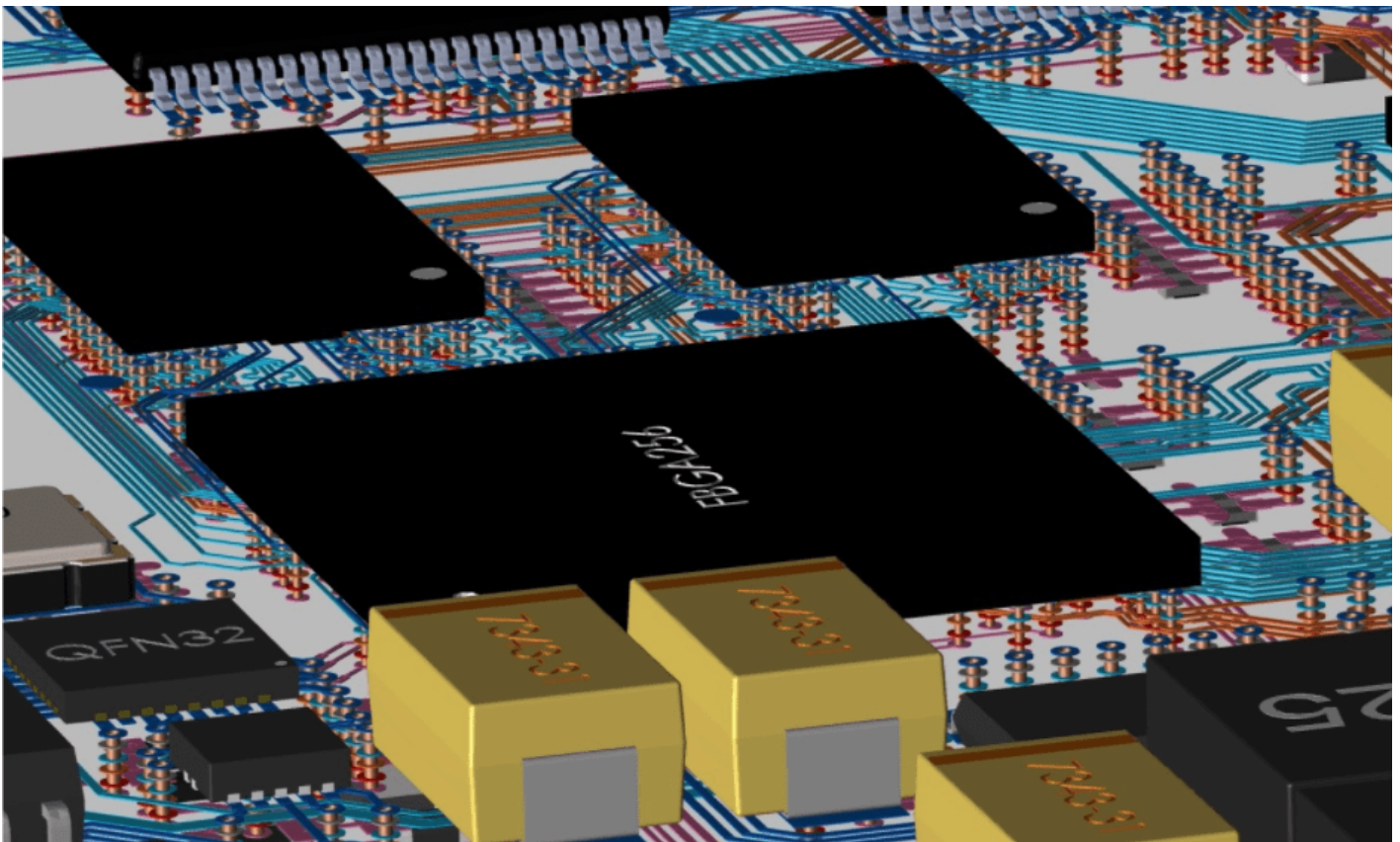


Effective PCB Trace Width and Spacing

CADENCE PCB SOLUTIONS

Key Takeaways

- The challenges of routing that designers are facing.
- Examples of trace width and spacing requirements.
- Using your PCB design CAD tools to help you.



A multi-layer circuit board showing the routing on the internal layers.

One of the advantages of being a PCB designer is getting to route traces in a printed circuit board layout. Long before Tetris or Candy Crush, trace routing gave designers their own unique puzzles to solve by untangling a rat's nest of

net connections in order to create a cleanly routed board. Routing traces on a circuit board can be a very rewarding accomplishment, but it also can get very complicated as the designs grow in size and complexity.

