive is not ready for interpolation.			
Description	The CANopen-drive is to be moved, although it is not in the "Operation enabled" state (see documentation of the drive manufacturer). Therefore, it cannot therefore follow the specified setpoints. Possible solutions: Check if the drive releases in the PLC were set Possibly the CANopen state must be gone through again after a drive error (Remove drive releases and set them again) Check whether the drive releases were set in the correct sequence ("Drive On" -> "Torque Permission" -> "Release Feedhold", see [HLI//Control commands of an axis]). Perhaps the releases were set too fast one after the other. Check the hardware release at the drive amplifier.		
Response	Class	5	Immediate stop of the concerned axis
Solution	Class	6	Possible solutions: • Check if the drive releases in the PLC were set • Possibly the CANopen state must be gone through again after a drive error (Remove drive releases and set them again) • Check whether the drive releases have been set in the correct order ("Drive On" ->"Torque Permission" -> "Release Feedhold", [HLI// Section: Control commands of an axis]). Perhaps the releases were set too fast one after the other. • Check the hardware release at the drive amplifier.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Current value [-]	
		Status word of the drive	
	%4:	Block number [-]	
		Block number of the NC program, in which the error occurred	
Error type	11, Error message from position controller.		

ID 70222

CANOpen drive reports an error.			
Description	An error has occurred in the CANOpen drive. Check the status of the drive. Further error diagnosis can be carried out with the help of the drive manufacturer's documentation. The axis is held in a controlled stop In a CNC-controlled error response (see P-AXIS-00254). On the other hand, if the drive executes the error response itself, the controller outputs no further command values to the drive (abrupt stop). If the cyclic process data contains the object 0x603F, the value 5 of the error message shows the drive error code.		
Response	Class	0	Controlled or abrupt stop of the axis
Solution	Class	6	Consult drive manufacturer's documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [-]	
		CANOpen drive status	
	%4:	Block number [-]	
		Block number of the NC program, in which the error occurred.	
	%5:	Current value [-]	
		Drive error code 0x603F (if configured in cyclic process data)	
Error type	11, Error message from position controller.		

ID 70223

Negative hardware limit switch of drive has been actuated.			
Description	The drive reports that the negative hardware limit switch is actuated. Possible causes: • Incorrect NC program. • Overshoot of axis is too great.		
Response	Class	5	Immediate stop of the affected axis. The drive executes its own error reaction depending on its drive setting.
Solution	Class	6	Correct the NC program and check controller settings
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Block number [-]	
		Block number of the NC program where the error occurred.	
	%4:	Current value [0.1 µm or 0.0001°]	
		Block number of the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70224

Positive hardware limit switch of drive has been actuated.

Description	The drive reports that the positive hardware limit switch is actuated. Possible causes: • Incorrect NC program. • Overshoot of axis is too great.		
Response	Class	5	Immediate stop of the affected axis. The drive executes its own error reaction depending on its drive setting.
Solution	Class	6	Correct the NC program and check controller settings
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis affected	
	%3:	Block number [-]	
		Block number of the NC program where the error occurred	
	%4:	Current value [0.1 µm or 0.0001°]	
		Current command position	
Error type	11, Error message from position controller.		

ID 70225

During active velocity monitoring maximum speed was exceeded.			
Description	The maximum permissible actual velocity P-AXIS-00311 was exceeded during active velocity monitoring.		
Response	Class	7	
Solution	Class	6	Check actual velocity or maximum permissible actual velocity P-AXIS-00311 and perform NC reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current velocity value.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible actual velocity P-AXIS-00311.	
Error type	11, Error message from position controller.		

ID 70229

Number of cycles for smooth coupling/decoupling the cross compensation too large.			
Description	Cross compensation correction values can be executed and distributed over several position controller cycles by means of a filter. Specify the number of filter cycles used in the parameter P-COMP-00026 in the compensation value list ([COMP// Section: Cross compensation]). However, the value found is too high. The number of filter cycles is reduced to the maximum permissible value (depending on application). See also[FCT-C5// Section: Cross compensation]		
Response	Class	1	Correct the parameter to the maximum possible number of cycles
Solution	Class	1	Correct the value of the parameter P-COMP-00026 in the compensation value list.

Parameter	%1:	Logical axis number [-]
		Logical axis number of the affected slave axis, see P-AXIS-00016
	%2:	Current value [-]
		Given number of filter cycles, see P-COMP-00026
	%3:	Corrected value [-]
		Number of maximum permissible filter cycles (application-dependent)
	%4:	Block number [-]
		Block number in the NC program where the error occurred.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70230

Gantry slave axis exceeds the maximum permissible path during referencing.			
Description	A gantry slave axis has exceeded the maximum permissible travel distance set in axis parameter P-AXIS-00284 for homing. The error message is output by the master axis of the gantry combination.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check why the homing was not completed in time. Possible reasons are: • Wrong position of reference cam. • Wrong wiring of reference cam or zero pulse signal of encoder. • Check the transfer of the reference cam signal on the HLI: Is the reference cam signal transferred to the correct axis? • Check homing parameters of the NC. • Check parameters of drive amplifier/terminal.
Parameter	%1:		Logical axis number [-]
			Logical axis number (P-AXIS-00016) of master axis
	%2:		Logical axis number [-]
			Logical axis number (P-AXIS-00016) of slave axis
	%3:		Current value [0.1 µm or 0.0001°]
			Distance moved by the slave axis.
	%4:		Limit value [0.1 µm or 0.0001°]
			Maximum permissible distance for slave axis see P-AXIS-00284.
	%5:		Current value [-]
			Relative distance the axis moved in the actual cycle, determines movement direction of the axis.
Error type	11, Error message from position controller.		

ID 70231

Axis shall be moved although drive does not supply valid current data.			
Description	An axis shall be moved although the drive does not deliver valid actual values.		
Response	Class	5	Controlled halt of the axis, the control loop is open.
Solution	Class	6	Possible causes: • Interruption of bus connection to drive. • The drive is not ready for cyclic data exchange. Check the bus connection to the drive and the drive status.

Parameter	%1:	Logical axis number [-]
		Logical axis number of drive see P-AXIS-00016
	%2:	Drive type [-]
		Drive type see P-AXIS-00020
	%3:	Current value [-]
		Valid indication for drive actual data.
	%4:	Block number [-]
		Block number of NC program.
Error type	11, Error message from position controller.	

ID 70242

The unit of the correction values is invalid.			
Description	An invalid unit for compensation values was specified in an axis compensation value list. P-COMP-00003, P-COMP-00008). However, the input in increments of 1 is only permissible for cross and plane compensation for path resolutions (measuring systems for master and slave axes) (see P-AXIS-00234, P-AXIS-00233). See [COMP// Section: General compensation value data]		
Response	Class	7	Warning, compensation is turned off for affected axis
Solution	Class	7	Cross compensation: Correct parameter P-COMP-00003 Plane compensation: Correct parameter P-COMP-00008.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Error value [-]	
		Faulty unit see P-COMP-00003, P-COMP-00008	
	%3:	Limit value [-]	
		Parameter value for incremental compensation values	
	%4:	Limit value [-]	
		Parameter value for metrical compensation values	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70243

Invalid resolution of slave axis for plane compensation.			
Description	The compensation values for plane compensation were specified in increments in the corresponding compensation value list. This is only permitted for a measuring system path resolution of one (see P-AXIS-00234, P-AXIS-00233). However, the slave axis (the axis, which has to be compensated) uses a different path resolution than one. See [COMP// Section: Plane compensation] and [FCT-C5//Plane compensation]		
Response	Class	7	Closed-loop controlled axis stop and turn off the plane compensation for the affected axis
Solution	Class	7	Change the value of parameter P-COMP-00008 in the compensation value list to one and enter the compensation values metrically (in 0.1 µm).

Parameter	%1:	Logical axis number [-]
		Logical axis number of the affected slave axis see P-AXIS-00016
	%2:	Error value [-]
		Path resolution of measuring system of slave axis, see P-AXIS-00233, P-AXIS-00234
	%3:	Expected value [-]
		Expected path resolution of the measuring system, see P-AXIS-00233, P-AXIS-00234
Error type	11, Error message from position controller.	

ID 70244

No compensation value list available for cross compensation.			
Description	An axis in the the axis parameters is selected for cross compensation (sag compensation) (P-AXIS-00047). However, the compensation value list containing the required correction values is missing (see [COMP// Section: Cross compensation]) [FCT-C5//Cross compensation]		
Response	Class	7	Closed-loop controlled axis stop and disable cross compensation
Solution	Class	7	If cross compensation is enabled for an axis, a compensation value list must also be specified for it in the start-up list P-STUP-00016, P-STUP-00017, P-STUP-00036) (see [COMP// Section: Cross compensation]). Example (extract from start-up list): zahl_kw 1 achs_kw[0] ..\listen\achskw3.lis achs_kw_log_ax_nr[0] 3
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
Error type	11, Error message from position controller.		

ID 70245

The command positions for cross compensation are not sorted in an ascending order.

Description	Compensation values are specified for cross compensation in the compensation value lists at defined interpolation points. between Between these points, the correction values are determined using linear interpolation. Therefore they must be arranged according to their position in an ascending order.
	A possible error cause can be using less interpolation points than the number given in the parameter P-COMP-00004 since the positions of the unused points are set to zero (nil).
	[FCT-C5//Section: Cross compensation])
	[COMP// Section: Cross compensation])
	
	Wrong example (extract from compensation data list)
	`kw.crosscomp.table[0].sollw 0`
	`kw.crosscomp.table[0].correction 0`
	`kw.crosscomp.table[1].sollw 100000`
	`kw.crosscomp.table[1].correction 1000`
	`kw.crosscomp.table[2].sollw 300000`
	`kw.crosscomp.table[2].correction 3000`
	`kw.crosscomp.table[3].sollw 200000`
	`kw.crosscomp.table[3].correction 2000`
	
	Corrected example (extract from compensation data list)
	`kw.crosscomp.table[0].sollw 0`
	`kw.crosscomp.table[0].correction 0`
	`kw.crosscomp.table[1].sollw 100000`
	`kw.crosscomp.table[1].correction 1000`
	`kw.crosscomp.table[2].sollw 200000`
	`kw.crosscomp.table[2].correction 2000`
	`kw.crosscomp.table[3].sollw 300000`
	`kw.crosscomp.table[3].correction 3000`		

Response	Class	7	Closed-loop controlled axis stop and disabling of cross compensation
Solution	Class	7	Correct compensation value table see P-COMP-00006
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Current value [-]	
		Position of last interpolation point see P-COMP-00006	
	%3:	Error value [-]	
		Position of current interpolation point see P-COMP-00006	
	%4:	Current value [-]	
		Index of actual interpolation point	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70246

Logical axis number of master axis for cross compensation is invalid.			
Description	Cross compensation was activated for the axis in the axis parameters. However, the logical axis number of the master axis specified in the compensation value list (the axis whose position affects the slave axis, see P-COMP-00005) is invalid. [FCT-C5//Cross compensation] [COMP// Section: Cross compensation]		
Response	Class	7	Closed-loop controlled axis stop and disabling of cross compensation
Solution	Class	7	Specify a valid master axis, see P-COMP-00005

Parameter	%1:	Logical axis number [-]
		Logical axis number of the affected slave axis see P-AXIS-00016
	%2:	Logical axis number [-]
		Invalid logical axis number of the master axis, see P-COMP-00005
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70247

Invalid slave axis type for cross compensation.			
Description	The axis is of an axis type (see P-AXIS-00018) that cannot be compensated with cross compensation. Both the master and the slave axes must be linear axes for cross compensation. [FCT-C5// Section: Cross compensation]		
Response	Class	7	Closed-loop controlled axis stop and disabling of cross compensation
Solution	Class	7	Check and correct the axis type of the slave axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the affected slave axis see P-AXIS-00016	
	%2:	Error value [-]	
		Incorrect axis type of slave axis see P-AXIS-00018	
	%3:	Expected value [-]	
		Expected axle type see P-AXIS-00018	
Error type	11, Error message from position controller.		

ID 70248

Invalid master axis type for cross compensation.			
Description	The axis is of an axis type (see P-AXIS-00018) that cannot be used with cross compensation. Both the master and the slave axes must be linear axes for cross compensation. See FCT-C5// Cross compensation		
Response	Class	7	Closed-loop controlled axis stop and disabling of cross compensation
Solution	Class	7	Check and correct the axis type of the master axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of master axis affected see P-COMP-00005	
	%2:	Error value [-]	
		Axis type of master axis see P-AXIS-00018	
	%3:	Expected value [-]	
		Expected axle type see P-AXIS-00018	
Error type	11, Error message from position controller.		

ID 70249

Invalid resolution of slave axis for cross compensation.

Description	Compensation values for cross compensation were specified in increments in the associated compensation value list. This is only permitted for a measuring system path resolution of one (see P-AXIS-00234, P-AXIS-00233). However, the slave axis to be compensated uses a path resolution unequal to one. See: [COMP// Section: Cross compensation]] [FCT-C5// Section: Cross compensation]		
Response	Class	7	Closed-loop controlled axis stop. The cross compensation is turned off for affected axis
Solution	Class	7	Change the value of parameter P-COMP-00003 in the compensation value list to one and enter the compensation values metrically (in 0.1 µm).

Parameter	%1:	Logical axis number [-]
		Logical axis number of the affected slave axis see P-AXIS-00016
	%2:	Error value [-]
		Path resolution of measuring system of slave axis, see P-AXIS-00233, P-AXIS-00234
	%3:	Expected value [-]
		Expected path resolution of the measuring system, see P-AXIS-00233, P-AXIS-00234
Error type	11, Error message from position controller.	

2.8.2 ID-range 70250-70499

ID 70250

Invalid resolution of master axis for cross compensation.			
Description	The compensation values for cross compensation were specified in increments in the corresponding compensation value list. This is only permitted for a measuring system path resolution of one (see P-AXIS-00234, P-AXIS-00233). However, the master axis whose position affects the slave axis uses a path resolution unequal to one. See: [COMP// Section: Cross compensation]] [FCT-C5// Section: Cross compensation]		
Response	Class	7	Controlled slave axis stop. And disable compensation for the slave axis
Solution	Class	7	Change the value of parameter P-COMP-00003 in the compensation value list to one and enter the compensation values metrically (in 0.1 µm).

Parameter		
	%1:	Logical axis number [-]
		Logical axis number of the slave axis P-AXIS-00016
	%2:	Logical axis number [-]
		Logical axis number of affected master axis, see P-AXIS-00016, P-COMP-00005
	%3:	Error value [-]
		Path resolution of measuring system of slave axis, see P-AXIS-00233, P-AXIS-00234
	%4:	Expected value[-]]
		Expected path resolution of the measuring system see P-AXIS-00233, P-AXIS-00234

Error type	
	11, Error message from position controller.

ID 70251

Invalid resolution of master axis for plane compensation.			
Description	The compensation values for plane compensation were specified in increments in the corresponding compensation value list. This is only permitted for path resolution of measuring system of one (see P-AXIS-00234, P-AXIS-00233). However, one of the master axes (the axes whose positions influences the slave axis) uses a different path resolutions unequal to one. See: [FCT-C5// Section: Plane compensation] [COMP// Section: Plane compensation]		
Response	Class	7	Controlled slave axis stop. And disable compensation for the slave axis
Solution	Class	7	Change the value of parameter P-COMP-00008 in the compensation value list to one and enter the compensation values metrically (in 0.1 µm).

Parameter	%1:	Logical axis number [-]
		Logical axis number of the slave axis P-AXIS-00016
	%2:	Logical axis number [-]
		Logical axis number of affected master axis see P-AXIS-00016, P-COMP-00014, P-COMP-00015
	%3:	Error value [-]
		Path resolution of measuring system of slave axis, see P-AXIS-00233, P-AXIS-00234
	%4:	Expected value[-]]
		Expected path resolution of the measuring system see P-AXIS-00233, P-AXIS-00234
Error type	11, Error message from position controller.	

ID 70252

Encoder axis can not be assigned with command values.			
Description	An axis that is configured as an encoder axis only was commanded to execute a motion.		
Response	Class	5	Immediate axis stop.
Solution	Class	6	Perform a reset. Check motion command or configuration setting for this axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020.	
	%3:	Current value [-]	
		BF LR command bitmask	
	%4:	Block number [-]	
		Block number of NC program or job number where the error occurred.	
Error type	11, Error message from position controller.		

ID 70253

Active program in channel: Switch on/off of axis compensation is delayed.			
Description	By updating the axis parameters, axis compensation was either switched off or on. To do this, it is necessary to modify the command and current positions in the position controller for this axis. In order not to impair the active program in the NC channel, axis compensation is only switched on or off at the end of the program. See parameters: P-AXIS-00175 for leadscrew error compensation P-AXIS-00047 for cross compensation P-AXIS-00174 for plane compensation		
Response	Class	1	Warning, activation or deactivation of compensation for the affected axis is delayed until program ends.
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
		Logical axis number of the affected slave axis see P-AXIS-00016
	%2:	Current value [-]
		State of axis compensation before updating the axis parameters (0 = disabled, 1 = enabled)
	%3:	Current value [-]
		State of axis compensation after actualising the axis parameters (0 = disabled, 1 = enabled)
Error type	11, Error message from position controller..	

ID 70254

Axis is to be compensated, although the required drive releases are missing.			
Description	One or several axis compensations are active for the affected axis. Its position should be corrected due to a position change of a master axis, However, the axis cannot move, since the required drive releases are missing. See [FCT-C5// Section:Overview]		
Response	Class	7	Controlled stop of affected axis.
Solution	Class	5	Set drive releases or turn axis compensation(s) off
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the affected slave axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of affected slave axis, see P-AXIS-00020	
	%3:	Block number [-]	
		Logical axis number of moved master axis P-COMP-00005, P-COMP-00014, P-COMP-00015	
	%4:	Logical axis number [-]	
		Block number of the NC program in the channel of the Master axis, in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70255

Axis is to be compensated, but a movement of the drive is not possible.			
Description	One or several axis compensations are active for the affected axis. Its position must be corrected due to a position change of a master axis, However, the axis cannot move, since an axis-specific or channel-specific feedhold is set for the axis by the PLC. [FCT-C5// Section: Overview]		
Response	Class	7	Controlled stop of affected axis.
Solution	Class	5	Remove feedhold or turn axis compensation(s) off

Parameter	%1:	Logical axis number [-]
		Logical axis number of the affected slave axis see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of affected slave axis, see P-AXIS-00020
	%3:	Block number [-]
		Logical axis number of moved master axis P-COMP-00005, P-COMP-00014, P-COMP-00015
	%4:	Logical axis number [-]
		Block number of the NC program in the channel of the Master axis, in which the error occurred.
Error type	11, Error message from position controller.	

ID 70256

Number of cycles for smooth coupling/decoupling of plane compensation too large.			
Description	The filter cycles specified for plane compensation in the compensation value list to activate the compensation value across several position controller time cycles (see P-COMP-00027) is greater than the maximum permissible number. It is reduced to the maximum permissible value. See: [COMP// Section: Plane compensation] [FCT-C5// Section: Plane compensation]		
Response	Class	1	Warning, reduction of filter cycles to maximum permissible value.
Solution	Class	1	Reduce number of filter cycles in compensation value list see P-COMP-00027
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the affected slave axis see P-AXIS-00016	
	%2:	Current value [-]	
		Number of specified filter cycles see P-COMP-00027	
	%3:	Corrected value [-]	
		Maximum permissible number of filter cycles	
	%4:	Block number [-]	
		Block number in the NC program at which the error occurred.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70259

Coefficient (pitch) of temperature compensation exceeds valid range.			
Description	The compensation values of temperature compensation are approximated by a linear straight line. The pitch of the straight line is specified in parameter P-AXIS-00274 in the axis parameter lists as a function of temperature. Alternatively, it can also be written using the CNC object “TEMPC::coefficient” of the corresponding axis. The CNC object can be found in the GEO task under the index group 0x120300 and index offset 0x<A_{ID}>0043. However, the current value of the pitch is outside the permitted value range. See [FCT-C5// Section: Temperature compensation]		
Response	Class	1	Warning, automatic correction of pitch to maximum or minimum permissible value.

Solution	Class	1	Correct value for pitch (via CNC object or by updating the axis parameter list).
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016)	
	%2:	Error value [-]	
		Incorrect value for P-AXIS-00274	
	%3:	Limit value [-]	
		Largest allowed value for pitch P-AXIS-00274	
	%4:	Limit value [-]	
		Smallest permissible value for pitch P-AXIS-00274	
	%5:	Corrected value [-]	
		Corrected value for parameter P-AXIS-00274	
Error type	11, Error message from position controller.		

ID 70261

Terminal reports an error.			
Description	The error code is set in the status byte delivered by an intelligent Bus Terminal. With this flag an internal error in the bus terminal is indicated.		
Response	Class	8	Error message output, controlled stop of axis.
Solution	Class	6	Eliminate error cause in the terminal, further information about the error cause can possibly be read at LEDs on the terminal or read out from the terminal using diagnostic software. Look at manufacturer documentation.
Parameter	%1:	Current value [-]	
		Logical axis number of affected axis (see P-AXIS-00016)	
	%2:	Current value [-]	
		Character string specifying the type of terminal, e.g.“KL2541“.	
	%3:	Current value [-]	
		Status value from the terminal that caused the error.	
Error type	-		

ID 70262

No position encoder configured for this axis.			
Description	No actual position value was configured in the cyclic value telegram of the axis, although the axis is not a spindle. Only spindles can be operated without position sensors.		
Response	Class	5	Error message output, immediate stop of the axis.
Solution	Class	7	Configure an actual position value in the cyclic telegram if the axis is not a spindle; if it is a spindle axis, assign the value 4 to the P-AXIS-00018 parameter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016)	
	%2:	Error value [-]	
		Configured axis type (P-AXIS-00018).	
	%3:	Logical axis number [-]	
Error type	-		

ID 70263

Homing method (homing_type) of master-/slave axis is different.			
Description	The entry P-AXIS-00299 is assigned differently for the master and slave axes of a gantry configuration.		
Response	Class	1	Error message output, correct P-AXIS-00299 of the slave axis to the value of the master axis..
Solution	Class	1	P-AXIS-00299 have the same value assigned to master and slave axes.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the slave axis (see P-AXIS-00016).	
	%2:	Logical axis number [-]	
		Logical axis number of the master axis (see P-AXIS-00016)	
	%3:	Error value [-]	
		Entry P-AXIS-00299 of the slave axis.	
	%4:	Error value [-]	
		Entry P-AXIS-00299 of the master axis.	
	%5:	Corrected value [-]	
		Corrected entry P-AXIS-00299 of the slave axis.	
Error type	-		

ID 70264

An axis collision was detected for an axis of a gantry system.			
Description	This is a follow-up error message of the collision monitor following an error message for axis collision (see ID 70092 [▶ 1026]: “Collision of two axes detected”). It is displayed due to the internal error handling in the controller if the affected axis is part of a gantry system. See [FCT-C3// Section: Description]		
Response	Class	7	Controlled stop of the gantry system
Solution	Class	6	see error message ID 70092 [▶ 1026] "Collision between two axes detected".
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
Error type	11, Error message from position controller.		

ID 70265

Parked drive to be moved.			
Description	An axis is to be interpolated although the command ‘Park axis’ was activated for this axis via the High Level Interface. See also MCControlBoolUnit_Mode0 for CNC build < 2800 Ir_mc_control.mode_0 for CNC Version > 2800		
Response	Class	5	Immediate stop of the axis.
Solution	Class	6	Reset the 'Park axis' command before interpolating the axis.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis see P-AXIS-00016
	%2:	Current value [-]
		Current value of state word 2 (ZSW2) of the drive.
	%3:	Block number [-]
		Block number in the NC program.
Error type	11, Error message from position controller.	

ID 70266

Drive shall be moved although encoder is parked.			
Description	An axis shall be moved although the encoder of the axis is parked via a HLI-command. See also MCControlBoolUnit_Mode0 for CNC build < 2800 Ir_mc_control.mode_0 for CNC Version > 2800		
Response	Class	5	Immediate stop of the axis.
Solution	Class	6	Reset 'Park encoder' command before axis interpolation [HLI// MCControlBoolUnit_Mode1].
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis see P-AXIS-00016	
	%2:	Block number [-]	
		Value of encoder state word (GX_ZSW)	
	%3:	Block number [-]	
		Block number in the NC program.	
Error type	11, Error message from position controller.		

ID 70267

Timeout while reading absolute position from drive.			
Description	When requesting the absolute position from an PROFIDRIVE, the value was not delivered within the timeout limit.		
Response	Class	5	Immediate stop of the axis.
Solution	Class	6	Consult drive manufacturer's documentation Check drive parameters Is the drive able to deliver absolute positions?
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected gantry axis see P-AXIS-00016	
	%2:	Current value [-]	
		Value of encoder state word GX_ZSW.	
	%3:	Upper limit value [µs]	
		Time-outs.	
Error type			

ID 70268

No actual position configured in the cyclic telegram for this operation mode.			
Description	For the axis, no actual position values are transferred from the drive to the controller. However, for the configured operation mode (e.g. for position control, see P-AXIS-00320), actual position values are required in the controller.		
Response	Class	5	Abort the controller start-up.
Solution	Class	7	Correct the configuration of the cyclic telegram so that actual positions are transferred from the drive.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		Configured operation mode, see P-AXIS-00320	
Error type	-		

ID 70269

No command velocity configured in the cyclic telegram for this operation mode.			
Description	The calculated commanded velocity in the controller system is not transferred to the drive for this axis. However, the commanded velocity must be transferred for the parameterised operation mode (e.g. position control, see P-AXIS-00320.		
Response	Class	5	Abort the controller start-up.
Solution	Class	7	Correct the configuration of the cyclic telegram so that the calculated command velocity is transferred to the drive.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		Configured operation mode, see P-AXIS-00320	
Error type	-		

ID 70271

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70272

Invalid name for EtherCAT master process.

Description	The specified process name of the EtherCAT for the real-time operation system INtime is invalid (see P-AXIS-00372). The process name is dependent on the network interface used. Valid names are: ECMeth0: EtherCAT Master uses network device "eth0" ECMeth1: EtherCAT Master uses network device "eth1"		
Response	Class	5	Correct process name P-AXIS-00372
Solution	Class	7	Abort the controller start-up.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Invalid process name	
Error type	-		

ID 70273

Different names for EtherCAT master process.			
Description	Different names for the EtherCAT Master were specified for two or more axes under the real-time operation system INtime (see P-AXIS-00372). However, only one EtherCAT master process is supported by the controller.		
Response	Class	5	Abort start-up
Solution	Class	7	Use identical process names P-AXIS-00372 for all axes
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Specified process name, see P-AXIS-00372	
	%3:	Expected value [-]	
		Expected process name, see P-AXIS-00372	
Error type	-		

ID 70274

Invalid memory identifier (memory_ident) for memory area.			
Description	The specified name for the shared memory with the process data of the EtherCAT master is invalid.		
Response	Class	5	Configuration of the process data is aborted
Solution	Class	7	Correct the memory identifier and restart the numerical controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected	
	%2:	Error value [-]	
		Incorrect identifier	
Error type	-		

ID 70275

Timeout during execution of a #DRIVE command.

Description	The execution of the programmed NC command #DRIVE was not acknowledged by the drive within the specified time. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Program is aborted
Solution	Class	6	Check in drive amplifier why command was not executed and correct cause.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Limit value [-]	
		Time limit.	
Error type	11, Error message from position controller.		

ID 70276

Invalid value for PARAM_SET in NC command #DRIVE.			
Description	An invalid parameter set number was entered in PARAM_SET in a #DRIVE command for parameter set switching See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Program abort.
Solution	Class	6	Enter correct parameter set number in NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Error value [-]	
		Erroneous parameter set number from NC program.	
	%3:	Lower limit value [-]	
		Minimal admissible parameter set number.	
	%4:	Upper limit value [-]	
		Maximal admissible parameter set number.	
Error type	11, Error message from position controller.		

ID 70277

Invalid value for PARAM_SET in NC command #DRIVE.			
Description	An invalid motor number was entered in MOTOR in a #DRIVE command for motor switching. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Program abort.
Solution	Class	6	Enter correct motor set number in NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Erroneous motor number from NC program.	
	%3:	Lower limit value [-]	
		Minimal admissible motor number.	
	%4:	Upper limit value [-]	
		Maximal admissible motor number.	
Error type	11, Error message from position controller.		

ID 70278

The #DRIVE NC command is not supported by this drive type.			
Description	The #DRIVE command was used for an axis whose drive type does not support this command. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Abort NC program.
Solution	Class	6	Change NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Drive type of the axis, see P-AXIS-00020.	
Error type	11, Error message from position controller.		

ID 70279

Programmed telegram element not found in configured telegram.			
Description	In a #DRIVE WR [] command, a telegram element name was specified in the element "TELEGR_ELEM" that is not configured in the cyclic setpoint telegram of the drive. See [PROG// Section: Drive-independent switching of drive functions]		
Response	Class	1	Program execution stop
Solution	Class	1	Correct NC program to address the correct telegram element, or add the telegram element to the configuration of the setpoint telegram.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [-]	
		In #DRIVE command programmed, unknown telegram element.	
	%3:	Error value [-]	
Error type	-		

ID 70280

Value of user ident out of range.			
Description	In a #DRIVE WR [... TELEGR_ELEM=XXX VAL= ...] command (see [PROG]), an attempt was made to assign a value to a telegram element XXX via VAL, which lies outside the permissible value range for this telegram element. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Program abort.
Solution	Class	6	Adjust value of VAL in the NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Incorrect value programmed in VAL element	
	%3:	Lower limit value [-]	
		Minimum admissible value.	
	%4:	Upper limit value [-]	
		Maximum permissible value	

Error type	11, Error message from position controller.

ID 70281

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70282

Drive error during initialization of zero pulse search.			
Description	An error in the drive occurred during the initialization of the zero pulse search. Consult the documentation of the drive manufacturer for a further error diagnosis.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Consult drive manufacturer's documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Current value [-]	
	%4:	Block number [-]	
		Block number of the NC program, in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70283

Drive error during zero pulse search.			
Description	An error occurred in the drive during the zero pulse search. Consult the drive manufacturer documentation for further error diagnosis.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Consult drive manufacturer documentation
Parameter	%1:	Current value [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Block number [-]	
		Block number of the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70284

Calculated absolute position out of permissible range.

Description	A new absolute position is calculated when the reference position is set via the HLI interface (see [HLI//Section: Control commands of an axis]) or when the absolute position is ready with Profidrive drives. The computed result however is greater than permissible. Possible causes: • The axis has a path resolution greater than one. The homing position specified in the PLC (see [HLI]) is larger/smaller than permissible when converted into increments. • The offset mode for setting the homing position (see P-AXIS-00278) is used. The current position of the drive + the specified offset to the homing position P-AXIS-00279 is too large or too small. • The absolute position read by the Profidrive exceeds the permissible value range. The conversion factor P-AXIS-00065 may have been specified incorrectly.		
Response	Class	7	Closed-loop controlled stop of axis, setting of homing position is aborted
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Calculated absolute position in increments.	
	%3:	Lower limit value [-]	
		Minimum permissible position value	
	%4:	Upper limit value [-]	
		Maximum permissible position value	
Error type	-		

ID 70285

Programmed function identifier in NC command #DRIVE not found.			
Description	With a #DRIVE WR SYN [... KEY=XXX VAL= ...] command, the programmed function identifier XXX was not found in the axis configuration data. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Program abort.
Solution	Class	6	Adjust used function ID in the NC program or add the function ID used in the NC program to the configuration data.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Unknown function ID value detected programmed in KEY element	
Error type	11, Error message from position controller.		

ID 70286

Communication type not supported by drive type.			
Description	The configured communication type for the drive function is not supported for this drive type.		
Response	Class	7	Program abort.

Solution	Class	6	Change configuration, use a communication type that is supported for this drive type.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Configured, not supported communication type: 0 - Cyclic communication 1 - Acyclic communication.	
	%3:	Current value [-]	
		Function ID of the drive function where the incorrect communication type was configured.	
	%4:	Drive type [-]	
		Drive type, see P-AXIS-00020.	
Error type	-		

ID 70287

Different data types in parameter list and configured telegram.			
Description	The data type configured for a drive function does not fit to the data type that was configured in the cyclic telegram for the transmission of the drive function.		
Response	Class	7	Program execution stop.
Solution	Class	6	Change configuration, the data type configured for the drive function must fit to the data type of the telegram element which is used for transmission of the drive function in the cyclic telegram.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [-]	
		Name of configured drive function with the wrong data type.	
	%3:	Current value [-]	
		Data type configured in the drive function.	
	%4:	Current value [-]	
		Data type configured in the cyclic telegram.	
Error type	-		

ID 70288

Configured data type is not supported.			
Description	The data type of drive function configured in the parameter list is not supported.		
Response	Class	1	Program abort.
Solution	Class	1	Change configuration, use data that is supported for this drive type.
Parameter	%1:	Current value [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Name of drive function	
	%3:	Error value [-]	
		Data type not supported.	
Error type	11, Error message from position controller.		

ID 70290

Error while writing a SERCOS parameter.			
Description	An error occurred while writing a SERCOS parameter via #DRIVE command. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	5	Program abort.
Solution	Class	6	Check number and value of SERCOS parameter written.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Number of the written SERCOS parameter.	
	%3:	Current value [-]	
		Value of the written SERCOS parameter.	
	%4:	Current value [-]	
		Name of the drive function.	
Error type	11, Error message from position controller.		

ID 70291

The function is not supported by this drive type.			
Description	The drive function programmed in #DRIVE is not supported by the drive type used. See [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Program abort.
Solution	Class	6	Change NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Error value [-]	
		Drive type, see P-AXIS-00020.	
	%3:	Error value [-]	
		Unsupported drive function that was programmed in the #DRIVE command.	
Error type	11, Error message from position controller.		

ID 70292

Reference cam signal is not configured in cyclic telegram.			
Description	The parameter P-AXIS-00321 was used to program that the reference cam signal should be read directly from the digital drive inputs. However, the elements required to transfer the digital inputs are not configured in the cyclic telegram.		
Response	Class	5	
Solution	Class	1	Configure elements for the transmission of the digital inputs in the cyclic actual value telegram of the drive.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Error value [0.1 µm or 0.0001°]	
		Parameterised name of the digital input used in P-AXIS-00321.	
	%3:	Drive type [-]	
		Drive type (P-AXIS-00020).	

Error type	-

ID 70293

Error when starting probing command in the drive.			
Description	An error occurred in a SERCOS drive when a measurement run was started.		
Response	Class	7	
Solution	Class	6	Detect the error cause in the drive by using the drive manufacturers diagnostic tools.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [-]	
		Return value 1 of the drive command.	
	%3:	Current value [-]	
		Return value 1 of the drive command.	
	%4:	Current value [-]	
		Command state.	
	%5:	Block number [-]	
		NC block number.	
Error type	11, Error message from position controller..		

ID 70294

Correction movement was aborted.			
Description	A correction movement of an axis was aborted due to one of the following reasons: During the compensation motion the PLC reset the • MCControlBoolUnit_FollowUp signal for CNC build < 2800 (see [HLI//Section: Axis control commands]) • Ir_mc_control.follow_up for CNC Version >2800 (see [HLI// Kapitel: Axis control commands]) or During the compensation motion the PLC reset the • MCControlBoolUnit_ReleaseFeedhold for CNC build < 2800 • Ir_mc_control.release_feedhold for CNC build > 2800 signal. [HLI// Section: Axis control commands]. The fieldbus did not deliver valid values.		
Response	Class	1	
Solution	Class	1	Change PLC program or fieldbus configuration.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Block number [-]	
		NC block number.	
Error type	11, Error message from position controller.		

ID 70295

No homing on zero pulse possible with the configured telegram.			
Description	Zero pulse homing is parameterised for the axis (P-AXIS-00084. However, the parameterised cyclic drive telegram cannot make a zero pulse search. The latched zero pulse value, the latch status word or the latch control word is not configured in the cyclic drive telegram. Zero pulse homing is disabled after a warning output.		
Response	Class	7	
Solution	Class	1	Based on the values 2 to 4 of the error message, define which elements are missing in the cyclic telegram and extend the telegram configuration accordingly.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [0.1 µm or 0.0001°]	
		Value is set to 1 if zero pulse value is configured in the cyclic telegram.	
	%3:	Current value [0.1 µm or 0.0001°]	
		Value is set to 1 if latch status word is configured in the cyclic telegram.	
	%4:	Current value [0.1 µm or 0.0001°]	
		Value is set to 1 if latch control word is configured in the cyclic telegram.	
	%5:	Corrected value [0.1 µm or 0.0001°]	
		Corrected value of P-AXIS-00084.	
Error type	11, Error message from position controller.		

ID 70296

The wr_ident[0] set up has no valid format.			
Description	The parameter identifier configured in the parameter list is not a valid SERCOS or CANopen parameter identifier.		
Response	Class	7	Program abort.
Solution	Class	6	Change parameter identifier in the parameter list. Valid parameter identifiers are of the form. [S|s|P|p]-[0..7]-[1 .. 04095]. Example: S-0-47 or P-7-3650.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Name of the drive function.	
	%3:	Error value [-]	
		Parameterised parameter identifier.	
Error type	-		

ID 70297

Motion range of axis is bigger than the value range of the absolute measurement system.

Description	For a PROFIDRIVE with absolute encoder, the motion range of the axis, which is defined by the software limit switches (P-AXIS-00177/P-AXIS-00178) is bigger than the value range of the absolute encoder, which is defined by P-AXIS-00092 and P-AXIS-00336. In this case the mapping of the encoder position to the axis position is not unique.		
Response	Class	5	System error, reboot controller is necessary
Solution	Class	7	Correct P-AXIS-00336. Refer to the drive documentation for the number of revolutions detectable by the absolute encoder. In many cases, this value can also be read out of the drive (P979 Subindex 5). Otherwise, reduce the axis motion range by resetting the software limit switches P-AXIS-00177/P-AXIS-00178 of the axis.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [0.1 µm or 0.0001°]
		Parameterised motion range of the axis.
	%3:	Current value [0.1 µm or 0.0001°]
		Maximum motion range of the axis that can be represented by the absolute encoder (P-AXIS-00336 * P-AXIS-00092).
	%4:	Current value [-]
		Value of P-AXIS-00336.
	%5:	Current value[-]
		Value of P-AXIS-00092.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70298

The operation mode of the drive could not be read.			
Description	The drive-controlled homing for CANopen requires switching the mode of operation in the drive to mode “homing”. The current operation mode in the drive must be read out before changing and for checking afterwards. However, one of these attempts to read failed.		
Response	Class	5	Abortion of homing
Solution	Class	6	Consult drive manufacturer's documentation

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of concerned axis, see P-AXIS-00020
	%3:	Block number [-]
		Block number in which the error occurred
Error type	11, Error message from position controller.	

ID 70299

The operation mode of the drive could not be changed.			
Description	The drive-controlled homing for CANopen requires switching the mode of operation in the drive to mode “homing”. Changing the mode of operation in the drive failed.		
Response	Class	5	Abort homing
Solution	Class	6	Consult drive manufacturer documentation

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	Block number [-]
		Block number where the error occurred
Error type	[-]	

ID 70300

A drive error occurred during homing.			
Description	An error has occurred in the drive during drive-controlled homing. Consult the documentation of the drive manufacturer for further error diagnosis.		
Response	Class	5	Abrupt axis stop
Solution	Class	6	Consult drive manufacturer documentation
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		Status word of the drive	
	%4:	Block number [-]	
		Block number where the error occurred	
Error type	[-]		

ID 70301

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70302

Drive-controlled homing could not be terminated.			
Description	For drive-controlled homing with CANopen, the operation mode in the drive must be switched to "Homing". After homing was completed, the original operation mode must be restored. If an error occurs, homing cannot be terminated and the axis cannot be moved further.		
Response	Class	5	Abrupt stop of the axis
Solution	Class	6	Command another reset. The controller attempts again to restore the original operation mode in the drive. Restart the controller. Verify that the correct mode of operation is active in the drive afterwards! Consult the documentation of the drive manufacturer.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
Error type		

ID 70304

Axis command positions inconsistent after abortion of a correction movement			
Description	A correction movement of the axis which was triggered by enabling the controller was aborted during an active program and then the axis was to be moved by the interpolator. If an axis correction movement is active at program start, the target position of this motion is used as a command position in order to initialise the axis positions. Due to the aborted correction movement, this target position is not reached and this leads to a permanent offset in the axis position. To prevent this position offset, an error message is output if the axis is to be moved to this state by the interpolator. An abort of a correction movement is triggered by: • PLC activates tracking mode • PLC resets the ‘feed enable’ drive control command PLC activates tracking mode by the PLC with the control unit • MCControlBoolUnit_FollowUp (CNC Version < 2800) • Ir_mc_control.follow_up (CNC Version > 2800 PLC resets the drive control command release feedhold signal with the control unit • MCControlBoolUnit_ReleaseFeedhold (CNC Builds < 2800) • Ir_mc_control.release_feedhold (CNC Version > 2800		
Response	Class	7	Axis stop
Solution	Class	6	Prevent abort of the correction movement, eventually start NC program when the correction movement has ended.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Block number [-]	
		Block number in the NC program.	
Error type	[-]		

ID 70305

PROFIDRIVE axis is not ready at the end of the control start-up.

Description	At start-up of the CNC, it is checked if the axis started up without error, and if the absolute positions could be read from the drive. This did not happen within the given time. For PROFIDRIVEs it is checked: • Axis is not in error state • Slave live sign is counting • The absolute position was read from the drive if it is set in the axis parameter list (see P-AXIS-00315)		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Correct cause for the error message with the information of this error message.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis see P-AXIS-00016
	%2:	Current value [-]
		Error identifier. A value other than zero (nil) indicates an error in the axis.
	%3:	Current value [-]
		Axis status. If the axis was set up to read the absolute position from the drive and could successfully read the position bit 0x2000 is set.
	%4:	Current value [-]
		Value of P-AXIS-00315
	%5:	Current value [-]
		A value of 1 is an indication that the sign of life of the slave is incremented.
Error type	-	

ID 70306

Error that cannot be corrected by reset. Reboot of controller required.			
Description	An error has occurred that cannot cleared by a reset. A controller reboot is required.		
Response	Class	5	Axis stop
Solution	Class	7	This error message is displayed during a reset. The actual error cause is displayed by another error message which occurred before this error message. Remedy the error using the information in this error message and then reset the controller. Possible causes of a non resettable error may include an incorrect configuration.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type, see P-AXIS-00020.
	%3:	Current value [-]
	%4:	Current value [-]
Error type	-	

ID 70307 / 70308

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70309

The latch function in the actuator is already activated at the start of the measurement run.			
Description	At start of measurement run, the touch probe function is already enabled in the drive, although it must be disabled according to the latch control word. Therefore touch probing is not possible.		
Response	Class	7	Controlled stop of axis and abortion of measurement
Solution	Class	6	Check status and control word of touch probing function Check parameterization of the touch probing function in the actuator
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Error value [-]	
		Status word of touch probing	
	%4:	Current value [-]	
		Control word of touch probing	
Error type	11, Error message from position controller.		

ID 70310

Activation of leadscrew error compensation not possible since it is not selected in axis parameter list.			
Description	To turn on leadscrew error compensation in the NC program for an axis, it must be activated in the axis parameter list (see P-AXIS-00175). See [FCT-C5// Section: Leadscrew error compensation]		
Response	Class	7	Stop axis, leadscrew error compensation remains inactive
Solution	Class	6	Activate leadscrew error compensation (see P-AXIS-00175)
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Parameter value P-AXIS-00175 to activate leadscrew error compensation	
Error type	11, Error message from position controller.		

ID 70311

Activation of leadscrew error compensation not possible since compensation table is invalid.	
Description	Could not activate leadscrew error compensation since no compensation value list exists or it contains errors. See also: [FCT-C5// Section: Leadscrew error compensation] [COMP// Section: Leadscrew error compensation]

Response	Class	7	Stop axis, leadscrew error compensation remains inactive
Solution	Class	6	Correct compensation value list]
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	11, Error message from position controller.		

ID 70312

Activation of leadscrew error compensation not possible since axis is not homed.			
Description	The axis must be homed in order to activate leadscrew error compensation. See FCT-C5]// Section: Leadscrew error compensation]		
Response	Class	1	Warning, lead screw error compensation remains inactive until axis is homed
Solution	Class	1	Home the axis (see [FCT-M1]) If an absolute measurement system is used, parameter P-AXIS-00014 can be set to 1 to mark the homing of the axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		TRUE, if axis is homed	
Error type	11, Error message from position controller.		

ID 70313

Activation of temperature compensation not possible since it is not selected in axis parameter list			
Description	To turn on temperature compensation in the NC program for an axis, it must be activated in the axis parameter list (see P-AXIS-00271). See FCT-C5]// Section: Temperature compensation]		
Response	Class	7	Stop axis, temperature compensation remains inactive
Solution	Class	6	Activate temperature compensation (see P-AXIS-00271)
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Parameter value P-AXIS-00271 to activate temperature compensation	
Error type	11, Error message from position controller.		

ID 70314

Activation of temperature compensation not possible since compensation table is invalid.			
Description	Could not activate temperature compensation since no compensation value list exists or it contains errors. See also: [FCT-C5// Section: Temperature compensation] [COMP// Section: General compensation value data]		
Response	Class	7	Stop axis, temperature compensation remains inactive
Solution	Class	6	Correct compensation table.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
Error type	11, Error message from position controller.	

ID 70315

Activation of temperature compensation not possible since axis is not referenced.			
Description	The axis must be homed in order to activate temperature compensation. See [FCT-C5// Section: Temperature compensation]		
Response	Class	1	Warning, temperature compensation remains inactive until axis is homed
Solution	Class	1	Home the axis (see [FCT-M1]) If an absolute measurement system is used, parameter P-AXIS-00014 can be set to 1 to mark the homing of the axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		TRUE, if axis is homed	
Error type	11, Error message from position controller.		

ID 70316

Activation of cross compensation not possible since it is not selected in axis parameter list			
Description	To turn on cross compensation in the NC program for an axis, it must be activated in the axis parameter list (see P-AXIS-00047). See [FCT-C5// Section: Cross compensation]		
Response	Class	7	Stop axis, cross compensation remains inactive
Solution	Class	6	Activate cross compensation (see P-AXIS-00047)
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Parameter value P-AXIS-00047 for activation of cross compensation	
Error type	11, Error message from position controller.		

ID 70317

Activation of cross compensation not possible since compensation table is invalid.			
Description	Could not activate cross compensation since no compensation value list exists or it contains errors. See also: [FCT-C5// Section: Cross compensation] [COMP// Section: Cross compensation]		
Response	Class	7	Stop axis, cross compensation remains inactive
Solution	Class	6	Correct compensation value list
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	

Error type	11, Error message from position controller.

ID 70318

Activation of cross compensation not possible since master axis is not referenced.			
Description	The master axis for cross compensation must be homed in order to activate compensation. The correction values for the slave axis are calculated depending on the position of the master axis. See [FCT-C5// Section: Cross compensation]		
Response	Class	1	Warning, cross compensation remains inactive until master axis is homed
Solution	Class	1	Home the master axis (see [FCT-M1]) If an absolute path measurement system is used for the master axis, parameter P-AXIS-00014 can be set to 1 for the master axis to mark homing of the axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of slave axis affected	
	%2:	Logical axis number [-]	
		Logical axis number P-COMP-00005 of master axis	
	%3:	Error value [-]	
		TRUE if axis is homed	
Error type	11, Error message from position controller.		

ID 70319

Activation of plane compensation not possible since it is not selected in axis parameter list			
Description	To turn on plane compensation in the NC program for an axis, it must be activated in the axis parameter list (see P-AXIS-00174). See [FCT-C5// Section: Plane compensation]		
Response	Class	7	Stop axis, plane compensation remains inactive
Solution	Class	6	Activate plane compensation (see P-AXIS-00174)
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Parameter value P-AXIS-00174 to activate plane compensation	
Error type	11, Error message from position controller.		

ID 70320

Activation of plane compensation not possible since compensation value list is invalid.			
Description	Could not activate plane compensation since no compensation value list exists or it contains errors. See also: [FCT-C5// Section: Plane compensation] [COMP// Section: Plane compensation]		
Response	Class	7	Stop axis, plane compensation remains inactive
Solution	Class	6	Correct Compensation value list see [COMP]

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
Error type	11, Error message from position controller.	

ID 70321

Activation of plane compensation not possible since master axes are not referenced.			
Description	The master axes for plane compensation must be homed in order to activate compensation. The correction values for the slave axis are calculated depending on the positions of the master axes. See [FCT-C5// Section: Plane compensation]		
Response	Class	1	Warning, plane compensation remains inactive until master axes are homed
Solution	Class	1	Home the master axis (see [FCT-M1]) If an absolute measurement system is used for the master axes, parameter P-AXIS-00014 can be set to 1 for the master axes to mark homing of the axes.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of slave axis affected	
	%2:	Logical axis number [-]	
		Logical axis number P-COMP-00014 of the first master axis	
	%3:	Current value [-]	
		TRUE if axis is homed	
	%4:	Logical axis number [-]	
		Logical axis number P-COMP-00015 of the second master axis	
	%5:	Current value [-]	
		TRUE if axis is homed	
Error type	11, Error message from position controller.		

ID 70322 / 70323

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70324

Encoder overflow offset bigger than encoder range.			
Description	The shift of the encoder overflow (P-AXIS-00354) is greater than the distance moved within one encoder revolution (P-AXIS-00355).		
Response	Class	5	Axis stop
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis see P-AXIS-00016
	%2:	Error value [0.1 µm or 0.0001°]
		Parameterised value of P-AXIS-00354.
	%3:	Limit value [0.1 µm or 0.0001°]
		Limit value for P-AXIS-00354.
	%4:	Corrected value [0.1 µm or 0.0001°]
		Corrected value of P-AXIS-00354
Error type	6, Error message by data transfer to control device.	

ID 70325

Axis movement from axis compensation while feedhold.			
Description	With cross and plane compensation, the position of the slave axis is corrected in dependency of the position(s) of the master axis (axes). The axis compensation moves the slave axis although feedhold is set for the axis, e.g. because a master axis is moved in another CNC channel. See [FCT-C5// Section: Overview]		
Response	Class	1	Warning
Solution	Class	1	Possible solutions: • Remove feedhold from slave axis • Stop the master axis (axes) if feedhold is set for slave axis • Turn axis compensation off during feedhold
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned slave axis	
	%2:	Drive type [-]	
		Drive type of affected slave axis, see P-AXIS-00020	
	%3:	Block number [-]	
		Logical axis number of moved master axis P-COMP-00005, P-COMP-00014, P-COMP-00015	
	%4:	Logical axis number [-]	
		Block number of the NC program in the channel of the Master axis, in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70326

Timeout during drive operation mode switch.			
Description	The operation mode of a drive could not be switched within the specified time.		
Response	Class	5	Abrupt stop for the defective axis, feedhold for the other axes.
Solution	Class	6	Use drive documentation to solve reason why mode switch was not executed.

Parameter	%1:	Logical axis number [-]
		Logical axis number of the axis.
	%2:	Current value [-]
		Operation mode requested by CNC
	%3:	Error value [-]
		Current drive operation mode.
	%4:	Limit value [-]
		Time limit
Error type	11, Error message from position controller.	

ID 70328

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	Error message output, the surplus actual positions are ignored.
Solution	Class	8	Requires controller restart.

ID 70329

Sensor signal: step of actual value oversized.			
Description	During active distance control (see [FCT-M3]) the encoder values of the sensor or probe system are monitored due to safety reasons to detect errors in the distance measurement. This error message is issued if the encoder value changes by more than the limit indicated by parameter P-AXIS-00417 within a CNC cycle.		
Response	Class	7	Controlled stop of affected axis. Distance control is turned off.
Solution	Class	6	Check the sensor/probe system
Parameter	%1:	Error value [1µm/s or 0.001°/s]	
		Actual value change of sensor/probe system	
	%2:	Upper limit value [1µm/s or 0.001°/s]	
		Permissible limit value P-AXIS-00417	
	%3:	Logical axis number [-]	
		Axis number P-AXIS-00016 of the concerned axis	
	%4:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis affected	
	%5:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 70330

Sensor is completely extended.			
Description	The current distance control offset (see [FCT-M3]) was frozen. However, the maximum excursion of the probe sensor was reached during axis motion (maximum excursion = P-AXIS-00420 + ½* P-AXIS-00421). A possible cause could be a hole in the workpiece.		
Response	Class	7	Immediate stop of the axis affected to avoid damage to the sensor. Distance control is turned off.

Solution	Class	6	Check the sensor and the workpiece
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current position value of sensor	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Lower limit value for the sensor position	
	%3:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%4:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis affected	
	%5:	Block number [-]	
		Block number in the CNC program where the error occurred	
Error type	-		

ID 70331

Tactile deviation too big.			
Description	The correction value offset calculated by the distance control (see FCT-M3) exceeds the permissible limit. Due to safety reasons, the maximum deviation caused by the distance control between the programmed position in the NC program and the real actual position of the axis can be limited with the parameter P-AXIS-00414. Possible error causes: 1. The deviation between command height (SET_POS) and actual height of the workpiece surface is too large. In this case, the command height should be compensated and the NC program should be regenerated with the new command height. 2. The workpiece may possibly contain a large indentation at this point, e.g. a hole. Deactivate distance control at the position to avoid a collision with the workpiece. In addition, there is a risk of possible damage to the probe if a (contacting) sensor is used. 3. Strong changes in height of workpiece surface. In this case, an attempt can be made to increase the permissible limit P-AXIS-00414. Possibly reduce path velocity so that distance control can follow the strong changes in height.		
Response	Class	7	Immediate axis stop to prevent damaging the sensor
Solution	Class	6	Check workpiece, command position SET_POS of workpiece surface or permissible correction value P-AXIS-00414
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current calculated correction offset	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Maximum permissible deviation P-AXIS-00414	
	%3:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%4:	Current value [0.1 µm or 0.0001°]	
		Current calculated correction offset after interpolator or limiter	
	%5:	Block number [-]	
		Block number in the CNC program where the error occurred	
Error type	-		

ID 70332

Distance control is still active at end of program.			
Description	The end of the NC program (M02/M30) was reached while distance control is still active (see [FCT-M3]). Distance control must be disabled before program end.		
Response	Class	7	Immediate axis stop. Distance control is turned off.
Solution	Class	6	Deactivate distance control at program end.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Block number [-]	
		Block number in the NC program	
Error type	11, Error message from position controller.		

ID 70333

Axis was released with active distance control.			
Description	The axis was released (#PUT AX) but distance control is still active (see [FCT-M3]). Before axis release, distance control must be switched off.		
Response	Class	7	Immediate axis stop. Distance control is turned off.
Solution	Class	6	Deactivate distance control before axis release
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Block number [-]	
		Block number in the CNC program where the error occurred	
	%3:	Error value [-]	
		Current correction offset of distance control	
	%4:	Current value [-]	
		Current state of distance control	
Error type	11, Error message from position controller.		

ID 70334

During deselection of distance control, it was activated again.			
Description	Distance control was deactivated and then re-activated immediately afterwards. After deactivation, wait until the existing compensation offset has finished before re-activating distance control. Alternatively, use the FREEZE command to freeze the current offset so that distance control can stop. [FCT-M3// Section: Programming]		
Response	Class	7	Immediate axis stop. Distance control is not turned on.
Solution	Class	6	Enable distance control only after the correction offset is zero or use the FREEZE command instead.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Block number [-]	
		Block number in the NC program in which the error occurred.	
Error type	11, Error message from position controller.		

ID 70335

Before activating distance control the command height must be specified.			
Description	Before distance control is activated, specify the command height of the workpiece surface SET_POS. Example: Z [DIST_CTRL ON **Before activating distance control the command height must be specified.**] See [FCT-M3// Section: Specifying the workpiece surface]		
Response	Class	7	Immediate axis stop. Distance control is not enabled.
Solution	Class	6	Specify the set height of the workpiece surface SET_POS
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Block number [-]	
		Block number in the NC program where the error occurred.	
Error type	11, Error message from position controller.		

ID 70336

Distance control is not enabled.			
Description	Distance control is not selected for this axis. Distance control must be activated in the axis parameter P-AXIS-00328 in order to send commands (ON, SET_POS) from the NC program or the PLC to the distance control function: `lr_param.distance_control_on 1` See [FCT-M3// Section: Distance control]		
Response	Class	7	Immediate stop of the concerned axis
Solution	Class	6	Correct the axis parameter P-AXIS-00328
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Block number [-]	
		Block number in the NC program	
Error type	11, Error message from position controller.		

ID 70337

Different settings of the time delay for deactivating the drive in case of PLC watchdog error in master and slave axis.			
Description	In parameter P-AXIS-00367, a time delay can be specified after which the drive releases are removed by the CNC at the latest in the event of a PLC watchdog error. The same value must be set for all axes belonging to a gantry system. See [HLI// Section: Watchdog mechanism] (CNC build < 2800) [HLI// Section: Watchdog mechanism] (CNC build > 2800)		
Response	Class	1	Warning and correct time delay for the slave axis
Solution	Class	1	Correct parameter P-AXIS-00367

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of gantry slave axis affected
	%2:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of gantry master axis affected
	%3:	Error value [µs]
		Time delay P-AXIS-00367 of the slave axis
	%4:	Error value [µs]
		Time delay P-AXIS-00367 of master axis
	%5:	Corrected value [µs]
		Corrected time delay P-AXIS-00367 slave axis
Error type	-	

ID 70338

No actual position configured for input axis of axis-specific transformation.			
Description	For an axis-specific actual value transformation, an axis was configured as an auxiliary input which has no position encoder configured.		
Response	Class	5	Error message output, the output variable of the transformation is set to 0 and tracking is set for this axis.
Solution	Class	7	Change configuration, only actual position values are possible as auxiliary input variables for an axis-specific transformation.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis with no position encoder configured.	
	%2:	Drive type [-]	
		Drive type of axis without configured position encoder.	
Error type	-		

ID 70339

Invalid axis number configured for input axis of axis-specific transformation.			
Description	For an axis-specific actual value transformation an auxiliary input variable with an unknown axis number was configured.		
Response	Class	5	Error message output, the output variable of the transformation is set to 0 and tracking is set for this axis.
Solution	Class	7	Change the configuration.
Parameter	%1:	Logical axis number [-]	
		Unknown logical axis number.	
Error type	-		

ID 70340

An error occurred during initialisation of the axis-specific actual value transformation.			
Description	An error occurred during initialisation of the axis-specific actual value transformation.		
Response	Class	5	Error message output, the output variable of the transformation is set to 0 and tracking is set for this axis.
Solution	Class	7	Change the configuration.

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis with actual value transformation.
	%2:	Logical axis number [-]
		Transformation id of actual value transformation.
Error type	-	

ID 70341

Homing is not supported for this kind of axis-specific transformation.			
Description	An axis-specific actual value transformation which does not support homing is active for this axis and homing was started.		
Response	Class	5	Error message output, abort program.
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
		Logical axis number.	
Error type	-		

ID 70342

Error in axis-specific transformation.			
Description	An axis-specific command value transformation is activated P-AXIS-00380. The error can only occur because missing or incomplete parameter settings occurred in an axis-specific command value transformation. For more information on this function, see [AXIS// Section: Command value transformation].		
Response	Class	5	Error message output, abort program.
Solution	Class	6	Check the parameter settings for the selected axis-specific transformation.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		Info variable	
	%3:		
		Current selected transformation ID.	
Error type	-		

ID 70343

For this operation mode the position lag has to be configured in the command value telegram.			
Description	An axis is to be position-controlled in DSC mode. For this operation mode the position lag must be configured in the cyclic command value telegram.		
Response	Class	5	Error message output, reset is locked.
Solution	Class	7	Add position lag to cyclic command value telegram or change operation mode of the axis.

Parameter	%1:	Logical axis number [-]
		Log. axis number of the axis (see also P-AXIS-00016).
	%2:	Drive type [-]
		Drive type of the axis (see also P-AXIS-00020).
	%3:	Current value [-]
		Operation mode configured for the axis (see also P-AXIS-00320).
Error type	-	

ID 70344

For this operation mode the position controller gain has to be configured in the command value telegram.			
Description	An axis is to be position-controlled in DSC mode. For this operation mode the position controller gain must be configured in the cyclic command value telegram.		
Response	Class	5	Error message output, reset is locked.
Solution	Class	7	Add position controller gain to cyclic command value telegram or change operation mode of the axis.
Parameter	%1:	Logical axis number [-]	
		Log. axis number of the axis (see also P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type of the axis (see also P-AXIS-00020).	
	%3:	Current value [-]	
		Operation mode configured for the axis (see also P-AXIS-00320).	
Error type	-		

ID 70345

Only spindles can be operated in open position loop mode.			
Description	One axis is to be operated without position controller (controlled operation). This is only possible for spindles. See also P-AXIS-00320.		
Response	Class	5	Error message output, reset is locked.
Solution	Class	7	Parameterise a different operation mode for the axis (see P-AXIS-00320) and restart the controller.
Parameter	%1:	Logical axis number [-]	
		Log. axis number of the axis (see also P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type of the axis (see also P-AXIS-00020).	
	%3:	Current value [-]	
		Operation mode configured for the axis (see also P-AXIS-00320).	
	%4:	Current value [-]	
		Internal operation mode.	
	%5:	Current value [-]	
		Configured axis type (see P-AXIS-00018)	
Error type	-		

ID 70346

For this operation mode the command position has to be configured in the command value telegram.			
Description	An axis is to be operated by specifying the command position with position controller in the drive. For this operation mode the command position must be configured in the cyclic command value telegram.		
Response	Class	5	Error message output, reset is locked.
Solution	Class	7	Add command position value to configuration of command value telegram or change operation mode of the axis.
Parameter	%1:	Logical axis number [-]	
		Log. axis number of the axis (see also P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type of the axis (see also P-AXIS-00020).	
	%3:	Current value [-]	
		Operation mode configured for the axis (see also P-AXIS-00320).	
Error type	-		

ID 70347

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70348

Delay time for acceleration feedforward is greater than cycle time.			
Description	The parameterised deceleration time for acceleration feedforward control (see also P-AXIS-00390) is greater than or equal to the interpolator cycle time.		
Response	Class	1	Error message output, the delay for acceleration feedforward is disabled (P-AXIS-00390 = 0).
Solution	Class	1	For P-AXIS-00390, parameterise a deceleration value smaller than the interpolator cycle time.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see P-AXIS-00016).	
	%2:	Current value [µs]	
		Incorrect value of P-AXIS-00390.	
	%3:	Corrected value [µs]	
		Corrected value of P-AXIS-00390.	
Error type	2, Error message by data transfer from parameter list into control device..		

ID 70349

Delay time for velocity feedforward is greater than cycle time.	
Description	The parameterised delay time for velocity feedforward (see also P-AXIS-00389) is greater than or equal to the interpolator cycle time.

Response	Class	1	Error message output; the delay time for velocity feedforward is disabled (P-AXIS-00389 = 0).
Solution	Class	1	For P-AXIS-00389, parameterise a deceleration value smaller than the interpolator cycle time.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see P-AXIS-00016).	
	%2:	Current value [µs]	
		Incorrect value of P-AXIS-00389.	
	%3:	Corrected value [µs]	
		Corrected value of P-AXIS-00389.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70350

Velocity feedforward with additive command value active, but no additive velocity command value configured.			
Description	In parameter P-AXIS-00223, velocity feedforward control was activated via an additive command value velocity (bit 0x100 is set), but no additive command value velocity was parameterised in the cyclic process data.		
Response	Class	5	Error message output, velocity feedforward control is disabled (bits 0x100 and 0x001 in P-AXIS-00223 are set to zero).
Solution	Class	6	Either delete bit 0x100 in P-AXIS-00223 or configure an additive command value velocity in the cyclic process data.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see P-AXIS-00016).	
	%2:	Current value [-]	
		Incorrect value of P-AXIS-00223.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00223.	
Error type	-		

ID 70351

Acceleration or jerk feedforward with additive command value active, but no configured additive torque command value.			
Description	In parameter P-AXIS-00223, acceleration feedforward control was activated via an additive torque command value (bit 0x200 is set), but no additive torque command value was parameterized in the cyclic process data.		
Response	Class	5	Error message output, acceleration feedforward control is disabled (bits 0x200 and 0x002 in P-AXIS-00223 are set to zero).
Solution	Class	6	Either clear bit 0x200 in P-AXIS-00223 or configure an additive torque command value in the cyclic process data.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see P-AXIS-00016).	
	%2:	Current value [-]	
		Incorrect value of P-AXIS-00223.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00223.	
Error type	-		

ID 70355

Time constant for balancing filter (P-AXIS-00361) is too small.			
Description	The cut-off frequency of the PT2 element defined by P-AXIS-00361 is greater than half the sampling rate (1/interpolator cycle time).		
Response	Class	1	If this error message occurs at controller start-up, P-AXIS-00361 is set to zero and feedforward balancing is disabled. If this error message occurs when an axis parameter list is updated the new, the incorrect value is discarded.
Solution	Class	1	Enter correct value for P-AXIS-00361; P-AXIS-00361 must be greater than the minimum value specified in parameter 5 of the error message.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see P-AXIS-00016).	
	%2:	Error value [-]	
		Cut-off frequency of the balancing filter in Hertz.	
	%3:	Limit value [-]	
		Maximal allowable cut-off frequency of the balancing filter.	
	%4:	Error value [µs]	
		Parameterised value of P-AXIS-00361.	
	%5:	Lower limit value [µs]	
		Minimum permissible value of P-AXIS-00361.	
Error type	-		

ID 70357

Invalid probing channel programmed.			
Description	With a NC command “MC_TouchProbeChannel” (see also [PROG] a channel number was specified but lies outside the valid range.		
Response	Class	7	Error message output, abort NC program.
Solution	Class	6	Use correct channel number.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
	%5:	Error value [-]	
Error type	11, Error message from position controller.		

ID 70358

X axis is not programmed after the #SYNC IN command.			
Description			
Response	Class	4	
Solution	Class	6	

Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	1, Error message from NC program.	

ID 70359

X axis is not programmed asynchronous after the #SYNC OUT command.			
Description			
Response	Class	4	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 70360

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70361

Length of output process data exceeds length of configured process data.			
Description	During an attempt to write data to the CNC output process data, the permitted process data length was exceeded.		
Response	Class	7	NC program abort, axis stop
Solution	Class	6	Either correct offset or length of data to be written.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Index of the incorrect data segment	
	%3:	Current value [-]	
		Offset of the incorrect data segment	
	%4:	Current value [-]	
		Length of the incorrect data segment	
	%5:	Current value [-]	
		Configured process data length	
Error type	-		

ID 70362

Writing process data while axis is moving is not permitted.			
Description	An attempt was made to write in the process data of the axis while the axis was interpolated. This is not permitted.		
Response	Class	7	Closed-loop controlled stop of axis
Solution	Class	6	Write process data only when axis at standstill.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 70363

Tool change with changing gear ratio only permitted at axis standstill.			
Description	If a tool with a gear stage is replaced [TOOL// Section: Dynamics and gear data], a gear change takes place in the position controller. Therefore, changing a tool with a gear stage is only permitted at standstill due to safety reasons.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Stop the spindle before changing the tool.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Interpolation end (TRUE/FALSE)	
	%3:	Current value [-]	
		Axis in position window (TRUE/FALSE)	
Error type	-		

ID 70364 / 70365

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70366

Permissible range exceeded during tool change with gear change.			
Description	A tool with a gear stage was replaced on a spindle axis [TOOL// Section: Dynamics and gear data]. However, a parameter value exceeded a valid value range during a gear change in the position controller.		
Response	Class	7	Closed-loop controlled stop of axis, gear change is aborted
Solution	Class	6	Correct the parameter value

Parameter	%1:	Upper limit value [-]
		Permissible limit value
	%2:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%3:	Error value [-]
		Invalid parameter value
	%4:	Corrected value [-]
		Corrected parameter value
	%5:	Current value [Increments / 0.1 um]
		Path resolution of the axis, see P-AXIS-00234, P-AXIS-00233
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70367

Tool change with changing gear ratio not permitted for gantry axes.			
Description	Replacing a tool with a gear stage (automatic gear changes in position controller, [TOOL// Section: Dynamics and gear data] is not permissible for gantry axes.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Correct the NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Operation mode P-AXIS-00015 of axis affected.	
Error type	-		

ID 70368

Change of motion direction only permitted at axis standstill.			
Description	The axis parameters P-AXIS-00230 and P-AXIS-00231 can be used to invert the sign of the actual position value or the manipulated variable/position setpoint in the position controller. The value of both parameters can be changed with a parameter update, if the operation mode P-AXIS-00015 ALLOW_RESOULTION_CHANGE = 0x00400000 is set for the axis. This is however permitted only at axis standstill due to safety reasons.		
Response	Class	7	Controlled stop of the axis, parameters are not accepted
Solution	Class	7	Stop the axis before updating the parameters P-AXIS-00230 and P-AXIS-00231.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value for sign reversal of actual position value P-AXIS-00230	
	%3:	Current value [-]	
		Current value of sign reversal of manipulated variable/set position P-AXIS-00231	
	%4:	Error value [-]	
	%5:	Error value [-]	

Error type	11, Error message from position controller.

ID 70369

Tool gear change not possible. Gear ratio not active!			
Description	This error message is a follow-up error. An update is executed for the axis parameters of a spindle axis for which a tool with a gear stage (see [TOOL// Section: Dynamics and gear data]) was just replaced. The gear stage can no longer be activated due to the changed parameters. The tool gear ratio is therefore no longer active.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	7	Correct parameter error, reselect tool
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Active tool gear ratio, see P-TOOL-00016, P-TOOL-00017	
	%3:	Current value [-]	
		Reversal of rotation direction by tool gear see P-TOOL-00018	
	%4:	Current value [-]	
		Path resolution of the axis see P-AXIS-00234, P-AXIS-00233	
Error type	-		

ID 70370

Setting homing position not possible because fieldbus does not deliver valid data.			
Description	The command to set the homing position was sent via the HLI; however, the fieldbus did not supply any valid position actual values for the axis. Setting the homing position is not possible in this situation. See Adopting the homing position [HLI// Section: MCControlBoolUnit_SetReferencePosition] for CNC build < 2800 [HLI// Section: Ir_mc_control.set_reference_position] for CNC Build > 2800		
Response	Class	7	Stop axis.
Solution	Class	6	Set homing position only if actual values valid for the axis are supplied by the fieldbus.
Parameter	%1:	Logical axis number [-]	
		Log. axis number of the axis (see also P-AXIS-00016).	
	%2:	Current value [-]	
		Flag indicates if fieldbus delivers valid actual positions.	
Error type	-		

ID 70371

Restoration of axis position not possible.

Description	The axis position could not be restored from the retain file at controller start-up. Possible reasons are: Version change in the CNC that affected the retain data. The retain data could not be loaded.		
Response	Class	7	Axis is not homed.
Solution	Class	6	Execute axis homing.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Flag that indicated if retain data is valid.	
Error type	-		

ID 70372

Shifting of reference offset with axes in control not possible.			
Description	An attempt has been made to change parameter P-AXIS-00341 although the axis is in control. This is not possible. The reference offset can only be changed for axes if they are disabled.		
Response	Class	2	The changed value of P-AXIS-00341 is not adopted.
Solution	Class	6	Disable axis and change P-AXIS-00341 again.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current value of P-AXIS-00341.	
	%3:	Error value [0.1 µm or 0.0001°]	
		New value of P-AXIS-00341.	
	%4:	Corrected value[-]	
		Corrected value of P-AXIS-00341.	
Error type	3, Error in the communication.		

ID 70373

Configuration mode activated while drive is powered up.			
Description	Configuration mode was activated although the axis is enabled.		
Response	Class	7	Error message output; configuration mode is enabled.
Solution	Class	6	Disable axis before enabling configuration mode.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see P-AXIS-00016).	
	%2:	Current value [-]	
	%3:	Error value [-]	
Error type	-		

ID 70374

Axis in configuration mode is to be interpolated.			
Description	Axis is to be moved although it is in configuration mode.		
Response	Class	7	Error message output, stop of axis
Solution	Class	6	Exit configuration mode before interpolation the axis.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see P-AXIS-00016).	
	%2:	Current value [-]	
Error type	-		

ID 70375

Inconsistent data for spindle revolution monitoring			
Description	Data for spindle speed monitoring is inconsistent. With spindle speed monitoring, limit speed and activation are stored inverted and non-inverted. To compare the two values, the second (inverted value) is re-inverted and then compared with the first value. If these two values differ this error message is output. Possible reasons for this error message are: • Configuration error, data was entered incorrectly in the configuration data. • Error in data transfer.		
Response	Class	5	Error message output, immediate stop of axis
Solution	Class	6	In case of a configuration error, correct the data and restart the controller. In case of a transmission error, reset the controller.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see P-AXIS-00016).	
	%2:	Current value [1µm/s or 0.001°/s]	
		First value.	
	%3:		
		Second (inverted) value, re-inverted, must result in same value as Kennung_2.	
	%4:		
		First value inverted, must result in same value as Kennung_5.	
	%5:		
		Second (inverted) value.	
Error type	11, Error message from position controller.		

ID 70376

Spindle speed monitoring: Maximum permissible speed exceeded.

Description	The actual spindle speed exceeds the parameterised maximum speed by at least 10%. Spindle speed is automatically limited to the maximum speed set in the tool data so this error message should never occur. Possible causes: • Incorrect configuration of velocity scaling either in the controller or in the drive. • Malfunction in the drive. • Insufficient parameter setting of speed or position control loop causing overshoot in the actual speed by more than 10%.		
Response	Class	5	Stop axis.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see P-AXIS-00016).	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current actual axis speed.	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum permissible axis speed.	
	%4:	Current value [-]	
		Current actual axis speed in increments per cycle.	
Error type	11, Error message from position controller.		

ID 70377

The actual value simulation was activated when speed monitoring was active.			
Description	Simulation of actual value positions was activated with active spindle speed monitoring. The actual value simulation is activated when the current spindle speed is higher than the maximum encoder velocity set in P-AXIS-00220. When the encoder velocity limit is exceeded, it is assumed that the encoder does not deliver valid actual values any more; monitoring the spindle speed is not possible in this case.		
Response	Class	7	Program abort
Solution	Class	6	Increase P-AXIS-00220 or reduce the maximum spindle speed.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
Error type	11, Error message from position controller.		

ID 70378

Invalid start position in axis-specific command position transformation.			
Description	When using axis-specific transformationsP-AXIS-00380, an invalid position jump was detected. A change of the two solution variants or wrong ACS axis position, when selected at the crank, leads to a position jump. The permissible position jump can be modified with transformation parameters; the default value is 5 degrees. This error message is generated if the current active position difference exceeds the default or adjusted parameter.		
Response	Class	5	Axis motion stop
Solution	Class	6	Correct parameter P-AXIS-00382

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of the axis affected
	%2:	Current value [-]
		Internal error code
	%3:	Current value [-]
		Parameterised transformation ID P-AXIS-00381
	%4:	Current value [0.1 µm or 0.0001°]
		MCS input coordinate of transformation
	%5:	Current value [0.1 µm or 0.0001°]
		ACS output coordinate of transformation
Error type	-	

ID 70379

Maximum distance for zero pulse search is smaller than the encoder range.			
Description	The maximum distance set in the parameter P-AXIS-00404 for zero pulse search or to search for encoder overflow is smaller than the distance covered by an encoder revolution. The distance covered in an encoder revolution can be defined by: 1. When the parameters P-AXIS-00092, P-AXIS-00362 and P-AXIS-00363 are used to parameterise path resolution, the parameter P-AXIS-00092 contains the number of encoder increments per motor revolution. 2. When homing on encoder overflow, the number of increments per encoder revolution is defined by the minimum value of P-AXIS-00355 and the modulo range (P-AXIS-00126 and P-AXIS-00127) of the axis.		
Response	Class	5	Correct the value of P-AXIS-00404 to the distance the axis moves during an encoder revolution; output of a warning.
Solution	Class	1	Correct the value of P-AXIS-00404 so that the axis can move at least one encoder revolution during the zero pulse search.

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of affected axis
	%2:	Current value [0.1 µm or 0.0001°]
		Distance moved within one encoder revolution.
	%3:	Error value [0.1 µm or 0.0001°]
		Set value of P-AXIS-00404.
	%4:	Corrected value [-]
		Corrected value of P-AXIS-00404.
Error type	6, Error message by data transfer to control device.	

ID 70380

Maximum permissible path exceeded for zero pulse search or encoder overflow search.

Description	During homing, the maximum permissible path set in parameter P-AXIS-00404 for searching the zero pulse or the encoder overflow was exceeded. Possible reasons for this error message are: • Invalid set value of P-AXIS-00404. • Encoder defective or incorrectly wired. • Incorrect parameterisation of drive amplifier with respect to encoder evaluation. • Incorrect configuration of process data between drive and CNC.		
Response	Class	7	Stop of axis, output of error message.
Solution	Class	6	Possible solutions: • Set correct value of P-AXIS-00404 • Replace encoder or wire correctly. • Correct parameterisation of encoder evaluation in drive amplifier • Check and correct configuration of process data between drive and CNC

Parameter	%1:	Logical axis number [-]
		Axis number (P-AXIS-00016).
	%2:	Current value [-]
		Maximum permissible path during zero pulse search (value of P-AXIS-00404)
	%3:	Limit value [-]
		Covered distance since start of zero pulse search.
	%4:	Current value [-]
		Current actual axis position in increments.
	%5:	Current value [-]
		Axis position at start of zero pulse search in increments.
Error type	1, Error message from NC program.	

ID 70381

Actual value request when back interpolation is active.			
Description	During the backward interpolation of an axis after tracking mode actual positions were requested with the command # CHANNEL_WR INIT [ACTPOS ....] .		
Response	Class	7	Program execution stop
Solution	Class	6	Change NC program sequence or PCL program. When actual positions are requested with the command #CHANNEL INT [ACTPOS...], they are tracked internally; it is therefore necessary to activate tracking mode via the HLI. [HLI// lr_mc_control.follow_up]

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of axis concerned.
	%2:	Block number [-]
		Block number in the NC program.
Error type	11, Error message from position controller.	

ID 70382

Maximum permissible axis velocity exceeded.

Description	The dynamic monitoring in the position controller has detected that the permissible axis velocity was exceeded. The permissible limit value can be specified in parameter P-AXIS-00407 .		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Possible solutions: • If the additional interface is used in the position controller (see [HLI// Section: External commanding of an axis]), check the specified command curve for velocity excesses. • Increase the permissible velocity limit value P-AXIS-00407 in dynamic monitoring if the maximum velocity of the drive is not exceeded as a result.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Error value[1µm/s or 0.001°/s]
		Incorrect axis velocity
	%3:	Upper limit value [1µm/s or 0.001°/s]
		Maximum permissible axis velocity (P-AXIS-00407 * P-AXIS-00212)
	%4:	Current value [1µm/s or 0.001°/s]
		Resulting velocity of additional interface of position controller (see [HLI// Chapter External commanding of an axis])
	%5:	Current value [-]
		Ratio between current and maximum permissible axis velocity P-AXIS-00212
Error type	11, Error message from position controller.	

ID 70383

The scaled limit value is bigger than the maximum value of the configured data type.			
Description	The scaled (see P-AXIS-00401) value of parameter P-AXIS-00409 is greater than the maximum value of the data type set with parameter P-AXIS-00399.		
Response	Class	1	Warning output, P-AXIS-00409 is automatically limited to the maximum value according to P-AXIS-00399.
Solution	Class	1	Adjust P-AXIS-00409.

Parameter	%1:	Logical axis number [-]
	%2:	Current value [-]
		Value of P-AXIS-00396 for identification of drive function.
	%3:	Error value [-]
		Parameterised value of P-AXIS-00409.
	%4:	Current value [-]
		Scaling factor (see P-AXIS-00401).
	%5:	Limit value [-]
		Maximum possible value for P-AXIS-00409 considering P-AXIS-00399 and P-AXIS-00401 simultaneously corrected value.
Error type	11, Error message from position controller.	

ID 70384

The scaled limit value is smaller than the minimum value of the configured data type.			
Description	The scaled (see P-AXIS-00401) value of parameter P-AXIS-00408 is smaller than the minimum value of the data type set with parameter P-AXIS-00399.		
Response	Class	1	Warning output, P-AXIS-00408 is automatically limited to the minimum value according to P-AXIS-00399.
Solution	Class	1	Adjust P-AXIS-00408.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Value of P-AXIS-00396 for identification of drive function.	
	%3:	Error value [-]	
		Parameterised value of P-AXIS-00408.	
	%4:	Current value [-]	
		Scaling factor (see P-AXIS-00401).	
	%5:	Limit value [-]	
		Minimum possible value for P-AXIS-00408 taking into account P-AXIS-00399 and P-AXIS-00401, simultaneously corrected value.	
Error type	11, Error message from position controller.		

ID 70385

The configured maximum value is less than the configured minimum value.			
Description	The parametrised minimum value (P-AXIS-00408) is greater than the parametrised maximum value (P-AXIS-00409).		
Response	Class	1	Warning output, P-AXIS-00408 and P-AXIS-00409 are swapped.
Solution	Class	1	P-AXIS-00408 and P-AXIS-00409 must be adjusted.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Value of P-AXIS-00396 for identification of drive function.	
	%3:	Upper limit value [-]	
		Parameterized value of P-AXIS-00409.	
	%4:	Lower limit value [-]	
		Parameterized value of P-AXIS-00408.	
Error type	11, Error message from position controller.		

ID 70386

The use of a DC brake is only possible for spindles.			
Description	Parameter P-AXIS-00410 was set to a value not equal to –1 (DC brake control was enabled), although the axis is not a spindle.		
Response	Class	1	Warning output, P-AXIS-00410 is set to –1.
Solution	Class	1	The use of a DC brake is only possible for spindles. Deactivate DC brake by setting P-AXIS-00410 to -1.

Parameter	%1:	Logical axis number [-]
	%2:	Current value [-]
		Configured axis type, see P-AXIS-00018.
	%3:	Error value [-]
		Configured value of P-AXIS-00410.
	%4:	Corrected value [-]
		Corrected value of P-AXIS-00410.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70387

Invalid bit number for control of DC brake.			
Description	An invalid value was used for the parameter P-AXIS-00410.		
Response	Class	1	Warning output, P-AXIS-00410 is set to –1 (DC brake control is disabled):
Solution	Class	1	Use correct bit number, see P-AXIS-00410.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
		Incorrect value of P-AXIS-00410.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00410.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70388

DC brake control is not supported for this drive type.			
Description	DC brake control was activated (P-AXIS-00410 was assigned a value not equal to -1), although the configured drive type does not support control of a DC brake.		
Response	Class	1	Warning output, P-AXIS-00410 is set to –1 (DC brake control is disabled):
Solution	Class	1	P-AXIS-00410 must be set to -1.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value [-]	
		Configured drive type (see P-AXIS-00020).	
	%3:	Error value [-]	
		Incorrect value of P-AXIS-00410.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00410.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70389

Axis reference due to bus failure removed. Position monitoring functions (e.g. software limit switch) not active!

Description	The number of permissible telegram failures on the EtherCAT fieldbus (see P-AXIS-00406) was exceeded. Axes without an absolute measurement system (see P-AXIS-00014) will be marked as not homed since position information may be lost due to the bus failure. The monitoring functions, e.g. software limit switch or gantry monitors, are deactivated for this axis. The axis must be re-homed for the monitoring functions to become effective (see [FCT-M1]). See: Software limit switch (SLS) monitoring [FCT-A2// Section: Description] Gantry monitors [FCT-C11// Section: Overview]		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Possible solutions: • Re-homing axis e.g. with NC command G74 • Check the fieldbus • Increase the permissible telegram failures (see P-AXIS-00406)

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Drive type [-]
		Drive type P-AXIS-00020 of the affected axis
	%3:	Current value [-]
		Last homing state of axis (TRUE = axis is homed)
	%4:	Corrected value [-]
		Homing state after bus failure (FALSE = axis is not homed)
	%5:	Current value [-]
		Value of bus failure counter
Error type	11, Error message from position controller.	

ID 70390

Changing measuring input for drive probing not possible for this drive type.			
Description	In the NC program, the #MEAS command can be used to change the probing input during position detection in the drive. However, this is not possible for some drive types e.g. like SERCOS since for these drives there must be several parameters changed in the drive itself. For further information, see: [PROG// Section: Settings for measurement]		
Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Correct the NC program

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Drive type [-]
		Drive type affected P-AXIS-00020
	%3:	Current value [-]
		Current probing method
	%4:	Error value [-]
		Programmed probing input
	%5:	Corrected value [-]
		Corrected probing input

Error type	11, Error message from position controller.

ID 70391

Changing measuring edge for drive probing not possible for this drive type.			
Description	The #MEAS command can be used to change the measuring edge (rising or falling) in the NC program. However, this is not possible with some drives (e.g. like SERCOS) when the probing value is latched in the drive, since in this case there must be several parameters changed in the drive itself. For further information, see: [PROG// Section: Settings for measurement]		
Response	Class	7	Controlled stop of axis affected
Solution	Class	6	Correct the NC program
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type affected P-AXIS-00020	
	%3:	Current value [-]	
		Current probing method	
	%4:	Error value [-]	
		Programmed probing input	
	%5:	Corrected value [-]	
		Corrected probing input	
Error type	11, Error message from position controller.		

ID 70393

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70394

Maximum admissible distance for homing exceeded.			
Description	During homing, the axis has moved further than the permissible distance set in parameter P-AXIS-00412 from the start position of homing. This is an indication that the homing sequence is disturbed, e. g. the reference cam was not found.		
Response	Class	7	Abort program, output error message.
Solution	Class	6	Remove reason for homing fault.

Parameter	%1:	Logical axis number [-]
	%2:	Limit value [0.1 µm or 0.0001°]
		Set value of P-AXIS-00412.
	%3:	Error value [0.1 µm or 0.0001°]
		Distance from start position of homing.
	%4:	Current value [-]
		Current command position.
	%5:	Current value [0.1 µm or 0.0001°]
		Modulo range of the axis.
Error type	1, Error message from NC program.	

ID 70395

Life sign error PROFIBUS slave.			
Description	A life sign error was detected in the Profibus slave. The expected valuer of the life sign counter does not match the value supplied by the slave. Since the absolute error count is below the limit defined by P-AXIS-00162, only this warning is output. The output of this warning can be suppressed by setting the axis parameter P-AXIS-00462.		
Response	Class	1	Warning output.
Solution	Class	1	Find and correct the cause of the life sign error. Possible reasons are: • Erroneous parametrisation of Profibus driver or Profibus slave. • Erroneous bus wiring. • EMC problems of bus wiring.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis concerned.	
	%2:	Current value [-]	
		Old value of life sign error counter	
	%3:	Current value [-]	
		Current value of life sign error counter	
	%4:	Upper limit value [-]	
		Error limit of life sign error counter (value of P-AXIS-00162)	
Error type	11, Error message from position controller.		

ID 70396

Denominator of axis coupling factor is zero (nil).			
Description	A axis coupling was set up over the HLI, with the value zero (nil) in the denominator for the fraction oft the coupling factor. See also [HLI// Section: Commanding axis couplings] for CNC build < 2800 [HLI// Section: Commanding axis couplings] for CNC build > 2800		
Response	Class	7	Error message output, axis coupling is not activated.
Solution	Class	6	Change PLC program; coupling factor denominator cannot be zero (nil).

Parameter	%1:	Logical axis number [-]
		Logical axis number.
	%2:	Current value [-]
		Index of the erroneous coupling definition on the HLI, starting with 1.
	%3:	Current value [-]
		Numerator of the coupling factor parameterized on the HLI.
	%4:	Error value [-]
		Denominator of coupling factor fraction, incorrect value.
Error type	-	

ID 70397

The coupling factor for the axis is greater than the permissible maximum value.			
Description	A coupling factor greater than the maximal permissible coupling factor was specified when the axis coupling was activated via the HLI See also [HLI// Section: Commanding axis couplings] for CNC build < 2800 [HLI// Section: Commanding axis couplings] for CNC build > 2800		
Response	Class	7	Error message output, axis coupling is not activated.
Solution	Class	6	The coupling factor must be smaller than the maximum permissible coupling factor.

Parameter	%1:	Logical axis number [-]
		Logical axis number.
	%2:	Current value [-]
		Index of invalid coupling specifications on the HLI starting with 1.
	%3:	Limit value [-]
		Limit value.
	%4:	Error value [-]
		Numerator of the coupling factor parameterised on the HLI.
	%5:	Error value [-]
		Denominator of the coupling factor parameterised on the HLI.
Error type	-	

ID 70398

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70399

Invalid transition for distance control.			
Description	An invalid state transition was commanded for distance control in the PLC. See [FCT-M3//Section: Distance control states and transitions]		
Response	Class	7	Controlled stop of axis affected

Solution	Class	6	Correct PLC program
Parameter	%1:	Logical axis number [-]	
	%2:	Block number [-]	
Error type	-		

ID 70400

Axis lost its reference.			
Description	With active reference monitoring, the bitwise AND-conjunction of the input data configured with P-AXIS-00426 and the checkmask set up with P-AXIS-00425 results in the value zero. This means the axis has lost its reference see [FCT-M1].		
Response	Class	7	Error message output, stop of axis.
Solution	Class	6	Remove cause for reference loss and reference axis again. Possible reasons are: • Reference loss caused by a change of drive parameters during drive start-up.→ Re-home axis after completion of parameterisation. • Undervoltage of the buffer battery for absolute encoders with battery buffering.→ Replace battery. • The encoder limit frequency of the encoder used by the drive was exceeded.→ Reduce axis velocity.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
		Value of the reference signal supplied by the drive (value of the input process date set in P-AXIS-00426).	
	%3:	Current value [-]	
		Checkmask calculated from P-AXIS-00425 .	
Error type	-		

ID 70401

Name for axis reference test not configured in cyclic input process data.			
Description	The input name set in P-AXIS-00426 was not found in the configured input process data of the drive. If the reference signal is transmitted in a real-time status bit of the drive, enter the value "S-0-0135" in parameter P-AXIS-00426. In this case the drive status word need not be set up explicitly in the input process data because it is always transferred automatically, see [FCT-M1].		
Response	Class	5	Warning output, axis reference test is disabled by assigning the value "" to P-AXIS-00426.
Solution	Class	1	Either change P-AXIS-00426 to a value configured in the process data of the drive, or extend the process data of the drive by the element entered in P-AXIS-00426.

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of axis concerned.
	%2:	Error value [-]
		Parameterised value in P-AXIS-00426.
Error type	-	

ID 70402

Setting/resetting of bits with #DRIVE command is not supported for this communication type.			
Description	In axis parameter P-AXIS-00399, 'BITARRAY_16' or 'BITARRAY_32' was configured as data type, although bitwise writing of drive values is not supported for the communication type configured in P-AXIS-00397. See also: [PROG// Section: Drive-independent switching of drive functions		
Response	Class	7	Error message output, evaluation of drive functions is aborted.
Solution	Class	6	Customise either P-AXIS-00399 or P-AXIS-00397.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Name of the configured drive function (P-AXIS-00396).	
	%3:	Drive type [-]	
		Drive type (P-AXIS-00020).	
	%4:	Error value [-]	
		Configured communication type (P-AXIS-00397).	
	%5:	Error value [-]	
		Configured data type (P-AXIS-00399).	
Error type	-		

ID 70403

The configured bitmask is too big for the configured data type.			
Description	The bitmask configured in P-AXIS-00429 is larger than the maximum value of the data type configured in P-AXIS-00399.		
Response	Class	7	Error message output, evaluation of drive functions is aborted.
Solution	Class	6	Adjust P-AXIS-00429.
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
		Name of the configured drive function (P-AXIS-00396).	
	%3:	Error value [-]	
		Configured value of P-AXIS-00429	
	%4:	Current value [-]	
		Configured data type (P-AXIS-00399).	
	%5:	Error value [-]	
		Permissible maximum value of P-AXIS-00429	
Error type	-		

ID 70404

Configured bitmask contains bits that are used also by the CNC.			
Description	The bitmask configured in P-AXIS-00429 for bitwise writing of drive values contains bits which are also used by the CNC.		
Response	Class	7	Error message output, evaluation of drive functions is aborted.
Solution	Class	6	Adjust P-AXIS-00429.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Name of the configured drive function (P-AXIS-00396).	
	%3:	Current value [-]	
		Configured value of P-AXIS-00429	
	%4:	Current value [-]	
		Configured data type (P-AXIS-00399).	
	%5:	Error value [-]	
		Permissible maximum value of P-AXIS-00429	
Error type	11, Error message from position controller.		

ID 70405

The telegram element configured for the #DRIVE command is used exclusively by the CNC.			
Description	The telegram element configured for bitwise writing (P-AXIS-00398) in a drive function is used exclusively by the CNC. See also: [PROG// Section: Drive-independent switching of drive functions		
Response	Class	1	Error message output, evaluation of drive functions is aborted.
Solution	Class	1	Adjust P-AXIS-00398.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Name of the configured drive function (P-AXIS-00396).	
	%3:	Error value [-]	
		Configured value of P-AXIS-00398	
Error type	-		

ID 70406

Number of configured user idents exceeds limit.			
Description	During the configuration of drive functions, the number of telegram elements (P-AXIS-00398) configured to be written in the cyclic process data exceeded the maximum limit See [FCT-A10// Section: Description]		
Response	Class	7	Program abort.
Solution	Class	6	Adjust value of VAL in the NC program.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis see P-AXIS-00016
	%2:	Drive type [-]
		Drive type (P-AXIS-00020).
	%3:	Limit value [-]
	%4:	Limit value [-]
	%5:	Current value [-]
		Name of drive function where the excess occurred.
Error type	-	

ID 70410

Target axis is its own source axis, calculation of command position is not possible.			
Description	When an axis coupling is activated via the HLI (see FCT-A9) it was detected that parameter P-AXIS-00436 was set for the axis and the axis is set by other active axis couplings where P-AXIS-00436 is also set for the respective target axis with which it is linked to itself as a source axis. In such coupling loops it is not always possible to map the drive position clearly to the interpolator position without ambiguity so that such coupling loops are prohibited.		
Response	Class	7	Error message output, axis coupling is not activated.
Solution	Class	6	Either break coupling loop by changing the axis coupling or set for at least one axis of the coupling loop parameter P-AXIS-00436 to zero.
Parameter	%1:	Logical axis number [-]	
		Axis number of axis for which the coupling loop was detected.	
	%2:	Logical axis number [-]	
		Axis number of the axis that closes the coupling loop.	
	%3:	Logical axis number [-]	
Error type	-		

ID 70411

Filter of additional command values interface could not be generated.			
Description	To smooth command values, a PT2 filter can be activated in the position controller for the additive command value interface. However, it is not possible to generate the filter with the current filter time constant P-AXIS-00438. The exact cause of the error can be found in the previous error message. See also: [HLI// Section: External axis commanding] for CNC build < 2800 [HLI// Section: External axis commanding] for CNC build > 2800		
Response	Class	7	Filter is deactivated
Solution	Class	6	Change time constant P-AXIS-00438 of PT2 filter

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	Current value [µs]
		Time constant P-AXIS-00438 of the PT2 filter used
	%4:	Error value [-]
Error type	-	

ID 70412 - 70419

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70420

Characteristic filter frequency greater than half the sampling rate.			
Description	According to Shannon's sampling theorem, the maximum cut-off frequency is (½*T_A= cycle time of the NC controller). The cut-off frequency of the PT2 filter is calculated from the time constant to obtain f_g = 1 / (2 * PI * P-AXIS-00438).		
Response	Class	7	The filter is deactivated.
Solution	Class	6	Possible solutions: • Increase time constant P-AXIS-00438 of PT2 filter • Reduce the cycle time of the NC controller
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of axis affected
	%2:		Current value [-]
			Number of filter affected
	%3:		Current value [-]
			Type of filter (see axis filter FCT-A7)
	%4:		Error value [Hz]
			Cut-off frequency of the filter
	%5:		Current value[-]
			Sampling rate of the NC (1 / T_A)
Error type	-		

ID 70421

Filter design with given parameters not possible.			
Description	It is not possible to design a stable filter for all filter parameters, e.g. due to the limited calculation accuracy.		
Response	Class	7	Filter is deactivated

Solution	Class	6	Change parameterisation: • Vary the filter time constant P-AXIS-00438 of the PT2 filter. • Change the cycle time of the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [-]	
		Number of concerned filter	
	%3:	Current value [-]	
		Type of filter (see axis filter FCT-A7)	
	%4:	Current value [-]	
		Order of the filter (see axis filter FCT-A7)	
	%5:	Current value [-]	
		Time constant of the PT2 filter (see P-AXIS-00438)	
Error type	-		

ID 70422

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70423

Stability reserve of designed filter not sufficient.			
Description	It is not possible to design a stable filter for all filter parameters, e.g. due to the limited calculation accuracy.		
Response	Class	7	Filter is deactivated
Solution	Class	6	Possible solutions: • Vary the filter time constant P-AXIS-00438 of the PT2 filter. • Change the cycle time of the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Number of filter affected	
	%3:	Current value [-]	
		Type of filter (see axis filter FCT-A7)	
	%4:	Current value [-]	
		Order of the filter (see axis filter FCT-A7)	
	%5:	Current value [-]	
		Time constant of the PT2 filter (see P-AXIS-00438)	
Error type	-		

ID 70424

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70425

Timeout after deactivation of additive command value interface. Filter not within position window.			
Description	To smooth command values, a PT2 filter can be activated in the position controller for the additive command value interface (see P-AXIS-00438). if the interface is disabled, the axis can run through the filter slightly. The CNC generates this warning message if the axis is not at standstill within a certain amount of time. See also: [HLI// Section: External axis commanding] for CNC build < 2800 [HLI// Section: External axis commanding] for CNC build > 2800		
Response	Class	1	Warning output
Solution	Class	1	Reduce time constant P-AXIS-00438 of PT2 filter
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Upper limit value [µs]	
		Elapsed time after deactivation of the additive command value interface	
	%3:	Upper limit value [-]	
		Elapsed CNC cycles after deactivation of the additive command value interface	
	%4:	Current value [0.1 µm or 0.0001°]	
		Position lag: Filter output - filter input	
	%5:	Current value[-]	
		Time constant P-AXIS-00438 of the PT2 filter used	
Error type	11, Error message from position controller.		

ID 70426

Timeout while reading or writing the drive operation mode.			
Description	Drive-controlled homing with CANopen drives requires that the CNC changes the operation mode in the object 0x6060 to homing mode (6) and afterwards back to the original operation mode. To do this, the CNC reads the active operation mode in 0x6061 via SDO and writes the desired operation mode to 0x6060. This error message occurs when the reading or writing operation does not terminate within a given time period.		
Response	Class	5	Stop axis immediately
Solution	Class	6	Check the fieldbus and the drive

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Drive type [-]
		Drive type P-AXIS-00020 of the affected axis
	%3:	Upper limit value [µs]
		Maximum time durance for the SDO read or write operation
	%4:	Upper limit value [-]
		Maximum CNC cycles for the SDO read or write operation
Error type	-	

ID 70427

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70428

No information about bus state (WC_SATE) available in the drive telegram.			
Description	In the cyclic data that the controller receives from the drive (process data), there is no information about the state of the EtherCAT fieldbus (WC_STATE, State of working counter). The axis cannot be moved after controller start-up (follow-up error P-ERR-50025 [▶ 778]). Example of axis parameter list: **antr.canopen.in[0].signal_nr 0** **antr.canopen.in[0].signal_len 2** **antr.canopen.in[0].nc_ref WCSTATE** antr.canopen.in[1].signal_nr 0 antr.canopen.in[1].signal_len 2 antr.canopen.in[1].nc_ref DRIVE_STATUS ...		
Response	Class	5	Immediate axis stop
Solution	Class	7	Correct configuration
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis.	
Error type	-		

ID 70429

Reading generic drive interface failed.			
Description			
Response	Class	3	
Solution	Class	6	

Error type	-

ID 70430

Distance between two interpolation points of plane compensation is zero (interval).			
Description	In the compensation list of Plane compensation, the distance between the interpolation points P-COMP-00009 was set to zero. Example: **kw.crosscomp2.interval 0** kw.crosscomp2.last_index_master1 100 kw.crosscomp2.last_index_master2 100 kw.crosscomp2.start_position_master1 -400000 kw.crosscomp2.start_position_master2 -700000 kw.crosscomp2.unit 1 kw.crosscomp2.n_cycles 10 kw.crosscomp2.master1_ax_nr 1 kw.crosscomp2.master2_ax_nr 2 Correct: **kw.crosscomp2.interval 100000** kw.crosscomp2.last_index_master1 100 kw.crosscomp2.last_index_master2 100 kw.crosscomp2.start_position_master1 -400000 kw.crosscomp2.start_position_master2 -700000 kw.crosscomp2.unit 1 kw.crosscomp2.n_cycles 10 kw.crosscomp2.master1_ax_nr 1 kw.crosscomp2.master2_ax_nr 2 See [COMP// Section: Plane compensation]		
Response	Class	7	Closed-loop controlled axis stop and deactivate plane compensation
Solution	Class	7	Correct parameter P-COMP-00009 for the distance.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of axis affected
	%2:		Error value [0.1 µm or 0.0001°]
			Invalid distance P-COMP-00009
	%3:		Lower limit value [0.1 µm or 0.0001°]
			Minimum permissible interpolation point distance P-COMP-00009
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70431

Compensation table of leadscrew error compensation incorrect.

Description	The leadscrew error compensation (see P-AXIS-00175) is activated for the axis. However, the compensation table contains errors; it is therefore not possible to activate compensation. See FCT-C5// Section: Leadscrew error compensation COMP// Section: Leadscrew error compensation		
Response	Class	7	Closed-loop controlled stop of axis, disable leadscrew error compensation
Solution	Class	7	Check previous warning message(s) and correct compensation value list of leadscrew error compensation
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	11, Error message from position controller.		

ID 70432

Parametrized greatest index of cross compensation table exceeds limits.			
Description	In the compensation value list, parameter P-COMP-00004 for cross compensation specifies the value (starting with 0) of the index of the last valid entry in the compensation value list for cross compensation. During interpretation of that list by the CNC it is checked if the specified index is between the limits that are provided for cross compensation. See [COMP// Section: Cross compensation]		
Response	Class	7	Closed-Loop controlled stop of axis, the cross compensation is turned off.
Solution	Class	7	Check the value in parameter P-COMP-00004 The value must be within the specified limits of the error message.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned slave axis	
	%2:	Error value [-]	
		Value specified by the operator in parameter P-COMP-00004 for the index of the last valid entry in the compensation value list for cross compensation.	
	%3:	Lower limit value [-]	
		Under limit value, P-COMP-00004 must be greater than or equal to this value.	
	%4:	Upper limit value [-]	
		Upper limit value, P-COMP-00004 must be less than or equal to this value.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70433

Parametrised greatest index of plane compensation table exceeds limits.

Description	In the compensation value list, the parameters P-COMP-00010 and P-COMP-00011 for plane compensation specify the value (starting with 0) of the index of the last valid entry in the compensation value list for plane compensation for the corresponding axis. When the CNC interpreted this list, it checked whether the specified indices are between the limits provided for plane compensation. See [COMP// Section: Cross compensation]		
Response	Class	7	Closed-loop controlled stop of axis, disable plane compensation
Solution	Class	7	Check the values of parameters P-COMP-00010, P-COMP-00011. The values must be within the two limits specified in the error message.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of slave axis affected	
	%2:	Error value [-]	
		Value specified by the operator in parameter P-COMP-00010 for the index of the last valid entry in the compensation value list for plane compensation for the first axis.	
	%3:	Lower limit value [-]	
		Lower limit value, the value of parameter P-COMP-00010 or P-COMP-00011 must be greater than or equal to this value.	
	%4:	Upper limit value [-]	
		Upper limit value, the value of parameters P-COMP-00010 or P-COMP-00011 must be less than or equal to this value.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70434

Coupled axis has to be homed after error.			
Description	An axis coupling is active via the HLI; the axis parameter P-AXIS-00436is set for the target axis of the axis coupling. One of the axes participating in the axis coupling was in error. All the participating axes must be homed in order to revert the coupling group to a consistent state. An attempt was made to move one of the axes of the coupling group without previously homing it.		
Response	Class	7	Error message output, abort NC program.
Solution	Class	6	An axis of the coupling group must be homed before it is moved.
Parameter	%1:	Logical axis number [-]	
		Axis number of the target axis of the coupling group.	
	%2:	Logical axis number [-]	
		Axis number of the source axis of the coupling group to be moved.	
Error type	11, Error message from position controller.		

ID 70435

Not all axis compensations required for program processing are active.

Description	The axis parameter P-AXIS-00465 specifies the required axis compensations. A necessary precondition was not met for at least one of these axis compensations. The corresponding axis compensation is not active. The currently effective compensations can be taken from parameter 2:		
	Bit	**Not all axis compensations required for program processing are active.**	
	0x1	Backlash compensation seeFCT-C5//Backlash compensation	
	0x2	Leadscrew error compensation see FCT-C5//Leadscrew error compensation	
	0x4	Temperature compensation see FCT-C5//Temperature compensation	
	0x8	Cross compensation see FCT-C5//Cross compensation	
	0x10	Plane compensation see FCT-C5//Plane compensation	
	Required preconditions of the compensations are: 1. Error-free compensation table see [COMP//Section: General compensation value data] 2. The axis must be homed for leadscrew error compensation and temperature compensation. 3. Cross and plane compensation require that the master axes are homed. 4. Cross and plane compensation require that the drive releases are set before starting the CNC program.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the required preconditions for the inactive compensation.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Bit-coded information of the axis compensation currently enabled for the axis.	
	%3:	Expected value [-]	
		Bit-coded information of the axis compensations required for the axis (see P-AXIS-00465)	
	%4:	Current value [-]	
		TRUE if axis is homed.	
	%5:	Current value [-]	
		TRUE if drive releases for the axis are set.	
Error type	11, Error message from position controller.		

ID 70436

Drive following error not configured in cyclic process data.			
Description	Parameter P-AXIS-00466 can be used to indicate that the CNC must use the position lag from the drive. However, use of the drive position lag requires that position control takes place in the drive (see P-AXIS-00320).		
Response	Class	1	Warning, lag is calculated in at the CNC.
Solution	Class	1	Correct parameter P-AXIS-00466 or change the operation mode P-AXIS-00320

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Drive type [-]
		Drive type P-AXIS-00320 of the concerned axis
	%3:	Error value [-]
		Current operation mode P-AXIS-00320
	%4:	Expected value [-]
		Expected operation mode P-AXIS-00320
	%5:	Corrected value [-]
		Corrected value for P-AXIS-00466
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70437

Drive following error cannot be used in this operation mode.			
Description	Parameter P-AXIS-00466 can be used to indicate that the CNC must use the position lag from the drive. However, the drive position lag is not contained in the cyclic process data: • SERCOS-Ident S-0-0189 • CANopen Objekt 0x60F4		
Response	Class	1	Warning, lag is calculated in at the CNC.
Solution	Class	1	Correct parameter P-AXIS-00466 or add drive position lag to the cyclic process data.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Corrected value [-]	
		Corrected value for P-AXIS-00466	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70438

Drive following error cannot be used with PROFIDdrive MC.			
Description	Parameter P-AXIS-00466 can be used to indicate that the CNC must use the position lag from the drive. However, this is not supported for PROFIdrive MC drives at the moment.		
Response	Class	1	Warning, the position lag error is calculated in the CNC
Solution	Class	1	Correct parameter P-AXIS-00466
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Corrected value [-]	
		Corrected value for P-AXIS-00465	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70439

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires controller restart.

ID 70440

Mode change while distance control is active.			
Description	The distance control function supports two different operation modes (see FCT-M3) 1. "Surface" mode: Control on the workpiece surface, HLI_DIST_CTRL_ON 2. "Distance" mode: Distance control:HLI_DIST_CTRL_ON_CONST_DIST If the distance control is active, a change of the operation mode is not allowed without previously turning the distance control off (HLI_DIST_CTRL_OFF) or freezing the actual offset (HLI_DIST_CTRL_FREEZE).		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	6	Correct the PLC commanding of the distance control
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Block number [-]	
		Block number in which the error occurred	
	%3:	Current value [-]	
		Current operation mode of the distance control	
	%4:	Error value [-]	
		Programmed operation mode of the distance control	
Error type	11, Error message from position controller.		

ID 70441

Distance control turned on without set distance given.			
Description	The distance control function (see FCT-M3) was enabled in “distance” mode (HLI_DIST_CTRL_ON_CONST_DIST) without specifying a set distance.		
Response	Class	7	Closed-Loop controlled stop of axis, the distance control is not turned on
Solution	Class	6	Specify a set distance when turning the distance control on
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Block number [-]	
		Block number in the NC program in which the error occurred.	
	%3:	Current value [-]	
		Programmed operation mode of the distance control	
Error type	11, Error message from position controller.		

ID 70442

Distance control not initialised. PLC command could not be executed.			
Description	In the PLC, the cyclic interface for distance control (see FCT-M3) was activated although distance control for the axis is not enabled.		
Response	Class	7	Closed-loop controlled axis stop
Solution	Class	7	Enable the distance control function (see P-AXIS-00328) or disable the PLC interface of distance control
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Value of parameter P-AXIS-00328	
	%3:	Current value [-]	
		TRUE = distance control function is initialised	
Error type	11, Error message from position controller.		

ID 70443 / 70444

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70445

Configured drive type is unknown.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
	%2:	Drive type [-]	
	%3:	Identification number [-]	
Error type	-		

ID 70446

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 70447

Configured drive interface type not implemented for the drive type.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
	%2:	Drive type [-]	
	%3:	Identification number [-]	
Error type	-		

ID 70448

Feature bit set but P-0-4078 not configured.			
Description	The process data element P-0-4078 belonging to feature bit was not configured in the cyclic process data.		
Response	Class	5	Error message output, stop of axis, error is not resettable.
Solution	Class	7	Either configure P-0-4078 in the cyclic process data or reset the feature bit. Restart the controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020	
	%3:	Current value [-]	
		Value of the feature bit	
	%4:	Current value [-]	
		Status bit array of the currently configured cyclic telegram	
Error type	-		

ID 70449

Feature bit set but P-0-4077 not configured.			
Description	The process data element P-0-4077 belonging to feature bit was not configured in the cyclic process data.		
Response	Class	5	Error message output, stop of axis, error is not resettable.
Solution	Class	7	Either configure P-0-4077 in the cyclic process data or reset the feature bit. Restart the controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020	
	%3:	Current value [-]	
		Value of the feature bit	
	%4:	Current value [-]	
		Status bit array of the commanded configured cyclic telegram.	

Error type	-

ID 70450

Timeout during SERCOS homing.			
Description	A timeout error occurred during homing controlled with fieldbus control word (P-0-4077) and fieldbus state word (P-0-4078).		
Response	Class	7	Error message output, abort NC program.
Solution	Class	6	Resolve cause for timeout.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Limit value [µs]	
		Limit for timeout	
	%3:	State [-]	
		Internal state where the timeout occurred: • 5 - The command change bit was reset too late in the fieldbus status word (P-0-4078) when the homing command was ended. • 6 - The status bit ‘axis homed‘ (0x0004) was reset too late in the fieldbus status word (P-0-4077) when the homing command was started.	
Error type	11, Error message from position controller.		

ID 70451

Given filter time constant is invalid.			
Description	The specified filter time constant P-AXIS-00357 is smaller or larger than permissible. The filter coefficients can therefore not be calculated. Permissible values are (where Tab_{Ab} = cycle time of the NC controller): For PT1 and PT2 filters: $T_{Ab} \leq$ P-AXIS-00357 $\leq$ MAX(UNS32) For time delay filters: $0 \leq$ P-AXIS-00357 $< 6 * T_{Ab}$		
Response	Class	7	The filter is deactivated.
Solution	Class	6	Correct parameter P-AXIS-00357
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [-]	
		Number of concerned filter	
	%3:	Current value [-]	
		Filter type P-AXIS-00204	
	%4:	Error value [µs]	
		Invalid time constant P-AXIS-00357	
	%5:	Upper limit value [µs]	
		Maximum permissible time constant P-AXIS-00357	
Error type	-		

ID 70452

Activation of axis coupling not possible, because axis position is not initialized.			
Description	With an axis where axis parameters P-AXIS-00436 and P-AXIS-00460 are set, the axis coupling was activated via the HLI (see FCT-A9), although the axis positions of the axes involved are not yet fully initialised (the fieldbus has not yet supplied actual values valid for all axes).		
Response	Class	7	Error message output, coupling is not activated.
Solution	Class	6	When activation the coupling wait until all axes get valid actual positions from the fieldbus.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the target axis of the coupling	
	%2:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of a source axis of the coupling	
	%3:	Current value [-]	
		Indication whether the axis in value 2 gets valid actual positions from the fieldbus.	
	%4:	Current value [-]	
Error type	-		

ID 70453

The CNC configured homing encoder does not fit to the homing encoder set up in the drive.			
Description	When setting up CNC controlled homing with drive support the homing encoder set up in the drive (S-0-147 bit 3 (bitmask 0x08)) does not fit to the encoder set up in axis parameter P-AXIS-00388 .		
Response	Class	7	Error message output, abort NC program
Solution	Class	6	Either set P-AXIS-00388 to the value 1, in which case the encoder parameterized in S-0-147 is used internally by the CNC, or adapt either P-AXIS-00388 or S-0-147. Valid combinations are: S-0-147 Bit 3 = 0 → P-AXIS-00388 = 2 S-0-147 Bit 3 = 1 → P-AXIS-00388 = 3
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Value of S-0-147 read from drive.	
	%3:	Error value [-]	
		Parameterized value of P-AXIS-00388.	
Error type	-		

ID 70454

Axis coupling for gantry slave is defined by master.			
Description	An axis coupling was commanded via the HLI for a gantry slave axis, although axis parameter P-AXIS-00486 is set for the associated master axis. When this parameter is set, the coupling definitions for the gantry slave are adopted from the master axis, it is not possible to set coupling definitions for the slave axis.		
Response	Class	7	Error message output, coupling definition is discarded.

Solution	Class	6	Either set P-AXIS-00486 to zero for the master axis or do not set a coupling definition for the slave axis.
Parameter	%1:	Logical axis number [-]	
		Axis number of gantry slave axis.	
	%2:	Current value [-]	
		Axis mode of gantry slave axis (P-AXIS-00015).	
	%3:	Current value [-]	
		Value of P-AXIS-00486 of the master axis.	
	%4:	Logical axis number [-]	
		Logical axis number of gantry master axis.	
Error type	-		

ID 70455

Output interface to generic drive is not available.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Drive type [-]	
	%3:	Identification number [-]	
Error type	-		

ID 70456

Input interface to generic drive is not available.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Drive type [-]	
	%3:	Identification number [-]	
Error type	-		

ID 70457

The motor of this axis must not be decoupled.			
Description			
Response	Class	5	
Solution	Class	6	

Parameter	%1:	Logical axis number [-]
Error type	11, Error message from position controller.	

ID 70458

Low-pass filter coefficients generated not possible with specified parameters.			
Description	With distance control (see FCT-M3), a low-pass filter was parameterised to smooth the measured distance values. However, no valid filter can be created with the given filter parameters.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check filter cut-off frequency. It must be less than 0.5 / GEO cycle time (Nyquist theorem). Try to reduce the filter order.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [-]	
		Return value of filter generation	
	%3:	Current value [-]	
		Filter order of low pass filter	
	%4:	Current value [Hz]	
		Characteristic frequency of low pas filter	
	%5:	Current value [µs]	
		Cycle time of GEO task	
Error type	11, Error message from position controller.		

ID 70459

Position overflow in axis-specific forward transformation.			
Description	The result of the axis-specific forward transformation (conversion of drive position to interpolator position) is larger than the maximum representable position. Possible reasons are: Incorrect parametrisation of the transformation Activation of the transformation at an inappropriate position		
Response	Class	5	Error message output, transformation is disabled.
Solution	Class	6	Check transformation parameters Check activation position
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Identifier of the axis-specific transformation (P-AXIS-00381)	
	%3:	Error value [0.1 µm or 0.0001°]	
		Input position of axis-specific transformation	
	%4:	Limit value [0.1 µm or 0.0001°]	
		Maximum possible output position of transformation	
Error type	-		

ID 70460

Argument of the exponential function in axis-specific forward transformation too big.			
Description	During the axis specific forward transformation (calculation of interpolator positions from axis positions) a to big argument of an exponential function was calculated. Possible reasons are: • Wrong parametrization of the transformation • Activation of the transformation at an inappropriate location		
Response	Class	5	Error message output, transformation is disabled.
Solution	Class	6	Check transformation parameter and activation position
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Identifier of the axis-specific transformation (P-AXIS-00381)	
	%3:	Error value [0.1 µm or 0.0001°]	
		Argument value of exponential function	
	%4:	Limit value [0.1 µm or 0.0001°]	
		Maximum possible argument value	
Error type	-		

ID 70461

An error occurred while writing a CANopen object.			
Description	An error occurred when trying to write an object to a CANopen drive during the execution of a #DRIVE command.		
Response	Class	5	Program abort
Solution	Class	6	In the configuration of the #DRIVE command, check the parameters of the object to be written (object number, subindex, object size, object value).
Parameter	%1:	Logical axis number [-]	
		Axis number	
	%2:	Current value [-]	
		Object number	
	%3:	Current value [-]	
		Subindex	
	%4:	Current value [-]	
		Object value	
	%5:	Current value [-]	
		Length of object in bytes.	
Error type	-		

ID 70462

Switching the parameter set is not possible because process data is incomplete.			
Description			
Response	Class	7	
Solution	Class	6	

Parameter	%1:	Logical axis number [-]
	%2:	Error value [-]
	%3:	Error value [-]
Error type	1, Error message from NC program.	

ID 70463

Could not open parameter file for Volumetric Compensation.			
Description	This error occurs when one of the parameter files for Volumetric Compensation could not be opened of it the diagnosis file could not be written. Possible reasons are incorrect path specifications in P-VOLC-00012 and P-VOLC-00016 or missing file access rights.		
Response	Class	-	
Solution	Class	-	Adjust paths and/or file rights, restart controller
Parameter	%1:	Filename	
		File path	
Error type	-		

ID 70464

Syntax error in parameter file for Volumetric Compensation.			
Description	Occurs when a parameter file could not be read due to a syntax error. A possible cause is the specification of the wrong file format in parameter P-VOLC-00013.		
Response	Class	-	
Solution	Class	-	Adjust parameter file or file format, update Volumetric Compensation.
Parameter	%1:	File name [-]	
		File path	
	%2:	Line number in file [-]	
		Line number	
	%3:	Column number in file [-]	
		Column number	
Error type	-		

ID 70465

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 70466

Unknown interpolation mode for Volumetric Compensation.			
Description	An unknown interpolation mode has been configured, see parameter P-VOLC-00014.		
Response	Class	3	
Solution	Class	6	Adjust interpolation mode, restart CNC.
Parameter	%1:	Compensation index [-]	
		ID of volumetric compensation	
Error type	11, Error message from position controller.		

ID 70467

Too many columns in a CSV file.			
Description	There are to many data in a record line oft he CSV file.		
Response	Class	-	
Solution	Class	-	Adjust CSV file, update Volumetric Compensation.
Parameter	%1:	Compensation index [-]	
		ID of Volumetric Compensation	
	%2:	File name [-]	
		Filepath	
	%3:	Error value [-]	
		Actual number of columns	
	%4:	Upper limit value [-]	
		Maximum number of columns	
Error type	-		

ID 70468

Unknown identifier.			
Description	An unknown identifier was found when parsing a parameter file.		
Response	Class	-	
Solution	Class	-	Adjust parameter file, update Volumetric Compensation.
Parameter	%1:	Compensation index[-]	
		ID of volumetric compensation	
	%2:	File name [-]	
		File path	
	%3:	Identifier [-]	
		Unknown identifier	
Error type	-		

ID 70469

An identifier occurred more than once.			
Description	While parsing a CSV parameter file, a column name occurred twice.		
Response	Class	-	
Solution	Class	-	Adjust parameter file, update Volumetric Compensation.

Parameter	%1:	Compensation index[-]
		ID of Volumetric Compensation
	%2:	File name [-]
		File path
	%3:	Identifier [-]
		Identifier
	%4:	Error value [-]
		Number of columns
	%5:	Current value [-]
		Column
Error type		

ID 70470

Too many records for Volumetric Compensation.			
Description	The parameter files of the volumetric compensation contain more data sets than can be read in or there is not enough memory for parameter splicing, see parameter P-STUP-00100 and function description [FCT-C26].		
Response	Class	-	
Solution	Class	-	Adjust parameter P-STUP-00100, restart controller.
Parameter	%1:	Compensation index[-]	
		ID of volumetric compensation	
	%2:	Error value [-]	
		Number of records read so far	
	%3:	Upper limit value [-]	
		Maximal number of records that may be loaded	
Error type	-		

ID 70471

The parameter values are not on a grid.			
Description	The GRID interpolation mode is enabled for volumetric configuration, but the parameter data is not on a grid, see P-VOLC-00014.		
Response	Class	-	
Solution	Class	-	Adjust parameter data or interpolation mode, restart CNC.
Parameter	%1:	Compensation index[-]	
		ID of volumetric compensation	
	%2:	Parameter name [-]	
		Critical parameter	
Error type	-		

ID 70472

Unknown axis identifier in kinematic chain of Volumetric Compensation.			
Description	An unknown axis identifier appears in the configuration of the kinematic chain, see P-VOLC-00007.		
Response	Class	3	

Solution	Class	6	Adjust kinematic chain, restart controller
Parameter	%1:	Compensation index[-]	
		ID of Volumetric Compensation	
	%2:	Kinematic chain [-]	
		Name of kinematic chain	
	%3:	Error value [-]	
		Unknown axis identifier	
Error type	11, Error message from position controller.		

ID 70473

Invalid logical axis number for axis in Volumetric Compensation.			
Description	The axis to be assigned to Volumetric Compensation is unknown in the controller, see P-VOLC-00001 to P-VOLC-00006.		
Response	Class	3	
Solution	Class	6	Adjust logical axis number, restart controller
Parameter	%1:	Compensation index[-]	
		ID of Volumetric Compensation	
	%2:	Axis name [-]	
		Identifier of the unknown axis	
	%3:	Error value [-]	
		Unknown logical axis number	
Error type	11, Error message from position controller.		

ID 70474

The parameter values are on a grid, but are not interpolated linearly.			
Description	This warning indicates that the parameter data provided lie on a grid, but are not linearly interpolated, see P-VOLC-00014. Instead, non-linear interpolation is used but it has some disadvantages. Therefore, it is recommended to switch over to linear interpolation.		
Response	Class	-	
Solution	Class	-	If necessary, change interpolation mode to GRID or AUTO, restart controller system
Parameter	%1:	Compensation index	
		ID of volumetric compensation	
	%2:	Parameter name [-]	
		Parameter	
Error type	-		

ID 70475

Insufficient memory for the Volumetric Compensation.			
Description	Insufficient memory was allocated to Volumetric Compensation. The memory requirement is set in the parameter P-STUP-00100. To fix the problem, you can try to decrease this parameter. However, this may require you to discard some parameter records.		
Response	Class	-	

Solution	Class	-	Adjust parameter P-STUP-00100, restart controller
Parameter	%1:	Compensation index [-]	
		ID of Volumetric Compensation	
Error type	-		

ID 70476

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70477

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 70478

The given grid for plane compensation is invalid.			
Description	Plane compensation (see FCT-C5//Plane compensation) can use the identical or a different grid for the two master axes. This is defined in parameter P-COMP-00031. However, the given value is incorrect.		
Response	Class	7	Error message, and disable plane compensation.
Solution	Class	7	Correct parameter P-COMP-00031.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Incorrect value P-COMP-00031	
	%3:	Error value [-]	
		Incorrect value P-COMP-00031	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70479

Volumetric Compensation: invalid compensation value.			
Description	The computed compensation value exceeds the limit specified in axis parameter P-AXIS-00525.		
Response	Class	7	Movement stop.
Solution	Class	6	Adapt the compensation parameter or increase the limit value P-AXIS-00525 then reset the controller. The error state cannot be exited without adapting the parameter.

Parameter	%1:	Logical axis number
		Logical axis number of axis affected
	%2:	Current value
		Incorrect value
	%3:	Limit value
		Maximum limit value
Error type	11, Error message from position controller.	

ID 70480

P-AXIS-00519 or P-AXIS-00779 set but no actual velocity configured in process data.			
Description	The parameters P-AXIS-00519 or P-AXIS-00779 were set but no actual velocity is configured in the process data.		
Response	Class	1	Output of a warning, P-AXIS-00519 and P-AXIS-00779 are set to zero.
Solution	Class	1	Either set P-AXIS-00779 to zero (nil) or configure the current velocity in the process data.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis.	
	%2:	Current value [-]	
		Current value of P-AXIS-00779	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00779	
Error type	-		

ID 70481

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires controller restart.

ID 70482

Drive instruction without commands received.			
Description	A #DRIVE command was received with no action entered (neither parameter record changeover nor motor switching nor writing of a KEY/VAL pair). #DRIVE commands can be sent by the NC program or by the PLC. If the command is sent by the NC program, the command syntax is checked in the decoder. Due to the existing syntax check in the decoder, this error can only be triggered by a command from the PLC: With this command, the input pins of the function block used (FB) are unassigned.		
Response	Class	7	Error message output, abort NC program.
Solution	Class	6	Check and modify the PLC program. Assign the input pins of the FB.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Error value [-]
		Command bitfield of drive command must not be zero (nil).
	%3:	Current value [-]
		When the command was triggered via the SPS interface the value of this entry is maximum_channel number + 1.
Error type	-	

ID 70483

Volumetric Compensation: invalid compensation ID.			
Description	An invalid Volumetric Compensation ID was specified for the update of parameter values.		
Response	Class	3	Abort current job.
Solution	Class	6	Reset CNC, repeat with valid ID
Parameter	%1:	Error value [-]	
		Invalid ID	
	%2:	Lower limit value [-]	
		Smallest valid ID	
	%3:	Upper limit value [-]	
		Largest valid ID	
Error type	11, Error message from position controller.		

