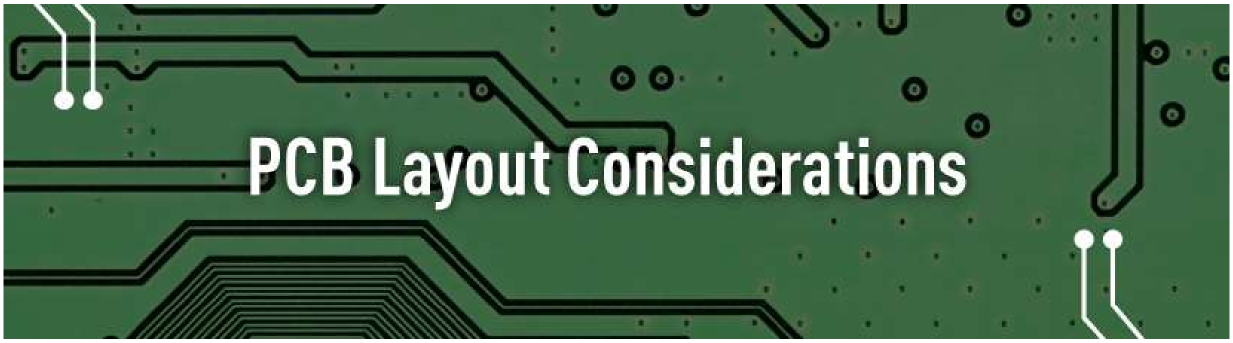


Jump To: Basic PCB Design Steps | Design Documentation | PCB Layout and Design Considerations | The Importance of Testing | Solving PCB Layout Problems With CAD | Choosing a PCB Supplier

Designing the layout of your printed circuit board is crucial to creating a reliable, cost-effective board. While circuit design and component selection are also essential, you should always make sure you leave enough time for PCB layout. A lot goes into determining the optimal PCB layout design, especially since today's boards are becoming more complex, compact and lightweight. The growing popularity of flexible PCBs complicates the process, too.

If you don't account for important PCB layout considerations, you may end up with a design that doesn't translate well to the real world. An inadequate layout can result in several problems such as electromagnetic interference, conflicts from components on either side of the board, limited board functionality and even total board failure. Plus, if you don't get the layout right the first time around, you will need to rework it, which can cause manufacturing delays and added costs.

So, what are the PCB layout design rules and considerations you need to keep in mind? Let's look at the steps of PCB layout design and identify some of the core considerations for each phase. Of course, there are other considerations you may want to keep in mind, too, but these are some of the most critical aspects of PCB layout design you should be aware of.

BASIC PCB DESIGN STEPS

PCB design plays a role in every step of the printed circuit board production process from the moment you know you need a PCB to final production. The basic design process includes six steps.

1. Concept

After identifying the need for a PCB, the next step is determining the board's final concept. This initial phase involves defining the functions the PCB will have and perform, its features, its interconnection with other circuits, its placement in the final product and its approximate dimensions. Also, consider the approximate temperature range the board will operate in and any other environmental concerns.

2. Schematic

The next phase is to draw the circuit schematic based on the final concept. This diagram includes all the information needed for the electrical components of the board to function appropriately, as well as details such as component names, value, rating and manufacturer part numbers.

While you're creating your schematic, you'll be creating your bill of materials. This BOM contains information on all of the components you need for your PCB. Always keep these two documents up to date.

3. Board-Level Block Diagram

Next, you will complete a board-level block diagram, a drawing describing the final dimensions of the PCB. Mark areas designated for each block, sections of components that are connected for electrical reasons or because of constraints. Keeping related components together will enable you to keep your traces short.

4. Component Placement

The next step is component placement, which determines where you will place each element on the board. Often, you may go through several rounds of refining component placement.

5. First-Pass Routing

Next, determine the routing and the routing priority for the circuit.

6. Testing

After you've completed the design, you should **conduct a series of tests** to ensure it meets all your needs. If it does, the design is complete. If not, you will go back to the phases where you need to make adjustments.

