What is Differential Impedance and Why do We Care?

Bert Simonovich

12/29/2024

Table 1 Release Control Record[1]:

Issue 01	April 14, 2020	Originally published Signal Integrity Journal
Issue 02	Dec. 29, 2024	Minor typo correction per change bar

[1] This document is an uncontrolled release. Latest issue can be obtained at LamsimEnterprises.com

What is Differential Impedance and Why do We Care?

What is differential impedance and why do we care? Simply put, differential impedance is the instantaneous impedance of a pair of transmission lines when two complimentary signals are transmitted with opposite polarity. For a printed circuit board (PCB) this is a pair of traces; also known as a differential pair. We care about maintaining the same differential impedance for the same reason we care about maintaining the same instantaneous impedance of a single-ended (SE) transmission line. That is to avoid reflections.

There is really nothing special about differential pairs, other than maintaining the correct differential impedance. But you must understand the implications of the spacing between the traces in a pair.

The differential impedance is simply twice the odd-mode impedance of each trace. SE impedance is the impedance of a single trace and only equals the odd-mode impedance when there is little or no intra-pair coupling between them. When the traces are brought closer together, the differential impedance is reduced, unless the line widths are adjusted to compensate. More about this later.

Figure 1 illustrates the effect on intra-pair coupling of a pair of edge-coupled stripline traces driven differentially. The top figure shows electro-magnetic fields surrounding a loosely coupled pair of traces 3.5 line-widths apart. The bottom figure shows a closely coupled pair at 1.5 line-widths apart. The red plus trace is current flowing into the page while the minus blue trace is current flowing out of the page.

The circular lines surrounding each trace are the magnetic fields representing loop inductance. The direction of rotation is based on current direction, using the right-hand rule. The electric field (e-field) lines are perpendicular to the magnetic field lines. They are a measure of capacitance.