ID 70484

The Volumetric Compensation cannot be updated, because it is still enabled.			
Description	A parameter update has been attempted for a Volumetric Compensation that is still enabled. First disable Volumetric Compensation before updating the parameters.		
Response	Class	3	Abort current job.
Solution	Class	6	Reset the controller, disable the compensation before updating the parameters.
Parameter	%1:		
		ID of volumetric compensation	
Error type	11, Error message from position controller.		

ID 70485

The Volumetric Compensation cannot be updated, because it is already in an update process.			
Description	This error can occur if an attempt is made to update a volumetric compensation from two different channels at the same time.		
Response	Class	3	The second update command is cancelled.
Solution	Class	6	Reorganize the two update commands.
Error type	11, Error message from position controller.		

ID 70487

The Volumetric Compensation cannot be updated, because it is in an error state.			
Description	The Volumetric Compensation could not be updated, because not enough memory has been allocated or the allocation failed earlier.		
Response	Class	3	The parameters are not updated.
Solution	Class	6	Remove cause for putting the compensation into error state, restart the CNC
Parameter	%1:	Compensation index [-]	
		ID of volumetric compensation	
Error type	11, Error message from position controller.		

ID 70488

After modification of absolute encoder position offset P-AXIS-00403, a position request (#CHANNEL INIT) is required.			
Description	The parameter antr.abs_pos_offset (P-AXIS-00403) is used for axes with absolute encoders to offset the position controller position in relation to the encoder position. If this parameter is changed e.g. via a list update or in the NC program via the #MACHINE DATA command, the command position in the position controller changes for this axis. Then, the CNC channel must be initialised with the new position. **After modification of absolute encoder position offset P-AXIS-00403, a position request (#CHANNEL INIT) required.** Change parameter P-AXIS-00403 only if there is no enabled NC program or when the axis is not assigned to an NC channel.		
Response	Class	7	Closed-loop controlled stop of affected axis.

Solution	Class	6	After updating parameter P-AXIS-00403, reinitialise the command positions in the channel ((#CHANNEL INIT [CMDPOS]).
			Example:
			`N00 G0 X100`
			`N10 #FLUSH WAIT`
			`N20 #MACHINE DATA \`
			`[AX=Y AXPARAM="antr.abs_pos_offset 1500000"]`
			`N30 #CHANNEL INIT [CMDPOS]`
			`N40 G0 X200`
			`N50 M30`

Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current position offset P-AXIS-00403	
	%3:	Current value [-]	
		Current relative command value change of the axis	
	%4:	Current value [-]	
		Control information for position controller	
Error type	11, Error message from position controller.		

ID 70489

Modification of absolute encoder position offset (P-AXIS-00403) permitted at axis standstill only.			
Description	The parameter antr.abs_pos_offset (P-AXIS-00420) is used for axes with an absolute encoders to offset the position controller position in relation to the encoder position. Changing this parameter via list update or #MACHINE DATA command is only permitted at standstill of the corresponding axis.		
Response	Class	7	Controlled stop of the concerned axis, the new parameters are discarded!
Solution	Class	6	Stop the axis before updating the parameter
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current position offset P-AXIS-00420	
	%3:	Error value [0.1 µm or 0.0001°]	
		New position offset P-AXIS-00420	
	%4:	Error value [-]	
		Current relative command position of axis	
	%5:	Current value [-]	
		Control information for position controller	
Error type	11, Error message from position controller.		

ID 70490

The Volumetric Compensation cannot be updated, because no memory has been allocated.			
Description	The parameter update of the compensation failed, since no memory has been allocated for it during controller start-up, see P-STUP-00100.		
Response	Class	3	
Solution	Class	6	Adjust P-STUP-00100, restart the controller.
Parameter	%1:	Compensation index [-]	
		ID of volumetric compensation	
Error type	11, Error message from position controller.		

ID 70491

The Volumetric Compensation cannot be enabled/disabled, because it is either not configured or in error.			
Description	The Volumetric Compensation cannot be enabled, because it is either not configured or in error.		
Response	Class	3	
Solution	Class	6	Remove cause for error, reset or restart CNC
Parameter	%1:	Compensation index [-]	
		ID of volumetric compensation	
Error type	11, Error message from position controller.		

ID 70492

The update of the Volumetric Compensation failed.			
Description	Follow-up error of an error during parameter update.		
Response	Class	3	
Solution	Class	6	Remove cause of problem, reset CNC
Error type	1, Error message from NC program.		

ID 70493

Dynamic limits of an axis involved in the Volumetric Compensation not configured.			
Description	The dynamic limits P-AXIS-00525 and P-AXIS-00526of the affected axis are not completely defined.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the parameters P-AXIS-00525 and P-AXIS-00526. Reset the controller
Parameter	%1:	Compensation index [-]	
		ID of volumetric compensation	
	%2:	Axis name [-]	
		Name of the affected axis	
	%3:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
Error type	11, Error message from position controller.		

ID 70494

Dynamic limits of a gantry axis involved in the Volumetric Compensation not configured.			
Description	The dynamic limits P-AXIS-00525 and P-AXIS-00526 of the affected gantry axis are not completely defined.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the parameters P-AXIS-00525 and P-AXIS-00526. Reset the controller
Parameter	%1:	Compensation index [-]	
		ID of Volumetric Compensation	
	%2:	Axis name [-]	
		Name of the affected axis	
	%3:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
Error type	11, Error message from position controller.		

ID 70495

Activation of friction compensation not possible since it is not selected in axis parameter list.			
Description	To enable friction compensation (see FCT-C25]) in the NC program for an axis, it must be activated in the axis parameter list (see P-AXIS-00522).		
Response	Class	7	Stop of axis, friction compensation remains inactive
Solution	Class	6	Activate friction compensation (see P-AXIS-00522)

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Error value [-]
		Parameter value P-AXIS-00522 to activate friction compensation
Error type	11, Error message from position controller.	

ID 70496

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	Axis stop
Solution	Class	8	Requires controller restart.

ID 70497

At end of program the functionality friction is still active.			
Description	A friction related function is still enabled at the end of the program.		
Response	Class	1	None
Solution	Class	1	Check and modify the NC program. End friction compensation with #FRICTION OFF before program end.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
Error type	11, Error message from position controller.		

ID 70498

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 70499

Identifier for additional 'Ready for Power' test not configured in cyclic input process data.			
Description	The input name set in P-AXIS-00711 was not found in the configured input process data of the drive. If the reference signal is transmitted in a real-time status bit of a SERCOS drive, enter the value "S-0-0135" in this parameter P-AXIS-00711. In this case the drive status word does not have to be configured explicitly in the drive process data because it is always transferred automatically.		
Response	Class	1	Warning output, additional 'ready for power' test is disabled by assigning an empty string to P-AXIS-00711.
Solution	Class	1	Either change P-AXIS-00711 to a value configured in the process data of the drive, or extend the process data of the drive by the element entered in P-AXIS-00711.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Error value [-]
		Set value of P-AXIS-00711.
Error type	-	

2.8.3 ID-range 70500-70749

ID 70500

Second encoder for distance control not configured. Distance control is disabled.			
Description	With distance control (see FCT-M3), height changes on the workpiece can be corrected with the help of a distance sensor. The axis therefore requires two encoder inputs for this function: 1. The actual position of the axis, e.g. the motor encoder 2. The sensor signal which is connected as actual value encoder 2 The CNC outputs this error message if the two encoders are not configured in the cyclic process data of the axis. Distance control is disabled for this axis.		
Response	Class	7	Controlled stop of the concerned axis, distance control is disabled.
Solution	Class	7	Configure the two encoders in the process data: 1. For SERCOS: S-0-0051, S-0-0053 2. For CANopen: 0x6064 (POS_ACT), 0x60E4 3. For Profidrive: G1_XIST1/2, G2_XIST1/2

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Drive type [-]
		Drive type P-AXIS-00020 of the affected axis
	%3:	Error value [-]
		Current number of configured encoders
	%4:	Expected value [-]
		Expected number of encoders for distance control
	%5:	Corrected value [-]
		FALSE = distance control is disabled
Error type	11, Error message from position controller.	

ID 70501

Given modulo range of drive encoder exceeds valid range of configured actual position.			
Description			
Response	Class	5	
Solution	Class	7	

Parameter	%1:	Logical axis number [-]
	%2:	Drive type [-]
	%3:	Error value [-]
	%4:	Upper limit value [-]
	%5:	Corrected value [-]

Error type	-

ID 70502

Axis should be interpolated during active frequency response measurement.			
Description	Axis shall be interpolated while the frequency response of the axis is measured from an external program or the axis is already interpolated at the start of the frequency measurement.		
Response	Class	7	Error message output, measurement abort.
Solution	Class	6	Don’t move axis during measurement of frequency response.
Parameter	%1:	Logical axis number [-]	
Error type	11, Error message from position controller.		

ID 70503

Axis is not ready for interpolation during active frequency response measurement.			
Description	The axis is not in control when starting or during a frequency response measurement.		
Response	Class	7	Error message output, measurement abort.
Solution	Class	6	Check if axis has all necessary PLC enable signals and if the power for the axis is switched on.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
Error type	11, Error message from position controller.		

ID 70504

Error during frequency response measurement occurred.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 70505

Position deviation during frequency response measurement exceeds permissible range.			
Description	During a frequency response measurement, command values are applied to an axis by an external software module. The external software module detected that the position deviation is greater than permitted and sets the CNC axis to error state.		
Response	Class	6	Abrupt stop for the defective axis, feedhold for the other axes.
Solution	Class	5	Check the external software module

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Drive type [-]
		Drive type P-AXIS-00020 of the affected axis
	%3:	Current value [-]
		Error code of the external software module
Error type	-	

ID 70506

The given index for the compensation is out of the value range.			
Description	An invalid index for compensation was specified at the request or on setting a data item for Volumetric Compensation from an NC program.		
Response	Class	7	The read/write access is cancelled.
Solution	Class	6	Correct compensation index
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 70507

The given index of the file for the compensation is out of the value range.			
Description	An invalid file index j was transferred when trying to read or write to the global variable V.G.VOLCOMP[i].FILE[j].		
Response	Class	7	Access to V.G.VOLCOMP[i].FILE[j] was cancelled.
Solution	Class	6	Correct file index j.
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 70508

A volumetric compensation is still enabled during program end.			
Description	This error is thrown if the Volumetric Compensation is associated to a channel (see parameter P-VOLC-00019) and is still enabled at the end of the program.		
Response	Class	7	Program is aborted, perform CNC reset.
Solution	Class	6	Disable the compensation before the program end or remove the channel association.
Parameter	%1:	Current value [-]	
		Index of the channel	
	%2:	Compensation index [-]	
		ID of volumetric compensation	

Error type	11, Error message from position controller.

ID 70509

Missing license for Volumetric Compensation.			
Description	A separate license is required for the use of volumetric compensation. Please contact your vendor of the CNC control.		
Response	Class	3	The Volumetric Compensation cannot be enabled.
Solution	Class	7	Enable license
Error type	11, Error message from position controller.		

ID 70510

With active Volumetric Compensation changing the compensation axis is not permitted.		
Description	Axes involved in Volumetric Compensation can be changed via a write access to the global variables V.G.VOLCOMP[i].X/Y/Z/A/B/C. However, this change is not allowed while the respective compensation is still enabled.	
Response	Class	3 Program abort.
Solution	Class	6 Disable compensation before changing the axis configuration.
Parameter	%1:	Compensation index [-]
		ID of Volumetric Compensation
	%2:	Axis name [-]
	%3:	Logical axis number [-]
		Previous axis number of the axis used
	%4:	Logical axis number [-]
		Axis number of the new axis to be used
	%5:	Block number [-]
Error type	11, Error message from position controller.	

ID 70511

Activation of measurement in current state of distance control not possible.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
	%2:	Error value [-]	
		Distance control state	
	%3:	Block number [-]	
		Current block number	
Error type	1, Error message from NC program.		

ID 70512

No measuring variable for the measured values specified.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
	%2:	Block number [-]	
		Current block number	
Error type	1, Error message from NC program.		

ID 70513

No minimum height difference for capturing the measured values specified.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
	%2:	Block number [-]	
		Current block number	
Error type	1, Error message from NC program..		

ID 70514

No minimum distance for capturing the measured values specified.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
	%2:	Block number [-]	
Error type	1, Error message from NC program.		

ID 70515

No maximum distance for capturing the measured values specified.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Block number [-]	

Error type	1, Error message from NC program.

ID 70516

Master axis not programmed for measurement run.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Block number [-]	
	%3:	Logical axis number [-]	
	%4:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 70517 / 70518

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70519

Too many measuring points. Measuring is aborted.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Block number [-]	
	%3:	Error value [-]	
	%4:	Upper limit value [-]	
Error type	-		

ID 70520

Activation of measurement not possible, distance control on master axis is already active.

Description	Distance control is active in the master axis when the measurement was activated; this is not allowed.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program and the program flow. Execute measurement before activating distance control.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
	%2:	Block number [-]	
		Block number in the NC program when the measurement is activated.	
	%3:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the master axis	
	%4:	Error value [-]	
		Control word of the master axis	
Error type	-		

ID 70521

Activation of distance control not possible because axis is part of active measurement.			
Description	The axis to be controlled is still actively involved in a measurement when distance control was activated.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the process. Only activate distance control after measurement.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
	%2:	Error value [-]	
		Distance control state	
	%3:	Block number [-]	
		Block number that activates distance control.	
	%4:	Current value [-]	
Error type	1, Error message from NC program.		

ID 70523

Permissible distance for movement from reference cam exceeded.			
Description	During homing, the maximum covered distance set in P-AXIS-00531 was exceeded when moving away from the reference cam. Possible causes for this error message are: • the reference switch is ‘frozen’. • P-AXIS-00531 setting is too small.		
Response	Class	7	Error message output, abort NC program.
Solution	Class	6	Possible remedies: • Check reference switch function • Set larger parameter in P-AXIS-00531

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of affected axis
	%2:	Error value [-]
		Distance covered on moving away from cam
	%3:	Limit value [-]
		Maximum permissible distance
	%4:	Current value [-]
		Current command position
Error type	11, Error message from position controller.	

ID 70524

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70525

Clamped axis shall be moved.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Current value [-]	
	%4:	Expected value [-]	
Error type	11, Error message from position controller.		

ID 70526

The axis should be compensated with a Volumetric Compensation but has no release.			
Description	An axis that is part of a Volumetric Compensation is about to receive a correction value but the axis is not released.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Reset CNC, issue axis release

Parameter	%1:	Compensation index [-]
		ID of Volumetric Compensation
	%2:	Logical axis number [-]
		Logical axis number (P-AXIS-00016 of axis affected
	%3:	Current value [* 10^-4 Incr]
		Preceding correction value
	%4:	Current value [* 10^-4 Incr]
		Current correction value
	%5:	Current value [* 10^-4 Incr]
		Following correction value
Error type	11, Error message from position controller.	

ID 70528

Volumetric Compensation: axis could not be compensated because it is not homed.			
Description	An axis that is part of a Volumetric Compensation is about to receive a correction value but the axis is not homed yet.		
Response	Class	3	
Solution	Class	6	Reset CNC, home axis
Parameter	%1:	ID of Volumetric Compensation	
	%2:	logical axis number	
Error type	11, Error message from position controller.		

ID 70529

Volumetric Compensation: axis could not be compensated because it is in an error state.			
Description	An axis that is part of a Volumetric Compensation is about to receive a correction value but the axis is in error state.		
Response	Class	3	
Solution	Class	6	Reset the controller
Parameter	%1:	ID of Volumetric Compensation	
	%2:	logical axis number	
	%3:	Channel number	
Error type	11, Error message from position controller.		

ID 70530

Volumetric Compensation: axis could not be compensated because it is moved by another channel.			
Description	An axis that is part of a Volumetric Compensation is about to be compensated by a command value correction but it is currently moving in another channel.		
Response	Class	3	

Solution	Class	6	Reset CNC, resolve conflict
Parameter	%1:	ID of Volumetric Compensation	
	%2:	logical axis number	
Error type	11, Error message from position controller.		

ID 70531

The Volumetric Compensation could not be inverted, therefore it cannot be enabled without movement.			
Description	A Volumetric Compensation is to be enabled without motion (NO MOVE) but the appropriate compensation value could not be computed. This problem occurs when the Volumetric Compensation parameter file contains error parameters that fluctuate too much. Therefore, check the content of the parameter file.		
Response	Class	3	
Solution	Class	6	Reset CNC, check parameter file of Volumetric Compensation
Parameter	%1:	Compensation index [-]	
		ID of Volumetric Compensation	
	%2	Block number [-]	
	%3:	Identifier [-]	
		Channel number	
Error type	11, Error message from position controller.		

ID 70532

The Volumetric Compensation cannot be enabled/disabled, because it is either not configured or in error.			
Description	This error is output if a Volumetric Compensation is to be enabled or disabled but it is in an error state. To enable or disable it, the Volumetric Compensation must be error-free.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Reset the controller
Parameter	%1:	Compensation index [-]	
		ID of volumetric compensation	
	%2:	State [-]	
		error state of the Volumetric Compensation	
	%3:	Block number [-]	
	%4:	Identifier [-]	
		Channel number (parameter is optional)	
Error type	11, Error message from position controller.		

ID 70533

Distance control active but encoder delivers no valid actual positions.

Description	An axis with active distance control (see [FCT-M3]) is to be moved although one of the sensors does not deliver valid positions. Value 2 of the error message indicates which of the two distance measuring sensors triggered the fault. Possible output values: 1 - Actual position of the axis 2 – Configured velocity of the axis 3 - Additive actual position value of the axis 4 - Second additive actual position of the axis		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Remedy the cause for sensor fault.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value[-]	
		Indication for erroneous sensor	
Error type	-		

ID 70534

Upper modulo limit in increments is outside valid range.			
Description			
Response	Class	1	Warning, no reaction.
Solution	Class	1	
Parameter	%1:	Logical axis number[-]	
	%2:	Error value [Incr.]	
	%3:	Lower limit value [Incr.]	
	%4:	Corrected value [Incr.]	
	%5:	Current value [Incr./0.1 µm]	
Error type	1, Error message from NC program.		

ID 70535

Lower modulo limit in increments is outside valid range.			
Description			
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
	%2:	Error value [-]
	%3:	Lower limit value [-]
	%4:	Corrected value [-]
	%5:	Current value [-]
Error type	1, Error message from NC program.	

ID 70536

The Volumetric Compensation can not be enabled or disabled, because the axis moves.			
Description	To enable Volumetric Compensation, every participating axis must either be • in a channel in which no program is running or • in no channel.		
Response	Class	3	Volumetric Compensation is not enabled.
Solution	Class	6	Only enable Volumetric Compensation if one of the two above conditions is met for each axis.
Parameter	%1	Compensation index [-]	
		ID of Volumetric Compensation	
	%2	Identifier [-]	
		Channel in which the relevant axis is located	
	%3	Logical axis number [-]	
		Logical axis number of the relevant axis	
Error type	11, Error message from position controller.		

ID 70537

An error occurred while reading a CANopen object.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1:	Logical axis number	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Current value [-]	
	%3:	Current value [-]	
	%4:	Current value [-]	
Error type	11, Error message from position controller.		

ID 70538

Moving to fixed stop already activated.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2	Error value [-]	
Error type	11, Error message from position controller.		

ID 70539

Moving to fixed stop not supported for configured drive type.			
Description	The drive type configured (P-AXIS-00020) does not currently support the Move to fixed stop function [FCT-M8]. Currently, the function can be used for SERVOS and CANopen drives.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	The Move to fixed stop function cannot be used for this drive type.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2	Drive type [-]	
		Drive type not supported P-AXIS-00020	
Error type	11, Error message from position controller.		

ID 70540

Configuration of drive idents for movement towards fixed stop incorrect.			
Description	The Move to fixed stop function [FCT-M8] cannot be activated since the configuration of the drive objects to be changed is incorrect. Please see the previous error message for the exact cause of the error.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and correct the parameterisation of the drive objects for Moving to fixed stop.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	11, Error message from position controller.		

ID 70541

Drive ident for torque limitation during fixed stop movement not configured.

Description	A drive object must be configured to limit torque for the Move to fixed stop function [FCT-M8]. The list of drive objects must contain an object with the name TORQUE_LIMIT (see P-AXIS-00719).
	
	Example for SERVOS drive;
	`antr.fixed_stop.drive_ident[0].id TORQUE_LIMIT`
	`antr.fixed_stop.drive_ident[0].wr_ident S_0_0092`		

Response	Class	7	Controlled halt of the axis, the control loop is closed. Transition to error state.
Solution	Class	6	Check and correct the parameterisation of the drive objects for Moving to fixed stop. Add the object for torque limitation.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
Error type	11, Error message from position controller.		

ID 70542-70543

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70544

Timeout during reading/writing drive data for fixed stop movement.			
Description	When the Move to fixed stop function is activated and deactivated [FCT-M8], drive objects must be changed. It is impossible to complete the read or write operations for an object within the specified time. Possible reasons are: 1. Problems with fieldbus transfer 2. Problems in the drive 3. Incorrect parameterisation of the drive objects		
Response	Class	6	Movement stop. Transition to error state. If the Move to fixed stop function is active during the error, it will remain active.
Solution	Class	6	Check fieldbus, drive and parameterisation.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Current value [-]	
		Internal identifier P-AXIS-00719 of the drive object that could not be read or written.	
Error type	11, Error message from position controller.		

ID 70545

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70546

Movement to fixed stop aborted by missing drive releases.			
Description	The drive release were reset during an active Move to fixed stop function [FCT-M8]. The start movement is aborted and clamping of the fixed stop that has already taken place cannot be maintained.		
Response	Class	6	Controlled halt of the axis, the control loop is closed. Transition to error state. The Move to fixed stop function is terminated.
Solution	Class	7	Check and modify the sequences in the NC program and the PLC. Deactivate the Move to fixed stop function before removing the drive releases.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2	Current value[-]	
		Fixed stop already detected.	
Error type			

ID 70547

Movement to fixed stop aborted from follow-up operation.			
Description	Tracking mode was activated during an active Move to fixed stop function [FCT-M8]. The approach motion is aborted and clamping of the fixed stop that has already taken place cannot be maintained.		
Response	Class	6	Controlled halt of the axis, the control loop is closed. Transition to error state. The Move to fixed stop function is terminated.
Solution	Class	6	Check and modify the sequences in the NC program and the PLC. Deactivate the Move to fixed stop function before activating tracking mode.
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2	Current value [-]	
		Fixed stop already detected	
Error type	-		

ID 70548

Permissible tolerance window exceeded during movement to fixed stop. Check the fixed stop.	
Description	After detection of the fixed stop [FCT-M8], the fixed stop can be monitored against a tolerance window (see P-AXIS-00713). The CNC generates this error message if the fixed stop breaks off and exits the tolerance window. Possible causes: • Fixed stop breaks off • Stop time too long

	Possible solutions: • Check the mechanical components of the fixed stop • Reduce the torque		
Response	Class	6	Controlled halt of the axis, the control loop is closed. Transition to error state. The Move to fixed stop function remains active.
Solution	Class	7	Check fixed stop, reduce torque
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Error value [0.1 10^-3 mm or ø]	
		Current position of the fixed axis	
	%3:	Current value [0.1 10^-3 mm or ø]	
		Position at first detection of the fixed stop	
	%4:	Lower limit value [0.1 10^-3 mm or ø]	
		Lower limit for fixed stop position	
	%5:	Upper limit value [0.1 10^-3 mm or ø]	
		Upper limit for fixed stop position	
Error type	11, Error message from position controller.		

ID 70549

Moving to fixed stop aborted by CNC reset.			
Description	This error message is output for safety reasons if Move to fixed stop [FCT-M8] is aborted by a CNC reset and the fixed stop was not yet detected. Before the machine can continue, make sure that the obstacle or the fixed stop is removed from the motion path. This message can be suppressed by the axis parameter P-AXIS-00715 Remove obstacle or fixed stop.		
Response	Class	6	Controlled halt of the axis, the control loop is closed. Transition to error state. Move to fixed stop is aborted
Solution	Class	7	Remove obstacle or fixed stop
Parameter	%1	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2	Current value [-]	
		Fixed stop detected	
Error type	11, Error message from position controller.		

ID 70550

Moving to fixed stop still active, fixed stop was detected.			
Description	This warning message is generated if the CNC triggers a reset after the fixed stop is detected. The Move to fixed stop function remains active [FCT-M8]. The function must be terminated explicitly after the fixed stop is detected since a backward motion from the fixed stop should take place after deactivation. This warning message can be suppressed by the parameter P-AXIS-00717. Example: `N10 G01 X0 F1000 X[FIXED_STOP OFF]`		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Deactivate Move to fixed stop

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [-]
		State of the Move to fixed stop function.
	%3:	Current value [-]
		State of the fixed stop detection function.
	%4:	Current value [0.1 10^-3 mm or ø]
		Position of the fixed stop.
Error type	11, Error message from position controller.	

ID 70551

Scaling factor for reading drive idents is smaller than permissible.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Error value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
Error type	11, Error message from position controller.		

ID 70552

Multiple drive idents for torque limitation given.			

Description	The drive torque must be limited for the [FCT-M8] Move to fixed stop function. For this reason, the function modifies a drive object. However, several torque limitation objects are specified in the configuration. The configuration should only contain one object with the name TORQUE_LIMIT.
	
	Example:
	`antr.fixed_stop.drive_ident[0].wr_ident TORQUE_LIMIT`
	…
	`antr.fixed_stop.drive_ident[2].wr_ident TORQUE_LIMIT`		

Response	Class	7	Controlled halt of the axis, the control loop is closed. The Move to fixed stop function cannot be used for this axis.
Solution	Class	6	Correct configuration of drive objects

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Error value [-]
		Index of double object
	%3:	Current value [-]
		Index of first object found
	%4:	Error value [-]
		Name of the double object P-AXIS-00721
	%5:	Current value [-]
		Name of the first object P-AXIS-00721
Error type	-	

ID 70553

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	
Solution	Class	8	Requires controller restart.

ID 70554

Programmed axis coupling still active at program end.			
Description	At program end, programmed axis couplings are still enabled with #GEAR LINK [.] for the affected axis.		
Response	Class	5	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	Disable axis coupling with #GEAR LINK OFF before program end.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis with active coupling	
	%2:	Error value [-]	
		Coupling state	
	%3:	Current value [-]	
		Coupling initiator	
Error type	11, Error message from position controller.		

ID 70555

Filter of additional command value interface is active and could not be updated.			
Description			
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Drive type [-]	
	%3:	Current value [-]	

Error type	-

ID 70556

Modulo range in increments is outside valid range.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
	%3	Lower limit value [-]	
	%4	Corrected value [-]	
	%5	Current value [Increments / 0.1 um]	
Error type	1, Error message from NC program.		

ID 70557

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires controller restart.

ID 70558

Missing drive releases for driving out gantry difference.			
Description			
Response	Class	5	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [0.1 10^-3 mm or ø]	
	%3	Current value [0.1 10^-3 mm or ø]	
	%4	Current value [-]	
Error type	1, Error message from NC program.		

ID 70559

Drive not ready for driving out gantry difference.			
Description			
Response	Class	5	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [0.1 10^-3 mm or ø]	
	%3	Current value [0.1 10^-3 mm or ø]	
	%4	Current value [-]	
Error type	1, Error message from NC program.		

ID 70560

Distance between two interpolation points of plane compensation is negative.			
Description	The interval size configured in the configuration list is negative. This error message is output if a value less than zero was configured for one of the parameters P-COMP-00009, P-COMP-00032 or P-COMP-00033. The parameter causing the error message is output in value 3 of this message.		
Response	Class	6	Controlled halt of the axis, the control loop is closed. Plane compensation is deactivated.
Solution	Class	7	Check and modify the incorrect parameter, assign a value greater than zero.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [0.1 10^-3 mm or ø]	
		Incorrect value for P-COMP-00009, P-COMP-00032 or P-COMP-00033	
	%3:	Current value [-]	
		Name of the incorrect value (P-COMP-00009, P-COMP-00032 or P-COMP-00033)	
Error type	-		

ID 70561

Reference position could not be written to drive.			
Description	The value read by the drive after writing P-AXIS-00152 does not match with P-AXIS-00152.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check why the parameter was not written, e,g, write protection for parameter in drive.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Error value [0.1 10^-7 mm or ø]
		Value that was reread by the drive
	%3	Expected value[0.1 10^-7 mm or ø]
		Value that was written to the drive.
Error type	11, Error message from position controller.	

ID 70562

The function for writing drive parameters is already in use.			
Description	When the write function was started for the homing position in the drive, the system detected that another write operation is active in the drive. For example, this may be a #DRIVE command or the‚Move to fixed stop‘ (X[FIXED_STOP….] function.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check program sequence
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
Error type	11, Error message from position controller.		

ID 70563

Axis is to be homed with frequency response measurement active.			
Description			
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
Error type	11, Error message from position controller.		

ID 70565

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 70568

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	

Solution	Class	8	Requires controller restart.

ID 70569

Changing gantry link only possible at standstill.			
Description	A change in gantry coupling is only possible when all the axes involved are at standstill.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	5	Stop the axes involved before sending the command to change the gantry coupling.
Parameter	%1:	Current value [-]	
		Logical axis number P-AXIS-00016 of the slave axis belonging to a gantry combination	
	%2:	Current value [-]	
		Channel ID	
	%3:	Current value [-]	
		Value for gantry change request. 1- Disable gantry coupling 2- Disable all gantry couplings 3- Restore gantry coupling 3- Restore all gantry couplings	
	%4:	Upper limit value [-]	
		Logical axis number P-AXIS-00016 of master axis belonging to a gantry combination	
Error type	11, Error message from position controller.		

ID 70571 / 70572

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 70574

Changing gantry coupling with move to fixed stop not possible.	

Description	When a gantry link is disabled or restored (see [FCT-C11]), the Move to fixed stop function [FCT-M8] is still active on at least one gantry axis.
	
	Example:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`…`
	`N100 #GANTRY RELEASE [AX=X] (error 70574)`
	`…`
	
	Possible solution:
	`N10 G01 X100 F1000 X[FIXED_STOP ON]`
	`…`
	`N090 G01 X0 F1000 X[FIXED_STOP OFF]`
	`N100 #GANTRY RELEASE [AX=X] (Error 70574)`
	`…`

Response	Class	7	Controlled halt of the axis, the control loop is closed. Transition to error state.
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Error value [-]	
		Move to fixed stop active	
	%3:	Current value [-]	
		Channel number of the commanding CNC channel	
	%4:	Current value [-]	
		Job to the gantry system	
	%5:	Logical axis number [-]	
		Logical axis number P-AXIS-00070 of master axis	
Error type	11, Error message from position controller.		

ID 70575

Command and actual operating modes are configured for different transfer types.			
Description	With CANopen drives, the command and actual operation modes can be transferred in the cyclic process data (PDO) or in the acyclic data (SDO). However, the same transfer type must be configured for both data elements, i.e. either both transfer per PDO or both transfer per SDO. If different transfer types are configured, the error message is output at controller start-up and at the start of drive-controlled homing.		
Response	Class	6	Movement stop. Error message output, abort program
Solution	Class	6	Configure the two CANopen objects in the cyclic process data or transfer both by SDO.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis.	
	%2:	Current value [-]	
		Actual operation mode (Object 0x6061) is configured in the cyclic process data (PDO)	
	%3:	Current value [-]	
		Command operation mode (Object 0x6060) is configured in the cyclic process data (PDO)	
Error type	11, Error message from position controller.		

ID 70576

Sensor is completely retracted.			
Description	The current distance control offset (see [FCT-M3]) was frozen. However, the maximum excursion of the probe sensor was reached during axis motion (maximum excursion = P-AXIS-00420 - ½* P-AXIS-00421). A possible cause could be an increase in height of the workpiece.		
Response	Class	7	Immediate stop of the axis affected to avoid damage to the sensor. Distance control is turned off.
Solution	Class	6	Check the sensor and the workpiece

Parameter	%1:	Current value [0.1 µm or 0.0001°]
		Current position value from sensor
	%2:	Limit value [0.1 µm or 0.0001°]
		Upper limit value for sensor position
	%3:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%4:	Drive type [-]
		Drive type P-AXIS-00020 of the axis affected
	%5:	Block number [-]
		Block number in the CNC program where the error occurred
Error type	-	

ID 70582

Function call of external compensation returns an error.			
Description	An error occurred when external compensation is used. For more information, refer to the parameter of the error message or contact the supplier of the external compensation.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check error text and error numbers of the parameters.
Parameter	%1:	Current value [-]	
		Return value of the function	
	%2:	Current value [-]	
		Error number 1	
	%3:	Current value [-]	
		Error number 2	
	%4:	Current value [-]	
		Error text	
Error type	11, Error message from position controller.		

ID 70583

Warning level of compensation value of external compensation exceeded.			
Description	The compensated value of an axis exceeded the warning threshold specified in P-AXIS-00743.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check the limit values.
Parameter	%1:	Current value [-]	
		Calculated compensation offset of the axis	
	%2	Limit value [-]	
		Limit value P-AXIS-00743 of the axis	
	%3	Logical axis number [-]	
		Logical axis P-AXIS-00016 whose value was exceeded	
Error type	11, Error message from position controller.		

ID 70584

Compensation value of external compensation has exceeded permissible limit.			
Description	The compensated value of an axis exceeded the error threshold specified in P-AXIS-00744.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the set limit values and compensation offsets
Parameter	%1:	Current value [-]	
		Calculated compensation offset of the axis	
	%2:	Limit value [-]	
		Limit value P-AXIS-00744 of the axis	
	%3:	Logical axis number [-]	
		Logical axis P-AXIS-00016 whose value was exceeded	
Error type	11, Error message from position controller.		

ID 70585

The external compensation of axis is only possible with specified limits.			
Description	The maximum compensation offset of the axis at the specified index is 0.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and modify the limit values of the affected axis.
Parameter	%1:	Current value [-]	
		Axis index	
Error type	11, Error message from position controller.		

ID 70586

Maximum permissible axis acceleration exceeded by external compensation.			
Description	The limit value specified in P-AXIS-00745 was exceeded for the minimum acceleration caused by the compensation offset.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the offset values of the external compensation. Large jumps in offset values must be smoothed out.
Parameter	%1:	Current value [-]	
		Current value of the acceleration	
	%2:	Limit value [-]	
		Entered limit value P-AXIS-00745	
Error type	11, Error message from position controller.		

ID 70587

Interface of external compensation is not enabled.			
Description	The NC command #EXTCOMP was used. In order to use external compensation, the interface must be enabled by the parameter P-STUP-00110. This is not the case.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Enable the interface with P-STUP-00110

Parameter	%1:	Current value [-]
		State of the interface
Error type	11, Error message from position controller.	

ID 70588

Axis is already active in another channel when external compensation is activated.			
Description	The external compensation has access to all axes irrespective of the channel. When external compensation was enabled, the system detected that one axis is already moved by another channel. Compensation cannot be enabled.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the enable sequence of the external compensation. The external compensation must be enabled when all axes are at standstill.
Parameter	%1:	Current value [-]	
		Index of the external compensation	
	%2	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the active axis	
	%3:	Identifier [-]	
		Channel ID	
Error type	11, Error message from position controller.		

ID 70589

Axis part of the external compensation is not ready to move.			
Description	When the external compensation is enabled, all the axes involved must have an enable.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the enable of all axes involved; enable all axes.
Parameter	%1:	Current value [-]	
		Index of the external compensation	
	%2:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the axis with no enable	
Error type	11, Error message from position controller.		

ID 70590

Axis part of the external compensation is in error state.			
Description	When the external compensation is enabled, one axis is already in error state.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and correct the preceding error. Re-enable the external compensation.

Parameter	%1:	Current value [-]
		Index of the external compensation
	%2:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of the axis in error state
	%3:	Identifier [-]
		Channel number to which the axis belongs
	%4:	Identifier [-]
		Channel ID
Error type	11, Error message from position controller.	

ID 70591

Mode P-AXIS-00753 for Improved Position Control is invalid.			
Description	The parameter P-AXIS-00753 defines the mode for the the Improved Position Control function. An invalid mode was specified. The value range is 0 to 3.		
Response	Class	1	Warning output and disable the Improved Position Control function.
Solution	Class	1	Check and correct the parameterisation of P-AXIS-00753. Update the parameters.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Instance [-]	
		Incorrect step specified in P-AXIS-00753.	
	%3:	Error value [-]	
		Configured value of P-AXIS-00753.	
	%4:	Upper limit value [-]	
		Maximum value for the mode.	
	%5:	Status [-]	
		The Improved Position Control function is disabled by P-AXIS-00758.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70592

Frequency P-AXIS-00751 for Improved Position Control is out of permissible range.			
Description	The Improved Position Control has the function of damping specific frequencies. The specified value for the damped frequency P-AXIS-00751 is outside the permissible limit values.		
Response	Class	1	Warning output and disable the Improved Position Control function.
Solution	Class	1	Check and correct the parameterisation of P-AXIS-00751. Update the parameters.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Instance [-]
		Incorrect step specified in P-AXIS-00751.
	%3:	Error value [Hz]
		Configured value of P-AXIS-00751.
	%4:	Limit value[Hz]
		Limit value of the frequency which was overshot or undershot.
	%5:	Status [-]
		The Improved Position Control function is disabled by P-AXIS-00758.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70593

Filter order P-AXIS-00750 for Improved Position Control is out of permissible range.			
Description	The Improved Position Control function operates with a band-pass filter. The specified filter order P-AXIS-00750 of the band-pass filter is outside the permissible limit values.		
Response	Class	1	Warning output and disable the Improved Position Control function.
Solution	Class	1	Check and correct the parameterisation of P-AXIS-00750. Update the parameters.
Parameter	%1:	Current value [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Instance [-]	
		Incorrect step specified in P-AXIS-00750.	
	%3:	Error value [-]	
		Configured value of P-AXIS-00750.	
	%4:	Limit value [-]	
		Limit value which was overshot or undershot.	
	%5:	Status [-]	
		The Improved Position Control function is disabled by P-AXIS-00758.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70594

Quality factor P-AXIS-00752 for Improved Position Control is out of permissible range.			
Description	The Improved Position Control function operates with a band-pass filter. The filter quality factor P-AXIS-00752 is outside the permissible limit values.		
Response	Class	1	Warning output and disable the Improved Position Control function.
Solution	Class	1	Check and correct the parameterisation of P-AXIS-00752. Update the parameters.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Instance [-]
		Incorrect step specified in P-AXIS-00752
	%3:	Error value [-]
		Configured value of P-AXIS-00752
	%4:	Limit value [-]
		Limit value which was overshot or undershot.
	%5:	Status [-]
		The Improved Position Control function is disabled by P-AXIS-00758.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70595

Filter for function Improved Position Control could not be generated.			
Description	Several filters must be initialised for the Improved Position Control function. It was not possible to initialise a relevant filter.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.. The Improved Position Control function is disabled.
Solution	Class	6	Check and correct the filter parameters
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Instance [-]	
		Step where it was impossible to initialise the filter.	
	%3:	Current value [-]	
		Further information why the filter could not be initialised.	
	%4:	Error value [-]	
		Filter type could not be initialised.	
	%5:	Status [-]	
		The Improved Position Control function is disabled by P-AXIS-00758.	
Error type	11, Error message from position controller.		

ID 70599

Damping P-AXIS-00756 for Improved Position Control is out of permissible range.			
Description	The Improved Position Control function requires a specified damping in order to simulate mechanical vibrations. The specified damping P-AXIS-00756 is outside the permissible limit values.		
Response	Class	1	Warning output and disable the Improved Position Control function.
Solution	Class	1	Check and correct P-AXIS-00756. Update the parameters.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Instance [-]
		Incorrect step specified in P-AXIS-00756.
	%3:	Error value [-]
		Configured value of P-AXIS-00756.
	%4:	Limit value [-]
		Limit value which was overshot or undershot.
	%5:	Status [-]
		The Improved Position Control function is disabled by P-AXIS-00758.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70600

Function Improved Position Control may not be activated for gantry slave axes.			
Description	The parameter P-AXIS-00758 was enabled for a gantry slave axis. This is not permitted. The Improved Position Control function may only be enabled for gantry master axes. Gantry slave axes are internally calculated based on the parameters of the associated master axis.		
Response	Class	7	Controlled halt of the axis, the control loop is closed. The Improved Position Control function is disabled..
Solution	Class	6	Check and disable the P-AXIS-00758 of the gantry slave axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis.	
	%3:	Error value [-]	
		Check whether the axis is a gantry slave axis.	
	%4:	Status [-]	
		The Improved Position Control function is disabled by P-AXIS-00758.	
Error type	11, Error message from position controller.		

ID 70601

Change of parameters for function Improved Position Control are only permitted in axis standstill.			
Description	Updating and changing parameters for the Improved Position Control function are only permitted when the axis is at standstill. The axis is not at standstill.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Carry out a reset. The new parameters are adopted after the axis is reset.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Error value [-]
		Identifier indicating that axis is interpolated.
	%3:	Current value [-]
		Value of the old parameter.
	%4:	Current value [-]
		Value of the new parameter.
Error type	11, Error message from position controller.	

ID 70602

Function Improved Position Control not possible during active operation mode P-AXIS-00320.			
Description	The parameter P-AXIS-00320 indicates the operation mode of the axis. The Improved Position Control function can only be used in CNC position control or drive-internal position control.		
Response	Class	7	Controlled halt of the axis, the control loop is closed. The Improved Position Control function is disabled..
Solution	Class	6	Check and correct P-AXIS-00320 Set operation mode via P-AXIS-00320 to “DRIVE_POSITION_CONTROL” or “CNC_POSITION_CONTROL”.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis.	
	%3:	Error value [-]	
		Operation mode P-AXIS-00320 of axis affected.	
	%4:	Status [-]	
		The Improved Position Control function is disabled by P-AXIS-00758.	
Error type	11, Error message from position controller.		

ID 70603

Numerator P-AXIS-00754 for Improved Position Control is zero or negative.			
Description	The Improved Position Control function outputs a weighted additive velocity. The numerator P-AXIS-00754 for the weighting factor may not be 0 or less than 0, otherwise the factor cannot be calculated correctly. The associated denominator of the weighting factor is defined in P-AXIS-00755.		
Response	Class	1	The Improved Position Control function is disabled.
Solution	Class	1	Check and correct P-AXIS-00754. Update the parameters.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Instance [-]
		Incorrect step specified in P-AXIS-00754.
	%3:	Error value [-]
		Configured value of P-AXIS-00754.
	%4:	Lower limit value [-]
		Lower limit value.
	%5:	Status [-]
		The Improved Position Control function is disabled by P-AXIS-00758.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70604

Denominator P-AXIS-00755 for Improved Position Control is zero or negative.			
Description	The Improved Position Control function outputs a weighted additive velocity. The denominator P-AXIS-00755 for the weighting factor may not be 0 or less than 0, otherwise the factor cannot be calculated correctly. The associated numerator of the weighting factor is defined in P-AXIS-00754.		
Response	Class	1	The Improved Position Control function is disabled.
Solution	Class	1	Check and correct P-AXIS-00755. Update the parameters.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Instance [-]	
		Incorrect step specified in P-AXIS-00755.	
	%3:	Error value [-]	
		Configured value of P-AXIS-00755.	
	%4:	Lower limit value [-]	
		Lower limit value.	
	%5:	Status [-]	
		The Improved Position Control function is disabled by P-AXIS-00758.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70605

Weighting factor for Improved Position Control exceeds maximum limit.			
Description	The Improved Position Control function outputs a weighted additive velocity. The weighting factor is defined in P-AXIS-00754 / P-AXIS-00755. The weighting factor exceeds the maximum value and is limited to the maximum value in order to prevent excessively high additive velocities.		
Response	Class	1	Warning output and weighting factor limited to maximum value.
Solution	Class	1	Check and modify P-AXIS-00754 and P-AXIS-00755 Update the parameters.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Instance [-]
		Step in which the weighting factor exceeds the upper limit value.
	%3:	Error value [-]
		Calculated weighting factor from P-AXIS-00754 / P-AXIS-00755.
	%4:	Corrected value [-]
		P-AXIS-00754 is set to the upper limit value.
	%5:	Corrected value [-]
		P-AXIS-00755 is set to 1.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70606

Velocity factor P-AXIS-00757 for Improved Position Control is out of permissible range.			
Description	The Improved Position Control function outputs a weighted additive velocity. To protect operators and the machine, the output additive velocity may not exceed an upper limit value. The parameter P-AXIS-00757 defines this limit and is also restricted for this reason. In addition, it is not permitted to specify a value less than 0 for P-AXIS-00757.		
Response	Class	1	Output the warning and limit of P-AXIS-00757
Solution	Class	1	Check and modify P-AXIS-00757. Update the parameters.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Error value [-]	
		Configured value of P-AXIS-00757.	
	%3:	Limit value [-]	
		If P-AXIS-00757 exceeds the maximum, the upper limit is output. If P-AXIS-00757 undershoots the minimum, the lower limit is output.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00757	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70607

Cyclic telegram of function Improved Position Control contains no additive command velocity.			
Description	The Improved Position Control function outputs a weighted additive velocity. Here, the additive velocity must be configured in the cyclic telegram. For SERCOS, the S-0-0037 parameter. Must be configured. For CANopen, the parameter “Additive velocity command value” must be configured in the process data.		
Response	Class	7	Controlled halt of the axis, the control loop is closed. The Improved Position Control function is deactivated.
Solution	Class	6	Check and modify the parameterisation for the additive velocity in the cyclic chart

Parameter	%1:	Current value [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Drive type [-]
		Drive type P-AXIS-00020 of the affected axis.
	%3:	Current value [-]
		Operation mode P-AXIS-00320 of concerned axis.
	%4:	Error value [-]
		Check whether an additive velocity is configured.
	%5:	Status [-]
		The Improved Position Control function is deactivated by P-AXIS-00758.
Error type	11, Error message from position controller.	

ID 70610

The table of plane compensation is not big enough.			
Description	The table for the compensation table is determined by specifying the maximum table entries (P-COMP-00061) for the plane compensation of an axis and specifying the last indices of the master axes (P-COMP-00010 and P-COMP-00011). The product of the indices is greater than the number of table entries available.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the parameters for plane compensation of this axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value [-]	
		Parameter of last index of master axis 1 P-COMP-00010	
	%3:	Current value [-]	
		Last parameterised index P-COMP-00011 of master axis 2	
	%4:	Error value [-]	
		Product of the two indices	
	%5:	Upper limit value [-]	
		Parameter of maximum number of table entries P-COMP-00061	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70612

Drive re-enabled during active rundown.			
Description	When measuring to a fixed stop, a measured value was acquired before the configured enable position was reached to acquire the measured value (P-AXIS-00776, P-AXIS-00777).		
Response	Class		Error message output, NC program aborted
Solution	Class		Check parameterisation for measuring to a fixed stop (P-AXIS-00774, P-AXIS-00775, P-AXIS-00776, P-AXIS-00777, P-AXIS-00778).

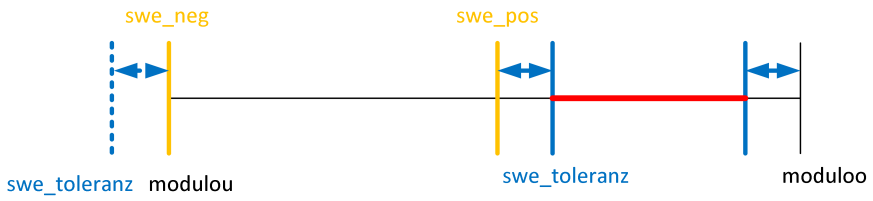
Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of affected axis
	%2:	Current value [0.1 10^-3 mm or ø]
		Measured value acquired
	%3:	Error value [-]
		Start position for acquisition of the measured value (a value of zero in connection with value 5 = FALSE means that the start position was not yet reached). The value is determined by P-AXIS-00776 and/or P-AXIS-00777.
	%4:	Current value [-]
		Direction of axis motion TRUE means that the axis was moving in a positive direction.
	%5:	Current value [-]
		Flag indicating whether the start position for acquiring the measured value was overshot. If the flag is TRUE, the start position is in value 3, otherwise it is zero.
Error type	-	

ID 70613

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 70614

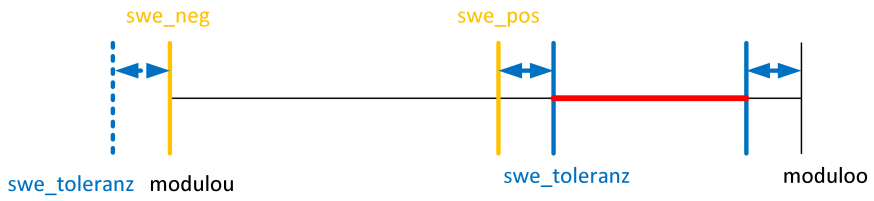
With positive axis velocity the actual position of modulo axis is out of range of the software limit switches.

Description	The current position of the modulo axis with a positive velocity has reached an invalid range defined by the software limit switches. This error only occurs if a software limit switch is shifted to a value outside the modulo range by the specified tolerance.   Possible causes: • Incorrect the NC program • Axis oscillations are too large. For further information, see [FCT-A2// Description]		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Possible solutions: • Check and correct the NC program • Check the gain factor (P-AXIS-00099) to reduce axis oscillations • Check and increase the tolerance P-AXIS-00179

Parameter		
	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Current value [0.1 10^-3 mm or ø]
		Current actual position of the axis
	%3:	Lower limit value [0.1 10^-3 mm or ø]
		Lower limit of the invalid modulo range defined by the software limit switches
	%4:	Current value [0.1 10^-3 mm or ø]
		Configured tolerance P-AXIS-00179 of the software limit switch
	%5:	Block number [-]
		Block number of the NC program where the error occurred.
Error type	-	

ID 70615

With negative axis velocity the actual position of modulo axis is out of range of the software limit switches.

Description	The current position of the modulo axis with a negative velocity has reached an invalid range defined by the software limit switches. This error only occurs if a software limit switch is shifted to a value outside the modulo range by the specified tolerance. See figure:   Possible causes: • Incorrect the NC program • Axis oscillations are too large. For further information, see [FCT-A2// Description]		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Possible solutions: • Check and correct the NC program • Check the gain factor (P-AXIS-00099) to reduce axis oscillations • Check and increase the tolerance P-AXIS-00179

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Current value [0.1 10^-3 mm or ø]
		Current actual position of the axis
	%3:	Lower limit value [0.1 10^-3 mm or ø]
		Lower limit of the invalid modulo range defined by the software limit switches
	%4:	Current value [0.1 10^-3 mm or ø]
		Configured tolerance P-AXIS-00179 of the software limit switch
	%5:	Block number [-]
		Block number of the NC program where the error occurred.
Error type	-	

ID 70616

No suitable process data during homing with zero pulse search.			
Description	A homing function was started containing parameters for a zero pulse search (see P-AXIS-00084). However, the control and status information for the zero pulse search are not configured in the drive process data.		
Response	Class	5	Abrupt stop for the defective axis, feedhold for the other axes.

Solution	Class	6	Either deactivate the zero pulse search (P-AXIS-00084) or supplement the process data.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Drive type [-]	
		Drive type (P-AXIS-00020) of affected axis	
Error type	11, Error message from position controller.		

ID 70617

While measuring on fixed stop the latch condition was valid before reaching the latch start position.			
Description	During Measuring to fixed stop, the latch condition set in the axis parameters P-AXIS-00774, P-AXIS-00775 and P-AXIS-00778 was already active although the minimum path set in P-AXIS-00776 and P-AXIS-00777 to activate the latch condition was not completed.		
Response	Class	2	Error message output abort program.
Solution	Class	3	Check and modify the axis parameters P-AXIS-00774, P-AXIS-00775 and P-AXIS-00778.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Amount of position lag	
	%3:	Current value [0.1 10^-3 mm or ø]	
		Position lag limit (P-AXIS-00774)	
	%4:	Upper limit value [0.1 10^-3 mm or ø]	
		Limit for movement during fixed stop detection (P-AXIS-00778)	
	%5:	Current value [0.1 10^-3 mm or ø]	
		Current value of movement during fixed stop detection.	
Error type	1, Error message from NC program.		

ID 70619

Generation of smoothing filter with given parameters not possible.			

Description	A filter for smoothing the measured distance values was parameterised for distance control (see FCT-M3) However, no valid filter can be created with the given filter parameters.
	
	`%Example`
	`;Select filter type`
	`N10 Z[DIST_CTRL FILTER_TYPE=KALMAN_MA]`
	`; Parametrise filter`
	`N20 Z[DIST_CTRL DISTC_N_CYCLES=30 KALMAN_SIGMA=1000]`
	`; Activate distance control`
	`N30 Z[DIST_CTRL ON CONST_DIST SET_DIST=1].`
	`; …`		

Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	Check and modify the required parameter.

Parameter	%1:	Logical axis number[-]
		Logical axis number (P-AXIS-00016) of axis affected
	%2:	Current value [-]
		Return value of filter generation
	%3:	Current value[-]
		Active filter type.
Error type	11, Error message from position controller.	

ID 70620

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 70621

The measured position is before the release position for probing in the movement block.			
Description	When measuring to a fixed stop, a measured value was acquired before the configured enable position for acquiring the measured value (P-AXIS-00776, P-AXIS-00777).		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check parameterisation for measuring travel to fixed stop (P-AXIS-00774, P-AXIS-00775, P-AXIS-00776, P-AXIS-00777, P-AXIS-00778).

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of concerned axis
	%2:	Current value [0.1 10^-3 mm or ø]
		Measured value acquired
	%3:	Current value [0.1 10^-3 mm or ø]
		Start position for acquisition of the measured value (a value of zero in connection with value 5 = FALSE, means that the start position is not yet reached). The value is determined by P-AXIS-00776 and/or P-AXIS-00777.
	%4:	Current value [-]
		Direction of axis motion, TRUE means that the axis was moving in a position direction.
	%5:	Current value [-]
		Flag indicating whether the start position for acquiring the measured value was overshot. If the flag is TRUE, the start position is in value 3, otherwise it is zero.
Error type	1, Error message from NC program.	

ID 70622

Master axis for crosstalk compensation not available.			
Description	The specified master axis (P-COMP-00063) for crosstalk compensation does not exist.		
Response	Class	7	Controlled stop of axis affected

Solution	Class	7	Specify a valid logical axis number, see P-COMP-00063
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected, see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Specified logical axis number of the master axis which does not exist. (see P-COMP-00063)	
	%3:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70625

No compensation value list available for crosstalk compensation.			
Description	The function crosstalk compensation (P-AXIS-00789) was selected for an axis in the axis parameters; however, the compensation value list containing the necessary correction values is missing.		
Response	Class	7	Controlled axis stop and disable crosstalk compensation
Solution	Class	7	If crosstalk compensation is enabled for an axis, a compensation value list must also be specified for it in the start-up list P-STUP-00016,, P-STUP-00017, P-STUP-00036). Example (extract from start-up list): zahl_kw 1 achs_kw[0] ..\listen\achskw3.lis achs_kw_log_ax_nr[0] 3
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected, see P-AXIS-00016	
Error type	11, Error message from position controller.		

ID 70626

The interpolation points for crosstalk compensation are not sorted in ascending order.			
Description	Compensation values are specified in the compensation value lists at defined interpolation points for crosstalk compensation. Compensation values are determined between these interpolation points by linear interpolation. Therefore, they must be arranged according to their position in an ascending order. A possible error cause can be the use of fewer interpolation points than the number given in the parameter P-COMP-00065 since the positions of the unused points are set to zero.		
Response	Class	7	Controlled stop of axis affected
Solution	Class	7	Correct compensation value list, see P-COMP-00066

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis affected, see P-AXIS-00016
	%2:	Current value [-]
		Position of last interpolation point, see P-COMP-00066
	%3:	Error value [-]
		Position of current interpolation point, see P-COMP-00066
	%4:	Current value [-]
		Index of current interpolation point
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70627

Invalid logical axis number of master axis for crosstalk compensation.			
Description	Crosstalk compensation was enabled for the axis in the axis parameters. However, the logical axis number of the master axis specified in the compensation value list (the axis whose position affects the slave axis, see P-COMP-00063) is invalid		
Response	Class	7	Controlled axis stop and disable crosstalk compensation
Solution	Class	7	Specify a valid master axis, see P-COMP-00063
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected, see P-AXIS-00016	
	%2:	Logical axis number [-]	
		Invalid logical axis number of the master axis, see P-COMP-00063	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70629

Number of cycles for coupling/decoupling for crosstalk compensation exceeds permissible limit.			
Description	The compensation values of crosstalk compensation can be executed and distributed over several position controller cycles by means of a filter. The number of filter cycles used is specified in parameter P-COMP-00064 in the compensation value list. However, the value found is too high. The number of filter cycles is reduced to the maximum permissible value.		
Response	Class	1	Correct the parameter to the maximum possible number of cycles
Solution	Class	1	Correct the value of the parameter P-COMP-00064 in the compensation value list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected, see P-AXIS-00016	
	%2:	Current value [-]	
		Given number of filter cycles, see P-COMP-00064	
	%3:	Corrected value [-]	
		Maximal number of filter cycles permitted	
	%4:	Block number [-]	
		Block number of the NC program where the error occurred	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70631

Activation of crosstalk compensation without selection via axis parameter not possible.			
Description	To enable crosstalk compensation for an axis in the NC program, it must be enabled in the axis parameter list (see P-AXIS-00789).		
Response	Class	7	Stop of axis, crosstalk compensation remains inactive.
Solution	Class	6	Enable crosstalk compensation (see P-AXIS-00789)
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Parameter value P-AXIS-00789 to enable crosstalk compensation	
Error type	11, Error message from position controller.		

ID 70632

Activation of crosstalk compensation not possible since compensation table is invalid.			
Description	Crosstalk compensation cannot be enabled since there is no compensation value list or the list contains errors.		
Response	Class	7	Stop of axis, crosstalk compensation remains inactive.
Solution	Class	6	Correct compensation table.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected, see P-AXIS-00016	
Error type	11, Error message from position controller.		

ID 70634

The programmed acceleration in #GEAR LINK command is greater than maximum acceleration.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [mm/s² or °/s²]	
	%3:	Upper limit value [mm/s² or °/s²]	
	%4:	Current value [-]	
	%5:		
Error type	-		

ID 70635

Error determining the current distributed clocks time.			
Description	The current time of Distributed Clocks in the fieldbus could not be determined.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.

Solution	Class	7	Check whether Distributed Clocks is active in the fieldbus.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	State [-]	
		Measuring methods used	
Error type	1, Error message from NC program.		

ID 70636

Timestamp could not be assigned to the distributed clocks times.			
Description	An interpolation is made between two cycles to calculate the position of the drive when a measuring event occurs. For this purpose, the high-resolution timestamp of the digital input terminal must be assigned to the Distributed Clocks times in the cycles before and after the timestamp. An error occurred on assigning the timestamp.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	Check whether Distributed Clocks is active in the fieldbus. Resynchronise Distributed Clocks.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	State [-]	
		Timestamp of the digital input terminal	
	%3:	State [-]	
		Timestamp with shift times considered	
	%4:	Lower limit value [-]	
		Smallest stored Distributed Clocks time	
	%5:	Upper limit value [-]	
		Largest stored Distributed Clocks time	
Error type	1, Error message from NC program.		

ID 70642

Axis is in control during homing although P-AXIS-00803 is set.			
Description	The parameter P-AXIS-00803 sets that no controller enables may be set during drive-controlled homing. This can be used if the axis is homed without axis motion, e.g. only by setting an actual position. The error message is output if P-AXIS-00803 is set and the axis is still enabled by the HLI while homing.		
Response	Class	2	Error message output and program abort.
Solution	Class	3	Set P-AXIS-00803 to zero or remove enable via the HLI while homing.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
Error type	-		

ID 70643

Drive delivers invalid data for axis at standstill.	
Description	The drive delivers invalid data over the fieldbus for an axis at standstill.

Response	Class	5	Controlled halt of the axis, the control loop is open.
Solution	Class	6	Check the bus connection to the drive and the drive status. Possible causes: • Interruption of bus connection to drive. • The drive is not ready for cyclic data exchange.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis	
	%3:	Current value [-]	
		Validity flag for drive actual data.	
	%4:	Block number [-]	
		Current block number in the NC program	
Error type	-		

ID 70644

Input position of axis-specific forward transformation out of permissible value range.			
Description	An axis-specific transformation is activated for the axis (P-AXIS-00370). Depending on the mapping rule, the transformation has a limited motion range. The axis position for the forward transformation is outside this valid range. Possible solutions are: • Check the parameter settings of the axis transformation (P-AXIS-00369) • Check the homing position of the axis		
Response	Class	2	Immediate stop of the axis affected
Solution	Class	3	Check the parameter settings of the axis transformation and axis homing position
Parameter	%1:	Logical axis number	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		ID of the axis transformation used P-AXIS-00370	
	%3:	Error value [0.1 10^-3 mm or ø]	
		Invalid axis position	
	%4:	Lower limit value [0.1 10^-3 mm or ø]	
		Minimum permissible axis position	
	%5:	Upper limit value [0.1 10^-3 mm or ø]	
		Maximum permissible axis position	
Error type	-		

ID 70645

Input position of axis-specific backward transformation out of permissible value range.			
Description	An axis-specific transformation is activated for the axis (P-AXIS-00370). Depending on the mapping rule, the transformation has a limited motion range. The axis position specified in the NC program is outside this valid range.		
Response	Class	2	Immediate stop of the axis affected
Solution	Class	3	Check and modify the NC program

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Current value [-]
		ID of the axis transformation used P-AXIS-00370
	%3:	Error value [0.1 10^-3 mm or ø]
		Invalid axis position
	%4:	Lower limit value [0.1 10^-3 mm or ø]
		Minimum permissible axis position
	%5:	Upper limit value [0.1 10^-3 mm or ø]
		Maximum permissible axis position
Error type	-	

ID 70647

No criteria configured for detection of a fixed stop.			
Description	No detection criteria were parameterised for a function that requires fixed stop detection.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	One of the detection criteria must be parameterised depending on the function. The following standard detection parameters can be used for this: • P-AXIS-00769 (position lag) • P-AXIS-00817 (velocity) • P-AXIS-00818 (torque)
Parameter	%1:	Logical axis number [-]	
		Logical axis number of drive see P-AXIS-00016	
Error type	11, Error message from position controller.		

ID 70648

Configured detection torque exceeds the maximum torque when referencing to fixed stop.			
Description	The detection torque is greater than the maximum torque that was configured for homing to a fixed stop.		
Response	Class	1	Warning output and value corrected to the maximum torque during homing to a fixed stop.
Solution	Class	1	Set value smaller than 1000 for P-AXIS-00347.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value [0.1%]	
		Parameterised detection torque.	
	%3:	Upper limit value [-]	
		Parameterised torque for homing to a fixed stop.	
	%4:	Corrected value	
		Corrected detection torque	
Error type	11, Error message from position controller.		

ID 70649

Homing to fixed stop for coupled axis not allowed.			
Description	When an axis coupling is active, it is not possible to home to a fixed stop.		
Response	Class	6	Controlled stop for the axis group.´
Solution	Class	6	• Select other homing method. • Decouple the axes before homing
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
Error type	11, Error message from position controller.		

ID 70650

Fixed stop detected prematurely.			
Description	A fixed stop was detected during homing to a fixed stop before the minimum distance (P-AXIS-00344) was covered.		
Response	Class	5	Abrupt stop for the defective axis, feedhold for the other axes.
Solution	Class	6	Check minimum distance for homing to a fixed stop and the detection criteria and perform NC reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Limit value [µm/s]	
		Detection velocity during homing to a fixed stop.	
	%3:	Limit value	
		Parameterised detection torque for homing to a fixed stop.	
	%4:	Limit value [0.1 µm]	
		Position lag for fixed stop detection.	
	%5:	Limit value [0.1 µm]	
		Minimum distance within which no fixed stop may be detected.	
Error type	11, Error message from position controller.		

ID 70651

Retraction distance after fixed stop detection for gantry axis to small.			
Description	An insufficient retraction distance was parameterised for a hard gantry axis during homing to a fixed axis. The retraction distance after fixed stop detection (P-AXIS-00348) must be greater than the resettable path distance (P-AXIS-00072) so that the gantry difference can be definitely extended.		
Response	Class	1	Warning output, correct the resettable path distance (P-AXIS-00348) of the slave axis onto the retraction distance (P-AXIS-00072) of the master axis. This makes the resettable path distance smaller.
Solution	Class	1	Parameterise a longer retraction distance (P-AXIS-00072) in the master axis.

Parameter	%1:	Logical axis number [-]
		Logical axis number of the master axis, see P-AXIS-00016
	%2:	Error value [0.1 µm]
		Retraction distance of the master axis, see P-AXIS-00348
	%3:	Logical axis number [-]
		Logical axis number of the slave axis P-AXIS-00016
	%4:	Error value[0.1 µm]
		Resettable path distance of the slave axis, see P-AXIS-00072
	%5:	Corrected value [0.1 µm]
		Corrected resettable path distance of the slave axis, see P-AXIS-00072
Error type	-	

ID 70652

Main encoder not found.			
Description	The main encoder was configured in the axis parameter list using P-AXIS-00823. The configured entry was not found.		
Response	Class	5	Abrupt stop for the defective axis, feedhold for the other axes. Position controller goes to error state that can be rectified by a reset.
Solution	Class	6	Check and modify P-AXIS-00823. The parameterised drive signal must match the data item of the parameterised nc_ref.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of master axis	
	%2:	Error value [-]	
		Main encoder signal which is to be used as the actual position value. P-AXIS-00823	
Error type	11, Error message from position controller.		

ID 70653

Main encoder can not be changed via list update.			
Description	An attempt was made to change the main encoder signal (P-AXIS-00823) using an axis list update. This is not possible. The controller must be restarted to change the main encoder signal.		
Response	Class	7	Controlled halt of the axis, the control loop is closed. The axis goes to error state. The main encoder signal is not adopted.
Solution	Class	6	Perform a reset. A change to the main encoder signal (P-AXIS-00823) requires a controller restart.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of drive affected	
	%2:	Error value [-]	
		New parameterised value for P-AXIS-00823	
	%3:	Corrected value [-]	
		Previous value of the main encoder signal (P-AXIS-00823)	
Error type	11, Error message from position controller.		

ID 70654

Secondary encoder can not be changed via list update.			
Description	An attempt was made to change the secondary encoder signal (P-AXIS-00824) using an axis list update. This is not possible. The controller must be restarted to change the secondary encoder signal.		
Response	Class	6	Controlled halt of the axis, the control loop is closed. The axis goes to error state. The secondary encoder signal is not adopted.
Solution	Class	6	Perform a reset. A change to the secondary encoder signal (P-AXIS-00824) requires a controller restart.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of drive affected	
	%2:	Error value [-]	
		New parameterised value for P-AXIS-00824	
	%3:	Corrected value [-]	
		Previous value of the secondary encoder signal (P-AXIS-00824)	
Error type	11, Error message from position controller.		

ID 70655

Secondary encoder not found.			
Description	A secondary encoder was configured in the axis parameter list using P-AXIS-00824. The configured entry was not found.		
Response	Class	1	Warning output.
Solution	Class	1	Check and modify the entry for the secondary encoder P-AXIS-00824. The parameterised drive signal must match the data item of the parameterised nc_ref.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of master axis	
	%2:	Error value [-]	
		Secondary encoder signal P-AXIS-00824 that is to be used.	
Error type	11, Error message from position controller.		

ID 70656

Drive encoder range can not be changed by a list update.			
Description	The modulo range of the drive encoder (P-AXIS-00528) cannot be changed by list update in its present state.		
Response	Class	1	Warning output; current value is retained
Solution	Class	1	Carry out the following steps to change the parameter by a list update: • the drive release be not set. • Set the value 0x00400000 in P-AXIS-00015

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Error value[-]
		New value for P-AXIS-00528 to be used from the list update.
	%3:	Corrected value [-]
		Current value of P-AXIS-00528
	%4:	Error value [-]
		State of tracking mode, expected value: True (1)
	%5:	Error value[-]
		State of drive release, expected value: False (0)
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70657

Adaptation axis for crosstalk compensation does not exist.			
Description	The logical axis number of the specified adaptation axis (P-COMP-00080) for crosstalk compensation does not exist.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	Check and modify the logical axis number in P-COMP-00080.
Parameter	%1:	Logical axis number [-]	
		Specified logical axis number of the adaptation axis which does not exist. (see P-COMP-00080)	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70658

Adaptation axes for crosstalk compensation must be different.			
Description	The specified adaptation axes for crosstalk compensation may not have the same logical axis number.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	Check and modify the logical axis numbers in P-COMP-00080 and P-COMP-00081
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
	%2:	Logical axis number [-]	
		Specified Logical axis number P-AXIS-00016 of the first adaptation axis (see P-COMP-00080).	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70659

Invalid acceleration in crosstalk compensation.			
Description	The specified acceleration in P-COMP-00066 for use with crosstalk compensation is zero. This is not permitted.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	Check and modify the acceleration in P-COMP-00066..

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of the axis affected
Error type	2, Error message by data transfer from parameter list into control device.	

ID 70660

Crosstalk compensation: Invalid order of positions of an adaptation axis.			
Description	The positions of the master axis in Mode 2 of crosstalk compensation (P-COMP-00082) must be specified in ascending order.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	7	Check and modify the sorting order of the compensation value list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 70661

Axis exchange while distance control is active and sensor source is 'VARIABLE'.			
Description	Distance control is active and the sensor signal is transferred via a channel-specific variable. An axis exchange is not permitted since otherwise the axis has no access to the variable.		
Response	Class	6	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program. Avoid using axis exchange or a global variable to transfer the sensor signal.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected.	
	%2:	Block number [-]	
		Block number of the axis exchange commanded	
	%3:	Current value [-]	
		Active distance control signal source	
Error type	-		

ID 70662

Calculated metric position out of permissible range.			
Description	The position set by the NC program is outside the permissible value range. (+-214.746m). See Setting an axis position in the channel (SET_POSITION)		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the position to be set. Select a position that is within the permitted metric value range.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of the axis affected
	%2:	Error value [-]
		Specified axis position
	%3:	Upper limit value [-]
		Maximum possible axis position
	%4:	Lower limit value [-]
		Minimum possible axis position
Error type	11, Error message from position controller.	

ID 70671

Drive does not follow command values.			
Description	The monitor of the drive command values reports an error.		
Response	Class	5	Controlled stop of defective axis Axes controlled by the CNC: Abrupt stop of defective axis.
Solution	Class	5	Check why the drive is not applying the command values. Possible causes: • Drive is in a limit • Drive is generating command values autonomously. If the drive should generate command values autonomously, tracking mode must first be enabled.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of axis affected
	%2:		Drive type [-]
			Drive type P-AXIS-00020 of the axis affected
Error type	11, Error message from position controller.		

ID 70681

Collision pair is already active.			
Description	The NC command #COLL MON ON[…] is used to monitor an axis for collision with another axis. The programmed collision pair is already active and can therefore not be programmed a second time.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify the NC program; remove the second activation of the collision pair.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of first programmed axis.
	%2:		Logical axis number[-]
			Logical axis number P-AXIS-00016 of second programmed axis.
Error type	11, Error message from position controller.		

ID 70682

For this axis collision monitoring is already active.

Description	The NC command #COLL MON ON[…] is used for an axis to monitor collision at axis level. Collision monitoring is already active for this axis.		
Response	Class	5	Abrupt stop for the defective axis, feedhold for the other axes.
Solution	Class	6	Check and modify NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the programmed axis	
Error type	11, Error message from position controller.		

2.9 Axis filter errors (ID-range 80000-89999)

2.9.1 ID-range 80000-80249

ID 80000

Dead time exceeds maximum value.			
Description	The dead time specified for the simulation drive is invalid. The parameter P-AXIS-00194 specifies this as a multiple of the cycle time for geometry processing.		
Response	Class	1	Warning, reduce the dead time to the maximum permissible value
Solution	Class	1	Correct the dead time in the axis parameter, see P-AXIS-00194
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Specified value for dead time, see P-AXIS-00194	
	%3:	Upper limit value [-]	
		Maximum permissible value for dead time	
Error type	-		

ID 80001 / 80004

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 80005

Number of axis group exceeds range or is zero.			
Description	When adopting the axis configuration, the system detected that the number of the axis group is invalid. Axis groups are defined in the channel parameter list (P-CHAN-00023). Only one axis group is supported by the controller.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct the configuration of the axis groups in the channel parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-CHAN-00035, P-AXIS-00016	
	%2:	Current value [-]	
		Invalid axis group number	
	%3:	Upper limit value [-]	
		Maximum permissible number of axis groups	
Error type	-		

ID 80006

Axis number exists already.

Description	The given logical axis number is already used for a different axis or spindle.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct axis configuration. See [CHAN// Section: Configuration of the path axes]
Parameter	%1:	Logical axis number [-]	
		Already existing logical axis number, see P-CHAN-00035	
	%2:	Current value [-]	
		Axis index	
Error type	-		

ID 80007

Number of axis in group is 0 or greater than permissible maximum.			
Description	The received axis number is greater than the maximum permissible number of axes in an axis group (application-dependent) or it is zero.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct axis configuration. See [CHAN// Section: Configuration of the path axes]
Parameter	%1:	Current value [-]	
		Invalid axis number	
	%2:	Upper limit value [-]	
		Maximum permissible number of axes in an axis group (application-dependent)	
Error type	-		

ID 80008

Axis index already used within group.			
Description	Two or more axes with the same index exist in the axis group.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct axis configuration See [CHAN // Section: Configuration of the path axes]
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [-]	
		Axis index	
	%3:	Current value [-]	
		Number of affected axis group	
Error type	-		

ID 80009

No axis specified within axis data of filter.			
Description	No axes are configured.		
Response	Class	3	Command is aborted
Solution	Class	6	Correct axis configuration

Error type	-

ID 80010

Number of axis groups not coherent.			
Description	The specified numbers of axis groups are not coherent.		
Response	Class	3	Abort job processing
Solution	Class	6	Correct the configuration of the axis groups. See [CHAN// Section: Axis group structure]
Parameter	%1:	Current value [-]	
		Expected axis group number	
Error type	-		

ID 80011

Number of axis not coherent within group.			
Description	The specified axis numbers are not contiguous within the axis group. Example (extract from channel parameters list): gruppe[0].achse[0].bezeichnung X AXIS1 ... gruppe[0].achse[2].bezeichnung A ... gruppe[0].achse[3].bezeichnung W1 ... Solution: gruppe[0].achse[0].bezeichnung X AXIS1 ... gruppe[0].achse[1].bezeichnung A ... gruppe[0].achse[2].bezeichnung W1 ...		
Response	Class	3	Command is aborted
Solution	Class	6	Correct axis configuration.
Parameter	%1:	Current value [-]	
		Number of axis group affected	
	%2:	Expected value [-]	
		Expected axis number	
Error type	-		

ID 80016 - 80043

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 80044

Attenuation greater or equal to 1.			
Description	The specified attenuation value for the simulation drive (P-AXIS-00020=4) is invalid. It must be smaller than one. Enter the attenuation value as a quotient of the parameters P-AXIS-00050 and P-AXIS-00051 in the axis parameter list. See [AXIS// Section: Drive type simulation]		
Response	Class	1	Warning, reduce attenuation value to maximum permissible value.
Solution	Class	1	Correct attenuation in the axis parameter
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Specified attenuation value, see P-AXIS-00050, P-AXIS-00051	
	%3:	Limit value [-]	
		Maximum permissible attenuation value	
	%4:	Corrected value [-]	
		Corrected attenuation value	
Error type	-		

2.10 Tool radius compensation error (ID-range 90000-99999)

2.10.1 ID-range 90000-90249

ID 90002 - 90005

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 90008

Selection of tool radius compensation within circular motion block.			
Description	After selection of the TRC a linear block must follow; a circular block is not possible.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program
Parameter	%1:	Block number [-]	
Error type	1, Error message from NC program.		

ID 90009 / 90010

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90011

Block memory for non-relevant channel blocks engaged.			
Description	Non-relevant blocks are blocks without motion data for the TRC. M functions are an example of non-relevant blocks: The number of consecutive non-relevant channel blocks is greater than the internal block memory allocated.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program. Interrupt the sequence of non-relevant channel blocks by a block with motion data.
Parameter	%1:	Upper limit value [-]	
		Number of the possible non-relevant blocks in sequence.	

Error type	1, Error message from NC program.

ID 90012

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90013

Deselection of tool radius compensation within circular motion block.			
Description	After deselecting the TRC, a linear block must follow; a circular block is not possible.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program
Error type	Abort NC program processing.		

ID 90014

Deselection of tool radius compensation not permitted using direct selection.			

Description	It is not possible to deselect tool radius compensation again after the first motion block when direct selection is selected.
	
	Example with error:
	`%wrk_90014.err`
	`N10 G00 X0 Y0 Z0`
	`N20 V.G.WZ_AKT.R=5`
	`N30 G01 X10 Y10 F1000`
	`N40 G41 G138 (direct TRC selection mode)`
	`N50 G01 X80`
	`N60 G40`
	`N70 G01 X90 Y0`
	`N99 M30`
	Corrected example:
	`%wrk_90014.kor`
	`N10 G00 X0 Y0 Z0`
	`N20 V.G.WZ_AKT.R=5`
	`N30 G01 X10 Y10 F1000`
	`N40 G41 G138`
	`N50 G01 X80`
	`N60 G139 G40 (indirect deselection of TRC)`
	`N70 G01 X90 Y0`
	`N99 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Possible changes to the NC program: 1. Execute deselection indirectly (corrected example) 2. Add additional motion block before direct deselection
Error type	Abort NC program processing.		

ID 90015

Change of selection mode not permitted using direct selection.			

Description	If TRC (G138) is selected directly, it is not possible to change the selected side after the first motion block.
	
	When indirect selection (G139) is used and then a motion bock follows without any relevant motion data in the current main plane, a change is made implicitly to direct selection mode. Therefore, this error may also occur with G139.
	Example:
	`V.G.WZ_AKT.R = 5`
	`G139 G90 G01 X10 Y10 F1000`
	`G42 G01 X10 Y10 ; implicit change from G139 → G138`
	`G41 G01 X30 Y12 ; the error 90015 is also output here.`
	`…`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Possible changes to the NC program: 1. Use indirect selection (G139) 2. Add additional motion block before changing selection side
Error type	Abort NC program processing.		

ID 90016

Change of selection mode not permitted within circular motion block.			
Description	A change of the selected side of the TRC is not possible when a circular block follows. A linear block must follow after the selection mode of the tool radius compensation is changed.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program
Error type	Abort NC program processing.		

ID 90017

Tool radius greater or equal to contour radius.		
Description	The radius of the tool used is greater or equal to the radius of the programmed contour.	
Response	Class 2	Abort NC program processing.
Solution	Class 6	Use a tool whose radius is smaller than the radius of the programmed contour.
Parameter	%1:	Current value
		Radius of the contour
	%2:	Current value
		Radius of the tool used.
	%3:	Current value
		Difference between circle radius and radius of the tool used
	%4:	Lower limit value
		Minimum difference between radii

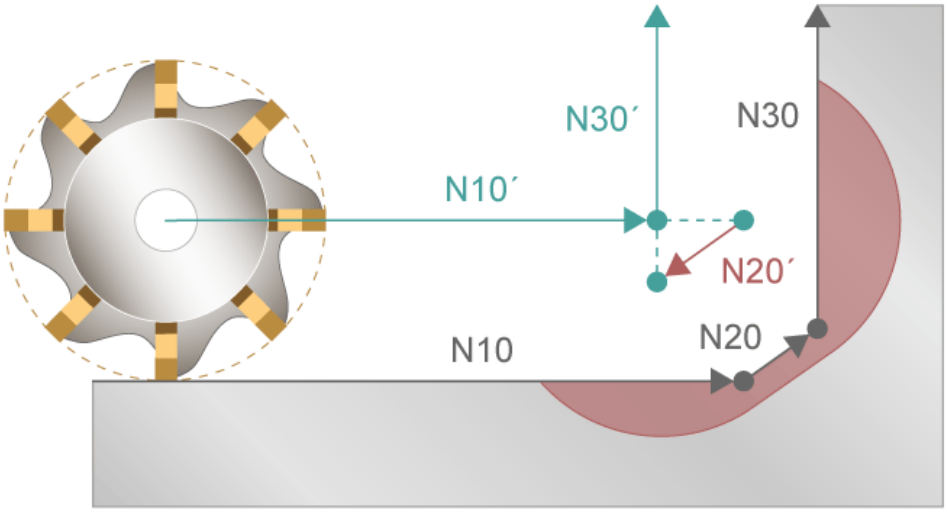
Error type	Abort NC program processing.

ID 90018

Difference between radii of starting and end points of the circle exceeds range.			
Description	Two radii are compared during circle programming for the tool radius compensation TRC. The first radius is the distance of the circle centre point from the starting point of the circle; the second radius is the distance between the circle centre point and the programmed end point of the circle. The difference between the two radii exceeds the permissible upper limit.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and correct the programmed circle parameters in the NC program. If necessary, increase the accuracy of circle programming.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
		Difference resulting from the distance from the circle centre point to the starting point and the distance from the circle centre point to the end point.	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
		Maximal value of the difference	
	%3:	Current value [0.1 µm or 0.0001°]	
		Distance: Circle centre point - circle starting point	
	%4:	Current value [0.1 µm or 0.0001°]	
		Distance: Circle centre point - circle end point	
Error type	1, Error message from NC program.		

ID 90019

Contour error due to compensation motion within linear motion block.

Description	To detect contour violations in linear blocks, the direction of the programmed motion block is compared with the direction of the corrected motion block. If the directions are opposite, this means a contour violation. The movement opposite to the programmed direction is called compensatory movement.  This can be caused by a relatively small motion block, e.g. smaller than the tool radius, which leads to contour violations when calculating the equidistant path. When using contour masking with G141, this error is not output because the contour masking process would delete the contour loop. In the figure above, the triangle would be removed with N20' and thus no contour damage occurs. [PROG// Section: Limits of TRC]		
Response	Class	2	Abort NC program processing
Solution	Class	6	Possible solutions are: • Use a tool with a smaller radius • Use contour masking (G141)
Error type	1, Error message from NC program.		

ID 90020 - 90025

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90026

Selection/deselection of tool radius compensation without valid feed rate.			
Description	No valid value exists for the feed rate. The feed rate must have a valid value for Tangential selection/deselection of TRC (G05) in rapid traverse.		
Response	Class	2	Abort NC program processing.

Solution	Class	6	Modify the NC program, program the feed rate
Error type	Abort NC program processing.		

ID 90027 - 90030

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90031

Tool radius exceeds range at channel block sequence circle-circle.			
Description	The tool radius exceeds the range at channel block sequence circle-circle; calculation of the intersection of the circles is not possible.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Use of a tool with a smaller radius
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Available width of tool diameter	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
		Tool diameter	
	%3:	Current value [0.1 µm or 0.0001°]	
		Diameters of the first programmed circle and/or circle segment	
	%4:	Current value [0.1 µm or 0.0001°]	
		Diameters of the second programmed circle and/or circle segment	
	%5:	Current value [0.1 µm or 0.0001°]	
		Distance between the circle centre points	
Error type	1, Error message from NC program.		

ID 90032 - 90035

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90036

Tool radius at channel block sequence circle-circle too large.

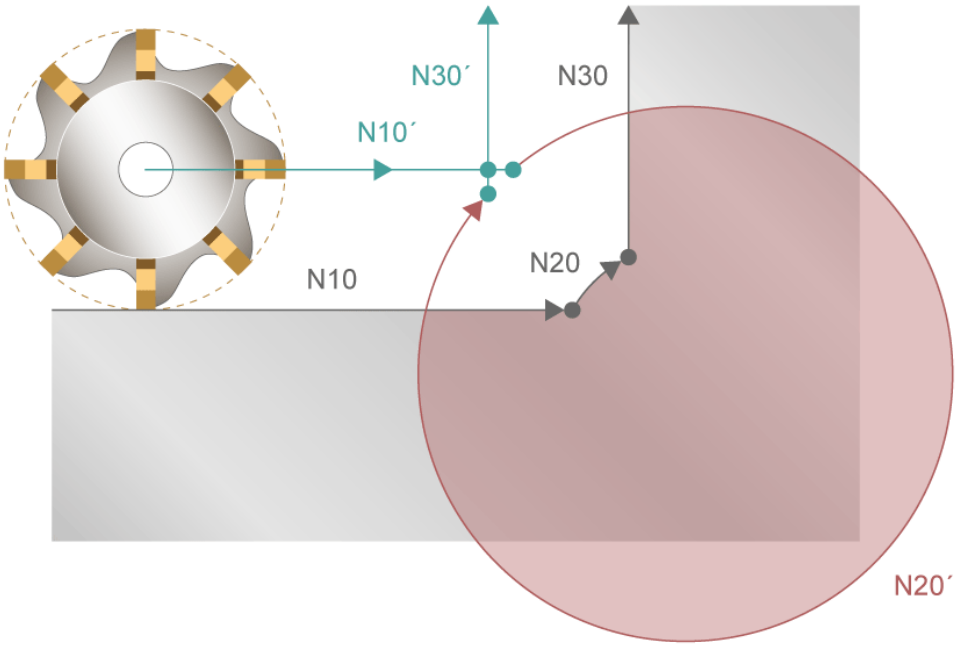
Description	No intersection was detected in the tool radius compensation at the block transition between two arcs to calculate parallel paths. Since the tool radius was selected too large, one of the two equidistant circles lies inside the other circle. Close the parallel path by using contour masking (G141) and this error is not output.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Possible solutions are: • Use a tool with smaller radius • Adapt the programmed circles • Use contour masking (G141)
Parameter	%1:		Current value [0.1 µm or 0.0001°]
			Difference between distance of centre points and difference between radii R1 and R2
	%2:		Upper limit value [0.1 µm or 0.0001°]
			Permissible tolerance for difference
Error type	1, Error message from NC program.		

ID 90037

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90038

Compensation motion within circular motion block in tool radius compensation.

Description	To detect contour violations in circular blocks, the corrected circular angle is compared with the original programmed angle. If the corrected circular angle is greater than the programmed one, the contour will be damaged. This is referred to as the compensation motion.  The figure above shows that if the direction of rotation is maintained at N20', the equidistant arc would damage the contour. This may be caused by a relatively small motion block, e.g. smaller than the tool radius. This leads to contour violations when calculating the equidistant path. When using contour masking with G141, this error is not output because the closed contour loop is deleted. In the figure above, this would remove the circular motion of N20' and thus no contour damage would occur. [PROG// Section : Limits of TRC]		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Possible solutions are: • Use a tool with smaller radius • Use contour masking (G141)
Parameter	%1:		Current value [0.1 µm or 0.0001°]
			Corrected circular angle
	%2:		Upper limit value [0.1 µm or 0.0001°]
			Maximum permissible circular angle
Error type	1, Error message from NC program.		

ID 90040

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90041

Removed non relevant channel block.			
Description	Remove a non-relevant transition set whose path motion is less than the minimum travel.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	
Error type	Abort NC program processing.		

ID 90042 - 90045

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90046

Radius within circular motion block is 0.			
Description	The radius of the circular block is zero (nil).		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modification of NC Program
Parameter	%1:	Current value [-]	
	%2:	Lower limit value [-]	
Error type	1, Error message from NC program.		

ID 90047

Changing the tool radius within circular motion block is not permitted.			
Description	It is not possible to change the tool radius in the selected TRC state before a circular block.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Difference between tool radii	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Maximum permissible difference between tool radii	

Error type	1, Error message from NC program.

ID 90048

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90049

End of program within selected state of tool radius compensation.			

Description	The NC program was ended in the selected state of tool radius compensation.
	
	Example with error:
	`%wrk_90049.err`
	`N10 G00 X0 Y0 Z0 G17`
	`N20 V.G.WZ_AKT.R=5`
	`N30 G01 X10 Y10 F10000`
	`N40 G41`
	`N50 G01 X80`
	`N60 G01 X90 Y20`
	`N80 G01 X90 Y0`
	`N99 M30`
	
	Corrected example:
	`%wrk_90049.kor`
	`N10 G00 X0 Y0 Z0 G17`
	`N20 V.G.WZ_AKT.R=5`
	`N30 G01 X10 Y10 F10000`
	`N40 G41`
	`N50 G01 X80`
	`N60 G01 X90 Y20`
	`N70 G40`
	`N80 G01 X90 Y0`
	`N99 M30`		

Response	Class	1	Output of warning The NC program is ended at the perpendicular point of the last motion block.
Solution	Class	1	Check and modify the NC program; deselect tool radius compensation with G40.
Error type	Abort NC program processing.		

ID 90050

Emptying of the NC channel within selected state of tool radius compensation not permitted.

Description	NC commands, which cause an emptying of the channel, are not permissible while TRC is active. These commands include, for example, the #FLUSH command.
	
	The error only occurs when the TRC is already deselected. This may occur when the direct or indirect deselect method is used and when G05 is used.
	In this two methods, tool radius compensation requires a motion block after G40 to reduce the offset built up by the tool radius.
	
	Test program with occurrence of the error:
	`N100 G139 ( indirect selection mode)`
	`N110 G41 G01X100 Y100 ( select TRC )`
	`N120 G01 X140 Y20 F2000`
	`N130 G03 X160 Y40 R20`
	`N140 G01 X200`
	`N150 G01 X220 Y20`
	`N160 G40 ( deselect TRC )`
	`N170 #FLUSH`
	`N180 G01 X0Y0`
	
	Corrected test program :
	`N100 G139 ( indirect selection mode)`
	`N110 G41 G01X100 Y100 ( select TRC )`
	`N120 G01 X140 Y20 F2000`
	`N130 G03 X160 Y40 R20`
	`N140 G01 X200`
	`N150 G01 X220 Y20`
	`N160 G40` **`G01 X0 Y0`** `( Deselect TRC )`
	`N170 #FLUSH`
	
	The decisive difference between the two test programs is the path motion after deselection with G40, since the distance between the tool to the programmed contour is only cancelled after this motion.
	
	In order to avoid the error, a motion block must be positioned between G40 and the command triggering the flushing of the channel when direct (G138) or indirect (G139) deselection are used.
	
	[PROG Section: Flushing NC channel]		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program. Insert a motion block after G40 similar to the example program above.
Error type	Abort NC program processing.		

ID 90051 - 90056

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90057

Adapting of feed rate impossible within G05 (tool radius exceeds range).

Description	The feedrate cannot be adapted with G11 when tool radius compensation is selected or deselected with G05. The tool radius is too large for this.		
Response	Class	1	NC program processing is continued
Solution	Class	1	Use a tool with a smaller radius
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Tool radius	
	%2:	Current value [0.1 µm or 0.0001°]	
		Theoretical radius of the tangential determined path	
	%3:	Current value [0.1 µm or 0.0001°]	
		Corrected radius of the tangential path	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 90059

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90062

No intersection point could be calculated between straight line and circle.			
Description	At a transition from a linear to circular motion block, no intersection for the equidistant course could be determined; the tool radius is too large.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Use a tool with a smaller radius.
Error type	1, Error message from NC program.		

ID 90063

No intersection point could be calculated between straight line and circle.			
Description	At a transition from a circular to a linear motion block, no intersection for the equidistant course could be determined; the tool radius is too large.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Use a tool with a smaller radius.
Error type	1, Error message from NC program.		

ID 90064

Motion block converted from circular to linear (tool radius too big).	
Description	The circular motion block is converted to a linear motion block. Reason for this conversion is that the corrected circle radius is close to zero. The cause is a tool radius that is too large.

Response	Class	1	NC program processing is continued
Solution	Class	1	Use of a tool with a smaller radius
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Corrected radius of the circular motion block	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 90065

Compensation movement found, enable contour masking process.			
Description	To detect contour violations in linear blocks, the direction of the programmed motion block is compared with the direction of the corrected motion block. If the directions are opposite, this means a contour violation. The movement opposite to the programmed direction is called compensation movement. This type of motion was detected and the contour masking process activated. See [PROG// Section: Limits of TRC] The principle of the outline fading out consists of removing the loop from the corrected path. For this purpose, the contour masking buffers a certain number of blocks corrected via the TRC. If the tool radius compensation recognizes a balance movement with active outline fading out, she marks the motion block and continues with the calculation of correction.		
Response	Class	1	Abort NC program processing.
Solution	Class	1	Possibility: Modification of NC Program Possibility: Use a tool with a smaller radius
Error type	1, Error message from NC program.		

ID 90066

Compensation movement found, impossible to enable cont.mask.-process.			
Description	A balance movement was recognized, the outline fading out process cannot not be activated. Cause: The tool radius is too large, meaning that the TRC cannot generate a contiguous equidistant contour.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Use a tool with a smaller radius
Error type	1, Error message from NC program.		

ID 90067 - 90073

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	

Solution	Class	8	Requires controller restart.

ID 90074

Contour masking process: Compensation movement could not be removed.			
Description	The contour masking process is unable to calculate an intersection point for the contour. This results in an overflow of an internal buffer allocated to it.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program: Modify the contour or use a tool with a smaller radius.
Parameter	%1:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 90075

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90076

Conversion from circular to linear in contour masking process.			
Description	Conversion of a circular motion block to a linear motion block because the minimum limit of motion paths was undershot.		
Response	Class	1	Warning
Solution	Class	1	
Parameter	%1:	Block number [-]	
		Number of converted block	
Error type	1, Error message from NC program.		

ID 90077

A loop was removed from the contour masking process.			
Description	A loop was removed because of contour masking. The reason for the loop is a compensation motion detected due to a direction reversal between a programmed and a corrected motion block.		
Response	Class	1	Warning
Solution	Class	1	
Parameter	%1:	Block number [-]	
	%2:	Block number [-]	

Error type	1, Error message from NC program.

ID 90079 - 90085

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90087

End of program in state ready of contour masking process.			
Description	Contour masking is still ready at the end of the program.		
Response	Class	1	NC program processing is continued
Solution	Class	1	Modify the NC program: Deselect contour masking with G140 before end of program
Parameter	%1:	Block number [-]	
Error type	1, Error message from NC program.		

ID 90088

End of program with active contour masking process. (contour error).			
Description	At the end of program contour masking process is still enabled.		
Response	Class	3	Abort NC program processing.
Solution	Class	6	Modification of NC Program: Insert an additional motion block, this allows the contour masking process the calculation of an intersection point of the contour. Following the deselection of the contour masking and the tool radius compensation before the end of the program.
Parameter	%1:	Block number [-]	
Error type	1, Error message from NC program.		

ID 90090

Flushing of the NC channel while contour masking process is active. (contour error).			
Description	The use of the commands for flushing the channel (#FLUSH) is prohibited when TRC is active and therefore also when contour masking is selected. [PROG// Section: Flushing NC channel]		
Response	Class	3	Abort NC program processing.

Solution	Class	6	Modify the NC program
Error type	1, Error message from NC program.		

ID 90091 - 90097

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 90098

Deselection of tool radius compensation while contour masking process is active.			
Description	TRC is deselected without first deactivating contour masking.		
Response	Class	1	NC program processing is continued
Solution	Class	1	Modify the NC program: Deselect contour masking before deselecting the TRC
Parameter	%1:	Block number [-]	
		Block number of TRC deselection.	
Error type	1, Error message from NC program.		

ID 90099

Deselection of contour masking while process is active.			
Description	Contour masking was deselected although a compensation motion was detected. The compensation motion caused the activation of the contour masking process. A compensation motion is a motion which is in the opposite direction to the programmed direction.		
Response	Class	3	Abort NC program processing.
Solution	Class	6	Possible changes to the NC program: 1. Use a tool with a smaller radius to prevent the compensation motion 2. Move deselection of contour masking (G140) in the NC program
Parameter	%1:	Block number [-]	
		NC block number of deselection of contour masking.	
Error type	-		

ID 90100

Contour masking process: Block buffer full and no more block output possible.			
Description	The contour masking process is unable to calculate an intersection point for the contour. This results in an overflow of an internal buffer allocated to it.		
Response	Class	1	NC program processing is continued
Solution	Class	1	Modify the NC program: Modify the contour or use a tool with a smaller radius.

Parameter	%1:	Current value [-]
Error type	1, Error message from NC program.	

ID 90101 - 90106

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90107

Selected G237 without enabling this functionality.			
Description	Use of perpendicular selection and deselection of TRC (G237) although this is not enabled.		
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 90109

Number of non-relevant buffered blocks was exceeded.			
Description	The number of non-relevant blocks, which are buffered with perpendicular selection and deselection, was exceeded. Non-relevant blocks contain no motion data for TRC blocks, e.g. technology functions.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program: Interrupt sequence of non-relevant blocks by a relevant block.
Error type	1, Error message from NC program.		

ID 90110 / 90111

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90112

Tool radius compensation: Block 1 and block 2 include an area.

Description	During the transition of blocks from linear to circular or from circular to linear, an overlap of the blocks arises. Both blocks have two common points and include an area that cannot be machined with active tool radius compensation.		
Response	Class	1	NC program processing is continued
Solution	Class	1	Check and modify the contour path in the NC program
Parameter	%1:	Current value [-]	
		Block number of the first motion block	
	%2:	Current value [-]	
		Block number of the second motion block	
Error type	1, Error message from NC program.		

ID 90114

The number of buffered blocks was exceeded with active inner corner selection.			
Description	The number of buffered blocks between selection and deselection of tool radius compensation (TRC) when using inner corner selection (G238) exceeds the permissible limit.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Reduce the number of the buffered blocks.
Parameter	%1:	Upper limit value [-]	
		Maximum number of blocks which can be buffered.	
Error type	1, Error message from NC program.		

ID 90115

During active inner corner selection changing side of selection is not permissible.			
Description	With active inner corner selection (G238) of TRC, changing the side of selection is not permissible.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Remove the inadmissible G function.
Parameter	%1:	Block number [-]	
		Number of the inadmissible programmed G function	
Error type	1, Error message from NC program.		

ID 90116

No relevant motion block available.			
Description	Using inner corner selection there is no motion block programmed, which contains movement information between selection and deselection of TRC.		
Response	Class	1	Abort NC program processing.
Solution	Class	1	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 90117 / 90118

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90119

The programmed contour is not closed.			
Description	When using the command inner corner selection of TRC, the following contour must be closed. Closed contour means that the endpoint of block of selection must be identical with the endpoint of the last programmed point before deselection of TRC. Single blocks and contours with only two motion blocks could not be selected with this method.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the first and the last motion while TRC is active. Change to different TRC selection method for individual blocks or for contours with only two motion blocks.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
		Distance of the endpoint of block of selection to the endpoint of the last motion block before deselection of TRC.	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
		Upper limit of distance.	
Error type	1, Error message from NC program.		

ID 90120 - 90122

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90123

Contour error through selection point of tool radius compensation.			
Description	The position of the selection point of tool radius compensation (TRC) is checked when Select inner corner (G238) is used. The check consists of a measurement of the distance between the selection point and the programmed contour element. The distance must be greater than the tool radius. When the error occurred, the distance was less than the tool radius.		
Response	Class	2	Abort NC program processing.

Solution	Class	6	Check and modify the NC program either by correcting the point of selection of TRC or correcting the contour element affected. A further option is to use a tool radius with a smaller radius.
Parameter	%1:	Block number [-]	
		Block number of the contour block affected.	
	%2:	Block number [-]	
		Block number of the selection block.	
	%3:	Error value [0.1 µm or 0.0001°]	
		Distance between the concerned element of contour and the selection point of TRC.	
	%4:	Upper limit value [0.1 µm or 0.0001°]	
		Minimum distance corresponds to the current tool radius.	
Error type	1, Error message from NC program.		

ID 90124

Programming a full circle is not permitted with active inner corner selection.			
Description	When inner corner selection (G238) is active, a full circle is programmed between selection and deselection of tool radius compensation (TRC). Programming a full circle represents a closed contour but this is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program.
Error type	1, Error message from NC program.		

ID 90125

Length of the selection block with inner corner selection is less than tool radius.			
Description	The length of the linear motion block after the inner corner selection of TRC is less than the tool radius. The reason for this is the position of the point where the TRC was selected. This point is too close to the programmed contour.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the position of the point where the TRC is selected.
Parameter	%1:	Block number [-]	
		Number of the block where TRC is selected.	
	%2:	Error value [-]	
		Length of the selection block.	
	%3:	Lower limit value [-]	
		Minimum length of selection block.	
Error type	1, Error message from NC program.		

ID 90126

The selected side for inner corner selection is wrong.			
Description	The selected side that was programmed is wrong. To reach the equidistant path, the tool must cross the programmed contour. In this case the contour is damaged.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Correct the side of selection.

Error type	1, Error message from NC program.

ID 90127

During active inner corner selection changing the tool radius is not permissible.			
Description	Changing the tool radius with active inner corner selection (G238) of the TRC, is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program. Modify the tool radius either before selection of the TRC or after the deselection of the TRC.
Parameter	%1:	Block number [-]	
		Number of the block the change of tool radius happened.	
	%2:	Expected value [-]	
	%3:	Current value [-]	
Error type	1, Error message from NC program.		

ID 90128

Inadmissible intersection between motion block and selection block.			
Description	When inner corner selection (G238) is active, the motion blocks between selection and deselection of TRC are checked for intersections with the approach block. An intersection of the blocks occurred as a result of this check and resulted in a contour violation.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program. Correct the point of selection to modify the approach block.
Parameter	%1:	Block number [-]	
	%2:	Block number [-]	
Error type	1, Error message from NC program.		

ID 90129

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90130

Angle exceeds upper limit at inner corner selection.

Description	Inner corner selection (G238) of the TRC checks the transition angle between selection block and the first motion block. The first motion block means the block that forms the distance between the tool radius and the programmed course. There is an upper limit for this transition angle and this was exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the point of selection of TRC.
Parameter	%1:	Error value [-]	
		Transition angle: selection block – first motion block	
	%2:	Upper limit value [-]	
		Upper limit of transition angle.	
Error type	1, Error message from NC program.		

ID 90131

Compensation motion of selection block of tool radius compensation not allowed.			
Description	A motion block was marked as a compensation motion for the contour masking process that cannot be eliminated. This motion block is either the first one after selection of TRC with active contour masking, or the first one after selection of contour masking with active TRC. The cause for compensation motion is either a motion block that is too short after selection of TRC or a tool radius that is too large. When the command #FLUSH CONTINUE is used in the NC program, the disputed motion block is placed directly before the #FLUSH CONTINUE command. This is caused by the TRC outputting the control block to flush the channel (#FLUSH CONTINUE) before the disputed motion block since the TRC retains this motion block for the interpolation point calculation that follows.		
Response	Class	3	Abort NC program processing.
Solution	Class	3	Check and modify the NC program and the tool used. • Use a tool with a smaller radius. • Move selection of TRC or contour masking in the NC program • When #FLUSH CONTINUE is used, place this command at a different position in the NC program
Parameter	%1:	Block number [-]	
		Number of the disputed motion block.	
Error type	1, Error message from NC program.		

ID 90132

The data item is out of data format.			
Description	Data item exceeds the permissible range of data.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program.

Parameter	%1:	Error value [-]
	%2:	Lower limit value [-]
	%3:	Upper limit value [-]
	%4:	Current value [-]
		Corresponding block number in NC program
Error type	1, Error message from NC program.	

ID 90133

The coordinate of the corrected point is outside of the permissible range.			
Description	The tool radius compensation calculates coordinates of the equidistant path. The error occurs when the coordinate of the corrected circle centre point exceeds the permissible range of data format.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program
Parameter	%1:		Error value [-]
			Calculated coordinate
	%2:		Lower limit value [-]
	%3:		Upper limit value [-]
	%4:		Block number [-]
			Corresponding block number in NC program
	%5:		Logical axis number [-]
			Logical axis no.
Error type	1, Error message from NC program.		

ID 90134

The coordinate of the corrected centre is outside of the permissible range.			
Description	The tool radius compensation calculates coordinates of the equidistant path. The error occurs when the coordinate of the calculated end point exceeds the permissible range of the data format. The overshoot of the data format also occurs with a transition block generated in the tool radius compensation when its corrected end point exceeds the data format. Example: When 2 linear channel blocks intersect at and extremely small angle between the blocks, the intersection of the equidistant points can deviate extremely far from the original programmed intersection, in extreme cases it exceeds the limit value of the permissible range.		
Response	Class	2	Abort NC program processing.

Solution	Class	6	Check and modify the NC program
Parameter	%1:	Error value [-]	
		Calculated coordinate	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
	%4:	Block number [-]	
		Corresponding block number in NC program	
	%5:	Logical axis number [-]	
		Logical axis no.	
Error type	1, Error message from NC program.		

ID 90135 / 90136

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90137

Tool radius compensation: Contour error due to selection block.			
Description	The length of the selection point of tool radius compensation is smaller than the tool radius. This means that the tool is tool close to the contour when tool radius compensation is selected and the contour will be damaged.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Move the point of selection of TRC.
Parameter	%1:	Block number [-]	
		Corresponding block number in NC program	
Error type	1, Error message from NC program.		

ID 90138 - 90140

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90141

Tool geometry compensation: used plane is not permissible.

Description	When tool geometry compensation (TGC) is used in combination with tool radius compensation (TRC), only the plane X/Y with G17 is permitted. The use of G18 or G19 is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program.
Error type	1, Error message from NC program.		

ID 90142

Tool geometry compensation: Contour error due to compensation motion within linear motion block.			
Description	When tool geometry compensation (TGC) is used in combination with tool radius compensation (TRC), a compensation motion occurred. This will cause a contour violation. Further information is also available under the error message P-ERR-90019 [▶ 1209].		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Use a tool with a smaller radius.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 90143

Selection of tool geometry compensation not possible without active transformation.			
Description	It is not possible to use the outline milling mode of tool geometry compensation (TGC) without a suitable active transformation.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program: Program #TRAFO ON before selecting TRC with #TGC ON
Error type	1, Error message from NC program.		

ID 90144

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90145

Deselection of tool radius compensation with mode G239 while contour masking process is active.			
Description	The contour masking process is active at the time the TRC is deselected with TRC mode G239. The contour masking process is attempting to compensate contour violations at this time and has not yet detected a suitable intersection point.		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Possible changes to the NC program: • Use a tool with a smaller radius to prevent the compensation motion • Move the deselection of the contour masking in the NC program
Parameter	%1:	Block number [-]	
		Number of the deselection block	
	%2:	Block number [-]	
		Block number of the block damaging the contour.	
Error type	1, Error message from NC program.		

ID 90146

Program end during active inner corner selection.			
Description	When inner corner selection (G238) is used, a program end (M30) was programmed in the NC program in active state. This is not permissible since the contour must be closed between the commands G41/G42 and G40. The active state of inner corner selection is quitted after TRC is deselected. A motion block after G40 is required for deselection.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modification of NC Program.
Error type	1, Error message from NC program.		

ID 90147

Tool radius is greater than the programmed radius of the full circle.			
Description	The programmed radius of the full circle is smaller than the radius of the tool used. The correction should be inside the full circle. This is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program or use a tool with a smaller radius.
Parameter	%1:	Current value [-]	
		Radius of the contour	
	%2:	Current value [-]	
		Radius of the tool used.	
	%3:	Current value [-]	
		Difference between circle radius and radius of the tool used	
	%4:	Lower limit value [-]	
		Minimum difference between radii	
Error type	1, Error message from NC program.		

ID 90148 - 90150

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90151

Generated circular motion block could not be corrected through TRC because of programmed #FLUSH CONTINUE.			
Description	The TRC tries to correct a self-generated circular transition set retroactively in order not to damage the contour. This circular transition block set has already been output due to the programming #FLUSH CONTINUE command and can therefore no longer be corrected.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the NC program and the tool used. Modify the NC program: reposition the #FLUSH CONTINUE command.
Error type	1, Error message from NC program.		

ID 90152

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90154

The coordinate is out of permissible range.			
Description	The permissible value of the coordinate exceeds the permissible data range.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	
Parameter	%1:	Error value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
	%4:	Block number [-]	
	%5:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 90155 / 90156

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90157

Circular selection of tool radius compensation with active tool geometry compensation not allowed.			
Description	Circular selection of TRC is not permitted when tool geometry compensation (TGC) is used in combination with tool radius compensation (TRC). Basically circular blocks are not possible when the TGC function is used.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program.
Error type	1, Error message from NC program.		

ID 90158

Circular motion block during active tool geometry compensation with tool radius compensation not allowed.			
Description	Circular blocks are not permitted when tool geometry compensation (TGC) is used in combination with tool radius compensation (TRC).		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program.
Error type	1, Error message from NC program.		

ID 90159

Contour loop detected.			
Description	A contour loop was detected. The use of contour masking is required to eliminate these automatically. The selection is made with G141.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Activation of contour masking with G141
Parameter	%1:	Block number [-]	
		NC block number of the compensation block	
	%2:	Block number [-]	
		NC block number of the first motion block which is not given out	
Error type	1, Error message from NC program..		

ID 90160

Compensation motion detected, no closed loop.

Description	A compensation motion was detected but not a closed loop. This compensation motion cannot be removed by using the contour masking method (G141). Modify the NC program or use a tool with a smaller radius to avoid compensation movements.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Block number [-]	
		NC block number of the compensation block	
	%2:	Block number [-]	
		NC block number of the first motion block which is not given out	
Error type	1, Error message from NC program.		

ID 90161

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90162

Compensation motion and contour loop detected, activation of contour masking too late.			
Description	Both compensation motion and a contour loop were detected. However, the activation of contour masking with G141 in the NC program occurs too late. Move the G141 command within the NC program so that it lies at least before the motion block with the block number output in parameter 2.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Modify the NC program. Shift the G141 command within the NC program
Parameter	%1:	Block number [-]	
		NC block number of the compensation block	
	%2:	Block number [-]	
		NC block number of the first motion block which is not output	
Error type	1, Error message from NC program.		

ID 90163

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90164

A contour error was detected at deselection of tool radius compensation with mode G239.			
Description	When deselecting TRC, a contour violation is detected. This can not be eliminated by activating the contour masking with G141.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Modify the NC program: Use a tool with a smaller radius to prevent contour damage
Parameter	%1:	Block number [-]	
		Block number of deselection block.	
	%2:	Block number [-]	
		Block number of contour damaging motion block	
Error type	1, Error message from NC program.		

ID 90165

Multipath programming not active when tool radius compensation is active.			

Description	A NC program was started with TRC in 2 paths.
	
	The following parameters must be set in order to use TRC in 2 paths:
	
	Correct
	Channel parameter list:
	`multi_path_configuration 1 # P-CHAN-00261`
	Activate the function:
	`configuration.tool_radius_comp.function MULTI_PATH # P-CHAN-00555`
	Alternatively, the function can be activated in the start-up list.
	`configuration.channel[0].tool_radius_comp.function MULTI_PATH # P-STUP-00080`
	
	Wrong:
	`configuration.tool_radius_comp.function 0`
	or in the start-up list
	`configuration.channel[0].tool_radius_comp.function 0`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Activate the function P-CHAN-00555 (or P-STUP-00080)
Error type	1, Error message from NC program.		

ID 90166

Wrong parameter or mode at selection time of tool radius compensation.	
Description	If TRC is selected with G41 or G42, the tool radius compensation settings and parameters are checked when 2-path configuration is active and online tool compensation is active. Online tool compensation is activated by #TRC [ONLINE=. INVERSE=..] Several settings for tool radius compensation are invalid in combination with the 2-path configuration.

Response	Class	2	Abort NC program processing.
Solution	Class	6	Remove or replace the invalid NC command or parameter in the NC program The invalid value may already be set in the channel parameters of a program start group P-CHAN-00063.
Parameter	%1:	Error value [-]	
		Invalid NC command or parameter.	
Error type	1, Error message from NC program.		

ID 90167 - 90171

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90172

Release a main axis in plane with active tool radius compensation not permitted.			
Description	A main axis of the active machining plane is to be be released without previously decreasing the tool radius. This error can only occur with the following NC program sequence: Wrong: N150 G139 G19 (Y-Z) N160 G41 G1 Y30 Z0 N170 G1 Y40 N180 G1 Y50 N190 G40 N200 **#PUT AX**[Z] (axis required in plane) N210 G01 Y70 (deselection block still required for TRC) Corrected: N150 G139 G19 (Y-Z) N160 G41 G1 Y30 Z0 N170 G1 Y40 N180 G1 Y50 N190 G40 N200 G01 Y70 (deselection block still required for TRC) N210 **#PUT AX**[Z]		
Response	Class	2	Abort NC program processing
Solution	Class	6	Modify the NC program.
Error type	1, Error message from NC program.		

ID 90173 - 90177

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90178

Illegal combination of parameters in multipath configuration.			

Description	The parameters for using single or multi path configurations are incorrect in the start-up list and the channel parameter list.
	
	In the channel parameter list the parameter
	`multi_path_configuration  0 # P-CHAN-00261`
	is incorrectly set for the use of multipath configurations.
	
	Correct setting for multipath configuration:
	Channel parameter list:
	`multi_path_configuration  1 # P-CHAN-00261`
	Activate the function:
	`configuration.tool_radius_comp.function  MULTI_PATH # P-CHAN-00550`
	Alternatively, the function can be activated in the start-up list:
	`configuration.channel[0].tool_radius_comp.function  MULTI_PATH # P-STUP-00080`
	
	Correct setting for single path configuration (default):
	Delete both parameters in each of the lists.		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Correct the parameters concerned in the channel parameter list (or start-up list.)
Error type	1, Error message from NC program.		

ID 90179

No suitable point was found on selection of tool radius compensation with mode G236.			
Description	When TRC was selected with mode G236, no valid selection point could be determined. This is possibly caused by short motion blocks with a corresponding large tool radius. The system uses the TRC direct selection mode (G138).		
Response	Class	1	NC program processing is continued
Solution	Class	1	Check and modify the motion blocks at TRC selection
Error type	1, Error message from NC program.		

ID 90180

No suitable point was found on deselection of tool radius compensation with mode G236.

Description	When TRC in mode G236 was deselected, no suitable point was found for the deselection. This is possibly caused by short motion blocks with a corresponding large tool radius. The TRC direct deselection mode (G138) was used.		
Response	Class	1	NC program processing is continued
Solution	Class	1	Check and modify motion blocks at deselection.
Error type	1, Error message from NC program.		

ID 90181

Circular selection of tool radius compensation with mode G236 and multipath programming is not allowed.			
Description	It is not possible to select TRC with G236 in a circular block when 2-path programming is used.		
Response	Class	2	Abort NC program processing
Solution	Class	3	Check and modify the NC program: • Select TRC with linear block • Switch to a different TRC selection mode
Error type	1, Error message from NC program.		

ID 90182

Circular deselection of tool radius compensation with mode G236 and multipath programming is not allowed.			
Description	It is not possible to deselect TRC with G236 in a circular block when 2-path programming is used.		
Response	Class	2	Abort NC program processing
Solution	Class	3	Check and modify the NC program: • Deselect TRC with linear block • Switch to a different TRC deselection mode
Error type	1, Error message from NC program.		

ID 90183

Circular selection of tool radius compensation with active online TRC is not allowed.			
Description	It is not allowed to select TRC in a circular block when online TRC is used.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program: • Select TRC with linear block • Deactivate online TRC
Error type	1, Error message from NC program.		

ID 90184

Circular deselection of tool radius compensation with active online TRC is not allowed.	
Description	It is not allowed to deselect TRC in a circular block when online TRC is used.

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program: • Deselect TRC with linear block • Deactivate online TRC
Error type	1, Error message from NC program.		

ID 90185 - 90187

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 90190

Tool radius is greater than the programmed radius of circle at selection of tool radius compensation.			
Description	The radius of the tool used is greater than the radius of the programmed circle after TRC is selected. This may result in contour violations and velocity drops at the block transition.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Modify the NC program or use a tool with a smaller radius.
Parameter	%1:	Current value [-]	
		Programmed radius of the contour	
	%2:	Current value [-]	
		Radius of the tool used.	
	%3:	Current value [-]	
		Difference between circle radius and radius of the tool used	
	%4:	Current value [-]	
		Minimum difference between radii	
Error type	1, Error message from NC program.		

ID 90191

Selection of tool radius compensation with mode G237 and active tool geometry compensation not allowed.			
Description	When the TRC is active, it is not possible to select the TRC with the G237 mode.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a different TRC selection mode
Error type	1, Error message from NC program.		

ID 90192

Deselection of tool radius compensation with mode G237/G239 and active tool geometry compensation not allowed.

Description	It is not possible to deselect the TRC with G237 or G239 mode when the TGC is active.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Use a different TRC mode for deselection.
Error type	1, Error message from NC program.		

ID 90193

Circular programming with G303 directly after deselection of tool radius compensation not allowed.			

Description	After TRC is deselected by default (G42), it is impossible to indirectly program an arc in space (G303).
	Default deselection of the TRC requires a motion block to reduce the tool radius again.
	
	Example with error:
	`N10 G42 G139`
	`N20 G01 X10 Y10 F5000`
	`;…`
	`N130 G40`
	`N140 G303 I50 J50 K50 X100 Y0 Z100`
	`;..`
	
	Corrected example:
	`N10 G42 G139`
	`N20 G01 X10 Y10 F5000`
	`; …`
	`N130 G40 X20 Y20`
	`N140 G303 I50 J50 K50 X100 Y0 Z100`
	`; …`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify NC program. Insert a motion block after G40
Error type	1, Error message from NC program.		

ID 90194

The equidistant circles are side by side and have no intersection.			
Description	No intersection was found at the transition between two circle elements during parallel path calculation. Since the tool radius was selected too large, the two equidistant circles lie adjacent to each other. Use the contour masking method (G141) so that this error message is not output since the parallel path is closed by the CNC.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Possible solutions: • Use a tool with a smaller radius • Use the contour masking method (G141)

Parameter	%1:	Current value [µm]
		Distance between centre points
	%2:	Current value [µm]
		Sum of remaining residual radii R1 and R2
	%3:	Current value [µm]
		Current tool radius
Error type	1, Error message from NC program.	

ID 90195-90199

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 90201

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 90202

Feed splitting not possible, programmed axis is not in the main plane.			
Description	When the feed splitting function is used, the programmed axis is not in the main plane. The specified axis for feed splitting must be in the corresponding main plane. G17: XY plane (default) G18: ZX plane G19: YZ plane		
Response	Class	1	Warning, no reaction., the feed is not split.
Solution	Class	1	Check and modify the specified axis
Parameter	%1:	Error value [-]	
		Logical axis number of the programmed axis	
	%2:	Expected value [-]	
		Logical axis number of the first axis of the main plane	
	%3:	Expected value [-]	
		Logical axis number of second axis of the main plane	
	%4:	Current value [-]	
		Select or deselect	
Error type	1, Error message from NC program.		

ID 90203

Feed splitting not possible, block length is too small.			
Description	When feed splitting, the length of the selected or deselected block is smaller than the length of the first segment. The block length is dependent on the feed splitting mode, either in the axis direction or the path direction.		
Response	Class	1	Warning, no reaction. The feed is not split.
Solution	Class	1	Check and modify the selected or deselected block. Or, reduce the value for DIST_SEG1.
Parameter	%1:	Error value [0.1 µm or 0.0001°]	
		Programmed value for DIST_SEG1	
	%2:	Error value [0.1 µm or 0.0001°]	
		Length of the selected or deselected block	
	%3:	Current value [-]	
		Select or deselect	
Error type	1, Error message from NC program.		

ID 90204

No license for use of edm functions.			
Description	The EDM license is missing. The license for the EDM functionality is required in order to use 2 different tool radii in 2-path programming.		
Response	Class	2	Abort NC program processing. Transition to error state.
Solution	Class	6	Possible solutions are: • Change the NC programming • Add the EDM license.
Error type	1, Error message from NC program.		

2.11 Axis management errors (ID-range 110000-119999)

2.11.1 ID-range 110000-110249

ID 110005 - 110033

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 110034

Illegal denominator for damping value.

Description	The damping value for simulation drives is specified by numerator and denominator. Damping value = P-AXIS-00051 / P-AXIS-00050 An invalid value was specified for the denominator.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00050 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00050	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00050	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110035

Illegal numerator for damping value.			
Description	The damping value for simulation drives is specified by numerator and denominator. Damping value = P-AXIS-00051 / P-AXIS-00050 An invalid value was specified for the numerator.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00051 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00051	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00051	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110036

Illegal denominator for natural frequency.			
Description	The natural frequency for simulation drives is specified by numerator and denominator. Natural frequency = P-AXIS-00062 / P-AXIS-00061 An invalid value was specified for the denominator.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00061 in the parameter list of the affected axis.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Error value [-]
		Configured value of P-AXIS-00061
	%3:	Limit value [-]
	%4:	Corrected value [-]
		P-AXIS-00061
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110037

Illegal numerator for natural frequency.			
Description	The natural frequency for simulation drives is specified by numerator and denominator. Natural frequency = P-AXIS-00062 / P-AXIS-00061 An invalid value was specified for the numerator.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00062 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00062	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00062	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110038

Illegal denominator for time constant.			
Description	The sampling time constant for simulation drives is specified by numerator and denominator. Sampling time constant = P-AXIS-00239/ P-AXIS-00238 An invalid value was specified for the numerator		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00238 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00238	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00238	

Error type	2, Error message by data transfer from parameter list into control device.

ID 110039

Illegal numerator for time constant.			
Description	The sampling time constant for simulation drives is specified by numerator and denominator. Sampling time constant = P-AXIS-00239/ P-AXIS-00238 An invalid value was specified for the numerator		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00239 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00239	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00239	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110040

Simulation drive: dead time exceeds maximum limit.			
Description	The dead time for simulation drives is specified by P-AXIS-00194. The given value is invalid.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00194 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00194	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00194	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110042

D/A converter resolution is illegal (numerator).	
Description	The resolution of the drive command value is specified by numerator and denominator. Damping value = P-AXIS-00129 / P-AXIS-00128 An invalid value was specified for the numerator.

Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00129 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00129	
	%3:	Corrected value [-]	
		P-AXIS-00129	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110043

D/A converter resolution is illegal (denominator).			
Description	The resolution of the drive command value is specified by numerator and denominator. Damping value = P-AXIS-00129 / P-AXIS-00128 An invalid value was specified for the numerator.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00128 in the parameter list of the affected axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
		Configured value of P-AXIS-00128	
	%3:	Corrected value [-]	
		P-AXIS-00128	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110048

SERCOS drive: Invalid type of protocol.			
Description	The SERCOS telegram type S-0-00015 was specified with an invalid value.		
Response	Class	1	Element is set to user-defined telegram type 7.
Solution	Class	1	Correct the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		P-AXIS-00188	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00188	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110051

Illegal axis mode for a translatory axis.			
Description	No linear axis mode P-AXIS-00015 (bit-coded) was specified for a translatory axis.		
Response	Class	1	The linear axis mode is automatically selected.
Solution	Class	1	Correct the axis parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		P-AXIS-00015	
	%3:	Corrected value [-]	
		P-AXIS-00015	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110052

Difference of modulo less than or equal to 0.			
Description	The difference between upper P-AXIS-00126 and lower modulo limit P-AXIS-00127 is smaller than 0. The modulo range for rotary axes or spindles must be at least 0.0001°.		
Response	Class	1	The upper and lower modulo limits are initialised by a default value.
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		P-AXIS-00126 - P-AXIS-00127	
	%3:	Limit value [0.1 µm or 0.0001°]	
	%4:	Corrected value [0.1 µm or 0.0001°]	
		P-AXIS-00126	
	%5:	Corrected value [0.1 µm or 0.0001°]	
		P-AXIS-00127	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110053

Axis mode invalid.			
Description	For a rotary axis, the axis mode P-AXIS-00015 (bit-coded) must be at least linear / optimised straightening / modulo.		
Response	Class	1	The linear axis mode is automatically selected.
Solution	Class	1	Correct the axis parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00015	
	%3:	Corrected value [-]	
		P-AXIS-00015	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110054

Axis position greater than the upper software limit switch.			
Description	The specified axis position P-AXIS-00017 is above the positive software limit switch P-AXIS-00178.		
Response	Class	1	
Solution	Class	6	Correct the value in axis parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		P-AXIS-00017	
	%3:	Limit value [0.1 µm or 0.0001°]	
		P-AXIS-00178	
	%4:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110055

Axis position smaller than the lower software limit switch.			
Description	The specified axis position P-AXIS-00017 is below the negative software limit switch P-AXIS-00177.		
Response	Class	1	
Solution	Class	6	Correct element in axis parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		P-AXIS-00017	
	%3:	Limit value [0.1 µm or 0.0001°]	
		P-AXIS-00177	
	%4:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110059

Limiting speed for standstill is greater than maximum axis velocity.			
Description	The specified velocity P-AXIS-00216 at which the axis is considered stationary is above the maximum axis velocity.		
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Incorrect value [0.1 µm or 0.0001°]
		P-AXIS-00216
	%3:	Limit value [0.1 µm or 0.0001°]
		P-AXIS-00212
	%4:	Corrected value [0.1 µm or 0.0001°]
		P-AXIS-00216
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110060

Unknown drive type.			
Description	The specified axis type P-AXIS-00018 is unknown. The following axis types are known: 1 – linear axis 2 – rotary axis 4 – spindle		
Response	Class	3	Stop current job.
Solution	Class	7	Set correct axis type in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		P-AXIS-00018	
Error type	-		

ID 110061

Number of revolutions in the event of modulo errors is less than 0.			
Description	The number of modulo revolutions P-AXIS-00125 is zero (nil). For an active modulo compensation P-AXIS-00120 the number must be greater than zero (nil).		
Response	Class	1	Initialization of Modulo revolutions with default value.
Solution	Class	1	Correction of the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00125	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00125	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110062

Modification of the logical axis number requires restart of the control.

