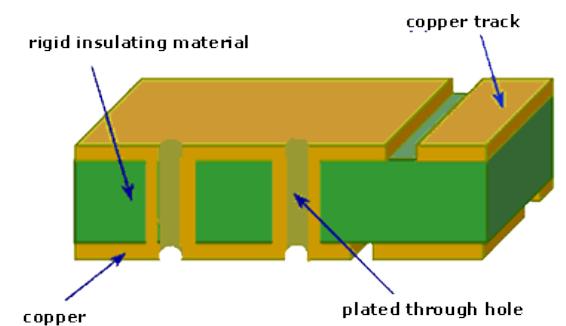
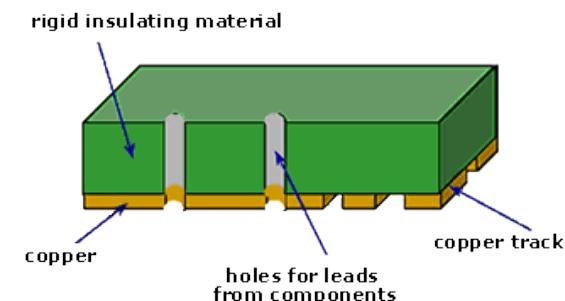


CpE 2303L
CpE Drafting and Design

Mechanical Design Considerations

Mechanical Design Considerations

- **Types of Boards**
 - Single-sided boards
 - Mostly used in applications where costs have to be minimum.
 - Double-sided boards
 - Can be made with or w/o PTH (Plated Through-Hole). Since PTH boards are expensive, they are chosen where circuit complexity and density necessitate their use.



Mechanical Design Considerations

- In the layout design, the conductors on the **component** side must be kept minimum in number to ensure sourceability.
- In PTH (Plated-Through Hole), via holes should be utilized only for through contacts and not for component mounting. The number of via holes should be kept minimum for economical and reliability reasons.

Mechanical Design Considerations

- The following must be considered in choosing how many sides/layer the PCB must have:
 - Component surface area (C)
 - Total PCB area (S)

Mechanical Design Considerations

Board Type	Single-sided	Double-sided PTH
Discrete components (ICs no more than 5% of the area)	2-3	1.5-2
Mixed (35 to 50%)	2.5-4	2-3
IC board (discrete components no more than 20 %)	4-6	2-3

The table shows the usual range of the ratio **S:C** for the most common types of PCB.

Main Mechanical Considerations

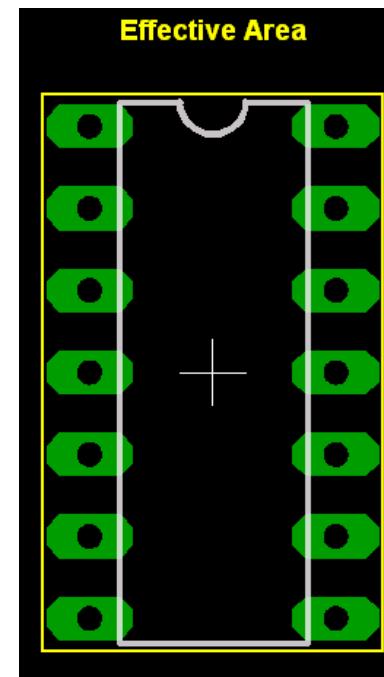
- Optimal board size compatible with the PCB manufacturing process;
- Position of board mounting holes, brackets, clamps, clips, shielding boxes and heat sinks;
- Proper fixation for heavy components;
- Proper hole diameter;

Main Mechanical Considerations

- Assembled board to withstand the mechanical stress and vibrations occurring in transportation;
- Type of installation of the board (vertical/horizontal);
- Method of cooling and
- Specific geographical requirements of components like front panel operated components such as push buttons, variable resistors etc.

Determining the Component Area

- The component area on the board is calculated by adding the area of each component.

