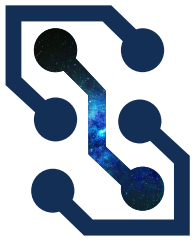
PCB LAYOUT

Steps for Successful Setup and Placement Processes



SEGMENTS OF SANITY
FROM THE GUIDE

PCB Layout: Steps for Successful Setup and Processes

In our previous schematic article, it might appear that with so much detailed information anyone could perform the layout phase. However, a detailed schematic is only the foundation for the PCB layout and not just anyone can create a successful PCB layout. Layout must be performed by someone who understands industry standards for design and manufacturing as well as the depth of knowledge required for incorporating design for excellence. This individual is analytical enough to be able to sort through vast amounts of data, but creative and free-thinking enough to see many alternative ways to complete a design and have the intuition to choose the best one. Simply put, a designer of PCBs is a person of renaissance. Here are some steps to place you on the path to successful layout setup and placement.

Component Footprint Creation

Sometimes it is up to the PCB designer to create new component footprints for a design. A simple, straightforward process based on satisfying the stakeholder need to create perfect solder joints which could pass inspection of IPCs J-STD-001. If the component footprints are not correct, Assembly Manufacturing Engineers (AMEs) have a more difficult time soldering all the component pins to the PCB lands.

Setting Up the Layout

Design automation helps perform the first step in the layout process using personal layout templates. These templates are configured to contain a designer's favorite settings for common design environment items. Often used for setting design units, layer colors, net colors, trace widths, and via sizing, they can be modified at any time during the layout to match design requirements.

Defining the Mechanical Constraints

Consider the complete definition of the PCB outline; there is more to a PCB outline than four lines to define a rectangle. In fact, PCB outlines are rarely defined as simple, rectangular shapes as PCB design continues to shrink electronics into the tightest useable spaces. Modern PCB outlines originating from advanced 3D packaging can contain complex curved edge requirements.

Parts Placement

Once the mechanical constraints of the PCB are set, it is time to bring in the data from the schematic. When the component footprints are imported, they are automatically connected by fine lines to show their pin's connectivity with other component pins. All these connections are based upon the previously established connectivity defined in the schematic. The imported part footprints usually appear to be piled up randomly off to the side of the defined PCB outline after being imported into database. A design with hundreds of components imported in this way will have so many connections crossed over one another, it will make the pile of parts and connections look like a 'rat's nest'. This is where the fun begins—the designer can begin adding organization to chaos. Most designers consider the placement stage of layout their reason for being—they know this first layout step will drive the success of the entire project because parts placement effects every manufacturing process and therefore every manufacturing stakeholder.

Group Imported Parts and Connections

After the parts are imported, they must be grouped. Using the schematic as a reference, a designer can quickly begin grouping the component footprints into clusters of circuits which can be organized so signal connections are at their

shortest distance between pins. A simple way to navigate and sort through parts during placement is to set up cross-probing functionality between the schematic and the layout. With cross-probing active and an organized flow, placement of the process of grouping is made easy.

Many things about the evolving layout will now begin to come into view. Component clusters will show the ratio of component area to PCB area outline. Many lines interconnecting the groups will eventually be converted into signal traces which will require area. Will all the lines be routed on the outer surfaces of the PCB? Will they fit once the components are all placed? Will more layers be required? These are all layout density questions to be addressed after determining all the parts will fit within the PCB outline, while still meeting requirements for Design for Excellence (DFx).

Position Groups onto the Board

Once the components are grouped per the schematic flow, begin organizing them. Groups should be organized according to their connectivity flow between each other and to their respective input or output connectors. Pay special attention to the voltage designations and try to organize similar voltage groups together, envisioning how a voltage plane may be used to connect to all nodes.

Placement Matters

Contrary to popular belief, placement matters because it affects performance and manufacturability. Checking in with a manufacturing stakeholder for wise council on DFM—even for placement of a three-cent chip resistor—can save thousands of dollars in cost and is a far better plan than using "artistic license."

DFx Tradeoffs—Design Density

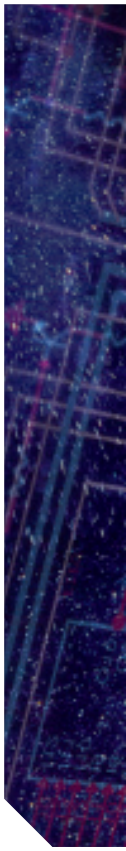
Consider design density early in the placement process. If the BOM and PCB outline has already been defined by the ME, using a CAD layout tool is the quickest way to begin this analysis. With the components already grouped, the designer can move the groups onto the layout. Hopefully all the groups will fit, and the designer can go on to refine the placement.

If the groups don't fit, a few logical steps can be taken to remedy this. Remember, the component groups are not set in stone. At this point, the designer will begin to get an idea for whether all the parts will fit on a single side of the layout or not. If it appears nudging individual parts in the groups will help all the parts to fit, then continue to smooth out the placement. If not, then it is time to consider utilizing the bottom (now secondary) side of the PCB. This is not a step to be considered lightly, as transitioning to a two-sided assembly brings the PCB design into another manufacturing class and will most certainly increase manufacturing costs.

Shaking Hands with Your Network

Understand the need for establishing and cultivating design and manufacturing contacts. Expanding your professional network will benefit you in the long run. There will always be design challenges and when required, turning to your network will help provide you with the best possible answer.

As a champion of DFx, it is the responsibility of the PCB design engineer to bring the two stakeholders together and negotiate compromise. Not every process will be perfect; DFx means the requirements of all stakeholders have been considered and the design has been optimized via stakeholder consensus.



PCB Layout: Steps for Successful Setup and Processes

Working with the ME Stakeholder

A mechanical engineer can only guestimate the board area required for the layout. Rarely is a complete schematic and BOM accessible to the ME while the outer packaging of the product is being designed. Early in the mechanical design layout phase, the ME will usually have access to interface components, like connectors and switches, and will naturally define a PCB outline based upon the area left over, after the mechanical design is complete.

Cross-Checking Tools and Placement

Use an ECAD tool to cross-check PCB layout with the mechanical layout. ECAD and MCAD tool industries are on convergent paths which someday will merge to become one. While rapid progress is being made, current projects are still designed by separate mechanical and electrical stakeholders who must be well-versed in communicating their perspectives.

Working with the Assembly Manufacturing Engineer Stakeholder

Once the placement is finalized and has been reviewed internally by the EE and ME, invite a review from the assembly stakeholder. Most EMS providers are happy to perform this review, but they will require the appropriate data to review effectively. It is important to call and ask what data they need.

Most likely, the EMS provider will be very happy to receive the layout in ODB++ or IPC-2581 formats. An assembly drawing (unfinished or not) will provide workmanship standards and peripheral information which may be used for the review.

Simplicity and Collaboration are the Keys to Success

The steps involved in placing parts onto a PCB design layout could be considered a lot like the steps involved in architecturally planning an urban construction project. However, keeping it simple by following the rules, determining where compromises must be made, and bringing together the right people for effective collaboration is a proven strategy for success in any scale of project, in any industry.layout in ODB++ or IPC-2581 formats. An assembly drawing (unfinished or not) will provide workmanship standards and peripheral information which may be used for the review.