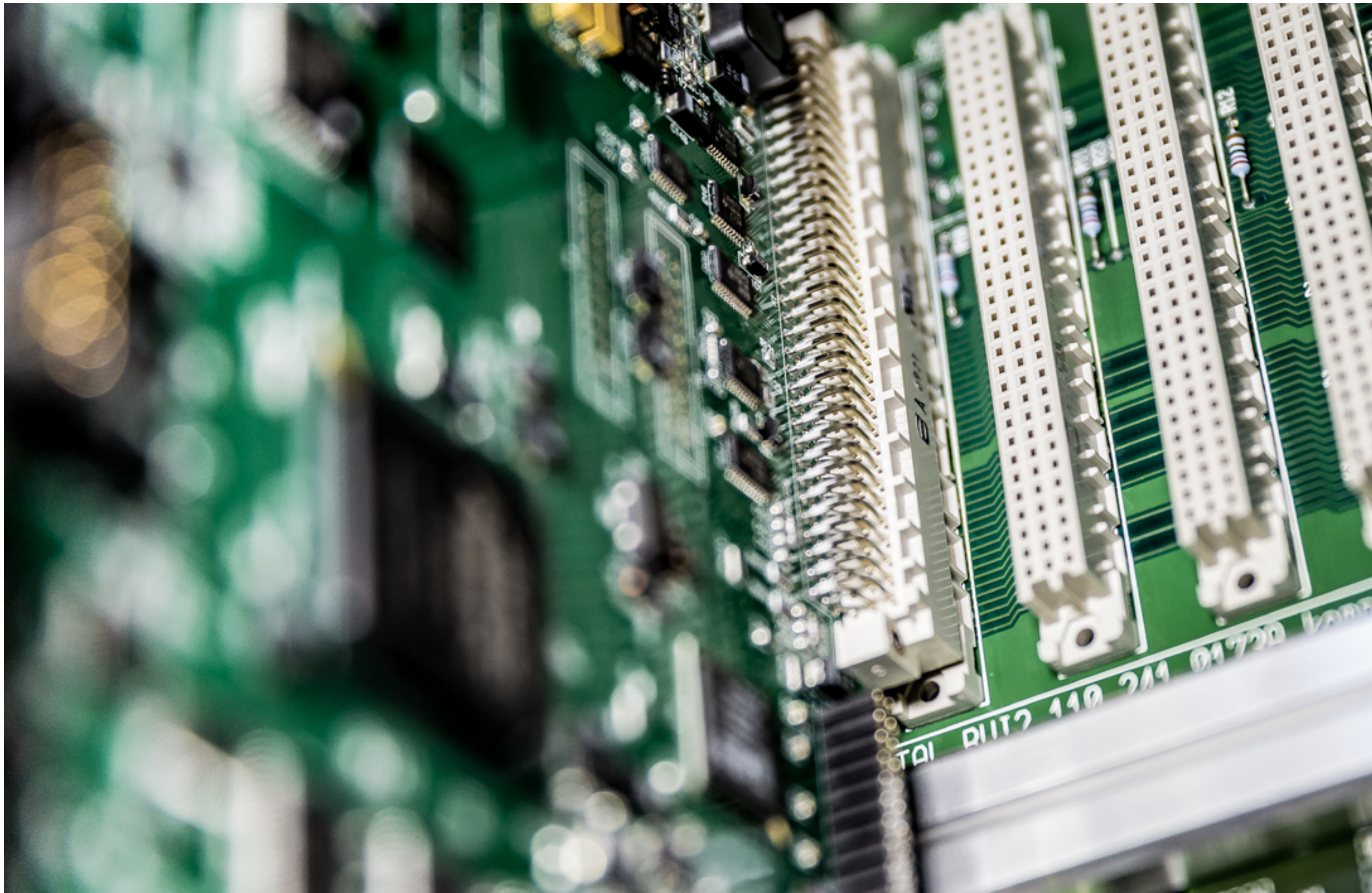
REFERENCE CHANNELS FOR RATING BOARD CONNECTORS

High Speed Data Transmission and Signal Integrity

Pushing Performance
Since 1945

Electronics devices are getting smaller and more modular. Oftentimes there are several PCBs inside a device, and board connectors are necessary for their interconnection. Application examples include backplane systems, industrial drives, control units, e-mobility charging stations and others.

