

Design Guide | EN

EJ8xxx

Signal distribution board for standard EtherCAT plug-in modules

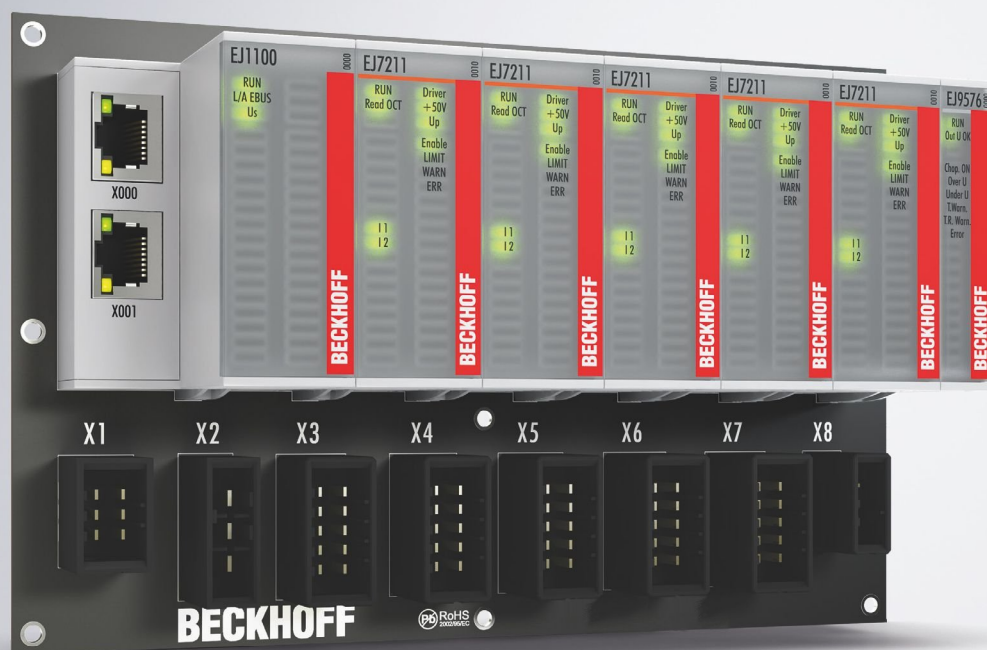


Table of contents

1 Foreword	5
1.1 Notes on the documentation	5
1.2 Safety instructions	6
1.3 Documentation issue status	7
1.4 Purpose and area of application	7
1.5 Guide through documentation	8
2 Technical data - Signal distribution board	9
3 Backplane mounting guidelines	10
3.1 Minimum distances for ensuring installability	10
3.2 Installation positions	11
4 Module placement	12
5 PCB - distances and footprint	14
6 Design of E-bus power supply	15
7 Power supply and potential groups	17
8 SGND Connection	19
9 Structure of the PCB layers	20
9.1 Top layer	21
9.2 Inner layer 1	22
9.3 Inner layer 2	24
9.4 Bottom layer	25
10 Routing guidelines	26
10.1 EMC guidelines	27
10.2 Impedance and Routing	27
11 Appendix	28
11.1 Support and Service	28

1 Foreword

1.1 Notes on the documentation

Intended audience

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 these components.

The qualified personnel is obliged to always use the currently valid documentation.

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.



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

Copyright

© Beckhoff Automation GmbH & Co. KG, Germany.

The reproduction, distribution and utilization of this document as well as the communication of its contents to others without express authorization 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.

Third-party brands

Trademarks of third parties may be used in this documentation. You can find the trademark notices here: <https://www.beckhoff.com/trademarks>

1.2 Safety instructions

Safety regulations

Please note the following safety instructions and explanations!

Product-specific safety instructions can be found on following pages or in the areas mounting, wiring, commissioning etc.

Exclusion of liability

All the components are supplied in particular hardware and software configurations appropriate for the application. Modifications to hardware or software configurations other than those described in the documentation are not permitted, and nullify the liability of Beckhoff Automation GmbH & Co. KG.

Personnel qualification

This description is only intended for trained specialists in control, automation and drive engineering who are familiar with the applicable national standards.

Signal words

The signal words used in the documentation are classified below. In order to prevent injury and damage to persons and property, read and follow the safety and warning notices.

Personal injury warnings

DANGER

Hazard with high risk of death or serious injury.

WARNING

Hazard with medium risk of death or serious injury.

CAUTION

There is a low-risk hazard that could result in medium or minor injury.

Warning of damage to property or environment

NOTICE

The environment, equipment, or data may be damaged.

Information on handling the product



This information includes, for example:
recommendations for action, assistance or further information on the product.

1.3 Documentation issue status

Version	Comment
6.5	<ul style="list-style-type: none">• Updated <i>PCB – clearances and footprint</i> chapter• Structural update
6.4	<ul style="list-style-type: none">• Updated <i>Technical data – signal distribution board</i> chapter• <i>SGND connection</i> chapter added• Updated <i>Voltages and potential groups</i> chapter• Structural update
6.3	<ul style="list-style-type: none">• Updated <i>Voltages and potential groups</i> chapter
6.2	<ul style="list-style-type: none">• New title page• Updated <i>Technical data – signal distribution board</i> chapter• <i>Voltages and potential groups</i> chapter added• Updated <i>Structure of the PCB levels</i> chapter• Updated <i>Routing guideline</i> chapter• Structural update
6.1	<ul style="list-style-type: none">• Updated <i>Structure of the PCB levels</i> chapter
6.0	<ul style="list-style-type: none">• Migration• Structural update

1.4 Purpose and area of application

This document is meant for developers who would like to create a backplane or an EJ distribution board upon which standard EtherCAT plug-in modules should be used.

This document gives information about the general design of an EJ distribution board for standard EtherCAT plug-in modules.

When secure EJ modules should be used along with standard EtherCAT plug-in modules please note the additional requirements and notes of the supplementary Design Guide [EJ-Backplane for TwinSAFE-modules](#).

1.5 Guide through documentation

NOTICE



Further components of this documentation

This documentation is part of the modular documentation concept for Beckhoff I/O components. The documentation listed in the following table is also required for the use and safe operation of the EtherCAT plug-in modules.

Title	Description
EJ backplane for TwinSAFE modules (PDF)	Notes on the design of an EJ distribution board that is the basis for using safety EJ modules with standard EJ modules <ul style="list-style-type: none"> • Additional requirements for the signal distribution board when using safety EJ modules
Documentation of the corresponding EtherCAT plug-in modules EJxxxx	<ul style="list-style-type: none"> • Technical data, • Pinout, • LEDs, • Notes on the principle of operation and • Description of configuration and parameterization
EtherCAT System Documentation (PDF)	<ul style="list-style-type: none"> • System overview • EtherCAT basics • Cable redundancy • Hot Connect • EtherCAT devices configuration
Infrastructure for EtherCAT/Ethernet (PDF)	Technical recommendations and notes for design, implementation and testing
Software Declarations I/O (PDF)	Open source software declarations for Beckhoff I/O components

The documentations can be viewed at and downloaded from the Beckhoff website (www.beckhoff.com) via:

- the “Documentation and Download” area of the respective product page,
- the [Download finder](#),
- the [Beckhoff Information System](#).

If you have any suggestions or proposals for our documentation, please send us an e-mail stating the documentation title and version number to: documentation@beckhoff.com

2 Technical data - Signal distribution board

Technical data	Signal distribution board
PCB layer stack	Multilayer PCB, min. 4 layers
PCB thickness	1.6 mm \pm 10 %
Board connector	Samtec: SSQ-120-01-L-D
Placing of modules [► 12]	Starting from the left: Coupler, power supply module, EJ-modules
E-bus power supply [► 15]	EJ1100 (2.2 A) EJ1101-0022 + (EJ9400 (2.5 A) or EJ9404 (12 A))
Refresh of E-bus power supply [► 16]	EJ9400 (2.5 A) EJ9404 (12 A)
Air gap and leakage distances [► 13] between E-bus- and field signals	typ. 1.2 mm
Differential impedance [► 27] of the LVDS traces	100 Ω
SGND connection	via mounting bolts
Mounting hole distances [► 10]	max. 100 mm
Reaching area [► 10]	92 mm
Module depth above PCB	min. 55 mm
Distance between PCB and mounting surface	min. 4 mm
Mounting position [► 11]	Standard

Description of potential groups	
U _{EBUS} : Power supply E-bus	The power supply U _{EBUS} is provided by the coupler and supplied from the supply voltage U _s of the EtherCAT coupler. U _{EBUS} : E-bus power supply 3.3 V GND: E-bus GND signal; Notice Don't connect with 0 V Up and 0 V Us!
Us: Power supply bus side	The power supply U _s is used to supply the bus coupler electronics and to generate voltage for the E-bus (U _{EBUS}). 0 V U _s : Bus side GND signal Notice Don't connect with GND (E-bus GND signal)! 24 V U _s : Bus side power supply 24 V
Up: Power supply field side	The peripheral voltage U _p supplies the electronics on the field side. 0 V U _p : GND signal field side Notice Don't connect with GND (E-bus GND signal)! 24 V U _p : Power supply field side 24 V
SGND: Shield Ground	Ground signal with shielding function in relation to the rest of the board Notice U _s , U _p and SGND must not be in direct contact with each other! Connect the SGND signal directly to the mounting plate via metal bolts (see chapter SGND Connection [► 19])

NOTICE

Damage to devices possible

Note the specifications and notes for the components used!

3 Backplane mounting guidelines

In order to avoid physical stress on the backplane in the module installation process, the signal distribution board should be mounted in the control cabinet as described in the following figure.