DESIGN DOCUMENTATION

As you go about creating your PCB, you'll develop numerous documents. These documents include:

- The Hardware Dimensional Drawings: Describes the size of the bare board
- The Schematic: Maps out the electrical features of the board
- The Bill of Materials: Describes the components needed for the project
- The Layout File: Describes the basic layout of the PCB
- The Component Placement File: Describes the location of the individual components.
- The Assembly Drawings and Instructions: Explains how to assemble the board
- The User Guides: While not required, they're useful for providing additional information to the user

- The Gerber File Set: The collection of output files of the layout that the PCB manufacturer will use to create the PCB

PCB LAYOUT AND DESIGN CONSIDERATIONS

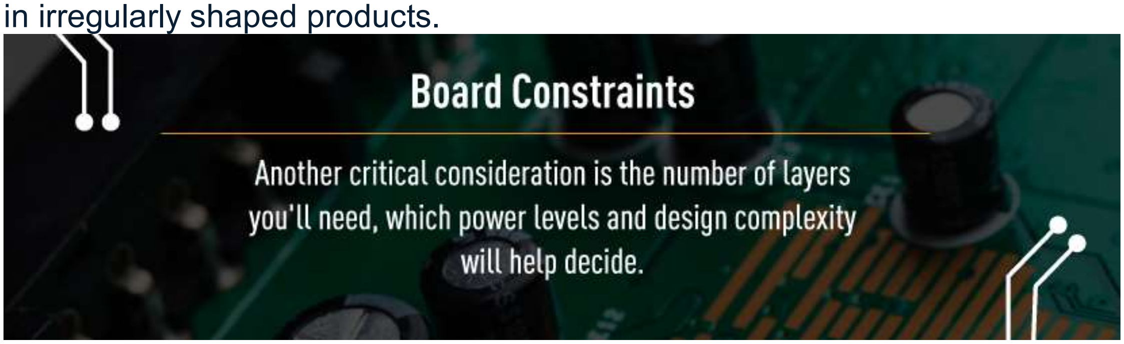
There's a lot to consider regarding PCB layout and design. Some considerations apply to the entire process, while some are specific to particular steps. Here are seven relevant factors to keep in mind.

1. Board Constraints

The first constraints you should look at are those associated with the **bare board**. Some of these basic constraints include the size and the shape of the board.

You will need to ensure you have adequate board area for the circuit. The size of the end product, the functionality the board must provide and other factors determine how large the board should be. Electronic products and the circuit boards they incorporate are becoming increasingly smaller. Before you start the design process, estimate the size of the board. If you do not have enough space for all the functionality required with a more straightforward design, you may need to use a multilayer or **high-density interconnect (HDI) design**.

The standard PCB is rectangular. This remains, overwhelmingly, the most common shape for PCBs. It is possible to create boards in other forms, however. PCB designers most often do this because of size constraints or use in irregularly shaped products.



Another critical consideration is the number of layers you'll need, which power levels and design complexity will help decide. It's best to figure out how many you need early in the layout design process. Adding more layers may increase production costs but enable you to include more tracks. This may be necessary for more complex boards with advanced functionality.

Use at least two vias to make layer transitions for all high-current paths. Using multiple vias at layer transitions increases reliability, improves thermal conductivity and reduces inductive and resistive losses.

2. Manufacturing Processes

You should also **consider the manufacturing processes** you'd like to employ to produce the board. Different methods have different limitations and constraints. You'll need to use reference holes or points that work with the manufacturing process on the board. Always ensure holes are clear of components.

Also, keep the board mounting method in mind. Different approaches may require you to leave different areas of the board open. Using multiple technology types, such as both through-hole and surface mount components, can increase the cost of your boards but may be necessary in some cases. Always check with your fabricator to make sure they have the capabilities to produce the type of board you need. Some might not, for instance, be able to manufacture boards with many layers — or those that use a flexible design.