Description	The parameter P-AXIS-00016 for unique identification of an axis in the system cannot be changed while the CNC kernel is running. The new logical axis number is not transferred; the previous active logical axis number remains valid.		
Response	Class	1	None.
Solution	Class	1	Only change the logical axis number P-AXIS-00016 in configuration mode or when the CNC is not running.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016	
	%2:	Current value [-]	
		Current value of P-AXIS-00016.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00016.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110063

Changing the drive type requires restart of the control.			
Description	The parameter P-AXIS-00020 for the axis drive type cannot be changed while the CNC kernel is running. The new drive type is not transferred; the previous active drive type remains valid.		
Response	Class	1	None.
Solution	Class	1	Only change the drive type P-AXIS-00020 in configuration mode or when the CNC is not running.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016.	
	%2:	Current value [-]	
		Current value of P-AXIS-00020.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00020.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110064

The logical axis number was assigned several times to axes.			
Description	Two axes in the system were assigned the same logical axis number P-AXIS-00016. This is not permitted. The data transfer for the second axis with the same logical axis number is rejected.		
Response	Class	1	None.
Solution	Class	1	Assign all axes in the system with different logical axis numbers P-AXIS-00016.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current logical axis number.	
	%3:	Corrected value [-]	
		Corrected logical axis number P-AXIS-00016.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110065

Unknown drive type.			
Description	The specified type of drive is unknown.		
Response	Class	3	
Solution	Class	7	P-AXIS-00020 must be checked
Parameter	%1:	Logical axis number [-]	
		s. P-AXIS-00016	
	%2:	Error value [-]	
		s. P-AXIS-00020	
Error type	-		

ID 110066

Missing gear with specified number.			
Description	An attempt was made to change a gear stage which is unknown. Example G112 X17 (Gear stage 17 does not exist)		
Response	Class	3	Abort current job.
Solution	Class	7	Check the number of gear stages and the gear stage that is to be changed.
Parameter	%1:	Current value [-]	
		Gear stage	
	%2:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%3:	Upper limit value [-]Upper limit value [-]	
		Maximum number of gears	
Error type	-		

ID 110067

Axis data for gear stage incomplete.			
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.		
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
		Incorrect bitmask	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110068

Hardware independent axis data for position controller incomplete.			
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.		
Response	Class	3	
Solution	Class	7	

Parameter	%1:	Logical axis number [-]
	%2:	Incorrect value [-]
		Incorrect bitmask
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110069

Hardware dependent axis data for position controller incomplete.		
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.	
Response	Class 3	
Solution	Class 7	
Parameter	%1:	Logical axis number [-]
	%2:	Incorrect value [-]
		Incorrect bitmask
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110070

Machine data for path controller incomplete.		
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.	
Response	Class 3	
Solution	Class 7	
Parameter	%1:	Logical axis number [-]
	%2:	Incorrect value [-]
		Incorrect bitmask
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110071

Axis data for acceleration and velocity feedforward control incomplete.		
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.	
Response	Class 3	
Solution	Class 7	
Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Incorrect value [-]
		Incorrect bitmask
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110073

Parameters for digital drive interface incomplete.		
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.	
Response	Class 3	

Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		Incorrect bitmask	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110074

Parameters for manual mode incomplete.			
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.		
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		Incorrect bitmask	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110075

Parameters for position controller and interpolator incomplete.			
Description	The machine data record was not completely initialised. This can only happen if this data record is transferred in binary form to the CNC.		
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		Incorrect bitmask	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110076

Axis number is 0.			
Description	The logical number of the axis parameter list must not be zero.		
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Error value [-]	
		P-AXIS-00016	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110077

Gear change position exceeds positive software limit switch.

Description	The gear change position P-AXIS-00078 is above the positive software limit switch P-AXIS-00178.		
Response	Class	1	The gear change position is set to the middle of the two software limit switches.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00078	
	%3:	Limit value [-]	
		P-AXIS-00178	
	%4:	Corrected value [-]	
		P-AXIS-00078	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110078

Gear change position exceeds lower software limit switch.			
Description	The gear change position P-AXIS-00078 is below the negative software limit switch P-AXIS-00177.		
Response	Class	1	The gear change position is set to the middle of the two software limit switches.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00078	
	%3:	Limit value [-]	
		P-AXIS-00177	
	%4:	Corrected value [-]	
		P-AXIS-00078	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110079

maximum speed is not allowed to be 0.			
Description	The maximum velocity P-AXIS-00212 of the axis has been set to 0. This speed must be set to greater than 0.		
Response	Class	1	Initialization of parameter with default value.
Solution	Class	1	Correction of the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Error value [1µm/s or 0,001°/s]	
		P-AXIS-00212	
	%3:	Corrected value [1µm/s or 0,001°/s]	
		P-AXIS-00212	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110080

Maximum velocity is greater than the maximum CNC internal velocity limit.			
Description	The maximum velocity P-AXIS-00212 of the axis exceeds the maximum velocity of the CNC.		
Response	Class	1	The maximum velocity is assigned the maximum velocity stored in the CNC.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00212	
	%3:	Limit value [1µm/s or 0.001°/s]	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00212	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110081

maximum acceleration is not allowed to be 0.			
Description	The maximum acceleration P-AXIS-00008 of the axis is 0.		
Response	Class	1	Initialization of parameter with minimum acceleration of CNC value.
Solution	Class	1	Correction of the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0,001°/s]	
		P-AXIS-00008	
	%3:	Corrected value [1µm/s or 0,001°/s]	
		P-AXIS-00008	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110082

Maximum acceleration is greater than the maximum CNC internal acceleration limit.			
Description	The maximum acceleration P-AXIS-00008 of the axis exceeds the maximum acceleration of the CNC.		
Response	Class	1	The maximum acceleration is assigned a default value.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [mm/s² or °/s²]	
		P-AXIS-00008	
	%3:	Limit value [mm/s² or °/s²]	
	%4:	Corrected value [mm/s² or °/s²]	
		P-AXIS-00008	

Error type	2, Error message by data transfer from parameter list into control device.

ID 110083

Invalid value for minimum ramp time.			
Description	The value for the minimum ramp time P-AXIS-00201 is invalid.		
Response	Class	1	Warning output. The value for P-AXIS-00201 is set to the specified limit value.
Solution	Class	1	Set the value for the minimum ramp time P-AXIS-00201 greater than the limit value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016.	
	%2:	Current value [1 µs]	
		Current value for P-AXIS-00201.	
	%3:	Limit value [1 µs]	
		Maximum limit value for P-AXIS-00201.	
	%4:	Corrected value [1 µs]	
		Corrected value for P-AXIS-00201.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110084

Rapid feed rate is not allowed to be 0.			
Description	The rapid feed acceleration P-AXIS-00209 of the axis has been set to 0. This speed must be set to greater than 0.		
Response	Class	1	Initialization of parameter with default value.
Solution	Class	1	Correction of the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0,001°/s]	
		P-AXIS-00209	
	%3:	Corrected value [1µm/s or 0,001°/s]	
		P-AXIS-00209	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110085

Rapid traverse velocity is greater than maximum feed rate.			
Description	The rapid traverse velocity P-AXIS-00209 was specified greater than the maximum axis velocity.		
Response	Class	1	The rapid traverse velocity is assigned the maximum velocity of the axis.
Solution	Class	1	Correct the value in the parameter list.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1µm/s or 0.001°/s]
		P-AXIS-00209
	%3:	Limit value [1µm/s or 0.001°/s]
		P-AXIS-00212
	%4:	Corrected value [1µm/s or 0.001°/s]
		P-AXIS-00209
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110086

Invalid denominator for distance resolution.			
Description	The denominator for distance resolution P-AXIS-00233 of the measurement system was zero. Specify a value unequal to zero here. Distance resolution is specified as a factor P-AXIS-00234 / P-AXIS-00233 in increments per 0.1 µm or per 0.0001°.		
Response	Class	1	Initialise resolution with default value
Solution	Class	1	Correct denominator in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00233	
	%3:	Corrected value [-]	
		P-AXIS-00233	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110087

Controller window is not allowed to be 0.			
Description	The size of the window P-AXIS-00236 was specified as zero. The window is used to display the “in position” state of an axis if the position lag is smaller than the specified window.		
Response	Class	1	Initialise the position window with default value.
Solution	Class	1	Correct denominator in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00236	
	%3:	Corrected value [-]	
		P-AXIS-00236	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110088

Acceleration step 1 is 0.	
Description	The acceleration P-AXIS-00011 of the first stage of linear slope (not jerk limited) was set to 0. The acceleration must be greater than 0.

Response	Class	1	Initialization of acceleration with default value.
Solution	Class	1	Correction of the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00011	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00011	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110089

Acceleration step 1 is greater than maximum acceleration.			
Description	The acceleration P-AXIS-00011 of the first step of the linear slope (not jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00011	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00011	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110090

Acceleration step 2 is 0.			
Description	The acceleration P-AXIS-00012 of the second stage of linear slope (not jerk limited) was set to 0. The acceleration must be greater than 0.		
Response	Class	1	Initialization of acceleration with default value.
Solution	Class	1	Correction of the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00012	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00012	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110091

Acceleration step 2 is greater than maximum acceleration.	
Description	The acceleration P-AXIS-00012 of the second step of the linear slope (not jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.

Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00012	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00012	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110092

Limiting acceleration step 1 is 0.			
Description	The limiting acceleration P-AXIS-00005 of the first stage of the linear slope (not jerk limited) was specified as 0. The acceleration must be greater than 0.		
Response	Class	1	Initialization of acceleration with default value.
Solution	Class	1	Correction of the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00005	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00005	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110093

Acceleration limit step 1 is greater than maximum acceleration.			
Description	The acceleration P-AXIS-00005 of the first step of the linear slope (not jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00005	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00005	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110094

The limiting acceleration level 2 is 0.

Description	The limiting acceleration P-AXIS-00006 of the second stage of the linear slope (not jerk limited) was specified as 0. The acceleration must be greater than 0.		
Response	Class	1	Initialization of acceleration with default value.
Solution	Class	1	Correction of the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00006	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00006	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110095

Acceleration limit step 2 is greater than maximum acceleration.			
Description	The acceleration P-AXIS-00006 of the second step of the linear slope (not jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00006	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00006	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110096

Switching feed rate is 0.			
Description	The changeover speed P-AXIS-00221 between acceleration levels 1 and 2 of the linear slope (not jerk limited) was set to 0. The value must be bigger 0.		
Response	Class	1	Initialization of acceleration with default value.
Solution	Class	1	Correction of the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0,001°/s]	
		P-AXIS-00221	
	%3:	Corrected value [1µm/s or 0,001°/s]	
		P-AXIS-00221	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110097

Changeover speed is greater than maximum axis velocity.

Description	The changeover speed P-AXIS-00221 between acceleration steps 1 and 2 of the linear slope (not jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00221	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		P-AXIS-00008	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00221	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110098

Switching feed rate is 0.			
Description	The changeover speed P-AXIS-00211 at the current limit between acceleration levels 1 and 2 of the linear slope (not jerk limited) was set to 0. The value must be bigger 0.		
Response	Class	1	Initialization of acceleration with default value.
Solution	Class	1	Correction of the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0,001°/s]	
		P-AXIS-00211	
	%3:	Corrected value [1µm/s or 0,001°/s]	
		P-AXIS-00211	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110099

Changeover speed is greater than maximum axis velocity.			
Description	The changeover speed P-AXIS-00211 at the current limit between acceleration steps 1 and 2 of the linear slope (not jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit velocity to maximum value
Solution	Class	1	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00211	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		P-AXIS-00008	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00211	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110100

Acceleration is 0.			
Description	The acceleration P-AXIS-00001 of the non-linear slope (jerk-limited) was specified with the value 0. The acceleration must be greater than 0.		
Response	Class	1	Initialise acceleration at default value.
Solution	Class	1	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00001	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00001	
Error type	2, Error message by data transfer from parameter list into control device. .		

ID 110101

Acceleration is greater than maximum axis acceleration.			
Description	The acceleration P-AXIS-00001 of the non-linear slope (jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00001	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00001	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110102

Deceleration is 0.			
Description	Deceleration P-AXIS-00002 of the non-linear slope (jerk limited) was specified with the value 0. The acceleration must be greater than 0.		
Response	Class	1	Initialise acceleration at default value.
Solution	Class	1	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00002	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00002	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110103

Deceleration is greater than maximum axis acceleration.			
Description	The deceleration P-AXIS-00002 of the non-linear slope (jerk limited) was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00002	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00002	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110104

Ramp time for acceleration up-gradation is too small.			
Description	The time period for up-gradation of maximum acceleration P-AXIS-00196 (ramp time) of the non-jerk limited slope was specified as smaller than the minimum ramp time P-AXIS-00201.		
Response	Class	1	Limit ramp time to minimum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1 µs]	
		P-AXIS-00196	
	%3:	Lower limit value [1 µs]	
		P-AXIS-00201	
	%4:	Corrected value [1 µs]	
		P-AXIS-00196	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110105

Ramp time for acceleration down-gradation is too small.			
Description	The time period for down-gradation of the max. acceleration P-AXIS-00195 (ramp time) of the non-jerk limited slope was specified as smaller than the minimum ramp time P-AXIS-00201.		
Response	Class	1	Limit ramp time to minimum value
Solution	Class	1	Correct the value in the parameter list

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1 µs]
		P-AXIS-00195
	%3:	Lower limit value [1 µs]
		P-AXIS-00201
	%4:	Corrected value [1 µs]
		P-AXIS-00195
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110106

Ramp time for deceleration up-gradation is too small.			
Description	The time period for up-gradation of the max. deceleration P-AXIS-00198 (ramp time) of the non-jerk limited slope was specified as smaller than the minimum ramp time P-AXIS-00201.		
Response	Class	1	Limit ramp time to minimum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1 µs]	
		P-AXIS-00198	
	%3:	Lower limit value [1 µs]	
		P-AXIS-00201	
	%4:	Corrected value [1 µs]	
		P-AXIS-00198	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110107

Ramp time for deceleration down-gradation is too small.			
Description	The time period for down-gradation of the max. deceleration P-AXIS-00197 (ramp time) of the non-jerk limited slope was specified as smaller than the minimum ramp time P-AXIS-00201.		
Response	Class	1	Limit ramp time to minimum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1 µs]	
		P-AXIS-00197	
	%3:	Lower limit value [1 µs]	
		P-AXIS-00201	
	%4:	Corrected value [1 µs]	
		P-AXIS-00197	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110110

Distance between software limit switches is 0.			
Description	The motion range between the two software limit switches is 0.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify the parameter settings of P-AXIS-00177 and P-AXIS-00178.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Parameterised value of P-AXIS-00177	
	%3:	Current value[0.1 10^-3 mm or ø]	
		Parameterised value of P-AXIS-00178	
	%4:	Error value [0.1 10^-3 mm or ø]	
		Actual motion range	
	%5:	Corrected value [0.1 10^-3 mm or ø]	
		Corrected range	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110111

Speed override is 0.			
Description	The parameter P-AXIS-00109 for the maximum axis specific velocity override is smaller than permissible minimum value. The parameter value is set to default minimum value.		
Response	Class	1	None.
Solution	Class	1	Set parameter P-AXIS-00109 greater than or equal to the lower limit value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current parameter value P-AXIS-00109.	
	%3:	Corrected value [-]	
		Corrected parameter value P-AXIS-00109.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110112

Velocity override exceeds limit.			
Description	The parameter P-AXIS-00109 for the maximum axis-specific velocity override is smaller than the permissible maximum value. The parameter value is set to default maximum value.		
Response	Class	1	None.
Solution	Class	1	Set parameter P-AXIS-00109 to smaller than or equal to the upper limit value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [-]
		Current parameter value P-AXIS-00109.
	%3:	Limit value [-]
		Maximum value for parameter.
	%4:	Corrected value [-]
		Corrected parameter value P-AXIS-00109.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110113

Measuring feed rate is 0.			
Description	The parameter P-AXIS-00215 for measurement feed motion is smaller than minimum permissible value. The parameter value is set to default minimum value.		
Response	Class	1	None.
Solution	Class	1	Set measurement feed greater than minimum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [1µm/s or 0,001°/s]	
		Current measurement feed P-AXIS-00215.	
	%3:	Corrected value [1µm/s or 0,001°/s]	
		Corrected measurement feed @@P-AXIS-00215.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110114

Measurement speed exceeds limit.			
Description	The parameter P-AXIS-00215 for measurement speed is greater than the maximum permissible value. The parameter value is set to default maximum value.		
Response	Class	1	None.
Solution	Class	1	Set measurement speed P-AXIS-00215 to smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current measurement speed P-AXIS-00215.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum value for measurement speed.	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		Corrected measurement speed P-AXIS-00215.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110115

Number of gear stage is 0.			
Description	The parameter P-AXIS-00079 for active gear stage after start-up of CNC is set to 0. The gear stage is set to default gear stage.		
Response	Class	1	None.
Solution	Class	1	Use gear stage greater than 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current gear stage P-AXIS-00079.	
	%3:	Corrected value [-]	
		Corrected gear stage P-AXIS-00079.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110117

Measurement using SERCOS protocol only for SERCOS type drives.			
Description	Invalid parameter setting for P-AXIS-00116 for SERCOS measurement.		
Response	Class	1	The parameter P-AXIS-00116 is assigned to default value.
Solution	Class	1	Only use parameter P-AXIS-00116 if drive type P-AXIS-00020 is also set to SERCOS.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [-]	
		Current value of P-AXIS-00116.	
	%3:	Drive type [-]	
		Current value of P-AXIS-00020	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00116.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110118

No measurement method was defined.			
Description	If the axis is marked as a measurement axis P-AXIS-00118, a measurement method must also be parameterised. The parameter P-AXIS-00117 is set to 1.		
Response	Class	1	None.
Solution	Class	1	Depending on the drive type P-AXIS-00020, at least one measurement method must be parameterised or P-AXIS-00118 must be set to 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Corrected value [-]	
		Current drive type P-AXIS-00020.	
	%3:		
		Corrected parameter P-AXIS-00117.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110119

Parameter may not be 0.			
Description	The parameter P-AXIS-00023 for controlling the display function in the position controller is assigned an invalid value. The parameter P-AXIS-00023 is set to the minimum value.		
Response	Class	1	None.
Solution	Class	1	Only assign values greater than 0 to parameter P-AXIS-00023.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [-]	
		Current value P-AXIS-00023.	
	%3:	Corrected value [-]	
		Corrected value P-AXIS-00023.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110120

Parameter value exceeds limit.			
Description	The parameter P-AXIS-00023 for controlling the display function in the position controller is assigned a high invalid value. Parameter P-AXIS-00023 is set to the maximum value.		
Response	Class	1	None.
Solution	Class	1	Only assign values smaller than the maximum value to parameter P-AXIS-00023.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [-]	
		Current value P-AXIS-00023.	
	%3:	Limit value [-]	
		Upper limit value.	
	%4:	Corrected value [-]	
		Corrected value P-AXIS-00023.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110121

Illegal mode of feedforward control.			
Description	An unknown mode was specified in the feedforward control setting. Only the values specified in the description (see P-AXIS-00223) may be set.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify the parameter P-AXIS-00223

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [-]
		Bit array P-AXIS-00223
	%3:	Limit value [-]
		P-AXIS-00223
	%4:	Corrected value [-]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110122

Numerator of velocity feed forward factor may not be 0.			
Description	The numerator P-AXIS-00228 of the factor for normalising the velocity feedforward control was assigned 0. This is not permitted. The normalisation factor is defined in P-AXIS-00228/ P-AXIS-00229		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify the parameter P-AXIS-00228.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Current value [-]	
		Parameterised value of P-AXIS-00228	
	%3:	Corrected value [-]	
		P-AXIS-00228	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110123

Denominator of velocity feed forward factor may not be 0.			
Description	The denominator P-AXIS-00229 of the factor for normalising the velocity feedforward control was assigned 0. This is not permitted. The normalisation factor is defined in P-AXIS-00228/ P-AXIS-00229		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify the parameter P-AXIS-00229.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Current value [-]	
		Parameterised value of P-AXIS-00229	
	%3:	Corrected value [-]	
		P-AXIS-00229	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110124

Handwheel resolution is smaller than lower limit.			
Description	When manual mode is active, the active axis-specific resolution P-AXIS-00025 must be greater than 0.		
Response	Class	1	None.
Solution	Class	1	Set handwheel resolution P-AXIS-00025 to greater than 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current resolution P-AXIS-00025.	
	%3:	Corrected value [-]	
		Corrected resolution P-AXIS-00025.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110125

Maximum velocity for manual mode may not be 0.			
Description	The maximum velocity P-AXIS-00213 with active manual mode is set to zero. The maximum velocity is set to default value.		
Response	Class	1	None.
Solution	Class	1	Set maximum velocity for manual mode P-AXIS-00213 to greater than 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current maximum velocity P-AXIS-00213.	
	%3:	Corrected value [1µm/s or 0.001°/s]	
		Corrected maximum velocity P-AXIS-00213.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110126

Maximum velocity for manual mode exceeds limit.			
Description	The maximum velocity P-AXIS-00213 in active manual mode is greater than the maximum value. The maximum velocity is set to default value.		
Response	Class	1	None.
Solution	Class	1	Set maximum velocity for manual mode P-AXIS-00213 to smaller than maximum value P-AXIS-00212.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current maximum velocity P-AXIS-00213.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Upper limit value of velocity.	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		Corrected maximum velocity P-AXIS-00213.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110127

Maximum acceleration for manual mode may not be 0.			
Description	The maximum acceleration P-AXIS-00009 for active manual mode is less than the minimum permissible value. The acceleration is set to minimum permissible value.		
Response	Class	1	None.
Solution	Class	1	Select manual mode acceleration to greater than minimum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Current setting of P-AXIS-00009.	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		Corrected assignment P-AXIS-00009.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110128

Maximum acceleration for manual mode exceeds limit.			
Description	The maximum acceleration P-AXIS-00009 with active manual mode was specified as greater than the maximum axis acceleration P-AXIS-00008. The acceleration for manual mode is set to P-AXIS-00008.		
Response	Class	1	None.
Solution	Class	1	Do not select manual mode acceleration greater than P-AXIS-00008.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Current setting of P-AXIS-00009.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Upper limit value for acceleration P-AXIS-00008.	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		Limited manual mode acceleration P-AXIS-00009.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110129

Percentage of manual mode maximum velocity in manual mode too small.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum velocity P-AXIS-00212 for the manual mode interpolator is determined by P-AXIS-00083. The percentage of the maximum velocity is set to a minimum value.		
Response	Class	1	None.
Solution	Class	1	Set percentage for manual mode P-AXIS-00083 to greater than minimum value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [0.1%]
		Current value of percentage P-AXIS-00083.
	%3:	Lower limit value [0.1%]
		Lower limit value for percentage value.
	%4:	Corrected value [0.1%]
		Corrected percentage P-AXIS-00083.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110130

Percentage of manual mode maximum velocity in manual mode too big.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum velocity P-AXIS-00212 for the manual mode interpolator is determined by P-AXIS-00083. The percentage of the maximum velocity is set to a default maximum value.		
Response	Class	1	None.
Solution	Class	1	Set percentage for manual mode P-AXIS-00083 to smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1%]	
		Current value of percentage P-AXIS-00083.	
	%3:	Upper limit value [0.1%]	
		Upper limit value for percentage value.	
	%4:	Corrected value [0.1%]	
		Corrected percentage P-AXIS-00083.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110131

Percentage of manual mode maximum acceleration in manual mode too small.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum acceleration P-AXIS-00008 for the manual mode interpolator is determined by P-AXIS-00082. The percentage of maximum acceleration is set to a minimum limit.		
Response	Class	1	None.
Solution	Class	1	Set percentage for manual mode P-AXIS-00082 to greater than minimum value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [0.1%]
		Current value of percentage P-AXIS-00082.
	%3:	Lower limit value [0.1%]
		Lower limit value for percentage value.
	%4:	Corrected value [0.1%]
		Corrected percentage value P-AXIS-00082.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110132

Percentage of manual mode maximum acceleration in manual mode too big.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum acceleration P-AXIS-00008 for the manual mode interpolator is determined by P-AXIS-00082. The percentage of maximum acceleration is set to maximum value.		
Response	Class	1	None.
Solution	Class	1	Set percentage for manual mode P-AXIS-00082 to smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1%]	
		Current percentage value P-AXIS-00082.	
	%3:	Upper limit value [0.1%]	
		Upper limit value for percentage value.	
	%4:	Corrected value [0.1%]	
		Corrected percentage value P-AXIS-00082.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110133

Percentage of interpolation maximum velocity in manual mode too small.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum velocity P-AXIS-00212 for the path interpolator is determined by P-AXIS-00095. The percentage of the maximum velocity is set to the minimum value.		
Response	Class	1	None.
Solution	Class	1	Set percentage for manual mode P-AXIS-00095 to greater than minimum value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [0.1%]
		Current percentage value P-AXIS-00095.
	%3:	Lower limit value [0.1%]
		Lower limit value for percentage value.
	%4:	Corrected value [0.1%]
		Corrected percentage value P-AXIS-00095.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110134

Percentage of interpolation maximum velocity in manual mode too big.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum velocity P-AXIS-00212 for the path interpolator is determined by P-AXIS-00095. The percentage of the maximum velocity is set to the maximum value.		
Response	Class	1	None.
Solution	Class	1	Set percentage for manual mode P-AXIS-00095 to smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1%]	
		Current percentage value P-AXIS-00095.	
	%3:	Upper limit value [0.1%]	
		Upper limit value for percentage value.	
	%4:	Corrected value [0.1%]	
		Corrected percentage value P-AXIS-00095.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110135

Percentage of interpolation maximum acceleration in manual mode too small.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum acceleration P-AXIS-00008 for the path interpolator is determined by P-AXIS-00094. The percentage of maximum acceleration is set to a minimum limit.		
Response	Class	1	None.
Solution	Class	1	Set percentage for path interpolator P-AXIS-00094 to greater than minimum value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [0.1%]
		Current percentage value P-AXIS-00094.
	%3:	Lower limit value [0.1%]
		Lower limit value for percentage value.
	%4:	Corrected value [0.1%]
		Corrected percentage value P-AXIS-00094.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110136

Percentage of interpolation maximum acceleration in manual mode too small.			
Description	In manual mode with parallel interpolation (G201), the axis dynamic values are distributed with a percentage value between the path interpolator and the axis-specific manual mode interpolator. The percentage of the maximum acceleration P-AXIS-00008 for the path interpolator is determined by P-AXIS-00094. The percentage of maximum acceleration is set to a maximum value.		
Response	Class	1	None.
Solution	Class	1	Set percentage value for path interpolator P-AXIS-00094 to smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1%]	
		Current percentage value P-AXIS-00094.	
	%3:	Upper limit value [0.1%]	
		Upper limit value for percentage value.	
	%4:	Corrected value [0.1%]	
		Corrected percentage value P-AXIS-00094.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110137

Filter time for "handwheel" operation mode may not be 0.			
Description	For manual mode in handwheel operation mode, the parameter P-AXIS-00069 sets the number of CNC cycles which are used to obtain a mean value for hand wheel increments. The filter time is set to a minimum value.		
Response	Class	1	None.
Solution	Class	1	Set filter time greater than minimum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1 µs]	
		Current filter time P-AXIS-00069.	
	%3:	Corrected value [1 µs]	
		Corrected filter time P-AXIS-00069.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110138

Filter time for "handwheel" operation mode exceeds limit.			
Description	For manual mode in handwheel operation mode, the parameter P-AXIS-00069 sets the number of CNC cycles which are used to obtain a mean value for hand wheel increments. The filter time is limited to the permissible maximum value.		
Response	Class	1	None.
Solution	Class	1	Set the filter time to smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1 µs]	
		Current filter time P-AXIS-00069.	
	%3:	Upper limit value [1 µs]	
		Corrected filter time P-AXIS-00069.	
	%4:	Corrected value [1 µs]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110139

Rapid traverse velocity for operation mode "continuous jog mode" may not be 0.			
Description	For manual mode in “continuous jog mode”, the parameter P-AXIS-00210 sets the rapid traverse velocity. The filter time is set to the permissible minimum value.		
Response	Class	1	None.
Solution	Class	1	P-AXIS-00210 must be greater than minimum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current rapid traverse velocity P-AXIS-00210.	
	%3:	Corrected value [1µm/s or 0.001°/s]	
		Corrected rapid traverse velocity P-AXIS-00210.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110140

Rapid traverse velocity for operation mode "continuous jog mode" exceeds limit.			
Description	For manual mode in “continuous jog mode”, the parameter P-AXIS-00210 sets the rapid traverse velocity. The rapid traverse velocity P-AXIS-00210 is set to the permissible maximum value.		
Response	Class	1	None.
Solution	Class	1	P-AXIS-00210 must be smaller than maximum value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [1µm/s or 0.001°/s]
		Current rapid traverse velocity P-AXIS-00210.
	%3:	Upper limit value [1µm/s or 0.001°/s]
		Maximum permissible velocity.
	%4:	Corrected value [1µm/s or 0.001°/s]
		Corrected rapid traverse velocity P-AXIS-00210.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110141

Incremental feed rate is greater than the maximum permissible feed rate.			
Description	With manual mode in “continuous jog mode”, the parameter P-AXIS-00077 defines the continuous jog velocity. The specified value is greater than the permissible maximum value and is limited to this value.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and correct the parameter P-AXIS-00077
Parameter	%1:	Logical axis number	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current velocity P-AXIS-00077.	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum permissible velocity.	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		Corrected velocity P-AXIS-00077.	
	%5:	Current value [-]	
		Array index of the incorrect entry	
Error type	2, Error message by data transfer from parameter list into control device.		
	[1µm/s or 0.001°/s]		

ID 110142

Negative jog incremental width is not permissible.			
Description	For manual mode in “jog incremental mode”, the parameter P-AXIS-00232 sets the jog incremental width. The jog incremental width is set to the minimum value.		
Response	Class	1	None.
Solution	Class	1	P-AXIS-00232 must be greater than minimum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current jog incremental width P-AXIS-00232.	
	%3:	Corrected value [0.1 µm or 0.0001°]	
		Corrected jog incremental width P-AXIS-00232.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110143

Incremental jog velocity is greater than the maximum permissible velocity.			
Description	For manual mode in "jog incremental mode", the parameter P-AXIS-00076 sets the jog velocity. The jog velocity is set to the maximum value.		
Response	Class	1	None.
Solution	Class	1	P-AXIS-00076 must be smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current jog velocity P-AXIS-00076.	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum permissible jog velocity.	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		Corrected jog velocity P-AXIS-00076.	
	%5:	Current value [-]	
		Array index of the incorrect entry	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110145

Positive relative manual mode offset limit negative.			
Description	The positive offset limit P-AXIS-00138 for the positive motion range in active manual mode is less than 0. The positive offset limit is corrected to the default value.		
Response	Class	1	None.
Solution	Class	1	Set positive offset limit P-AXIS-00138 greater than 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Current value P-AXIS-00138.	
	%3:	Current value [-]	
		Corrected value P-AXIS-00138.	
	%4:	Corrected value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110147

Maximum homing velocity is greater than the permissible maximum velocity.			
Description	The fast velocity set in the axis parameter list for CNC-controlled homing exceeds the maximum velocity of the axis.		
Response	Class	1	The homing velocity is limited to the maximum velocity.
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1µm/s or 0.001°/s]
		P-AXIS-00219
	%3:	Limit value [1µm/s or 0.001°/s]
		P-AXIS-00213
	%4:	Corrected value [1µm/s or 0.001°/s]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110148

Minimum homing velocity is greater than the permissible maximum velocity.			
Description	The slow velocity set in the axis parameter list for CNC-controlled homing exceeds the maximum velocity of the axis.		
Response	Class	1	The homing velocity is limited to the maximum velocity.
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00218	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00213	
	%4:	Corrected value [1µm/s or 0.001°/s]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110149

Minimum velocity is greater than maximum homing velocity.			
Description	The slow velocity set in the axis parameter list for CNC-controlled homing exceeds the fast homing velocity.		
Response	Class	1	The slow homing velocity is limited to the fast homing velocity.
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00218	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00219	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00218	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110150

Gain factor is 0.

Description	The proportional gain of the axis was specified as 0. This is an invalid value.		
Response	Class	1	Value is initialised with default value.
Solution	Class	1	Correct the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [0.01/sec]	
		P-AXIS-00099	
	%3:	Corrected value [0.01/s]	
		P-AXIS-00099	
	%4:	Current value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110151

Denominator at resolution of D/A converter may not be 0.			
Description	The resolution of the digital/analogue converter (denominator) was set to 0. This value is invalid.		
Response	Class	1	Value is initialised with default value.
Solution	Class	1	Correct the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00128	
	%3:	Corrected value [-]	
		P-AXIS-00128	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110154

Dynamic position lag exceeds range of permissible values.			
Description	The factor P-AXIS-00167 of position lag monitoring was too large. The factor has a different meaning depending on position lag monitoring P-AXIS-00172.		
Response	Class	1	The position lag factor is limited to the maximum value.
Solution	Class	1	Correct value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00167	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00167	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110162

List interpreter cannot evaluate the predefined list.			
Description	CNC wants evaluate an axis parameter list or compensation value list, which should be exist as file, but detects that the file doesn't exist in path.		
Response	Class	1	
Solution	Class	7	If the message occurs during the control start-up, check the path specifications in parameters P-STUP-00015 and P-STUP-00017 of the start-up list. If the message appears after an order to access a file, check the path send by the order.
Error type	-		

ID 110163 - 110181

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 110182

Factor for spindle speed is 0.			
Description	The specified factor for speed monitoring P-AXIS-00217 is zero.		
Response	Class	1	Value is initialised by a default.
Solution	Class	1	Correct the value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		P-AXIS-00217	
	%3:	Corrected value [-]	
		P-AXIS-00217	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110183

Changing the axis type requires restart of the control.			
Description	The parameter P-AXIS-00018 for the type of axis cannot be changed while the NC kernel is running. The new axis type is not adopted; the previous active axis type remains valid.		
Response	Class	1	None.
Solution	Class	1	Only change the axis type P-AXIS-00018 in configuration mode or when the NC is not running.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [-]
		Current value of P-AXIS-00018.
	%3:	Corrected value [-]
		Corrected value of P-AXIS-00018.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110184 - 110198

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 110199

Received unknown axis.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 110200 - 110209

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 110210

Unknown type of axis corrected to translatory axis type.			
Description	The axis type specified in the parameter list P-AXIS-00018 is unknown.		
Response	Class	1	The type is set to P-AXIS-00018 =1.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00018	
	%3:	Corrected value [-]	
		P-AXIS-00018	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110211

Corrected unknown drive type to drive type simulation.			
Description	The specified type of drive is unknown.		
Response	Class	1	The type is set to P-AXIS-00020 = 4
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00020	
	%3:	Corrected value [-]	
		P-AXIS-00020	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110212

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 110213

Unknown characteristic acceleration curve.			
Description	An unknown type P-AXIS-00202 of the acceleration curve was specified. Following types are allowed: 0 – off 1 – hyperbole 4th grade 2 – polynomial 3rd degrees 3 – Corresponding to asynchronous motor 1/n or 1/n²		
Response	Class	1	Turn off type.
Solution	Class	1	Correct list entry.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Error value [-]	
		P-AXIS-00202	
	%3:	Corrected value [-]	
		P-AXIS-00202	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110214

Limit speed is smaller than permissible value.			
Description	The limit speed P-AXIS-00130 of the acceleration curve was specified too high.		
Response	Class	1	Turn off type.
Solution	Class	1	Correct list entry.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Incorrect value [1µm/s or 0.001°/s]
		P-AXIS-00130
	%3:	Lower limit value [1µm/s or 0.001°/s]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110215

Acceleration limit is greater than permissible value.			
Description	The acceleration limit P-AXIS-00007 of the curve-controlled acceleration was specified too high. It may not be higher than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit the value
Solution	Class	1	Adapt the list entry
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00007	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00008	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00007	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110216

Acceleration minimum is smaller than permissible value.			
Description	Minimum acceleration limit P-AXIS-00010 of the curve-controlled acceleration was specified too high. It may not be higher than the constant acceleration P-AXIS-00007.		
Response	Class	1	Limit the value
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00010	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00007	
	%4:	Incorrect value [1µm/s or 0.001°/s]	
		P-AXIS-00010	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110217

Invalid number of sections within list for leadscrew error compensation.

Description	In the compensation value list for leadscrew error compensation, parameter P-COMP-00020 specifies the number of entries that were specified in the compensation value list for this type of compensation. When the CNC interpreted this list, it check whether the parameterised number of entries exceeds the lower or upper number of entries that are necessary or to use the leadscrew error compensation correctly.		
Response	Class	3	The CNC reaction depends on whether the lower or upper limit is violated for the number of entries in the compensation value list. If the value for P-COMP-00020 is • **0** The CNC uses 2 entries internally, both initialised by 0. • **1** The CNC uses an additional entry internally initialised with the values of the first entry. • **> upper limit** The CNC uses only the maximum number of entries that can be processed (parameter 2 of the error message) for leadscrew error compensation.
Solution	Class	7	Check the value in parameter P-COMP-00020. It must be ≥ 2 and ≤ than the value in parameter 2 of the error message.

Parameter	%1:	Logical axis number [-]
		Number of entries in the compensation value list for leadscrew error compensation specified by the operator in parameter P-COMP-00020.
	%2:	Incorrect value [-]
		Indicates how many entries are used for leadscrew error compensation after this message.
	%3:	Lower limit value [-]
	%4:	Upper limit value [-]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110218

Illegal grating of correction value intervals.			
Description	The step size between 2 interpolation points from the list of compensation values for leadscrew error compensation is specified in parameter P-COMP-00018. The message is displayed because the value for the interval is < 0.		
Response	Class	3	Correction of the value
Solution	Class	7	Check and modify the parameter P-COMP-00018. The value must be greater than 0.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Error value [Incr.]
		The CNC uses this value for parameter P-COMP-00009 instead of the value indicated by the user.
	%3:	Lower limit value [Incr.]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110219

Maximum velocity of manual mode is greater than the maximum axis velocity.			
Description	The set maximum manual mode velocity P-AXIS-00213 is higher than the general maximum axis velocity P-AXIS-00212.		
Response	Class	1	Velocity is limited to maximum axis velocity.
Solution	Class	1	Correct parameter in axis list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [1µm/s or 0.001°/s]	
		P-AXIS-00213	
	%3:	Corrected value [1µm/s or 0.001°/s]	
	%4:	Instance [-]	
		Current gear stage P-AXIS-00136	
	%5:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum axis velocity P-AXIS-00212	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110220

Rapid traverse velocity in manual mode is greater than the maximum axis velocity.			
Description	The maximum rapid traverse velocity set in manual mode P-AXIS-00210 is higher than the general maximum axis velocity P-AXIS-00212.		
Response	Class	1	Velocity is limited to maximum axis velocity.
Solution	Class	1	Correct parameter in axis list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00210	
	%3:	Corrected value [1µm/s or 0.001°/s]	
	%4:	Instance [-]	
		Current gear stage P-AXIS-00136	
	%5:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum axis velocity P-AXIS-00209	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110221

Continuous jog velocity exceeds maximum axis velocity.			
Description	The maximum velocity set for continuous jog mode P-AXIS-00077 is higher than the general maximum axis velocity P-AXIS-00213.		
Response	Class	1	Velocity is limited to maximum axis velocity.
Solution	Class	1	Correct parameter in axis list.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1µm/s or 0.001°/s]
		P-AXIS-00077
	%3:	Corrected value [1µm/s or 0.001°/s]
	%4:	Instance [-]
		Current gear stage P-AXIS-00136
	%5:	Upper limit value [1µm/s or 0.001°/s]
		Maximum axis velocity in manual mode P-AXIS-00213
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110222

Incremental jog velocity exceeds maximum axis velocity.			
Description	The maximum velocity set for incremental jog mode P-AXIS-00076 is higher than the general maximum axis velocity P-AXIS-00213.		
Response	Class	1	Velocity is limited to maximum axis velocity.
Solution	Class	1	Correct parameter in axis list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00076	
	%3:	Corrected value [1µm/s or 0.001°/s]	
	%4:	Instance [-]	
		Current gear stage P-AXIS-00136	
	%5:	Upper limit value [1µm/s or 0.001°/s]	
		Maximum axis velocity in manual mode P-AXIS-00213	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110225 - 110227

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires controller restart.

ID 110228

Logical axis number in compensation data list and in command PDU not equal.			
Description	CNC is commanded to parametrise an axis with values from a specified list. This message is output because the number of the logical axis in the list is not the same as the number of the axis.		
Response	Class	3	Abort interpretation of the list.

Solution	Class	7	Check which list is to be used to parameterise the axis. Possibly change change the number of the logical axis in the list or use a different list.
Parameter	%1:	Current value [-]	
		Number of a logical axis. This number comes from the list that was to be interpreted.	
	%2:	Expected value [-]	
		Number of logical axis which is to be parametrised with the values contained in list.	
Error type	-		

ID 110229

Logical axis number in compensation value list and in start-up list not equal.			
Description	A logical axis is to use a compensation value list. This is why this compensation value list is interpreted. The system detects that the number of the logical axis of the compensation value list does not match the number of the logical axis that is to use this list.		
Response	Class	3	Abort interpretation of the compensation value list.
Solution	Class	7	Check which compensation value list is to be used for the axis. Possibly change change the number of the logical axis in the list or use a different list.
Parameter	%1:	Current value [-]	
		Number of a logical axis. This number (P-COMP-00001) comes from the compensation value list that was to be used.	
	%2:	Expected value [-]	
		Number of the logical axis to be used by the compensation value list (P-STUP-00036).	
Error type	-		

ID 110230

Number of gear stages of a C axis out of range.			
Description	In an axis with spindle P-AXIS-00018 axis type, the parameter for the default CAX gear stage P-AXIS-00052 has an invalid value. In the rotary C axis, the #CAX command is used to change to the default CAX gear stage when the CAX function is selected. The CAX gear stage is set to default value. For more information on the CAX function, see [PROG// Section: C axis machining].		
Response	Class	1	None.
Solution	Class	1	Use correct entry for default CAX gear stage P-AXIS-00052.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current default CAX gear stage P-AXIS-00052.	
	%3:	Limit value [-]	
		Maximum value of gear stage.	
	%4:	Corrected value [-]	
		Corrected default CAX gear stage P-AXIS-00052.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110231

A translatory axis cannot be a C-axis.			
Description	The C axis setting was specified for a translatory axis in axis mode P-AXIS-00015 (bit-coded). This is just possible for rotary axis or spindles.		
Response	Class	1	The linear axis mode is automatically selected. C-axis bit is cleared.
Solution	Class	1	Correction of axis parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Error value [-]	
		P-AXIS-00015	
	%3:	Corrected value [-]	
		P-AXIS-00015	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110232

Number of hardware-specific data record out of permissible range.			
Description	The specified default hardware number of the gear stage is not valid.		
Response	Class	1	Hardware number is set to default value.
Solution	Class	1	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00054	
	%3:	Upper limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00054	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110233 / 110234

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires controller restart.

ID 110236

Leadscrew error compensation activated without compensation value list.

Description	The CNC detects an inconsistent parameterisation regarding leadscrew error compensation. In an axis configuration list, the leadscrew error compensation was activated by the parameter P-AXIS-00175. The CNC now expects the information regarding the path and the name by which a corresponding compensation value list can be found. To do this, the data in parameters P-STUP-00016, P-STUP-00017, P-STUP-00036 must be parameterised correctly in start-up list. The parameters P-COMP-00001 and P-COMP-00020 must also be entered correctly for leadscrew error compensation in the appropriate compensation value list.		
Response	Class	1	CNC disables leadscrew error compensation.
Solution	Class	7	Check the parameters P-STUP-00016 P-STUP-00017 P-STUP-00036 P-COMP-00001 P-COMP-00020 for data consistency. If leadscrew error compensation is not required, set the parameter P-AXIS-00175 to the value 0.
Parameter	%1:	Logical axis number [-]	
		Parameter P-AXIS-00016 from an axis configuration list where the leadscrew error compensation was activated by P-AXIS-00175.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110237

Order of filter too high.			
Description	The specified order of the filter is too high.		
Response	Class	1	Warning, the filter order is limited to the maximum permissible value.
Solution	Class	1	Correct the filter order P-AXIS-00140
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Number of filter affected	
	%3:	Current value [-]	
		Filter type P-AXIS-00204	
	%4:	Incorrect value [-]	
		Invalid filter order P-AXIS-00140	
	%5:	Corrected value [-]	
		Corrected filter order P-AXIS-00140	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110238

Filter prototype not permitted.	
Description	The filter prototype specified is unknown. Only the specified filter prototypes may be set.

Response	Class	1	The filter prototype is set to 1 (critical damping).
Solution	Class	1	Correct the filter prototype P-AXIS-00153
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Number of filter affected	
	%3:	Current value [-]	
		Invalid filter prototype P-AXIS-00153	
	%4:	Lower limit value [-]	
		Corrected filter prototype P-AXIS-00153	
	%5:	Upper limit value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110239

Filter type not permitted.			
Description	The specified filter type is unknown. You can only set the specified filter types.		
Response	Class	3	The filter is deactivated.
Solution	Class	7	Correct the filter type P-AXIS-00204
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Number of filter affected	
	%3:	Incorrect value [-]	
		Invalid filter type	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110240

Filter share exceeds permissible range.			
Description	This numerical value determines the signal component that is passed through the filter.		
Response	Class	1	The signal share is limited to the maximum value (0% or 100%).
Solution	Class	1	Correct value of signal share P-AXIS-00164
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [0.1%]	
		Number of filter affected	
	%3:	Current value [0.1%]	
		Invalid value for signal share P-AXIS-00164	
	%4:	Corrected value [-]	
		Corrected value for signal share P-AXIS-00164	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110241

Acceleration at current limit is 0.			
Description	Rapid traverse acceleration P-AXIS-00004 of the non-linear slope (jerk-limited) was specified with the value 0. The acceleration must be greater than 0.		
Response	Class	1	Initialise acceleration at default value.
Solution	Class	1	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00004	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00004	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110242

Acceleration at current limit is greater than maximum axis acceleration.			
Description	The rapid feed acceleration P-AXIS-00004 of the non-linear slope (jerk limited) was specified to be greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00004	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00004	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110243

Ramp time for acceleration at current limit too small.			
Description	The time period P-AXIS-00200 (ramp time) for up-gradation and down-gradation of the maximum current limit acceleration P-AXIS-00004 of the non-jerked limited slope was specified as less than the minimum ramp time P-AXIS-00201.		
Response	Class	1	Limit ramp time to minimum value
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1 µs]	
		P-AXIS-00200	
	%3:	Lower limit value [1 µs]	
		P-AXIS-00201	
	%4:	Corrected value [1 µs]	
		P-AXIS-00200	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110244

Logical axis number of master axis is missing for gantry operation.			
Description	The axis is marked as a slave axis of a gantry combination in P-AXIS-00015 parameter, but there is a missing entry for the lead master axis P-AXIS-00070.		
Response	Class	3	None.
Solution	Class	7	Use correct entry for logical axis number P-AXIS-00070 of lead master axis in the gantry combination.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Logical axis number [-]	
		Current logical axis number of the master axis P-AXIS-00070.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110246

Logical axis number for collision axis missing.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
	%2:	Logical axis number [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

2.11.2 ID-range 110250-110499

ID 110250

Speed limit N1 greater N2.			
Description	Speed N1 P-AXIS-00241 is greater than speed N2 P-AXIS-00242. This is not possible.		
Response	Class	1	N1 is limited by N2
Solution	Class	1	Correct the list entry.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [1µm/s or 0.001°/s]	
		P-AXIS-00241	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00242	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00241	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110251

Acceleration limit is greater than permissible maximum value.			
Description	The maximum acceleration P-AXIS-00240 of the curve-controlled acceleration was specified too high. It may not be higher than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00240	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00008	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00240	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110253

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 110254

Master spindle not found.			
Description	The master axis to which the slave should be synchronised (camming or gearing) is not available.		
Response	Class	6	Axis is stopped immediately.
Solution	Class	6	Correction of command for camming or gearing.
Parameter	%1:		
	%2:	Error value [-]	
Error type	1, Error message from NC program.		

ID 110264

Unknown entry in configuration list for PROFIBUS telegram			
Description	The number of increments per revolution is specified as 0.		
Response	Class	3	Parameter is set to 1.
Solution	Class	6	Set correct parameter, check configuration of telegram.
Parameter	%1:	Logical axis number [-]	
		see P-AXIS-00016	
	%2:	Incorrect value [-]	
	%3:	Current value [-]	
	%4:	Current value [-]	
	%5:	Current value [-]	
Error type	-		

ID 110267

Minimum position lag is greater than maximum position lag.			
Description	The minimum position lag P-AXIS-00169 was specified greater than the maximum position lag P-AXIS-00168.		
Response	Class	1	The minimum position lag is limited to the maximum value.
Solution	Class	1	Correct value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [0.1 µm or 0.0001°]	
		P-AXIS-00169	
	%3:	Limit value [0.1 µm or 0.0001°]	
		P-AXIS-00168	
	%4:	Corrected value [0.1 µm or 0.0001°]	
		P-AXIS-00169	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110268

Inadmissible value for time base at normalisation of velocity.			
Description	The specified value for the time base is invalid.		
Response	Class	3	Abort
Solution	Class	7	Correct parameter list of axis.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		P-AXIS-00207	
	%3:	Limit value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110269

Inadmissible value of numerator at velocity resolution.			
Description	The specified value for velocity resolution (numerator) is invalid.		
Response	Class	3	Abort
Solution	Class	7	Correct parameter list of axis.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00206	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110270

Inadmissible value of denominator at velocity resolution.			
Description	The specified value for velocity resolution (denominator) is invalid.		
Response	Class	3	Abort
Solution	Class	7	Correct parameter list of axis.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00205	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110271

Inadmissible mode for output of position setpoint.			
Description	The mode (drive-dependent, linear, modulo) for using the position setpoint is specified incorrectly.		
Response	Class	3	Abort
Solution	Class	7	Correct the value in axis parameter list.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Incorrect value [-]
		P-AXIS-00123
	%3:	Limit value [-]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110272

Inadmissible mode for input of actual position value.			
Description	The mode (drive-dependent, linear, modulo) for using actual position value is specified incorrectly.		
Response	Class	3	Abort
Solution	Class	1	Correct the value in axis parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00122	
	%3:	Limit value [-]	
	%4:	Limit value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110273

Invalid value for P-AXIS-00092, value will be corrected.			
Description	The number of increments per revolution is specified as 0.		
Response	Class	1	Parameter is set to 1.
Solution	Class	1	Set correct parameter.
Parameter	%1:	Logical axis number [-]	
		see P-AXIS-00016	
	%2:	Incorrect value [increment/revolution]	
		see P-AXIS-00092	
	%3:	Corrected value [increment/revolution]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110274

Value for parameter P-AXIS-00065 is greater than permitted maximum value.			
Description	Value specifies the resolution of the position controller for PROFIBUS drives.		
Response	Class	1	Value is limited to maximum value.
Solution	Class	1	Correct parameter list of axis.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Incorrect value [-]
		P-AXIS-00065
	%3:	Limit value [-]
	%4:	Corrected value [-]
		P-AXIS-00065
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110275

Number of delay cycles is greater than permitted maximum value.			
Description	The number of delay cycles between actual positions and commanded positions is specified too high because of the runtime of the digital drive.		
Response	Class	3	Abort
Solution	Class	7	Correct the value in axis parameter list.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Limit value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110276

Invalid geometry ramp time.			
Description	The geometry ramp time P-AXIS-00199 was specified smaller than the minimum ramp time P-AXIS-00201.		
Response	Class	1	Geometry ramp time is limited to minimum value.
Solution	Class	1	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1 µs]	
		P-AXIS-00199	
	%3:	Limit value [1 µs]	
		P-AXIS-00201	
	%4:	Corrected value [1 µs]	
		P-AXIS-00199	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110277

After initialisation of one axis, it is not allow to work with default axis MDS.

Description	The CNC got an order to take the axis parameter data set from the standard configuration. This message is displayed when the assignment is made at a time when the CNC has already accepted its specific axis parameter data set for the axis. At that moment it's not allowed to overwrite the actual data set by the standard data set.		
Response	Class	3	Order is rejected.
Solution	Class	7	You have to check if it's necessary to interpret and take over the standard axis parameter data set at the chosen moment.
Parameter	%1:	Current value [-]	
		Number of the logical axis for which the order was received to interpret and use the standard axis parameter data set.	
Error type	-		

ID 110278

Changing digital drive type requires restart of the control.			
Description	An attempt was made to change the node type of a SERCOS bus node by updating the axis parameter list. Changing the node type of a SERCOS node requires a restart of the control.		
Response	Class	1	Error message output, old node type is retained.
Solution	Class	1	Restart control to change node type.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110281

Changing gantry master axis requires restart of the control.			
Description	The parameter P-AXIS-00070 containing the logical axis number of the lead master axis cannot be changed while the CNC kernel is running. The new parameter value is not transferred; the previously active value remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry of the logical axis number P-AXIS-00070 only in configuration mode or when the CNC is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00070.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00070.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110282

Changing the possibility for direct access of cam requires restart of the control.

Description	The parameter P-AXIS-00036 indicating direct access to cam signals via driver functions cannot be changed while the CNC kernel is running. The new parameter value is not transferred, the previously active value remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry P-AXIS-00036 only in configuration mode or when CNC is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00036.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00036.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110283

Changing the hardware cam id requires restart of the control.			
Description	The parameter P-AXIS-00037 indicating the unique hardware code for cam signals cannot be changed while the CNC kernel is running. The new hardware identification is not transferred; the previous active hardware identification remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry P-AXIS-00037 only in configuration mode or when the CNC is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00037.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00037.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110286

Changing the counter channel requires restart of the control.			
Description	The parameter P-AXIS-00041 defining the physical counter channel on I/O hardware cannot be changed while the CNC kernel is running. The new parameter value is not transferred, the previously active value remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry P-AXIS-00041 only in configuration mode or when the CNC is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00041.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00041.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110288

Changing the hardware counter id requires restart of the control.			
Description	The parameter P-AXIS-00042 indicating the unique hardware identification of the counter cannot be changed while the CNC kernel is running. The new hardware identification is not transferred; the previous active hardware identification remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry P-AXIS-00042 only in configuration mode or when the CNC is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00042.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00042.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110292

Changing the D/A channel requires restart of the control.			
Description	The parameter P-AXIS-00048 defining the D/A channel on I/O hardware cannot be changed while the CNC kernel is running. The new parameter value is not transferred, the previously active value remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry P-AXIS-00048 only in configuration mode or when the CNC is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00048.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00048.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110294

Changing the D/A hardware id requires restart of the control.			
Description	The parameter P-AXIS-00049 indicating the system-wide unique hardware identification for the D/A area cannot be changed while the CNC kernel is running. The new hardware identification is not transferred; the previous active hardware identification remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry P-AXIS-00049 only in configuration mode or when the CNC is disabled.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [-]
		Current value of P-AXIS-00049.
	%3:	Corrected value [-]
		Corrected value of P-AXIS-00049.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110297

Changing the hardware number requires restart of the control.			
Description	The parameter P-AXIS-00136, indicating the number of the hardware-specific parameter set cannot be changed while the CNC kernel is running. The modified number is not transferred; the previous number remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the sign P-AXIS-00136 in configuration mode or when the CNC is inactive.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00136.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00136.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110298

Changing sign of actual value requires restart of the control.			
Description	The parameter P-AXIS-00230 indicating the sign of actual value cannot be changed while the CNC kernel is running. The modified sign value is not transferred; the previous sign value remains valid.		
Response	Class	0	None.
Solution	Class	0	Change the sign P-AXIS-00230 in configuration mode or when the CNC is inactive.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		Current value of P-AXIS-00230.	
	%3:		
		Corrected value of P-AXIS-00230.	
Error type	-		

ID 110299

Changing sign of the control value requires restart of the control.	
Description	The parameter P-AXIS-00231 indicating the sign of a manipulated variable cannot be changed while the CNC kernel is running. The modified sign value is not transferred; the previous sign value remains valid.

Response	Class	0	None.
Solution	Class	0	Change the sign P-AXIS-00231 in configuration mode or when the CNC is inactive.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		Current value of P-AXIS-00231.	
	%3:		
		Corrected value of P-AXIS-00231.	
Error type	-		

ID 110300

Logical axis number in the default axis parameter list must be 0.			
Description	The logical number (P-AXIS-00016) in the default axis parameter list must be 0. If the parameter is not specified in the list, it is assigned the value 0.		
Response	Class	3	Abort current job. Start-up is aborted.
Solution	Class	7	Check P-AXIS-00016. Remove the entry
Parameter	%1:	Error value [-]	
		P-AXIS-00016	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110301

Change of resolution numerator requires restart of control.			
Description	The parameter P-AXIS-00234 for distance resolution numerator cannot be changed while the CNC kernel is running. The new numerator value is not transferred; the previous numerator value remains valid.		
Response	Class	0	None.
Solution	Class	0	Change numerator P-AXIS-00234 in configuration mode or when the CNC is inactive.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		Current value of P-AXIS-00234.	
	%3:		
		Corrected value of P-AXIS-00234.	
Error type	-		

ID 110302

Change of resolution denominator requires restart of control.			
Description	The parameter P-AXIS-00233 for the denominator of distance resolution cannot be changed while the CNC kernel is running. The new denominator value is not transferred; the previous denominator value remains valid.		
Response	Class	0	None.

Solution	Class	0	Change denominator P-AXIS-00233 in configuration mode or when the CNC is inactive.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		Current value of P-AXIS-00233.	
	%3:		
		Corrected value of P-AXIS-00233.	
Error type	-		

ID 110303

The list contains an unknown element.			
Description	During interpretation of a list, an unknown list element is detected. More information about type of list, unknown element etc. you can get inspecting the additional data in the message.		
Response	Class	1	Interpretation of list is continued.
Solution	Class	1	Remove or modify the unknown list element in the corresponding list.
Error type	-		

ID 110304

Request for spindle interface, but axis is not a spindle.			
Description	A spindle is defined in the channel parameter list (SDA parameter) and the logical axis number for the spindle is parametrised by parameter P-CHAN-00036. In the axis configuration list with the corresponding logical axis number (P-AXIS-00016), the axis type (P-AXIS-00018) is not the value for a spindle (0x0004).		
Response	Class	2	CNC start-up is continued but no axes are available in the channel where the spindle was parametrised.
Solution	Class	7	When the axis with this logical axis number is a spindle, set the value for the axis type P-AXIS-00018 to the value for a spindle (0x0004). If the spindle has a different logical axis number, change the parameter P-CHAN-00036 .
Parameter	%1:	Expected value [-]	
		The parameter P-AXIS-00018 in the axis configuration list must have this value if the axis is to be configured as a spindle.	
	%2:	Incorrect value [-]	
		Current value of parameter P-AXIS-00018 in the axis configuration list.	
	%3:	Logical axis number [-]	
		The parameter P-AXIS-00016 in the axis configuration list where the parameter P-AXIS-00018 has the invalid value.	
Error type	3, Error message from communication.		

ID 110305

Changing P-AXIS-00065 requires restart of the control.

Description	The parameter P-AXIS-00065 defining the conversion factor between Profibus interface and CNC internal format cannot be changed while the CNC kernel is running. The new conversion factor is not transferred; the previous active factor remains valid.		
Response	Class	1	None.
Solution	Class	1	Change the entry P-AXIS-00065 only in configuration mode or when the CNC is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00065.	
	%3:	Corrected value [-]	
		Current value of P-AXIS-00065.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110306

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 110307

Hardware description with that index not present.			
Description	The default hardware description P-AXIS-00054 is not known.		
Response	Class	3	Hardware number is set to the highest known hardware description.
Solution	Class	7	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		Gear number	
	%3:	Current value [-]	
		P-AXIS-00054	
	%4:	Limit value [-]	
		Highest known hardware description	
	%5:	Corrected value [-]	
		P-AXIS-00054	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110308

Modulo difference is greater than the permissible maximum value.			
Description	The difference between the upper P-AXIS-00126 and lower modulo limits P-AXIS-00127 exceeds the maximum 32-bit range. The modulo range for rotary axes or spindles must be smaller than the maximum 32-bit range (2147483647).		
Response	Class	1	The upper and lower modulo limits are initialised by a default value.

Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		P-AXIS-00126 - P-AXIS-00127	
	%3:	Limit value [0.1 µm or 0.0001°]	
	%4:	Corrected value [0.1 µm or 0.0001°]	
		P-AXIS-00126	
	%5:	Corrected value [0.1 µm or 0.0001°]	
		P-AXIS-00127	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110309

Factor for speed monitoring is greater than the permissible maximum value.			
Description	The specified factor P-AXIS-00217 for the speed monitoring tolerance band is greater than 100%.		
Response	Class	1	Value is initialised by a default.
Solution	Class	1	Correct the value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		P-AXIS-00217	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
		P-AXIS-00217	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110310

Modulo difference is less than or equal to 0 after weighting with distance resolution.			
Description	The distance between the upper and lower modulo limits weighted by the resolution is too small.		
Response	Class	1	Parameter is limited to maximum value range.
Solution	Class	1	Correct modulo limits or check current resolution.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Incorrect value [-]	
		Difference between upper and lower modulo limits weighted with resolution (P-AXIS-00233, P-AXIS-00234)	
	%3:	Corrected value [-]	
		Corrected to minimum difference between upper and lower modulo limits	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Lower modulo limit P-AXIS-00127	
	%5:	Upper limit value [0.1 µm or 0.0001°]	
		Upper modulo limit P-AXIS-00126	

Error type	-

ID 110311

Reduced maximum velocity P-AXIS-00214 is greater than the maximum axis velocity.			
Description	The reduced maximum velocity for G01 P-AXIS-00214 was specified greater than the maximum axis velocity.		
Response	Class	1	The velocity is assigned the maximum axis velocity.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00214	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00212	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00214	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110312

Invalid type for position lag monitoring selected.			
Description	The selected type of position lag monitoring P-AXIS-00172 is unknown.		
Response	Class	1	Position lag monitoring is deactivated (type = 0)
Solution	Class	1	Correct value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00172	
	%3:	Corrected value [-]	
		P-AXIS-00172	
	%4:	Current value [-]	
		Gear stage	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110316

Reduced maximum velocity P-AXIS-00155 is greater than the maximum axis velocity.			
Description	The reduced maximum velocity for G00 motions P-AXIS-00155 was specified higher than the maximum axis velocity.		
Response	Class	1	The velocity is assigned the maximum axis velocity.
Solution	Class	1	Correct the value in the parameter list.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1µm/s or 0.001°/s]
		P-AXIS-00155
	%3:	Limit value [1µm/s or 0.001°/s]
		P-AXIS-00212
	%4:	Corrected value [1µm/s or 0.001°/s]
		P-AXIS-00155
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110317

SGN32 Underflow: Negative software limit switch - tolerance less than MIN_SGN32.			
Description	The negative software limit switch (SLS) P-AXIS-00177 is out of range. The software limit switch including the tolerance band P-AXIS-00179 must be greater than –2147483648 (minimum signed 4-byte numeral).		
Response	Class	1	Value is set to minimum value.
Solution	Class	1	Correct value in parameter list.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [0.1 µm or 0.0001°]
		SLS P-AXIS-00177
	%3:	Current value [0.1 µm or 0.0001°]
		Tolerance P-AXIS-00179
	%4:	Corrected value [0.1 µm or 0.0001°]
		P-AXIS-00177
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110318

SGN32 Overflow: Positive software limit switch + tolerance greater than MAX_SGN32.			
Description	The positive software limit switch (SLS) P-AXIS-00178 is out of range. The software limit switch including the tolerance band P-AXIS-00179 must be greater than 2147483647 (maximum signed 4-byte numeral).		
Response	Class	1	Value is set to minimum value.
Solution	Class	1	Correct value in parameter list.

Parameter	%1:	Logical axis number [0.1 µm or 0.0001°]
		P-AXIS-00016
	%2:	Current value [0.1 µm or 0.0001°]
		SLS P-AXIS-00178
	%3:	Current value [0.1 µm or 0.0001°]
		Tolerance P-AXIS-00179
	%4:	Corrected value [0.1 µm or 0.0001°]
		P-AXIS-00178
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110319

Unknown entry in configuration of input telegram.			
Description	At CNC kernel start-up, cyclic drive data are assigned to the internal working data of the CNC in the axis parameter list. This cyclic drive data contains data from the drive to the CNC and vice versa. An unknown entry was detected in this section of the axis parameter list. In the following example, either value 99999 in entry antr.terminal.in[1].signal_nr or value XXXXXX in entry antr.terminal.in[1].nc_ref is invalid. Example: antr.terminal.in[1].signal_nr 99999 antr.terminal.in[1].signal_len 2 antr.terminal.in[1].nc_ref XXXXXX		
Response	Class	3	Error message output, invalid entry is ignored.
Solution	Class	6	Correct invalid entry in the parameter list.
Parameter	%1:	Current value [-]	
		P-AXIS-00020	
	%2:	Logical axis number [-]	
		P-AXIS-00016	
	%3:	Current value [-]	
		Index of invalid entry, in the example above value 1.	
	%4:	Current value [-]	
		Signal number of invalid entry, in the example above 99999.	
	%5:	Current value [-]	
		Signal name of invalid entry, in the example above XXXXXX.	
Error type	-		

ID 110320

Unknown entry in configuration of output telegram.			
Description	At CNC kernel start-up, cyclic drive data are assigned to the internal working data of the CNC in the axis parameter list. This cyclic drive data contains data from the drive to the CNC and vice versa. An unknown entry was detected in this section of the axis parameter list. In the following example, either value 99999 in entry antr.terminal.out[1].signal_nr or value XXXXXX in entry antr.terminal.out[1].nc_ref is invalid. Example: antr.terminal.out[1].signal_nr 99999 antr.terminal.out[1].signal_len 2 antr.terminal.out[1].nc_ref XXXXXX		
Response	Class	3	Error message output, invalid entry is ignored.
Solution	Class	6	Correct invalid entry in the parameter list.

Parameter	%1:	Current value [-]
		P-AXIS-00020
	%2:	Current value [-]
		P-AXIS-00016
	%3:	Logical axis number [-]
		Index of the invalid entry, in the above example 1.
	%4:	Current value [-]
		Signal number of the incorrect entry, in the above example 99999.
	%5:	Current value [-]
		Signal name of the incorrect entry, in the above example XXXXXX.
Error type	-	

ID 110321

Emergency deceleration must not be 0.			
Description	The maximum acceleration P-AXIS-00003 of the axis was set to 0. This acceleration must be specified greater than 0.		
Response	Class	1	Emergency stop deceleration is assigned with maximum axis acceleration.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00003	
	%3:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00003	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110322

Emergency deceleration is greater than maximum deceleration.			
Description	Emergency stop deceleration P-AXIS-00003 of the axis exceeds maximum axis acceleration.		
Response	Class	1	Limit emergency stop deceleration to maximum axis acceleration.
Solution	Class	1	Correct the value in the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00003	
	%3:	Limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00003	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110323

Terminal KL5111 cannot be used for measuring.			
Description	For a terminal axis with the KL5111 encoder terminal, the parameter P-AXIS-00118 was set to 1. It is not possible to measure with the terminal since it has no measuring input.		
Response	Class	1	Error message output, P-AXIS-00118 is set to zero (nil).
Solution	Class	1	Parameter P-AXIS-00118 at zero (nil).
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Value of P-AXIS-00118	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00118	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110324

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 110326

Axis for duplication (clone) is internally unknown.			
Description	For easier configuration, an axis could be a copy (clone) of another axis. This means, when the axis is initialised, the clone just uses the data of its master as template. The axis parameter P-AXIS-00040 should specify the logical axis number whose axis parameter data record is to be cloned. This error is output since there is no logical axis with the corresponding axis number.		
Response	Class	3	Job processing aborted
Solution	Class	1	Check the value of the axis parameter P-AXIS-00040.
Parameter	%1:	Logical axis number [-]	
		The logical axis with this number should be the result of the cloning operation of the axis parameter data record (P-AXIS-00016).	
	%2:	Incorrect value [-]	
		This is the logical axis number of the master axis whose axis parameters are to be used as the template for cloning but it does not exist (P-AXIS-00040).	
Error type	-		

ID 110327

Invalid number for SERCOS real-time bit [1... 2].			
Description	A value in the axis parameter list was specified for the parameter P-AXIS-00060 but it is outside the permissible value range.		
Response	Class	1	Warning output, automatic correction of the value.

Solution	Class	1	Enter valid value for P-AXIS-00060 in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number.	
	%2:	Error value [-]	
		Incorrect value of P-AXIS-00060.	
	%3:	Upper limit value [-]	
		Maximum permissible value for P-AXIS-00060.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00060.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110328

Percentage value P-AXIS-00289 for camming/gearing is greater than 100%.			
Description	The value of the percentage window for the “Speed reached” acknowledgement P-AXIS-00289 with camming/gearing is greater than 100%. The dimension for this parameter is 0.1%.		
Response	Class	1	Warning output. Value of P-AXIS-00289 is automatically set to 100%.
Solution	Class	1	Set value of P-AXIS-00289 to smaller or equal to 100%.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current value of P-AXIS-00289	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00289.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110329

Timeout value P-AXIS-00291 is smaller than permissible value of P-AXIS-00290.			
Description	The value of P-AXIS-00291 is smaller than the value of P-AXIS-00290. For the “Speed reached” acknowledgement, the axis time P-AXIS-00290 must be permanently within the percentage window P-AXIS-00289. P-AXIS-00291 is the timeout for the “Speed reached” acknowledgement.		
Response	Class	1	Warning output. Value of P-AXIS-00290 is automatically set to P-AXIS-00291.
Solution	Class	1	Set value of P-AXIS-00290 smaller than P-AXIS-00291.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00290.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00290.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110330

Deceleration step 1 greater than the maximum axis acceleration.			
Description	The deceleration P-AXIS-00282 of the first step of the linear slope (not jerk limited) was specified greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit deceleration to maximum value.
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00282	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00282	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110331

Deceleration step 2 greater than the maximum axis acceleration.			
Description	The deceleration P-AXIS-00283 of the second step of the linear slope (not jerk limited) was specified greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit deceleration to maximum value.
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00283	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00283	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110332

Feedhold acceleration is greater than permissible maximum axis acceleration.			
Description	The feedhold acceleration P-AXIS-00024 of the linear slope was specified as greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct the value in the parameter list

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1mm/s^2 or 1°/s^2]
		P-AXIS-00024
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]
		P-AXIS-00008
	%4:	Corrected value [1mm/s^2 or 1°/s^2]
		P-AXIS-00024
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110333

Ramp time for feedhold acceleration is too small.			
Description	The time period P-AXIS-00081 (ramp time) for up-gradation and down-gradation of the maximum feedhold acceleration P-AXIS-00053 of the non-jerk limited slope was specified as less than the minimum ramp time P-AXIS-00201.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1 µs]	
		P-AXIS-00081	
	%3:	Lower limit value [1 µs]	
		P-AXIS-00201	
	%4:	Corrected value [1 µs]	
		P-AXIS-00081	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110334

Reduced speed zone: upper range limit is greater than upper software limit switch.			
Description	The positive range limit for Reduced Speed in the security zones P-AXIS-00085, P-AXIS-00097 is greater than the positive software limit switch. The positive range limit is set to the positive software limit switch position.		
Response	Class	1	None.
Solution	Class	1	Select positive range limit smaller than positive software limit switch.
Parameter	%1:	Logical axis number [0.1 µm or 0.0001°]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current positive range limit.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Positive software limit switch P-AXIS-00178.	
	%4:	Corrected value [0.1 µm or 0.0001°]	
		Corrected positive range limit.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110335

Reduced speed zone: lower range limit is less than lower software limit switch.			
Description	The negative range limit for Reduced Speed in the security zones P-AXIS-00093, P-AXIS-00105 is smaller than the negative software limit switch. The negative range limit is set to the negative software limit switch position.		
Response	Class	1	None.
Solution	Class	1	Select negative range limit greater than negative software limit switch.
Parameter	%1:	Logical axis number [0.1 µm or 0.0001°]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current negative range limit.	
	%3:	Current value [0.1 µm or 0.0001°]	
		Negative software limit switch P-AXIS-00177.	
	%4:	Lower limit value [0.1 µm or 0.0001°]	
		Corrected negative range limit.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110336

Limit deceleration step 1 greater than maximum axis acceleration.			
Description	The current limit delay P-AXIS-00280 of the first step of the linear slope (non jerk limited) was specified to be greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit deceleration to maximum value.
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00280	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00280	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110337

Limit deceleration step 2 greater than maximum axis acceleration.			
Description	The current limit delay P-AXIS-00281 of the second step of the linear slope (non jerk limited) was specified to be greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit deceleration to maximum value.
Solution	Class	1	Correct the value in the parameter list

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1mm/s^2 or 1°/s^2]
		P-AXIS-00281
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]
		P-AXIS-00008
	%4:	Corrected value [1mm/s^2 or 1°/s^2]
		P-AXIS-00281
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110339

Terminal KL5001 cannot be used for measuring.			
Description	A terminal of the KL5001 type cannot be used for measuring since it has no external latch input. The measurement function is disabled for this axis (see P-AXIS-00118).		
Response	Class	1	Correction of parameter P-AXIS-00118
Solution	Class	1	Correct parameter P-AXIS-00118 or use other terminal type
Parameter	%1:	Logical axis number [-]	
		For logical axis number of concerned axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Specified value of parameter P-AXIS-00118	
	%3:	Corrected value [-]	
		Corrected value for parameter P-AXIS-00118	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110340

Velocity dependent position loop gain for SERCOS or CANopen drives not possible.			
Description	Position control is generally executed in the drive with SERCOS or CANopen drives. In this case, the “velocity-dependent position controller gain” function in the controller cannot be used. It is deactivated for this axis.		
Response	Class	1	Correct parameter P-AXIS-00244
Solution	Class	1	Correct parameter P-AXIS-00244
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Corrected value [-]	
		Corrected value of parameter P-AXIS-00244	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110341

Velocity-dependent proportional factor: second velocity limit must be greater than the first velocity limit.			
Description	The second velocity limit P-AXIS-00248 of dynamic Kv adaptation is smaller than the first velocity limit P-AXIS-00246.		
Response	Class	1	Value is set slightly higher than the first velocity limit.
Solution	Class	1	Correct parameter list of axis.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [1µm/s or 0.001°/s]
		P-AXIS-00248
	%3:	Corrected value [1µm/s or 0.001°/s]
		P-AXIS-00248
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110342

Axis number for gantry master axis has to be 0.			
Description	The axis is marked as the master axis of a gantry combination in the parameter P-AXIS-00015. At the same time, this axis has an entry for a leading master axis P-AXIS-00070. This is not permitted.		
Response	Class	3	Abort current job.
Solution	Class	7	The entry of the logical axis number P-AXIS-00070 must be assigned 0 in the master axis of the gantry combination.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Logical axis number [-]	
		Current entry of the logical axis number for P-AXIS-00070.	
	%3:	Corrected value [-]	
		New entry for P-AXIS-00070	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110343

If global feedforward control is switched off, P-AXIS-00223 must be set to 0.			
Description	If feedforward control was disabled by P-AXIS-00256, e.g. in the default axis parameter list, feedforward control must also be disabled in the individual setting of the axis P-AXIS-00223.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify the corresponding parameter
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		P-AXIS-00223	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110344

Drive-controlled measurement only possible with digital drives.

Description	The use of parameter P-AXIS-00269 requires a digital drive. Parameter P-AXIS-00269 is set to the default value.		
Response	Class	1	None.
Solution	Class	1	Only use parameter P-AXIS-00269 for digital drives corresponding to the set drive type P-AXIS-00020.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [-]	
		Current parameter P-AXIS-00269.	
	%3:	Drive type [-]	
		Current drive type P-AXIS-00020.	
	%4:	Corrected value [-]	
		Corrected parameter P-AXIS-00269.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110345 - 110347

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 110348

Terminal KL2521 can not be used for probing.			
Description	A terminal of the KL2521 type cannot be used for measuring since it has no external latch input. The measurement function is disabled for this axis (see P-AXIS-00118).		
Response	Class	1	Correction of parameter P-AXIS-00118
Solution	Class	1	Correct parameter P-AXIS-00118 or use other terminal type
Parameter	%1:	Logical axis number [-]	
		For logical axis number of concerned axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Specified value of parameter P-AXIS-00118	
	%3:	Corrected value [-]	
		Corrected value for parameter P-AXIS-00118	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110349

Zero pulse cannot be homed with terminal KL2521.			
Description	A terminal of the KL2521 type cannot be used to execute homing with zero pulse search since it has no external latch input. Therefore, no zero pulse search is carried out for this axis during homing (see P-AXIS-00084).		
Response	Class	1	Correct parameter P-AXIS-00084
Solution	Class	1	Correct parameter P-AXIS-00084 or use other terminal type

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [-]
		Specified value of parameter P-AXIS-00084
	%3:	Corrected value [-]
		Corrected value of parameter P-AXIS-00084
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110350

Maximum velocity for axis which is not homed was set to 0.			
Description	The maximum velocity P-AXIS-00268 for the non-homed axis is 0.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Correct the parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00268	
	%3:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00268	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110351

Maximum velocity for axis which is not homed is greater than maximum axis velocity.			
Description	The maximum velocity P-AXIS-00268 for the non-homed axis is greater than the maximum axis velocity.		
Response	Class	1	Value is limited to maximum velocity.
Solution	Class	1	Correct the parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00268	
	%3:	Limit value [1µm/s or 0.001°/s]	
		P-AXIS-00212	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		P-AXIS-00268	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110352

Axis mode not permitted for spindle.			
Description	The axis mode P-AXIS-00015 is not set to MODULO (= 4) for the spindle even though this is required.		
Response	Class	1	Value is set to MODULO
Solution	Class	1	Correct the value in parameter list.

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016
	%2:	Current value [-]
		P-AXIS-00015
	%3:	Corrected value [-]
		P-AXIS-00015
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110353

Unknown axis mode corrected.			
Description	The specified axis mode P-AXIS-00015 is unknown. Following values are possible: 1 : LINEAR 2 : optimised positioning 4 : MODULO		
Response	Class	1	Value is set to MODULO for spindles or rotary axes. Otherwise, the value is set to LINEAR
Solution	Class	1	Correct the value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00015	
	%3:	Corrected value [-]	
		P-AXIS-00015	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110354

Permanent activation and deactivation were both parameterised for axis feedfoward control.			
Description	The parameterisation of the axis specified that feedforward control should be both permanently activated (P-AXIS-00255) and deactivated (P-AXIS-00256).		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00255	
	%3:	Current value [-]	
		P-AXIS-00256	
	%4:	Corrected value [-]	
		P-AXIS-00255	
Error type	2, Error message on data transfer of parameter lists to controller.		
	Feedforward control is permanently deactivated by default.		

ID 110355

Function feedforward control not enabled.

Description	An attempt was made to enable feedforward control permanently, even though the function is not enabled in the current version.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00255	
	%3:	Limit value [-]	
		Maximum available number of interpolation points to calculate feedforward control in this version	
	%4:	Expected value [-]	
		Required number of interpolation points to calculate feedforward control.	
	%5:	Corrected value [-]	
		P-AXIS-00255	
Error type	2, Error message on data transfer of parameter lists to controller.		
	Feedforward control is not permanently enabled.		

ID 110356

Feedforward control permanently enabled but no mode selected.			
Description	Feedforward control was permanently switched on without specifying the feedforward control type. Velocity feedforward control or acceleration feedforward control of type P-AXIS-00233 must at least be specified.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00255	
	%3:	Current value [-]	
		Feedforward control type P-AXIS-00233	
	%4:	Corrected value [-]	
		P-AXIS-00255	
Error type	2, Error message on data transfer of parameter lists to controller.		
	Feedforward control is not permanently enabled.		

ID 110357 / 110358

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 110359

Feedhold acceleration is greater than maximum permissible axis acceleration.

Description	Feedhold acceleration P-AXIS-00259 with active manual mode was specified to be greater than the maximum axis acceleration P-AXIS-00008. Feedhold acceleration for manual mode is set to P-AXIS-00008.		
Response	Class	1	None.
Solution	Class	1	Do not select feedhold acceleration greater than P-AXIS-00008.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Current setting of P-AXIS-00259.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Upper limit value for acceleration P-AXIS-00008.	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		Limited feedhold acceleration P-AXIS-00259.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110360

The value of the parameter 'op_mode_for_velocity_control' is invalid.			
Description	The specified SERCOS mode for automatic switching to velocity controlled mode is not valid.		
Response	Class	1	
Solution	Class	1	Check P-AXIS-00264
Parameter	%1:	Logical axis number [-]	
		see P-AXIS-00016	
	%2:	Incorrect value [-]	
		see P-AXIS-00264	
	%3:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110361

The characteristic frequency of axis filter exceeds minimum limit.			
Description	The characteristic frequency (see P-AXIS-00067) of the specified axis filter is too low. The interpretation of the characteristic frequency is dependent on the filter type: • Low-pass filter: Beginning of cut-off frequency range (ideal filter) • High-pass filters: Beginning of cut-off frequency range (ideal filter) • Band-pass and band-stop filters: middle frequency		
Response	Class	1	Correct parameter P-AXIS-00067 to lowest permissible frequency
Solution	Class	1	Increase the characteristic frequency P-AXIS-00067 of the axis filter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Number of filter affected	
	%3:	Current value [-]	
		Specified value for the characteristic filter frequency P-AXIS-00067 in Hertz.	
	%4:	Corrected value [-]	
		Corrected value for characteristic filter frequency P-AXIS-00067 in Hertz.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110362

Number of cycles for coupling/decoupling temperature compensation exceeds maximum limit.			
Description	The specified number of cycles for coupling(decoupling the temperature compensation value is set by the parameter *lr_param.temp_comp_n_cycles* (P-AXIS-00275). The specified value is too high. See [FCT-C5// Temperature compensation]		
Response	Class	1	Parameter is limited to maximum value range.
Solution	Class	1	Correct parameter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Error value [-]	
		Parameterised value of P-AXIS-00275	
	%3:	Upper limit value [-]	
		Maximum value for P-AXIS-00275	
	%4:	Corrected value [-]	
		New value for P-AXIS-00275	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110363

Unknown mode for setting the reference position.			
Description	The parameter *kenngr.set_refpos_mode* (P-AXIS-00278) is assigned an invalid mode. The parameter is assigned the default value for the mode.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00278. Use a valid mode.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
	%2:	Current value [-]	
		Current parameter P-AXIS-00278.	
	%3:	Corrected value [-]	
		Corrected parameter P-AXIS-00278.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110364

Maximum distance is smaller than minimum distance during homing to fixed stop.			
Description	The value for *kenngr.homing.torq_max_distance* (P-AXIS-00345) may not be smaller than the value for *kenngr.homing.torq_min_distance* (P-AXIS-00344).		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and modify the entered values for P-AXIS-00345 and P-AXIS-00344. P-AXIS-00345 must be greater than P-AXIS-00344.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [0.1 10^-3 mm or ø]
		Parameterised value of *kenngr.homing.torq_max_distance* (P-AXIS-00345).
	%3:	Current value [0.1 10^-3 mm or ø]
		Parameterised value of *kenngr.homing.torq_min_distance* (P-AXIS-00344).
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110365

Minimum distance during homing to fixed stop is less than 0.			
Description	The value for *kenngr.homing.torq_min_distance* (P-AXIS-00344) may not be less than 0.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00344 is set to 0.
Solution	Class	1	Check and modify P-AXIS-00344. Set a value greater than zero.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [0.1 10^-3 mm or ø]
		Parameterised value of *kenngr.homing.torq_min_distance* (P-AXIS-00344).
	%3:	Corrected value [0.1 10^-3 mm or ø]
		Corrected value of *kenngr.homing.torq_min_distance* (P-AXIS-00344).
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110366

Maximum distance during homing to fixed stop is less than 0.			
Description	The value for *kenngr.homing.torq_max_distance* (P-AXIS-00345) may not be less than 0.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00345 is set to 0.
Solution	Class	1	Check and modify P-AXIS-00345. Set a value greater than zero.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [0.1 10^-3 mm or ø]
		Parameterised value of *kenngr.homing.torq_max_distance* (P-AXIS-00345).
	%3:	Corrected value [0.1 10^-3 mm or ø]
		Corrected value of *kenngr.homing.torq_max_distance* (P-AXIS-00345).
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110367

Velocity limit during homing to a fixed stop is less than 0.			
Description	The value for *kenngr.homing.torq_detect_velocity_limit* (P-AXIS-00347) may not be less than 0.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00347 is set to 0.

Solution	Class	1	Check and modify P-AXIS-00347. Set a value greater than zero.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [0.1%]	
		Parameterised value of *kenngr.homing.torq_detect_velocity_limit* (P-AXIS-00347).	
	%3:	Corrected value [0.1%]	
		Corrected value of *kenngr.homing.torq_detect_velocity_limit* (P-AXIS-00347).	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110368

Retraction distance during homing to a fixed stop less than 0.			
Description	The value for *kenngr.homing.torq_retraction_distance* (P-AXIS-00348) may not be less than 0.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00348 is set to 0.
Solution	Class	1	Check and modify P-AXIS-00348. Set a value greater than 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Parameterised value of *kenngr.homing.torq_retraction_distance* (P-AXIS-00348).	
	%3:	Corrected value [0.1 10^-3 mm or ø]	
		Corrected value of *kenngr.homing.torq_retraction_distance* (P-AXIS-00348).	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110369

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 110371

Torque homing with this drive type not possible.			
Description	The homing type "TORQ" was entered for the axis in axis parameter P-AXIS-00299, although the drive type used does not support this homing type.		
Response	Class	1	Warning output, homing type is set to the default homing type of each drive (displayed in value 5 of the error message).
Solution	Class	1	Check and modify the drive type.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Drive type [-]
		Drive type
	%3:	Error value [-]
		Parameterised value of P-AXIS-00299.
	%4:	Error value [-]
		Meaning of the value of P-AXIS-00299.
	%5:	Corrected value [-]
		Corrected value of P-AXIS-00299.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110372

Parameter for activation of modulo compensation ineffective, parameter set internally.			
Description	Setting the parameter P-AXIS-00120 for activating modulo compensation has no effect because it is set internally.		
Response	Class	1	-
Solution	Class	1	-
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 110373

Permissible value range is exceeded.			
Description	When converting a position to increments with the set resolution of the measurement system, the maximum value range was exceeded.		
Response	Class	1	Parameter is limited to maximum value range.
Solution	Class	1	Correct parameter. Check resolution
Parameter	%1:	Upper limit value [increments]	
	%2:	Logical axis number [-]	
		P-AXIS-00016	
	%3:	Incorrect value [increments]	
		Incorrect value including resolution factor	
	%4:	Corrected value [increments]	
	%5:	Current value [increments/0.1µm]	
		Set resolution, see. P-AXIS-00233, P-AXIS-00234	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110374

Command position for DSE corrected to MODE_ACC_LINEAR.

Description	With DSE, P-AXIS-00122 and P-AXIS-00123 must be set to MODE_ACC_LINEAR (linear handling of values) to scale actual and command positions. Only then can display data match in actual and command positions and position lag.		
Response	Class	1	Warning output. P-AXIS-00123 is set to MODE_ACC_LINEAR.
Solution	Class	1	Set value for P-AXIS-00123 to MODE_ACC_LINEAR.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current value for P-AXIS-00123.	
	%3:	Corrected value [-]	
		Corrected value for P-AXIS-00123.	
Error type	-		

ID 110376

Acceleration during homing to a fixed stop is greater than the maximum permissible axis acceleration.			
Description	The value of parameter *getriebe[i].homing.torq_move_acceleration* (P-AXIS-00334) is greater than the maximum permissible axis acceleration P-AXIS-00008.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00334 is limited to the value of P-AXIS-00008.
Solution	Class	1	Check and modify P-AXIS-00334.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [mm/s² or °/s²]	
		Current value for *getriebe[i].homing.torq_move_acceleration* (P-AXIS-00334).	
	%3:	Limit value [mm/s² or °/s²]	
		Maximum permissible axis acceleration P-AXIS-00008.	
	%4:	Corrected value [mm/s² or °/s²]	
		Corrected value for *getriebe[i].homing.torq_move_acceleration* (P-AXIS-00334).	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110378

Torque limit value during homing to a fixed stop is greater than 100%.			
Description	The value of torq_move_torque_limit for the maximum torque is greater than 1000, i.e. it is greater than 100%.		
Response	Class	1	Warning output. The value of torq_move_torque_limit is set to 1000 (100%) automatically.
Solution	Class	1	Choose value of torq_move_torque_limit less or equal than 1000.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1%]	
		Current value of torq_move_torque_limit.	
	%3:	Limit value [0.1%]	
		1000 (100%)	
	%4:	Corrected value [0.1%]	
		Corrected value of torq_move_torque_limit.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110379

Torque limit for detection during homing to a fixed stop is greater than 100%.			
Description	The value of torq_detect_torque_limit is greater than 1000, i.e. it is greater than 100%.		
Response	Class	1	Warning output. The value of torq_detect_torque_limit is set to 1000 (100%) automatically.
Solution	Class	1	Choose value of torq_move_torque_limit less or equal than 1000.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [0.1%]	
		Current value of torq_detect_torque_limit.	
	%3:	Limit value [0.1%]	
		1000 (100%)	
	%4:	Corrected value [0.1%]	
		Corrected value of torq_detect_torque_limit.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110380

Numerator of acceleration feed forward factor may not be 0.			
Description	The numerator of the factor for normalising the acceleration feedforward control *vorsteuer.vs_a_faktor* (P-AXIS-00225) was assigned 0. The associated denominator is assigned by *vorsteuer.vs_a_nenner* (P-AXIS-00226).		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00225.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Current value [-]	
		Parameterised value for *vorsteuer.vs_a_faktor* (P-AXIS-00225)	
	%3:	Corrected value [-]	
		New value for P-AXIS-00225	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110381

Denominator of acceleration feed forward factor may not be 0.			
Description	The denominator of the factor for normalising acceleration feedforward control *vorsteuer.vs_a_nenner* (P-AXIS-00226) was assigned 0. The associated numerator is assigned by *vorsteuer.vs_a_fakor* (P-AXIS-00225).		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of concerned axis
	%2:	Current value [-]
		Parameterised value for *vorsteuer.vs_a_nenner* (P-AXIS-00226)
	%3:	Corrected value [-]
		New value for P-AXIS-00226
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110382

The numerator for the resolution of the D/A converter is 0.			
Description	The resolution of the digital/analogue converter (denominator) was set to 0. This value is invalid.		
Response	Class	1	Value is initialised with default value.
Solution	Class	1	Correct parameter list of axis.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00129	
	%3:	Corrected value [-]	
		P-AXIS-00129	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110384

This drive type does not support the parametrised homing type.			
Description	The homing type (e.g. drive-controlled homing) to be used for the axis is specified in the parameter P-AXIS-00299. The actual drive type, however, does not support the specified type of homing.		
Response	Class	1	Use the default setting (see P-AXIS-00299)
Solution	Class	1	Change the homing type P-AXIS-00299
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of axis affected, see P-AXIS-00020	
	%3:	Current value [-]	
		Axis operation mode, see P-AXIS-00320	
	%4:	Error value [-]	
		Specified homing type, see P-AXIS-00299	
	%5:	Corrected value [-]	
		Corrected homing type, see P-AXIS-00299	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110385

Acceleration minimum is smaller than minimum permissible value.

Description	Minimum acceleration limit P-AXIS-00010 of the curve-controlled acceleration was specified too high. This value must not be higher than the minimum acceleration.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1µm/s or 0.001°/s]	
		P-AXIS-00010	
	%3:	Limit value [1µm/s or 0.001°/s]	
	%4:	Incorrect value [1µm/s or 0.001°/s]	
		P-AXIS-00010	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110386

Numerator for distance resolution is 0.			
Description	The numerator for the distance resolution P-AXIS-00234 of the measurement system was specified as zero. A value greater than zero must be specified. Distance resolution is specified as a factor P-AXIS-00234 / P-AXIS-00233 in increments per 0.1 µm or per 0.0001°.		
Response	Class	1	Initialise resolution with default value
Solution	Class	1	Correct value in parameter list
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00234	
	%3:	Corrected value [-]	
		P-AXIS-00234	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110388

Maximum path distance for homing of gantry slave axis is less than 0.			
Description	A value smaller than zero for P-AXIS-00284 was entered in the axis parameter list. The value is corrected automatically by adopting the set value.		
Response	Class	1	The set value is re-used.
Solution	Class	1	Check value of P-AXIS-00284.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current set value, see P-AXIS-00284	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Lower limit value for P-AXIS-00284 (0)	
	%4:	Corrected value [-]	
		Corrected value.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110389

Feedhold acceleration of jerk limiting slope is greater than maximum permissible axis acceleration.			
Description	The feedhold acceleration P-AXIS-00053 of the jerk-limited slope was specified to be greater than the maximum axis acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Correct value in parameter list.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00053	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		P-AXIS-00053	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110390

First entry for adjustment of position controller proportional gain was specified as 0.			
Description	The first proportional gain P-AXIS-00245 of the dynamic kv adjustment was specified as 0. This is an invalid value.		
Response	Class	1	Value is initialised with default value.
Solution	Class	1	Correct parameter list of axis.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00245	
	%3:	Corrected value [-]	
		P-AXIS-00245	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110391

Second entry for adjustment of position controller proportional gain was specified as 0.			
Description	The second proportional gain P-AXIS-00247 of the dynamic kv adjustment was specified as 0. This is an invalid value.		
Response	Class	1	Value is initialised with default value.
Solution	Class	1	Correct parameter list of axis.
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00247	
	%3:	Corrected value [-]	
		P-AXIS-00247	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110392

Number of cycles for driving out backlash at inversion of direction too great.			
Description	Different compensation tables for bilateral leadscrew error compensation (see FCT-C5//Leadscrew error compensation) are specified for the two axis motion directions. When the motion direction is reversed, the compensation value difference (backlash) is extended over several position controller cycles by means of a filter. The number of cycles is specified in parameter P-AXIS-00243. The specified value is, however, greater than the permissible maximum value.		
Response	Class	1	Correct the filter cycles to permissible maximum value
Solution	Class	1	Correct the value of parameter P-AXIS-00243
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Incorrect value [-]	
		Specified number of filter cycles, see P-AXIS-00243	
	%3:	Limit value [-]	
		Maximum permissible number of filter cycles, see P-AXIS-00243	
	%4:	Corrected value [-]	
		Corrected number of filter cycles, see P-AXIS-00243	
Error type	-		

ID 110393

Acceleration for homing is greater than maximum permissible axis acceleration.			
Description	The acceleration for homing P-AXIS-00285 with jerk-limited slope was specified as greater than the maximum permissible acceleration P-AXIS-00008.		
Response	Class	1	Limit acceleration to maximum value
Solution	Class	1	Set acceleration for homing smaller than maximum value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Current acceleration P-AXIS-00285.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible acceleration P-AXIS-00008.	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		Corrected acceleration P-AXIS-00285.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110394

Ramp time for homing is smaller than minimum permissible ramp time.			
Description	The ramp time for homing P-AXIS-00286 with jerk-limited slope was specified smaller than the minimum ramp time P-AXIS-00201.		
Response	Class	1	Limit ramp time to minimum value
Solution	Class	1	Correct the value in the parameter list

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis affected P-AXIS-00016.
	%2:	Current value [1 µs]
		Current ramp time P-AXIS-00286.
	%3:	Lower limit value [1 µs]
		Minimum permissible ramp time P-AXIS-00201.
	%4:	Corrected value [1 µs]
		Corrected ramp time P-AXIS-00286.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110397

Camming/gearing: Unknown type for mean value calculation.			
Description	The type for P-AXIS-00300 for mean value calculation is not permitted..		
Response	Class	1	Warning output. The type for P-AXIS-00300 is corrected to MV_NONE.
Solution	Class	1	Adjust type for P-AXIS-00300 accordingly.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00300.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00300.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110398

Camming/gearing: Camming/Gearing: Illegal number of values for means value calculation.			
Description	The number of values P-AXIS-00301 to be used for calculating the mean value of master actual value filtering is not permitted.		
Response	Class	1	Warning output. The value for P-AXIS-00301 is corrected automatically.
Solution	Class	1	Adjust the value for P-AXIS-00301 accordingly.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00301.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00301.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110399

Illegal measurement method. Only one method may be selected.			
Description	Invalid parameter setting for P-AXIS-00117 was assigned to the measurement. The parameter P-AXIS-00117 is assigned the default value.		
Response	Class	1	None.
Solution	Class	1	Only use parameter P-AXIS-00117 if P-AXIS-00115 is disabled.

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis affected P-AXIS-00016.
	%2:	Current value [-]
		Current parameter P-AXIS-00117.
	%3:	Current value [-]
		Current parameter P-AXIS-00115.
	%4:	Current value [-]
		Corrected parameter P-AXIS-00117.
	%5:	Corrected value [-]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110400

Illegal measurement method. Only one method may be selected.			
Description	Invalid parameter setting for P-AXIS-00117 was assigned to the measurement. The parameter P-AXIS-00117 is assigned the default value.		
Response	Class	1	None.
Solution	Class	1	Only use parameter P-AXIS-00117 if P-AXIS-00115 is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [-]	
		Current parameter P-AXIS-00117.	
	%3:	Current value [-]	
		Current parameter P-AXIS-00115.	
	%4:	Corrected value [-]	
		Corrected parameter P-AXIS-00117.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110401

Illegal measurement method. Only one method may be selected.			
Description	Invalid parameter setting for P-AXIS-00117 was assigned to the measurement. The parameter P-AXIS-00117 is assigned the default value.		
Response	Class	1	None.
Solution	Class	1	Only use parameter P-AXIS-00117 if P-AXIS-00115 is disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [-]	
		Current parameter P-AXIS-00269.	
	%3:	Current value [-]	
		Current parameter P-AXIS-00117.	
	%4:	Corrected value [-]	
		Corrected parameter P-AXIS-00117.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110402

Illegal measurement method. Only one method may be selected.			
Description	Invalid parameter setting for P-AXIS-00115 was assigned to the measurement. The parameter P-AXIS-00115 is assigned the default value.		
Response	Class	1	None.
Solution	Class	1	Only use parameter P-AXIS-00115 if P-AXIS-00269 is disabled.
Parameter	%1:		
		Logical axis number of axis affected P-AXIS-00016.	
	%2:		
		Current parameter P-AXIS-00269.	
	%3:	Current value [-]	
		Current parameter P-AXIS-00115.	
	%4:	Corrected value [-]	
		Corrected parameter P-AXIS-00115.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110403

The maximum velocity of slave axis may not be set to 0.			
Description	During velocity synchronisation (camming and gearing) and with synchronisation of camming, the maximum velocity P-AXIS-00302 for a slave axis may not be assigned zero.		
Response	Class	1	Warning output. The value for the maximum slave velocity P-AXIS-00302 is automatically set to the maximum permissible axis velocity P-AXIS-00212.
Solution	Class	1	Set value for P-AXIS-00302 greater than zero.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value of P-AXIS-00302.	
	%3:	Corrected value [1µm/s or 0.001°/s]	
		Corrected value of P-AXIS-00302.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110404

The maximum acceleration of slave axis may not be set to 0.			
Description	During synchronisation (camming and gearing) and with synchronisation of camming, the maximum acceleration P-AXIS-00303 of a slave axis may not be assigned zero.		
Response	Class	1	Warning output. The value for the maximum permissible slave acceleration P-AXIS-00303 is automatically set to the maximum permissible axis acceleration P-AXIS-00008.
Solution	Class	1	Set value for P-AXIS-00303 greater than zero.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [1mm/s^2 or 1°/s^2]
		Current value of P-AXIS-00303.
	%3:	Corrected value [1mm/s^2 or 1°/s^2]
		Corrected value of P-AXIS-00303.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110405

Maximum for weighted acceleration is greater than maximum permissible axis acceleration.			
Description	The maximum acceleration for weighting using the functions G130/G131 (see P-AXIS-00292) is greater than the maximum permissible axis acceleration (see P-AXIS-00008).		
Response	Class	1	Reduce acceleration P-AXIS-00292 to the maximum axis acceleration
Solution	Class	1	Correct parameter P-AXIS-00292
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Specified maximum for weighted acceleration, see P-AXIS-00292	
	%3:	Limit value [1mm/s^2 or 1°/s^2]	
		Maximum axis acceleration, see P-AXIS-00008	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
		Corrected maximum for weighted acceleration, see P-AXIS-00292	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110406

Minimum for weighted ramp time is smaller than minimum permissible ramp time.			
Description	The minimum for ramp time P-AXIS-00293 with weighting using function G132/G133 is smaller than the minimum permissible ramp time P-AXIS-00201.		
Response	Class	1	Increase ramp time P-AXIS-00293 to the minimum permissible ramp time.
Solution	Class	1	Increase ramp time P-AXIS-00293
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [1 µs]	
		Specified minimum for weighted ramp time, see P-AXIS-00293	
	%3:	Limit value [1 µs]	
		Minimum permissible ramp time, see P-AXIS-00201	
	%4:	Corrected value [1 µs]	
		Corrected minimum for weighted ramp time, see P-AXIS-00293	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110409

Phasing: velocity is inadmissible.

Description	The phasing velocity P-AXIS-00305 for synchronisation with camming may not be zero or greater than half the maximum permissible axis velocity P-AXIS-00212.		
Response	Class	1	Warning output. The value of P-AXIS-00305 is automatically set to half the maximum permissible axis velocity P-AXIS-00212 .
Solution	Class	1	Set value for P-AXIS-00305 greater than zero and less than half the maximum permissible axis velocity P-AXIS-00212.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value of P-AXIS-00305.	
	%3:	Corrected value [1µm/s or 0.001°/s]	
		Corrected value of P-AXIS-00305.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110410

Phasing: acceleration is inadmissible.			
Description	The acceleration P-AXIS-00306 for phase synchronisation with camming may not be zero or greater than half the maximum permissible axis acceleration P-AXIS-00008.		
Response	Class	1	Warning output. The value of P-AXIS-00306 is automatically set to half the maximum permissible axis acceleration P-AXIS-00008 .
Solution	Class	1	Set value for P-AXIS-00306 greater than zero and less than half the maximum permissible axis acceleration P-AXIS-00008.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Current value of P-AXIS-00306.	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		Corrected value of P-AXIS-00306.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110411

Phasing: deceleration is inadmissible.			
Description	The deceleration P-AXIS-00307 for phase synchronization with camming may not be zero or greater than half the maximum permissible axis acceleration P-AXIS-00008.		
Response	Class	1	Warning output. The value of P-AXIS-00307 is automatically set to half the maximum permissible axis acceleration P-AXIS-00008 .
Solution	Class	1	Set value for P-AXIS-00307 greater than zero and less than half the maximum permissible axis acceleration P-AXIS-00008.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Current value of P-AXIS-00307.	
	%3:	Corrected value [1mm/s^2 or 1°/s^2]	
		Corrected value of P-AXIS-00307.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110412

Coefficient (pitch) of temperature compensation exceeds valid range.			
Description	The compensation values of temperature compensation are approximated by a linear straight line. The pitch of the straight line is specified in parameter P-AXIS-00274 as a function of temperature. However, the current value specified in the axis parameter list is outside the permissible value range. See FCT-C5// Section: Temperature compensation		
Response	Class	1	Warning, automatic correction of parameter P-AXIS-00274 to maximum or minimum permissible value.
Solution	Class	1	Correct the value of parameter P-AXIS-00274 in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016)	
	%2:	Incorrect value [-]	
		Invalid value for P-AXIS-00274	
	%3:	Limit value [-]	
		Largest allowed value for pitch P-AXIS-00274	
	%4:	Limit value [-]	
		Smallest permissible value for pitch P-AXIS-00274	
	%5:	Corrected value [-]	
		Corrected value for parameter P-AXIS-00274	
Error type	-		

ID 110413

The velocity during homing to a fixed stop is greater than the maximum velocity of unreferenced axes.			
Description	The velocity *getriebe[i].homing.torq_move_velocity* (P-AXIS-00333) may not be greater than the permissible maximum velocity *getriebe[i].vb_not_referenced* (P-AXIS-00268) for unreferenced axes in relative and endless movements.		
Response	Class	1	Warning output. The value of P-AXIS-00333 is automatically set to the maximum permissible velocity P-AXIS-00268 for unreferenced axes.
Solution	Class	1	Set the value for P-AXIS-00333 smaller than the maximum permissible velocity P-AXIS-00268 for unreferenced axes.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value for *torq_move_velocity* (P-AXIS-00333)	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum velocity P-AXIS-00268 for unreferenced axes.	
	%4:	Corrected value	
		Parameter P-AXIS-00268 corrected from P-AXIS-00333.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110414

This terminal can not be used for probing.			
Description	The axis was set as probing axis by assigning the value 1 to P-AXIS-00118, although the used terminal does not support this function.		
Response	Class	1	Warning output, corrected value of P-AXIS-00118 is set to 0.

Solution	Class	1	Assign the value 0 to parameter P-AXIS-00118 in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis identifier of concerned axis (see P-AXIS-00016).	
	%2:	Current value [-]	
		Incorrect value for P-AXIS-00118.	
	%3:	Corrected value [-]	
		Corrected value for P-AXIS-00118.	
	%4:	Current value [-]	
		String containing the terminal type.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110415

This Terminal can not be used for homing with zero pulse.			
Description	Homing with zero pulse search was performed for the axis by assigning the value 0 to P-AXIS-00084, although the terminal used does not support this function. For more information on parameterising homing. FCT-M1// Section: Homing with zero pulse without reference cam.		
Response	Class	1	Warning output, corrected value of P-AXIS-00084 is set to 1.
Solution	Class	1	Assign the value 1 to parameter P-AXIS-00084 in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016).	
	%2:	Current value [-]	
		Incorrect value for P-AXIS-00084.	
	%3:	Corrected value [-]	
		Corrected value for P-AXIS-00084.	
	%4:	Current value [-]	
		String containing the terminal type.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110416

Homing with evaluation of encoder overflow is not supported.			
Description	The 'Encoder overflow evaluation' homing method was parameterised for this axis by assigning P-AXIS-00294 a value of 1, although this homing method is not available for the type of drive used.		
Response	Class	3	Warning output, correct value of P-AXIS-00294 to 0.
Solution	Class	7	Assign the value 0 to parameter P-AXIS-00294 in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type of the axis (see P-AXIS-00020).	
	%3:	Current value [-]	
		Incorrect value for P-AXIS-00294.	
	%4:	Corrected value [-]	
		Corrected value for P-AXIS-00294.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110417

Invalid value for parameter probing_input_number was corrected.			
Description	The axis was set as probing axis (P-AXIS-00118) and an invalid value was specified for the probing input to be used at the drive (parameter P-AXIS-00295).		
Response	Class	1	Warning output, correct value of P-AXIS-00295 to 1.
Solution	Class	1	In the axis parameter list, assign a valid value (1 or 2) to parameter P-AXIS-00295. In the drive, the latch function must be assigned to the digital input used by assigning the value 26 to drive parameter IN1MODE or IN2MODE.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016).	
	%2:	Incorrect value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110418

Value for parameter encoder_range is outside permissible range.			
Description	Parameter P-AXIS-00296 is out of the permissible range.		
Response	Class	1	Warning output, correct the value of P-AXIS-00296. If P-AXIS-00296 was configured below the minimum value, P-AXIS-00296 is set to the minimum value, otherwise, if the maximum value was exceeded, P-AXIS-00296 is limited to the maximum value.
Solution	Class	1	Set a valid value for P-AXIS-00295 in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016).	
	%2:	Current value [-]	
		Index of the invalid entry in the lr_hw[] array. A value of 0 means lr_hw[0].encoder_range is incorrect.	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110419

Changing of parameter encoder_range requires restart of the control.	
Description	The parameter P-AXIS-00296 for the range of the position sensor cannot be changed while the CNC kernel is running. The modified value is not adopted; the previous value remains valid.

Response	Class	0	None.
Solution	Class	0	Change the sign P-AXIS-00296 in configuration mode or when the CNC is inactive.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of axis affected.
	%2:		
			Current value of P-AXIS-00296.
	%3:		
			Corrected value of P-AXIS-00296.
Error type	-		

ID 110420

Invalid value for parameter max_reference_position_offset entered.			
Description	The value of parameter P-AXIS-00298 (permissible distance to homing to suppress a gantry error) is outside the permissible data range.		
Response	Class	1	Warning output, value corrected to 0.
Solution	Class	1	Change parameter P-AXIS-00298.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of axis affected.
	%2:		Lower limit value [0.1 µm or 0.0001°]
			Permissible minimum value of P-AXIS-00298.
	%3:		Upper limit value [0.1 µm or 0.0001°]
			Permissible maximum value of P-AXIS-00298.
	%4:		Incorrect value [0.1 µm or 0.0001°]
			Current (incorrect) value of P-AXIS-00298.
	%5:		Corrected value [0.1 µm or 0.0001°]
			Corrected value of P-AXIS-00298.
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110421

Maximum velocity for homing is greater than the maximum velocity of non-homed axes.			
Description	The fast homing velocity P-AXIS-00219 may not be greater than the maximum permissible velocity P-AXIS-00268 for non-homed axes.		
Response	Class	1	Warning output. The value of P-AXIS-00219 is automatically set to the maximum permissible velocity P-AXIS-00268 for non-homed axes.
Solution	Class	1	Set the value for P-AXIS-00219 smaller than the maximum permissible velocity P-AXIS-00268 for non-homed axes.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of axis affected.
	%2:		Current value [1µm/s or 0.001°/s]
			Current value of P-AXIS-00219.
	%3:		Limit value [1µm/s or 0.001°/s]
			Maximum velocity P-AXIS-00268 for non-homed axes.
	%4:		Corrected value [1µm/s or 0.001°/s]
			Velocity value corrected to P-AXIS-00268 .

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110422

Minimum velocity for homed is greater than the maximum velocity of non-homed axes.			
Description	The slow homing velocity P-AXIS-00218 may not be greater than the maximum permissible velocity P-AXIS-00268 for non-homed axes.		
Response	Class	1	Warning output. The value of P-AXIS-00218 is automatically set to the maximum permissible velocity P-AXIS-00268 for non-homed axes.
Solution	Class	1	Set the value for P-AXIS-00218 smaller than the maximum permissible velocity P-AXIS-00268 for non-homed axes.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Current value of P-AXIS-00218.	
	%3:	Limit value [1µm/s or 0.001°/s]	
		Maximum velocity P-AXIS-00268 for non-homed axes.	
	%4:	Corrected value [1µm/s or 0.001°/s]	
		Velocity value corrected to P-AXIS-00268 .	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110423

Line could not be written to file.			
Description	When changing an axis parameter, the changed line could not be written to the file. Possible causes are: • The file is write protected. • The file is blocked by another process. • Insufficient access rights.		
Response	Class	3	Abort job processing.
Solution	Class	6	Remedy possible problems and process reset after that. Problem solution: • Remove write protection. • Wait until the blocking process releases the file or ends the blocking process • Check access rights.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	5, Error message by access on files.		

ID 110424

Could not delete file.

Description	The file could not be deleted. Possible causes are: • The file is write-protected. • The file is blocked by another process. • Insufficient access rights.		
Response	Class	3	Abort job processing.
Solution	Class	6	Remedy possible problems and then execute reset. Problem solution: • Remove write protection. • Wait until the blocking process releases the file or end the blocking process. • Check access rights.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	5, Error message by access to files.		

ID 110425

Could not change file name, check file properties and access rights.			
Description	The file name could not be renamed. Possible causes are: • The file is write-protected. • The file is blocked by another process. • Insufficient access rights.		
Response	Class	3	Abort job processing.
Solution	Class	6	Remedy possible problems and then execute reset. Problem solution: • Remove write protection. • Wait until the blocking process releases the file or end the blocking process. • Check access rights.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	5, Error message by access to files.		

ID 110426

The given parameter set number is invalid.			
Description	The specified parameter set number is too large. Check for maximum number of alternative parameter set ids.		
Response	Class	3	Abort job processing.
Solution	Class	3	Correct value of parameter set number.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Incorrect value [-]
		Invalid parameter set number.
	%3:	Upper limit value [-]
		Maximum number of alternative parameter sets.
Error type	-	

ID 110427

There are no changed parameter values available.			
Description	There are no changed parameter values available.		
Response	Class	3	Job processing aborted.
Solution	Class	3	-
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
Error type	-		

ID 110428

Saving not possible, since parameter set is not activated.			
Description	The change parameter values cannot be saved because the current parameter set was not activated.		
Response	Class	3	Abort job processing.
Solution	Class	3	Activate current parameter set for saving.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Current parameter set.	
Error type	-		

ID 110429

File name is too long.			
Description	The file name of the backup list file, including the path, is too long.		
Response	Class	3	Abort job processing.
Solution	Class	6	Adapt length of file name or path.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Current value [-]
		Value of the actual parameter set id.
	%3:	Limit value [-]
		Maximum length of the file name, including path.
	%4:	Error value [-]
		Current length of the file name, including path.
Error type	5, Error message by access on files.	

ID 110430

Activation of parameter set not possible, since there are no changed values available.			
Description	The current parameter set cannot be activated because there are no changed parameter values.		
Response	Class	3	Abort job processing.
Solution	Class	3	Activation of parameter set is not possible until there are any changed values available.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	-		

ID 110431

Saving not possible, since there are no changed parameter values available.			
Description	Saving is not possible because no parameter values was changed.		
Response	Class	1	-
Solution	Class	1	-
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	-		

ID 110432

The string of the read parameter value is too long.			
Description	The string of the read parameter value is too long.		
Response	Class	3	Abort job processing.
Solution	Class	6	Adapt length of parameter value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [-]
		Current parameter set number.
	%3:	Error value [-]
		Length of the read parameter value.
	%4:	Limit value [-]
		Maximum permissible length of the parameter value.
Error type	-	

ID 110433

The parameter identifier could not be found.			
Description	The entered parameter identifier is not valid.		
Response	Class	3	Abort job processing.
Solution	Class	3	Enter valid parameter identifier.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	-		

ID 110434

Inconsistent axis parameters due to errors during updating process.			
Description	Inconsistent axis parameters due to errors during updating process.		
Response	Class	3	Abort job processing.
Solution	Class	7	Correct axis parameters.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 110435 / 110436

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 110437

Velocity monitoring parameter was corrected from zero to P-AXIS-00268.

Description	The value for actual velocity monitoring *getriebe[i].vb_monitor* (P-AXIS-00311) may not be zero. The value was corrected to the maximum permissible velocity *getriebe[i].vb_not_referenced* (P-AXIS-00268) for non-homed axes.		
Response	Class	1	Warning output. Value is corrected to P-AXIS-00268.
Solution	Class	1	Value for P-AXIS-00311 must be greater than zero (nil).
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1µm/s or 0.001°/s]	
		Parameterised value of P-AXIS-00311.	
	%3:	Corrected value [1µm/s or 0.001°/s]	
		New value of P-AXIS-00311.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110438

Changed parameter values are already activated.			
Description	The changed parameter values have already been activated.		
Response	Class	1	-
Solution	Class	1	-
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	-		

ID 110439

The given parameter type is invalid.			
Description	The ID for the specified parameter or list type is invalid.		
Response	Class	3	Abort job processing.
Solution	Class	3	Correct parameter or list type.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 110440

No parameter identifier specified.			
Description	No parameter identifier has been specified.		
Response	Class	3	Abort job processing.
Solution	Class	3	Use valid parameter identifier.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current parameter set number.	

Error type	-

ID 110441

No parameter value specified.			
Description	No parameter value has been specified.		
Response	Class	3	Abort job processing.
Solution	Class	3	Use valid parameter value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current parameter set number.	
Error type	-		

ID 110442

Line for parameter configuration list too long or end of line token missing.			
Description	The line for the parameter configuration is too long or the end of line token is missing.		
Response	Class	3	Abort job processing.
Solution	Class	6	Adjust line length or insert end of line token.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current parameter set number.	
	%3:	Error value [-]	
		Current line length.	
	%4:	Limit value [-]	
		Maximum line length.	
Error type	-		

ID 110443

The axis management could not open the file.			
Description	The axis management could not open the file. Possible causes are: • The file is write-protected. • The file is blocked by another process. • Insufficient access rights. • The file does not exist.		
Response	Class	3	Abort job processing.
Solution	Class	6	Remedy possible problems and then execute reset. Problem solution: • Remove write protection. • Wait until the blocking process releases the file or end the blocking process. • Check access rights.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [-]
		Current parameter set number.
Error type	-	

ID 110444

A line could not be read from file.			
Description	A line could not be read from file. Possible reasons are: • The file is write protected. • The file is blocked by another process. • Insufficient access rights.		
Response	Class	3	Abort of order processing.
Solution	Class	6	Remedy possible problems and process reset after that. Problem solution: • Remove write protection. • Wait until the blocking process has released the file or blocking process has been terminated. • Check access rights.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of concerned axis.
	%2:		Current value [-]
			Current parameter set number.
Error type	5, Error message by access on files.		

ID 110445

Phasing: jerk inadmissible.			
Description	The value for the phasing jerk j_phasing may not be zero or greater than the maximum jerk. The maximum jerk is then half the value of the maximum acceleration divided by the minimum permissible ramp time.		
Response	Class	1	Warning output. The value of j_phasing is automatically set to the maximum jerk.
Solution	Class	1	Set value of j_phasing greater than zero and less than the maximum jerk.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of axis affected.
	%2:		Current value [1mm/2^3 or 1°/s^3]
			Current value of j_phasing.
	%3:		Corrected value [1mm/2^3 or 1°/s^3]
			Value of j_phasing corrected to the maximum jerk.
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110446

Jerk for velocity synchronisation inadmissible.			
Description	The value of the jerk j_vel_sync for velocity synchronisation may not be zero or greater than the maximum jerk. The maximum jerk is then half the value of the maximum acceleration divided by the minimum permissible ramp time.		
Response	Class	1	Warning output. The value of j_vel_sync is automatically set to the maximum jerk.
Solution	Class	1	Set value of j_vel_sync greater than zero (nil) and less than the maximum jerk.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/2^3 or 1°/s^3]	
		Current value of j_vel_sync.	
	%3:	Corrected value [1mm/2^3 or 1°/s^3]	
		Value of j_vel_sync corrected to the maximum jerk.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110447

Invalid value for parameter P979_3 was corrected.			
Description	The 'reading absolute position from drive' function (P-AXIS-00315) was activated in the axis parameter list, and no value or a value outside the permissible value range was assigned to parameter P-AXIS-00316. This parameter is initialised with –1 by default. This error message is output if no entry was made in the axis parameter list.		
Response	Class	1	The value of P-AXIS-00316 is automatically corrected.
Solution	Class	1	Change value of P-AXIS-00316 or enter an entry in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Incorrect value [-]	
		Current (incorrect) value of P-AXIS-00316.	
	%3:	Lower limit value [-]	
		Permissible minimum value of P-AXIS-00316.	
	%4:	Upper limit value [-]	
		Permissible maximum value of P-AXIS-00316.	
	%5:	Corrected value [-]	
		Corrected value of P-AXIS-00316.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110448

Invalid value for parameter P979_4 was corrected.	
Description	The 'reading absolute position from drive' function (P-AXIS-00315) was activated in the axis parameter list, and no value or a value outside the permissible value range was assigned to parameter P-AXIS-00317. This parameter is initialised with –1 by default. This error message is output if no entry was made in the axis parameter list.

Response	Class	1	The value of P-AXIS-00317 is automatically corrected.
Solution	Class	1	Change value of P-AXIS-00317 or enter an entry in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Incorrect value [-]	
		Current (incorrect) value of P-AXIS-00317.	
	%3:	Lower limit value [-]	
		Permissible minimum value of P-AXIS-00317.	
	%4:	Upper limit value [-]	
		Permissible maximum value of P-AXIS-00317.	
	%5:	Corrected value [-]	
		Corrected value of P-AXIS-00317.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110449

Invalid value for parameter P-AXIS-00318 was corrected.			
Description	The ‘Reading absolute position from drive’ function in the axis parameter list was activated by *antr.profibus.read_abs_pos_mode* (P-AXIS-00315) and no value or a value outside the permissible value range was assigned to the parameter *antr.profibus.read_abs_pos_from_drive* (P-AXIS-00318).		
Response	Class	1	The value of P-AXIS-00318 is automatically corrected.
Solution	Class	1	Change value of P-AXIS-00318 or enter an entry in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Error value [-]	
		Parameterised value of P-AXIS-00318.	
	%3:	Upper limit value [-]	
		Permissible maximum value of P-AXIS-00318.	
	%4:	Corrected value [-]	
		New value of P-AXIS-00318.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110450

Modification of parameter p1042 requires restart of the control.			
Description	Parameter P-AXIS-00316 cannot be changed while the CNC kernel is running. The new value is not adopted; the previous active value remains valid.		
Response	Class	1	None.
Solution	Class	1	Reboot the controller after changing P-AXIS-00316.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00316.	
	%3:	Corrected value [-]	
		Corrected (old) value of P-AXIS-00316.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110451

Modification of parameter p1043 requires restart of the control.			
Description	Parameter P-AXIS-00317 cannot be changed while the CNC kernel is running. The new value is not adopted; the previous active value remains valid.		
Response	Class	1	None.
Solution	Class	1	Reboot the controller after changing P-AXIS-00317.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of P-AXIS-00317.	
	%3:	Corrected value [-]	
		Corrected (old) value of P-AXIS-00317.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110452

Modification of parameter P-AXIS-00318 requires restart of the control.			
Description	The parameter *antr.profibus.read_abs_pos_mode* (P-AXIS-00318) cannot be changed while the CNC is running. The modified sign value is not transferred, the previous value remains valid.		
Response	Class	1	None.
Solution	Class	1	Reboot the controller after changing P-AXIS-00318.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current value of P-AXIS-00318.	
	%3:	Corrected value [-]	
		Corrected (old) value of P-AXIS-00318.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110453

Modification of parameter P-AXIS-00315 requires restart of the control.			
Description	The parameter *antr.profibus.read_abs_pos_from_drive* (P-AXIS-00315) cannot be changed while the CNC is running. The modified sign value is not transferred, the previous value remains valid.		
Response	Class	1	None.
Solution	Class	1	Reboot the controller after changing P-AXIS-00315.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current value of P-AXIS-00315.	
	%3:	Corrected value [-]	
		Corrected (old) value of P-AXIS-00315.	

Error type	2, Error message by data transfer from parameter list into control device.

ID 110454

Value for quantization of measurement system increments is zero.			
Description	The value for the quantization of measurement system increments is zero. It must be greater zero. The default setting consists of one increment. For spindle revolution monitoring (see FCT-S1// Section: Spindle monitoring and during the check of the exact stop of axes (see P-AXIS-00236), a check is made whether the current values are within a specified tolerance. Parameter P-AXIS-00323 is the minimum size of the tolerance window after conversion of the tolerance limits into the resolution of the measurement system (default setting: 1 increment).		
Response	Class	1	Correct parameter P-AXIS-00323 to default value 1
Solution	Class	1	Correct parameter P-AXIS-00323
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Incorrect value [-]	
		Specified value, see P-AXIS-00323	
	%3:	Corrected value [-]	
		Corrected value see P-AXIS-00323	
Error type	-		

ID 110455

A specification for a link with P-AXIS-00101 is not allowed for gantry slave axis.			
Description	The parameter P-AXIS-00101 can be used to connect a logical axis of the interpolator to a physical axis in the position controller. However, this is not permitted tor a gantry slave axis (see [FCT-C11//: Description]) since it cannot be moved independently of the gantry master axis and is therefore not available in the interpolator.		
Response	Class	3	Start-up aborted
Solution	Class	7	Check the parameter P-AXIS-00101 and set to zero.
Parameter	%1:	Error value [-]	
		Specified value for parameter P-AXIS-00101	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110456

Changing the drive type requires restart of the control.			
Description	The operation mode (see P-AXIS-00320) of an axis cannot be changed while the controller is running, e.g. when updating a parameter list. The controller must be stopped and restarted in order to change the operation mode.		
Response	Class	1	Warning, operation mode is corrected to the old value
Solution	Class	1	Stop and restart the controller

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of concerned axis, see P-AXIS-00020
	%3:	Incorrect value [-]
		New operation mode for axis, see P-AXIS-00320
	%4:	Corrected value [-]
		Currently used operation mode for axis, see P-AXIS-00320
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110457

This drive type does not support the parameterised operation mode.			
Description	The configured operation mode (see P-AXIS-00320) is not supported for the drive type used.		
Response	Class	1	Warning and correct the operation mode
Solution	Class	1	Select an operation mode which is supported for the drive type
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Drive type [-]	
		Drive type of concerned axis, see P-AXIS-00020	
	%3:	Incorrect value [-]	
		New operation mode for axis, see P-AXIS-00320	
	%4:	Corrected value [-]	
		New operation mode for axis, see P-AXIS-00320	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110459

Activation of conventional and adaptive feedforward control at the same time not possible.			
Description	In the parameter *vorsteuer.vorsteuerung* (P-AXIS-00223), both conventional feedforward control (bits 0 ... 3) and adaptive feedforward control (bits 4 ... 6) were activated.		
Response	Class	1	Warning and correction of parameter. Feedforward control is deactivated.
Solution	Class	1	Check and modify the parameter P-AXIS-00223.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Current value [-]	
		Parameterised value of P-AXIS-00223	
	%3:	Corrected value [-]	
		New value of P-AXIS-00223	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110460

Limiting control velocity (P-AXIS-00220) equals zero expected for spindles without sensor.

Description	Parameter *getriebe[i].vb_regelgrenze* (P-AXIS-00220) describes the limiting velocity above which the measurement system delivers invalid values (e.g. spindles at high speeds). When this velocity is exceeded, the position controller of the axis is switched to open-loop controlled mode since the axis cannot be controlled without current values. For a sensor-less spindle, the limiting velocity must be set to zero.		
Response	Class	1	Warning and correct parameter P-AXIS-00220
Solution	Class	1	Spindle without encoder: Set the limiting velocity P-AXIS-00220 to zero. Spindle with encoder: Actual values must be included in the cyclic telegram for this axis.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Drive type [-]
		Drive type of axis affected, see P-AXIS-00020
	%3:	Incorrect value [-]
		Specified limiting velocity for this axis, see P-AXIS-00220
	%4:	Expected value [-]
		Expected limiting velocity for this axis, see P-AXIS-00220
	%5:	Corrected value [-]
		Corrected value for limiting velocity, see P-AXIS-00220
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110461

Number of configured user idents exceeds limit.			
Description	During the configuration of drive functions, the number of telegram elements (P-AXIS-00398) written in the cyclic process data exceeded the maximum limit See [FCT-A10// Section: Description]		
Response	Class	3	Program abort.
Solution	Class	6	Adjust value of VAL in the NC program.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [-]
		Drive type (P-AXIS-00020).
	%3:	Current value [-]
	%4:	Limit value [-]
	%5:	Current value [-]
Error type	-	

ID 110462

User ident configured with zero length.

