MCL Bootcamp

Classification of PCBs & Their Uses
Through-hole vs Surface Mount Technology

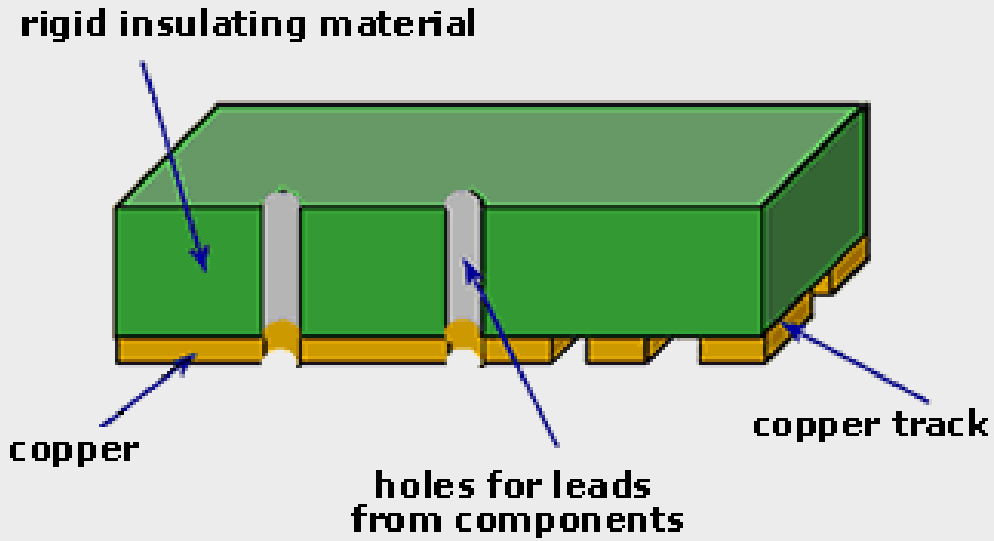
Classification System

- Based on the number of planes or layers of wiring, the presence or absence of plated-through-holes
- Advantage to this system is that is directly related to the board specification
- 3 Categories
  - Single-sided boards
  - Double-sided boards
  - Multi-layer boards

Single-Sided PCBs

- Wiring is only on one side of the insulating substrate
- Side that contains the circuit pattern is called the ‘solder side’
- Other side is called the ‘component side’
- When are they used?
  - Simple circuitry
  - When manufacturing costs need to be kept to a minimum
- Represent a large volume of PCBs

