* CAD (Computer Aided Design) FOR PRINTED WIRING BOARDS
* DESIGN FOR MANUFACTURABILITY

Continued..
Wiring Board Configurations

- Single-sided
- Double-sided
- Double-sided with layer interconnections – (via or plated through hole)
- Multilayer
- Flexible and Rigid-flex
- High Density PWB
Wiring Board Configurations
PCB CAD PACKAGE
INTRODUCTION and DEMO

Introduction
Any good CAD Software provides complete electronics design solution.

For Demo purpose in this course, we are utilizing OrCAD Software. We will proceed to understand how to design a typical PCB using OrCAD Capture and OrCAD Layout modules.

* Please note: you can use your own software tool at your institution and try the demo of the systematic process steps illustrated and explained here.
Any CAD Provides

- Schematic Capture
- Layout netlist
- Placing components
- Routing
- Cross probing
- Gerber, DXF, reports
- Edit Gerber

The first step is to build the schematic diagram of the circuit, and layout is used to design the circuit board. During this process, manufacturing aspects are considered without fail.
Design Flow

The Schematic Capture tool is used to create schematic using available symbols from the symbol libraries and interconnect with the wire tool. This means you are adding different components on to your board and connecting them with wires.

Generally, the following are the main categories of the components used:

* Resistors, Capacitors, Inductors
* Diodes, Transistors, FETs, LEDs
* Connectors, Headers, electromechanical components
* DIP ICs, BGA ICs and others like QFN, CSP
In Schematic the symbols which we use are IEEE symbols or ANSI symbols which will have only the symbol view with number of pins, not the actual size and dimension of the component.

For Example

\[74LS08\] Quad 2 input AND Gate

Institute of Electrical and Electronic Engineers (IEEE)  
American National Standards Institute (ANSI)

IEEE Symbol for 7408

\[\text{7408}\]

Both IEEE and ANSI symbols are functionally the same.
Some Basic Broad Process Steps:

- Collect data sheets for all components that will be used.
- Decide which footprints need to be used. Footprint is a packaging view of the component that includes the holes through your board or pads for surface mount devices. You will find SMT footprint and Through-hole footprint for components. Select the footprint that meets the mechanical requirements for the component that you have chosen.
- Attach the name of the selected footprint to the component symbol by editing the properties for each component.
- Using ‘place wire’ tool create a electrical connection between the components on the board.
- Once you complete the schematic you have to generate the net list and import it to PCB Layout to complete the board layout. Layout will automatically insert footprints into the board based on the information given earlier.
- Now place the components, define power and ground planes, route physical wires using this tool. Once the board layout is completed and routing of traces are done, we create technology files, universally called, ‘Gerber files’. These are used by the manufacturer to photoplot the masks and manufacture the board thereafter.
- Finally verify the board for errors; verification is done also at all stages.
Example of CAD process using OrCAD (v9)

In any s/w tool, the library is the heart of the tool. Library contains both schematic symbols and part libraries (footprints).

For example some components are shown here with symbol and footprint details.
- Schematic symbol for Resistor is available in symbol library of OrCAD Capture.
- Thru hole and SMD footprints are available in footprint library of OrCAD Layout.
In the case of a diode, for example, the symbol pin names are given as 1 & 2 and Cathode & Anode. But in the package footprint pad names are given as C & A, 1 & 2. In case of mismatch of pin & pad names, edit either in the symbol library or footprint library to match the reference designations.

In case of a Zener diode, the symbol pin count with footprint pad count does not match, say. Edit symbol to add missing pin by edit >part method.
In case of transistors the schematic symbols are represented in three different forms with pin names: emitter, base, collector, or C,B,E or they can be represented with numbers as 1,2, and 3 in symbol library. And they are also available as thru’ hole and SMD packages in the footprint library of the Layout module.

