Design Guide

EJ8xxx

Signal-Distribution-Board for standard EtherCAT plug-in modules

Version: 6.1
Date: 2019-11-22

BECKHOFF

Table of contents

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.
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®, 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 covered, including but not limited to the following patent 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

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.

Description of instructions

In this documentation the following instructions are used.
These instructions must be read carefully and followed without fail!

⚠ DANGER

Serious risk of injury!

Failure to follow this safety instruction directly endangers the life and health of persons.

⚠ WARNING

Risk of injury!

Failure to follow this safety instruction endangers the life and health of persons.

⚠ CAUTION

Personal injuries!

Failure to follow this safety instruction can lead to injuries to persons.

NOTE

Damage to environment/equipment or data loss

Failure to follow this instruction can lead to environmental damage, equipment damage or data loss.

Tip or pointer

This symbol indicates information that contributes to better understanding.

1.3 Documentation issue status

Version	Comment
6.1	• Update chapter *Structure of the PCB layers*
6.0	• Migration • Update structure

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.

2 Technical data - Signal distribution board

Technical data	Signal distribution board
PCB layer stack	Multilayer PCB, min. 4 layers
PCB thickness	1.6 mm ± 10 %
Board connector	Samtec: SSQ-120-01-L-D
Placing of modules [▶ 11]	Starting from the left: Coupler, power supply module, EJ-modules
E-bus power supply [▶ 19]	EJ1100 (2.2 A) EJ1101-0022 + (EJ9400 (2.5 A) or EJ9404 (12 A)) CX + EK1110-0043
Refresh of E-bus power supply [▶ 20]	EJ9400 (2.5 A) EJ9404 (12 A)
Air gap and leakage distances [▶ 12] between E-bus- and field signals	typ. 1.2 mm
Differential impedance [▶ 23] of the LVDS traces	100 Ω
SGND connection	via mounting bolts
Mounting hole distances	max. 100 mm
Reaching area [▶ 9]	92 mm
Module depth above PCB	min. 55 mm
Distance between PCB and mounting surface	min. 4 mm
Mounting position	Standard

NOTE

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 backplane pcb should be mounted in the switch cabinet as described in the following figure.

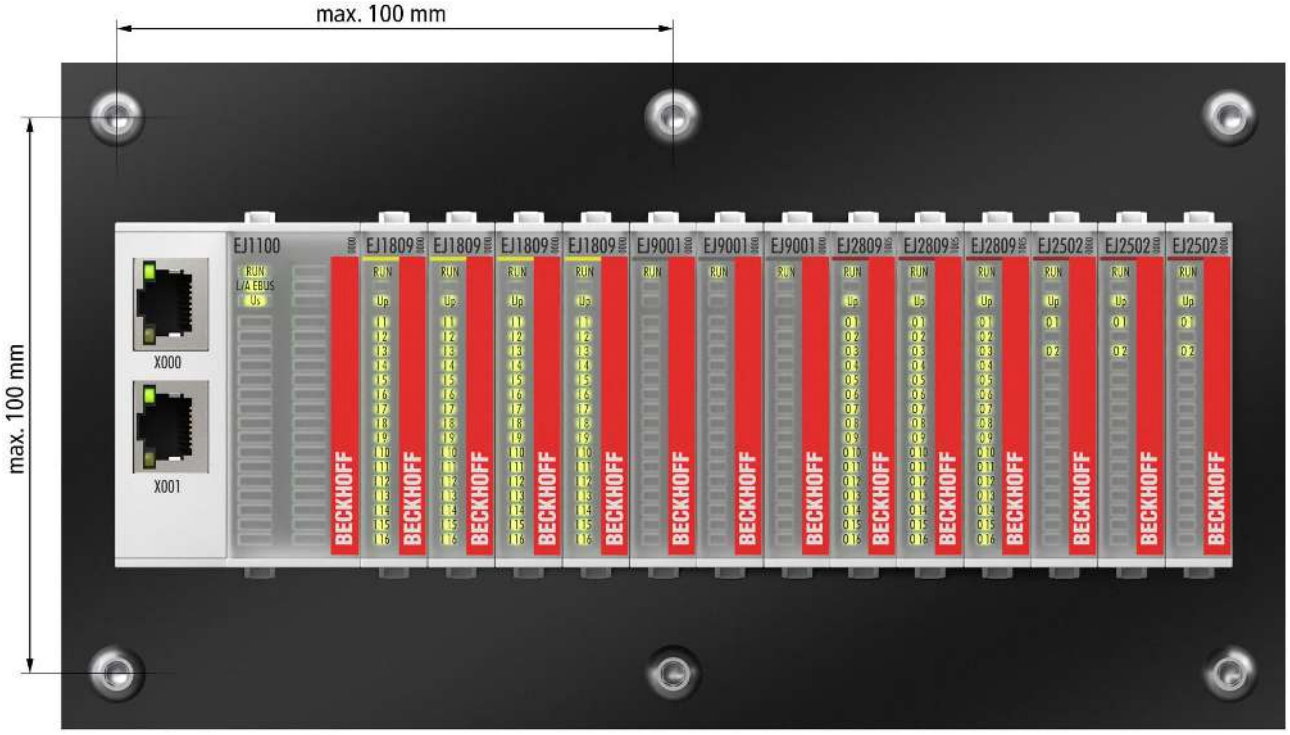


Fig. 1: Maximum distances between mounting holes and PCB

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.

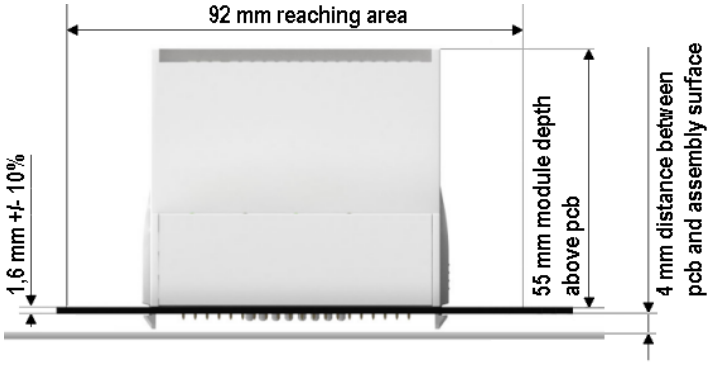


Fig. 2: Mounting distances EJ module - PCB

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 [▶ 10]) 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

NOTE

Constraints regarding installation position and operating temperature range

Please refer to the technical data 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.

