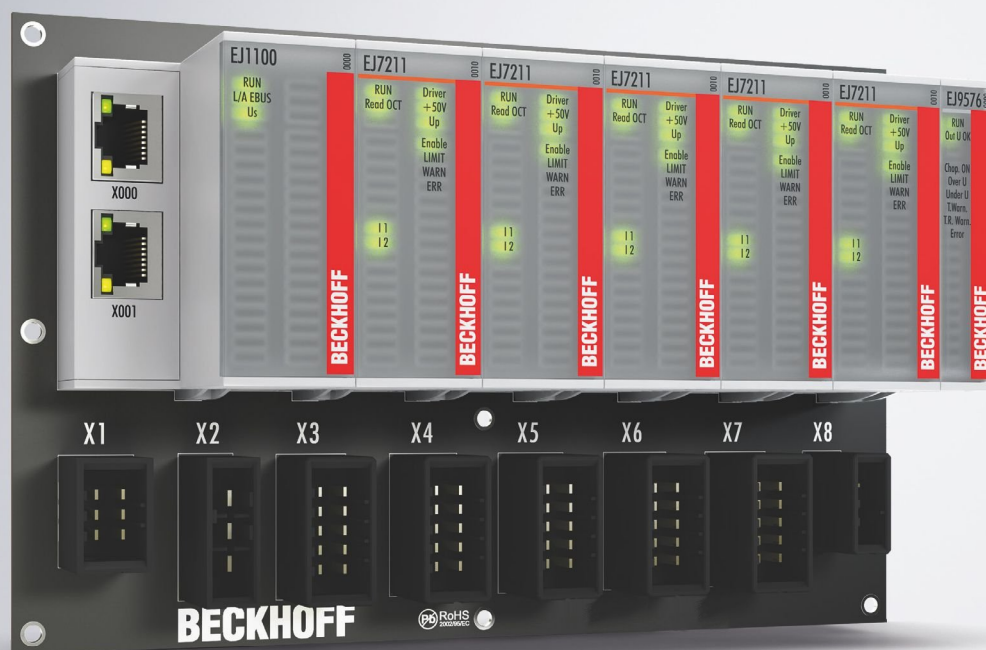


Design Guide | EN

## EJ8xxx

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





# Table of contents

<b>1 Foreword</b>	<b>5</b>
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
<b>2 Technical data - Signal distribution board</b>	<b>8</b>
<b>3 Backplane mounting guidelines</b>	<b>9</b>
3.1 Minimum distances for ensuring installability	9
3.2 Installation positions	10
<b>4 Module placement</b>	<b>11</b>
<b>5 PCB - distances and footprint</b>	<b>13</b>
<b>6 Design of E-bus power supply</b>	<b>14</b>
<b>7 Power supply and potential groups</b>	<b>16</b>
<b>8 SGND Connection</b>	<b>18</b>
<b>9 Structure of the PCB layers</b>	<b>19</b>
9.1 Top layer	20
9.2 Inner layer 1	21
9.3 Inner layer 2	23
9.4 Bottom layer	24
<b>10 Routing guidelines</b>	<b>25</b>
10.1 EMC guidelines	26
10.2 Impedance and Routing	26
<b>11 Appendix</b>	<b>27</b>
11.1 Support and Service	27



# 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®, 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 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

© 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.

## 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
6.4	<ul style="list-style-type: none"><li>• Update chapter <i>Technical data - Signal distribution board</i></li><li>• Chapter <i>SGND Connection</i> added</li><li>• Update chapter <i>Power supply and potential groups</i></li><li>• Update structure</li></ul>
6.3	<ul style="list-style-type: none"><li>• Update chapter <i>Power supply and potential groups</i></li></ul>
6.2	<ul style="list-style-type: none"><li>• New title page</li><li>• Update chapter <i>Technical data - Signal distribution board</i></li><li>• Chapter <i>Power supply and potential groups</i> added</li><li>• Update chapter <i>Structure of the PCB layers</i></li><li>• Update chapter <i>Routing guidelines</i></li><li>• Update structure</li></ul>
6.1	<ul style="list-style-type: none"><li>• Update chapter <i>Structure of the PCB layers</i></li></ul>
6.0	<ul style="list-style-type: none"><li>• Migration</li><li>• Update structure</li></ul>

## 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 $\pm$ 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 [► 14]	EJ1100 (2.2 A) EJ1101-0022 + (EJ9400 (2.5 A) or EJ9404 (12 A))
Refresh of E-bus power supply [► 15]	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 [► 26] of the LVDS traces	100 $\Omega$
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

Description of potential groups	
U <sub>EBUS</sub> : Power supply E-bus	<p>The power supply U<sub>EBUS</sub> is provided by the coupler and supplied from the supply voltage U<sub>s</sub> of the EtherCAT coupler.</p> <p>U<sub>EBUS</sub>: E-bus power supply 3.3 V</p> <p>GND: E-bus GND signal;</p> <p><b>Note</b> Don't connect with 0 V Up and 0 V Us!</p>
U <sub>s</sub> : Power supply bus side	<p>The power supply U<sub>s</sub> is used to supply the bus coupler electronics and to generate voltage for the E-bus (U<sub>EBUS</sub>).</p> <p>0 V U<sub>s</sub>: Bus side GND signal</p> <p><b>Note</b> Don't connect with GND (E-bus GND signal)!</p> <p>24 V U<sub>s</sub>: Bus side power supply 24 V</p>
U <sub>p</sub> : Power supply field side	<p>The peripheral voltage U<sub>p</sub> supplies the electronics on the field side.</p> <p>0 V U<sub>p</sub>: GND signal field side</p> <p><b>Note</b> Don't connect with GND (E-bus GND signal)!</p> <p>24 V U<sub>p</sub>: Power supply field side 24 V</p>
SGND: Shield Ground	<p>Ground signal with shielding function in relation to the rest of the board</p> <p><b>Note</b> U<sub>s</sub>, U<sub>p</sub> 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 [► 18])</p>

### 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 signal distribution board should be mounted in the control cabinet as described in the following figure.