Figure Source: Wikimedia Commons
The schematic symbol and footprint view of a capacitor is shown below:

Surface Mount (SMD):
- 1206, 0805, 0603, 0402 (in x 100)

Dimensions:
- 60 mils
- 120 mils

Figure Source: Wikimedia Commons
The foot print information for through hole and SMD ICs are defined as below:

**Thru hole mount IC**

* DIP specifies Dual inline package
* 100 specifies pitch between the two pins
* 14 indicates the number of pins
* W and L indicates the width and length of the component.

**Surface Mount IC**

* SOG indicates surface mount gull wing component
* 50 specifies pitch between the two pins
* 14 indicates the number of pins
* W and L indicates the width and length of the component.

Figure Source: Wikimedia Commons
PACKAGE FOOTPRINT DETAILS ARE OBTAINED FROM DATASHEETS OF DEVICES SUPPLIED BY MANUFACTURERS
Working with Capture

New Project

Name:
assignment-1

Create a New Project Using:
- Analog or Mixed-Signal Circuit Wizard
- PC Board Wizard
- Programmable Logic Wizard
- Schematic

Tip for New Users:
Schematic Wizard is the fastest way to create blank schematic project.

Location:
F:\orcad video class\Demo

Browse...
Right side of your design window we can see tool palette button provides an **electrical** or **graphical function**, like adding wires and parts or placing graphic objects.
When you place a part on a schematic page, the part can be placed without a reference designator assigned to it or it can be automatically assigned.

To specify automatic reference designators Options>go to Options>preferences>select miscellaneous> Auto Reference > enable automatically reference placed parts.

Add components to the design by selecting Place >Part > choose required components from different libraries.

Or  click the Place Part tool on the tool palette
You can type a selective search string to instantly locate a specific part or search by using part search command.

Use **Place > Wire** from the place menu or click the **wire tool** on the tool palette for interconnecting components. You can cross the wires without connecting them.
To add additional schematic pages go to the Project manager window, select Schematic and right click for new page. You can name the page by the rename command.
A Schematic folder is simply a collection of schematic pages, which are logically connected by off-page connectors. Nets can be simply continued to the next page.
Editing component

Give reference designators, value and footprint information to the components by edit > component > properties. Select a component and right click the mouse you get pop up window. In pop up window select edit properties.

In the editor window
In that select PCB Footprint column and enter the name of the footprint like SM/R_1206 (this is SMD type component footprint). These footprint names are available in Library manager.

SAVE FILE
File > Save as > file name .dsn
**Netlist creation**
Using a schematic capture tool, such as OrCAD Capture, you create a layout-compatible netlist.

Go to **project manager window** select the design file and goto Tools > choose Create Netlist > in Create Netlist window select Layout and give file name and path and enable Run ECO to Layout.

If you change the netlist after back annotating in Layout and Capture, Layout’s AutoECO utility automatically updates the board file.
Introduction to Layout

PCB Design/Layout

Once the schematic is completed and approved by you, the PCB layout can be started.

The following steps are performed to create the PCB:

Opening Layout

You can start a new design from Layout’s session frame or from the design window.

Next go to Start > Layout plus > select file > New > in load template file window type the .TCH file name (It can define the board layers, default grids, spacing, track widths, design rules) and select > Open.
Load netlist

Save file

Footprint

AutoECO cannot find a footprint for component J1 from part name CON2.

Please choose one of the options below:

- Link existing footprint to component ...
- Create or modify footprint library ...
- Defer remaining edits until completion ...

OK
In case of mismatch of pin & pad names edit the symbol and use it.

Note: Before we start doing any schematic select a right symbol and Footprint to the Component. The Symbol pin name & pin count should match with Footprint pad name and pad count. If you take a diode the symbol pin names are given as 1 and 2 and Cathode & Anode. But in the Footprint pad names are given as C & A, 1 & 2. Any parts not in the OrCAD Layout standard library are created at this time. Once the footprints have been inputted, they are randomly placed on the PCB.