25Gbps Automotive Ethernet ECU PCB: MDI Design Implementation and Insertion Loss Characterization

Jamila J. Borda[1], Kirsten Matheus[1] and Friedel Gerfers[2]
[1]*Department of Communications Network Technologies, BMW Group, Munich, Germany*
[2]*Department of Computer Engineering and Microelectronics: Mixed Signal Circuit Design, Technische Universitaet (TU) Berlin, Berlin, Germany*

Keywords: Automotive Multi-Gigabit Ethernet, MultiGBASE-T1, 2.5GBASE-T1, 5GBASE-T1, 10GBASE-T1, 25GBASE-T1, ECU, PCB, MDI, Insertion Loss, Signal Attenuation.

Abstract: Physical Layer (PHY) Signal Integrity (SI) aspects of an Automotive Ethernet communication channel are characterized using Radio Frequency (RF) parameters. With increasing Automotive Ethernet data rates, communication channel signal attenuations (i.e., Insertion Loss (IL)) are significantly worsened. At 25Gbps data rate, the communication in cars faces various electrical limits and all components (i.e., segments) of the communication channel have to be optimized in order to reach the expected performance requirements. One such component is the Electronic Control Unit (ECU) Printed Circuit Board (PCB) Media Independent Interface (MDI). Consequently, for such high-speed links, ECU PCB electrical and material properties have an impact on the overall IL. Considering the stringent Automotive Ethernet channel electrical requirements, this study proposes and characterizes ECU PCB MDI design concepts for a 25Gbps in-vehicle Ethernet connectivity. Furthermore, the design concepts are manufactured on test boards to characterize the corresponding MDI signal IL budget. The characterizations are conducted using RF test bench measurement and a defined simulation approach. Lastly, in relation to test bench measurements, this study investigates and characterizes to what extent simulations can serve as either an alternative or a coexisting option for in-vehicle 25Gbps MDI IL characterizations, validations, and qualifications.

1 INTRODUCTION

Increasing in-vehicle communication channel operating frequencies are accompanied with increasing in-vehicle system sensitivity to ElectroMagnetic Interference (EMI) and channel/components signal ILs. This results from steeper signal rise/fall SI timing requirements and higher Nyquist frequencies (i.e., signal bandwidth) of the ever-increasing data rates (Borda J. , 2022). Consequently, essential PHY system performance aspects of an Ethernet communication channel, namely system design and implementation, are impacted by these effects (Borda J. , 2022).

As a channel segment, the PCB MDI has a meaningful contribution to the overall channel performance in relation to SI and ElectroMagnetic Compatibility (EMC) characteristics. Right after the cable (i.e., link segment) IL, the PCB MDI serves as the second largest contributing factor to the overall channel IL. Link segment characterization for 25Gbps is referenced to the study in (Borda J. J., 2022). Hence, the channel PHY system design and implementation have to have a subsequent focus on the ECU PCB. The MDI network comprises of passive (and application dependent active) components. For MultiGBASE-T1 data rates (2.5GBASE-T1/2.5Gbps, 5GBASE-T1/5Gbps, 10GBASE-T1/10Gbps, 25GBASE-T1/25Gbps), the MDI network is reduced to Alternating Current (AC) coupling capacitors (for blocking undesired direct current) and Electrostatic Discharge (ESD) protection device on each of the differential traces (IEEE 802.3ch, 2020) (IEEE 802.3cy (Unpublished), 2022). Figure 1 provides an overview of the electrical Automotive Ethernet communication channel with the defined MDI segment and several other components (i.e., within this study interchangeably also described as “segments”). For a copper-based/electrical transmission, a Single Communication Channel (SCC) extends through the ECU PCB MDI up to the PHY transceiver (typically having a Ball Grid Array (BGA) design type) MDI

Borda, J., Matheus, K. and Gerfers, F.
25Gbps Automotive Ethernet ECU PCB: MDI Design Implementation and Insertion Loss Characterization.
DOI: 10.5220/0011756000003479
In *Proceedings of the 9th International Conference on Vehicle Technology and Intelligent Transport Systems (VEHITS 2023)*, pages 74-84
ISBN: 978-989-758-652-1; ISSN: 2184-495X
Copyright © 2023 by SCITEPRESS – Science and Technology Publications, Lda. Under CC license (CC BY-NC-ND 4.0)

differential pair pads/pins (IEEE 802.3ch, 2020). The MDI header connector can also be based on a multi-port system. This is either because more than a single differential pair port of the same interface is available, or because "hybrid" multi-port connectors with a blend of various communication interfaces (single or/and differential based signals) as well as power lines are being used (OPEN Alliance, 2020) (OPEN Alliance, 2022).

Figure 1: Electrical Automotive Ethernet SCC showing the MDI within the ECUs (Borda J. , 2022).

Several PCB electrical and material design properties must be well defined and characterized to ensure an optimum ECU PCB MDI signal IL budget and subsequently SI retention for a 25Gbps Automotive Ethernet channel:

(1) This starts off with having a proper selection of **dielectric materials** to be used for the complete ECU PCB design to accommodate both the high-speed design and Automotive requirements. One of its electrical properties i.e., loss tangent (also known as dissipation factor) plays a significant contribution to the PCB MDI signal IL.

(2) Another contributing factor to the PCB MDI signal IL is the **design concept** used **for the MDI differential pair trace layout**. Primarily, one differentiates between microstrip (including embedded) and Stripline PCB trace designs (Mittal, 2021). ECU PCBs are typically densely populated with several passive and active components occupying the MDI segment. A trace layout design structure targeting an optimum signal IL budget must therefore be tailored to ensure SI compliance.