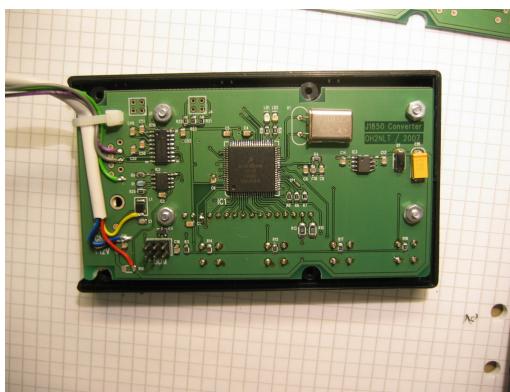
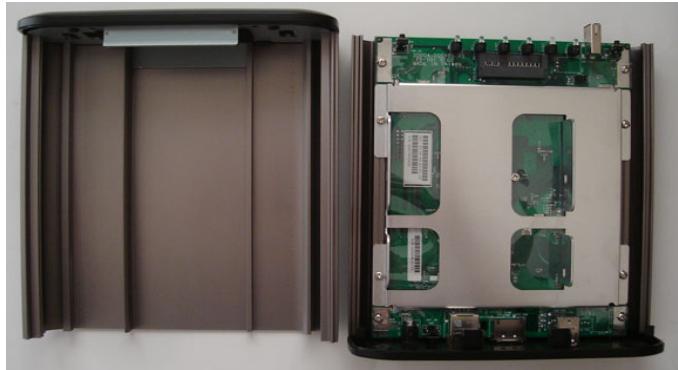


Area of DIP component



Area of an axial component

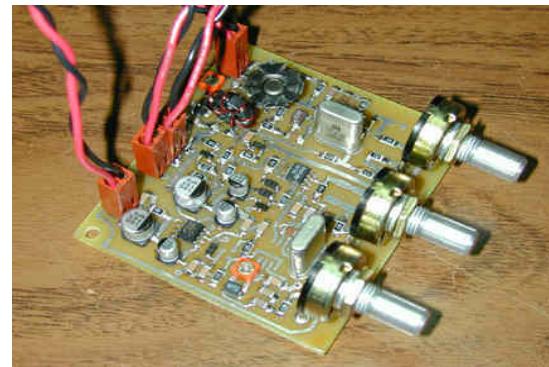
Volume Computing



- Unless there is a mandatory requirement to use a standard enclosure, the enclosure or casing should be designed to fit the system to avoid forcing the system in to an enclosure size that may dictate the use of a non-standard or non-optimal board size.
- It is important to know the maximum volume that the board can occupy in the worst condition including safety clearances.

Accessibility of Adjustable Components

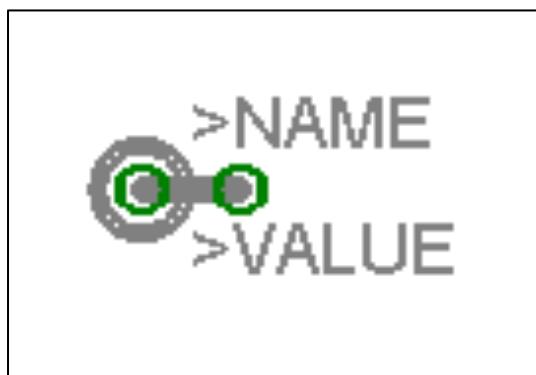
- Adjustable components like variable resistors are common in many printed circuit board assemblies. These components should be mounted on a PCB in such a manner that there is an easy access to such components.



Horizontal/Vertical Mounting of Components



Horizontal Mounting



Vertical Mounting

- Axial components can be mounted vertically and horizontally. However, vertical mounting requires less surface area which reduces the PCB size.
- Vertical mounting has several disadvantages like:
 - lower reliability
 - Increased difficulty in component forming and manual assembly
- Therefore, vertical mounting should be adopted only when area is limited and there is unlimited volume.

