An Introduction to Electronics Systems Packaging

Prof. G. V. Mahesh

Department of Electronic Systems Engineering

Indian Institute of Technology, Bangalore

Module No. 05

Lecture No. 21

Design Flow considerations

Beginning a circuit design with schematic

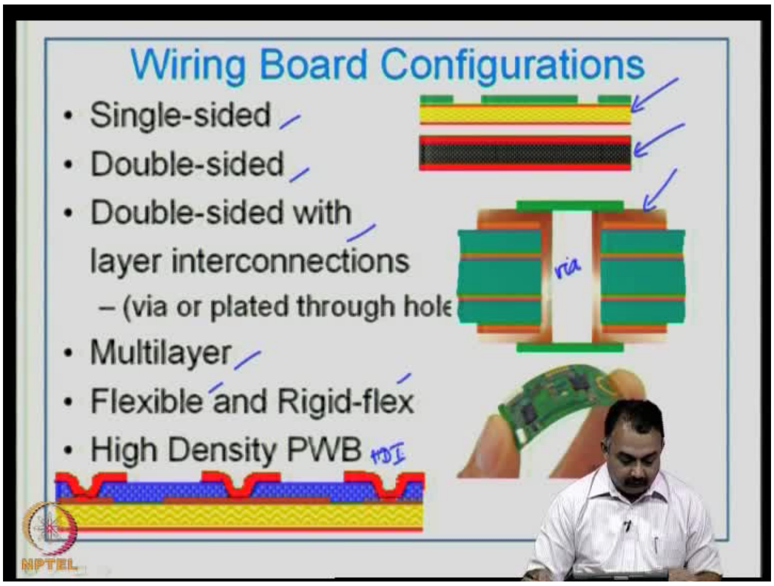
Work and component layout

This chapter is on computer aided design for printed wiring boards. We are also looking side by side into the aspects on design for manufacturing or you can say, design for manufacturability. I also said that I will brief you on the term called design for reliability.

We also briefly defined what these terms are: that is, DFM, DFR and DFT - design for testability.

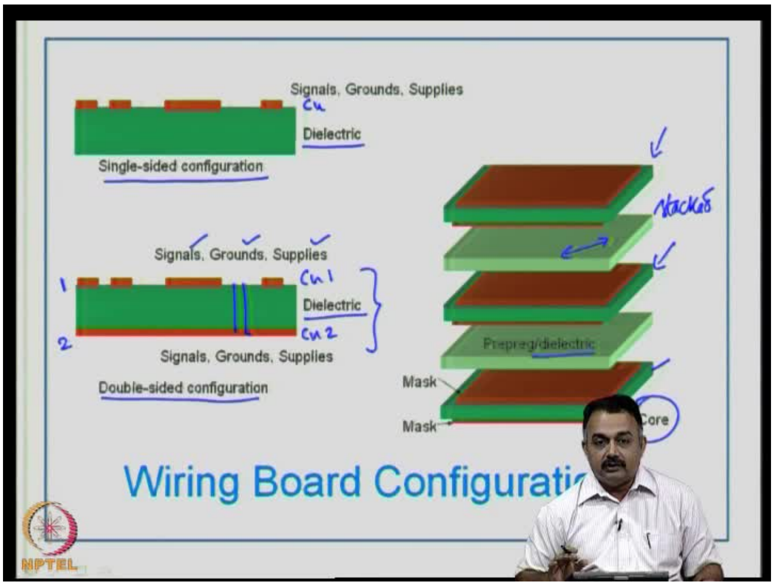
The last two hours on CAD would have given you ample information about the importance of CAD for making this second level interconnect to happen. The second level interconnect is on the printed circuit board or the printed wiring board; and essentially there are two parts to it; one is the manufacturing of the printed wiring board and it is followed by this stuffing of the components on the board. So, you have what is known as the printed circuit board assembly. Design, definitely relates to both manufacturing and assembly. We are trying to see how well we can understand the various opportunities that are available to create a well-structured board.

(Refer Slide Time: 01:50)



We have briefly seen, as you can see here, towards the end of the last class, what are the printed wiring board configurations. We saw what is meant by a single sided printed wiring board, which means that in a structure like this, there will be copper on one side and the assembly of components will be done on that particular copper side. The other side will not have copper. Double sided refers to typically, two sides of the structure, the thick laminate having copper on both sides; as you can see here, in this picture. Connections can be established between the top copper and the bottom copper; and you can use via structures, you can create 2 layers, 4 layers, 6 layers, 8 layers or a multilayer configuration, like what you see here, in the next picture. As I mentioned before this (Refer Slide Time: 03:05) is the via structure that interconnects different copper layers in the entire structure. This is a very cumbersome, complex process, which we are going to see in detail shortly. Therefore, you have a single sided board, a doubled sided board without via interconnections; double sided boards with via interconnections; multilayer board with via interconnections, between different sets of electrical layers; then you have rigid and rigid flex boards; and high density interconnect structures - HDI.

(Refer Slide Time: 04:13)



The reason for putting this figure again is that we are going to demo the CAD today, in this particular class; you will see that there will be opportunities for you to design any of these kind of structures during your design work. As a designer, you must also be aware of the stacking of electrical layers, as you see here, in this picture. I have briefly shown this picture in the last class also.