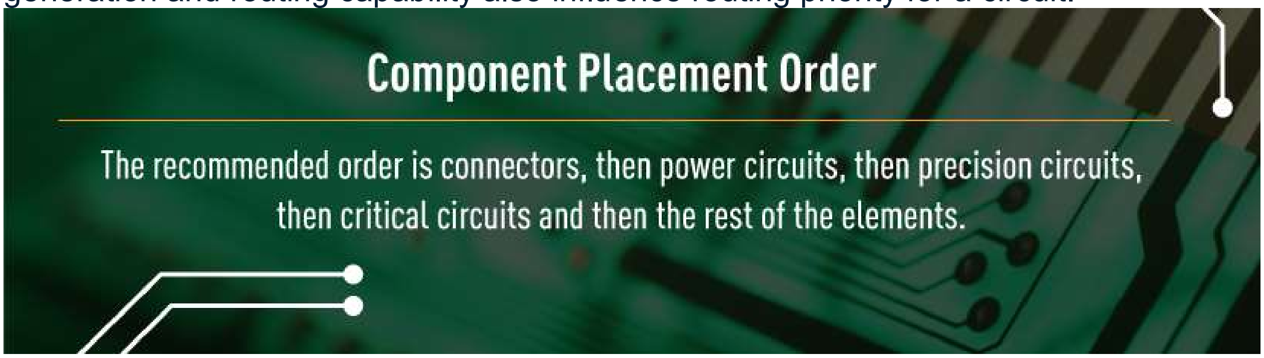
3. Materials and Components

Consider during the layout phase the **materials and components** you plan to use for your board. You'll first need to make sure the desired items are accessible. Some materials and parts are hard to find, while others are so expensive they're cost-prohibitive. Different components and materials may also come with different designs needs.

Take time to ensure you've chosen the optimal materials and components for your board, and also that you've designed a board that plays to those items' strengths.

4. Component Placement Order

One of the most fundamental PCB design guidelines involves the order in which you place components on the board. The recommended order is connectors, then power circuits, then precision circuits, then critical circuits and then the rest of the elements. Power levels, noise susceptibility, generation and routing capability also influence routing priority for a circuit.



5. Orientation

When placing components, try to orient those that are similar to one another in the same direction. This will make the soldering process more efficient and help prevent mistakes from occurring during it.

6. Placement

Try not to place parts on the solder side of the PCB that will sit behind plated through-hole parts.

7. Organization

Logically organizing your components can reduce the number of required assembly steps, increasing efficiency and reducing costs. Aim to put all your surface mount components on one side of the board and all your through-hole components on the top side.

POWER, GROUND AND SIGNAL TRACE CONSIDERATIONS

The above tips focused on PCB component placement. For those components to work as desired, you also need to route the power, ground and signal traces. Completing this step efficiently will help ensure your signals have a reliable path to travel to keep your board functioning properly. Here are five factors to keep in mind.

1. Power and Ground Planes

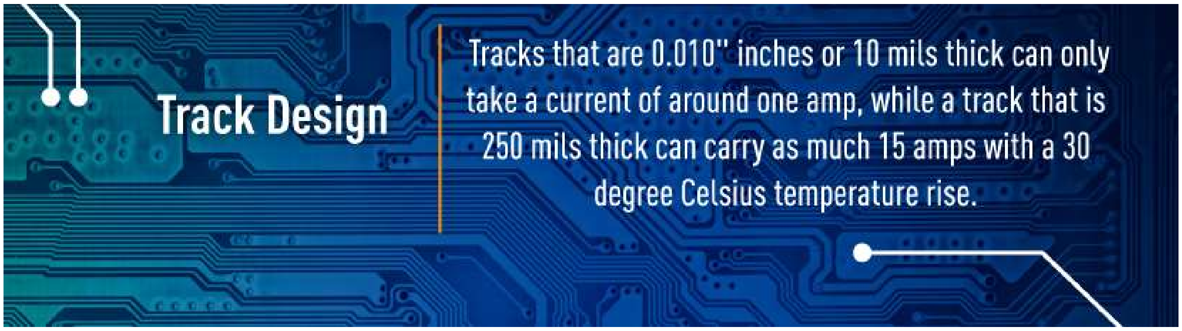
One fundamental PCB layout design rule is to keep your power and **ground planes** internally within your board. They should also be centered and symmetrical to prevent bowing and twisting of your board. Bowing can cause components to move out of position and potentially damage the board. Other recommendations include using common rails for each supply, making sure you have reliable, extensive traces and avoiding creating daisy chains to connect components.

High voltage in power circuits can interfere with low-voltage and current control circuits. You can use the placement of your power ground and control ground to help minimize this interference. Try to keep your grounds for each power supply stage separated. If you need to place some together, make sure they're toward the end of your supply path. If your ground plane is in the middle layer of your board, include a small impedance path to prevent power circuit interference.