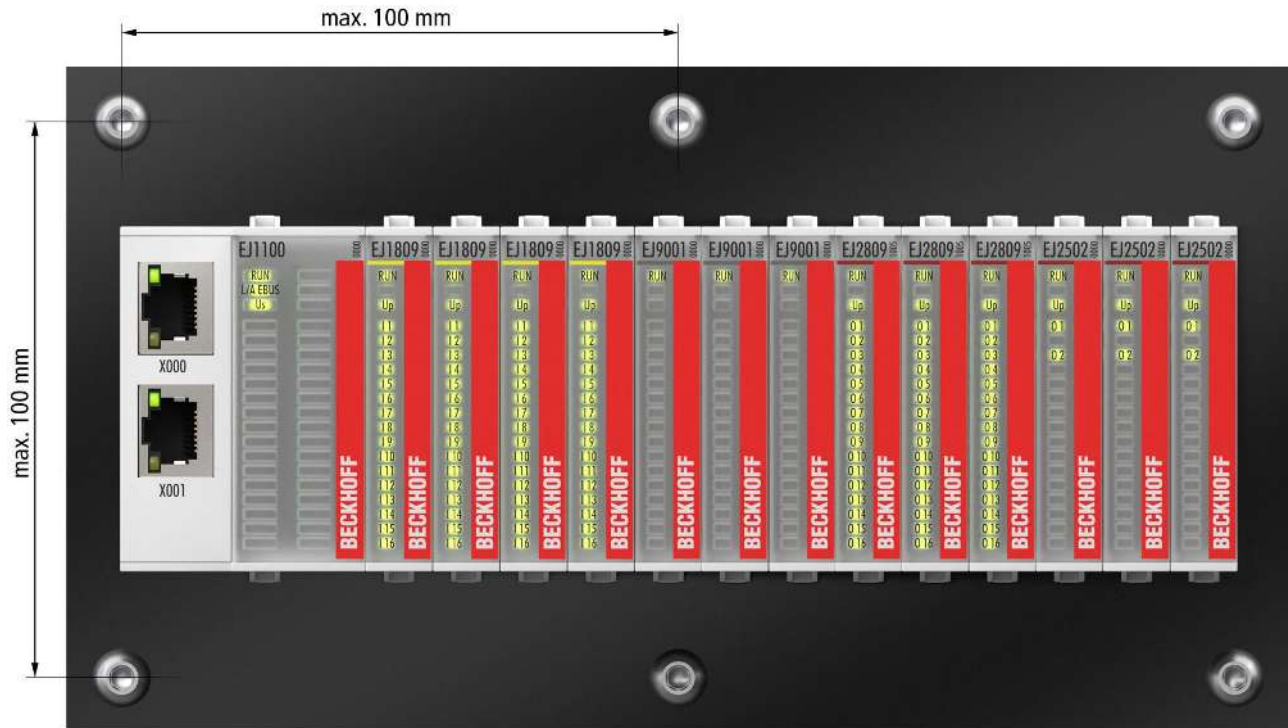


Fig. 1: Maximum distances between mounting holes and PCB

3.1 Minimum distances for ensuring installability

Note the dimensions shown in the following diagram for the design of the signal distribution board to ensure safe latching and simple assembly / disassembly of the modules.

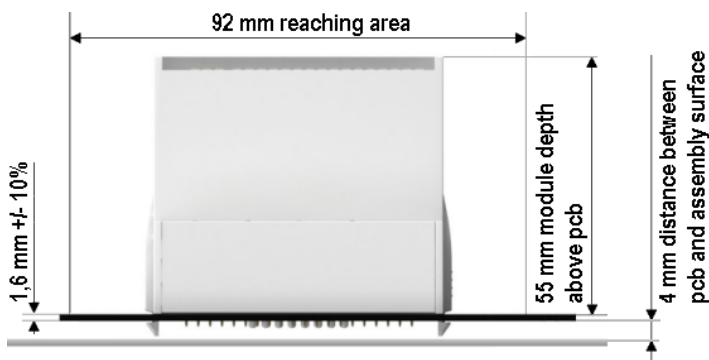


Fig. 2: Mounting distances EJ module - PCB

i Observing the reaching area

A minimum reaching area of 92 mm is required for assembly / disassembly, in order to be able to reach the mounting tabs with the fingers.

Adherence to the recommended minimum distances for ventilation (see [section Installation position](#) [▶ 11](#)) ensures an adequate reaching area.

The signal distribution board must have a thickness of 1.6 mm and a minimum distance of 4 mm from the mounting surface, in order to ensure latching of the modules on the board.

3.2 Installation positions

NOTICE

Constraints regarding installation position and operating temperature range

Please refer to the [technical data](#) [► 9] for the installed components to ascertain whether any restrictions regarding the mounting position and/or the operating temperature range have been specified. During installation of modules with increased thermal dissipation, ensure adequate distance above and below the modules to other components in order to ensure adequate ventilation of the modules during operation!

The standard installation position is recommended. If a different installation position is used, check whether additional ventilation measures are required.

Ensure that the specified conditions (see Technical data) are adhered to!

Optimum installation position (standard)

For the optimum installation position the signal distribution board is installed horizontally, and the fronts of the EJ modules face forward (see Fig. "Recommended distances for standard installation position"). The modules are ventilated from below, which enables optimum cooling of the electronics through convection. "From below" is relative to the acceleration of gravity.

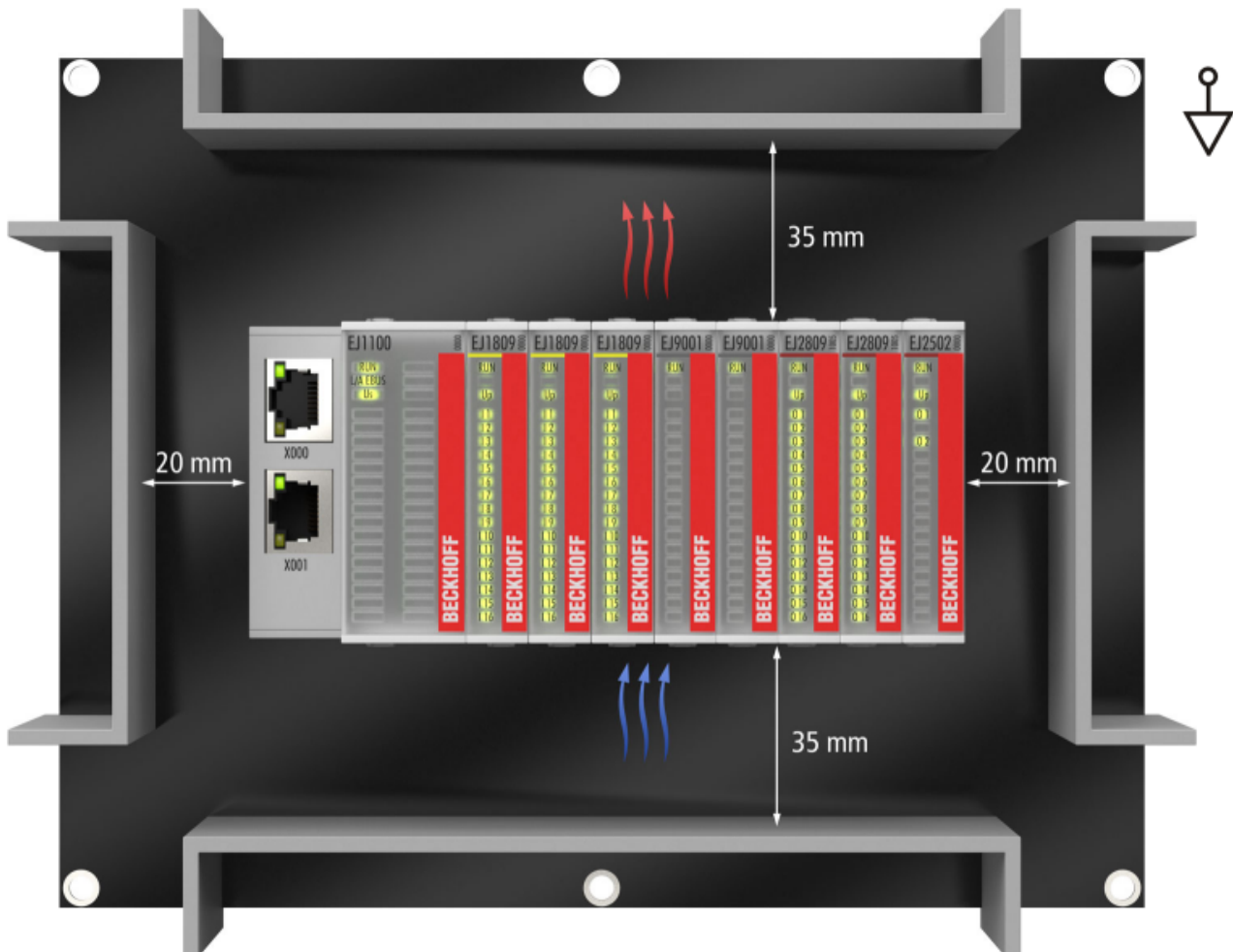


Fig. 3: Recommended distances for standard installation position

Compliance with the distances shown in Fig. "Recommended distances for standard installation position" is recommended. The recommended minimum distances should not be regarded as restricted areas for other components. The customer is responsible for verifying compliance with the environmental conditions described in the technical data. Additional cooling measures must be provided, if required.

4 Module placement

The EJ-Module line shall begin on the left side of the single distribution board with the coupler (or RJ45 plugs) followed by a power supply and IO-modules.

