CAD 1203  
Chapter 7  
Computer Generated Printed Circuit Boards

**Required Printed Circuit Drawings**
- Logic or schematic drawing
- Layout drawing
- Artwork
- Drill and trim drawing
- Assembly drawing
- Optional drawings (silkscreen/solder mask)

**Design Data**
- In addition to the schematic and the components list, the following additional information must be provided to the PCB designer.
  - Voltage and amperage rating
  - Heat sink requirements for heat dissipating components.
  - Grounding requirements
  - Board size and mounting requirements
  - Layer requirements
  - Schematic/logic diagram

**Designing the PCB Board**
- Primary Considerations
  - Components size.
  - Method by which components will be mounted to the board.
  - Size necessary for mounting holes.
  - Diameter of solder pads
- (Solder pads are used to provide a surface for the technician to attach the components of the circuits. Size is dictated by the diameter of the component lead wires)

**Conductor Spacing Standards**

**Dedicated Electronic Circuit Software**
- Design tool
- A library of symbols allows the designer to quickly add components from the library to the schematic
- As the designer/drafter creates the schematic, the software aides in generating PCB layouts.
Software for Designing PCBs

- http://www.expresspcb.com/
- http://zone.ni.com/devzone/cda/tut/pid/6880#toc0
- http://www.pad2pad.com/
- http://www.lvr.com/pcbs.htm

Software for Designing PCBs

- Just as a designer will add lines to a mechanical drawing, the designer will add wires in the pcb design software.

A symbols and parts library is generated by the software as the pcb board is designed and will contain all of the necessary attributes for components.

PCB Design Software

Component Attributes

- Part Name/ Part Number
- Library Reference Number
- Part Description
- Number of Pins
- Cost
- Footprint, (dimensions, shape, size)
- Symbols
- Power Pin Information
- Ratings, such as capacitance, and resistance
- Reference Numbers, (R1, C1 etc...)

14 Pin NAND Gate

Symbolic Representation from Component Library

Component Library Dimension Data
Printed Circuit Boards

- The size and shape of the board will be determined very early in the design process.
- As the distribution of components is planned a decision on the number of layers required must be determined.
- The PCB layout software will work from the list of required components, (Net list) to develop the appropriate interconnections.

Printed Circuit Board Substrate

- Each layer is a thin sheet of substrate.
- Layers are composed of an insulating material with a conductive material bonded to it.
- The conductive layer is etched to create the electrical routing.
- When connections are required between layers, a via, (plated through hole) is added.

Routing the Board

- The term routing refers to establishing the conductive paths between components.
- The routing software will generate the conductor lines for the pcb.
- The software will interpret the schematic and logic diagrams and establish the interconnections between components.

Routing the Board

- Routing will be done over a grid which has a spacing of 25 mils in both directions.
- The majority of components should fit on this grid spacing.
- When it cannot be avoided it is acceptable to locate components outside of this grid.

Component Placement

- The designer must be cautious in choosing where they locate the required components.
- Components that have many interconnections to each other should be located as close to each other as feasible.
- Components should be located in a manner that results in the interconnections being as close to vertical or horizontal as possible.

Component Placement

- The components should be evenly distributed over the area of the pcb.
- An optimally designed circuit board will have the components located in an arrangement that will result in the interconnections requiring a minimal amount of surface area of the board.
Component Placement

Jumbled mess with many long tangled connections

Component Placement

Same application as the previous slide. U3 has been moved to make connections more direct

Component Placement

Notice how moving U3 to the left of U1 reduces the length of the connection paths

Routing of the Board

- Locate the widest lines first, (this will include the power and ground lines).
- Be sure to exclude those areas of the board that should be left clear for board mounting screws, or other mechanical limitations.
- The spacing clearance values specify the line to line clearances as opposed to center to center distances.

Routing

The ground and power lines are typically the widest

Routing of Board

Areas that will contain hardware must be excluded from routing.
Routing Schedule

- The routing schedule provides the specifications for connecting the components.
- The routing schedule will show where the ends of the connecting lines are to be connected, without determining the actual routing path.

Routing System

- Routing software greatly reduces the time required by the designer to create/design a PCB.
- Icons, pulldown menus, and keystrokes should allow the user to navigate about the work area.

Layer Controls

- Layers must be established for component sides, circuit sides, the ground plane, etc…
- If the board is to be a single sided board, the designer will specify the layer to be used and turn the other layers off.

Pad and Via Type

- Vias are pads that are used to connect the inner layers of the circuit board.
- Vias may be “blind”, (embedded within the layers) or go completely through the board.
- The designer will specify the type of via within the board specifications.

Power and Ground Plane Masks

- The power or ground mask is used to make a conduction layer of a board that is dedicated to a single layer or level.
- This layer will be a solid sheet of conductor and will only be connected to those components that require a ground or power source.
- Power and ground plane masks are used to conserve space. By providing a dedicated layer for this routing, the overall footprint of the board is reduced.

Power and Ground Plane Masks

- Power and ground plane masks are used to conserve space.
- Since nearly all of the components must be connected to the power or ground, this greatly reduces the area required for routing.
- By providing a dedicated layer for this routing, the overall footprint of the board is reduced.
Printed Circuit Board Layers

Location of Components

- As the components are located on the PCB, their exact location, angle of orientation and attributes are registered in a components tables, (similar to a parts list but containing much more information)
- This table will also contain the specifics for the project, (board part number, final product description and model number)

Example Component Reference Table

Lab Assignment

- Chapter 7 review questions