How to Order Printed Circuit Boards from BITTELE

Website: 7pcb.com

Units: Metric or English

Discount: Members of the Hardware Academy get a 5% discount on larger orders. For more details see the Academy discounts spreadsheet.

PCB Fabrication:

PCB Specifications:

Board Size: This is the dimensions of your individual board which is a critical factor in the PCB cost.

Quantity: Start with 5 boards on your first version, then slowly increase as your confidence in the design improves.

Thickness: This is the final thickness of the board. The most common thickness is 1.6mm (0.062”). If space or weight are critical then you will select a thinner board. If strength of the board is critical then a thicker board will be best.

Layers: This is the number of routing layers used in your PCB design. The simplest PCB uses 2 layers (top and bottom). Most moderate complexity designs (i.e. microcontrollers) require 4 or

6 layers, and more complex designs (i.e. microprocessors) requires 8 or more layers. The number of layers directly impacts the cost of the PCB, and is always an even number.

Material: The is the base material used in the PCB between stacked routing layers. ***FR-4*** is by far the most common material which is sufficient for most designs.

FR-4 comes in different Tg versions where Tg is known as the glass transition temperature. It is the temperature at which the material begins to lose its rigidness and becomes more rubbery. For most designs ***TG140*** is sufficient. Stepping up to ***TG170*** adds considerable cost.

If your design has custom RF circuitry (including antenna feedlines) then a RF specialized base material may be needed. Rogers R4000 series of materials are the most commonly used material for RF designs. ***R4003C*** is a halogen-free material which is more environmentally friendly than ***R4350B***. Both have fairly similar RF related specifications.

If your design is high power then a base material with a metal core may be necessary for improved power dissipation. This is especially common in high power LED lighting applications. It does not appear as if Bittele offers this as part of their normal quoting process so you may need to inquire with them or use PCBWay.

Number of Designs: Printed circuit boards are not manufactured individually. Instead they are produced in larger panels containing multiple copies of your board. This allows more efficient production for smaller boards.

After PCB assembly (soldering of components) or perhaps after final testing is completed the individual boards on each panel are then separated or depaneled.

In most cases you will not need to worry about panelization and the manufacturer will supply you fully assembled boards that have already been separated.

It is possible to have different boards combined together on the same panel. If your product requires multiple different boards then you can have them all on the same panel to save costs. Although there is an additional fee for having a panel with different board designs.

Solder mask: Solder mask is protective layer that is added to the outer layers of the board to prevent oxidation and solder bridges (shorts) between adjacent solder pads. Green is by far the most common color, but you can also select for other colors.

Silkscreen: Silkscreen is a layer for adding text and simple images (logo) to your board. It most commonly adds the component identifiers.

Surface finish: The surface finish is a coating added to exposed copper pads on the outer layers of the board to prevent corrosion and improve solderability. In the past HASL was the dominate choice but it has mostly been replaced by ENIG (Immersion Gold) because it is lead-free. Immersion silver (Ag) is also a lead-free finishing option.

Copper weight: This is the weight of the copper layer spread out over 1 square foot. It really is a measure of the copper thickness. For example, 1oz of copper equivalates to 1.4 mils of copper thickness. Thicker copper traces are primary used for high power designs so as to require less wide traces to carry the same amount of current.

Impedance control: Impedance control is necessary for some more complex high-frequency digital or analog designs. It consists of measuring the impedance of certain traces when the PCB is manufactured to make sure they are withing the design limits for proper signal propagation. It adds considerable cost so it is only used when absolutely necessary. It is not necessary for most designs.

Advanced Specs:

Blind/Buried Vias: *Blind vias* and *buried vias* are only used when small size is critical because they significantly increase the board cost. Normal vias are through-hole meaning then go through all layers, even the layers where they are not needed. These vias can take up unnecessary space on the layers where they are not connected. A blind via connects from an outer layer to an inner layer so you can only see one end visibly on the final board. A buried via instead connects between two inner layers and is totally invisible on the final board.