In order to avoid electromagnetical interferences on the E-bus it is not recommended to route IO-connection signals through the E-bus routing area marked in the following figures.

Notes for routing

i Follow the instructions for routing in chapter [Design of power supply](#) [▶ 16] and [Routing guidelines](#) [▶ 26]!

Example with coupler EJ1100

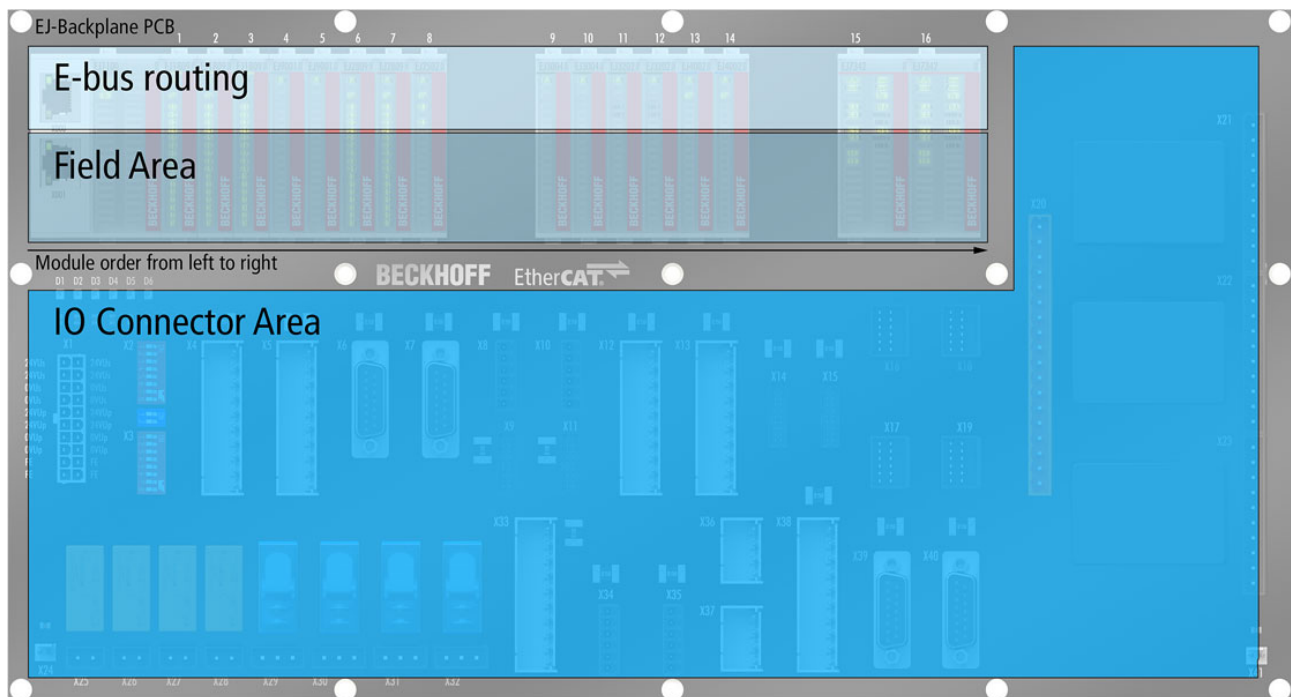


Fig. 4: EJ module line starting with coupler EJ1100

Example with coupler EJ1101-0022 and power supply module EJ9400

An additional power supply module (e.g. EJ9400) and RJ45 sockets are required when using coupler EJ1101-0022. The RJ45 sockets should be placed near the coupler. Crossing the EtherCAT RX/TX lines between the coupler and the modular jacks with signals that may carry electromagnetical interference shall be avoided.

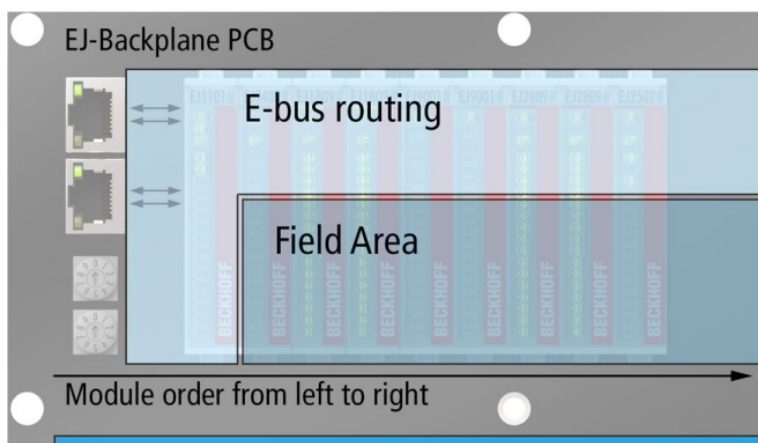


Fig. 5: EJ module line starting with coupler EJ1101-0022 (optional RJ45 sockets)

Clearances and creepage distances

Between field and E-bus signals clearances and creepage distances have to be taken care of. A Clearance of 1.2 mm is recommended.

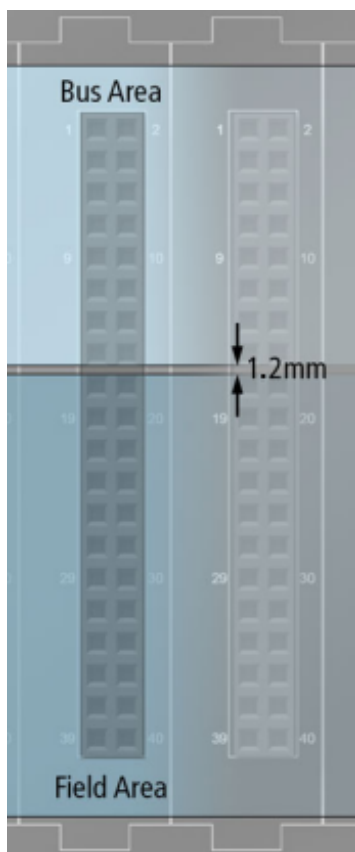


Fig. 6: Clearance between bus- and field area

5 PCB - distances and footprint

In the following figure the footprint, position of coding pins (A), holes for the connector pins (B) and locking holes (C) are shown.

The locking holes and the holes for the upper left contact pin of the module connector (B1) are in x-direction 0.03 mm away.

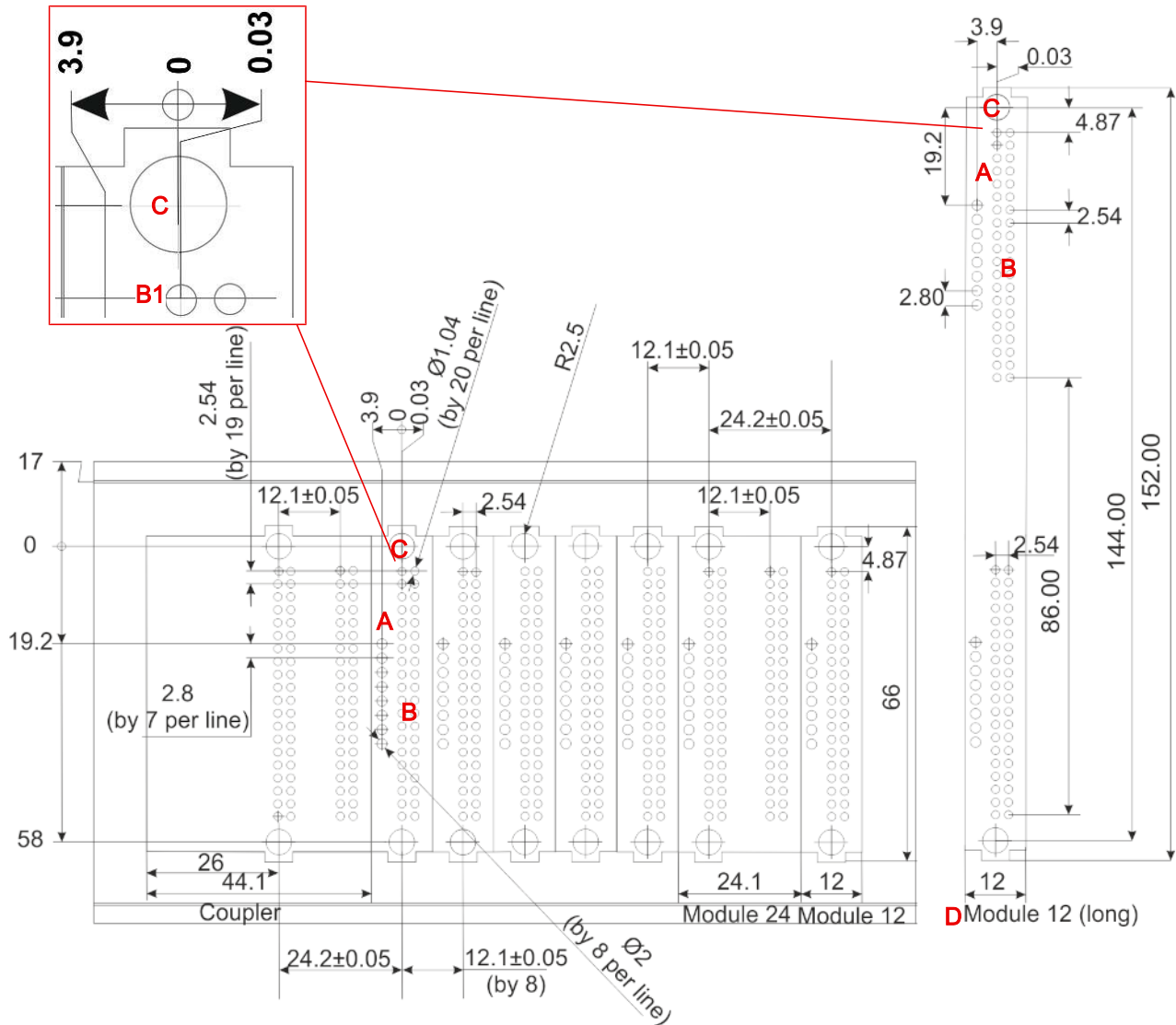


Fig. 7: Backplane layout, dimensions in mm

Long EtherCAT plug-in modules

The distance to neighboring modules should be at least 12.1 mm, measured from the centre of one opening to the next.

