
What Affects The Price Of Boards?

- **Number of layers:** Limit the number of layers as much as possible. If higher quantities are anticipated, spend the extra time and money in engineering to minimize layer count. Layout engineering is a one time expense, manufacturing is not!
- **Quantities ordered:** Order the higher quantities whenever possible. You may be better off maintaining a higher inventory of say 3 or 5 months worth. Smaller runs mean your board house is less likely to panelize and maximize the number of images on a panel. Larger runs are simply more efficient. If your board manufacturer has to stock them until you need them, you can bet there is a hidden charge somewhere.
- **Delivery time:** Quick delivery greatly influences board prices. Allow as much time as schedule permits. Don't ask for a rush only to find out a special order chip or transformer has a 10 week delivery and you cannot proceed with assembly. Longer lead times allow your board house flexibility for other quick turn work, so yours can be at lower cost.
- **Number of hole sizes:** Limit the number of hole sizes. Each additional size adds to the cost. Layout designers have considerable flexibility since hole sizes can be .007 - .015 larger than component lead sizes.
- **Small hole size:** Try not to use hole sizes smaller than .028. They limit stack drilling, and drilling one board at a time greatly increases machine time. Drills also break more often and copper plating becomes more difficult. Holes .021 and smaller must be stack drilled 1 deep. Holes .022 thru .027 can be stack drilled 2 deep. Holes .028 and larger can be stack drilled 3 deep.
- **Annular ring:** Try to provide pads that are .020 larger than hole size. Planned annular ring of .010 is good design criteria. Allowances must be made for etching, lamination, drill position, front to back layer registration and film stability. Smaller annular ring is certainly possible, but at the expense of lower yields and increased manufacturing difficulty which results in higher cost.
- **Hole tolerance:** If possible, do not specify hole size tolerances any smaller than +/- .003. Boards are usually pre-drilled .003 to .004 larger than the finished holes size to allow for copper plating inside the hole. Drilling and copper plating is a very critical step in manufacturing circuit boards. Tight tolerances just increase problems and lowers yields. A lot of material and labor go into a multilayer before its drilled and plated. If it's out of tolerance, it's all scrap.
- **Copper weight:** Do not specify 2 oz. copper unless you are sure you need it. 2 oz. material cost about 40% more than 1 oz.
- **Via and Component Pads:** Keep vias and component pads a minimum of .050 away from connector gold-plated fingers. This will simplify the tape process and prevent exposed bare copper after hot air solder leveling.
- **Internal cut-outs:** Avoid these if at all possible. These and very large holes require special routing. Also, make the board as small as possible.
- **Mil-P-5511OD boards:** If you require Mil boards, look into manufacturing the prototypes to best commercial standards such as IPC600. The quality will probably be the same, but at much lower cost.
- **Standards:** Adopt PCB design, artwork and manufacturing standards: By adopting standards, you will insure your product can be manufactured for the lowest cost possible. Your artwork should be of such quality that literally any board house can build it without problems. Also provide the manufacturer with a drill file, a fabrication detail drawing, and gerber data instead of film whenever possible. This allows first generation photo and encourages panelization for reduced unit cost. We would be most happy to meet with your design layout group and review these standards at no cost.