What it basically means is that in a single sided configuration is like this, you will have a dielectric layer and you will have a copper layer, in which the signals the ground and the supply lines will be designed. Because it is a single sided board, the area of the board might be larger.

In a double sided configuration, you have dielectric structure, as usual, which separates the copper 1 layer and the copper 2 layer here; and you can see you are able to spread the signal lines, the ground and the supply lines between the two electrical layers; the top and the bottom. Of course, you can interconnect these two by using via structures and plating those via structures to establish reliable copper interconnections.

The figure here basically shows that in a multilayer structure, you will have possibilities of having these kinds of core. If I call this as one core structure; you can have a core structure like this; another one here; and another one here and these (Refer Slide Time: 06:02) can be stacked and separated by these dielectric materials; what you see here is a dielectric material; it is also known as a prepreg.

When you start with your design work, you need to understand what kind of board you are making; and more importantly feature sizes, for example, you should know what is the finished thickness of your board, whether it is a single sided board or a double sided board or a multi layered board; what is the finished thickness of the entire structure and what are the individual thicknesses of each of the cores that you have built the structure with; the thickness of copper in each layer; the thickness of dielectric in each layer, separating the two coppers; and these kind of information need to be passed on to the manufacturer.

Therefore I wish to emphasize here again that DFM not just from these perspectives; there are more angles to looking at design for manufacturing because finally, after all, you require a reliable board that has been manufactured with good tolerance, with good specifications and reliability.

(Refer Slide Time: 07:25)

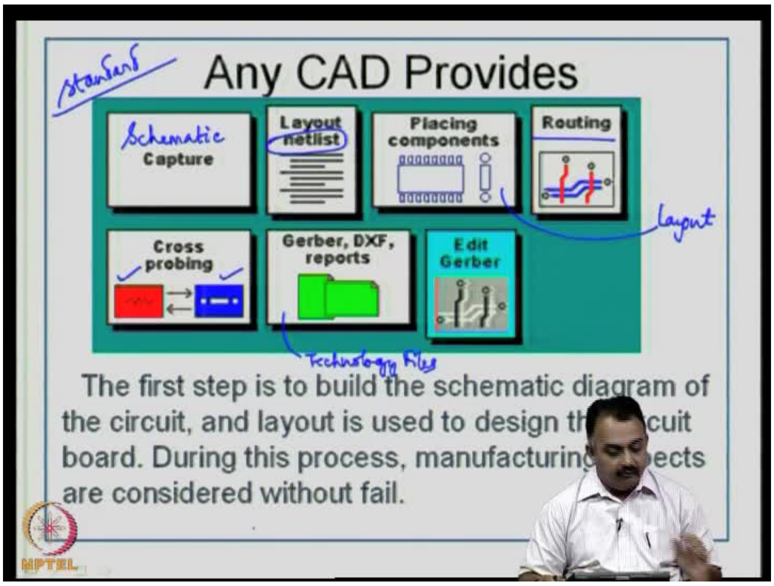
Now, we will enter into a sample demo, so that you will understand a CAD package flow much better; so, I have titled here CAD package introduction and demo.

What I would basically like to tell you is that any good software provide complete EDA solution - Electronics Design Automation solution. There are many software packages available; you need to select a package that is complete in all aspects. If you are going to work with some kind of a medium dense or a high dense board, then you need to look at the various modules that are available in this package; how you can work with it and utilize it to the maximum. For demo purposes, in this course, we are going to use OrCAD software. We will proceed to understand how to design a typical printed wiring board or a printed circuit board using OrCAD capture tool and OrCAD layout module

Now, it does not mean that you cannot use your software tool; if you have any other package at your institution or organization. All we need to interact here is that we will understand the basic flow process, starting from the capture module to a routing module, which will be very similar in most of the packages, except that the menu and the commands will be different; it does not matter; you can use your own software tool. Look at this demo and implement it in your package. Obviously, there is going to be online help in any software package today, including access to current libraries of your package.

So, you can go ahead and look at this demo and also implement it or try it using your own package.

(Refer Slide Time: 09:38)



Any CAD software will provide these basic modules; what are these? There will be a schematic capture; this is the starting point for any CAD tool. Then, we will proceed to what is known as a netlist generation, because schematic is basically grouping together various components and interconnecting these components, based on the circuit diagram.

Then there will be a layout module; so the placing of components will typically be in a layout module; you invoke that layout module and utilize the information that you have generated from the schematic and the netlist, and start placing the components into the board area that your product is designed for. Once you have placed the components, you can do a routing process. What does routing mean? It means showing you, exactly, how copper will be depicted in your printed wiring board, once the board is fabricated, (Refer time: 11:00) with all the exact dimensions of pads and the tracks; and various design rules will be implemented during this routing program.

The layout and routing are very integral; the netlist is the feeder information for the placement of components and the routing of the components.

Some modules will have cross probing; that is, you can always interact between the schematic and the layout or the routing package; and always back annotate and make corrections to the schematic; and then get it updated into your placement module, thereby making it most updated.