The technical drawings can be downloaded from the [download finder](#). The drawings are named as described in the adjacent illustration

Positon of coding pins

Amount of contact pins

Housing width in mm

```
> ej_12_16pin_code13
```


6 Design of E-bus power supply

⚠ WARNING

Power supply from SELV / PELV power supply unit!

SELV / PELV circuits (safety extra-low voltage / protective extra-low voltage) according to IEC 61010-2-201 must be used to supply this device.

Notes:

- SELV / PELV circuits may give rise to further requirements from standards such as IEC 60204-1 et al, for example with regard to cable spacing and insulation.
- A SELV supply provides safe electrical isolation and limitation of the voltage without a connection to the protective conductor, a PELV supply also requires a safe connection to the protective conductor.

The signal distribution board should have a power supply designed for the maximum possible current load of the module string. Information on the current required from the E-bus supply can be found for each module in the respective documentation in section “Technical data”, online and in the catalog. The power requirement of the module string is displayed in the TwinCAT System Manager.

E-bus power supply with EJ1100 or EJ1101-0022 and EJ940x

The EJ1100 Bus Coupler supplies the connected EJ modules with the E-bus system voltage of 3.3 V. The coupler can accommodate a load up to 2.2 A. If a higher current is required, a combination of the coupler EJ1101-0022 and the power supply units EJ9400 (2.5 A) or EJ9404 (12 A) should be used. The EJ940x power supply units can be used as additional supply modules in the module string.

Depending on the application, the following combinations for the E-bus supply are available:

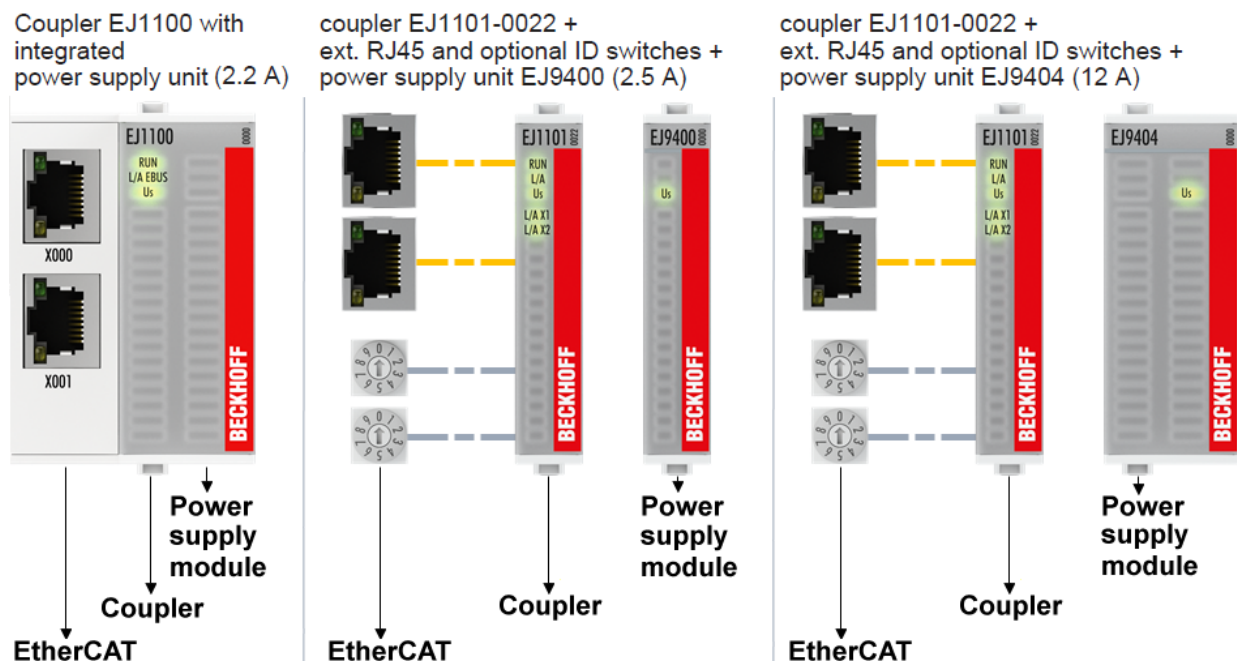


Fig. 8: E-bus power supply with EJ1100 or EJ1101-0022 + EJ940x

In the EJ1101-0022 coupler, the RJ45 plugs and optional ID switches are external and can be positioned anywhere on the signal distribution board, as required. This facilitates feeding through a housing.

Additional E-bus power supply with EJ940x

The power supply modules EJ940x can be used as additional supply modules in the module line.

When adding an additional power supply module in an EJ line only the supply voltages for the E-bus (U_{EBUS}) have to be separated into two or more nets. The E-bus GND is common for all EtherCAT plug-in modules in the design.

As examples, two designs with additional power supply modules are shown below.

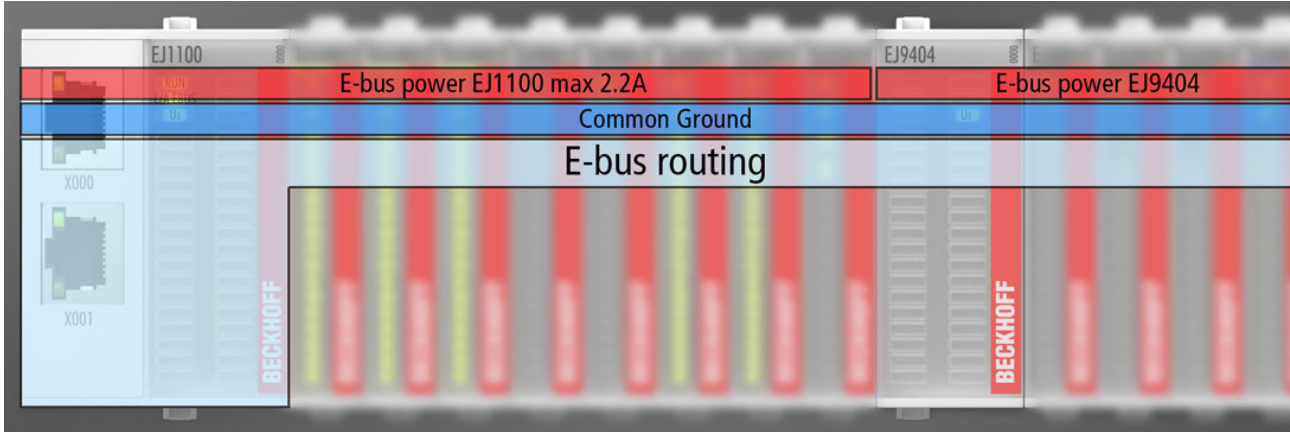


Fig. 9: Example with Coupler EJ1100, with integrated power supply (2.2 A), additional power supply with EJ9404 (12 A)

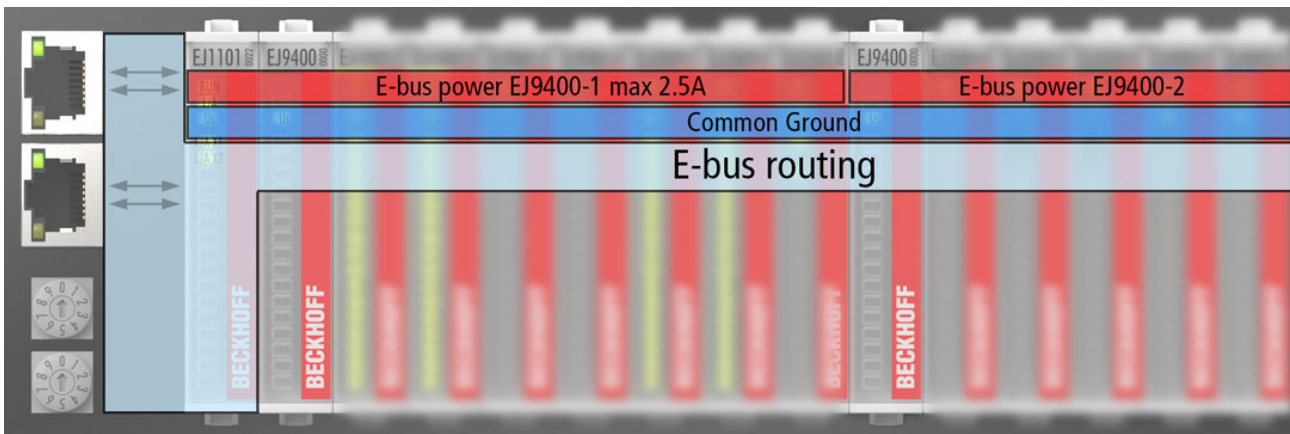


Fig. 10: Example with Coupler EJ1101-0022, power supply module EJ9400 (2.5 A), additional power supply with EJ9400 (2.5 A)

7 Power supply and potential groups

The board is supplied via two galvanically isolated 24 V power supplies:

- The power supply U_s is used to supply the bus coupler electronics and to generate voltage for the E-Bus (U_{EBUS} : 3.3 V)
- The peripheral voltage U_p supplies the electronics on the field side.