You should also keep your digital and analog grounds separate in a similar fashion. Try only to have analog lines cross your analog ground to reduce capacitive coupling.

2. Track Design

This step also involves connecting signal traces according to your schematic. You always want your traces to be as short and direct as possible. If you have horizontal trace routing on one side of the PCB, place vertical traces on the other side.



Your board may require multiple nets with different currents, which will determine the net width you need. Using a **trace width calculator** can help with this step. Thin tracks can only carry so much current. Tracks that are 0.010″ inches or 10 mils thick can only take a current of around one amp, while a track that is 250 mils thick can carry as much 15 amps with a 30 degree Celsius temperature rise.

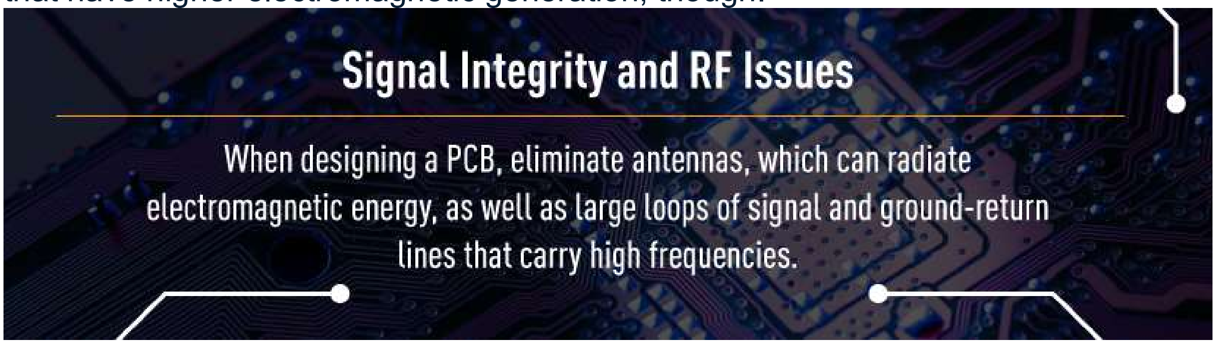
3. Pad and Hole Dimension

You'll also need to determine pad and hole dimensions early in the PCB design process. As the size of the pads and holes decreases, getting the right pad-to-hole size ratio becomes more crucial. It's especially critical when working with via holes. The **bare PCB manufacturer** may be able to provide guidelines on the standards and aspect ratio they require.
Another important consideration is the shape of the PCB pads. PCB footprints can vary according to the manufacturing process. **Wave soldering** typically requires larger footprints than infra-red reflow soldering does, for instance.

4. Signal Integrity and RF Issues

PCB layout design plays a crucial role in ensuring signal integrity and preventing electrical problems such as interference, often referred to as radio-frequency interference or electromagnetic interference.
Avoiding these problems has a lot to do with how you route your traces. To prevent signal issues, avoid running tracks parallel to each other. Parallel tracks will have more crosstalk, which can cause various problems that are difficult to fix once you've built the PCB. If tracks need to cross over each

other, make sure they do so at right angles. This will reduce capacitance and mutual inductance between the lines, decreasing crosstalk in turn.
Using semiconductor components that generate low electromagnetic radiation can also help with signal integrity. Sometimes, other needs may require parts that have higher electromagnetic generation, though.



When designing a PCB, eliminate antennas, which can radiate electromagnetic energy, as well as large loops of signal and ground-return lines that carry high frequencies. You must position integrated circuits carefully to achieve short interconnect lines.
Placing a close ground grid over the PCB is another essential **RF PCB** layout design guideline that helps to ensure that return lines are close to the signal lines. This keeps the effective antenna area relatively small. **In a multilayer board**, you can achieve this with a ground plane.