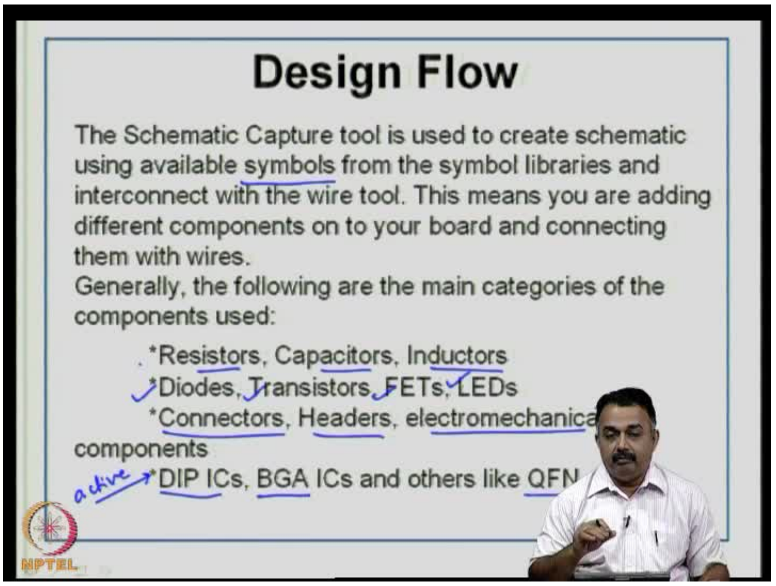
This is an example of a very simple standard package. There maybe packages with all the high end simulation tools EMI analysis, thermal analysis or signal integrity and so on; but generally, if you talk of a low dense board, where you are not going to really look at simulation tools; and where you have given enough tolerances in your design; this will be the major pathway that you will go through.

Finally, you will generate what are known as technology files that are sent to the manufacturers in the form of Gerber files, DXF, and you can also generate documents or reports for your future use.

There will also be an edit Gerber package that you can work with or the manufacturer can work with, on a finished board or a routed board; to add logos and some other manufacturing information; to add fiducials and other markers that are required for the manufacturing and assembly of printed circuit boards.

The first step is to start with the schematic and end up with the layout or a routing; and during this process your mind is always into thinking about manufacturing and assembly. Unless you think about manufacturing and assembly, the entire process step can end up as a futile exercise because you have not interacted with the manufacturer.

(Refer Slide Time: 13:48)



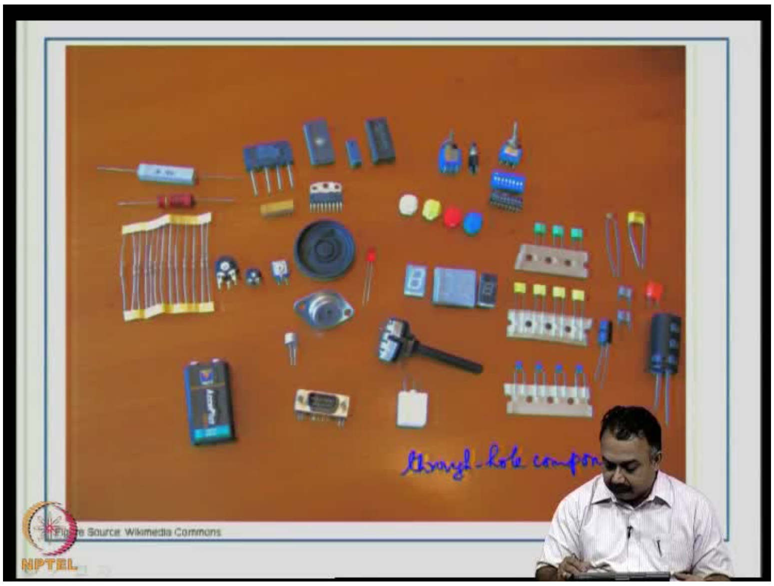
Let us look at the design flow as an example, which can be implemented in any software package. The schematic capture tool is used to create a schematic diagram using available symbols. In a schematic capture tool, you are worried about bringing symbols from the libraries. So you will use available symbols from the symbol libraries and interconnect them with the wiring tool; so, once you bring a symbol and if you have put all the symbols from that particular diagram, the next job is to interconnect them. This means you are adding different components onto your board and connecting them with wires

Generally, the following are the main categories of components used. Typically, you will have, first, the active devices: DIP packages - Dual Inline Packages, ball grid arrays, CSPs, QFN - Quad Flat No lead package; then finally, after the active devices are brought, because this is your main concern - the microprocessor or the micro controller or other memory products that you are going to use in your circuit diagram will be your first concern, you will bring them onto the schematic screen; you will add the other devices like the diodes, transistors, FETs, light emitting diodes and so on.

Then comes the passive devices – resistors, capacitors, inductors, which are again very important; if I can give an example of a medium dense board, if generally they will have about thirty percent of active devices and seventy percent of passive devices; and they will also occupy, the passives, about seventy percent or sixty percent of the board area; so, passives cannot be ignored, because they are there are different varieties of resistors and capacitors which you would like to use; and which you would like to choose carefully for a particular application.

Then comes the important world of connectors, headers and other electromechanical components, which will occupy a lot of space; and along with this they also need to be placed in certain areas of the board, which are very essential from the physical access during repair and rework; and during normal use, in the case of connectors. You cannot place a connector in the centre of the board; you have to place connector at the edges and therefore, you have to carefully choose the exact location, because sometimes, your board can be a daughter board, which will go and get connected to a motherboard. Therefore, for connectors and electromechanical components, you have to look at from the issue of weight; can it be really mounted on the board? , or should there be outside the board area?