Many of today's digital electronics in devices operates at increasingly high data rates. While searching for connector solutions for an application, engineers will need to know a connector's signal integrity performance, for example if the connector supports a specific data protocol or special data rates. With these metrics engineers can compare different connector solutions to each other and narrow down their connector options.



PEOPLE. POWER. PARTNERSHIP.

It is important to consider that a connector is one component in a PCB system. The SI performance of a system depends on a variety of factors - so when it comes to digital data transmission, the components of the complete channel must be matched to each other. A bad PCB design can cause bit failures even with the best connectors. On the other hand, a low-speed connector can support high data rates if the channel is designed accordingly.

To allow a fair comparison of different board connector data rates HARTING has defined a set of reference channels.

These channels simulate the complete transmission path from transmitter to receiver including crosstalk and reflections. The channel performance will be assigned to the connector, which provides both a reference for channel design and a metric to compare different connectors to each other. The metrics will be provided in table format so engineers can use it as a practical reference.

Preliminary remarks:

Please note that for most multipin connectors it is not possible to route all connections on the same layer. The maximum routing capability of the footprint sets the number of layers needed to route all necessary signals. The stub length then varies depending on the layer. The GND-to-GND spacing of the stripline setup is assumed to be 0.4 mm allowing 4-5-4 routing with 0.9 mm (35 mil) clearance. A 3-4-3 routing, which is increasingly applied in dense electronics, is not used for the sake of comparability.

Material notes:

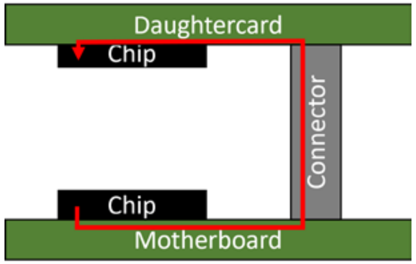
The PCB materials used in the channel simulation shall reflect the application specific needs in regards to Insertion Loss, i.e. higher data rates need lower loss materials. The low performance material used in simulation is assumed to have dk=4.3 / df=0.023, mid performance material has dk=3.6 / df=0.009, and high performance material has dk=3.4 / df=0.006.

REFERENCE CHANNEL DEFINITIONS

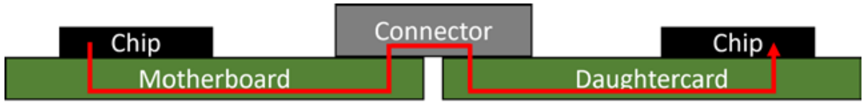
Single Interface Short Channel

In a Single Interface Short Channel there is a direct signal path from the transmitter located on the Motherboard to the receiver on the Daughtercard. The channel includes 1 mated connector interface connecting the Motherboard to the Daughtercard.

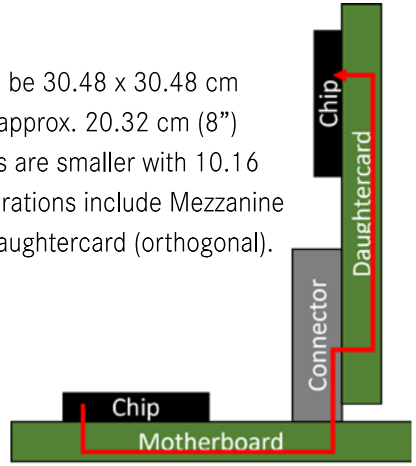
This channel refers to systems which are assumed to be 30.48 x 30.48 cm (12”x12”) max. Switching is centralized, resulting in approx. 20.32 cm (8”) max trace length on the Motherboard. Daughtercards are smaller with 10.16 cm (4”) maximum trace length. Possible PCB configurations include Mezzanine (parallel), Extender (coplanar) and Motherboard-to-Daughtercard (orthogonal).