In today's PCB layouts, each net may have different and unique routing characteristics that have to be applied to it in order for it to function as designed. In addition to specific areas that nets can be routed in or layers that they can be routed on, there are also different PCB trace width and spacing rules that have to be managed as well.

Here are some of the challenges facing designers today as they route their circuit boards, as well as some methods you can use to **successfully route** according to the required rules and constraints.

The Challenges Posed by Today's PCB Routing Technologies

At one time, routing traces on a printed circuit board was a fairly straightforward procedure. Your traces were all assigned to a default width and spacing, except for where power and ground connections connected to their vias, which had to be made wider. Any other trace widths that might be

required would be few and could be easily handled by manually changing them during routing. As circuit board technology grew, however, the trace width and spacing requirements became much more complicated.

Now, on any given board you may see some or all of the following:

- Controlled impedance routing with specified widths and spacings.
- Sensitive high-speed traces that need to be isolated from other routing with wider spacing.
- Analog routing which may have a different default width spacing requirement.
- **Power and ground** connections which will need a wider trace.
- Power supplies which could have multiple trace widths depending on the circuit.
- Extra spacing between analog and digital routing to keep them isolated from each other.

In addition to these trace width and spacing requirements based on the functionality of the circuit, there also may be different width and spacing requirements dictated by their location. Some of these are as follows:

- Connectors may require nets to use a smaller trace width to snake their way between closely spaced pins.
- Fine-pitch parts such as quad flat packages (QFP) or small-outline packages (SOP) may need their trace widths reduced for escape routing.
- Ball grid arrays (BGA) may also require shrunken trace widths to route in and around the pins and vias of the package.

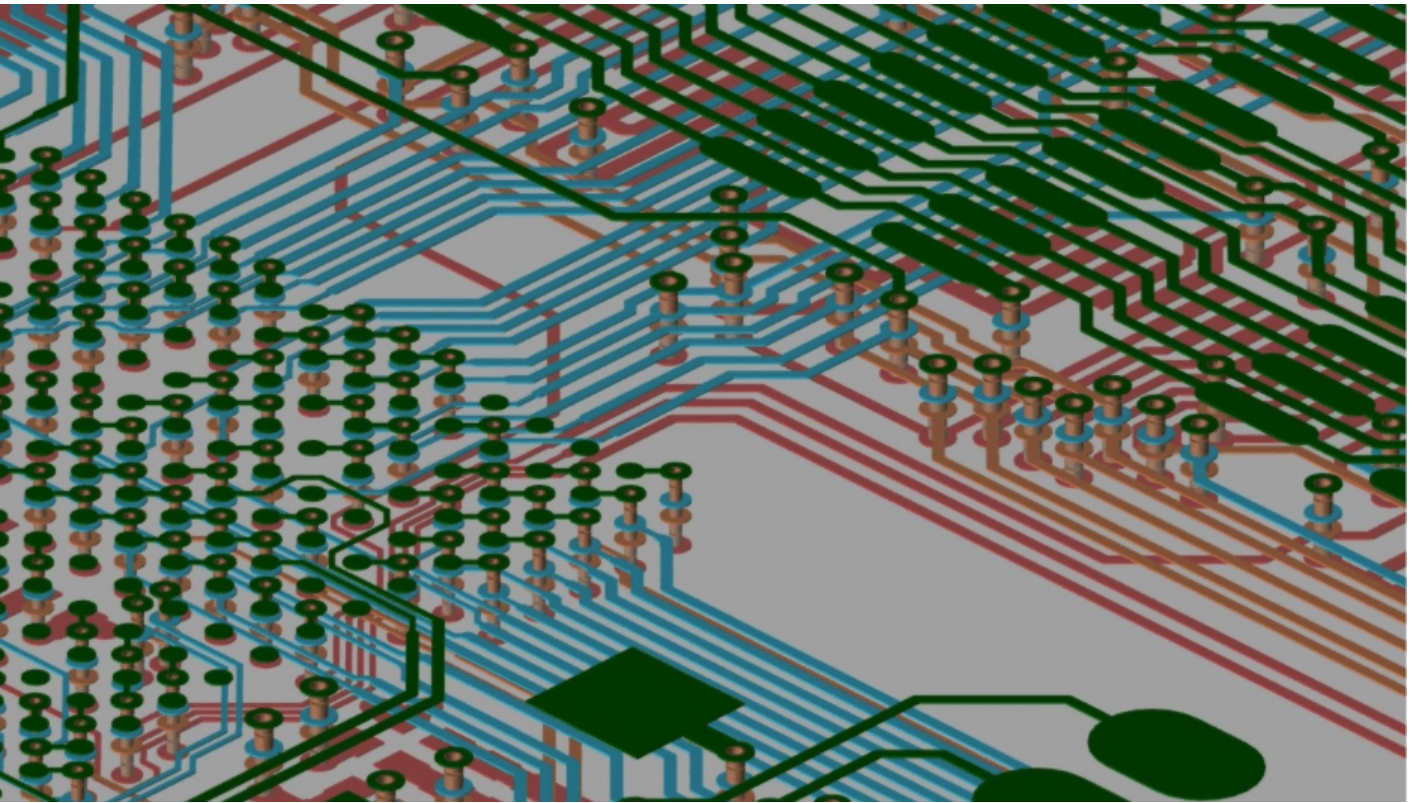
Another part of PCB routing is the vias used for transitioning between layers. When circuit boards used to be laid out manually with tape and dollies, a designer could simply take an Exacto knife and trim down a via pad to squeeze a trace by it. In today's PCB design CAD systems, however, a more precise method is needed. Designers will use an array of different **via types and sizes** for their routing.