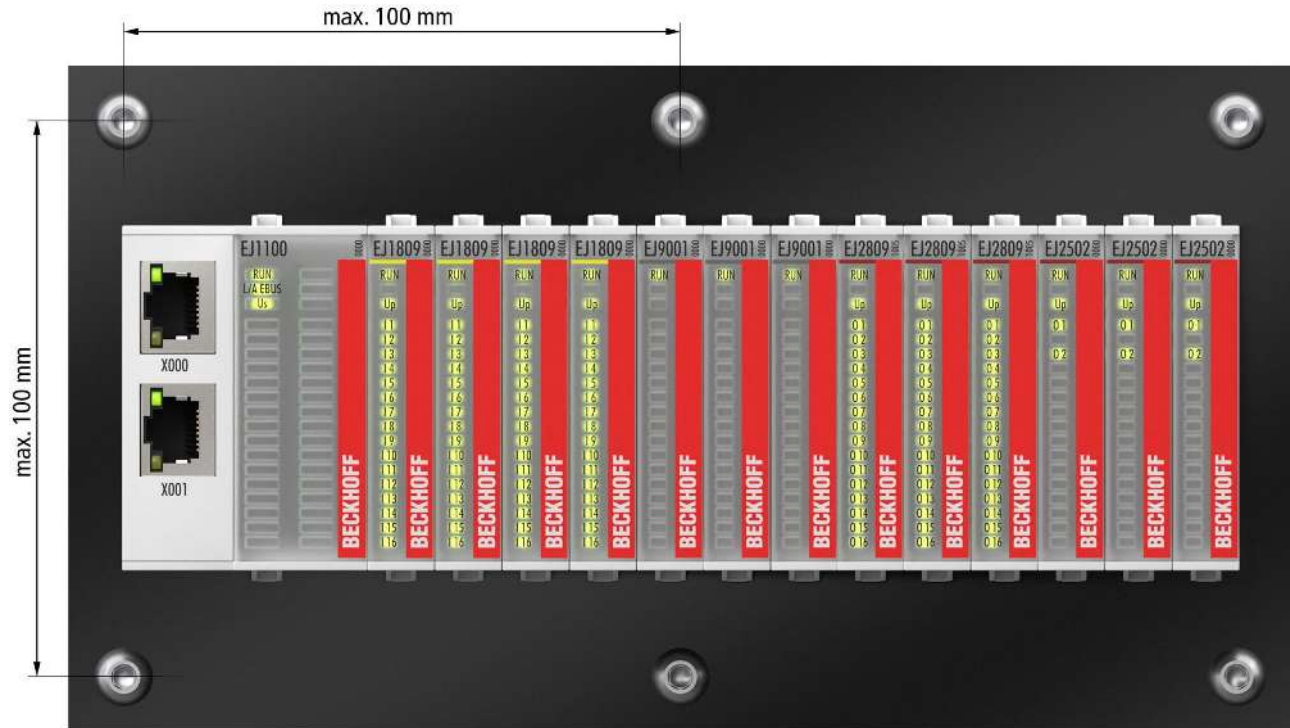


Fig. 1: Maximum distances between mounting holes and PCB

#### 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.

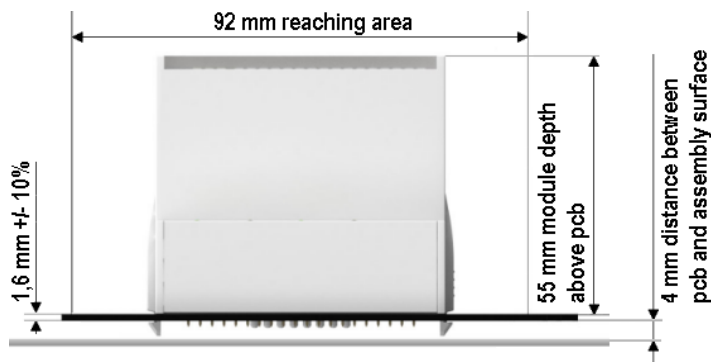


Fig. 2: Mounting distances EJ module - PCB

##### **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](#) [p. 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.

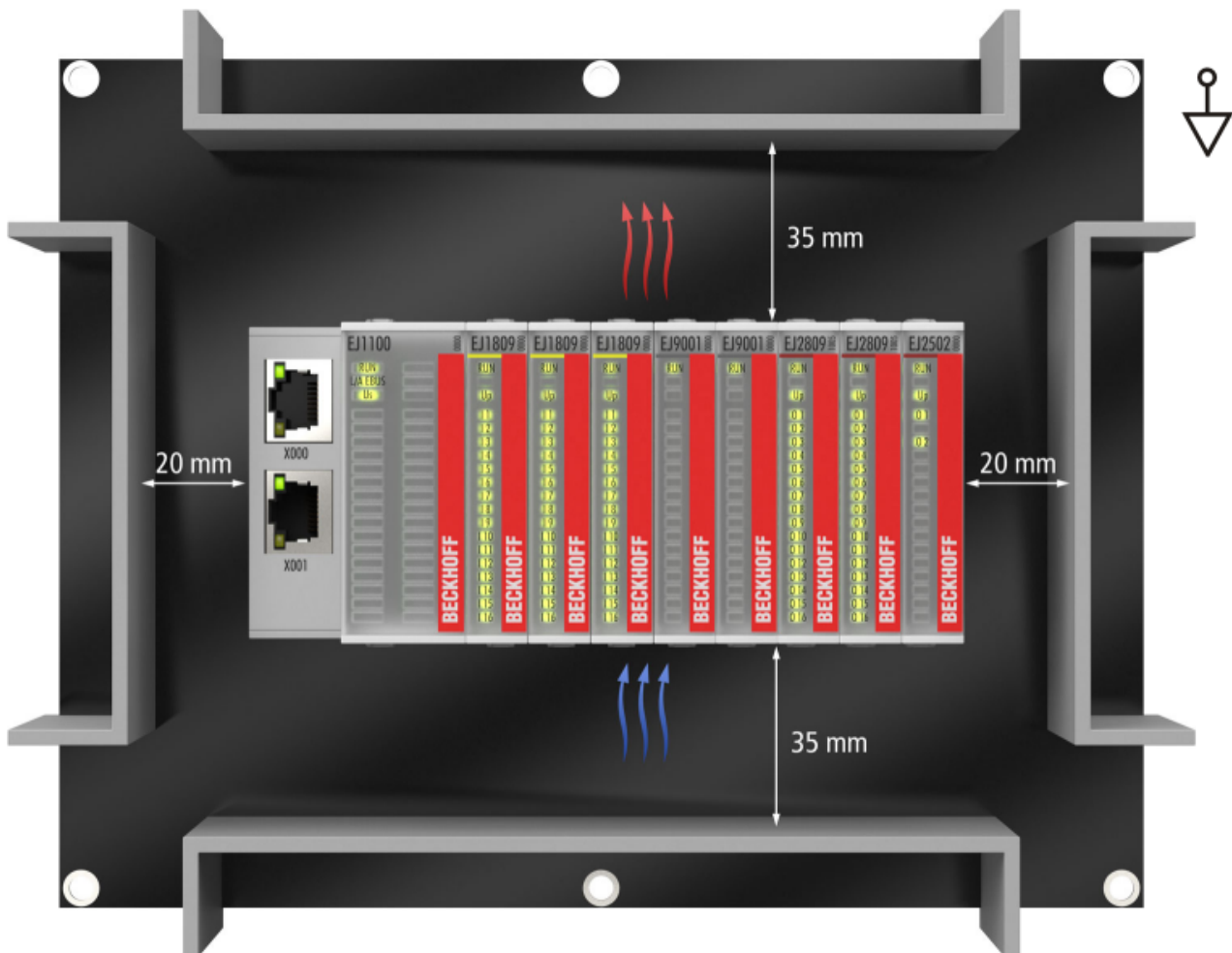


Fig. 3: Recommended distances for standard installation position

Compliance with the distances shown in Fig. "Recommended distances for standard installation position" is 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 connectors) followed by a power supply and IO-modules.

In order to avoid electromagnetic 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](#) [► 15] and [Routing guidelines](#) [► 25]!