SGND (Shield Ground) is a ground signal with shielding function in relation to the rest of the board.

NOTICE

Damage to devices possible

- U_s , U_p and SGND must not be in direct contact with each other!
- Don't connect E-bus GND signal (GND) with 0 V U_p and 0 V U_s !
- The SGND connection to the mounting plate shall be implemented as metal bolts building a direct connection between the signal distribution board and mounting plate (see chapter [SGND Connection](#) [► 19]).

Example with the EJ1100 EtherCAT coupler

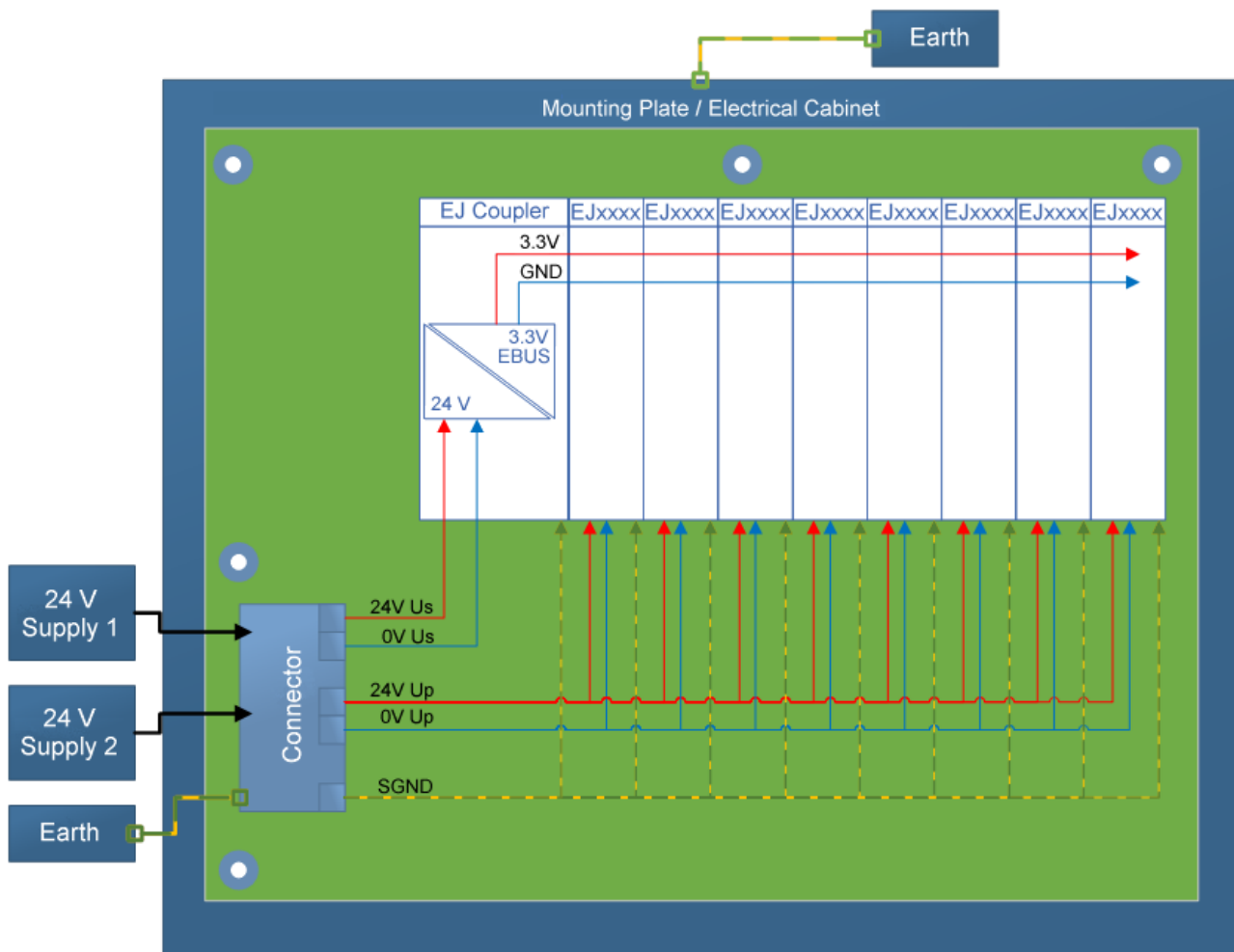


Fig. 11: Power supply of the EtherCAT plug-in modules via the EJ1100 EtherCAT coupler

Example with the EJ1101-0022 EtherCAT coupler and power supply module EJ940x

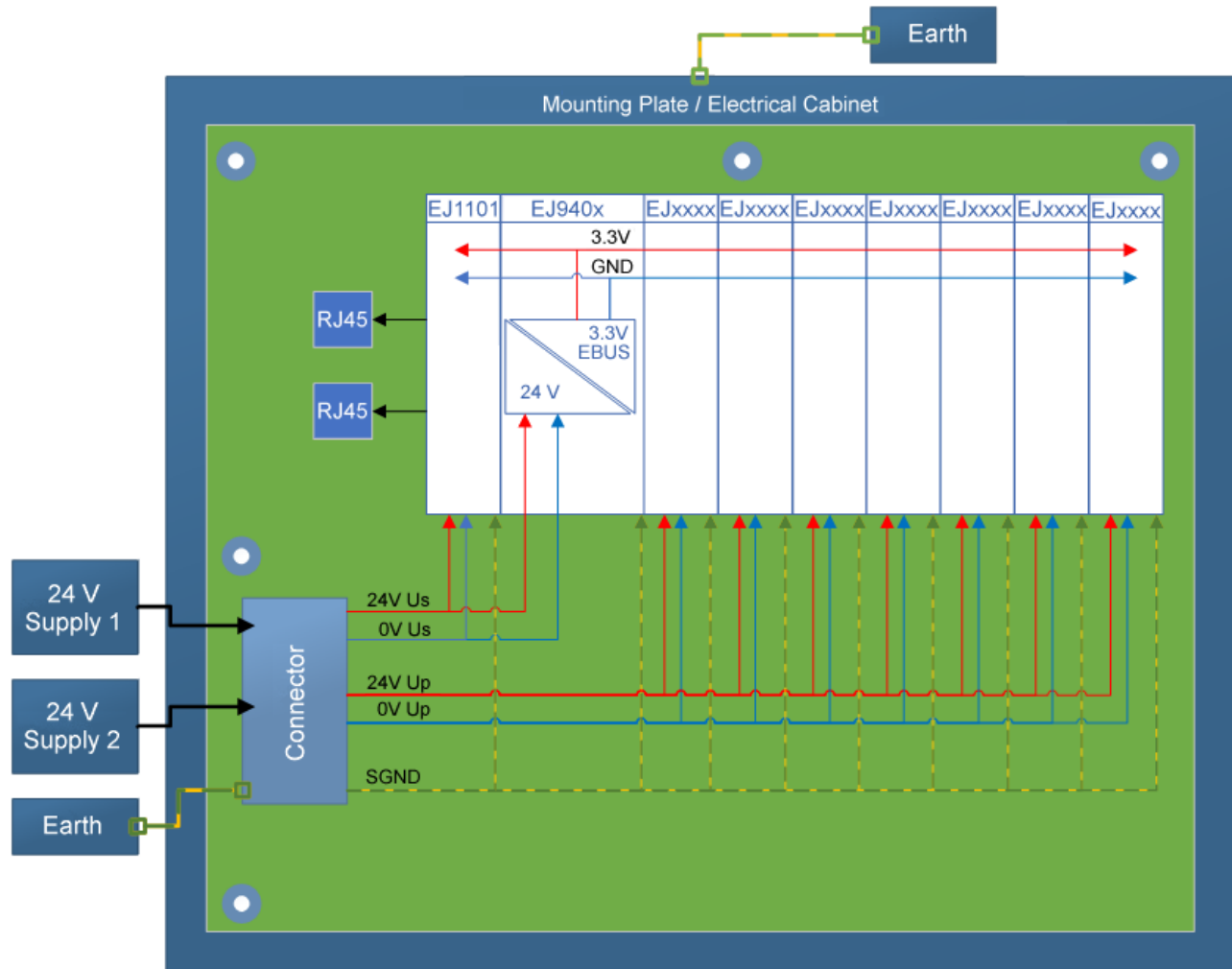


Fig. 12: Power supply of the EtherCAT plug-in modules via the EJ1101-0022 EtherCAT coupler and power supply module EJ940x

NOTICE

Note on routing

- Read the notes on routing in chapter [Module placement](#) [► 12], [Design of E-bus power supply](#) [► 15] and [Routing guidelines](#) [► 26]!
- Observe additional notes in chapter *pinout* of the documentations for the modules used.

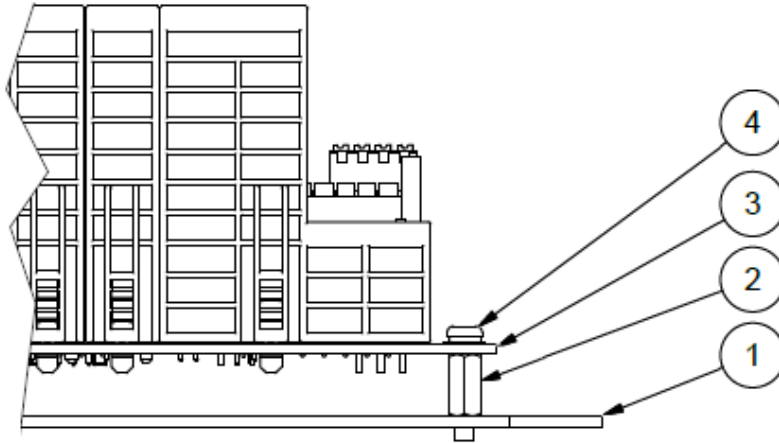
8 SGND Connection

NOTICE

Notes on the SGND connection

The copper rings around the holes are connected to SGND (see chapter [Top layer](#) [► 21]).