Vias are broken into three types:

- Thru-hole
- Blind and buried
- Micro

Thru-hole vias are the standard via used in the design of a circuit board. They are mechanically drilled and go all the way through the board. A blind or buried via is also drilled mechanically, but it will either only go partially through the board or start and stop on internal layers. These vias require more fabrication steps to create, as the individual layer pairs of the board have to be drilled and aligned prior to laminating the board together. Microvias are created with a laser, which allows them to be much smaller than a mechanically drilled hole, and they typically span only two layers of the board. They are well suited for via-in-pad applications and high-density interconnect (HDI) designs that require smaller traces and vias.

These are some of the challenges facing designers as they route complex printed circuit boards. Next, we will look in greater detail at some specific examples of width and spacing requirements.



3D CAD layers showing the trace routing on a circuit board.

PCB Trace Width and Spacing Examples

PCB trace widths and spacings can affect the circuit board in many ways. Here are four areas to consider when deciding what width and spacing values to use:

Electrical Performance and Signal Integrity

Most digital routing on a circuit board will use a default value for its trace widths and spacings, but some nets will require different sizes. Controlled impedance nets, for instance, will need their trace widths calculated based on the configuration of the **board layer stackup**. Sensitive high-speed traces may require larger spacing values to prevent crosstalk and other signal integrity problems.

Analog routing may also require unique trace widths and spacings depending on the purpose of the circuitry. In some cases, the default trace width may be reduced in tight and constricted areas, but care has to be taken that this isn't

extended across the board. If the trace gets so thin that it gets compromised during PCB fabrication, then the signal integrity of the entire board could be compromised.

Power and Ground Routing

Traces used to route power and ground need to be wider to conduct greater amounts of current. If the traces are too thin, they can get hot and even burn through. Power traces routed on the internal layers of the board also need to be wider for heat dispersal than those routed on the external layers, as exposure to the air will provide more cooling. For those traces that are used in power supply circuitry, it is important to keep them as short and as wide as possible to handle the current, as well as to **reduce inductance and noise** in the circuit. It is important to increase the spacing for traces that are carrying higher current to prevent the power from arcing between them.