Single-Sided PCBs

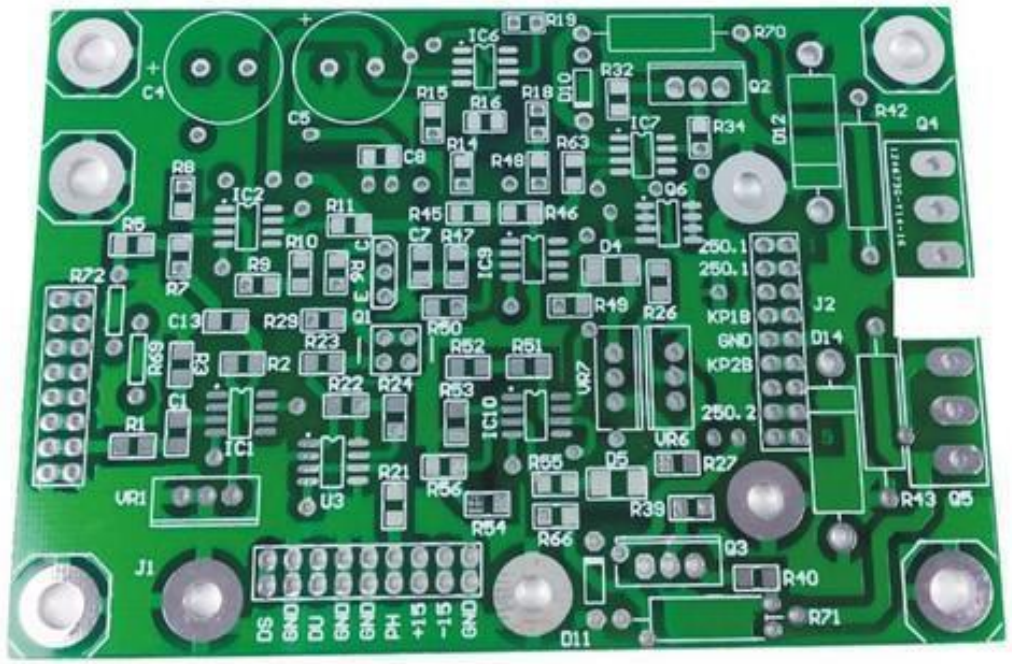


Single-Sided PCBs

- Manufactured mostly by the ‘print and etch’ method or by the ‘diecut’ technique by using a die that carries an image of the wiring pattern an the die is either photo-engraved or machine-engraved
- Normally components are used to jump over conductor tracks, but if this is not possible, jumper wires are used
  - The number of jumper wires on a board cannot be accepted beyond a small number due to costs, which resulted in the need for double-sided boards

Double-Sided PCBs

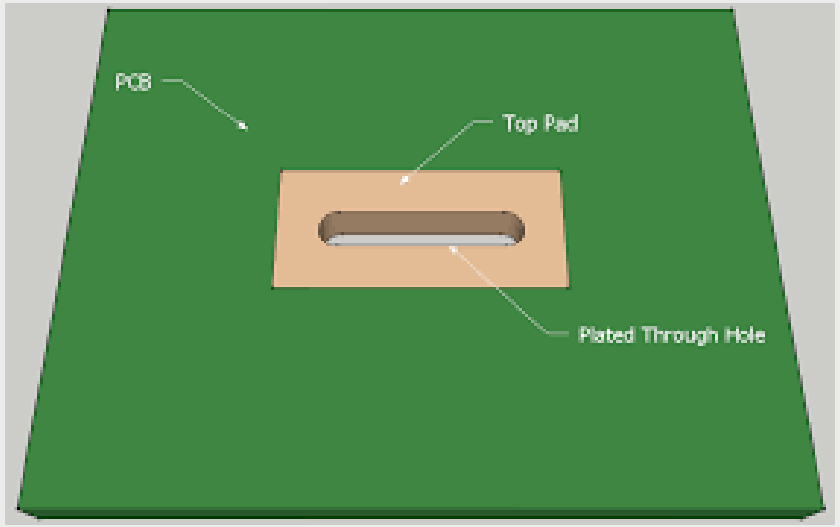
- Boards with circuitry on both sides of the insulating material
  - The circuit pattern is available on both the component side and the solder side
- The component density and the conductor lines are higher than the single-sided boards
- May have both
  - Plated through-hole connection (PTH)
  - Non-plated through-hole connection (non-PTH)