(3) The overall PCB **layer stackup** concept serves as an additional characteristic aspect contributing to PCB MDI signal IL. To reach an optimum PCB MDI IL, several layer stackup design concepts must be thoroughly thought out. Considering Automotive channel requirements, these primarily include, (1) how to define the layer stackup (including signal and power planes, layer counts), (2) defining optimum differential pair trace lengths, width and spacing, and (3) design concepts for current return paths and the proper usage of vias to ensure compliance to SI.

To the authors' best knowledge, the aforementioned ECU PCB design aspects are yet to be thoroughly investigated and characterized in the Automotive industry for 25Gbps electrical Automotive Ethernet connectivity. This therefore calls for having these aspects adapted and, in some cases, the associated requirements and specifications in regard to PHY ECU PCB MDI system design and implementation need to be newly defined. Furthermore, this study serves as an essential baseline for the PHY system design technical feasibility study for the deployment of 25Gbps Automotive Ethernet.

Initial investigations of this study focus on defining optimum PCB MDI design concepts for 25Gbps data rate. Here, multiple PCB variants are defined considering several electrical properties. To emulate an ECU PCB, the design concepts are then implemented to be used as Device Under Test (DUT) for MDI IL characterization. Subsequent chapters target the actual characterization of the PCB MDI IL. Section 5 of this study discusses and defines the technical feasibility of deploying ECU PCB MDI simulations in correlation with a conventional test bench measurement approach.

2 THEORETICAL BACKGROUND

Figure 2 shows the side-view of the MDI within the ECU PCB defining the region on which the PCB MDI RF characterization takes place.

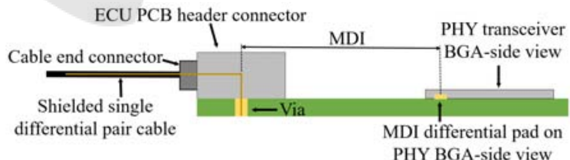


Figure 2: Overview of the MDI within the ECU PCB.

Validation and characterization of ECU PCB MDI design implementation primarily focuses on two system performance parametric categories, namely SI retention and compliance to EMC. The associated electrical properties cover RF, transient, and channel transmission line characteristic parameters (Borda J. J., 2022) (OPEN Alliance, 2022) (OPEN Alliance, 2020). For SI, IL is the primary essential parameter used to characterize the ECU PCB MDI in relation to the Automotive Ethernet communication overall

channel requirements (IEEE 802.3ch, 2020) (IEEE 802.3cy (Unpublished), 2022). Signal differential pair trace characteristic impedance, propagation delay, and an EMC-compliant PCB design are typically additionally characterized at the PCB-level independent of the communication channel. These particular characterizations are however not further covered within the scope of this study.

2.1 ECU PCB Total Losses

Generally, the overall ECU PCB total loss (i.e., IL) α_t is a sum of dielectric losses α_d, signal conductor losses α_c, leakage α_l and radiation losses α_r as described in Equation (1 (Borda J. J., 2022) (Polar Instruments, 2022) (Coonrod , 2013).

$$\alpha_t = \alpha_d + \alpha_c + \alpha_l + \alpha_r \quad (1)$$

Dielectric Losses: Dielectric losses are associated with the dissipation factor D_f (i.e., loss tangent). The governing equation for this is described and referenced in (Borda J. J., 2022) and (Mittal, 2021). Dielectric losses increase with operating frequency due to changing ElectroMagnetic fields. These changes cause the dielectric molecules to vibrate faster and hence leading to more energy loss. For high-speed transmission, lower loss tangents favour lower energy losses in the dielectric material (Knack, 2020).

Signal Conductor Losses: Conductor losses are associated with numerous variables. Amongst these are losses due to Skin-effect. At higher frequencies, i.e., in the GHz range, the signal current flows predominately on the conductor surface and the current density decays exponentially towards the center of the conductor, which causes the signal attenuation (Borda J. J., 2022) (Polar Instruments, 2022) (Coonrod , 2013). Surface roughness of a conductor is another issue contributing to PCB conductor losses. This happens when a signal conductor surface is rough causing longer wave propagation paths and consequently creating more losses. The losses are linked to parasitic inductances from surface inductance of the current following in partial loops in the copper metal profile (Coonrod , 2013).

Leakage Losses: Leakage losses are usually in relation to semiconductor grade materials. With the materials used in PCB technology generally having very high-volume resistance, leakage losses have an almost negligible contribution to the total PCB losses (Coonrod , 2013).

Radiation Losses: The occurrence of radiation losses is design dependent. Energy lost from a PCB circuit or from a transmission line radiated off to the surrounding environment typically describes radiation losses. These losses are at their maximum in impedance transitions and signal transmitting areas of the PCB. Radiation losses depend on the operating frequency, the dielectric constant, and the PCB substrate thickness (Coonrod , 2013).

Characterization of IL is conducted using Mixed-Mode S-Parameter analysis. A description of these S-Parameters is not covered within the scope of this study. However, these are referenced to descriptions found in (Borda J. J., 2022) and (Fan, Lu, Wai, & Lok, 2003).

2.2 ECU PCB MDI Insertion Loss

For 25GBASE-T1, the PCB MDI IL is specified based on maximum and minimum MDI differential pair trace lengths. These trace lengths are 76.2mm and 25.4mm respectively (IEEE 802.3ch, 2020) (IEEE 802.3cy (Unpublished), 2022). The governing equations for the maximum PCB MDI IL, $IL_{MDI-PCB,max}$ and minimum IL $IL_{MDI-PCB,min}$ are described in Equation (2) and Equation (3) respectively (IEEE 802.3cy (Unpublished), 2022). $IL_{MDI-PCB,max}$ and $IL_{MDI-PCB,min}$ defining equations for the lower MultiGBASE-T1 data rates are described in (IEEE 802.3ch, 2020).

$$IL_{MDI-PCB,max}(f) \leq 0.09144\left(f/_{1000}\right) + 0.51054\left(f/_{1000}\right)^{0.45} \; [dB] \quad (2)$$

$$IL_{MDI-PCB,min}(f) \geq 0.03048\left(f/_{1000}\right) + 0.17018\left(f/_{1000}\right)^{0.45} \; [dB] \quad (3)$$

Table 1 shows an overview of $IL_{MDI-PCB,max}$ for the various MultiGBASE-T1 data rates. These PCB MDI IL values solely cover a single PCB. The total PCB MDI IL to be considered in the complete channel is usually twice of this, i.e., $2IL_{MDI-PCB,max}$ rates (Borda J. J., 2022) (IEEE 802.3ch, 2020) (IEEE 802.3cy (Unpublished), 2022).