Circuit Board Fabrication

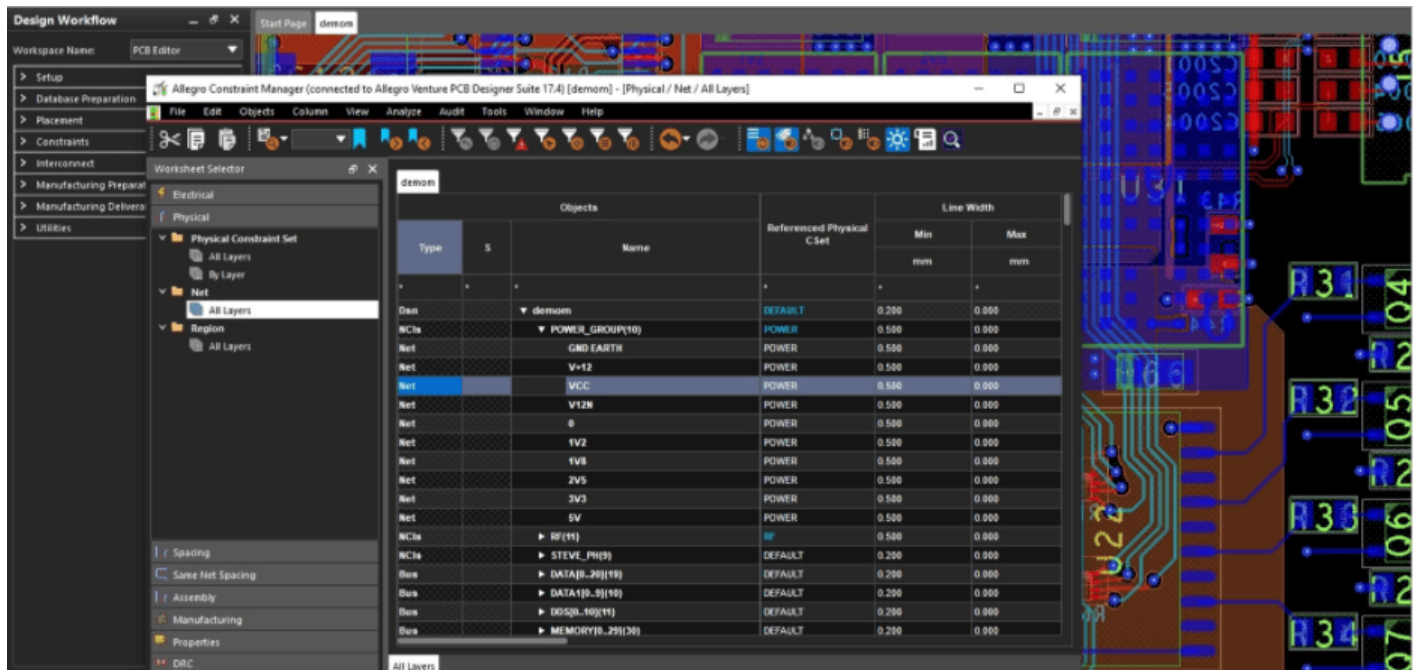
The wider a trace is, the easier it is to fabricate. The fabrication etching process will have a greater effect on traces that are long and isolated, so it is better to make them wider when possible. A 20 mil trace will have a much greater tolerance for losing metal than a 3 mil trace would. Trace widths are also dependent on the weight of the copper being used to build the layer of the board. If that layer of the board requires a greater copper weight due to higher current requirements, the fabricator may not be able to etch thinner trace widths on it as well.

Circuit Board Assembly

Traces that are too wide can affect how easily the components will solder during PCB assembly. The wide traces used for power and ground nets can also act as a heat sink, leading to uneven heating and poor solder joints. Small two-pin parts that are connected to a large area of metal on one pad and a thin trace on the other may be unbalanced enough that the component will be pulled up on end during solder reflow. This effect is known as “tombstoning”

and will force manual rework of the board for corrections. An abundance of metal under a BGA can also cause problems during soldering as well, but these errors are more difficult to find due to the size of the BGA on the board.

With all of the potential problems that can happen if the right trace widths and spacings are not maintained, PCB layout engineers need all the help that they can get. Fortunately, there are utilities in the design tools that can help with this.



The design rules and constraints being set up for power net trace widths.