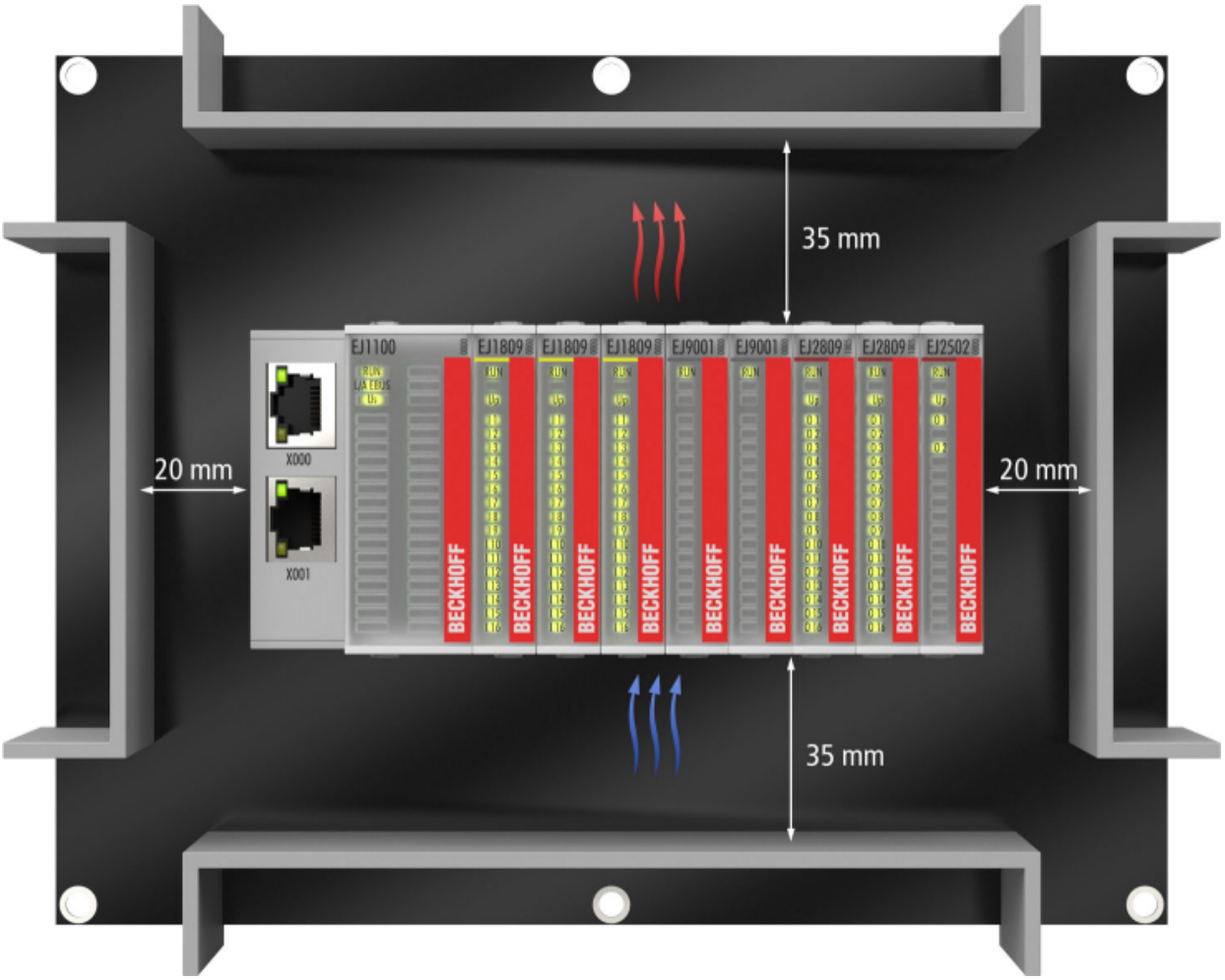


Fig. 3: Recommended distances for standard installation position

Compliance with the distances shown in Fig. *"Recommended distances for standard installation position"* is recommend. 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 connectors) 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

Follow the instructions for routing in chapter Design of power supply [▶ 20] and Routing guidelines [▶ 22]!

Example with coupler EJ1100

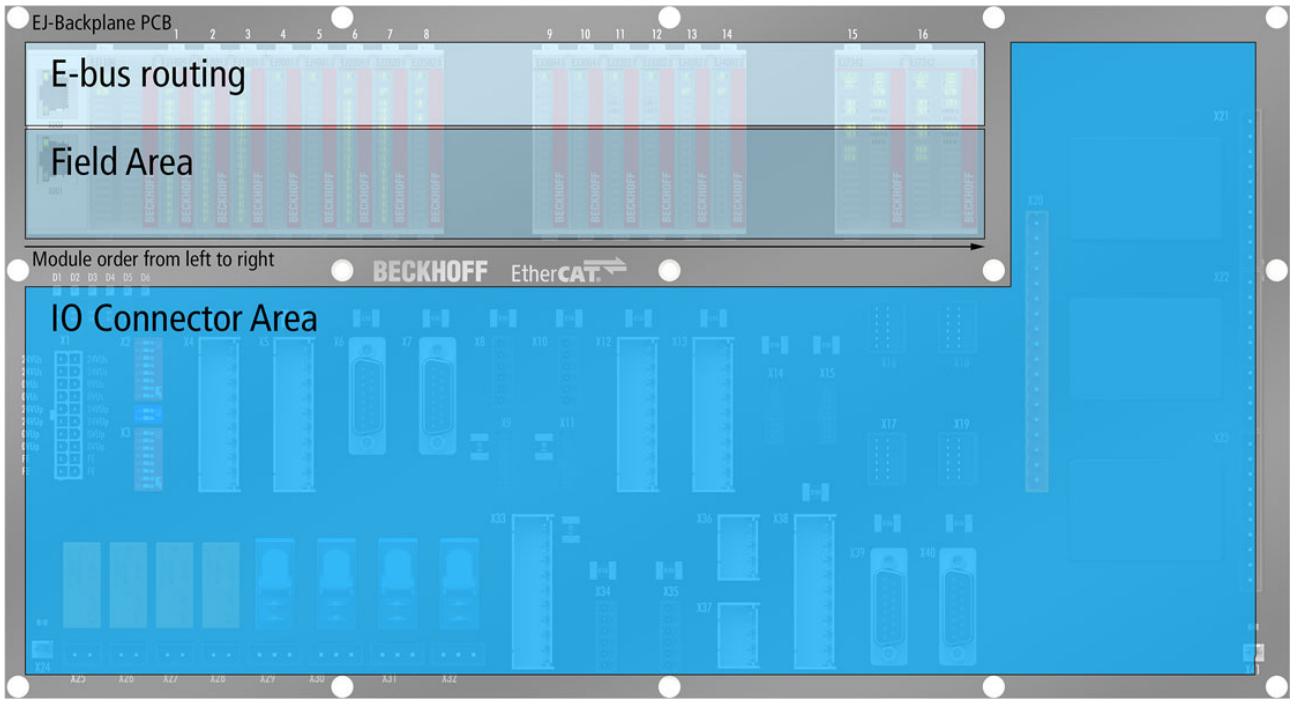


Fig. 4: EJ module line starting with coupler EJ1100

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.