Table 1: An overview of the maximum specified PCB MDI IL for MultiGBASE-T1 data rates.

MultiGBASE-T1 PHY Technology	Nyquist Frequency [MHz]	$IL_{MDI-PCB,\ max}$ @ Nyquist Frequency [dB]
2.5GBASE-T1	703.125	0.6948
5GBASE-T1	1406.25	1.1238
10GBASE-T1	2812.5	1.8717
25GBASE-T1	7031.25	1.871

3 PCB MDI DESIGN CONCEPT

ECU PCB overall and MDI PHY electrical aspects must be considered to ensure SI (in this study with focus on an optimum IL). PCB design aspects within this study cover in-vehicle communication channel PHY in combination with high-speed transmission requirements for a 25Gbps connectivity.

Overall ECU PCB: For a 25Gbps connectivity, prior to defining the PCB MDI design, the PCB design in terms of the required PCB stackup, layer count, and dielectric material (with loss tangent) to be used are characterized with focus on high-speed signal transmission. Here, in addition to essential high-speed design considerations, for instance propagation delay, routing guidelines, current return paths, via definition, and impedance matching, the signal IL is also thoroughly considered at the forefront (Johnson & Graham, 1993). A typical ECU PCB would have a minimum of 8 layers. These layers are also described in Figure 3 for the PCB design concept defined in this study. Here, the layer stackup is defined in a manner that targets a low signal IL within the PCB MDI for the 25Gbps differential pair traces. The dielectric materials used are designated in Figure 3 as *Dielectric1* through *Dielectric 7*. For three separate 25Gbps PCB designs, a single dielectric material (1 through 7) was selected with focus on a low loss tangent as an essential parameter to ensure low signal IL.

Table 2 shows the dielectric materials used within this study based on the corresponding loss tangents at 10GHz.

Table 2: Overview of the dielectric materials and loss tangents for 25Gbps PCB MDI design.

Dielectric Material	Loss Tangent, @10GHz
Rogers (RO4003)	0.0027
Megtron6	0.0040
EM827(I)	0.0280

Figure 4 provides an overview of the implemented ECU PCB design for this study. Multiple MDI differential pair traces are implemented on a single PCB. All implemented PCB designs originate from a single PCB panel.

The purpose of including the SubMiniature version A (SMA) ports is described in Section 4.

ECU PCB MDI: this segment of the PCB is defined in consideration of the overall ECU PCB component spacing requirements. The MDI differential pair trace lengths are typically designed in the range between ~24mm through ~28mm. In this study two differential pair trace lengths have been defined. One being ~27.05mm emulating an ECU PCB MDI case, and a longer trace length of 150mm. Generally, the higher the trace length the higher the signal IL (The Sierra Circuits Team, 2021). Hence, the longer trace length aids in further analyzing the MDI IL

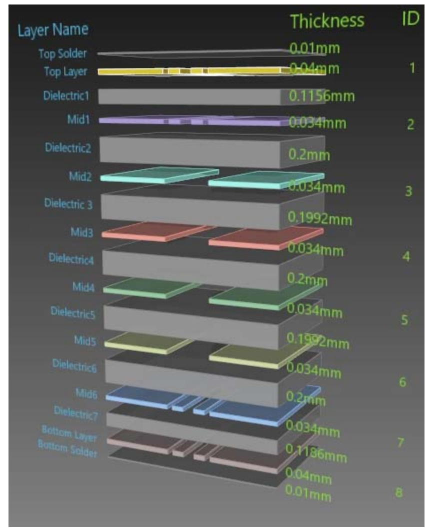


Figure 3: Defined ECU PCB layer stackup from a design tool.

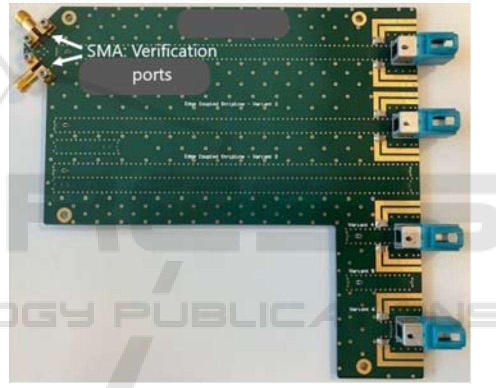


Figure 4: Exemplary of the multiple implemented ECU PCB MDI design test boards.

Figure 5: PCB MDI top layer description.

SCITEPRESS SCIENCE AND TECHNOLOGY PUBLICATIONS

properties. On each differential pair trace, an Alternating Current (AC) coupling capacitors of 100nF are included. A capacitive shield plane to ground termination is also considered in the MDI design. Refer to Figure 5 for a top layer description of the PCB layout.

Furthermore, this study defines two MDI differential trace layout structures to emulate typical options of an ECU PCB design. These trace structures are described in Figure 6 as Edge-Coupled Embedded Microstrip (Str. A) and Edge-Coupled Stripline (Str. B).

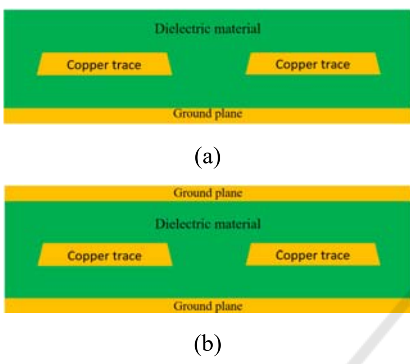


Figure 6: ECU PCB MDI trace structures: (a) Edge-Coupled Embedded Microstrip and (b) Edge-Coupled Stripline.