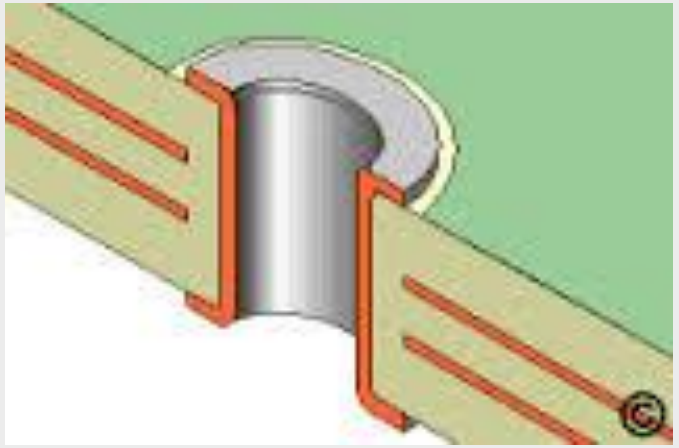


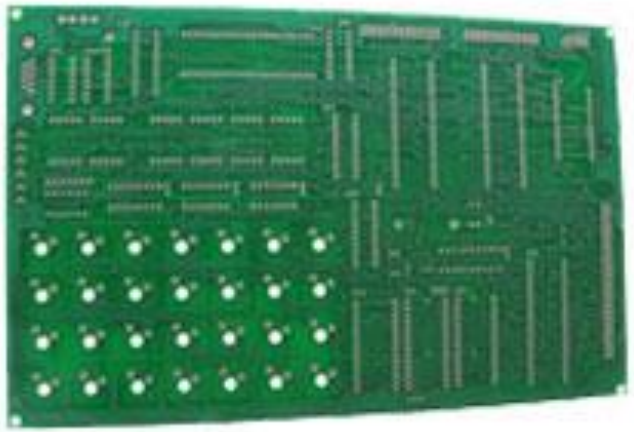
Double-Sided PCBs with Plated Through-Holes

- Circuitry is connected by metallizing the wall of a hole in the substrate that intersects the circuitry on both sides
- Most PCBs produced use this technology
  - Popular where the circuity complexity and density is high

Double-Sided PCBs







Double-Sided PCBs Without Plated Through-Holes

- An extension of a single-sided board
- Cost is considerably lower because plating can be avoided
- Through contacts are made by soldering the component leads on both sides of the board, wherever required
- Interconnection is made by a jumper wire
  - A formed insulated solid lead wire is placed though the hole, clinched and soldered to the conductor pad on each side of the board
  - Different types of eyelets can also be used for interconnection

