USB Impedance

In a differential pair, there exists four kinds of characteristic impedance:

* Single-ended impedance (): The impedance of the single trace in isolation, when it's excited by a signal between the trace and the ground plane. In single-ended signaling, this is the only kind of impedance.
* Odd-mode impedance (): The impedance of the single trace in a differential pair, when the differential pair as a whole is excited by a differential-mode (out of phase) signal.
* Even-mode impedance (): The impedance of the single trace in a differential pair, when the differential pair as a whole is excited by a common-mode (in phase) signal.
* Differential impedance (): The impedance of the entire differential pair as a whole (two traces), when the differential pair as a whole is excited by a differential-mode (out of phase) signal. By definition, it's always 2⋅Zodd.

<https://electronics.stackexchange.com/questions/311310/understanding-usb-differential-and-single-ended-impedance-requirements>

In the context of USB signal routing on a printed circuit board (PCB), differential pairs are used to transmit data signals. These pairs consist of two conductive traces that carry the same signal but with opposite polarities, which helps to reduce noise and electromagnetic interference (EMI), improving signal integrity. The way these differential pairs are routed—whether as tightly-coupled or loosely-coupled—can significantly affect their performance. Here's a detailed explanation of both concepts:

### Tightly-Coupled Differential Pairs

- \*\*Definition:\*\* In tightly-coupled differential pairs, the two traces are placed very close to each other relative to their distance from the reference plane (usually a ground or power plane within the PCB). This close proximity enhances the coupling between the traces.

- \*\*Benefits:\*\* The strong electromagnetic coupling between the traces in a tightly-coupled pair reduces the impact of external noise and EMI, making them more immune to interference. It also decreases the differential impedance and increases the phase velocity of the signal, which can be beneficial for maintaining signal integrity over longer distances.

- \*\*Design Considerations:\*\* To achieve tight coupling, designers must consider the trace width, the spacing between the traces, and the thickness of the PCB substrate. The goal is to maintain a consistent impedance while minimizing the space the pair occupies on the PCB. This often requires precise control over the manufacturing process to ensure that the spacing and widths remain within tight tolerances.

- \*\*Applications:\*\* Tightly-coupled differential pairs are typically used in high-speed data communication applications where signal integrity is critical, such as USB 3.0/3.1, HDMI, and Ethernet.

### Loosely-Coupled Differential Pairs

- \*\*Definition:\*\* Loosely-coupled differential pairs have a greater distance between the two traces compared to their distance from the reference plane. This results in weaker electromagnetic coupling between the traces.

- \*\*Benefits:\*\* Loosely-coupled pairs are easier to route on PCBs, especially in complex designs where space is a constraint or when traces need to navigate around obstacles. They are less susceptible to manufacturing variations since the wider spacing is more tolerant to deviations in trace width or spacing.

- \*\*Design Considerations:\*\* While easier to implement, loosely-coupled pairs may require additional design considerations to maintain signal integrity, such as impedance matching and controlling the length of the traces to prevent timing mismatches. The reduced coupling can make them more susceptible to external noise and EMI.

- \*\*Applications:\*\* Loosely-coupled differential pairs are often used in applications where the speed of the data transmission is lower, or the design complexity and PCB real estate do not allow for tightly-coupled routing. They can still be effective in USB 2.0 and other lower-speed interfaces.

### Conclusion

The choice between tightly-coupled and loosely-coupled differential pairs depends on the specific requirements of the USB interface (such as the data rate), the available PCB space, and the desired resilience against noise and EMI. Tightly-coupled pairs are generally preferred for high-speed data transmission due to their superior signal integrity, but they require more precise manufacturing and design. Loosely-coupled pairs offer more flexibility in routing and are more tolerant to manufacturing variations, making them suitable for lower-speed or more complex PCB designs.