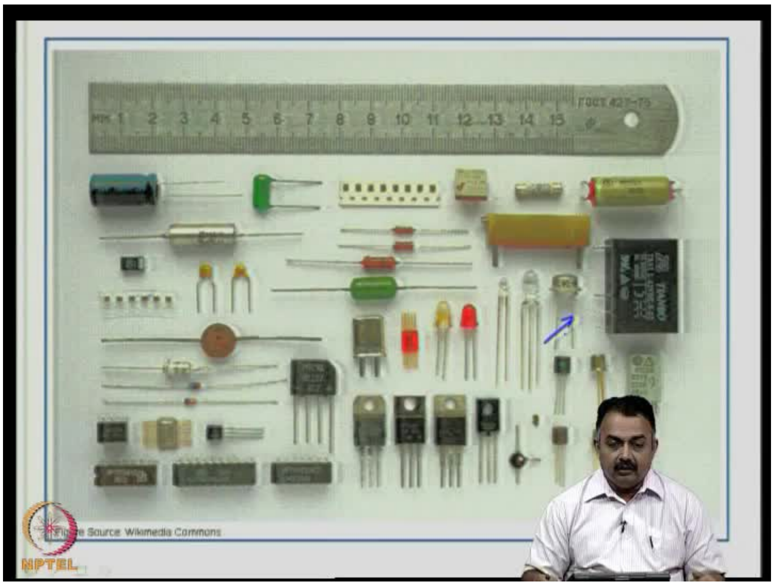
(Refer Slide Time: 17:13)



This is just to give you a glimpse of the varieties of components that you can see in the market today, which you might use; various electromechanical components, large devices, batteries and in some cases your printed wiring board can itself have housing for a small alkaline battery; then you have various shapes of axial capacitors; cylindrical capacitors; different sizes of resistors; displays – here, you can see a 7 segment display; then, you can see various active device formats.

In this particular picture, you will see, most of them are through-hole components; so this example here, is basically a highlight of through-hole components.

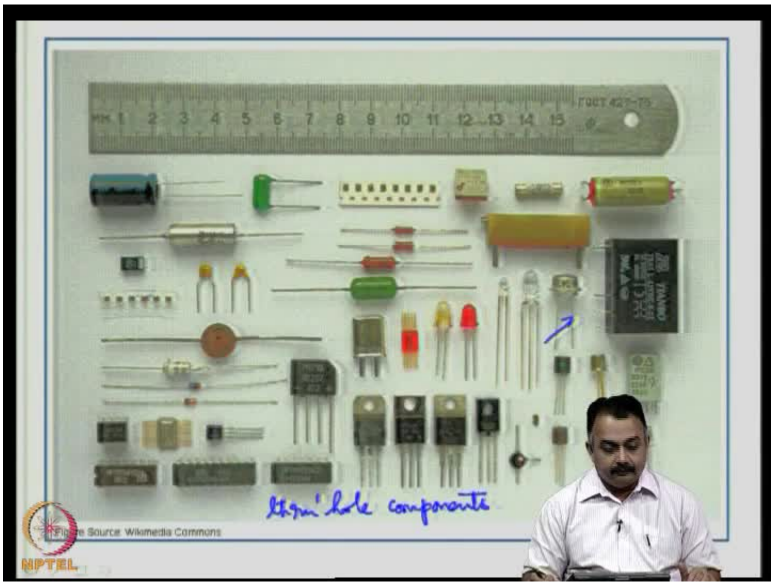
(Refer Slide Time: 18:16)



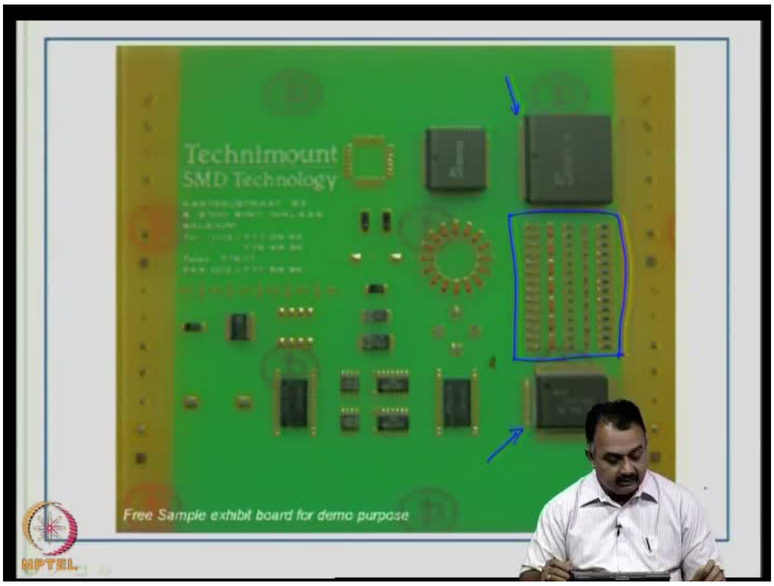
This is another picture showing you a wide variety of a capacitors; few transistors; few integral passive devices; you can see large capacitors; capacitors with different materials - you can have capacitors that are made up of, for example, barium titanate, some could be tantalum based and so on; different TO cans, as you can see here, are being used even today; different diodes and so on