#### Example with coupler EJ1100

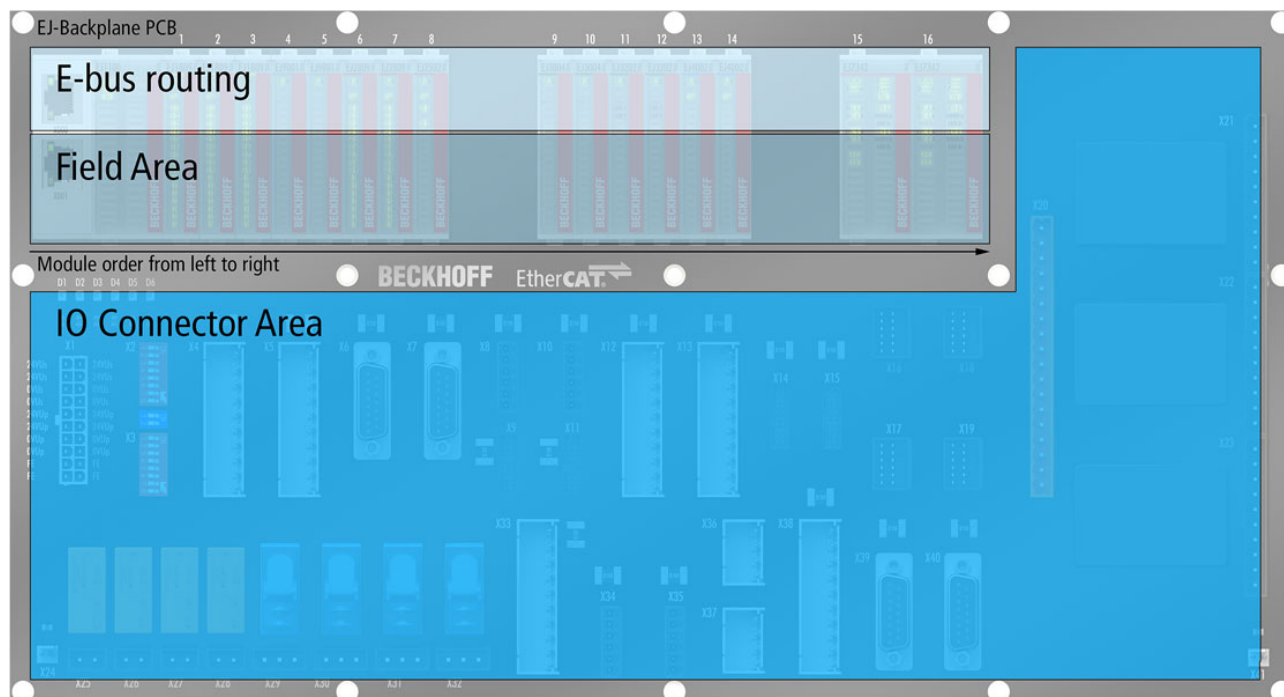


Fig. 4: EJ module line starting with coupler EJ1100

#### 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 electromagnetic interference shall be avoided.

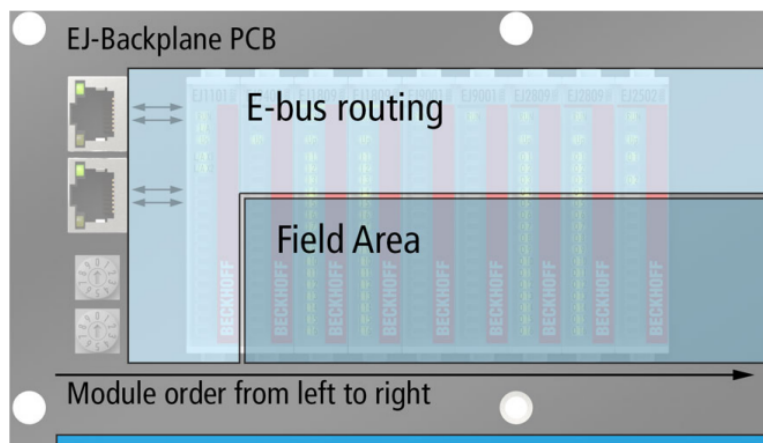


Fig. 5: EJ module line starting with coupler EJ1101-0022 (optional RJ45 sockets)

**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.

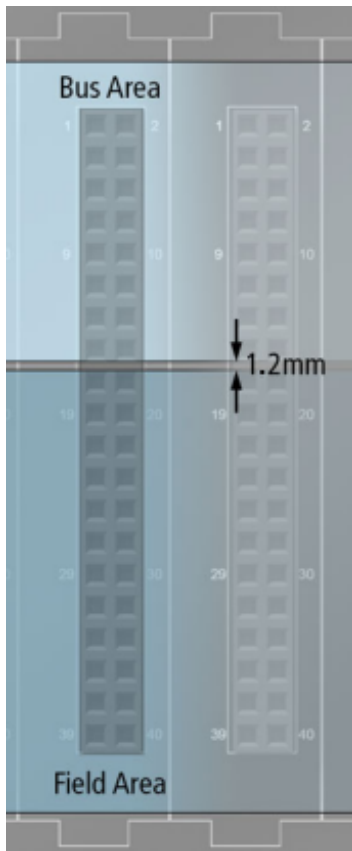


Fig. 6: Clearance between bus- and field area

## 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.

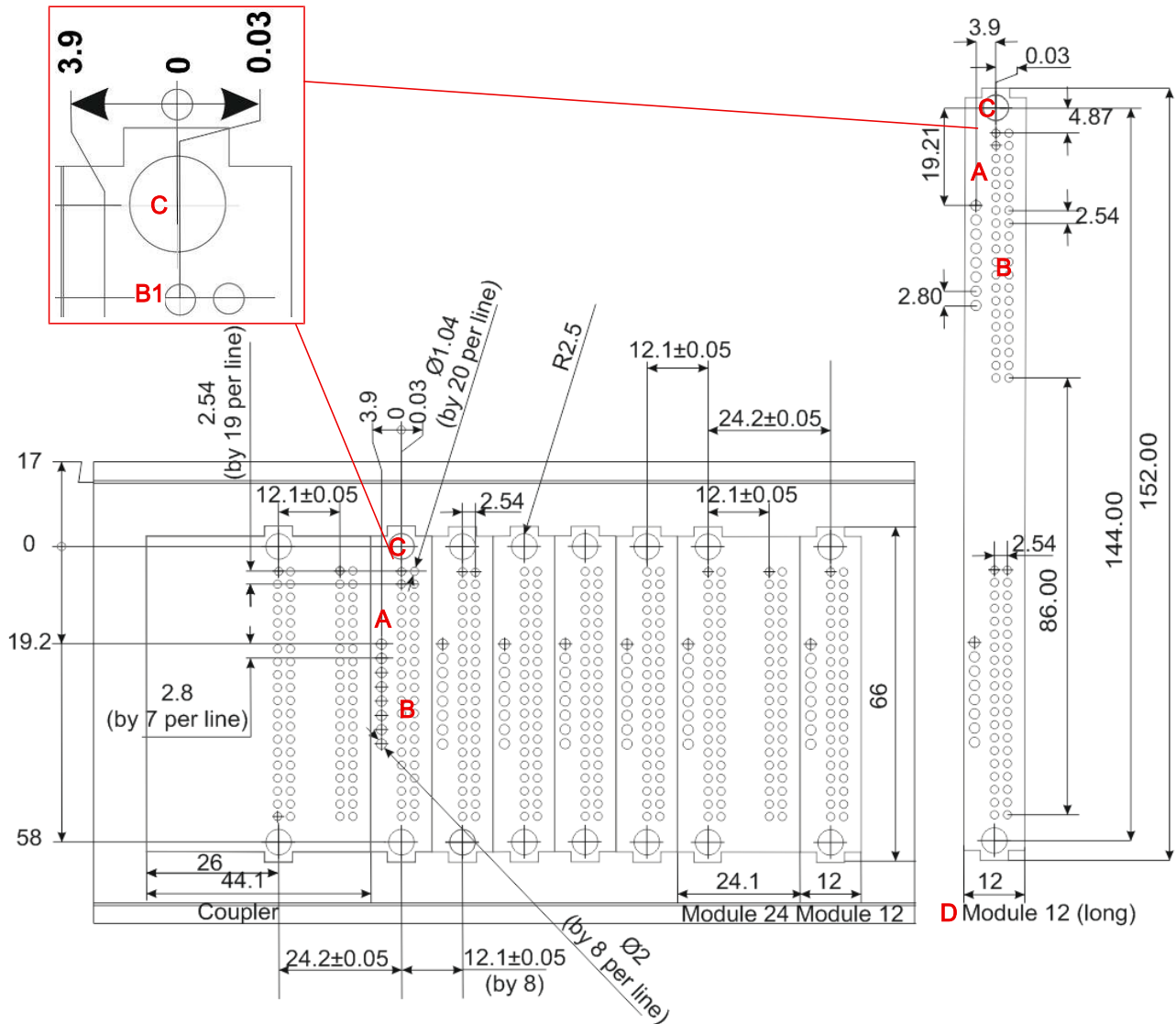


Fig. 7: Backplane layout, dimensions in mm

## 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 (Safety Extra Low Voltage) supply provides safe electrical isolation and limitation of the voltage without a connection to the protective conductor, a PELV (Protective Extra Low Voltage) 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:

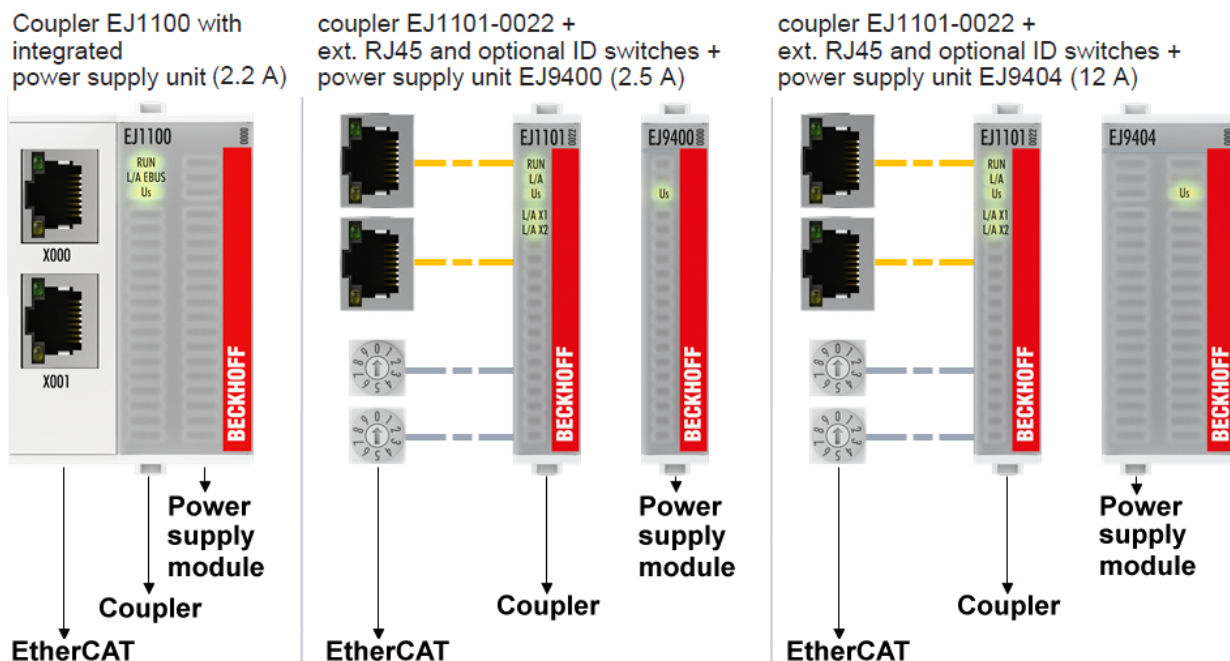


Fig. 8: E-bus power supply with EJ1100 or EJ1101-0022 + EJ940x

In the EJ1101-0022 coupler, the RJ45 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 ( $U_{\text{EBUS}}$ ) 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.

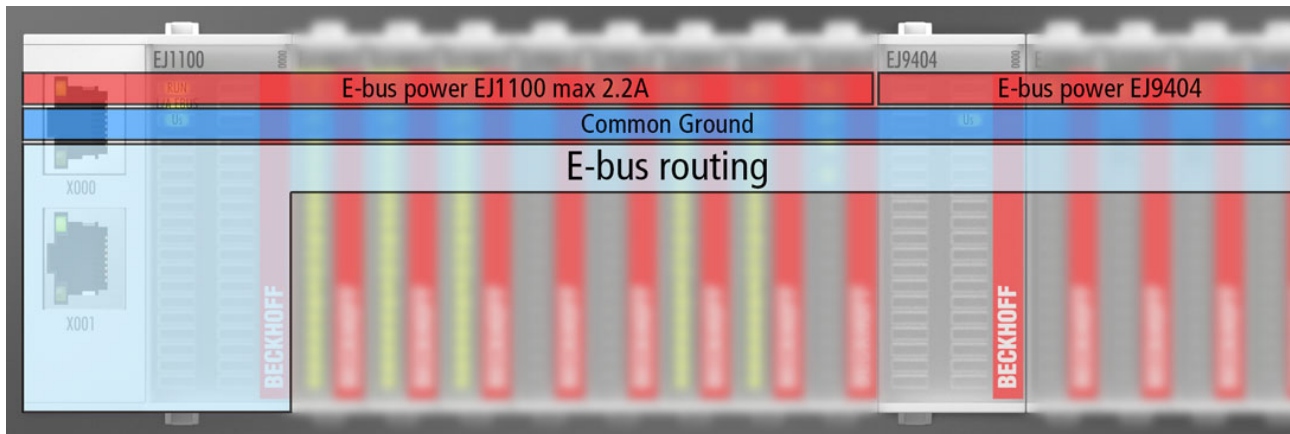


Fig. 9: Example with Coupler EJ1100, with integrated power supply (2.2 A), additional power supply with EJ9404 (12 A)

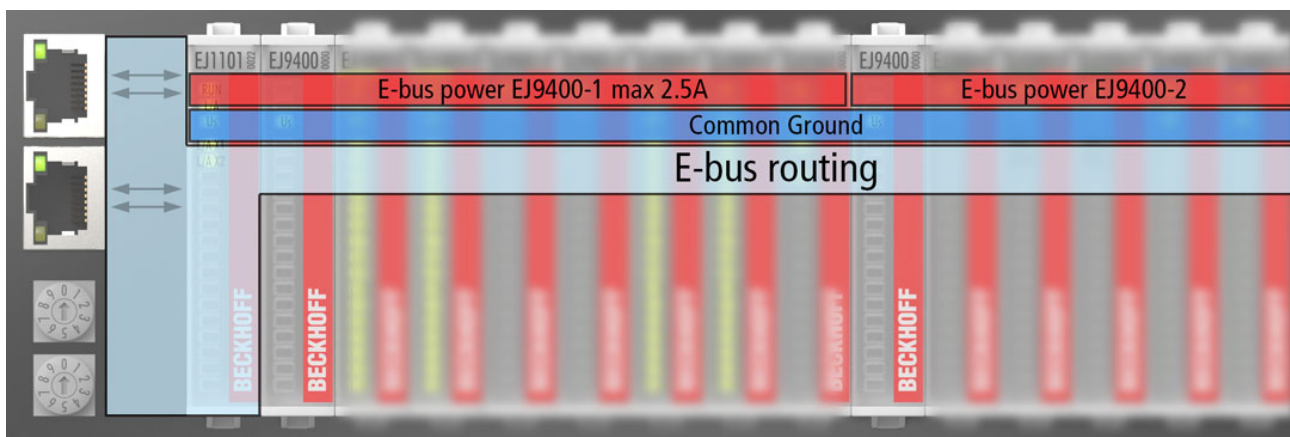


Fig. 10: Example with Coupler EJ1101-0022, power supply module EJ9400 (2.5 A), additional power supply with EJ9400 (2.5 A)

