Just How Wide Should a PCB 50 Ohm Trace Width Be?

CADENCE PCB SOLUTIONS

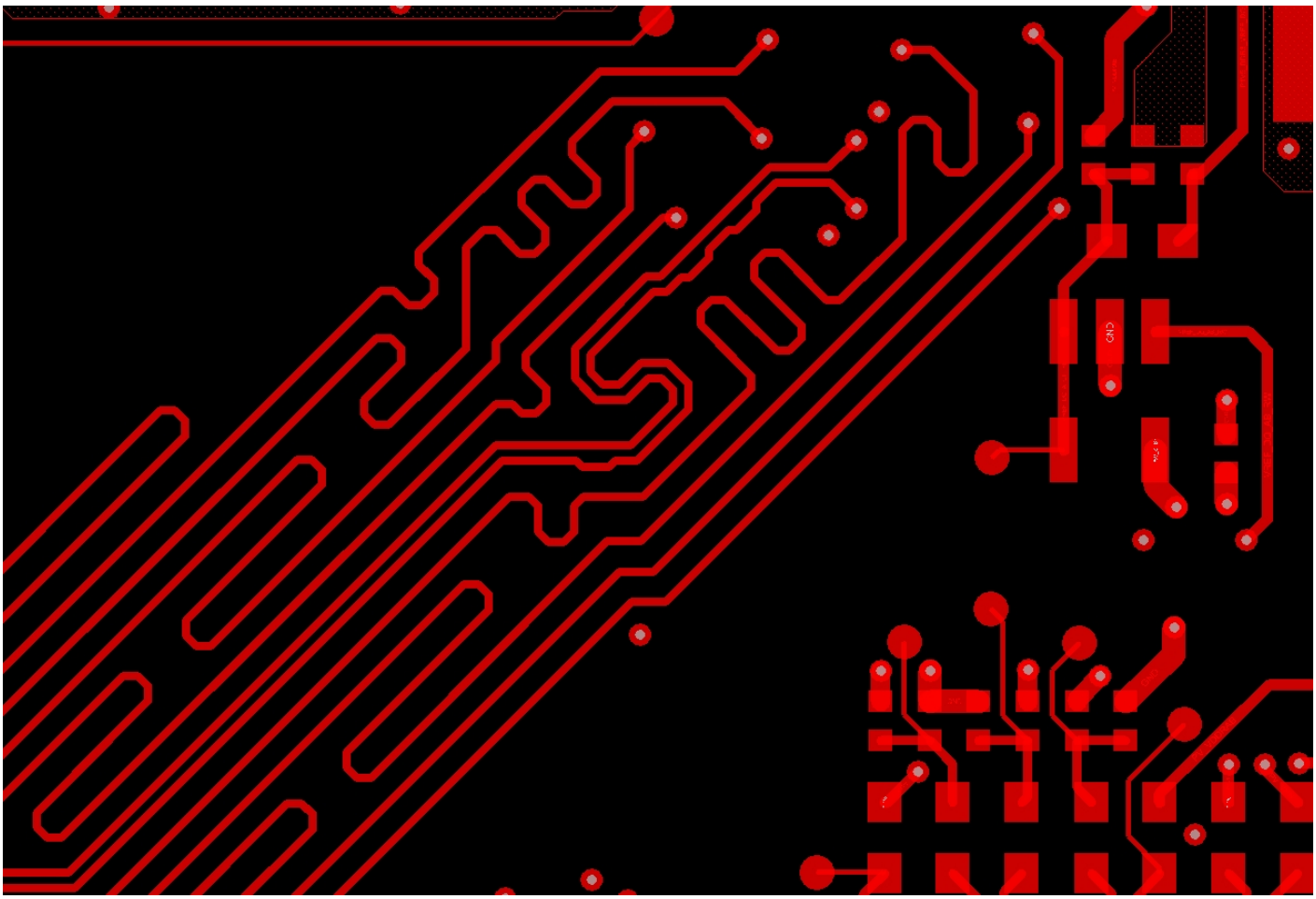
Although it's over 35 years old and very out of date, the movie "Mr. Mom" is still a charming and funny film to watch. In it actor Michael Keaton goes from a successful career one day to being out of work the next. While his wife pursues her own career he tries to run the house and take care of their three small kids. Although confident at first that this wouldn't be a problem, he soon realizes that he is in way over his head. In the middle of the grocery store while trying to figure out what to purchase and at the same time looking for a child who has wandered off, he finally confesses that he has no idea what he is doing.

If we're honest, we've probably had similar moments where we really didn't know what we were doing either. One of those can be the first time we have to route some PCB 50 ohm trace widths. Every other trace width is specified by what net it is assigned to or what layer it is on, but 50 ohm lines are different. Their trace widths are determined by a number of different variables. Let's take a look at what exactly a 50 ohm trace is, and how you should go about figuring out what trace width to route it at.

What Is Important About PCB 50 Ohm Trace Widths?

As high speed transmission line speeds have increased, signal rise and fall times have decreased. This leads to the characteristics of the **transmission line creating impedance**, and for the best signal integrity the impedance must be consistent across the entire length of the line. If there is an impedance mismatch between the source and the load, then the signal will start reflecting, and all of that bounce will degrade the signal. It can get to the point where the receiving device may sense a different clock pulse then what was actually intended.

The key thing then is to ensure that a consistent impedance is held along the entire length of the line, and that is why controlled impedance routing is critical in high-speed PCB designs. There will be different impedance values for different circumstances, but a characteristic impedance of 50 ohms is a **sweet spot** for efficient signal transfer. By holding this impedance you will minimize the chance of signal reflection and give your design the **best signal integrity performance**. 50 ohm trace width routing is an important part of high-speed PCB design and the next step is determining what its width should be.