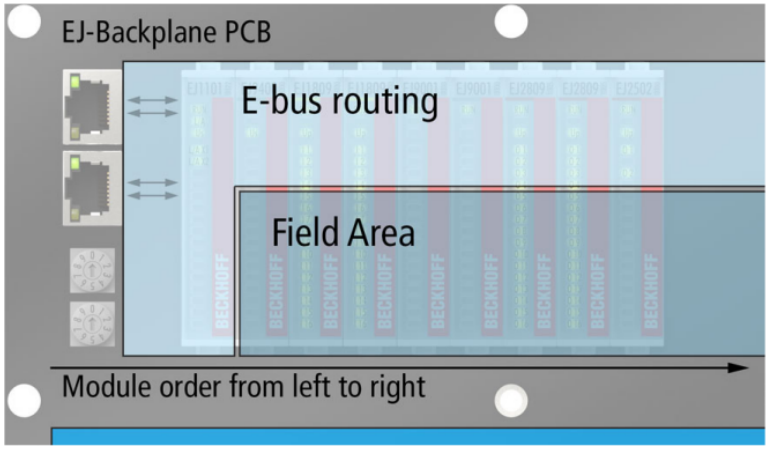


Fig. 5: EJ module line starting with coupler EJ1101-0022 (optional RJ45 sockets)

Clearances and creepage distances

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

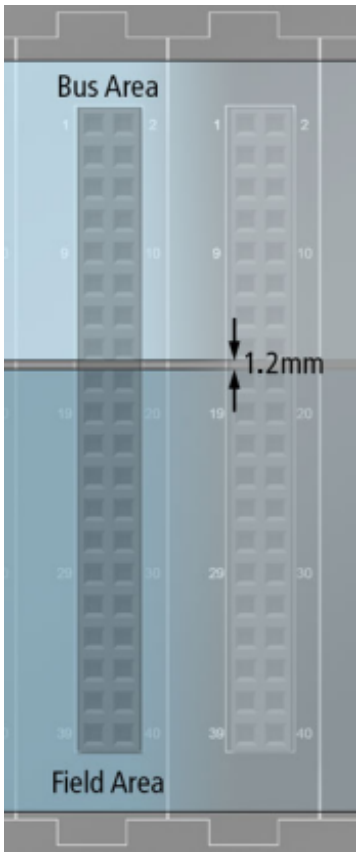


Fig. 6: Clearance between bus- and field area

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.

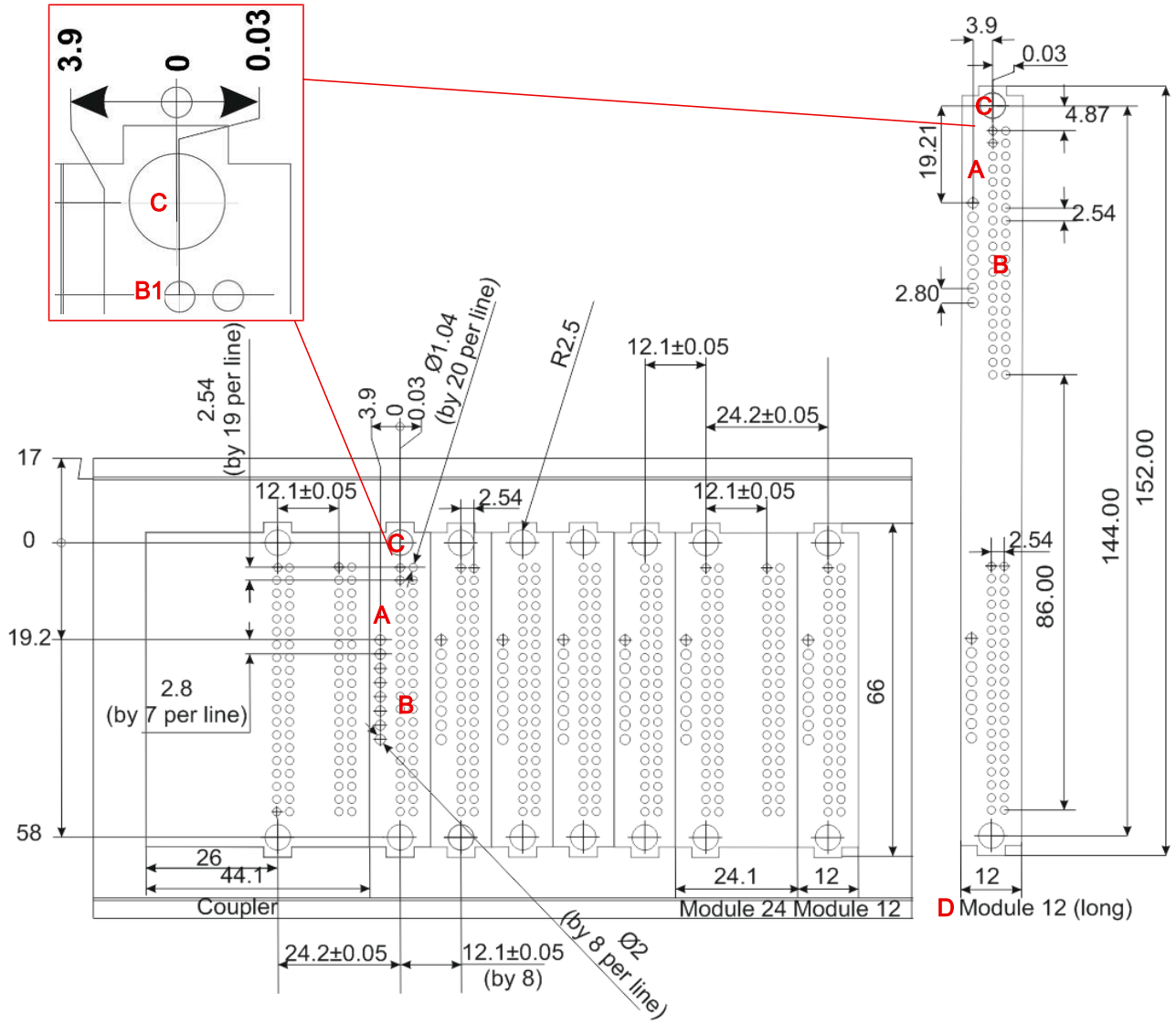
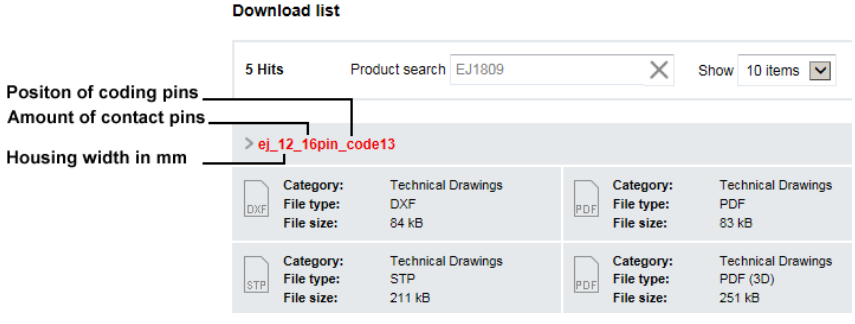


