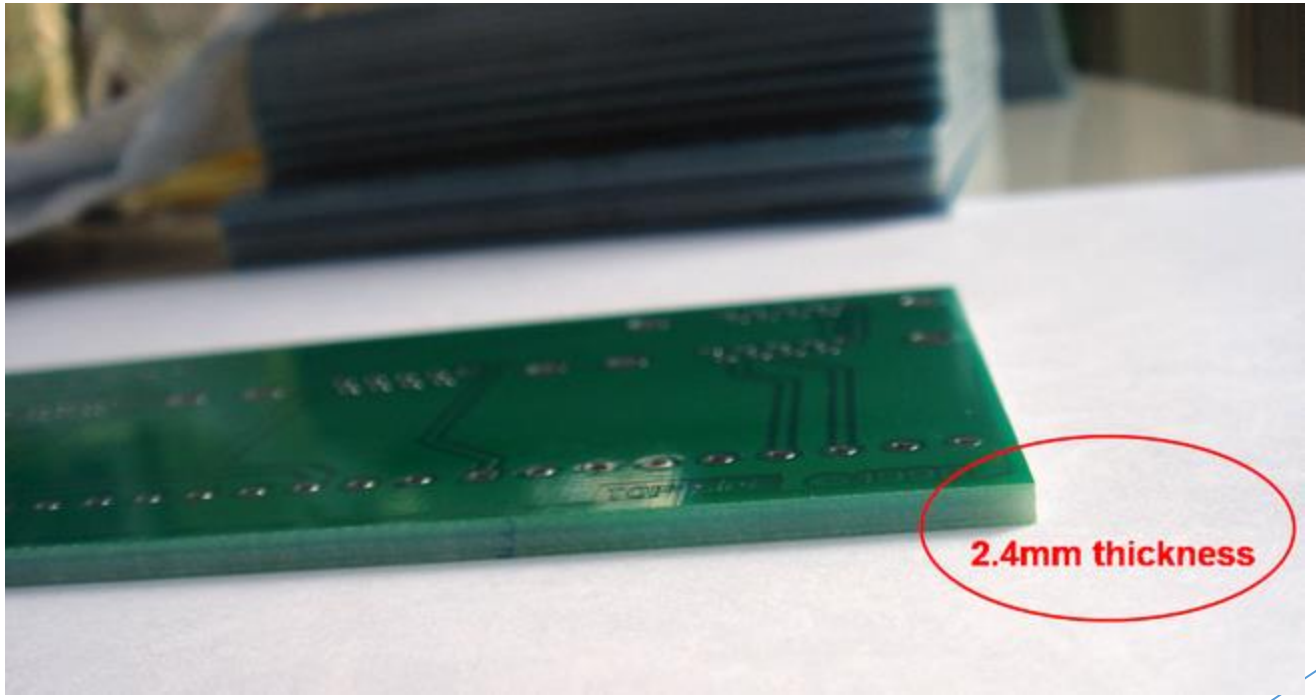
MCL Bootcamp

Board Thickness & Limitations, Aspect Ratio, Copper Weight, Trace & Space

Thickness of Board

- No standard rule for the thickness of a board
- Occasional limiting factor is the diameter of the smallest hole, especially when the holes are plated through



Final Board Thickness

- Will depend on the number of conductor layers and on the electrical layer-to-layer spacing requirements of the design
- For multi-layer boards, the increase in cost is not directly proportional to the increase in the number of conductor layers
- Thickness can vary from 0.50 to 6.5 mm, but most rigid boards have thicknesses of 1.57mm or 0.062”
- Importance – thickness and width of a trace determines the amount of current (amps) the circuit can carry
  - Thickness and width of a trace is also used in the calculation of trace impedance (ohms) in RF and high speed digital circuits

Factors Influencing Board Thickness

- Space – thinner boards take less space in your packaging
  - Very small devices will use a much thinner board to save space
    - Bluetooth headsets
- Connections – using PCB edge connectors, or PCB trace based USB A connectors, etc., require the PCB to be the right thickness to fit in the mating portion of the connector
- Impedance matching – every two (or more) layer board will act like a capacitor between traces on adjacent layers
  - The thickness of the board defines the thickness of the dielectric, and hence the value of the capacitance
  - For high frequency signals where impedance matching is critical that capacitance has to be taken into account, and getting the right dielectric thickness or distance between the defined layers

Factors Influencing Board Thickness

- Flexibility – thinner boards can flex; that can pose problems for certain machines if the board is too big, or its panelized wrong
  - V-Scores (V-grooves) are not a good idea with very thin boards and small panels are better than large, so its up to the fab house what they set as an upper limit on size for those
  - Boards that can flex can cause broken connections with components (especially BGAs), so a thin board will need good support from the casing its in to avoid that
- Component compatibility – some through-hole components are designed for specific board thicknesses
  - Ex: press-fit connectors
- Weight – thinner boards are lighter and can be cheaper to ship and also result in lighter end products

