In this article, we outline the “Golden Rules of PCB design,” most of which have stayed consistent for 25 years.

Despite increasing levels of semiconductor integration and readily available systems-on-chips for many applications, in addition to the increasing availability of highly-featured development boards, electronics often still require a custom PCB. Even for “one-off” developments, the humble PCB still performs an important role. It’s a physical platform for a design, and the most flexible for pulling an electronics system together. In this article, we outline ten best practices of PCB design, most of which have stayed consistent for 25 years. These rules are in no particular order, can generally be applied to any PCB design project, and should prove as a useful guide both to veteran design engineers as well as makers alike.

Rule #1: Get the right grid.
Find a grid spacing that suits as many of your components as possible and use it throughout. Although multiple grids may seem appealing, a little additional thought at the early stages of the layout can avoid spacing difficulties and will maximize board use. Many devices are available in different package sizes, so use that to your advantage. Furthermore, as the polygon is an important shape when adding copper to your board, and boards with multiple grids will often produce polygon-fill discrepancies, not standardizing on one grid can make your life tougher than necessary.

Rule #2: Keep trace lengths as short and direct as possible
This rule applies even if it means going back over parts of the layout again to optimize track lengths. This applies particularly in analog and high-speed digital circuitry where impedance and parasitic effects will always play a part in limiting your system performance.

Rule #3: Whenever possible, use a power plane to manage the distribution of power lines and ground.
Using pours on the power plane is a quick and easy option in most PCB design software. It applies plenty of copper to common connections and helps ensure power flows as effectively as possible with minimal impedance or voltage drop, and that ground return paths are adequate. If possible, run multiple supply lines in the same area of the board and remember that if the ground plane is run over a large section of one layer, it can have a positive impact on cross-talk between lines running above it on an adjacent layer.

Rule #4: Group related components and test points together.
For example, place the discrete components needed for an opamp close to that device so the bypass capacitors and resistors are co-located with it. This helps with the track lengths in Rule #2, and it also makes testing and fault-finding easier.
Rule #5: Panelize your PCB by replicating the board you need several times on a larger board.  
Using a size which best suits the equipment used by your manufacturer will improve the cost of prototypes and manufacturing. Start by laying out the board as one panel. Ask your board house what size panel they prefer. Then, after your design rules have been corrected, do your best to step and repeat your design multiple times within the preferred panel size.

Rule #6: Consolidate your component values.  
As a designer, you will have picked some discrete components that could be a higher or lower value and work just the same. Consolidating on a smaller range of standard values makes the BOM simpler and probably less expensive. It also makes stock decisions easier in the long run if you have a range of PCBs based on your preferred device values.

Rule #7: Design rule check (DRC) as often as you can bear it.  
The DRC function on PCB software takes a little time, but checking as you go can save hours on more complex designs, and it is a good habit to adopt. Every layout decision is important, but the DRC keeps the most important ones top-of-mind. Rule #8: Use the silkscreen wisely.

Rule #8: Use the silkscreen wisely.  
The silkscreen can be used to portray a wealth of useful information to the board builder, as well as the service or test engineer, installer, or device operator. Clear labels depicting functions and test points are obvious, but orientation of components and connectors should also be considered wherever possible. Even if annotation ends up under your components following board assembly, it’s still a good practice. Full use of silk screening on both sides of the board streamlines production and can reduce re-work.

Rule #9: Decoupling caps are not optional!  
Do not try and optimize your design by avoiding decoupling power lines and trusting the absolute limits of component data sheets. Capacitors are inexpensive and robust; take the time to fit them in wherever possible and remember Rule #6 - use a range of standard values to keep the inventory neat.

Rule #10: Generate your own PCB manufacturing data and verify it before sending it out to be fabricated.  
Most board houses will be happy to do this for you, but if you output your own Gerber files first and use a free viewer to verify it looks as you envisioned, then you can avoid misunderstanding. You may even catch an error inadvertently included before it’s set forever in fiberglass, resin, and copper.

As circuit designs are more widely shared, and reference designs are relied upon more and more by in-house teams, we believe it’s important that basic rules like these remain in printed circuit design. Keeping sight of the basics means developers retain the flexibility to add value to their products and extract the most from every board they make. Finally, anyone new to board design will accelerate their learning-curve and confidence when the basics are “designed in.”

Edwin Robledo is an Electronic Engineer with his degree from the Polytech University of Puerto Rico, and has been member of the CadSoft team for over 20 years. Currently he is General Manager of CadSoft Computer for the US. See his profile.

Mark Toth is Business Development Manager of the Americas for CadSoft Computer, a partner of U.S.-based Newark element14, and a division of global electronics distributor, Premier Farnell. See his profile.
Also See:

- PDN issues occur in the simplest of circuits
- Component placement tips and strategies
- PCB Design Basics: Example design flow