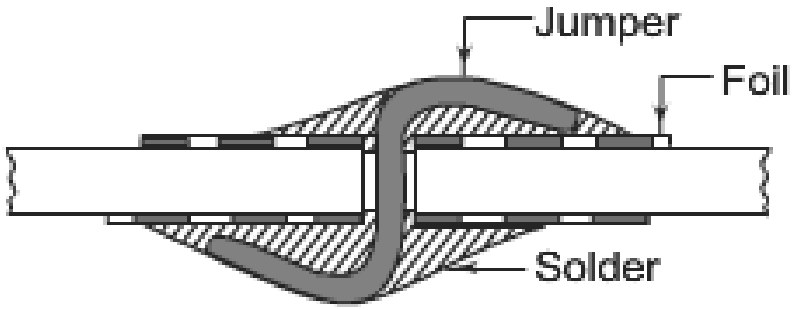


Fig. 1.3 Interconnection with clinched jumper

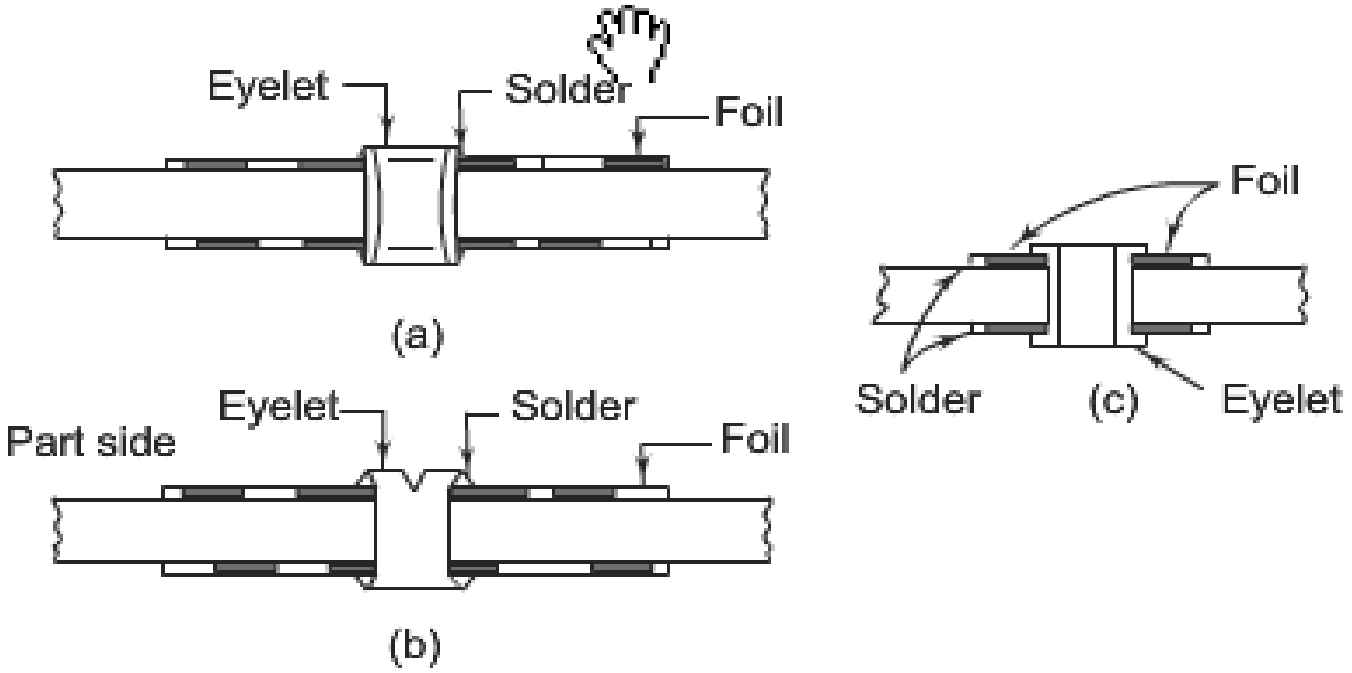


Fig. 1.4 Interconnections with (a) funnel-flanged eyelet (b) split funnel-flanged eyelet (c) fused-in-place eyelet

Multilayer (MLB) PCBs

- Boards with circuity on both sides, plus layers of circuits inside the base material
  - Used in situations where the density of connections needed is too high to be handled by two layers or where there are other reasons
    - Accurate control of line impedance
- Further defined by the number of layers
  - As few as 3 to as many as 60 (or more)

Multilayer (MLB) PCBs

- Makes use of more than two printed circuit boards with a thin layer of prepreg material placed between each layer
- The printed circuit on the top board is similar to a conventional printed circuit board assembly except that the components are placed much closer to avoid having many terminals which necessitates the use of additional layers for the required interconnections
  - The electrical circuit is completed by interconnecting the different layers with plated through-holes, placed traverse to the board at appropriate places