Mezzanine



Extender



Motherboard-to-Daughtercard

FR4 Material with 18 µm (0.5 oz) copper is used for both PCBs.

Motherboard:

- Trace length: 20.32 cm (8”)
- Routing [mil]: 4-5-4
- Material: See material notes
- Signal skew at the end of the channel <5 ps
- Differential Impedance mismatch of the traces ±6 Ω
- Overall PCB thickness: 2.3 mm
- Min stub length: 0.3 mm
- Max stub length: add 0.4 mm per additional required layer (based on routing capability)

Daughtercard:

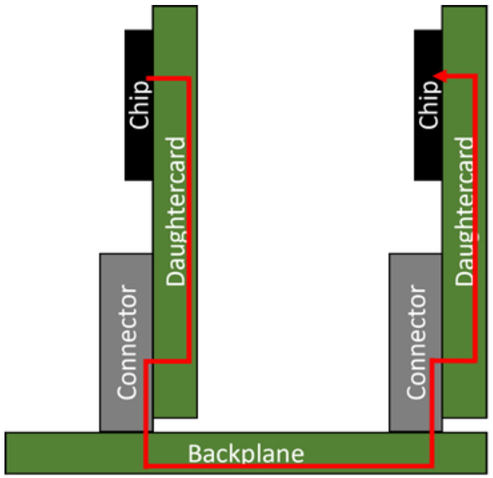
- Trace length: 10.16 cm (4”)
- Routing [mil]: 4-5-4
- Material: See material notes
- Signal skew at the end of the channel <5 ps
- Differential Impedance mismatch of the traces ±6 Ω
- Overall PCB thickness: 1.6 mm
- Min stub length: 0.3 mm
- Max stub length: add 0.4 mm per additional required layer (based on routing capability)

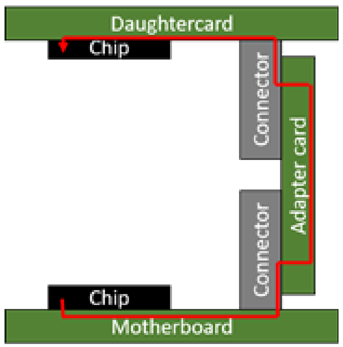
Double Interface Short Channel

The Double Interface Short Channel is based on devices using several PCBs. It accounts for the increasingly compact design of PCB applications and features smaller PCBs and shorter traces than the Double Interface Long Channel.

The signal path passes 2 mated connector interfaces from the transmitter PCB to the receiver PCB, through an intermediate PCB. The intermediate PCB may be a backplane or an adapter card.

The intermediate PCB (i.e. Backplane/Adapter card) is assumed to have approx. 12.7 cm (5”) long traces and the controller is located on one of the plug-in cards on one end of the intermediate PCB. The plug-in cards are assumed to have the high-speed silicon placed close to the connector.





A special case of Double Interface Channel is the Mezzanine configuration with the help of an adapter card.

Daughtercard/Motherboard:
- Trace length: 10.16 cm (4”)
- Routing [mil]: 4-5-4
- Material: See material notes
- Signal skew at the end of the channel <5 ps
- Differential Impedance mismatch of the traces ±6 Ω
- Overall PCB thickness: 2.3 mm
- Min stub length: 0.3 mm
- Max stub length: add 0.4 mm per additional required layer (based on routing capability)