(Refer Slide Time: 18:58)

(Refer Slide Time: 19:04)

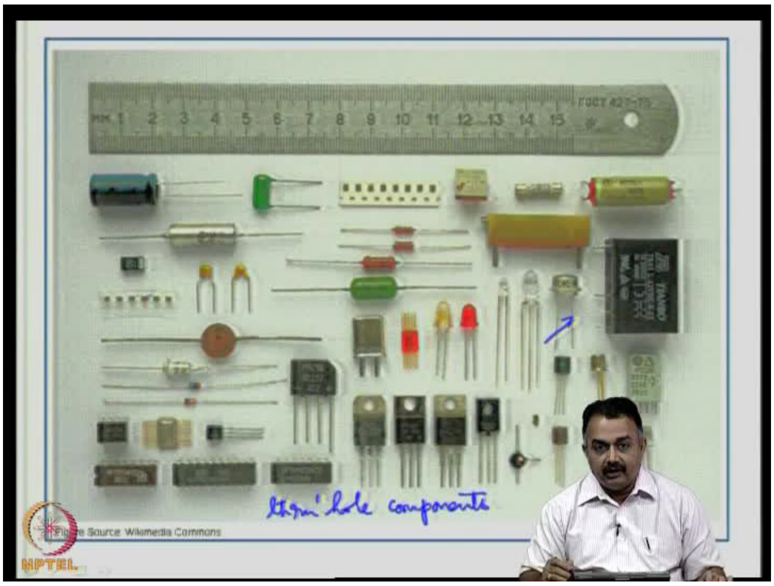


(Refer Slide Time: 19:19)

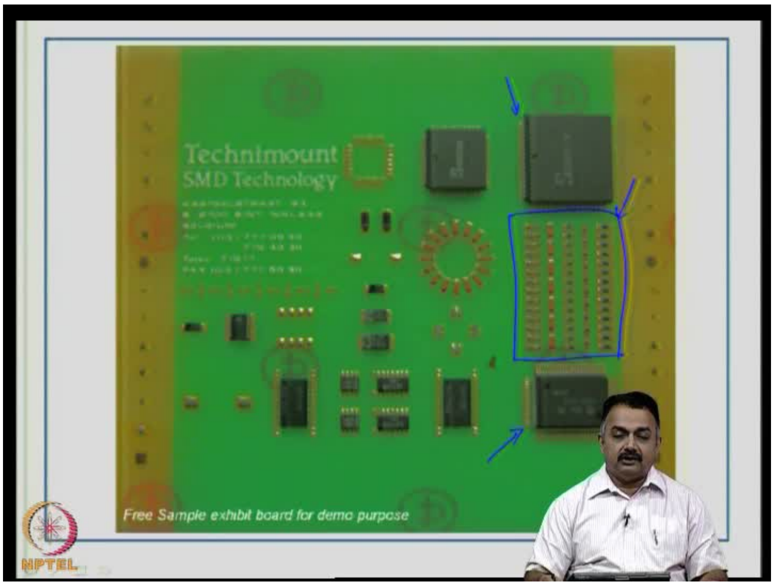


But today, as a matter of fact, these are being switched over to surface mount format. The earlier slide shows you, typically most of them depicted are through-hole components. This picture (Refer Slide Time: 19:21) will give you an example of surface mount technology or surface mount devices; this is basically an arrangement, an exhibit board which shows you very small low profile components. All them are surface mount, but different lead formats you can see here: some of them are gullwing type; some of them are small outline packages. I am sure that you are aware of these different formats, after going through the packages chapter.

(Refer Slide Time: 20:04).



(Refer Slide Time: 20:10)



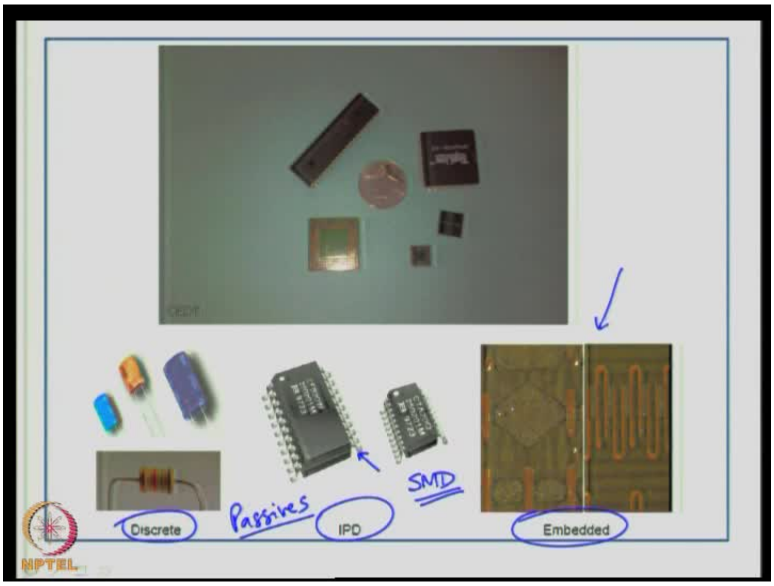
And here, you can see at the middle, there are various capacitors and resistors, very small in size compared to your through-hole capacitors, which are fairly large and they occupy a large area on the board; in contrast, surface mount devices will reduce the board size. That is the exact reason why people are reengineering their through-hole boards to surface mount technology.

