FUNDAMENTALS
OF ROUTING & PLANES

SEGMENTS OF SANITY FROM THE GUIDE

EMA Design Automation®

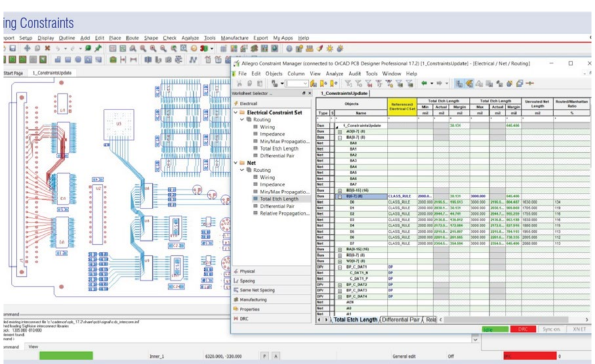
Fundamentals of Routing and Planes

In our last article, we established the PCB stack-up is more than just a thickness diagram; this lays the foundation for routing and utilization of copper planes within the layout. There are four important topics to consider during completion of the layout:

- Design Rules and Defining Vias
- Utilizing Planes and Pours
- Impedance Control and SI
- Routing

Design Rules

To successfully complete a PCB layout, it is very important to set the design rules to match the estimated producibility class of the design. PCB design rules can be set in most layout tools which allow generic settings to be saved. Design Rules Checking (DRC) tools will check against these rules in real-time, providing you feedback as you go. Having this real-time feedback is invaluable as a designer. Trying to keep track of all the rules and how they interrelate manually is not a task you want to take on.



Just about everything pertaining to copper in a PCB design is controllable in the design rules portion of a PCB layout tool. The pre-set design rule values or “default settings” are set by a layout tool provider who knows nothing about the new layout a designer may be working on. There can be hundreds, if not thousands, of control variants in the tool’s default settings which can cause catastrophic manufacturing conflicts if not re-set to match the requirements of a new project.

Once the connectivity of the layout is synchronized with the schematic, it is essential for a designer to surmise which general design rule constraints will best control the new design and reset them for the required result.

In general, there are five basic areas which will need to be configured to match the design you are working on. Four of the five category settings directly affect manufacturability: copper plane clearance, part outline clearance, drill (hole) clearance, legend (markings) clearance, and trace length clearance. The last setting category affects performance: trace length clearance.

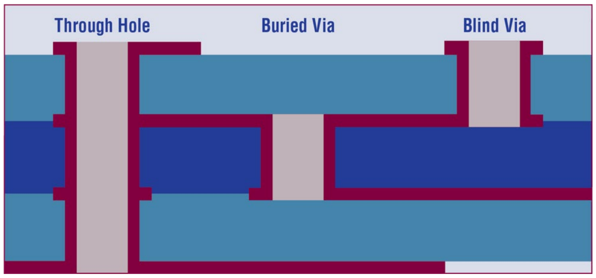
It will likely take some time for a new designer to learn how to control all the design rule settings. However, concentrating on these five categories first should help. Additionally, it is always a good idea to check in with the appropriate manufacturing process stakeholder when initially setting these values. For best DFM, never set values to the stakeholder’s minimum or maximum capability.

Design rules in a PCB layout tool can be a blessing if understood and set properly. However, they can also be a curse if not carefully considered and reset from the default settings.

Defining Vias

Via size requirements need to be considered at the beginning of a layout. Vias are small holes in the PCB which are plated to make interstitial connections to multiple circuit layers. In other words, making connections from one side of the board to the other or to inner layers requires a via.

There are three general types of mechanically-drilled vias: through-hole, buried, and blind.



Drilled Vias
Mechanically-drilled via holes are kept small for space concerns and have a special relationship with PCB thickness due to the process by which the holes are plated.

Laser-formed Vias
Due to the conical shape of a hole formed by the laser while burning through material, designers must pay very close attention to the width / depth relationship capabilities of the process. A 1:2 ratio between the width and depth is considered manufacturable.

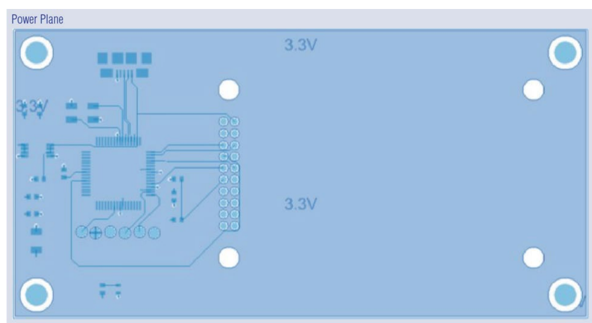
Selecting the right type of via
Selecting the proper via type will not only help complete trace routing but can also help in alternate applications for uses like EMI shielding and thermal management. However, choosing any via type other than a common thru-hole via with a modest hole diameter/board thickness aspect ratio will most certainly incur a cost adder as well.