A screenshot of a computer

Description automatically generated

(Prepreg 7628 has a relative permittivity ϵr​ around 4.6, assuming standard FR-4 material unless specified otherwise)

When designing a PCB with JLCPCB and using their standard FR4 material (Tg 140C), it's important to note that this material is widely used in the industry and typically has a dielectric constant (ϵr​) around 4.3 to 4.8 for frequencies relevant to USB signal transmission. For our calculations and considerations, we'll use an average value of 4.6, which is a common approximation for FR4 material unless JLCPCB provides a specific dielectric constant value for their FR4-Standard Tg 140C material.

I have a 4-layer PCB with the following stackup:

```

L1 Copper: 0.035mm thick

Prepreg: 0.2104mm thick

L2 Copper: 0.0152mm thick

Core Material: 1.065mm thick

L3 Copper: 0.0152mm thick

Prepreg: 0.2104mm thick

L4 Copper: 0.035mm thick

```

Assume the permittivity of the Prepreg and Core Material to be 4.6.

The manufacturing constraints are:

- Minimal trace width: 0.2mm

- Minimal spacing between traces: 0.2mm

I want to route a differential pair of USB signals. I will put them on layer L2 and sandwich them between a GND plane on layer L1 and a GND plane on layer L3.

If the USB signals have a width of 0.2mm and are spaced 0.2mm apart, what would be their differential impedance?

To calculate the differential impedance of a differential pair of USB signals with the given stackup and constraints, we can use an appropriate model or formula for differential pairs. One commonly used approximation for differential impedance (\(Z\_{diff}\)) of a pair of traces in a PCB is given by:

A black text with a white background

Description automatically generated with medium confidence\[ Z\_{diff} = 2 \times Z\_0 \times \sqrt{1 + \frac{d}{s + 2h}} \]

where:

- \(Z\_0\) is the single-ended impedance of one trace,

- \(d\) is the distance between the centers of the two traces (i.e., the sum of the trace width and the spacing between the traces),

- \(s\) is the spacing between the traces, and

- \(h\) is the height of the dielectric material above the reference plane.

First, we need to calculate the single-ended impedance (\(Z\_0\)) of one trace using the microstrip model, as the differential pair is routed on layer L2 with ground planes on both L1 and L3. The formula for the impedance of a microstrip is more complex and depends on the width of the trace (\(W\)), the thickness of the dielectric (\(H\)), the permittivity of the dielectric (\(\epsilon\_r\)), and other factors. A simplified version of the formula can be:

A math problem with numbers

Description automatically generated with medium confidence\[ Z\_0 = \frac{87}{\sqrt{\epsilon\_r+1.41}} \cdot \ln\left( \frac{5.98H}{0.8W + T} \right) \]

where:

- \(W\) is the trace width,

- \(H\) is the height above the reference plane (thickness of the dielectric),

- \(T\) is the thickness of the trace, and

- \(\epsilon\_r\) is the relative permittivity of the dielectric material.

Given:

- \(W = 0.2\,mm\) (trace width),

- \(H = 0.2104\,mm\) (thickness of the Prepreg above L2),

- \(T = 0.0152\,mm\) (thickness of L2 Copper),

- \(\epsilon\_r = 4.6\) (permittivity of the Prepreg and Core Material),

- \(s = 0.2\,mm\) (spacing between the traces).

Let's calculate the single-ended impedance (\(Z\_0\)) and then the differential impedance (\(Z\_{diff}\)).

The calculated single-ended impedance (\(Z\_0\)) of one trace is approximately \(30.39 \, \Omega\), and the differential impedance (\(Z\_{diff}\)) of the USB signal pair, with each having a width of \(0.2 \, mm\) and spaced \(0.2 \, mm\) apart, is approximately \(77.93 \, \Omega\).

This differential impedance is within the typical range for USB differential pair signaling, which often aims for around \(90 \, \Omega\) in differential impedance, but variations can be acceptable depending on the specific USB standard and the tolerances allowed by the system design.