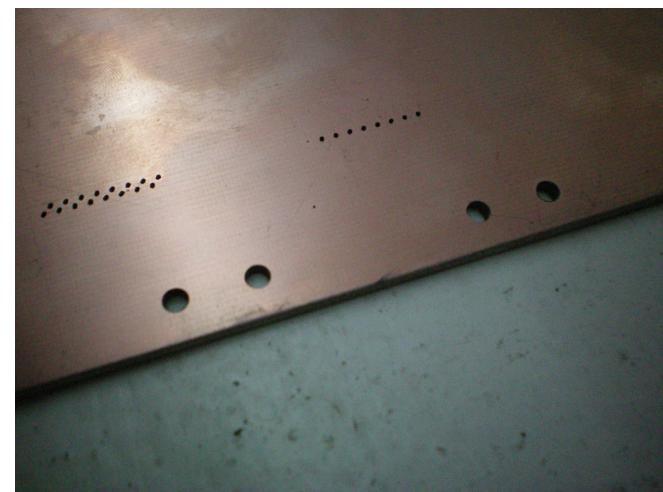


Board Size

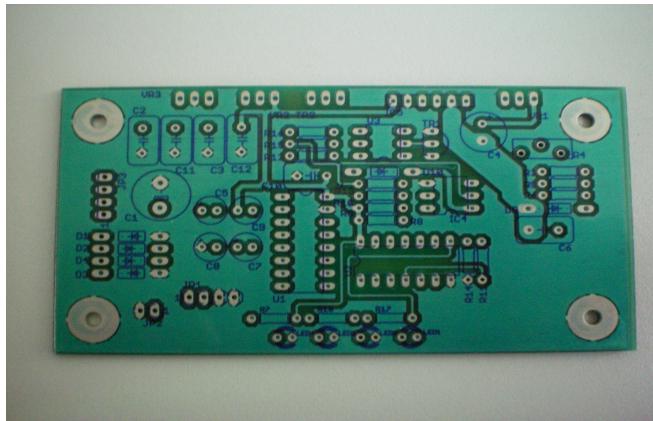
- Factors that affect board size:
 - casing or enclosure
 - require height restrictions
 - tooling holes and keep-out areas for assembly and manufacturing

Geometrical Constraints

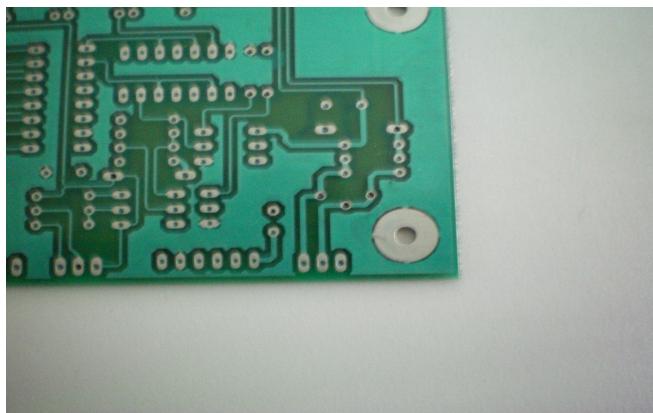
- Locating Holes (for assembly)
 - Preferred diameter is 3.17 mm
 - Close to the edge of the board w/ the widest possible span
 - Center should be 1.5 mm times the hole diameter from the edge of the board, but not less than 2 mm
 - Safety area should not be of circular form



Accessibility of Adjustable Components



PCB with four mounting holes.

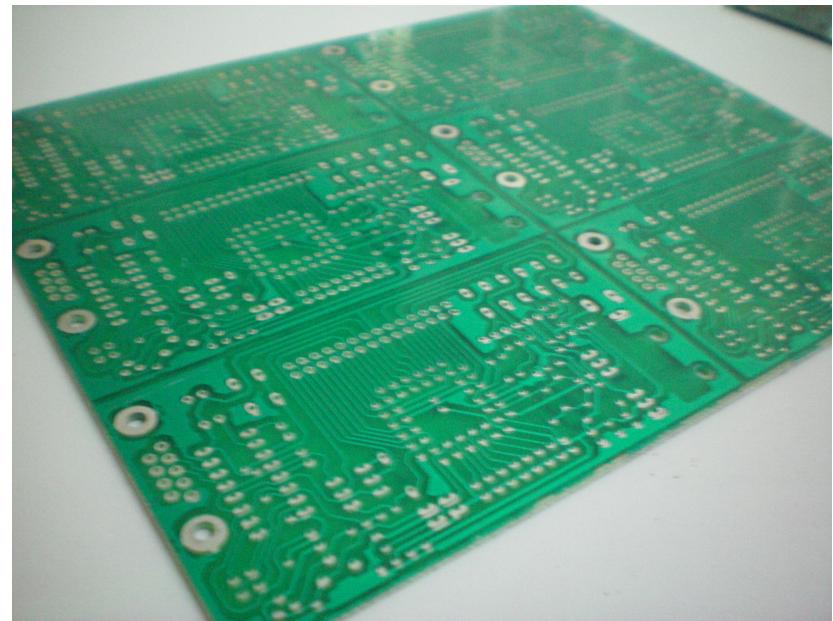


Sufficient edge clearance.