## 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.

### NOTE

#### 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](#) [► 18]).

#### Example with the EJ1100 EtherCAT coupler

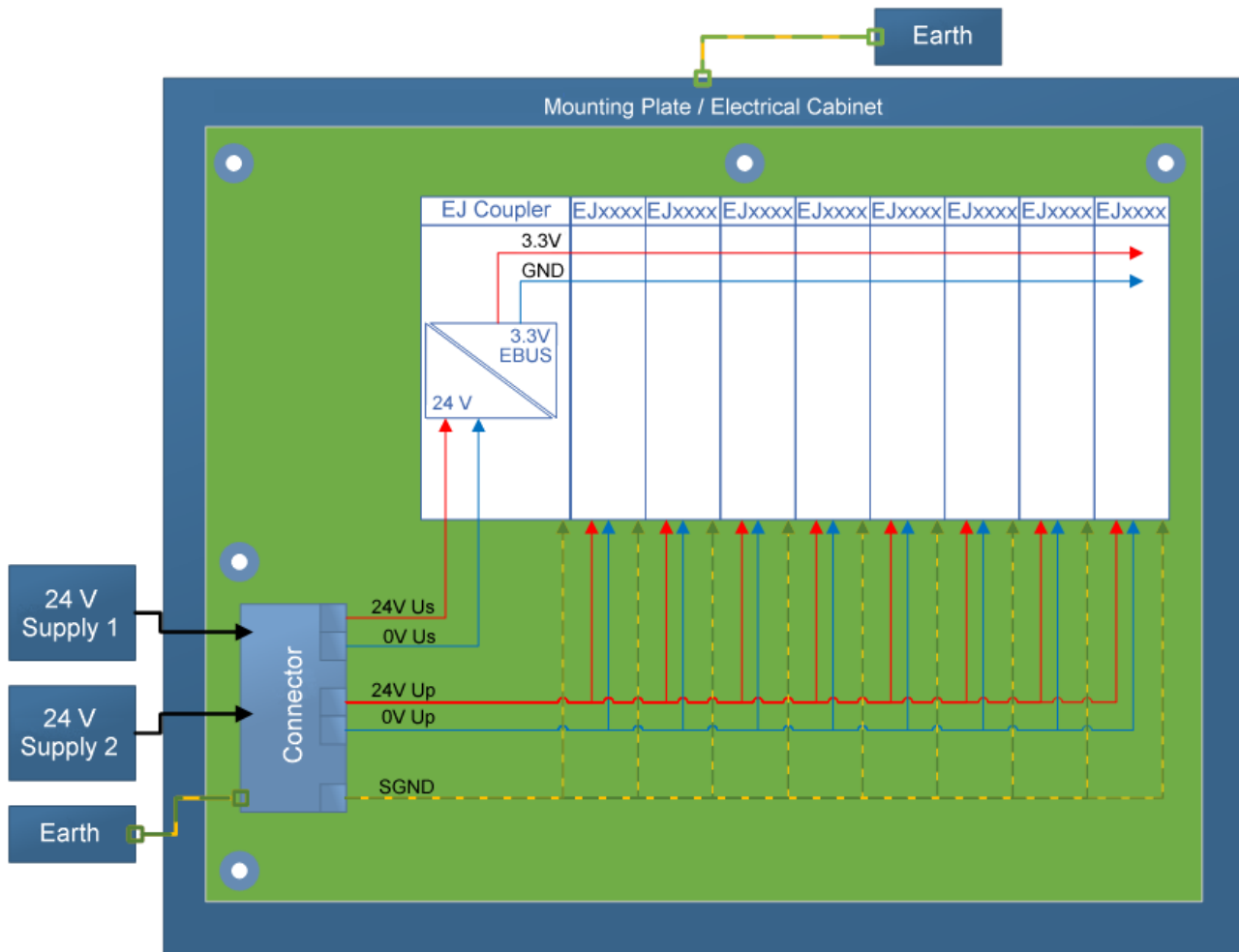


Fig. 11: Power supply of the EtherCAT plug-in modules via the EJ1100 EtherCAT coupler



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

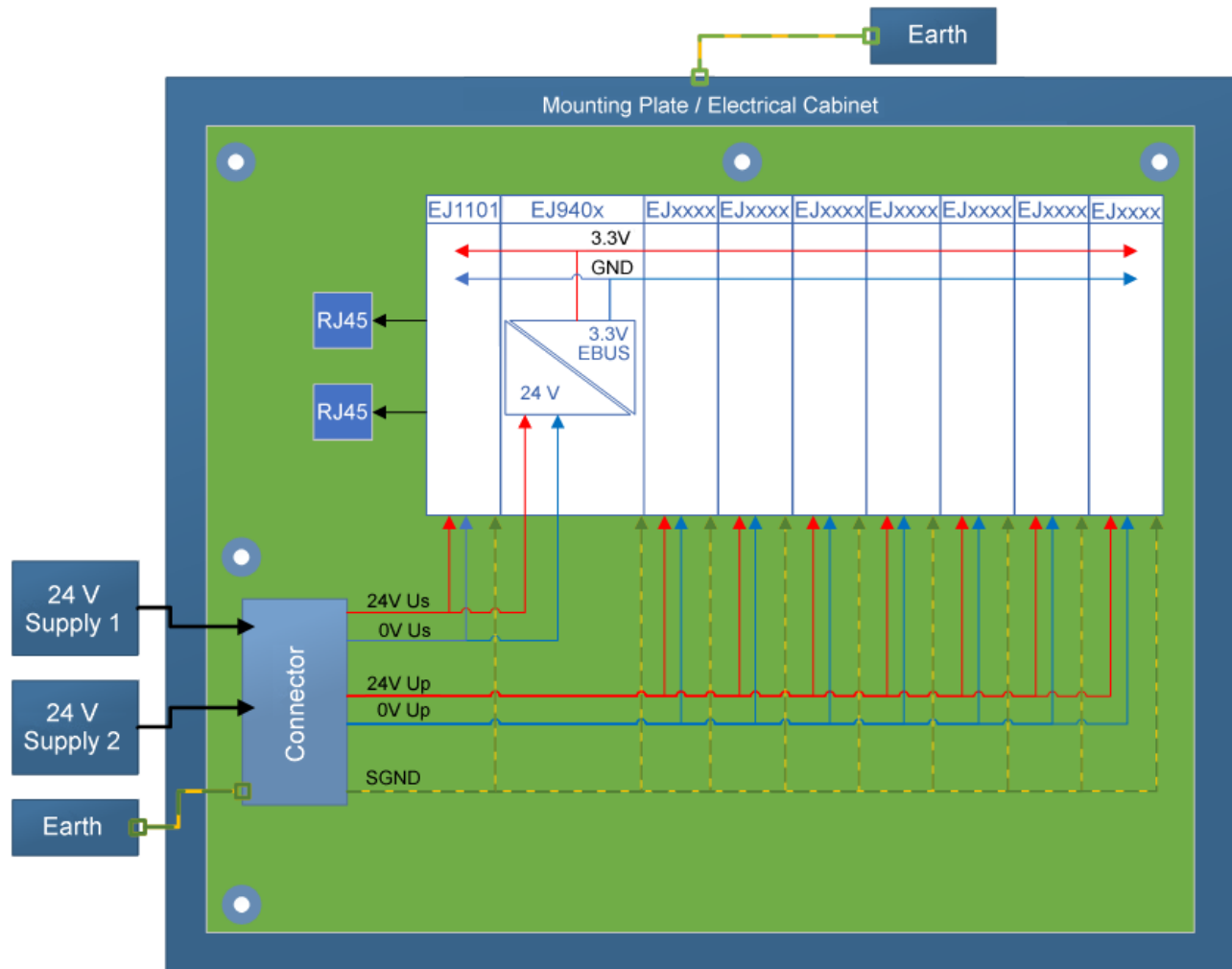


Fig. 12: Power supply of the EtherCAT plug-in modules via the EJ1101-0022 EtherCAT coupler and power supply module EJ940x

**NOTE**

**Note on routing**

- Read the notes on routing in chapter [Module placement](#) [► 11], [Design of E-bus power supply](#) [► 14] and [Routing guidelines](#) [► 25]!
- Observe additional notes in chapter *pinout* of the documentations for the modules used.

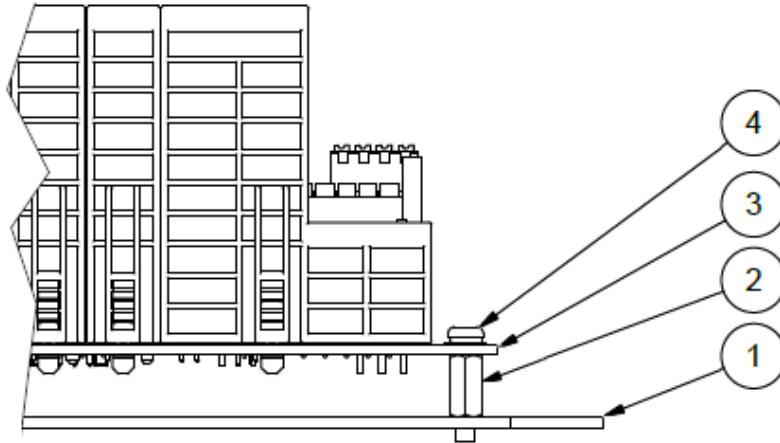
## 8 SGND Connection

### NOTE

#### Notes on the SGND connection

The copper rings around the holes are connected to SGND (see chapter [Top layer](#) [► 20]).