Fig. 7: Backplane layout, dimensions in mm

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



6 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.

NOTE

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 mm ±10%.

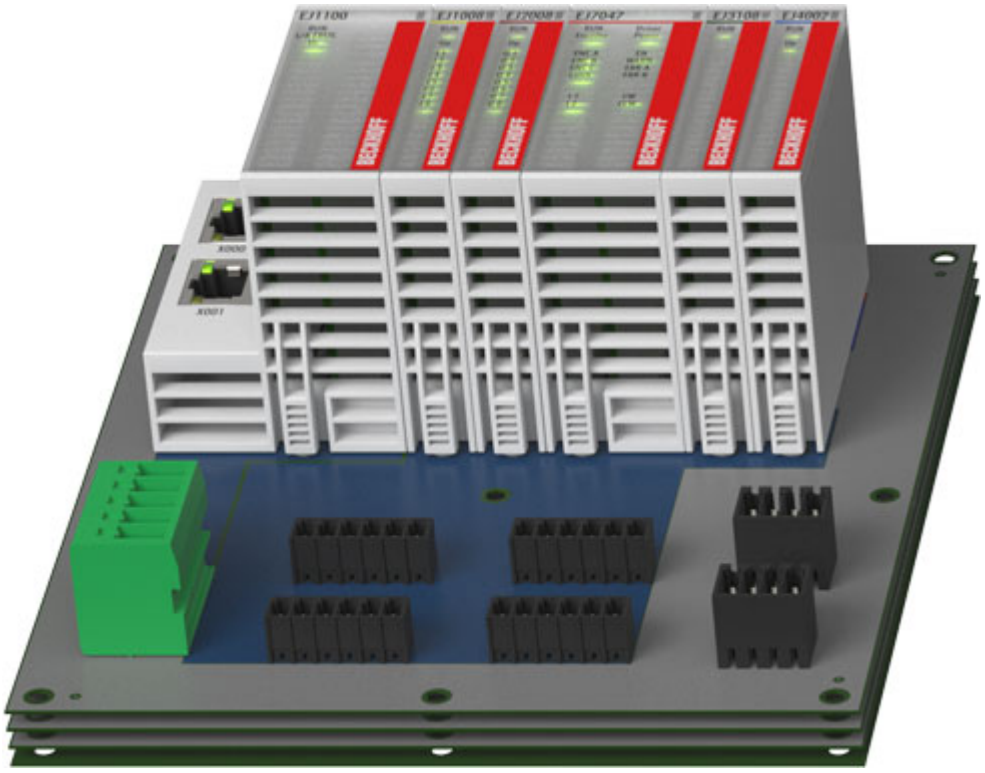


Fig. 8: Requirements of the pcb (min. 4 layers, max. 1.6 mm thickness)

The following figures show an example for a PCB with 4 layers with the routing in the individual layers.

NOTE

Note on routing

- Read the notes on routing in chapter Module placement [▶ 11], Design of power supply [▶ 19] and Routing guildelines [▶ 22]!
- If necessary, read the routing instructions in chapter *pinout* in the documentations of the modules used.