The MDI differential pair trace width and spacing implemented vary with the dielectric material used as well as the trace structure. The trace width considered is 114µm and 133µm for Str. A and 101µm and 103µm for Str. B. As for the differential pair trace spacing, this is 135µm, 130µm, and 150µm for Str. A and 205µm, 210µm, and 230µm for Str. B. All differential pair traces have a 100Ω impedance with a tolerance of ±10%.

4 MDI IL CHARACTERIZATION

This section is an extension of the analysis conducted in (Borda J. J., 2022). Within this study, three MDI IL characterization criteria for a 25Gbps Ethernet transmission are defined. These criteria are based on (1) the dielectric material (also according to the loss tangent), (2) the differential trace structure and (3) the differential trace length. MDI IL characterization is conducted using the ECU PCB design concept in accordance with the description in Section 3. For this, mixed-mode S-Parameter test bench measurements were conducted at room temperature conditions using a 40GHz four-port Vector Network Analyzer (VNA). The measurements were conducted with a start frequency f_{start} of 1MHz and linear steps f_{steps} of 1MHz (10000 linear points).

With the specified Nyquist frequency of 25Gbps being at 7.03125GHz, the corresponding test stop frequency f_{stop} is set to 10GHz. IEEE 802.3cy specifies MDI IL limits of up to 9GHz for a 25Gbps Ethernet channel. For this study, the VNA is set for an output power of -5dBm and Intermediate Frequency (IF) Bandwidth (BW) to 1kHz (Borda J. J., 2022) (IEEE 802.3cy (Unpublished), 2022).

As described in the test setup of the preceding study in (Borda J. J., 2022), the measurements are performed by setting 0.8mm pitch differential precision test probes placed directly on the PHY BGA pads (set I). For this study, the other end of the test probes (set II) has a physical contact on the side of the ECU PCB header connector pins (also refer to Figure 7). Both sides of the precision test probe cables are then directly connected to a 40GHz VNA. The specified IL for the precision differential test probes used is less than ~3dB. However, de-embedding of these test probes is performed also considering pin-test probe contact mismatch (and probe length) (Borda J. J., 2022).

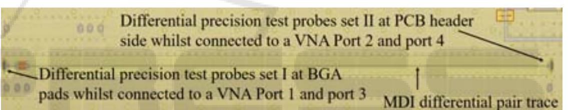
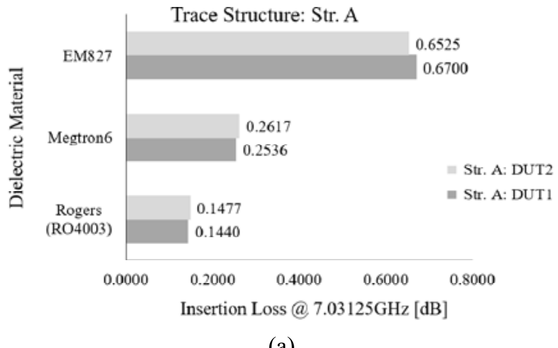


Figure 7: Positioning of the differential precision test probes on the MDI.

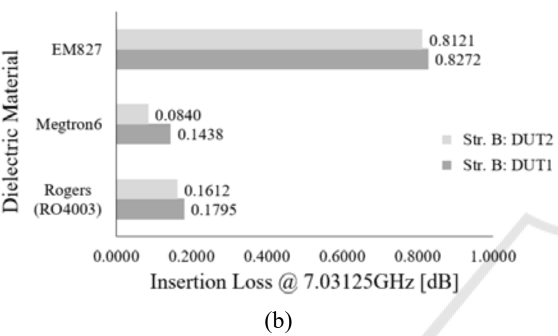
Additional verification measurements are conducted using implemented SMA verification ports (refer to Figure 4). These measurements serve the purpose to validate the test conducted using the precision differential test probes.

4.1 PCB Dielectric Material

Three variants of the PCB MDI test board designs were implemented using the three different dielectric materials defined in Table 2. To ensure measurement reproducibility, for each of these PCB MDI variants, two DUTs were considered for the analysis. Additionally, the two differential pair trace structures Str. A and Str. B were considered separately on each of the three PCB variants. Figure 8 displays the outcome of the MDI IL at 7.03125GHz from the three PCB variants with Str. A and Str. B on a ~27.05mm differential pair trace length.



(a)



(b)

Figure 8: Influence of PCB dielectric material on MDI IL: (a) Trace Str. A and (b) trace Str. B.

In Figure 8 MDI IL variations between DUTs are dependent on the dielectric material used and vary between the values ~0.0175dB, ~0.0081dB, and ~0.0037dB. These variations between DUTs are attributed to (1) PCB fabrication tolerances and (2) tolerances in contacting the precision test probes onto the PCB test regions. The influence of dielectric material on MDI IL is dominated with EM827 followed by Megtron6 and lastly RO4003. These observations are explained by their corresponding loss tangents ($D_{f,EM827} > D_{f,Megtron6} > D_{f,RO4003}$) and their impact on the dielectric losses. The maximum MDI IL in Figure 8 is based on EM927 with Str. B. Megtron6 and RO4003 exhibit lower MDI ILs with a dependency on the differential pair trace structure deployed. This dependency is further discussed in Section 4.2.

4.2 MDI Differential Pair Trace Structure

Based on the outcome of the three dielectric materials on the MDI IL discussed in Section 4.1, this section characterizes the individual impact of the differential pair trace structures Str. A and Str. B on MDI IL using the shorter trace lengths. Here, the maximum (max.) MDI IL from each pair of DUTs for the same trace structure is considered. These maximum MDI IL values are summarized in Table 3 and Figure 9.