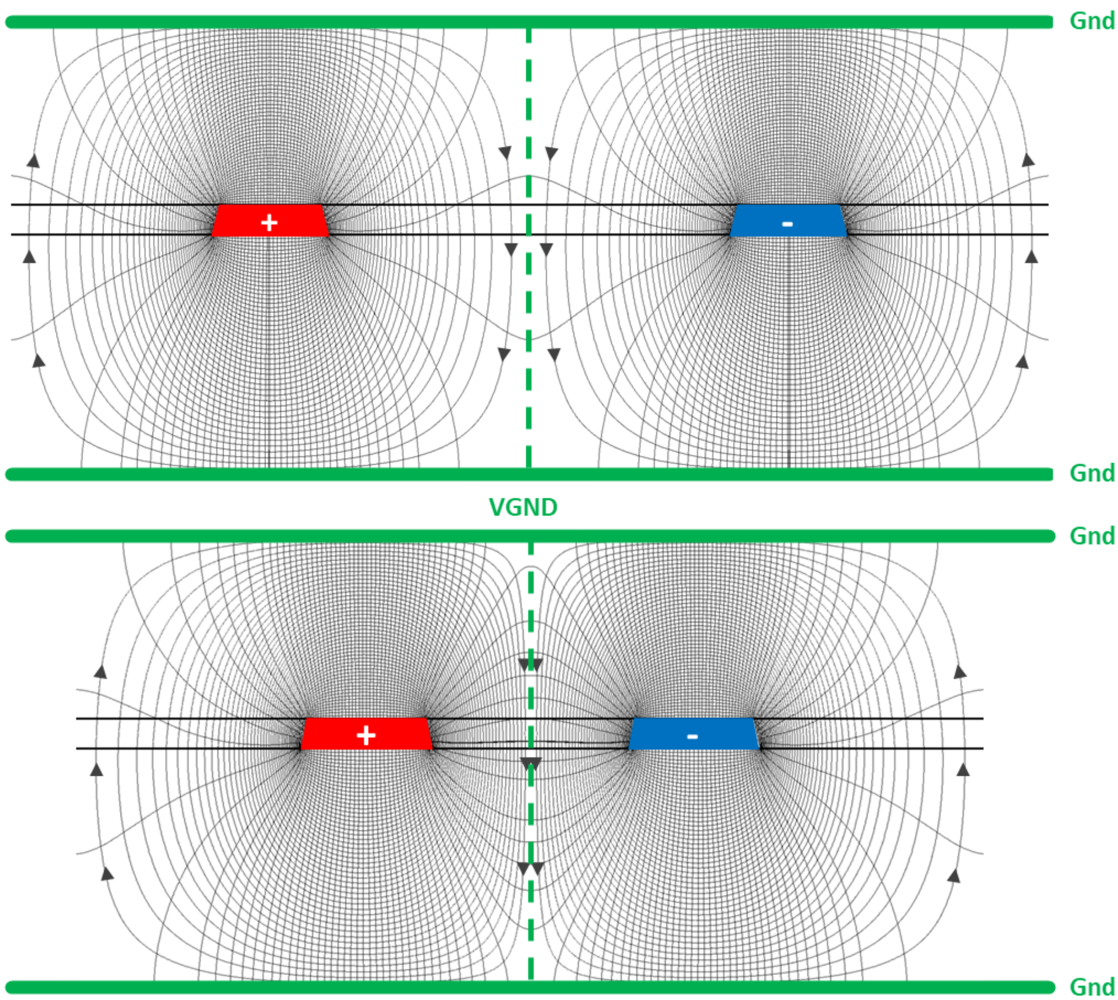


Figure 1 Effect on intra-pair coupling of a pair of edge-coupled stripline traces driven differentially. Top figure shows electro-magnetic fields surrounding a loosely coupled pair of traces 3.5 line-widths apart. Bottom figure shows a closely coupled pair at 1.5 line-widths apart.

When the traces are loosely coupled, the electric and magnetic field lines are fairly symmetrical around each trace, and are mirror images of one another about the center line between them. Most of the respective e-field coupling is to the reference ground planes. As the traces are moved closer to one another, the counter-rotating rings compress about the centerline, lowering the inductance. At the same time, more of the e-field lines along the inside edge of each trace tend to couple to one another, increasing the capacitance.

Because of the way the EM-fields interact along the centerline, we can think of it as a virtual ground (VGND) reference plane. They behave exactly the same way as if there is a solid reference plane between them.

Odd-Mode Impedance

Consider a pair of equal width microstrip line traces, labeled 1 and 2, with a constant spacing between them as shown in Figure 2. Assuming lossless transmission lines, each individual trace, when driven in isolation, will have a SE characteristic impedance *Zo,* defined by the self-loop inductance (L11, L22) and self-capacitance (C11, C22) with respect to the GND reference plane.

When the pair of traces are driven differentially, the mode of propagation is odd. The electromagnetic field interaction is shown in Figure 1. When the intra-pair spacing is close, there will be electromagnetic coupling defined by the mutual inductance (Lm) and mutual capacitance (Cm).

The proximity of the traces to a reference plane influences the amount of electromagnetic coupling between traces. The closer the traces are to the reference plane, the lower the self-loop inductance and stronger self-capacitance; resulting in a lower mutual inductance, and weaker mutual capacitance between traces. The end result is a lower differential impedance.