Top layer

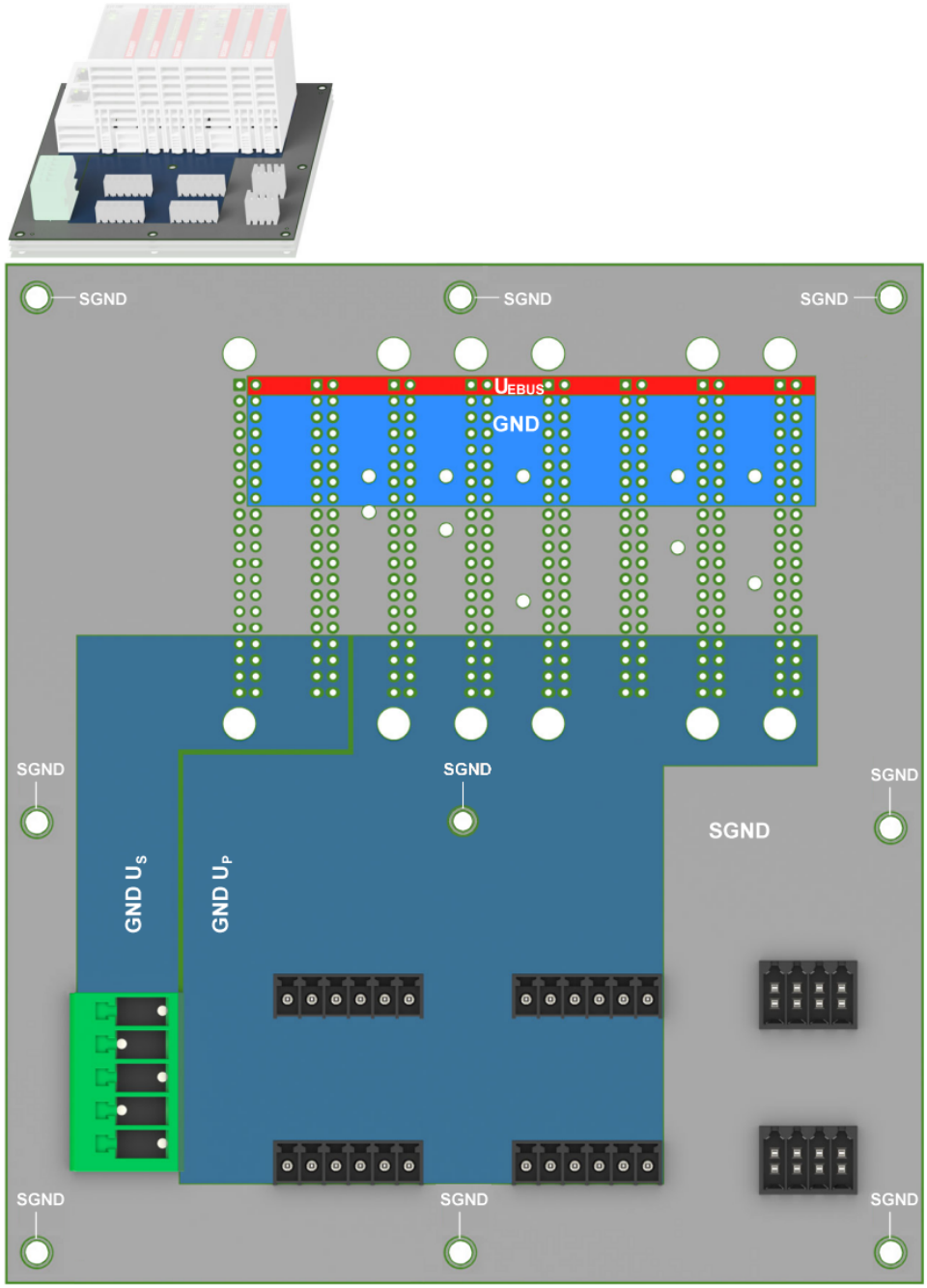


Fig. 9: Top layer

- Keep 0 V Us power supply as close as possible to the coupler in order to avoid unnecessary antennas.
- 0 V Us/Up and 24 V Us/Up should be routed at different layers.
- The SGND shield ground pins may be connected and routed on the top layer.
- SGND connection to the control cabinet shall be implemented as metal bolts building a direct connection between back plane and control cabinet. The copper rings around the holes are connected to SGND.
  A cable based SGND connection to the control cabinet shall be avoided.
- It is recommended to route the signals SGND, 0 V Us/Up and 24 V Us/Up as an area.

Inner layer 1

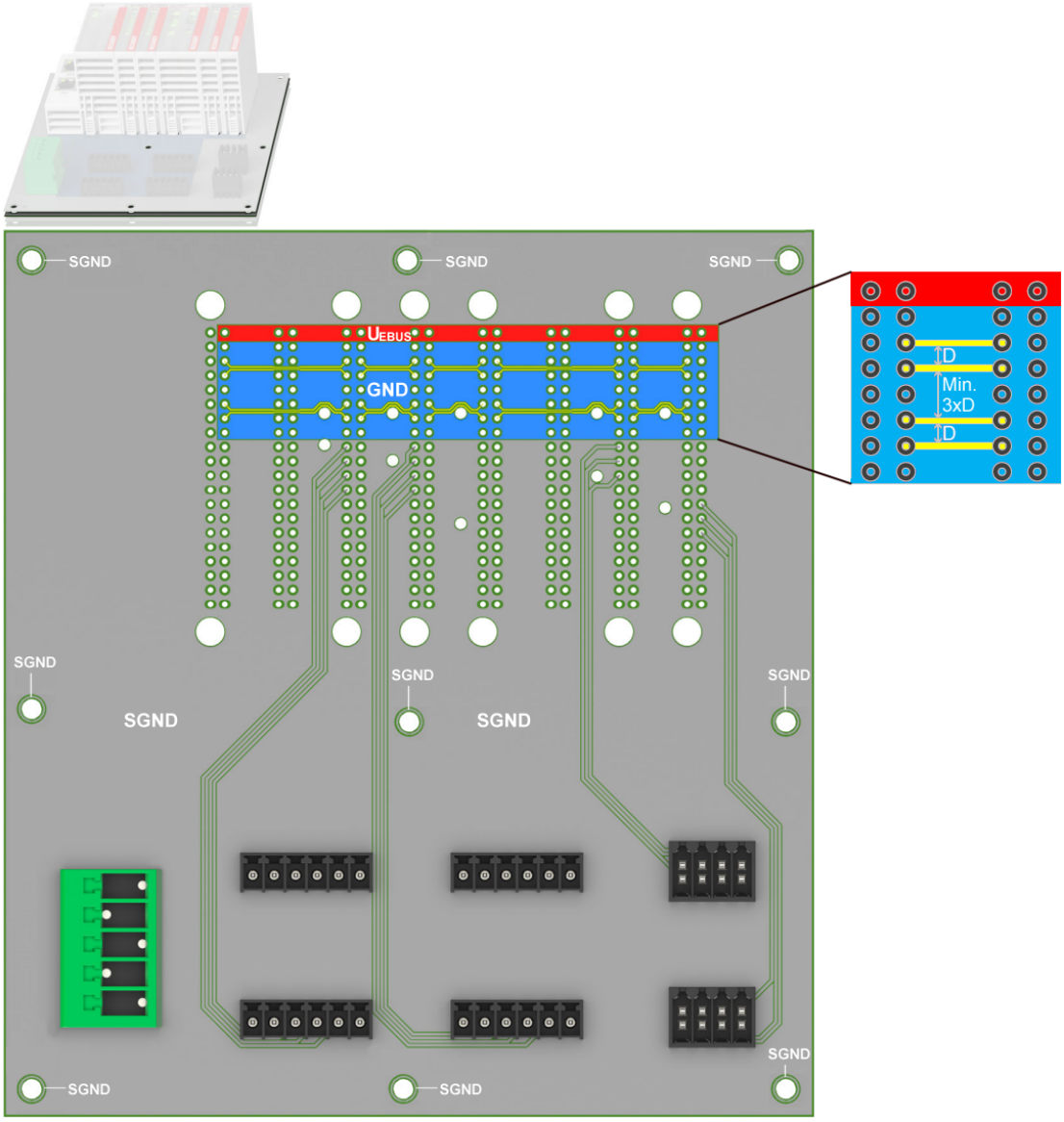


Fig. 10: Inner layer1

- 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 (**v**ertical **i**nterconnect **a**ccess), 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 **L**ow **V**oltage **D**ifferential **S**ignaling (LVDS)".
- It is recommended to route SGND as an area.