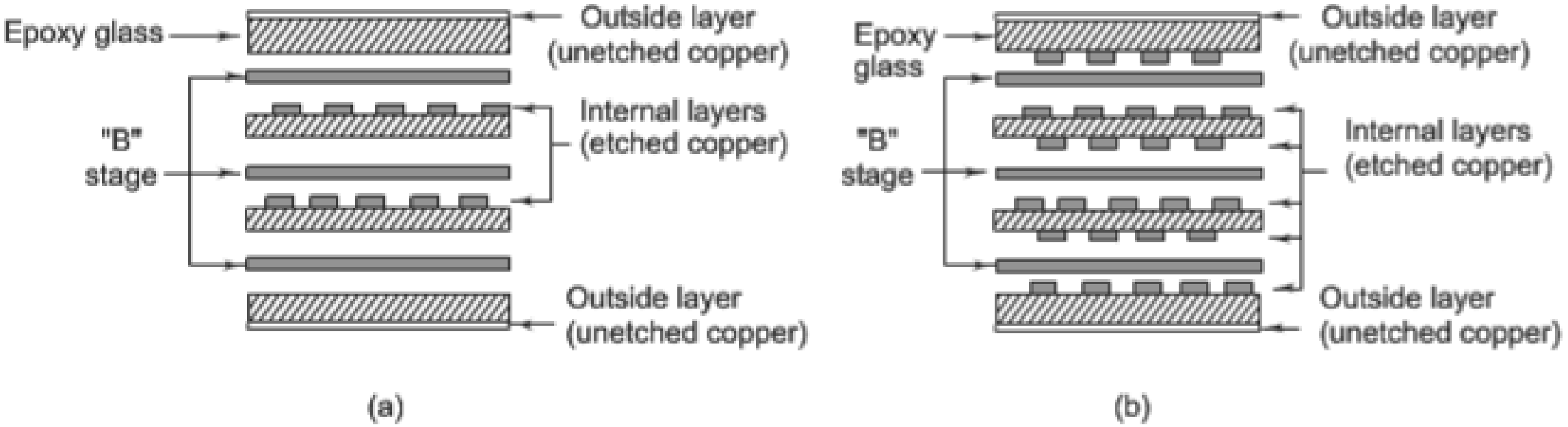


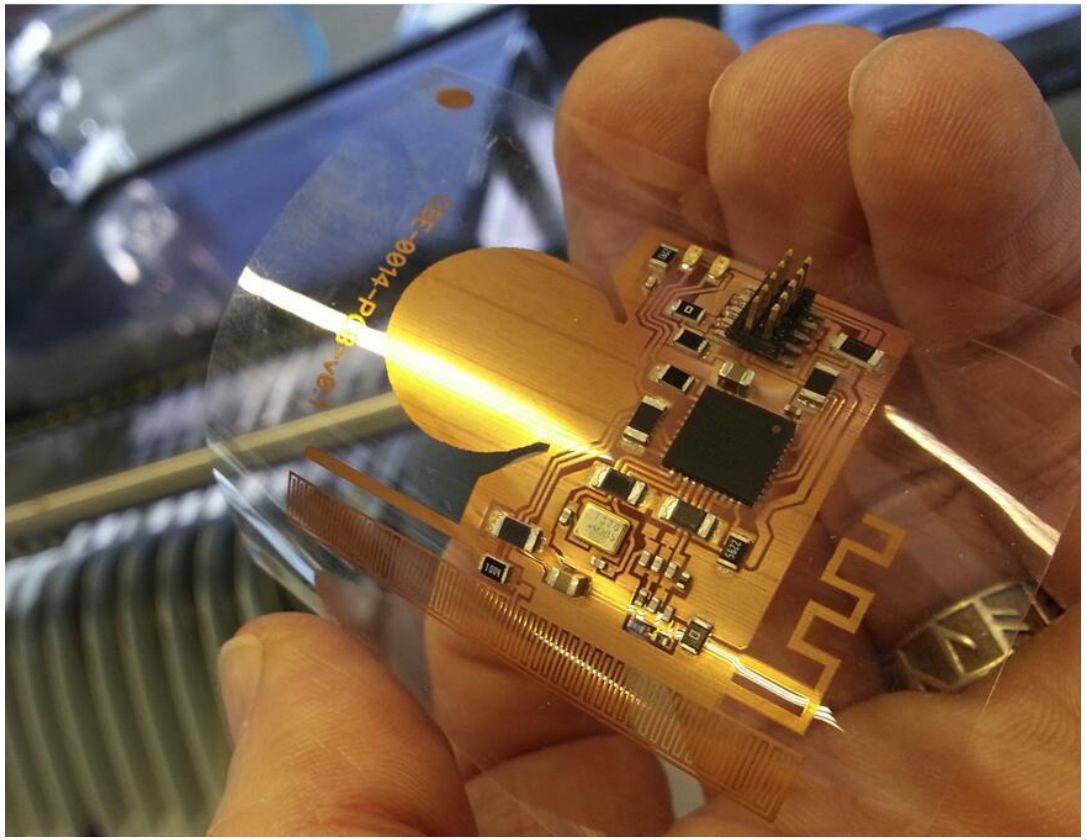
Fig. 1.6 Multi-layer lay-up details (a) four-layer board (b) eight-layer board

Multilayer (MLB) PCBs

- When are multilayers necessary?
  - When weight and volume savings in interconnections are a major concern such as with military or aerospace applications
  - When the complexity of interconnection in the subsystems requires complicated or expensive wiring or harnessing
  - When frequency requirements calls for careful control and uniformity of impedances
  - When coupling or shielding of a large number of connections is necessary
- With the development in mass lamination technology, 4- and 6-layer boards can be made with almost the same ease a double-sided boards
- With the improvement in reliability and reduction in cost, the use of multilayer boards is no longer limited to high technology