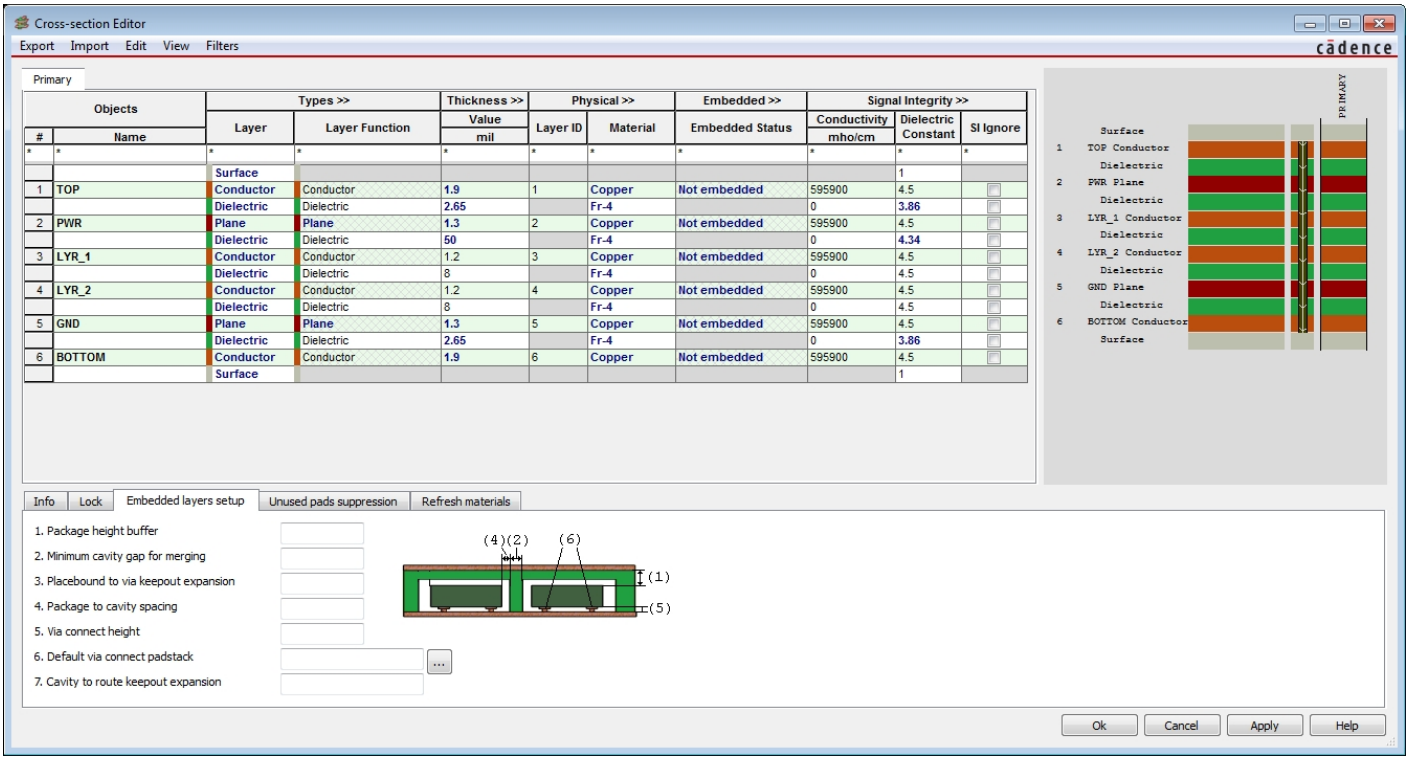
High-speed design included controlled impedance 50 ohm routing and tuned routing like this

How Do I Create a Trace That Is 50 Ohms?

As designers we are used to routing specific trace widths to meet the minimum manufacturing requirements for external and internal layers, or to route specific widths for power and ground nets. These values are usually pretty well set in stone, but in the case of controlled impedance routing we are setting our trace widths to meet an impedance value instead of these other specifications. Yes, we still have to meet the manufacturing requirements for our routing, but the actual trace width will be driven by the need to match the impedance. This means that the actual trace width for a 50 ohm line could vary from design to design.

There are many factors that go into calculating the width of an impedance controlled trace. Different board materials, the dielectric layer thicknesses and constant (Dk), and the thickness of the metal traces all need to be part of the calculations. Additionally the layer stackup of the board will have an effect as your transmission lines need an uninterrupted return path for the signal.

Different layers in your design may need different trace widths to hold the target impedance value. And just as the target impedance value will ultimately determine the trace width that you use, you can also if needed change the target impedance value by changing the trace width. All you need are the tools to make these impedance controlled trace width calculations.



Using a cross section editor like this can be very helpful in setting up 50 ohm trace widths

Tools that Can Help You Determine Your 50 Ohm Trace Widths

The first step in making your trace width calculations for impedance controlled routing is to consult with your contract manufacturer (CM) before you start the layout. Your CM will be working with the board fabricator to arrive at the right combination of materials and board layer stackup for your design. They can also help you with the calculations that you need, and can often send a complete board layer stackup profile to you for your PCB layout tools in an **IPC-2581 format**.

The next step is to arm yourself with **PCB design tools** that can work with these impedance values and give you the trace width information that you need to route with. **OrCAD PCB Designer** is one of the most versatile PCB design systems that you can get, and it has the impedance calculator technology you need to determine precisely what trace widths you need for your 50 ohm controlled impedance routing.

If you're looking to learn more about how Cadence has the solution for you, **talk to us and our team of experts**.