- SGND connection to the mounting plate shall be implemented as metal bolts building a direct connection between signal distribution board and mounting plate (see following figure).



Direct connection to the mounting plate via metal bolts

- 1 Mounting plate
- 2 Bolt e. g. DIA M3x10 (Steel, galvanized)
- 3 Signal distribution board
- 4 Combination screw e. g. M3x8 T10 (Steel, galvanized)

9 Structure of the PCB layers

Requirements of the PCB

A multilayer PCB with at least four Layers is recommended for EJ-Backplane, in order to allow complete covering of the differential pairs with copper (GND net) from both sides of the PCB.

NOTICE

Avoid damage of backplane and components!

Short circuit condition has to be taken into account for cross section configuration.

The snap in mechanism of the EJ-Modules is designed for a PCB thickness of $1.6 \text{ mm} \pm 10\%$.

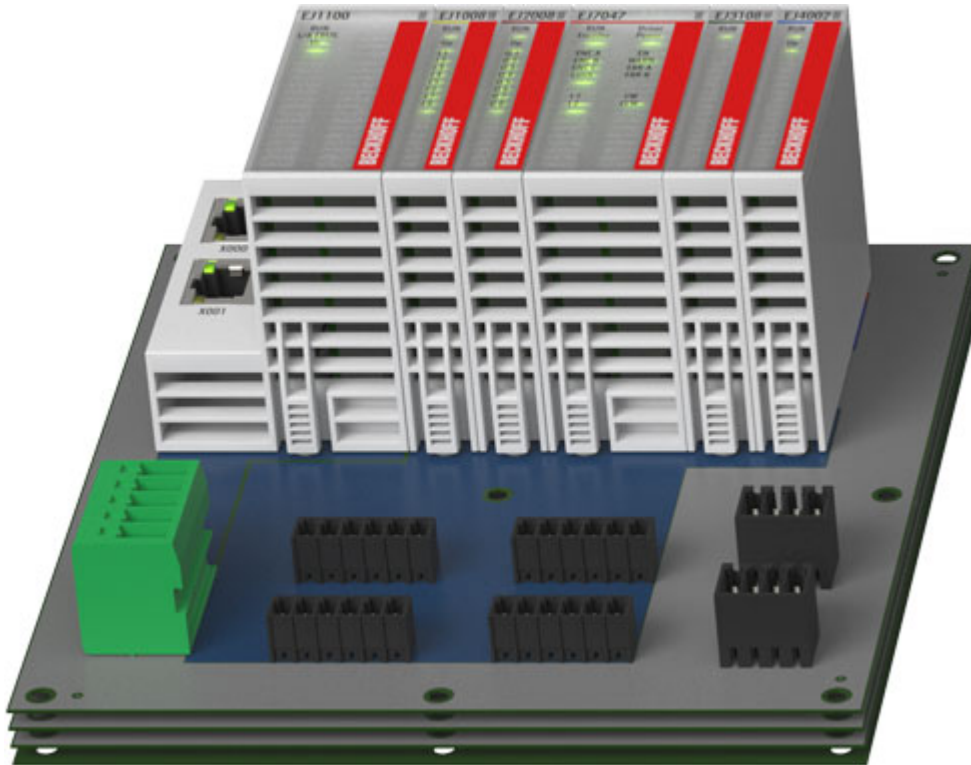


Fig. 13: Requirements of the PCB (min. four layers, max. 1.6 mm thickness)

The following chapters show an example for a PCB with four layers with the routing in the individual layers.

NOTICE

Note on routing

- Read the notes on routing in chapter [Module placement](#) [► 12], [Design of E-bus power supply](#) [► 15] and [Routing guidelines](#) [► 26]!
- Observe additional notes in chapter *pinout* of the documentations for the modules used.

9.1 Top layer

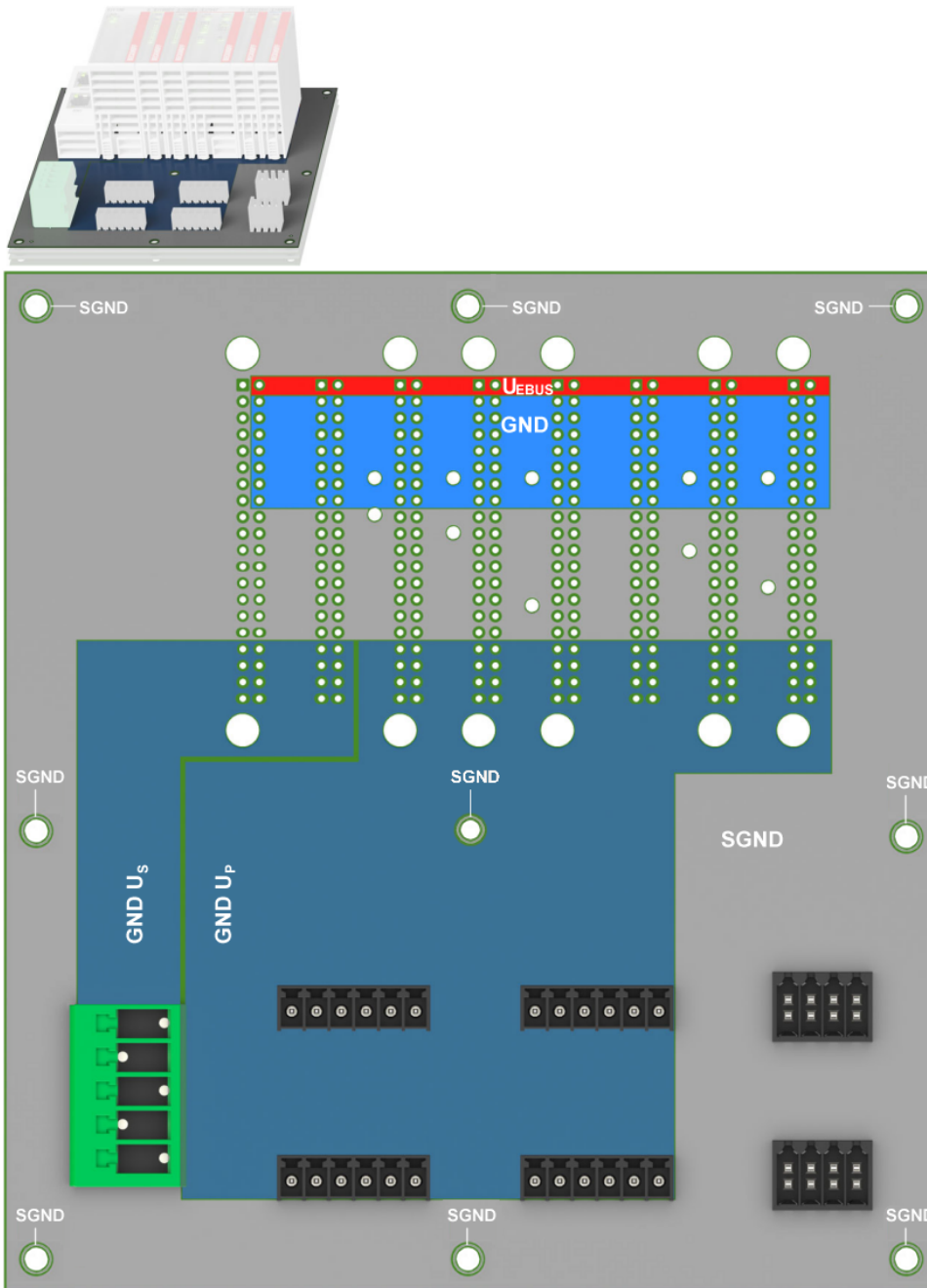


Fig. 14: Example for routing the top layer of a four-layer PCB

- Keep 0 V U_S power supply as close as possible to the coupler in order to avoid unnecessary antennas.
- 0 V U_S/U_P and 24 V U_S/U_P should be routed at different layers.
- The SGND shield ground pins may be connected and routed on the top layer.
- SGND connection to the mounting plate shall be implemented as metal bolts building a direct connection between signal distribution board and mounting plate (see chapter [SGND Connection](#) [19]). The copper rings around the holes are connected to SGND.
- It is recommended to route the signals SGND, 0 V U_S/U_P and 24 V U_S/U_P as an area.

i Layout for U_P power supply

In the figure above, the areas for the U_P supply are shown as an example. The actual layout must be adapted to the respective application.