Table 3: Overview max. MDI IL based on varying MDI differential pair trace structures.

Dielectric Material	Max. DUT IL @ 7.03125GHz [dB]	
	Str. A	Str. B
Rogers (RO4003)	0.147	0.1795
Megtron6	0.2617	0.1438
EM827	0.6700	0.8272

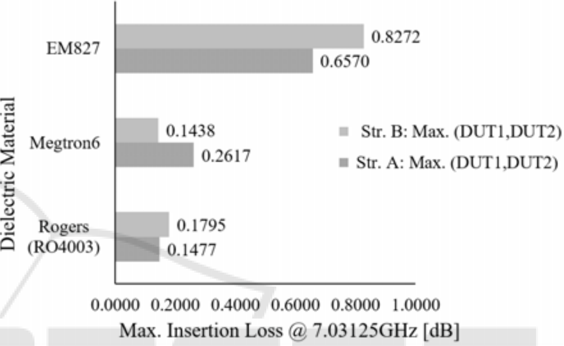


Figure 9: Impact of MDI differential pair trace structures on IL.

In terms of the impact of the differential pair trace structures and independent of the dielectric material used, in two cases Str. B exhibits higher MDI IL values compared to Str. A. This is however with an exception when Megtron6 dielectric material is considered on the PCB design. The higher MDI IL on Str. B is associated with conductor losses regarding copper surface roughness with no radiation losses. And in the case of an optimum 25Gbps Ethernet PCB design, Str. B trace structure would additionally have a copper plane to protect the differential trace pair signal conductors from undesired signal crosstalk. With this, Str. B is better suited for a 25Gps Ethernet channel compared to Str. A. Furthermore, the maximum MDI IL (0.872dB) for Str. B on a 27.07mm trace length lies within the specified limits (< 1.8717 per 76.2mm trace) for a 25Gbps MDI.

4.3 MDI Differential Pair Trace Length

Placement options for various required circuit components are typically very limited in terms of available space on the ECU PCB. This leads to cases

were variations in MDI differential trace lengths between different ECU implementations is expected.

This subsection characterizes the impact of the MDI differential trace length on the MDI IL. Table 4 and Table 5 lists down the maximum measured MDI IL at 7.03125GHz on the 150mm trace length based on the two previously defined differential pair trace structures (Str. A and Str. B) and the three dielectric materials implemented in the design concept as per Section 3.

Table 4: Max. MDI IL comparison at 7.03125GHz: Short versus long differential pair trace for Str. A.

Dielectric Material	Max. MDI IL [dB]	
	Short Trace Length (27.05mm)	Long Trace Length (150mm)
Rogers (RO4003)	0.1477	2.9620
Megtron6	0.2617	2.341
EM827	0.6570	7.2932

Table 5: Max. MDI IL comparison: Short versus long differential pair trace for Str. B.

Dielectric Material	Max. MDI IL [dB]	
	Short Trace Length (27.05mm)	Long Trace Length (150mm)
Rogers (RO4003)	0.1795	4.9277
Megtron6	0.1438	3.5855
EM827	0.8272	7.1115

The maximum MDI IL values in Table 4 and Table 5 are based on measurements conducted on two DUTs of similar design type. Here, a comparison with the corresponding implemented shorter differential pair trace length (~27.05mm) is conducted. A higher increase in MDI IL values is observed with Str. B due to having an even higher increase in conductor losses compared to Str. A (as addressed in Subsection 4.2).

With the maximum IEEE 802.3cy specified MDI IL being at 1.8717dB for a 76.02mm differential pair trace length, actual MDI design implementation tend to exhibit variations due to non-ideal PCB implementation aspects typically not considered in such specifications. Based on the designed ECU PCB MDI concept in this study, this subsection further investigates the maximum possible MDI differential pair trace length in correlation with the existing MDI IL limit of 1.8717dB for a 25Gbps MDI. For this characterization the following analysis approach is defined within this study:

(1) At 7.03125GHz, for each PCB MDI design variant with the three different dielectric materials, the maximum MDI ILs based on the two DUTs is plotted against the two corresponding differential pair trace lengths (27.05mm and 150mm).
(2) From (1) the graph trend line and the governing equation is determined.
(3) The equations in (2) are then used to determine additional maximum MDI ILs for various differential pair trace lengths considering the limit of 1.8717dB.
(4) The step in (3) is conducted for each PCB MDI design variant and trace structure type separately.
(5) From (4) a trend graph is determined on how the MDI IL would vary dependent on how the trace length will be chosen. It also determines the trace length options implementable to ensure compliance to the limit of 1.8717dB.

As an outcome of the aforementioned analysis and based on step (3), the governing equations for the MDI ILs exhibit an exponential functional trend, $IL = Ce^{tracelengthvariable}$ with C being the function coefficient. For Equation (4) through Equation (9), the decimal function coefficients are rounded up to one significant figure.

Figure 10 provides the range of the differential pair trace lengths for an optimum 25Gbps MDI IL with Rogers (RO4003) as the dielectric material. The governing equations are described in Equation (4) for Str. A and Equation (5) for Str. B. Here, T_l stands for differential pair trace length.

$$IL_{MDI,Str.\mathbf{A},RO4003}(T_l) = 0.077e^{0.0244(T_l)}\ [dB] \quad (4)$$

$$IL_{MDI,Str.\mathbf{B},RO4003}(T_l) = 0.087e^{0.0269(T_l)}\ [dB] \quad (5)$$

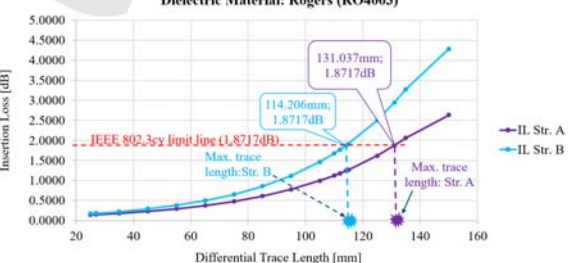


Figure 10: PCB MDI design variant with RO4003: Range of MDI IL limit compliant differential pair trace lengths.