Via in Pad: Normally vias are not permitted inside of solder pads because the via hole acts to wick away solder thus potentially leaving a bad solder connection. However, in some cases such as with high pin count BGA packages with a small pin-to-pin pitch, the limited space requires the use of vias in the pads. Some chip manufacturers will also specify vias in a pad to improve the heat transfer properties.

Min Trace Width/Spacing: This is minimum allowable width for any routing traces/tracks and the minimum space allowed between traces. This is measured in mils (a mil is a thousandth of an inch). 6 mils is probably the most common, but many designs where space is critical may need to go down to 4 or 5 mils. Once you go below 6 mils the price begins to increase significantly.

Inspection Process: This is the inspection process followed for the PCB fabrication. Class 3 adds some additional cost.

Gold Fingers: If your board has edge connectors then you will want those to be gold plated for corrosion and abrasion resistance.

Soldermask Sides: Do you need soldermask on only one side or both sides. See above for description of soldermask.

Silkscreen Sides: Do you need silkscreen on only one side or both sides. See above for description of silkscreen.

Custom Stackup: The stackup is the order in which the various board routing layers are stacked.

Skip V-Scoring: V-scoring is a method for depaneling individual boards from the panel. A V-shaped grove is cut about a third of the way through the board on both sides. This allows the individual boards to be easily broken apart.

Half-cut/Castellated Holes: Castellated holes are commonly used on PCB modules which are designed to be soldered onto another PCB board.

Edge Plating: This is copper plating that runs along at least one edge of the board.

Countersinks/Counterbores: These are drilling methods that allow a fastener (such as a screw or bolt) to sit flush with the top surface of the board.

Date-Code Marking: This is just a marking on the board that specifies the manufacture date.

UL Marking: PCBs are flame retardant. In fact, the FR in the popular base material FR-4 refers to flame retardant. This falls under certification UL94V-0 which can be stamped on the board if preferred.

Silkscreen Clipping: The silkscreen text and drawings sometimes may overlap SMT pads, vias, and other features. You can choose to have this overlapping text/drawings clipped. When designing the silkscreen layer it's best to ensure it doesn't overlap with any of these features so no clipping is necessary.

PCB Assembly:

Number of SMT Pads: This is the number of component pads that require Surface-Mount Technology (SMT). For example, an SMT capacitor will require two pads.

Number of Through Holes: This is the number of through-holes required. For example, a through-hole resistor will require two through holes. Through-hole parts are rarely used on modern electronic designs except in cases where a higher strength attachment is necessary such as for a connector that will experience regular forces.

Double Sided SMT Assembly: This is whether your design have components attached on a single side or both sides of the board.

Lead-Free: You can choose to use leaded or lead-free solder during your assembly process. Lead-free is required in order to obtain RoHS certification. For your own health and the health of the environment I always recommend lead-free.

BGA/LGA/WLCSP Components: This is the number of chips in your design that are packaged in BGA, LGA, or WLCSP packages.

QFN/DFN/SON Components: This is the number of chips in your design that are packaged in leadless (meaning no leads, don't confuse with lead-free) packages such as QFN, DFN, and SON.

Smallest Package: This is the size of your smallest passive components (resistors, capacitors, and inductors). These are standard package numbers where 1206 is the largest and 0201 is the smallest. For most designs I recommend you go no smaller than 0402.

Inspection Standard: This is the inspection process followed for the PCB assembly. Class 3 adds some additional cost.

All Parts: Total number of parts to be assembled on to the board. This is extracted automatically from the BOM.

BOM Lines: This is the number of unique components that will need to be soldered onto your board. For example, if your design uses five identical capacitors in various places that would count as one unique component. This count includes every unique chip, resistor, capacitor, inductor, connector, etc. This is extracted automatically from the BOM.

HARDWARE ACADEMY