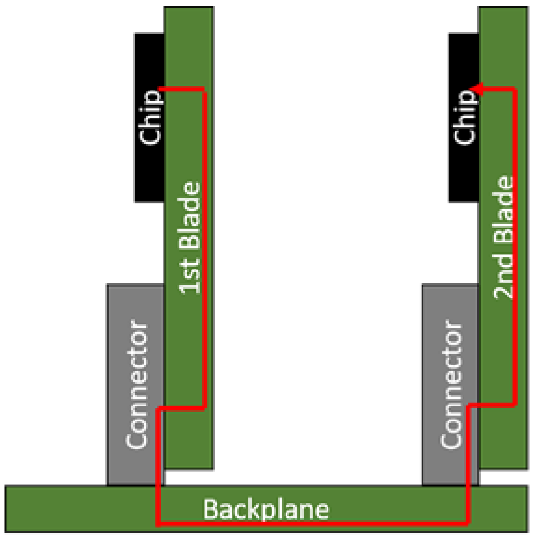
Backplane/Adapter card:
- Trace length: 12.7 cm (5”)
- Routing [mil]: 4-5-4
- Material: See material notes
- Signal skew at the end of the channel <5 ps
- Differential Impedance mismatch of the traces ±6 Ω
- Overall PCB thickness: 4.0 mm
- Min stub length: 0.3 mm
- Max stub length: add 0.4 mm per additional required layer (based on routing capability)

Double Interface Long Channel

The Double Interface Long channel is based on the classic backplane PCB configuration with one backplane and several daughtercards, or “Blades”. It features bigger PCBs and longer traces than the Double Interface Short Channel.

The signal path from the transmitter (on 1st Blade) to the receiver (on 2nd Blade) includes 2 mated connector interfaces. The backplane is an intermediate PCB.

With the switch being in the center of the Backplane, the longest signal path will be approx. 38.1 cm (15”). Trace length on the Daughtercards is assumed to be 12.7 cm (5”) max. FR4 Material with 18 µm (0.5 oz) copper is used (Nelco 4000-6: Dk=4,3; DF=0.023) for the PCB material of both Backplane and Daughtercard.



Blade:

- Trace length: 12.7 cm (5”)
- Routing [mil]: 4-5-4
- Material: See material notes
- Signal skew at the end of the channel <5 ps
- Differential Impedance mismatch of the traces ±6 Ω
- Overall PCB thickness: 2.3 mm
- Min stub length: 0.3 mm
- Max stub length: add 0.4 mm per additional required layer (based on routing capability)

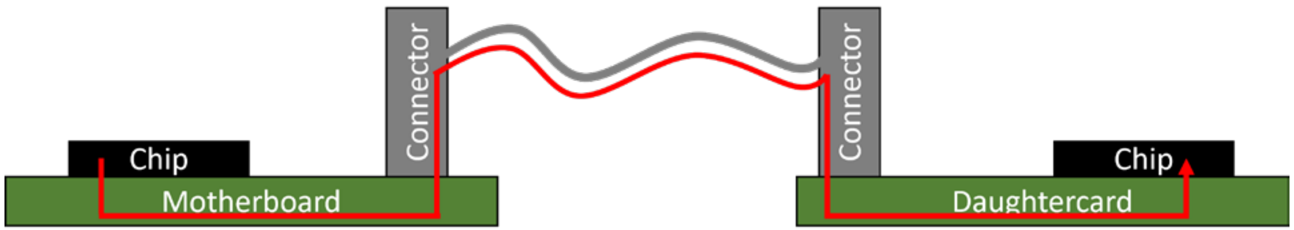
Backplane:

- Trace length: 38.1 cm (15”)
- Routing [mil]: 4-5-4
- Material: See material notes
- Signal skew at the end of the channel <5 ps
- Differential Impedance mismatch of the traces ±6 Ω
- Overall PCB thickness: 4.0 mm
- Min stub length: 0.3 mm
- Max stub length: add 0.4 mm per additional required layer (based on routing capability)

Cable Channel

In a cable channel two PCBs are connected by a cable. The SI performance of cable-based channels strongly depends on the specific cable parameters.

Main differences are e.g., the conductor diameter and the cable length. To account for the different cable parameters, they will be left as variables in the general channel definition. Instead, they will be defined individually for each connector series (based on the most relevant use cases) and they will be provided with the results for each connector. Both ends of the cable are connected to a PCB through a connector. So, the signal path from transmitter (on 1st PCB) to receiver (on 2nd PCB) includes 2 mated connector interfaces, connected by a cable.