9.2 Inner layer 1

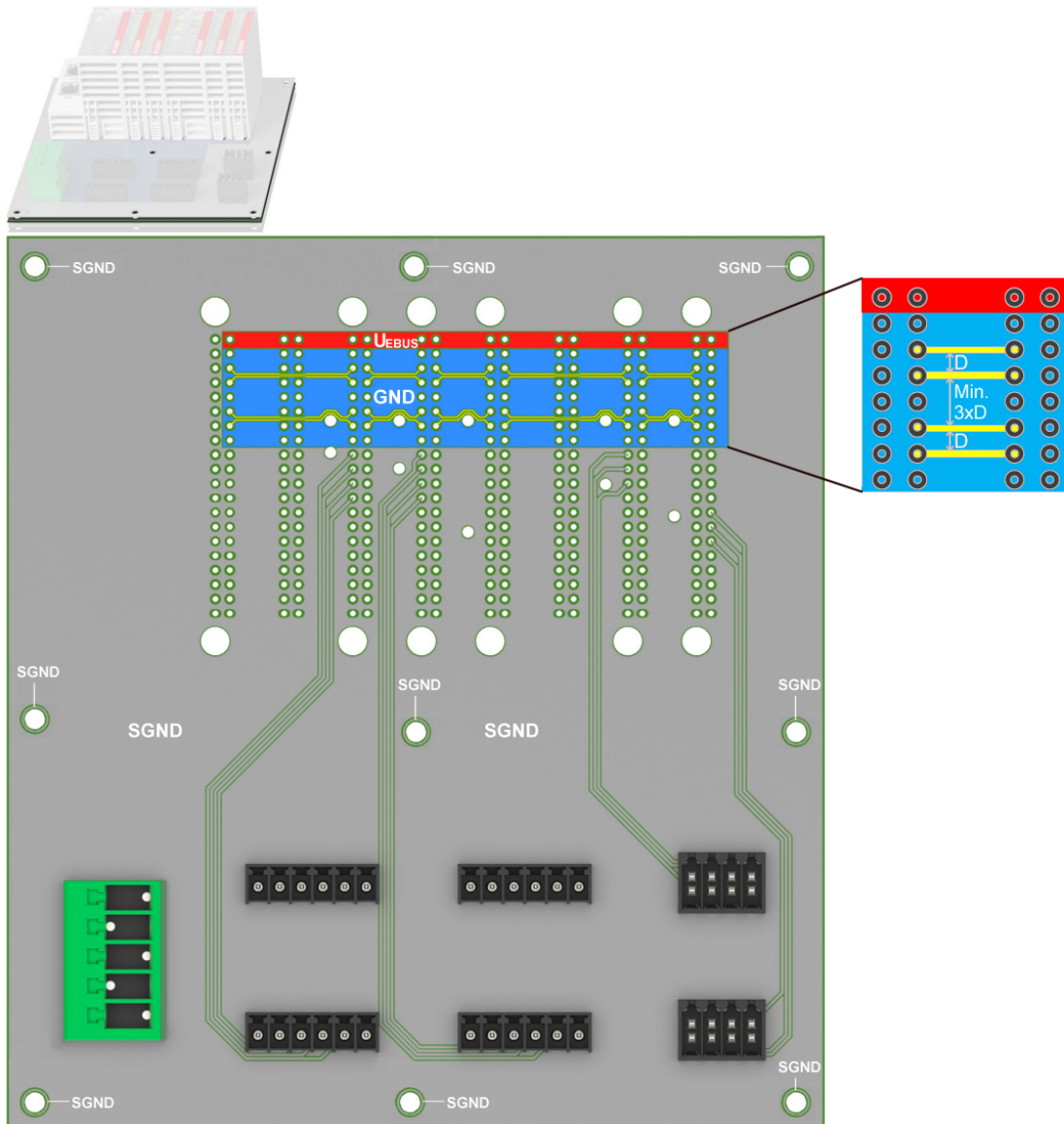


Fig. 15: Example for routing the inner layer1 of a four-layer PCB

- The E-bus traces have to be routed in inner layers, in order to allow complete covering of the differential pairs with copper (GND net) from both sides of the PCB.
- On the E-bus TX and RX routing layer free space between the signals shall be filled with copper connected to GND.
- Impedance and Routing
 - The differential impedance of the LVDS traces shall be 100Ω.
 - Width and spacing of the differential signal are depending on the concrete layer stack up and have to be calculated individually.
 - The differential signals should be routed as edge coupled traces.
 - The distance between the differential pairs should be three times larger than their inner distance (see Figure above).
 - Differential pairs should be routed without Vias (vertical interconnect access), in order to avoid impedance jumps.
 - Maximum values for uncoupled trace and overall trace length can be found in the specification for LVDS signals ANSI/TIA/EIA-644 "Electrical Characteristics of Low Voltage Differential Signaling (LVDS)".
- It is recommended to route SGND as an area.

Physical Communication Layer

The EtherCAT plug-in modules use the E-bus for backplane communication.

The E-bus physical layer uses **Low Voltage Differential Signaling** (LVDS) according to the ANSI/TIA/EIA-644 „Electrical Characteristics of Low Voltage Differential Signaling (LVDS) Interface Circuits” standard.

The E-bus has a data rate of 100 Mbit/s to accomplish the Fast Ethernet data rate.

9.3 Inner layer 2

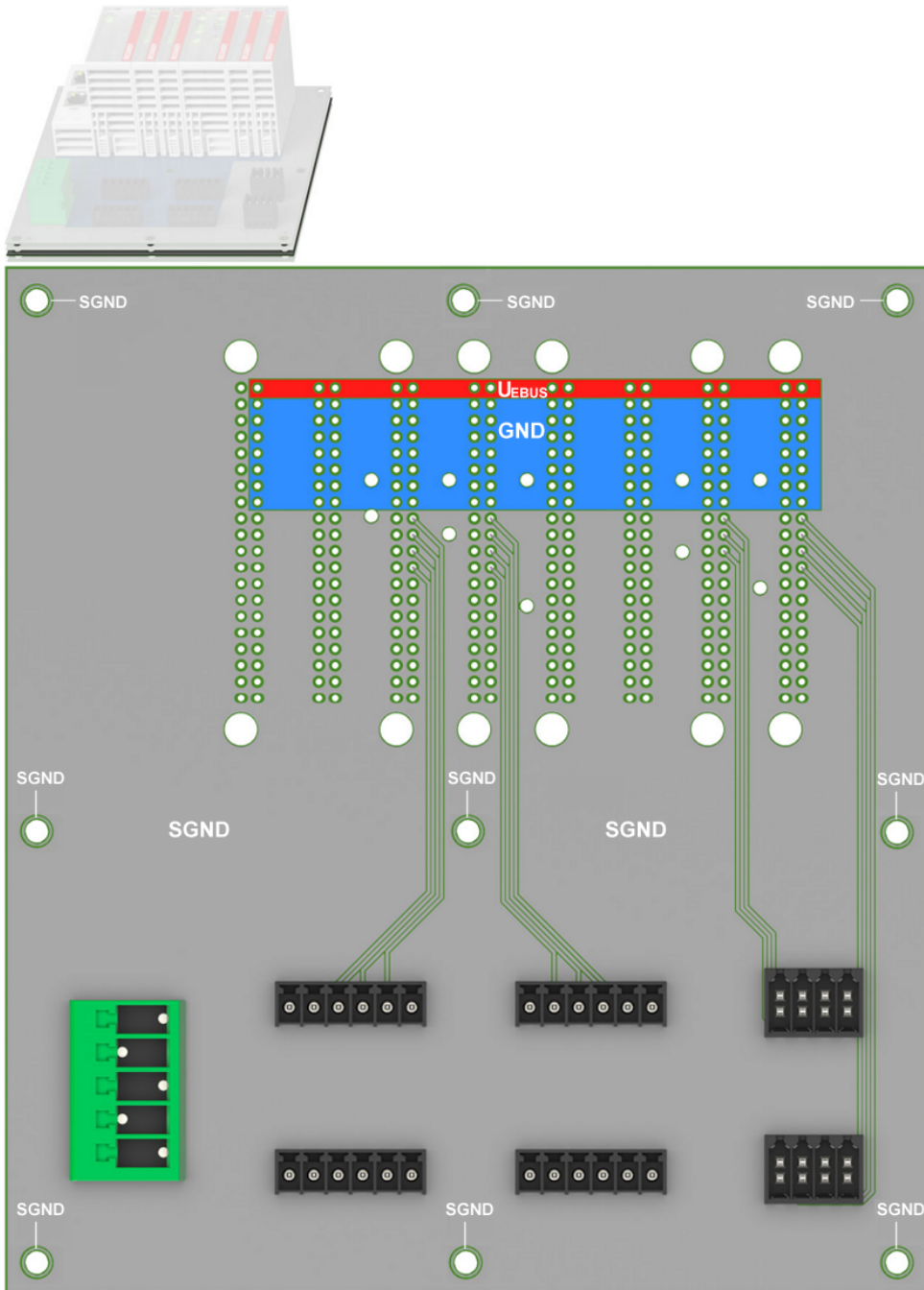


Fig. 16: Example for routing the inner layer 2 of a four-layer PCB

- I/O signals should be routed in the inner layers, as covering of signal lines from both sides with SGND can improve insensibility against EMC disturbances.
- Additionally the space between signal lines and signal groups should be filled with copper on SGND potential.
- It is recommended to route SGND as an area.