Description	A telegram element with data length 0 was configured in the output telegram of the drive.		
Response	Class	3	Program abort.
Solution	Class	6	Change signal length of telegram element.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Drive type, see P-AXIS-00020.	
	%3:	Current value [-]	
		Index of the incorrect telegram segment.	
	%4:	Current value [-]	
		Name of incorrect telegram element.	
	%5:	Current value [-]	
		Signal number of incorrect telegram element.	
Error type	-		

ID 110463

Additive torque command is greater than nominal torque.			
Description	The additive torque command value entered in the axis parameter list (see P-AXIS-00324) is greater than 1000. This means that the output torque command value would be greater than the nominal torque of the motor.		
Response	Class	1	Warning output, automatic correction of P-AXIS-00324 to 0.
Solution	Class	1	Assign a value > 1000 to P-AXIS-00324.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (see P-AXIS-00016).	
	%2:	Lower limit value [0.1 µm or 0.0001°]	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
	%4:	Incorrect value [0.1 µm or 0.0001°]	
	%5:	Corrected value [0.1 µm or 0.0001°]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110464

Additive torque command is not supported by this drive type.			
Description	The output of an additive torque command value (see P-AXIS-00324) was configured in the axis parameter list although the drive type set (see P-AXIS-00020) does not support this.		
Response	Class	1	Warning output, output of an additive torque command value is disabled by setting P-AXIS-00324 to 0.
Solution	Class	1	Remove P-AXIS-00324 from the axis parameter list or assign it the value 0.

Parameter	%1:	Logical axis number [-]
		Logical axis number (see P-AXIS-00016).
	%2:	Current value [-]
		Drive type (see P-AXIS-00020).
	%3:	Incorrect value [-]
		Parameterised value of P-AXIS-00324.
	%4:	Corrected value [-]
		Corrected value of P-AXIS-00324.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110465

Denominator for torque scaling must not be 0.			
Description	The denominator of the torque scaling factor (see P-AXIS-00326) was assigned the value 0.		
Response	Class	1	Warning output, torque scaling factor is set internally to 0.
Solution	Class	1	Assign P-AXIS-00326 a value unequal to 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (see P-AXIS-00016).	
	%2:	Incorrect value [-]	
		Incorrect value of P-AXIS-00326.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00326.	
	%4:	Corrected value [-]	
		Incorrect value of P-AXIS-00325.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110466

Illegal measurement method. Only one method may be selected.			
Description	In addition to the parameter P-AXIS-00330, one of the parameters P-AXIS-00269,P-AXIS-00117, P-AXIS-00115, P-AXIS-00116 or P-AXIS-00257 was enabled. If P-AXIS-00330 is not enabled, no other measurement method can be enabled.		
Response	Class	1	Error message output, the parameters P-AXIS-00269, P-AXIS-00117, P-AXIS-00115, P-AXIS-00116 and P-AXIS-00257 are set to zero.
Solution	Class	1	Configure measurement method correctly.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		Value of P-AXIS-00330.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110467

Invalid value for P-AXIS-00321.

Description	Parameter *antr.reference_cam_signal* (P-AXIS-00321) specified a digital input to read the reference cam signal that does not match the drive type used (see also P-AXIS-00321).		
Response	Class	1	Warning and correction of parameter. Correct P-AXIS-00321 to the value “PLC”.
Solution	Class	1	Check and modify P-AXIS-00321; the drive type used must support this value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Drive type [-]	
		Drive type (P-AXIS-00020).	
	%3:	Error value [-]	
		Incorrect value of P-AXIS-00321.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00321.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110468

Homing on hardware limit switch without reversal is not possible.			
Description	The parameters P-AXIS-00329 and P-AXIS-00157 are set simultaneously. This is not possible because homing on hardware limit switches is executed in principle with reversal.		
Response	Class	3	Warning output, correct P-AXIS-00157 to 0.
Solution	Class	1	Assign the value 0 to either P-AXIS-00157 or P-AXIS-00329.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (see P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type (see P-AXIS-00020).	
	%3:	Current value [-]	
		Incorrect value of P-AXIS-00157.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00157.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110469

Homing on hardware limit switch without cam is not possible.			
Description	The parameters P-AXIS-00329 and P-AXIS-00156 are set simultaneously.		
Response	Class	3	Warning output, correct P-AXIS-00156 to 0.
Solution	Class	1	Assign the value 0 to either P-AXIS-00156 or P-AXIS-00329.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (see P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type (see P-AXIS-00020).	
	%3:	Current value [-]	
		Incorrect value of P-AXIS-00156.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00156.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110470

Homing on hardware limit switch is only possible with CNC-controlled homing.			
Description	Homing to hardware limit switch (P-AXIS-00329) and a homing type (P-AXIS-00299) unequal to "CNC_CONTROLLED " was parameterised.		
Response	Class	3	Warning output, correct P-AXIS-00329 to 0.
Solution	Class	1	Either execute CNC-controlled homing (P-AXIS-00299 = CNC_CONTROLLED) or assign the value 0 to P-AXIS-00329.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (see P-AXIS-00016).	
	%2:	Current value [-]	
		Drive type (see P-AXIS-00020).	
	%3:	Current value [-]	
		Incorrect value of P-AXIS-00329.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00329.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110471

Invalid value of numerator for feed constant.			
Description	Axis parameter P-AXIS-00362 was set to 0.		
Response	Class	3	Error message output, correct P-AXIS-00362 to 100000.
Solution	Class	7	Enter the valid value for P-AXIS-00362. Valid value range is 1 … MAX_UNS32.
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110472

Invalid value of denominator for feed constant.			
Description	Axis parameter P-AXIS-00363 was set to 0.		
Response	Class	3	Error message output, correct P-AXIS-00363 to 1.
Solution	Class	7	Enter the valid value for P-AXIS-00363. Valid value range is 1 … MAX_UNS32.

Parameter	%1:	Logical axis number [-]
	%2:	Incorrect value [-]
	%3:	Lower limit value [-]
	%4:	Upper limit value [-]
	%5:	Corrected value [-]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110473

Denominator of scaling factor for jerk feedforward may not be zero.			
Description	The denominator of the scaling factor for the jerk feedforward control (P-AXIS-00338) was set to zero.		
Response	Class	1	Warning output, correct P-AXIS-00338 to 100.
Solution	Class	1	Set P-AXIS-00338 to a value unequal to zero.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (see P-AXIS-00016).	
	%2:	Incorrect value [-]	
		Incorrect value of P-AXIS-00338.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00338.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110474

Backlash in leadscrew error compensation exceeds range of values.			
Description	With double-sided leadscrew error compensation (P-COMP-00021), the difference (backlash) between the compensation value in the positive direction (see P-COMP-00023) and the compensation value in the negative direction (see P-COMP-00024) must be extended when reversing the direction. However, the specified backlash in the compensation value list exceeds the permissible value range.		
Response	Class	3	Transfer of the compensation value list to the controller is aborted.
Solution	Class	7	Correct the compensation value list
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis (see P-AXIS-00016)	
	%2:	Current value [-]	
		Array index of the affected compensation value pair.	
	%3:	Current value [increments]	
		Compensation value in positive direction (P-COMP-00023)	
	%4:	Current value [increments]	
		Compensation value in negative direction (P-COMP-00024)	
	%5:	Incorrect value [increments]	

Error type	-

ID 110475

Signal length configured in user ident is not supported.			
Description	The length of the configured telegram element is not supported by the NC kernel.		
Response	Class	3	Program abort.
Solution	Class	6	Change signal length of telegram element.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Drive type, see P-AXIS-00020.	
	%3:	Current value [-]	
		Index of incorrect telegram element.	
	%4:	Current value [-]	
		Name of incorrect telegram element.	
	%5:	Incorrect value [-]	
		Signal number of incorrect telegram element.	
Error type	-		

ID 110476

Compensation value change in leadscrew error compensation exceeds range of values.			
Description	The difference between successive compensation values (see P-COMP-00023, P-COMP-00024) in the compensation value list for leadscrew error compensation is greater than permissible. If metric input in 0.1 µm is used (see P-COMP-00017), the compensation values in the controller are multiplied by the resolution of the axis (see P-AXIS-00234/P-AXIS-00233).		
Response	Class	3	Leadscrew error compensation is not activated.
Solution	Class	7	Correct the compensation value list of the leadscrew error compensation. Reduce the distance between interpolation points (see P-COMP-00018) or insert additional interim points in the compensation value list to reduce the compensation value change per interval
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Array index of the affected compensation value in the compensation value list	
	%3:	Current value [increments]	
		Current compensation value in increments (see P-COMP-00023/P-COMP-00024))	
	%4:	Current value [increments]	
		Current compensation value in increments (see P-COMP-00023/P-COMP-00024))	
	%5:	Incorrect value [increments]	
		Difference between both compensation values in increments	
Error type	-		

ID 110477

Position of interpolation point in leadscrew error compensation exceeds range of values.			
Description	The interpolation points in the compensation value list of equidistant leadscrew error compensation are defined by the start position (see P-COMP-00019), the distance between interpolation points (see P-COMP-00018) and the number of entries (see P-COMP-00020). However, the end position of the compensation value list calculated from this data (P-COMP-00019 + P-COMP-00020 * P-COMP-00018) is greater than permissible.		
Response	Class	3	Leadscrew error compensation is not activated.
Solution	Class	7	Correct the compensation value list of the leadscrew error compensation. Reduce the number of entries P-COMP-00020 in the compensation value list Reduce the distance P-COMP-00018 between interpolation points
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [increments]	
	%3:	Upper limit value [increments]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110478

Compensation value in leadscrew error compensation exceeds range of values.			
Description	If the compensation values are entered metrically in 0.1 µm (see P-COMP-00017), the controller converts the values into increments by multiplying them with the resolution of the axis (see P-AXIS-00234/P-AXIS-00233). However, the resulting value exceeds the permissible value range.		
Response	Class	3	Leadscrew error compensation is not activated.
Solution	Class	7	Correct the compensation value list of the leadscrew error compensation.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [increments]	
		Affected compensation value in increments (see P-COMP-00023/P-COMP-00024))	
	%3:	Lower limit value [increments]	
		Minimum permissible value	
	%4:	Upper limit value [increments]	
		Maximum permissible value	
	%5:	Current value [-]	
		Table index of affected compensation value	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110479

Distance between interpolation points in leadscrew error compensation exceeds range of values.

Description	During leadscrew error compensation, if the distance between compensation values (see P-COMP-00018) is entered metrically in 0.1 µm (see P-COMP-00017) , the controller converts the value into increments by multiplying it by the resolution of the axis (see P-AXIS-00234/P-AXIS-00233). The resulting value exceeds the range of values.		
Response	Class	3	Leadscrew error compensation is not activated.
Solution	Class	7	Correct the compensation value list of the leadscrew error compensation. Reduce the distance between interpolation points (see P-COMP-00018) in the compensation value list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [increments]	
		Distance between Interpolation points in increments	
	%3:	Lower limit value [increments]	
		Minimum permissible distance between interpolation points in increments.	
	%4:	Upper limit value [increments]	
		Maximum permissible distance between interpolation points in increments.	
	%5:	Current value [0.1 µm or 0.0001°]	
		Distance between Interpolation points P-COMP-00018 in compensation value list	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110480

Distance between interpolation points in leadscrew error compensation exceeds range of values.			
Description	During leadscrew error compensation, if the interpolation points P-COMP-00025 in the compensation value list are entered metrically in 0.1 µm (see P-COMP-00017), the positions in the controller are converted into increments by multiplying them by the resolution of the axis (see P-AXIS-00234 / P-AXIS-00233). However, the resulting distance between two interpolation points exceeds the permissible value range.		
Response	Class	3	Leadscrew error compensation is not activated.
Solution	Class	7	Correct the compensation value list of the leadscrew error compensation. Insert additional interpolation points P-COMP-00025 in the compensation value list to reduce the distance between the interpolation points.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		List index of affected interpolation point	
	%3:	Current value [increments]	
		Position of affected interpolation point in increments	
	%4:	Current value [increments]	
		Position of next interpolation point in increments	
	%5:	Incorrect value [increments]	
		Distance between the two interpolation points in increments.	
Error type	-		

ID 110481

Modification of parameter P-AXIS-00326 requires restart of the control.			
Description	Parameter *antr.torque_scale_denom* (P-AXIS-00326) is to be changed by updating an axis parameter list. To update P-AXIS-00326, the controller must be rebooted.		
Response	Class	1	Program abortWarning, no reaction. The value change is not adopted.
Solution	Class	1	Restart the controller after changing P-AXIS-00326.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		The value of P-AXIS-00326 is new in the axis parameter list.	
	%3:	Corrected value [-]	
		Old value of P-AXIS-00326.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110482

Invalid range of values for homing with evaluation of encoder overflow.			
Description	Axis parameter P-AXIS-00355 is out of value range.		
Response	Class	1	Warning output, correct this parameter to 0; this activates a drive type-dependent default assignment.
Solution	Class	1	Enter the correct value for P-AXIS-00355.
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110483

Different latch methods for probing/zero pulse search in telegram configured.

Description	Different latch variants (e.g. manufacturer-specific or according to CiA402) may not be combined with latch positions in the drive for probing or zero pulse search. The latch method is defined in the controller based on the configured cyclic command and actual value telegrams. Objects that belong to different latch methods cannot be combined. Example of a configuration of the cyclic telegram for probing according to the CANopen CiA402 profile: Command value telegram: • Touch probe status (0x60B9) • Touch probe position 1 positive value (0x60BA) • ... Actual value telegram: • Touch probe function (0x60B8) • ...		
Response	Class	3	
Solution	Class	6	Correct the configuration of the cyclic telegram
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:		
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Incorrect value [-]	
		Object belonging to the first latch method	
	%4:		
		Object belonging to the second latch method	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110484

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 110485

Invalid value for latch input used in zero pulse search is corrected.			
Description	Parameter P-AXIS-00364 was assigned an invalid value.		
Response	Class	1	Error message output and correct parameter to 1.
Solution	Class	1	Enter a valid value for P-AXIS-00364 (value range 1 .. 2).

Parameter	%1:	Logical axis number [-]
	%2:	Incorrect value [-]
	%3:	Lower limit value [-]
	%4:	Upper limit value [-]
	%5:	Corrected value [-]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110486

This drive type does not support feedforward balancing.			
Description	Feedforward balancing was enabled (P-AXIS-00361 ≠ 0) although it does not support this drive type.		
Response	Class	1	Warning and correction of parameter. Feedforward balancing is disabled.
Solution	Class	1	Disable feedforward balancing (assign the value 0 to P-AXIS-00361).
Parameter	%1:	Logical axis number	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [µs]	
	%3:	Corrected value [µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110487

Changing the operation mode "gantry master axis" requires restart of the control.			
Description	The operation of an axis as "master axis of a gantry link" (see P-AXIS-00015) may not be changed while the CNC kernel is running. The new value is not adopted; the old setting remains valid.		
Response	Class	0	Warning and correct parameter P-AXIS-00015
Solution	Class	0	Only change the operation mode “master axis of gantry link“ in configuration mode or if the CNC is not running.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:		
		Current axis operation mode P-AXIS-00015	
	%3:		
		Corrected P-AXIS-00015 operation mode of the axis.	
Error type	-		

ID 110488

Changing the operation mode "gantry slave axis" requires restart of the control.			
Description	The operation of an axis as "slave axis of a gantry link" (see P-AXIS-00015) may not be changed while the CNC kernel is running. The new value is not adopted; the old setting remains valid.		
Response	Class	0	Warning and correct parameter P-AXIS-00015
Solution	Class	0	Only change the operation mode “slave axis of gantry link“ in configuration mode or if the CNC is not running.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:		
		Current axis operation mode P-AXIS-00015	
	%3:		
		Corrected P-AXIS-00015 operation mode of the axis.	
Error type	-		

ID 110489

Invalid value for parameter P-AXIS-00353 is corrected.			
Description	The axis was parameterised as an edge banding axis *antr.edge_bending_input_nbr* (P-AXIS-00098) and an invalid value was specified for the edge banding input to be used at the drive (parameter P-AXIS-00353).		
Response	Class	1	Warning and correction of parameter. P-AXIS-00353 is corrected to 1
Solution	Class	1	In the axis parameter list, assign to parameter P-AXIS-00353 a value valid for the drive type used (P-AXIS-00020). In the drive, the latch function must be assigned to the digital input used.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Error value [-]	
	%3:	Lower limit value [-]	
	%4:	Upper limit value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110490

Filter quality is less than the minimum permissible value.			
Description	The filter quality P-AXIS-00080 (reciprocal of the filter bandwidth) of the specified axis filter is too small. The reciprocal filter bandwidth is used for band-stop and band-pass filters.		
Response	Class	1	Correct the filter quality P-AXIS-00080 to the value 1
Solution	Class	1	Increase the P-AXIS-00080 filter quality of the axis filter

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [-]
		Number of filter affected
	%3:	Current value [-]
		Incorrect value for filter quality P-AXIS-00080
	%4:	Incorrect value [-]
		Corrected value for filter quality P-AXIS-00080
	%5:	Corrected value [-]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110491

Value of filter time constant invalid.			
Description	The time constant P-AXIS-00357 of the PT1 axis filter is zero.		
Response	Class	1	Correct time constant P-AXIS-00357 to 10 milliseconds
Solution	Class	1	Increase the time constant P-AXIS-00357 of the filter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
	%2:	Current value [-]	
		Number of filter affected	
	%3:	Current value [-]	
		Filter type P-AXIS-00204	
	%4:	Incorrect value [1 µs]	
		Invalid value for time constant P-AXIS-00357	
	%5:	Corrected value [1 µs]	
		Corrected value for time constant P-AXIS-00357	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110496

Position window of axis filter too small.			
Description	The tolerance window P-AXIS-00351 specified for the axis filters was too small. The program waits for the position window to be reached, provided the axis filters are active (| unfiltered command value – filtered command value| < P-AXIS-00351) at reset, axis position request or exact stop. The permissible tolerance window cannot be specified as too small due to the restricted computational accuracy.		
Response	Class	1	Warning output, automatic correction of P-AXIS-00351.
Solution	Class	1	Correct the value in the parameter list
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		Incorrect value for position window P-AXIS-00351	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Smallest permissible value for position window P-AXIS-00351	
	%4:	Corrected value [0.1 µm or 0.0001°]	
		Corrected value of P-AXIS-00351	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110497

Velocity monitoring: factor for warning greater than for error.			
Description	Tolerance factor for excessive velocity warning is set higher than for an error.		
Response	Class	1	Value for warning is set to value for error. If velocity limit is exceeded, no warning is generated before an error occurs
Solution	Class	1	Set smaller value.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [0.1%]	
	%3:	Limit value [0.1%]	
	%4:	Corrected value [0.1%]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110498

Acceleration monitoring: factor for warning greater than for error.			
Description	Tolerance factor for excessive acceleration warning is set higher than for an error.		
Response	Class	1	Value for warning is set to value for error. If acceleration limit is exceeded, no warning is generated before an error occurs
Solution	Class	1	Set smaller value.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [0.1%]	
	%3:	Limit value [0.1%]	
	%4:	Corrected value [0.1%]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110499

Jerk monitoring: factor for warning greater than for error.			
Description	Tolerance factor for excessive jerk warning is set higher than for an error.		
Response	Class	1	Value for warning is set to value for error. If jerk limit is exceeded, no warning is issued before an error occurs.
Solution	Class	1	Set smaller value.

Parameter	%1:	Logical axis number [-]
	%2:	Current value [0.1%]
	%3:	Limit value [0.1%]
	%4:	Corrected value [0.1%]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110387

Adjusted distance resolution is less than 1.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Current value [-]	
	%4:	Corrected value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

2.11.3 ID-range 110500-110749

ID 110500

Only one HSC-filter is allowed per axis.			
Description	During start-up, the check of filter parameters of an axis detects that more than one HSC filters are configured. This is not permitted.		
Response	Class	1	The filter of the axis configured incorrectly is disabled. The start-up is continued
Solution	Class	1	Check and modify the configuration. Configure only one HSC filter on the first filter entry (filter[**0**].).
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the affected axis where the filter was incorrectly configured.	
	%2:	Current value [-]	
		Filter entry where the filter was configured.	
	%3:	Error value [-]	
		Filter prototype (P-AXIS-00153) which was configured in this filter:	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110501

Only first axis filter can be a HSC-filter.			
Description	During start-up, the check of filter parameters of an axis detects that an HSC filter is configured which is not placed on the first filter entry (filter[0].). This is not permitted.		
Response	Class	1	The filter of the axis configured incorrectly is disabled. The start-up is continued

Solution	Class	1	Modify the filter configuration of the axis so that the HSC filter is placed on the first filter entry (filter[**0**].).
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Index of axis filter	
	%3:	Error value [-]	
		Filter prototype : P-AXIS-00153	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110502

Unknown transformation id for axis-specific current position transformation configured.			
Description	The transformation id P-AXIS-00381 specified for the axis-specific actual value transformation is unknown.		
Response	Class	3	Input transformation is deactivated.
Solution	Class	7	Change transformation id.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
	%2:	Error value [-]	
		Unknown transformation id.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110503

Enable/disable of axis-specific current values transformation requires restart of the control.			
Description	An attempt was made to change the value of P-AXIS-00380 with an update of the axis parameter list. This is not possible; a change of P-AXIS-00380 requires a restart of the controller.		
Response	Class	1	Warning output, the value of P-AXIS-00380 not changed.
Solution	Class	0	Change parameter and restart controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Current value [-]	
		Changed value of P-AXIS-00380	
	%3:	Corrected value [-]	
		Old value of P-AXIS-00380	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110504

Changing the id of axis-specific current values transformation requires restart of the control.			
Description	An attempt was made to change the transformation id of the actual value transformation P-AXIS-00381 by updating the axis parameter list. This is not possible; a change requires a controller restart.		
Response	Class	0	Warning output, the value of P-AXIS-00381 is not changed.
Solution	Class	0	Change parameter and restart controller.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis
	%2:	Current value [-]
		Changed value of P-AXIS-00381
	%3:	Corrected value [-]
		Old value of P-AXIS-00381
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110505

Changing parameters of axis-specific current values transformation requires restart of the control.			
Description	An attempt was made to change the value of P-AXIS-00382 with an update of the axis parameter list. This is not possible; a change of P-AXIS-00382 requires a controller restart.		
Response	Class	1	Warning output, the value of P-AXIS-00382 is not changed.
Solution	Class	1	Change parameter and restart controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Current value [-]	
		Changed value of P-AXIS-00382	
	%3:	Corrected value [-]	
		Old value of P-AXIS-00382	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110506

Changing parameters of axis-specific current values transformation requires restart of the control.			
Description	An attempt was made to change the value of P-AXIS-00371 with an update of the axis parameter list. This is not possible, a change of P-AXIS-00371 requires a restart of the controller.		
Response	Class	0	Warning output, the value of P-AXIS-00371 is no changed.
Solution	Class	0	Change parameter and restart controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Current value [-]	
		Changed value of P-AXIS-00371	
	%3:	Corrected value [-]	
		Old value of P-AXIS-00371	
Error type	-		

ID 110507

Parameter id for axis-specific current position transformation invalid.			
Description	A parameter P-AXIS-00382 in the axis-specific actual value transformation has an invalid value.		
Response	Class	3	Warning output, the value of P-AXIS-00382 is not changed.

Solution	Class	7	Change parameter and restart controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Current value [-]	
		Parameterised transformation id P-AXIS-00381	
	%3:	Current value [-]	
		Changed value of P-AXIS-00382	
	%4:	Incorrect value [-]	
		Old value of P-AXIS-00382	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110508

Probing is not supported for this transformation.			
Description	The axis is configured as probing axis (see P-AXIS-00118) and, at same time, an axis-specific feedback value transformation is configured for the axis that does not support probing.		
Response	Class	3	Error message output; probing is disabled.
Solution	Class	7	P-AXIS-00118 must be disabled.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Current value [-]	
		Transformation if P-AXIS-00381 of the configured current value transformation.	
	%3:	Incorrect value [-]	
		Value of P-AXIS-00118.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00118.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110509

Inadmissible transformation id by command value transformation.			
Description	The transformation id parameter P-AXIS-00369 was assigned a value outside the valid value range.		
Response	Class	1	Warning output, corrected P-AXIS-00369 is set to 0.
Solution	Class	1	Set correct value for P-AXIS-00369
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Current value [-]	
		Parameterised transformation id P-AXIS-00369	
	%3:	Upper limit value [-]	
	%4:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110510

Inadmissible scaling factor for motor speed.			
Description	For a PROFIDRIVE, the scaling factor for motor speed (P-AXIS-00379) was set to zero in the parameter list. Only values greater than 0 are allowed.		
Response	Class	1	Error message output; the value is corrected to the default value (200000):
Solution	Class	1	Read out the correct value of P-AXIS-00379 from the drive (SIMODRIVE 611: P880, SINAMICS: P2000) and enter in the parameter list.
Parameter	%1:	Logical axis number [-]	
		axis number of the axis (see also P-AXIS-00016).	
	%2:	Current value [-]	
		Parameterised value of P-AXIS-00379.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00379	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110511

Only spindles can be operated without position control.			
Description	One axis is to be operated without position controller (controlled operation). This is only possible for spindles. See also P-AXIS-00320..		
Response	Class	1	Error message output, reset is locked.
Solution	Class	1	Parameterise a different operation mode for the axis (see P-AXIS-00320) and restart the controller.
Parameter	%1:	Logical axis number [-]	
		Log. axis number of the axis (see also P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type of the axis (see also P-AXIS-00020).	
	%3:	Incorrect value [-]	
		Operation mode configured for the axis (see also P-AXIS-00320).	
	%4:	Current value [-]	
		Configured axis type (see P-AXIS-00018)	
	%5:	Corrected value [-]	
		Corrected operation mode.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110512

Homing on encoder overflow and homing on zero pulse can not be active at the same time.			
Description	Homing to encoder overflow (see P-AXIS-00294) and homing to zero pulse search (see P-AXIS-00386) have been activated simultaneously.		
Response	Class	1	Error message output, reference to encoder overflow (P-AXIS-00294) is disabled.
Solution	Class	1	Set either P-AXIS-00294 or P-AXIS-00386 to zero (nil).

Parameter	%1:	Logical axis number [-]
		Log. Axis number (see also P-AXIS-00016).
	%2:	Current value [-]
		Value of P-AXIS-00386.
	%3:	Current value [-]
		Value of P-AXIS-00294.
	%4:	Corrected value [-]
		Corrected value of P-AXIS-00294.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110513

Homing on zero pulse is only possible with CNC controlled homing.			
Description	Parameter P-AXIS-00386 was set to activate a CNC-controlled homing with drive support, but the type of homing (see parameter P-AXIS-00299) is not set to "CNC_CONTROLLED".		
Response	Class	1	Error message output, P-AXIS-00286 is set to zero.
Solution	Class	1	Either parameterise a CNC-controlled homing by setting P-AXIS-00299 to the value "CNC_CONTROLLED", or deactivate the drive support by assigning the value 0 to parameter P-AXIS-00386.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:	Current value [-]	
		Parameterised value of P-AXIS-00299.	
	%3:	Current value [-]	
		Parameterised value from P-AXIS-00386.	
	%4:	Corrected value	
		Corrected value of P-AXIS-00386.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110514

Invalid value for assignment of control and status bits for homing.			
Description	Parameter P-AXIS-00387 was assigned an invalid value.		
Response	Class	1	Error message output and correct P-AXIS-00387 to the value 1.
Solution	Class	1	The permissible value range forP-AXIS-00387 is [1, 16].
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:	Incorrect value [-]	
		Incorrect value of P-AXIS-00387	
	%3:	Lower limit value [-]	
		Permissible minimum value of P-AXIS-00387	
	%4:	Upper limit value [-]	
		Permissible maximum value of P-AXIS-00387	
	%5:	Corrected value [-]	
		Corrected value of P-AXIS-00387	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110515

Using real-time bits for probing and homing at the same time not possible.			
Description	The axis is to be used for probing, so the real-time status and control bits of the SERCOS control and status word are assigned (P-AXIS-00118). At the same time, the real-time control and status bits are to be used for the drive-supported CNC-controlled homing, (P-AXIS-00387 has the value 1). This double use of the real-time bits is not possible.		
Response	Class	1	Probing is deactivated.
Solution	Class	1	Configure transfer of the control and status bits for homing via signal control and signal status word and adapt P-AXIS-00387 accordingly.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:	Incorrect value [-]	
		Value of P-AXIS-00387.	
	%3:	Incorrect value [-]	
		Value of P-AXIS-00118.	
	%4:	Incorrect value [-]	
		Corrected value of P-AXIS-00118.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110516

This parameter is not supported for this drive type.			
Description	Parameter P-AXIS-00388 was set to a value unequal to 0 although it is not supported for the drive type used.		
Response	Class	3	P-AXIS-00388 is set to zero.
Solution	Class	1	Assign the value zero to P-AXIS-00388.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:		
		Configured drive type (P-AXIS-00020).	
	%3:	Incorrect value [-]	
		Value of P-AXIS-00388.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00388.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110517

Invalid reference value for scaling of acceleration feedforward.			
Description	Parameter P-AXIS-00392 was set to zero.		
Response	Class	1	P-AXIS-00392 is set to the value 1.
Solution	Class	1	Set the correct value for P-AXIS-00392.

Parameter	%1:	Logical axis number [-]
		Log. axis number (see P-AXIS-00016).
	%2:	Incorrect value [-]
		Incorrect value of P-AXIS-00392.
	%3:	Lower limit value [-]
		Limit value of P-AXIS-00392.
	%4:	
		Corrected value of P-AXIS-00392.
	%5:	Corrected value [-]
Error type	2, Error message by data transfer from parameter list into control device. .	

ID 110518

Invalid value for motor load.			
Description	Parameter P-AXIS-00391 was set to zero.		
Response	Class	1	P-AXIS-00391 is set to the default value (1e-6).
Solution	Class	1	Set the correct value for P-AXIS-00391.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see P-AXIS-00016).	
	%2:	Incorrect value [motor load in kg or kg*m^2]	
		Incorrect value of P-AXIS-00391.	
	%3:	Corrected value [-]	
		Limit value of P-AXIS-00391.	
	%4:		
		Corrected value of P-AXIS-00391.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110519

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 110520

Not enough memory available for the compensation list.

Description	There are two possible causes for this error message: 1. A compensation value list was assigned in P-STUP-00036 for axis compensations. However, the CNC was unable to allocated the required memory from the operation system. A possible solution is to delete the compensation value list or increase the available memory. 2. A compensation value list was not assigned to the axis in the start-up list with P-STUP-00036 but a compensation value list was still loaded during the controller runtime. A solution in this case is to add a compensation value list for this axis P-STUP-00036 in the start-up list.		
Response	Class	3	Start-up of CNC is aborted
Solution	Class	7	Delete compensation value list or increase available memory.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 110521

Homing on zero pulse is not enabled.			
Description	Parameter P-AXIS-00386 was set, but homing with zero pulse search (see P-AXIS-00084) is not active.		
Response	Class	1	Error message output, P-AXIS-00386 is disabled.
Solution	Class	1	Set P-AXIS-00084 to zero.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see also P-AXIS-00016).	
	%2:	Error value [-]	
		Value of P-AXIS-00084.	
	%3:	Current value[-]	
		Value of P-AXIS-00386.	
	%4:	Error value [-]	
		Corrected value of P-AXIS-00386.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110522

CNC controlled homing with zero pulse search in the drive is not supported for spindles.			
Description	The use of P-AXIS-00386 in connection with spindles or SAIs (single-axis interpolators) is currently not possible.		
Response	Class	1	Error message output, P-AXIS-00386 is disabled.
Solution	Class	1	Set P-AXIS-00386 to zero.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:	Current value [-]	
		Value of P-AXIS-00018.	
	%3:	Current value [-]	
		Value of P-AXIS-00250.	
	%4:	Incorrect value [-]	
		Value of P-AXIS-00386.	
	%5:	Corrected value [-]	
		Corrected value of P-AXIS-00386.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110523

Software limit switch was adjusted because of resolution settings.			
Description	The positive software limit switch P-AXIS-00178 or the negative software limit switch P-AXIS-00177 is out of range. This range is defined by the distance resolution of the measurement system.		
Response	Class	1	Initialise the software limit switch
Solution	Class	1	Correct software limit switch in parameter list
Parameter	%1:	Logical axis number [-]	
		Log. axis number P-AXIS-00016	
	%2:	Incorrect value [-]	
		Software limit switches P-AXIS-00177/P-AXIS-00178	
	%3:	Current value [-]	
		Numerator of distance resolution P-AXIS-00234.	
	%4:	Current value [-]	
		Denominator of the distance resolution P-AXIS-00233.	
	%5:	Corrected value [-]	
		Corrected value of software limit switch.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110524

Modulo range exceeds data range because of resolution settings.			
Description	Modulo range defined through the parameters P-AXIS-00127 and P-AXIS-00126 was exceeded. This is caused by a change in distance resolution. This is set via parameters P-AXIS-00234 and P-AXIS-00233.		
Response	Class	3	None
Solution	Class	7	Check and modify the modulo limits and the parameters of distance resolution.
Parameter	%1:	Logical axis number [-]	
		Log. axis number P-AXIS-00016	
	%2:	Current value [-]	
		P-AXIS-00126	
	%3:	Limit value [-]	
		Maximum possible value for P-AXIS-00126 with constant distance resolution	
	%4:	Current value [-]	
		P-AXIS-00127	
	%5:	Limit value [-]	
		Minimum possible value for P-AXIS-00127 with constant distance resolution	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110528

The parametrised controller dead band is bigger than the position window.

Description	The parameterised position controller dead band (P-AXIS-00395) is bigger than the position window (P-AXIS-00236) set for the axis.		
Response	Class	1	Error message output, P-AXIS-00395 is limited to the value of P-AXIS-00236.
Solution	Class	1	Either reduce P-AXIS-00395 or increase P-AXIS-00236.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		Value of P-AXIS-00395.	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Value of P-AXIS-00236.	
	%4:	Corrected value [0.1 µm or 0.0001°]	
		Corrected value of P-AXIS-00395.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110529

Controller deadband is zero (nil) after incremental conversion.			
Description	The set up position controller deadband scaled in encoder increments (P-AXIS-00395) is zero (nil), the parameter is ineffective.		
Response	Class	1	Warning output.
Solution	Class	1	P-AXIS-00395 must be increased.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:	Current value [0.1 µm or 0,0001°]	
		Value of P-AXIS-00395 in encoder increments.	
	%3:	Current value [-]	
		Set up encoder resolution.	
	%4:	Current value [0.1 µm or 0,0001°]	
		Metric value of P-AXIS-00395.	
Error type	-		

ID 110530

During parameter update axis is not in standstill.			
Description	A change attempt was made, e.g. by updating the list or by the #MACHINE DATA [] command. The axis must be at standstill. This is not the case; the axis is interpolated.		
Response	Class	1	The change to the axis parameters is not executed.
Solution	Class	1	Stop the axis before executing a change in parameters.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
Error type	6, Error message by data transfer to control device.		

ID 110531

Inadmissible operation mode by manual mode.

Description	An invalid value was assigned to axis parameter P-AXIS-00139.		
Response	Class	1	Warning output, P-AXIS-00139 is set to zero.
Solution	Class	1	Parameterise the correct value for P-AXIS-00139.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Limit value [-]	
	%4:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110532

This parameter is not supported for this drive type.			
Description	Parameter P-AXIS-00403 was assigned a value unequal to 0 although the drive type set (see P-AXIS-00020) does not support this function (setting an offset to the drive absolute position).		
Response	Class	1	Warning output, P-AXIS-00403 is set to 0.
Solution	Class	1	Set P-AXIS-00403 to 0.
Parameter	%1:	Logical axis number [-]	
		Log. axis number (see also P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type (see also P-AXIS-00020):	
	%3:	Incorrect value [-]	
		Parameterised value of P-AXIS-00403.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00403.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110533

Sign of actual value and set value must be changed jointly.			
Description	In the operation mode of an axis P-AXIS-00015, if ALLOW_RESOLUTION_CHANGE = 0x400000 is set, the value of parameters P-AXIS-00230 and P-AXIS-00231 for the sign reversal of manipulated variable and actual value can be changed by updating the axis parameter list. However, in order to prevent a runaway of the drive from running through, these two parameters can only be changed together.		
Response	Class	1	Warning, no acceptance of new parameter P-AXIS-00230 and P-AXIS-00231
Solution	Class	1	Only change sign of manipulated variable and actual value jointly.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Current value [-]
		Current value for sign reversal of actual value P-AXIS-00230
	%3:	Current value [-]
		Current value for sign reversal of manipulated variable P-AXIS-00231
	%4:	Error value [-]
		New, invalid value for sign reversal of current value P-AXIS-00230
	%5:	Error value [-]
		New, invalid value for sign reversal manipulated variable P-AXIS-00231
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110538

Changing the coarsening factor of encoder resolution requires a restart of the control.			
Description	The axis parameter P-AXIS-00405 can be used to coarsen the encoder resolution of the drive. After changing the parameter P-AXIS-00405, a controller restart is necessary.		
Response	Class	0	Warning, the old parameter value P-AXIS-00405 remains valid.
Solution	Class	0	After changing the parameter P-AXIS-00405, restart the controller.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of axis affected
	%2:		
			New value for coarsening factor P-AXIS-00405
	%3:		
			Old but still valid value for coarsening factor P-AXIS-00405
Error type	-		

ID 110539

This drive type does not support a coarsening factor for encoder resolution.			
Description	The axis parameter P-AXIS-00405 can be used to coarsen the encoder resolution of the drive. However, this is not supported for the given drive type.		
Response	Class	1	Warning, parameter P-AXIS-00405 is set to zero.
Solution	Class	1	Correct parameter P-AXIS-00405.
Parameter	%1:		Logical axis number [-]
			Logical axis number P-AXIS-00016 of axis affected
	%2:		Drive type [-]
			Drive type P-AXIS-00020 of the affected axis
	%3:		Incorrect value [-]
			Incorrect value for coarsening factor P-AXIS-00405
	%4:		Corrected value [-]
			Corrected value for coarsening factor P-AXIS-00405
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110540

Coarsening factor for encoder resolution exceeds valid range of values.			
Description	The axis parameter P-AXIS-00405 can be used to coarsen the encoder resolution of the drive. The given value is however invalid.		
Response	Class	1	Warning and correct parameter P-AXIS-00405
Solution	Class	1	Correct parameter P-AXIS-00405
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Incorrect value [-]	
		Incorrect value for coarsening factor P-AXIS-00405	
	%4:	Upper limit value [-]	
		Maximum permissible value for coarsening factor P-AXIS-00405	
	%5:	Corrected value [-]	
		Corrected value for coarsening factor P-AXIS-00405	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110541

Coarsening of encoder resolution requires drive command/actual values without modulo treatment.			
Description	The axis parameter P-AXIS-00405 can be used to coarsen the encoder resolution of the drive. However, this requires drive command values and actual values for which no upper module calculation is performed (see P-AXIS-00122, P-AXIS-00123).		
Response	Class	1	Warning, parameter P-AXIS-00405 is set to zero.
Solution	Class	1	Correct parameter P-AXIS-00405. Check if the drive supports a linear coordinate system.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [-]	
		Incorrect value for coarsening factor P-AXIS-00405	
	%3:	Current value [-]	
		Normalisation of drive command position P-AXIS-00122	
	%4:	Current value [-]	
		Normalisation of drive actual position P-AXIS-00123	
	%5:	Corrected value [-]	
		Corrected value for coarsening factor P-AXIS-00405	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110543

Parameter P-AXIS-00408 only used for EtherCAT drive bus.			
Description	The parameter "permissible_telegram_failures" P-AXIS-00406 specifies how many drive telegrams may fail before the CNC considers the drive bus to be interrupted. This parameter is only supported for drives with EtherCAT fieldbus (CANopen or SERCOS over EtherCAT).		
Response	Class	1	Warning, value of parameter P-AXIS-00406 is corrected and set to zero.

Solution	Class	1	Correct the value of parameter P-AXIS-00406.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Incorrect value [-]	
		Incorrect value for P-AXIS-00406	
	%4:	Corrected value [-]	
		Corrected value for P-AXIS-00406	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110544

Parameter P-AXIS-00408 greater than permissible maximum.			
Description	In the parameter permissible_telegram_failures (see P-AXIS-00409), the number of permitted bus failures for an EtherCAT fieldbus can be specified before the controller reports an error. However, the given value exceeds valid range.		
Response	Class	1	Warning and correct the parameter
Solution	Class	1	Correct parameter P-AXIS-00409
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Incorrect value [-]	
		Incorrect value for permissible bus errors P-AXIS-00409	
	%4:	Upper limit value [-]	
		Maximum permissible parameter value	
	%5:	Corrected value [-]	
		Corrected value for permissible bus errors P-AXIS-00409	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110545

Monitoring of homing distance is only possible for spindles and rotary axes.			
Description	Parameter P-AXIS-00412 was set to a value other than zero although the axis does not have the axis type spindle or rotary axis (see also P-AXIS-00018). Parameter P-AXIS-00412 can only be used together with spindles or rotary axes.		
Response	Class	3	Warning output, P-AXIS-00412 is set to zero.
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
		Set value of P-AXIS-00412.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00412.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110546

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 110547

Distance control not supported for this drive type.			
Description	In the axis parameter list of the axis, distance control is activated by setting P-AXIS-00328 to 1, but the configured drive type (see P-AXIS-00020) does not support distance control.		
Response	Class	1	Warning output, P-AXIS-00328 is set to zero (distance control is disabled).
Solution	Class	1	Disable distance control or use a different drive type.
Parameter	%1:	Logical axis number [-]	
	%2:	Drive type [-]	
	%3:	Incorrect value [-]	
	%4:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110548

Axis reference test is not supported for this drive type.			
Description	Parameter P-AXIS-00426 was assigned a value so that reference monitoring is activated although the drive type set in P-AXIS-00020 does not support reference monitoring. See FCT-M1// Section: Reference loss(reference monitoring)		
Response	Class	1	Reference monitoring is deactivated.
Solution	Class	1	Assign P-AXIS-00426 the value "".
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
		Current value of P-AXIS-00426.	
	%3:	Current value [-]	
		Current value of P-AXIS-00020.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00426.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110549

Invalid bit number for additional 'Ready for Power test'.			
Description	Reference monitoring was activated but the bit number configured in P-AXIS-00425 is invalid for reference monitoring. See FCT-M1// Section: Reference loss (reference monitoring) The smallest permissible value for P-AXIS-00425 is 0. Depending on the length of the configured telegram element used to transfer the referenced information, the maximum value of this parameter is:		
	Telegram element length	Maximum value for P-AXIS-00425	
	2	15	
	4	31	
	This error message also occurs if no value was configured for P-AXIS-00425.		
Response	Class	3	Reference monitoring is deactivated.
Solution	Class	1	Assign a valid value to P-AXIS-00425 or disable reference monitoring.
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
		Current value of P-AXIS-00425.	
	%3:	Limit value [-]	
		Limit value for P-AXIS-00425.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00425.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110550

Changing of P-AXIS-00426 requires restart of the control.			
Description	An attempt was made to change the parameter *antr.reference_check.element_name* (P-AXIS-00426) by updating lists or using the NC command #MACHINE DATA[ ]; this is not possible.		
Response	Class	1	Output a warning; the new value is not accepted.
Solution	Class	1	Save changed configuration and restart controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Current value [-]	
		New value of P-AXIS-00426.	
	%3:	Corrected value [-]	
		Current value of P-AXIS-00426.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110551

Path parameter for dynamic weighting are not sorted in ascending order.	
Description	When configuring distance-dependent dynamic weighting, the path parameters (P-AXIS-00432) are not configured in ascending order.

Response	Class	1	Warning output, the dynamic weighting factors of the entry with the smaller path parameter are replaced by the dynamic weighting factors of the entry with the larger path parameter.
Solution	Class	1	Sort the entries for the dynamic weighting factors in ascending order.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [0.1 10^-3 mm or ø]	
	%3:	Current value [0.1 10^-3 mm or ø]	
	%4:	Corrected value [0.1 10^-3 mm or ø]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110552

Invalid number format in list or permissible range of values exceeded.			
Description	During interpretation of a list parameter, an invalid number format is detected or the interpreted parameter value exceeds the permissible value range. More information about type of list, invalid number formats etc. is contained in the data of the message.		
Response	Class	1	Interpretation of list is continued. The parameter value is limited to the maximum or minimum permissible value or set to zero if the number format is invalid.
Solution	Class	1	Correct the number format in the corresponding parameter list.
Error type	-		

ID 110553

Link to axis not allowed (link_to) with multi axis link (multi_link[]).			
Description	The parameter P-AXIS-00101 can be used to connect a logical axis of the interpolator to a physical axis in the position controller. This is not permitted in conjunction with gear coupling and automatic calculation of the input variable P-AXIS-00436.		
Response	Class	3	Start-up aborted
Solution	Class	7	Set parameter P-AXIS-00101 to zero or deactivate gear link
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis	
	%2:	Logical axis number [-]	
		Logical axis number of the lead axis P-AXIS-00383	
	%3:	Current value [-]	
		Value of P-AXIS-00436	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110557

Coarsening of encoder resolution for encoder with modulo positions not possible.	
Description	

Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
	%3:	Current value [-]	
	%4:	Current value [-]	
	%5:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110558

Denominator of scaling factor for predictive command velocity is zero.			
Description	The denominator (P-AXIS-00469) of the scaling factor for predictive command velocity is 0. This is not permitted. The scaling factor is made up of: Factor = P-AXIS-00468 / P-AXIS-00469		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00469
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Error value [-]	
		Parameterised value of P-AXIS-00469	
	%3:	Corrected value [-]	
		New value for P-AXIS-00469	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110559

Denominator of scaling factor for predictive command acceleration is zero.			
Description	The denominator (P-AXIS-00471) of the scaling factor for predictive command velocity is 0. This is not permitted. The scaling factor is made up of: Factor = P-AXIS-00470 / P-AXIS-00471		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-AXIS-00471
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Error value [-]	
		Parameterised value of P-AXIS-00471	
	%3:	Corrected value [-]	
		New value for P-AXIS-00471	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110560

Position window for rapid traverse cannot be used with spindles.

Description	For a spindle (see P-AXIS-00018), a value > 0 was parameterised for the axis parameter P-AXIS-00472. The parameter P-AXIS-00472 cannot be used with spindles because spindles cannot be moved at rapid traverse.		
Response	Class	1	Warning output, P-AXIS-00472 is set to zero.
Solution	Class	1	Assign P-AXIS-00472 a value of <= 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value [-]	
		Parameterised value of P-AXIS-00018.	
	%3:	Incorrect value [-]	
		Parameterised value of P-AXIS-00472.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00472.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110561

The parametrised controller dead band is bigger than the rapid traverse position window.			
Description	The parameterised position controller dead band (P-AXIS-00395) is greater than the position window for rapid traverse (P-AXIS-00472) set for the axis.		
Response	Class	1	Error message output, P-AXIS-00395 is limited to the value of P-AXIS-00472 .
Solution	Class	1	Either reduce P-AXIS-00395 or increase P-AXIS-00472.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (see also P-AXIS-00016).	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		Value of P-AXIS-00395.	
	%3:	Limit value [0.1 µm or 0.0001°]	
		Value of P-AXIS-00472.	
	%4:	Corrected value [0.1 µm or 0.0001°]	
		Corrected value of P-AXIS-00395.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110562

Maximum time of drive reset is less than the permissible minimum value.			
Description	The parametrised value of P-AXIS-00484 is lower than the permissible minimum value.		
Response	Class	1	Warning output, P-AXIS-00484 is limited to minimum value.
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
		Logical axis number	
	%2:	Incorrect value [1 µs]	
		Parameterised value of P-AXIS-00484.	
	%3:	Lower limit value [1 µs]	
		Permissible minimum value of P-AXIS-00484.	
	%4:	Corrected value [1 µs]	
		Corrected value of P-AXIS-00484.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110563

Relative homing of gantry slave axis is not supported for this drive type.			
Description	For a gantry slave axis, the 'homing relative to encoder overflow' referencing method was parameterised by assigning P-AXIS-00393 a value of 1 although this homing method is not available for the type of drive used.		
Response	Class	3	Warning output, corrected value of P-AXIS-00393 is set to 0.
Solution	Class	7	Assign the value 0 to parameter P-AXIS-00393 in the axis parameter list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016).	
	%2:	Drive type [-]	
		Drive type of the axis (see P-AXIS-00020).	
	%3:	Current value [-]	
		Incorrect value for P-AXIS-00393.	
	%4:	Corrected value [-]	
		Corrected value for P-AXIS-00393.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110564

Relative homing of gantry slave axis is not supported for this drive type.			
Description	The parameter P-AXIS-00488 can delay the output of the position lag message if the permissible position lag is exceeded (see [FCT-A1// Section:Type 4: Velocity-independent method]). This permits short-term excesses of the permissible position lag, e.g. in dynamic phases. However, the maximum specifiable time delay is limited for safety reasons.		
Response	Class	1	Warning output and limit the parameter P-AXIS-00488 to the maximum permitted delay time
Solution	Class	1	Correct parameter P-AXIS-00488
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [1 µs]	
		Invalid parameter value P-AXIS-00488	
	%3:	Upper limit value [1 µs]	
		Maximum permitted value for parameter P-AXIS-00488	
	%4:	Corrected value [1 µs]	
		Corrected parameter value P-AXIS-00488	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110565

Axis is no modulo axis in C axis machining.	
Description	To use the transformation functions #FACE, #CYL[], #CYL ORI LATERAL, a rotary "CAX" modulo axis is required with the setting in axis mode "kenngr.achsmode". In this case, this axis is not configured as a modulo axis. The #CYL[] function cannot be used.

Response	Class	1	No reaction
Solution	Class	1	Use correct MDS axis setting with “kenngr.achsmode 0x204”
Parameter	%1:	Logical axis number [-]	
		Logical axis number of C axis	
	%2:	Current value [-]	
		Current axis mode	
	%3:	Corrected value [-]	
		Corrected axis mode	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110566

Invalid mode of velocity override.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
	%3:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110567

Change of P-AXIS-00362 requires restart of the control.			
Description	An attempt was made to change the axis parameter list entry P-AXIS-00362 by updating the axis parameter list but the ALLOW_RESOLUTION_CHANGE bit in P-AXIS-00015 is not set. Since a change of P-AXIS-00362 changes the resolution of the axis, this change is rejected and a restart of the controller is required.		
Response	Class	0	Warning output, changed value is not adopted.
Solution	Class	0	Restart of the controller with changed parameter P-AXIS-00362
Parameter	%1:		
		Axis number	
	%2:		
		Incorrect value	
	%3:		
		Corrected value	
Error type	-		

ID 110568

Change of P-AXIS-00363 requires restart of the control.			
Description	An attempt was made to change the axis parameter list entry P-AXIS-00363 by updating the axis parameter list but the ALLOW_RESOLUTION_CHANGE bit in P-AXIS-00015 is not set. Since a change of P-AXIS-00363 changes the resolution of the axis, this change is rejected and a restart of the controller is required.		
Response	Class	0	Warning output, changed value is not adopted.
Solution	Class	0	Restart of the controller with changed parameter P-AXIS-00363

Parameter	%1:	
		Axis number
	%2:	
		Incorrect value
	%3:	
		Corrected value
Error type	-	

ID 110569

Positive absolute ACS manual mode limit is smaller than negative limit.			
Description	The absolute positive movement limit P-AXIS-00493 for the motion range in active manual mode is smaller than the negative software limit switch P-AXIS-00492. Monitoring is disabled.		
Response	Class	1	None.
Solution	Class	1	Set positive movement limit P-AXIS-00493 greater than the negative movement limit P-AXIS-00492.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value for positive offset limit P-AXIS-00493.	
	%3:	Current value [-]	
		Current value of negative offset limit P-AXIS-00492.	
	%4:	Corrected value [-]	
		Corrected value for positive offset limit P-AXIS-00493.	
	%5:	Corrected value [-]	
		Corrected value for negative offset limit P-AXIS-00492.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110570

Negative absolute ACS manual mode limit is smaller than negative limit switch.			
Description	The absolute negative movement limit P-AXIS-00492 for the motion range in active manual mode is smaller than the negative software limit switch P-AXIS-00177. The negative absolute movement limit is set to the negative software limit switch.		
Response	Class	1	None.
Solution	Class	1	Set negative movement limit P-AXIS-00492 greater than negative software limit switch.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of negative movement limit P-AXIS-00492.	
	%3:	Limit value [-]	
		Current value for negative software limit switch P-AXIS-00177.	
	%4:	Corrected value [-]	
		Corrected value for negative movement limit P-AXIS-00492.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110571

Positive absolute ACS manual mode limit is greater than positive limit switch.			
Description	The absolute positive movement limit P-AXIS-00493 for the motion range in active manual mode is greater than the negative software limit switch P-AXIS-00178. The absolute positive movement limit is set to the positive software limit switch.		
Response	Class	1	None.
Solution	Class	1	Set positive movement limit P-AXIS-00493 smaller than positive software limit switch.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value for positive movement limit P-AXIS-00493.	
	%3:	Limit value [-]	
		Current value for positive software limit switch P-AXIS-00178.	
	%4:	Corrected value [-]	
		Corrected value for positive movement limit P-AXIS-00493.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110572

Number of filter cycles for distance control exceeds valid range.			
Description	The distance control (see FCT-M3]) uses an average value filter to smooth the sensor signal. However, the number of specified filter cycles for this filter exceeds the maximum permissible value.		
Response	Class	1	Warning output and limit the filter cycles to the maximum permissible value
Solution	Class	1	Correct parameter P-AXIS-00413
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Incorrect value [-]	
		Incorrect value for P-AXIS-00413	
	%3:	Corrected value [-]	
		Corrected value P-AXIS-00413	
	%4:	Upper limit value [-]	
		Maximum permissible number of filter cycles P-AXIS-00413	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110573

Switch on delay for I position controller exceeds maximum value.			
Description	The value set for parameter P-AXIS-00497 is greater than the maximum permissible value.		
Response	Class	1	Warning output, limit value to maximum value.
Solution	Class	1	Reduce the value of P-AXIS-00497

Parameter	%1:	Logical axis number [-]
		Axis number
	%2:	Incorrect value [1 µs]
		Set value of P-AXIS-00497
	%3:	Upper limit value [1 µs]
		Permissible maximum value.
	%4:	Corrected value [1 µs]
		Corrected value of P-AXIS-00497
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110574

Integral time for I position controller exceeds maximum value.			
Description	The value set for parameter P-AXIS-00495 is greater than the maximum permissible value.		
Response	Class	1	Warning output, P-AXIS-00495 is set to zero.
Solution	Class	1	Reduce the value of P-AXIS-00495
Parameter	%1:	Logical axis number [-]	
		Axis number	
	%2:	Incorrect value [1 µs]	
		Set value of P-AXIS-00495	
	%3:	Upper limit value [1 µs]	
		Permissible maximum value.	
	%4:	Corrected value [1 µs]	
		Corrected value of P-AXIS-00495	
	%5:	Current value [-]	
		Gear index	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110575

Integral time for I position controller is below minimum value.			
Description	The value set for parameter P-AXIS-00495 is lower than the minimum value permissible value.		
Response	Class	1	Warning output, P-AXIS-00495 is set to zero.
Solution	Class	1	Increase value of P-AXIS-00495.
Parameter	%1:	Logical axis number [-]	
		Axis number	
	%2:	Incorrect value [1 µs]	
		Set value of P-AXIS-00495	
	%3:	Lower limit value [1 µs]	
		Permissible minimum value.	
	%4:	Corrected value [1 µs]	
		Corrected value of P-AXIS-00495	
	%5:	Current value [-]	
		Gear index	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110576

Limit for I position controller output exceeds maximum.			
Description	The value set for parameter P-AXIS-00496 is greater than the maximum permissible value.		
Response	Class	1	Warning output, P-AXIS-00496 is limited to the maximum value.
Solution	Class	1	Reduce the value of P-AXIS-00496
Parameter	%1:	Logical axis number [-]	
		Axis number	
	%2:	Incorrect value [1 µs]	
		Set value of P-AXIS-00496	
	%3:	Upper limit value [1 µs]	
		Permissible maximum value.	
	%4:	Corrected value [1 µs]	
		Corrected value of P-AXIS-00496	
	%5:	Current value [-]	
		Gear index	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110577

Invalid value for Parameter P-AXIS-00388.			
Description	An invalid value was assigned to axis parameter P-AXIS-00388. For SERCOS drives the values [1, 2, 3] are valid.		
Response	Class	3	Warning output, value of P-AXIS-00388 is set to 1.
Solution	Class	1	Assign a valid value to axis parameter P-AXIS-00388.
Parameter	%1:	Logical axis number [-]	
	%2:		
		Drive type (P-AXIS-00020).	
	%3:	Incorrect value [-]	
		Parameterised value of P-AXIS-00388	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00388.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110578

Error reaction DRIVE_ERROR_FROM_MESSAGE configured but error code not in cyclic telegram.			
Description			
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Logical axis number [-]
	%2:	Current value [-]
	%3:	Incorrect value [-]
	%4:	Current value [-]
	%5:	Corrected value [-]
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110579

P-AXIS-00511 may not be 0.			
Description	Parameter P-AXIS-00511 was set to zero; this value is not permitted.		
Response	Class	1	Error message output, value corrected to 1.
Solution	Class	1	Warning output, automatic correction of the value.
Parameter	%1:	Logical axis number [-]	
		Axis number	
	%2:	Incorrect value [-]	
		Set value of P-AXIS-00511.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00511.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110580

P-AXIS-00512 may not be 0.			
Description	Parameter P-AXIS-00512 was set to zero; this value is not permitted.		
Response	Class	1	Error message output, value corrected to 1.
Solution	Class	1	Warning output, automatic correction of the value.
Parameter	%1:	Logical axis number [-]	
		Axis number	
	%2:	Incorrect value [-]	
		Set value of P-AXIS-00512.	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00512.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110581

Changing of P-AXIS-00511 requires restart of the control.			
Description	The value P-AXIS-00511 is to be changed by updating a parameter list; this is only permitted if the "ALLOW_RESOLUTION_CHANGE" **bit** is set in P-AXIS-00015.		
Response	Class	0	Error message output, new value is not adopted.
Solution	Class	0	In parameter P-AXIS-00015 set the "ALLOW_RESOLUTION_CHANGE" bit or change P-AXIS-00511 by controller restart.

Parameter	%1:	
		Axis number
	%2:	
		Set value of P-AXIS-00511.
	%3:	
		Corrected (old) value of P-AXIS-00511.
Error type	-	

ID 110582

Change of P-AXIS-00512 requires restart of the control.			
Description	The value P-AXIS-00512 is to be changed by updating a parameter list; this is only permitted if the "ALLOW_RESOLUTION_CHANGE" bit is set in P-AXIS-00015.		
Response	Class	0	Error message output, new value is not adopted.
Solution	Class	0	In parameter P-AXIS-00015 set the "ALLOW_RESOLUTION_CHANGE" bit or change P-AXIS-00512 by controller restart.
Parameter	%1:		
		Axis number	
	%2:		
		Set value of P-AXIS-00512.	
	%3:		
		Corrected (old) value of P-AXIS-00512.	
Error type	-		

ID 110583

Only the last filter can be a delay filter for positions.			
Description	By using parameter P-AXIS-00513 in conjunction with a delay filter, the position setpoint output can be delayed at a deceleration relative to the velocity and feedforward values. When multiple filters per axis are used, the delay filter must be the last filter in the filter chain. If this condition is not fulfilled, the system issues this error message.		
Response	Class	3	Warning output, delay filter is disabled.
Solution	Class	7	Delay filter must be configured as last filter.
Parameter	%1:	Logical axis number [-]	
		Log. axis number.	
	%2:	Current value [-]	
		Index of last filter	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110584

Suppression of homing only possible for axis with absolute path measurement system.			
Description	Axis parameter P-AXIS-00299 can be used to suppress homing for an axis with the IGNORE_ABS_POS setting. However, this is only permitted for axes with an absolute path measurement system (see P-AXIS-00014).		
Response	Class	1	Warning and correct homing type P-AXIS-00299

Solution	Class	1	Correct the homing type P-AXIS-00299 or set the parameter P-AXIS-00014 to the value 1 if an absolute path measurement system is present.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Current value [-]	
		Current value of parameter P-AXIS-00014 for the identifier of an absolute path measurement system	
	%4:	Incorrect value [-]	
		Incorrect setting for parameter P-AXIS-00299 homing type	
	%5:	Corrected value [-]	
		Corrected setting for P-AXIS-00299 homing type	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110585

Selected source for the probing signal is invalid.			
Description	Parameter P-AXIS-00516 "kenngr.measure.signal" defines the probing signal source for use with a measurement run (e.g. G100). However, the setting found is invalid.		
Response	Class	1	Warning and correct parameter P-AXIS-00516
Solution	Class	1	Correct the parameter P-AXIS-00516 for the probing signal source
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis	
	%3:	Incorrect value [-]	
		Incorrect setting for probing signal source P-AXIS-00516	
	%4:	Corrected value [-]	
		Corrected setting for P-AXIS-00516 probing signal source	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110586

Parameterised probing signal source not supported for this drive type.			
Description	Parameter P-AXIS-00516 "kenngr.measure.signal" defines the probing signal source for use with a measurement run (e.g. G100). However, the given setting is unsupported with the drive type of the axis.		
Response	Class	1	Warning and correct parameter P-AXIS-00516
Solution	Class	1	Correct the parameter P-AXIS-00516 for the probing signal source
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis	
	%3:	Incorrect value [-]	
		Incorrect setting for probing signal source P-AXIS-00516	
	%4:	Corrected value [-]	
		Corrected setting for P-AXIS-00516 probing signal source	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110587

Selected probing input is not supported by the drive.			
Description	Parameter P-AXIS-00517 "kenngr.measure.input" defines the number of the probing input used for a measurement run (e.g. G100). However, the setting found is invalid because the selected probing input is not available on the drive. Depending on the drive type, permissible values for the probing input are 1 or 2.		
Response	Class	1	Warning and correct parameter P-AXIS-00517
Solution	Class	1	Parameter P-AXIS-00517 correct number of probing input
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Current value [-]	
		Selected probing signal source P-AXIS-00516	
	%3:	Incorrect value [-]	
		Incorrect setting for probing input P-AXIS-00517	
	%4:	Upper limit value [-]	
		Maximum permissible value for P-AXIS-00517 for this drive type	
	%5:	Corrected value [-]	
		Corrected setting for P-AXIS-00517 probing input	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110588

The given relevant probing signal edge is invalid.			
Description	Parameter P-AXIS-00518 "kenngr.measure.edge" defines the relevant probing signal edge for a measurement run (e.g. G100). However, the setting found is invalid. Correct values are POS (positive edge) or NEG (negative edge).		
Response	Class	1	Warning and correct parameter P-AXIS-00517
Solution	Class	1	Correct the parameter P-AXIS-00518 for the relevant probing signal edge.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis	
	%3:	Current value [-]	
		Selected probing signal source P-AXIS-00516	
	%4:	Incorrect value [-]	
		Incorrect value for relevant probing signal edge P-AXIS-00518	
	%5:	Corrected value [-]	
		Corrected value for relevant probing signal edge P-AXIS-00518	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110589

Parameter P-AXIS-00519 can only be used for spindles.

Description	Parameter P-AXIS-00519 was set for an axis other than a spindle (see P-AXIS-00018).		
Response	Class	1	Warning output, P-AXIS-00519 is corrected to zero.
Solution	Class	1	P-AXIS-00519 for non spindle axes shall be set to zero (nil).
Parameter	%1:	Logical axis number [-]	
		Log. Axis number (P-AXIS-00016)	
	%2:	Current value [-]	
		Axis type (P-AXIS-00018)	
	%3:	Error value [-]	
		Current value of P-AXIS-00519	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00519	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110590

The set positions for leadscrew error compensation are not sorted in ascending order.			
Description	Compensation value lists for leadscrew error compensation can be specified at defined interpolation points in addition to equidistant grids. Compensation values are determined between these interpolation points by linear interpolation. Therefore, they must be arranged according to their position in an ascending order. A possible error cause can be the use of fewer interpolation points than the number given in the parameter P-COMP-00020 since the positions of the unused points are set to zero. Example (extract from compensation data list):		
	kw.ssfk.table[0].setpoint	0	
	kw. ssfk.table[0].pos	0	
	kw. ssfk.table[1].setpoint	100000	
	kw. ssfk.table[1].pos	1000	
	kw. ssfk.table[2].setpoint	300000	
	kw. ssfk.table[2].pos	3000	
	kw. ssfk.table[3].setpoint	200000	
	kw. ssfk.table[3].pos	2000	
	Correct:		
	kw.ssfk.table[0].setpoint	0	
	kw. ssfk.table[0].pos	0	
	kw. ssfk.table[1].setpoint	100000	
	kw. ssfk.table[1].pos	1000	
	kw. ssfk.table[2].setpoint	200000	
	kw. ssfk.table[2].pos	2000	
	kw. ssfk.table[3].setpoint	300000	
	kw. ssfk.table[3].pos	3000	
Response	Class	3	Leadscrew error compensation is disabled
Solution	Class	7	Correct compensation value list, see P-COMP-00025. Check parameter kw_nr_max P-COMP-00020.

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [-]
		Index of the incorrect interpolation point in the compensation value list
	%3:	Incorrect value
		Incorrect position of current interpolation point, see P-COMP-00025
	%4:	Current value
		Position of previous interpolation point, see P-COMP-00025
	%5:	Lower limit value
		Smallest permissible position of current interpolation point, see P-COMP-00025
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110591

The velocities for friction compensation are not sorted in ascending order.			
Description	For friction compensation (see [FCT-C25]), lists with velocities and associated torques must be specified in the compensation value lists. The velocity values must be greater than zero and be specified in ascending order.		
	Example (extract from compensation data list):		
	frict_comp.table[0].in	0	
	frict_comp.table[0].out	0	
	frict_comp.table[1].in	100	
	frict_comp.table[1].out	10	
	frict_comp.table[2].in	300	
	frict_comp.table[2].out	20	
	frict_comp.table[3].in	200	
	frict_comp.table[3].out	20	
	Correct:		
	frict_comp.table[0].in	0	
	frict_comp.table[0].out	0	
	frict_comp.table[1].in	100	
	frict_comp.table[1].out	10	
	frict_comp.table[2].in	200	
	frict_comp.table[2].out	20	
	frict_comp.table[3].in	300	
	frict_comp.table[3].out	20	
Response	Class	3	Friction compensation is disabled
Solution	Class	7	Check and correct the parameters in the compensation value list. frict_comp.table[.].in

Parameter	%1:	Logical axis number [-]
		Logical axis number of affected axis, see P-AXIS-00016
	%2:	Current value [-]
		Index of the incorrect velocity value in the compensation value list
	%3:	Error value [1µm/s or 0.001°/s]
		Incorrect velocity value
	%4:	Current value [1µm/s or 0.001°/s]
		Last correctly entered velocity value

Error type	2, Error message by data transfer from parameter list into control device.

ID 110592

The parametrization of friction compensation is not correct.			
Description	For friction compensation (see FCT-C25), parameters must be set in the compensation value lists. At least one of these is wrong.		
Response	Class	3	Friction error compensation will be disabled
Solution	Class	7	Correct compensation data list s. @@P-COMP-00025.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis see P-AXIS-00016	
	%2:	Current value [-]	
		Index of the incorrect interpolation point	
	%3:	Current value [-]	
		frict_comp.table_entries	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110593

Additional 'Ready for Power test' is not supported for this drive type.			
Description	Parameter P-AXIS-00709 was assigned a value unequal to -1, thus activating the additional 'ready for power' test, although the drive type set in P-AXIS-00020 does not support this test.		
Response	Class	1	Error message output, the additional “ready for power” test is disabled.
Solution	Class	1	Assign P-AXIS-00709 the value “-1“.
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
		Current value of P-AXIS-00709.	
	%3:	Current value [-]	
		Current value of P-AXIS-00020.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00709.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110594

Invalid bit number for additional 'Ready for Power test'.

Description	Additional 'Ready for Power' test was enabled but the bit number configured in P-AXIS-00709 for this test is invalid. The smallest permissible value for P-AXIS-00709 is 0. Depending on the length of the configured telegram element used to transfer the additional ready for power test information, the maximum value of this parameter is:		
	Telegram element length	Maximum value for P-AXIS- 00709	
	2	15	
	4	31	
	This error message is also output if no value is configured for P-AXIS-00709.		
Response	Class	3	Error message output, ready for power test is disabled.
Solution	Class	1	P-AXIS-00709 must have a permissible value assigned, or disable the ‘Ready for Power' test.
Parameter	%1:	Logical axis number [-]	
	%2:	Incorrect value [-]	
		Current value of P-AXIS-00709.	
	%3:	Limit value [-]	
		Limit value for P-AXIS-00709.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00709.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110595

Value for drive encoder modulo range is outside permissible range.			
Description	The parameter P-AXIS-00528 specifies the value range of the drive encoder,if the encoder supplies modulo positions. The CNC uses this value to handle encoder overflow. The given value is however invalid.		
Response	Class	1	Warning message output and correct the parameter
Solution	Class	1	Correct parameter P-AXIS-00528
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [increments]	
		Incorrect value for modulo range P-AXIS-00528	
	%3:	Lower limit value [increments]	
		Minimum permissible value for P-AXIS-00528	
	%4:	Upper limit value [increments]	
		Maximum permissible value for P-AXIS-00528	
	%5:	Corrected value [increments]	
		Corrected value for encoder modulo range P-AXIS-00528	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110596

Modification of drive encoder modulo range requires a restart of the control.

Description	The parameter P-AXIS-00528 specifies the value range of the drive encoder,if the encoder supplies modulo positions. The CNC uses this value to handle encoder overflow. Changing the parameter value during runtime is not possible; it requires a restart of the controller.		
Response	Class	0	Warning message output, old parameter value remains active.
Solution	Class	0	Restart the controller after changing the parameter P-AXIS-00528.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected	
	%2:		
		Old value for encoder value range P-AXIS-00528	
	%3:		
		New encoder modulo value range P-AXIS-00528	
Error type	-		

ID 110597

List interpretation for logical axis number zero commanded.			
Description	The user commanded the interpretation of an axis parameter or axis compensation list for the logical axis number zero.		
Response	Class	3	Negative acknowledge of the command, list is not updated.
Solution	Class	7	Transfer correct axis number before updating the list.
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
		List name of the list to be interpreted.	
Error type	-		

ID 110598

Rapid traverse velocity for operating mode "incremental jog mode" may not be zero.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [1µm/s or 0.001°/s]	
	%3:	Corrected value [1µm/s or 0.001°/s]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110600

Invalid value for maximum movement distance from reference cam.	
Description	An invalid value was assigned to axis parameter P-AXIS-00531. The following restrictions apply: P-AXIS-00531 ≥ 0 P-AXIS-00531 ≤ modulo range for axes where a modulo calculation is executed.

Response	Class	1	Warning output, the value is corrected automatically. If P-AXIS-00531 < 0, the corrected value is 0; if P-AXIS-00531 is greater than the modulo range, it is limited to the modulo range.
Solution	Class	1	Correct P-AXIS-00531.
Parameter	%1:	Logical axis number [-]	
		Axis number	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		Incorrect value of P-AXIS-00531.	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Minimum permissible value of P-AXIS-00531.	
	%4:	Upper limit value [0.1 µm or 0.0001°]	
		Maximum permissible value of P-AXIS-00531.	
	%5:	Corrected value [0.1 µm or 0.0001°]	
		Corrected value of P-AXIS-00531.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110601

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 110602

Weighting factor for velocity of distance control for lowering movement exceeds maximum permissible value.			
Description	The velocity (P-AXIS-00415) used for the lowering movement of the distance control (see FCT-M3) can be weighted with the parameter P-AXIS-00533 in per mille [0.1%]. However, the specified weighting factor is greater than permissible.		
Response	Class	1	Warning, weighting factor for velocity P-AXIS-00533 is corrected.
Solution	Class	1	Correct parameter P-AXIS-00533.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [0.1%]	
		Invalid weighting value P-AXIS-00533	
	%3:	Upper limit value [0.1%]	
		Maximum permissible velocity weighting P-AXIS-00533	
	%4:	Corrected value [0.1%]	
		Corrected value for velocity weighting P-AXIS-00533	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110603

Weighting factor for acceleration of distance control for lowering movement exceeds maximum permissible value.

Description	The acceleration (P-AXIS-00416) used for the lowering movement of the distance control [FCT-M3] can be weighted with the factor P-AXIS-00534 in per mille [0.1%]. However, the given weighting factor is greater than permissible.		
Response	Class	1	Warning, weighting factor for acceleration P-AXIS-00534 is corrected.
Solution	Class	1	Correct parameter P-AXIS-00534.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [0.1%]	
		Invalid weighting value P-AXIS-00534	
	%3:	Upper limit value [0.1%]	
		Maximum permissible acceleration weighting P-AXIS-00534	
	%4:	Corrected value [0.1%]	
		Corrected value for acceleration weighting P-AXIS-00534	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110604

Velocity for distance control must be greater than zero.			
Description	For distance control (see FCT-M3]), the velocity used to adjust the height differences can be set in parameter P-AXIS-00415 [1 µm/s]. The given acceleration value must be greater zero.		
Response	Class	1	Warning, velocity P-AXIS-00415 is corrected.
Solution	Class	1	Correct parameter P-AXIS-00415.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Incorrect value [-]	
		Invalid value of velocity P-AXIS-00415	
	%3:	Lower limit value [-]	
		Minimum permissible velocity P-AXIS-00415	
	%4:	Corrected value [-]	
		Corrected velocity value P-AXIS-00415	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110605

Acceleration for distance control must be greater/equal to zero.			
Description	For distance control (see FCT-M3), the acceleration used to compensate for height differences can be set in parameter P-AXIS-00416 [1 mm/s^2]. The given acceleration value must be greater or equal zero. If the value is zero, the maximum acceleration of the axis P-AXIS-00008 is used.		
Response	Class	1	Warning, acceleration P-AXIS-00416 is corrected.
Solution	Class	1	Correct parameter P-AXIS-00416.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Incorrect value [-]
		Invalid acceleration value P-AXIS-00416
	%3:	Lower limit value [-]
		Minimum permissible acceleration P-AXIS-00416
	%4:	Corrected value [-]
		Corrected acceleration value P-AXIS-00416
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110606

Invalid combination of P-AXIS-00704 and P-AXIS-00261 configured.			
Description	Axis parameters P-AXIS-00704 and P-AXIS-00261 were assigned contradictory values. If P-AXIS-00261 is set, only the values 'DEFAULT' or 'EDGE_TRIGGERED' may be used for P-AXIS-00704.		
Response	Class	1	Warning output, P-AXIS-00704 is set to 'EDGE_TRIGGERED'.
Solution	Class	1	If the edge detection of the 'gantry_on' signal is to be used, use the following setting: P-AXIS-00704 = 'EDGE_TRIGGERED' and P-AXIS-00261 = 0. When edge detection is not to be used, set P-AXIS-00261 = 0 and, depending on the required function, either P-AXIS-00704 = 'DEFAULT' or P-AXIS-00704 = 'ONLY_CONTROL_UNIT'.
Parameter	%1:	Logical axis number[-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value[-]	
		Parameterised value of P-AXIS-00261.	
	%3:	Error value[-]	
		Parameterised value of P-AXIS-00704.	
	%4:	Corrected value[-]	
		Corrected value of P-AXIS-00261.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110607

Functionality 'Disabling drive controller for position lag error' is not supported for this drive type.			
Description	Parameter P-AXIS-00537 was set to "ON", although the configured drive type does not support this function.		
Response	Class	1	Warning output, correct P-AXIS-00537.
Solution	Class	1	Set P-AXIS-00537 to "OFF" or "DEFAULT".
Parameter	%1:	Logical axis number [-]	
		Axis number (P-AXIS-00016)	
	%2:	Drive type [-]	
		Drive type (P-AXIS-00020)	
	%3:	Incorrect value [-]	
		Parameterised value of P-AXIS-00537.	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00537.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 110608

The process data for the 'value_latched' signal was not found in the cyclic process data.			
Description	When configuring the SERCOS real-time status bits for the signal 'measuring probe ½', the process data element configured in P-AXIS-00677 (probe 1) or P-AXIS-00683 (probe 2) was not found in the configured cyclic process data.		
Response	Class	1	An error message is output containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset in order to prevent the execution of measurement run or edge banding with incompletely configured probes.
Solution	Class	1	Correct P-AXIS-00677/or P-AXIS-00683.
Parameter	%1:	Logical axis number [-]	
		Log. Axis number	
	%2:	Incorrect value [-]	
		Incorrect value of P-AXIS-00677/or P-AXIS-00683	
	%3:	Lower limit value [-]	
		Drive type	
	%4:	Upper limit value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110609

An invalid bit number was entered for the 'value_latched' signal.			
Description	When configuring the SERCOS real-time status bits for the signal 'value latched probe 1/2', an invalid bit number was entered either in P-AXIS-00678 (probe 1) or in P-AXIS-00684 (probe 2).		
Response	Class	3	An error message is output containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset in order to prevent the execution of measurement run or edge banding with incompletely configured probes.
Solution	Class	1	Correct P-AXIS-00678/or P-AXIS-00684.
Parameter	%1:		
		Log. axis number	
	%2:		
		Incorrect value of P-AXIS-00678/or P-AXIS-00684	
	%3:		
		Permissible minimum value.	
	%4:		
		Permissible maximum value.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110610

The process data for the 'probe_actuated' signal was not found in the cyclic process data.

Description	When configuring the SERCOS realtime status bits for the signal 'measuring probe ½' actuated', the process data element configured in P-AXIS-00679 (probe 1) or P-AXIS-00685 was not found in the configured cyclic process data.		
Response	Class	1	Output of an error message containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset, in order to prevent the execution of measurement run or edge bending with incompletely configured probes.
Solution	Class	1	P-AXIS-00679/ or P-AXIS-00685 must be corrected.
Parameter	%1:		
	Log. Axis number		
	%2:		
	Incorrect value of P-AXIS-00679/or P-AXIS-00685		
	%3:		
	Drive type		
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110611

An invalid bit number was entered for the 'probe_actuated' signal.			
Description	When configuring the SERCOS real-time status bits for the signal 'measuring probe ½', an invalid bit number was entered either in P-AXIS-00680 (probe 1) or P-AXIS-00686 (probe 2).		
Response	Class	3	An error message is output containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset in order to prevent the execution of measurement run or edge banding with incompletely configured probes.
Solution	Class	1	Correct P-AXIS-00680/or P-AXIS-00686.
Parameter	%1:		
	Log. axis number		
	%2:		
	Incorrect value of P-AXIS-00680/or P-AXIS-00686		
	%3:		
	Permissible minimum value.		
	%4:		
	Permissible maximum value.		
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110612

The process data for the 'start_probing' signal was not found in the cyclic process data.			
Description	When configuring the SERCOS realtime status bits for the signal 'measuring probe ½ enable', the process data element configured in P-AXIS-00675 (probe 1) or P-AXIS-00681 (probe 2) was not found in the configured cyclic process data.		
Response	Class	1	Output of an error message containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset, in order to prevent the execution of measurement run or edge bending with incompletely configured probes.
Solution	Class	1	Correct P-AXIS-00675/or P-AXIS-00681.

Parameter	%1:	
		Log. Axis number
	%2:	
		Incorrect value of P-AXIS-00675/or P-AXIS-00681
	%3:	
		Drive type
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110613

The process data for the 'start_probing' signal was not found in the cyclic process data.			
Description	When configuring the SERCOS real-time control bits for the signal 'measuring probe 1/2 enable', an invalid bit number was entered either in P-AXIS-00676 (probe 1) or P-AXIS-00682 (probe 2).		
Response	Class	3	An error message is output containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset in order to prevent the execution of measurement run or edge banding with incompletely configured probes.
Solution	Class	1	Correct P-AXIS-00676/or P-AXIS-00682.
Parameter	%1:		
		Log. axis number	
	%2:		
		Incorrect value of P-AXIS-00676/or P-AXIS-00682	
	%3:		
		Permissible minimum value.	
	%4:		
		Permissible maximum value.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110614

Parameter P-AXIS-00539 is not supported for this drive type.			
Description	Parameter P-AXIS-00539 was assigned a value other than zero (nil), although the drive type used does not support this parameter.		
Response	Class	1	Error message output, P-AXIS-00539 is set to zero (nil).
Solution	Class	1	P-AXIS-00539 shall be assigned the value zero (nil
Parameter	%1:	Logical axis number [-]	
		Log. Axis number	
	%2:	Drive type [-]	
		Drive type	
	%3:	Error value [-]	
		Incorrect value of P-AXIS-00539	
	%4:	Corrected value [-]	
		Corrected value of P-AXIS-00539	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110615

Invalid value for parameter P-AXIS-00539 is corrected.			
Description	Parameter P-AXIS-00539 was assigned an invalid value. The permissible value range depends on the drive type:		
	Drive type	Value range	
	Profidrive	0 .. 2	
	SERCOS	0 .. 2	
	all others	0	
	A value of 0 for P-AXIS-00539 means that the parameter is not used.		
Response	Class	1	Error message output, P-AXIS-00539 is set to zero.
Solution	Class	1	Assign P-AXIS-00539 a permissible value assigned
Parameter	%1:	Logical axis number [-]	
		Log. axis number	
	%2:	Incorrect value [-]	
		Incorrect value of P-AXIS-00539	
	%3:	Lower limit value [-]	
		Permissible minimum value of P-AXIS-00539	
	%4:	Upper limit value [-]	
		Permissible maximum value of P-AXIS-00539	
	%5:	Corrected value [-]	
		Corrected value of P-AXIS-00539	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110617

Process data element for the 'probing_command_start' signal was not found in the cyclic process data.			
Description	When configuring the SERCOS real-time control bits for the signal 'Command start probing', the process data element configured in P-AXIS-00699 was not found in the configured cyclic process data.		
Response	Class	1	An error message is output containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset in order to prevent the execution of measurement run or edge banding with incompletely configured probes.
Solution	Class	1	Correct P-AXIS-00699.
Parameter	%1:		
		Log. axis number	
	%2:		
		Incorrect value of P-AXIS-00699	
	%3:		
		Drive type	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110618

An invalid bit number was configured for the 'probing_command_start' signal.	
Description	When configuring the SERCOS real-time control bits for the signal 'command start probing', an invalid bit number was entered in P-AXIS-00700.

Response	Class	3	An error message is output containing the axis parameters P-AXIS-00118 (measurement axis) and P-AXIS-00098 (kasto_achse) are reset in order to prevent the execution of measurement run or edge banding with incompletely configured probes.
Solution	Class	1	Correct P-AXIS-00700.
Parameter	%1:		
		Log. axis number	
	%2:		
		Incorrect value of P-AXIS-00700	
	%3:		
		Permissible minimum value.	
	%4:		
		Permissible maximum value.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110619

Given setting 'Disabling drive controller in case of bus error' is not supported for this drive type.			
Description	The parameter P-AXIS-00542 is used for CANopen drives to set the behaviour in case of a bus error. However, the parameter was set for a different drive type and to a value ("ON" or “OFF”) although the configured drive type does not support this setting.		
Response	Class	1	Warning output, correct P-AXIS-00542.
Solution	Class	1	Set P-AXIS-00542 to "DEFAULT".
Parameter	%1:	Current value [-]	
		Parameter number P-AXIS-00542	
	%2:	Logical axis number [-]	
		Axis number (P-AXIS-00016)	
	%3:	Drive type [-]	
		Drive type (P-AXIS-00020)	
	%4:	Incorrect value [-]	
		Parameterised value of P-AXIS-00542.	
	%5:	Corrected value [-]	
		Corrected value of P-AXIS-00542.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110620

Reduced acceleration for manual mode exceeds permissible limit value.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
	%4:	Corrected value [1mm/s^2 or 1°/s^2]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 110621

Invalid parameterisation of spindle homing. Homing switch evaluation is activated.			
Description	The parameterisation of the spindle homing is invalid. Homing without a homing switch (kenngr.ref_ohne_nocken 1) must be performed without direction reversal (kenngr.ref_ohne_rev 1). Valid settings for these two parameters are:		
		ref_ohne_rev = FALSE	**ref_ohne_rev = TRUE**
	ref_ohne_nocken = FALSE	conventional homing	flying homing with cam
	Invalid parameterisation of spindle homing. Homing switch evaluation is activated.	impermissible setting	flying homing without cam

Response	Class	1	Spindle homing is executed with evaluation of homing switch
Solution	Class	1	Correct parameter P-AXIS-00156 or P-AXIS-00157

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [-]
		Axis type P-AXIS-00018 of the affected axis.
	%3:	Incorrect value [-]
		Homing with/without cam P-AXIS-00156.
	%4:	Incorrect value [-]
		Homing with/without direction reversal P-AXIS-00157.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 110622

Invalid value for P-AXIS-00583..			
Description	Parameter P-AXIS-00583 was assigned an invalid value.		
Response	Class	1	Warning output, P-AXIS-00583 is corrected automatically to the permissible maximum value.
Solution	Class	1	Check and modify P-AXIS-00583. Use a valid value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [-]
		Index of invalid entry of P-AXIS-00583.
	%3:	Error value [-]
		Incorrect value of P-AXIS-00583.
	%4:	Upper limit value [-]
		Permissible maximum value of P-AXIS-00583.
	%5:	Corrected value [-]
		Corrected value of P-AXIS-00583
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110624

Feedforward without delay is only possible with velocity feedforward.

Description	The axis parameter P-AXIS-00566 was set and simultaneously the acceleration and/or jerk feedforward was enabled in P-AXIS-00223. Only a velocity feedforward is possible when P-AXIS-00566 is set.		
Response	Class	1	Warning output, the acceleration and/or jerk feedforward is disabled in P-AXIS-00223.
Solution	Class	1	Either set P-AXIS-00566 to zero or disable the acceleration and jerk feedforward in P-AXIS-00223.
Parameter	%1:	Logical axis number [-]	
		Logical axis number	
	%2:	Error value [-]	
		Incorrect value of P-AXIS-00223	
	%3:	Corrected value [-]	
		Corrected value of P-AXIS-00223	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110625

Parameter P-AXIS-00584 is not supported for this drive type.			
Description	Parameter P-AXIS-00584 is not supported for this drive type.		
Response	Class	1	Warning and automatic correction of parameter
Solution	Class	1	Correct parameter.
Parameter	%1:	Logical axis number [-]	
		Logical axis number	
	%2:	Drive type [-]	
		Drive type	
	%3	Error value[-]	
		Parameterised value of P-AXIS-00584	
	%4	Corrected value [-]	
		Corrected value of P-AXIS-00584	
Error type	2, Error message by data transfer from parameter list into control device.		

ID110626

Invalid parameterisation of P-AXIS-00571. Parameter is corrected.			
Description	The Vibration Guard mode (P-AXIS-00571) was configured with an invalid value.		
Response	Class	1	The default value is used for the Vibration Guard mode and the start-up is continued.
Solution	Class	1	Check and modify P-AXIS-00571 of the affected axis. Use a valid value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis where the invalid Vibration Guard mode was configured.	
	%2:	Error value[-]	
		Invalid Vibration Guard mode that was configured.	
	%3	Corrected value [-]	
		Corrected value of P-AXIS-00571	
Error type	2, Error message by data transfer from parameter list into control device.		

ID110627

Invalid parameterisation of P-AXIS-00568. Parameter is corrected.			
Description	The damping factor of the Vibration Guard (P-AXIS-00568) was configured with an invalid value.		
Response	Class	1	The default value is used for the Vibration Guard damping factor and the start-up is continued.
Solution	Class	1	Check and modify P-AXIS-00568 of the affected axis. Use a valid value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number where the invalid Vibration Guard mode was configured.	
	%2:	Error value[-]	
		Invalid damping factor that was configured.	
	%3	Corrected value [-]	
		Corrected value of P-AXIS-00568	
Error type	2, Error message by data transfer from parameter list into control device.		

ID110628

If vib_guard- or filter_fir-parameters are configured, then filter[0].prototype!=5 is neededIf vib_guard- or filter_fir-parameters are configured, then filter[0].prototype!=5 is needed			
Description	The Vibration Guard or the FIR filter in an axis was configured together with HSC filters. This is not allowed.		
Response	Class	1	Vibration Guard or FIT filters will be disabled. The start-up is continued
Solution	Class	1	Check and modify the configuration. Do not configure any HSC filters. Instead, configure either only FIR filters or only Vibration Guard.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis where the HSC filter was configured together with Vibration Guard or FIR filters.	
	%2:	Error value [-]	
		Filter prototype used to configure the HSC filter	
Error type	2, Error message by data transfer from parameter list into control device.		

ID110629

Permanent activation of HSC filters not possible.			
Description	On checking the filter parameters, the system detected that an attempt was made in an axis to activate an HSC filter via filter[0].enable 1 permanently. Permanent activation of HSC filters is not possible, only of FIR filters.		
Response	Class	1	Parameter for permanent activation is reset and the start-up is continued.
Solution	Class	1	Check and modify the parameters. Permanent activation can only be used for FIT filters.

Parameter	%1:	Logical axis number [-]
		Logical axis number of the axis where the invalid parameter was configured.
	%2:	Error value[-]
		Value set for permanent activation for P-AXIS-00573.
	%3	Error value[-]
		Filter prototype (P-AXIS-00153) which was configured for this axis.
Error type	2, Error message by data transfer from parameter list into control device.	

ID110630

FIR filter and Vibration Guard cannot be combined. Vibration Guard not active.			
Description	On checking the filter parameters, the system detected that an attempt was made in an axis to activate the Vibration Guard although FIR filters are already configured in the same axis. The simultaneous use of Vibration Guard and FIR filters is not allowed.		
Response	Class	1	Vibration Guard is deactivated and the start-up is continued.
Solution	Class	1	Check and modify the parameters. In order to activate the Vibration Guard using the axis configuration, do not configure any FIR filters, i.e. the parameter filter_fir.type P-AXIS-00586 may not be assigned a value unequal to 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis where the invalid combination was configured.	
	%2:	Error value[-]	
		P-AXIS-00588 for permanent activation of the Vibrations Guard	
	%3	Corrected value [-]	
		Permanent activation P-AXIS-00588	
Error type	2, Error message by data transfer from parameter list into control device.		

ID110632

Invalid value for filter share factor.			
Description	The filter share factor (P-AXIS-00590) was configured with an invalid value. The value value range is from 0 to 100%. With a configured value less than 0, the value is corrected to 0 so that the filter is deactivated.		
Response	Class	1	The filter share factor P-AXIS-00590 is corrected and the start-up is continued.
Solution	Class	1	Check and modify the parameters. Use a valid value for the filter share factor P-AXIS-00590
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis where the invalid combination was configured.	
	%2:	Error value[%]	
		Incorrectly parameterised filter share factor P-AXIS-00590	
	%3	Corrected value [%]	
		Corrected filter share factor	
Error type	2, Error message by data transfer from parameter list into control device.		

ID110633

Invalid frequency value.			
Description	On checking the filter parameters, the system detected that an invalid value was specified in an axis for the limit frequency of the FIR filter (P-AXIS-00585) or the machine natural frequency (P-AXIS-00589).		
Response	Class	1	The corresponding parameter is set to the default value and the start-up is continued.
Solution	Class	1	Check and modify the parameters (P-AXIS-00585) and (P-AXIS-00589). Use a valid value
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the axis where the invalid combination was configured.	
	%2:	Error value[Hz]	
		Incorrect frequency specified.	
	%3	Corrected value[Hz]	
		Corrected frequency value	
Error type	2, Error message by data transfer from parameter list into control device.		

ID110634

Activation of FIR filters by P-AXIS-00573 is not effective without setting a valid filter type (P-AXIS-00586).			
Description	On checking the filter parameters, the system detected that an attempt was made to activate a FIR filter in the list using P-AXIS-00573 without configuring a FIR filter type (P-AXIS-00586). It is only possible to use FIR filters if a corresponding FIR filter type is defined.		
Response	Class	1	No reaction; FIR filter is inactive and the start-up is continued.
Solution	Class	1	Check and modify the parameter. Assign FIR filter type (P-AXIS-00586) with a valid value unequal to 0.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected	
	%2:	Error value[-]	
		Value of P-AXIS-00573	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110636

Activation/deactivation of single axis function in the path interpolator requires restart of the control.			
Description	The single axis function in the path interpolator can only be activated and deactivated by the axis parameter P-AXIS-00457 by rebooting the controller. The previous setting remains active.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Change the entry P-AXIS-00457 only in configuration mode or when the CNC is disabled.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [-]
		Commanded value of P-AXIS-00457
	%3:	Corrected value [-]
		Corrected value of P-AXIS-00457
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110637

The value of filter parameter P-AXIS-00593 exceeds the range of values.			
Description	A filter quality of less than 0 (P-AXIS-00593) was selected. A filter quality of less than 0 is not defined or not possible.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Correct the parameter, enter a valid value for P-AXIS-00593 in the axis affected.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Parameterised value for P-AXIS-00593 in the axis parameter list.	
	%3:	Corrected value [-]	
		Corrected value for P-AXIS-00593	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110638

P-COMP-00059 (max_points) for table size of the leadscrew compensation must not be changed.			
Description	The parameter max_points (P-COMP-00059) sets how many inputs are reserved for the table of the leadscrew error compensation [FCT-C5]. This value may not be changed after start-up.		
Response	Class	6	List interpretation is aborted
Solution	Class	6	Set value of max_points to the original value and re-interpret compensation value lists
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file	
Error type	-		

ID 110639

P-COMP-00060 (max_points) for table size of the cross compensation must not be changed.

Description	The parameter P-COMP-00060 (max_points) sets how many inputs are reserved for the table of the cross compensation [FCT-C5]. This value may not be changed after start-up.		
Response	Class	6	List interpretation is aborted
Solution	Class	6	Set value of P-COMP-00060 (max_points) to the original value and re-interpret compensation value lists
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110640

P-COMP-00061 (max_points) for table size of the plane compensation must not be changed.			
Description	The parameter P-COMP-00061 (max_points) sets how many inputs are reserved for the table of the plane compensation [FCT-C5]. This value may not be changed after start-up.		
Response	Class	6	List interpretation is aborted
Solution	Class	6	Set value of P-COMP-00061 (max_points) to the original value and re-interpret compensation value lists
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110641

P-COMP-00062 (max_points) for table size of the friction compensation must not be changed.			
Description	The parameter P-COMP-00062 (max_points) sets how many inputs are reserved for the table of the friction compensation [FCT-C25). This value may not be changed after start-up.		
Response	Class	6	List interpretation is aborted
Solution	Class	3	Set value of P-COMP-00062 (max_points) to the original value and re-interpret compensation value lists
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110642

Could not allocate enough memory for table of leadscrew compensation.

Description	The parameter P-COMP-00059 (max_points) sets how many inputs are reserved for the table of the leadscrew error compensation [FCT-C5]. Not enough memory could be allocated for this request.		
Response	Class	6	List interpretation is aborted.
Solution	Class	6	Either: • make more memory space available, or • reduce the value of P-COMP-00059 (max_points) and then reboot the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file	
	%3:	Error value [-]	
		Requested number of table inputs	
	%4:	Limit value [-]	
		Maximum number of table inputs available	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110643

Could not allocate enough memory for table of cross compensation.			
Description	The parameter P-COMP-00060 (max_points] sets how many inputs are reserved for the table of the cross compensation [FCT-C5). Not enough memory could be allocated for this request.		
Response	Class	6	List interpretation is aborted.
Solution	Class	6	Either: • make more memory space available, or • reduce the value of P-COMP-0060 (max_points) and then reboot the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file	
	%3:	Error value [-]	
		Requested number of table inputs	
	%4:	Limit value [-]	
		Maximum number of table inputs available	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110644

Could not allocate enough memory for table of plane compensation.

Description	The parameter P-COMP-00061 (max_points) sets how many inputs are allocated to the table of the plane compensation [FCT-C5]. Not enough memory could be allocated for this request.		
Response	Class	6	List interpretation is aborted.
Solution	Class	6	Either: • make more memory space available, or • reduce the value of P-COMP-00061 (max_points) and then reboot the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file	
	%3:	Error value [-]	
		Requested number of table inputs	
	%4:	Limit value [-]	
		Maximum number of table inputs available	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110645

Could not allocate enough memory for friction compensation table.			
Description	The parameter P-COMP-00062 (max_points) sets how many inputs are reserved for the friction compensation) table. Not enough memory could be allocated for this request.		
Response	Class	6	List interpretation is aborted.
Solution	Class	6	Either: • make more memory space available, or • reduce the value of P-COMP-00062 (max_points) and then reboot the controller
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis affected.	
	%2:	Current value [-]	
		Name of the list file	
	%3:	Error value [-]	
		Requested number of table inputs	
	%4:	Limit value [-]	
		Maximum number of table inputs available	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110646

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	

Solution	Class	8	Requires controller restart.

ID 110647

The table of plane compensation is not big enough.			
Description	The number of inputs defined by max_points (P-COMP-00061) is available for the plane compensation table. The two parameters last_index_master1 (P-COMP-00010) and last_index_master2 (P-COMP-00011) describe the dimensions of the compensation table. The following must apply to the table: *(last_index_master1 + 1) * (last_index_master2 + 1) <= max_points* This warning is output if max_points is not large enough to fit the entire compensation table, i.e. if *(last_index_master1 + 1) * (last_index_master2 + 1) is > max_points* .		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Either: • increase max_points and reboot the controller, or • reduce last_index_master1 and/or last_index_master2 and reload the compensation value lists.
Parameter	%1:	Current value [-]	
		Dimension of the compensation table in the first master axis, last_index_master1 + 1 (P-COMP-00010)	
	%2:	Current value [-]	
		Dimension of the compensation table in the second master axis, last_index_master2 + 1 (P-COMP-00011)	
	%3:	Error value [-]	
		Parameterised value for max_points (P-COMP-00061)	
Error type	-		

ID 110648

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 110649

Kp factor P-AXIS-00759 for distance control is out of permissible range.			
Description	The Kp factor P-AXIS-00759 for distance control may not be outside the limit values.		
Response	Class	1	Warning and correction of parameter.P-AXIS-00759 is automatically limited to maximum with values that are too large or limited to minimum if the values are too small.
Solution	Class	1	Check and modify P-AXIS-00759

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Error value [-]
		Incorrect value of P-AXIS-00759
	%3:	Lower limit value [-]
		Lower limit value of P-AXIS-00759
	%4:	Upper limit value [-]
		Upper limit value of P-AXIS-00759
	%5:	Corrected value [-]
		Corrected value of P-AXIS-00759
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110651

Acceleration P-AXIS-00760 in tracking mode is greater than P-AXIS-00008.			
Description	The parameterised acceleration for P-AXIS-00760 is greater than the permitted maximum acceleration (P-AXIS-00008). The value is limited.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-CHAN-00760. For P-AXIS-00760 set a value ≤ P-AXIS-00008
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016	
	%2:	Error value [mm/s² or °/s²]	
		Parameterised value of P-AXIS-00760	
	%3:	Limit value [mm/s² or °/s²]	
		Permissible maximum value (P-AXIS-00008)	
	%4:	Corrected value [mm/s² or °/s²]	
		Corrected value of P-AXIS-00760	
	%5:	Current value [-]	
		Gear index	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110652

Position controller and interpolator position cannot be restored at the same time.			
Description	The parameter P-AXIS-00402 restores the last interpolator position of the axis for PLC axis couplings at controller start-up. On the other hand, the parameter P-AXIS-00761 loads the last position controller position of the axis. The two parameters are therefore mutually exclusive.		
Response	Class	1	The function to restore the last position controller position is disabled.
Solution	Class	1	Correct parameter P-AXIS-00402 or P-AXIS-00761.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Drive type [-]
		Drive type P-AXIS-00020 of the affected axis
	%3:	Current value [-]
		Current value of P-AXIS-00402 to restore the interpolator position
	%4:	Error value [-]
		Current value of P-AXIS-00761 to restore the last position controller position
	%5:	Corrected value [-]
		Corrected value for P-AXIS-00761 to restore the last position controller position
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110653

Restoring of axis positions for Profidrive not supported.			
Description	The parameter P-AXIS-00761 activates the restore function for the last position controller position at controller start-up. However, this function is not supported for PROFIdrive drives.		
Response	Class	1	The function to restore the last position controller position is disabled.
Solution	Class	1	Correct parameter P-AXIS-00761.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type P-AXIS-00020 of the affected axis	
	%3:	Error value [-]	
		Invalid value for P-AXIS-00761 to restore the last position controller position	
	%4:	Corrected value [-]	
		Corrected value for P-AXIS-00761 to restore the last position controller position	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110654

Tn factor P-AXIS-00764 for distance control is out of permissible range.			
Description	The I-Tn integral action time P-AXIS-00764 for distance control may not be outside the limit values.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00764 is automatically limited to maximum with values that are too large or to minimum if the values are too small.
Solution	Class	1	Check and modify P-AXIS-00764

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Error value [-]
		Incorrect value of P-AXIS-00764
	%3:	Lower limit value [-]
		Lower limit value of P-AXIS-00764
	%4:	Upper limit value [-]
		Upper limit value of P-AXIS-00764
	%5:	Corrected value [-]
		Corrected value of P-AXIS-00764
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110655

Tv factor P-AXIS-00765 for distance control is out of permissible range.			
Description	The I-Tv derivative action time P-AXIS-00765 for distance control may not be outside the limit values.		
Response	Class	1	Warning and correction of parameter.P-AXIS-00765 is automatically limited to maximum with values that are too large or limited to minimum if the values are too small.
Solution	Class	1	Check and modify P-AXIS-00765
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Error value [-]	
		Incorrect value of P-AXIS-00765	
	%3:	Lower limit value [-]	
		Lower limit value of P-AXIS-00765	
	%4:	Upper limit value [-]	
		Upper limit value of P-AXIS-00765	
	%5:	Corrected value [-]	
		Corrected value of P-AXIS-00765	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110656

Weighting P-AXIS-00766 for feed forward control is out of permissible range.			
Description	The value for weighting velocity feedforward control P-AXIS-00766 is outside the permissible limit values.		
Response	Class	1	Value is correct to upper limit value.
Solution	Class	1	Check and correct P-AXIS-00766 Update parameters.

Parameter	%1:	Logical axis number[-]
		Logical axis number P-AXIS-00016 of concerned axis
	%2:	Error value [-]
		Incorrect value of P-AXIS-00766.
	%3:	Upper limit value [-]
		Upper limit value of P-AXIS-00766.
	%4:	Corrected value [-]
		Corrected value of P-AXIS-00766.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110657

Measuring to a fixed stop, relative start value is too big.			
Description	A value of > 1000 was specified for the P-AXIS-00777 parameter. The P-AXIS-00777 parameter specifies the activation position of the fixed stop search in 0.1% referred to the length of the measurement block. A value > 1000 is not possible.		
Response	Class	1	Warning output.
Solution	Class	1	P-AXIS-00777 must be assigned a value of less than 1000.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [0.1%]	
		Incorrect value of P-AXIS-00777.	
	%3:	Upper limit value [0.1%]	
		Upper limit value.	
Error type	-		

ID 110658

Movement on fixed stop, relative start value is too big.			
Description	A value of > 1000 was specified for the P-AXIS-00772 parameter. The P-AXIS-00772 parameter specifies the activation position of the fixed stop search in 0.1% referred to the length of the measuring block for Measuring travel to fixed stop. A value > 1000 is not possible.		
Response	Class	1	Warning output.
Solution	Class	1	P-AXIS-00772 must be assigned a value of less than 1000.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [0.1%]	
		Incorrect value of P-AXIS-00777.	
	%3:	Upper limit value [0.1%]	
		Upper limit value.	
Error type	-		

ID 110659

Axes with position restoration may not use identifier for absolute measurement system.

Description	The parameter P-AXIS-00761 activates the restore function for the last position controller position at controller start-up. This function is used for axes whose measurement system only provides an absolute position within a limited range (e.g. within one motor revolution) or whose absolute encoder overflows in the motion range. For these axes, the identification code "absolute measurement system" P-AXIS-00014 must therefore not be set, otherwise an incorrect reference position is used under certain circumstances, e.g. if remanent data is lost. If the axis measurement system actually supplies an absolute position within the entire motion range, it can be used directly. An automatic position restoration is not necessary.		
Response	Class	1	The function to restore the last position controller position is disabled
Solution	Class	1	Check and modify the parameters P-AXIS-00014 and P-AXIS-00761
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value [-]	
		Value for P-AXIS-00761 to restore the last position controller position	
	%3:	Error value [-]	
		Incorrect identifier P-AXIS-00014 for absolute measurement system	
	%4:	Corrected value [-]	
		Corrected identifier P-AXIS-00014 for absolute measurement system	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110660

Smoothing factor P-AXIS-00784 for distance control is out of permissible range.			
Description	The smoothing factor P-AXIS-00784 for the exponential average filter of the distance control may not exceed the specified limits.		
Response	Class	1	Output the warning and limit P-AXIS-00784
Solution	Class	1	Check and modify P-AXIS-00784.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of concerned axis	
	%2:	Error value [-]	
	%4:	Upper limit value [-]	
	%5:	Corrected value [-]	
		If P-AXIS-00784 exceeds the maximum, the upper limit is output. If P-AXIS-00784 undershoots the minimum, the lower limit is output.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110661

Kalman-Sigma P-AXIS-00783 for distance control is out of permissible range.			
Description	The degree of uncertainty of the recorded measured values P-AXIS-00783 for the Kalman filter of the distance control may not exceed the specified limits.		
Response	Class	1	Output the warning and limit P-AXIS-00783
Solution	Class	1	Check and modify P-AXIS-00783. Update the parameters

Parameter	%1:	Logical axis number [-]
		Logical axis number (P-AXIS-00016) of concerned axis
	%2:	Error value [-]
	%4:	Upper limit value [-]
	%5:	Corrected value [-]
		If P-AXIS-00783 exceeds the maximum, the upper limit is output. If P-AXIS-00783 undershoots the minimum, the lower limit is output.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110663

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 110664

The order P-AXIS-00507 of the low pass filter of the distance control is out of permissible range.			
Description	The parameter P-AXIS-00507 indicates the order of the low-pass filter of the distance control. This may not be outside the specified limits.		
Response	Class	1	Output the warning and limit P-AXIS-00507
Solution	Class	1	Check and modify P-AXIS-00507. Update the parameters.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [-]	
	%3:	Upper limit value [-]	
		If P-AXIS-00507 exceeds the maximum, the upper limit is output. If P-AXIS-00507 undershoots the minimum, the lower limit is output.	
	%4:	Corrected value [-]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110665

Tv factor P-AXIS-00765 for distance control is out of permissible range.			
Description	The parameter P-AXIS-00508 indicates the cut-off frequency of the low-pass filter of the distance control. This may not be less than the specified value.		
Response	Class	1	Output the warning and limit P-AXIS-00508
Solution	Class	1	Check and modify P-AXIS-00508. Update the parameters.

Parameter	%1:	Logical axis number [-]
	%2:	Error value [Hz]
	%3:	Lower limit value [Hz]
	%4:	Corrected value [Hz]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110666

Delay time P-AXIS-00787 for spindle revolution monitoring is bigger than permissible.			
Description	The delay time set in P-AXIS-00787 to output an error message for spindle revolution monitoring exceeds the maximum value allowed.		
Response	Class	1	Warning output. Correct the value for the maximum permissible value.
Solution	Class	1	Correct value of P-AXIS-00787.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [µs]	
		Configured value	
	%3:	Limit value [µs]	
	%4:	Corrected value [µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110667

Maximum velocity P-AXIS-00788 for delay of spindle revolution monitoring is bigger than permissible.			
Description	The limit speed set in P-AXIS-00788 to delay the output of an error message for spindle revolution monitoring exceeds the maximum value allowed.		
Response	Class	1	Output a warning to correct the value of the maximum value allowed.
Solution	Class	1	Correct value of P-AXIS-0788
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis	
	%2:	Error value [1µm/s or 0.001°/s]	
		Configured value	
	%3:	Limit value [1µm/s or 0.001°/s]	
	%4:	Corrected value [1µm/s or 0.001°/s]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110668

Change of P-AXIS-00708 requires restart of the control.			
Description	The change in the parameter conv_sync.is_master (P-AXIS-00708) requires a controller reboot.		
Response	Class	1	Warning output, the value of P-AXIS-00708 is not changed.
Solution	Class	1	Change parameter and restart controller.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Current value [-]	
		Changed value of P-AXIS-00708	
	%3:	Corrected value [-]	
		Old value of P-AXIS-00708	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110669

Switching drive simulation on/off requires restart of the control.			
Description	The parameter P-AXIS-00790 for switching the drive type to simulation or to switch back to the configured drive type cannot be changed during CNC runtime. The new drive type is not adopted; the previous active drive type remains valid.		
Response	Class	2	None
Solution	Class	3	Switch over to simulation with P-AXIS-00790 in configuration mode or when the CNC is inactive.
Parameter	%1:	Logical axis number [-]	
	%2:		
		Current value of P-AXIS-00790	
	%3:		
		Corrected value of P-AXIS-00790	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110673

Parameter P-AXIS-00803 can only be used for drive controlled homing.			
Description	The parameter kenngr.homing_without_drive_enable (P-AXIS-00803) can only be used with drive types that support drive-controlled homing.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00803 is set to zero.
Solution	Class	1	Check the strategy for homing.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis effected.	
	%2:	Current value [-]	
		Drive type used (P-AXIS-00020) of the axis.	
	%3:	Error value [-]	
		Parameterised value of P-AXIS-00803.	
	%4:	Corrected value [-]	
		Corrected value for P-AXIS-00803.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110674

Parameter P-AXIS-00803 is not supported for this drive type.			
Description	The parameter kenngr.homing_without_drive_enable (P-AXIS-00803) is not supported for this drive type.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00803 is set to zero.
Solution	Class	1	Check the strategy for homing.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of axis effected.	
	%2:	Drive type [-]	
		Drive type used (P-AXIS-00020) of the axis.	
	%3:	Error value [-]	
		Parameterised value of P-AXIS-00803.	
	%4:	Corrected value [-]	
		Corrected value for P-AXIS-00803.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110675

The configured bit in P-AXIS-00815 is not supported.			
Description	The parameterised bit in “antr.probe_actuated_bit“ (P-AXIS-00815) to support the measurement status is not supported.		
Response	Class	1	Warning output; the value is reset. Monitoring via the HLI is not supported.
Solution	Class	1	Bit 6/7 must be parameterised for CANopen, see P-AXIS-00815
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of drive affected	
	%2:	Current value [-]	
		Invalid parameterised value for P-AXIS-00815	
	%3:	Corrected value [-]	
		Corrected value for P-AXIS-00815	
Error type	-		

ID 110678

Change of P-AXIS-00795 requires restart of the control.			
Description	A change in the parameter P-AXIS-00795 `pos_lag_model.pos_lag_batch_num` requires a controller restart.		
Response	Class	1	Output of warning; adopt the new value.
Solution	Class	1	The new value is adopted; a controller restart is required.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value for P-AXIS-00795.	
	%3:	Corrected value [-]	
		New value for P-AXIS-00795.	

Error type	-

ID 110679

Changing of P-AXIS-00796 requires restart of the control.			
Description	A change in the parameter P-AXIS-00796 `pos_lag_model.units` requires a controller restart.		
Response	Class	1	Output of warning; adopt the new value.
Solution	Class	1	The new value is adopted; a controller restart is required.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Current value for P-AXIS-00796.	
	%3:	Corrected value [-]	
		New value for P-AXIS-00796.	
Error type	-		

ID 110680

Change of P-AXIS-00794 requires restart of the control.			
Description	A change in the parameter P-AXIS-00794 `pos_lag_model.window_size` requires a controller restart.		
Response	Class	1	Output of warning; adopt the new value.
Solution	Class	1	The new value is adopted; a controller restart is required.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value for P-AXIS-00794.	
	%3:	Corrected value [-]	
		New value for P-AXIS-00794.	
Error type	-		

ID 110681

Invalid value for inverse rotation direction for spindle P-AXIS-00490 .			
Description	The spindle rotation direction can be inverted for various functions by using P-AXIS-00490. An invalid parameter was assigned to this parameter.		
Response	Class	1	Warning and correction of parameter. P-AXIS-00490 is set to FALSE.
Solution	Class	1	Check P-AXIS-00490. Use a valid value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Current value [-]
		Current hexadecimal value of parameter P-AXIS-00490
	%3:	Upper limit value [-]
		Maximum limit of parameter P-AXIS-00490
	%4:	Corrected value [-]
		Corrected value of parameter P-AXIS-00490
Error type	2, Error message by data transfer from parameter list into control device.	

ID 110682

Velocity limit during homing to a fixed stop is higher than 100%.			
Description	The value for kenngr.homing.torq_detect_velocity_limit (P-AXIS-00347) must be less than 1000.		
Response	Class	1	Warning and value corrected to 1000.
Solution	Class	1	Set value for P-AXIS-00347 less than 1000.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of drive, see P-AXIS-00016	
	%2:	Current value [0.1%]	
		Parameterised incorrect value of P-AXIS-00347	
	%3:	Corrected value [0.1%]	
		Corrected value of P-AXIS-00347	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110683

Torque limit during homing to a fixed stop is greater than 100%.			
Description	The value for kenngr.homing.torq_detect_torque_limit (P-AXIS-00820) must be less than 1000.		
Response	Class	1	Warning and value corrected to 1000 [0.1%].
Solution	Class	1	Set value for P-AXIS-00820 less than 1000
Parameter	%1:	Logical axis number [-]	
		Logical axis number of drive, see P-AXIS-00016	
	%2:	Current value [0.1%]	
		Parameterised incorrect value of P-AXIS-00820	
	%3:	Corrected value [0.1%]	
		Corrected value of P-AXIS-00820	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110684

Default parameter for torque detection for functions with detection of fixed stop greater than 100%.			
Description	The default limit for torque for functions that detect a fixed stop (P-AXIS-00818) must be smaller than 1000.		
Response	Class	1	Warning and correct the value to 1000 [0.1%].

Solution	Class	1	Set value for P-AXIS-00818 less than 1000.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of drive, see P-AXIS-00016	
	%2:	Current value [0.1%]	
		Parameterised incorrect value of P-AXIS-00818	
	%3:	Corrected value [0.1%]	
		Corrected value of P-AXIS-00818	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110685

Default parameter for velocity detection for functions with detection of fixed stop greater than 100%.			
Description	The default limit for velocity for functions that detect a fixed stop (P-AXIS-00817) must be smaller than 1000.		
Response	Class	1	Warning and correct the value to 1000 [0.1%]
Solution	Class	1	Set value for P-AXIS-00817 less than 1000.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of drive see P-AXIS-00016	
	%2:	Current value [0.1%]	
		Parameterised incorrect value of P-AXIS-00817	
	%3:	Corrected value [0.1%]	
		Corrected value of P-AXIS-00817	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110688

Main and secondary encoders cannot use the same signal.			
Description	The same signal was used for the main and secondary encoders. This is not permitted.		
Response	Class	1	Warning output. The value for the secondary encoder is not used.
Solution	Class	1	Check and modify the parameter settings of P-AXIS-00823 and P-AXIS-00824. Parameterise different signals.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the drive.	
	%2:	Error value [-]	
		Parameterised signal of the main encoder.	
	%3:	Error value [-]	
		Parameterised signal of the secondary encoder.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110689

No main and secondary encoders can be parameterised for this drive type.			
Description	No main and secondary encoders can be parameterised for this drive type.		
Response	Class	1	Warning, no reaction.

Solution	Class	1	Check and modify P-AXIS-00823 and P-AXIS-00824 Do not parameterise both parameters for the drive type used.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of drive affected.	
	%2:	Error value [-]	
		Parameterised signal of the main encoder.	
	%3:	Corrected value [-]	
		Parameterised signal of the secondary encoder.	
	%4:	Drive type [-]	
		Drive type P-AXIS-00020 of the axis.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110690

When using P-AXIS-00403 WC_STATE must be configured in the process data.			
Description	The axis parameter P-AXIS-00403 was configured for an axis of type TERMINAL but there is no signal in the process data to indicate that the process data loaded is valid (WC_STATE). The controller can therefore not detect the correct time to set the actual position and to include the calculation of the offset configured in P-AXIS-00403.		
Response	Class	1	Warning output, P-AXIS-000403 is set to zero.
Solution	Class	0	Either .. • configure the data item WC_STATE in the cyclic process data or .. • set the parameter P-AXIS-00403 to zero. .,. and restart the control.
Parameter	%1:	Logical axis number	
		Logical axis number P-AXIS-00016 of axis	
	%2:	Drive type	
		Drive type P-AXIS-00020 of the axis	
	%3:	Error value	
		Parameterised value of P-AXIS-00403	
	%4:	Corrected value	
		New corrected value of P-AXIS-00403	
Error type	-		

ID 110691

Reference monitoring is not supported by this drive type.			
Description	Reference monitoring (*antr.reference_check.**) is not possible for this drive type.		
Response	Class	1	Warning output, parameters are reset to their default values.
Solution	Class	1	--
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110694

P-AXIS-00237 exceeds maximum permissible value.			
Description	The maximum distance (P-AXIS- 00237 - getriebe[i].wsi_meldung) between lubrication pulses was parameterised too large. The permissible maximum value depends on the resolution of the drive and the operation mode of the axis (P-AXIS-00015).		
Response	Class	1	Output a warning to correct the value of the maximum value allowed.
Solution	Class	1	Parameterise a small distance between two lubrication pulses. (P-AXIS-00237)
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Error value [-]	
		Parameterised distance between two lubrication impulses.	
	%3:	Corrected value [-]	
		Corrected maximum value depending on the configuration.	
Error type	-		

ID 110700

Parameter for writing reference position P-AXIS-00584 invalid.			
Description	The parameter P-AXIS-00584 (kenngr.write_reference_position_to_drive) is used to write the homing position from P-AXIS-00152 to the drive before drive-controlled homing. The value specified for the parameter P-AXIS-00584 is invalid.		
Response	Class	1	Warning message output and deactivate function.
Solution	Class	1	Check and modify the settings in P-AXIS-00584
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [-]	
		Incorrect value for P-AXIS-00584	
	%3:	Upper limit value[-]	
		Permissible maximum value for P-AXIS-00584	
	%4:	Corrected value [-]	
		Corrected value for P-AXIS-00584	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 110703

Monitoring touch probe signal not possible due to invalid bit information in P-AXIS-00815.			
Description	The touch probe signal is to be checked at the start of the CANopen measurement run (see P-AXIS-00834). However, P-AXIS-00815 does not specify which bit must be evaluated in the “Touch probe status“ object 0x60B9 or the bit value is invalid.		
Response	Class	1	Output of warning; monitoring the touch probe signal P-AXIS-00834 is disabled.
Solution	Class	1	Check and correct the parameterisation of P-AXIS-00815.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Error value [-]
		Incorrect bit setting in parameter P-AXIS-00815
	%3:	Current value [-]
		Current setting of P-AXIS-00834
	%4:	Corrected value [-]
		Corrected setting of P-AXIS-00834
Error type	2, Error message by data transfer from parameter list into control device.	

2.12 Path preparation error (ID-range 120000-129999)

2.12.1 ID-range 120000-120249

ID 120000 / 120001

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120002

Positive software limit switch exceeded.			
Description	The axis command position reached or exceeded the positive software limit switch (see P-AXIS-00178) of the axis. When using circular motion blocks, the maximum command value of the specified axis at this block is monitored. Offsets, e.g. zero offsets, are already included in the calculation. For more information on software limit switches, see [FCT-A2// Section: Description].		
Response	Class	2	Abort NC program
Solution	Class	6	Include software limit switch when programming. Shift software limit switch by G99. Change software limit switch P-AXIS-00178 in axis configuration.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		Current command value of the axis. For circular block: maximum command value for specified axis for this motion block.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Positive software limit switch P-AXIS-00178.	
	%4:	Current value [0.1 µm or 0.0001°]	
		Start point of the axis in the current motion block (in the ACS coordinate system)	
	%5:	Current value [0.1 µm or 0.0001°]	
		Target point of the axis in the current motion block (in the ACS coordinate system)	
Error type	1, Error message from NC program.		

ID 120003

Negative software limit switch exceeded.			
Description	The axis command position has reached or exceeded the negative software limit switch (see P-AXIS-00177) of the axis. When using circular motion blocks, the minimum command value of the specified axis at this block is monitored. Offsets, e.g. zero offsets, are already included in the calculation. For more information on software limit switch, see [FCT-A2// Section: Description].		
Response	Class	2	Abort NC program

Solution	Class	6	Include software limit switch when programming. Shift software limit switch by G98. Change software limit switch P-AXIS-00177 in axis configuration
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current command value of the axis. For circular block: minimum command value for specified axis for this motion block	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Negative software limit switch P-AXIS-00177.	
	%4:	Current value [0.1 µm or 0.0001°]	
		Start point of the axis in the current motion block (in the ACS coordinate system)	
	%5:	Current value [0.1 µm or 0.0001°]	
		Target point of the axis in the current motion block (in the ACS coordinate system)	
Error type	1, Error message from NC program.		

ID 120006 - 120008

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120011

Positive software limit switch smaller than actual position.			
Description	The programmed positive software limit switch (G99) is smaller than the current axis position. Limit switch is invalid and not saved. For more information on software limit switch see [FCT-A2// Section: Description].		
Response	Class	2	Warning. Program is continued
Solution	Class	1	Correct the programming of the positive software limit switch (G99). Correct the current axis position.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
	%2:	Current value [0.1 µm or 0,0001°]	
		Current axis position.	
	%3:	Upper limit value [0.1 µm or 0,0001°]	
		For the positive software limit switch programmed value.	
Error type	1, Error message from NC program.		

ID 120015

Negative software limit switch greater than current position.

Description	The programmed negative software limit switch (G98) is smaller than the current axis position. Limit switch is invalid and not saved. For more information on software limit switch, see [FCT-A2// Section: Description].		
Response	Class	2	Warning. Program is continued
Solution	Class	1	Correct programming of the negative software limit switch (G98). Correct current position of the axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of axis affected P-AXIS-00016.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Programmed value for the negative software limit switch.	
Error type	1, Error message from NC program.		

ID 120016 - 120018

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120019

Unknown acceleration weighting mode of non-linear slope.			
Description	When acceleration weighting mode was programmed with #SET SLOPE PROFIL[], an invalid parameter was used. For more information on slope, see [PROG// Selecting operation mode].		
Response	Class	2	Program abort.
Solution	Class	6	Program valid mode.
Parameter	%1:	Current value [-]	
		Programmed mode for acceleration weighting.	
Error type	1, Error message from NC program.		

ID 120020

Unknown ramp time weighting mode of non-linear slope.			
Description	When the ramp time weighting mode was programmed with #SET SLOPE PROFIL[], an invalid parameter was used. For more information on slope, see [PROG// Selecting operation mode].		
Response	Class	2	Program abort.
Solution	Class	6	Program valid mode.
Parameter	%1:	Current value [-]	
		Programmed mode for ramp time weighting.	

Error type	1, Error message from NC program.

ID 120021

Unknown slope type.			
Description	When the profile type was programmed in #SLOPE[TYPE...] or #SET SLOPE PROFIL [...], an invalid parameter was used for the slope type . For more information on slope profiles, see [PROG// Selecting operation mode].		
Response	Class	2	Program abort.
Solution	Class	6	Program valid profile type.
Parameter	%1:	Current value [-]	
		Programmed profile type.	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 120022 - 120024

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120025

Maximum difference between radii exceeded.			
Description	When circles are programmed G02, G03 or it may occur also when CNC code is generated in a CAD/CAM system, the calculated radius of circle start point and circle end point may vary. The resulting radius difference between the circle start point and circle end point should not exceed a limit value. The error may occur quite often in conjunction with inactive circle centre point compensation (G164). For further information see [PROG// Controlling centre point offset in circle].		
Response	Class	2	Program abort.
Solution	Class	6	Check and correct programmed circle end point in CNC program.
Parameter	%1:	Incorrect value [0.1 µm or 0.0001°]	
		Current radius difference between circle start point and circle end point.	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
		Permissible radius difference	
	%3:	Current value [0.1 µm or 0.0001°]	
		Circle start point radius	
	%4:	Current value [0.1 µm or 0.0001°]	
		Circle end point radius	
Error type	1, Error message from NC program.		

ID 120026

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120027

Acceleration step 1 is greater than maximum acceleration.			
Description	The axis acceleration can be weighted with G functions G130, G131. The weighted axis acceleration value step 1 P-AXIS-00011 of linear slope exceeds limit of maximum permissible axis acceleration P-AXIS-00008. The weighted value is limited to the maximum permissible axis acceleration. For further information, see [PROG// Section: Acceleration weighting].		
Response	Class	1	None
Solution	Class	1	Keep programmed acceleration weighting within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted value of acceleration level 1 P-AXIS-00011.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible axis acceleration P-AXIS-00008.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120028

Acceleration step 1 is 0.			
Description	The axis acceleration can be weighted with G functions G130, G131. The weighted axis acceleration value step 1 P-AXIS-00011 of linear slope is 0. The weighted value is limited to the minimum permissible axis acceleration. For further information see [PROG// Section: Acceleration weighting].		
Response	Class	1	None.
Solution	Class	1	Use Acceleration weighting in valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted value of acceleration level 1 P-AXIS-00011.	
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]	
		Minimum permissible axis acceleration.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 120029

Acceleration step 2 is greater than maximum acceleration.