- Mounting Holes – safety areas need to be provided around the fixing holes, keeping in view the size of the washers and screws, unless insulating washers are used.
- Edges – Usually a 2-5 mm safety strip is provided along edges. This is necessary because trimming of the board could cause de-lamination of copper areas to close to the edge.

Dimensions and Tolerances

- The most commonly adapted dimensional tolerances are +0.25 mm and +0.50 mm
- Dimensional tolerances are needed to reduce manufacturing cost by allowing smaller panels on a larger board.



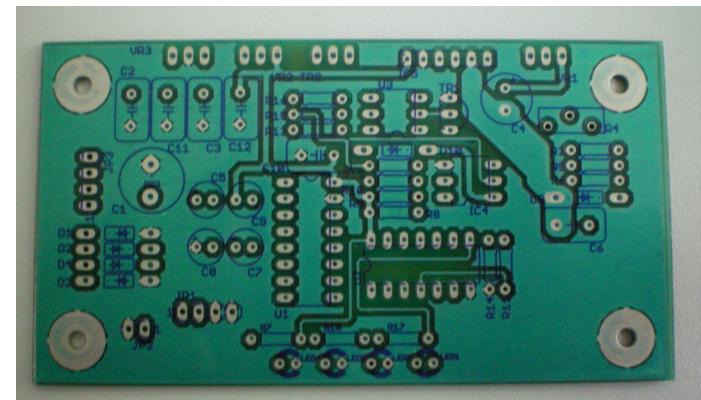
**Large PCB with several panels.
(out side manufacturing
clearance cut-off)**

Partitioning

- Partitioning is splitting the circuit into two or more boards.
- Advantages of partitioning are:
 - Easier layout design
 - Manufacture testing and maintenance
- It also allow boards to be manufactured on a small-scale manufacturing plants (ex. USC's PCB Prototyping Lab) in which the machines are only capable of a maximum board size of 200 mm x 300 mm.

Board Mounting Techniques

- There are several techniques in mounting the PCB to the chassis or to the next assembly.
- To provide good mechanical stability, the board should supported within 25mm of the board edge on at least three sides.
- As general reference, boards between 0.031 inch and 0.062 inch (0.785 mm and 1.57mm) thick should be supported at intervals of at least 10 cm.

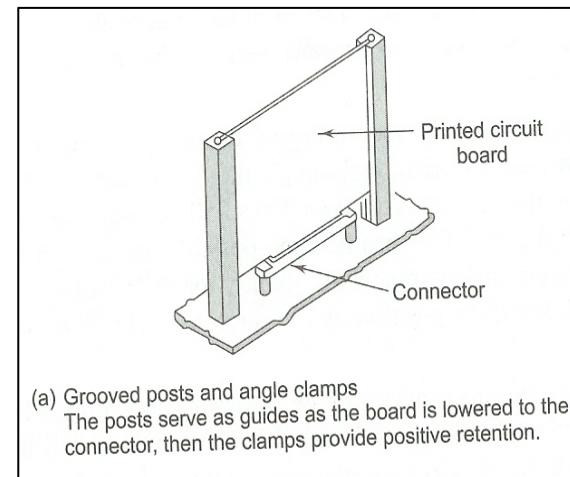


Board Mounting Techniques

- The choice of board mounting technique would depend upon the following factors:
 - Board size and shape
 - Input/output terminations
 - Board removal requirements
 - Heat dissipation requirements
 - Required shielding
 - Type of mounting hardware
 - Available equipment space and
 - Type of circuit and its relation with other circuits.