Inner layer 2

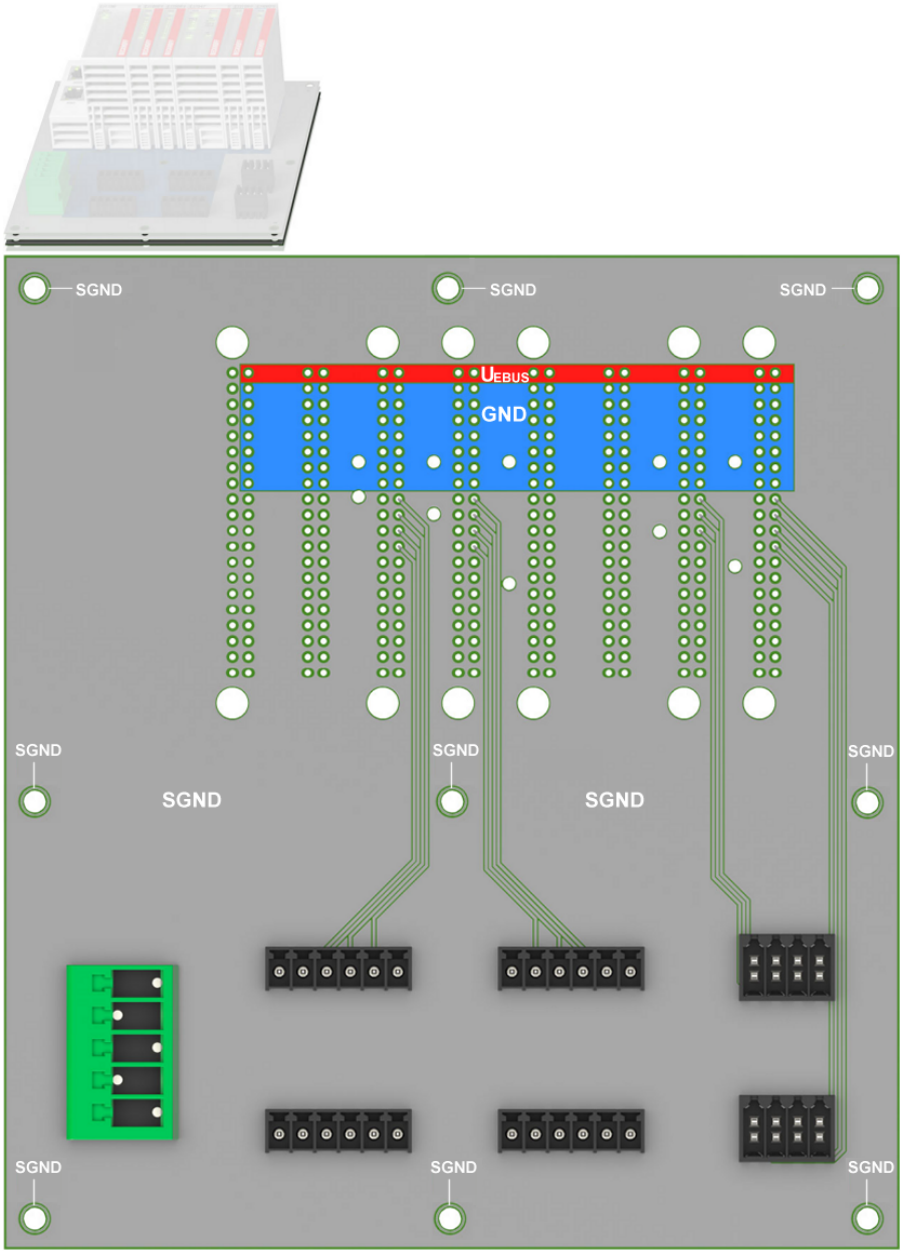


Fig. 11: Inner layer 2

- 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.

Bottom layer

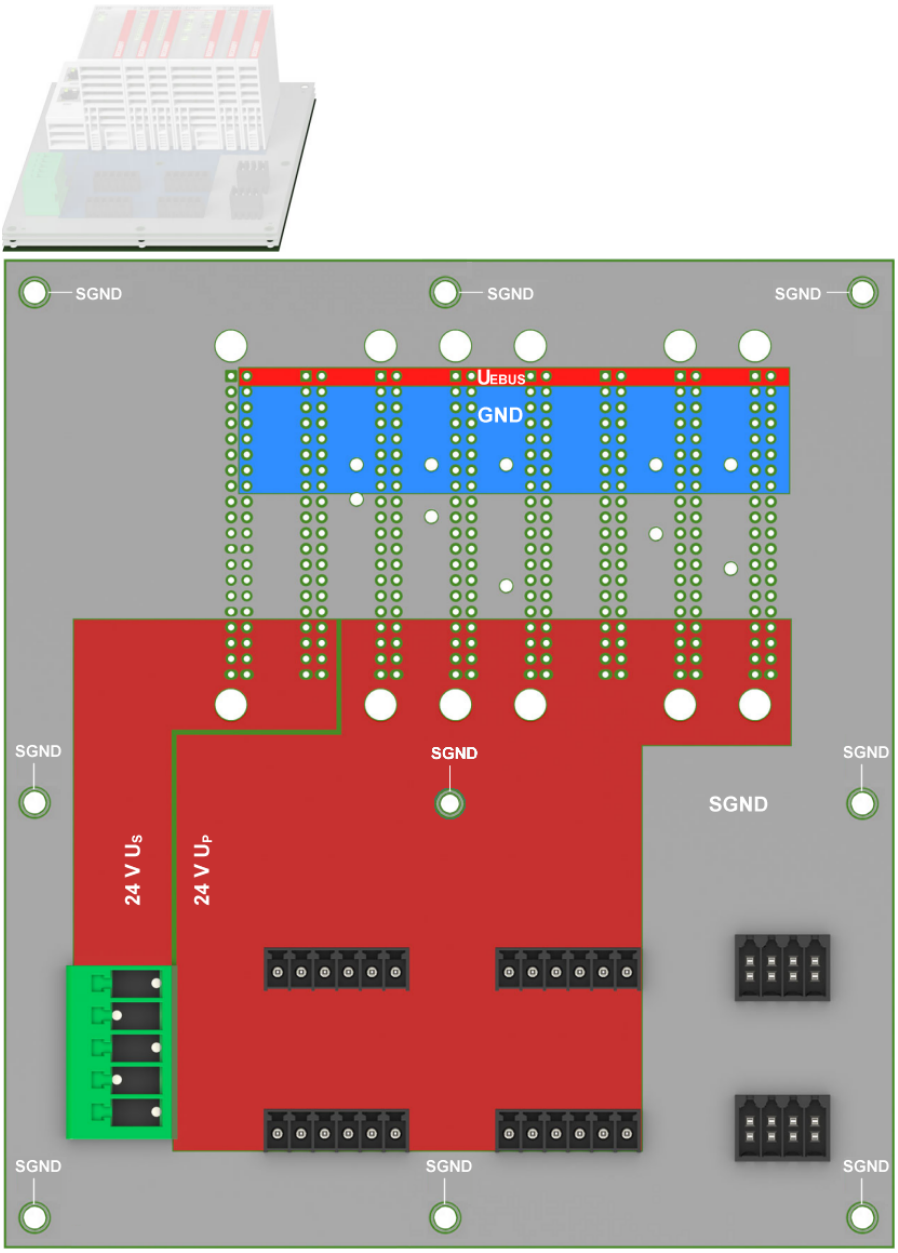


Fig. 12: Bottom Layer

- 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.