The first thing is that they are reliable; they have been in the market for over 2 decades now; the surface mount devices, the technology is improving and the sizes are still reducing, still shrinking; therefore, the capacity or the capability to handle these at the industry level is increasing; there are various electronics manufacturing services that provide you excellent assembly services for surface mount devices, even of the order of very small footprints that you see here in this marked area - the resistors and capacitors which, sometimes it will be very difficult to do by yourself, in the lab or for prototyping. But if you want to do it in large volumes, then you have to depend on the EMS services, because they can increase the throughput and also reduce the cost.

This picture is basically an arrangement of various surface mount devices. As a designer, the reason why I have put these 3 slides here, is that you may be dealing with different types of these components in your board; your board can be entirely through-hole, or it can be a mix of through-hole technology and surface mount technology, or it can be a 100 percent surface mount, with the exception of some through-hole components like connectors and a few electromechanical devices. You can still make a 100 percent surface mount device, if you can find the corresponding surface mount components.

Therefore, the design issue starts from the schematic itself; then the layout; and looking at this board is going for a surface mount technology assembly; what are the precautions I have to take?; how do I interact with the manufacturer, as well as the assembly person into getting the proper inputs for assembling a surface mount technology board?

(Refer Slide Time: 22:49)

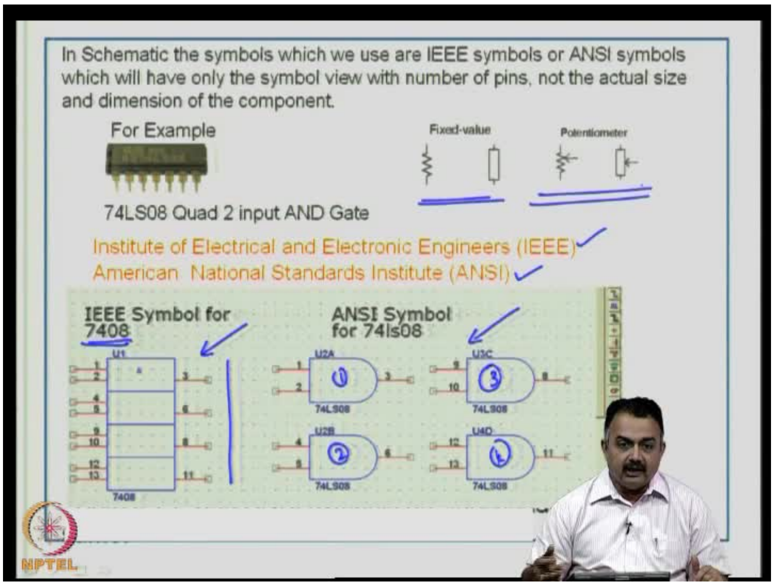


This is again, an example, showing that today, we are moving away from discrete passive devices; these are passive components; we started with discrete devices; we have worked with the integrated passive devices, which means the passive devices are similar looking to a DIP package, but they are integrated inside this housing; and the IO connections are typically in the form of lead frame structures that you see here.

And today, we are moving into; of course, there is surface mount here, which cannot be ignored; but we are still moving further into embedded passives - this is the picture showing you that the resistors and capacitors can be generated in situ, while you fabricate the printed wiring board on an organic substrate or a ceramic substrate; and therefore, you can do away with using discrete and integral passive devices.

This also eliminates the assembly process in your board; it will also remove 20 to 30 percent of board area or footprint area that you normally require for resistors and capacitors; and therefore, your board density can go up.

(Refer Slide Time: 24:25)



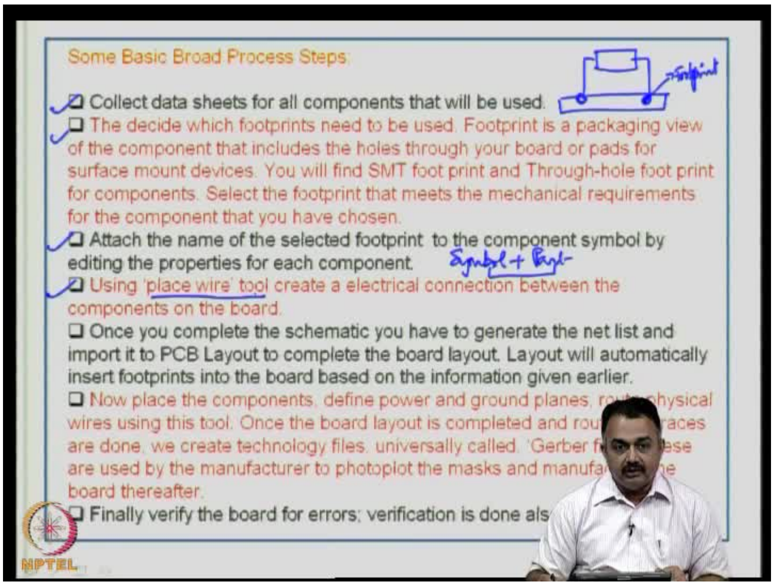
We will now move into looking at what are the ingredients in a software package. In a schematic, the symbols which we use are IEEE symbols or ANSI symbols IEEE stands for Institute of Electrical and Electronic Engineers and ANSI stands for American National Standards Institute.