PCB parameters::

- Trace length: 5.1 cm (2”)
- Routing [mil]: 4-5-4
- Material: See material notes
- Signal skew at the end of the channel <5 ps
- Differential Impedance mismatch of the traces ±6 Ω
- Overall PCB thickness: 1.6 mm
- Min stub length: 0.3 mm
- Max stub length: add 0.4 mm per additional required layer (based on routing capability)

Possible parameters defined individually are:

- Cable type: e.g. ribbon cable, single wires, etc
- Wire Gauge / conductor diameter
- Cable length: 0.3 m, 0.5 m, etc
- Dielectric material: PTFE, PVC, TPE, etc
- Signal skew at the end of the channel <50 ps
- Differential Impedance mismatch of the traces 5
- ...

RESULT

A Board Connector's Signal Integrity Performance

After a connector is simulated in the relevant reference channels, the results are evaluated and summarized. The possible data rate and recommended protocols are presented in a table enabling the user to get a quick overview of the possible applications for the connector. The simulation results for each simulated board connector will be included in the technical information provided.

For each data transmission protocol, the table specifies whether the connector is "suitable" or "not recommended". As an example, the *har*-flex® HD-Card Edge can be used for data transmission with up to 25 Gbit/s for the following protocols.

Example:

har-flex® HD-Card Edge Family Product Data Rate: 25Gbit/s				
Protocol	**Serial Data Rate / Pair**	**Single Interface Short Channel**	**Double Interface Short Channel**	**Double Interface Long Channel**
400GBASE-KR4	56 Gbit/s	Not recommended	Not recommended	Not recommended
100GBASE-KR4	25 Gbit/s	suitable	Not recommended	Not recommended
40GBASE-KR4	10 Gbit/s	suitable	suitable	suitable
Infiniband HDR	50 Gbit/s	Not recommended	Not recommended	Not recommended
Infiniband HDR	25.8 Gbit/s	Not recommended	Not recommended	Not recommended
Infiniband FDR	14 Gbit/s	suitable	suitable	suitable
PCIe Gen 4	16 Gbit/s	suitable	suitable	suitable
PCIe Gen 3	8 Gbit/s	suitable	suitable	suitable
PCIe Gen 2	5 Gbit/s	suitable	suitable	suitable
USB 3.1	5 Gbit/s	suitable	suitable	suitable
USB 3.0	5 Gbit/s	suitable	suitable	suitable
Hypertransport 3	5.2 Gbit/s	suitable	suitable	suitable
SATA 3.2	16 Gbit/s	suitable	suitable	suitable
SAS 4.0	22.5 Gbit/s	Not recommended	Not recommended	Not recommended
SAS 3.0	12 Gbit/s	suitable	suitable	suitable
SAS 2.0	6 Gbit/s	suitable	suitable	suitable

[1] Mid Performance Material [2] High Performance Material

The HARTING Technology Group

Intelligent high-performance connection technology forms the basis of industrial application and product technology. The HARTING range of products and solutions comprises connectors, device connection technology, network components and ready-to-use system cables.

HARTING products connect and network devices, machines and facilities via data, signal and power. These offerings enable solutions for the following markets: automation technology, energy, transport technology, machine construction, medical technology and infrastructure. HARTING also produces electro-magnetic components for the automobile industry, charging technology and cables for electronic vehicles and provides solutions for the fields of Industrie 4.0 (Integrated Industry/Industrial Internet), housing technology and shop systems.

HARTING Technology Group's worldwide presence includes 14 production plants and 44 sales offices. Some 6,200 employees generated sales of EUR 1.059 million in 2021/22.

Further information available via:

HARTING Electronics GmbH
Marienwerderstraße 3 | 32339 Espelkamp
Tel. +49 (0)57 72 47-0
info@HARTING.com | www.HARTING.com

Version 1.1 | May 2023