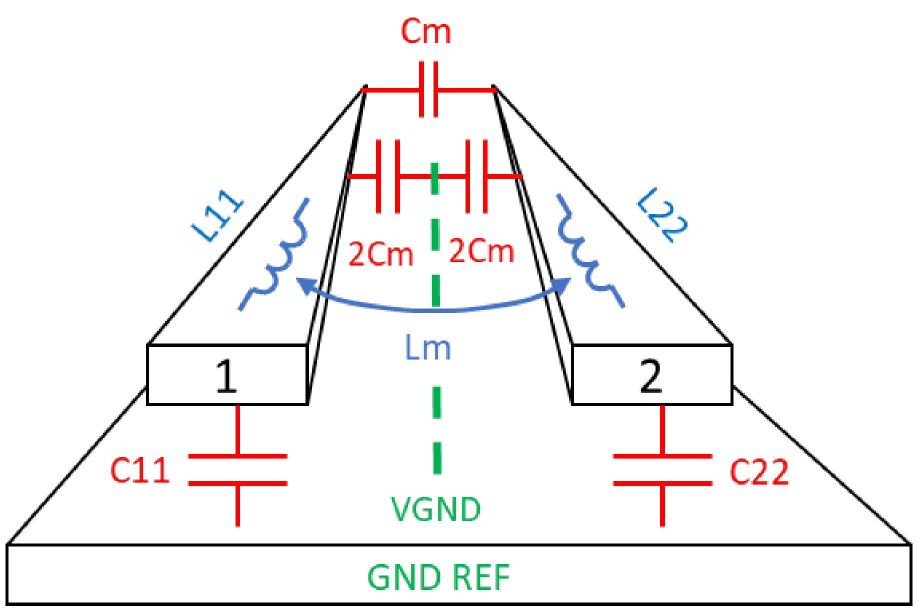


Figure 2 Pair of microstrip traces showing self-loop inductance (L11, L22), self-capacitance (C11, C22), mutual capacitance (Cm) and mutual inductance (Lm) when line 1 and line 2 are driven differentially.

A 2D field solver is usually used to extract the parameters for a given geometry. Once the resistance, inductance, conductance and capacitance (RLGC) parameters are extracted, an L C matrix can be set up as follows:

$$\begin{matrix} \mathbf{L11} & \mathbf{L12} \\ \mathbf{L21} & \mathbf{L22} \end{matrix} \qquad \begin{matrix} \mathbf{C11} & \mathbf{C12} \\ \mathbf{C21} & \mathbf{C22} \end{matrix}$$

The self-loop inductance and self-capacitance for trace 1 and 2 are L11, C11, L22, C22 respectively. In a perfectly symmetrical differential pair, the off-diagonal (12, 21) terms in each matrix are the mutual inductance and mutual capacitance respectively. The LC matrix can be used to determine the odd-mode impedance. It can be calculated by the following equation [1]:

Equation 1

$$Z_{odd} = \sqrt{\frac{L_s - L_m}{(C_s + 2C_m)}}$$

Where:

Z_{odd} = odd mode impedance

L_s = self-loop inductance = $L_{11} = L_{22}$

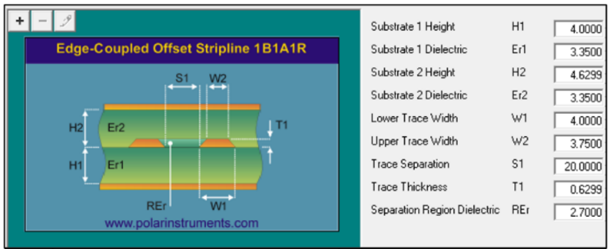
C_s = self-capacitance = $C_{11} = C_{22}$

L_m = mutual inductance = $L_{12} = L_{21}$

C_m = mutual capacitance = $|C_{12}|=|C_{21}|$

Example

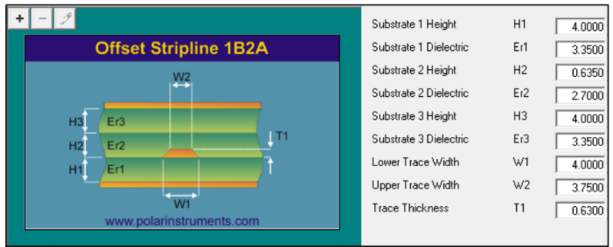
Polar SI9000 field solver is used to compare a loosely coupled pair, with 4 mil traces, separated by 20 mil space, vs a single ended transmission line with the same dielectric thickness, shown in Figure 3. The LC matrix was extracted at 10GHz. As can be seen, the odd-mode impedance of the loosely coupled pair equals the characteristic impedance of the SE trace, and thus differential impedance would be the same.