7 Design of power supply

⚠ WARNING

Power supply

A SELV/PELV power supply must be used to supply power for the EJ coupler and modules. Couplers and modules have to be connected to SELV/PELV circuits exclusively.

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 (see Current consumption of the EJ modules from the E-bus).

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:

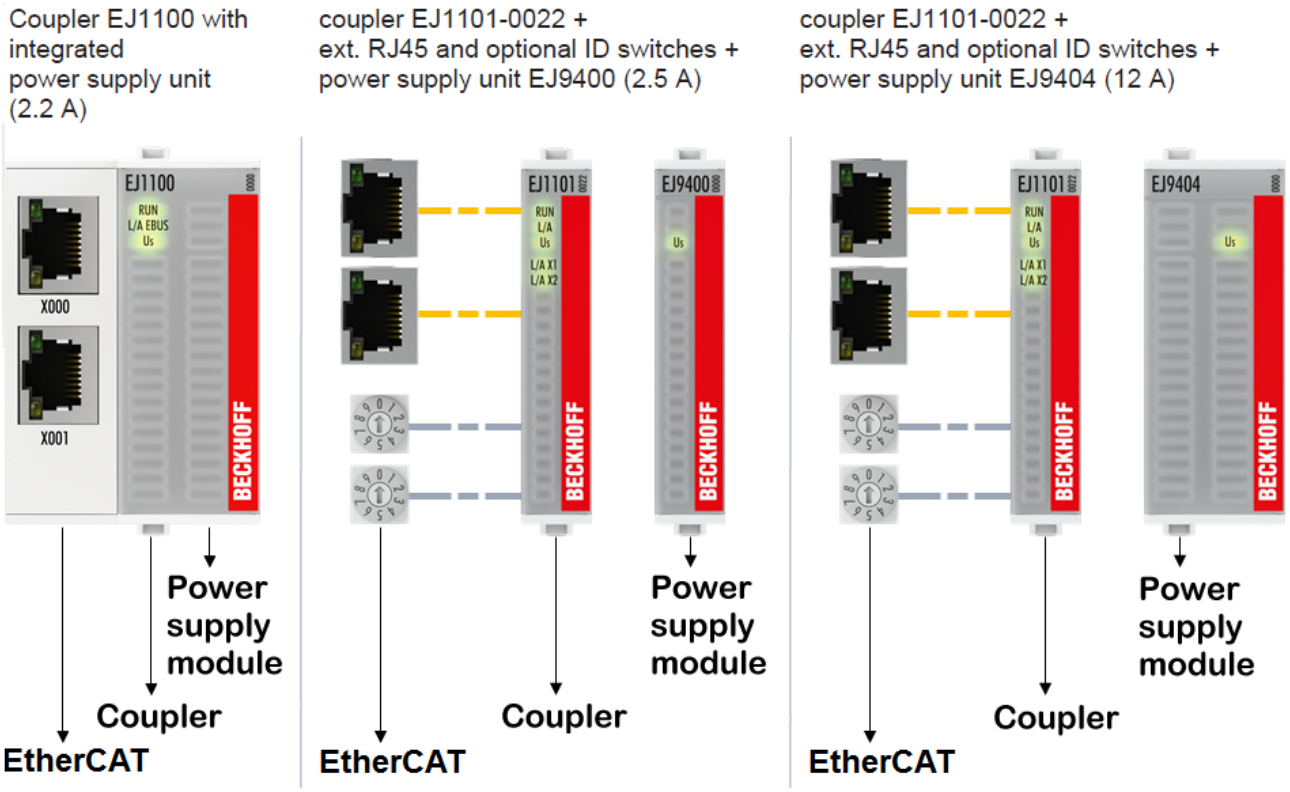


Fig. 13: E-bus power supply with EJ1100 or EJ1101-0022 + EJ940x

In the EJ1101-0022 coupler, the RJ45 connectors 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.

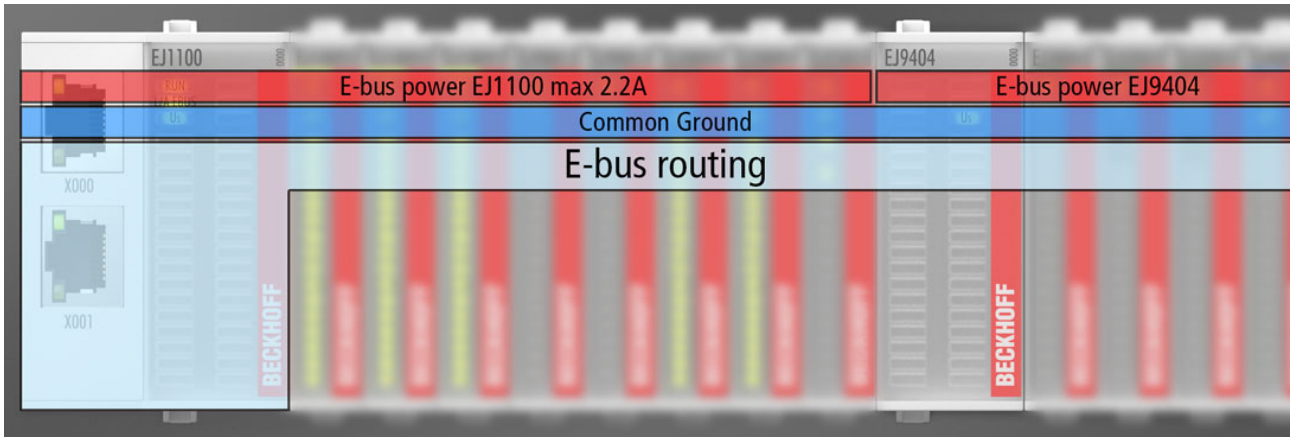


Fig. 14: Example with Coupler EJ1100, with integrated power supply (2.2 A), additional power supply with EJ9404 (12 A)