Base Material Thickness

- Power electronic devices dissipate a certain amount of heat, in order to do so they usually need suitable heat sinks
  - If the heat sink directly mounted to the board, the entire PCB will be heated up to the same temperature
  - Therefore, the base material selected must withstand the continuous operation of the end product
- Most used epoxy laminate has a thickness of 1.6mm (0.062”); however, 2.4mm (0.093”) and 3.2mm (0.125”) thicknesses will meet the mechanical property requirements for mounting heavier components
  - Ex: heat sinks, pulse transformers, chokes, etc.
- For flexible boards, the base material is usually very thin, in the range of 0.05mm

Base Material Thickness

- The base copper weight chosen impacts many critical features of the PCB
  - Achievable trace and space width dimension
  - Recommended minimum width of the annular rings
  - The amount of trace reduction – more base copper equates to higher trace reduction
    - trace reduction is compensation – during manufacturing the fab house has to make etch compensations based on the thickness of copper
      - During the etching process, each layer will have some variance in trace width
      - The more base copper, the larger amount of etch compensation that is needed

Copper Foil Thickness

- A copper clad laminate with a 35 μm (1 oz.) standard thickness is preferable for low power circuits
- For high power circuits, normally a copper clad laminate with a thickness of 70 μm (2 oz.) or above copper foil is commonly used

Thickness of Finished PCBs

- Extensive range of finished PCB thicknesses can be accomplished with the combination of different core thicknesses and sheets of prepreg
- Standard thickness of 1.57mm (0.062 inch)
  - Not unusual to have a thickness tolerance of ± 10% per IPC standards
- Core thickness, prepreg thicknesses must be considered for trace impedance

How to Measure Thickness of a PCB

- Measured from outermost feature to outermost feature
  - copper to copper (typically with solder mask over the copper)

PCB Limitations Due to Thickness

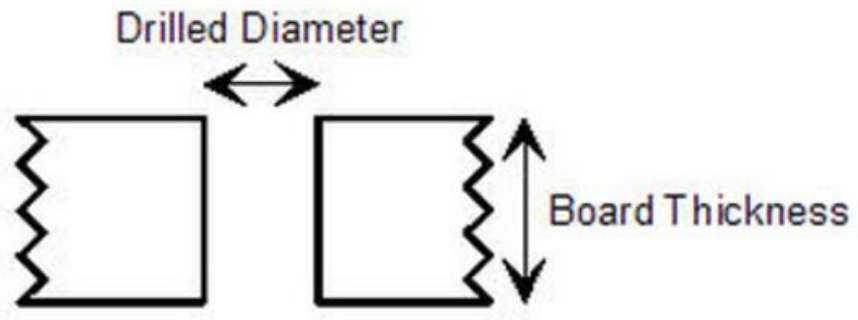
- Aspect ratio
- Final enclosure

Aspect Ratio

- The thickness of a PCB divided by the diameter size of a drilled hole
- Importance relates to the plating process
  - Used as a guide to ensure that a manufacturer does not exceed the mechanical capability of the drilling equipment or the available drill sizes
  - Plating solutions must flow through the hole efficiently in order to achieve acceptable copper plating
  - Hole sizes that are small compared with the board thickness can result in unsatisfactory plating
- The larger the aspect ratio, the more difficult it is to achieve reliable plating

Aspect Ratio (cont'd)

- Example: 0.7mm finished hole size in a 1.57mm PCB
  - 1.57mm / 0.8 mm is 2:1 regarded as a very safe ratio
  - NOTE: the drill diameter size used to calculate the ratio is 0.1mm larger than the finished hole size this is to allow for plating down the barrel of the hole
    - This value to will vary between manufacturer
    - All manufacturers should be able to achieve an aspect ratio of up to 8:1
- Very small holes in thick boards (10:1 ratio) are possible



Aspect Ratio (cont'd)

- In today's industry, standard aspect ratio is 10:1, that is
  - If board thickness is 1.57mm, hole size must be >=0.157mm
  - If board thickness is 2.0mm, hole size must be >=0.20mm
  - If board thickness is 3.0mm, hole size must be >=0.30mm

- Its recommended using 0.2mm vias or larger as a standard practice, this will help keep cost down, reliability up, and ease the pains of manufacturing

Copper Weight

- Many designs require specific copper thicknesses to accommodate the current requirements of the design
- Copper weight – the ounces of copper present in one square foot of area
  - Indicates the overall **finished** thickness of copper on the layer
  - Oz/ft^2

Trace & Space

- Copper spacing is the minimum air gap between any two adjacent copper features
- Trace width – the minimum width of a copper feature, usually traces
- Proper distances between PCB traces are critical to avoid arcing between electrical conductors
- There is a variety of industry and safety standards that prescribe different spacing requirements depending on the voltage, application, and other factors
- The minimum space between conductors is determined by the peak voltage difference
- Designs that must meet UL approval will have more controlled spacing requirements
- Humidity and pollution may also affect performance, so it may be wise to maintain extra clearance if the design will allow it