Whatever be the package as I said, a symbol is a symbol; it is similar in all packages; so go to your symbol library bring out the component and have a view of the symbol of a particular package; for example, if you look at 7408 or 74ls08, the symbol is brought onto the screen. Here, this is an IEEE symbol and here, this is a ANSI symbol, but today, most packages would like to use IEEE symbol, because that is now the current standard for symbols.

What you can see here is, in the ANSI symbols, there are the different gates; so you can see pin number 1, 2, 3 grouped together; then you have 4, 5, 6 grouped together. So, these are the different gates; the pin numbers are shown, but here, they are integrated and you can also see the pin arrangement. This is the IEEE symbol and for example, for 74ls08, this is the IEEE symbol that you will see and these are the symbols for example, of the passive devices, like your resistor or your potentiometer and so on, that you will see on the screen.

Both IEEE and ANSI symbols are functionally the same; so, whether you are using an ANSI symbol or the IEEE symbols, the functionalities built into it are the same.

(Refer Slide Time: 26:44)



Now, let us look at the broad process steps in a work environment; or when you start with the circuit design, what are the things that you will do:

Collect data sheets for all components that will be used

Then you decide which footprints need to be used. By footprint, you mean the exact mechanical dimensions of the pins; or in the case of a BGA package, it will be a solder ball footprint, which sits on your printed wiring board or a printed circuit board, after the board has been fabricated. So you need exact dimensions of these IO points or the pins; otherwise, you will be utilizing unnecessary board area. So decide on which footprints need to be used. Footprint is therefore, a packaging view of the component that includes the holes through your board or pads for surface mount devices. So, if you have a resistor, for example, a through-hole resistor, then you should have the information about the hole size so that goes on to your board. This is a footprint. You will have footprints for every pin in every package. You will find SMT footprint separately, through-hole footprint separately in your package library. Select the footprint that needs to be used; and that meets the mechanical requirements correctly, for the component that you have chosen; these are the 2 important steps.

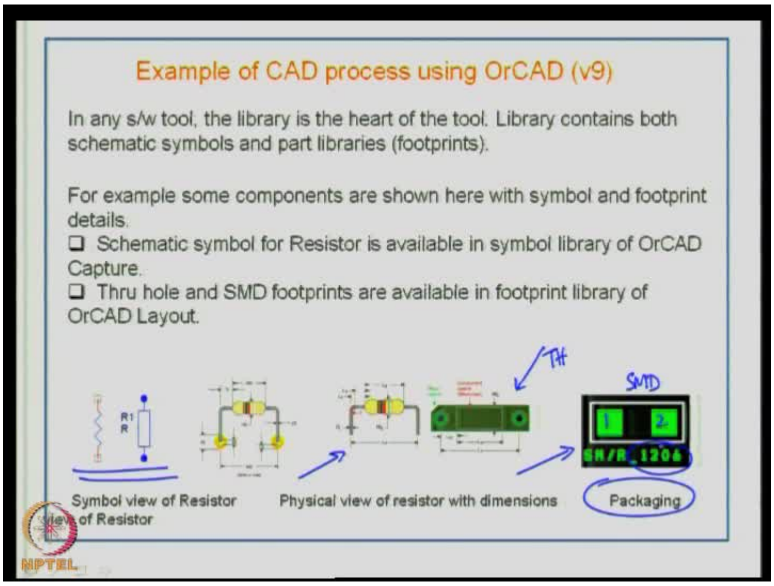
The third one is: attach the name of the selected footprint to the component symbol, by editing the properties for each component. You have a symbol; then you have selected a part; now, you have to attach them or link them, so that this information is stored between the two entities, right from the schematic to your routing.

Then using the place wire tool, which means you are going to start the interconnections in the schematic, create an electrical connection between the components on the board. Once you complete the schematic, you have to generate the netlist, which I said, is very important and import it into the PCB layout to complete the board layout. Layout will automatically insert the footprints, because you have already tagged them with the symbol; so, once this process transition takes place, the information is also moved over into the layout section. Now, place the components according to your wish, because as I said earlier, your design and my design are definitely going to be different. If you are working separately for a day, on a particular design; and we now come and compare our work; obviously, your layout and my layout is going to be different, but both are acceptable if we follow certain design rules.

Therefore, in the layout area you place the components; define the power and ground planes for a multilayer board; route the physical wire using this tool; once the board layer is complete and routing of traces is also complete, we create what is known as the technology files, also called Gerber files - it is a universal format file, which is sent to the manufacturer to photoplot the masks that you require for every electrical layer; and therefore, this is a very critical step and this done by some other agencies outside, or you can have a small photoplotter in your lab, which can do the job.

You need to be aware of the post processing activities, if you want to create a mask and then the board is sent for manufacturing. After that it is sent for assembly and at every point of time, in this module, from schematic to layout to routing, you can always check the board for errors; you always have a design rule check tool in your software package; use that and verify the board and make sure all the entities are in correct format.

(Refer Slide Time: 31:32)



Now, the example of the CAD process that we are going about, is using OrCAD version 9. In any software, the library is the heart of the tool; so if you are spending money on a CAD software, please analyze that the library is excellent, because you are going to interact with the library entities very often; you are going to bring the symbol, you are going to bring the footprint for various packages and you do not want to spend too much time on creating your symbol or creating your own part and by part I mean footprint.