Figure 11 provides the range of the differential pair trace lengths for an optimum 25Gbps MDI IL for PCB MDI design variant with Megtron6 (Meg6) as the dielectric material. The governing trend equations are described in Equation (6) for Str. A and Equation (7) for Str. B.

$$IL_{MDI,Str.\mathbf{A},Meg6}(T_l) = 0.162e^{0.0178(T_l)}\ [dB] \quad (6)$$

$$IL_{MDI,Str.\mathbf{B},Meg6}(T_l) = 0.071e^{0.0261(T_l)}\ [dB] \quad (7)$$

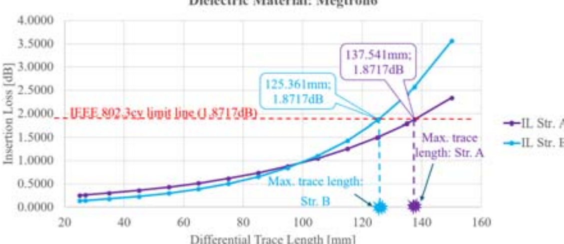


Figure 11: PCB MDI design variant with Megtron6: Range of MDI IL limit compliant differential pair trace lengths.

Figure 12Figure 11 provides the range of the differential pair trace lengths for an optimum 25Gbps MDI IL for PCB MDI design variant with EM827 as the last dielectric material. The governing trend equations are described in Equation (8) for Str. A and Equation (9) for Str. B.

$$IL_{MDI,Str.\mathbf{A},EM827}(T_l) = 0.387e^{0.0196(T_l)}\ [dB] \quad (8)$$

$$IL_{MDI,Str.\mathbf{B},EM827}(T_l) = 0.515e^{0.017(T_l)}\ [dB] \quad (9)$$

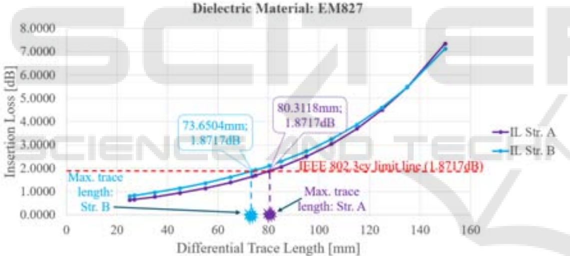


Figure 12: PCB MDI design variant with EM827: Range of MDI IL limit compliant differential pair trace lengths.

Table 6 summarizes the various maximum MDI IL limit compliant differential pair trace lengths for all the PCB MDI design variants defined within this study. A trace length greater 76.02mm trace length is implementable for Rogers and Megtron6 dielectric materials. However, on the other hand, EM824 based PCB MDI design variant exhibits significant lower maximum trace lengths, with 80.3118mm for Str. A and 73.6504mm for Str. B trace structures. This is attributed to (1) the higher loss tangent of this dielectric material compared to the remaining two and consequently (2) the higher MDI IL that was already obtained at a shorter trace length of 27.05mm.

Table 6: 25Gbps MDI IL limit compliant maximum differential pair trace length.

Dielectric Material	Max. trace length for MDI IL < 1.8717dB at 7.03125GHz [mm]	
	Str. A	Str. B
Rogers (RO4003)	131.037	114.206
Megtron6	137.541	125.361
EM827	80.3118	73.6504

4.4 Worst- and Best-Case MDI PCB Design

Based on the characterization results in Sections 4.1, 4.2, and 4.3, this subsection defines and presents the Worst-Case (WC) and Best Case (BC) ECU PCB MDI design for a 25Gbps Ethernet communication channel. Table 7 provides an overview.

Table 7: WC and BC ECU PCB MDI design parameters: According to 25Gbps PCB MDI IL limit of 1.8717dB at 7.03125GHz.

Design Parameter	Design for max. 1.8717dB MDI IL at 7.03125GHz		
	WC	BC	
Trace Structure	Str. B	Str. A	Str. B
Dielectric Material	EM827	Rogers (RO4003)	Megtron6
Trace Length [mm]	73.6504	131.037	125.361

In Table 7, the two BC ECU PCB MDI designs provide more options for design variations. In this case both Str. A and Str. B can be considered with more flexibility in terms of the differential pair trace length to be implemented. Though, application based, mostly the trace lengths would be around ~28mm and seldom beyond this value. As characterized in previous subsections of this study, the WC ECU PCB MDI design has a disadvantage in terms of the higher MDI IL it exhibits even at shorter trace lengths of 27.05mm.

Overall, a dielectric material with a low loss tangent (equal or less 0.0027) is recommended for a 25Gbps Automotive Ethernet MDI to attain an optimum BC MDI IL similar to the characterized cases for Rogers (RO4003) and Megtron6. As for the differential pair trace structure selection, this will depend on whether one goes for Roger (RO4003) or Megtron6 design variant. Nevertheless, Str. B remains preferable over Str. A due to its optimum overall high-speed characteristics. (Coonrod , 2013).

SCITEPRESS SCIENCE AND TECHNOLOGY PUBLICATIONS

5 SIMULATIONS FOR ECU PCB MDI IL CHARACTERIZATION

While test bench measurements are the standard means for communication channel PHY validation and qualification of "*lower*" (<10Gbps) data rates in the automotive industry, reliable and meaningful channel PHY simulations are not at all used at the car manufacturers for this purpose, yet.