	L		C	
	1	**2**	**1**	**2**
1	7.81E-09	2.00E-12	3.08E-12	6.65E-16
2	2.00E-12	7.81E-09	6.65E-16	3.08E-12

$$Z_{odd} = \sqrt{\frac{L_s - L_m}{(C_s + 2C_m)}} = \sqrt{\frac{7.81\text{E-}9 - 2\text{E-}12}{(3.08\text{E-}12 + 2(6.65\text{E-}16))}} = 50.34\Omega$$

$$Z_{diff} = 2 \times 50.34 = 100.68\Omega$$



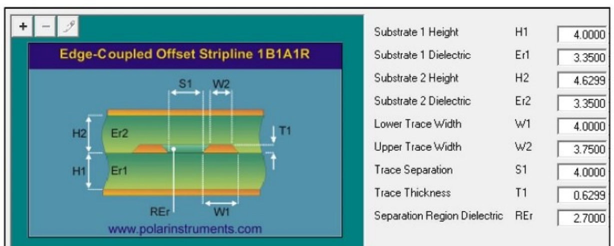
	L		C	
	1	**2**	**1**	**2**
1	7.82E-09	0	3.05E-12	0
2	0	7.81E-09	0	3.05E-12

$$Z_O = \sqrt{\frac{L_s}{C_s}} = \sqrt{\frac{7.81\text{E-}9}{3.08\text{E-}12}} = 50.61\Omega$$

$$Z_{diff} = 2 \times 50.61 = 101.22\Omega$$

Figure 3 Comparison of a loosely coupled pair (left), with 4 mil traces, separated by 20 mil space, vs a single ended transmission line (right) with the same dielectric thickness. Odd-mode impedance of the loosely coupled pair equals the characteristic impedance of the SE trace

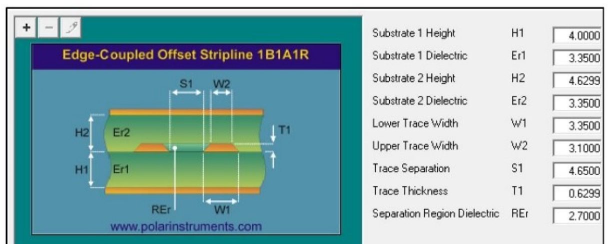
But if you route a pair of traces with close coupling, the odd-mode impedance is less than the SE impedance for the same trace width unless you adjust the line-width. For example, on the left side of **Figure 4**, a 4-4-4 mil geometry has a differential impedance of 91 Ohms. In order to get 100 Ohms differential, the line width must be reduced to 3.35 mils and space adjusted to 4.65 mils to keep the same 12 mil center-center pitch, shown on right.



	L		C	
	1	**2**	**1**	**2**
1	7.78E-09	7.28E-10	2.84E-12	2.76E-13
2	7.28E-10	7.78E-09	2.76E-13	2.84E-12

$$Z_{odd} = \sqrt{\frac{L_s - L_m}{(C_s + 2C_m)}} = \sqrt{\frac{7.78\text{E-}9 - 7.28\text{E-}10}{(2.84\text{E-}12 + 2(2.76\text{E-}13))}} = 45.56\Omega$$

$$Z_{diff} = 2 \times 45.56 = 91.12\Omega$$



	L		C	
	1	**2**	**1**	**2**
1	8.48E-09	6.71E-10	2.64E-12	2.13E-13
2	6.71E-10	8.48E-09	2.13E-13	2.64E-12

$$Z_{odd} = \sqrt{\frac{L_s - L_m}{(C_s + 2C_m)}} = \sqrt{\frac{8.48\text{E-}9 - 6.71\text{E-}10}{(2.64\text{E-}12 + 2(2.13\text{E-}13))}} = 50.45\Omega$$

$$Z_{diff} = 2 \times 50.45 = 100.90\Omega$$

Figure 4 Comparison of 4-4-4 mil geometry (left) vs 3.35-4.65-3.35 geometry (right) to achieve 100 Ohm differential impedance for the same center-center pitch.

But it doesn't end there.