Description	The axis acceleration can be weighted with G functions G130, G131. The weighted axis acceleration value step 2 P-AXIS-00012 of linear slope exceeds limit of maximum permissible axis acceleration P-AXIS-00008. The weighted value is limited to the maximum permissible axis acceleration. For further information see [PROG// Section: Acceleration weighting].		
Response	Class	1	None.
Solution	Class	1	Keep programmed acceleration weighting within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [mm/s² or °/s²]	
		Weighted value of acceleration level 2 P-AXIS-00012.	
	%3:	Upper limit value [mm/s² or °/s²]	
		Maximum permissible axis acceleration P-AXIS-00008.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 120030

Acceleration step 2 is 0.			
Description	The axis acceleration can be weighted with G functions G130, G131. The weighted axis acceleration value step 2 P-AXIS-00012 of linear slope is 0. The weighted value is limited to the minimum permissible axis acceleration. For further information see PROG//Section: Acceleration weighting].		
Response	Class	1	None.
Solution	Class	1	Use Acceleration weighting in valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted value of acceleration level 2 P-AXIS-00012.	
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]	
		Minimum permissible axis acceleration.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 120031

Acceleration of non-linear slope is greater than maximum value.			
Description	The axis acceleration can be weighted with G functions G130, G131. The weighted axis acceleration value P-AXIS-00001 of non-linear slope exceeds limit of maximum permissible axis acceleration P-AXIS-00008. The weighted value is limited to the maximum permissible axis acceleration. For further information see [PROG// Section: Acceleration weighting].		
Response	Class	1	None.
Solution	Class	1	Keep programmed acceleration weighting within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted value of acceleration P-AXIS-00001.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible axis acceleration P-AXIS-00008.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 120032

Acceleration of non-linear slope is 0.			
Description	The axis acceleration can be weighted with G functions G130, G131. The weighted axis acceleration value P-AXIS-00001 of non-linear slope is 0. The weighted value is limited to the minimum permissible axis acceleration. For further information see [PROG// Section: Acceleration weighting].		
Response	Class	1	None.
Solution	Class	1	Keep programmed acceleration weighting within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted value of acceleration P-AXIS-00001.	
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]	
		Minimum permissible axis acceleration.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120033

Deceleration of non-linear slope is greater than maximum value.			
Description	The axis acceleration can be weighted with G functions G130, G131. The weighted deceleration value P-AXIS-00002 of non-linear slope exceeds limit of maximum permissible axis acceleration P-AXIS-00008. The weighted value is limited to the maximum permissible axis acceleration. For further information, see [PROG// Section: Acceleration weighting].		
Response	Class	1	None.
Solution	Class	1	Keep programmed acceleration weighting within valid range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted value of acceleration P-AXIS-00002.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible axis acceleration P-AXIS-00008.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120034

Deceleration of non-linear slope is 0.			
Description	The axis acceleration can be weighted with G functions G130, G131. The weighted deceleration value P-AXIS-00002 of non-linear slope is 0. The weighted value is limited to the minimum permissible axis acceleration. For further information see [PROG// Section: Acceleration weighting].		
Response	Class	1	None.
Solution	Class	1	Keep programmed acceleration weighting within valid range.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [1mm/s^2 or 1°/s^2]
		Weighted value of the deceleration P-AXIS-00002.
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]
		Minimum permissible axis acceleration.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 120035

Ramp time for acceleration up-gradation is smaller than minimum value.			
Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00196 of the non-linear slope is smaller than minimum value. The weighted value is limited to the minimum permissible ramp time P-AXIS-00201 . For further information, see [PROG// Section: Ramp time weighting].		
Response	Class	1	None.
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00196.	
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]	
		Minimum permissible ramp time P-AXIS-00201.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120036

Ramp time for acceleration down-gradation is smaller than minimum value.			
Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00195 of the non-linear slope is smaller than minimum value. The weighted value is limited to the minimum permissible ramp time P-AXIS-00201 . For further information, see [PROG// Section: Ramp time weighting].		
Response	Class	1	None
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00195.	
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]	
		Minimum permissible ramp time P-AXIS-00201.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120037

Ramp time for deceleration up-gradation is smaller than minimum value.

Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00198 of the non-linear slope is smaller than minimum value. The weighted value is limited to the minimum permissible ramp time P-AXIS-00201 . For further information, see [PROG// Section: Ramp time weighting].		
Response	Class	1	None.
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00198.	
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]	
		Minimum permissible ramp time P-AXIS-00201.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120038

Ramp time for deceleration down-gradation is smaller than minimum value.			
Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00197 of the non-linear slope is smaller than minimum value. The weighted value is limited to the minimum permissible ramp time P-AXIS-00201 . For further information, see [PROG// Section: Ramp time weighting].		
Response	Class	1	None.
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00197.	
	%3:	Lower limit value [1mm/s^2 or 1°/s^2]	
		Minimum permissible ramp time P-AXIS-00201.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120039

Ramp time for acceleration up-gradation is greater than maximum value.			
Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00196 of non-linear slope is greater than maximum value. The weighted value is limited to the maximum permissible ramp time. For further information, see [PROG// Section: Ramp time weighting].		
Response	Class	1	None.
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00196.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible axis ramp time.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120040

Ramp time for acceleration down-gradation is greater than maximum value.			
Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00195 of non-linear slope is greater than maximum value. The weighted value is limited to the maximum permissible ramp time. For further information see [PROG// Section: Ramp time weighting].		
Response	Class	1	None.
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00195.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible axis ramp time.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120041

Ramp time for deceleration up-gradation is greater than maximum value.			
Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00198 of non-linear slope is greater than maximum value. The weighted value is limited to the maximum permissible ramp time. For further information see [PROG// Section: Ramp time weighting].		
Response	Class	1	None.
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00198.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible axis ramp time.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120042

Ramp time for deceleration down-gradation is greater than maximum value.			
Description	The ramp time can be weighted with G functions G132, G133. The weighted ramp time value P-AXIS-00197 of non-linear slope is greater than maximum value. The weighted value is limited to the maximum permissible ramp time. For further information see [PROG// Section: Ramp time weighting].		
Response	Class	1	None.
Solution	Class	1	Keep the programmed ramp time weighting within the permitted range.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [1mm/s^2 or 1°/s^2]	
		Weighted ramp time value P-AXIS-00197.	
	%3:	Upper limit value [1mm/s^2 or 1°/s^2]	
		Maximum permissible axis ramp time.	

Error type	2, Error message on data transfer of parameter lists to controller.

ID 120050 - 120093

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120094

Tangential transition not possible.			
Description	With an active Akima Spline function (G151), the spline was interrupted by a motion block with stationary main axes and moving tracking axes. In this case, a tangential transition to the following circular block is not possible.		
Response	Class	1	None.
Solution	Class	1	If tracking axis motion is programmed, also program main axis motion.
Error type	1, Error message from NC program.		

ID 120095

Deselection only possible with active spline interpolation.			
Description	Spline interpolation can only be deselected after this function was selected. For more information on spline interpolation, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, select spline function with G151 .
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120096

Selection of spline not possible in this state.			
Description	Activation of spline interpolation only possible if function is inactive. For more information on spline interpolation, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, deselect spline function with G150.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120097

Specified transition type of Akima spline is not permitted.			
Description	The CNC command #SET ASPLINE MODE[] used an invalid parameter for the type of transition. For more information on spline interpolation, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Use correct parameter value for transition mode.
Parameter	%1:	Current value [-]	
		Current parameter.	
	%2:	Lower limit value [-]	
		Lower parameter limit.	
	%3:	Upper limit value [-]	
		Upper parameter limit.	
Error type	1, Error message from NC program.		

ID 120099

Invalid start tangent with Akima spline.			
Description	When the #AKIMA STARTVECTOR[..] or #SET ASPLINE STARTTANG[..] command was used, the components of the start tangent vector were not correctly specified. For more information on spline interpolation, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and use correct start tangent vector components.
Error type	1, Error message from NC program.		

ID 120100

Invalid target tangent with Akima spline.			
Description	When the #AKIMA ENDVECTOR[..] or #SET ASPLINE ZIELTANG[.. command was used, the components of the target tangent vector were not correctly specified. For more information on spline interpolation, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and use correct target vector components.
Error type	1, Error message from NC program.		

ID 120101

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120102

Flushing buffers not allowed while spline interpolation is active.			
Description	With active spline interpolation (G151), no function selection or deselection is possible if they result in the flushing of CNC internal buffer. These are, for example, the following CNC commands / functions: -#FLUSH, #FLUSH CONTINUE, #FLUSH WAIT -#CHANNEL INIT, #SET DEC LR SOLL -#GET CMDPOS, #SET IPO SOLLPOS -#CS ON[], CS OFF -#TRAFO ON/OFF -G200 -Reading synchronous V.E variables For more information on spline interpolation and the listed commands, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Before using the commands listed above, deselect spline interpolation with #SPLINE OFF or with G150
Error type	1, Error message from NC program.		

ID 120104

While spline interpolation is active initialisation of axis position not possible.			
Description	With active spline interpolation (G151), no function selection or deselection is possible if they result in initialisation of the axis positions in the CNC channel. These are, for example, the following CNC commands / functions: - #CHANNEL INIT, #SET DEC LR SOLL - #CS ON[], CS OFF - #TRAFO ON/OFF - G200 For more information on spline interpolation and the listed commands, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	
Solution	Class	6	
Error type	-		

ID 120105

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120106

End of program while active spline interpolation.			
Description	Before program end (M30), the spline interpolation (G151) must be deselected. For more information on spline interpolation see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program execution stop.
Solution	Class	6	Before ending the program, deselect spline interpolation via G150.
Error type	1, Error message from NC program.		

ID 120107

Too many non-relevant blocks in sequence while spline is active.			
Description	The CNC program contains too many non-relevant blocks between two motion blocks to be connected by spline interpolation. Non-relevant blocks are all CNC program blocks that contain no motion. These are, for example, blocks resulting from M function subroutine calls or parameter calculations. With too many non-relevant blocks, internal buffer memory is not sufficient and spline interpolation cannot be executed. The spline is activated by: • G151 or #SPLINE ON (B spline or Akima) • #HSC ON [BSPLINE …] For more information on spline interpolation, see [PROG// Section: B spline interpolation] [PROG// Section: Akima spline interpolation]		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, reduce the number of M functions, subroutine calls and parameter calculations between spline interpolation motion blocks.
Error type	1, Error message from NC program.		

ID 120108 - 120111

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120112

No relevant motion block before activation of spline interpolation.

Description	A motion block with motion path must exist before spline interpolation (G151) is activated with tangential transition (#AKIMA TRANS[...]).
	
	`Example:`
	`%spline`
	`N05 G00 X-100`
	`N10 G01 X0 Y0 F2000 ( 100mm motion path in X )`
	`N15 #AKIMA TRANS[START=TANGENTIAL END=TANGENTIAL]`
	`N10 G151 X3 Y25`
	`N20 X15 Y15`
	`...`
	
	For more information on spline interpolation, see [PROG// Section: Akima spline interpolation]		

Response	Class	2	Program abort.
Solution	Class	6	Program motion block with motion path before selecting spline interpolation.
Error type	1, Error message from NC program.		

ID 120115

Positive software limit switch exceeded.			
Description	With active spline interpolation, the programmed contour element is outside the positive software limit switch. The affected contour element is not traversed. All previous blocks are still moved up to the detected contour element. The position of the software limit switch is defined by parameter P-AXIS-00178 and can be changed in the NC program by G99. See also [FCT-A2// Section: Description].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and offsets, program contour within software limit switches
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
	%2:	Incorrect value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Position software limit switch P-AXIS-00178.	
Error type	1, Error message from NC program.		

ID 120116

Negative software limit switch exceeded.			
Description	With spline interpolation function, the programmed contour element is outside the negative software limit switch. The affected contour element is not traversed. All previous blocks are still moved up to the detected contour element. The position of the software limit switch is defined by parameter P-AXIS-00177 and can be changed in the NC program by G99. See also FCT-A2// Section: Description].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and offsets, program contour within software limit switches

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis.
	%2:	Incorrect value [0.1 µm or 0.0001°]
		Current axis position.
	%3:	Lower limit value [0.1 µm or 0.0001°]
		Negative software limit switch P-AXIS-00177.
Error type	1, Error message from NC program.	

ID 120117

Spline interpolation requires at least 2 axes.			
Description	Before activation of spline interpolation at least 2 axes must be available in CNC channel. For more information on spline interpolation and axis exchange see [PROG// Section: Akima spline interpolation] [PROG// Section: Axis exchange commands]		
Response	Class	2	Program execution stop.
Solution	Class	6	Make sure that there are at least 2 axes before CNC start-up by configuring the axes accordingly, or if there is already an axis in the system, exchange it for a channel via #CALL AX.
Parameter	%1:	Current value [-]	
		Current active number of axes.	
	%2:	Lower limit value [-]	
		Minimum permissible number of axis.	
Error type	1, Error message from NC program.		

ID 120120

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120125

Tangent data of Akima spline are incomplete.			
Description	When the #SET ASPLINE STARTTANG[], #SET ASPLINE ZIELTANG[] commands are used, the components of the start and target tangent vectors were not completely and correctly specified. #AKIMA STARTVECTOR corresponds to #SET ASPLINE STARTTANG[] #AKIMA ENDVECTOR corresponds to #SET ASPLINE ZIELTANG[] For more information on spline interpolation, see [PROG// Section: Akima spline interpolation].		
Response	Class	-	Program abort.

Solution	Class	-	Check CNC program and use correct and complete start and target tangent vectors with 3 components .
Error type	-		

ID 120126

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120129

Corner deviation of polynomial contouring smaller than permissible minimum value.			
Description	When polynomial contouring ((#CONTOUR MODE[]) was parameterised, the programmed value for corner deviation (keyword PATH_DEV) is smaller than the minimum permissible value. For further information see PROG//Section:Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Program corner deviation greater than minimum value.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current corner deviation.	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Minimum permissible corner deviation.	
Error type	1, Error message from NC program.		

ID 120130

Percentage velocity value of polynomial contouring smaller than permissible minimum value.			
Description	When polynomial contouring ((#CONTOUR MODE[]) was parameterised, the programmed value for percentage velocity (keyword VEL) is smaller than the minimum permissible value. For further information see [PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Program percentage velocity greater than minimum value.
Parameter	%1:	Current value [0.1%]	
		Current percentage velocity.	
	%2:	Lower limit value [0.1%]	
		Minimum permissible percentage velocity.	
Error type	1, Error message from NC program.		

ID 120131

Corner distance of polynomial contouring smaller than permissible minimum value.

Description	When polynomial contouring ((#CONTOUR MODE[]) was parameterised, the programmed value for corner distance (keywords PRE_DIST, POS_DIST) is smaller than the minimum permissible value. For further information see [PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Program corner distance greater than minimum value.
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
		Current corner distance	
	%2:	Limit value [0.1 µm or 0.0001°]	
		Minimum permissible corner distance.	
Error type	1, Error message from NC program.		

ID 120133 / 120139

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120140

Too many non-relevant blocks in sequence while polynomial contouring is active.			
Description	In the CNC program, there are too many non-relevant blocks between two motion blocks to be connected by polynomial contouring (G61, G261). Non-relevant blocks are all CNC program blocks that contain no motion. These are, for example, blocks that result from subroutine calls or parameter calculations. Too many non-relevant blocks exceed the capacity of the internal buffer memory and polynomial contouring cannot be calculated. For further information, see [PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, reduce subroutine calls and parameter calculation between polynomial contouring motion blocks.
Error type	1, Error message from NC program.		

ID 120141 / 120143

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120144

Positive software limit switch exceeded.

Description	With active polynomial contouring, the programmed contour element is outside the positive software limit switch. The affected contour element is not traversed. All previous blocks are still moved up to the detected contour element. The position of the software limit switch is defined by parameter P-AXIS-00178 and can be changed in the NC program by G99. See also [PROG// Section: Polynomial smoothing], [FCT-A2// Section: Description].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and offsets, program contour within software limit switches.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Position software limit switch P-AXIS-00178.	
Error type	1, Error message from NC program.		

ID 120145

Negative software limit switch exceeded.			
Description	With active polynomial contouring, the programmed contour element is outside the negative software limit switch. The affected contour element is not traversed. All previous blocks are still moved up to the detected contour element. The position of the software limit switch is defined by parameter P-AXIS-00177 and can be changed in the NC program by G99. See also [PROG// Section: Polynomial smoothing], [FCT-A2// Section: Description].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and offsets, program contour within software limit switches.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Negative software limit switch P-AXIS-00177.	
Error type	1, Error message from NC program.		

ID 120146

Tangential transition with automatic contouring mode not permitted.			
Description	A tangential block transition is programmed in polynomial contouring (G61, G261) with automatic contouring (keyword AUTO_VEL). For more information on contouring, see [PROG// Section: Polynomial contouring].		
Response	Class	-	Program abort.
Solution	Class	-	With automatic contouring mode type as described above, block transition in main axes should not be tangential in the main axes.
Error type	-		

ID 120147

Curvature too large for polynomial contouring.			
Description	With active polynomial contouring (G61, G261), no polynomial contouring with contouring modes (keywords AUTO_DEF, AUTO_VEL) is possible because of an excessive curvature. For further information, see [PROG// Section: Polynomial contouring].		
Response	Class	-	Program abort.
Solution	Class	-	Check blocks length of the adjacent motion blocks during polynomial contouring and extend them if possible.
Error type	-		

ID 120148

At least 2 main axes must be available.			
Description	At least 2 main axes must exist to use polynomial contouring (G61, G261). For further information, see [PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Create CNC configuration with at least 2 main axes in the channel parameter list or by axis exchange.
Parameter	%1:	Current value [-]	
		Current number of axes.	
	%2:	Limit value [-]	
		Necessary number of axes.	
Error type	1, Error message from NC program.		

ID 120149 - 120166

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120167

Programmed cutting speed less than or equal to 0.			
Description	An invalid cutting speed was programmed with the C word for the turning function Constant cutting speed (G96). For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	2	Program abort.
Solution	Class	6	Program a cutting speed S greater than 0.
Error type	1, Error message from NC program.		

ID 120169

Programmed maximum spindle speed less than or equal to 0.

Description	For the turning function Constant cutting speed (G96) the permissible maximum spindle speed can be also limited with the command G196. The programmed value after G196 must be greater than 0. For more information on turning functions, see [PROG// Section: Turning functions].		
Response	Class	2	Program abort.
Solution	Class	6	Program maximum speed greater than 0.
Error type	1, Error message from NC program.		

ID 120170

Axis number of face turning axis is 0.			
Description	For the turning function Constant cutting speed (G96) a face turning axis must be marked and available in the MDS via the parameter P-AXIS-00015 (0x41). For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	2	Program execution stop.
Solution	Class	6	Check MDS and correct entry of facing axis in P-AXIS-00015.
Error type	1, Error message from NC program.		

ID 120171

Missing face turning axis.			
Description	For the turning function Constant cutting speed (G96) a face turning axis must be marked in the MDS via P-AXIS-00015 (0x41). For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	2	Program execution stop.
Solution	Class	6	Check axis configuration, MDS and correct face turning axis settings for P-AXIS-00015.
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 120172

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120173

Missing pitch axis.			
Description	With turning function thread cutting (G33), the pitch axis which is programmed via I, J or K parameter must be available as main axis in CNC channel. For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	2	Program execution stop.

Solution	Class	6	Check axis configuration, depending on programmed I, J, or K parameters, main axis 1, 2 or 3 must be available in CNC channel.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of pitch axis.	
Error type	1, Error message from NC program.		

ID 120174

No motion programmed in pitch axis.			
Description	With the thread cutting (G33) turning function, a motion path must exist in the thread pitch axis programmed with I, J or K. For more information on turning functions, see [PROG// Section: Turning functions].		
Response	Class	2	Program abort.
Solution	Class	6	A real thread pitch greater than 0 must be programmed in the thread pitch axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of thread pitch axis.	
Error type	1, Error message from NC program.		

ID 120175

Axis number of pitch axis is 0.			
Description	With turning function thread cutting (G33), the pitch axis which is programmed via I, J or K parameter must be available as main axis in CNC channel. For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	2	Program execution stop.
Solution	Class	6	Check axis configuration, depending on programmed I, J, or K parameters, main axis 1, 2 or 3 must be available in CNC channel.
Error type	1, Error message from NC program.		

ID 120176

Pitch can not be reached due to limit in axis dynamic.			
Description	With turning function thread cutting (G33), the requested pitch cannot be realized due to axes dynamic limiting. For more information on turning functions see [PROG// Section: Turning functions].		
Response	Class	1	None.
Solution	Class	1	Use smaller pitch value in CNC program.
Parameter	%1:	Current value [1µm/s or 0,001°/s]	
		Required path velocity of face turning and longitudinal turning axis to realize requested pitch.	
	%2:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible path velocity of face turning and longitudinal turning axis.	
Error type	1, Error message from NC program.		

ID 120178

Unknown ramp time weighting mode for non-linear slope.			
Description	An impermissible value was used when the effect was defined for the ramp time weighting by the NC command #SET SLOPE PROFIL[] or the channel parameter P-CHAN-00073.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Use permissible value for the effect of the ramp time weighting.
Parameter	%1:	Current value [-]	
		Current value for the effect of ramp time weighting.	
	%2:	Lower limit value [-]	
		Minimum value for effect of ramp time weighting.	
	%3:	Upper limit value [-]	
		Maximum value for effect of ramp time weighting.	
Error type	1, Error message from NC program.		

ID 120179

Unknown acceleration weighting mode for non-linear slope.			
Description	An impermissible value was used when the effect was defined for acceleration weighting by the command #SET SLOPE PROFIL[] or the channel parameter P-CHAN-00001.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Use permissible value for effect of acceleration weighting.
Parameter	%1:	Current value [-]	
		Current value for effect of acceleration weighting.	
	%2:	Lower limit value [-]	
		Minimum value for effect of acceleration weighting.	
	%3:	Upper limit value [-]	
		Maximum value for effect of acceleration weighting.	
Error type	1, Error message from NC program.		

ID 120180

Deselection of Akima spline in this state not possible.			
Description	With active spline function (G151), a motion block is programmed which has no main axis motion path. For more information on spline functions, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Program tracking axis motion only with main axis motion.
Error type	1, Error message from NC program.		

ID 120183

Tangent specification of Akima spline not complete.

Description	When using the #SET ASPLINE STARTTANG[], #SET ASPLINE ZIELTANG commands, invalid parameter values were programmed for components of start and end tangent vectors. The NC commands used were replaced by the following new NC commands: • #SET ASPLINE STARTTANG by #AKIMA STARTVECTOR • #SET ASPLINE ZIELTANG by #AKIMA ENDVECTOR For more information on spline functions, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and use correct start and end tangent vector components.
Error type	1, Error message from NC program.		

ID 120185 - 120192

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120193

End of program while active spline interpolation.			

Description	At program end (M30), the spline function is still enabled.
	The function has to be deselected before end of CNC program.
	
	`Example:`
	`%spline`
	`N10 G151 X3 Y25`
	`N20 X15 Y15`
	`N30 X23 Y12`
	`N40 X25 Y25`
	`N50 X30 Y35`
	`N60 G150 X50 Y37.5`
	`N70 M30`
	
	For more information on spline function, see
	[PROG// Section: Akima spline interpolation].		

Response	Class	1	None.
Solution	Class	1	Deactivate activated function before program end.
Error type	1, Error message from NC program.		

ID 120199

Change of coordinate system with active spline interpolation not possible.

Description	An active coordinate system cannot be changed or disabled when the spline function is active.
	`Example of correct activation/deactivation:`
	`%spline_ks`
	`N05 #CS ON[0,0,0,0,0,30]`
	`N10 G151 X3 Y25`
	`N20 X15 Y15`
	`N30 X23 Y12`
	`N40 X25 Y25`
	`N50 X30 Y35`
	`N60 G150 X50 Y37.5`
	`N70 #CS OFF`
	`N70 M30`
	
	For more information on spline functions, see [PROG// Section: Akima spline interpolation].		

Response	Class	2	Program abort.
Solution	Class	6	First deactivate spline function then change or deactivate the coordinate system.
Error type	1, Error message from NC program.		

ID 120200 / 120202

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120204

Change of coordinate system with active polynomial contouring not possible.			
Description	If the polynomial contouring function G61, G261 is active, the Cartesian transformation cannot be selected or deselected by the commands #CS ON[] or #CS OFF. For more information on polynomial contouring, see [PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Before selecting or deselecting the Cartesian transformation, polynomial contouring must be deactivated via G260.
Error type	1, Error message from NC program.		

ID 120206

End of program while polynomial contouring is active.			
Description	Before program end (M30) polynomial contouring (G61, G161) must be deselected. For more information on polynomial contouring, see [PROG// Section: Polynomial contouring].		
Response	Class	2	None.
Solution	Class	1	Before program end, deselect polynomial contouring via G260.
Parameter	%1:	Current value [-]	

Error type	1, Error message from NC program.

ID 120207

Inadmissible parameter in parameter setting of polynomial contouring.			
Description	An invalid contouring parameter was programmed in the parameter setting for polynomial contouring (G61, G261). Problem can only occur when using old syntax! For more information on polynomial contouring, see[PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Program correct parameter for contouring type, use new command syntax.
Parameter	%1:	Current value [-]	
		Current programmed contouring mode.	
	%2:	Lower limit value [-]	
		Minimum value for contouring mode.	
	%3:	Upper limit value [-]	
		Maximum value for contouring mode.	
Error type	1, Error message from NC program.		

ID 120208 - 120212

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120213

Corrected programmed feed rate due to stroke of probe.			
Description	In combination with the measuring function G100, different measurement types can be selected via the machine channel parameters (P-CHAN-00057) or CNC programming (#MEAS MODE). With measurement modes 2 und 4, the feedrate measurement is automatically reduced so that value of braking distance can be stopped within the probe stroke P-AXIS-00086. For more information on measurement function see [PROG// Section: Measuring functions].		
Response	Class	1	None

Solution	Class	1	Check permissible stroke of measuring probe P-AXIS-00086 and dynamic characteristics [FCT-D1// Section: Dynamic variables] measurement type type 2: Reduce measurement feed in axis parameter list P-AXIS-00215. measurement type type 4: Reduce measurement F word in CNC program.
Parameter	%1:	Corrected value [1µm/s or 0,001°/s]	
	%2:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 120214

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120215

Stroke of probe (P-AXIS-00086) is 0.			
Description	If the measuring function G100 is used, certain measurement types can check and include the stroke of the measuring probe P-AXIS-00086 during the measurement run. In this case, this parameter must be assigned in the MDS. The path velocity is then reduced during the measurement run to such an extent that it can be stopped within the probe stroke after the measuring probe has responded. For detailed information on measuring function G100, see FCT-C4// Section: Introduction [PROG// Section: Measuring functions].		
Response	Class	2	Program abort.
Solution	Class	6	Set MDS parameter P-AXIS-00086 according to the permissible stroke of the measuring probe.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 120216

Length of block is too small for polynomial contouring.			
Description	For processing a contour line with polynomial contouring (G61, G161), a minimum block length is required. If the block is too short, the contour cannot be contoured. For more information on polynomial contouring, see[PROG// Section: Polynomial contouring].		
Response	Class	1	None
Solution	Class	1	Check block length with interpolation points.

Parameter	%1:	Current value [0.1 µm or 0.0001°]
		Current block length.
	%2:	Lower limit value [0.1 µm or 0.0001°]
		Minimum block length for polynomial contouring.
Error type	1, Error message from NC program.	

ID 120237 - 120248

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

2.12.2 ID-range 120250-120499

ID 120251 / 120253

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120254

Circles not allowed while spline interpolation is active.			
Description	Depending on the selected spline type, only specific path conditions are permitted. For example, when B spline interpolation is active, only linear motion blocks G00 and G01 are permitted. For more information on spline functions, see [PROG// Section: B spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Either use Akima spline type or do not program circular blocks.
Error type	1, Error message from NC program.		

ID 120255

Polynomial contouring not allowed while spline interpolation is active.			
Description	Depending on the selected spline type, only specific path conditions are permitted. For example, when Akima or B spline interpolation is active, no HSC function or polynomial contouring is permitted. For more information on spline functions, see [PROG// Section: Akima spline interpolation] .[PROG// Section: B spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, the HSC function, polynomial contouring and spline should not be enabled at the same time.
Error type	1, Error message from NC program.		

ID 120256 - 120264

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120265

Axis exchange commands are not permitted while spline interpolation is active.

Description	When spline interpolation (G151) is active, an attempt was made to change the number of axes (e.g. using axis exchange commands #PUT AX[], #CALL AX[]). This is only permitted when the spline function is inactive. For more information on spline interpolation, see [PROG// Section: Akima spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Deactivate spline function before programming axis exchange via G150.
Error type	1, Error message from NC program.		

ID 120266

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120267

Braking distance is too small to reach the correct corner velocity.			
Description	With the corner deceleration function G12, G13, the braking distance from the programmed path velocity to corner velocity is either greater than the block path distance or the corner distance parameter. For more information on corner deceleration [PROG// Section : Corner deceleration]		
Response	Class	2	Program abort.
Solution	Class	6	Reduce programmed feed F or increase corner distance parameter.
Parameter	%1:	Current value [-]	
		Programmed path feed.	
	%2:	Limit value [-]	
		Maximum permissible path feed for braking to corner velocity.	
Error type	1, Error message from NC program.		

ID 120268

Change of spline type not permitted while spline interpolation is active.			
Description	The spline type cannot be changed while Akima or B spline interpolation is active. For more information on spline functions, see [PROG// Section: Akima spline interpolation] .[PROG// Section: B spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Before changing spline type in CNC program, deactivate spline function.
Error type	1, Error message from NC program.		

ID 120269 / 120270

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120272

Leaving specified working area.			
Description	For non-Cartesian machine kinematics (e.g. tripod), additional workspace monitoring is executed based on a cylinder depending on the machine dimensions. For more information on kinematic transformations, see [KITRA// Kinematic transformations].		
Response	Class	2	Program abort.
Solution	Class	6	Keep programmed path motion in CNC program in permitted range depending on radius, see below.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Current radius resulting from programmed contour.	
	%2:	Limit value [0.1 10^-3 mm or ø]	
		Permissible radius.	
Error type	1, Error message from NC program.		

ID 120275 - 120284

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120285

Axis exchange not permitted during active spline interpolation.			
Description	No axis exchanges are permitted while Akima or B spline interpolation is active. For more information on spline function, see [PROG// Section: Akima spline interpolation] .[PROG// Section: B spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Deactivate the spline function before releasing or requesting axes in the CNC program.
Error type	1, Error message from NC program.		

ID 120287 - 120292

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120293

State change of transformation is not permitted during active spline interpolation.			
Description	While spline interpolation is active, it is not permitted to select or deselect a kinematic transformation using the #TRAFO ON/OFF command. For more information on spline function, see [PROG// Section: Akima spline interpolation] .[PROG// Section: B spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	The spline function must be deactivated before a kinematic transformation can be selected or deselected.
Error type	1, Error message from NC program.		

ID 120294

State change of transformation is not permitted during polynomial contouring.			
Description	While polynomial contouring G61, G261 is active, it is not permitted to activate or deactivate a kinematic transformation using the #TRAFO ON/OFF command. For more information on polynomial contouring, see[PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Before selecting or deselecting a kinematic transformation, deactivate polynomial contouring using G260 .
Error type	1, Error message from NC program.		

ID 120296

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120297

Flushing buffers not allowed while polynomial contouring is active.

Description	While polynomial contouring (G61, G161), is active, it is not possible to execute any selection or deselection functions which may cause flushing of the CNC channel. These include the following CNC commands / functions, for example: - #FLUSH, #FLUSH CONTINUE, #FLUSH WAIT - #CHANNEL INIT, #SET DEC LR SOLL - #GET CMDPOS, #SET IPO SOLLPOS - #CS ON[], CS OFF - #TRAFO ON/OFF - G200 - Reading synchronous V.E variables For more information on polynomial contouring interpolation and the listed commands, see [PROG// Section: Polynomial contouring]. [PROG// Section: Flushing NC channel]		
Response	Class	2	Program abort.
Solution	Class	6	Do not use any of the commands listed above when polynomial contouring is active.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120299 - 120318

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120319

Buffer full: too many non-relevant blocks programmed in sequence.			
Description	The CNC program contains too many non-relevant blocks between two motion blocks. Non-relevant blocks are all CNC program blocks that do not contain any motion. These are, for example, blocks resulting from M function subroutine calls or parameter calculations.		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, reduce the number of M functions, subroutine calls and parameter calculations between motion blocks.
Error type	1, Error message from NC program.		

ID 120320 - 120328

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	

Solution	Class	8	Requires restart of controller-

ID 120332

De-/Selection of synchronous operation during active spline interpolation.			
Description	While active Akima or B-spline interpolation no activation or deactivation of synchronous operation via #ENABLE AX LINK[], #DISABLE AX LINK[] is allowed. For more information on spline function, see [PROG// Section: Akima spline interpolation]. [PROG// Section: B spline interpolation]		
Response	Class	2	Program execution stop.
Solution	Class	6	Deactivate the spline function before selecting/deselecting synchronous mode.
Error type	1, Error message from NC program.		

ID 120333

Selecting/deselecting synchronous operation while polynomial contouring is active.			
Description	When polynomial contouring G61, G261 is active, it is not permitted to activate or deactivate synchronous operation using the #ENABLE AX LINK[], #DISABLE AX LINK[] commands. For more information on polynomial contouring, see [PROG// Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Before selecting and deselecting synchronous operation, polynomial contouring must be deactivated using G260.
Error type	1, Error message from NC program.		

ID 120334 - 120345

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120346

Buffer full: too many non-relevant blocks programmed in sequence.			
Description	The CNC program contains too many non-relevant blocks between two motion blocks. Non-relevant blocks are all CNC program blocks that do not contain any motion. These are, for example, blocks resulting from M function subroutine calls or parameter calculations.		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, reduce the number of M functions, subroutine calls and parameter calculations between motion blocks.
Error type	1, Error message from NC program.		

ID 120347 - 120369

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120370

Selecting/deselecting spline interpolation not permitted while free-form surface mode is active.			
Description	When the HSC function (#HSC[]) is active, it is not permitted to also select Akima or B spline interpolation with the (G151) command or to deselect these functions using the (G150) command. For more information on spline functions, see [PROG// Section: Akima spline interpolation] .[PROG// Section: B spline interpolation].		
Response	Class	2	Program abort.
Solution	Class	6	Before selecting the spline function, deselect the HSC function via #HSC[OFF].
Error type	1, Error message from NC program.		

ID 120371

Buffer full: too many non-relevant blocks programmed in sequence.			
Description	In the CNC program there are too many non-relevant blocks between two motion blocks to be connected via the HSC function (#HSC ON[..]). Non-relevant blocks are all CNC program blocks that contain no motion. These are, for example, blocks resulting from M functions, subroutine calls, dwell times or parameter calculations. With too many non-relevant blocks, the internal buffer memory is not sufficiently large to execute a polynomial calculation or a spline curve. For more information on the HSC function, see [PROG// Section: Standard HSC programming]		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program. In some cases, the problem can be solved by appropriately placing the HSC function selection and deselection outside the ranges using non-relevant blocks.. Otherwise, reduce the number of M functions, subroutine calls and parameter calculations between motion blocks.
Error type	1, Error message from NC program.		

ID 120372 - 120374

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120377

Selecting free-form surface mode not permitted while spline generation is active.			
Description	Before selecting the HSC function (#HSC[]), disable spline interpolation (G151). For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Disable spline interpolation via G150.
Error type	1, Error message from NC program.		

ID 120378

Changing the mode of HSC processing not permitted in active state.			
Description	The HSC mode cannot be changed when the HSC function (#HSC[]) is active. For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Before changing the HSC mode, deselect the HSC function via #HSC[OFF].
Error type	1, Error message from NC program.		

ID 120379

Maximum contour error is 0 or negative.		
Description	The HSC function (#HSC[]) used an invalid parameter value for contour error (keyword CONTERROR). For more information on HSC function, see [PROG// Section: Standard HSC programming].	
Response	Class	2 Program execution stop.
Solution	Class	6 Use parameter value in valid range.
Parameter	%1:	Current value [-]
		Current parameter value.
Error type	1, Error message from NC program.	

ID 120380 - 120388

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120389

Positive software limit switch exceeded.

Description	In HSC mode, the programmed contour element is outside the positive software limit switch. The affected contour element is not traversed. All previous blocks are still moved up to the detected contour element. The position of the software limit switch is defined by parameter P-AXIS-00178 and can be changed in the NC program by G99. See also [PROG// Section: Set the positive software limit switch] [FCT-A2//: Description].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and offsets, program contour within software limit switches
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Position software limit switch P-AXIS-00178.	
Error type	1, Error message from NC program.		

ID 120390

Negative software limit switch exceeded.			
Description	In HSC mode, the programmed contour element is outside the negative software limit switch. The affected contour element is not traversed. All previous blocks are still moved up to the detected contour element. The position of the software limit switch is defined by parameter P-AXIS-00177 and can be changed in the NC program by G98. See also [PROG// Section: Set negative software limit switch] [FCT-A2//: Description].		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program and offsets, program contour within software limit switches
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current axis position.	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
		Negative software limit switch P-AXIS-00177.	
Error type	1, Error message from NC program.		

ID 120391

Invalid value for cosine of angle between blocks.			
Description	The HSC function (#HSC[]) used an invalid parameter value (keyword COS_PHI_MIN) for the cosine of the block angle. For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Program the parameter value within the valid range.

Parameter	%1:	Current value [-]
		Current parameter value.
	%2:	Lower limit value [-]
		Lower parameter limit.
	%3:	Upper limit value [-]
		Upper parameter limit.
Error type	1, Error message from NC program.	

ID 120392

Factor for block length is 0 or negative.			
Description	The HSC function (#HSC[]) used an invalid parameter value (keyword FACT_BLOCK_LEN) for the block length factor. For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Program the parameter value within the valid range.
Parameter	%1:	Current value [-]	
		Current parameter value.	
Error type	1, Error message from NC program.		

ID 120393

Invalid value for order of block length filter.			
Description	The HSC function (#HSC[]) used an invalid parameter value (keyword BL_FILTER_ORDER) for the order of the block length filter. For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Program the parameter value within the valid range.
Parameter	%1:	Current value [-]	
		Current parameter value.	
	%2:	Lower limit value [-]	
		Lower parameter limit.	
	%3:	Upper limit value [-]	
		Upper parameter limit.	
Error type	1, Error message from NC program.		

ID 120394

Maximum factor angle between blocks is 0 or negative.			
Description	The HSC function (#HSC[]) used an invalid parameter value (keyword FACT_ANGLE) for angle factor. For more information on HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program execution stop.
Solution	Class	6	Use parameter value in valid range.

Parameter	%1:	Current value [-]
		Current parameter value.
Error type	1, Error message from NC program.	

ID 120395

Maximum product of block length and angle is 0 or negative.			
Description	The HSC function (#HSC[]) used an invalid parameter value (keyword FACT_PROD_ANGLE_LEN) for the product of the block length and angle. For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Program the parameter value within the valid range.
Parameter	%1:	Current value [-]	
		Current parameter value.	
Error type	1, Error message from NC program.		

ID 120396

Invalid value for angle filter order.			
Description	The HSC function (#HSC[]) used an invalid parameter value (keyword ANGLE_FILTER_ORDER) for the order of the angle filter. For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Program the parameter value within the valid range.
Parameter	%1:	Current value [-]	
		Current parameter value.	
	%2:	Lower limit value [-]	
		Lower parameter limit.	
	%3:	Upper limit value [-]	
		Lower parameter limit.	
Error type	1, Error message from NC program.		

ID 120397 - 120404

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120405

Tool data are inconsistent.

Description	The specified tool geometry is not known. The geometry must be a ball end cutter or a shaft end cutter.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the tool geometry of the tool in the tool data list.
Parameter	%1:	Current value [-]	
		Tool number	
	%2:	Current value [0.1 µm or 0,0001°]	
		Tool radius	
	%3:	Current value [-]	
		Wrong tool geometry	
	%4:	Current value [0.1 µm or 0,0001°]	
		Internal state of tool geometry compensation	
Error type	1, Error message from NC program.		

ID 120406

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120407

Unknown function block.			
Description	The received function block is not valid in the current state of tool geometry compensation. Only linear blocks are permitted in the selected state of TGC.		
Response	Class	2	Abort NC program.
Solution	Class	3	Check selection and deselection of TGC.
Parameter	%1:	NC block number [-]	
		Internal state of tool geometry compensation	
Error type	1, Error message from NC program.		

ID 120422 - 120424

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120425

At least one axis is missing for the coordinate system.

Description	At least 3 coordinate axes are required for the orientation interpolation. Currently, the channel has not enough axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the NC program and, if necessary, check the axis exchange sequence.
Parameter	%1:	Incorrect value [-]	
		Current number of axes in channel	
	%2:	Limit value [-]	
		Required number of axes	
Error type	1, Error message from NC program.		

ID 120428 - 120430

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120433

While HSC function is active with mode 2 the programmed function is not supported.			
Description	When the HSC function (#HSC[]) is active in spline mode, the following functions cannot be programmed. - G100, measurement function. - G95, feed rate per revolution. - G61, G261 polynomial contouring - G96, constant cutting speed. - G33, thread cutting. For more information on the HSC function [PROG// Section: Standard HSC programming]		
Response	Class	2	Program abort.
Solution	Class	6	Deselect the HSC function (#HSC[OFF]) before programming the functions listed above
Error type	1, Error message from NC program.		

ID 120434

While HSC function is active with mode 2 flushing NC channel is not permitted.

Description	When the HSC function (#HSC[]) is active in spline mode, it is not possible to select or deselect functions that lead to a flushing of CNC channel. These are, for example, the following CNC commands / functions: - #FLUSH, #FLUSH CONTINUE, #FLUSH WAIT - #CHANNEL INIT, #SET DEC LR SOLL - #GET CMDPOS, #SET IPO SOLLPOS - #CS ON[], CS OFF - #TRAFO ON/OFF - G200 - Reading synchronous V.E variables For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	Deselect the HSC function with #HSC[OFF] before using the commands listed above
Error type	1, Error message from NC program.		

ID 120435

While HSC function is active with mode 2 dwell time programming is not permitted.			
Description	When the HSC function (#HSC[]) is active in spline mode, it is not permitted to program a dwell time (G04). For more information on the HSC function, see [PROG// Section: Standard HSC programming].		
Response	Class	2	Program abort.
Solution	Class	6	When dwell time is to be programmed, deselect the HSC function with #HSC[OFF] previously.
Error type	1, Error message from NC program.		

ID 120438

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120439

Buffer full: too many non-relevant blocks programmed in sequence.			
Description	The CNC program contains too many irrelevant blocks between two motion blocks in combination with the tangential tracking (#CAXTRACK ON) function. Non-relevant blocks are all CNC program blocks that contain no motion. These are, for example, blocks resulting from M function subroutine calls or parameter calculations. For more information on the HSC function, see [PROG// Section: Automatic axis tracking (C axis tracking)].		
Response	Class	2	Program abort.

Solution	Class	6	Check CNC program, reduce the number of M functions, subroutine calls and parameter calculations between motion blocks.
Error type	1, Error message from NC program.		

ID 120440 - 120458

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120459

Corner deviation of tracking axis while polynomial contouring is active exceeds limit.			
Description	During polynomial contouring (G61, G261), the specified corner deviation of the tracking axis was not maintained. For more information on contouring, see [PROG// Section: Polynomial contouring].		
Response	Class	1	None.
Solution	Class	1	Increase permissible corner deviation in tracking axis.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the tracking axis.	
	%2:	Current value [0.1 µm or 0.0001°]	
		Current deviation of tracking axis.	
	%3:	Upper limit value [0.1 µm or 0.0001°]	
		Permissible deviation of tracking axis.	
	%4:	NC block number [-]	
		Block number of contouring block 1	
	%5:	NC block number [-]	
		Block number of contouring block 2	
Error type	1, Error message from NC program.		

ID 120460

Intermediate position during polynomial contouring cannot be determined.			
Description	In polynomial contouring using the G61, G261 functions, either block length of the contouring blocks are too small or the parameter for corner deviation or corner distance in the #CONTOUR MODE[] command is too small to calculate the polynomial contour line. For more information on polynomial contouring, see[PROG// Section: Polynomial contouring], FCT-D3// Section: Method 4, Polynomial contouring		
Response	Class	1	No reaction.
Solution	Class	1	Check block length of both contouring blocks. Depending on the mode, select a greater corner distance or corner deviation in the CNC command #CONTOUR MODE[].

Parameter	%1:	Incorrect value [0.1 µm or 0.0001°]
		Sum of corner distances of the first and second polynomial contouring blocks.
	%2:	NC block number [-]
		Block number of first polynomial contouring block
	%3:	NC block number [-]
		Block number of second polynomial contouring block
Error type	1, Error message from NC program.	

ID 120462

Feed axis transformation with selected axis group not possible.			
Description	The restrictions must be taken into account for the definition of feed axis groups (#FGROUP) in connection with polynomial and circular motions. Regardless of the #FGROUP command, the following commands / functions always use the first two or three axes as path feed axes: - Polynomial contouring, G61, G261 - Spline function, G151 - HSC function, #HSC[ON] With circular motion G02, G03, either only circular axes (and helical axis) are feed axes or the other axes are feed axes. For more information on feed groups, [PROG// Section: Definition of feed axes].		
Response	Class	1	None.
Solution	Class	1	Define feed axis group with the first two or three axes depending on axis configuration.
Error type	1, Error message from NC program.		

ID 120463

Corner distance for iterative calculation of intermediate point too small.			
Description	In polynomial contouring via functions G61, G261, the parameter of corner deviation or corner distance in the #CONTOUR MODE[] command is too small to calculate the polynomial contour line. The error can also occur when the contour elements are near the resolution limit of the CNC. In this case, contouring cannot be used. For more information on polynomial contouring, see [PROG// Section: Polynomial contouring], FCT-D3].		
Response	Class	2	Program abort.
Solution	Class	7	Depending on the mode, select a greater corner distance or corner deviation in the CNC command #CONTOUR MODE[].

Parameter	%1:	Incorrect value [0.1 µm or 0.0001°]
		Detected corner distance.
	%2:	Lower limit value [0.1 µm or 0.0001°]
		Lower limit value for corner distance.
	%3:	NC block number
		First block of the transition.
	%4:	NC block number
		Second block of the transition.
Error type	1, Error message from NC program.	

ID 120464 - 120468

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120469

Circle angle below minimum.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 120470

No oscillation path.			

Description	Invalid oscillation movement programmed.
	`Wrong:`
	`%osc`
	`N10 X[OSC_ON 1ST_POS=50 2ND_POS=50 FEED=1000]`
	`N20 M30`
	
	`Right:`
	`%osc`
	`N10 X[OSC_ON 1ST_POS=-100 2ND_POS=100 FEED=1000]`
	`N20 M30`
	
	For more information on oscillation movement, see [FCT-A8// Section: Description], [PROG// Section: Oscillating axes].		

Response	Class	2	Program execution stop.
Solution	Class	6	Check oscillation zero position and excursion, or oscillation reversal positions. There must be an oscillation path distance resulting from the difference between the oscillation reversal positions.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Current value [0.1 µm or 0,0001°]
		Oscillation reversal position 1.
	%3:	Current value [0.1 µm or 0,0001°]
		Oscillation reversal position 2.
Error type	1, Error message from NC program.	

ID 120471

Oscillating motion outside software limit switches.			
Description	The programmed motion of the oscillating axis exceeds software limit switches. For more information on oscillating motion, see [FCT-A8// Section: Description [PROG// Section: Oscillating axes].		
Response	Class	2	Program abort.
Solution	Class	6	Check oscillation zero position and excursion or the oscillation reversal positions. Program oscillating motion inside software limits.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [0.1 10^-3 mm or ø]
		Oscillation reversal position 1.
	%3:	Current value [0.1 10^-3 mm or ø]
		Oscillation reversal position 2.
	%4:	Lower limit value [0.1 10^-3 mm or ø]
		Negative software limit switch P-AXIS-00177
	%5:	Upper limit value [0.1 10^-3 mm or ø]
		Positive software limit switch P-AXIS-00178
Error type	1, Error message from NC program.	

ID 120472

Ambiguous axis position on selection of a kinematic transformation.			
Description	With a kinematic transformation was selected, the machine axis positions must be inside the valid solution range of these transformation functions.		
Response	Class	2	Program abort.
Solution	Class	6	Position axes within valid range when kinematic transformation is not active.

Parameter	%1:	Logical axis number [-]
		Logical axis number of axis affected P-AXIS-00016.
	%2:	Current value [0.1 µm or 0.0001°]
		Current invalid axis position.
	%3:	Limit value [0.1 µm or 0.0001°]
		Upper limit of valid range.
	%4:	Limit value [0.1 µm or 0.0001°]
		Upper limit of valid range.

Error type	1, Error message from NC program.

ID 120473

Active area can not be overwritten, area is not saved.			
Description	An attempt was made to overwrite an active area. Overwrite areas means to define an area with the same existing ID. See also [PROG// Defining a control area] and [FCT-C14// Description]. An area already exists with the commanded ID and it is currently activated. To overwrite an existing area, it should be inactive. **Example with error:** %areaoverwrite N10 #CONTROL AREA ON [**ID2**] N20 #CONTROL AREA BEGIN [**ID2** WORK …… (< error **Corrected example:** %areaoverwrite N10 #CONTROL AREA ON [**ID2**] N15 #CONTROL AREA OFF [**ID2**] N20 #CONTROL AREA BEGIN [**ID2** WORK.. (< overwrite		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify NC program; deactivate the area to be overwritten.
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

ID 120474

Maximum number of areas exceeded, area is not saved.			
Description	The maximum number of stored areas is limited. This limit was exceeded. The limit value is defined by a constant in the CNC. If you need to have a greater number of areas stored, contact your controller manufacturer. For this reason, the assigned area could not be saved. See also [FCT-C14].		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program; clear the areas not required using the command #CONTROL AREA CLEAR[ID xx]
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Limit value [-]	
		Maximum number of defined areas.	
	%3:	Current value [-]	
		Number of defined areas.	
Error type	1, Error message from NC program.		

ID 120475

Z plane definition incorrect, minimum greater than maximum.			
Description	The description of the commanded areas is incorrect. The minimum value of the Z planes is greater than the maximum value. Constant limit values for the Z direction can be specified as limit values in the Z plane. See also [PROG// Monitoring the workspace and protection space] and [FCT-C14// Description]. **Example:** %zmin_zmax N10 #CONTROL AREA BEGIN…. MIN_EXCUR=55 MAX_EXCUR=-20] (> error **Solution:** %zmin_zmax N10 #CONTROL AREA BEGIN…. MIN_EXCUR=-20 MAX_EXCUR=55] (> correct		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Correct the limit values for the Z plane.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Lower limit value [0.1 10^-3 mm or ø]	
		Lower limit of the Z plane	
	%3:	Upper limit value [0.1 10^-3 mm or ø]	
		Upper limit of the Z plane	
Error type	1, Error message from NC program.		

ID 120476

Start and end points must be the same for defining a polygonal area.			
Description	When defining polygonal areas, the last dot of the polygon must coincide with the first dot of the polygon. A continuous polygon can be then guaranteed. See [PROG// Monitoring the workspace and protection space]. **Example:** %start not end N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X70 Y70 G01 F10000 G90 N30 X70 Y0 N40 X0 Y0 N50 X0 Y70 **N60 X75 Y75** N70 #CONTROL AREA END (> error **Solution:** %start not end N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X70 Y70 G01 F10000 G90 N30 X70 Y0 N40 X0 Y0 N50 X0 Y70 **N60 X70 Y70** N70 #CONTROL AREA END (> correct		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program.

Parameter	%1:	Identification number [-]
		Area ID
	%2:	Expected value [0.1 10^-3 mm or ø]
		Starting point of area definition.
	%3:	Current value [0.1 10^-3 mm or ø]
		End point of area definition.
Error type	1, Error message from NC program.	

ID 120477

Definition of polygon is invalid. Overlapping is not permitted.			
Description	The polygonal area must be a convex envelope. This means that individual polygon lines may not intersect each other. **Example:** %poly intersects N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X65 Y0 N30 X-65 Y0 **N40 X65 Y65** **N50 X-65 Y65** N60 X65 Y0 N100 #CONTROL AREA END (> error **Solution:** %poly intersects N10 #CONTROL AREA POLY BEGIN….MAX_EXCUR=55] N20 X65 Y0 N30 X-65 Y0 **N40 X-65 Y65** **N50 X65 Y65** N60 X65 Y0 N100 #CONTROL AREA END (> error		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Error value [0.1 10^-3 mm or ø]	
		Intersection of polygon lines	
Error type	1, Error message from NC program.		

ID 120478

No area with this ID available.			
Description	You programmed an area command with an invalid area ID. The programmed area ID does not exist.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program. Correct the area ID.
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

ID 120479

The first block at the start of a circular area definition must be linear.			
Description	The first motion block after starting a definition of a circular area must be linear. A circular area definition consists of a linear block as starting point and a circular block as circle description. See also [PROG// Section: Monitoring the workspace and protection space] and [FCT-C14// Section: Description]. **Example with error:** %circular area N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU… N20 **G02 G162 I70 J150** N30 G01 X150 Y200 F10000 N40 #CONTROL AREA END N50 M30 **Corrected example:** %circular area N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU… N20 **G01 X150 Y200 F10000** (start point) N30 G02 G162 I70 J150 N40 #CONTROL AREA END N50 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Expected value [-]	
		Intersection of polygon lines	
	%3:	Error value [-]	
Error type	1, Error message from NC program.		

ID 120480

The second block to define a circular area must be circular.			
Description	The second motion block after starting a definition of a circular area must be a circular block for the circular description of the area. A circular area definition consists of two motion blocks, one linear at the starting point, and one circular to describe the circle See also [PROG// Monitoring the workspace and protection space] and [FCT-C14// Description]. **Example:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 **N30 G01 X150 Y200 F10000** N40 #CONTROL AREA END N50 M30 (> error **Solution:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 (start point) **N30 G02 G162 I70 J150** N40 #CONTROL AREA END N50 M30 (> correct)		
Response	Class	2	Abort NC program processing.

Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

ID 120481

A linear and a circular block are required for a circular area.			
Description	A circular area consists of a linear block as starting point and a circular block as circular description. The second motion block after starting a definition of a circular area must be followed by a circular block as circle description. See also [PROG// Monitoring the workspace and protection space] **Example with error:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 **N30 G01 X150 Y200 F10000** N40 #CONTROL AREA END N50 M30 **Corrected example:** %circular area N10 #CONTROL AREA START [ID51 PROT CIRC MIN_EXCU… N20 G01 X150 Y200 F10000 (start point) **N30 G02 G162 I70 J150** N40 #CONTROL AREA END N50 M30		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Error value [-]	
		Intersection of polygon lines	
Error type	1, Error message from NC program.		

ID 120482

A full circle without end point is required for a circular area.

Description	A circular area consists of a linear block as starting point and a circular block as circular description.
	A circular block must be a programmed full circle since areas can only consist of a full circle.
	See also [PROG// Monitoring the workspace and protection space]
	
	Example with error:
	`%circular area`
	`N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU…`
	`N20 G01 X150 Y200 F10000`
	`N30 G03 G162 I50 J0 X250 Y200 F10000`
	`N40 #CONTROL AREA END`
	`N50 M30`
	
	`Corrected example:`
	`%circular area`
	`N10 #CONTROL AREA BEGIN [ID51 PROT CIRC MIN_EXCU…`
	`N20 G01 X150 Y200 F10000 (start point)`
	`N30 G03 G162 I70 J150`
	`N40 #CONTROL AREA END`
	`N50 M30`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID	
	%2:	Initial value [0.1 10^-3 mm or ø]	
		Starting point for the circle description.	
	%3:	Final value [0.1 10^-3 mm or ø]	
		Target point for the circle description.	
Error type	1, Error message from NC program.		

ID 120483

Point violates protection area.			
Description	The start or end point of the assigned movement violates an activated protection space. A protective space must not be violated by a tool centre point. It may consist of a circular or polygonal area with a constant third plane. See also FCT-C14.		
Response	Class	6	Stop all axes immediately
Solution	Class	6	Dependence defined, activated protection spaces
Parameter	%1:	Identification number [-]	
		Area Id number of violated protection area.	
	%2:	Current value [-]	
		Point of violation.	
	%3:	Initial value [0.1 µm or 0,0001°]	
	%4:	Initial value [0.1 µm or 0,0001°]	
Error type	1, Error message from NC program.		

ID 120484

Point outside workspace.

Description	The start or end point of the commanded motion lies outside the activated workspace. A workspace may not be exited by a tool centre point. It may consist of circular or polygonal areas with a constant third plane. See also FCT-C14..		
Response	Class	6	Abrupt stop for all axes.
Solution	Class	6	Take into account the defined and activated workspaces
Parameter	%1:	Current value [-]	
		Bitmask value of the participating axes in the workspace. e.g. 7 or binary 0b111 if the first three axes participate in the workspace.	
	%2:	Start or target value [0.1 10^-3 mm or ø]	
		Point that is outside the workspace.	
	%3:	Start or target value [0.1 10^-3 mm or ø]	
		Point in the active coordinate system if the workspace affects the first three axes and a Cartesian coordinate system is active.	
Error type	1, Error message from NC program.		

ID 120485

Motion path violates protection area.			
Description	The programmed movement path between start point and target point violates an active protection area. A protective space must not be violated by a tool centre point. It may consist of a circular or polygonal area with a constant third plane. See also FCT-C14.		
Response	Class	2	Stop all axes immediately
Solution	Class	6	Dependence defined, activated protection spaces
Parameter	%1:	Identification number [-]	
		Area Id number of violated protection area.	
	%2:	Initial value [0.1 µm or 0,0001°]	
		Start point of movement.	
	%3:	End value [0.1 µm or 0,0001°]	
		Target point of movement.	
	%4:	Error value [0.1 µm or 0,0001°]	
		Intersection point violates protection area.	
Error type	1, Error message from NC program.		

ID 120486

Motion path exits workspace.			
Description	Start point and target point of programmed motion path is outside active valid workspace. A workspace may not be exited by a tool centre point. A workspace may consist of a circular or polygonal area with constant third plane. See also FCT-C14.		
Response	Class	2	Stop all axes immediately
Solution	Class	6	Take into account the defined activated workspaces

Parameter	%1:	Logical axis number [-]
		Area Id number of exited workspace.
	%2:	Incorrect value [0.1 µm or 0.0001°]
		Start point of motion.
	%3:	Upper limit value [0.1 µm or 0.0001°]
		End point of motion.
	%4:	Current value [0.1 µm or 0.0001°]
		Intersection point of motion path on exiting the workspace.
	%5:	Current value [0.1 µm or 0.0001°]
Error type	1, Error message from NC program.	

ID 120487

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 120488

Maximum number of points allowed for polygon exceeded.			
Description	The number of points to define a polygonal area is restricted to one limit value. This limit was exceeded. A polygonal area consists of a number of points representing a closed polygon. The start point must be the same as the end point. See also [FCT-C14//Polygonal control areas].		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program. Reduce the number of points in the area definition
Parameter	%1:	Identification number [-]	
		Affected area ID	
	%2:	Limit value [-]	
		Maximum number of points for a polygonal area.	
	%3:	Error value [-]	
		Current number of points for a polygonal area.	
Error type	1, Error message from NC program.		

ID 120489

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 120490

Enable tool geometry compensation not possible because of inactive kinematic transformation.			
Description	Function in preparation.		
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 120491

Kinematic transformation without orientation vector active.		
Description	Function not completed.	
Response	Class	2
Solution	Class	3
Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	1, Error message from NC program.	

ID 120492

Invalid plane normal vector in tool geometry compensation.		
Description	Function not completed.	
Response	Class	2
Solution	Class	3
Error type	1, Error message from NC program.	

ID 120493

Invalid path normal vector in tool geometry compensation.		
Description	Function not completed.	
Response	Class	2
Solution	Class	3
Error type	1, Error message from NC program.	

ID 120494

Invalid path tangent vector in tool geometry compensation.		
Description	Function not completed.	
Response	Class	2
Solution	Class	3
Error type	1, Error message from NC program.	

ID 120495 / 120496

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120498

No workspace monitoring possible with activated coordinate system.			
Description	When a machine coordinate system (#CS ON) is active, workspace monitoring cannot take place given the current development status. This function will be developed in the future.		
Response	Class	6	Movement stop.
Solution	Class	6	Disable the CS function
Error type	1, Error message from NC program.		

ID 120499

Active workspace or protection space cannot be cleared.			
Description	An active workspace or protection area cannot be cleared with the "CLEAR" command. This is intended to prevent the deletion of currently monitored areas. **Example:** %areaclear N10 #CONTROL AREA ON [**ID2**] N20 #CONTROL AREA CLEAR[**ID2**] ..... (Error **Solution:** %areaclear N10 #CONTROL AREA ON [**ID2**] N15 #CONTROL AREA OFF [**ID2**] N20 #CONTROL AREA CLEAR [**ID2**]< clear		
Response	Class	1	Continue NC program processing., the area is not cleared.
Solution	Class	1	Deactivate the workspace or protection space using the command #CONTROL AREA OFF [ID xx] before deleting it.
Parameter	%1:	Identification number [-]	
		Area ID	
Error type	1, Error message from NC program.		

2.12.3 ID-range 120500-120749

ID 120500

Not all three main axes of (A)CS known. No workspace monitoring possible.			
Description	The first three main axes must be defined in order to use workspace monitoring. They are then used as X, Y and Z coordinates for workspace monitoring.		
Response	Class	2	Abort NC program processing
Solution	Class	6	Check your axis configuration
Error type	1, Error message from NC program.		

ID 120501 - 120503

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120504

List interpreter can not evaluate given list.			
Description	During the controller start-up, the list interpreter cannot evaluate a file path or list name entered in the start-up list. Possibly some of these data are wrong.		
Response	Class	3	Start-up of the control is aborted.
Solution	Class	7	Check the notation of the corresponding file path or list name and correct as required.
Error type	-		

ID 120505

Transfer scene and collision data in path preparation failed.			
Description	During controller start-up or update, the list interpreter cannot evaluate the collision/ scene list due to invalid or unknown information.		
Response	Class	3	Controller start-up or data update is aborted.
Solution	Class	7	Check the entries in the collision/scene list and correct if necessary.
Error type			

ID 120506 / 120507

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120508

The list contains an unknown element.			
Description	During interpretation of the collision-/scene-lists, an unknown list element is detected.		
Response	Class	3	Start-up of the control is continued.
Solution	Class	7	Remove or modify the unknown list element in the corresponding list.
Error type			

ID 120509

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120512

Distance between two monitored solids in collision detection smaller than warning level.			
Description	When the axes are moved, the minimum distance between two monitored bodies is below the specified minimum limit. The minimum distance was specified in the definition of collision pairs in #COLL DEF[PAIR<nam>... WARNING<distance>].		
Response	Class	1	Warning output
Solution	Class	1	Adapt path motion or increase warning level.
Parameter	%1:	Current value [-]	
		Name of first monitored object	
	%2:	Current value [-]	
		Name of second monitored object	
	%3:	Current value [0.1 µm or 0.0001°]	
		Current distance between the two bodies	
	%4:	Limit value [0.1 µm or 0.0001°]	
		Default limit value for output of a warning	
Error type	1, Error message from NC program.		

ID 120513

Distance between two monitored solids in collision detection smaller than minimum value.			
Description	When the axes are moved, the minimum distance between two monitored bodies is below the specified minimum limit. The minimum distance was specified in the definition of collision pairs in #COLL DEF[PAIR<nam>... ERROR<distance>].		
Response	Class	2	Abort the motion or stop and “wait for motion” to continue
Solution	Class	0	Adapt path motion

Parameter	%1:	Current value [-]
		Name of first monitored object
	%2:	Current value [-]
		Name of second monitored object
	%3:	Current value [0.1 µm or 0.0001°]
		Current distance between the two bodies
	%4:	Limit value [0.1 µm or 0.0001°]
		Specified limit value
Error type	1, Error message from NC program.	

ID 120514

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120515

Cancellation of request without axis registration.			
Description	A wrong axis was requested and this request already stopped e.g. by a reset.		
Response	Class	1	Warning, no reaction
Solution	Class	1	Check NC-program regarding axes exchange
Parameter	%1:	Logical axis number [-]	
		Number of wrong axis, requested before	
Error type	3, Error message from communication.		

ID 120516

No active collision detection.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120517

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120518

Leading angle outside of the permissible range of value.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Incorrect value [0.1 µm or 0.0001°]	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 120519

Incline angle outside of the permissible range of value.			
Description			
Response	Class	2	
Solution	Class	3	
Parameter	%1:	Incorrect value [0.1 µm or 0.0001°]	
	%2:	Upper limit value [0.1 µm or 0.0001°]	
	%3:	Lower limit value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 120521

Incline angle changes sign.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
	%2:	Current value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 120522 / 120523

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120524

Intermediate point of contour mode not within start/ end/ corner point.			
Description	The calculated intermediate point of the contouring curve should be within the convex envelope of the start/target and end points. This is currently not the case.		
Response	Class	2	Warning output
Solution	Class	1	Check contouring curve
Parameter	%1:	Incorrect value [0.1 µm or 0.0001°]	
		X coordinate of intermediate point	
	%2:	Y coordinate of intermediate point	
	%3:	Incorrect value [0.1 µm or 0.0001°]	
		Z coordinate of intermediate point	
	%4:	Number of previous block	
	%5:	NC block number [0.1 µm or 0.0001°]	
		Number of following block	
Error type	1, Error message from NC program.		

ID 120525

Synchronous operation in mirror link mode with polynomials not permitted.			
Description	When defining an axis coupling via #SET AX LINK or #AX LINK [NBR], mirroring was defined by the coupling factor = -1. This mirroring is not possible with an active polynomial curve.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the NC program and activate either mirroring or polynomial contouring.
Error type	1, Error message from NC program.		

ID 120526 - 120528

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120529

NC command #SCENE PRINT [FILE=<file>] not available.			
Description	The NC command #SCENE PRINT [FILE=<file>] can only be executed synchronously.		
Response	Class	3	Abort NC command
Solution	Class	1	Modify NC program accordingly
Error type	1, Error message from NC program.		

ID 120531

Unknown file extension of geometry file in collision detection.			
Description	The file extension of the enveloping body was incorrectly specified in the command #SCENE ADD[GOBJECT<>... FILE<name >]. The extension “dcp” is expected.		
Response	Class	2	Abort NC program
Solution	Class	6	Check correct name of file
Parameter	%1:	Error value	
		Name of geometrical description of collision object	
Error type	1, Error message from NC program.		

ID 120532

The geometry file specified in geometry object was not found.			
Description	The file of the enveloping body was not found in the command #SCENE ADD[GOBJECT<>... FILE<name >].		
Response	Class	2	Abort NC program
Solution	Class	6	Check correct name of file
Parameter	%1:	Error value	
		File names	
	%2:	Current value	
		Name of graphical object	
	%3:	Current value	
		Path to enveloping body file	
Error type	1, Error message from NC program.		

ID 120533

The geometry file has an incorrect format (decomposed).			
Description	The file of the enveloping body in the command #SCENE ADD[GOBJECT<>... FILE<name >] has an incorrect data format.		
Response	Class	2	Abort NC program
Solution	Class	6	Check the data format of the file, possibly carry out decomposition again.
Parameter	%1:	Error value	
		File names	
	%2:	Current value	
		Name of graphical object	
Error type	1, Error message from NC program.		

ID 120536

Existing active pair cannot be redefined.			
Description	An attempt was made to define a collision pair by the command #COLL DEF [PAIR=<>...] which already exists.		
Response	Class	2	Abort NC program
Solution	Class	6	Check NC program

Parameter	%1:	Current value
		Name of first graphical object
	%2:	Current value
		Name of second graphical object
	%3:	Current value
		Name of collision pair
Error type	1, Error message from NC program.	

ID 120537

Graphical object for defining collision pair not found in scene database.			
Description	A graphical object specified in the collision list does not exist. The collision pair could not be generated.		
Response	Class	1	Break of execution
Solution	Class	1	
Parameter	%1:	Error value [-]	
		Name of graphical object	
	%2:	Name of collision pair	
Error type	-		

ID 120539

Maximum number of permissible collisions pairs reached.			
Description	The maximum number of collision pairs was reached. No more pairs can be defined.		
Response	Class	2	Abort execution
Solution	Class	6	Reduce the number of collision pairs
Parameter	%1:	Current value	
		Current number of pairs	
	%2:	Error value	
		Maximum number of pairs	
Error type	1, Error message from NC program.		

ID 120540

Maximum number of geometry objects in collision detection reached.			
Description	The maximum number of graphical objects is reached. No more graphical objects can be defined.		
Response	Class	2	Abort execution
Solution	Class	6	Reduce number of graphical elements
Parameter	%1:	Current value	
		Current number of graphical elements	
	%2:	Incorrect value	
		Maximum number of graphical elements	
Error type	1, Error message from NC program.		

ID 120542

Parameter change during active B-spline interpolation not possible.			
Description	An attempt was made to change the parameterisation (e.g. #HSC [BSPLINE TRACK_DEV=5 MERGE=0 AUTO_OFF_PATH=1]) while B spline contour smoothing is active. This is currently only possible when the spline is disabled.		
Response	Class	1	The new value of the parameter is not adopted
Solution	Class	1	Modify the NC program
Parameter	%1:	NC block number [-]	
		Block number of parameter change	
	%2:	Current value [-]	
		Parameter type: 0x0040 PATH_DEV 0x0080 TRACK_DEV 0x0100 MAX_PATH_LENGTH 0x0200 MAX_ANGLE 0x0400 AUTO_OFF_G00 0x0800 MERGE_DEV 0x1000 MERGE_RETRY 0x2000 MERGE_WINDOW 0x4000 PATH_SPLIT or AUTO_OFF_PATH 0x8000 TRACK_SPLIT or AUTO_OFF_TRACK 0x10000 LIMIT_CORNER_DIST	
	%3:	Current value [-]	
		Current parameter value	
	%4:	Incorrect value [-]	
		New value of parameter	
Error type	1, Error message from NC program.		

ID 120543

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120544

In #HSC command the given number of blocks for merge is too big.			
Description	The number of blocks considered for the merge process of the B spline is too high (see #HSC [... MERGE_WINDOW<number>]).		
Response	Class	1	New parameter for merge window is discarded.
Solution	Class	1	Reduce size of merge window

Parameter	%1:	NC block number [-]
		Block number of parameter change
	%2:	Current value [-]
		Current parameter value
	%3:	Limit value [-]
		Maximum size of merge window
	%4:	Corrected value [-]
		Limited new parameter value
Error type	1, Error message from NC program.	

ID 120545 - 120547

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120548

Buffer full: too many non-relevant blocks programmed in sequence.			
Description	In the CNC program, there are too many non-relevant blocks between two motion blocks to be processed during pre-segmentation. Non-relevant blocks are all CNC program blocks that do not contain any motion. These are, for example, blocks resulting from M function subroutine calls or parameter calculations. With too many non-relevant blocks, internal buffer memory is not sufficient and spline interpolation cannot be executed.		
Response	Class	2	Program abort.
Solution	Class	6	Check CNC program, reduce the number of M functions, subroutine calls and parameter calculations between spline interpolation motion blocks.
Error type	1, Error message from NC program.		

ID 120549 / 120550

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120551

Geometry object is too complex for collision detection.			
Description	The number of surfaces or edges of the graphical object exceeds the internal maximum value. This limitation prevents the collision detection from handling graphical objects that are too high-resolution.		
Response	Class	2	Abort NC program.

Solution	Class	6	Reduce resolution of graphical object
Parameter	%1:	Current value	
		Name of graphical object	
	%2:	Current value	
		Current number of surfaces and edges	
	%3:	Limit value	
		Maximum number of surfaces and edges	
Error type	1, Error message from NC program.		

ID 120552 / 120553

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120554

Mirror link operation mode not allowed with oscillating axis.			
Description	The oscillation function (Z[OSC ON..]) is part of the axis-specific functions of the NC channel. The axis moves independently and asynchronously in relation to path axes. Therefore, path functions such as Soft Gantry (#ENABLE AX LINK) are not fully supported in this axis operation mode. In this case, Soft Gantry mode was activated with a coupling factor of -1 (mirror link mode) and the master axis is an oscillating axis. Soft Gantry mode in this case is only permitted if the coupling factor is 1. For more information on soft gantry mode see [PROG// Section: Extended programming of axis couplings].		
Response	Class	2	Program abort
Solution	Class	6	Check CNC program and correct the operation mode (#SET AXLINK[]).
Parameter	%1:	Logical axis number [-]	
		Logical axis number of the soft gantry slave axis affected.	
	%2:	Current value [-]	
		Currently selected link operation mode of soft gantry.	
Error type	1, Error message from NC program.		

ID 120555 / 120556

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	Program execution stop
Solution	Class	8	Requires restart of controller-

ID 120557

Could not allocate new B spline segment.

Description	When internally processing the B spline segments, the new segment could not be read in because there is no more memory available internally.		
Response	Class	2	Program abort
Solution	Class	6	If necessary, force output of the segments by reducing the MERGE_WINDOW P-CHAN-00127.
Error type	1, Error message from NC program.		

ID 120558

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120559

Maximum number of buffer slots reached. Oldest segment is released.			
Description	During preparation of B spline segments for merge, the maximum number of segments was reached. See [PROG// Section: B spline interpolation]		
Response	Class	1	Warning
Solution	Class	1	If necessary, force output of the segments by reducing the MERGE_WINDOW P-CHAN-00127.
Parameter	%1:	Current value [-]	
		Number of segments	
	%2:	Upper limit value [-]	
		Maximum number of segments	
	%3:	Current value [-]	
		Number of forced segments output	
	%4:	NC block number [-]	
	%5:	NC block number [-]	
Error type	1, Error message from NC program.		

ID 120560 / 120561

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120562

Error during writing in file.			
Description	An error occurred while writing to scene/collision report file. The action was triggered by a previous #SCENE PRINT SYN [FILE=<name>] or #COLL PRINT [FILE=<name>].		
Response	Class	1	Warning
Solution	Class	1	Check name and path of output file
Parameter	%1:	Current value [-]	
		Name of the report file	
Error type	1, Error message from NC program.		

ID 120563

The weighting factor for jerk is smaller than permissible minimum value.			
Description	The HSC factor specified in the channel list for the weighting of jerk P-CHAN-00143 (see hsc.gen.jerk_weighting in [0.1%]) is too small.		
Response	Class	1	Warning
Solution	Class	1	Value is limited to minimum
Parameter	%1:	Current value [0.1%]	
		Specified value	
	%2:	Limit value [0.1%]	
	%3:	Corrected value [0.1%]	
Error type	-		

ID 120564

The minimum block length for dynamic segmentation is smaller than permissible minimum value.			
Description	The HSC value specified in the channel list for the minimum block length of relevant blocks P-CHAN-00140 (see: hsc.gen.min_segment_length in [0.1um]) was too small. Blocks below this length are not specified in finer resolution in the dynamic consideration.		
Response	Class	1	Warning
Solution	Class	1	Value is limited to minimum
Parameter	%1:	Current value [0.1 µm or 0.0001°]	
	%2:	Limit value [0.1 µm or 0.0001°]	
	%3:	Corrected value [0.1 µm or 0.0001°]	
Error type			

ID 120566

Polynomial contouring mode is only possible in default configuration of feed axes.

Description	For G61 contouring mode, the feed rate must be mapped by the first three coordinate system axes. See also [PROG//Section: Polynomial contouring].		
Response	Class	2	Program abort.
Solution	Class	6	Check default setting in channel list P-CHAN-00011 or #FGROUP programming.
Parameter	%1:	Logical axis number [-]	
		Axis with incorrect setting	
	%2:	Incorrect value [-]	
		Axis participates / does not participate in feed rate mapping.	
	%3:	Expected value [-]	
Error type	1, Error message from NC program.		

ID 120567

Changing definition of feed axis impossible while polynomial contouring is active.			
Description	Feed rate mapping on axes may not be changed while contouring mode is active (see #FGROUP).		
Response	Class	2	
Solution	Class	6	Check default setting in channel list P-CHAN-00011 or #FGROUP programming.
Parameter	%1:	Logical axis number [-]	
	%2:	NC block number [-]	
		Number of previous block in block transition	
	%3:	Current value [-]	
		Setting in the current block that indicates whether the axis is involved in feed rate mapping.	
	%4:	Expected value [-]	
		Setting in the previous NC block that indicates whether the axis is involved in feed rate mapping.	
Error type	1, Error message from NC program.		

ID 120568 / 120569

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120570

Weighting factor for jerk is greater than permissible maximum value.			
Description	The HSC factor specified in the channel list for the weighting of the jerk P-CHAN-00143 (see hsc.gen.jerk_weighting in [0.1%]) is too big.		
Response	Class	1	Warning

Solution	Class	1	Value is limited to maximum value.
Parameter	%1:	Current value [0.1%]	
		Specified value	
	%2:	Limit value [0.1%]	
	%3:	Corrected value [0.1%]	
Error type	-		

ID 120571

Weighting factor for jerk with kinematic transformation smaller than permissible minimum value.			
Description	The HSC factor specified in the channel list for the weighting of jerk with kinematic transformation P-CHAN-00142 (see hsc.gen.jerk_weighting_5ax in [0.1%]) is too small.		
Response	Class	1	Warning
Solution	Class	1	Value is limited to minimum
Parameter	%1:	Current value [0.1%]	
		Specified value	
	%2:	Limit value [0.1%]	
	%3:	Corrected value [0.1%]	
Error type			

ID 120572

Weighting factor for jerk with kinematic transformation greater than permissible maximum value.			
Description	The HSC factor specified in the channel list for weighting jerk with kinematic transformation P-CHAN-00142 (see hsc.gen.jerk_weighting_5ax in [0.1%]) is too large.		
Response	Class	1	Warning
Solution	Class	1	Value is limited to maximum value.
Parameter	%1:	Current value [0.1%]	
		Specified value	
	%2:	Limit value [0.1%]	
	%3:	Corrected value [0.1%]	
Error type			

ID 120575

Invalid activation of tool radius compensation when tool geometry compensation is active in face compensation mode.
Description

Response	Class	2	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120576

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120577

Front tracking inside collision monitoring was reset.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120578

Error during calculation of the contouring curve in mode DIST_SOFT.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 120579

Plane change in work or protection area definition not permitted.

Description	The machining plane may not be changed during the definition of a work or protection area (see [FCT-C14// Definition of work and protection areas]). **Example with error:** #CONTROL AREA START [ID1 PROT CIRC MIN_EXCUR=-50 MAX_EXCUR=50] G01 Y100 Z100 F1000 **G19** G02 G162 Y100 Z100 J-100 F1000 #CONTROL AREA END **Corrected example:** **G19** #CONTROL AREA START [ID1 PROT CIRC MIN_EXCUR=-50 MAX_EXCUR=50] G01 Y100 Z100 F1000 G02 G162 Y100 Z100 J-100 F1000 #CONTROL AREA END		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Identification number [-]	
		Identifier of affected work or protection area	
Error type	1, Error message from NC program.		

ID 120580 / 120581

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120582

Maximum number of transformed work or protection areas exceeded.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Identification number [-]	
	%2:	Incorrect value [-]	
	%3:	Upper limit value [-]	
Error type	1, Error message from NC program.		

ID 120583

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120585

License limitation: number of simultaneously moved axes exceeded.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Limit value [-]	
Error type	1, Error message from NC program.		

ID 120586

Necessary function of path preparation not enabled in CNC-configuration list.			
Description	In the start-up parameters, the CNC system is configured in terms of the number of channels, number of axes in the system, etc. The number of axes in the system is set by the CNC. In addition, the active functions of path planning of each channel can also be defined with P-STUP-00060. With this parameters, the system can be tuned for minimizing runtime and memory size in multi channel systems, depending on the requirements of the user. All required basic functions are activated in the default setting. In the present case, a function was used in the NC program that is deactivated in the start-up list or is not activated as a default function.		
Response	Class	2	Abort NC program processing.
Solution	Class	7	Correction of NC program or configuration setting according to P-STUP-00060
Parameter	%1:	Expected value [-]	
		Missing function in P-STUP-00060	
	%2:	Expected value [-]	
		Missing internal function- ID	
	%3:	Current value [-]	
		Currently active internal functions	
	%4:	State [-]	
		Control block-ID of programmed function	
Error type	1, Error message from NC program.		

ID 120587 - 120600

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120603

Lift axis must not be programmed during lift movement.			
Description	The lift axis is commanded by the controller during the lift movement. Therefore, it is not possible to program the motions of the lift axis during the lift movement.		
Response	Class	2	Lift movement is not performed. Stop all axes.
Solution	Class	6	Remove the programmed movements for the lift axis between LIFT_START and LIFT_END.
Error type	1, Error message from NC program.		

ID 120604

No further start of lift movement may be programmed during active lift movement.			
Description	Another lift movement was programmed between LIFT_START and LIFT_END. This is not permitted.		
Response	Class	2	Lift movement is not performed. All axis stop.
Solution	Class	6	Remove second lift movement.
Parameter	%1:		
	Block number of LIFT_START of the first lift movement		
Error type	1, Error message from NC program.		

ID 120605

Different axis by <axis>[LIFT_START] and <axis>[LIFT_END] programmed.			
Description	A lift movement started for a certain axis must also be ended with this axis. Please check the programming syntax of LIFT_START and LIFT_END		
Response	Class	2	Lift movement is not performed. All axis stop.
Solution	Class	6	Correct the syntax
Parameter	%1:		
	Logical axis index of lift axis at LIFT_START		
	%2:		
	Logical axis index of lift axis at LIFT_END		
Error type	1, Error message from NC program.		

ID 120606

Programmed NC block is not allowed during active lift movement.			
Description	The programmed command is not allowed during a lift movement. Please remove the impermissible command.		
Response	Class	2	Lift movement is not executed. Stop all axes.
Solution	Class	6	Remove command
Error type	1, Error message from NC program.		

ID 120607

Axis exchange of active lift axis not allowed.	
Description	

Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120608

Lift START/END point not found to insert linear axis movement.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120611

Akima spline is not allowed in lift segments.			
Description	Akima spline interpolation is currently not permitted for lift segments. Please change the type of interpolation.		
Response	Class	2	Lift movement is not performed. All axis stop.
Solution	Class	6	Change geometry interpolation method
Error type	1, Error message from NC program.		

ID 120614

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120615

M/H functions with synchronisation type MNS_SNS with and without path-related pre-output in the same NC block not possible.			
Description	M and H functions with synchronisation type MNS_SNS with and without path-related pre-output are not possible in the same NC block. Either N/H functions with or without path-related pre-output may be used in an NC line.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program.
Error type	1, Error message from NC program.1, Error message from NC program.		

ID 120617/120618

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 120619

Calculated contour deviation of spline segment greater than maximum value.			
Description			
Response	Class	1	
Solution	Class	0	
Error type	1, Error message from NC program.		

ID 120621

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120622

Maximum lift height POS_LIMIT not specified.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120623

Maximum lift height POS_LIMIT is above positive software limit switch.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120624

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120625

M/H functions with synchronisation type MNS_SNS with pre-output and #FLUSH CONTINUE in the same NC block not possible.			
Description	The function “Path controlled M code output” is enabled (FCT_M_PRE_OUTPUT in MDS parameter for controller start-up). Programming is not allowed when the #FLUSH CONTINUE function is active.		
Response	Class	2	Program abort
Solution	Class	6	Check and correct the NC program.
Error type	1, Error message from NC program.		

ID 120626

Tracking axis on main axis index.			
Description	The axis for tangential tracking tangential tracking P-CHAN-00095 is on a main axis index. See also [PROG// Section: Automatic axis tracking (C axis tracking)]		
Response	Class	2	Abort NC program
Solution	Class	6	Check configuration, only tracking axis with axis index > 2 allowed.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of concerned axis P-AXIS-00016.	
	%2:	Current value [-]	
		Axis index of the tracking axis.	
Error type	1, Error message from NC program.		

ID 120627

Rapid mode acceleration is greater than maximum value.			
Description	Rapid mode acceleration can be weighted with the function G231. Weighting is possible up to the maximum value a_max. Greater values are limited automatically.		
Response	Class	1	Warning.
Solution	Class	1	Reduce weighting value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [mm/s² or °/s²]	
		Weighted rapid mode acceleration value.	
	%3:	Limit value [mm/s² or °/s²]	
		Maximum axis acceleration P-AXIS-00008.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 120628

Rapid mode acceleration is zero.			
Description	Rapid mode acceleration can be weighted with the function G231. Weighting is possible up to a minimum value. Smaller values are limited automatically.		
Response	Class	1	Warning.
Solution	Class	1	Increase weighting value.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Current value [mm/s² or °/s²]
		Weighted rapid mode acceleration value.
	%3:	Limit value [mm/s² or °/s²]
		Minimum acceleration value.[mm/s² or °/s²]
Error type	2, Error message by data transfer from parameter list into control device.	

´

ID 120629

Rapid mode ramp time is smaller than minimum value.			
Description	Rapid mode ramp time can be weighted with the function G233. Weighting is possible up to a minimum value. Smaller values are limited automatically.		
Response	Class	1	Warning.
Solution	Class	1	Increase weighting value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [mm/s² or °/s²]	
		Weighted rapid mode ramp time value.	
	%3:	Limit value [mm/s² or °/s²]	
		Minimum ramp time value.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 120630

Rapid mode ramp time is greater than maximum value.			
Description	Rapid mode ramp time can be weighted with the function G233. Weighting is possible up to a maximum value. Greater values are limited automatically.		
Response	Class	1	Warning.
Solution	Class	1	Increase weighting value.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [mm/s² or °/s²]	
		Weighted rapid mode ramp time value.	
	%3:	Limit value [mm/s² or °/s²]	
		Maximum ramp time value.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 120631 - 120639

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	

Solution	Class	8	Requires restart of controller-

ID 120640

Buffer FIFO full.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120641 - 120643

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120644

Internal error during distance calculation of collision monitoring.		
Description		
Response	Class	2
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	1, Error message from NC program.	

ID 120645

Internal error during distance calculation of collision monitoring.		
Description		
Response	Class	2
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	1, Error message from NC program.	

ID 120646

Internal error during activation of collision monitoring.		
Description		
Response	Class	2
Solution	Class	6

Parameter	%1:	Incorrect value
	%2:	Incorrect value
	%3:	Current value
	%4:	Current value
Error type	1, Error message from NC program.	

ID 120647

Collision pair definition by #COLL RESTORE incorrectly reset.		
Description		
Response	Class	1
Solution	Class	3
Parameter	%1:	Incorrect value [-]
	%2:	Incorrect value [-]
	%3:	Incorrect value [-]
	%4:	Current value [-]
	%5:	Expected value [-]
Error type	-	

ID 120648

Solid figure definition by #SCENE RESTORE incorrectly reset.		
Description		
Response	Class	1
Solution	Class	3
Parameter	%1:	Incorrect value [-]
	%2:	Incorrect value [-]
Error type	-	

ID 120649

Coordinate plane definition by #SCENE RESTORE incorrectly reset.		
Description		
Response	Class	1
Solution	Class	3
Parameter	%1:	Incorrect value [-]
	%2:	Incorrect value [-]
Error type	-	

ID 120650 / 120651

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120652

Measurement during active polynomial contouring not possible.			

Description	While polynomial contouring is active, it is impossible to execute a measurement run.
	
	Example:
	`N010 G261`
	`N020 G0 Z100`
	`N030 G0 Z200 Y50`
	`N040 G100 X100 F1000 (=> Error 120652)`
	`…`
	`N980 G260`
	`N990 M30`
	
	Correct:
	`N010 G261`
	`N020 G0 Z100`
	`N030 G0 Z200 Y50`
	`N050 G260`
	`N040 G100 X100 F1000`
	`N050 G261`
	`…`
	`N980 G260`
	`N990 M30`		

Response	Class	2	Abort NC program.
Solution	Class	6	Correct NC program, deactivate polynomial contouring during measurement run
Parameter	%1:	NC block number [-]	
		Block number in which the error occurred.	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120653 / 120654

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120655

Programmed difference too small to heterodyne automatically the Contour Mode.			
Description			
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Current value [0.1 µm or 0.0001°]
	%2:	Limit value [0.1 µm or 0.0001°]
Error type	1, Error message from NC program.	

ID 120656

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 120658

This NC function is not allowed during active geometry filtering.			
Description	The function #HSC ON[SURFACE] is used in the NC program. NC functions that result in NC channel initialisation are not permitted in this active geometry filter. This includes e.g.: • Measurements [G100] • #FLUSH WAIT • #CHANNEL INIT • Read synchronous V.E. variables		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program; change the NC program flow, e.g. deselect with #HSC OFF before e.g. the measuring process.
Error type	1, Error message from NC program.		

ID 120659 - 120661

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 120662

Smoothing is already active. This NC command has no effect.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120663

Error initialising kinematic transformation.

Description	When kinematic transformation is selected with command #TRAFO ON, #RTCP ON leads to an initialisation error in the kinematic transformation. For more information on kinematic transformations, see [KITRA// Kinematic transformations], [PROG// Section: 5-axis function].		
Response	Class	2	Program abort.
Solution	Class	6	Check selected kinematic identifier, check and correct kinematic transformation parameters in the channel parameters or tool management.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120665

Maximum number of multiple rotations of section tube per block exceeded.			
Description	With active function tube profile machining #CYL[] the number of rotations of rotary axis is exceeded in programmed path motion. This can only occur with the relative programming (G91) of path motion. Positioning always takes place on the shortest distance using absolute programming (G90). For further information on the function, see [FCT-M5// Round pipe and section tube machining]		
Response	Class	2	Program execution stop.
Solution	Class	6	Check movement programming, split movement in more than one block.
Parameter	%1:	Current value [-]	
		Number of rotations in programmed movement command	
	%2:	Limit value [-]	
		Maximum number of rotations per movement command	
Error type	1, Error message from NC program.		

ID 120666

Too many axes in channel for geometry filtering. Please use a different function.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120667

Invalid additional axis movement by transformation during active lift movement.			
Description	The lift axis is additionally moved during the lift movement by a kinematic transformation. This is not permitted See [FCT-A11// Section: Overview]		
Response	Class	2	Lift movement is not performed. Stop all axes.
Solution	Class	6	Either deactivate kinematic transformation before the lift movement or remove the lift movement (LIFT_START , LIFT_END).

Error type	1, Error message from NC program.

ID 120668

No axes in channel for geometry filtering. Please deactivate this function.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120669

Changing number of axes while geometry filtering is active not permitted.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120670

Smoothing is not active but will be disabled. This NC command has no effect.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120671

Tube projection, programmed contour exceeds Y limits.			
Description	In tube processing, the contour must be programmed within the permissible limits in the projection plane. The symmetrical limit value is programmed in #CYL[] command; the default value is 0.25*radius of tube. This case is a contour description which exceeds the permissible limits. For further information on this function, see [FCT-M5// Tube processing].		
Response	Class	2	Program abort.
Solution	Class	6	Keep programmed path motion in the CNC program within the allowed range depending on limits, see below.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Current Y position of programmed contour.	
	%2:	Lower limit value [0.1 10^-3 mm or ø]	
		Permissible positive Y value.	
	%3:	Upper limit value [0.1 10^-3 mm or ø]	
		Permissible negative Y value.	
Error type	1, Error message from NC program.		

ID 120672

A transition block was inserted with path smoothing.			
Description			
Response	Class	2	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120674

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120676

Intersection calculation during Cartesian forward transformation impossible.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120677

Intersection calculation during Cartesian backward transformation impossible.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120678

Flushing buffers not allowed between SYNC IN and SYNC OUT.

Description	NC commands resulting in NC channel flushing may not be programmed between SYNC IN and SYNC OUT. These are, for example, the following CNC commands / functions: • #FLUSH, #FLUSH CONTINUE, #FLUSH WAIT • #CHANNEL INIT, #SET DEC LR SOLL • #GET CMDPOS, #SET IPO SOLLPOS • #CS ON[], CS OFF • #TRAFO ON/OFF • G200 • Read synchronous V.E . variables Excerpt from the NC program: … N130 X=39.819 A[**SYNC IN** G01 POS=130 DIST=70] N140 X=1.257 N160 X=200 A[**SYNC OUT** G91 G0 POS160] … See also Synchronising an axis in coordinated motion (SYNC IN / OUT)		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Program affected NC command after SYNC OUT.
Error type	1, Error message from NC program.		

ID 120679

Program end while SYNC IN is active.			
Description	With Synchronising an axis in coordinated motion (SYNC IN / OUT) the area must be ended by SYNC OUT. In this case, the program was simply ended; this is not permissible.		
Response	Class	2	1, Error message from NC program.
Solution	Class	3	Check and modify the NC program. End synchronisation by SYNC_OUT.
Error type	1, Error message from NC program.		

ID 120680

Timeout while waiting for SYNC OUT.			
Description	The start and end of the range must be known to adapt the feed rate in the synchronisation area. If SYNC OUT is not found, program processing is aborted. See also Synchronising an axis in coordinated motion (SYNC IN / OUT)		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check whether SYNC OUT was programmed. If required, delete programmed blocks that are not absolutely necessary between SYNC IN and SYNC OUT.
Error type	1, Error message from NC program.		

ID 120681

Invalid superimposed movement with independent axis.			
Description	During active tube profile machining ((#CYL[EDGES=...]), an independent axis was programmed and is an input axis of the tube transformation. Crossing the profile rounding of the tube leads to invalid superimposed movement. Programming of independent movement is only allowed on plane sections of profile.		
Response	Class	2	Program abort.
Solution	Class	6	Check and correct the NC program.
Error type	1, Error message from NC program.		

ID 120682

Dynamics calculation: calculated feasible maximal velocity to small.			
Description			
Response	Class	2	
Solution	Class	7	
Parameter	%1:	Current value [1µm/s or 0,001°/s]	
	%2:	Limit value [1µm/s or 0.001°/s]	
	%3:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120683

Maximum feasible path feed in synchronisation area too small!			
Description	Due to the geometrical conditions, it is not possible to traverse at the required velocity for synchronous machining with the synchronisation area.		
Response	Class	2	
Solution	Class	3	Check the constancy of your programming; possibly use smoothing methods to correct inconstancies.
Error type	1, Error message from NC program.		

ID 120684

Transition velocity smaller than synchronisation velocity.			
Description	Due to the geometrical conditions, it is not possible to continue at the required velocity for synchronous machining within the synchronisation area.		
Response	Class	2	
Solution	Class	3	Check the constancy of your programming, especially for kinks at the transition to the synchronisation area.
Error type	1, Error message from NC program.		

ID 120685

Invalid tool orientation.			
Description			
Response	Class	2	

Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120686 - 120688

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120689

Motion path violates protection area. PCS coordinates.			
Description	The programmed movement path between start point and target point violates an active protection area. A protective space must not be violated by a tool centre point. It may consist of a circular or polygonal area with a constant third plane. See also FCT-C14. This is a subsequent error message from P-ERR-120485. [▶ 1491] It is output if a protection space is violated with active Cartesian transformation #(A)CS. The message contains the start, target point and the violating point, where the movement path enters the protection area in PCS coordinates.		
Response	Class	6	Stop all axes immediately
Solution	Class	6	Dependence defined, activated protection spaces
Parameter	%1:	Identification number [-]	
		Area Id number of violated protection area.	
	%2:	Current value [-]	
		Start point of movement in PCS coordinate system.	
	%3:	Initial value [0.1 µm or 0,0001°]	
		Target point of movement in PCS coordinate system.	
	%4:	End value [0.1 µm or 0,0001°]	
		Intersection point violates protection area in PCS coordinate system.	
	%5:	Error value [0.1 µm or 0,0001°]	
Error type	1, Error message from NC program.		

ID 120690

Motion path exits workspace. PCS coordinates.			
Description	Start point and target point of programmed motion path is outside active valid workspace. A workspace may not be exited by a tool centre point. A workspace may consist of a circular or polygonal area with constant third plane. See also FCT-C14. This is a follow-on error message from P-ERR-120486 [▶ 1491]. It is output if the workspace is exited when Cartesian transformation #(A)CS is active. The message contains the start, target point and the violating point where the motion path enters the protection area in PCS coordinates.		
Response	Class	6	Stop all axes immediately

Solution	Class	6	Take into account the defined activated workspaces
Parameter	%1:	Identification number [-]	
		Area Id number of exited workspace.	
	%2:	Current value [-]	
		Start point of movement in PCS coordinate system.	
	%3:	Initial value [0.1 µm or 0.0001°]	
		Target point of movement in PCS coordinate system.	
	%4:	Initial value [0.1 µm or 0.0001°]	
		Intersection of the motion path when leaving the workspace in the PCS coordinate system.	
	%5:	Incorrect value [0.1 µm or 0.0001°]	
Error type	1, Error message from NC program.		

ID 120691

Only linear motion blocks G00/G01 are allowed for definition of polygonal areas.			

Description	An invalid motion block was used in the definition of a polygonal control area of the workspace monitoring function (see [FCT-C14// Description]). Only linear motion blocks G00 or G01 are permitted for defining polygonal control areas.
	
	Example:
	`N010 G17 F10000`
	`N020 #CONTROL AREA BEGIN`
	`[ID=1 PROT POLY MIN_EXCUR = 0 AX_EXCUR = 200]`
	`N030 G01 X0 Y0`
	`N040 G01 X100 Y0`
	`N050 G02 X100 Y100 J+50 (error 120691)`
	`N060 G01 X100 Y0`
	`N070 G01 X0 Y0`
	`N080 #CONTROL AREA END`
	
	Correct:
	`N010 G17 F10000`
	`N020 #CONTROL AREA BEGIN`
	`[ID=1 PROT POLY MIN_EXCUR = 0 AX_EXCUR = 200]`
	`N030 G01 X0 Y0`
	`N040 G01 X100 Y0`
	`N050 G01 X100 Y100`
	`N060 G01 X100 Y0`
	`N070 G01 X0 Y0`
	`N080 #CONTROL AREA END`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the definition of the control area in the NC program.
Parameter	%1:	Identification number [-]	
		Area ID of affected control area	
	%2:	Error value [-]	
		Incorrect type of motion block	
Error type	1, Error message from NC program.		

ID 120692

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120693

Programmed pre-output of M-function not possible, reserved memory too small.			
Description	NC blocks must be buffered in order to use the pre-output of M functions. The pre-output of this M function is not possible since the memory parameters are too small. The memory is specified using P-CHAN-00603 (alternatively P-STUP-00061).		
Response	Class	1	Warning, no reaction.. The M function is not pre-output.
Solution	Class	1	Check and modify P-CHAN-00603.
Parameter	%1:	Expected value [0.1 10^-3 mm or ø]	
		Set path of the pre-output	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Distance covered since program start	
	%3:	Upper limit value [0.1 10^-3 mm or ø]	
	%4:	Current value[-]	
	%5:	Block number [-]	
Error type	1, Error message from NC program.		

ID 120694

Circle contour overlapping on lateral surface.			
Description	During active function tube profile machining (#CYL[]), a circle was programmed but its maximum range exceeds the profile circumference of the workpiece. This would result in an invalid contour overlap.		
Response	Class	2	Program abort.
Solution	Class	6	Check and correct the NC program; maximum circle limits should be inside profile circumference.
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
	%2:	Limit value [0.1 10^-3 mm or ø]	
Error type	1, Error message from NC program.		

ID 120695

SYNC IN missing before SYNC OUT.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 120696

Start position outside of range of software limit switches.			
Description	The start point of the measuring block is outside the range limited by the software limit switches. This is not permitted.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the start point of the measuring block.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current position of the axis	
	%3:	Lower limit value [0.1 10^-3 mm or ø]	
		Negative software limit switch (P-AXIS-00177)	
	%4:	Upper limit value [0.1 10^-3 mm or ø]	
		Positive software limit switch (P-AXIS-00178)	
Error type	1, Error message from NC program.		

ID 120697

Independent axis movement during active spline interpolation not permitted.			

Description	No independent axis motion may be programmed during active spline interpolation.
	
	Example:
	`N010 #HSC ON [BSPLINE]`
	`N020 G0 X100 Y200 Z300 A[INDP_ASYN G0 G90 POS180]` □ **error!**
	`N030 …`
	`N100 #HSC OFF`
	`N110 M30`		

Response	Class	2	Program abort.
Solution	Class	6	Correct the NC program: • Deselect spline interpolation before independent axis movement, e.g. #HSC OFF and then select it again. • Replace independent axis motion by path motion, z.B.: N020 G0 X100 Y200 Z300 **Independent axis movement during active spline interpolation not permitted.**
Parameter	%1:		
		Current state of spline interpolator	
	%2:		
		Current NC block identifier	
	%3:		
		Block number of affected NC block	
Error type	1, Error message from NC program.		

ID 120698

Contour mode change during active DIST_SOFT polynomial contouring not permitted.

Description	The polynomial contouring mode is not permitted when the polynomial contouring function is active. If it is necessary to change the mode, deselect the contouring mode first (G260) and then switch it on again (G261).
	
	Example:
	`N010 #CONTOUR MODE [DEV PATH_DEV=0.2]`
	`N020 G261`
	`N030 G0 X100 Y100`
	`N040 #CONTOUR MODE [DIST_SOFT] □ Error 120698`
	`N050 G0 X0 Y0`
	`N060 G260`
	`N070 M30`
	
	Solution:
	`N010 #CONTOUR MODE [DEV PATH_DEV=0.2]`
	`N020 G261`
	`N030 G0 X100 Y100 G260`
	`N035 N040 #CONTOUR MODE [DIST_SOFT]`
	`N050 G0 X0 Y0 G261`
	`N060 G260`
	`N070 M30`		

Response	Class	2	Program abort.
Solution	Class	6	Correct the NC program
Parameter	%1:		
		Current mode of polynomial contouring	
	%2:		
		New commanded mode for polynomial contouring	
	%3:		
		Current state of polynomial contouring	
	%4:		
		Block number of affected NC block	
Error type	1, Error message from NC program.		

ID 120700

Dynamic calculation: Transition velocity smaller than programmed E feed rate.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [1µm/s or 0.001°/s]	
	%2:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120701

Maximum lift height POS_LIMIT is smaller than negative software limit switch.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120702 - 120704

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120705

Lifting axis is a transformation axis. Lift movement therefore not possible.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120706

Start of lift movement not possible since a synchronisation movement is already active.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120707

Start of synchronisation movement not possible since a lift movement is already active.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120708

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120709

Position change due to position request during lift movement not allowed.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120710

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120711

Lift and synchronisation movements with active HSC slope not possible.			
Description	When a lift movement is programmed with the “Advanced lifting” function, it is not permitted to use the HSC slope.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Change the slope type, e.g. #SLOPE [TYPE=STEP]
Error type	1, Error message from NC program.		

ID 120712

G61 was used to program too many blocks without motion between two motion blocks.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120713 / 120714

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 120715

STROKE sequence is too long.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 120716

Function not allowed within STROKE definition.			
Description			
Response	Class	2	

Solution	Class	3	
Error type	1, Error message from NC program.		

ID 120717

STROKE sequence moves axis in active plain.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 120718

Invalid measurement run over profile rounding.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120719

Error in Cartesian transformation.			
Description			
Response	Class	2	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 120720

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120721

Distance in plane is zero.			
Description			
Response	Class	2	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 120722 - 120728

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120729

Tangent initialisation of following geometric block not possible.			
Description			
Response	Class	2	
Solution	Class	1	
Parameter	%1:	NC block number [-]	
	%2:	Current value [-]	
	%3:	Upper limit value [-]	
	%4:	Class [-]	
Error type	1, Error message from NC program.		

ID 120730 / 120731

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120732

Error in spline generator. Calculation of spline curve not possible.			
Description	An error occurred during the calculation of the spline curve. The exact cause of the error can be found in the previous error message.		
Response	Class	2	Program execution stop.
Solution	Class	6	Correct NC program
Error type	-		

ID 120733

Feed axis transformation only with main axis and selected additional axis possible, group was adapted.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
Error type	1, Error message from NC program.		

ID 120734

Axis of the third control area direction (EXCUR) may not be used for the definition of the base plane.			

Description	A workspace or protection area in the workspace monitoring function (see [FCT-C14// Defining workspace and protection areas]) is defined by a base plane (polygon or circle) and an excursion EXCUR in the 3rd dimension. The axis defining the third excursion direction of the control area may not be part of the base plane definition:
	
	Example with error:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360 EXCUR_AX=X]`
	`G01 F1000 G90`
	`X0 Y0 (error 120734)`
	`X0 Y100`
	`X100 Y100`
	`X100 Y0`
	`X0 Y0`
	`#CONTROL AREA END`
	
	Corrected example:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360 EXCUR_AX=Z]`
	`G01 F1000 G90`
	`X0 Y0`
	`X0 Y100`
	`X100 Y100`
	`X100 Y0`
	`X0 Y0`
	`#CONTROL AREA END`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Current value [-]	
		ID of affected control area (area ID)	
	%2:	Current value [-]	
		Axes of the base plane	
	%3:	Error value [-]	
		3rd dimension axis	
Error type	1, Error message from NC program.		

ID 120735

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120736

The definition of the control area base plane requires the specification of two axes.

Description	Normally, the control areas of the workspace monitoring function (see [FCT-C14// Description]) refer to the first three Cartesian main axes. If a workspace or protection area is defined with tracking axes, both axes must be specified to define the base plane.
	See [PROG// Definition of a control area] and
	
	Example with error:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X2 = 0 (error 120736)`
	`X2 = 0 Y2=100`
	`X2 = 100 Y2=100`
	`X2 = 100 Y2=0`
	`X2 = 0 Y2=0`
	`#CONTROL AREA END`
	
	Corrected example:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X2 = 0 Y2=0`
	`X2 = 0 Y2=100`
	`X2 = 100 Y2=100`
	`X2 = 100 Y2=0`
	`X2 = 0 Y2=0`
	`#CONTROL AREA END`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program
Parameter	%1:	Current value [-]	
		ID of affected workspace or protection space	
	%2:	Error value [-]	
		Given axes in the area definition	
Error type	1, Error message from NC program.		

ID 120737 - 120739

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120740

Not all axes required for workspace monitoring are available. No monitoring possible.

Description	The relevant axes of all active workspace and protective areas must be available in the NC channel when workspace monitoring is used (see FCT-C14). Otherwise, control area monitoring is not possible. This error message is generated with a path motion.
	See also [PROG// Section: Definition of a control area]
	
	Example:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X = 0 Y=0`
	`X = 0 Y=100`
	`X = 100 Y=100`
	`X = 100 Y=0`
	`X = 0 Y=0`
	`#CONTROL AREA END`
	
	`#PUT AX [Y]`
	`…`
	`#CONTROL AREA ON [ID1]`
	`G0 X100 (Error 120740)`
	
	Correct:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X = 0 Y=0`
	`X = 0 Y=100`
	`X = 100 Y=100`
	`X = 100 Y=0`
	`X = 0 Y=0`
	`#CONTROL AREA END`
	
	`#PUT AX [Y]`
	`…`
	`#CONTROL AREA ON [ID1]`
	`Not all axes required for workspace monitoring are available. No monitoring possible.`
	`G0 X100`		

Response	Class	2	Output error message and abort NC program
Solution	Class	6	Correct the NC program, request all relevant axes first
Parameter	%1:	Current value [-]	
		ID of affected workspace or protection space	
	%2:	Incorrect value [-]	
		Available axes in the NC channel	
	%3:	Expected value [-]	
		Required axis for workspace monitoring	
	%4:	Current value [-]	
		TRUE = Cartesian transformation #(A)CS enabled	
Error type	1, Error message from NC program.		

ID 120741

Not all three axes of the control area are present. No monitoring possible.

Description	In the definition of a workspace monitoring function (see FCT-C14) not all axes in the NC channel which are used for the definition of a workspace or protection area are known.
	See also [PROG// Section: Definition of a control area]
	
	Example:
	`#PUT AX [Z]`
	`…`
	`G17 (-> EXCUR in direction of Z axis, error 120741)`
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G0 G90`
	`X = 0 Y=100`
	`X = 0 Y=100`
	`X = 100 Y=100`
	`X = 100 Y=0`
	`X = 0 Y=0`
	`#CONTROL AREA END`
	
	Correct:
	`#PUT AX [Z]`
	`…`
	`#CALL AX [Z,3,2]`
	`G17 (-> EXCUR in direction of Z axis)`
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G0 G90`
	`X = 0 Y=100`
	`X = 0 Y=100`
	`X = 100 Y=100`
	`X = 100 Y=0`
	`X = 0 Y=0`
	`#CONTROL AREA END`		

Response	Class	2	Output error message and abort NC program
Solution	Class	6	Correct the NC program, request relevant axes first
Parameter	%1:	Current value [-]	
		ID of affected workspace or protection area	
	%2:	Current value [-]	
		Required axes	
	%3:	Incorrect value [-]	
		Index of the missing axis	
Error type	1, Error message from NC program.		

ID 120742

Independent axis movement not permitted for axes belonging to an active work or protection area.

Description	Axes belonging to an active workspace or protection area may not be moved by an independent axis movement (see FCT-C14). This also includes lift movements.
	For further information, see:
	• For programming: [PROG// Section: Definition of a control area]
	• For lift movements: [FCT-A11//Section: Overview]
	
	Example:
	`#CONTROL AREA BEGIN [ID1 PROT POLY MIN_EXCUR=0 MAX_EXCUR=360]`
	`G01 F1000 G90`
	`X = 0 Y = 0`
	`X = 0 Y = 100`
	`X = 100 Y = 100`
	`X = 100 Y = 0`
	`X = 0 Y = 0`
	`#CONTROL AREA END`
	`#CONTROL ARA ON [ID1]`
	`Independent axis movement not permitted for axes of an active work or protection area.`		

Response	Class	2	Output error message and abort NC program
Solution	Class	6	Correct the NC program
Parameter	%1:	Current value [-]	
		ID of affected workspace or protection space	
	%2:	Current value [-]	
		Relevant axes for control area	
	%3:	Incorrect value [-]	
		Currently axis moving independently	
Error type	1, Error message from NC program.		

ID 120743

Axis for definition of cylindrical work or protection area not permitted.			

Description	When defining cylindrical workspace or protection area, only main axes may be used to specify the base plane (see [FCT-C14// Cylindrical control areas]).
	See also [PROG// Definition of a control area]
	
	Example with error:
	`G17 G0 G90`
	`#CONTROL AREA BEGIN [ID1 PROT CIRC MIN_EXCUR=0 MAX_EXCUR=360]`
	`G0 X0 X2=0 (error 120743)`
	`G02 X0 X2=0 I+100`
	`#CONTROL AREA END`
	
	Corrected example:
	`G17 G0 G90`
	`#CONTROL AREA BEGIN [ID1 PROT CIRC MIN_EXCUR=0 MAX_EXCUR=360]`
	`G0 X0 Y0`
	`G02 X0 Y0 I+100`
	`#CONTROL AREA END`		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the area definition in the NC program.

Parameter	%1:	Identification number [-]
		ID of affected workspace or protection space
	%2:	Error value [-]
		Given axes
	%3:	Expected value [-]
		Permitted axes for definition of a cylindrical control area
Error type	1, Error message from NC program.	

ID 120745

Normal vector is zero.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120746

Radius factor of virtual circle is negative.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120747

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120748

Number of active MCS / IMCS areas exceeds maximum value.			
Description	The number of MCS / IMCS active at the same time was exceeded when workspace monitoring was active.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify NC program.
Parameter	%1:	Current value [-]	
		Number of active MCS / IMCS	
	%2:	Limit value [-]	
		Maximum number of MCS / IMCS that may be active at the same time	

Error type	1, Error message from NC program.

ID 120749-120750

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

2.12.4 ID-range 120750-120999

ID 120750

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120751

Too many non-relevant blocks, no transition curve can be calculated.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120752 - 120755

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120756

System error. Please contact control supplier.			
Description			
Response	Class	2	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 120757

Error in dynamic DLL occurred.			
Description			
Response	Class	2	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 120758

Error in dwell time calculation for SYNC IN.	
Description	

Response	Class	2	
Solution	Class	7	
Error type	1, Error message from NC program.		

ID 120759

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120760

Surface tolerance is programmed to zero, value is raised to minimum.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120761

Inconsistency in circular block.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
	%2:	Current value [0.1 10^-3 mm or ø]	
Error type	1, Error message from NC program.		

ID 120762

More than one contouring mode enabled. Please select only one.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 120763

Error in dynamic DLL. Default values are used.			
Description			
Response	Class	1	Warning, no reaction.
Solution	Class	1	

Error type	1, Error message from NC program.

ID 120764

Singularity of transformation in movement.			
Description			
Response	Class	2	Abort NC program processing.
Solution	Class	5	Check and modify the NC programming
Parameter	%1:	NC block number [-]	
	%2:	Row number in file [-]	
Error type			

ID 120765

#SLOPE[TYPE=STEP] not possible with PTP smoothing.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120766

Invalid channel block type in shift sequence.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120767

Detected NC block with axis polynomial and no main axis path. Block was removed.			
Description	Axis polynomials can only be interpolated in connection with motion of other axes in the channel. For this reason, the programmed block is removed and processing is continued with the next block.		
Response	Class	1	Continue NC program processing.
Solution	Class	1	Check and modify the programming.
Error type	1, Error message from NC program..		

ID 120768

Invalid channel block type inside #SHIFT BEGIN/ END.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120769

Missing #SHIFT END in the shift sequence before end of program.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120770

Invalid #SHIFT ON outside #SHIFT BEGIN/ END sequence.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120771-120776

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 120778-120780

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires restart of controller-

ID 120781

Too many blocks without movement programmed (#CAXTRACK). Calculation interrupted.

Description	At least two motion blocks must be known in order to calculate a dressing motion. Too many non motion-relevant blocks were programmed between the motion blocks. See Tracking the C axis with block search (#CAXTRACK)		
Response	Class	1	Output of warning
Solution	Class	1	Check and modify the NC program; delete non-relevant motion blocks.
Error type	1, Error message from NC program.		

ID 120782

Error in splitting of block. Please contact control supplier.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 120783

#STROKE DEF CAXTRACK: Movements inside active plane not allowed.			
Description	Axis motions within the sequence may only be programmed outside the machining plane. Example of machining plane XY (G17): Motions of the X and Y axes are not allowed.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check the required motion. Check the selected machining plane before #STROKE DEF CAXTRACK.
Error type	1, Error message from NC program.		

ID 120784

#STROKE DEF CAXTRACK: Position of Z axis at start and end of sequence not identical.			
Description	An automatic stroke movement must end at the same position at which the stroke movement started. See Splitting up motion path and programming (#STROKE DEF, #PUNCH ON/OFF, #NIBBLE ON/OFF)		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the Z positions at the start and end of the sequence in the NC program.
Error type	1, Error message from NC program.		

ID 120785

#STROKE DEF CAXTRACK: Missing #STROKE DEF CAXTRACK ALIGN BLOCK.			
Description	The position for a required dressing motion was not found in the sequence.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC programming

Error type	1, Error message from NC program.		

ID 120786

#STROKE DEF CAXTRACK: Programmed NC block in sequence not allowed.			
Description	The scope of possible NC commands is restricted in the sequence definition. The command used is not supported.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program. Remove the invalid NC command.
Error type	1, Error message from NC program.		

ID 120787

#STROKE DEF CAXTRACK: Programmed sequence too long.			
Description	Only a limited number of NC commands is allowed between the start and end of the sequence definition. The permitted number of NC commands was exceeded.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program; shorten the sequence.
Error type	1, Error message from NC program.		

ID 120788

#STROKE DEF CAXTRACK: Polynomial contouring in sequence not allowed.			
Description	Polynomial contouring is not allowed within a stroke sequence.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; remove the polynomial contouring in the stroke sequence.
Error type	1, Error message from NC program.		

ID 120789

#CAXTRACK: NC command not allowed with optimised C axis tracking.			
Description	The scope of possible NC commands is limited while Automatic axis tracking (C axis tracking) (#CAXTRACK) is active and optimised mode is used. The command used is not supported.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the NC program; remove the invalid command.
Error type	1, Error message from NC program.		

ID 120790

Invalid control info.	
Description	

Response	Class	2	
Solution	Class	6	
Parameter	%1:	Error value [-]	
Error type	1, Error message from NC program.		

ID 120791

Non convex geometry not allowed for extension.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
Error type	1, Error message from NC program.		

ID 120792

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class		
Solution	Class	8	Requires restart of controller.

ID 120793

Geometry element in invalid zone.			
Description	The face transformation #FACE[..] is active. A kinematic parameter for the Y offset of the tool can be set using this transformation. If the kinematic parameter HD4 is assigned a value unequal to 0, the result may be that not all PCS positions can be programmed or approached within the radius about the centre of rotation of the C axis. These positions are in an invalid range. In this case, a geometry element was programmed in an invalid range.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program. Only program positions within the valid range.
Parameter	%1	Current value [0.1 10^-3 mm or ø]	
		Distance to the invalid range	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current radius in relation to centre of rotation	
	%3:	Current value [0.1 10^-3 mm or ø]	
		Y offset, kinematic parameter HD4	
Error type	1, Error message from NC program.		

ID 120794 / 120795

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 120796

It is not recommended to use #HSC[SURFACE] and #SEGMENTATION[CIR] at the same time.			
Description	In the NC program, #HSC[SURFACE] was used in combination with #SEGMENTATION[CIR]. Recommendation: Use #HSC[SURFACE CIR_MODE=1]		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and modify the NC program.
Error type	-		

ID 120797

Compliance with the axis tolerance could not be ensured. Reduce cycle time if possible.			
Description	The tolerance monitor was unable to find a correct adaptation for the required dynamic. The actual deviation may therefore deviate from the required tolerance! Possible cause: the resolution is too large due to the cycle time.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Reduce the cycle time.
Error type	1, Error message from NC program.		

ID 120798

The interior angle in the linear corner segment of the orbit is smaller than 60 degrees.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120799

Using the combination of ANGPOS and START_STROKE with #CAXTRACK not allowed.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	1, Error message from NC program.		

ID 120800

Illegal control bit field of TRC.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 120801

Initialisation of the kinematic of the TCP velocity limit failed.			
Description	The parameters of the kinematics for the velocity limits are incorrectly parameterised.		
Response	Class	4	Movement stop.
Solution	Class	6	Check, correct and update the kinematic parameters.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 120804

Illegal control bit field of TRC.			
Description			
Response	Class	2	
Solution	Class	3	
Error type	1, Error message from NC program.		

ID 120805

Weighted feedhold acceleration is greater than maximum acceleration.		
Description		
Response	Class	2
Solution	Class	3
Parameter	%1:	Logical axis number [-]
	%2:	Current value [mm/s² or °/s²]
	%3:	Upper limit value [mm/s² or °/s²]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 120806

Weighted feedhold acceleration is smaller than minimum value.			
Description			
Response	Class	2	

Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [mm/s² or °/s²]	
	%3:	Lower limit value [mm/s² or °/s²]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 120807-120809

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 120810

Too many movement blocks in #SPLINE sequence for REDUCED_BSPLINE programmed.			

Description	If the reduced B spline type is used, only a limited of motion blocks may be programmed.
	For example,
	`N010 #SPLINE TYPE REDBSPLINE`
	`N020 #SPLINE ON`
	`N030 Y0.018078 Z-0.0127`
	`N040 Y0.057658 Z-0.009547`
	`N050 Y0.0767 Z-0.0029934`
	`N060 Y0.0767 Z0`
	`N065 Y0.0765 Z0.002 (delete)`
	`N070 #SPLINE OFF`
	The number of motion blocks was exceeded.		

Response	Class	6	Abort NC program processing.
Solution	Class	3	Check NC program and the number of motion blocks
Parameter	%1:	Current value [-]	
		Number of motion blocks	
	%2:	Upper limit value [-]	
		Maximum number of motion blocks	
Error type	1, Error message from NC program.		

ID 120811

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 120812

Monitoring levels in activated areas has to be identical.			

Description	Starting position: several areas are active.
	The activated workspace and protection areas do not have identical monitoring planes.
	Example with error:
	`%area`
	`%Lsub_area_def_imcs`
	`#CONTROL AREA BEGIN [ID=1 WORK MONITOR_LVL = "IMCS" POLY MIN_EXCUR=0MAX_EXCUR=100]`
	`N20 G01F1000G90X50Y50`
	`N30 X-50`
	`N50 Y-50`
	`N60 X50`
	`N70 X50 Y50`
	`N80 #CONTROL AREA END`
	`M29`
	
	`%Lsub_area_def_mcs`
	`#CONTROL AREA BEGIN [ID=2 PROT MONITOR_LVL = "MCS" POLY MIN_EXCUR=0MAX_EXCUR=100]`
	`N20 G01F1000G90X50Y50`
	`N30 X55`
	`N50 Y55`
	`N60 X50`
	`N70 X50 Y50`
	`N80 #CONTROL AREA END`
	`M29`
	
	`%main`
	`LL sub_area_def_imcs`
	`LL sub_area_def_mcs`
	`#CONTROL AREA ON ALL`
	`; …`
	`M30`
	
	The two lines must be corrected, either both must contain
	`MONITOR_LVL = "IMCS" or MONITOR_LVL = "MCS"`
	.		

Response	Class	2	Abort NC program processing.
Solution	Class	6	Check and modify the NC program. Only areas with identical monitoring level may be activated simultaneously.
Parameter	%1:	Identification number [-]	
		Identification number of the control area	
	%2:	Expected value [-]	
		Active monitoring level	
	%3:	Current value [-]	
		Monitoring level of the areas	
Error type	1, Error message from NC program.		

ID 120813

Derivation calculation error, kinematic/Cartesian transformation not possible.			
Description	An error occurred in calculating the derivation of the kinematic or Cartesian transformation. Please refer to the previous transformation errors.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the target points.

Parameter	%1:	Block number [-]
		Block number where which the calculation error occurred.
	%2:	Current value [-]
		State of coordinate system. Value 1: active, value 0: inactive.
	%3:	Current value [-]
		Number of axes in the coordinate system.
	%4:	Current value [-]
		Address of the transformation matrix. (Internal data item)
	%5:	Current value [-]
		Degree of polynomial used
Error type	1, Error message from NC program.	

ID 120814

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 120815

Deviation between the defined and actual pre-output position of the M/H function exceeds the permissible tolerance.			
Description	Due to a limitation in the microjoint function, an M/H function with a pre-output cannot be output at the required position. The path and the number of blocks can be limitations. The number of blocks can be affected by P-CHAN-00603. With contour-changing functions, e.g. contouring, the number of blocks is increased compared to the programmed blocks. The specified tolerance value (P-CHAN-00760) was exceeded for such a deviation from the required position.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Increase tolerance (P-CHAN-00760) if possible. Otherwise, check what limitation obstructs the planned pre-output of the M/H function,.or increase P-CHAN-00603.

Parameter	%1:	Expected value [0.1 µm]
		Specified tolerance value for permitted deviation.
	%2:	Current value [0.1 µm]
		Current actual deviation.
Error type	1, Error message from NC program.	

ID 120818 / 120819

System error [▶ 12]	
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.

Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 120820

Path deviation (Cartesian) too large after conversion to ACS.			
Description	When a block is converted into an ACS block, an exceeded parameterised path deviation (Cartesian) was detected.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Increase the permissible path deviation (Cartesian).
Parameter	%1:	Current value [0.1 10^-3 mm or ø]	
		Programmed path deviation	
	%2:	Upper limit value [0.1 10^-3 mm or ø]	
		Maximum permissible path deviation	
Error type	1, Error message from NC program.		

ID 120821

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	Abort NC program processing.
Solution	Class	8	Requires controller restart.

ID 120822

Torque of the robot axis is invalid or missing.			
Description	When the load model (FCT-C48) is used, the maximum torque must be specified for the axis using P-AXIS-00798 for all robot axes.		
Response	Class	2	Abort NC program processing.
Solution	Class	6	Check the axis torques specified P-AXIS-00798
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected	
	%2:	Error value [Nm]	
		Incorrect axis torque	
Error type	1, Error message from NC program.		

ID 120823

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 120824

Path deviation (orientation) too large after conversion to ACS.			
Description	When a block is converted into an ACS block, an exceeded parameterised path deviation (orientation) was detected.		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Increase the permissible path deviation (orientation).
Parameter	%1:	Current value [mm or degrees]	
		Programmed path deviation	
	%2:	Upper limit value [mm or degrees]	
		Maximum permissible path deviation	
Error type	1, Error message from NC program.		

ID 120825 / 120826

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 120831

Warning: HSC[SURFACE] MAX_ANGLE too big, value corrected.			
Description	When the NC command #HSC[SURFACE …] is used, the keyword MAX_ANGLE can be used to define the maximum knee angle for transitions between two linear blocks. The specified angle is too large and is corrected.		
Response	Class	1	Warning is output and the parameter is corrected.
Solution	Class	1	Check and modify the NC program; correct the angle value.
Parameter	%1:	Error value [0.1 10^-3 mm or ø]	
		Specified angle value	
	%2:	Corrected value [0.1 10^-3 mm or ø]	
		Corrected angle value	
Error type	1, Error message from NC program.		

2.13 Communication error (ID-range 130000-139999)

2.13.1 ID-range 130000-130249

ID 130000 - 130006

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 130007

Internal participant engaged maximum number of buffer locations.			
Description	While sending a PDU, the system participant cannot get any PDU-buffer any more.		
Response	Class	3	Warning output.
Solution	Class	1	Wait, because later there may be any PDU-buffer again, and then repeat instruction.
Parameter	%1:	Current value [-]	
		Current number of PDU buffers.	
	%2:	Expected value [-]	
		Maximum number of PDU buffers for this system participant.	
Error type	-		

ID 130008 - 130104

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 130105

Timeout for writing PDU to receiver.			
Description	The receiver doesn't answer and doesn't take the message. Possible error cause is that the PLC is logged onto the HLI for messages but does not access the messages fast enough.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	3, Error message from communication.		

ID 130106

Timeout for number of attempts to send PDU to receiver.			
Description	The receiver does no answer and does not take the message. Possible error cause is that the PLC is logged onto the HLI for messages but does not access the messages fast enough.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	3, Error message from communication.		

ID 130107 - 130119

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 130120

Invalid receiver of PDU message.			
Description	The message cannot be delivered to the receiver.		
Response	Class	3	
Solution	Class	3	Modify the NC program. Check the programmed receiver in the #MSG command and correct the receiver. Only the documented receivers are possible.
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

ID 130121

Message cannot be handled by receiver.			
Description	The programmed receiver does not understand the message received. Message is rejected.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	3, Error message from communication.		

ID 130122 - 130124

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 130125

PDU receiver not announced or unknown.			
Description	The receiver specified in the message is not known to the controller or has not yet been registered.. The message is undeliverable and ignored. Possible error cause is the programming of an unknown receiver by explicit programming of the receiver ID in command #MSG.		
Response	Class	3	
Solution	Class	1	Check the programmed receiver in the #MSG command and correct the receiver. Only the documented receivers are possible.
Parameter	%1:	Current value [-]	
	%2:	Incorrect value [-]	
	%3:	Current value [-]	
	%4:	Current value [-]	
	%5:	Current value [-]	
Error type	3, Error message from communication.		