Board Guiding and Retaining

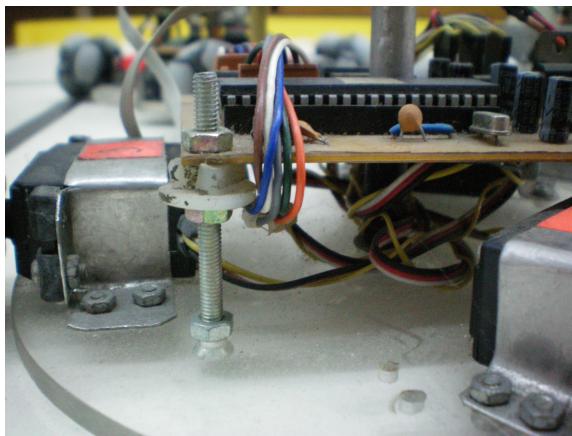
- The most convenient method for use with plug-in circuit assemblies is that of card guides.
- It provide a quick connect/disconnect capability.



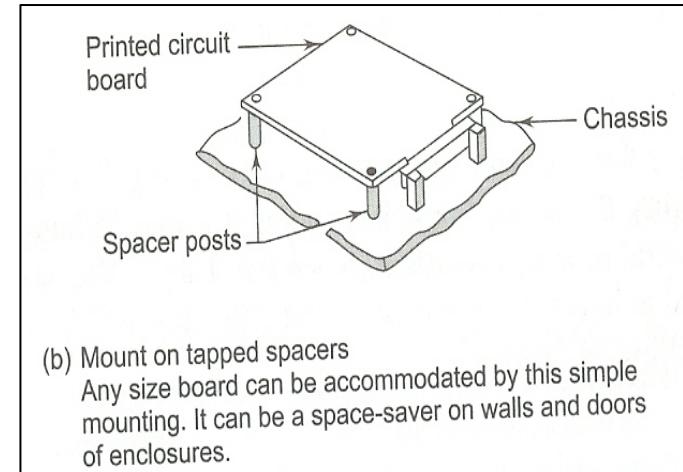
An example of a card guide PCB.

Board Guiding and Retaining

If electrical interface does not require a connector or a card guide is not practical, then mounting holes may be provided on the board so that it can be installed with screws, Stand-offs or other mechanical fasteners.

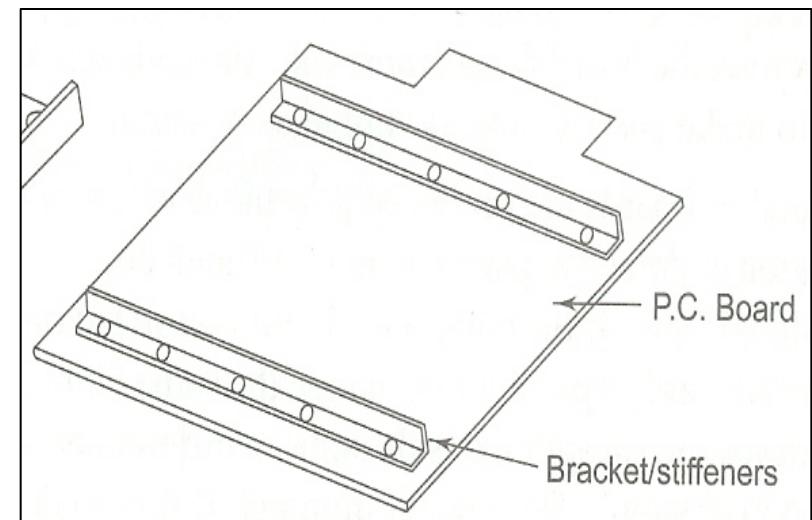
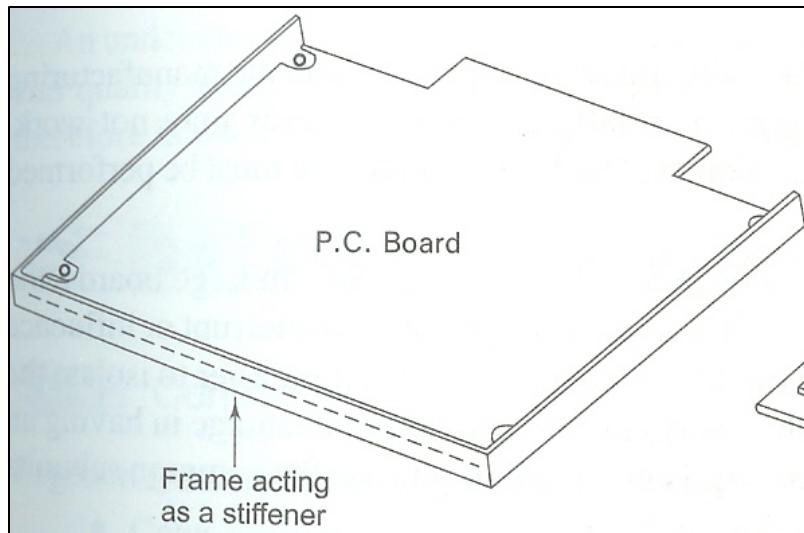