As with most things in the PCB manufacturing industry, as via size shrinks or the complexity goes up, the "yield" of the PCB run may go down as some part features, such as vias, fail to pass inspection. Blind vias and blind vias will incur cost adders because of the unique processes involved.

Vias are often a necessity in the design. Knowing the different types, benefits, and potential costs will help you make sure you select the best via for your needs avoiding cost and manufacturability issues downstream.

Utilizing Planes and Pours

Power planes are the most effective way to distribute power to almost every part of the PCB. Power planes are simply formed by adding layers of copper foil to the stack-up and connecting them to power or ground. Smaller surface mount technology (SMT) parts located any place on the board surfaces can connect to a plane by use of a via. Larger SMT parts requiring more power can use multiple vias connected to the power planes. By their physical nature, thru-hole parts will easily connect to the planes as required. However, for high-current requirements, designers sometimes add support vias around a component pin to increase the current path to the pin.



Besides distribution, copper planes can be used for many other purposes such as:
- EMI reduction return path
- Added capacity
- EMI shielding and ESD guard band support
- Thermal management– heat sinking

Impedance Control and Signal Integrity

Industries requiring quick digital applications such as telecommunications, video, computing, and others rely on impedance control to help minimize signal degradation and improve signal integrity. A PCB design engineer needs to recognize controlled impedance requirements on a schematic and be able to manipulate the design layout to achieve them.

Fundamentals of Routing and Planes

Controlled Impedance

Impedance control on a PCB is required when performance dictates signal degradation in a circuit cannot vary beyond a specified amount expressed in ohms. Impedance control requirements for PCB layout can be addressed physically in the layout by identifying which nets in the design are to be controlled by their specific impedance value. Always seek approval from the PCB fabricator after estimating solutions.

Signal Integrity

Certain circuits require routing parameters which must be held for the device signals to run properly. Signal routing parameters are documented in the component manufacturer's data sheet. There are several parameters to be aware of when performing placement of the chip, its related components, and when completing the routing. These include:

- Lengths and routing order of critical signals
- Hub routing constraints where a signal routes to a given point, then splits off to several length-matched destinations
- Distance to adjacent signals could allow for unwanted signal coupling or "cross-talk"
- Impedance control, either single-ended or differential
- Reference plane running beneath signal routing layers

Sometimes PCB electrical performance considerations conflict with manufacturing considerations. A designer must be able to quickly assess the appropriate compromise. Often companies will have an SI expert or consultant who specializes in these types of high-speed signals to ensure proper operation. While every designer may not be an SI expert, having a solid understanding of the critical signals and the general requirements needed to ensure acceptable signal integrity will help keep issues to a minimum.

Routing

PCB routing is a highly subjective topic due to the myriad of various constraints which may be present within the PCB design. Not only does the PCB layout be routed, it must perform. Subjectivity aside, there are a few considerations which must be implemented in the interest of both electrical performance and DFM. These include:

- Fanout
- Routing Power
- Routing Critical Lines

There is a saying in the PCB design industry: "90% of the routing on a PCB takes only 10% of the time, while 10% of the remaining routing can take 90% of the time." This perspective describes a designer's experience toward the completion of a layout when the fun of routing turns over to an ever-increasing challenge. Suddenly there is a realization all the fun routes have eaten up all the routing space. The routing challenge will most certainly turn into defeat and a complete do-over if the remaining 10% of the routing includes any critical signals. Keep routing fun and challenging—consider critical net noise, clearance, and establishing shortest routing paths first. At the same time, consider avoiding adjacent layer parallelism and eliminating circuitous routing paths. Avoid complete routing do-overs by routing critical lines first.

Conclusion

It might be easy to sum up the process as simply "fitting the parts on the board and hooking them together." However, the intent of this article has been to give a new designer some basic routing points to consider before, or instead of, simply activating an auto-routing routine. Successful routing and plane utilization has a direct effect on circuit performance. Implementing set-up, routing and establishing copper planes in a PCB layout can really make the difference between a dot-to-dot hook-up technician and a designer. Metaphorically, this comparison may be likened to the steps a skilled photographer would implement to create a classic photograph. Without understanding how to properly setup and manipulate the subject matter, lighting, and exposure controls, a camera operated by the click of novice may only yield a cheesy snapshot.

For more, download The Hitchhikers Guide to PCB Design.