ID 130126

Undeliverable PDUs are reinstated. PDU dismissed.			
Description	The specified receiver doesn't understand the message. He has sent the message back to the sender, who notices the message undeliverable. Possible error cause is, that e.g. with a #MSG a receiver was explicitly specified, who cannot do anything with this message. The receiver didn't understand the message		
Response	Class	3	
Solution	Class	3	
Error type	3, Error message from communication.		

ID 130127

Participant already announced.	
Description	Every communication device must register once after controller start-up in order to send or receive messages. Double registration is not allowed; the user can only register again after the next controller start-up.

Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	-		

ID 130128 - 130133

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 130134

New SIGNAL/WAIT-event could not be saved.			
Description	Too many #SIGNAL or #WAIT have been used, and have not yet been acknowledged. The available controller memory is full. Depending on the configuration, 50 or 100 open signals and wait are permitted, see @@[SYSP].		
Response	Class	3	
Solution	Class	7	Modify the CNC program and use less synchronization events. Deleting synchronizations that are no longer required .
Parameter	%1:	Current value [-]	
Error type	3, Error message from communication.		

ID 130135 / 130136

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 130137

Unknown PDU code. PDU discarded.			
Description			
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Current value [-]	

Error type	3, Error message from communication.

2.14 Drive management error (ID-range 140000-149999)

2.14.1 ID-range 140000-140249

ID 140000 - 140016

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 140017

Index of field data requested exceeds the limit of field definition.			
Description	The maximum permissible length of an operating data item ident was exceeded on reading the ident.		
Response	Class	1	Warning output.
Solution	Class	1	Check length of the operating data item ident in the drive.
Parameter	%1:	Current value [-]	
		Current length of the IDN.	
	%2:	Limit value [-]	
		Maximum permissible length of an IDN.	
	%3:	Current value [-]	
		Index of the SERCOS axis.	
	%4:	Identification number [-]	
		Number of the ident to be read.	
Error type	-		

ID 140018

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 140021

Start of procedure command not possible.			
Description	An error occurred during the execution of a command.		
Response	Class	1	Warning output.
Solution	Class	1	Read error diagnosis of drive (ident 95).

Parameter	%1:	Status [-]
		Logical axis number (P-AXIS-00016).
	%2:	Logical axis number [-]
		Number of the incorrect command.
	%3:	Identification number [-]
Error type	-	

ID 140023

The service channel is already busy with another command.			
Description	When executing a SERCOS command, the system detected that the service channel is already assigned with another command.		
Response	Class	1	Warning output.
Solution	Class	1	Abort old command before starting new command.
Error type	-		

ID 140024

A command which is requested to be erased is already erased.			
Description	A SERCOS command that is to be erased is already erased.		
Response	Class	1	Warning output
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
	%2:	Identification number [-]	
Error type	-		

ID 140025

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 140026

No device with this logical number available.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 140027

Abort of procedure command not possible, because there is no command active.			
Description	No SERCOS procedure commands can be aborted because there are no active commands present.		
Response	Class	1	Warning output.
Solution	Class	1	No further action necessary.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016).	
Error type	-		

ID 140028

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 140029

An error occurred while switching to phase 0.			
Description	An error occurred on switching in Phase 0.		
Response	Class	1	Warning output.
Solution	Class	1	Check SERCOS ring: Is ring closed? Power supply for all drives switched on?
Parameter	%1:		
		Target phase (0).	
Error type	-		

ID 140030

An error occurred while switching to phase 1.			
Description	The SERCOS ring could not be switched to phase 1.		
Response	Class	1	Warning output.
Solution	Class	1	
Parameter	%1:	Target phase (1).	
Error type	-		

ID 140031

An error occurred while switching to phase 2.			
Description	The SERCOS ring could not be switched to phase 2.		
Response	Class	1	Warning output.
Solution	Class	1	

Parameter	%1:	Target phase (2).
Error type	-	

ID 140032

Error while reading ident.			
Description	There was an error while trying to read a SERCOS ident from a drive.		
Response	Class	1	Warning output.
Solution	Class	1	Check if the ident that was tied to read is present in the drive.
Parameter	%1:		
		Logical axis number (P-AXIS-00016).	
	%2:		
		Number of the identity to be read.	
Error type	-		

ID 140033

Error while reading slave arrangement.			
Description	SEROS- ident 96 (SLKN) could not be read from the drive.		
Response	Class	1	Warning output.
Solution	Class	1	Check if SERCOS ident 96 is supported by the drive.
Parameter	%1:		
		Logical axis number (P-AXIS-00016).	
Error type	-		

ID 140034

Error while time slot calculation.			
Description	An error occurred during the time slot calculation for the cyclic SERCOS telegram.		
Response	Class	1	Warning output.
Solution	Class	1	Check time slot parameter of the drive.
Parameter	%1:		
		Logical axis number (P-AXIS-00016).	
	%2:		
		Erroneous time slot value.	
	%3:		
		Internal error code.	
Error type	-		

ID 140035

Error while writing an ident.

Description	An error occurred while writing a SERCOS ident. Possible reasons are: • Could not find ID • ID is write-protected by password (see ID Ident is write protected by a password (see Ident 279, list of password protected data) • Ident can only be written in phase 2.		
Response	Class	1	Warning output.
Solution	Class	1	Cancel password protection (Ident 267) or switch to phase 2 before writing the ident.
Parameter	%1:		
		Logical axis number (P-AXIS-00016).	
	%2:		
		Ident to be written.	
Error type	-		

ID 140036

Error while writing a drive telegram.			
Description	An error occurred while writing the drive telegram in the drive. Possible reasons are: • An error occurred during the transmission of the idents. • An invalid value for ident 15 (telegram type) was written. • A value which cannot be transferred cyclically is entered in the configuration list of the drive telegram (ID 16) (see ID 187, list of configurable data in drive telegram). • A value which cannot be transferred cyclically is entered in the configuration list of the master data telegram (ID 24) (see ID 188, list of configurable data in MDT).		
Response	Class	1	Warning output
Solution	Class	1	Correction of the invalid values
Parameter	%1:		
		Logical axis number (P-AXIS-00016).	
	%2:		
		Ident when the error occurred.	
Error type	-		

ID 140037

Error while writing MDT start address (IDN 9).			
Description	An error occurred while writing ident 9.		
Response	Class	1	Warning output.
Solution	Class	1	
Parameter	%1:		
		Logical axis number (P-AXIS-00016).	
Error type	-		

ID 140038

Error in transition check for communication phase 3 (command 127).			
Description	The SERCOS command 127 (switch-over preparation in phase 3) returns an error. This means that an invalid value was found when checking the parameters valid in phase 3. A list with the idents of the invalid parameters can be read in ident 21 (list of invalid idents in phase 3).		
Response	Class	1	Warning output
Solution	Class	1	Read ident 21 (list of invalid idents in phase 3) and correct invalid values.
Parameter	%1:		
	Logical axis number (P-AXIS-00016).		
	%2:		
	Faulty command (Ident 127).		
	%3:		
	Return code 1 (internal error code)		
	%4:		
	Return code 2 (service channel error code)		
	%5:		
	Data status of the procedure command		
Error type	-		

ID 140039

An error occurred while switching to phase 3.			
Description	The SERCOS ring could not be switched to phase 3.		
Response	Class	1	Warning output.
Solution	Class	1	Check the parameters valid in phase 3 with the command Preparation for changeover in phase 3 (Ident 127) and, if necessary, correct the faulty parameters.
Parameter	%1:		
	Target phase (3).		
Error type	-		

ID 140040

Error when preparing for transition in phase 4 (command 128).			
Description	The SERCOS command 128 (switch-over preparation in phase 4) returns an error. This means that an invalid value was found when checking the parameters valid in phase 4. A list with the idents of the invalid parameters can be read in ident 22 (list of invalid idents in phase 4).		
Response	Class	1	Warning output
Solution	Class	1	Read ident 22 (list of invalid idents in phase 4) and correct invalid values.

Parameter	%1:	
		Logical axis number (P-AXIS-00016).
	%2:	
		Faulty command (Ident 128).
	%3:	
		Return code 1 (internal error code)
	%4:	
		Return code 2 (service channel error code)
	%5:	
		Data status of the procedure command
Error type	-	

ID 140041

An error occurred while switching to phase 4.			
Description	The SERCOS ring could not be switched to phase 4.		
Response	Class	1	Warning output.
Solution	Class	1	Check the parameters valid in phase 4 with the command Preparation for changeover in phase 4 (Ident 128) and, if necessary, correct the faulty parameters.
Parameter	%1:	Target phase (3).	
Error type	-		

ID 140042

PDU with unknown logical axis number received.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
Error type	3, Error message from communication.		

ID 140043

PDU with invalid logical axis number received.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
Error type	3, Error message from communication.		

ID 140047

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 140048

At the moment, it is only possible to switch to phase 2. Instruction was corrected.			
Description	Since the service channel is still blocked by an action that was commanded before, a communication phase switch can only be done to communication phase 2. Therefore the phase switch is stopped in communication phase 2.		
Response	Class	1	Warning output.
Solution	Class	1	Abort the previous service channel command in communication phase 2 and then start phase switch again.
Parameter	%1:	Current value [-]	
		Internal code for the commanded communication phase.	
	%2:	Corrected value [-]	
		Internal code for communication phase 2.	
Error type	3, Error message from communication.		

ID 140050

SERCOS I/O channel type is not supported yet.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Identification number [-]	
	%2:	Current value [-]	
Error type	-		

ID 140051

Note: Manufacturer specific SERCOS-I/O channel type.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Identification number [-]	
	%2:	Current value [-]	
Error type	-		

ID 140052

Transferred address of a SERCOS command (via PDU) is unknown.			
Description	The SERCOS service channel command cannot be executed because the content of the command cannot be read.		
Response	Class	1	Warning output.
Solution	Class	1	

Parameter	%1:	Current value [-]
		Memory address to the content of the SERCOS command.
Error type	-	

ID 140053

Multiple use of same SERCOS drive address (configuration error).			
Description	In the configuration data, two SERCOS axes are configured with the same drive address.		
Response	Class	3	Error message output, but nevertheless the operation is continued.
Solution	Class	6	Check drive address in the configuration data.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of first axis P-AXIS-00016).	
	%2:	Current value [-]	
		Drive address of first axis (P-AXIS-00019).	
	%3:	Logical axis number [-]	
		Logical axis number of second axis P-AXIS-00016).	
	%4:	Current value [-]	
		Drive address of second axis (P-AXIS-00019).	
Error type	-		

ID 140054

Timeout: Not terminated procedure command is aborted.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Status [-]	
	%2:	Logical axis number [-]	
	%3:	Identification number [-]	
	%4:	Current value [-]	
	%5:	Upper limit value [-]	
Error type	-		

ID 140055

Invalid SERCOS service channel.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
	%3:	Logical axis number [-]	
	%4:	Current value [-]	
	%5:	Current value [-]	

Error type	-

ID 140056

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 140057

No callback function registered to handle Sercos-over-EtherCAT service channel write request.			
Description	To use the SoE service channel together with the CNC SDK, a corresponding callback function must be stored to process a write request. This error occurs with a service channel write request from the CNC if no corresponding callback function was stored.		
Response	Class	3	Abort current job.
Solution	Class	6	To eliminate the error message, a corresponding callback method for processing a write request must be registered. Further information on this is contained in the API documentation of the CNC SDK
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected.	
	%2:	Identification number [-]	
		Identification number of read request.	
	%3:	Logical channel number [-]	
		Channel number of channel affected.	
Error type	Fatal error by NC program or operator action.		

ID 140058

No callback function registered for handling a SERCOS-over-EtherCAT service channel read request.			
Description	To use the SoE service channel together with the CNC SDK, a corresponding callback function must be stored to process a read request. This error occurs with a service channel read request from the CNC if no corresponding callback function was stored.		
Response	Class	3	Abort current job.
Solution	Class	6	To eliminate the error message, a corresponding callback method for processing a read request must be registered. Further information on this is contained in the API documentation of the CNC SDK.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected.	
	%2:	Identification number [-]	
		Identification number of read request.	
	%3:	Logical channel number [-]	
		Channel number of channel affected.	
Error type	Fatal error by NC program or operator action.		

ID 140059

An error occurred while processing a SERCOS-over-EtherCAT service channel read request.			
Description	To use the SoE service channel together with the CNC SDK, a corresponding callback function must be stored to process a read request. This error occurs if an error has occurred during the processing of a service channel read request from the CNC within the callback function.		
Response	Class	3	Abort current job. Warning output.
Solution	Class	3	To eliminate the error message, the user must check why an error occurred in the callback function that he registered.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected.	
	%2:	Identification number [-]	
		Identification number of read request.	
	%3:	Error value [-]	
		Return value of the callback function	
	%4:	Logical channel number [-]	
		Channel number of channel affected.	
Error type	-		

ID 140060

An error occurred while processing a SERCOS-over-EtherCAT service channel write request.			
Description	To use the SoE service channel together with the CNC SDK, a corresponding callback function must be stored to process a write request. This error occurs if an error has occurred during the processing of a service channel write request from the CNC within the callback function.		
Response	Class	3	Abort current job. Warning output.
Solution	Class	1	To eliminate the error message, the user must check why an error occurred in the callback function that he registered.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected.	
	%2:	Identification number [-]	
		Identification number of read request.	
	%3:	Error value [-]	
		Return value of the callback function	
	%4:	Logical channel number [-]	
		Channel number of channel affected.	
Error type	-		

2.15 Manual mode error (ID-range 150000-159999)

2.15.1 ID-range 150000-150249

ID 150000

List interpreter can not evaluate given list.			
Description	The list with the configuration data for manual operation mode cannot be interpreted because a directory path or file name was specified in parameter P-STUP-00013 of the start-up list which does not contain a corresponding file.		
Response	Class	3	Abort current job.
Solution	Class	1	Specify the correct directory path for parameter P-STUP-00013.
Error type	2, Error message by data transfer from parameter list into control device.		

ID 150001

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150002

Can not write to FIFO of manual mode.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 150003

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150004

Resolution of handwheel is 0.

Description	The CNC uses the formula P-AXIS-00025 **Resolution of handwheel is 0.** P-MANU-00006 / P-MANU-00005 the motion path of the axis is calculated per handwheel increment in the unit 0.1µm/increment. The result of the calculation is 0.		
Response	Class	1	
Solution	Class	1	Check the values for the parameters of the formula: P-AXIS-00025 – see axis parameter P-MANU-00006 – see manual mode parameter
Parameter	%1:	Logical axis number [-]	
		Number of logical axis whose parameters lead to this message.	
Error type	-		

ID 150005

Number of handwheel increments per cycle exceeds range.			
Description	Parameter P-MANU-00007 defines the maximum number of increments delivered by the handwheel per CNC cycle. This assists the plausibility check of the handwheel hardware. The program detected that more handwheel increments were supplied than defined by the parameter P-MANU-00007.		
Response	Class	1	The number of increments in the current cycle is limited to the value of parameter P-MANU-00007.
Solution	Class	1	Check value for parameter P-MANU-00007 Check hardware.
Parameter	%1:	Logical axis number [-]	
		Number of logical axis which is commanded by the handwheel.	
	%2:	Current value [-]	
		Number of increments that occurred during on CNC cycle.	
	%3:	Limit value [-]	
		Maximum permissible number of handwheel increments defined by parameter P-MANU-00007.	
	%4:	Current value [-]	
		Counter handwheel input interface	
	%5:	Current value [-]	
		Counter handwheel working data	
Error type	-		

ID 150006

Continuous jog velocity is 0.			
Description	An axis was activated in manual mode so that it is to be moved in continuous jog mode. The velocity at which the jog is run in continuous jog mode was either taken from parameter P-AXIS-00077 of the axis parameter data record or specified by another application. The message occurs because the continuous jog velocity is 0.		
Response	Class	1	

Solution	Class	1	Check the values for parameters P-AXIS-00077 in the axis parameter data record. Check velocity given by external application.
Parameter	%1:	Logical axis number [-]	
		Number of logical axis which should be moved in continuous jog mode.	
Error type	-		

ID 150007

Incremental jog width or velocity is 0.			
Description	The CNC is in manual mode and an axis is to be moved in incremental jog mode. For this purpose, the path distance and the motion velocity are assigned to the axis. These are either specified from the axis parameter data record using P-AXIS-00232, P-AXIS-00076 or commissioned by an external application. The value for path distance or path velocity was commanded the value 0.		
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Logical axis number [-]	
		Number of logical axis which was incorrectly commanded.	
Error type	-		

ID 150008

Offset limit reached in jog mode.			
Description	The CNC is in manual operation mode state. The incremental jog mode is activated for the axis to be moved. The axis reached the motion area restricted by the manual operation offset limits. They are defined by: • the parameters P-AXIS-00137, P-AXIS-00138 of the axis parameter data record • or by programming #MANUAL LIMITS[...] or #SET OFFSET[...] in the NC program For more information on manual mode offset limits: [FCT-A4// Section: Description]		
Response	Class	1	Output of warning, stop the axis at the corresponding offset limit.
Solution	Class	1	Motion is only possible in the opposite direction.
Parameter	%1:	Logical axis number [-]	
		Number of logical axis, which exceeds manual mode offset limit.	
	%2:	Current value [Incr.]	
		Sum of all increments commanded by control elements during manual mode.	
	%3:	Limit value [Incr.]	
		Depending on the manual offset limit value exceeded: P-AXIS-00137 or P-AXIS-00138	
Error type	-		

ID 150009

Programmed jog velocity is too small.			
Description	The velocity commanded for moving an axis in incremental jogging mode results in a value of less than 1 increment/CNC cycle in the CNC-internal conversion to increments/CNC cycle.		
Response	Class	1	CNC calculates inside with feed rate 1 increment/CNC-cycle.
Solution	Class	1	Check the values for P-AXIS-00076 in the axis parameter data set. Check values for feed rate given by external application.
Parameter	%1:	Logical axis number [-]	
		Number of logical axis which was commanded with to less feed rate.	
	%2:	Current value [1µm/s or 0,001°/s]	
		Commanded feed rate for moving axis in incremental jog mode.	
Error type	-		

ID 150010

Unknown axis number.			
Description	The NC channel is in manual mode and a manual operation mode should be activated for an axis. The axis must therefore be assigned to this manual mode type. The axis is assigned to a manual mode type by its logical axis number. CNC detects that there is no axis with such a logical axis number.		
Response	Class	1	
Solution	Class	1	Use a logical axis number that exist within the CNC system.
Parameter	%1:	Logical axis number [-]	
		Number of logical axis which was commanded	
Error type	-		

ID 150011

Unknown hand wheel number.			
Description	A command has been sent to the CNC to assign a handwheel to an axis so that the axis will respond in manual operation mode to the handwheel input. The handwheel is then specified by its logical handwheel number. See [HLI// Section: Activating control elements] For CNC build < 2800 [HLI// Section: Activating control elements] For CNC build > 2800 The message is displayed because none of the handwheels defined in the manual operation data set has the commanded logical handwheel number as value for parameter P-MANU-00002. Message occurs independent if axis still is in manual mode or not.		
Response	Class	3	
Solution	Class	3	Check if correct logical handwheel number was used. Afterwards use another logical handwheel number or adapt the manual mode data set.
Parameter	%1:	Logical control element number [-]	
		Logical handwheel number which was commanded to CNC.	

Error type	-

ID 150012

Unknown inching key number.			
Description	A command has been sent to the CNC to assign a key to an axis so that this axis will respond in manual operation mode to the input from this key. The button is then specified by its logical button number (). The manual mode state machine of the axis is at that moment in state ACTIVE or READY. See also [HLI// Section: Activating control elements] For CNC build < 2800 [HLI// Section: Activating control elements] For CNC build > 2800 The message as displayed because none of the keys defined in the manual operation data set has the instructed logical key number as value for parameter P-MANU-00009.		
Response	Class	3	
Solution	Class	3	Check if correct logical key number was used. Afterwards use another logical key number or adapt the manual mode data set.
Parameter	%1:	Logical control element number [-]	
		Logical key number which was commanded to CNC.	
Error type	-		

ID 150013 - 150023

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150024

Missing axis.			
Description	Manual mode for an axis was deselected with the NC command G202 although no manual mode was activated for this axis.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and modify the NC program and the program flow. Either activate manual for the axis beforehand or remove G202 from the NC program.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
Error type	-		

ID 150025

Missing programming of manual mode parameter.			
Description	Before activation of manual mode for axis by G200 or G201, at least one parameter set (P-AXIS-00025, P-AXIS-00076, P-AXIS-00232) for the handwheel/continuous/ incremental operation modes has to be initialized. Initialization can be done in the parameter list of the manual operation mode, or by explicitly programming #JOG INCR, #JOG CONT, #HANDWHEEL or #SET TIP, #SET JOG , #SET HR in the NC program.		
Response	Class	3	Axis will not be prepared for manual mode
Solution	Class	7	Correct initialization of parameters in axis list or NC program.
Parameter	%1:	Logical axis number [-]	
		For logical axis number of concerned axis, see P-AXIS-00016	
Error type	-		

ID 150026 - 150032

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150033

Error while interpreting the list of manual mode parameters.			
Description	This message occurs if the interpretation of manual mode parameters results in an error. This general message follows a message which describes the cause in detail why the interpretation of manual mode parameters failed.		
Response	Class	3	Manual mode parameters are not transferred to the CNC.
Solution	Class	1	Rectify the cause which led to preceding error message.
Parameter	%1:	Current value [-]	
		Internal state of state machine which represents the interpretation of manual mode parameters.	
Error type	-		

ID 150034 - 150038

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150039

Axis number is 0 in handwheel mode.

Description	The NC channel is in manual mode and an axis is to be moved by handwheel by specifying command values. The axis must then be linked to the corresponding handwheel. The axis is assigned to the handwheel by specifying its logical axis number (P-AXIS-00016). When the CNC accepts the data, it detects that the logical axis number is 0. This value is not permitted for logical axis numbers.		
Response	Class	3	
Solution	Class	6	Indicate the logical axis number parameterised in the axis machine data record.
Parameter	%1:	Logical axis number [-]	
		Incorrect logical axis number 0.	
Error type	-		

ID 150040

Resolution of handwheel is 0.			
Description	CNC is in manual mode and in ENABLED state. Message occurs because the CNC received the command to move a specific axis according to input of a handwheel. When the CNC accepts the data, it detects that distance resolution of 0 µm per handwheel increment is parameterised for the specific axis. This parameter is either taken from the axis parameter data record (P-AXIS-00025) or was commanded by another application.		
Response	Class	1	Warning The default value is not accepted.
Solution	Class	1	Change the values of the parameter P-AXIS-00025 to values unequal to 0. or Change the values specified by an external application to values unequal to 0.
Parameter	%1:	Current value [-]	
		When the distance resolution is used from the axis parameter data record, it is the index of the parameter P-AXIS-00025 with the value 0. If an external application specified the distance resolution, it is the value that was specified.	
Error type	-		

ID 150041

Invalid index of resolution.			
Description	The CNC is in manual mode and in ENABLED state. An axis is now to be prepared so that it reacts to the command input of a handwheel. The distance resolution is specified by the value of the parameter P-AXIS-00025. The parameter to use is specified by its index. When the CNC accepts the data, it detects that there is no value was defined for distance resolution for this index.		
Response	Class	1	Warning
Solution	Class	1	Correct value of index to value that exists within in the axis parameter data record.

Parameter	%1:	Current value [-]
		Value of index that does not exist in the axis parameter data record.
Error type	-	

ID 150042

State transition from ready to enabled not possible.			
Description	The **handwheel** manual mode is to be activated for an axis but the axis is not ready. Each axis has a state machine that manages various phases of the axis in manual mode. The manual mode state machine must be in READY (1) or ENABLED (2) state if you want to assign a manual mode and manual input device to an axis. See also :[HLI// Section: Manual mode], [HLI// Section: Handwheel mode] for CNC builds < 2800] [HLI// Section: Handwheel mode] for CNC builds > 2800] The message occurs because the manual mode state machine of axis is not in a READY (1) or ENABLED (2) state, but an attempt was made to activate the **handwheel**manual mode for the axis.		
Response	Class	1	Warning The manual mode configuration is maintained as it was before this attempt to select the **handwheel** mode for the axis.
Solution	Class	1	Place the manual mode state machine of the axis in READY state. Before assigning a manual mode and a manual input device to the axis, either parameterise the NC program or program a manual block G200 or G201 [PROG// Section: Manual mode]. or change over the operation mode of the NC channel explicitly in the HLI or GUI to Manual mode\Enabled. See [HLI// Section: Parameterising manual mode] for CNC builds < 2800] [HLI// Section: Parameterising manual mode] for CNC builds > 2800]
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016 of the axis for which the **State transition from ready to enabled not possible.**mode is to be enabled.	
Error type	-		

ID 150043

State transition from enabled to ready not possible.			
Description	Each axis has a state machine that manages various phases of the axis in manual mode. The manual mode state machine must be in ENABLED mode [HLI//Section: Manual mode] if the manual mode type is deselected for an axis or if the assigned manual input device is disabled. The message occurs because the manual mode state machine of the axis is not in ENABLED state, an attempt was made to deselect a manual mode type for the axis.		
Response	Class	1	Warning

Solution	Class	1	In principle, check whether the timing to deselect the manual mode type or deactivate the manual input device was correctly selected.. Check whether the deactivation command was sent for the required axis and so check the logical axis number which was also sent in the command. Otherwise, ensure that the axis is in ENABLED state [HLI//Section: Manual mode].
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016 of the axis for which a manual mode type was commanded.	
Error type	-		

ID 150044 - 150046

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150047

Axis number is 0 in continuous jog mode.			
Description	The NC channel is in manual mode and an axis is to move in continuous jog mode. Continuous jog mode must be enabled for the axis and linked to a key. Select the axis by specifying its logical axis number (P-AXIS-00016). When the CNC accepts the data, it detects that the logical axis number is 0. This value is not permitted for logical axis numbers.		
Response	Class	1	Warning
Solution	Class	1	Indicate the logical axis number parameterised in the axis machine data record.
Parameter	%1:	Logical axis number [-]	
		Incorrect logical axis number 0.	
Error type	-		

ID 150048

Invalid selection of operation mode with continuous jog mode.	
Description	The continuous jog mode manual mode is to be activated for an axis but the axis is not ready. Each axis has a state machine that manages various phases of the axis in manual mode. This manual operation mode machine state must be READY (1) or ENABLED (2) ([HLI//Manual mode] if you want to assign a manual mode type and a manual input device to an axis. The message occurs because the manual mode state machine of the axis is not in READY (1) or ENABLED (2) state, but an attempt was made to activate the continuous jog mode manual mode for the axis.

Response	Class	1	Warning The manual mode configuration is maintained as it was before this attempt to select continuous jog mode for the axis.
Solution	Class	1	Place the manual mode state machine of the axis in READY state. Before assigning a manual mode and a manual input device to the axis, • either parameterise the NC program or program a manual block G200 or G201 • or change over the operation mode of the NC channel explicitly in the HLI or GUI to Manual mode\Enabled. [HLI// Section: Operation modes] and [HLI// Section: Manual mode]
Parameter	%1:	Logical axis number [-]	
		P-AXIS-00016 of the axis for which the continuous jog mode is to be activated.	
Error type	-		

ID 150049

Continuous jog mode velocity activated at 0.			
Description	The CNC is in manual mode and in ENABLED state. The message is issued because the CNC was commanded to move a certain logical axis at a certain keystroke in continuous jog mode. The velocity in continuous jog mode is either taken from the axis parameter data record, or specified by an external application. When the CNC accepts the data, it detects that the commanded continuous jog velocity is 0.		
Response	Class	1	Warning The value 0 is not adopted.
Solution	Class	1	Change the continuous jog velocity parameterised in the parameters of P-AXIS-00077. or Change value for continuous jog velocity specified by an external application.
Parameter	%1:	Current value [-]	
		Index of the parameter P-AXIS-00077 which has the value 0 for jog velocity.	
Error type	-		

ID 150050

Continuous jog velocity is greater than maximum velocity.			
Description	The CNC is in manual mode and in ENABLED state. The message is issued because the CNC was commanded to move a certain logical axis at a certain keystroke in continuous jog mode. When the CNC accepts the data, it detects that the commanded continuous jog velocity exceeds the maximum allowed feed rate during manual operation mode.		
Response	Class	1	The CNC replaces the command value by the limit value.

Solution	Class	1	The maximum permissible velocity limit in manual mode depends on whether it is a pure manual mode (G200) or a parallel interpolation of axis motions takes place (G201). Pure manual mode The parameter P-AXIS-00077 of the axis parameter data record, or the value of the commanded continuous jog velocity if commanded by another application, must be changed so that these values are smaller than P-AXIS-00213. Manual mode with parallel interpolation The parameterP-AXIS-00077 of the axis parameter data record or, if commanded by another application, the value for the commanded continuous jog velocity must be changed so that these values are smaller than P-AXIS-00212 **Continuous jog velocity is greater than maximum velocity.** P-AXIS-00083/100. In both cases it is also possible to change the parameters that define the limit value.

Parameter	%1:	Current value [1µm/s or 0.001°/s]
		Commanded velocity for moving in continuous jog mode.
	%2:	Limit value [1µm/s or 0.001°/s]
		Maximal allowed velocity in manual mode.
	%3:	Corrected value [1µm/s or 0.001°/s]
		The commanded value is replaced by this value.
Error type	-	

ID 150051

Invalid index of continuous jog velocity.			
Description	CNC is in manual mode and in ENABLED state. This message is issued because the CNC was commanded to adopt the motion velocity value in continuous jog mode (P-AXIS-00077) from the axis parameter data record. It is possible to parameterise several motion velocities in the axis parameter data record and therefore the value used is selected by a index. When the CNC accepts the data, it detects that no entry exists in the axis parameter data record for the transferred index.		
Response	Class	1	Warning The active motion velocity for continuous jog mode at this time is maintained.
Solution	Class	1	Change the index to command the activation of continuous jog mode or Add an entry to the axis parameter data record for P-AXIS-00077 under the corresponding index.

Parameter	%1:	Current value [-]
		Index of motion velocity for continuous jog mode for which no entry exists in the axis parameter data record.
Error type	-	

ID 150052

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 150053

Logical axis number is 0 in incremental jog mode.			
Description	The NC channel is in manual mode and an axis is to move in incremental jog mode. The axis must therefore be enabled for the incremental jog mode axis and linked to a key. Select the axis by specifying its logical axis number (P-AXIS-00016). When the CNC accepts the data, it detects that the logical axis number is 0. This value is not permitted for logical axis numbers.		
Response	Class	1	Warning
Solution	Class	1	Indicate the logical axis number parameterised in the axis machine data record.
Parameter	%1:	Logical axis number [-]	
		Incorrect logical axis number 0.	
Error type	-		

ID 150054

Unable to select jog mode.			
Description	The incremental jog manual mode is to be activated for an axis, but the axis is not ready. Each axis has a state machine that manages various phases of the axis in manual mode. The manual mode state machine must be in READY (1) or ENABLED (2) mode [HLI//Section: Manual mode] so that a manual mode and a manual input device can be assigned.. The message occurs because the manual mode state machine of the axis is not in READY (1) or ENABLED (2) state, but an attempt was made to activate the incremental jog manual mode for an axis.		
Response	Class	1	Warning The configuration of the manual mode is maintained as it was before the attempt to select the incremental jog mode for the axis.
Solution	Class	1	Place the manual mode state machine of the axis in READY state. Before assigning a manual mode and a manual input device to the axis, either parameterise the NC program or program a manual block G200 or G201. or the operation mode\operation mode of the NC channel must be changed over explicitly in the HLI or GUI to manual mode\Enabled See also [HLI//Section: Operation modes] and [HLI// Section: Manual mode]

Parameter	%1:	Logical axis number [-]
		P-AXIS-00016 of the axis for which the incremental jog mode is to be enabled.
Error type	-	

ID 150055 / 150056

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150057

Distance or velocity is 0 in jog mode.			
Description	An external application requests the CNC to activate the incremental jog manual mode for an axis. The value for jog velocity and jog incremental jog width is checked and then enabled. The CNC detects that either the jog velocity or the jog incremental width or both are 0.		
Response	Class	1	Warning Manual mode parameters are not transferred to the CNC. Therefore the axis cannot be moved by keys.
Solution	Class	1	Change the values of parameters P-AXIS-00076 and P-AXIS-00232 in the axis parameter data record to values unequal to 0 if the parameters were selected during activation by specifying an index parameter. or Change the commanded values for jog velocity and jog incremental width to values unequal to 0 if they were explicitly commanded.
Parameter	%1:	Current value [-]	
		Value >= 0 is an index of the parameter data record of incremental jog mode in the axis parameter data record which was commanded. -1 means that values for jog incremental width and jog velocity are commanded explicitly.	
Error type	-		

ID 150058

Incremental jog velocity greater than maximum velocity.

Description	An external application requests the CNC to activate the incremental job mode for an axis. The value for incremental jog velocity is checked and then enabled. The CNC detects that the currently activated incremental jog velocity is greater than the maximum velocity limit for the axis. The limit value is for manual mode without parallel interpolation (G200) of the value of parameter P-AXIS-00213 of the axis parameter data record. manual mode with parallel interpolation (G201) of the value resulting from P-AXIS-00212 * P-AXIS-00083. Value of P-AXIS-00212 is dependent on the active gear stage.		
Response	Class	1	Warning Limit value for incremental jog velocity is used.
Solution	Class	1	Command value for incremental jog velocity that is less than limit. or adapt the described parameters.
Parameter	%1:	Current value [1µm/s or 0.001°/s]	
		Commanded incremental jog velocity. Takes effect if incremental jog mode is activated.	
	%2:	Limit value [1µm/s or 0.001°/s]	
		Maximum permissible incremental jog velocity calculated from the parameters described above.	
Error type	-		

ID 150059

Invalid enable index in jog mode.			
Description	An external application requests the CNC to activate the incremental job mode for an axis. Therefore, the application commands an index of a parameter data record during activation. The data record contains values for incremental jog velocity and incremental jog width. The CNC detects that the axis parameter data record does not contain a parameter data record for incremental jog mode with this index.		
Response	Class	1	Warning The manual mode parameters are not transferred to the CNC. Axis could not be moved in manual mode.
Solution	Class	1	Please note that only a limited number of parameter data records can be stored in the axis parameter data record for incremental jog mode. If the incorrect index is still within this limit, an additional parameter data record with the corresponding index can be created. Otherwise, use an existing index for activation.
Parameter	%1:	Current value [-]	
		Value of the index for which there is no parameter data record for incremental jog mode in the axis parameter data record.	
Error type	-		

ID 150060

System error [▶ 12]

Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 150061

Repeatedly pressed the same inching key.			
Description	One of the two incremental jog or continuous jog manual operating modes is activated for an axis. If these manual modes are active, movement of axis is controlled by input from keys. Pressing a button is to start axis motion. Therefore, the CNC is informed of the start of this button press and the required motion direction of the axis via the available interfaces. If the CNC receives another message about a button press after the motion starts, this message appears before the CNC was informed of the end of the previous button press. The message appears irrespective of whether the axis motion direction commanded by the repeated button press matches the previous button press or not. See also: [HLI// Section: Enforcing a button press] For CNC build < 2800 [HLI// Section: Enforcing a button press] For CNC build > 2800		
Response	Class	1	Warning
Solution	Class	1	This message can generally be avoided if the previous keystroke is terminated by the assignment of "End keystroke" (0) before a new keystroke is assigned.
Error type	3, Error message from communication.		

ID 150063 - 150080

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 150087

Upper limit for handwheel increments greater than maximum value.			
Description	The manual mode parameters are read and interpreted by the CNC. The program detects that a value for the P-MANU-00007 parameters was specified in the manual mode parameters. This value exceeds the specified limit for the maximum number of handwheel increments per CNC cycle fixed in the CNC system.		
Response	Class	3	Manual mode parameters are not transferred to the CNC.
Solution	Class	6	Correct the value for P-MANU-00007 so that it is <= the limit value (see parameter 2 of the message). Reinterpret manual mode parameters by restarting CNC system.

Parameter	%1:	Current value [-]
		Maximum number of handwheel increments (P-MANU-00007) allowed by the handwheel per CNC cycle defined in the manual mode parameter record.
	%2:	Limit value [-]
		Maximum number of handwheel increments per CNC cycle taken into account by the CNC system.
	%3:	Logical identifier of control element [-]
		Logical identifier of control element where the value of parameter P-MANU-00007 exceeds the value set in the CNC system.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 150088

Number of handwheels does not match with the number of logical handwheels.			
Description	When interpreting the manual mode parameters, the system detects that there are more or less handwheel parameter data records than specified in the parameter P-MANU-00001.		
Response	Class	3	Manual mode parameters are not adopted in the CNC, but the CNC start-up is continued using default parameters.
Solution	Class	6	Correct value for parameter P-MANU-00001 or Add missing handwheel parameter data records or remove surplus records.
Parameter	%1:	Current value [-]	
		Value of parameter P-MANU-00001 in manual mode parameter data record.	
	%2:	Expected value [-]	
		During interpretation of manual mode parameters, handwheel parameter records were found.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 150089

Number of inching keys does not match with number of logical inching keys.			
Description	When interpreting manual mode parameters, the system detects that there are fewer handwheel parameter records than specified in parameter P-MANU-00008.		
Response	Class	3	Manual mode parameters are not adopted in the CNC, but the CNC start-up is continued using default parameters.
Solution	Class	6	Correct value for parameter P-MANU-00008 or Add missing key parameter data records.
Parameter	%1:	Current value [-]	
		Value of parameter P-MANU-00008 in manual mode parameter data record.	
	%2:	Expected value [-]	
		Key parameter records were found during interpretation of manual mode parameters.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 150090

Invalid resolution of handwheel parameterised.			
Description	In the manual mode parameter data record, the value 0 was parameterised for at least one of the parameters P-MANU-00005 or P-MANU-00006. Together these parameters define a handwheel resolution.		
Response	Class	3	Manual mode parameters are not transferred to the CNC and start-up of CNC system is aborted.
Solution	Class	6	Correct the values of P-MANU-00005 or P-MANU-00006.
Parameter	%1:	Current value [-]	
		Value of parameter P-MANU-00005 or P-MANU-00006.	
	%2:	Logical identifier of control element [-]	
		Logical control identifier of the handwheel with the incorrect parameter.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 150091

Multiple assignment of logical hand wheel number.			
Description	After interpretation of the manual mode parameters, a plausibility check is carried out. The CNC detects that the same logical handwheel number (@@P-MANU-00002) was used for several handwheels when parameterizing handwheels.		
Response	Class	3	Manual mode parameters aren't taken over into CNC, but CNC start-up is continued using default parameters.
Solution	Class	6	Correct values of logical handwheel numbers, which are used by several handwheel.
Parameter	%1:	Current value [-]	
		Logical handwheel number (P-MANU-00002) used for several handwheels.	
	%2:	Current value [-]	
		Index of handwheel parameter set containing logical handwheel number that is used several times.	
	%3:	Current value [-]	
		Index of handwheel parameter set even containing logical handwheel number that is used several times.	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 150092

Multiple assignment of logical inching key number.			
Description	After interpretation of the manual mode parameters, a plausibility check is carried out. The CNC detects that the same logical key number (P-MANU-00009) was used for several keys when keys were parameterized.		
Response	Class	3	Manual mode parameters aren't taken over into CNC, but CNC start-up is continued using default parameters.
Solution	Class	6	Correction of the values for the logical key number in the manual operation data set that were used several times.

Parameter	%1:	Current value [-]
		Logical key number (P-MANU-00009) used for multiple keys.
	%2:	Current value [-]
		Index of keys parameter set containing logical key number that is used several times.
	%3:	Current value [-]
		Index of key parameter set even containing logical key number that is used several times.
Error type	2, Error message by data transfer from parameter list into control device.	

ID 150096

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150099

Upper limit for handwheel increments may not be 0.			
Description	The manual mode parameters are read and interpreted by the CNC. The program detects that the value 0 was specified in the manual mode parameters for one of the P-MANU-00007 parameters.		
Response	Class	3	Manual mode parameters are not transferred to the CNC.
Solution	Class	6	Correct the value for P-MANU-00007 so that it is > 0. Reinterpret manual mode parameters by restarting CNC system.
Parameter	%1:	Current value [-]	
		Value specified in the manual mode parameter record for P-MANU-00007.	
	%2:	Limit value [-]	
		Limit value that was exceeded.	
	%3:	Logical identifier of control element [-]	
		Logical identifier of control element where the value of parameter P-MANU-00007 exceeds the limit value.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 150100

Number of handwheels greater than maximum value.			
Description	The manual mode parameters are read and interpreted by the CNC. The program detects that a value was specified in the manual mode parameters for the parameter P-MANU-00001. This value exceeds the fixed limit for the maximum number of handwheels specified in the CNC system and managed by the CNC system.		
Response	Class	3	Manual mode parameters are not transferred to the CNC.
Solution	Class	6	Correct the value for P-MANU-00001 so that it is <= the limit value (see parameter 2 of the message). Reinterpret manual mode parameters by restarting CNC system.

Parameter	%1:	Current value [-]
		Maximum number of handwheels (P-MANU-00001) defined in the manual mode parameter record and managed by the CNC system.
	%2:	Limit value [-]
		Maximum number of handwheels managed in the CNC system and fixed in the CNC system.
Error type	2, Error message on data transfer of parameter lists to controller.	

ID 150101

Number of inching keys greater than maximum value.			
Description	The manual mode parameters are read and interpreted by the CNC. The program detects that a value for the parameter P-MANU-00008 was specified in the manual mode parameters. This value exceeds the fixed limit for the maximum number of keys specified in the CNC system and managed by the CNC system.		
Response	Class	3	Manual mode parameters are not transferred to the CNC.
Solution	Class	6	Correct the value for P-MANU-00008 so that it is <= the limit value (see parameter 2 of the message). Reinterpret manual mode parameters by restarting CNC system.
Parameter	%1:	Current value [-]	
		Maximum number of keys (P-MANU-00008) defined in the manual mode parameter record and managed by the CNC system.	
	%2:	Limit value [-]	
		Fixed value in CNC system for maximum number of keys managed in the CNC system.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 150103

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 150104

Inadmissible value for handwheel counter address (P-MANU-00004).			
Description	The manual mode parameters are read and interpreted by the CNC. The CNC detects that a value for manual mode parameters P-MANU-00004 exceeds the CNC system internal limit. Parameter P-MANU-00004 indicates the address to which the handwheel counters are sent to the CNC. When this message is displayed, the high-level interface to the PLC was defined as the transfer location (see P-MANU-00003). The HLI interface selected to transfer the handwheel counter reading is defined in parameter P-MANU-00004 .		
Response	Class	3	Manual mode parameters are not transferred to the CNC.

Solution	Class	6	Correct the value for P-MANU-00004, P-MANU-00004 <= the limit value (see parameter 2 of the message). Reinterpret manual mode parameters by restarting CNC system.
Parameter	%1:	Current value [-]	
		Index of interface to transfer handwheel counter reading to CNC defined in manual mode parameter data record.	
	%2:	Limit value [-]	
		Maximal allowed index value.	
	%3:	Logical identifier of control element [-]	
		Identifier of logical control element whose parameters caused the message.	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 150105

Live cycle of key push finished.			
Description	One of the two incremental jog or continuous jog manual operation modes is activated for the axis. If these manual modes are active, movement of axis is controlled by input from keys. A time period (number of CNC interpolator cycles) has been transmitted with the message to the CNC that a corresponding key has been pressed, at the end of which the CNC automatically generates the "End of keystroke" command and stops the path motion of the axis unless the corresponding "End of keystroke" message has previously been sent to the CNC. Message occurs because after a “begin key stroke”-event time expires the given interval to send corresponding “end of key stroke”-event.		
Response	Class	1	Warning CNC internal creates the “end of key stroke”-event. With continuous jog mode enabled, all axes to which this key is assigned and which are moved via this key will stop. With incremental jog mode enabled, the axes will continue to move further if there is any distance to go of the jog step size. Therefore the event doesn't affect the movement of the axes.
Solution	Class	1	To prevent this output of this message, the corresponding “End of button press” event must be sent to the CNC within the commanded “Duration of button press” after the message of a “Start of button press” event is sent to the CNC. As a remedy here, command • a longer “Duration of button press” to be sent to the CNC • or inform the CNC of the “End of button press” event at an earlier time than before.
Parameter	%1:	Logical control element number [-]	
		P-MANU-00009: Number of logical button pair for which the “Duration of button press” expires.	
Error type	-		

ID 150106

Unexpected repeat of keystroke.

Description	The refresh of the keystroke was only received after the lifetime of the keystroke has expired. The movement first has to be started by a normal key press. Or the direction of the refresh key press is opposite to the last key press.		
Response	Class	1	Refresh is not considered
Solution	Class	1	Trigger refresh faster or correct the direction of the refresh
Parameter	%1:	Logical identifier of control element [-]	
		Actual number of the key	
	%2:	Current value [-]	
		Current state of the key (–1 - backward, 0 – not pressed, 1 – forward)	
	%3:	Incorrect value [-]	
		New reported key press	
Error type	-		

ID 150107

The list contains an unknown element.			
Description	During interpretation of the manual mode parameter data set, an unknown list element is detected		
Response	Class	1	Start-up of the control is continued.
Solution	Class	1	Remove or modify the unknown list element from manual mode parameter data set.
Error type	-		

ID 150108 - 150111

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 150112

Parameters for handwheel are not initialized.			
Description	Before activation of manual mode for axis by G200 or G201, the parameter set (P-AXIS-00025) for the operation modes handwheel has to be initialized. The values must be initialized in the axis parameter list.		
Response	Class	1	Warning
Solution	Class	1	Correct initialization of parameters in axis list or NC program.
Parameter	%1:	Logical axis number [-]	
		For logical axis number of concerned axis, see P-AXIS-00016	
Error type	-		

ID 150113

Parameters for continuous jog are not initialised.

Description	Before enabling manual mode for axis with G200 or G201, initialise at least one parameter record (P-AXIS-00077) for the continuous jog modes. The values must be initialised in the axis parameter list.		
Response	Class	1	Warning
Solution	Class	1	Specify correct parameters in axis list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
Error type	-		

ID 150114

Parameters for incremental jog are not initialised.			
Description	Before enabling manual mode with G200 or G201, initialise at least one parameter record P-AXIS-00076, P-AXIS-00232) for the handwheel modes. The values must be initialised in the axis parameter list.		
Response	Class	1	Warning
Solution	Class	1	Specify correct parameters in axis list.
Parameter	%1:	Logical axis number [-]	
		Logical axis number of affected axis, see P-AXIS-00016	
Error type	-		

ID 150115

Current distance exceeds permissible value range.			
Description	The numerical range is exceeded in the calculation of the current distance, consisting of the product of handwheel increments and resolution.		
Response	Class	1	Output of warning; correct the value
Solution	Class	1	Check the settings for handwheel increments.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
	%2:	Error value [0.1 10^-3 mm or ø]	
	%3:	Corrected value [0.1 10^-3 mm or ø]	
Error type	-		

2.16 Utility error (as of ID area 220000)

2.16.1 ID-range 220000-220249

ID 220000 - 220011

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 220012

Configuration of MTS failed.			
Description	An error occurred when attempting to connect to a remote message transport system.		
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
		ID of message transport system	
	%2:	Current value [-]	
		ID	
	%3:	Current value [-]	
		Individual error code	
Error type	-		

ID 220013

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 220015

OSACA platform parameter error.			
Description	The syntax of the specified OSACA configuration parameters was incorrect.		
Response	Class	3	
Solution	Class	1	
Error type	5, Error message by access on files.		

ID 220016

Specified hardware ID is unknown.			
Description	The hardware ID specified for Signal/Wait is unknown.		
Response	Class	3	
Solution	Class	7	
Error type			

ID 220017

Message could not be set to specified hardware.(connecting error)			
Description	An error occurred when sending a message (signal/wait) to the specified hardware.		
Response	Class	3	
Solution	Class	0	
Error type	-		

ID 220018

Message could not be sent to the specified hardware.(timeout)		
Description	A timeout error occurred when sending a message (signal/wait) to the specified hardware.	
Response	Class	3
Solution	Class	1
Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	-	

ID 220019

Start-up of OSACA stack is impossible.		
Description	An error occurred during start-up of the OSACA stack.	
Response	Class	2
Solution	Class	7
Parameter	%1:	Current value [-]
Error type	-	

ID 220020

System error [▶ 12]		
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.	
Response	Class	-

Solution	Class	8	Requires controller restart.

ID 220021

Specified hardware ID is unknown.			
Description			
Response	Class	3	
Solution	Class	0	
Error type	-		

ID 220022

Specified hardware ID is unknown in CORBA naming service.			
Description			
Response	Class	3	
Solution	Class	0	
Parameter	%1:		
	%2:	Current value [-]	
Error type			

ID 220027 / 220028

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 220029

Invalid logical axis number in retain data.			
Description	The retain data contains data for a logical axis which does not exist in the current configuration.		
Response	Class	2	Warning output. The retain data for this axis is not used.
Solution	Class	3	Check the cause for the invalid retain data. Possible causes: • The configuration was changed. • Problems during shutdown causing incorrect data to be written.
Parameter	%1:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of affected axis	
Error type	-		

2.16.2 ID-range 230000-230249

ID 230000 - 230039

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 230040

Communication operation interrupted.			
Description	This error message show, that the communication process needs more time as determined. It is possible, that the connection between HMI and control is broken.		
Response	Class	3	Abort job processing.
Solution	Class	1	Check the communication process between HMI and control.
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 230041

Simulation of an error PDU.			
Description	This error message show, that the communication process needs more time as determined. It is possible, that the connection between HMI and control is broken.		
Response	Class	3	Warning
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	-		

2.16.3 ID-range 240000-240249

ID 240000 - 240006

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 240007

Cam table manager: Invalid pointer.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 240008

Cam table manager: Insufficient memory, table cannot be created.			
Description	Cam table cannot be created because there is not enough memory left.		
Response	Class	3	Cam table is not created.
Solution	Class	1	Resize cam table, or delete cam tables which are not in use.
Error type	-		

ID 240009

Cam table manager: Maximum number of tables exceeded.			
Description	The maximum number of available cam tables is exceeded.		
Response	Class	3	Cam table is not loaded.
Solution	Class	1	Reduce number of cam tables to be used.
Error type	-		

ID 240010

Cam table manager: Table is locked, table cannot be created.			
Description	The cam table is locked and can therefore not be created. Reason: Cam table was already loaded or is in use.		
Response	Class	3	Cam table is not loaded.
Solution	Class	1	Unload and reload cam table.
Error type	-		

ID 240011

Cam table manager: Invalid table type, table cannot be created.	
Description	The table type specified in the table file is invalid. Cam table cannot be created.

Response	Class	3	Cam table is not loaded.
Solution	Class	1	Check table type and correct it.
Error type	-		

ID 240012

Cam table manager: Table is already used, table cannot be created.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240013

Cam table manager: Invalid table id, table cannot be created.			
Description	The specified table id is not valid.		
Response	Class	3	Cam table is not loaded.
Solution	Class	1	Check id type and correct it.
Error type	-		

ID 240014

Cam table manager: Table does not exist, table cannot be created.			
Description	Cam table cannot be changed because it is not loaded or does not exist.		
Response	Class	3	-
Solution	Class	1	Load or create cam table.
Error type	-		

ID 240015

Cam table manager: Table is locked and cannot be created.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240016

Cam table manager: Line number exceeded, table cannot be changed.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240017

Cam table manager: Column number exceeded, table cannot be changed.	
Description	

Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240018

Cam table manager: Table size exceeded, table cannot be changed.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240019

Cam table manager: Invalid access mode, table cannot be changed.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240020

Cam table manager: Table is not ready to use.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240021

Cam table manager: Table is blocked and cannot be deleted.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240022

Cam table manager: Table does not exist and cannot be deleted.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240023

Cam table manager: Invalid table type, table cannot be changed.			
Description			

Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240024

Cam table manager: Invalid table type, table cannot be read.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240025

Cam table manager: Table does not exist, table cannot be read.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240026

Cam table manager: Line number exceeded, table cannot be read.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240027

Cam table manager: Column number exceeded, table cannot be read.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240028

Cam table manager: Table size exceeded, table cannot be read.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240029

Cam table manager: Access to table cell not possible.	
Description	

Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240030

Cam table manager: Invalid job type.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240031

Cam table manager: Invalid file name.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240032

Cam table manager: Table file cannot be opened.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240033

Cam table manager: Invalid table function type.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240034

Cam table manager: Invalid table header element type.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240035

Cam table manager: Too many lines in cam table file.

Description	The number of lines specified in the CAM table file is too large. The actual number of lines is smaller.		
Response	Class	1	Warning output.
Solution	Class	1	Correct number of lines.
Error type	-		

ID 240036

CAM table interpreter: timeout writing to FIFO.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240037

CAM table interpreter: timeout reading from FIFO.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240038

CAM table interpreter: Invalid pointer to table manager.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240039

CAM table interpreter: Invalid PDU command.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240040 - 240042

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 240043

CAM table interpreter: Table file wasn’t found.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240044

CAM table interpreter: Invalid table type.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240045

CAM table interpreter: Invalid value for table lines.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240046

CAM table interpreter: Invalid value for table data.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240047

CAM table interpreter: Keyword for table ID not found.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240048

CAM table interpreter: Keyword for table type not found.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240049

CAM table interpreter: Keyword for line number of table not found.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240050

CAM table interpreter: Too few columns with values within the table.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240051

CAM table interpreter: Too few lines with values within the table.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240052

CAM table interpreter: Unknown keyword found within the table.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240053

CAM table interpreter: Invalid value for table ID specified.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

ID 240054 - 240056

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 240057

CAM table interpreter: Unknown receiver for command.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 240058

CAM table interpreter: Invalid value for interpolation type within table.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message on data transfer to controller.		

2.16.4 ID-range 250000-250249

ID 250000

Invalid PDU code for event manager.			
Description	The communication participant event manager (handling of #SIGNAL and #WAIT) has received a message packet that it cannot process. Possible cause for this error message is that the #MSG command explicitly specifies a receiver. Example: #MSG 35 [“Message from the nc program”]		
Response	Class	3	
Solution	Class	1	The keywords specified in the programming manual should be used instead of numbers. See [PROG// Section Messages from the NC program].
Parameter	%1:	Incorrect value [-]	
	%2:	Current value [-]	
Error type	3, Error message from communication.		

ID 250001

Maximum number of non-acknowledged SIGNAL/WAIT exceeded.			
Description	Too many #SIGNAL or #WAIT signals were programmed and they have not yet been acknowledged. The memory limit is reached.		
Response	Class	3	
Solution	Class	6	When programming, make sure that the limits documented in [SYSP] are not exceeded. Modify the CNC programs accordingly.
Parameter	%1:	Limit value [-]	
		Maximum number of open #SIGNAL/#WAIT signals of the controller.	
	%2:	Current value [-]	
		Number of open #SIGNAL/#WAIT signals.	
	%3:	Limit value [-]	
	%4:	Current value [-]	
Error type	-		

ID 250002

Unnecessary SIGNAL in list detected.

Description	If #SIGNAL and #WAIT are used with receiver specified, the program sequence must be considered. The programmed #WAIT sequence does not match the programmed #SIGNAL sequence. Wrong: #SIGNAL [ID4711 CH1] #SIGNAL [ID9999 CH1] … #WAIT [ID9999 CH1] #WAIT [ID4711 CH1] Correct: #SIGNAL [ID4711 CH1] #SIGNAL [ID9999 CH1] … #WAIT [ID4711 CH1] #WAIT [ID9999 CH1]		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Incorrect value [-]	
	%2:	Expected value [-]	
	%3:	Incorrect value [-]	
Error type	-		

ID 250003

SIGNAL/WAIT receipt cannot send.			
Description	The acknowledgement of an open #WAIT event cannot be sent repeatedly. Please check the communication connection.		
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	3, Error message from communication.		

ID 250004

Connection to given hardware at #SIGNAL-programming unknown.	
Description	The controller programmed with the keyword HW in the #SIGNAL/#WAIT command is unknown. Please check the configuration. Example: #SIGNAL [ID10 HW5]

Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 250005

Acknowledgement of driver to remote hardware during reset timed out.			
Description	The configured remote hardware (#SIGNAL/#WAIT function) does not report in time within the specified time. A timeout occurred.		
Response	Class	3	
Solution	Class	1	Check the configuration and the availability of the remote hardware.
Error type	-		

ID 250008

Given channel number if SIGNAL&WAIT is above maximum number of CNC channels.			
Description	The programmed channel number is greater than the maximum number of CNC channels. The order cannot be executed. Example: Wrong: #SIGNAL [ID1 CH100] Correct: #SIGNAL [ID1 CH1]		
Response	Class	3	
Solution	Class	1	Please modify the NC program.
Parameter	%1:	Incorrect value [-]	
	%2:	Limit value [-]	
	%3:	Current value [-]	
	%4:	Current value [-]	
Error type	-		

ID 250009

Number of unacknowledged BROADCAST SIGNAL/WAIT exceeds limit.			
Description	Too many BROADCAST #SIGNAL or #WAIT were programmed,and they have not yet been acknowledged. The memory limit is reached.		
Response	Class	3	
Solution	Class	6	When programming, make sure that the limits documented in [SYSP] are not exceeded. Modify the NC programs accordingly.

Parameter	%1:	Limit value [-]
		Maximum number of open #SIGNAL / #WAIT signals of the controller.
	%2:	Current value [-]
		Number of open #SIGNAL / #WAIT .
	%3:	Limit value [-]
	%4:	Current value [-]
Error type	-	

ID 250010 - 250012

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

2.16.5 ID-range 260000-260249

ID 260001

AT-telegram was missing two times.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Logical axis number [-]	
	%3:	Current value [-]	
	%4:	Current value [-]	
Error type	-		

ID 260002

MST telegram was missing two times, signal MSTMISS, MSTEARLY, MSTLATE.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Ring number [-]	
	%2:	Error value [-]	
	%3:	Error value [-]	
	%4:	Current value [-]	
Error type	-		

ID 260003

Physical transmission is disturbed.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Ring number [-]	
	%2:	Incorrect value [-]	
	%3:	Current value [-]	
Error type	-		

ID 260004

SERCOS transmission error, signal INT_PROGERR.			
Description			
Response	Class	1	
Solution	Class	6	

Parameter	%1:	Current value [-]
	%2:	Incorrect value [-]
	%3:	Incorrect value [-]
	%4:	Current value [-]
	%5:	Incorrect value [-]
Error type	-	

ID 260005

HS timeout error occurred.		
Description		
Response	Class	1
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	Logical axis number [-]
	%3:	Status [-]
Error type	-	

ID 260007

General time-out error occurred.		
Description		
Response	Class	1
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Logical axis number [-]
	%4:	Limit value [-]
	%5:	Current value [-]
Error type		

ID 260008

Selected phase is wrong, phase 1 after 0 not possible.		
Description		
Response	Class	1
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	State [-]
	%3:	Incorrect value [-]

Error type	-

ID 260009

Selected phase is wrong, phase 2 after 1 not possible.		
Description		
Response	Class	1
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	State [-]
	%3:	Incorrect value [-]
Error type	-	

ID 260010

Selected phase is wrong, phase 3 after 2 not possible.		
Description		
Response	Class	1
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	State [-]
	%3:	Incorrect value [-]
Error type	-	

ID 260011

Selected phase is wrong, phase 4 after 3 not possible.		
Description		
Response	Class	1
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	State [-]
	%3:	Incorrect value [-]
Error type	-	

ID 260012

Selected phase does not exist.		
Description		
Response	Class	1
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	State [-]
	%3:	Incorrect value [-]
Error type	-	

ID 260013

Phase switch exceeds time-out.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Limit value [-]	
Error type	-		

ID 260016

Checked address could not be found.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Incorrect value [-]	
Error type	-		

ID 260017

Phase switch to CP1 not possible (HS-TIMEOUT). Check drive addresses.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Incorrect value [-]	
	%3:	Incorrect value [-]	
	%4:	Incorrect value [-]	
Error type	-		

ID 260018

Received telegram wrong in CP1.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Error value [-]	
	%4:	Logical axis number [-]	
Error type	-		

ID 260019

Switch back from phase 1 to phase 0 not possible.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Incorrect value [-]	
	%3:	Incorrect value [-]	
	%4:	Limit value [-]	
Error type	-		

ID 260020

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

ID 260022

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

ID 260025

Calculation of T1.m not possible (TATAT time slot missing).			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Logical axis number [-]	
Error type	-		

ID 260026

MDT cannot be sent in permitted time (T2 too big).			
Description			
Response	Class	1	
Solution	Class	6	

Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Limit value [-]
	%4:	Incorrect value [-]
Error type	-	

ID 260028 - 260030

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

ID 260031

Identification 0002 in SERCON 410B not found.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Expected value [-]	
	%3:	Expected value [-]	
Error type	-		

ID 260032

SERCON 410B cannot be reset.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Incorrect value [-]	
	%2:	Expected value [-]	
Error type	-		

ID 260033

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

ID 260034

Read data are longer than commanded data length.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Incorrect value [-]	
	%2:	State [-]	
	%3:	Expected value [-]	
	%4:	Incorrect value [-]	
Error type	-		

ID 260035

Received HS bit is wrong.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	-		

ID 260036

In this moment the service channel are not open.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	[-]	
	%2:	Identification number [-]	
	%3:	Error value [-]	
	%4:	Status [-]	
Error type	-		

ID 260037

Wrong access of element 0.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260038

IDENT number not exist.			
Description			
Response	Class	1	

Solution	Class	1	
Error type	-		

ID 260039

Invalid access to element 1.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260040

Element of the name from IDENT not exist.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260041

The transfer of the name is too short.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260042

The transfer of the name is too long.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260043

The name cannot be changed.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260044

The name is currently write-protected.			
Description			
Response	Class	1	

Solution	Class	1	
Error type	-		

ID 260045

The transfer of the attribute is too short.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260046

The transfer of the attribute is too long.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260047

It is impossible to change the attribute.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260048

The attribute is currently write-protected.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260049

The unit element of IDENT not exist.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260050

The transfer of the unit is too short.			
Description			
Response	Class	1	

Solution	Class	1	
Error type	-		

ID 260051

The transfer of the unit is too long.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260052

It is impossible to change the unit.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260053

The unit is currently write-protected.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260054

Minimum value element from IDENT not exist.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260055

The transfer of the minimum value is too short.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260056

The transfer of the minimum value is too long.			
Description			
Response	Class	1	

Solution	Class	1	
Error type	-		

ID 260057

It is impossible to change the minimum value.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260058

The minimum value is currently write-protected.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260059

Maximum value element from IDENT not exist.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260060

The transfer of the maximum value is too short.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260062

It is impossible to change the maximum value.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260063

The maximum value is currently write-protected.			
Description			
Response	Class	1	

Solution	Class	1	
Error type	-		

ID 260064

The transfer of the date is too short.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260065

The transfer of the data item is too long.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260066

It is impossible to change the data item.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260067

The data item is currently write-protected.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260068

The date is under the minimum value.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260069

The data item is greater than the maximum value.			
Description			
Response	Class	1	

Solution	Class	1	
Error type	-		

ID 260070

Incorrect date value.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260071

Service data error is not coded.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260074

Incorrect reception of single bits, RDIST signal.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Ring number [-]	
	%2:	Incorrect value [-]	
	%3:	Current value [-]	
Error type	-		

ID 260075

Transfer of MST returns early, MSTEARLY signal.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Ring number [-]	
	%2:	Incorrect value [-]	
	%3:	Incorrect value [-]	
	%4:	Current value [-]	
Error type	-		

ID 260076

Transfer of MST returns late, MSTLATE signal.	
Description	

Response	Class	1	
Solution	Class	6	
Parameter	%1:	Ring number [-]	
	%2:	Incorrect value [-]	
	%3:	Incorrect value [-]	
	%4:	Current value [-]	
Error type	-		

ID 260077

Requested address too big.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Incorrect value [-]	
	%2:	Current value [-]	
Error type	-		

ID 260078

A - parameter too big.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Incorrect value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 260079

Y - parameter too big.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260080

Index from a searching IDENT too big.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Incorrect value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 260081

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

ID 260082

Wrong element instructed.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Error value [-]	
	%2:	State [-]	
	%3:	Ring number [-]	
Error type	-		

ID 260083

Impossible to initialized SERCOS-master.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 260084

Order from service channel by SERCOS CP under 2.			
Description			
Response	Class	1	
Solution	Class	6	
Parameter	%1:	Error value [-]	
	%2:	Error value [-]	
	%3:	Limit value [-]	
Error type	-		

ID 260085

Time slot parameter information.			
Description			
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Current value [1 µs]
Error type	-	

ID 260086

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

ID 260089

MDT-Configuration of SERCOS telegram 7 wrong.			
Description	A specified element of the SERCOS master data telegram (MDT) is not known. All configured telegram entries have to be known by the CNC to be linked to the corresponding CNC element. If you want to configure elements in the drive telegram, which are not commanded by the CNC, just add these additional elements after linkage of CNC axis to physical axis. Example: Telegram configuration MDT = S-0-0047, AT = S-0-0051 in the physical axis Link logical CNC axis to physical axis, insert additional SI-control word P-0-3210 in drive telegram		
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
		see P-AXIS-00016	
	%2:	Current value [-]	
		see P-AXIS-00132	
Error type	-		

ID 260090

AT-Configuration of SERCOS telegram 7 wrong.	
Description	A specified element of the SERCOS drive telegram is unknown. All configured telegram entries have to be known by the CNC to be linked to the corresponding CNC element. If you want to configure elements in the drive telegram, which are not linked to the CNC, just add this additional elements after linkage of CNC axis to physical axis. Example: Telegram configuration MDT = S-0-0047, AT = S-0-0051 in the physical axis Link logical CNC axis to physical axis insert additional SI-status word P-0-3215 in drive telegram

Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
		see P-AXIS-00016	
	%2:	Current value [-]	
		see P-AXIS-00131	
	%3:	Incorrect value [-]	
Error type	-		

ID 260091

SERCOS, calculation of total MDT length failed.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 260092

Operation data write protected by a password			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260093

Date item is write-protected for service channel access.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260094

Invalid indirect addressing: (e.g. data container, list handling).			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260095

Data item is write-protected due to other settings.			
Description			
Response	Class	1	

Solution	Class	1	
Error type	-		

ID 260096

Procedure command is already enabled.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260097

Procedure command not interruptible.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260098

Procedure command in current state not executable.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260099

Procedure command not executable (wrong parameters).			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 260100

Invalid slave identifier.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Logical axis number [-]
Error type	-	

ID 260101

RT thread definition contains a function more than once.			
Description	Several functions can be assigned to the definition of a thread using P-RTCF-00012. It is not permitted to assign the same function to a thread multiple times.		
Response	Class	3	Abort current job.
Solution	Class	6	Check and modify the thread parameters.
Parameter	%1:	Current value [-]	
		Thread index	
	%2:	Current value [-]	
		Index of specified function	
Error type	6, Error message by data transfer to control device.		

2.16.6 ID-range 270000-270249

ID 270000 / 270001

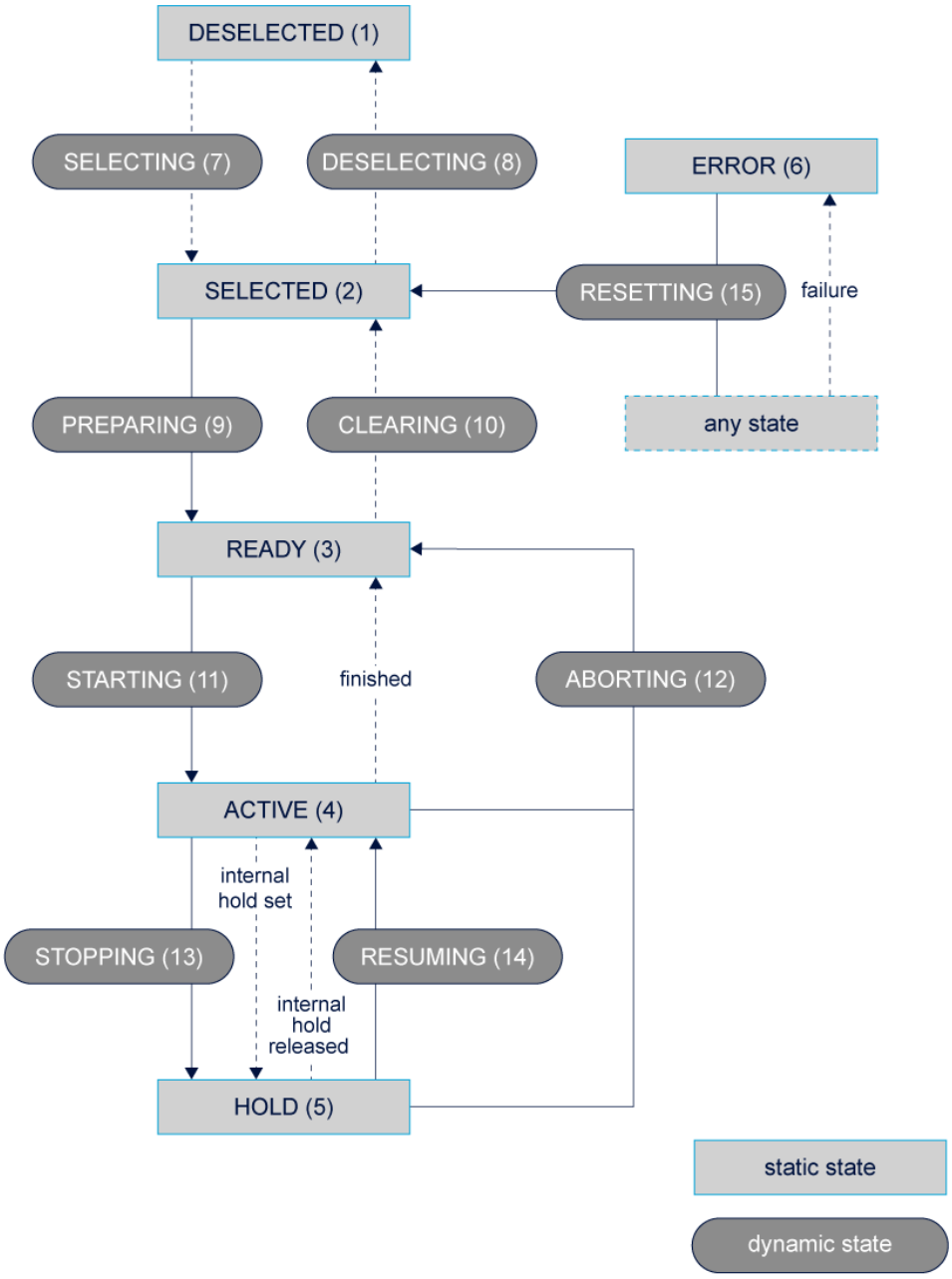
System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270018

Object does not exist.			
Description	A communication object required for controller start-up is missing in the parameter list for the software module Channel or HMI. This error can only occur if the start-up list has been changed. The newly specified parameter lists for HMI objects and BF channel are incomplete.		
Response	Class	3	Abort current job.
Solution	Class	7	Check and modify the parameters P-STUP-00024 to P-STUP-00027. Reset to the value described in the documentation.
Error type	-		

ID 270021

Transition for this mode in current state not allowed.

Description	The selected transition is not permissible in the current operation mode. The possible individual transitions are described in [HLI// Section; Operation modes].  General state graph for operation modes To make handling easier, there is a possibility in the mode state commanding interface to include the required state in state graphs in any case. Here, the controller executes the required transitions itself.		
Response	Class	1	
Solution	Class	1	Correct selection of the transition that is permissible in the particular state; change command in PLC or user interface.

Parameter	%1:	
		Selected transition
	%2:	
		Current operation mode
	%3:	
		Current operation mode using the mode-state interface
Error type	-	

ID 270022

Unknown state or mode.			
Description	The desired state in the order for switching the state or the operation mode is not known. The possible states are in [HLI// Section: Operation modes] that follows.		
Response	Class	1	The order is rejected
Solution	Class	1	Change order via HMI or PLC
Parameter	%1:		
		Active operation mode	
	%2:		
		New unknown state in the active operation mode	
Error type	-		

ID 270023

Chosen mode is invalid.			
Description	The selected operation mode is not known. Possible operation modes of the controller are described in [HLI// Section: Operation modes].		
Response	Class	1	The order is rejected
Solution	Class	1	Change the order and new order via HMI or PLC.
Parameter	%1:		
		Chosen operation mode	
	%2:		
		Current operation mode	
Error type	-		

ID 270024

Object cannot be written.			
Description	The request cannot be executed. The necessary communication object was not writeable. This can be caused by multiple requests, that cannot be processed at the appropriate speed.		
Response	Class	3	The order is rejected
Solution	Class	1	Send new request by HMI or PLC

Parameter	%1:	Current value
		Number of object
	%2:	Current value
		Name of object
	%3:	Instance
		Active operation mode
	%4:	State
		State oft the active operation mode
	%5:	Transition
		Transition ID
Error type	-	

ID 270029 / 270030

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270031

BF-Pin cannot connect with HLD from an axis.			
Description	A BF could not be connected to the High Level Driver of the PLC interface.		
Response	Class	3	Command is aborted.
Solution	Class	7	System restart.
Error type	-		

ID 270032

BF-Pin cannot connect with HLD from CTM-plug.			
Description	A BF could not be connected to the CTM.		
Response	Class	3	Command is aborted.
Solution	Class	7	Restart the system.
Error type	-		

ID 270033

Check reset of NC-channel without explicit order over MCM.			
Description	The NC-channel was commanded to do a reset, although the MCM did not command any.		
Response	Class	1	Warning output.
Solution	Class	1	Check reset command.
Parameter	%1:	Current MCM-Mode.	
	%2:	Current MCM-State.	
	%3:	Flag to signal if BF BAVO is started.	
	%4:	Flag to signal if BF BAHN is started.	

Error type	-

ID 270034 - 270036

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270037

The PLC must be running in PLC RUN mode at this time.			
Description	After the controller has started-up, a cyclic check is performed to determine which control units of the HLI are newly registered or deregistered. Accordingly, the internal simulation is switched on or off, and the HMI requests are forwarded to the PLC or directly enforced. The parameter P-STUP-00032 can be used to specify that a one-time check is only made at the start-up time to determine which units are operated by the PLC. A later registration is not possible. The prerequisite for this, is that the PLC is already running before the CNC start-up.		
Response	Class	3	Transition to error state, no communication with the PLC possible.
Solution	Class	7	Set parameter P-STUP-00032 to 0.
Error type	-		

ID 270039 / 270040

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270041

Initialisation of work data from HLDX (in PLC_TEST_MODE) failed.			
Description	The initialization of the axis specific working data of the High Level Driver for the PLC interface failed.		
Response	Class	3	Command is aborted.
Solution	Class	7	Restart the system.
Error type	-		

ID 270042

Initialisation of work data from HLDX (in PLC_TEST_MODE) failed.	
Description	

Response	Class	3	
Solution	Class	7	
Error type	-		

ID 270045

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270046

Interpreter cannot evaluate the PLC-I/O-mapping list.			
Description	There was an error during interpretation of the SERCOS PLC-I/O mapping list, e.g. wrong token.		
Response	Class	3	Command is aborted.
Solution	Class	1	Check PLC-I/O mapping list.
Error type	-		

ID 270047

Error while checking the new mapping list.			
Description	An error was detected in the parameter of the new PLC I/O mapping list.		
Response	Class	3	Command is aborted.
Solution	Class	1	Check parameter in the new PLC I/O mapping list.
Parameter	%1:	Current value [-]	
		Current state of the interpreter.	
Error type	-		

ID 270048

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270049

No mapping declaration for SERCOS Ident (output) found.			
Description	There is no mapping rule configured in the PLC I/O mapping list for the cyclic SERCOS output ident.		
Response	Class	3	Command is aborted.
Solution	Class	1	Check PLC I/O mapping list.

Parameter	%1:	Current value [-]
		Incorrect SERCOS ident.
Error type	-	

ID 270050

No mapping declaration for SERCOS Ident (input) found.			
Description	There is no mapping rule configured in the PLC /O mapping list for the cyclic SERCOS input ident.		
Response	Class	3	Command is aborted.
Solution	Class	1	Check PLC I/O mapping list.
Parameter	%1:	Current value [-]	
		Incorrect SERCOS ident.	
Error type	-		

ID 270051

Global transfer memory does not exist.			
Description	The number of the I/O devices could not be read, because there was no access to the global transfer memory.		
Response	Class	3	Command is aborted.
Solution	Class	7	Restart the system.
Parameter	%1:	Current value [-]	
		Size of PLC I/O transfer memory.	
Error type	-		

ID 270052

The I/O interface of the PLC cannot be allocated.			
Description	The shared memory to transport the cyclic PLC I/O data between NC and PLC could not be allocated.		
Response	Class	3	Command is aborted.
Solution	Class	7	Restart the system.
Parameter	%1:	Current value [-]	
		Size of PLC I/O transfer memory.	
Error type	-		

ID 270053

CNC monitors PLC - No sign of life of PLC.	
Description	The CNC monitors the PLC to see whether it gives a cyclic life sign. The monitor responds and reports that the PLC gave no life sign within the agreed time interval. One possible cause may be an unfavourable parameterisation of task priorities. When the watchdog mechanism is used, the PLC task must have a high priority. See also: [HLI// Section: Watchdog mechanism] for CNC builds < 2800 [HLI// Section: Watchdog mechanism] for CNC builds > 2800

Response	Class	3	Abort current job.
Solution	Class	6	The PLC activated the monitor via the HLI. The PLC also specified the time interval within which it gives a life sign via the HLI. Therefore, check the PLC cycle time and compare it with the time interval within which the PLC is to give a cyclic life sign. The time interval must have a greater value than the PLC cycle time. Then restart the controller.
Parameter	%1:	Error value [-]	
		Last state of the cycle counter with valid sign of life of the PLC	
	%2:	Current value [-]	
		Current state of the cycle counter	
	%3:	Current value [µs]	
		CNC cycle time	
	%4:	Current value [µs]	
		Parameterised time of the watchdog (see control unit)	
Error type	-		

ID 270054

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270055

Interpreter cannot evaluate the device list.			
Description	There is an error in the parameter of the device list.		
Response	Class	1	Command is aborted.
Solution	Class	1	Check the parameter of the device list.
Error type	-		

ID 270056 / 270057

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270059

When there is a transition, the current operation mode is not adopted.

Description	When an operation mode change is commanded, the starting operation mode does not match the current operation mode. The operation mode changeover is not carried out. See starting operation mode: [HLI// Section: Output operating mode] for CNC build < 2800 [HLI// Section: Output operating mode] for CNC build < 2800 The values entered in the mode state unit [HLI// Section: Operation modes] must be corrected. Possibilities: Enter the correct operation mode Set the value "0" for the current operation mode. The operation mode switch will be executed regardless of the current operation mode.		
Response	Class	3	The transition will not be done
Solution	Class	1	Correct the state of the starting operation mode. Create new request
Parameter	%1:		
		Current operation mode	
	%2:		
		Expected Mode	
Error type	-		

ID 270060

The current state differs from the expected one.			
Description	When an operation mode change is commanded, the ‘Starting state of the operation mode on operation mode change’ does not correspond to the current state. See: [HLI// Starting state of the operation mode on operation mode change] CNC Version < 2800 [HLI// Starting state of the operation mode on operation mode change] CNC Version > 2800 Switch-over of the operation mode/state is not executed. The values entered in the mode state unit [HLI// Section: Operation modes] must be corrected. Possibilities: Enter the correct state Set the value "0" for the current status. The operation mode switch will be executed regardless for the current operation mode/state.		
Response	Class	3	The operation mode switch is not executed.
Solution	Class	1	Correct the ‘Starting state of the operation mode on operation mode change’. Create new request

Parameter	%1:	
		Current state
	%2:	
		Expected State
Error type	-	

ID 270061

Given destination state in transition not allowed.			
Description	A job to change the state or an operation mode to change the operation mode itself requires specification of the required target state. The job that leads to this message contains a target state which is not permissible for the required operation mode. The following target states are permissible for the automatic (2), manual block input (3), manual (4) and homing (5) operation modes		
	Operation mode state		**Value**
	Deselected		1
	Selected		2
	Ready		3
	Active		4
	Hold		5
	Resetting		15
	The permissible target states for the Standby (1) operation mode are:		
	Operation mode state		**Given destination state in transition not allowed.**
	Deselected		1
	Selected		2
	Ready		3
Response	Class	3	The transition will not be executed
Solution	Class	1	Correct the destination state and start new request
Parameter	%1:		
			Selected destination state
	%2:		
			Current operation mode
Error type	-		

ID 270062

Destination state could not be reached in an appropriate time.			
Description	Switching to the specified target state could not be done in the appropriate time. The timeout aborts the transition.		
Response	Class	1	The transition is not completely done.
Solution	Class	1	Renewed assignment of the transition.
Parameter	%1:		Counter number of trials
Error type	-		

ID 270063 - 270069

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	Command is aborted.
Solution	Class	8	Requires restart of controller-

ID 270070

An error message could not be written to the HLI (channel interface).			
Description	The PLC reported receipt of the error messages at the HLI; however, it executes them in a job that is too slow. As a result, not all error messages can be sent to the PLC. The information is incomplete. Reading error messages should be executed in a faster job to ensure that all error messages can be received.		
Response	Class	-	Creating a new error message
Solution	Class	-	Increase the priority of the PLC task and the cycle time.
Parameter	%1:	Current value [-]	
		Error ID of the error message that was not transmitted	
Error type	-		

ID 270071

An error message could not be written to the HLI (axis interface).			
Description	The PLC reported receipt of the error messages at the HLI; however, it executes them in a job that is too slow. As a result, not all error messages can be sent to the PLC. The information is incomplete. Reading error messages should be executed in a faster job to ensure that all error messages can be received.		
Response	Class	3	Creating a new error message
Solution	Class	7	Increase the priority of the PLC task and the cycle time.
Parameter	%1:	Current value [-]	
		Error ID of the error message that was not transmitted	
	%2:	Logical axis number [-]	
Error type	-		

ID 270073

No PDU buffer for SERCOS service available.			
Description	The PLC High Level Driver has run out of PDU buffer and can therefore not send the SECOS service channel request.		
Response	Class	3	Command is aborted.
Solution	Class	7	Reset.
Error type	-		

ID 270074

Output of PDU for SERCOS service failed.			
Description	The PLC High Level Driver could not send the SERCOS service channel command.		
Response	Class	3	Command is aborted.
Solution	Class	7	Reset.
Parameter	%1:	Logical axis number [-]	
Error type	-		

ID 270075

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270076

CNC reset triggered by user.			
Description	A CNC reset was triggered by the HMI or PLC. This message will be created if a transition causes a reset of the control. Error events can be synchronised by evaluating this message. Older error messages are no longer valid.		
Response	Class	-	-
Solution	Class	-	-
Error type	-		

ID 270077

Activation job for manual mode was not processed by PLC (control unit not empty).

Description	The CNC tries to forward a job from another application (e.g. HMI) to the PLC. The job contains data on which input device (e. g. key, handwheel) commands a specific axis in manual mode. This job to assign an input device to an axis, however, could not be written within a defined number of CNC cycles via the associated control unit of the interface to the PLC because this control unit is still occupied by a previously transmitted data. Check the following points in the PLC application: 1. The **request semaphore** in the control unit must be reset by the PLC after each transfer of data from an activation request for manual mode. 2. The PLC cyclically checks the Control Unit for newly received data. 3. If the PLC is not to handle an activation request for manual mode, the **Enable** flag of the corresponding Control Unit is set to False. See [HLI// Section: Activating control elements for manual mode] for CNC builds < 2800 [HLI// Section: Activating control elements for manual mode] for CNC builds > 2800		
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 270078

Continuous jog parameter was not processed by PLC (control unit not empty).			
Description	The CNC tries to send to the PLC the parameterisation data for continuous jog mode received from another application (e.g. HMI). However, it was not possible to send this data to the PLC via the interface within a defined number of CNC cycles. The reason for this is that the associated control unit is still occupied by previously sent parameterisation data.		
Response	Class	3	Abort current job.
Solution	Class	1	The PLC application must be checked immediately to ensure that 1. the **request semaphore** in the control unit is reset by the PLC after the PLC adopted the parametrisation data. 2. the PLC checks the control unit cyclically for new jobs received. 3. if the PLC should not handle the parameterisation of the continuous jog mode, the **Enable**Flag must be set to FALSE for the associated control unit. See [HLI// Section: Continuous jog mode (by button press)] CNC Build < 2800 [HLI// Section: Continuous jog mode (by button press)] CNC Build > 2800
Parameter	%1:	Current value [-]	
		Number of CNC cycles in which the CNC tried to write the data into the interface to the PLC.	
	%2:	Limit value [-]	
		Maximal number of CNC cycles the CNC should try to write data into the interface to the PLC.	
Error type	-		

ID 270079

Incremental jog parameter was not processed by PLC (control unit not empty).			
Description	The CNC attempts to send to the PLC the incremental jog mode data received from another application (e.g. HMI). However, this data could not be transmitted to the PLC via the interface within a defined number of CNC cycles because the associated control unit is still occupied by a previously transmitted job for parameterisation.		
Response	Class	3	
Solution	Class	1	The PLC application must be checked immediately to ensure that 1. the **request semaphore** is reset by the PLC in the control unit after each extraction of parameterisation data. 2. the PLC checks the control unit for new parameter data received. 3. if the PLC is not to handle a parameterisation job for incremental jog mode, the **Enable** flag of the corresponding Control Unit is set to FALSE. See [HLI// Section: Incremental jog mode)] for CNC builds < 2800 [HLI// Section: Incremental jog mode)] for CNC builds > 2800".
Parameter	%1:		
		Number of CNC cycles in which the CNC attempted to write the data in the interface to the PLC.	
	%2:		
		Maximum number of CNC cycles the CNC should try to write data in the interface to the PLC.	
Error type	-		

ID 270080

Handwheel parameter was not processed by PLC (control unit not empty).			
Description	The CNC tries to send the handwheel parameterization data received from another application (e.g. HMI) to the PLC. This data couldn't be transferred to the PLC via the interface within a defined number of CNC cycles, because the related control unit is occupied by a former order to parametrize handwheel mode.		
Response	Class	3	
Solution	Class	1	The PLC application has to be checked, to be sure that 1. the **request semaphore** of the control unit is set to FALSE after each transfer of parametrization data into the PLC. 2. the PLC cyclical checks, if there are new parametrization data in the control unit. 3. if you don't want to handle handwheel parametrization by the PLC the **enable** flag of the concerning control unit is set to FALSE. See [HLI// Section: Handwheel mode] for CNC build < 2800 [HLI// Section: Handwheel mode] for CNC build > 2800
Parameter	%1:		
		Number of CNC cycles in which the CNC tried to write the data into the interface to the PLC.	
	%2:		
		Maximal number of CNC cycles the CNC should try to write data into the interface to the PLC.	

Error type	-

ID 270081

Key press was not processed by PLC (control unit not empty).			
Description	The CNC wants to pass on the data to the PLC after a button press in another application (e.g. HMI) and write it to the PLC interface. However, this data could not be written to the corresponding Control Unit of the interface to the PLC within a defined number of CNC cycles, because this Control Unit is still occupied by previously transmitted data.		
Response	Class	3	
Solution	Class	1	The PLC application has to be checked, to be sure that 1. the **request semaphore** of the control unit is set to FALSE after each transfer of data concerning a pressed key into the PLC. 2. the PLC cyclical checks, if there are new data concerning a pressed key in the control unit. 3. if you don't want to handle keys by the PLC the **enable** flag of the concerning control unit is set to FALSE. See [HLI// Section: Enforcing a button press] For CNC build < 2800 [HLI// Section: Enforcement of a keystroke] for CNC build > 2800
Parameter	%1:		
		Number of CNC cycles in which the CNC tried to write the data into the interface to the PLC.	
	%2:		
		Maximal number of CNC cycles the CNC should try to write data into the interface to the PLC.	
Error type	-		

ID 270082

Rapid mode key press was not processed by PLC (control unit not empty).			
Description	The CNC wants to pass on the data to the PLC after the press of a rapid traverse button in another application (e.g. HMI) and write it to the PLC interface. However, this data could not be written to the corresponding Control Unit of the interface to the PLC within a defined number of CNC cycles, because this Control Unit is still occupied by previously transmitted data.		
Response	Class	3	

Solution	Class	1	The PLC application has to be checked, to be sure that 1. the **request semaphore** of the control unit is set to FALSE after each transfer of data concerning a pressed rapid key into the PLC. 2. the PLC cyclical checks, if there are new data concerning a pressed rapid key in the control unit. 3. if you don't want to handle rapid key by the PLC, the **enable** flag of the concerning control is set to FALSE. See [HLI// Section: Rapid traverse velocity during path motion] For CNC build < 2800 [HLI// Section: Rapid traverse velocity during path motion] For CNC build > 2800“
Parameter	%1:		
		Number of CNC cycles in which the CNC tried to write the data into the interface to the PLC.	
	%2:		
		Maximal number of CNC cycles the CNC should try to write data into the interface to the PLC.	
Error type	-		

ID 270083 / 270085

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 270086

File name is too long.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 270091

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270092

BF-Pin cannot connect with HLD from I/O-station.

Description	A BF could not be connected to the High Level Driver of the PLC I/O-interface.		
Response	Class	3	Command is aborted.
Solution	Class	7	Restart the system.
Error type	-		

ID 270093

Index of PLC I/O station exceeds limits.			
Description			
Response	Class	3	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	-		

ID 270094

Index of PLC I/O slots exceeds limits.			
Description			
Response	Class	3	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	-		

ID 270095

Index of PLC I/O data exceeds limits.			
Description			
Response	Class	3	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Lower limit value [-]	
	%3:	Upper limit value [-]	
Error type	-		

ID 270096

Trace of IPO data in zero channel configuration not possible.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 270097

The list contains an unknown element.			
Description	During interpretation of the mapping list of the IO devices, an unknown list element is detected.		
Response	Class	1	
Solution	Class	1	Remove or modify the unknown list element in the corresponding list.
Error type	-		

ID 270098 - 270102

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270103

Job list full. PLC job can not be entered.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 270104

Timeout: Message dismissed by HLD.			
Description	When using the #MSG command, a message will be dismissed. This is caused by too many messages.		
Response	Class	1	Warning
Solution	Class	1	Reduce the number of messages or modify the #MSG[..] command to #MSG SYN_ACK[...] in the NC program. [PROG// Section: Programming a message]
Error type	-		

ID 270105 / 270106

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 270109

Initialisation of channel can not be finished within expected time.

Description			
Response	Class	3	
Solution	Class	1	
Parameter	%1:	Limit value [-]	
	%2:	State [-]	
	%3:	State [-]	
Error type	-		

ID 270110

Unknown block type in EXT_TO_IPO interface.			
Description	When using the external command interface of the spindle/a SAI, an unknown command type was used. For external commanding of a spindle, see [HLI// Section: External spindle command].		
Response	Class	1	Warning output, command is discarded.
Solution	Class	1	Change PLC program:
Parameter	%1:	Logical axis number [-]	
		Achsnummer	
	%2:	Error value [-]	
		Unknown type	
Error type	-		

ID 270111

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

2.16.7 ID-range 280000-280249

2.16.8 ID-range 280250-280499

ID 280304

DataFactory stopped reading job manager response data.			
Description	The termination response of a GCM job cannot be sent to the DataFactory. This and all following responses are lost until DataFactory get enabled.		
Response	Class	-	
Solution	Class	-	Information to the machine manufacturer
Error type	-		

2.16.9 ID-range 280500-280749

ID 280600

No free space for new job/part.			
Description	Internal GCM error: The administration of jobs or #START commands is full. No new job can be stored.		
Response	Class	-	Please inform the controller producer.
Solution	Class	-	To clear up all stuck jobs, please execute the PLCopen P4 FB MCV_GrpResetForced from the PLC in all GCM master channels.
Error type	-		

ID 280601

Undefined internal state.			
Description	Internal Error: Unknown state while executing #START command.		
Response	Class	-	Please inform the controller producer.
Solution	Class	-	
Error type	-		

ID 280602

Job/Part ID unknown.			
Description	Internal Error: When decoding the current NC command, no active workpiece/job ID found. The actual CNC program is aborted.		
Response	Class	-	Please inform the controller producer.
Solution	Class	-	
Error type	-		

ID 280603

Response for non commanded step.			
Description	The PLC sent an acknowledgement for which no job can be found. This error occurs when external (slave) software communicates incorrectly with the GCM. Basically, the GCM sends a request containing GCM-specific data to the slave. The data must be returned in the response to determine the associated master job. If this data is missing, the message is rejected and the error message is generated. If the message does not belong to a pending master job, it may be that it is not terminated.		
Response	Class	0	
Solution	Class	0	

Parameter	%1:	Current value [-]
		Refereed Job ID of the response.
	%2:	Current value [-]
		ID of the commanded decoder.
	%3:	Current value [-]
		Logical number of responding PLC function.
Error type	-	

ID 280604

Wrong decoder ID.			
Description	A acknowledgement was sent from the PLC. However, the designated decoder is invalid. Since the message belongs to a pending master job, it is probably not terminated. The program must be aborted.		
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Job ID of the response	
	%2:	Current value [-]	
		ID of the commanded decoder	
	%3:	Current value [-]	
		Log. Number of the PLC function sending the acknowledgement.	
Error type	-		

ID 280605

WAIT information: the requested step is unknown or already terminated.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280606

Job manager initialisation error.			
Description	System error: GCM internal FIFOs could not be initialised.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280607

Initialization error while creating new step list.			
Description			
Response	Class	-	
Solution	Class	-	

Error type	-

ID 280608

Responded job/part ID unknown.			
Description	A acknowledgement was sent from the PLC. However, the job ID is invalid. Since the message belongs to a pending master job, it is probably not terminated. The program must be aborted.		
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Job ID of the response	
	%2:	Current value [-]	
		ID of the commanded decoder	
	%3:	Current value [-]	
		Log. Number of the PLC function sending the acknowledgement.	
Error type	-		

ID 280609

LOCK inside already active LOCK block.			
Description	The #LOCK command was commanded multiple times. #LOCK is already enabled. Correct the GCM program.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280610

Current no active LOCK block but UNLOCK called.			
Description	The #UNLOCK command was commanded multiple times. #LOCK already inactive. Correct the GCM program.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280611

Current decoder / channel not configured for job manager commands.			
Description	GCM commands cannot be processed since the channel is not configured as a GCM master. Blocked commands are: #LOCK, #UNLOCK, #START, #WAIT CMD.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280612

Wrong configuration: Master and Slave ID are identical			
Description	A GCM master channel of a GCM group is also configured as GCM slave. That is impossible. gcm.group[x].master[x].channel_id and gcm.group[x].CNC_slave[x].channel_id cannot reference to the same channel.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		Master channel number master[x]	
	%3:	Current value [-]	
		Slave channel number CNC_slave[y]	
Error type	-		

ID 280613

LOCK block not closed.			
Description	At the end of a GCM program, a #LOCK block is still opened. There is a missing #UNLOCK. Correct the GCM program.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280614

Job/Part ID already removed by reset.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280615

Job/Part ID already used.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280616

Job/Part ID not terminated.	
Description	The program detected an already open job/program after start of new job/program. The job is terminated and an "abort" message sent to the PLC interface.

Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280700

No master in job manager configuration.			
Description	Configuration error: At least a gcm.group[x].master[x].channel_id declaration is missing. Example: CCM group 0, first master – channel 1 gcm.group[0].master[0].log_id 4711 # log. ID of masters -> V.G.IP_NR gcm.group[0].master[0].channel_id 1 # Channel 1 is master		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	-		

ID 280701

No slave in job manager configuration.			
Description	Configuration error: At least one gcm.group[x].cnc_slave[x].channel_id or gcm.group[x].plc_slave[x] specification is missing. Example: CCM group 0, first slave – channel 2 gcm.group[0].cnc_slave[0].log_id 1 # #START[CNC CHANNEL = <1> ...] gcm.group[0].cnc_slave[0].channel_id 2 # Channel 2 is log. slave "1"		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	-		

ID 280702

'Index' gaps in the configuration list for job manager decoders.			
Description	Configuration error: In a given GCM group the "master[x]" index list has gaps. Set the index/[x] values in a continuous order.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
Error type	-		

ID 280703

'Index' gaps in the configuration list for CNC slaves.			
Description	Configuration error: In a given GCM group "CNC_slave[x]" index list has gaps. Set the [x] values in a continuous order.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
Error type	-		

ID 280704

'Index' gaps in the configuration list for PLC slaves.			
Description	Configuration error: In a given GCM group the "PLC_slave[x]" index list has gaps. Set the [x] values in a continuous order.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
Error type	-		

ID 280706

Missing 'coding' entry in PLC slave list.			
Description	Configuration error: In a GCM group the "PLC_slave[x]" setting has no "coding" entry. Example: plc_slave[1].coding NR #Encoded with the #syntax specified "gcm.coding[NR].list".		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		"PLC_slave" index	
Error type	-		

ID 280707

Configured master interface unknown.			
Description	Configuration error: A "master[x]" referenced channel does not exist.		
Response	Class	3	
Solution	Class	6	

Parameter	%1:	Current value [-]
		GCM group
	%2:	Current value [-]
		Master index[x]
Error type	-	

ID 280708

Configured CNC slave interface unknown.			
Description	Configuration error: The "CNC_slave[x]" referenced channel does not exist.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		Slave index: cnc_slave[x]	
Error type	-		

ID 280709

Size of master interfaces exceeded.			
Description	Configuration error: There are too many "master[x]" index configuration defaults defined. The maximum value is lower than usable CNC channels.		
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Count of master entries	
	%2:	Current value [-]	
		Count of usable channels	
Error type	-		

ID 280710

Size of slave interfaces exceeded.			
Description	Configuration error: There are too many "cnc_slave[x]" configuration defaults. The maximum value is lower than usable CNC channels.		
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Count of configuration entries	
	%2:	Current value [-]	
		Count of usable channels	
Error type	-		

ID 280711

Corrupted job stack.

Description	Internal Error. Current read job stack is corrupt. Abort!		
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Error of interface type: 1: Shortcut interface, job start 2: Shortcut Interface, Job stop 3: PLCopen channel interface 4: CTM interface	
	%2:	Current value [-]	
		Interface number	
Error type	-		

ID 280712

Job stack lower than expected.			
Description	Internal Error. Current job stack is corrupted. Abort		
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Interface type affected by error: 1: Shortcut interface, job start 2: Shortcut Interface, Job stop 3: PLCopen channel interface 4: CTM Interface	
	%2:	Current value [-]	
		Interface number	
	%3:	Current value [-]	
		Stack size	
	%4:	Current value [-]	
		Expected size	
Error type			

ID 280713

Job stack full. No space for new commanded job..			
Description	Internal Error. Current job stack is corrupted. Abort		
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
		Interface number	
	%2:	Current value [-]	
		Expected stack size	
	%3:	Current value [-]	
		Actual stack size	
Error type	-		

ID 280714

#START command inconsistent for CNC/PLC type and coding configuration			
Description	The coding oft the #START parameters do no match to the data structures of the targets.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280714

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	2	
Solution	Class	8	Requires restart of controller-

ID 280715

Command #START inconsistency to CNC/PLC type and coding configuration.			
Description	The coding of the #START parameters do no match to the data structures of the targets.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 280716

Configured PLC slave interface unknown.			
Description	Configuration error: The user index specified in plc_slave[x].hli_index is unknown.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		Slave index: plc_slave[x]	
Error type	-		

ID 280717

Configured Master interface multiple used.			
Description	Configuration error: A master channel cannot be used more than once as a source.		
Response	Class	3	
Solution	Class	6	

Parameter	%1:	Current value [-]
		GCM group
	%2:	Current value [-]
		Master index 1: master[x]
	%3:	Current value [-]
		Master index 2: master[y]
Error type	-	

ID 280718

Configured CNC slave interface multiple used.			
Description	Configuration error: A channel can be used as slave only once.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		Slave index 1: cnc_slave[x]	
	%3:	Current value [-]	
		Slave index 2: cnc_slave[y]	
Error type	-		

ID 280719

Configured PLC slave interface multiple used.			
Description	Configuration error: A PLC channel can be used as slave only once.		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		Slave index 1: cnc_slave[x]	
	%3:	Current value [-]	
		Slave index 2: cnc_slave[y]	
Error type	-		

ID 280720

Configured interface not available.			
Description	Internal Error: The expected interface is empty (NULL)		
Response	Class	-	
Solution	Class	-	

Parameter	%1:	Current value [-]
		Interface type: 10: Master response 11: Master abort 12: Master command block 20: PLC Slave command 21: PLC Slave response 22: PLC Slave abort 30: Channel Slave command 31: Channel Slave response 32: Channel Slave abort
	%2:	Current value [-]
		Interface number
Error type	-	

ID 280721

Configured master already used in another job manager group.			
Description	Configuration error: A master channel can be used only once.		
Response	Class	0	
Solution	Class	0	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		Master index: master[x]	
Error type	-		

ID 280722

Configured slave already used in another job manager group.			
Description	Configuration error: A slave channel can be used only in one group.		
Response	Class	0	
Solution	Class	0	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Current value [-]	
		Slave index: cnc_slave[x]	
Error type	-		

ID 280723

Slave is used as a master in another job manager group.			
Description	Configuration error: The slave configured in the GCM group is configured in a different group to the master. Cascaded GCM (groups) are not permissible.		
Response	Class	0	
Solution	Class	0	

Parameter	%1:	Current value [-]
		GCM group
	%2:	Current value [-]
		Slave index 1: cnc_slave[x]
Error type	-	

ID 280724

Configured job manager group is inadmissible.			
Description	Configuration error: The used GCM group index is wrong. It is too big or zero (nil).		
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
		GCM group	
	%2:	Upper limit value [-]	
		Upper limit value for group[x]	
Error type	-		

2.16.10 ID-range 292000-292249

ID 292000

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 292001

Kinematic transformation: Division by 0.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with wrong or incomplete parameters has lead to a calculation error. If machine is not referenced this error also can occur because of wrong or invalid axis positions. For more information on kinematic transformation see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA// Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check axis positions, axis configuration and parameters of kinematic transformation.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (for diagnosis by specialist only)	
Error type	-		

ID 292002

Kinematic transformation: Negative root.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with wrong or incomplete parameters has lead to a calculation error. If machine is not homed, this error also can occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check axis positions, axis configuration and parameters of kinematic transformation.

Parameter	%1:	Current value [-]
		ID of incorrectly configured kinematic
	%2:	Current value [-]
		Internal ID of error cause (only for diagnosis by specialists)
Error type	-	

ID 292003

Kinematic transformation: Illegal argument for ASIN.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with wrong or incomplete parameters has lead to a calculation error. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check axis positions, axis configuration and parameters of kinematic transformation.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292004

Kinematic transformation: Illegal argument for ACOS.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with wrong or incomplete parameters has lead to a calculation error. If machine is not homed, this error also can occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check axis positions, axis configuration and parameters of kinematic transformation.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292005

Kinematic transformation: Illegal kinematic ID.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with invalid kinematic identifier. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Use valid kinematic identifier in CNC command [#KIN ID or, if selection is done by tool selection, in machine parameters of tool.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292006

Kinematic transformation: Transformation not available.			

Description	With this kinematic type, an auxiliary function is not supported with active kinematic transformation (#RTCP ON, #TRAFO ON).
	Example: Use of #TOOL ORI CS in tripod kinematics.
	`Wrong:`
	`%trafo`
	`N10 KIN ID[12]`
	`N20 #TRAFO ON`
	`N20 #TOOL ORI CS`
	`N30 G01 G90 X0 Y0 Z100 F2000`
	`N40 M30`
	
	`Correct:`
	`%trafo`
	`N10 KIN ID[1]`
	`N20 #TRAFO ON`
	`N20 #TOOL ORI CS`
	`N30 G01 G90 X0 Y0 Z100 F2000`
	`N40 M30`
	For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		

Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Do not use the utility function with this kinematic type.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292007

Kinematic transformation: Illegal vector.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with wrong or incomplete parameters has lead to a calculation error. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check axis positions, axis configuration and parameters of kinematic transformation.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292008

Kinematic transformation: Illegal angle phi.			
Description	With active kinematic transformation (#RTCP ON, #TRAFO ON), moving axis to invalid positions. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Move in valid axis position range.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292009

Kinematic transformation: Inadmissible number of transformation axes.	
Description	The used CNC version does not support the selected kinematic transformation. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].

Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Contact CNC manufacturer.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292010

Kinematic transformation: Inadmissible kinematic parameter.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with wrong or incomplete parameters has lead to a calculation error. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check and correct parameters of kinematic transformation.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292011

Kinematic transformation: Inadmissible axis position.			
Description	Selection of kinematic transformation (#RTCP ON, #TRAFO ON) with wrong or incomplete parameters has lead to a calculation error. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction].		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check axis positions, axis configuration and parameters of kinematic transformation.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	

Error type	-

ID 292012

Kinematic transformation: Illegal argument for ATAN.			
Description	With active kinematic transformation (#RTCP ON, #TRAFO ON), moving axis to invalid positions or using wrong kinematic parameters. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction]		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Check and correct parameter settings of kinematics, move only to valid axis positions.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292013

Kinematic transformation: Illegal argument for TAN.			
Description	With active kinematic transformation (#RTCP ON, #TRAFO ON), moving axis to invalid positions. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction]		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Move in valid axis position range.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292014

Kinematic transformation: Invalid WCS orientation.

Description	With active kinematic transformation (#RTCP ON, #TRAFO ON), moving to invalid orientations. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction]		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Move in valid axis position range.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292015

Kinematic transformation: Iteration counter overflow.			
Description	Iterative transformation algorithm with active kinematic transformation (#RTCP ON, #TRAFO ON) does not provide a valid result. Can occur, for example, if moving to a singular axis position was done. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction]		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Move in valid axis position range.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292016

Kinematic transformation: Singular matrix.	
Description	Iterative transformation algorithm with active kinematic transformation (#RTCP ON, #TRAFO ON) does not provide a valid result. Can occur, for example, if moving to a singular axis position was done. If the machine is not homed, this error may also occur because of wrong or invalid axis positions. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA//Introduction]

Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Move in valid axis position range.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292019

Requested kinematic transformation has not been configured.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292020 / 292021

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 292022

Interface to kinematic transformation has not been configured.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292023

Backward transformation after forward transformation results in different position.		
Description		
Response	Class	-
Solution	Class	-
Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Current value [-]
Error type	-	

ID 292024

Directory of the TcCOM CNC interface objects does not exist.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292025

ISGCtrl not yet initialised.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292026

Specified kinematic ID is not configured.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292027

Specified kinematic-transformation could not saved internal.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292030

Error by request of configuration data of kinematic transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292031

Error by initialisation of kinematic transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292032

Error in the kinematic forward transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292033

Error in the kinematic backward transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292034

Current MCS input position of the kinematic forward transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292035

Current PCS output position of the kinematic forward transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292036

Current PCS input position of the kinematic backward transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292037

Current MCS output position of the kinematic forward transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292038

Requested tool radius compensation is not configured.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292039

ID of tool radius compensation is not configured.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292040

Kinematic transformation: Invalid axis configuration programmed.			
Description	When kinematic transformations (#RTCP ON, #TRAFO ON) are active, an impermissible axis configuration was parameterised. For more information on kinematic transformation, see [PROG// Section: Rotation Tool Centre Point (RTCP)] [KITRA// Kinematic transformations].		
Response	Class	-	Depending on module where error occurs - Program abort - Axes motion stop
Solution	Class	-	Check and correct parametrisation for kinematic transformation.
Parameter	%1:	Current value [-]	
		Number of the selected kinematic	
	%2:	Error value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292041

Kinematic transformation: Too many additional inputs.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292042

System error [▶ 12]	
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.

Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 292043

Kinematic transformation: Invalid axis pointer.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292044

CNC interface version is too old. Incompatible with the current TcCOM object.			
Description	The interface version of of the CNC and the TcCOM object are not compatible.		
Response	Class	-	The TcCOM object cannot be loaded.
Solution	Class	-	Update of CNC version.
Parameter	%1:	Current value [-]	
		Transformation ID used.	
	%2:	Current value [-]	
		Major version number of the TcCOM object	
	%3:	Error value [-]	
		Major version number of the CNC interface.	
	%4:	Error value [-]	
		Minor version number of the CNC interface.	
Error type			

ID 292045

Selected orientation mode or rotation sequence unsupported by the transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Incorrect value [-]	
	%2:	Incorrect value [-]	
	%3:	Current value [-]	
Error type	-		

ID 292046

CNC base interface of the transformation not available.			
Description			
Response	Class	-	
Solution	Class	-	

Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Incorrect value [-]
Error type	-	

ID 292047

CNC interface version is too old. Incompatible with the current TcCOM object of online tool radius compensation.			
Description	The version number of the TcCOM object for online tool compensation does not match the version number of the CNC interface. The CNC version is obsolete.		
Response	Class	-	The TcCOM object cannot be loaded.
Solution	Class	-	Update the CNC.
Parameter	%1:	Current value [-]	
		Transformation ID used.	
	%2:	Current value [-]	
		Major version number of the TcCOM object	
	%3:	Error value [-]	
		Major version number of the CNC interface.	
	%4:	Error value [-]	
		Minor version number of the CNC interface.	
Error type			

ID 292048

CNC base interface of online tool radius compensation not available.			
Description			
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Incorrect value [-]	
Error type	-		

ID 292049

CNC interface version is too old. Incompatible with the current TcCOM object of geometric feed adaptation.			
Description	The version number of the TcCOM object for geometric feed adaptation does not match the version number of the CNC interface. The CNC version is obsolete.		
Response	Class	-	The TcCOM object cannot be loaded.
Solution	Class	-	Update the CNC.

Parameter	%1:	Current value [-]
		Transformation ID used.
	%2:	Current value [-]
		Major version number of the TcCOM object for geometric feed adaptation
	%3:	Current value [-]
		Minor version number of the TcCOM object for geometric feed adaptation
	%4:	Error value[-]
		Major version number of the CNC interface.
	%5:	Error value[-]
		Minor version number of the CNC interface.
Error type	-	

ID 292050

CNC basic interface of geometric feed adaptation not available.			
Description			
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Incorrect value [-]	
Error type	-		

ID 292051

CNC interface version is too old. Incompatible with the current TcCOM object of dynamic contour control.			
Description	The version number of the TcCOM object for dynamic contour control does not match the version number of the CNC interface. The CNC version is obsolete.		
Response	Class	-	The TcCOM object cannot be loaded.
Solution	Class	-	Update the CNC version.
Parameter	%1:	Current value [-]	
		Transformation ID used.	
	%2:	Current value [-]	
		Major version number of the TcCOM object for dynamic contour control	
	%3:	Current value [-]	
		Minor version number of the TcCOM object for dynamic contour control	
	%4:	Error value[-]	
		Major version number of the CNC interface.	
	%5:	Error value[-]	
		Minor version number of the CNC interface.	
Error type	-		

ID 292052

CNC basic interface of dynamic contour control not available.

Description			
Response	Class	-	
Solution	Class	-	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
	%3:	Incorrect value [-]	
Error type	-		

ID 292053

Kinematic transformation: Singular position.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292054

Kinematic transformation: Boundary singularity.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292055

Kinematic transformation: Elbow singularity.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292056

Kinematic transformation: Shoulder singularity.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292057

Kinematic transformation: Wrist singularity.
Description

Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292058

General error in external kinematic transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292059

User-specific error in external kinematic transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 292060

Couple kinematics: Unsupported member kinematic.			
Description	A configured member kinematic is not supported by the couple kinematics.		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Configuration of supported member kinematics. A list of member kinematics is in the supplied documentation
Parameter	%1:	Current value [-]	
		Kinematic ID of unsupported kinematic	
	%2:	Error value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292061

Couple kinematics: TCP cannot be reached.

Description	The programmed TCP of the couple kinematics cannot be reached. This may be because of missing degrees of freedom or the turning axis of a member kinematic is programmed separately and is in conflict with the programmed TCP. Example: 1. A configured couple kinematic consists of linear axes. However, a rotation axis of the TCP is programmed in the NC program. The TCP with the configured member kinematics cannot be reached as the rotary degree of freedom is missing. 2. Couple kinematic consisting of a robot and the same rotation axis (e.g. C axis) is programmed with different values for the robot and the TCP (e.g. C_TCP=45 C_ROB=10).		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Add an additional kinematic which supplies the required degree of freedom or change the NC program so that the programming of member kinematics and TCP are not mutually excluded.
Parameter	%1:	Current value [-]	
		Kinematics ID of the couple kinematic	
	%2:	Error value [-]	
		Internal ID of error cause (for diagnosis by specialist only)	
Error type	-		

ID 292062

Couple kinematics: Too many member kinematics configured.			
Description	Too many member kinematics are configured for a couple kinematic. The number of member kinematics results from the sum of member kinematics per group.		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Reduce the number of member kinematics configured. The maximum number of partial kinematics is contained in the documentation (FCT-C35 Couple kinematics).
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Error value [-]	
		Internal ID of error cause (for diagnosis by specialist only)	
Error type	-		

ID 292063

Couple kinematics: Group ID is configured multiple times.			
Description	Several groups in a couple kinematic are configured with the same ID (P-CHAN-00447).		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Change the double ID so that all group IDs are unique within a couple kinematic.

Parameter	%1:	Current value [-]
		ID of incorrectly configured kinematic
	%2:	Error value [-]
		Internal ID of error cause (only for diagnosis by specialists)
Error type	-	

ID 292064

Couple kinematics: ID of a member kinematics cannot be found in the channel parameter list.			
Description	In a couple kinematic, a member kinematic in the kinematic chain is configured with an ID that cannot be found in the channel parameter list.		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Change the ID in the kinematic chain of the couple kinematic or change the ID of the corresponding kinematic so that they both match.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Error value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292065

Couple kinematics: ID of a member kinematic in the priority list cannot be found in the kinematic chain.			
Description	An element in the priority list (P-CHAN-00450) of a couple kinematic cannot be found in the associated kinematic chain (P-CHAN-00449).		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Correct the invalid ID in the priority list into an ID that exists in the kinematic chain.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Error value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292066

Couple kinematics: The group holding the workpiece cannot be found			
Description	A tool CS (P-CHAN-00448) is configured for a group of a couple kinematic. Its name does not match any existing group names (P-CHAN-00447) for a couple kinematic.		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.

Solution	Class	-	Correct the name of the tool CS group so that is matches an existing group name.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (for diagnosis by specialist only)	
Error type	-		

ID 292067

Couple kinematics: Element in kinematic chain has no entry in the move priority list.			
Description	An element in the kinematic chain (P-CHAN-00449) of a couple kinematic cannot be found in the associated priority list (P-CHAN-00450).		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Entry in the priority list for the missing element.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (for diagnosis by specialist only)	
Error type	-		

ID 292068

Couple kinematics: Element of the move priority list is used multiple times.			
Description	Entries in the priority list (P-CHAN-00450) may not be used multiple times in a couple kinematic.		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Remove or rename the incorrect entry.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic	
	%2:	Current value [-]	
		Internal ID of error cause (for diagnosis by specialist only)	
Error type	-		

ID 292069

Couple kinematics: Element of the kinematic chain is used multiple times.			
Description	Entries in the kinematic chain (P-CHAN-00449) may not be used multiple times in a couple kinematic.		
Response	Class	-	Depending on the module where error occurs: • Abort NC program processing. • Movement stop.
Solution	Class	-	Remove or rename the incorrect entry.

Parameter	%1:	Current value [-]
		ID of incorrectly configured kinematic
	%2:	Current value [-]
		Internal ID of error cause (for diagnosis by specialist only)
Error type	-	

ID 292070

CNC interface version is too old. Incompatible with current TcCOM object of external compensation.			
Description	The interface version of the CNC is compared with that of the TcCOM object of the external compensation. The system detected that the interface version of the CNC is too old.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Update the CNC to obtain the compatibility of the interface.
Parameter	%1:	Current value [-]	
		Error number	
	%2:	Current value [-]	
		Major version number of the TcCOM object interface	
	%3:	Current value [-]	
		Minor version number of the TcCOM object interface	
	%4:	Error value [-]	
		Major version number of the CNC interface.	
	%5:	Error value [-]	
		Minor version number of the CNC interface.	
Error type	11, Error message from position controller.		

ID 292071

CNC base interface of external compensation is not available.			
Description	The interface version of the TcCOM object of the external compensation cannot be determined.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Please contact the controller manufacturer.
Parameter	%1:	Current value [-]	
		Error number	
	%2:	Current value [-]	
		Compensation index	
	%3:	Error value [-]	
Error type	11, Error message from position controller.		

ID 292072

Kinematic transformation: Invalid configuration of lateral offset.			
Description			
Response	Class	2	

Solution	Class	3	
Parameter	%1:		
	%2:		
Error type	-		

ID 292073

Exception during request of configuration data of kinematic transformation.			
Description	When a TcCOM object was used for a user-defined transformation, an exception occurred in the GetDimensions() function, request for configuration data.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify parameters in the GetDimensions() function.
Error type	-		

ID 292074

Exception during initialisation of kinematic transformation.			
Description	When a TcCOM object was used for a user-defined transformation, an exception occurred in the TrafoSupported() function.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify parameters in the TrafoSupported() function.
Error type	-		

ID 292075

Exception in the kinematic forward transformation.			
Description	When a TcCOM object was used for a user-defined transformation, an exception occurred in the Forward() function, calculation of the kinematic forward transformation.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify parameters in the Forward() function.
Error type	-		

ID 292076

Exception in the kinematic backward transformation.			
Description	When a TcCOM object was used for a user-defined transformation, an exception occurred in the Backward() function, calculation of the kinematic backward transformation.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify parameters in the Backward() function.
Error type	-		

ID 292080

Wrist position in non-reachable area (cylinder around C1 axis).			
Description	If the parameter HD32 of the robot kinematic KIN_TYP_45 is unequal to zero, this results in a cylindrical area about the C1 rotary axis that cannot be physically reached by the robot wrist (intersection A1-B3-A2 rotary axis). The programmed position and orientation result in a wrist position that is within the unreachable range.		
Response	Class	2	Abort NC program processing.
Solution	Class	5	Check and modify the NC program, e.g. increase the distance between the workpiece and the C1 rotary axis.
Error type	1, Error message from NC program.		

ID 292081

Position outside of workspace.			
Description	The programmed position and orientation result in a position that is unreachable.		
Response	Class	2	Abort NC program processing.
Solution	Class	5	Check and modify the NC program. The programmed position and orientation must be within the reachable working range of the robot.
Error type	1, Error message from NC program.		

ID 292082

Kinematic transformation: No valid status&turn solution found..			
Description	The programmed values for Status & Turn result in a contradiction. See also PROG//Status & Turn (IS, IT)		
Response	Class	2	Abort NC program processing.
Solution	Class	5	Check and modify the NC program. The values programmed for Status&Turn (IS, IT) must be consistent.
Error type	1, Error message from NC program.		

ID 292083

Status/Pose not valid.			
Description	The kinematic was called with an invalid status/pose. See also PROG://Status & Turn (IS, IT)		
Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the pose in the NC program.
Error type	1, Error message from NC program.		

ID 292084

Non-reachable position with current pose (position too close to robot base).

Description	The robot cannot reach the position or orientation of the current pose. The robot is configured so that there is an unreachable area about the B1 axis. See also PROG//Status & Turn (IS, IT)		
Response	Class	2	Abort NC program processing.
Solution	Class	5	Check and modify the NC program. Select a different pose/a different status (for example, approach via #PTP using Status&Turn). The programmed position and orientation must be within the reachable workspace of the robot.
Error type	1, Error message from NC program.		

ID 292085

Couple kinematics: All groups are configured with group ID 0.			

Description	All groups in a couple kinematic have the ID 0 (P-CHAN-00447).
	
	Incorrect configuration
	`trafo[0].id 1`
	`trafo[0].type 45`
	…
	`trafo[1].id 2`
	`trafo[1].type 91`
	…
	`trafo[2].id 3`
	`trafo[2].type 210`
	`trafo[2].group[0].id 0`
	`trafo[2].group[0].workpiece_cs 200`
	`trafo[2].group[0].chain[0] 1`
	`trafo[2].group[1].id 0`
	`trafo[2].group[1].chain[0] 2`
	
	Corrected entries:
	`trafo[2].group[0].id 100`
	…
	`trafo[2].group[0].id 200`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the couple kinematic. At least the first entry of a group of a couple kinematic must be configured with an ID unequal to 0.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic, based on the above example, ID 3 of trafo[2].id	
	%2:	Error value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292086

Couple kinematics: Gap in group configuration found.

<table>
<tr><td>Description</td><td colspan="3">All groups with an ID unequal to 0 must follow consecutively in the list of all groups.
No gaps are allowed within the group configuration.

Incorrect configuration
<pre>
trafo[0].id   1
trafo[0].type   45
…
trafo[1].id   2
trafo[1].type   91
…
trafo[2].id   3
trafo[2].type   210
trafo[2].group[0].id   100
trafo[2].group[0].chain[0] 1
…
trafo[2].group[1].id   0
trafo[2].group[1].chain[0] 2
…
trafo[2].group[2].id   200
trafo[2].group[2].chain[0] 2
</pre>
Correction:
<pre>
trafo[2].group[1].id   200
trafo[2].group[1].chain[0] 2
…
~~trafo[2].group[2].id   200~~
~~trafo[2].group[2].chain[0] 2~~
</pre>
Unassigned entries of group[x].id are initialised with 0.</td></tr>
<tr><td>Response</td><td>Class</td><td>2</td><td>Abort NC program processing.</td></tr>
<tr><td>Solution</td><td>Class</td><td>3</td><td>Check and modify the configuration of the couple kinematic.
Remove the groups with ID 0 and modify the indices of the groups with configured IDs. All groups with configured IDs must follow each other consecutively.
The first groups in the list (index 0) must already contain an ID unequal to 0.</td></tr>
<tr><td rowspan="4">Parameter</td><td>%1:</td><td colspan="2">Current value [-]</td></tr>
<tr><td></td><td colspan="2">ID of incorrectly configured kinematic,
based on the above example, ID 3 of trafo[2].id</td></tr>
<tr><td>%2:</td><td colspan="2">Error value [-]</td></tr>
<tr><td></td><td colspan="2">Internal ID of error cause (only for diagnosis by specialists)</td></tr>
<tr><td>Error type</td><td colspan="3">-</td></tr>
<tr><td></td><td colspan="3"></td></tr>
</table>

ID 292087

Couple kinematics: Group has no member kinematics.

Description	A group is configured but contains no member kinematics in the kinematic chain (P-CHAN-00449).
	Incorrect configuration
	`trafo[0].id 1`
	`trafo[0].type 45`
	…
	`trafo[1].id 2`
	`trafo[1].type 91`
	…
	`trafo[2].id 3`
	`trafo[2].type 210`
	`trafo[2].group[0].id 100`
	`trafo[2].group[0].workpiece_cs 200`
	`trafo[2].group[0].chain[0] 0`
	…
	`trafo[2].group[0].chain[1] 0`
	…
	Corrected entries:
	`trafo[2].group[0].chain[0] 1`
	`trafo[2].group[0].chain[1] 2`		

Response	Class	2	Abort NC program processing.
Solution	Class	3	Check and modify the configuration of the couple kinematic. At least one group of a couple kinematic must be configured with an ID unequal to 0.
Parameter	%1:	Current value [-]	
		ID of incorrectly configured kinematic, based on the above example, ID 3 of trafo[2].id	
	%2:	Error value [-]	
		Internal ID of error cause (only for diagnosis by specialists)	
Error type	-		

ID 292088

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 292089

Exception in WCS to matrix transformation.			
Description	When a TcCOM object was used for a user-defined transformation with orientations, an exception occurred in the McsToPosOri() function and the calculation of MCS coordinates to position and orientation.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify the McsToPosOri() function.
Error type	-		

ID 292090

Exception in matrix to WCS transformation.

Description	When a TcCOM object was used for a user-defined transformation with orientations, an exception occurred in the PosOriToMcs() function and the calculation of position and orientation to MCS coordinates.		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and modify the PosOriToMcs() function.
Error type	-		

ID 292091

Error in WCS to matrix transformation.			
Description	An error occurred in the TcCom transformation function PosOriToMcs().		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and eliminate the error in the PosOriToMcs() function.
Parameter	%1:	Current value [-]	
		Kinematic ID where the error occurred.	
	%2:	Current value [-]	
		ret_value1; user-defined.	
	%3:	Current value [-]	
		ret_value2; user-defined.	
	%4:	Current value [-]	
		ret_text[24]; user-defined.	
	%5:	Current value [-]	
		Function where the error occurred.	
Error type	-		

ID 292092

Error in matrix to WCS transformation.			
Description	An error occurred in the TcCom transformation function McsToPosOri().		
Response	Class	4	Controlled stop for the axis group.´
Solution	Class	6	Check and eliminate the error in McsToPosOri() function.
Parameter	%1:	Current value [-]	
		Kinematic ID where the error occurred.	
	%2:	Current value [-]	
		ret_value1; user-defined.	
	%3:	Current value [-]	
		ret_value2; user-defined.	
	%4:	Current value [-]	
		ret_text[24]; user-defined.	
	%5:	Current value [-]	
		Function where the error occurred.	
Error type	-		

2.16.11 ID-range 293000-293249

ID 293000

HW device type for counter interface unknown.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:		
	%2:	Current value [-]	
	%3:	Limit value [-]	
Error type	-		

ID 293001

HW device type for D/A interface unknown.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:		
	%2:	Current value [-]	
	%3:	Limit value [-]	
Error type	-		

ID 293002

HW device type for cam interface unknown.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:		
	%2:	Current value [-]	
	%3:	Limit value [-]	
Error type	-		

ID 293003

HW device type for counter interface not in permissible range.			
Description			
Response	Class	3	
Solution	Class	7	

Parameter	%1:	
	%2:	Current value [-]
	%3:	Limit value [-]
Error type	-	

ID 293004

HW device type for D/A interface not in permissible range.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:		
	%2:	Current value [-]	
	%3:	Limit value [-]	
Error type	-		

ID 293101

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	6	
Solution	Class	8	Requires restart of controller-

ID 293102

The directory of the COM CNC interface object does not exist.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 293103

ISGCtrl not yet initialised.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 293104

Specified ID of COM interface is not configured.			
Description			
Response	Class	-	
Solution	Class	-	

Error type	-

ID 293105

Specified COM interface cannot be stored internally.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 293109

Specified COM interface for dynamic contour control is not available in current CNC.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 293110

Specified COM interface for geometric feed adaptation is not available in current CNC.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 293111

Specified COM interface for online TRC is not available in current CNC.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

2.16.12 ID-range 294000-294249

ID 294000

Drive error No. Trip.			
Description	The KSD of the concerned axis reports a common internal error state. Watch the next errors for detailed information. If this message appears without follow-up messages, the error appeared in the KSD for such a short time that the DSE was unable to read the error number in the KSD. In this case, the error memory in the affected KSD must be read out. To do this, open the DSE/RDW tool and export the code points of the affected KSD to a logfile. The code points 162, 163 and 164 supply detailed information on error history. No. 161 reports the actual error state of the KSD.		
Response	Class	7	Contour-accurate stop and lock all active commands.
Solution	Class	6	Contact KUKA service team.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		DSE drive address.	
	%3:	Current value [-]	
		DSE error id.	
Error type	11, Error message from position controller.		