- 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.

#### 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  $1.6 \text{ mm} \pm 10\%$ .

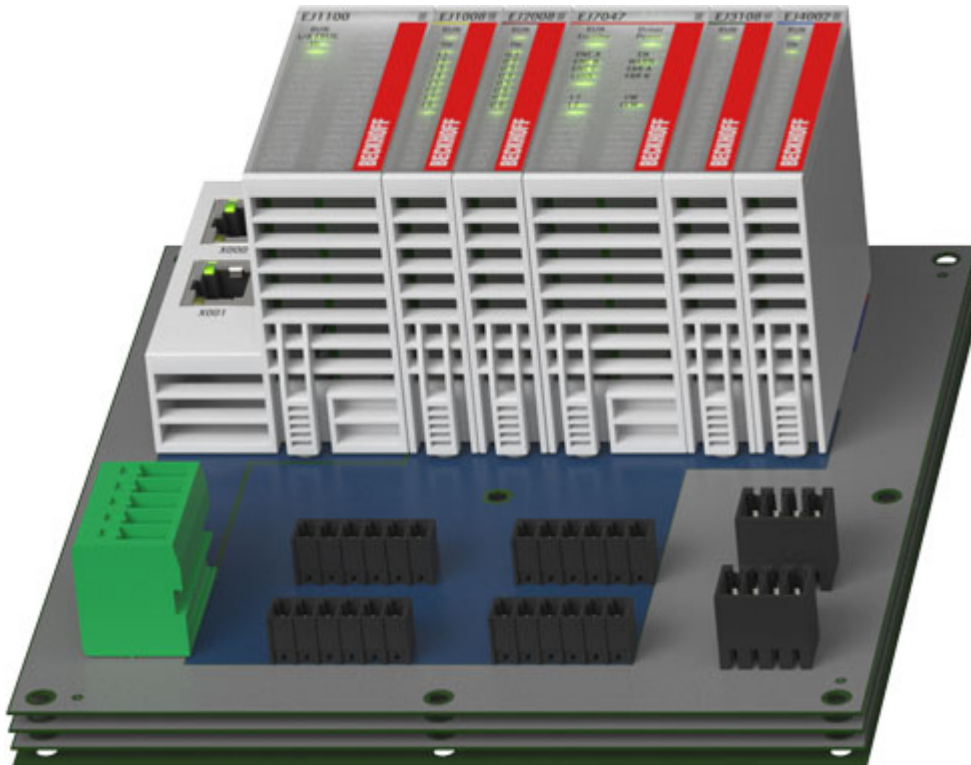


Fig. 13: Requirements of the PCB (min. four layers, max. 1.6 mm thickness)

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

#### NOTE

##### Note on routing

- Read the notes on routing in chapter [Module placement](#) [► 11], [Design of E-bus power supply](#) [► 14] and [Routing guidelines](#) [► 25]!
- Observe additional notes in chapter *pinout* of the documentations for the modules used.

## 9.1 Top layer

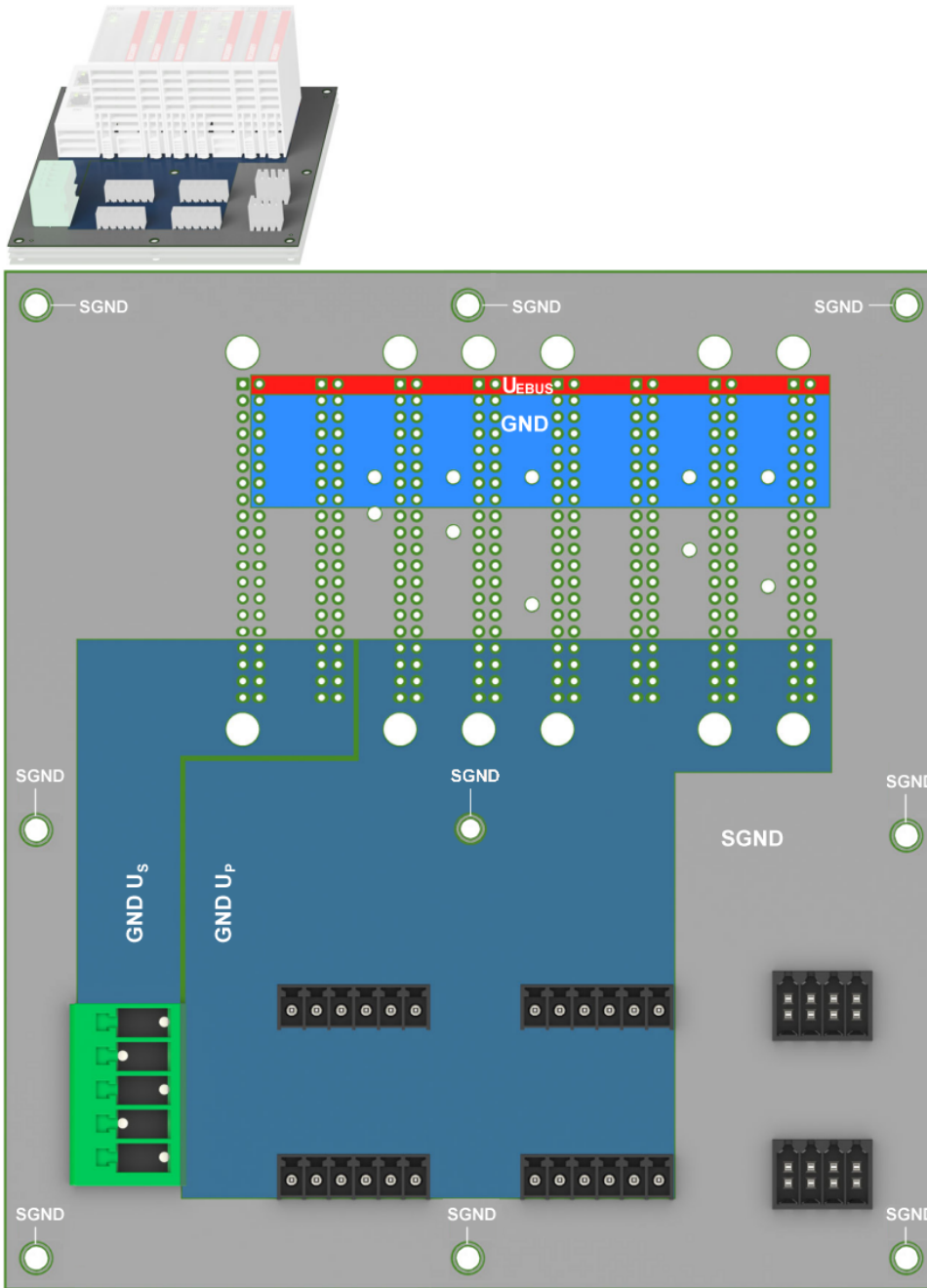


Fig. 14: Example for routing the top layer of a four-layer PCB

- Keep 0 V 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 mounting plate shall be implemented as metal bolts building a direct connection between signal distribution board and mounting plate (see chapter [SGND Connection](#) [18]). The copper rings around the holes are connected to SGND.
- It is recommended to route the signals SGND, 0 V Us/Up and 24 V Us/Up as an area.

### **i** Layout for Up power supply

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.