For some industry standards, there is usually a very short reach (VSR) spec which has a maximum channel loss defined. For example, the IEEE 802.3 CAUI-4 chip-module (C2M) spec budgets 7.5 dB at 12.89 GHz Nyquist frequency from the chip's pins to a faceplate module's pins, e.g. small form-factor pluggable (SFP) module. Because of modern top-of-rack routers and switches, it is not unusual to have 10 or more inches between the main switch chip and SFP module, the differential pair geometry design becomes important to satisfy both differential impedance and IL.

Reduced line width and tighter coupling results in higher loss over the length of the channel. Using the above examples, differential IL is plotted in Figure 5 for all three differential pairs. Loose coupling is shown in green; tight coupling without line width adjustment (Tight1) is shown in red, while tight coupling with line width adjustment (Tight2) is shown in blue.

As you can see, there is about a half dB difference at 12.89 GHz between loose coupling and both tight coupling examples over 10.6 inches. Tight coupling lowers insertion loss (IL), regardless if line width is adjusted to meet differential impedance. In this example there is only 0.1 dB delta between Tight1 and Tight2, which suggests most of the higher loss is due to tighter coupling.

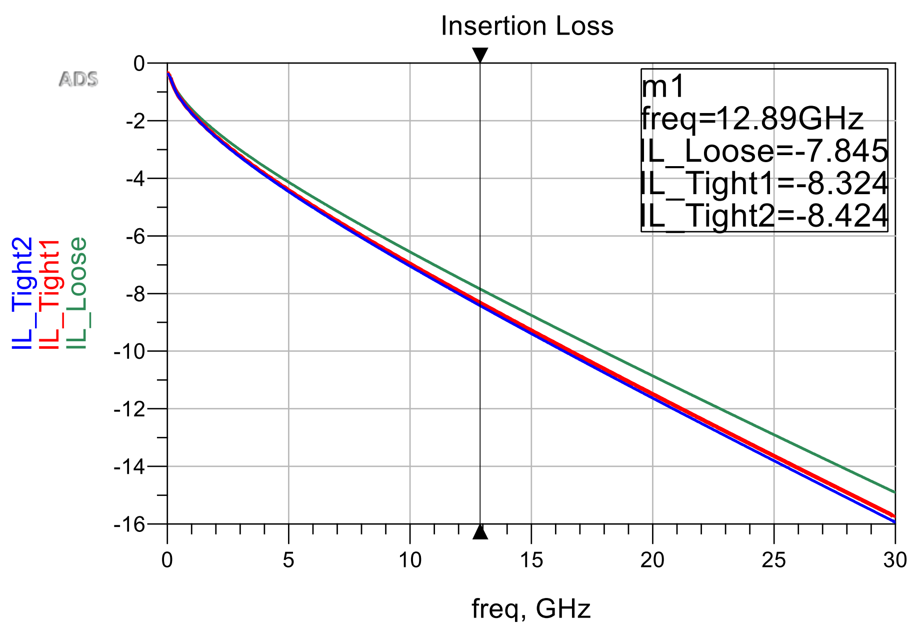


Figure 5 Differential IL comparison of loose coupling (green); Tight1 coupling without line width adjustments (red) and Tight2 coupling with line width adjustment (blue).

This can be explained by reviewing SE to differential mixed-mode conversion. Given a 4-port S-parameter, with SE port order as shown in Figure 6, the differential IL is determined by;

Equation 2

$$SDD21 = 0.5\left[S21 - S23 + S43 - S41\right]$$

Where:

SDD21 = the differential IL defined by the ratio of the differential signaling coming out of port 2 to the differential signal going into port 1

S21 = the SE IL defined by the ratio of the SE signaling coming out of port 2 to the SE signal going into port 1

S43 = the SE IL defined by the ratio of the SE signaling coming out of port 4 to the SE signal going into port 3

S23 = far-end crosstalk coupling from port 3 to port 2

S41 = far-end crosstalk coupling from port 1 to port 4

As you can see from **Equation 2**, when the traces get closer together, and the coupling terms get larger, differential IL increases.