There are several existing and expected challenges in terms of system level PHY test bench validations and qualifications for beyond 10Gbps in- vehicle connectivity. These primarily include (1) tolerances in contacting the precision differential test probes onto the PCB contact pads, (2) increased costs on the required test infrastructure which includes DUTs as well as test setup equipment, (3) more time required to conduct the tests especially if multiple DUTs have to be analyzed, and lastly (4) increased measurement artifacts for high-speed signals considering the steeper rise/fall signal timing requirements and the impact of the additional capacitance from the test probes on SI.

It is therefore imperative to conduct a feasibility analysis for an alternative approach to validate and qualify not only the PHY channel as a whole but within this study, the ECU PCB MDI design. An alternative aspect to be discussed is therefore the usage of simulations in place of test bench measurements. Essentially, this analysis focuses on what simulations can cover and whether a full replacement or coexistence with test bench measurements is expected/reasonable within the Automotive industry.

For this characterization, mixed-mode S-Parameter analysis of the MDI differential pair trace is performed using ANSYS SIwave and HFSS 3D. Here, the same PCB MDI designs from measurements in Section 4 are deployed.

5.1 Ports and Reference Plane

The initial part of the analysis defines the reference plane (also serving as the current return path) and ports for the differential pair traces. To emulate actual test bench measurement setup, the MDI simulation ports are defined at the Top Layer (TL). The corresponding reference ports are set at the mid 2 Layer (mid2). With the preference of a least impedance (i.e., least inductance) path for high frequency transmission, mid2 also serves as the current return path for both Str. A and Str. B (Archambeault, 2008).

In addition to an optimum selection of the MDI AC coupling capacitance value, the Equivalent Series Resistance (ESR) and Equivalent Series Inductance (ESL) values play a key role. These values exhibit ideal characteristics in a simulation compared to test bench measurements. This is due to the actual occurrence of the capacitor component tolerances on actual test bench measurements. The ESR and ESL are also influenced by the non-ideal characteristics of the PCB layout, for instance, trace parameters and dielectric constant variations (Borda J. J., 2022). The study in (Borda J. J., 2022) provides a further description of how the MDI port definition is conducted for the simulation.

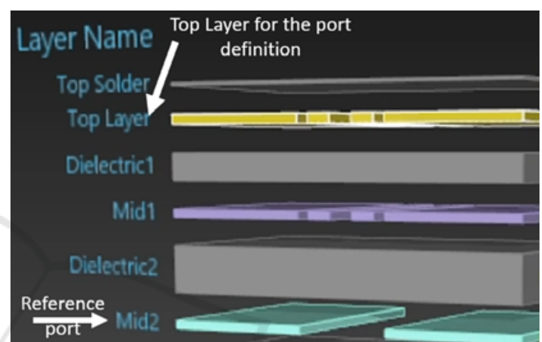


Figure 13: PCB layer selection for port and reference.

5.2 Correlation Between Measurements and Simulations

As there are numerous variables to consider when looking at the correlation between MDI IL test measurements versus simulation, the question remains, what is reasonable to compare between these two. Figure 14 displays the MDI IL correlation between test bench measurements and simulations for Str. A and Str. B for a 27.05mm differential pair trace.

From Figure 14, independent of the dielectric material used, MDI IL deviations between simulations and measurements are observed. These deviations vary between a minimum of 0.179dB and a maximum of 0.8290dB.

Similar to test bench measurements, MDI IL from simulations also exhibits an increase in value in this particular order based on the dielectric material Rogers (RO4003), Megtron6, and EM827. The reason behind this trend was discussed in Section 4.1. Table 8 and Table 9 provide an overview of the MDI IL between simulations and measurements.

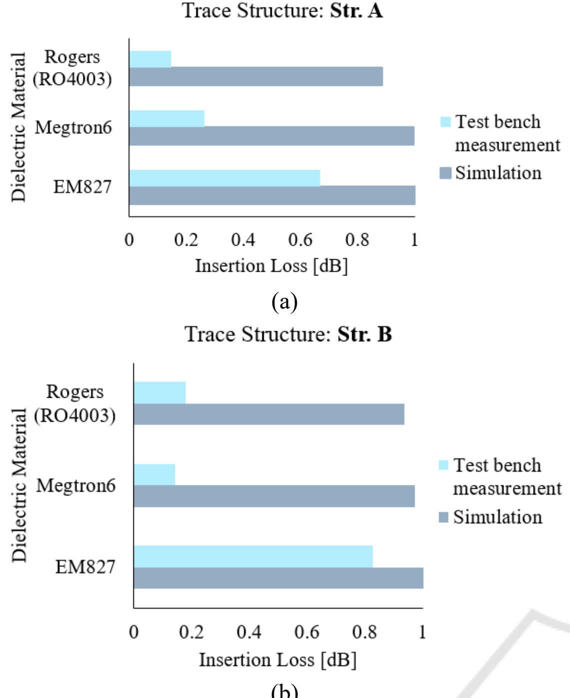


Figure 14: PCB MDI IL trend correlation of measurements versus simulations: (a) Trace Str. A and (b) trace Str. B.

Table 8: MDI IL: Simulation versus measurements for Str. A.

Dielectric Material	Simulation versus Measurements, MDI IL Str. A [dB]	
	Simulations	Measurements
Rogers (RO4003)	0.8844	0.1470
Megtron6	0.9938	0.2617
EM827	1.0154	0.6700

Table 9: MDI IL: Simulation versus measurements for Str. B.

Dielectric Material	Simulation versus Measurements, MDI IL Str. B [dB]	
	Simulations	Measurements
Rogers (RO4003)	0.9369	0.1795
Megtron6	0.9728	0.1438
EM827	1.0063	0.8272

A higher valued MDI IL from measurements leads to an improved correlation to simulation results. Exemplary to this is with EM827. Whereas MDI IL based on Rogers and Megtron6 exhibit high deviations between simulations and measurements (refer to Table 8 and Table 9).