## 9.2 Inner layer 1

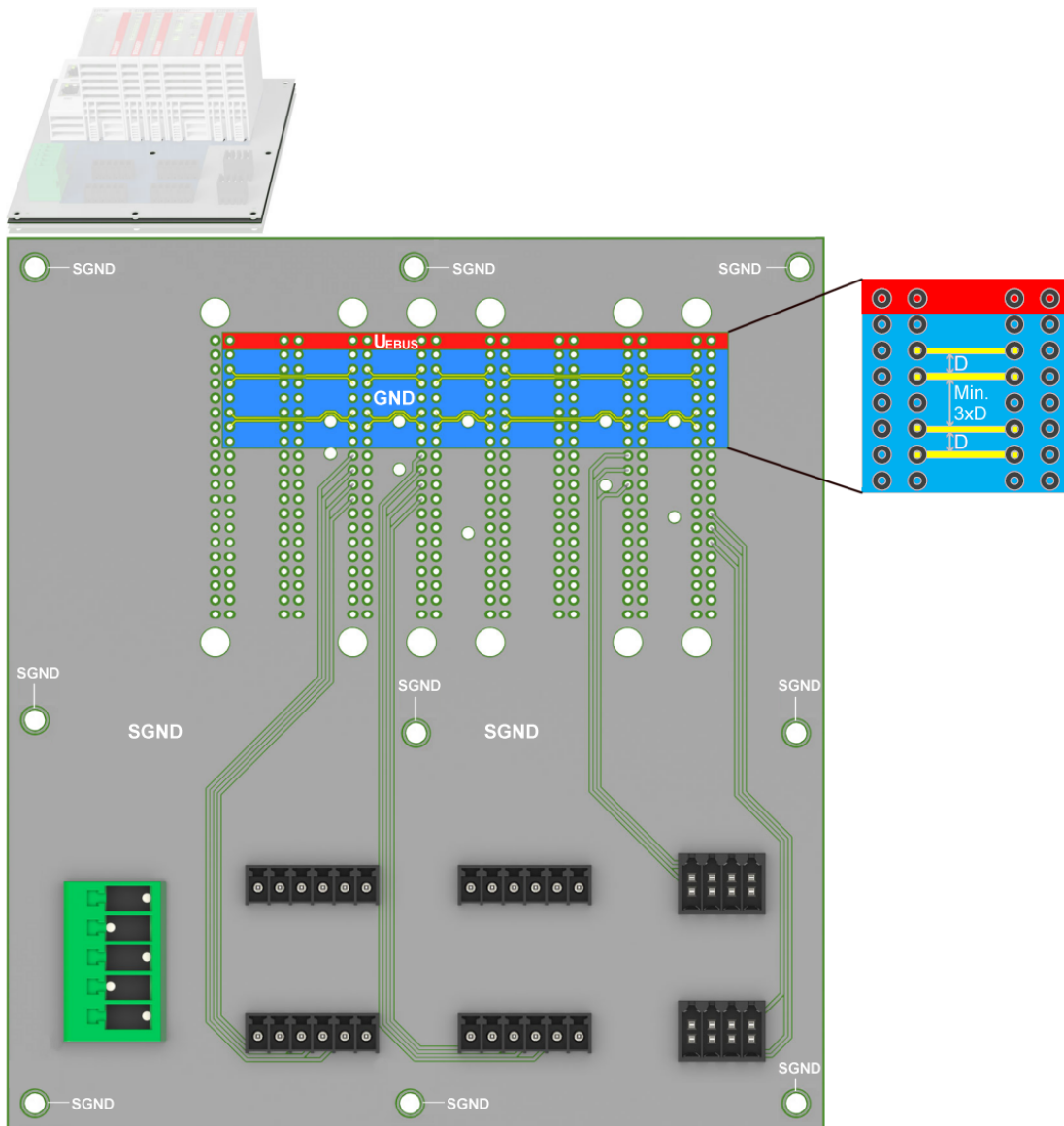


Fig. 15: Example for routing the inner layer1 of a four-layer PCB

- 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

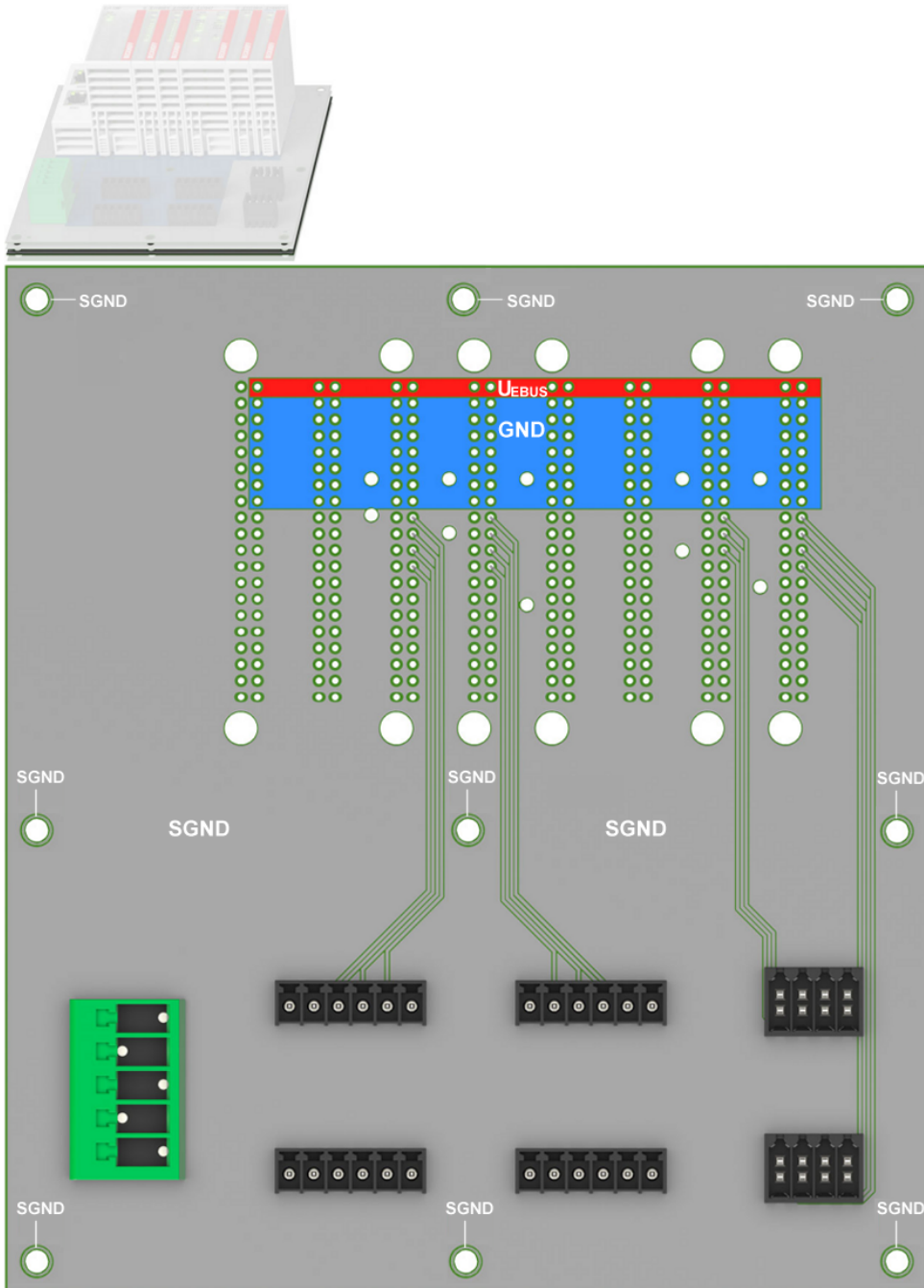


Fig. 16: Example for routing the inner layer 2 of a four-layer PCB

- 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

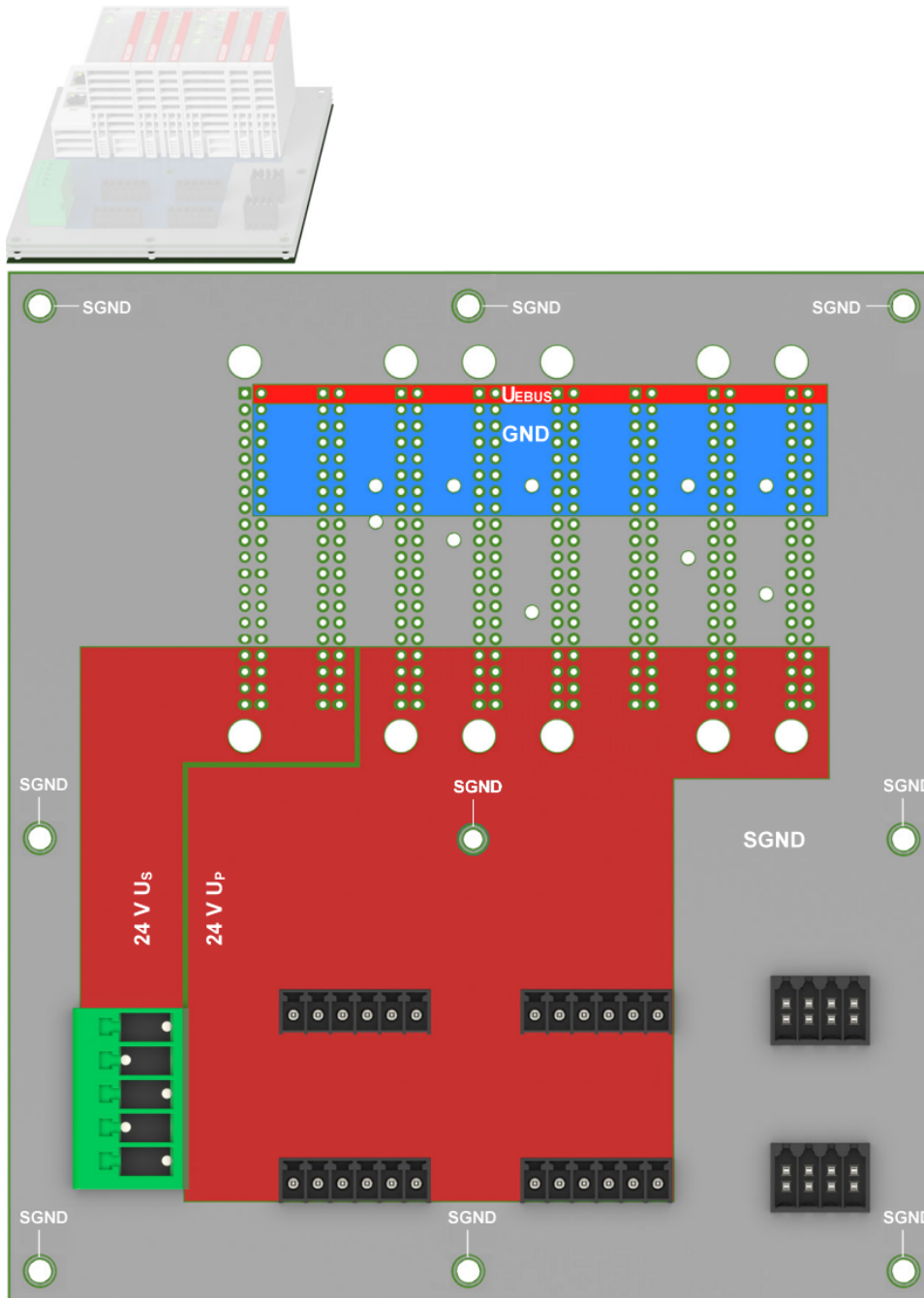


Fig. 17: Example for routing the bottom layer of a four-layer PCB

- 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.

### **i** Layout for Up power supply

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  $U_{\text{EBUS}}$  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

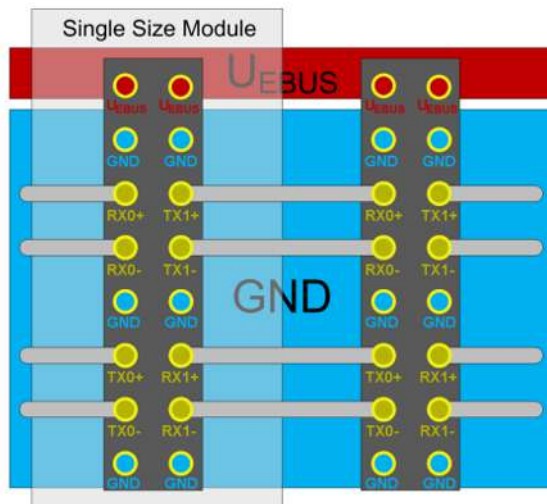


Fig. 18: Routing for 12-mm-modules

### 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.

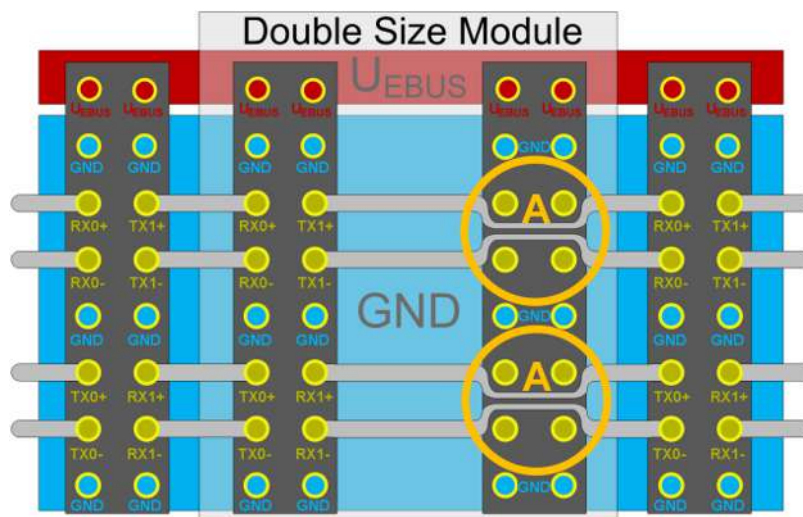


Fig. 19: Routing for 24-mm modules

### 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 [► 18] 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 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 mounting plate shall be implemented as metal bolts building a direct connection between signal distribution board and mounting plate (see chapter [SGND connection](#) [► 18]).