Trace & Space

- Electrical spacing rules become significant from a product safety perspective when the normal operating voltage is greater than 30VAC or 60VDC
- Creepage – the shortest path between two conductive parts, or between a conductive part and the bounding surface of the equipment, measured along the surface of the insulation
  - the parameters for pad-to-pad, pad-to-trace, trace-to-trace spacing and any spacing rule that is applied between conductive elements over an insulating surface is in regards to creepage
- Clearance – the shortest path between two conductive parts, or between a conductive part and the bonding surface of the equipment, measured through air
  - Commonly used to describe all spacing

Clearance vs Creepage

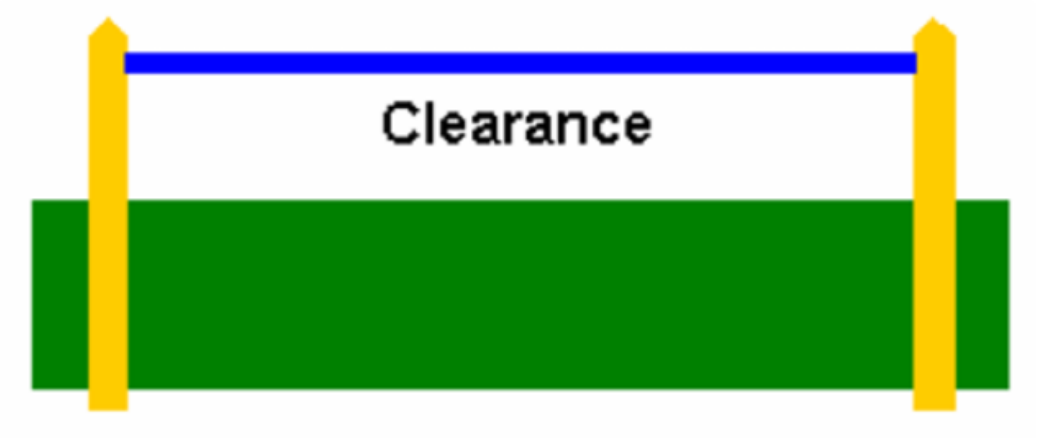
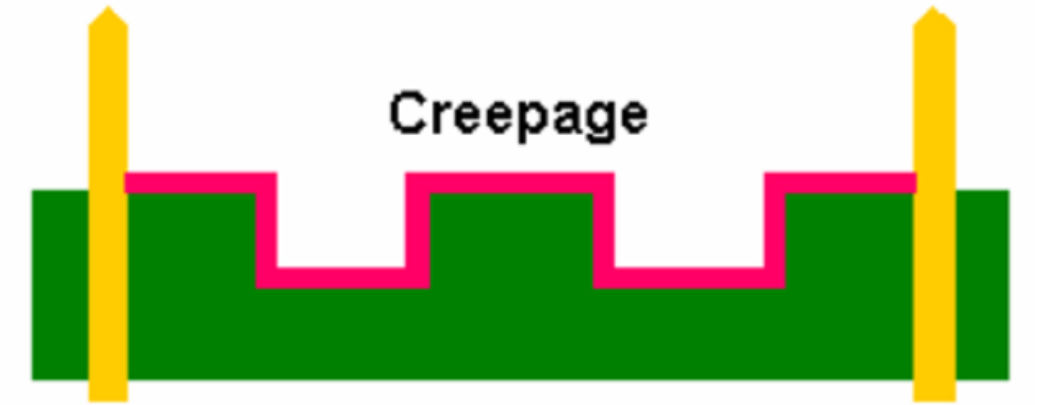


Figure 1: Clearance (In Air)



Implications of increasing the copper weight in regards to trace & space

- During manufacturing, as the copper weight is increased, the minimum trace and space also has to increase
  - Due to it's more difficult to etch through higher copper weights with precision
    - Ex: its more difficult to cut through 30 pieces of paper using scissors than it is to precisely cut through a few – there will be a greater variance on the 30 pieces than on a few
  - Typical trace and space requirements are below:
    - 1 oz. finished (5 mil trace/5 mil space) but can go to 3/3
      - 8 mil pad to pad spacing
    - 2 oz. finished (8 mil trace/8 mil space) but can go to 6/6
      - 9 mil pad to pad spacing
    - 3 oz. finished (10 mil trace/12 mil space) but can go to 8/10
      - 10 mil pad to pad spacing
    - 4 oz. finished (12 mil trace/14 mil space)
      - 11 mil pad to pad spacing

    *Solder mask dam information (only for green mask, for all others add another 1 mil to the information above)

Questions
Thoughts
Ideas