9.4 Bottom layer

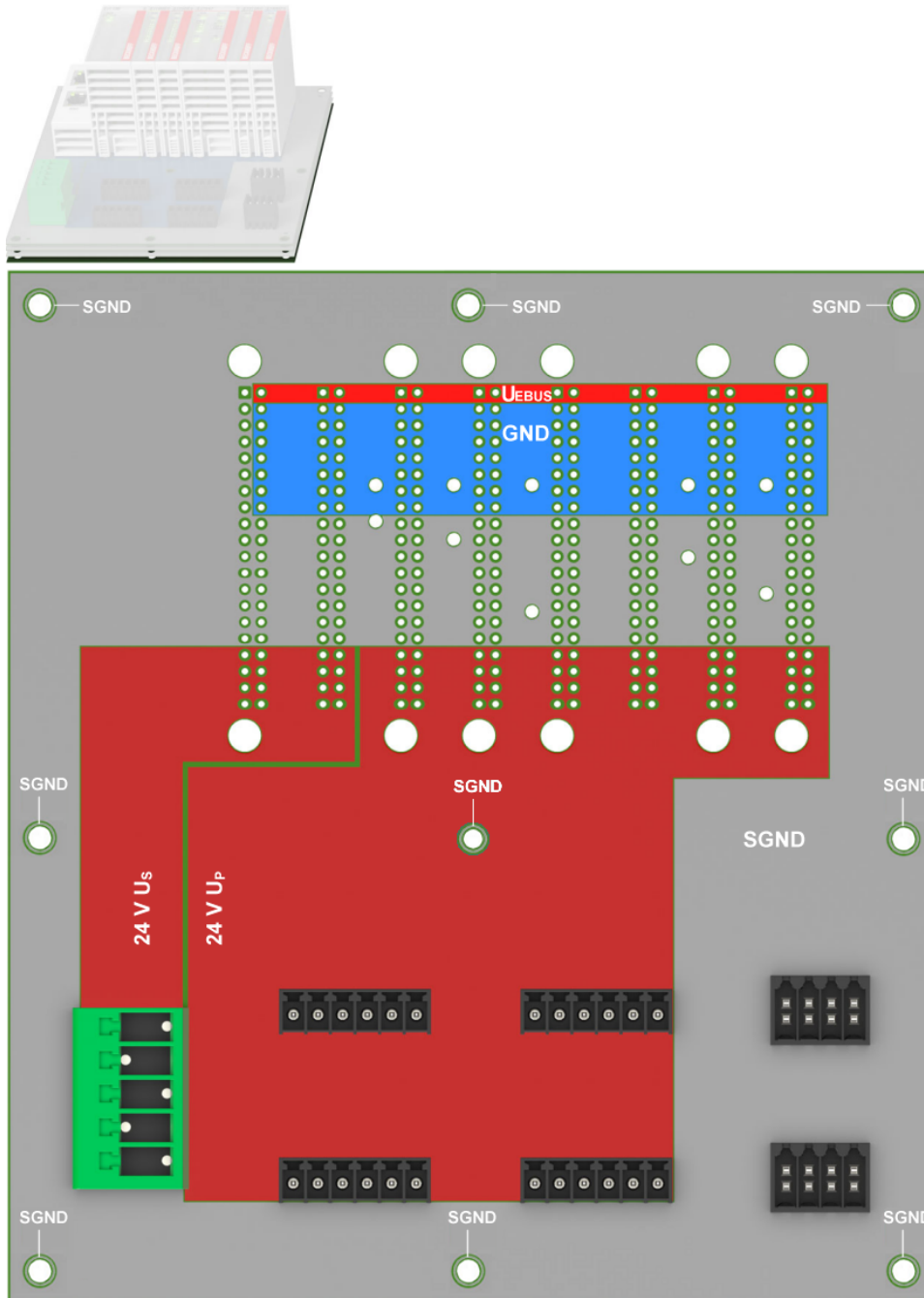


Fig. 17: Example for routing the bottom layer of a four-layer PCB

- Keep 24 V Us power supply as close as possible to the EJ1100 coupler in order to avoid unnecessary antennas.
- 0 V Us/Up and 24 V Us/Up should be routed at different layers.
- 24 V Us should be galvanically separated from 24 V Up.
- It is recommended to route the signals SGND, 0 V Us/Up and 24 V Us/Up as an area.

● Layout for Up power supply

1

In the figure above, the areas for the Up supply are shown as an example. The actual layout must be adapted to the respective application.

10 Routing guidelines

- Ground and U_{EBUS} power supply can be routed to partial surfaces within a plane.
- The differential E-bus signals have to be routed on internal layers.
- On the E-bus TX and RX routing layer free space between the signals shall be filled with copper connected to GND.

12-mm modules E-bus routing

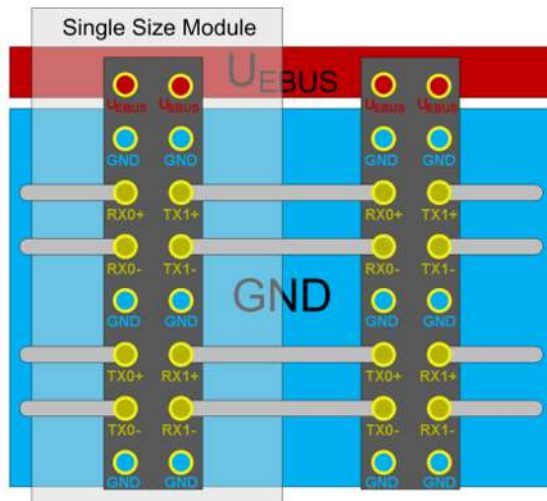


Fig. 18: Routing for 12-mm-modules

24-mm modules E-bus routing (e.g. EJ7342)

24-mm modules, where the E-bus has to be connected to the left or right connector either (e.g. EJ7342), shall be routed in a way shown in the following Figure.

In the area of the module connectors, the trace width and spacing may be reduced, if necessary (see following fig. (A)).

If available in the design software the option *Unused Pad Suppression* may be helpful to generate more routing space between the connector pins.

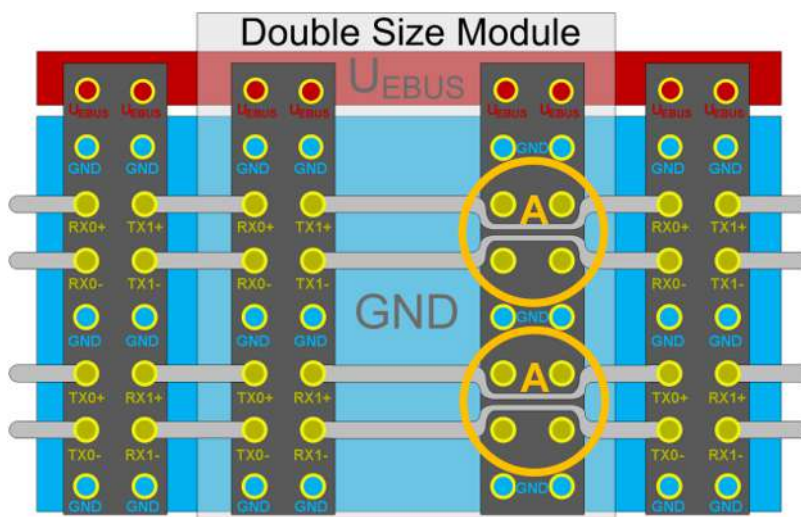


Fig. 19: Routing for 24-mm modules

Signal ground routing

The shield ground pins may be connected and routed on the top layer of the PCB (see fig. *Top layer*). Ensure proper connection of the SGND signal [► 19] with the mounting plate and the control cabinet!

10.1 EMC guidelines

EMC stability can be improved by the following points:

- Covering of signal lines from both sides with SGND can improve insensitivity against EMC disturbances. Additionally the space between signal lines and signal groups should be filled with copper on SGND potential.
- Keep Us power supply as close as possible to the EJ1100 coupler in order to avoid unnecessary antennas.
- SGND connection to the mounting plate shall be implemented as metal bolts building a direct connection between signal distribution board and mounting plate (see chapter [SGND connection](#) [► 19]).

10.2 Impedance and Routing

The following points should be taken in to account during the PCB design phase:

- The E-bus traces have to be routed in inner layers.
- The differential impedance of the LVDS traces shall be 100 Ω .
- Width and spacing of the differential signal are depending on the concrete layer stack up and have to be calculated individually.
- The differential signals should be routed as edge coupled traces.
- The distance between the differential pairs should be three times larger than their inner distance (see following Figure (D)).
- No GND area may form between the differential lines (D). This usually results from the required impedance.
- Differential pairs should be routed without Vias (vertical interconnect access), in order to avoid impedance jumps.
- Maximum values for uncoupled trace and overall trace length can be found in the specification for LVDS signals ANSI/TIA/EIA-644 „Electrical Characteristics of Low Voltage Differential Signaling (LVDS)

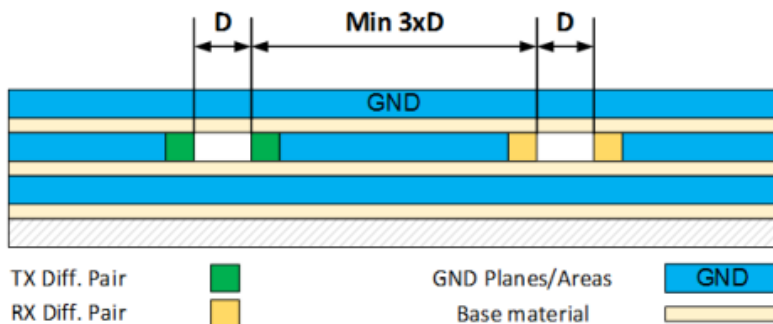


Fig. 20: Differential pair spacing

NOTICE

Avoid shor circuits

Pay attention to short circuits when configuring the cross-section!

11 Appendix

11.1 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.

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 her internet pages: www.beckhoff.com

You will also find further documentation for Beckhoff components there.

Support

The Beckhoff 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
web: www.beckhoff.com/support

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
web: www.beckhoff.com/service

Headquarters Germany

Beckhoff Automation GmbH & Co. KG

Hülshorstweg 20
33415 Verl
Germany

Phone: +49 5246 963 0
e-mail: info@beckhoff.com
web: www.beckhoff.com

Trademark statements

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.

More Information:
www.beckhoff.com/EJ8xxx

Beckhoff Automation GmbH & Co. KG
Hülshorstweg 20
33415 Verl
Germany
Phone: +49 5246 9630
info@beckhoff.com
www.beckhoff.com