The deviations in the observed MDI IL between simulations and measurements can be attributed to numerous factors. Primarily these factors include: (1) Capacitance ESR and ESL variations between the simulation model (ideal case) and actual non-ideal capacitance used on the PCB MDI boards, (2) tolerances in contacting measurement test probes versus simulation port placement, (3) reference ground used on the simulations can deviate to that automatically used on measurements. On measurements this is freely determined by a low impedance path and not direct test probe placement. The definition of an optimum reference ground addresses the most crucial part of both simulation and measurements. This is even more the case when dealing with Edge-Coupled Embedded Microstrip trace structures where the vias and transitions between multiple layers of the PCB have to be considered.

Can simulations be deployed for 25Gpbs MDI IL characterization? A general answer to this question is, yes, however, the following conditions have to be considered:

(1) Means to consider non-ideal characteristics of the passive MDI components have to be modelled as a standalone pre-analysis.
(2) Methodologies to correlate a test bench versus simulation reference ground definition throughout a trace structure have to be defined.
(3) Baseline test bench measurements serve as pre-requisite (i.e., coexistence).
(4) Only a non-quantitative comparison in terms of the MDI IL trend between different dielectric materials and design variants provides a reasonable simulation analysis.
(5) Recommendable for designs with higher loss tangent range based dielectric materials.
(6) With (3), MDI IL simulations are reasonable for longer (≥70mm) differential pair trace lengths. In this case, the high-speed properties become even more visible.

6 CONCLUSION

For 25Gbps Automotive Ethernet connectivity, this study has defined and characterized several essential ECU PCB MDI aspects:

(1) Design concept for the PCB MDI.
(2) ECU PCB MDI IL budget characterization using RF test bench measurements on the design concepts based in (1).
(3) Feasibility of a simulation approach in coexistence with RF test bench measurements for MDI IL characterization.

SCITEPRESS SCIENCE AND TECHNOLOGY PUBLICATIONS

This study provides the baseline requirements and specifications for the ECU PCB MDI system design and implementation for 25Gbps in-vehicle connectivity. Furthermore, this study serves as an essential framework towards Automotive industry standardization projects in regard to ECU PCB MDI design and RF characterization as well as qualification. Future studies are planned to characterize signal attenuation in relation to crosstalk on high densely populated 25Gbps ECU PCB MDI design.

REFERENCES

Archambeault, B. (2008). *Design Tip: Resistive versus Inductive Return Current Paths, IEEE.* Retrieved from https://www.ewh.ieee.org/soc/emcs/acstrial/newsletters/spring08/design_tips.pdf

Borda, J. (2022). Which is more Suitable for Next Generation Automotive Ethernet Connectivity: Copper or Optical Fiber? *Automotive Ethernet Congress* . Retrieved 06 2022

Borda, J. J. (2022). Beyond 10Gbps Electrical Automotive Ethernet Channel Insertion. *IEEE Intelligent Vehicles Symposium.* Aachen, Germany.

Coonrod , J. (2013). *Insertion Loss Comparisons of Common High Frequency PCB Constructions.* Retrieved January 2022, from Rogers Cooperation: https://www.circuitinsight.com/pdf/insertion_loss_comparisons_pcb_constructions_ipc.pdf

Fan, W., Lu, A., Wai, L., & Lok, B. (2003). Mixed-Mode S-Parameter Characterization of Differential Structures. *IEEE XPlore*, 1,2,3. Retrieved January 23, 2022

IEEE 802.3ch. (2020). IEEE P802.3ch™: 2020, Standard for Ethernet Amendment: Physical Layer Specifications and Management Parameters for 2.5 Gb/s, 5 Gb/s, and 10 Gb/s Automotive Electrical Ethernet. IEEE 802.3ch. Retrieved June 2022

IEEE 802.3cy (Unpublished). (2022, 07 01). *IEEE P802.3cy™/D2.0 Draft Standard for Ethernet Amendment 9:Physical Layer Specifications and Management Parameters for 25 Gb/s Electrical Automotive Ethernet.* Retrieved 08 2022, from https://www.ieee802.org/3/cy/private/802.3cy_D2p0.pdf

Johnson, H., & Graham, M. (1993). *High-Speed Digital Design, A Handbook of Black Magic.* Prentice Hall PTR.

Knack, K. (2020, 10 06). *https://resources.altium.com/.* Retrieved from https://resources.altium.com/p/increasingly-important-role-loss-tangents-pcb-laminates

Mittal, A. (2021, March 8). Losses in PCB Transmission Lines,https://www.protoexpress.com/blog/losses-in-pcb-transmission-lines/. Retrieved January 23, 2022, from https://www.protoexpress.com/blog/losses-in-pcb-transmission-lines/

OPEN Alliance. (2020). 1000BASE-T1 EMC Measurement Specification for Transceivers v2.1. Retrieved 06 2022

OPEN Alliance. (2020). TC9, Channel and Components Requirements for 1000BASE-T1 Link Segment Type A (STP) v2.0. Retrieved 06 2022

OPEN Alliance. (2020). 1000BASE-T1 EMC Measurement Specification for Transceivers v2.1. Retrieved 06 2022

OPEN Alliance. (2022). TC9 (Unpublished) Channel and Component Requirements for Fully Shielded 1000BASE-T1 and 2.5G/5G/10GBASE-T1 Link Segments v0.7. Retrieved June 2022

Polar Instruments. (2022). *AP8155 , https:// www.polarinstruments.com/support/si/AP8155.html.* Retrieved January 2022, from https://www.polarinstruments.com/support/si/AP8155.html

The Sierra Circuits Team. (2021, 06 14). *How to Reduce Signal Attenuation in Hig-Speed PCBs.* Retrieved from https://www.protoexpress.com/: https://www.protoexpress.com/blog/how-to-reduce-signal-attenuation-high-speed-pcbs/

SCITEPRESS SCIENCE AND TECHNOLOGY PUBLICATIONS