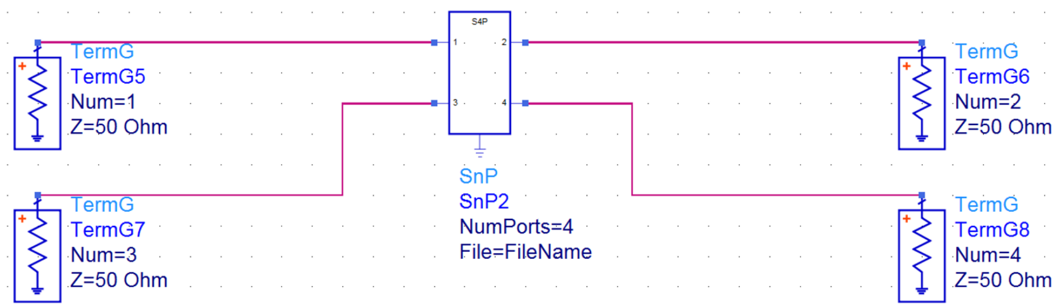


Figure 6 SE 4-port S-parameter port labeling.

Figure 7 plots differential TDR of all three examples. The steeper monotonic rise of the blue trace is due to higher resistive loss of 3.35 mil traces compared to the 4 mil traces in the other two examples.

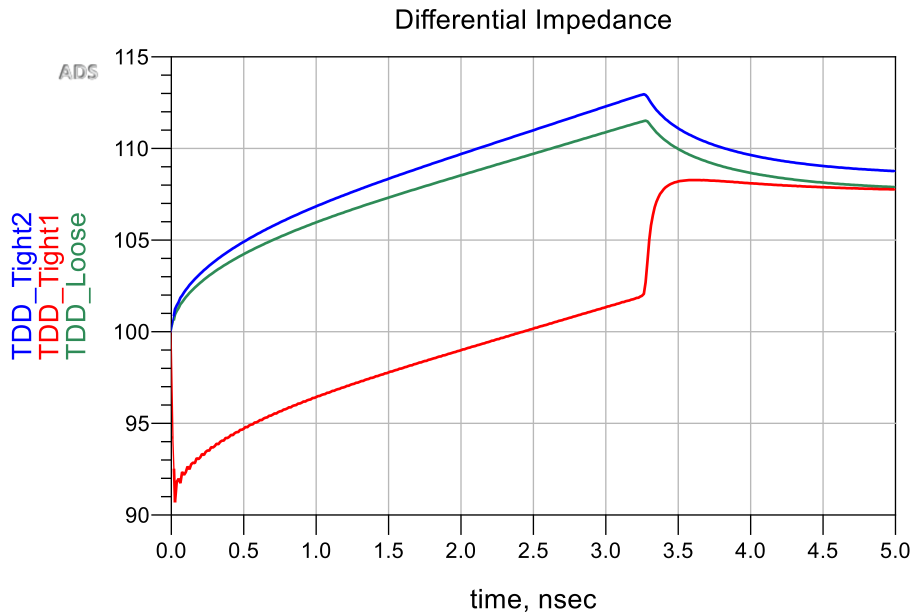


Figure 7 Differential TDR comparison of loose coupling (green); Tight1 coupling without line width adjustments (red) and Tight2 coupling with line width adjustment (blue).

To summarize then, it doesn't matter if a differential pair is tightly coupled or loosely coupled. Properly engineered, both can be designed to properly match the output driver impedance. But as we have seen, each will have advantages and disadvantages.

Tighter coupling gives you better routing density at the expense of higher loss. Loose coupling allows for easier routing around obstacles and less loss. But in either case, they must be designed and measured for differential impedance.

So why is this important?

PCB fabrication shops use impedance as a metric to determine if the board has been fabricated to specification. Because the odd-mode impedance of a tightly spaced pair of traces depends on driving both traces differentially, you will not be able to determine the differential impedance by just measuring SE impedance of a tightly coupled pair like you could with two uncoupled traces.

References

[1] E. Bogatin, "Signal Integrity Simplified", 3rd edition, Prentice Hall PTR, 2018
[2] Keysight Advanced Design System (ADS) [computer software], (Version 2020)
[3] Polar Instruments Si9000e [computer software] Version 2017