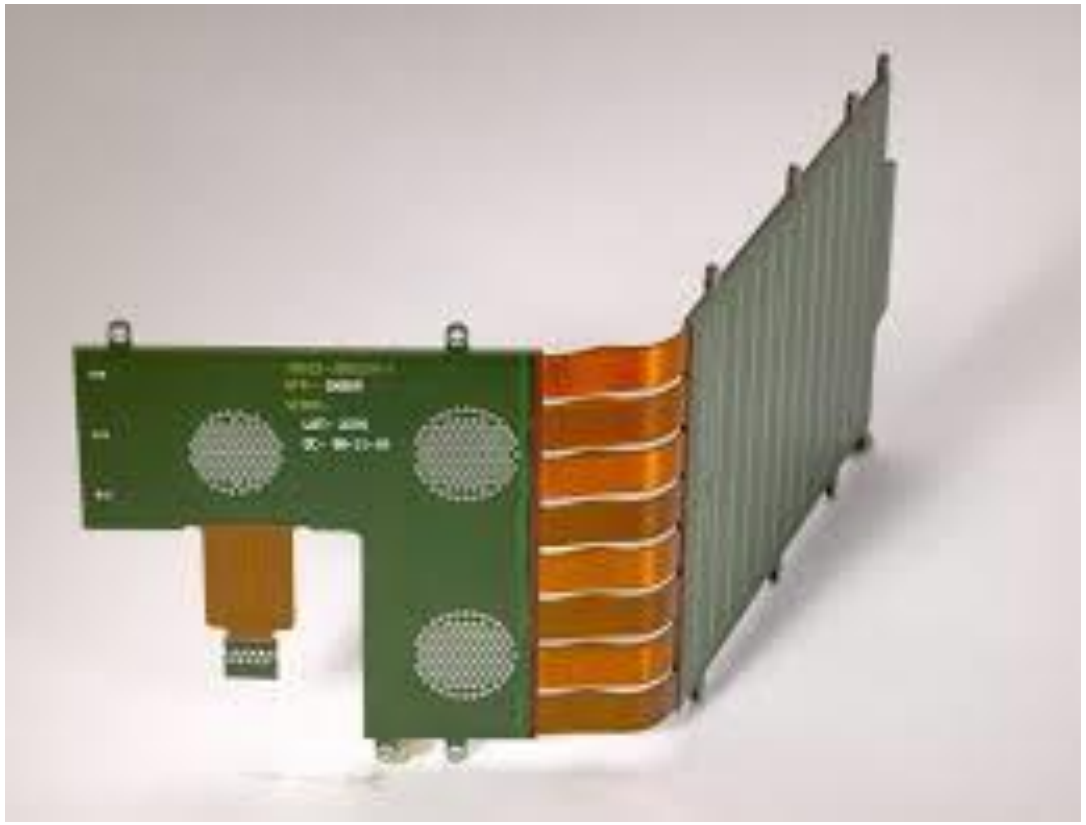
Rigid vs Flex Boards

- Boards can also be classified based on the type of insulating substrate used
  - Rigid or flexible
- Flexible boards are made out of polyamide or polyester
  - base material is very thin
  - Only make-up about 15% of worldwide market



Flexible PCBs

- May be single-sided, double-sided (PTH or non-PTH) or multilayer
- In its purest form, a flex circuit is a vast array of conductors bonded to a thin, dielectric film
- Benefits
  - Reduced wiring errors
  - Elimination of mechanical connectors
  - Unparalleled design flexibility
  - Higher circuit density
  - More robust operating temperature range
  - Stronger signal quality
  - Improved reliability and impedance control
  - Size and weight reduction



Rigid Flex Boards

- Constitute a combination of rigid and flexible boards usually bonded together
- 3D structures that have flexible parts connecting the rigid bards, which usually support copper components
- When used?
  - Need volumetrically efficient packaging
- Gaining use in electronic equipment
- Most rigid flex boards consist of multiple layers of flexible circuit substrates attached to one or more rigid board externally and/or internally, depending upon the design
- Designs are more challenging as designed in a 3D space