5. Thermal Issues

Thermal issues can impact many different parts of the design process. Larger boards and those with higher component density and higher processing speeds tend to have more heat-related problems. For smaller boards, they might not be a concern, but for more advanced ones, they can be a significant challenge.
To prevent heat-related problems, you need to allow heat to dissipate. First, identify components that generate a lot of heat. You should be able to find each component’s thermal resistance ratings in its datasheet. Then, you can follow the recommended guidelines for diverting heat from that component.
Ensure you leave sufficient space around all components that may get hot. The more heat they create, the more area they will need to cool off. It’s also vital not to place critical components near heat sources.
Ideally, the entire board will have the same operating temperature. Use thermally conductive planes to dissipate heat across a wide area, which

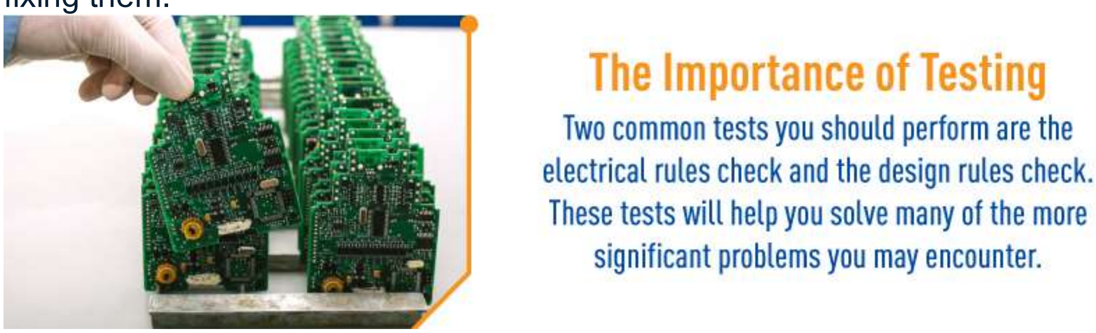
speeds the rate at which temperature decreases by increasing the surface area used for heat transfer.

If thermal issues are substantial for your board, you may need to include cooling fans, heat sinks and thermal reliefs, which are critical for wave soldering on multilayer boards and assemblies with high copper content. You can create heat sinks using a heat sink paste, a polymer filled with finely dispersed solid particles. You can apply this paste using **screen or stencil printing**. After a drying or baking process, it becomes fixed and acts as a heat sink.

It's always advisable to use thermal reliefs on through-hole components, which slows the rate at which heat sinks through the component plates. As a general rule, use a thermal relief pattern any time a via or hole connects to a ground or power plane. You may also want to use teardrops where traces and pads meet to provide additional support and reduce thermal stress.

THE IMPORTANCE OF TESTING

Throughout the PCB design process, as well as the rest of the PCB manufacturing process, you should continuously check your work. Catching problems early on will help minimize their impact and reduce the costs of fixing them.



Two common tests you should perform are the electrical rules check and the design rules check. These tests will help you solve many of the more significant problems you may encounter.

Once you can pass your ERC and DRC tests without any problems, you should check the routing of every signal and compare your board to your schematic in detail.

SOLVING PCB LAYOUT PROBLEMS WITH CAD

Today, most PCB designers use advanced computer-aided design (CAD) software systems to create their PCBs. Similarly, manufacturers use computer-aided manufacturing software. Using these systems can help you solve many of the layout problems you may encounter. Some of the advantages of using these software systems include:

- Simple, Semi-Automated Design Processes: CAD programs allow you to drag and drop components where you need them to go on your design. Many systems will even create the traces for you, and then you can move, add or remove components or reroute them as needed. This approach can increase the efficiency and accuracy of your design process.
- Design Validation: Before you send your design to the manufacturing stage, you can test it using a CAD system to verify your tolerances, compatibility, component placement and other aspects. Many systems can even catch basic errors in real time, minimizing or eliminating their impact.
- Manufacturing File Generation: You can generate Gerber files and other files formats you may need to send to the manufacturer with a CAD system. Creating these files directly from the design software can help increase their accuracy and ensure a smooth transition to the manufacturing stage.
- Documentation: You can also use these systems to generate and save detailed documentation related to component use, error reports, design status, version control and more, which can assist with future projects.
- Rule Creation: Some of these programs allow you to create and store custom rule sets, which you can share with designers to improve the functionality of the software.
- Template Creation: You can even create templates for use in future projects. Once you create a design, you can save it and reuse it as a template for other projects.
- Increased Efficiency and Reduced Costs: Incorporating computer-aided design into your operations can improve the efficiency and accuracy of your design process, which reduces overall costs.