PCB with screw separator.



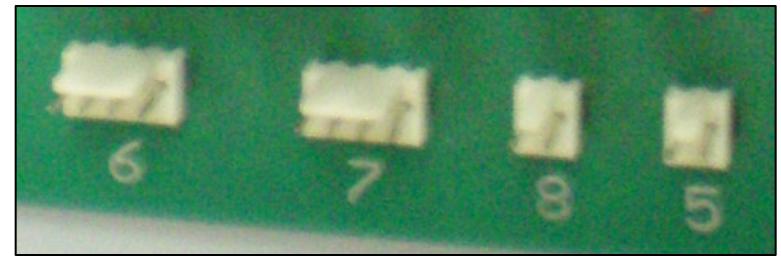
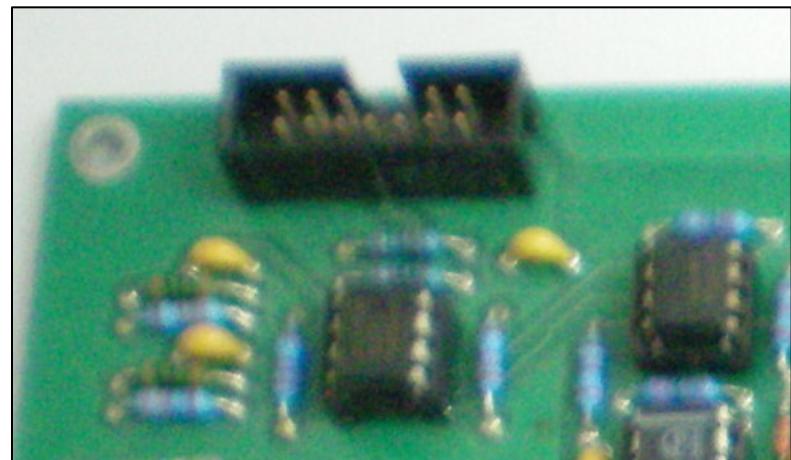
Board Guiding and Retaining

- Stiffeners – used to ensure that the board remains flat under mechanical stresses.



Input/Output Terminations

- The most common method of providing electrical interface between the PCB and the associated equipment is by the use of connectors, terminals and cables.
- The type of interface to be used for any particular board is generally decided during the mechanical design of the equipment.



Connectors of Laser Photoplotter PCB.

Testing and Servicing

- Depending upon the complexity, the quality of incoming components and manufacturing process, there is a likelihood that a certain number of assembled boards may not work.
- Therefore, the board design must take into consideration the level at which the test must be performed and to make such testing as simple as possible.



Mechanical Stress

- Small size PCB of less than 100 mm x 150 mm and the standard thickness of 1.6mm will hardly pose any problem in mechanical strength if assembled w/ the usual electronic components.

Board Thickness

- There is no standard rule for the optimum thickness neither for the printed wiring nor the number of multilayer conductive layers.
- The limiting factor for printed wiring thickness is the diameter of the smallest hole, especially when the holes are plated though.

Important Specification Standards

- ANSI/IPC 2221 “Generic Design” – identifies the generic physical design principles as well as the material selection.
- IPC-6012 – specifies the performance requirements of the finished rigid printed boards.

CpE 2303L
CpE Drafting and Design

End of Lecture

This material is prepared by Van B. Patiluna with contents from the reference textbook and other sources.

References:

R.S. Khandpur. Printed Circuit Boards: Design, Fabrication, Assembly and Testing. New York: McGraw-Hill, 2006.