## 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  $\Omega$ .
- 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)

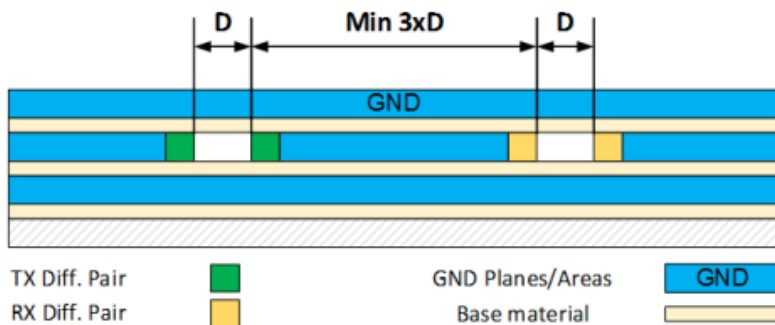


Fig. 20: Differential pair spacing

### NOTE

#### 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](http://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](mailto:support@beckhoff.com)  
web: [www.beckhoff.com/support](http://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](mailto:service@beckhoff.com)  
web: [www.beckhoff.com/service](http://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](mailto:info@beckhoff.com)  
web: [www.beckhoff.com](http://www.beckhoff.com)



More Information:  
**[www.beckhoff.com/EJ8xxx](http://www.beckhoff.com/EJ8xxx)**

Beckhoff Automation GmbH & Co. KG  
Hülshorstweg 20  
33415 Verl  
Germany  
Phone: +49 5246 9630  
[info@beckhoff.com](mailto:info@beckhoff.com)  
[www.beckhoff.com](http://www.beckhoff.com)