ID 294001

Encoder cable failure.			
Description	Possible causes: • Encoder cable defective. • Loose connection. • Supply voltage missing. • Encoder defective.		
Response	Class	7	• Dynamic braking. • Output “$ALARM_STOP” has signal level 0. • All active commands inhibited.
Solution	Class	6	• Check encoder cable. • Check connections. • Check supply voltage. • Check encoder and exchange if necessary.
Error type	11, Error message from position controller.		

ID 294002

Regulator limit exceeded.			
Description	The position lag is too large. The axis does not follow the command value.		
Response	Class	7	• Maximum braking. • Active commands inhibited. • “$ALARM_STOP” output has signal level 0.

Solution	Class	6	• Check the machine data $IN_POS_MA (axis positioning window) and $FOL_ERR_MA (factor for tracking error monitoring). • Check manipulator and motor. • Check mechanics and motor.
Error type	11, Error message from position controller.		

ID 294003

Motor blocked.			
Description	The command torque value supplied by the speed controller exceeds the max. permissible command torque value by more than 30%.		
Response	Class	7	Speed stop and locking of all active commands.
Solution	Class	6	Rectify cause of torque overload (reduce velocity, acceleration).
Error type	11, Error message from position controller.		

ID 294004

Collision monitoring.			
Description	The motor torque is not located in the specified monitoring range.		
Response	Class	7	Contour-accurate stop and lock all active commands.
Solution	Class	6	• Verify load data. • Increase size of range if required.
Error type	11, Error message from position controller.		

ID 294005

Watchdog interpolation cycle.			
Description	DSE did not get a new position value from kernel system for this axis.		
Response	Class	7	• Dynamic braking. • All active commands inhibited. • Output “ALARM_STOP” has signal level 0.
Solution	Class	6	• Check plug connection between DSE and MFC as well as between MFC and motherboard for proper contact. • If this does not work, exchange DSE or MFC.
Error type	11, Error message from position controller.		

ID 294006

Overcurrent.			
Description	Intermediate circuit current monitoring safeguard triggered by overcurrent.		
Response	Class	7	Speed stop and lock all active commands.
Solution	Class	6	• Rectify error. • Reset the error by pressing the hardware acknowledgement button “ACK” on the DSE I module.

Error type	11, Error message from position controller.

ID 294007

Brake fault.			
Description	Brake cable monitoring device has signalled short-circuit, overloading or break in connection.		
Response	Class	7	Contour-accurate stop and stop program processing.
Solution	Class	6	• Rectify error. • Reset the error by pressing the hardware acknowledgement button "ACK" on the DSE I module.
Error type	11, Error message from position controller.		

ID 294008

Error Sync input drive module.			
Description	Synchronization error in the DSE drive module.		
Response	Class	7	Trajectory control stop. All active commands inhibited.
Solution	Class	6	Eliminate error. Reset the fault on the DSE-module.
Parameter	%1:		
	Logical axis number P-AXIS-00016 of concerned axis.		
	%2:		
	DSE drive address.		
	%3:		
	DSE error id.		
Error type	11, Error message from position controller.		

ID 294009

Synchronisation error drive.			
Description	An synchronization error occurred within the drive module.		
Response	Class	7	Speed stop and locking of all active commands.
Solution	Class	6	Eliminate synchronization error and perform NC reset.
Error type	11, Error message from position controller.		

ID 294010

Monitoring of the actual velocity.			
Description	Possible causes: • Fault in control loop: • Motor cables of two axes interchanged. • Error in the axis assignment. • Power module defective.		
Response	Class	7	• Speed stop and locking of all active commands.

Solution	Class	6	Possible remedial measures: • Check motor cables. • Eliminate fault in the control loop / error in the axis assignment. • Check power module and exchange if necessary. • In the case of an misaligned resolver, readjust the robot.
Error type	11, Error message from position controller.		

ID 294011

Servo bus fault after.			
Description	Possible causes: • Defective bus cable. • Defective bus module. • Defective bus driver.		
Response	Class	7	Maximum braking. All active commands inhibited.
Solution	Class	6	Change the element.
Error type	11, Error message from position controller.		

ID 294012

Motor cable.			
Description	The DSE-drive reports heavy load on the motor cable.		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Check the motor. Eliminate error. Reset the fault on the DSE-module.
Parameter	%1:		
	Logical axis number P-AXIS-00016 of concerned axis.		
	%2:		
	DSE drive address.		
	%3:		
	DSE error id.		
Error type	11, Error message from position controller.		

ID 294013

Motor cable.			
Description	The DSE-drive reports a short circuit in the motor cable.		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Check the motor cable. Eliminate error. Reset the fault on the DSE-module.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	
		DSE drive address.
	%3:	
		DSE error id.
Error type	11, Error message from position controller.	

ID 294014

Overcurrent.			
Description	The DSE drive reports an overcurrent.		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Check the motor. Eliminate error. Reset the error on the DSE module.
Parameter	%1:		
	Logical axis number P-AXIS-00016 of axis affected.		
	%2:		
	DSE drive address.		
	%3:		
	DSE error id.		
Error type	11, Error message from position controller.		

ID 294015

Failure of motor phase.			
Description	Power module or intermediate circuit voltage discharged.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Switch on drives. Check power module and DSE.
Error type	11, Error message from position controller.		

ID 294016

Heat sink temperature.			
Description	Thermostatic switch on the heat sinks of the servo output stage signals that the temperature of the output stage transistors is too high.		
Response	Class	7	Contour-accurate stop. All active commands inhibited.
Solution	Class	6	Rectify cause of the overheating (e.g. reduce the load).
Error type	11, Error message from position controller.		

ID 294017

Drive error No. 71.			
Description	The DSE drive reports error No.71.		
Response	Class	7	Stop the axis. All active commands inhibited.

Solution	Class	6	Contact the service personnel. Eliminate error. Reset the error on the DSE module.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		DSE drive address.	
	%3:		
		DSE error id.	
Error type	11, Error message from position controller.		

ID 294018

Wrong drive parameter PR1.			
Description	The DSE-drive reports parameter error No.1.		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the error on the DSE module.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		DSE drive address.	
	%3:		
		DSE error id.	
Error type	11, Error message from position controller.		

ID 294020

Drive error No. 79.			
Description	The DSE drive reports error No. 79 (communication error EEPROM control).		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the error on the DSE module.
Parameter	%1:		
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:		
		DSE drive address.	
	%3:		
		DSE error id.	
Error type	11, Error message from position controller.		

ID 294021

Drive error No. 80.			
Description	The DSE drive reports error No.80 (communication error EEPROM power).		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the error on the DSE module.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	
		DSE drive address.
	%3:	
		DSE error id.
Error type	11, Error message from position controller.	

ID 294022

Drive error No. 91.			
Description	The DSE drive reports error No.91 (external error).		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the error on the DSE module.
Parameter	%1:		
	Logical axis number P-AXIS-00016 of axis affected.		
	%2:		
	DSE drive address.		
	%3:		
	DSE error id.		
Error type	11, Error message from position controller.		

ID 294023

(Old) drive error No. 105.			
Description	The DSE drive reports error No.105.		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the error on the DSE module.
Parameter	%1:		
	Logical axis number P-AXIS-00016 of axis affected.		
	%2:		
	DSE drive address.		
	%3:		
	DSE error id.		
Error type	11, Error message from position controller.		

ID 294024

Drive error No. 106.			
Description	The DSE drive reports error No.106 (check sum error in DSE drive).		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the error on the DSE module.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	
		DSE drive address.
	%3:	
		DSE error id.
Error type	11, Error message from position controller.	

ID 294025

Axis without power.			
Description	Denoted axis is without power.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Check hardware.
Error type	11, Error message from position controller.		

ID 294026

Drives Error No.[error number], see parameter 4.			
Description	The DSE drive reports an error, specified more precisely by error parameter no. 4.		
Response	Class	7	Trajectory control stop. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the fault on the DSE-module.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of concerned axis.
	%2:		
			DSE drive address.
	%3:		
			DSE error id.
	%4:		Current value [-]
			Error Number.
Error type	11, Error message from position controller.		

ID 294027

General drive error, see parameter 4.			
Description	The DSE-drive reports a general error.		
Response	Class	7	Stop the axis. All active commands inhibited.
Solution	Class	6	Contact the service personnel. Eliminate error. Reset the fault on the DSE-module.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	
		DSE drive address.
	%3:	
		DSE error id.
	%4:	Current value [-]
		Error Number.
Error type	11, Error message from position controller.	

ID 294028

Single brake module of drive deactivated.			
Description	The single break module is activated in the axis servo file, although there is no module available in the moment.		
Response	Class	1	Warning output.
Solution	Class	1	Deactivate single break module in the axis servo file.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of concerned axis.
	%2:		
			DSE drive address.
	%3:		
			DSE error id.
Error type	11, Error message from position controller.		

ID 294029

Brake channel closed.			
Description	Brakes have not been opened because not all axis of this brake channel are active.		
Response	Class	1	Example of error messages: • Manipulated variable, or • Motor blocked.
Solution	Class	1	• Control all axes at one brake channel, or • Use individual brake control.
Error type	11, Error message from position controller.		

ID 294030

Slave: wrong amplifier.			
Description	The amplify parameter is wrong in the DSE drive controller.		
Response	Class	7	Contour-accurate stop. All active commands inhibited.
Solution	Class	6	Eliminate error. Reset the error on the DSE module.

Parameter	%1:	
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	
		DSE drive address.
	%3:	
		DSE error id.
Error type	11, Error message from position controller.	

ID 294031

Slave: Deviation warning.			
Description	The position deviation between master and slave drive exceeds the warning limit.		
Response	Class	1	Warning output.
Solution	Class	1	Reduce speed.
Parameter	%1:		
			Logical axis number P-AXIS-00016 of concerned axis.
	%2:		
			DSE drive address.
	%3:		
			DSE error id.
Error type	11, Error message from position controller.		

ID 294032

Slave: Deviation alarm.			
Description	Follow-up message for the status message: "Deviation alarm master-slave <...>".		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Acknowledge message.
Error type	11, Error message from position controller.		

ID 294033

Slave: Critical deviation.			
Description	Follow-up message for the status message: "Deviation critical master-slave <...>".		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Acknowledge message.
Error type	11, Error message from position controller.		

ID 294034

Slave: Speed deviation.			
Description	Follow-up message for the status message: "Speed deviation master-slave <...>".		
Response	Class	7	Motion stop. All active commands inhibited.

Solution	Class	6	Acknowledge message.
Error type	11, Error message from position controller.		

ID 294035

ASR error.			
Description	The system deviation has exceeded a limit value due to excessive slip.		
Response	Class	7	Maximum braking. All active commands inhibited.
Solution	Class	6	Check the hardware.
Error type	11, Error message from position controller.		

ID 294036

KR3 error.		
Description	There is an error in the KR3 module.	
Response	Class	7 Trajectory control stop. All active commands inhibited.
Solution	Class	6 Eliminate error. Reset the fault on the DSE-module.
Parameter	%1:	
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	
		DSE drive address.
	%3:	
		DSE error id.
Error type	11, Error message from position controller.	

ID 294037

DSE watchdog error.		
Description	The maximum permissible calculation time was exceeded.	
Response	Class	7 Trajectory control stop. All active commands inhibited.
Solution	Class	6 Eliminate error. Reset the fault on the DSE-module.
Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Drive type [-]
		Drive type of the axis.
	%3:	NC block number [-]
		Block number in the NC program.
Error type	11, Error message from position controller.	

ID 294038

Unknown DSE-drive error ID.			
Description	The DSE has sent an unknown drive-error ID.		
Response	Class	7	Immediate stop of the axis.
Solution	Class	6	Remedy error case and do nc reset.

Error type	11, Error message from position controller.

ID 294039

Control enable is blocked, this may also be caused by errors of other axes.			
Description	Control enable is blocked, this may be cause by error of other axes.		
Response	Class	7	Abrupt axis stop, feed hold for the other axes.
Solution	Class	6	Rectify blocked control enabling an do nc reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Drive type [-]	
		Drive type of the axis.	
	%3:	Status [-]	
		State of flag ‘stop_drive’	
	%4:	Status [-]	
		Status for flag ‘emergency_stop’.	
Error type	11, Error message from position controller.		

ID 294040

DSE drive was stopped because of an error in another drive.			
Description	The DSE drive was stopped due to an error in another axis.		
Response	Class	7	Abrupt axis stop, feed hold for the other axes.
Solution	Class	6	Remedy error in other axis and do nc reset.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Drive type [-]	
		Drive type of the axis.	
	%3:	NC block number [-]	
		Block number in the NC program.	
Error type	11, Error message from position controller.		

ID 294041

Control enable is blocked, because drive has no power.			
Description	Control enabling is blocked because drive has no power.		
Response	Class	7	Abrupt axis stop, feed hold for the other axes.
Solution	Class	6	Switch on power for drive.
Error type	11, Error message from position controller.		

ID 294042

Control enable is blocked because hardware re-initialisation is running.			
Description	Control enable is currently blocked because hardware re-initialisation is running.		
Response	Class	7	Abrupt axis stop, feedhold for the other axes.
Solution	Class	6	Wait until re-initialisation is finished.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected.
	%2:	Drive type [-]
		Drive type of the axis.
	%3:	NC block number [-]
		Block number in the NC program.
Error type	11, Error message from position controller.	

ID 294043

Fan error. Change module as soon as possible for avoiding breakdown.			
Description	Fan on power converter of a Cobra control is defect.		
Response	Class	1	There is no sufficient cooling of the converter.
Solution	Class	1	Check for free rotatability of fan. Change fan.
Error type	11, Error message from position controller.		

ID 294044

Deviation in absolute position value DSE - RDC.			
Description	The absolute position value, calculated on DSE, differs from the value on RDC for more than 1/4 resolver turn.		
Response	Class	7	Ramp-down braking. All active commands inhibited.
Solution	Class	6	-
Error type	11, Error message from position controller.		

ID 294045

Encoder cable failure external.			
Description	Encoder cable external position input defective. Loose connection. Supply voltage missing. Encoder defective.		
Response	Class	7	Dynamic braking. All active commands inhibited. Output $ALARM_STOP has signal level 0.
Solution	Class	6	Check encoder cable. Check connections. Check supply voltage. Check encoder and exchange if necessary.
Error type	11, Error message from position controller.		

ID 294046

Force controller error.

Description	Force sensor signal does not fit to movement: Actual force changes and position remains constant. Position changes and actual force remains constant.		
Response	Class	1	Dynamic braking. All active commands inhibited. Output “$ALARM_STOP” has signal level 0.
Solution	Class	1	Check force sensor. Check force sensor cable. Check RDC plug in board (KSK). Check gripper bearings and guides.
Error type	11, Error message from position controller.		

ID 294047

Master-slave maximum position exceeded.			
Description	Maximum position of the slave of 8,000,000 increments exceeded.		
Response	Class	1	Calculation errors in actual velocity, following error and deviation.
Solution	Class	1	Change working area of the axis so that the position is always in +-8000000 increments.
Error type	11, Error message from position controller.		

ID 294048

Allowed maximum force exceeded.			
Description	The force probe signal is greater than"FORCE_MAX + UPPER_LIMIT_TOL" after force control "($Force[axis] > FORCE_LIM)" is switched on.		
Response	Class	7	Dynamic braking. All active commands inhibited. Output “$ALARM_STOP” has signal level 0.
Solution	Class	6	Check force sensor value "$Force_act[axis]" using the menu function "Display-Variable-Single" and correct if necessary. Check force sensor and cable. Trace test group ””Force control” to determine if the controller overshoots too much.
Error type	11, Error message from position controller.		

ID 294049

I2T monitoring: Current limit of motor cable exceeds 95 per cent.			
Description	The current load of the motor cable exceeds 95 percent.		
Response	Class	1	Warning output.
Solution	Class	1	Reduce the motor load by suitable measured (reduced load, reduce acceleration, etc.).
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type of the axis.	
	%3:	Current value [0.1%]	
		Current load of motor cable.	

Error type	11, Error message from position controller.

ID 294050

Current limit of the motor cable exceeded 100 per cent (wait about 60s).			
Description	Current load of motor cable exceeds 100 percent.		
Response	Class	7	Guided stop. Active commands inhibited.
Solution	Class	6	Wait approx. 60 s.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Drive type [-]	
		Drive type of the axis.	
	%3:	Current value [0.1%]	
		Current load of motor cable.	
Error type	11, Error message from position controller.		

ID 294051

I2T monitoring: Current load of motor cable dropped below 100 percent.			
Description	The current load of the motor cable has again dropped below 100 percent.		
Response	Class	1	Warning output.
Solution	Class	1	-
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Drive type [-]	
		Drive type of the axis.	
	%3:	Current value [0,1%]	
		Momentary current load of motor cable.	
Error type	11, Error message from position controller.		

ID 294053

Maximum acceleration exceeded.			
Description	The safety controller has detected an axis acceleration that was greater than the configured safety parameter.		
Response	Class	1	The robot stops.
Solution	Class	1	The message has to be acknowledged then the robot can move on.
Error type	11, Error message from position controller.		

ID 294054

Maximum speed exceeded.			
Description	The safety controller has detected an axis speed that was greater than the configured safety parameter.		
Response	Class	1	The robot stops.

Solution	Class	1	The message has to be acknowledged then the robot can move on.
Error type	11, Error message from position controller.		

ID 294055

Brake test failed.			
Description	The brake test checks the torque of the mechanical holding brakes. The torque of the brake of the mentioned axis is under the necessary torque limit.		
Response	Class	1	The robot must not be operated until the braking torque of the holding brake is o.k. again.
Solution	Class	1	The brake or motor must be replaced.
Error type	11, Error message from position controller.		

ID 294056

Gripper moves force controlled under FORCE_LIM.			
Description	The gripper opens further than the position corresponding to FORCE_LIM. This results in a stop to prevent uncontrolled opening of the gripper. Note: These monitors are only enabled during force control.		
Response	Class	7	Dynamic braking. Active commands inhibited. Output “$ALARM_STOP” has signal level 0.
Solution	Class	6	Check force sensor. Check force sensor cable. Check RDC plug in board (KSK). Trace test group 11-Force control to determine if the controller oscillates.
Error type	11, Error message from position controller.		

ID 294057

Increase in force values in force control is missing.			
Description	The force sensor value does not rise when a force-controlled point is approached. Possible causes may be: Force sensor, KSK, cables, RDC,... broken or not right connected. Gun tips don't touch the part.		
Response	Class	7	A stop occurs to prevent damage to the clip and the component. Dynamic braking. Active commands inhibited. Output “$ALARM_STOP” has signal level 0. The force controller is not activated.
Solution	Class	6	Check force sensor. Check force sensor cable. Check RDC plug in board (KSK).
Error type	11, Error message from position controller.		

ID 294058

Measuring range of force sensor exceeded.

Description	The resolution of the force sensor is -1500 to +6500 increments. Are these values exceeded we do not have a correct signal any more. The reason can be a reset of the sensor under pressure or long time drift in the force signal.		
Response	Class	1	The value of the force sensor is wrong.
Solution	Class	1	Check if the force value in increments is in between +- 50 INC with an open gripper. Check if the reset works properly and is not set under pressure.
Error type	11, Error message from position controller.		

ID 294059

Deviation of Resolver shut down position from actual position.			
Description	Resolver shut down position is not identical with the actual position.		
Response	Class	1	Mastering of this axis will be discarded.
Solution	Class	1	Check brakes. Set mastering.
Error type	11, Error message from position controller.		

ID 294060

The motor temperature indicates greater than 155 degrees centigrade.			
Description	The motor temperature of the displayed axis is more than 155°C (signal output by PTC element in the motor winding.)		
Response	Class	7	Contour-accurate stop. All active commands inhibited.
Solution	Class	6	Reduce mechanical load. Rectify any faults.
Error type	11, Error message from position controller.		

ID 294061

Command speed monitoring exceeded.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

2.16.13 ID-range 294500-294749

ID 294500

KPS: Servo bus fault DSEx.			
Description	Servo bus fault.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Check bus cable, bus module, or bus driver. Do reset.
Parameter	%1:	Logical axis number [-]	
		KPS number.	
	%2:	Current value [-]	
		DSE error id.	
	%3:	Current value [-]	
Error type	11, Error message from position controller.		

ID 294501

KPS: Servo bus fault after PMx.			
Description	Servo bus fault after power module.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Check power module. Do reset.
Parameter	%1:	KPS number.	
	%2:	DSE error id.	
Error type	11, Error message from position controller.		

ID 294502

KPS: Servo bus fault after DSEx, no automatic data storage possible.			
Description	Servo bus fault after DSE drive.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Check DSE drive. Do reset.
Parameter	%1:	KPS number.	
	%2:	DSE error id.	
Error type	11, Error message from position controller.		

ID 294503

KPS: Transmission error x. DSE-RDW.			
Description	Damaged cable or connector between DSE and RDW. Cable not connected or connected incorrectly. Data transmission malfunction.		
Response	Class	7	Dynamic braking. All active commands inhibited. Output “$ALARM_STOP” has signal level 0.

Solution	Class	6	Check DSE and RDW power supply (LEDs). Check cable (also screen!) and plug-in connections. Check serial DSE - RDW interface. Check grounding connection of robot and cabinet.
Error type	11, Error message from position controller.		

ID 294504

KPS: Heat sink temperature.			
Description	Overtemperature in the power module.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Reduce power. Do reset.
Parameter	%1:	KPS number.	
	%2:	DSE error id.	
Error type	11, Error message from position controller.		

ID 294505

KPS: Overvoltage PMx.			
Description	Intermediate circuit voltage is too high. Possible causes are: Defective ballast resistor. Ballast fuse.		
Response	Class	7	Maximum braking. All active commands inhibited.
Solution	Class	6	Eliminate error. Reset by pressing the hardware acknowledgement button "ACK" on the front control panel.
Error type	11, Error message from position controller.		

ID 294506

KPS: Under voltage PMx.			
Description	The internal operating voltage (+15V) of the servo output stage is monitored. If the servo output stage is not ready, the intermediate circuit voltage may be too low. This occurs cyclically.		
Response	Class	7	Maximum braking. All active commands inhibited.
Solution	Class	6	Eliminate error. Reset by pressing the hardware acknowledgement button ACK on the DSE I module.
Error type	11, Error message from position controller.		

ID 294507

KPS: Watchdog power module PMx.	
Description	Power module watchdog has dropped out. (Triggered by DSE after delay of 0.125 ms)

Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Check DSE. Check power module.
Error type	11, Error message from position controller.		

ID 294508

KPS: Overtemperature feedback resistor PMx.			
Description	The ballast resistor has got too hot, because the robot is accelerated and braked in the motion program without a pause.		
Response	Class	7	Ramp-down braking. All active commands inhibited.
Solution	Class	6	Reduce acceleration. Introduce wait times.
Error type	11, Error message from position controller.		

ID 294509

KPS: Buffer battery voltage low.			
Description	The voltage of the backup battery is too low.		
Response	Class	1	Next time the controller is switched off, it will no longer be possible to write completely to the hard disk the robot data that needs to be saved.
Solution	Class	1	Exchange the battery.
Error type	11, Error message from position controller.		

ID 294510

KPS: Cabinet temperature too high PMx.			
Description	The cabinet temperature is too high.		
Response	Class	7	Ramp-down braking. All active commands inhibited.
Solution	Class	6	Check the function of the fans in the control cabinet. Check whether the ventilation openings on the cabinet are unimpeded and whether enough air is circulating. Reduce the ambient temperature. If necessary, choose a different installation site.
Error type	11, Error message from position controller.		

ID 294511

KPS: Ballast switch energized for too long PMx.			
Description	The energy of the braked axis exceeded the limit.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Check power module ballast resistor and exchange if necessary. Set braking ramp less steep.
Error type	11, Error message from position controller.		

ID 294512

KPS: Bus voltage charging unit is defective PMx.			
Description	The optocoupler on the KPS is not registering current flow.		
Response	Class	7	Motion stop. All active commands inhibited. Intermediate circuit cannot be loaded.
Solution	Class	6	Exchange affected KPS.
Error type	11, Error message from position controller.		

ID 294513

KPS: K1 contact fail to open.			
Description	K1 contactor is welded.		
Response	Class	7	Ramp-down braking. All active commands inhibited. The energy supply is not switched off.
Solution	Class	6	Exchange contactor, KPS.
Error type	11, Error message from position controller.		

ID 294514

KPS: Drive error PMx No.		
Description	Drive error in power module (for power module number see parameter 3)	
Response	Class	7 Motion stop. All active commands inhibited.
Solution	Class	6 Consult service personnel. Do reset.
Parameter	%1:	
		KPS number.
	%2:	
		DSE error id.
	%3:	Current value [-]
Error type	11, Error message from position controller.	

ID 294523

KPS: Drive bus DSE x, device y unknown.			
Description	Unknown device detected in DSE drive bus.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Check devices in DSE drive bus. Do reset.
Parameter	%1:	KPS number.	
	%2:	DSE error id.	
Error type	11, Error message from position controller.		

ID 294525

DSE error list is full.

Description	The DSE error memory is full, so the next DSE error cannot be saved.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Eliminate error. Do reset.
Error type	11, Error message from position controller.		

ID 294526

KPS: Fan temperature error.			
Description	One of the fans in the cabinet is defective.		
Response	Class	7	Ramp-down braking. All active commands inhibited.
Solution	Class	6	Replace defective fan.
Error type	11, Error message from position controller.		

ID 294527

KPS: KR3 power module error.		
Description	There is an error in the KR3 power module.	
Response	Class 7	Motion stop. All active commands inhibited.
Solution	Class 6	Check power modules. Do reset.
Parameter	%1:	KPS number.
	%2:	DSE error id.
Error type	11, Error message from position controller.	

ID 294528

DSE homing error.		
Description	There was an error in the homing sequence of a DSE drive.	
Response	Class 7	Motion stop. All active commands inhibited.
Solution	Class 6	Eliminate error. Do reset.
Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of concerned axis.
	%2:	Drive type [-]
		Drive type of the axis.
	%3:	NC block number [-]
		NC block number.
	%4:	State [-]
		Return value of subroutine.
Error type	11, Error message from position controller.	

ID 294529

KPS: Unknown error ID.			
Description	In the KPS there was an error that is not yet known to the system.		
Response	Class	7	Motion stop. All active commands inhibited.
Solution	Class	6	Eliminate error. Do reset.

Parameter	%1:	KPS number.
	%2:	DSE error id.
Error type	11, Error message from position controller.	

ID 294530

Multiple use of same DSE drive address (configuration error).			
Description	The DSE drive address specified in the axis list is already in use. Multiple use of drive addresses is not possible.		
Response	Class	3	Job processing aborted.
Solution	Class	7	Use free drive address.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%2:	Current value [-]	
		Currently used drive address.	
Error type	-		

ID 294533

Too many DSE-drives configured.			
Description	The number of configured DSE drives is greater than the maximum number of DSE drivers.		
Response	Class	3	Abort job processing.
Solution	Class	7	Consider maximum number of DSE drives.
Parameter	%1:	Current value [-]	
		Configured number of DSE drives.	
	%2:	Limit value [-]	
		Maximum number of DSE drives.	
Error type	-		

ID 294534

DSE drive address too big (configuration error).			
Description	DSE drive address is too big.		
Response	Class	3	Abort job processing.
Solution	Class	7	Choose drive address less than maximum available drive address.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of axis affected.	
	%2:	Current value [-]	
		Current value of the drive address.	
	%3:	Limit value [-]	
		Maximum available drive address.	
Error type	-		

ID 294535

DSE is not ready, start-up of DSE is not completed yet.

Description	DSE is not ready, start-up of DSE is not completed yet.		
Response	Class	3	Abort job processing.
Solution	Class	7	Wait until DSE start up is finished.
Error type	-		

ID 294536

DSE DPRAM SW version does not fit.			
Description	DSE DPRAM SW version does not fit.		
Response	Class	3	Job processing aborted.
Solution	Class	7	-
Parameter	%1:	Current value [-]	
		Error code.	
Error type	-		

ID 294538

DSE measuring channel is occupied.			
Description	The DSE measuring channel is occupied.		
Response	Class	7	Stop axis motion immediately.
Solution	Class	6	Check measuring channel. Do reset.
Parameter	%1:	Current value [-]	
		Actual value of the measuring channel.	
	%2:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%3:	Drive type [-]	
		Drive type of the axis.	
	%4:	NC block number [-]	
		NC block number.	
Error type	11, Error message from position controller.		

ID 294539

Invalid DSE measuring channel.			
Description	The denoted DSE measuring channel is not valid.		
Response	Class	7	Stop axis motion immediately.
Solution	Class	6	Check measuring channel. Do reset.
Parameter	%1:	Current value [-]	
		Actual value of the measuring channel.	
	%2:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%3:	Drive type [-]	
		Drive type of the axis.	
	%4:	NC block number [-]	
		NC block number.	
Error type	11, Error message from position controller.		

ID 294540

DSE: The axis already has an active job in the measuring channel.			
Description	The DSE axis has already an active job in this measuring channel.		
Response	Class	7	Stop axis motion immediately.
Solution	Class	6	Check measuring channel. Do reset.
Parameter	%1:	Current value [-]	
		Measuring channel.	
	%2:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of concerned axis.	
	%3:	Drive type [-]	
		Drive type of the axis.	
	%4:	NC block number [-]	
		NC block number.	
Error type	11, Error message from position controller.		

ID 294541

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	Stop axis motion immediately.
Solution	Class	8	Requires restart of controller-

ID 294542

KPS: Current too high.			
Description	The KPS is overloaded.		
Response	Class	7	The KPS shuts down.
Solution	Class	6	Rectify causes for too high current drain: Reteach robot path. Slow down velocity.
Error type	11, Error message from position controller.		

ID 294543

KPS: Main phase loss.			
Description	Phase loss within the 3-phase-voltage.		
Response	Class	7	The KPS cannot offer the required power.
Solution	Class	6	Check for loose connection Check for cable break.
Error type	11, Error message from position controller.		

ID 294544

KPS: Drives disabled.	
Description	Drives disabled on the KPSi of a Cobra-Control.

Response	Class	7	Robot stops.
Solution	Class	6	Eliminate error of the KPSi.
Error type	11, Error message from position controller.		

ID 294545

Emergency stop (local).			
Description	The local emergency stop button on the KCP triggered.		
Response	Class	7	Ramp-down braking for all axes.
Solution	Class	6	Release emergency stop button. NC reset.
Parameter	%1:	Logical axis number [-]	
		Current value of safety signal.	
	%2:	Current value [-]	
Error type	11, Error message from position controller.		

ID 294546

Emergency stop.			
Description	Emergency stop was triggered.		
Response	Class	7	Ramp-down braking for all axes.
Solution	Class	6	Release emergency stop button. NC reset.
Error type	11, Error message from position controller.		

ID 294547

Safety gate open.			
Description	The safety gate is open.		
Response	Class	7	Controlled motion stop Control enable blocked.
Solution	Class	6	Close safety gate and do nc reset.
Error type	11, Error message from position controller.		

ID 294548

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	7	Stop axis motion immediately.
Solution	Class	8	Requires restart of controller-

ID 294549

Number of DSE drives does not fit with global.servo (configuration error).	
Description	The number of axes set up in the configuration list (hochlauf.lis) does not fit the number of DSE drives set up in global.servo.

Response	Class	1	Abort job processing.
Solution	Class	1	Correct the number of axes or DSE drives.
Parameter	%1:	Current value [-]	
		Number DSE drives in the configuration list.	
Error type	-		

ID 294550

KPS: Assertion failed.			
Description	There was an assertion failure in the KPS.		
Response	Class	1	Warning output.
Solution	Class	1	Check KPS.
Parameter	%1:	KPS number.	
	%2:	DSE error id.	
Error type	11, Error message from position controller.		

ID 294551

KPS: Working area exceeded.			
Description	One or more axis of the robot have exceeded there axis range limits.		
Response	Class	1	The robot stops and can only move on at reduced speed.
Solution	Class	1	The robot must be moved back into the axis range.
Error type	11, Error message from position controller.		

ID 294552

KPS: Brake test required.			
Description	The holding torque of the mechanical brakes have to be checked every 8 hours in order to guarantee the safety of the robot.		
Response	Class	1	The robot stops if the time of the periodic test is exceeded.
Solution	Class	1	The robot can move on if the test has been made.
Error type	11, Error message from position controller.		

ID 294553

KPS: Maximum Cartesian speed exceeded.			
Description	The safety controller has detected a Cartesian speed that has exceeded the configured safety parameter.		
Response	Class	1	The robot stops.
Solution	Class	1	The message must be acknowledged. Then the robot can move on.
Error type	11, Error message from position controller.		

ID 294554

KPS: Safe Cartesian workspace exceeded.			
Description	The robot has left the Cartesian workspace that was defined in the safety parameters.		
Response	Class	1	The robot cannot move on.

Solution	Class	1	The robot must return to the workspace, or the next workspace to which the robot has moved misleadingly must be enabled.
Error type	11, Error message from position controller.		

ID 294555

KPS: Mastering test failed.			
Description	In order to guarantee a correct robot adjustment, it must be referenced by moving to a defined point. The mastering test has failed in this case. Possible reasons are: an incorrect mastering a defective referencing sensor a displaced referencing sensor.		
Response	Class	1	The robot may only be moved at reduced speed.
Solution	Class	1	check mastering and correct if applicable check defect referencing sensor and change if applicable check position of referencing sensor and correct if applicable
Error type	11, Error message from position controller.		

ID 294556

KPS: Mastering test required.			
Description	In order to guarantee correct robot alignment, the alignment must be referenced by moving to a defined position. This must be done periodically every 8 hours.		
Response	Class	1	If the mastering has not been checked within this time period, the robot can only move with reduced speed.
Solution	Class	1	The robot can return to normal operation if the reference run has been carried out successfully.
Error type	11, Error message from position controller.		

ID 294557

KPS: Safe RDW system error.			
Description	An internal error occurred on the Safe RDW.		
Response	Class	1	Drives ready signal cannot be set.
Solution	Class	1	The problem may originate from the software or hardware.
Error type	11, Error message from position controller.		

ID 294558

KPS: Safety parameters incorrect.			
Description	Data in the set safety parameters are not correct or missing.		
Response	Class	1	Drives ready signal cannot be set.
Solution	Class	1	The correct safety parameters must be downloaded.
Error type	11, Error message from position controller.		

ID 294559

KPS: Safety position violated.

Description	The robot has moved, although the safety position stop is enabled.		
Response	Class	1	The robot is stopped and can only move at reduced speed.
Solution	Class	1	The safety position monitoring must be deactivated or the robot must return to that position in which the safety position monitoring has been activated.
Error type	11, Error message from position controller.		

ID 294560

KPS: Position tracker transmission error DSE - RDW.			
Description	Damaged cable or plug connection between DSE and RDW, which are only used for position trackers. Cable not connected or connected incorrectly. Data transmission malfunction.		
Response	Class	1	Output signal $POS_TRACKER_ERROR is set.
Solution	Class	1	Check DSE and RDW power supply (LEDs). Check cable (screen!) and plug-in connections. Check serial DSE - RDW interface. Check grounding connection.
Error type	11, Error message from position controller.		

ID 294561

KPS: Position tracker encoder cable failure.			
Description	Encoder cable position tracker input defective. Loose connection. Supply voltage missing. Encoder of position tracker defective.		
Response	Class	1	Output signal $POS_TRACKER_ERROR is set.
Solution	Class	1	Check position tracker encoder cable. Check connections. Check supply voltage. Check position tracker encoder and exchange if necessary.
Error type	11, Error message from position controller.		

ID 294562

KPS: Internal DSE error.		
Description	The KPS reported an internal DSE error.	
Response	Class	1 Warning output.
Solution	Class	1 Check KPS.
Parameter	%1:	KPS number.
	%2:	DSE error id.
Error type	11, Error message from position controller.	

ID 294563

KPS: Safe retraction enabled.

Description	The drive is activated by a safe input on the safe RDW. The robot can be moved at a safely reduced speed without any further monitoring being active. This permits the robot to manoeuvre out of a safety violation or e.g. execute homing and brake test to restore the safe state.		
Response	Class	1	Safe reduced speed is active, other moving monitoring functions are not active.
Solution	Class	1	Motion with normal speed is only possible after deactivating the "Safe Retraction".
Error type	11, Error message from position controller.		

ID 294564

KPS: Safe Input/Output incorrect.			
Description	One or all inputs or outputs are electrically defective.		
Response	Class	1	Drives ready signal cannot be set.
Solution	Class	1	Check the I/O wiring.
Error type	11, Error message from position controller.		

ID 294565

KPS: Safety mode not possible.			
Description	The Safe RDC cannot monitor the robot.		
Response	Class	1	Drives ready signal cannot be set.
Solution	Class	1	Boot Safe RDC, load correct safety parameters, master robot axis, ...
Error type	11, Error message from position controller.		

ID 294566

SW version of DSE Library does not fit.			
Description	The software version of the DSE Library does not fit the expected version.		
Response	Class	3	Abort job processing.
Solution	Class	7	Use expected version of the DSE Library.
Parameter	%1:	Expected value [-]	
		Expected software version of the DSE Library.	
	%2:	Current value [-]	
		Used software version of the DSE Library.	
Error type	-		

ID 294567

KPS: IPO-Stop request by RDC.			
Description	The RDC recognized an error condition that requires an interpolator stop.		
Response	Class	7	Axes are stopped.
Solution	Class	6	e.g. analyze taught path regarding valid work envelope.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294568

KPS: Ramp stop request by RDC.			
Description	The RDC recognized an error condition that requires a ramp stop.		
Response	Class	7	Axes are stopped.
Solution	Class	6	e.g. analyze taught path regarding valid work envelope.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294569

KPS: Short circuit braking request by RDC.			
Description	The RDC recognized an error condition that requires a short circuit braking.		
Response	Class	7	Axes are stopped.
Solution	Class	6	e.g. analyze taught path regarding valid work envelope.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294570

KPS: Axis range monitoring has issued a stop.			
Description	Axis range monitoring on SafeRDW issues a stop reaction.		
Response	Class	7	Axes are stopped.
Solution	Class	6	Do not exceed axis range.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294571

KPS: Calibration reference switch defect.			
Description	The reference switch has been triggered although no calibration has been requested.		
Response	Class	1	Warning output.
Solution	Class	1	Maybe the reference switch is defect.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294572

KPS: Stop2 issued by SafeRDC.			
Description	The SafeRDC state demands a stop 2 reaction.		
Response	Class	7	Axes are stopped.
Solution	Class	6	-

Parameter	%1:	KPS number
	%2:	Original KPS error code
Error type	11, Error message from position controller.	

ID 294573

KPS: Workspace violated.			
Description	One or more axis of the robot have violated there axis range limits.		
Response	Class	7	The axis stops and can only move on with safe retraction active.
Solution	Class	6	The axis must be moved back into the workspace.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294574

KPS: Acknowledge stop because work envelope is exceeded.			
Description	An axis exceeded work envelope.		
Response	Class	7	The axis stops and can only move on with safe retraction active.
Solution	Class	6	The axis must be moved back into the workspace.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294575

KPS: Safety coefficients missing - auto/external mode blocked.			
Description	At least for one axis not at least 3 safety coefficients $SR_BRK_POLY[,] in R1/$machine.dat are defined.		
Response	Class	1	Automatic mode is locked.
Solution	Class	1	Set $SR_BRK_POLY[,] in R1/$machine.dat correct.
Parameter	%1:	KPS number	
	%2:	Original KPS error code	
Error type	11, Error message from position controller.		

ID 294576

KPS: Operating mode mismatch with input T1 on SafeRDC.			
Description	The operating mode is inconsistent with the input T1 on the SafeRDW		
Response	Class	7	Drives enable not possible.
Solution	Class	6	-Check connection from KCP to CI3-Tech-Board. -Check connection from CI3-Tech-Board to SafeRDC. -Is the correct KCP used? -Is the correct CI3-Tech-Board used?
Parameter	%1:	KPS number	
	%2:	Original KPS error code	

Error type	11, Error message from position controller.

ID 294577

KPS: Maximum safe reduced Cartesian speed exceeded.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294578

KPS: RDC: Flash sectors defective.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	11, Error message from position controller.		

ID 294579

KPS: More then one tool enabled on the Safe RDC.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	11, Error message from position controller.		

ID 294580

KPS: Stop by more than one active tool on the Safe RDC.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294581

KPS: Maximum global Cartesian speed limit exceeded.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294582

KPS: Maximum Cartesian speed limit in T1 mode exceeded.			
Description			
Response	Class	7	
Solution	Class	6	

Error type	11, Error message from position controller.

ID 294583

KPS: Maximum Cartesian speed limit for monitoring range exceeded.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294584

KPS: Limit of orientation angle of tool exceeded.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	11, Error message from position controller.		

ID 294585

KPS: Stop issued by orientation angle limitation.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294586

KPS: Stop by violated Safety position.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294587

KPS: Failure safety input.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	11, Error message from position controller.		

ID 294588

KPS: Stop by failure safety input.			
Description			
Response	Class	7	
Solution	Class	6	

Error type	11, Error message from position controller.

ID 294589

KPS: Failure safety output.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	11, Error message from position controller.		

ID 294590

KPS: Stop by failure safety output.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294591

KPS: Acknowledge Enable safety output.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294592

KPS: Acknowledge stop because of cross compare error.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294593

KPS: Monitoring range exceeded.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	11, Error message from position controller.		

ID 294594

KPS: Stop before monitoring range was violated.			
Description			
Response	Class	7	
Solution	Class	6	

Error type	11, Error message from position controller.

ID 294595

KPS: Monitoring range violated.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	11, Error message from position controller.		

ID 294596

KPS: Stop by violated monitoring range.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 294597

EMI watchdog error.		
Description		
Response	Class	7
Solution	Class	6
Parameter	%1:	Current value [-]
Error type	11, Error message from position controller.	

ID 294598

Configuration of EMI index wrong.		
Description		
Response	Class	3
Solution	Class	7
Parameter	%1:	Incorrect value [-]
	%2:	Limit value [-]
Error type	-	

ID 294600

Error in CNC EMI driver.		
Description		
Response	Class	7
Solution	Class	6
Error type	11, Error message from position controller.	

ID 294601

SW version of EMI interface does not fit.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Expected value [-]	
	%2:	Current value [-]	
Error type	-		

ID 294602

Cannot open EMI-PLC command interface.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

2.16.14 ID-range 295000-295249

ID 295000

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires restart of controller-

ID 295001

Homing error of generic drive.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 295002

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	5	
Solution	Class	8	Requires restart of controller-

ID 295003

Position lag error in generic drive interface.		
Description		
Response	Class	0
Solution	Class	0
Parameter	%1:	Logical axis number [-]
	%2:	Identification number [-]
	%3:	NC block number [-]
	%4:	Current value [-]
	%5:	Current value [-]
Error type	11, Error message from position controller.	

ID 295004

Homing error in generic drive interface.			
Description			
Response	Class	0	
Solution	Class	0	
Error type	11, Error message from position controller.		

ID 295005

Error in generic drive interface.			
Description			
Response	Class	0	
Solution	Class	0	
Error type	11, Error message from position controller.		

ID 295006

Error while unhoming a generic drive.		
Description		
Response	Class	7
Solution	Class	6
Parameter	%1:	Identification number [-]
	%2:	NC block number [-]
	%3:	Current value [-]
Error type	11, Error message from position controller.	

ID 295007

Homing is only possible in operation mode T1.		
Description		
Response	Class	7
Solution	Class	6
Parameter	%1:	Identification number [-]
	%2:	NC block number [-]
	%3:	Current value [-]
Error type	11, Error message from position controller.	

ID 295008

Error while aborting homing procedure of generic drive.		
Description		
Response	Class	7
Solution	Class	6
Parameter	%1:	Identification number [-]
	%2:	NC block number [-]
	%3:	Current value [-]
Error type	11, Error message from position controller.	

ID 295009

Error while aborting unhoming a generic drive.		
Description		
Response	Class	7
Solution	Class	6

Parameter	%1:	Identification number [-]
	%2:	NC block number [-]
	%3:	Current value [-]
Error type	11, Error message from position controller.	

ID 295010

Selected operation mode is not available.			
Description			
Response	Class	-	
Solution	Class	-	
Error type			

ID 295011

ISGNC Wrapper: Thread could not be started.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	System restart.
Error type	-		

ID 295012

ISGNC Wrapper: Check for ProConOS failed.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	- Check if ProConOS is running and activate ProConOS - System restart.
Error type	-		

ID 295013

ISGNC Wrapper: Check for CNC adapter failed.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	- Check if CNC adapter is running and activate CNC adapter. - System restart.
Error type	-		

ID 295014

ISGNC Wrapper: Shared memory for HLI could not be created.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	
Error type	-		

ID 295015

ISGNC Wrapper: HLI could not be registered to ProConOS.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	
Error type	-		

ID 295016

ISGNC Wrapper: EMI-/PLC Platform Interface could not be registered to ProConOS.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	
Error type	-		

ID 295017

ISGNC Wrapper: Initialisation of kernel failed.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	
Error type	-		

ID 295018

ISGNC Wrapper: Error in boot phase 'AttachingData'.			
Description			
Response	Class	-	ISG kernel stopped.
Solution	Class	-	
Error type	-		

ID 295019

Invalid measuring channel.		
Description	The measure channel is invalid.	
Response	Class	7
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	Identification number [-]
	%3:	Lower limit value [-]
	%4:	Upper limit value [-]
Error type	11, Error message from position controller.	

ID 295020

The axis already has an active job in the measuring channel.

Description	The measure channel is already use by another measuring task.		
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Identification number [-]	
	%3:	Drive type [-]	
	%4:	NC block number [-]	
Error type	11, Error message from position controller.		

ID 295021

Measuring channel is occupied.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Identification number [-]	
	%3:	Drive type [-]	
	%4:	NC block number [-]	
Error type	11, Error message from position controller.		

ID 295022

Service channel error while measuring.			
Description			
Response	Class	7	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Logical axis number [-]	
	%3:	Current value [-]	
	%4:	NC block number [-]	
Error type	11, Error message from position controller.		

ID 295023

Error on coupling or decoupling a generic drive.			
Description			
Response	Class	5	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	11, Error message from position controller.		

ID 295024

Drive is in RampStopped state.	
Description	

Response	Class	0	
Solution	Class	0	
Error type	11, Error message from position controller.		

ID 295025

Warning in generic drive interface.			
Description			
Response	Class	0	
Solution	Class	0	
Error type	11, Error message from position controller.		

ID 295026

Could not load kernel.			
Description	The “ISGkernel.o” kernel module could not be loaded.		
Response	Class	-	
Solution	Class	-	Check whether the kernel module is present. Check whether there is enough memory available. Check whether RAM disc is full.
Error type	-		

ID 295027

Could not connect axis interface.			
Description	Could not connect axis interface.		
Response	Class	-	
Solution	Class	-	Check configuration for motion axes.
Error type	-		

ID 295028

Could not connect measuring channel.			
Description	Could not connect measuring channel.		
Response	Class	-	
Solution	Class	-	Check configuration for measuring channels.
Error type	-		

ID 295029

Could not connect diagnosis interface.			
Description	Could not connect diagnosis interface.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 295030

Could not connect non-cyclic interface.			
Description	Non-cyclic interface could not be connected.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 295031

Could not disconnect axis interface.			
Description	Could not disconnect axis interface.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 295032

Measuring channel could not be disconnected.			
Description	Measuring channel could not be disconnected.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 295033

Could not disconnect diagnosis interface.			
Description	Could not disconnect diagnosis interface.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 295034

Could not disconnect non-cyclic interface.			
Description	Non-cyclic interface could not be disconnected.		
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 295035

Axis locked by drive driver.			
Description	Drive driver locked by drive driver. IFeedLock = Locked.		
Response	Class	0	
Solution	Class	0	
Error type	11, Error message from position controller.		

ID 295036

General error.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 295037

Axis reset required after returning from hibernation.			
Description			
Response	Class	0	
Solution	Class	0	
Error type	11, Error message from position controller.		

ID 295038

The CNC EMI interface must be enabled when a tool is changed.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 295039

NC program start during active tool change is not permitted.			
Description			
Response	Class	7	
Solution	Class	6	
Error type	11, Error message from position controller.		

ID 295040

TorqueLimit not available while TorqueFreeStop is active.			
Description			
Response	Class	0	
Solution	Class	0	
Error type	11, Error message from position controller.		

ID 295041

EMI/PLatform PLC Interface version does not match expected version.			
Description			
Response	Class	-	
Solution	Class	-	
Error type			

ID 295042

HLI version does not match expected version.			
Description			
Response	Class	-	
Solution	Class	-	
Error type			

ID 295043

The ProConOS firmware FB could not be installed.			
Description			
Response	Class	-	
Solution	Class	-	
Error type			

2.16.15 ID-range 296000-296249

ID 296000

Specified error log file could not be opened.			
Description	The file specified in P-STUP-00170 cannot be opened to log the error messages. • Incorrect file path specified. • The file is write protected		
Response	Class	3	Abort current job.
Solution	Class	3	Check and modify P-STUP-00170.
Parameter	%1:	Current value [-]	
		Parameterised file path of log file.	
Error type	-		

ID 296001

ProfiDrive drive does not respond within the expected time.			
Description			
Response	Class	3	
Solution	Class	3	
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 296002

Too many error messages. No space left in index directory.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
	%2:	Upper limit value [-]	
Error type	-		

ID 296003

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 296004

Maximum number of error IDs to be filtered was exceeded.

Description	The maximum number of error IDs to be filtered is limited. Error IDs can be filtered by platform, channel or axis. (P-STUP-00186, P-CHAN-00378, P-AXIS-00627) This number was exceeded. `error_filter[0].reason 20048, 14, 15, 17, 18, 1234`		
Response	Class	1	Output a warning; the error ID is not filtered.
Solution	Class	1	Check and modify the filter specified.. Specify an additional filter.
Parameter	%1:	Current value [-]	
		Indication to localise the invalid entry: • Start-up list - for start-up list • Channel - for channel parameter • Axis - for axis parameter list	
	%2:	Current value [-]	
		If the channel or axis parameter list is specified, the channel ID or the logical axis number is output.	
	%3:	Current value [-]	
		Filter index where the error occurred. error_filter[**0**].reason	
	%4:	Current value [-]	
		Specified number of error IDs	
	%5:	Upper limit value [-]	
		Maximum number of error IDs	
Error type	-		

ID 296005

Maximum number of error texts to be filtered was exceeded.			
Description	The maximum number of error texts to be filtered is limited. Error texts can be filtered by platform, channel or axis. (P-STUP-00186, P-CHAN-00378, P-AXIS-00627) This number was exceeded.		
Response	Class	1	Output a warning; the error text is not filtered.
Solution	Class	1	Check and modify the filter specified.. Specify an additional filter.
Parameter	%1:	Current value [-]	
		Indication to localise the invalid entry: • Start-up list - for start-up list • Channel - for channel parameter • Axis - for axis parameter list	
	%2:	Current value [-]	
		If the channel or axis parameter list is specified, the channel ID or the logical axis number is output.	
	%3:	Current value [-]	
		Filter index where the error occurred. error_filter[**0**].reason	
	%4:	Current value [-]	
		Specified number of error texts	
	%5:	Upper limit value [-]	
		Maximum number of error texts	
Error type	-		

ID 296006

Maximum number of characters of an error text to be filtered was exceeded.			
Description	The maximum number of error texts to be filtered was exceeded. Error texts can be filtered by platform, channel or axis. (P-STUP-00186, P-CHAN-00378, P-AXIS-00627)		
Response	Class	1	Output a warning; the error text to be filtered is not accepted.
Solution	Class	1	Check and modify the error text to be filtered.
Parameter	%1:	Current value [-]	
		Indication to localise the invalid entry: • Start-up list - for start-up list • Channel - for channel parameter Axis - for axis parameter list	
	%2:	Current value [-]	
		If the channel or axis parameter list is specified, the channel ID or the logical axis number is output.	
	%3:	Current value [-]	
		Filter index where the error occurred. error_filter[**0**].reason	
	%4:	Current value [-]	
		Number of characters in the specified text	
	%5:	Upper limit value [-]	
		Maximum number of characters in an error text	
Error type	-		

ID 296007

Maximum number of error IDs to be filtered with conditional filter activation was exceeded.			
Description	The maximum number of error IDs to be filtered with conditional filter activation is limited. Error IDs can be filtered conditionally by platform, channel or axis. (P-STUP-00190, P-CHAN-00382, P-AXIS-00631) This number was exceeded.		
Response	Class	1	Output a warning; the error ID is not filtered.
Solution	Class	1	Check and modify the filter specified.. Specify an additional filter.
Parameter	%1:	Current value [-]	
		Indication to localise the invalid entry: • Start-up list - for start-up list • Channel - for channel parameter • Axis - for axis parameter list	
	%2:	Current value [-]	
		If the channel or axis parameter list is specified, the channel ID or the logical axis number is output.	
	%3:	Current value [-]	
		Filter index where the error occurred. error_filter[**0**].conditional_param	
	%4:	Current value [-]	
		Specified number of conditional error IDs	
	%5:	Upper limit value [-]	
		Maximum number of conditional error IDs	

Error type	-

ID 296008

Maximum number of error texts to be filtered with conditional filter activation was exceeded.			
Description	The maximum number of error texts to be filtered with conditional filter activation is limited. Error texts can be filtered conditionally by platform, channel or axis. (P-STUP-00190, P-CHAN-00382, P-AXIS-00631) This number was exceeded.		
Response	Class	1	Output a warning; the error text is not filtered.
Solution	Class	1	Check and modify the filter specified.. Specify an additional filter.
Parameter	%1:	Current value [-]	
		Indication to localise the invalid entry: • Start-up list - for start-up list • Channel - for channel parameter Axis - for axis parameter list	
	%2:	Current value [-]	
		If the channel or axis parameter list is specified, the channel ID or the logical axis number is output.	
	%3:	Current value [-]	
		Filter index where the error occurred. error_filter[**0**].conditional_param	
	%4:	Current value [-]	
		Specified number of conditional error texts	
	%5:	Upper limit value [-]	
		Maximum number of conditional error texts	
Error type	-		

ID 296009

Maximum number of characters of an error text to be filtered with conditional filter activation was exceeded.			
Description	The maximum length of an error text to be filtered with conditional filter activation was exceeded. Error texts can be filtered conditionally by platform, channel or axis. (P-STUP-00190, P-CHAN-00382, P-AXIS-00631)		
Response	Class	1	Output a warning; the error text to be filtered is not accepted.
Solution	Class	1	Check and modify the error text to be filtered.

Parameter	%1:	Current value [-]
		Indication to localise the invalid entry: • Start-up list - for start-up list • Channel - for channel parameter Axis - for axis parameter list
	%2:	Current value [-]
		If the channel or axis parameter list is specified, the channel ID or the logical axis number is output.
	%3:	Current value [-]
		Filter index where the error occurred. error_filter[**0**].conditional_param
	%4:	Current value [-]
		Number of characters in the specified text
	%5:	Upper limit value [-]
		Maximum number of characters in an error text
Error type	-	

ID 296010

Parameterised error text file could not be opened.			
Description	Error text files can be specified by the following parameters: P-STUP-00168 P-STUP-00169 A file specified by these parameters cannot be opened.		
Response	Class	1	Warning, no reaction.
Solution	Class	1	Check and modify the parameters in the start-up list.
Parameter	%1:	Current value [-]	
		Name of the file that cannot be opened.	
Error type	-		

ID 296011

User-specific error ID in TcCOM object outside permissible value range.			
Description	Sections are defined for user-specific error messages for individual TcCOM objects. The error ID used by the user is outside the permissible value range.		
Response	Class	3	Abort current job.
Solution	Class	3	Check and modify the user-specific error message in the TcCOM object.
Parameter	%1:	Current value [-]	
		Error ID of internal CNC error	
	%2:	Error value [-]	
		User-specific error ID used	
	%3:	Lower limit value [-]	
		Lower limit value for user-specific error ID in the TcCOM object	
	%4:	Upper limit value [-]	
		Upper limit value for user-specific error ID in the TcCOM object	
Error type	-		

2.16.16 ID-range 297000-297249

ID 297012

Only linear motion blocks G00/G01 can be used to define polygonal control areas.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Identification number [-]	
Error type	1, Error message from NC program.		

ID 297014

Plane change in work or protection area definition not permitted.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Identification number [-]	
	%3:	Current value [-]	
Error type	1, Error message from NC program.		

ID 297015

Axis of the third control area direction (EXCUR) may not be used for the definition of the base plane.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%3:	Incorrect value [-]	
Error type	1, Error message from NC program.		

ID 297016

The definition of the control area base plane requires the specification of two axes.			
Description			
Response	Class	2	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	1, Error message from NC program.		

2.16.17 ID-range 301000-301249

ID 301000

Specified 'LINKPOINT' unknown.			
Description	The LINKPOINT specified in the NC program does not exist in the current scene.		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction
Parameter	%1:	Name of incorrect LINKPOINT	
Error type	1, Error message from NC program.		

ID 301001

Specified 'Mountpoint' does not exist.			
Description	When a kinematic chain is defined by #SCENE ADD[LINKPOINT=<> MOUNTPOINT=<>...], a LINKPOINT attempts to set to a different non-existing LINKPOINT (MOUNTPOINT).		
Response	Class	2	Abort NC program processing.
Solution	Class	6	NC program correction
Parameter	%1:	Name of current LINKPOINT	
	%2:	Name of invalid MOUNTPOINT	
Error type	1, Error message from NC program.		

ID 301002

No memory space to create an additional linkpoint.			
Description	When an attempt is made to define an additional linkpoint using #SCENE ADD[LINKPOINT=<> MOUNTPOINT=<>...], this could not be created due to lack of memory space.		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check and reduce their number if necessary
Parameter	%1:		
		Name of the current LINKPOINT	
	%2:		
		Name of invalid MOUNTPOINT	
Error type	1, Error message from NC program.		

ID 301004

No memory to create further axis as linkpoint input.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 301005

Group could not be generated for new graphic object.			
Description	When an attempt is made to add a graphic object #SCENE ADD [GOBJECT=<> GROUP=<>] or #GROUP ADD [GROUP=<> GOBJECT=<>] to a group, the new group could not created.		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the groups and reduce their number if necessary
Parameter	%1:		
		Name of incorrect group	
	%2:		
		Name of current graphic object	
Error type	1, Error message from NC program.		

ID 301006

Could not create new graphic object.			
Description	New graphic object could not be created using #SCENE ADD [GOBJECT=<>] since maximum number is reached.		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the graphic objects and reduce their number if necessary
Parameter	%1:		
		Name of graphic object to be created	
	%2:		
		Maximum number of graphic objects	
Error type	1, Error message from NC program.		

ID 301007

Graphic object could not be inserted in group.			
Description	Could not add another graphic object in the group since the group is already full.		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the group association of the graphic object.
Parameter	%1:		
		Name of graphic object group	
	%2:		
		Name of graphic object to be created	
	%3:		
		Maximum number of graphic objects in the group	
Error type	1, Error message from NC program.		

ID 301008

Could not find graphic object ‘LINKPOINT’ specified.

Description	Could not find the graphic object LINKPOINT specified in the command #SCENE ADD [GOBJECT=<> LINKPOINT=<>].		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the name of the LINKPOINT.
Parameter	%1:		
		Name of LINKPOINT	
	%2:		
		Name of the graphic object to be created.	
Error type	1, Error message from NC program.		

ID 301009

Could not find graphic object specified.			
Description	The graphic object specified in the NC command is currently unknown.		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the name of the graphic object.
Parameter	%1:		
		Name of graphical object	
Error type	1, Error message from NC program.		

ID 301010

Could not find group specified.			
Description	The specified group of the command #SCENE MOVE [GROUP=<>] is unknown.		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the group name.
Parameter	%1:	Name of the group	
Error type	1, Error message from NC program.		

ID 301011

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	4	
Solution	Class	8	Requires restart of controller-

ID 301012

Geometry object could not have more group definitions.			
Description	When group association was output, the maximum group count to which a graphic object belongs was exceeded. The function can be output by the command #SCENE PRINT[DATAFACTORY / FILE]. Group association can be specified by the two NC commands #GROUP ADD [GROUP<> GOBJECT<>] and #SCENE ADD [GOBJECT<> GROUP<>].		
Response	Class	1	Warning

Solution	Class	1	NC program correction: check the group name.
Parameter	%1:		
		Name of graphical object	
	%2:		
		Current count of group association of graphic object	
	%3:		
		Maximum number of groups to which a graphic object may belong.	
Error type	1, Error message from NC program.		

ID 301013

No memory to create further collision pair.			
Description	The maximum number of pairs was exceeded when an attempt was made to define a new collision pair using #COLL DEF [PAIR=<>].		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the collision definition.
Parameter	%1:		
		Name of collision pair	
	%2:		
		Name of first graphic object	
	%3:		
		Name of second graphic object	
	%4:		
		Maximum number of pairs	
Error type	1, Error message from NC program.		

ID 301014

No memory space to manage additional collision group.			
Description	The maximum number of definitions was exceeded when an attempt is made to define a new collision pair using #COLL DEF [PAIR=<>].		
Response	Class	2	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the collision definition.
Parameter	%1:		
		Name of collision definition	
Error type	1, Error message from NC program.		

ID 301015

Type of collision pair specified is unknown.			
Description			
Response	Class	2	
Solution	Class	6	
Error type	1, Error message from NC program.		

ID 301016

Name was not specified when a graphic object was created.			
Description	No name was specified when a new graphic object was created using #SCENE ADD [GOBJECT=<>].		
Response	Class	4	Stop motion and abort NC program.
Solution	Class	6	NC program correction: check the definition of the graphic object.
Error type	1, Error message from NC program.		

ID 301017

Specified 'LINKPOINT' not moved over an axis.			
Description	No valid linkpoint was specified when specifying the arm length of a rotary axis. An arm length can only be specified for a moved rotary axis. Syntax of the NC command **#COLL SET [LINKPOINT**<*name*> **ARM_LEN**<*length*>] **]**		
Response	Class	4	Stop axis and abort NC program.
Solution	Class	6	Correct linkpoint specified in NC command
Error type	1, Error message from NC program.		

ID 301018

Graphic object could not be added to group, maximum number of groups per object reached.			
Description	The specified object could not be added again to the group since the group already contains too many objects. Syntax of the NC command **#GROUP ADD [GROUP**<*name*> { **GOBJECT**<*name*> } **]**		
Response	Class	4	Stop the axis and abort the NC program.
Solution	Class	6	Check group and group objects
Parameter	%1:		
		Name of the group	
	%2:		
		Name of object	
	%3:		
		Maximum number of objects in the group	
Error type	1, Error message from NC program.		

ID 301019

Implicit LINKPOINT=GROUND cannot be created explicitly.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 301020

Scene DB must be initialised although it was not configured.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	3, Error message from communication.		

2.16.18 ID-range 310000-399999

ID 310000

Activation of measurement not possible, distance control on master axis is already active.			
Description	Distance control is active in the master axis when the measurement was activated; this is not allowed.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program and the program flow. Execute measurement before activating distance control.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected in which the measurement is to be executed.	
	%2:	Block number [-]	
		Block number in the NC program when the measurement is activated.	
	%3:	Logical axis number [-]	
		Logical axis number (P-AXIS-00016) of the master axis	
	%4:	Error value [-]	
		Control word of the master axis	
Error type	-		

ID 310001

Generation of low pass filter coefficients with given parameters not possible.			
Description	With distance control (see FCT-M3), a low-pass filter was parameterised to smooth the measured distance values. However, no valid filter can be created with the given filter parameters. For axis-specific distance control: • Filter type P-AXIS-00782 • Order of the filter P-AXIS-00507 • Filter cut-off frequency P-AXIS-00508 For 3D distance control: • Filter type P-CHAN-00825 • Order of the filter P-CHAN-00816 • Filter cut-off frequency P-CHAN-00817 Or, the low-pass filter can be programmed in the NC program for axis-specific and 3D distance control.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check filter cut-off frequency. It must be less than 0.5 / GEO cycle time (Nyquist theorem). Try to reduce the filter order.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of the axis affected, parameter is only output for axis-specific distance control.
	%2:	Current value [-]
		Return value of filter generation
	%3:	Current value [-]
		Filter order of low pass filter
	%4:	Current value[Hz]
		Characteristic frequency of low-pass filter
	%5:	Current value [µs]
		Cycle time of GEO task
Error type	-	

ID 310002

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 310003

Generation of smoothing filter with given parameters not possible.			

Description	A filter for smoothing the measured distance values was parameterised for distance control (see FCT-M3) However, no valid filter can be created with the given filter parameters.
	Example of axis-specific distance control:
	`%Example`
	`;Select filter type`
	`N10 Z[DIST_CTRL FILTER_TYPE=KALMAN_MA]`
	`; Parametrise filter`
	`N20 Z[DIST_CTRL DISTC_N_CYCLES=30 KALMAN_SIGMA=1000]`
	`; Activate distance control`
	`N30 Z[DIST_CTRL ON CONST_DIST SET_DIST=1]`
	; …
	Example of 3D distance control:
	`%Example`
	`;Select filter type`
	`N10 #DIST CTRL [FILTER_TYPE=KALMAN_MA]`
	`; Parametrise filter`
	`N20 #DIST CTRL [DISTC_N_CYCLES=30 KALMAN_SIGMA=1000]`
	`; Activate distance control`
	`N30 #DIST CTRL ON [CONST_DIST SET_DIST=1]`
	; …
	Alternatively, the filter can also be defined by the corresponding axis parameters for axis-specific distance control. By analogy, this also applies to the channel parameters for 3D distance control.		

Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the required parameter.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Current value [-]
		Return value of filter generation
	%3:	Current value [-]
		Active filter type.
Error type	-	

ID 310004

Too many measuring points. Measurement is aborted.			
Description	When the distance is measured using distance control, the number of measuring points exceeds the size of the array of the V.E. variable.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Adapt the size of the V.E. array or reduce the number of measuring points.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number of the measurement	
	%3:	Error value [-]	
		Number of measuring points	
	%4:	Upper limit value [-]	
		Maximum permissible number of measuring points	
Error type	-		

ID 310005

Activation of distance control not possible because axis is part of active measurement.			
Description	With axis-specific distance control, the axis to be controlled is still actively involved in a measurement when distance control was activated. With 3D distance control, an axis to be controlled is still involved in the measurement.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the process. Only activate distance control after measurement.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [-]	
		Distance control state	
	%3:	Block number [-]	
		Block number that activates distance control.	
	%4:	Current value [-]	

Error type	-

ID 310006

Activation of measurement in current state of distance control not possible.			
Description	In order to activate the measurement function of distance control, it must be in “INACTIVE” or “FREEZE” state.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	5	Distance control must be returned to a valid state before activating the measurement function. Axis-specific distance control: • Z[DIST_CTRL OFF/FREEZE] 3D distance control: • #DIST CTRL/ FREEZE
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [-]	
		Distance control state	
	%3:	Block number [-]	
		Current block number	
Error type	-		

ID 310007

No measuring variable for the measured values specified.			
Description	A valid V.E. variable must be specified as array in order to read the recorded measuring points from the CNC.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	5	Store valid V.E. variable in the keyword “MEMORY”.
Parameter	%1:	Logical axis number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control.	
	%2:	Block number [-]	
		Current block number	
Error type	-		

ID 310008

No minimum height difference for capturing the measured values specified.			
Description	A valid minimum height difference must be specified in order to use the measurement function of distance control.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Program the minimum height difference in the keyword “MIN_MEAS_HEIGHT“.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control
	%2:	Block number [-]
		Current block number
Error type	-	

ID 310009

No minimum distance for capturing the measured values specified.			
Description	A valid minimum distance of the measured values must be programmed in order to use the measurement function of distance control.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Program the minimum distance in the keyword “MIN_MEAS_DIST“.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310010

No maximum distance for capturing the measured values specified.			
Description	A valid maximum distance must be programmed in order to use the measurement function of distance control.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Program the maximum valid distance in the keyword “MAX_MEAS_DIST“.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310011

Master axis not programmed for measurement run.			
Description	A master axis must be specified in order to use the measurement function of distance control.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Program the master axis in the keyword “AX1“ or “AX2“.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Block number [-]
		Block number in the NC program
	%3:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis AX1
	%4:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis AX2
Error type	-	

ID 310012 / 310013

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 310014

Invalid transition for distance control.			
Description	An invalid state transition was commanded for distance control in the PLC. This error can only occur with axis-specific distance control. For further information, see [FCT-M3//Section: Distance control states and transitions]		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the SPS program.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310015

Distance control is not enabled.			

Description	Distance control is not selected. Distance control must be activated before commands can be specified for distance control (ON, SET_POS) in the NC program or in the PLC :
	For axis-specific distance control using P-AXIS-00328:
	`lr_param.distance_control_on 1`
	For 3D distance control using P-CHAN-00500:
	`configuration.decoder.function FCT_3D_DIST_CTRL`		

Response	Class	7	Controlled halt of the axis, the control loop is closed.

Solution	Class	6	Check the activation. Activate the function depending on the distance control used.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310016

Distance control active but encoder delivers no valid actual positions.			
Description	An axis or several axes with active distance control is to be moved although one of the sensors does not deliver valid values. Value 2 of the error message indicates which of the two distance measuring sensors triggered the error. Possible output values: 1 - Actual position of the axis/axes 2 – Configured velocity 3 - Additive actual position value of the axis/axes 4 - Second additive actual position of the axis		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and rectify the sensor fault
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Current value [-]	
		Identifier of faulty sensor	
Error type	-		

ID 310017

Tactile deviation too big.			
Description	The compensation value offset calculated by distance control exceeds the permissible limit. Due to safety reasons, the maximum deviation caused by distance control between the programmed position in the NC program and the real actual position of the axis can be limited. With axis-specific distance control, this is achieved using P-AXIS-00414 (kenngr.distc.max_deviation) and for 3D distance control using P-CHAN-00801 (dist_ctrl[i].max_deviation).		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check workpiece, set position SET_POS of the workpiece surface or set distance SET_DIST to the workpiece surface. Check permissible compensation value P-AXIS-00414 or P-CHAN-00801.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Current value [0.1 10^-3 mm or ø]
		Current sensing deviation
	%3:	Upper limit value [0.1 10^-3 mm or ø]
		Parameterised maximum deviation P-AXIS-00414 or P-CHAN-00801
	%4:	Current value [0.1 10^-3 mm or ø]
		Current offset due to distance control
	%5:	Block number [-]
		Block number in the NC program
Error type	-	

ID 310018

Sensor signal: step of actual value oversized.			
Description	During active distance control (see [FCT-M3) the encoder values of the sensor or probe system are monitored due to safety reasons to detect errors in the distance measurement. This error message is issued if the encoder value changes by more than the specified limit within a CNC cycle. For axis-specific distance control, the limit is defined using P-AXIS-00417 and for 3D distance control using P-CHAN-00804.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the sensor/probe system
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [1µm/s or 0.001°/s]	
		Actual value change of sensor/probe system	
	%3:	Upper limit value [1µm/s or 0.001°/s]	
		Permissible limit value P-AXIS-00417 or P-CHAN-00804	
	%4:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310019

Sensor is completely extended.			
Description	The current offset of the distance control (see [FCT-M3]) was frozen. With axis-specific distance control, the moved axis has reached the maximum deviation of the probe sensor. (maximum deviation = P-AXIS-00420 + ½* P-AXIS-00421) With 3D distance control, the TCP reaches the maximum deviation of the probe sensor. (maximum deviation = P-CHAN-00807 + ½* P-CHAN-00808) A possible cause could be a hole in the workpiece.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.

Solution	Class	6	Check the sensor and the workpiece
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current position value of sensor	
	%3:	Lower limit value [0.1 10^-3 mm or ø]	
		Lower limit value for the sensor position	
	%4:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310020

Sensor is completely retracted.			
Description	The current offset of the distance control (see [FCT-M3]) was frozen. With axis-specific distance control, the moved axis has reached the minimum deviation of the probe sensor. (minimum deviation = P-AXIS-00420 - ½* P-AXIS-00421) With 3D distance control, the TCP reaches the maximum deviation of the probe sensor. (minimum deviation = P-CHAN-00807- ½* P-CHAN-00808) A possible cause could be an increase in height of the workpiece.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check the sensor and the workpiece
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Current value [0.1 10^-3 mm or ø]	
		Current position value of sensor	
	%3:	Upper limit value [0.1 10^-3 mm or ø]	
		Upper limit value for sensor position	
	%4:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310021

Distance control is still active at end of program.			
Description	The end of the NC program M02/M30) was reached while distance control is still active. Distance control must be disabled before program end.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the process. Disable distance control before program end

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Block number [-]
		Block number in the NC program
Error type	-	

ID 310022

Axis was released with active distance control.			
Description	When axis-specific distance control is active, one or several axes are to be released and their distance controlled. This is not permitted.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the process. Release the axis after deselecting distance control
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
	%3:	Error value [0.1 10^-7 mm or ø]	
		Current offset due to distance control	
	%4:	Current value [-]	
		Current state of distance control	
Error type	-		

ID 310023

During deselection of distance control, it was activated again.			
Description	Distance control was deactivated and then re-activated immediately afterwards. After deactivation, wait until the existing compensation offset has finished before re-activating distance control. Alternatively, distance control can be disabled with the addition NO_MOVE without clearing the existing compensation offset. See: • Programming axis-specific distance control • Programming 3D distance control		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the process. Possibly disable distance control using OFF NO_MOVE to avoid clearing the existing compensation offset.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	

Error type	-

ID 310024

Before activating distance control the command height must be specified.			

Description	Before distance control is activated in “surface” mode, specify the command height of the workpiece surface SET_POS.
	Example of axis-specific distance control:
	`Z [DIST_CTRL ON SET_POS=100]`
	See: [FCT-M3//Specifying the workpiece surface]
	
	Example of 3D distance control:
	`#DIST CTRL ON [SURFACE SET_POS=100 … ]`		

Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program; set the command height before or on activating distance control.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
Error type	-		

ID 310025

Mode change while distance control is active.			
Description	Axis-specific distance control supports two different operation modes (see FCT-M3// Operation mode of distance control) • "Surface" mode: Control on the workpiece surface, with axis-specific distance control: Default 3D distance control: SURFACE • "Distance" mode: Control of the distance SET_DIST with axis-specific distance control: CONST_DIST 3D distance control: CONST_DIST If the distance control is active, a change in these operation modes is not allowed without previously disabling distance control (OFF) or freezing the current offset (FREEZE).		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the process. First disable distance control before activating the other mode.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Block number [-]
		Block number in the NC program
	%3:	Current value [-]
		Current operation mode of the distance control
	%4:	Error value [-]
		New programmed operation mode of distance control
Error type	-	

ID 310026

Distance control turned on without set distance given.			

Description	Axis-specific distance control function (see FCT-M3//Specifying the distance) was enabled in “distance” mode without specifying a set distance.
	
	Example of axis-specific distance control:
	`Z [DIST_CTRL ON CONST_DIST SET_DIST=100]`
	
	Example of 3D distance control:
	`#DIST CTRL ON [CONST_DIST SET_DIST=100]`		

Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the process. Specify a set distance when enabling.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
	%3:	Current value [-]	
		Programmed operation mode of distance control	
Error type	-		

ID 310027

Distance control turned on without valid sensor variable.			
Description	A VARIABLE was selected as sensor source for distance control. A valid variable must be programmed for this sensor source using the keyword VARIABLE.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program; use a valid sensor variable.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Block number [-]
		Block number in the NC program
	%3:	Current value [-]
		Current sensor source
Error type	-	

ID 310028

Change of sensor source while distance control is active not allowed.			

Description	A change in sensor source with axis-specific and 3D distance control is not allowed.
	`Z[DIST_CTRL SENSOR_SOURCE=… ]`
	or
	`#DIST CTRL[SENSOR_SOURCE=… ]`		

Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and modify the NC program; use a valid sensor variable.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Block number [-]	
		Block number in the NC program	
	%3:	Current value [-]	
		Current sensor source	
Error type	-		

ID 310029

Distance control turned on with sensor source 'SECOND_ENCODER', but no second encoder configured.			
Description	Distance control was activated with the sensor source “SECOND_ENCODER“. • Axis-specific distance control The sensor signal must be transferred by a second encoder for this sensor source. The second encoder was not configured • 3D distance control it is not allowed to activate 3D distance control using the sensor source “SECOND_ENCODER“.		
Response	Class	7	Controlled halt of the axis, the control loop is closed.
Solution	Class	6	Check and configure a second encoder or change the sensor source.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Block number [-]
		Block number in the NC program
	%3:	Current value [-]
		Current sensor source
Error type	-	

ID 315000

Velocity for distance control must be greater than zero.			
Description	For distance control (see FCT-M3), the velocity used to compensate the distance can be adjusted. For axis-specific distance control using P-AXIS-00415 and for 3D distance control using P-CHAN-00802. The parameter used must be greater than zero.		
Response	Class	1	Output a warning and correct the parameter to the minimum value.
Solution	Class	1	Check and correct the parameter accordingly. • Axis-specific distance control P-AXIS-00415 • 3D distance control P-CHAN-00802.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [µm/s]	
		Parameterised velocity	
	%3:	Lower limit value [µm/s]	
		Minimum velocity	
	%4:	Corrected value [µm/s]	
		Corrected velocity	
Error type	-		

ID 315001

Acceleration for distance control must be greater/equal to zero.			
Description	For distance control (see FCT-M3), the velocity used to compensate the distance can be adjusted. For axis-specific distance control using P-AXIS-00416 and for 3D distance control using P-CHAN-00803. The parameter used must be greater than zero.		
Response	Class	1	Output a warning and correct the parameter to the minimum value.
Solution	Class	1	Check and correct the parameter. • Axis-specific distance control P-AXIS-00416 • 3D distance control P-CHAN-00803

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Error value [mm/s²]
		Parameterised acceleration
	%3:	Lower limit value [mm/s²]
		Minimum acceleration
	%4:	Corrected value [mm/s²]
		Corrected acceleration.
Error type	-	

ID 315002

Number of filter cycles for distance control exceeds valid range.			
Description	Distance control (see FCT-M3) uses an average value filter to smooth the sensor signal. However, the number of specified filter cycles for this filter exceeds the maximum permissible value. For axis-specific distance control using P-AXIS-00413, and for 3d distance control using P-CHAN-00800.		
Response	Class	1	Output a warning and correct the parameter to the minimum value.
Solution	Class	1	Check and correct the parameter accordingly. • Axis-specific distance control P-AXIS-00413 • 3D distance control P-CHAN-00800
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [-]	
		Parameterised value for the number of filter cycles	
	%3:	Corrected value [-]	
		Correct the value for the number of filter cycles	
	%4:	Upper limit value [-]	
		Maximum value for the number.of filter cycles	
Error type	-		

ID 315003

The cut-off frequency of the low-pass filter for distance control is out of permissible range.			
Description	The cut-off frequency can be defined for the low-pass filter for distance control (see FCT-M3). For axis-specific distance control using P-AXIS-00508, and for 3d distance control using P-CHAN-00817. The parameter may not be smaller than the specified limit value.		
Response	Class	1	Output a warning and correct the parameter to the minimum value.
Solution	Class	1	Check and correct the parameter accordingly. • Axis-specific distance control P-AXIS-00508 • 3D distance control P-CHAN-00817

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Error value [Hz]
		Parameterised cut-off frequency
	%3:	Lower limit value [Hz]
		Minimum cut-off frequency
	%4:	Corrected value [Hz]
		Corrected cut-off frequency
Error type	-	

ID 315004

The order of the low-pass filter for distance control is out of permissible range.			
Description	The order can be defined for the low-pass filter for distance control (see FCT-M3). For axis-specific distance control using P-AXIS-00507, and for 3d distance control using P-CHAN-00816. The parameter in each case is outside the permissible range		
Response	Class	1	Warning is output and correct the parameter.
Solution	Class	1	Check and correct the parameter accordingly. • Axis-specific distance control P-AXIS-00507 • 3D distance control P-CHAN-00816
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [-]	
		Parameterised order of the low-pass filter	
	%3:	Lower limit value [-]	
		Minimum order of the low-pass filter	
	%4:	Upper limit value [-]	
		Maximum order	
	%5:	Corrected value [-]	
		Corrected order of the low-pass filter	
Error type	-		

ID 315005

Weighting factor for velocity of distance control for lowering movement exceeds maximum permissible value.

Description	The velocity used for lowering movement and feed motion can be weighted for distance control [FCT-M3]. However, the specified weighting factor is greater than permissible. For axis-specific distance control: • Velocity P-AXIS-00415 • Weighting factor P-AXIS-00533 3D distance control: • Velocity P-CHAN-00802 • Weighting factor P-CHAN-00819		
Response	Class	1	Warning, weighting factor for velocity is corrected.
Solution	Class	1	Check and correct the weighting factor accordingly.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [0.1%]	
		Incorrect weighting factor	
	%3:	Upper limit value [0.1%]	
		Maximum permissible velocity weighting	
	%4:	Corrected value [0.1%]	
		Corrected value for velocity weighting	
Error type	-		

ID 315006

Weighting factor for acceleration of distance control for lowering movement exceeds maximum permissible value.			
Description	The acceleration used for lowering movement and feed motion can be weighted for distance control [FCT-M3]. However, the specified weighting factor is greater than permissible. For axis-specific distance control: • Acceleration P-AXIS-00416 • Weighting factor P-AXIS-00534 3D distance control: • Acceleration P-CHAN-00803 • Weighting factor P-CHAN-00820		
Response	Class	1	Warning, weighting factor for acceleration is corrected.
Solution	Class	1	Check and correct the weighting factor accordingly.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Error value [0.1%]
		Incorrect weighting factor
	%3:	Upper limit value [0.1%]
		Maximum permissible acceleration weighting
	%4:	Corrected value [0.1%]
		Corrected value for acceleration weighting
Error type	-	

ID 315007

Maximum distance change for distance control is out of permissible range.			
Description	The parameterised value for the maximum distance change of the Kalman filter for distance control [FCT-M3] is outside the permissible limit value. For axis-specific distance control: (*kenngr.distc.kalman_max_dist_change*) For 3D distance control: (*dist_ctrl[i].kalman_max_dist_change*)		
Response	Class	1	Warning, parameter for maximum distance change is corrected
Solution	Class	1	Check and correct the parameter accordingly
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [-]	
		Incorrect maximum distance change	
	%3:	Lower limit value [-]	
		Minimum value for maximum distance change	
	%4:	Corrected value [-]	
		Corrected value for maximum distance change	
Error type	-		

ID 315008

Smoothing factor for distance control is out of permissible range.			
Description	The smoothing factor for the exponential averaging filter for distance control [FCT-M3] may not exceed the specified limits. Parameterisation is executed using: • P-AXIS-00784 for axis-specific distance control • P-CHAN-00827 for 3D distance control		
Response	Class	1	Output the warning and limit smoothing factor.
Solution	Class	1	Check and correct the parameter for the smoothing factor.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Error value [-]
		Incorrectly parameterised value of smoothing factor
	%3:	Lower limit value [-]
		Minimum value of smoothing factor
	%4:	Upper limit value [-]
		Maximum value of smoothing factor
	%5:	Corrected value [-]
		Corrected value of smoothing factor
Error type	-	

ID 315009

Kalman sigma for distance control is out of permissible range.			
Description	The degree of uncertainty of the recorded measured values [FCT-M3]for the Kalman filter of distance control may not exceed the specified limits. Parameterisation is executed using: • P-AXIS-00783 for axis-specific distance control • P-CHAN-00826 for 3D distance control		
Response	Class	1	Warning output and value is corrected for Kalman sigma
Solution	Class	1	Check and correct the parameter for Kalman sigma.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [-]	
		Incorrectly parameterised value for Kalman sigma	
	%3:	Lower limit value [-]	
		Minimum value for Kalman sigma	
	%4:	Upper limit value [-]	
		Maximum value for Kalman sigma	
	%5:	Corrected value [-]	
		Corrected value for Kalman sigma	
Error type	-		

ID 315010

Kp factor for distance control is out of permissible range.			
Description	The Kp factor [FCT-M3] for distance control may not be outside the limit values. Parameterisation is executed using • P-AXIS-00759 for axis-specific distance control • P-CHAN-00821 for 3D distance control		
Response	Class	1	Warning is output and correct the Kp factor.
Solution	Class	1	Check and correct the parameter for the Kp factor.

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Error value [-]
		Incorrectly parameterised value of Kp factor
	%3:	Lower limit value [-]
		Minimum value of Kp factor
	%4:	Upper limit value [-]
		Maximum value of Kp factor
	%5:	Corrected value [-]
		Corrected value of Kp factor
Error type	-	

ID 315011

Tn factor for distance control is out of permissible range.			
Description	The I-Tn integral action time for distance control [FCT-M3] may not be outside the limit values. Parameterisation is executed using: • P-AXIS-00764 for axis-specific distance control • P-CHAN-00822 for 3D distance control		
Response	Class	1	Warning is output and correct the Tn factor.
Solution	Class	1	Check and correct the parameter for the Tn factor.
Parameter	%1:	Logical axis number [-] or Logical channel number [-]	
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.	
	%2:	Error value [-]	
		Incorrectly parameterised value of Tn factor	
	%3:	Lower limit value [-]	
		Minimum value of Tn factor	
	%4:	Upper limit value [-]	
		Maximum value of Tn factor	
	%5:	Corrected value [-]	
		Corrected value of Tn factor	
Error type	-		

ID 315012

Tv factor for distance control is out of permissible range.			
Description	The D-Tv derivative action time for distance control [FCT-M3] may not be outside the limit values. Parameterisation is executed using: • P-AXIS-00765 for axis-specific distance control • P-CHAN-00823 for 3D distance control		
Response	Class	1	Warning is output and correct the Tv factor.
Solution	Class	1	Check and correct the parameter for the Tv factor

Parameter	%1:	Logical axis number [-] or Logical channel number [-]
		Logical axis number P-AXIS-00016 of the axis affected for axis-specific distance control, logical channel number P-CHAN-00400 for 3D distance control.
	%2:	Error value [-]
		Incorrectly parameterised value of Tv factor
	%3:	Lower limit value [-]
		Minimum value of Tv factor
	%4:	Upper limit value [-]
		Maximum value of Tv factor
	%5:	Corrected value [-]
		Corrected value of Tv factor
Error type	-	

2.16.19 ID-range 400000-400249

ID 400000

TcCOM kinematic transformation: Input parameter reference is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400001

TcCOM kinematic transformation: Reference of of input vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400002

TcCOM kinematic transformation: Reference of output vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400003

TcCOM kinematic transformation: Type of parameter is unknown.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400004

TcCOM kinematic transformation: Wrong dimension of input vector.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400005

TcCOM kinematic transformation: Wrong dimension of output vector.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400006

TcCOM kinematic transformation: Value of input vector is out of range.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400007

TcCOM kinematic transformation: Division by zero.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400008

TcCOM kinematic transformation: Negative argument of square root operation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400009

TcCOM kinematic transformation: Argument for arcsine out of range.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400010

TcCOM kinematic transformation: Argument for arccosine out of range.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400011

TcCOM kinematic transformation: Argument for arctangent out of range.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400012

TcCOM kinematic transformation: Argument for tangent lies outside of value range.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400013

TcCOM kinematic transformation: Overflow during iteration.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 400014

TcCOM kinematic transformation: singularity in kinematic transformation.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

2.16.20 ID-range 401000-401249

ID 401000

TcCOM online TRC: Input parameter reference is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 401001

TcCOM online TRC: Reference of of Input vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 401002

TcCOM online TRC: Reference of Output vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

2.16.21 ID-range 402000-402249

ID 402000

TcCOM dynamic contour control: Input parameter reference is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 402001

TcCOM dynamic contour control: Reference of of Input vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 402002

TcCOM dynamic contour control: Reference of Output vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

2.16.22 ID-range 403000-403249

ID 403000

TcCOM geo feed adapt: Input parameter reference is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 403001

TcCOM geo feed adapt: Reference of of Input vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 403002

TcCOM geo feed adapt: Reference of Output vector is NIL.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

2.16.23 ID-range 404000-405005

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

2.16.24 ID-range 1000000-1000249

ID 1000000 - 1000043

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000044

PDU job to decoder interpret. Could not execute MDS list.		
Description		
Response	Class	3
Solution	Class	6
Parameter	%1:	Current value [-]
	%2:	[-]
	%3:	Instance [-]
Error type	-	

ID 1000045 - 1000085

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000086

Error while interpret the RT operating system settings.		
Description		
Response	Class	3
Solution	Class	7
Error type	-	

ID 1000087

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000090

The specified start-up procedure is unknown.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value [-]
	%2:	Corrected value [-]
Error type	-	

ID 1000091 - 1000098

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000099

Logical axes number dues not exist.			
Description	No axis is configured with the specified logical axis number. The order to accept the parameter by the control is rejected.		
Response	Class	3	The order to accept the parameter by the control is rejected.
Solution	Class	6	New order with correct logical axis number.
Parameter	%1:	Current value [-]	
		Unknown logical axis number	
Error type	-		

ID 1000100 / 1000101

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000116

MCM mode object does not exist.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	5, Error message by access on files.		

ID 1000117

The data type of IDENT is wrong.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value [-]
	%2:	Corrected value [-]
	%3:	Logical axis number [-]
Error type	-	

ID 1000119

The possible length of IDENT exceeds the limit.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value [-]
	%2:	Corrected value [-]
	%3:	Logical axis number [-]
Error type	-	

ID 1000120

The size of IDENT list is greater than 64 bytes.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value [-]
	%2:	Corrected value [-]
	%3:	Logical axis number [-]
Error type	-	

ID 1000121

The name of the file list is too long.			
Description			
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Corrected value [-]	
	%3:	Logical axis number [-]	
Error type	-		

ID 1000122

Cannot open the named file.			
Description			
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Logical axis number [-]	
Error type	-		

ID 1000123

Invalid modificator.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Corrected value [-]	
	%3:	Logical axis number [-]	
Error type	-		

ID 1000124

Invalid phase for IDENT.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Corrected value [-]	
	%3:	Logical axis number [-]	
Error type	-		

ID 1000127 - 1000129

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000130

Interpretation of hardware configuration list failed.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1000131

Timeout while waiting for acknowledge of all hardware initialisation tasks.			
Description			
Response	Class	1	
Solution	Class	7	
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 1000132

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000133

Initialisation of interrupt logic not available.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	3, Error in the communication.		

ID 1000134

License server not available.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1000135

License file not found.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1000136

Syntax error in license file.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1000137

Internal error during license key check.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1000138

License key for function not valid.			
Description			
Response	Class	3	
Solution	Class	7	

Error type	-

ID 1000139

License key expired.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1000140

Error on allocation of memory from the table administrator.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1000141

Function to call specified twice in list.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	6, Error message by data transfer into control device.		

ID 1000142 - 1000144

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000145

The minimum timer period of the operation system could not be read.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [1 µs]	

Error type	2, Error message by data transfer from parameter list into control device.

ID 1000146

The time slice is too small.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Error value [1 µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 1000147

The time slice is not an integral multiple of the timer value			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Error value [1 µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 1000148

One second divided by the time slice must be an integral value.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Error value [µs]
	%2	Corrected value [µs]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 1000149

The cycle time is too small.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Error value [1 µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 1000150

The ratio between cycle time and time slice is too small.			
Description			
Response	Class	3	
Solution	Class	1	

Parameter	%1:	Current value [µs]
	%3:	Corrected value [µs]
Error type	2, Error message by data transfer from parameter list into control device.	

ID 1000151

NC parameter minimum Windows time is smaller than one time slice.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [µs]	
	%3:	Corrected value [µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 1000152

The Windows minimum time is too big.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [µs]	
	%2:	Corrected value [µs]	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 1000153

The cycle time is no integral multiple of the parametrised time slice.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [1 µs]	

%2:	Corrected value [1 µs]
Error type	2, Error message by data transfer from parameter list into control device.

ID 1000154

NC parameter interrupt number is not specified.			
Description			
Response	Class	3	

Solution	Class	7	
Error type	2, Error message by data transfer from parameter list into control device.		

ID 1000155

Interrupt vector not attachable.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1000156

Cannot create semaphore "SEM_RAISE_IRQ".			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1000157

Cannot create semaphore "SEM_DROP_IRQ".			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1000158

Cycle time of interpolation task different to NC cycle time.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Expected value [-]	
Error type	6, Error message by data transfer into control device.		

ID 1000159

Too many axes for spindles configured.			
Description	The maximum number of spindles was exceeded. Please contact the controller manufacturer if more spindles are required.		
Response	Class	3	No start-up possible
Solution	Class	7	Reduce number of spindles
Parameter	%1:	Current value [µs]	
		Number of configured spindles	
	%2:	Limit value [µs]	
		Maximum number of spindles possible	

Error type	-

ID 1000160 - 1000163

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000164

Type of variable must not be specified as subelement of structure.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message by data transfer into control device.		

ID 1000165

Flexible array must content at least two subelement.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	6, Error message by data transfer into control device.		

ID 1000166

Error by parametrization of the event-manager with the hardware config.		
Description		
Response	Class	3
Solution	Class	7
Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	-	

ID 1000167

Error by parametrization of the decoder with the hardware configuration.		
Description		
Response	Class	3
Solution	Class	7
Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	-	

ID 1000168

Error: Parametrisation of the driver for transmission to the remote hardware.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	-		

ID 1000170 - 1000172

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000173

More SERCOS devices are defined than system can manage.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1000174

Specified number of channels exceeds the maximum number.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%3:	Corrected value [-]	
Error type	2, Error message on data transfer of parameter lists to controller.		

ID 1000175

Error while update of PLCopen resolution units.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	

Error type	-

ID 1000176

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000177

Axis with logical number zero is not permissible.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Logical axis number [-]	
	%2:	Current value [-]	
Error type	-		

ID 1000178

Structure member alignment of external variables invalid.			
Description			
Response	Class	1	
Solution	Class	7	
Parameter	%1:	Incorrect value [-]	
	%2:	Corrected value [-]	
Error type	-		

ID 1000179

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1000180

Invalid CPU number.			
Description	Could not find the specified CPU number.		
Response	Class	-	The assignment is made via the operating system.
Solution	Class	-	Check entry P-RTCF-00015 in parameter file and correct if necessary.

Parameter	%1:	Current value [-]
		Parameterised CPU number P-RTCF-00015
	%2:	Limit value [-]
		Maximum number of CPUs
Error type	6, Error message by data transfer to control device.	

ID 1000181

Setting the thread affinity failed.			
Description	Failure on assigning the specified CPU number to the thread.		
Response	Class	1	The assignment is made via the operating system.
Solution	Class	1	Check entry P-RTCF-00015 in parameter file and correct if necessary.
Parameter	%1:	Current value [-]	
		Specified CPU number P-RTCF-00015	
	%2:	Limit value [-]	
		Maximum number of CPUs	
Error type	6, Error message by data transfer to control device.		

2.16.25 ID-range 1001000-1001249

ID 1001082 - 1001084

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001085

Drive reports an error.			
Description	The SERCOS drive reports an error. Consult the drive manufacturer documentation for further error diagnosis.		
Response	Class	3	
Solution	Class	6	
Error type	-		

ID 1001087 - 1001089

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001090

Logical axis number not permitted.			
Description			
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	7, Error message by overflow of NC block.		

ID 1001095

Unknown ID in List 16 or 24.			
Description			
Response	Class	-	
Solution	Class	-	

Error type	-

ID 1001099 / 1001100

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001102

Block-interface for manual-mode not available.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1001103

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001105

The fibre-optic transmission rate is wrong.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Corrected value [-]	
Error type	-		

ID 1001106

The number of SERCOS rings is wrong.			
Description			
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Error value [-]
	%2:	Limit value [-]
	%3:	Corrected value [-]
Error type	-	

ID 1001107

The option does not exist.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Error value [-]
	%2:	Corrected value [-]
Error type	-	

ID 1001108

The time value is out of range.		
Description		
Response	Class	3
Solution	Class	6
Parameter	%1:	Error value [-]
	%2:	Limit value [-]
	%3:	Limit value [-]
Error type	-	

ID 1001109

The optional AT time value calculation does not exist.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Incorrect value [-]
	%2:	Corrected value [-]
Error type	-	

ID 1001110

The optional digital drive driver is not available.
Description

Response	Class	3	
Solution	Class	6	
Parameter	%1:	Corrected value [-]	
Error type	-		

ID 1001111 / 1001112

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 1001113

Illegal computer address.			
Description			
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Error value [-]	
	%2:	Error value [-]	
Error type	-		

ID 1001114

The computer address cannot be set.			
Description			
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Error value [-]	
Error type	-		

ID 1001115

Error by interpretation of CTM-DDL.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
	%2:	Current value [-]	
Error type	-		

ID 1001116

CTM index out of range.

Description			
Response	Class	3	
Solution	Class	6	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1001117

The list contains an unknown element.			
Description	The start-up list includes an unknown list element, which cannot calculated.		
Response	Class	1	Unknown list element will be ignored.
Solution	Class	1	Remove the unknown list element or correct the wrong entry.
Error type	-		

ID 1001118

The list contains an unknown element.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 1001119

The list contains an unknown element.			
Description	The axis parameter list includes an unknown list element, which cannot be interpreted.		
Response	Class	1	Unknown list element will be ignored.
Solution	Class	1	Remove the unknown list element or correct the wrong entry.
Error type	-		

ID 1001120

The list contains an unknown element.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value [-]
	%2:	Current value [-]
Error type	-	

ID 1001121

The same log. axis number is used several times.	
Description	Several configured axes contain the identical logical axis number. However, a logical axis number must be assigned uniquely within the entire system.

Response	Class	3	Axis cannot be configured.
Solution	Class	6	Abort the numerical control start-up.
Parameter	%1:	Logical axis number [-]	
		Multiple used logical axis number.	
	%2:	Current value [-]	
		Axis with this index include the same logical axis number.	
	%3:	Current value [-]	
		Axis with this index include the same logical axis number.	
Error type	-		

ID 1001122

Axis for duplication (clone) is internally unknown.			
Description	The specified logical axis number is unknown (parameter clone_of). No duplication can be done.		
Response	Class	3	No transfer of the axis parameter possible.
Solution	Class	7	Set the correct logical axis number.
Parameter	%1:	Logical axis number [-]	
	%2:	Error value [-]	
		Unknown logical axis number	
Error type	6, Error message by data transfer to control device.		

ID 1001125

Number of channels does not match to configuration.			
Description	For certain configurations (P-STUP-00007) only a set of number of channel is possible. In this case the configured number of channels does not match with the actual configuration.		
Response	Class	1	Correct the entry number for channels.
Solution	Class	1	Correct the start-up list.
Parameter	%1:	Expected value [-]	
		Expected number of channels.	
	%2:		
		Configured number of channels, P-STUP-00001	
	%3:	Corrected value [-]	
		Corrected number of channels.	
Error type	6, Error message by data transfer to control device.		

ID 1001129

Maximum number of axes exceeded, no license available.			
Description	No license available for configured number of axes. Please contact control vendor.		
Response	Class	3	No start-up of control possible.
Solution	Class	7	Load missing license key or reduce number of axes.

Parameter	%1:	Current value [-]
	%2:	Current value [-]
	%3:	Limit value [-]
Error type	-	

ID 1001130

Maximum number of channels exceeded, no license available.			
Description	No license available for configured number of channels. Please contact control vendor.		
Response	Class	3	No start-up of control possible.
Solution	Class	7	Load missing license key or reduce number of channels.
Parameter	%1:	Current value [-]	
	%2:	Limit value [-]	
Error type	-		

ID 1001131

Filename is invalid. No external HMI object list can be used.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Error value [-]	
	%3:	Corrected value [-]	
Error type	-		

ID 1001132

Invalid mode. No external HMI object list can be used.			
Description			
Response	Class	1	
Solution	Class	1	

Parameter	%1:	Current value [-]
	%2:	Error value [-]
	%3:	Corrected value [-]
Error type	-	

ID 1001133

File name invalid. No external BF channel object list can be used.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Error value [-]	
	%3:	Corrected value [-]	
Error type	-		

ID 1001134

Invalid mode. No external BF channel object list can be used.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
	%2:	Error value [-]	
	%3:	Corrected value [-]	
Error type	-		

ID 1001135

List contains unknown elements.			
Description	During interpretation of an IO device list, an unknown list element is detected.		
Response	Class	1	
Solution	Class	1	Remove or modify the unknown list element in the corresponding list.
Error type	-		

ID 1001136

No valid license for CNC export version.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001137

Valid license only for CNC export version.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001138

Invalid number format in list or permissible range of values exceeded.			
Description	During interpretation of an axis parameter, an invalid number format is detected or the interpreted parameter value exceeds the permissible value range.		
Response	Class	1	Interpretation of list is continued. The parameter value is limited to the maximum or minimum permissible value or set to zero if the number format is invalid.
Solution	Class	1	Correct the number format in the corresponding parameter list.
Error type	-		

ID 1001139

Maximum number of channels exceeded.			
Description	The number of parameterised channels exceeds the permitted upper limit.		
Response	Class	1	Start-up is executed with the permitted upper limit. Channels are created up to the specified upper limit.
Solution	Class	7	Check and correct configuration.
Parameter	%1:	Current value [-]	
		Number of parameterised channels	
	%2:	Limit value [-]	
		Permitted upper limit	
Error type	-		

ID 1001140

Maximum number of axes exceeded.			
Description	The number of parameterised axes exceeds the permitted upper limit.		
Response	Class	1	Start-up is executed with the permitted upper limit. Axes are created up to the specified upper limit.
Solution	Class	7	Check and correct configuration.

Parameter	%1:	Current value [-]
		Number of parameterised axes
	%2:	Limit value [-]
		Permitted upper limit
Error type	-	

ID 1001141

Invalid number format in list or permissible range of values exceeded.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001142

No license for use of ESA function available.		
Description		
Response	Class	1
Solution	Class	1
Parameter	%1:	Current value [-]
Error type	-	

ID 1001143

Configured maximum number of decoder macros exceeds limit.			
Description	The parameter P-CHAN-00509 defines the total number of NC macros, The value assigned in the parameter is too large.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-CHAN-00509
Parameter	%1:	Current value [-]	
		Parameterised value of P-CHAN-00509	
	%2:	Corrected value [-]	
		Corrected value for P-CHAN-00509	
Error type	-		

ID 1001144

Number of predefined NC macros exceeds limit.			
Description	The parameter P-CHAN-00510 defines the maximum number of predefined NC macros. The value assigned in the parameter is too large.		
Response	Class	1	Warning and correction of parameter.
Solution	Class	1	Check and modify P-CHAN-00510

Parameter	%1:	Current value [-]
		Parameterised value of P-CHAN-00510
	%2:	Corrected value [-]
		Corrected value for P-CHAN-00510
Error type	-	

ID 1001145

Configured maximum length of decoder macros symbols exceeds limit.			
Description	The parameter P-CHAN-00511 defines the maximum limit of characters for the name of NC macros. However, the parameterised value exceeds the permitted limit.		
Response	Class	1	P-CHAN-00511 is corrected to the maximum.
Solution	Class	1	Check and modify P-CHAN-00511
Parameter	%1:	Current value [-]	
		Parameterised value of P-CHAN-00511	
	%2:	Corrected value [-]	
		Corrected value of P-CHAN-00511	
Error type	-		

ID 1001146

Configured maximum number length of decoder macros NC-code exceeds limit.			
Description	The parameter P-CHAN-00512 defines the maximum limit of characters for the content of NC macros. However, the parameterised value exceeds the permitted limit.		
Response	Class	1	P-CHAN-00512 is corrected to the permitted limit.
Solution	Class	1	Check and modify P-CHAN-00512
Parameter	%1:	Current value [-]	
		Parameterised value of P-CHAN-00512	
	%2:	Corrected value [-]	
		Corrected value for P-CHAN-00512	
Error type	-		

ID 1001147

Memory allocation for mds interpretation failed.			
Description	The operating system cannot allocate the requested memory area.		
Response	Class	3	Abort current job.
Solution	Class	6	Check whether the operating system has sufficient memory. If necessary, expand the memory.
Parameter	%1:	Error value [-]	
		Memory address	
	%2:	Current value [-]	
		Size of the requested memory area	
	%3:	Current value [-]	
		Number of parameterised channels	

Error type	-

ID 1001148

No license for use of function 'processing on moving system/workpiece'.			
Description	A license is required for the ‘Machining on moved system/workpiece’ functionality. The error occurred because the configuration parameter FCT_DLM was configured in P-CHAN-00650 but the license is missing.		
Response	Class	1	The functionality is not enabled.
Solution	Class	1	Add the required license.
Parameter	%1:	Current value [-]	
	%2:	Corrected value [-]	
Error type	6, Error message by data transfer to control device.		

ID 1001149

Number of configured string labels exceeds the upper limit.			
Description	The number of string labels is defined in the channel parameter P-CHAN-00515. The configured value is greater than permitted.		
Response	Class	1	Output of warning: Apply the permitted maximum value.
Solution	Class	1	Check and modify P-CHAN-00515
Parameter	%1:	Current value [-]	
		Configured value for P-CHAN-00515	
	%2:	Corrected value [-]	
		Corrected value for P-CHAN-00515	
Error type	-		

ID 1001150

Number of configured expression labels exceeds the upper limit.			
Description	The number of expression labels is defined in the channel parameter P-CHAN-00516. The configured value is greater than permitted.		
Response	Class	1	Output of warning: Apply the permitted maximum value.
Solution	Class	1	Check and modify P-CHAN-00516
Parameter	%1:	Current value [-]	
		Configured value for P-CHAN-00516	
	%2:	Corrected value [-]	
		Corrected value for P-CHAN-00516	
Error type	-		

ID 1001151

No license for use of function 'additive manufacturing'.			
Description	The contour look-ahead function is part of the ‘extended interpolation’ license pack.		
Response	Class	1	The functionality is not enabled.
Solution	Class	1	Add the license for ‘extended interpolation'.
Parameter	%1:	Current value [-]	
	%2:	Corrected value [-]	
Error type	-		

ID 1001152

No license for use of function 'extended interpolation'.			
Description	The 2-path programming function is part of the ‘extended interpolation’ license pack.		
Response	Class	1	The functionality is not enabled.
Solution	Class	1	Add the license for ‘extended interpolation'.
Error type	-		

ID 1001153

The configured length of the diagnosis program trace exceeds the upper limit.			
Description	The parameter P-CHAN-00524 to configure the length of the diagnosis program trace of the decoder is greater than the permitted maximum value.		
Response	Class	1	P-CHAN-00524 is corrected to the permitted limit.
Solution	Class	1	Check and modify P-CHAN-00524..
Parameter	%1:	Current value [-]	
		Parameterised value for P-CHAN-00524	
	%2:	Corrected value [-]	
		Corrected value for P-CHAN-00524	
Error type	-		

ID 1001154

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	-	
Solution	Class	8	Requires controller restart.

ID 1001200

PDU-job to interpret data list fails in decoder.			
Description			
Response	Class	1	

Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	-		

2.16.26 ID-range 1001750-1001999

ID 1001815

Initialization of ISG-8-axes hardware version V2 failed.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1001816

Initialisation of R&W hardware (ring) failed.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001817

Initialization of SERCOS-hardware (ring) failed.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001818

Initialization of ISG-8-axes hardware version V2 failed.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001819

No SERCOS hardware configuration but there is a SERCOS axis configured.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001820

Driver for ISG hardware selected, but not supported by control.
Description

Response	Class	3	
Solution	Class	7	
Error type	6, Error message by data transfer to control device.		

ID 1001821

Driver for stepper hardware selected, but not supported by control.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	6, Error message by data transfer to control device.		

ID 1001822

Driver for CIM hardware selected, but not supported by control.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	6, Error message by data transfer to control device. .		

ID 1001825 / 1001826

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001827

Phase switch selected but SERCOS ring is not closed.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 1001828

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001829

Initialization of ISG-8-axes hardware version V3 failed.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1001830

Initialization of R&W MCU 3T PCI hardware failed.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	-		

ID 1001831

R&W MCU 3T PCI card found, but not booted.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 1001832

R&W MCU 3T PCI board found and booted, but DPRAM pointer invalid.			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 1001833

Cycle time of R&W MCU 3T PCI board does not match NC cycle time			
Description			
Response	Class	-	
Solution	Class	-	
Error type	-		

ID 1001834

R&W MCU 3T PCI board: physical HLI address is zero.			
Description			
Response	Class	-	

Solution	Class	-	
Error type	-		

ID 1001835 / 1001836

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001837

Initialization of driver of ISG-8-axes hardware version V3 failed.			
Description			
Response	Class	3	
Solution	Class	7	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1001838

I/O-operation of driver of ISG-8-axes hardware version V3 failed.			
Description			
Response	Class	1	
Solution	Class	1	
Parameter	%1:	Current value [-]	
Error type	-		

ID 1001839

SERCOS hardware selected, but not supported by control.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	6, Error message by data transfer to control device. .		

ID 1001840

DSE hardware selected, but not supported by control.			
Description			
Response	Class	3	
Solution	Class	7	
Error type	6, Error message by data transfer to control device. .		

ID 1001841

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001842

No license for physical drives.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	-		

ID 1001843

Cam table manager: Table file cannot be opened.			
Description			
Response	Class	1	
Solution	Class	1	
Error type	5, Error message by access on files.		

ID 1001844

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001845

Maximum number of job manager exceeded.		
Description		
Response	Class	3
Solution	Class	1
Parameter	%1:	Upper limit value [-]
Error type	6, Error message by data transfer to control device.	

ID 1001848

Specified channel-ID out of range.

Description	The specified value for the channel ID (P-CHAN-00400) of the channel is greater than the maximum permissible value.		
Response	Class	1	Warning is output and correct P-CHAN-00400
Solution	Class	1	Check and modify the value P-CHAN-00400
Parameter	%1:	Current value [-]	
		Parameterised value of P-CHAN-00400	
	%2:	Upper limit value [-]	
		For maximum value, see CNC system parameters/ 2.4	
Error type	6, Error message by data transfer to control device.		

ID 1001849

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	3	
Solution	Class	8	Requires restart of controller-

ID 1001850

Channel ID specified multiple times. Using default IDs.			
Description	The parameter P-CHAN-00400 assigns unique numbers to the CNC channels. However, an identical number was assigned to two different channels in the channel parameter list. For this reason, the controller uses the default numbering for start-up. Channel ID P-CHAN-00400 = channel index + 1		
Response	Class	3	Abort current job., use default numbering
Solution	Class	1	Check and modify the channel IDs in P-CHAN-00400
Parameter	%1:	Error value[-]	
		Double channel numbers P-CHAN-00400	
	%2:	Current value[-]	
		Channel numbers already assigned P-CHAN-00400	
	%3:	Current value[-]	
		Channel index of the invalid channel	
	%4:	Current value[-]	
		Channel index of the previous channel	
Error type	6, Error message by data transfer to control device.		

ID 1001851

MulteCore-context of axis different to context of platform.			
Description	When the MultiCore function [FCT-C39] is used, the context entries of **all** axes must be identical with the context entry of the GEOs of the first channel.		
Response	Class	3	Job processing aborted
Solution	Class	6	Check and modify the context entry of the axis affected.

Parameter	%1:	Logical axis number [-]
		Logical axis number P-AXIS-00016 of axis affected
	%2:	Error value [-]
		Incorrect context entry of the axis
	%3:	Expected value [-]
		Expected context entry of the GEOs of the first channel.
Error type	-	

2.16.27 ID-range 1003000-1003249

ID 1003001 / 1003002

System error [▶ 12]			
Description	This error message informs about internal states, error solutions and the place of error in the source. Please give the complete error message data to the CNC manufacturer.		
Response	Class	1	
Solution	Class	8	Requires restart of controller-

ID 1003003

Basic initialization of CANopen driver failed.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 1003004

Setup of CANopen driver failed.			
Description			
Response	Class	3	
Solution	Class	1	
Error type	-		

ID 1003005

Wrong constant definition from interface to ext. tool management.			
Description			
Response	Class	3	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Expected value [-]	
Error type	-		

ID 1003006

Wrong data definition from interface to ext. tool management.			
Description			
Response	Class	3	
Solution	Class	1	

Parameter	%1:	Error value [-]
	%2:	Expected value [-]
Error type	-	

ID 1003007

OCCI communication error by data transfer to ext. tool management.			
Description			
Response	Class	3	
Solution	Class	1	
Parameter	%1:	Error value [-]	
Error type	-		

ID 1003008

Wrong data length transmitted by external tool management.			
Description			
Response	Class	3	
Solution	Class	1	
Parameter	%1:	Error value [-]	
	%2:	Expected value [-]	
Error type	-		

3 Support and Service

Beckhoff and their partners around the world offer comprehensive support and service, making available fast and competent assistance with all questions related to Beckhoff products and system solutions.

Download finder

Our download finder contains all the files that we offer you for downloading. You will find application reports, technical documentation, technical drawings, configuration files and much more.

The downloads are available in various formats.

Beckhoff's branch offices and representatives

Please contact your Beckhoff branch office or representative for local support and service on Beckhoff products!

The addresses of Beckhoff's branch offices and representatives round the world can be found on our internet page: www.beckhoff.com

You will also find further documentation for Beckhoff components there.