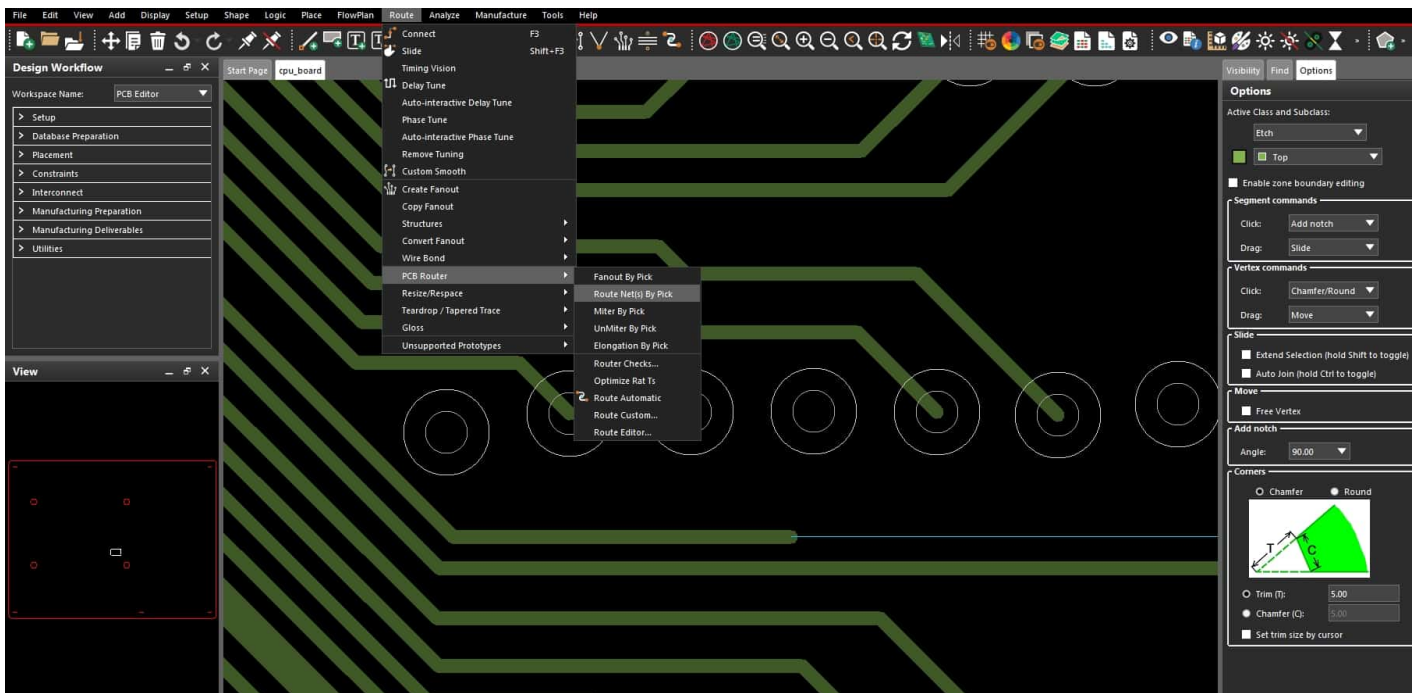
Design Rules and Constraints

As we have seen, there are a lot of different trace widths and spacings that need to be managed in the layout of a printed circuit board today. These will include width and spacing assignments for individual nets, groups of nets (buses), and power and ground nets. Additionally, the correct vias need to be used in each of these assignments, and in some cases, there may be more than one set of values that have to be assigned to an individual net or group. The way PCB design tools handle this task is to use a design rules and constraint management system.

Design rules and constraints have advanced a long way since the early days of PCB design CAD tools when there was a limited amount of control available. Now, with **constraint management systems** like the one pictured above, you can use the spreadsheet-style interface to set up a variety of rules and constraints. As you can see, the nets are assigned with different trace widths. If you were to scroll over to the right in this menu, you would also find settings for trace spacing, via assignments, and much more.

Constraint management systems like these give designers complete control over the trace routing rules for their designs. Additionally, these systems will also control other design values like signal timing, component spacing, and manufacturability settings for solder paste and silkscreens.

Today's CAD tools offer a lot more than just rules and constraint management. however; they have a wide range of routing tools that can help a designer's efficiency as well.



Some of the different routing options available in the Cadence Allegro PCB Editor.

Putting the Full Power of Your PCB Design Tools to Work

Although a PCB layout will have many nets that require different approaches to how they are routed, you no longer have to manually route all of these nets yourself. The design tools will have several trace routing features and functions that can help. Take the **routing capabilities** in Cadence Allegro, for instance, which offer the designer plenty of routing aids and tools to use, such as:

- Visual cues to show you the best channels while manually routing.
- Heads up display that will track your routing constraints while you route.
- Push, shove, hug, and slide options to assist you with manual routing.
- A scribble routing feature that allows you to sketch out your route manually, and then let the automated engine do the heavy lifting of routing the traces in.
- Contour routing features that help to create curved traces at any angle.
- Timing functionality that allows you to add and edit trace tuning segments in your routing.
- Auto-routing engines to add single traces, via fanouts, or fully route the entire board.
- Auto-interactive routing to manually organize and maneuver bundles of nets for the auto-router to finalize.
- Routing clean-up features such as glossing the traces for clean-up or adding tear-drops to the pad and trace connections to enhance manufacturability.

Cadence design tools will give you a wealth of power when routing your circuit board so that you have complete control over your PCB trace width and spacing settings, as well as so many others. Additionally, there is lots more practical information on PCB routing that you can read about in this **E-book**.

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