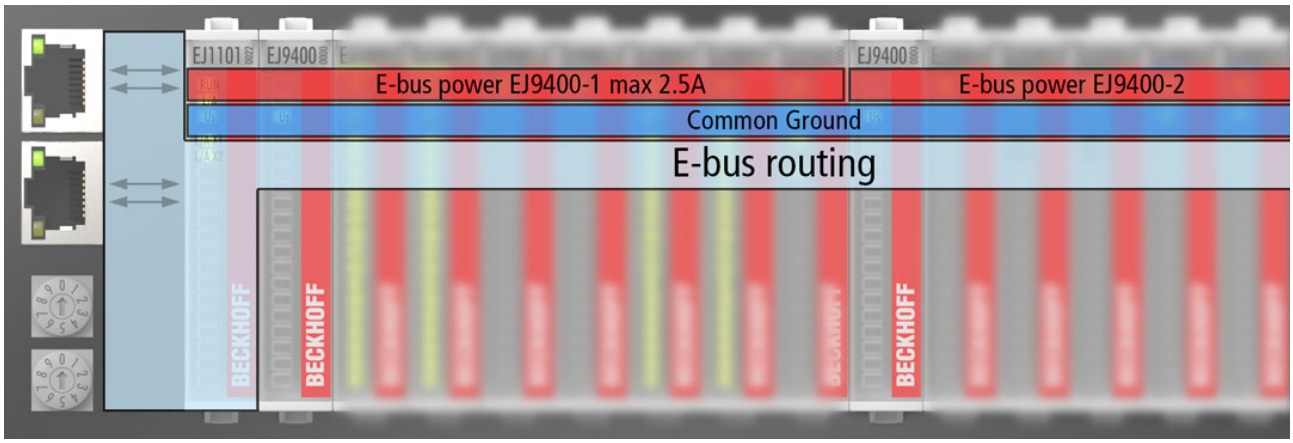


Fig. 15: Example with Coupler EJ1101-0022, power supply module EJ9400 (2.5 A), additional power supply with EJ9400 (2.5 A)

8 Physical Communication Layer

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

The E-bus physical layer uses **L**ow **V**oltage **D**ifferential **S**ignaling (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 Routing guildelines

- Ground and U_{EBUS} power supply shall be routed as planes on separate layers.
- 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

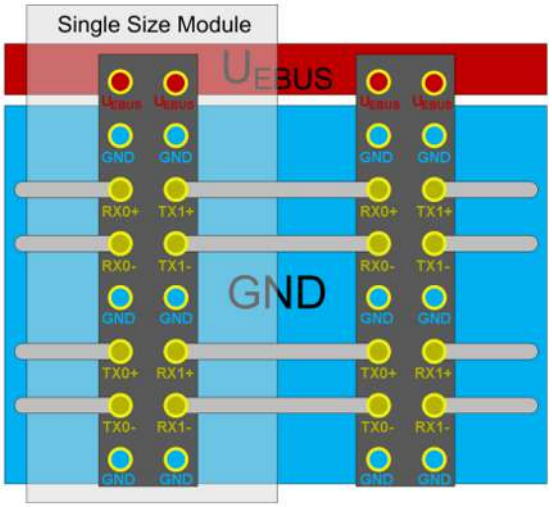


Fig. 16: Routing for 12-mm-modules

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.

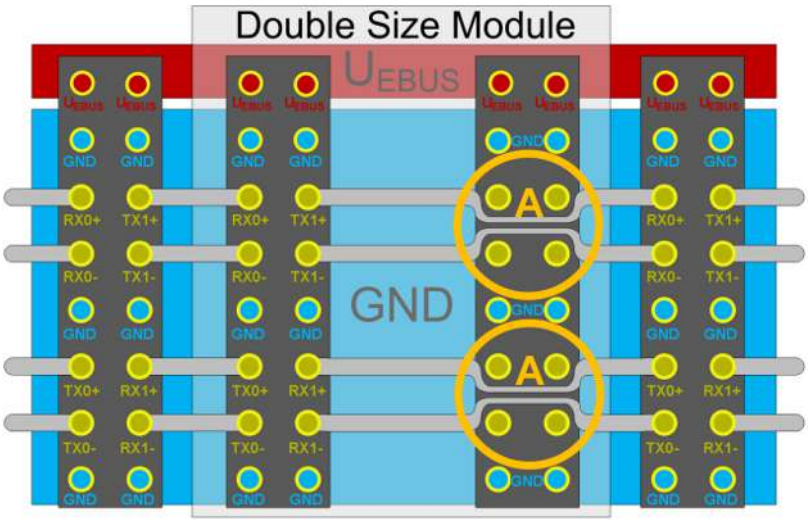


Fig. 17: Routing for 24-mm modules

In the area of the module connectors, the trace width and spacing may be reduced, if necessary (see figure above (A)).
If available in the design software the option *Unused Pad Suppression* may be helpful to generate more routing space between the connector pins.

Signal ground routing

The signal ground pins may be connected and routed on the top layer of the PCB as shown in figure Top layer [▶ 15] .

Ensure proper connection of the SGND signal with the control cabinet!

9.1 EMC guidelines

EMC stability can be improved by the following points:

- 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.
- Keep Us power supply as close as possible to the EJ1100 coupler in order to avoid unnecessary antennas.
- SGND connection to the control cabinet shall be implemented as metal bolts building a direct connection between back plane and control cabinet. A cable based SGND connection to the control cabinet shall be avoided.

9.2 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)).
- Differential pairs should be routed without Vias (**v**ertical **i**nterconnect **a**ccess), 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 **L**ow **V**oltage **D**ifferential **S**ignaling (LVDS)

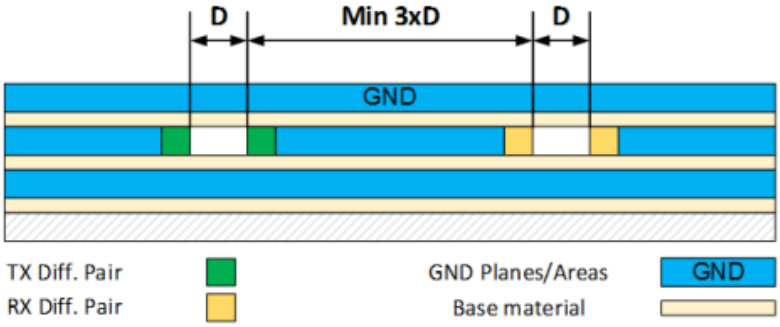


Fig. 18: Differential pair spacing

NOTE

Avoid shor circuits

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

10 Appendix

10.1 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:
http://www.beckhoff.com

You will also find further documentation for Beckhoff components there.

Beckhoff Headquarters

Beckhoff Automation GmbH & Co. KG

Huelshorstweg 20
33415 Verl
Germany

Phone:	+49 5246 963 0
Fax:	+49 5246 963 198
e-mail:	info@beckhoff.com

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
Fax:	+49 5246 963 9157
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
Fax:	+49 5246 963 479
e-mail:	service@beckhoff.com

List of illustrations