Beckhoff Support

Support offers you comprehensive technical assistance, helping you not only with the application of individual Beckhoff products, but also with other, wide-ranging services:

- support
- design, programming and commissioning of complex automation systems
- and extensive training program for Beckhoff system components

Hotline:	+49 5246 963-157
e-mail:	support@beckhoff.com

Beckhoff Service

The Beckhoff Service Center supports you in all matters of after-sales service:

- on-site service
- repair service
- spare parts service
- hotline service

Hotline:	+49 5246 963-460
e-mail:	service@beckhoff.com

Beckhoff Headquarters

Beckhoff Automation GmbH & Co. KG

Huelshorstweg 20
33415 Verl
Germany

Phone:	+49 5246 963-0
e-mail:	info@beckhoff.com
web:	www.beckhoff.com

Index

P

P-ERR-0070560 1169
P-ERR-1000044 1776
P-ERR-1000086 1776
P-ERR-1000090 1777
P-ERR-1000099 1777
P-ERR-1000116 1778
P-ERR-1000117 1778
P-ERR-1000119 1778
P-ERR-1000120 1778
P-ERR-1000121 1779
P-ERR-1000122 1779
P-ERR-1000123 1779
P-ERR-1000124 1779
P-ERR-1000130 1780
P-ERR-1000131 1780
P-ERR-1000133 1780
P-ERR-1000134 1781
P-ERR-1000135 1781
P-ERR-1000136 1781
P-ERR-1000137 1781
P-ERR-1000138 1781
P-ERR-1000139 1782
P-ERR-1000140 1782
P-ERR-1000141 1782
P-ERR-1000145 1782
P-ERR-1000146 1783
P-ERR-1000147 1783
P-ERR-1000148 1783
P-ERR-1000149 1783
P-ERR-1000150 1783
P-ERR-1000151 1784
P-ERR-1000152 1784
P-ERR-1000153 1784
P-ERR-1000154 1784
P-ERR-1000155 1785
P-ERR-1000156 1785
P-ERR-1000157 1785
P-ERR-1000158 1785
P-ERR-1000159 1785
P-ERR-1000164 1786
P-ERR-1000165 1786
P-ERR-1000166 1786
P-ERR-1000167 1786
P-ERR-1000168 1787
P-ERR-1000173 1787
P-ERR-1000174 1787
P-ERR-1000175 1787
P-ERR-1000177 1788
P-ERR-1000178 1788
P-ERR-1000180 1788
P-ERR-1000181 1789
P-ERR-1001085 1790
P-ERR-1001090 1790
P-ERR-1001095 1790
P-ERR-1001102 1791
P-ERR-1001105 1791
P-ERR-1001106 1791
P-ERR-1001107 1792
P-ERR-1001108 1792
P-ERR-1001109 1792
P-ERR-1001110 1792
P-ERR-1001113 1793
P-ERR-1001114 1793
P-ERR-1001115 1793
P-ERR-1001116 1793
P-ERR-1001117 1794
P-ERR-1001118 1794
P-ERR-1001119 1794
P-ERR-1001120 1794
P-ERR-1001121 1794
P-ERR-1001122 1795
P-ERR-1001125 1795
P-ERR-1001129 1795
P-ERR-1001130 1796
P-ERR-1001131 1796
P-ERR-1001132 1796
P-ERR-1001133 1797
P-ERR-1001134 1797
P-ERR-1001135 1797
P-ERR-1001136 1798
P-ERR-1001137 1798
P-ERR-1001138 1798
P-ERR-1001139 1798
P-ERR-1001140 1798
P-ERR-1001141 1799
P-ERR-1001142 1799
P-ERR-1001143 1799
P-ERR-1001144 1799
P-ERR-1001145 1800
P-ERR-1001146 1800
P-ERR-1001147 1800
P-ERR-1001148 1801
P-ERR-1001149 1801
P-ERR-1001150 1801
P-ERR-1001151 1802
P-ERR-1001152 1802
P-ERR-1001153 1802
P-ERR-1001200 1802
P-ERR-1001815 1804
P-ERR-1001816 1804
P-ERR-1001817 1804
P-ERR-1001818 1804
P-ERR-1001819 1804
P-ERR-1001820 1804
P-ERR-1001821 1805
P-ERR-1001822 1805
P-ERR-1001827 1805
P-ERR-1001829 1806
P-ERR-1001830 1806
P-ERR-1001831 1806
P-ERR-1001832 1806
P-ERR-1001833 1806
P-ERR-1001834 1806
P-ERR-1001837 1807
P-ERR-1001838 1807
P-ERR-1001839 1807
P-ERR-1001840 1807
P-ERR-1001842 1808

P-ERR-1001843 1808
P-ERR-1001845 1808
P-ERR-1001848 1808
P-ERR-1001850 1809
P-ERR-1001851 1809
P-ERR-10030 49
P-ERR-1003003 1811
P-ERR-1003004 1811
P-ERR-1003005 1811
P-ERR-1003006 1811
P-ERR-1003007 1812
P-ERR-1003008 1812
P-ERR-1030 22
P-ERR-1033 22
P-ERR-1034 23
P-ERR-1035 23
P-ERR-1036 23
P-ERR-110034 1242
P-ERR-110035 1243
P-ERR-110036 1243
P-ERR-110037 1244
P-ERR-110038 1244
P-ERR-110039 1245
P-ERR-110040 1245
P-ERR-110042 1245
P-ERR-110043 1246
P-ERR-110048 1246
P-ERR-110051 1247
P-ERR-110052 1247
P-ERR-110053 1247
P-ERR-110054 1248
P-ERR-110055 1248
P-ERR-110059 1248
P-ERR-110060 1249
P-ERR-110061 1249
P-ERR-110062 1249
P-ERR-110063 1250
P-ERR-110064 1250
P-ERR-110065 1251
P-ERR-110066 1251
P-ERR-110067 1251
P-ERR-110068 1251
P-ERR-110069 1252
P-ERR-110070 1252
P-ERR-110071 1252
P-ERR-110073 1252
P-ERR-110074 1253
P-ERR-110075 1253
P-ERR-110076 1253
P-ERR-110077 1253
P-ERR-110078 1254
P-ERR-110079 1254
P-ERR-110080 1255
P-ERR-110081 1255
P-ERR-110082 1255
P-ERR-110083 1256
P-ERR-110084 1256
P-ERR-110085 1256
P-ERR-110086 1257
P-ERR-110087 1257
P-ERR-110088 1257
P-ERR-110089 1258
P-ERR-110090 1258
P-ERR-110091 1258
P-ERR-110092 1259
P-ERR-110093 1259
P-ERR-110094 1259
P-ERR-110095 1260
P-ERR-110096 1260
P-ERR-110097 1260
P-ERR-110098 1261
P-ERR-110099 1261
P-ERR-110100 1262
P-ERR-110101 1262
P-ERR-110102 1262
P-ERR-110103 1263
P-ERR-110104 1263
P-ERR-110105 1263
P-ERR-110106 1264
P-ERR-110107 1264
P-ERR-110110 1265
P-ERR-110111 1265
P-ERR-110112 1265
P-ERR-110113 1266
P-ERR-110114 1266
P-ERR-110115 1267
P-ERR-110117 1267
P-ERR-110118 1267
P-ERR-110119 1268
P-ERR-110120 1268
P-ERR-110121 1268
P-ERR-110122 1269
P-ERR-110123 1269
P-ERR-110124 1270
P-ERR-110125 1270
P-ERR-110126 1270
P-ERR-110127 1271
P-ERR-110128 1271
P-ERR-110129 1271
P-ERR-110130 1272
P-ERR-110131 1272
P-ERR-110132 1273
P-ERR-110133 1273
P-ERR-110134 1274
P-ERR-110135 1274
P-ERR-110136 1275
P-ERR-110137 1275
P-ERR-110138 1276
P-ERR-110139 1276
P-ERR-110140 1276
P-ERR-110141 1277
P-ERR-110142 1277
P-ERR-110143 1278
P-ERR-110145 1278
P-ERR-110147 1278
P-ERR-110148 1279
P-ERR-110149 1279
P-ERR-110150 1279
P-ERR-110151 1280
P-ERR-110154 1280
P-ERR-11016 50
P-ERR-110162 1281
P-ERR-110182 1281
P-ERR-110183 1281
P-ERR-110199 1282
P-ERR-11020 50

P-ERR-110210 1282
P-ERR-110211 1283
P-ERR-110213 1283
P-ERR-110214 1283
P-ERR-110215 1284
P-ERR-110216 1284
P-ERR-110217 1284
P-ERR-110218 1285
P-ERR-110219 1286
P-ERR-110220 1286
P-ERR-110221 1286
P-ERR-110222 1287
P-ERR-110228 1287
P-ERR-110229 1288
P-ERR-110230 1288
P-ERR-110231 1289
P-ERR-110232 1289
P-ERR-110236 1289
P-ERR-110237 1290
P-ERR-110238 1290
P-ERR-110239 1291
P-ERR-110240 1291
P-ERR-110241 1292
P-ERR-110242 1292
P-ERR-110243 1292
P-ERR-110244 1293
P-ERR-110246 1293
P-ERR-110250 1294
P-ERR-110251 1294
P-ERR-110254 1295
P-ERR-110264 1295
P-ERR-110267 1295
P-ERR-110268 1296
P-ERR-110269 1296
P-ERR-110270 1296
P-ERR-110271 1296
P-ERR-110272 1297
P-ERR-110273 1297
P-ERR-110274 1297
P-ERR-110275 1298
P-ERR-110276 1298
P-ERR-110277 1298
P-ERR-110278 1299
P-ERR-110281 1299
P-ERR-110282 1299
P-ERR-110283 1300
P-ERR-110286 1300
P-ERR-110288 1301
P-ERR-110292 1301
P-ERR-110294 1301
P-ERR-110297 1302
P-ERR-110298 1302
P-ERR-110299 1302
P-ERR-110300 1303
P-ERR-110301 1303
P-ERR-110302 1303
P-ERR-110303 1304
P-ERR-110304 1304
P-ERR-110305 1304
P-ERR-110307 1305
P-ERR-110308 1305
P-ERR-110309 1306
P-ERR-110310 1306
P-ERR-110311 1307
P-ERR-110312 1307
P-ERR-110316 1307
P-ERR-110317 1308
P-ERR-110318 1308
P-ERR-110319 1309
P-ERR-110320 1309
P-ERR-110321 1310
P-ERR-110322 1310
P-ERR-110323 1311
P-ERR-110326 1311
P-ERR-110327 1311
P-ERR-110328 1312
P-ERR-110329 1312
P-ERR-110330 1313
P-ERR-110331 1313
P-ERR-110332 1313
P-ERR-110333 1314
P-ERR-110334 1314
P-ERR-110335 1315
P-ERR-110336 1315
P-ERR-110337 1315
P-ERR-110339 1316
P-ERR-110340 1316
P-ERR-110341 1316
P-ERR-110342 1317
P-ERR-110343 1317
P-ERR-110344 1317
P-ERR-110348 1318
P-ERR-110349 1318
P-ERR-110350 1319
P-ERR-110351 1319
P-ERR-110352 1319
P-ERR-110353 1320
P-ERR-110354 1320
P-ERR-110355 1320
P-ERR-110356 1321
P-ERR-110359 1321
P-ERR-110360 1322
P-ERR-110361 1322
P-ERR-110362 1323
P-ERR-110363 1323
P-ERR-110364 1323
P-ERR-110365 1324
P-ERR-110366 1324
P-ERR-110367 1324
P-ERR-110368 1325
P-ERR-110371 1325
P-ERR-110372 1326
P-ERR-110373 1326
P-ERR-110374 1326
P-ERR-110376 1327
P-ERR-110378 1327
P-ERR-110379 1328
P-ERR-110380 1328
P-ERR-110381 1328
P-ERR-110382 1329
P-ERR-110384 1329
P-ERR-110385 1329
P-ERR-110386 1330
P-ERR-110387 1371
P-ERR-110388 1330
P-ERR-110389 1331

P-ERR-110390 1331
P-ERR-110391 1331
P-ERR-110392 1332
P-ERR-110393 1332
P-ERR-110394 1332
P-ERR-110397 1333
P-ERR-110398 1333
P-ERR-110399 1333
P-ERR-110400 1334
P-ERR-110401 1334
P-ERR-110402 1335
P-ERR-110403 1335
P-ERR-110404 1335
P-ERR-110405 1336
P-ERR-110406 1336
P-ERR-110409 1336
P-ERR-110410 1337
P-ERR-110411 1337
P-ERR-110412 1338
P-ERR-110413 1338
P-ERR-110414 1338
P-ERR-110415 1339
P-ERR-110416 1339
P-ERR-110417 1340
P-ERR-110418 1340
P-ERR-110419 1340
P-ERR-110420 1341
P-ERR-110421 1341
P-ERR-110422 1342
P-ERR-110423 1342
P-ERR-110424 1342
P-ERR-110425 1343
P-ERR-110426 1343
P-ERR-110427 1344
P-ERR-110428 1344
P-ERR-110429 1344
P-ERR-110430 1345
P-ERR-110431 1345
P-ERR-110432 1345
P-ERR-110433 1346
P-ERR-110434 1346
P-ERR-110437 1346
P-ERR-110438 1347
P-ERR-110439 1347
P-ERR-110440 1347
P-ERR-110441 1348
P-ERR-110442 1348
P-ERR-110443 1348
P-ERR-110444 1349
P-ERR-110445 1349
P-ERR-110446 1350
P-ERR-110447 1350
P-ERR-110448 1350
P-ERR-110449 1351
P-ERR-110450 1351
P-ERR-110451 1352
P-ERR-110452 1352
P-ERR-110453 1352
P-ERR-110454 1353
P-ERR-110455 1353
P-ERR-110456 1353
P-ERR-110457 1354
P-ERR-110459 1354
P-ERR-110460 1354
P-ERR-110461 1355
P-ERR-110462 1355
P-ERR-110463 1356
P-ERR-110464 1356
P-ERR-110465 1357
P-ERR-110466 1357
P-ERR-110467 1357
P-ERR-110468 1358
P-ERR-110469 1358
P-ERR-110470 1359
P-ERR-110471 1359
P-ERR-110472 1359
P-ERR-110473 1360
P-ERR-110474 1360
P-ERR-110475 1361
P-ERR-110476 1361
P-ERR-110477 1362
P-ERR-110478 1362
P-ERR-110479 1362
P-ERR-110480 1363
P-ERR-110481 1364
P-ERR-110482 1364
P-ERR-110483 1364
P-ERR-110485 1365
P-ERR-110486 1366
P-ERR-110487 1366
P-ERR-110488 1367
P-ERR-110489 1367
P-ERR-110490 1367
P-ERR-110491 1368
P-ERR-110496 1368
P-ERR-110497 1369
P-ERR-110498 1369
P-ERR-110499 1369
P-ERR-110500 1371
P-ERR-110501 1371
P-ERR-110502 1372
P-ERR-110503 1372
P-ERR-110504 1372
P-ERR-110505 1373
P-ERR-110506 1373
P-ERR-110507 1373
P-ERR-110508 1374
P-ERR-110509 1374
P-ERR-110510 1375
P-ERR-110511 1375
P-ERR-110512 1375
P-ERR-110513 1376
P-ERR-110514 1376
P-ERR-110515 1377
P-ERR-110516 1377
P-ERR-110517 1377
P-ERR-110518 1378
P-ERR-110520 1378
P-ERR-110521 1379
P-ERR-110522 1379
P-ERR-110523 1380
P-ERR-110524 1380
P-ERR-110528 1380
P-ERR-110529 1381
P-ERR-110530 1381
P-ERR-110531 1381

P-ERR-110532 1382
P-ERR-110533 1382
P-ERR-110538 1383
P-ERR-110539 1383
P-ERR-110540 1384
P-ERR-110541 1384
P-ERR-110543 1384
P-ERR-110544 1385
P-ERR-110545 1385
P-ERR-110547 1386
P-ERR-110548 1386
P-ERR-110549 1387
P-ERR-110550 1387
P-ERR-110551 1387
P-ERR-110552 1388
P-ERR-110553 1388
P-ERR-110557 1388
P-ERR-110558 1389
P-ERR-110559 1389
P-ERR-110560 1389
P-ERR-110561 1390
P-ERR-110562 1390
P-ERR-110563 1391
P-ERR-110564 1391
P-ERR-110565 1391
P-ERR-110566 1392
P-ERR-110567 1392
P-ERR-110568 1392
P-ERR-110569 1393
P-ERR-110570 1393
P-ERR-110571 1394
P-ERR-110572 1394
P-ERR-110573 1394
P-ERR-110574 1395
P-ERR-110575 1395
P-ERR-110576 1396
P-ERR-110577 1396
P-ERR-110578 1396
P-ERR-110579 1397
P-ERR-110580 1397
P-ERR-110581 1397
P-ERR-110582 1398
P-ERR-110583 1398
P-ERR-110584 1398
P-ERR-110585 1399
P-ERR-110586 1399
P-ERR-110587 1400
P-ERR-110588 1400
P-ERR-110589 1400
P-ERR-110590 1401
P-ERR-110591 1402
P-ERR-110592 1403
P-ERR-110593 1403
P-ERR-110594 1403
P-ERR-110595 1404
P-ERR-110596 1404
P-ERR-110597 1405
P-ERR-110598 1405
P-ERR-110600 1405
P-ERR-110602 1406
P-ERR-110603 1406
P-ERR-110604 1407
P-ERR-110605 1407
P-ERR-110606 1408
P-ERR-110607 1408
P-ERR-110608 1409
P-ERR-110609 1409
P-ERR-110610 1409
P-ERR-110611 1410
P-ERR-110612 1410
P-ERR-110613 1411
P-ERR-110614 1411
P-ERR-110615 1412
P-ERR-110617 1412
P-ERR-110618 1412
P-ERR-110619 1413
P-ERR-110620 1413
P-ERR-110621 1414
P-ERR-110622 1414
P-ERR-110624 1414
P-ERR-110625 1415
P-ERR-110626 1415
P-ERR-110627 1416
P-ERR-110628 1416
P-ERR-110629 1416
P-ERR-110630 1417
P-ERR-110632 1417
P-ERR-110633 1418
P-ERR-110634 1418
P-ERR-110636 1418
P-ERR-110637 1419
P-ERR-110638 1419
P-ERR-110639 1419
P-ERR-110640 1420
P-ERR-110641 1420
P-ERR-110642 1420
P-ERR-110643 1421
P-ERR-110644 1421
P-ERR-110645 1422
P-ERR-110647 1423
P-ERR-110649 1423
P-ERR-110651 1424
P-ERR-110652 1424
P-ERR-110653 1425
P-ERR-110654 1425
P-ERR-110655 1426
P-ERR-110656 1426
P-ERR-110657 1427
P-ERR-110658 1427
P-ERR-110659 1427
P-ERR-11066 50
P-ERR-110660 1428
P-ERR-110661 1428
P-ERR-110664 1429
P-ERR-110665 1429
P-ERR-110666 1430
P-ERR-110668 1431
P-ERR-110669 1431
P-ERR-110673 1431
P-ERR-110674 1432
P-ERR-110675 1432
P-ERR-110678 1432
P-ERR-110679 1433
P-ERR-110680 1433
P-ERR-110681 1433
P-ERR-110682 1434

P-ERR-110683 1434
P-ERR-110684 1434
P-ERR-110685 1435
P-ERR-110688 1435
P-ERR-110689 1435
P-ERR-110690 1436
P-ERR-110691 1436
P-ERR-110694 1437
P-ERR-110700 1437
P-ERR-110703 1437
P-ERR-11074 51
P-ERR-1110667 1430
P-ERR-11203 52
P-ERR-11204 52
P-ERR-11205 53
P-ERR-11206 53
P-ERR-120002 1439
P-ERR-120003 1439
P-ERR-120011 1440
P-ERR-120015 1440
P-ERR-120019 1441
P-ERR-120020 1441
P-ERR-120021 1442
P-ERR-120025 1442
P-ERR-120027 1443
P-ERR-120028 1443
P-ERR-120029 1443
P-ERR-120030 1444
P-ERR-120031 1444
P-ERR-120032 1445
P-ERR-120033 1445
P-ERR-120034 1445
P-ERR-120035 1446
P-ERR-120036 1446
P-ERR-120037 1446
P-ERR-120038 1447
P-ERR-120039 1447
P-ERR-120040 1448
P-ERR-120041 1448
P-ERR-120042 1448
P-ERR-120094 1449
P-ERR-120095 1449
P-ERR-120096 1449
P-ERR-120097 1450
P-ERR-120099 1450
P-ERR-120100 1450
P-ERR-120102 1451
P-ERR-120104 1451
P-ERR-120106 1452
P-ERR-120107 1452
P-ERR-120112 1452
P-ERR-120115 1453
P-ERR-120116 1453
P-ERR-120117 1454
P-ERR-120125 1454
P-ERR-120129 1455
P-ERR-120130 1455
P-ERR-120131 1455
P-ERR-120140 1456
P-ERR-120144 1456
P-ERR-120145 1457
P-ERR-120146 1457
P-ERR-120147 1458
P-ERR-120148 1458
P-ERR-120167 1458
P-ERR-120169 1458
P-ERR-120170 1459
P-ERR-120171 1459
P-ERR-120173 1459
P-ERR-120174 1460
P-ERR-120175 1460
P-ERR-120176 1460
P-ERR-120178 1461
P-ERR-120179 1461
P-ERR-120180 1461
P-ERR-120183 1461
P-ERR-120193 1462
P-ERR-120199 1462
P-ERR-120204 1463
P-ERR-120206 1463
P-ERR-120207 1464
P-ERR-120213 1464
P-ERR-120215 1465
P-ERR-120216 1465
P-ERR-120254 1467
P-ERR-120255 1467
P-ERR-120265 1467
P-ERR-120267 1468
P-ERR-120268 1468
P-ERR-120272 1469
P-ERR-120285 1469
P-ERR-120293 1470
P-ERR-120294 1470
P-ERR-120297 1470
P-ERR-120319 1471
P-ERR-120332 1472
P-ERR-120333 1472
P-ERR-120346 1472
P-ERR-120370 1473
P-ERR-120371 1473
P-ERR-120377 1474
P-ERR-120378 1474
P-ERR-120379 1474
P-ERR-120389 1474
P-ERR-120390 1475
P-ERR-120391 1475
P-ERR-120392 1476
P-ERR-120393 1476
P-ERR-120394 1476
P-ERR-120395 1477
P-ERR-120396 1477
P-ERR-120405 1477
P-ERR-120407 1478
P-ERR-120425 1478
P-ERR-120433 1479
P-ERR-120434 1479
P-ERR-120435 1480
P-ERR-120439 1480
P-ERR-120459 1481
P-ERR-120460 1481
P-ERR-120462 1482
P-ERR-120463 1482
P-ERR-120469 1483
P-ERR-120470 1483
P-ERR-120471 1484
P-ERR-120472 1484

P-ERR-120473 1485
P-ERR-120474 1485
P-ERR-120475 1486
P-ERR-120476 1486
P-ERR-120477 1487
P-ERR-120478 1487
P-ERR-120479 1488
P-ERR-120480 1488
P-ERR-120481 1489
P-ERR-120482 1489
P-ERR-120483 1490
P-ERR-120484 1490
P-ERR-120485 1491
P-ERR-120486 1491
P-ERR-120488 1492
P-ERR-120490 1493
P-ERR-120491 1493
P-ERR-120492 1493
P-ERR-120493 1493
P-ERR-120494 1493
P-ERR-120498 1494
P-ERR-120499 1494
P-ERR-120500 1495
P-ERR-120504 1495
P-ERR-120505 1495
P-ERR-120508 1496
P-ERR-120512 1496
P-ERR-120513 1496
P-ERR-120515 1497
P-ERR-120516 1497
P-ERR-120518 1498
P-ERR-120519 1498
P-ERR-120521 1498
P-ERR-120524 1499
P-ERR-120525 1499
P-ERR-120529 1499
P-ERR-120531 1500
P-ERR-120532 1500
P-ERR-120533 1500
P-ERR-120536 1500
P-ERR-120537 1501
P-ERR-120539 1501
P-ERR-120540 1501
P-ERR-120542 1502
P-ERR-120544 1502
P-ERR-120548 1503
P-ERR-120551 1503
P-ERR-120554 1504
P-ERR-120557 1504
P-ERR-120559 1505
P-ERR-120562 1506
P-ERR-120563 1506
P-ERR-120564 1506
P-ERR-120566 1506
P-ERR-120567 1507
P-ERR-120570 1507
P-ERR-120571 1508
P-ERR-120572 1508
P-ERR-120575 1508
P-ERR-120577 1509
P-ERR-120578 1509
P-ERR-120579 1509
P-ERR-120582 1510
P-ERR-120585 1511
P-ERR-120586 1511
P-ERR-120603 1512
P-ERR-120604 1512
P-ERR-120605 1512
P-ERR-120606 1512
P-ERR-120607 1512
P-ERR-120608 1513
P-ERR-120611 1513
P-ERR-120615 1513
P-ERR-120619 1514
P-ERR-120622 1514
P-ERR-120623 1514
P-ERR-120625 1515
P-ERR-120626 1515
P-ERR-120627 1515
P-ERR-120628 1515
P-ERR-120629 1516
P-ERR-120630 1516
P-ERR-120640 1517
P-ERR-120644 1517
P-ERR-120645 1517
P-ERR-120646 1517
P-ERR-120647 1518
P-ERR-120648 1518
P-ERR-120649 1518
P-ERR-120652 1519
P-ERR-120655 1519
P-ERR-120658 1520
P-ERR-120662 1520
P-ERR-120663 1520
P-ERR-120665 1521
P-ERR-120666 1521
P-ERR-120667 1521
P-ERR-120668 1522
P-ERR-120669 1522
P-ERR-120670 1522
P-ERR-120671 1522
P-ERR-120672 1523
P-ERR-120676 1523
P-ERR-120677 1523
P-ERR-120678 1523
P-ERR-120679 1524
P-ERR-120680 1524
P-ERR-120681 1525
P-ERR-120682 1525
P-ERR-120683 1525
P-ERR-120684 1525
P-ERR-120685 1525
P-ERR-120689 1526
P-ERR-120690 1526
P-ERR-120691 1527
P-ERR-120693 1528
P-ERR-120694 1528
P-ERR-120695 1528
P-ERR-120696 1529
P-ERR-120697 1529
P-ERR-120698 1529
P-ERR-120700 1530
P-ERR-120701 1530
P-ERR-120705 1531
P-ERR-120706 1531
P-ERR-120707 1531

P-ERR-120709 1531
P-ERR-120711 1532
P-ERR-120712 1532
P-ERR-120715 1532
P-ERR-120716 1532
P-ERR-120717 1533
P-ERR-120718 1533
P-ERR-120719 1533
P-ERR-120721 1533
P-ERR-120729 1534
P-ERR-120732 1534
P-ERR-120733 1534
P-ERR-120734 1535
P-ERR-120736 1535
P-ERR-120740 1536
P-ERR-120741 1537
P-ERR-120742 1538
P-ERR-120743 1539
P-ERR-120745 1540
P-ERR-120746 1540
P-ERR-120748 1540
P-ERR-120751 1542
P-ERR-120756 1542
P-ERR-120757 1542
P-ERR-120758 1542
P-ERR-120760 1543
P-ERR-120761 1543
P-ERR-120762 1543
P-ERR-120763 1543
P-ERR-120764 1544
P-ERR-120765 1544
P-ERR-120766 1544
P-ERR-120767 1544
P-ERR-120768 1545
P-ERR-120769 1545
P-ERR-120770 1545
P-ERR-120781 1545
P-ERR-120782 1546
P-ERR-120783 1546
P-ERR-120784 1546
P-ERR-120785 1546
P-ERR-120786 1547
P-ERR-120787 1547
P-ERR-120788 1547
P-ERR-120789 1547
P-ERR-120790 1547
P-ERR-120791 1548
P-ERR-120793 1548
P-ERR-120796 1549
P-ERR-120797 1549
P-ERR-120798 1549
P-ERR-120799 1549
P-ERR-120800 1550
P-ERR-120801 1550
P-ERR-120804 1550
P-ERR-120805 1550
P-ERR-120806 1550
P-ERR-120810 1551
P-ERR-120812 1552
P-ERR-120813 1552
P-ERR-120815 1553
P-ERR-120820 1554
P-ERR-120822 1554
P-ERR-120824 1555
P-ERR-120831 1555
P-ERR-12138 656
P-ERR-130007 1556
P-ERR-130105 1556
P-ERR-130106 1557
P-ERR-130120 1557
P-ERR-130121 1557
P-ERR-130125 1558
P-ERR-130126 1558
P-ERR-130127 1558
P-ERR-130134 1559
P-ERR-130137 1559
P-ERR-13100 54
P-ERR-13101 54
P-ERR-13102 54
P-ERR-13103 54
P-ERR-13104 55
P-ERR-13105 55
P-ERR-13106 55
P-ERR-13107 56
P-ERR-13108 56
P-ERR-13109 57
P-ERR-13110 57
P-ERR-13111 57
P-ERR-13112 58
P-ERR-13113 58
P-ERR-13114 58
P-ERR-13115 59
P-ERR-13116 59
P-ERR-13120 59
P-ERR-13121 59
P-ERR-13122 60
P-ERR-13123 60
P-ERR-13124 60
P-ERR-13125 60
P-ERR-13126 61
P-ERR-13127 61
P-ERR-13128 61
P-ERR-13140 61
P-ERR-13141 62
P-ERR-13142 62
P-ERR-13160 62
P-ERR-13161 62
P-ERR-13162 62
P-ERR-13163 63
P-ERR-13164 63
P-ERR-13165 63
P-ERR-13166 64
P-ERR-13167 64
P-ERR-13168 64
P-ERR-13169 64
P-ERR-13170 65
P-ERR-13171 65
P-ERR-13172 65
P-ERR-13173 65
P-ERR-13180 66
P-ERR-13181 66
P-ERR-13182 66
P-ERR-13183 66
P-ERR-13184 67
P-ERR-13185 67
P-ERR-13186 67

P-ERR-13187 67
P-ERR-13188 67
P-ERR-13189 68
P-ERR-13190 68
P-ERR-13191 68
P-ERR-13192 68
P-ERR-13193 69
P-ERR-13194 69
P-ERR-13195 69
P-ERR-13196 69
P-ERR-13200 69
P-ERR-13201 70
P-ERR-13202 70
P-ERR-13203 70
P-ERR-13204 70
P-ERR-13205 71
P-ERR-13206 71
P-ERR-13207 71
P-ERR-13208 71
P-ERR-13209 71
P-ERR-13210 72
P-ERR-13211 72
P-ERR-13220 72
P-ERR-13221 73
P-ERR-13222 73
P-ERR-13223 73
P-ERR-13224 73
P-ERR-13225 74
P-ERR-13226 74
P-ERR-13240 74
P-ERR-13241 75
P-ERR-13242 75
P-ERR-13243 75
P-ERR-13244 76
P-ERR-13245 76
P-ERR-13246 76
P-ERR-13247 76
P-ERR-13248 77
P-ERR-13260 78
P-ERR-13261 78
P-ERR-13262 78
P-ERR-13263 78
P-ERR-13264 79
P-ERR-13265 79
P-ERR-13266 79
P-ERR-13280 79
P-ERR-13281 79
P-ERR-13282 80
P-ERR-13283 80
P-ERR-13284 80
P-ERR-13285 80
P-ERR-13286 80
P-ERR-13287 81
P-ERR-13288 81
P-ERR-13289 81
P-ERR-13300 81
P-ERR-13301 82
P-ERR-13302 82
P-ERR-13303 82
P-ERR-13304 82
P-ERR-13305 82
P-ERR-13306 83
P-ERR-13307 83
P-ERR-13308 83
P-ERR-13309 83
P-ERR-13310 84
P-ERR-13311 84
P-ERR-13312 84
P-ERR-13313 84
P-ERR-13314 85
P-ERR-13315 85
P-ERR-13316 85
P-ERR-13317 85
P-ERR-13318 85
P-ERR-13319 86
P-ERR-13320 86
P-ERR-13321 86
P-ERR-13322 86
P-ERR-13323 87
P-ERR-13324 87
P-ERR-13325 87
P-ERR-13326 88
P-ERR-13327 88
P-ERR-13328 89
P-ERR-13329 89
P-ERR-13330 89
P-ERR-13340 89
P-ERR-13341 90
P-ERR-13342 90
P-ERR-13343 90
P-ERR-13344 90
P-ERR-13345 91
P-ERR-13346 91
P-ERR-13360 91
P-ERR-13361 92
P-ERR-13362 92
P-ERR-13363 92
P-ERR-13364 92
P-ERR-13365 93
P-ERR-13366 93
P-ERR-13380 93
P-ERR-13381 93
P-ERR-13382 94
P-ERR-13383 94
P-ERR-13384 94
P-ERR-13385 94
P-ERR-13400 94
P-ERR-13401 95
P-ERR-13402 95
P-ERR-13403 95
P-ERR-13404 95
P-ERR-13405 95
P-ERR-13406 96
P-ERR-13407 96
P-ERR-13408 96
P-ERR-13409 96
P-ERR-13410 97
P-ERR-13411 97
P-ERR-13412 97
P-ERR-13413 97
P-ERR-13414 97
P-ERR-13415 98
P-ERR-13416 98
P-ERR-13417 98
P-ERR-13418 98
P-ERR-13419 98

P-ERR-13420 99
P-ERR-13421 99
P-ERR-13422 99
P-ERR-13423 99
P-ERR-13424 99
P-ERR-13425 100
P-ERR-13426 100
P-ERR-13427 100
P-ERR-13428 100
P-ERR-13429 101
P-ERR-13430 101
P-ERR-13431 101
P-ERR-13432 101
P-ERR-13433 101
P-ERR-13434 102
P-ERR-13435 102
P-ERR-13436 102
P-ERR-13437 102
P-ERR-13438 102
P-ERR-13439 103
P-ERR-13440 103
P-ERR-13441 103
P-ERR-13443 103
P-ERR-13444 104
P-ERR-13445 104
P-ERR-13446 104
P-ERR-13447 104
P-ERR-13448 105
P-ERR-13449 105
P-ERR-13450 105
P-ERR-13451 105
P-ERR-13452 105
P-ERR-13453 106
P-ERR-13454 106
P-ERR-13455 106
P-ERR-13456 106
P-ERR-13457 106
P-ERR-13458 107
P-ERR-13459 107
P-ERR-13460 107
P-ERR-13461 107
P-ERR-13462 107
P-ERR-13501 109
P-ERR-13502 109
P-ERR-13503 109
P-ERR-13504 109
P-ERR-13505 109
P-ERR-13506 110
P-ERR-13507 110
P-ERR-13508 110
P-ERR-13509 110
P-ERR-13551 110
P-ERR-13552 111
P-ERR-13553 111
P-ERR-13554 111
P-ERR-13555 112
P-ERR-13556 112
P-ERR-13557 112
P-ERR-13558 112
P-ERR-13559 113
P-ERR-13560 113
P-ERR-13561 113
P-ERR-13562 114
P-ERR-13563 114
P-ERR-13600 114
P-ERR-13601 114
P-ERR-13602 115
P-ERR-13603 115
P-ERR-13604 115
P-ERR-13605 115
P-ERR-13606 116
P-ERR-14000 117
P-ERR-14001 117
P-ERR-140017 1561
P-ERR-14002 117
P-ERR-140021 1561
P-ERR-140023 1562
P-ERR-140024 1562
P-ERR-140026 1562
P-ERR-140027 1563
P-ERR-140029 1563
P-ERR-140030 1563
P-ERR-140031 1563
P-ERR-140032 1564
P-ERR-140033 1564
P-ERR-140034 1564
P-ERR-140035 1564
P-ERR-140036 1565
P-ERR-140037 1565
P-ERR-140038 1566
P-ERR-140039 1566
P-ERR-140040 1566
P-ERR-140041 1567
P-ERR-140042 1567
P-ERR-140043 1567
P-ERR-140048 1568
P-ERR-140050 1568
P-ERR-140051 1568
P-ERR-140052 1568
P-ERR-140053 1569
P-ERR-140054 1569
P-ERR-140055 1569
P-ERR-140057 1570
P-ERR-140058 1570
P-ERR-140059 1571
P-ERR-140060 1571
P-ERR-14020 117
P-ERR-14021 118
P-ERR-14040 118
P-ERR-14041 118
P-ERR-14042 119
P-ERR-14043 119
P-ERR-14060 119
P-ERR-14061 119
P-ERR-14062 120
P-ERR-14063 120
P-ERR-14064 121
P-ERR-14065 121
P-ERR-14066 121
P-ERR-14067 121
P-ERR-14068 121
P-ERR-14069 122
P-ERR-14080 122
P-ERR-14081 122
P-ERR-14082 122
P-ERR-14083 122

P-ERR-14084 123
P-ERR-14085 123
P-ERR-14086 123
P-ERR-14087 123
P-ERR-14088 123
P-ERR-14089 124
P-ERR-14090 124
P-ERR-14091 124
P-ERR-14092 124
P-ERR-14093 124
P-ERR-14094 125
P-ERR-14095 125
P-ERR-14096 125
P-ERR-14097 125
P-ERR-14098 126
P-ERR-14099 126
P-ERR-14100 126
P-ERR-14101 126
P-ERR-14102 126
P-ERR-14103 127
P-ERR-14104 127
P-ERR-14105 127
P-ERR-14106 127
P-ERR-14107 127
P-ERR-14108 128
P-ERR-14109 128
P-ERR-14110 128
P-ERR-14111 128
P-ERR-14112 128
P-ERR-14113 129
P-ERR-14114 129
P-ERR-14300 130
P-ERR-14301 130
P-ERR-14302 130
P-ERR-14303 130
P-ERR-14304 130
P-ERR-14305 131
P-ERR-14306 131
P-ERR-14307 131
P-ERR-14308 131
P-ERR-14309 132
P-ERR-14310 132
P-ERR-14311 132
P-ERR-14312 132
P-ERR-14313 132
P-ERR-14314 133
P-ERR-14315 133
P-ERR-14316 133
P-ERR-14317 133
P-ERR-14318 133
P-ERR-14319 134
P-ERR-14320 134
P-ERR-14321 134
P-ERR-14322 134
P-ERR-14323 135
P-ERR-14324 135
P-ERR-14325 135
P-ERR-14326 135
P-ERR-14327 136
P-ERR-14328 136
P-ERR-14329 136
P-ERR-14330 136
P-ERR-14331 137
P-ERR-14332 137
P-ERR-14333 137
P-ERR-14334 137
P-ERR-14335 138
P-ERR-14336 138
P-ERR-14337 138
P-ERR-14338 138
P-ERR-14339 139
P-ERR-14340 139
P-ERR-14341 139
P-ERR-14342 139
P-ERR-14343 140
P-ERR-14344 140
P-ERR-14345 140
P-ERR-14346 140
P-ERR-14347 141
P-ERR-14348 141
P-ERR-14349 141
P-ERR-14350 142
P-ERR-14351 142
P-ERR-14352 142
P-ERR-14353 142
P-ERR-14354 143
P-ERR-14355 143
P-ERR-14356 167
P-ERR-14357 143
P-ERR-14358 144
P-ERR-14359 144
P-ERR-14360 144
P-ERR-14361 144
P-ERR-14362 145
P-ERR-14363 145
P-ERR-14364 145
P-ERR-14365 145
P-ERR-14366 145
P-ERR-14368 146
P-ERR-14369 146
P-ERR-14370 146
P-ERR-14371 146
P-ERR-14372 147
P-ERR-14373 147
P-ERR-14374 147
P-ERR-14375 147
P-ERR-14376 148
P-ERR-14377 148
P-ERR-14378 148
P-ERR-14386 148
P-ERR-14388 148
P-ERR-14389 149
P-ERR-14390 149
P-ERR-14391 149
P-ERR-14392 149
P-ERR-14393 150
P-ERR-14394 150
P-ERR-14395 150
P-ERR-14396 151
P-ERR-14397 151
P-ERR-14398 151
P-ERR-14399 151
P-ERR-14400 152
P-ERR-14401 152
P-ERR-14402 152
P-ERR-14403 152

P-ERR-14404 153
P-ERR-14405 153
P-ERR-14406 153
P-ERR-14407 153
P-ERR-14408 154
P-ERR-14409 154
P-ERR-14410 154
P-ERR-14411 154
P-ERR-14412 155
P-ERR-14413 155
P-ERR-14414 155
P-ERR-14415 155
P-ERR-14416 156
P-ERR-14418 156
P-ERR-14419 156
P-ERR-14420 156
P-ERR-14421 156
P-ERR-14422 157
P-ERR-14423 157
P-ERR-14424 157
P-ERR-14425 157
P-ERR-14426 158
P-ERR-14427 158
P-ERR-14428 158
P-ERR-14429 158
P-ERR-14430 158
P-ERR-14431 159
P-ERR-14432 159
P-ERR-14433 159
P-ERR-14434 159
P-ERR-14435 160
P-ERR-14436 160
P-ERR-14437 160
P-ERR-14438 160
P-ERR-14439 160
P-ERR-14440 161
P-ERR-14441 161
P-ERR-14442 161
P-ERR-14443 161
P-ERR-14444 161
P-ERR-14445 162
P-ERR-14447 162
P-ERR-14448 162
P-ERR-14449 162
P-ERR-14450 162
P-ERR-14451 163
P-ERR-14452 163
P-ERR-14453 163
P-ERR-14461 164
P-ERR-14462 164
P-ERR-14463 164
P-ERR-14477 164
P-ERR-14478 165
P-ERR-14479 165
P-ERR-14480 165
P-ERR-14481 165
P-ERR-14482 166
P-ERR-14483 166
P-ERR-14484 166
P-ERR-14485 166
P-ERR-14486 167
P-ERR-150000 1572
P-ERR-150002 1572
P-ERR-150004 1572
P-ERR-150005 1573
P-ERR-150006 1573
P-ERR-150007 1574
P-ERR-150008 1574
P-ERR-150009 1575
P-ERR-150010 1575
P-ERR-150011 1575
P-ERR-150012 1576
P-ERR-150024 1576
P-ERR-150025 1577
P-ERR-150033 1577
P-ERR-150039 1577
P-ERR-150040 1578
P-ERR-150041 1578
P-ERR-150042 1579
P-ERR-150043 1579
P-ERR-150047 1580
P-ERR-150048 1580
P-ERR-150049 1581
P-ERR-150050 1581
P-ERR-150051 1582
P-ERR-150053 1583
P-ERR-150054 1583
P-ERR-150057 1584
P-ERR-150058 1584
P-ERR-150059 1585
P-ERR-150061 1586
P-ERR-150087 1586
P-ERR-150088 1587
P-ERR-150089 1587
P-ERR-150090 1588
P-ERR-150091 1588
P-ERR-150092 1588
P-ERR-150099 1589
P-ERR-150100 1589
P-ERR-150101 1590
P-ERR-150104 1590
P-ERR-150105 1591
P-ERR-150106 1591
P-ERR-150107 1592
P-ERR-150112 1592
P-ERR-150113 1592
P-ERR-150114 1593
P-ERR-150115 1593
P-ERR-1599 24
P-ERR-1624 24
P-ERR-1700 25
P-ERR-1701 25
P-ERR-1702 26
P-ERR-1703 26
P-ERR-1704 26
P-ERR-1705 27
P-ERR-1706 27
P-ERR-1708 27
P-ERR-1709 28
P-ERR-20003 168
P-ERR-20007 168
P-ERR-20008 169
P-ERR-20010 169
P-ERR-20011 169
P-ERR-20012 170
P-ERR-20013 170

P-ERR-20014 171
P-ERR-20015 171
P-ERR-20016 172
P-ERR-20017 172
P-ERR-20019 172
P-ERR-20020 172
P-ERR-20022 173
P-ERR-20023 173
P-ERR-20024 174
P-ERR-20025 174
P-ERR-20028 175
P-ERR-20029 175
P-ERR-20030 175
P-ERR-20032 176
P-ERR-20033 176
P-ERR-20034 177
P-ERR-20035 177
P-ERR-20036 178
P-ERR-20037 178
P-ERR-20038 179
P-ERR-20041 179
P-ERR-20042 180
P-ERR-20044 180
P-ERR-20045 180
P-ERR-20048 181
P-ERR-20049 181
P-ERR-20050 181
P-ERR-20051 182
P-ERR-20052 182
P-ERR-20054 183
P-ERR-20057 184
P-ERR-20063 184
P-ERR-20064 184
P-ERR-20065 185
P-ERR-20066 185
P-ERR-20075 185
P-ERR-20076 186
P-ERR-20078 186
P-ERR-20079 186
P-ERR-20083 187
P-ERR-20085 187
P-ERR-20087 188
P-ERR-20088 189
P-ERR-2009 29
P-ERR-20092 189
P-ERR-20097 190
P-ERR-20100 190
P-ERR-20101 191
P-ERR-20103 192
P-ERR-20105 192
P-ERR-20106 192
P-ERR-20107 193
P-ERR-20108 193
P-ERR-20109 193
P-ERR-20110 194
P-ERR-20111 194
P-ERR-20112 194
P-ERR-20113 195
P-ERR-20114 195
P-ERR-20115 195
P-ERR-20116 196
P-ERR-20117 196
P-ERR-20118 196
P-ERR-20119 197
P-ERR-20120 197
P-ERR-20121 197
P-ERR-20122 197
P-ERR-20123 198
P-ERR-20124 198
P-ERR-20125 199
P-ERR-20126 199
P-ERR-20127 199
P-ERR-20128 200
P-ERR-20129 200
P-ERR-20130 200
P-ERR-20131 201
P-ERR-20133 201
P-ERR-20136 201
P-ERR-20137 202
P-ERR-20138 202
P-ERR-20147 203
P-ERR-20149 203
P-ERR-20151 203
P-ERR-20154 204
P-ERR-20155 204
P-ERR-20156 205
P-ERR-20157 205
P-ERR-20158 205
P-ERR-20160 206
P-ERR-20161 207
P-ERR-20162 207
P-ERR-20165 207
P-ERR-20166 208
P-ERR-20167 208
P-ERR-20169 209
P-ERR-20170 209
P-ERR-20171 210
P-ERR-20172 210
P-ERR-20175 211
P-ERR-20176 211
P-ERR-20177 212
P-ERR-20178 212
P-ERR-20180 212
P-ERR-20181 213
P-ERR-20182 213
P-ERR-20183 214
P-ERR-20186 214
P-ERR-20187 215
P-ERR-20188 216
P-ERR-20191 216
P-ERR-20192 216
P-ERR-20193 217
P-ERR-20194 217
P-ERR-20195 218
P-ERR-20196 218
P-ERR-20197 218
P-ERR-20198 219
P-ERR-20200 219
P-ERR-20203 220
P-ERR-20205 220
P-ERR-20206 221
P-ERR-20207 221
P-ERR-20209 222
P-ERR-20211 222
P-ERR-20212 223
P-ERR-20214 223

P-ERR-20215 224
P-ERR-20216 224
P-ERR-20217 225
P-ERR-20218 225
P-ERR-20219 226
P-ERR-20220 226
P-ERR-20221 227
P-ERR-20222 227
P-ERR-20223 228
P-ERR-20224 229
P-ERR-20225 229
P-ERR-20226 230
P-ERR-20227 230
P-ERR-20228 231
P-ERR-20229 231
P-ERR-2023 29
P-ERR-20230 232
P-ERR-20231 232
P-ERR-20232 232
P-ERR-20233 233
P-ERR-20234 233
P-ERR-20235 234
P-ERR-20236 235
P-ERR-20237 235
P-ERR-20238 235
P-ERR-20239 236
P-ERR-2024 29
P-ERR-20240 237
P-ERR-20241 237
P-ERR-20242 238
P-ERR-20243 238
P-ERR-20244 239
P-ERR-20245 239
P-ERR-20260 241
P-ERR-20263 241
P-ERR-20264 241
P-ERR-20265 242
P-ERR-20267 242
P-ERR-20282 243
P-ERR-20283 243
P-ERR-20288 243
P-ERR-20290 244
P-ERR-20296 244
P-ERR-20297 244
P-ERR-2030 30
P-ERR-2031 30
P-ERR-20312 245
P-ERR-20317 245
P-ERR-2032 30
P-ERR-2033 30
P-ERR-20339 246
P-ERR-2034 30
P-ERR-20347 246
P-ERR-2035 30
P-ERR-20351 246
P-ERR-20352 247
P-ERR-20353 247
P-ERR-20354 248
P-ERR-2036 31
P-ERR-20360 249
P-ERR-20361 249
P-ERR-20362 249
P-ERR-20364 249
P-ERR-20365 250
P-ERR-20367 250
P-ERR-20368 251
P-ERR-20369 251
P-ERR-2037 31
P-ERR-20371 252
P-ERR-20372 253
P-ERR-20373 253
P-ERR-20375 253
P-ERR-20376 254
P-ERR-20377 255
P-ERR-2038 31
P-ERR-20381 255
P-ERR-20385 256
P-ERR-20386 256
P-ERR-20387 256
P-ERR-2039 31
P-ERR-20392 257
P-ERR-20394 257
P-ERR-20396 257
P-ERR-20397 258
P-ERR-20398 258
P-ERR-20399 258
P-ERR-2040 31
P-ERR-20400 259
P-ERR-20401 259
P-ERR-20402 260
P-ERR-20403 260
P-ERR-20407 260
P-ERR-20408 261
P-ERR-20409 261
P-ERR-2041 32
P-ERR-20410 261
P-ERR-20418 262
P-ERR-20419 262
P-ERR-2042 32
P-ERR-20420 262
P-ERR-20423 262
P-ERR-20424 263
P-ERR-20425 263
P-ERR-20426 263
P-ERR-20427 264
P-ERR-20428 264
P-ERR-20429 265
P-ERR-2043 32
P-ERR-20430 265
P-ERR-20434 265
P-ERR-20435 266
P-ERR-20436 266
P-ERR-20437 266
P-ERR-20438 267
P-ERR-20439 267
P-ERR-2044 32
P-ERR-20440 268
P-ERR-20441 268
P-ERR-20442 268
P-ERR-20450 269
P-ERR-20457 270
P-ERR-2046 33
P-ERR-20461 270
P-ERR-20464 271
P-ERR-20465 271
P-ERR-20466 271

P-ERR-20467 271
P-ERR-20468 272
P-ERR-20469 272
P-ERR-20470 273
P-ERR-20472 273
P-ERR-20475 274
P-ERR-20476 274
P-ERR-20477 275
P-ERR-20478 275
P-ERR-2048 33
P-ERR-20480 275
P-ERR-20481 276
P-ERR-20482 276
P-ERR-20483 276
P-ERR-20485 277
P-ERR-20486 277
P-ERR-20487 278
P-ERR-20488 278
P-ERR-2049 34
P-ERR-20490 278
P-ERR-20491 279
P-ERR-20493 279
P-ERR-20494 279
P-ERR-20495 280
P-ERR-20496 280
P-ERR-20498 281
P-ERR-20499 281
P-ERR-2050 34
P-ERR-20500 282
P-ERR-20506 282
P-ERR-20509 282
P-ERR-2051 34
P-ERR-2052 34
P-ERR-20521 282
P-ERR-20524 283
P-ERR-2053 35, 36
P-ERR-20535 283
P-ERR-2054 35
P-ERR-20545 283
P-ERR-20546 284
P-ERR-20547 285
P-ERR-20551 285
P-ERR-2056 35
P-ERR-20566 285
P-ERR-20567 286
P-ERR-20568 287
P-ERR-20569 287
P-ERR-2057 36
P-ERR-20570 288
P-ERR-20571 288
P-ERR-20574 288
P-ERR-20577 289
P-ERR-20578 290
P-ERR-20582 290
P-ERR-20583 290
P-ERR-20584 291
P-ERR-20585 291
P-ERR-20587 292
P-ERR-2059 36
P-ERR-20590 292
P-ERR-20591 293
P-ERR-20592 294
P-ERR-20593 294
P-ERR-20595 295
P-ERR-20601 295
P-ERR-20603 296
P-ERR-20604 296
P-ERR-20605 296
P-ERR-20606 297
P-ERR-20607 297
P-ERR-20608 298
P-ERR-20610 298
P-ERR-20612 299
P-ERR-20613 299
P-ERR-20614 299
P-ERR-20615 300
P-ERR-20616 300
P-ERR-20617 300
P-ERR-20618 301
P-ERR-20619 301
P-ERR-2062 36
P-ERR-20620 301
P-ERR-20622 302
P-ERR-20624 302
P-ERR-20625 303
P-ERR-20626 303
P-ERR-20629 303
P-ERR-2063 37
P-ERR-20630 304
P-ERR-20633 304
P-ERR-20634 304
P-ERR-20635 305
P-ERR-20636 305
P-ERR-20637 305
P-ERR-20638 305
P-ERR-20639 306
P-ERR-20640 306
P-ERR-20641 306
P-ERR-20642 306
P-ERR-20643 307
P-ERR-20644 307
P-ERR-20648 307
P-ERR-2065 37
P-ERR-20650 308
P-ERR-20651 308
P-ERR-20652 309
P-ERR-20653 309
P-ERR-20654 310
P-ERR-20657 310
P-ERR-20658 311
P-ERR-20659 311
P-ERR-20660 311
P-ERR-20662 312
P-ERR-20663 312
P-ERR-20664 313
P-ERR-20665 314
P-ERR-20666 314
P-ERR-20668 314
P-ERR-20669 315
P-ERR-20671 315
P-ERR-20675 316
P-ERR-20676 316
P-ERR-20680 317
P-ERR-20683 318
P-ERR-20684 318
P-ERR-20685 319

P-ERR-20686 319
P-ERR-20687 319
P-ERR-20689 320
P-ERR-20690 320
P-ERR-20691 321
P-ERR-20692 321
P-ERR-20693 321
P-ERR-20695 321
P-ERR-20699 322
P-ERR-20701 322
P-ERR-20702 323
P-ERR-20703 323
P-ERR-20704 323
P-ERR-20705 324
P-ERR-20706 324
P-ERR-20707 324
P-ERR-20711 325
P-ERR-20712 325
P-ERR-20713 326
P-ERR-20715 326
P-ERR-20716 326
P-ERR-20718 327
P-ERR-20721 327
P-ERR-20723 328
P-ERR-20724 328
P-ERR-20726 328
P-ERR-20734 329
P-ERR-20737 329
P-ERR-20738 330
P-ERR-20739 330
P-ERR-20740 330
P-ERR-20741 331
P-ERR-20743 331
P-ERR-20745 331
P-ERR-20746 331
P-ERR-20747 332
P-ERR-20748 332
P-ERR-20751 333
P-ERR-20752 333
P-ERR-20756 334
P-ERR-20758 334
P-ERR-20760 335
P-ERR-20762 335
P-ERR-20764 335
P-ERR-20765 336
P-ERR-20766 336
P-ERR-20770 337
P-ERR-20771 337
P-ERR-20772 337
P-ERR-20773 338
P-ERR-20774 338
P-ERR-20775 339
P-ERR-20776 339
P-ERR-20777 339
P-ERR-20780 340
P-ERR-20781 340
P-ERR-20782 340
P-ERR-20786 341
P-ERR-20788 341
P-ERR-20789 342
P-ERR-20790 342
P-ERR-20791 342
P-ERR-20792 342
P-ERR-20794 343
P-ERR-20795 343
P-ERR-20796 343
P-ERR-20797 344
P-ERR-20798 344
P-ERR-20799 344
P-ERR-20800 344
P-ERR-20801 345
P-ERR-20802 345
P-ERR-20803 346
P-ERR-20804 346
P-ERR-20805 346
P-ERR-20806 346
P-ERR-20809 346
P-ERR-20810 347
P-ERR-20812 347
P-ERR-20813 347
P-ERR-20814 348
P-ERR-20816 348
P-ERR-20819 348
P-ERR-20820 349
P-ERR-20821 349
P-ERR-20822 349
P-ERR-20823 350
P-ERR-20824 350
P-ERR-20825 350
P-ERR-20826 350
P-ERR-20829 351
P-ERR-20831 352
P-ERR-20833 352
P-ERR-20834 352
P-ERR-20836 353
P-ERR-20837 353
P-ERR-20838 353
P-ERR-20839 354
P-ERR-20840 354
P-ERR-20841 354
P-ERR-20842 355
P-ERR-20844 355
P-ERR-20845 355
P-ERR-20846 356
P-ERR-20847 356
P-ERR-20851 356
P-ERR-20852 357
P-ERR-20856 357
P-ERR-20857 357
P-ERR-20860 357
P-ERR-20861 358
P-ERR-20862 358
P-ERR-20863 358
P-ERR-20864 359
P-ERR-20865 359
P-ERR-20868 359
P-ERR-20870 359
P-ERR-20872 360
P-ERR-20875 360
P-ERR-20876 361
P-ERR-20877 361
P-ERR-20879 361
P-ERR-20880 361
P-ERR-20881 362
P-ERR-20885 362
P-ERR-20886 362

P-ERR-20887 362
P-ERR-20889 363
P-ERR-20890 363
P-ERR-20891 364
P-ERR-20893 364
P-ERR-20894 364
P-ERR-20895 364
P-ERR-20897 365
P-ERR-20899 365
P-ERR-20900 365
P-ERR-20901 365
P-ERR-20902 366
P-ERR-20903 366
P-ERR-20904 367
P-ERR-20906 367
P-ERR-20908 368
P-ERR-20909 368
P-ERR-20910 368
P-ERR-20912 368
P-ERR-20913 368
P-ERR-20915 369
P-ERR-20916 369
P-ERR-20917 369
P-ERR-20918 369
P-ERR-20919 370
P-ERR-20920 370
P-ERR-20921 371
P-ERR-20924 371
P-ERR-20925 371
P-ERR-20927 372
P-ERR-20930 372
P-ERR-20931 372
P-ERR-20932 372
P-ERR-20933 373
P-ERR-20934 373
P-ERR-20939 374
P-ERR-20940 374
P-ERR-20941 374
P-ERR-20942 375
P-ERR-20943 375
P-ERR-20944 375
P-ERR-20953 376
P-ERR-20954 376
P-ERR-20955 376
P-ERR-20956 376
P-ERR-20958 377
P-ERR-20959 377
P-ERR-20961 377
P-ERR-20962 378
P-ERR-20963 378
P-ERR-20965 378
P-ERR-20966 378
P-ERR-20967 379
P-ERR-20971 379
P-ERR-20972 380
P-ERR-20974 380
P-ERR-20977 380
P-ERR-20978 381
P-ERR-20980 381
P-ERR-20981 381
P-ERR-20984 382
P-ERR-20985 382
P-ERR-20986 382
P-ERR-20990 382
P-ERR-20991 383
P-ERR-20994 383
P-ERR-20995 383
P-ERR-20996 384
P-ERR-20997 384
P-ERR-20998 384
P-ERR-20999 385
P-ERR-2100 37
P-ERR-21000 386
P-ERR-21020 386
P-ERR-21022 386
P-ERR-21023 387
P-ERR-21024 387
P-ERR-21027 387
P-ERR-21028 388
P-ERR-21029 388
P-ERR-21035 389
P-ERR-21036 389
P-ERR-21037 389
P-ERR-21038 389
P-ERR-21039 390
P-ERR-21040 390
P-ERR-21041 390
P-ERR-21042 390
P-ERR-21044 391
P-ERR-21045 391
P-ERR-21047 392
P-ERR-21050 392
P-ERR-21056 393
P-ERR-21057 393
P-ERR-21058 393
P-ERR-21059 393
P-ERR-21060 394
P-ERR-21061 394
P-ERR-21067 394
P-ERR-21068 394
P-ERR-21071 395
P-ERR-21072 395
P-ERR-21076 395
P-ERR-21077 396
P-ERR-21078 396
P-ERR-21079 396
P-ERR-21080 396
P-ERR-21081 397
P-ERR-21083 397
P-ERR-21085 397
P-ERR-21086 397
P-ERR-21087 398
P-ERR-21088 398
P-ERR-21089 398
P-ERR-21090 399
P-ERR-21094 399
P-ERR-21095 399
P-ERR-21096 400
P-ERR-21097 400
P-ERR-21098 400
P-ERR-21099 400
P-ERR-21100 401
P-ERR-21101 401
P-ERR-21102 402
P-ERR-21103 402
P-ERR-21104 402

P-ERR-21105 402
P-ERR-21106 403
P-ERR-21107 403
P-ERR-21110 403
P-ERR-21111 403
P-ERR-21120 404
P-ERR-21121 404
P-ERR-21122 404
P-ERR-21123 405
P-ERR-21126 405
P-ERR-21128 405
P-ERR-21129 406
P-ERR-21130 406
P-ERR-21131 406
P-ERR-21132 407
P-ERR-21134 407
P-ERR-21135 407
P-ERR-21136 407
P-ERR-21137 408
P-ERR-21138 408
P-ERR-21139 408
P-ERR-21141 408
P-ERR-21142 409
P-ERR-21143 409
P-ERR-21144 409
P-ERR-21146 410
P-ERR-21147 410
P-ERR-21151 410
P-ERR-21152 410
P-ERR-21157 411
P-ERR-21158 411
P-ERR-21159 411
P-ERR-21164 412
P-ERR-21165 412
P-ERR-21167 412
P-ERR-21168 412
P-ERR-21169 413
P-ERR-21170 413
P-ERR-21171 413
P-ERR-21173 414
P-ERR-21174 414
P-ERR-21175 414
P-ERR-21176 415
P-ERR-21177 415
P-ERR-21178 415
P-ERR-21179 415
P-ERR-21180 416
P-ERR-21182 416
P-ERR-21184 417
P-ERR-21185 417
P-ERR-21187 417
P-ERR-21188 418
P-ERR-21189 418
P-ERR-21190 418
P-ERR-21192 419
P-ERR-21193 419
P-ERR-21196 420
P-ERR-21197 420
P-ERR-21198 420
P-ERR-21199 421
P-ERR-21200 421
P-ERR-21201 421
P-ERR-21202 421
P-ERR-21203 422
P-ERR-21204 422
P-ERR-21205 423
P-ERR-21206 423
P-ERR-21207 423
P-ERR-21208 423
P-ERR-21209 424
P-ERR-21210 424
P-ERR-21211 424
P-ERR-21212 424
P-ERR-21213 425
P-ERR-21215 425
P-ERR-21216 425
P-ERR-21217 425
P-ERR-21218 426
P-ERR-21219 426
P-ERR-21220 426
P-ERR-21230 426
P-ERR-21231 427
P-ERR-21232 427
P-ERR-21234 427
P-ERR-21235 428
P-ERR-21236 428
P-ERR-21237 428
P-ERR-21238 428
P-ERR-21239 429
P-ERR-21240 429
P-ERR-21241 430
P-ERR-21242 430
P-ERR-21243 430
P-ERR-21244 430
P-ERR-21247 431
P-ERR-21248 431
P-ERR-21249 431
P-ERR-21250 433
P-ERR-21251 433
P-ERR-21252 433
P-ERR-21253 433
P-ERR-21254 434
P-ERR-21255 434
P-ERR-21256 434
P-ERR-21257 435
P-ERR-21258 435
P-ERR-21259 435
P-ERR-21260 435
P-ERR-21261 436
P-ERR-21262 436
P-ERR-21273 436
P-ERR-21274 437
P-ERR-21275 437
P-ERR-21276 437
P-ERR-21277 437
P-ERR-21279 438
P-ERR-21280 438
P-ERR-21281 439
P-ERR-21282 439
P-ERR-21283 440
P-ERR-21284 440
P-ERR-21285 440
P-ERR-21286 441
P-ERR-21287 441
P-ERR-21288 441
P-ERR-21289 442

P-ERR-21290 442
P-ERR-21291 442
P-ERR-21292 443
P-ERR-21293 443
P-ERR-21294 443
P-ERR-21295 443
P-ERR-21296 444
P-ERR-21297 444
P-ERR-21298 444
P-ERR-21299 444
P-ERR-21300 445
P-ERR-21301 445
P-ERR-21302 445
P-ERR-21303 446
P-ERR-21304 446
P-ERR-21305 447
P-ERR-21306 447
P-ERR-21307 447
P-ERR-21308 448
P-ERR-21309 448
P-ERR-21310 448
P-ERR-21311 449
P-ERR-21312 449
P-ERR-21313 450
P-ERR-21314 450
P-ERR-21315 451
P-ERR-21316 451
P-ERR-21317 451
P-ERR-21318 452
P-ERR-21319 452
P-ERR-21320 452
P-ERR-21321 452
P-ERR-21322 453
P-ERR-21323 453
P-ERR-21326 454
P-ERR-21327 454
P-ERR-21328 455
P-ERR-21329 455
P-ERR-21330 455
P-ERR-21332 456
P-ERR-21333 456
P-ERR-21335 456
P-ERR-21336 457
P-ERR-21337 457
P-ERR-21338 457
P-ERR-21339 457
P-ERR-21340 457
P-ERR-21341 458
P-ERR-21342 458
P-ERR-21343 458
P-ERR-21344 459
P-ERR-21345 459
P-ERR-21346 459
P-ERR-21347 459
P-ERR-21348 460
P-ERR-21349 460
P-ERR-21350 460
P-ERR-21351 461
P-ERR-21352 461
P-ERR-21353 461
P-ERR-21354 462
P-ERR-21355 462
P-ERR-21356 462
P-ERR-21357 462
P-ERR-21358 463
P-ERR-21359 463
P-ERR-21360 463
P-ERR-21362 464
P-ERR-21363 464
P-ERR-21365 465
P-ERR-21369 465
P-ERR-21371 466
P-ERR-21372 466
P-ERR-21373 466
P-ERR-21374 466
P-ERR-21375 467
P-ERR-21376 467
P-ERR-21377 467
P-ERR-21379 468
P-ERR-21380 468
P-ERR-21381 469
P-ERR-21382 469
P-ERR-21383 470
P-ERR-21384 470
P-ERR-21385 471
P-ERR-21386 471
P-ERR-21387 471
P-ERR-21388 472
P-ERR-21389 472
P-ERR-21390 473
P-ERR-21391 474
P-ERR-21392 474
P-ERR-21394 475
P-ERR-21395 475
P-ERR-21396 475
P-ERR-21397 475
P-ERR-21398 476
P-ERR-21399 476
P-ERR-21400 477
P-ERR-21401 477
P-ERR-21403 478
P-ERR-21404 478
P-ERR-21405 479
P-ERR-21406 479
P-ERR-21407 479
P-ERR-21408 480
P-ERR-21409 480
P-ERR-21410 481
P-ERR-21411 481
P-ERR-21412 481
P-ERR-21413 482
P-ERR-21414 482
P-ERR-21415 482
P-ERR-21416 483
P-ERR-21417 483
P-ERR-21418 483
P-ERR-21419 484
P-ERR-21420 484
P-ERR-21422 484
P-ERR-21423 433
P-ERR-21424 484
P-ERR-21425 485
P-ERR-21426 485
P-ERR-21427 486
P-ERR-21429 486
P-ERR-21430 487

P-ERR-21431 487
P-ERR-21432 487
P-ERR-21433 488
P-ERR-21434 488
P-ERR-21435 489
P-ERR-21436 489
P-ERR-21437 490
P-ERR-21438 490
P-ERR-21439 490
P-ERR-21440 491
P-ERR-21441 491
P-ERR-21442 492
P-ERR-21443 492
P-ERR-21444 492
P-ERR-21445 493
P-ERR-21446 493
P-ERR-21447 494
P-ERR-21451 494
P-ERR-21452 494
P-ERR-21453 495
P-ERR-21454 495
P-ERR-21455 495
P-ERR-21456 496
P-ERR-21457 496
P-ERR-21458 496
P-ERR-21459 497
P-ERR-21460 497
P-ERR-21461 498
P-ERR-21462 498
P-ERR-21463 498
P-ERR-21464 499
P-ERR-21465 499
P-ERR-21466 499
P-ERR-21467 499
P-ERR-21468 500
P-ERR-21469 500
P-ERR-21470 501
P-ERR-21471 501
P-ERR-21472 502
P-ERR-21474 503
P-ERR-21476 503
P-ERR-21477 503
P-ERR-21478 504
P-ERR-21479 504
P-ERR-21480 504
P-ERR-21481 504
P-ERR-21482 505
P-ERR-21483 505
P-ERR-21484 505
P-ERR-21485 506
P-ERR-21486 506
P-ERR-21487 506
P-ERR-21488 506
P-ERR-21489 507
P-ERR-21490 507
P-ERR-21491 507
P-ERR-21492 508
P-ERR-21493 508
P-ERR-21494 508
P-ERR-21495 509
P-ERR-21496 509
P-ERR-21497 509
P-ERR-21498 510
P-ERR-21499 510
P-ERR-21500 511
P-ERR-21501 511
P-ERR-21502 512
P-ERR-21504 513
P-ERR-21505 513
P-ERR-21506 514
P-ERR-21507 514
P-ERR-21508 514
P-ERR-21509 514
P-ERR-21510 515
P-ERR-21511 515
P-ERR-21512 516
P-ERR-21513 517
P-ERR-21514 518
P-ERR-21515 518
P-ERR-21516 519
P-ERR-21517 519
P-ERR-21518 519
P-ERR-21519 520
P-ERR-21520 521
P-ERR-21521 521
P-ERR-21522 522
P-ERR-21523 522
P-ERR-21524 522
P-ERR-21525 522
P-ERR-21526 523
P-ERR-21527 523
P-ERR-21528 524
P-ERR-21529 525
P-ERR-21530 525
P-ERR-21531 525
P-ERR-21532 526
P-ERR-21533 526
P-ERR-21534 526
P-ERR-21535 527
P-ERR-21537 527
P-ERR-21538 527
P-ERR-21539 528
P-ERR-21540 528
P-ERR-21541 528
P-ERR-21542 529
P-ERR-21543 529
P-ERR-21544 529
P-ERR-21545 530
P-ERR-21546 530
P-ERR-21547 530
P-ERR-21548 531
P-ERR-21549 531
P-ERR-21550 532
P-ERR-21551 532
P-ERR-21552 532
P-ERR-21553 533
P-ERR-21555 533
P-ERR-21556 534
P-ERR-21557 534
P-ERR-21558 534
P-ERR-21559 534
P-ERR-21560 534
P-ERR-21561 535
P-ERR-21562 535
P-ERR-21563 535
P-ERR-21564 535

P-ERR-21565 536
P-ERR-21566 536
P-ERR-21567 536
P-ERR-21568 536
P-ERR-21569 537
P-ERR-21570 537
P-ERR-21571 537
P-ERR-21572 537
P-ERR-21573 538
P-ERR-21574 538
P-ERR-21575 538
P-ERR-21576 539
P-ERR-21577 539
P-ERR-21578 539
P-ERR-21579 540
P-ERR-21580 540
P-ERR-21581 540
P-ERR-21582 541
P-ERR-21585 541
P-ERR-21586 541
P-ERR-21588 541
P-ERR-21589 542
P-ERR-21590 542
P-ERR-21591 543
P-ERR-21592 543
P-ERR-21593 543
P-ERR-21594 543
P-ERR-21595 544
P-ERR-21596 544
P-ERR-21597 544
P-ERR-21598 544
P-ERR-21599 545
P-ERR-21600 545
P-ERR-21601 545
P-ERR-21602 545
P-ERR-21604 546
P-ERR-21605 546
P-ERR-21608 546
P-ERR-21609 547
P-ERR-21610 547
P-ERR-21611 547
P-ERR-21612 547
P-ERR-21614 547
P-ERR-21617 548
P-ERR-21618 548
P-ERR-21620 548
P-ERR-21621 548
P-ERR-21622 548
P-ERR-21623 549
P-ERR-21624 549
P-ERR-21625 549
P-ERR-21627 549
P-ERR-21628 549
P-ERR-21629 550
P-ERR-21630 550
P-ERR-21631 551
P-ERR-21633 551
P-ERR-21634 551
P-ERR-21635 551
P-ERR-21636 551
P-ERR-21637 552
P-ERR-21638 552
P-ERR-21640 552
P-ERR-21641 552
P-ERR-21643 553
P-ERR-21644 553
P-ERR-21646 553
P-ERR-21647 553
P-ERR-21648 554
P-ERR-21649 554
P-ERR-21650 554
P-ERR-21651 555
P-ERR-21655 555
P-ERR-21656 556
P-ERR-21657 556
P-ERR-21658 557
P-ERR-21659 558
P-ERR-21660 558
P-ERR-21663 559
P-ERR-21664 559
P-ERR-21665 559
P-ERR-21666 559
P-ERR-21667 559
P-ERR-21668 559
P-ERR-21669 560
P-ERR-21670 560
P-ERR-21673 560
P-ERR-21674 560
P-ERR-21675 561
P-ERR-21676 561
P-ERR-21678 561
P-ERR-21680 561
P-ERR-21683 561
P-ERR-21684 562
P-ERR-21687 562
P-ERR-21688 562
P-ERR-21689 563
P-ERR-21690 563
P-ERR-21691 563
P-ERR-21692 563
P-ERR-21693 564
P-ERR-21694 564
P-ERR-21695 564
P-ERR-21696 564
P-ERR-21697 564
P-ERR-21698 565
P-ERR-21701 565
P-ERR-21702 565
P-ERR-21703 565
P-ERR-21705 565
P-ERR-21706 566
P-ERR-21709 566
P-ERR-21710 566
P-ERR-21713 566
P-ERR-21714 567
P-ERR-21717 567
P-ERR-21718 568
P-ERR-21719 568
P-ERR-21720 568
P-ERR-21721 568
P-ERR-21723 568
P-ERR-21724 569
P-ERR-21725 569
P-ERR-21726 569
P-ERR-21727 569
P-ERR-21729 570

P-ERR-21730 570
P-ERR-21731 570
P-ERR-21732 570
P-ERR-21734 570
P-ERR-21736 571
P-ERR-21737 571
P-ERR-21738 571
P-ERR-21739 572
P-ERR-21743 572
P-ERR-21744 572
P-ERR-21745 572
P-ERR-21748 572
P-ERR-21749 573
P-ERR-21750 574
P-ERR-21751 574
P-ERR-21752 574
P-ERR-21756 575
P-ERR-21757 575
P-ERR-21758 575
P-ERR-21766 576
P-ERR-21767 576
P-ERR-21768 576
P-ERR-21771 577
P-ERR-21772 577
P-ERR-21773 577
P-ERR-21775 578
P-ERR-21776 578
P-ERR-21777 578
P-ERR-21778 578
P-ERR-21779 579
P-ERR-21780 579
P-ERR-21781 579
P-ERR-21782 579
P-ERR-21783 580
P-ERR-21784 580
P-ERR-21785 580
P-ERR-21786 581
P-ERR-21788 581
P-ERR-21789 581
P-ERR-21790 581
P-ERR-21791 582
P-ERR-21792 582
P-ERR-21793 582
P-ERR-21795 582
P-ERR-21796 583
P-ERR-21797 583
P-ERR-21798 583
P-ERR-21799 584
P-ERR-21800 584
P-ERR-21801 584
P-ERR-21802 584
P-ERR-21803 584
P-ERR-21804 584
P-ERR-21805 585
P-ERR-21806 585
P-ERR-21807 585
P-ERR-21808 585
P-ERR-21809 585
P-ERR-21810 585
P-ERR-21811 586
P-ERR-21812 586
P-ERR-21813 586
P-ERR-21814 587
P-ERR-21815 587
P-ERR-21816 587
P-ERR-21819 587
P-ERR-21820 588
P-ERR-21821 588
P-ERR-21822 588
P-ERR-21823 589
P-ERR-21826 589
P-ERR-21827 589
P-ERR-21828 589
P-ERR-21829 589
P-ERR-21832 590
P-ERR-21833 590
P-ERR-21834 591
P-ERR-21836 591
P-ERR-21837 591
P-ERR-21840 591
P-ERR-21841 592
P-ERR-21842 592
P-ERR-21843 592
P-ERR-21844 593
P-ERR-21845 593
P-ERR-21846 593
P-ERR-21847 594
P-ERR-21848 594
P-ERR-21849 595
P-ERR-21850 595
P-ERR-21851 595
P-ERR-21852 596
P-ERR-21853 596
P-ERR-21855 596
P-ERR-21857 597
P-ERR-21858 597
P-ERR-21859 597
P-ERR-21860 597
P-ERR-21861 597
P-ERR-21862 598
P-ERR-21863 598
P-ERR-21864 598
P-ERR-21865 598
P-ERR-21866 599
P-ERR-21867 599
P-ERR-21868 599
P-ERR-21870 600
P-ERR-21871 600
P-ERR-21872 600
P-ERR-21873 600
P-ERR-21874 601
P-ERR-21876 601
P-ERR-21877 601
P-ERR-21878 601
P-ERR-21879 602
P-ERR-21880 602
P-ERR-21881 602
P-ERR-21882 603
P-ERR-21883 603
P-ERR-21885 603
P-ERR-21886 604
P-ERR-21887 604
P-ERR-21889 604
P-ERR-21891 604
P-ERR-21892 604
P-ERR-21893 605

P-ERR-21894 605
P-ERR-21895 607
P-ERR-21897 607
P-ERR-21898 608
P-ERR-21903 608
P-ERR-21904 609
P-ERR-21905 609
P-ERR-21907 611
P-ERR-21908 611
P-ERR-21909 611
P-ERR-21910 612
P-ERR-21912 613
P-ERR-21913 613
P-ERR-21914 614
P-ERR-21915 614
P-ERR-21916 614
P-ERR-21917 615
P-ERR-21918 615
P-ERR-21919 615
P-ERR-21920 615
P-ERR-21921 616
P-ERR-21925 616
P-ERR-21927 616
P-ERR-21928 616
P-ERR-21929 617
P-ERR-21931 617
P-ERR-21932 617
P-ERR-21935 618
P-ERR-21936 618
P-ERR-21944 618
P-ERR-21945 619
P-ERR-21946 619
P-ERR-21947 619
P-ERR-21950 620
P-ERR-21952 620
P-ERR-21955 620
P-ERR-21956 620
P-ERR-21964 621
P-ERR-21966 621
P-ERR-21967 622
P-ERR-21968 622
P-ERR-21969 623
P-ERR-21970 623
P-ERR-21972 624
P-ERR-21974 624
P-ERR-21978 624
P-ERR-21979 625
P-ERR-21980 625
P-ERR-21981 625
P-ERR-21982 626
P-ERR-21984 626
P-ERR-21987 626
P-ERR-21988 627
P-ERR-21989 627
P-ERR-21990 627
P-ERR-21991 628
P-ERR-21992 628
P-ERR-21993 628
P-ERR-21994 628
P-ERR-21995 629
P-ERR-21996 629
P-ERR-21997 629
P-ERR-21998 630
P-ERR-21999 630
P-ERR-22000 631
P-ERR-22001 631
P-ERR-220012 1594
P-ERR-220015 1594
P-ERR-220016 1595
P-ERR-220017 1595
P-ERR-220018 1595
P-ERR-220019 1595
P-ERR-22002 631
P-ERR-220021 1596
P-ERR-220022 1596
P-ERR-220029 1596
P-ERR-22003 631
P-ERR-22004 631
P-ERR-22006 632
P-ERR-22007 632
P-ERR-22008 632
P-ERR-22009 632
P-ERR-22012 633
P-ERR-22014 633
P-ERR-22015 634
P-ERR-22016 634
P-ERR-22018 634
P-ERR-22019 634
P-ERR-22020 635
P-ERR-22021 635
P-ERR-22022 635
P-ERR-22023 635
P-ERR-22024 635
P-ERR-22026 636
P-ERR-22027 636
P-ERR-22028 636
P-ERR-22029 637
P-ERR-22030 637
P-ERR-22031 637
P-ERR-22043 637
P-ERR-22053 638
P-ERR-22060 638
P-ERR-22062 639
P-ERR-22066 639
P-ERR-22067 639
P-ERR-22068 640
P-ERR-22073 640
P-ERR-22077 640
P-ERR-22079 641
P-ERR-22080 641
P-ERR-22081 641
P-ERR-22083 642
P-ERR-22084 642
P-ERR-22088 643
P-ERR-22089 643
P-ERR-22090 643
P-ERR-22091 644
P-ERR-22092 644
P-ERR-22093 644
P-ERR-22094 645
P-ERR-22095 645
P-ERR-22096 645
P-ERR-22097 646
P-ERR-22099 646
P-ERR-22100 646
P-ERR-22101 647

P-ERR-22102 647
P-ERR-22103 647
P-ERR-22104 647
P-ERR-22105 648
P-ERR-22106 648
P-ERR-22107 648
P-ERR-22108 648
P-ERR-22109 649
P-ERR-22110 649
P-ERR-22111 649
P-ERR-22112 650
P-ERR-22114 650
P-ERR-22115 650
P-ERR-22116 650
P-ERR-22117 650
P-ERR-22118 651
P-ERR-22121 651
P-ERR-22122 651
P-ERR-22123 651
P-ERR-22124 652
P-ERR-22125 652
P-ERR-22126 652
P-ERR-22127 653
P-ERR-22128 653
P-ERR-22129 654
P-ERR-22130 654
P-ERR-22131 655
P-ERR-22132 655
P-ERR-22134 655
P-ERR-22135 655
P-ERR-22136 656
P-ERR-22137 656
P-ERR-22139 657
P-ERR-22140 657
P-ERR-22141 658
P-ERR-22142 658
P-ERR-22143 658
P-ERR-22144 659
P-ERR-22145 659
P-ERR-22146 659
P-ERR-22147 659
P-ERR-22148 660
P-ERR-22149 660
P-ERR-22150 660
P-ERR-22151 660
P-ERR-22153 661
P-ERR-22155 661
P-ERR-22156 661
P-ERR-22157 662
P-ERR-22158 662
P-ERR-22159 663
P-ERR-22162 663
P-ERR-22163 663
P-ERR-22164 663
P-ERR-22165 664
P-ERR-22166 664
P-ERR-22167 664
P-ERR-22168 664
P-ERR-22169 665
P-ERR-22170 665
P-ERR-22172 665
P-ERR-22173 666
P-ERR-22174 666
P-ERR-22175 666
P-ERR-22178 666
P-ERR-22179 667
P-ERR-22180 667
P-ERR-22181 668
P-ERR-22182 668
P-ERR-22183 668
P-ERR-22189 668
P-ERR-22190 669
P-ERR-22191 669
P-ERR-22192 670
P-ERR-22193 670
P-ERR-22194 670
P-ERR-22195 671
P-ERR-22200 671
P-ERR-22201 672
P-ERR-22203 672
P-ERR-22204 672
P-ERR-22205 673
P-ERR-22206 673
P-ERR-22207 673
P-ERR-22209 674
P-ERR-22210 674
P-ERR-22213 674
P-ERR-22214 675
P-ERR-22215 675
P-ERR-22216 675
P-ERR-22217 676
P-ERR-22218 676
P-ERR-22219 676
P-ERR-22224 677
P-ERR-22225 677
P-ERR-22226 677
P-ERR-22229 678
P-ERR-22230 678
P-ERR-22231 678
P-ERR-22233 678
P-ERR-22234 679
P-ERR-22235 679
P-ERR-22236 679
P-ERR-22237 679
P-ERR-22241 680
P-ERR-22242 680
P-ERR-22243 680
P-ERR-22244 681
P-ERR-22245 681
P-ERR-22246 681
P-ERR-22247 682
P-ERR-22248 682
P-ERR-22250 683
P-ERR-22253 683
P-ERR-22254 683
P-ERR-22255 684
P-ERR-22256 684
P-ERR-22257 684
P-ERR-22258 685
P-ERR-22261 685
P-ERR-22262 686
P-ERR-22263 686
P-ERR-22264 687
P-ERR-22265 688
P-ERR-22266 688
P-ERR-22267 688

P-ERR-22269 688
P-ERR-22272 689
P-ERR-22273 689
P-ERR-22274 690
P-ERR-22275 690
P-ERR-22276 690
P-ERR-22277 691
P-ERR-22278 691
P-ERR-22279 691
P-ERR-22280 692
P-ERR-22281 692
P-ERR-22282 692
P-ERR-22283 693
P-ERR-22284 693
P-ERR-22285 694
P-ERR-22286 694
P-ERR-22287 694
P-ERR-22288 695
P-ERR-22303 695
P-ERR-22306 695
P-ERR-22307 695
P-ERR-22310 696
P-ERR-22311 696
P-ERR-22312 696
P-ERR-22313 696
P-ERR-22314 697
P-ERR-22315 697
P-ERR-22316 697
P-ERR-22317 697
P-ERR-22318 697
P-ERR-22319 698
P-ERR-22320 698
P-ERR-22321 698
P-ERR-22322 698
P-ERR-22323 698
P-ERR-22324 699
P-ERR-22325 699
P-ERR-22326 699
P-ERR-22327 699
P-ERR-22328 699
P-ERR-22329 700
P-ERR-22330 700
P-ERR-22331 700
P-ERR-22332 700
P-ERR-22333 701
P-ERR-22334 701
P-ERR-22335 701
P-ERR-22336 701
P-ERR-22337 702
P-ERR-22338 702
P-ERR-22339 702
P-ERR-22340 703
P-ERR-22341 703
P-ERR-22342 703
P-ERR-22343 703
P-ERR-22344 704
P-ERR-22345 704
P-ERR-22346 704
P-ERR-22347 704
P-ERR-22348 705
P-ERR-230040 1597
P-ERR-230041 1597
P-ERR-240007 1598
P-ERR-240008 1598
P-ERR-240009 1598
P-ERR-240010 1598
P-ERR-240011 1598
P-ERR-240012 1599
P-ERR-240013 1599
P-ERR-240014 1599
P-ERR-240015 1599
P-ERR-240016 1599
P-ERR-240017 1599
P-ERR-240018 1600
P-ERR-240019 1600
P-ERR-240020 1600
P-ERR-240021 1600
P-ERR-240022 1600
P-ERR-240023 1600
P-ERR-240024 1601
P-ERR-240025 1601
P-ERR-240026 1601
P-ERR-240027 1601
P-ERR-240028 1601
P-ERR-240029 1601
P-ERR-240030 1602
P-ERR-240031 1602
P-ERR-240032 1602
P-ERR-240033 1602
P-ERR-240034 1602
P-ERR-240035 1602
P-ERR-240036 1603
P-ERR-240037 1603
P-ERR-240038 1603
P-ERR-240039 1603
P-ERR-240043 1604
P-ERR-240044 1604
P-ERR-240045 1604
P-ERR-240046 1604
P-ERR-240047 1604
P-ERR-240048 1604
P-ERR-240049 1605
P-ERR-240050 1605
P-ERR-240051 1605
P-ERR-240052 1605
P-ERR-240053 1605
P-ERR-240057 1606
P-ERR-240058 1606
P-ERR-250000 1607
P-ERR-250001 1607
P-ERR-250002 1607
P-ERR-250003 1608
P-ERR-250004 1608
P-ERR-250005 1609
P-ERR-250008 1609
P-ERR-250009 1609
P-ERR-260001 1611
P-ERR-260002 1611
P-ERR-260003 1611
P-ERR-260004 1611
P-ERR-260005 1612
P-ERR-260007 1612
P-ERR-260008 1612
P-ERR-260009 1613
P-ERR-260010 1613
P-ERR-260011 1613

P-ERR-260012 1613
P-ERR-260013 1614
P-ERR-260016 1614
P-ERR-260017 1614
P-ERR-260018 1614
P-ERR-260019 1615
P-ERR-260025 1615
P-ERR-260026 1615
P-ERR-260031 1616
P-ERR-260032 1616
P-ERR-260034 1617
P-ERR-260035 1617
P-ERR-260036 1617
P-ERR-260037 1617
P-ERR-260038 1617
P-ERR-260039 1618
P-ERR-260040 1618
P-ERR-260041 1618
P-ERR-260042 1618
P-ERR-260043 1618
P-ERR-260044 1618
P-ERR-260045 1619
P-ERR-260046 1619
P-ERR-260047 1619
P-ERR-260048 1619
P-ERR-260049 1619
P-ERR-260050 1619
P-ERR-260051 1620
P-ERR-260052 1620
P-ERR-260053 1620
P-ERR-260054 1620
P-ERR-260055 1620
P-ERR-260056 1620
P-ERR-260057 1621
P-ERR-260058 1621
P-ERR-260059 1621
P-ERR-260060 1621
P-ERR-260062 1621
P-ERR-260063 1621
P-ERR-260064 1622
P-ERR-260065 1622
P-ERR-260066 1622
P-ERR-260067 1622
P-ERR-260068 1622
P-ERR-260069 1622
P-ERR-260070 1623
P-ERR-260071 1623
P-ERR-260074 1623
P-ERR-260075 1623
P-ERR-260076 1623
P-ERR-260077 1624
P-ERR-260078 1624
P-ERR-260079 1624
P-ERR-260080 1624
P-ERR-260082 1625
P-ERR-260083 1625
P-ERR-260084 1625
P-ERR-260085 1625
P-ERR-260089 1626
P-ERR-260090 1626
P-ERR-260091 1627
P-ERR-260092 1627
P-ERR-260093 1627
P-ERR-260094 1627
P-ERR-260095 1627
P-ERR-260096 1628
P-ERR-260097 1628
P-ERR-260098 1628
P-ERR-260099 1628
P-ERR-260100 1628
P-ERR-260101 1629
P-ERR-270018 1630
P-ERR-270021 1630
P-ERR-270022 1632
P-ERR-270023 1632
P-ERR-270024 1632
P-ERR-270031 1633
P-ERR-270032 1633
P-ERR-270033 1633
P-ERR-270037 1634
P-ERR-270041 1634
P-ERR-270042 1634
P-ERR-270046 1635
P-ERR-270047 1635
P-ERR-270049 1635
P-ERR-270050 1636
P-ERR-270051 1636
P-ERR-270052 1636
P-ERR-270053 1636
P-ERR-270055 1637
P-ERR-270059 1637
P-ERR-270060 1638
P-ERR-270061 1639
P-ERR-270062 1639
P-ERR-270070 1640
P-ERR-270071 1640
P-ERR-270073 1640
P-ERR-270074 1641
P-ERR-270076 1641
P-ERR-270077 1641
P-ERR-270078 1642
P-ERR-270079 1643
P-ERR-270080 1643
P-ERR-270081 1644
P-ERR-270082 1644
P-ERR-270086 1645
P-ERR-270092 1645
P-ERR-270093 1646
P-ERR-270094 1646
P-ERR-270095 1646
P-ERR-270096 1646
P-ERR-270097 1647
P-ERR-270103 1647
P-ERR-270104 1647
P-ERR-270109 1647
P-ERR-270110 1648
P-ERR-280304 1650
P-ERR-280600 1651
P-ERR-280601 1651
P-ERR-280602 1651
P-ERR-280603 1651
P-ERR-280604 1652
P-ERR-280605 1652
P-ERR-280606 1652
P-ERR-280607 1652
P-ERR-280608 1653

P-ERR-280609 1653
P-ERR-280610 1653
P-ERR-280611 1653
P-ERR-280612 1654
P-ERR-280613 1654
P-ERR-280614 1654
P-ERR-280615 1654
P-ERR-280616 1654
P-ERR-280700 1655
P-ERR-280701 1655
P-ERR-280702 1655
P-ERR-280703 1656
P-ERR-280704 1656
P-ERR-280706 1656
P-ERR-280707 1656
P-ERR-280708 1657
P-ERR-280709 1657
P-ERR-280710 1657
P-ERR-280711 1657
P-ERR-280712 1658
P-ERR-280713 1658
P-ERR-280714 1659
P-ERR-280715 1659
P-ERR-280716 1659
P-ERR-280717 1659
P-ERR-280718 1660
P-ERR-280719 1660
P-ERR-280720 1660
P-ERR-280721 1661
P-ERR-280722 1661
P-ERR-280723 1661
P-ERR-280724 1662
P-ERR-292001 1663
P-ERR-292002 1663
P-ERR-292003 1664
P-ERR-292004 1664
P-ERR-292005 1665
P-ERR-292006 1665
P-ERR-292007 1666
P-ERR-292008 1666
P-ERR-292009 1666
P-ERR-292010 1667
P-ERR-292011 1667
P-ERR-292012 1668
P-ERR-292013 1668
P-ERR-292014 1668
P-ERR-292015 1669
P-ERR-292016 1669
P-ERR-292019 1670
P-ERR-292022 1670
P-ERR-292023 1670
P-ERR-292024 1671
P-ERR-292025 1671
P-ERR-292026 1671
P-ERR-292027 1671
P-ERR-292030 1671
P-ERR-292031 1671
P-ERR-292032 1672
P-ERR-292033 1672
P-ERR-292034 1672
P-ERR-292035 1672
P-ERR-292036 1672
P-ERR-292037 1672
P-ERR-292038 1673
P-ERR-292039 1673
P-ERR-292040 1673
P-ERR-292041 1673
P-ERR-292043 1674
P-ERR-292044 1674
P-ERR-292045 1674
P-ERR-292046 1674
P-ERR-292047 1675
P-ERR-292048 1675
P-ERR-292049 1675
P-ERR-292050 1676
P-ERR-292051 1676
P-ERR-292052 1676
P-ERR-292053 1677
P-ERR-292054 1677
P-ERR-292055 1677
P-ERR-292056 1677
P-ERR-292057 1677
P-ERR-292058 1678
P-ERR-292059 1678
P-ERR-292060 1678
P-ERR-292061 1678
P-ERR-292062 1679
P-ERR-292063 1679
P-ERR-292064 1680
P-ERR-292065 1680
P-ERR-292066 1680
P-ERR-292067 1681
P-ERR-292068 1681
P-ERR-292069 1681
P-ERR-292070 1682
P-ERR-292071 1682
P-ERR-292072 1682
P-ERR-292073 1683
P-ERR-292074 1683
P-ERR-292075 1683
P-ERR-292076 1683
P-ERR-292080 1684
P-ERR-292081 1684
P-ERR-292082 1684
P-ERR-292083 1684
P-ERR-292084 1684
P-ERR-292085 1685
P-ERR-292086 1685
P-ERR-292087 1686
P-ERR-292089 1687
P-ERR-292090 1687
P-ERR-292091 1688
P-ERR-292092 1688
P-ERR-293000 1689
P-ERR-293001 1689
P-ERR-293002 1689
P-ERR-293003 1689
P-ERR-293004 1690
P-ERR-293102 1690
P-ERR-293103 1690
P-ERR-293104 1690
P-ERR-293105 1691
P-ERR-293109 1691
P-ERR-293110 1691
P-ERR-293111 1691
P-ERR-294000 1692

P-ERR-294001 1692
P-ERR-294002 1692
P-ERR-294003 1693
P-ERR-294004 1693
P-ERR-294005 1693
P-ERR-294006 1693
P-ERR-294007 1694
P-ERR-294008 1694
P-ERR-294009 1694
P-ERR-294010 1694
P-ERR-294011 1695
P-ERR-294012 1695
P-ERR-294013 1695
P-ERR-294014 1696
P-ERR-294015 1696
P-ERR-294016 1696
P-ERR-294017 1696
P-ERR-294018 1697
P-ERR-294020 1697
P-ERR-294021 1697
P-ERR-294022 1698
P-ERR-294023 1698
P-ERR-294024 1698
P-ERR-294025 1699
P-ERR-294026 1699
P-ERR-294027 1699
P-ERR-294028 1700
P-ERR-294029 1700
P-ERR-294030 1700
P-ERR-294031 1701
P-ERR-294032 1701
P-ERR-294033 1701
P-ERR-294034 1701
P-ERR-294035 1702
P-ERR-294036 1702
P-ERR-294037 1702
P-ERR-294038 1702
P-ERR-294039 1703
P-ERR-294040 1703
P-ERR-294041 1703
P-ERR-294042 1703
P-ERR-294043 1704
P-ERR-294044 1704
P-ERR-294045 1704
P-ERR-294046 1704
P-ERR-294047 1705
P-ERR-294048 1705
P-ERR-294049 1705
P-ERR-294050 1706
P-ERR-294051 1706
P-ERR-294053 1706
P-ERR-294054 1706
P-ERR-294055 1707
P-ERR-294056 1707
P-ERR-294057 1707
P-ERR-294058 1707
P-ERR-294059 1708
P-ERR-294060 1708
P-ERR-294061 1708
P-ERR-294500 1709
P-ERR-294501 1709
P-ERR-294502 1709
P-ERR-294503 1709
P-ERR-294504 1710
P-ERR-294505 1710
P-ERR-294506 1710
P-ERR-294507 1710
P-ERR-294508 1711
P-ERR-294509 1711
P-ERR-294510 1711
P-ERR-294511 1711
P-ERR-294512 1712
P-ERR-294513 1712
P-ERR-294514 1712
P-ERR-294523 1712
P-ERR-294525 1712
P-ERR-294526 1713
P-ERR-294527 1713
P-ERR-294528 1713
P-ERR-294529 1713
P-ERR-294530 1714
P-ERR-294533 1714
P-ERR-294534 1714
P-ERR-294535 1714
P-ERR-294536 1715
P-ERR-294538 1715
P-ERR-294539 1715
P-ERR-294540 1716
P-ERR-294542 1716
P-ERR-294543 1716
P-ERR-294544 1716
P-ERR-294545 1717
P-ERR-294546 1717
P-ERR-294547 1717
P-ERR-294549 1717
P-ERR-294550 1718
P-ERR-294551 1718
P-ERR-294552 1718
P-ERR-294553 1718
P-ERR-294554 1718
P-ERR-294555 1719
P-ERR-294556 1719
P-ERR-294557 1719
P-ERR-294558 1719
P-ERR-294559 1719
P-ERR-294560 1720
P-ERR-294561 1720
P-ERR-294562 1720
P-ERR-294563 1720
P-ERR-294564 1721
P-ERR-294565 1721
P-ERR-294566 1721
P-ERR-294567 1721
P-ERR-294568 1722
P-ERR-294569 1722
P-ERR-294570 1722
P-ERR-294571 1722
P-ERR-294572 1722
P-ERR-294573 1723
P-ERR-294574 1723
P-ERR-294575 1723
P-ERR-294576 1723
P-ERR-294577 1724
P-ERR-294578 1724
P-ERR-294579 1724
P-ERR-294580 1724

P-ERR-294581 1724
P-ERR-294582 1724
P-ERR-294583 1725
P-ERR-294584 1725
P-ERR-294585 1725
P-ERR-294586 1725
P-ERR-294587 1725
P-ERR-294588 1725
P-ERR-294589 1726
P-ERR-294590 1726
P-ERR-294591 1726
P-ERR-294592 1726
P-ERR-294593 1726
P-ERR-294594 1726
P-ERR-294595 1727
P-ERR-294596 1727
P-ERR-294597 1727
P-ERR-294598 1727
P-ERR-294600 1727
P-ERR-294601 1728
P-ERR-294602 1728
P-ERR-295001 1729
P-ERR-295003 1729
P-ERR-295004 1729
P-ERR-295005 1730
P-ERR-295006 1730
P-ERR-295007 1730
P-ERR-295008 1730
P-ERR-295009 1730
P-ERR-295010 1731
P-ERR-295011 1731
P-ERR-295012 1731
P-ERR-295013 1731
P-ERR-295014 1731
P-ERR-295015 1732
P-ERR-295016 1732
P-ERR-295017 1732
P-ERR-295018 1732
P-ERR-295019 1732
P-ERR-295020 1732
P-ERR-295021 1733
P-ERR-295022 1733
P-ERR-295023 1733
P-ERR-295024 1733
P-ERR-295025 1734
P-ERR-295026 1734
P-ERR-295027 1734
P-ERR-295028 1734
P-ERR-295029 1734
P-ERR-295030 1735
P-ERR-295031 1735
P-ERR-295032 1735
P-ERR-295033 1735
P-ERR-295034 1735
P-ERR-295035 1735
P-ERR-295036 1736
P-ERR-295037 1736
P-ERR-295038 1736
P-ERR-295039 1736
P-ERR-295040 1736
P-ERR-295041 1736
P-ERR-295042 1737
P-ERR-295043 1737
P-ERR-296000 1738
P-ERR-296001 1738
P-ERR-296002 1738
P-ERR-296004 1738
P-ERR-296005 1739
P-ERR-296006 1740
P-ERR-296007 1740
P-ERR-296008 1741
P-ERR-296009 1741
P-ERR-296010 1742
P-ERR-296011 1742
P-ERR-297012 1743
P-ERR-297014 1743
P-ERR-297015 1743
P-ERR-297016 1743
P-ERR-30014 706
P-ERR-301000 1744
P-ERR-301001 1744
P-ERR-301002 1744
P-ERR-301004 1744
P-ERR-301005 1745
P-ERR-301006 1745
P-ERR-301007 1745
P-ERR-301008 1745
P-ERR-301009 1746
P-ERR-301010 1746
P-ERR-301012 1746
P-ERR-301013 1747
P-ERR-301014 1747
P-ERR-301015 1747
P-ERR-301016 1748
P-ERR-301017 1748
P-ERR-301018 1748
P-ERR-301019 1748
P-ERR-301020 1749
P-ERR-310000 1750
P-ERR-310001 1750
P-ERR-310003 1751
P-ERR-310004 1752
P-ERR-310005 1752
P-ERR-310006 1753
P-ERR-310007 1753
P-ERR-310008 1753
P-ERR-310009 1754
P-ERR-310010 1754
P-ERR-310011 1754
P-ERR-310014 1755
P-ERR-310015 1755
P-ERR-310016 1756
P-ERR-310017 1756
P-ERR-310018 1757
P-ERR-310019 1757
P-ERR-310020 1758
P-ERR-310021 1758
P-ERR-310022 1759
P-ERR-310023 1759
P-ERR-310024 1760
P-ERR-310025 1760
P-ERR-310026 1761
P-ERR-310027 1761
P-ERR-310028 1762
P-ERR-310029 1762
P-ERR-315000 1763

P-ERR-315001 1763
P-ERR-315002 1764
P-ERR-315003 1764
P-ERR-315004 1765
P-ERR-315005 1765
P-ERR-315006 1766
P-ERR-315007 1767
P-ERR-315008 1767
P-ERR-315009 1768
P-ERR-315010 1768
P-ERR-315011 1769
P-ERR-315012 1769
P-ERR-3210 39
P-ERR-3314 39
P-ERR-3315 40
P-ERR-3316 40
P-ERR-400000 1770
P-ERR-400001 1770
P-ERR-400002 1770
P-ERR-400003 1770
P-ERR-400004 1771
P-ERR-400005 1771
P-ERR-400006 1771
P-ERR-400007 1771
P-ERR-400008 1771
P-ERR-400009 1771
P-ERR-40001 707
P-ERR-400010 1772
P-ERR-400011 1772
P-ERR-400012 1772
P-ERR-400013 1772
P-ERR-400014 1772
P-ERR-40002 707
P-ERR-40003 707
P-ERR-40004 708
P-ERR-40005 708
P-ERR-40006 708
P-ERR-40007 709
P-ERR-40008 710
P-ERR-40009 710
P-ERR-40010 711
P-ERR-400100 736
P-ERR-400101 736
P-ERR-400102 737
P-ERR-40011 711
P-ERR-40014 712
P-ERR-40015 712
P-ERR-40016 713
P-ERR-40017 714
P-ERR-40018 714
P-ERR-40019 715
P-ERR-40020 715
P-ERR-40021 716
P-ERR-40022 716
P-ERR-40023 716
P-ERR-40024 717
P-ERR-40025 717
P-ERR-40026 718
P-ERR-40027 718
P-ERR-40028 718
P-ERR-40032 719
P-ERR-40033 719
P-ERR-40034 719
P-ERR-40035 720
P-ERR-40037 720
P-ERR-40038 720
P-ERR-40039 721
P-ERR-40040 721
P-ERR-40041 722
P-ERR-40042 723
P-ERR-40044 723
P-ERR-40045 723
P-ERR-40046 723
P-ERR-40047 724
P-ERR-40048 724
P-ERR-40049 725
P-ERR-40051 725
P-ERR-40052 726
P-ERR-40053 726
P-ERR-40055 726
P-ERR-40056 726
P-ERR-40059 727
P-ERR-40066 727
P-ERR-40067 727
P-ERR-40068 727
P-ERR-40069 728
P-ERR-40070 728
P-ERR-40071 728
P-ERR-40072 728
P-ERR-40073 729
P-ERR-40074 729
P-ERR-40075 729
P-ERR-40076 729
P-ERR-40077 730
P-ERR-40078 730
P-ERR-40079 730
P-ERR-40080 730
P-ERR-40081 731
P-ERR-40084 731
P-ERR-40085 731
P-ERR-40086 731
P-ERR-40087 732
P-ERR-40088 732
P-ERR-40089 732
P-ERR-40090 733
P-ERR-40091 733
P-ERR-40092 734
P-ERR-40093 734
P-ERR-40094 735
P-ERR-40095 735
P-ERR-40097 735
P-ERR-40098 736
P-ERR-40099 736
P-ERR-401000 1773
P-ERR-401001 1773
P-ERR-401002 1773
P-ERR-40103 737
P-ERR-402000 1774
P-ERR-402001 1774
P-ERR-402002 1774
P-ERR-403000 1775
P-ERR-403001 1775
P-ERR-403002 1775
P-ERR-4119 42
P-ERR-4120 42
P-ERR-4121 42

P-ERR-4122 43
P-ERR-4123 43
P-ERR-4124 43
P-ERR-4125 44
P-ERR-4126 44
P-ERR-4127 44
P-ERR-4128 45
P-ERR-4129 45
P-ERR-4130 45
P-ERR-4131 46
P-ERR-4132 46
P-ERR-4133 46
P-ERR-4134 46
P-ERR-4135 47
P-ERR-4136 47
P-ERR-4137 47
P-ERR-4142 48
P-ERR-4143 48
P-ERR-4144 48
P-ERR-4145 48
P-ERR-4146 49
P-ERR-42000 738
P-ERR-42012 738
P-ERR-42014 738
P-ERR-42015 739
P-ERR-42016 739
P-ERR-44000 740
P-ERR-44001 740
P-ERR-44002 740
P-ERR-44003 741
P-ERR-44004 741
P-ERR-44005 741
P-ERR-44006 741
P-ERR-44007 742
P-ERR-44008 742
P-ERR-44009 742
P-ERR-44010 743
P-ERR-44011 743
P-ERR-44012 743
P-ERR-44013 744
P-ERR-44014 744
P-ERR-44015 744
P-ERR-44016 745
P-ERR-44017 745
P-ERR-44018 745
P-ERR-44019 746
P-ERR-44020 746
P-ERR-44021 747
P-ERR-44022 747
P-ERR-44024 747
P-ERR-44025 748
P-ERR-44026 748
P-ERR-44027 749
P-ERR-44028 749
P-ERR-44029 749
P-ERR-44030 749
P-ERR-44031 750
P-ERR-44032 750
P-ERR-44033 750
P-ERR-44034 750
P-ERR-44035 751
P-ERR-44036 751
P-ERR-44037 751
P-ERR-44038 751
P-ERR-44039 752
P-ERR-44040 752
P-ERR-44043 752
P-ERR-44044 752
P-ERR-44045 753
P-ERR-44046 753
P-ERR-44047 753
P-ERR-44049 754
P-ERR-44050 754
P-ERR-44052 754
P-ERR-44053 755
P-ERR-44054 755
P-ERR-44055 755
P-ERR-44056 755
P-ERR-44057 755
P-ERR-44058 756
P-ERR-44059 756
P-ERR-44060 757
P-ERR-44062 757
P-ERR-44064 757
P-ERR-44065 758
P-ERR-44066 758
P-ERR-44069 758
P-ERR-44070 758
P-ERR-44075 759
P-ERR-44076 759
P-ERR-44077 760
P-ERR-44078 760
P-ERR-44079 760
P-ERR-44082 760
P-ERR-44083 761
P-ERR-44084 761
P-ERR-45057 775
P-ERR-46000 762
P-ERR-46001 762
P-ERR-46002 762
P-ERR-46003 762
P-ERR-46004 763
P-ERR-46005 763
P-ERR-46006 763
P-ERR-46007 763
P-ERR-46008 764
P-ERR-46009 764
P-ERR-46010 764
P-ERR-46016 765
P-ERR-46017 765
P-ERR-46018 765
P-ERR-46019 765
P-ERR-46020 766
P-ERR-46021 766
P-ERR-46022 766
P-ERR-46025 767
P-ERR-46028 767
P-ERR-46029 767
P-ERR-46030 768
P-ERR-46032 768
P-ERR-46033 768
P-ERR-46034 768
P-ERR-46035 769
P-ERR-46036 769
P-ERR-46037 769
P-ERR-46038 770

P-ERR-46039 770
P-ERR-46040 770
P-ERR-46041 770
P-ERR-46042 771
P-ERR-46043 771
P-ERR-46044 771
P-ERR-46045 772
P-ERR-46046 772
P-ERR-46047 772
P-ERR-46048 772
P-ERR-46049 773
P-ERR-46050 773
P-ERR-46051 773
P-ERR-46052 774
P-ERR-46053 774
P-ERR-46054 774
P-ERR-46055 774
P-ERR-46056 775
P-ERR-46058 775
P-ERR-50010 777
P-ERR-50011 777
P-ERR-50025 778
P-ERR-50041 778
P-ERR-50042 779
P-ERR-50043 779
P-ERR-50045 780
P-ERR-50050 780
P-ERR-50052 781
P-ERR-50053 781
P-ERR-50057 782
P-ERR-50058 782
P-ERR-50059 782
P-ERR-50061 783
P-ERR-50063 783
P-ERR-50065 784
P-ERR-50067 784
P-ERR-50092 785
P-ERR-50116 785
P-ERR-50118 786
P-ERR-50119 786
P-ERR-50120 787
P-ERR-50121 787
P-ERR-50122 788
P-ERR-50123 788
P-ERR-50130 789
P-ERR-50168 790
P-ERR-50169 790
P-ERR-50172 790
P-ERR-50180 790
P-ERR-50181 791
P-ERR-50182 791
P-ERR-50183 791
P-ERR-50185 792
P-ERR-50193 792
P-ERR-50206 793
P-ERR-50214 793
P-ERR-50215 794
P-ERR-50233 794
P-ERR-50253 796
P-ERR-50254 796
P-ERR-50260 796
P-ERR-50262 797
P-ERR-50263 797
P-ERR-50278 798
P-ERR-50280 798
P-ERR-50350 799
P-ERR-50351 799
P-ERR-50354 800
P-ERR-50355 800
P-ERR-50356 800
P-ERR-50357 801
P-ERR-50358 801
P-ERR-50359 802
P-ERR-50360 802
P-ERR-50361 803
P-ERR-50362 803
P-ERR-50384 804
P-ERR-50388 804
P-ERR-50389 805
P-ERR-50398 805
P-ERR-50403 806
P-ERR-50404 806
P-ERR-50411 807
P-ERR-50412 807
P-ERR-50417 807
P-ERR-50418 808
P-ERR-50429 808
P-ERR-50430 809
P-ERR-50431 809
P-ERR-50438 809
P-ERR-50443 810
P-ERR-50449 811
P-ERR-50450 811
P-ERR-50452 812
P-ERR-50453 812
P-ERR-50454 812
P-ERR-50455 813
P-ERR-50462 813
P-ERR-50463 813
P-ERR-50464 814
P-ERR-50465 814
P-ERR-50473 815
P-ERR-50474 815
P-ERR-50475 815
P-ERR-50487 816
P-ERR-50494 816
P-ERR-50498 817
P-ERR-50499 817
P-ERR-50500 818
P-ERR-50503 818
P-ERR-50504 818
P-ERR-50505 819
P-ERR-50506 819
P-ERR-50512 820
P-ERR-50513 820
P-ERR-50515 821
P-ERR-50516 821
P-ERR-50519 822
P-ERR-50529 822
P-ERR-50533 823
P-ERR-50534 823
P-ERR-50540 824
P-ERR-50541 824
P-ERR-50543 825
P-ERR-50544 825
P-ERR-50545 825

P-ERR-50557 826
P-ERR-50558 827
P-ERR-50562 827
P-ERR-50564 828
P-ERR-50567 828
P-ERR-50568 829
P-ERR-50569 829
P-ERR-50570 830
P-ERR-50571 830
P-ERR-50572 830
P-ERR-50573 831
P-ERR-50593 832
P-ERR-50594 832
P-ERR-50595 832
P-ERR-50597 833
P-ERR-50599 833
P-ERR-50600 833
P-ERR-50601 833
P-ERR-50604 834
P-ERR-50605 835
P-ERR-50606 835
P-ERR-50607 835
P-ERR-50608 836
P-ERR-50611 837
P-ERR-50612 838
P-ERR-50613 838
P-ERR-50616 839
P-ERR-50618 839
P-ERR-50619 840
P-ERR-50625 840
P-ERR-50626 841
P-ERR-50627 841
P-ERR-50629 841
P-ERR-50631 841
P-ERR-50633 842
P-ERR-50634 842
P-ERR-50635 842
P-ERR-50638 843
P-ERR-50639 843
P-ERR-50640 844
P-ERR-50641 844
P-ERR-50642 844
P-ERR-50643 844
P-ERR-50644 845
P-ERR-50649 845
P-ERR-50653 846
P-ERR-50655 846
P-ERR-50660 847
P-ERR-50663 847
P-ERR-50664 848
P-ERR-50665 848
P-ERR-50666 848
P-ERR-50673 849
P-ERR-50674 849
P-ERR-50675 849
P-ERR-50676 850
P-ERR-50677 850
P-ERR-50678 850
P-ERR-50681 851
P-ERR-50684 851
P-ERR-50685 851
P-ERR-50686 851
P-ERR-50688 852
P-ERR-50689 852
P-ERR-50692 853
P-ERR-50693 853
P-ERR-50694 853
P-ERR-50695 854
P-ERR-50697 854
P-ERR-50698 855
P-ERR-50706 855
P-ERR-50708 856
P-ERR-50713 856
P-ERR-50717 857
P-ERR-50718 857
P-ERR-50720 857
P-ERR-50721 858
P-ERR-50724 859
P-ERR-50725 859
P-ERR-50726 859
P-ERR-50731 861
P-ERR-50733 861
P-ERR-50734 861
P-ERR-50735 862
P-ERR-50736 862
P-ERR-50737 862
P-ERR-50738 862
P-ERR-50745 862
P-ERR-50746 863
P-ERR-50747 863
P-ERR-50748 863
P-ERR-50750 864
P-ERR-50752 864
P-ERR-50761 864
P-ERR-50762 864
P-ERR-50763 865
P-ERR-50766 865
P-ERR-50767 866
P-ERR-50768 866
P-ERR-50771 866
P-ERR-50773 866
P-ERR-50774 867
P-ERR-50778 868
P-ERR-50779 868
P-ERR-50780 869
P-ERR-50781 869
P-ERR-50782 870
P-ERR-50783 870
P-ERR-50784 871
P-ERR-50785 871
P-ERR-50787 873
P-ERR-50788 873
P-ERR-50789 874
P-ERR-50790 874
P-ERR-50791 875
P-ERR-50792 875
P-ERR-50793 876
P-ERR-50794 876
P-ERR-50795 877
P-ERR-50796 878
P-ERR-50798 878
P-ERR-50806 879
P-ERR-50807 879
P-ERR-50810 879
P-ERR-50812 879
P-ERR-50819 880

P-ERR-50820 880
P-ERR-50821 880
P-ERR-50822 880
P-ERR-50824 881
P-ERR-50833 881
P-ERR-50837 882
P-ERR-50839 882
P-ERR-50841 882
P-ERR-50842 883
P-ERR-50843 883
P-ERR-50849 883
P-ERR-50852 884
P-ERR-50854 884
P-ERR-50855 885
P-ERR-50856 885
P-ERR-50857 885
P-ERR-50858 885
P-ERR-50861 886
P-ERR-50863 886
P-ERR-50870 886
P-ERR-50871 886
P-ERR-50873 887
P-ERR-50874 887
P-ERR-50875 888
P-ERR-50876 888
P-ERR-50877 888
P-ERR-50878 888
P-ERR-50879 889
P-ERR-50881 889
P-ERR-50882 889
P-ERR-50884 890
P-ERR-50885 890
P-ERR-50886 891
P-ERR-50889 891
P-ERR-50891 892
P-ERR-50892 892
P-ERR-50893 892
P-ERR-50894 893
P-ERR-50895 893
P-ERR-50896 893
P-ERR-50898 893
P-ERR-50899 893
P-ERR-50900 894
P-ERR-50901 894
P-ERR-50902 894
P-ERR-50905 894
P-ERR-50909 895
P-ERR-50910 895
P-ERR-50911 895
P-ERR-50913 896
P-ERR-50914 896
P-ERR-50915 896
P-ERR-50916 896
P-ERR-50917 897
P-ERR-50918 897
P-ERR-50922 898
P-ERR-50924 898
P-ERR-50926 899
P-ERR-50927 899
P-ERR-50929 900
P-ERR-50930 900
P-ERR-50931 900
P-ERR-50932 901
P-ERR-50933 901
P-ERR-50934 902
P-ERR-50938 902
P-ERR-50939 903
P-ERR-50940 903
P-ERR-50941 903
P-ERR-50942 903
P-ERR-50943 903
P-ERR-50947 904
P-ERR-50948 905
P-ERR-50950 905
P-ERR-50951 905
P-ERR-50953 906
P-ERR-50954 906
P-ERR-50956 907
P-ERR-50961 907
P-ERR-50962 908
P-ERR-50963 908
P-ERR-50964 909
P-ERR-50965 909
P-ERR-50966 909
P-ERR-50967 910
P-ERR-50968 910
P-ERR-50973 910
P-ERR-50974 911
P-ERR-50975 911
P-ERR-50976 911
P-ERR-50977 912
P-ERR-50978 912
P-ERR-50987 913
P-ERR-50988 913
P-ERR-50989 914
P-ERR-50991 914
P-ERR-50995 915
P-ERR-50997 916
P-ERR-50999 916
P-ERR-51000 917
P-ERR-51001 917
P-ERR-51002 917
P-ERR-51004 918
P-ERR-51006 918
P-ERR-51007 919
P-ERR-51012 919
P-ERR-51013 920
P-ERR-51014 920
P-ERR-51015 920
P-ERR-51016 920
P-ERR-51019 921
P-ERR-51021 921
P-ERR-51023 922
P-ERR-51024 922
P-ERR-51025 922
P-ERR-51026 923
P-ERR-51027 923
P-ERR-51028 923
P-ERR-51029 924
P-ERR-51031 924
P-ERR-51033 925
P-ERR-51034 925
P-ERR-51035 925
P-ERR-51036 925
P-ERR-51037 926
P-ERR-51040 926

P-ERR-51041 927
P-ERR-51043 927
P-ERR-51044 928
P-ERR-51045 928
P-ERR-51046 928
P-ERR-51047 929
P-ERR-51048 929
P-ERR-51049 929
P-ERR-51050 930
P-ERR-51051 931
P-ERR-51052 931
P-ERR-51053 931
P-ERR-51054 931
P-ERR-51055 931
P-ERR-51056 932
P-ERR-51057 932
P-ERR-51058 932
P-ERR-51065 933
P-ERR-51066 933
P-ERR-51068 933
P-ERR-51069 934
P-ERR-51070 934
P-ERR-51072 935
P-ERR-51073 936
P-ERR-51079 936
P-ERR-51081 936
P-ERR-51082 937
P-ERR-51083 937
P-ERR-51084 937
P-ERR-51085 938
P-ERR-51086 938
P-ERR-51087 939
P-ERR-51088 939
P-ERR-51089 939
P-ERR-51090 940
P-ERR-51091 940
P-ERR-51092 941
P-ERR-51093 941
P-ERR-51098 942
P-ERR-51101 942
P-ERR-51102 943
P-ERR-51103 943
P-ERR-51109 944
P-ERR-51111 944
P-ERR-51112 945
P-ERR-51114 945
P-ERR-51115 946
P-ERR-51116 946
P-ERR-51117 946
P-ERR-51118 947
P-ERR-51119 947
P-ERR-60006 948
P-ERR-60018 948
P-ERR-60019 949
P-ERR-60020 949
P-ERR-60032 950
P-ERR-60034 950
P-ERR-60040 951
P-ERR-60053 951
P-ERR-60054 951
P-ERR-60085 952
P-ERR-60100 952
P-ERR-60101 953
P-ERR-60102 953
P-ERR-60103 953
P-ERR-60104 954
P-ERR-60105 954
P-ERR-60106 955
P-ERR-60107 956
P-ERR-60131 956
P-ERR-60132 956
P-ERR-60137 957
P-ERR-60138 957
P-ERR-60139 957
P-ERR-60141 958
P-ERR-60142 959
P-ERR-60146 959
P-ERR-60147 959
P-ERR-60154 960
P-ERR-60155 960
P-ERR-60157 961
P-ERR-60158 961
P-ERR-60159 962
P-ERR-60162 962
P-ERR-60164 963
P-ERR-60167 964
P-ERR-60172 964
P-ERR-60173 965
P-ERR-60174 965
P-ERR-60175 965
P-ERR-60176 965
P-ERR-60177 966
P-ERR-60181 966
P-ERR-60182 966
P-ERR-60183 967
P-ERR-60184 967
P-ERR-60186 967
P-ERR-60188 968
P-ERR-60189 968
P-ERR-60190 969
P-ERR-60191 970
P-ERR-60192 971
P-ERR-60193 971
P-ERR-60194 972
P-ERR-60195 973
P-ERR-60196 974
P-ERR-60197 974
P-ERR-60198 974
P-ERR-60199 975
P-ERR-60200 975
P-ERR-60201 975
P-ERR-60207 976
P-ERR-60208 977
P-ERR-60209 977
P-ERR-60210 977
P-ERR-60211 978
P-ERR-60212 978
P-ERR-60213 978
P-ERR-60214 979
P-ERR-60224 979
P-ERR-60226 979
P-ERR-60227 980
P-ERR-60228 980
P-ERR-60229 981
P-ERR-60230 981
P-ERR-60231 981

P-ERR-60232 982
P-ERR-60234 982
P-ERR-60235 982
P-ERR-60237 983
P-ERR-60239 983
P-ERR-60240 984
P-ERR-60241 984
P-ERR-60243 985
P-ERR-60244 985
P-ERR-60246 985
P-ERR-60247 986
P-ERR-60248 986
P-ERR-60250 987
P-ERR-60251 987
P-ERR-60252 987
P-ERR-60253 988
P-ERR-60254 988
P-ERR-60255 988
P-ERR-60257 989
P-ERR-60260 989
P-ERR-60261 990
P-ERR-60262 990
P-ERR-60263 991
P-ERR-60264 991
P-ERR-60265 991
P-ERR-60266 991
P-ERR-60267 992
P-ERR-60268 992
P-ERR-60269 992
P-ERR-60273 993
P-ERR-60274 994
P-ERR-60284 995
P-ERR-60285 995
P-ERR-60286 995
P-ERR-60287 995
P-ERR-60290 996
P-ERR-60291 996
P-ERR-60292 997
P-ERR-60294 997
P-ERR-60296 998
P-ERR-60298 998
P-ERR-60299 999
P-ERR-60300 999
P-ERR-60302 1000
P-ERR-60303 1000
P-ERR-60304 1001
P-ERR-60306 1001
P-ERR-60311 1002
P-ERR-60313 1003
P-ERR-60314 1003
P-ERR-60315 1004
P-ERR-60316 1004
P-ERR-60322 1005
P-ERR-60324 1006
P-ERR-60328 1006
P-ERR-60332 1007
P-ERR-60333 1007
P-ERR-60334 1008
P-ERR-70001 1009
P-ERR-70008 1009
P-ERR-70009 1010
P-ERR-70010 1010
P-ERR-70011 1010
P-ERR-70012 1011
P-ERR-70013 1011
P-ERR-70014 1012
P-ERR-70020 1012
P-ERR-70021 1013
P-ERR-70022 1014
P-ERR-70023 1015
P-ERR-70024 1016
P-ERR-70030 1017
P-ERR-70031 1017
P-ERR-70033 1018
P-ERR-70034 1018
P-ERR-70035 1019
P-ERR-70039 1020
P-ERR-70040 1020
P-ERR-70049 1020
P-ERR-70081 1021
P-ERR-70082 1022
P-ERR-70083 1022
P-ERR-70084 1023
P-ERR-70085 1023
P-ERR-70086 1023
P-ERR-70088 1024
P-ERR-70090 1024
P-ERR-70091 1025
P-ERR-70092 1026
P-ERR-70093 1026
P-ERR-70099 1027
P-ERR-70100 1027
P-ERR-70101 1028
P-ERR-70104 1029
P-ERR-70105 1029
P-ERR-70106 1030
P-ERR-70111 1030
P-ERR-70112 1031
P-ERR-70113 1031
P-ERR-70114 1031
P-ERR-70115 1031
P-ERR-70116 1031
P-ERR-70120 1032
P-ERR-70121 1032
P-ERR-70123 1033
P-ERR-70124 1033
P-ERR-70125 1033
P-ERR-70126 1034
P-ERR-70127 1034
P-ERR-70151 1035
P-ERR-70152 1035
P-ERR-70159 1036
P-ERR-70160 1036
P-ERR-70162 1037
P-ERR-70164 1038
P-ERR-70166 1038
P-ERR-70167 1039
P-ERR-70168 1039
P-ERR-70169 1040
P-ERR-70170 1040
P-ERR-70171 1040
P-ERR-70172 1040
P-ERR-70177 1041
P-ERR-70178 1041
P-ERR-70179 1042
P-ERR-70182 1042

P-ERR-70183 1043
P-ERR-70184 1043
P-ERR-70185 1044
P-ERR-70186 1044
P-ERR-70187 1044
P-ERR-70190 1045
P-ERR-70191 1045
P-ERR-70192 1045
P-ERR-70193 1046
P-ERR-70194 1046
P-ERR-70195 1047
P-ERR-70197 1047
P-ERR-70198 1048
P-ERR-70199 1048
P-ERR-70200 1048
P-ERR-70202 1049
P-ERR-70204 1049
P-ERR-70205 1050
P-ERR-70206 1050
P-ERR-70207 1051
P-ERR-70208 1051
P-ERR-70209 1052
P-ERR-70210 1052
P-ERR-70211 1053
P-ERR-70212 1053
P-ERR-70215 1053
P-ERR-70216 1054
P-ERR-70217 1054
P-ERR-70218 1054
P-ERR-70220 1054
P-ERR-70221 1055
P-ERR-70222 1056
P-ERR-70223 1056
P-ERR-70224 1056
P-ERR-70225 1057
P-ERR-70229 1057
P-ERR-70230 1058
P-ERR-70231 1058
P-ERR-70242 1059
P-ERR-70243 1059
P-ERR-70244 1060
P-ERR-70245 1060
P-ERR-70246 1061
P-ERR-70247 1062
P-ERR-70248 1062
P-ERR-70249 1062
P-ERR-70250 1064
P-ERR-70251 1064
P-ERR-70252 1065
P-ERR-70253 1065
P-ERR-70254 1066
P-ERR-70255 1066
P-ERR-70256 1067
P-ERR-70259 1067
P-ERR-70261 1068
P-ERR-70262 1068
P-ERR-70263 1069
P-ERR-70264 1069
P-ERR-70265 1069
P-ERR-70266 1070
P-ERR-70267 1070
P-ERR-70268 1071
P-ERR-70269 1071
P-ERR-70272 1071
P-ERR-70273 1072
P-ERR-70274 1072
P-ERR-70275 1072
P-ERR-70276 1073
P-ERR-70277 1073
P-ERR-70278 1074
P-ERR-70279 1074
P-ERR-70280 1074
P-ERR-70282 1075
P-ERR-70283 1075
P-ERR-70284 1075
P-ERR-70285 1076
P-ERR-70286 1076
P-ERR-70287 1077
P-ERR-70288 1077
P-ERR-70290 1078
P-ERR-70291 1078
P-ERR-70292 1078
P-ERR-70293 1079
P-ERR-70294 1079
P-ERR-70295 1080
P-ERR-70296 1080
P-ERR-70297 1080
P-ERR-70298 1081
P-ERR-70299 1081
P-ERR-70300 1082
P-ERR-70302 1082
P-ERR-70304 1083
P-ERR-70305 1083
P-ERR-70306 1084
P-ERR-70309 1085
P-ERR-70310 1085
P-ERR-70311 1085
P-ERR-70312 1086
P-ERR-70313 1086
P-ERR-70314 1086
P-ERR-70315 1087
P-ERR-70316 1087
P-ERR-70317 1087
P-ERR-70318 1088
P-ERR-70319 1088
P-ERR-70320 1088
P-ERR-70321 1089
P-ERR-70324 1089
P-ERR-70325 1090
P-ERR-70326 1090
P-ERR-70329 1091
P-ERR-70330 1091
P-ERR-70331 1092
P-ERR-70332 1093
P-ERR-70333 1093
P-ERR-70334 1093
P-ERR-70335 1094
P-ERR-70336 1094
P-ERR-70337 1094
P-ERR-70338 1095
P-ERR-70339 1095
P-ERR-70340 1095
P-ERR-70341 1096
P-ERR-70342 1096
P-ERR-70343 1096
P-ERR-70344 1097

P-ERR-70345 1097
P-ERR-70346 1098
P-ERR-70348 1098
P-ERR-70349 1098
P-ERR-70350 1099
P-ERR-70351 1099
P-ERR-70355 1100
P-ERR-70358 1100
P-ERR-70359 1101
P-ERR-70361 1101
P-ERR-70362 1102
P-ERR-70363 1102
P-ERR-70366 1102
P-ERR-70367 1103
P-ERR-70368 1103
P-ERR-70369 1104
P-ERR-70370 1104
P-ERR-70371 1104
P-ERR-70372 1105
P-ERR-70373 1105
P-ERR-70374 1106
P-ERR-70375 1106
P-ERR-70376 1106
P-ERR-70377 1107
P-ERR-70378 1107
P-ERR-70379 1108
P-ERR-70380 1108
P-ERR-70381 1109
P-ERR-70382 1109
P-ERR-70383 1110
P-ERR-70384 1111
P-ERR-70385 1111
P-ERR-70386 1111
P-ERR-70387 1112
P-ERR-70388 1112
P-ERR-70389 1112
P-ERR-70390 1113
P-ERR-70391 1114
P-ERR-70394 1114
P-ERR-70395 1115
P-ERR-70396 1115
P-ERR-70397 1116
P-ERR-70399 1116
P-ERR-70400 1117
P-ERR-70401 1117
P-ERR-70402 1118
P-ERR-70403 1118
P-ERR-70404 1119
P-ERR-70405 1119
P-ERR-70406 1119
P-ERR-70410 1120
P-ERR-70411 1120
P-ERR-70420 1121
P-ERR-70421 1121
P-ERR-70423 1122
P-ERR-70425 1123
P-ERR-70426 1123
P-ERR-70428 1124
P-ERR-70429 1124
P-ERR-70430 1125
P-ERR-70431 1125
P-ERR-70432 1126
P-ERR-70433 1126
P-ERR-70434 1127
P-ERR-70435 1127
P-ERR-70436 1128
P-ERR-70437 1129
P-ERR-70438 1129
P-ERR-70440 1130
P-ERR-70441 1130
P-ERR-70442 1131
P-ERR-70445 1131
P-ERR-70447 1132
P-ERR-70449 1132
P-ERR-70450 1133
P-ERR-70451 1133
P-ERR-70452 1134
P-ERR-70453 1134
P-ERR-70454 1134
P-ERR-70455 1135
P-ERR-70456 1135
P-ERR-70457 1135
P-ERR-70458 1136
P-ERR-70459 1136
P-ERR-70460 1137
P-ERR-70461 1137
P-ERR-70462 1137
P-ERR-70463 1138
P-ERR-70464 1138
P-ERR-70466 1139
P-ERR-70467 1139
P-ERR-70468 1139
P-ERR-70469 1139
P-ERR-70470 1140
P-ERR-70471 1140
P-ERR-70472 1140
P-ERR-70473 1141
P-ERR-70474 1141
P-ERR-70475 1141
P-ERR-70478 1142
P-ERR-70479 1142
P-ERR-70480 1143
P-ERR-70482 1143
P-ERR-70483 1144
P-ERR-70484 1144
P-ERR-70485 1144
P-ERR-70487 1145
P-ERR-70488 1145
P-ERR-70489 1146
P-ERR-70490 1146
P-ERR-70491 1146
P-ERR-70492 1147
P-ERR-70493 1147
P-ERR-70494 1147
P-ERR-70495 1147
P-ERR-70497 1148
P-ERR-70499 1148
P-ERR-70500 1150
P-ERR-70501 1150
P-ERR-70502 1151
P-ERR-70503 1151
P-ERR-70504 1151
P-ERR-70505 1151
P-ERR-70506 1152
P-ERR-70507 1152
P-ERR-70508 1152

P-ERR-70509 1153
P-ERR-70510 1153
P-ERR-70511 1153
P-ERR-70512 1154
P-ERR-70513 1154
P-ERR-70514 1154
P-ERR-70515 1154
P-ERR-70516 1155
P-ERR-70519 1155
P-ERR-70520 1155
P-ERR-70521 1156
P-ERR-70523 1156
P-ERR-70525 1157
P-ERR-70526 1157
P-ERR-70528 1158
P-ERR-70529 1158
P-ERR-70530 1158
P-ERR-70531 1159
P-ERR-70532 1159
P-ERR-70533 1159
P-ERR-70534 1160
P-ERR-70535 1160
P-ERR-70536 1161
P-ERR-70537 1161
P-ERR-70538 1162
P-ERR-70539 1162
P-ERR-70540 1162
P-ERR-70541 1162
P-ERR-70544 1163
P-ERR-70546 1164
P-ERR-70547 1164
P-ERR-70548 1164
P-ERR-70549 1165
P-ERR-70550 1165
P-ERR-70551 1166
P-ERR-70552 1166
P-ERR-70554 1167
P-ERR-70555 1167
P-ERR-70556 1168
P-ERR-70558 1168
P-ERR-70559 1169
P-ERR-70561 1169
P-ERR-70562 1170
P-ERR-70563 1170
P-ERR-70569 1171
P-ERR-70574 1171
P-ERR-70575 1172
P-ERR-70576 1172
P-ERR-70582 1173
P-ERR-70583 1173
P-ERR-70584 1174
P-ERR-70585 1174
P-ERR-70586 1174
P-ERR-70587 1174
P-ERR-70588 1175
P-ERR-70589 1175
P-ERR-70590 1175
P-ERR-70591 1176
P-ERR-70592 1176
P-ERR-70593 1177
P-ERR-70594 1177
P-ERR-70595 1178
P-ERR-70599 1178
P-ERR-70600 1179
P-ERR-70601 1179
P-ERR-70602 1180
P-ERR-70603 1180
P-ERR-70604 1181
P-ERR-70605 1181
P-ERR-70606 1182
P-ERR-70607 1182
P-ERR-70610 1183
P-ERR-70612 1183
P-ERR-70614 1184
P-ERR-70615 1185
P-ERR-70616 1186
P-ERR-70617 1187
P-ERR-70619 1187
P-ERR-70621 1188
P-ERR-70622 1188
P-ERR-70625 1189
P-ERR-70626 1189
P-ERR-70627 1190
P-ERR-70629 1190
P-ERR-70631 1191
P-ERR-70632 1191
P-ERR-70634 1191
P-ERR-70635 1191
P-ERR-70636 1192
P-ERR-70642 1192
P-ERR-70643 1192
P-ERR-70644 1193
P-ERR-70645 1193
P-ERR-70647 1194
P-ERR-70648 1194
P-ERR-70649 1195
P-ERR-70650 1195
P-ERR-70651 1195
P-ERR-70652 1196
P-ERR-70653 1196
P-ERR-70654 1197
P-ERR-70655 1197
P-ERR-70656 1197
P-ERR-70657 1198
P-ERR-70658 1198
P-ERR-70659 1198
P-ERR-70660 1199
P-ERR-70661 1199
P-ERR-70662 1199
P-ERR-70671 1200
P-ERR-70681 1200
P-ERR-70682 1200
P-ERR-80000 1202
P-ERR-80005 1202
P-ERR-80006 1202
P-ERR-80007 1203
P-ERR-80008 1203
P-ERR-80009 1203
P-ERR-80010 1204
P-ERR-80011 1204
P-ERR-80044 1205
P-ERR-90008 1206
P-ERR-90011 1206
P-ERR-90013 1207
P-ERR-90014 1207
P-ERR-90015 1208

P-ERR-90016 1208
P-ERR-90017 1208
P-ERR-90018 1209
P-ERR-90019 1209
P-ERR-90026 1210
P-ERR-90031 1211
P-ERR-90036 1211
P-ERR-90038 1212
P-ERR-90041 1214
P-ERR-90046 1214
P-ERR-90047 1214
P-ERR-90049 1215
P-ERR-90050 1215
P-ERR-90057 1216
P-ERR-90062 1217
P-ERR-90063 1217
P-ERR-90064 1217
P-ERR-90065 1218
P-ERR-90066 1218
P-ERR-90074 1219
P-ERR-90076 1219
P-ERR-90077 1219
P-ERR-90087 1220
P-ERR-90088 1220
P-ERR-90090 1220
P-ERR-90098 1221
P-ERR-90099 1221
P-ERR-90100 1221
P-ERR-90107 1222
P-ERR-90109 1222
P-ERR-90112 1222
P-ERR-90114 1223
P-ERR-90115 1223
P-ERR-90116 1223
P-ERR-90119 1224
P-ERR-90123 1224
P-ERR-90124 1225
P-ERR-90125 1225
P-ERR-90126 1225
P-ERR-90127 1226
P-ERR-90128 1226
P-ERR-90130 1226
P-ERR-90131 1227
P-ERR-90132 1227
P-ERR-90133 1228
P-ERR-90134 1228
P-ERR-90137 1229
P-ERR-90141 1229
P-ERR-90142 1230
P-ERR-90143 1230
P-ERR-90145 1230
P-ERR-90146 1231
P-ERR-90147 1231
P-ERR-90151 1232
P-ERR-90154 1232
P-ERR-90157 1233
P-ERR-90158 1233
P-ERR-90159 1233
P-ERR-90160 1233
P-ERR-90162 1234
P-ERR-90164 1235
P-ERR-90165 1235
P-ERR-90166 1235
P-ERR-90172 1236
P-ERR-90178 1237
P-ERR-90179 1237
P-ERR-90180 1237
P-ERR-90181 1238
P-ERR-90182 1238
P-ERR-90183 1238
P-ERR-90184 1238
P-ERR-90190 1239
P-ERR-90191 1239
P-ERR-90192 1239
P-ERR-90193 1240
P-ERR-90194 1240
P-ERR-90202 1241
P-ERR-90203 1242
P-ERR-90204 1242

More Information:
www.beckhoff.com/TF5200

Beckhoff Automation GmbH & Co. KG
Hülshorstweg 20
33415 Verl
Germany
Phone: +49 5246 9630
info@beckhoff.com
www.beckhoff.com