Mounting the Components

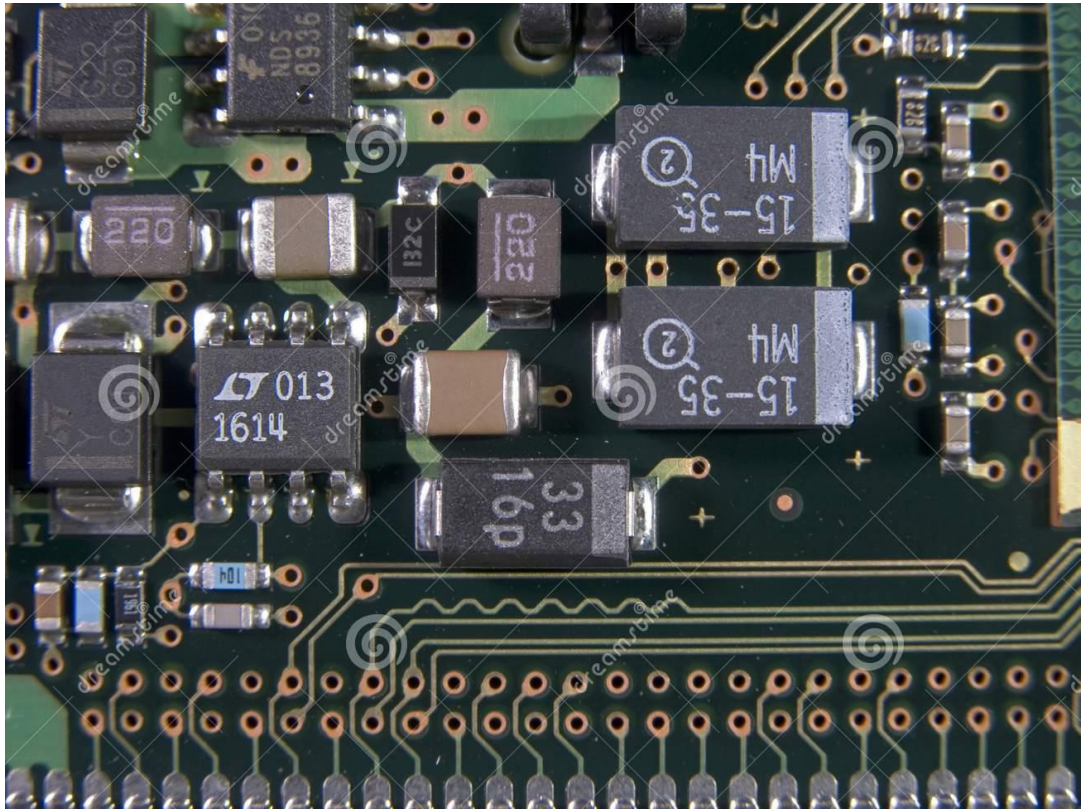
- Components provide the brains that activate and monitor the electrical current that passes through the board
- Come in many different shapes and sizes depending on the purpose for which they are used
- Components are mounted in 2 ways:
  - Plated through-hole technology (PTH)
  - Surface-mount technology (SMT)

Plated Through-Hole Technology

- The first PCBs used through-hole technology
- Mounts electronic components by leads inserted through holes on one side of the board and soldered onto copper traces on the other side.
- Single-sided boards have an unplated component side
- Double-sided boards can have components soldered on both sides
- Replaced all earlier electronics assembly techniques
- From the second generation of computers in the 1950’s until surface-mount technology became popular in the late 1980’s, every component on a typical PCB was a through-hole component

Plated Through-Hole Technology

- Adds to manufacturing costs by requiring so many holes to be drilled accurately, limits area for traces on layers immediately below the top layer on multilayer boards since the holes must pass through all the layers to the opposite side



Surface-Mount Technology

- Emerged in the 1960's but gained momentum in the 1980's
- Became widely used in the 1990's
- Components designed to have small metal tabs or end caps that could be soldered directly onto the PCB surface, instead of wire leads to pass through holes
- Components became smaller and placement on both sides of the board became more common than with through-hole mounting
  - Allows for smaller PCB assemblies and much higher circuit densities
- High degree of automation in assembly, reduces labor costs and increased production rates

Questions