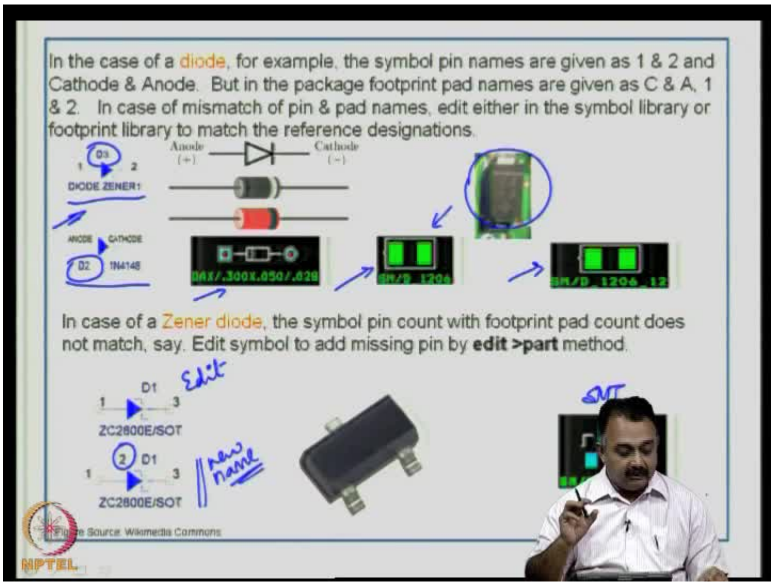
Today, the software companies which provide these CAD tools provide you with online help and online updates of library components. For example, some components are shown here with symbol and footprint details.

Schematic symbol for a resistor is available in the OrCAD capture and the footprint information for these will be available in the OrCAD layout. This is true for any active device or a passive device.

Now, symbol view of a resistor is shown here; now what is shown here is the physical view of the resistor with the dimensions, which is available in a data, sheet but in your pah library, you will end up with something like this. Essentially, this is for a through-hole format and this is a surface mount.

In a through-hole format footprint, you will see pad details; body outline; keep off area drill information; that is, where exactly and what size the drill holes have to drilled, in the pin area, for inserting your resistor; here, since it is a surface mount format, you do not require drilling. If you look at this picture here, you see pin number 1 and pin number 2 here of the two resistors and you can see the white area, which is basically telling you that it is the maximum body area of the package; and obviously you cannot do routing in those areas, so, that is like a keep of distance. You can see dimensional details, so, this 1206 has a meaning; it basically indicates the size of the resistor package. We will see that shortly. So, these are the kind of information that you will get from a footprint library.

(Refer Slide Time: 34:31)



In the case of a diode, for example, the symbol pin names are given as 1 and 2, and cathode and anode, but in the package footprint, pad names are given as C and A; 1 and 2. You must be able to look at these kind of numbering naming systems.

In the case of mismatch between the pin and the pad names that are either given in the symbol or the footprint library, you can edit, either the symbol library or the footprint library, to match the reference designations that you want. You can save it in your own file name and if you are going to use this particular symbol very frequently and if it is not present in the host library, it is better to edit and keep one for yourself.

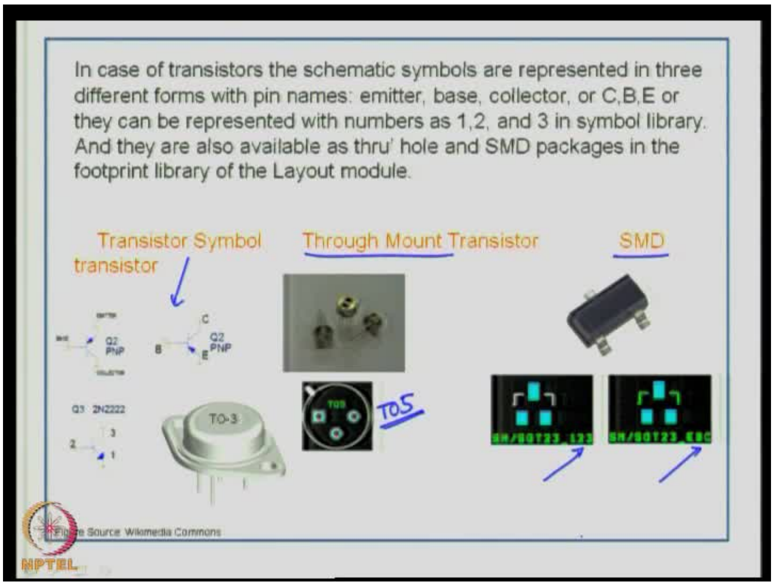
You can see here, this is the Zener diode; pin numbers 1 and 2 are indicated; some cases it is anode and cathode that is indicated; D2 represents the reference designation of the diode D1, D2, D3 and so on. That will be in a sequence in the entire schematic paper.

Then, you can see, these are the footprint information; pin number 1 is indicated as a square and pin number 2 is a normal circular fashion. This is a surface mount footprint; this again is a surface mount; this is a surface mount device; and therefore, you have to bring the symbol for this particular component, a diode, like it is depicted here; and then attach or link it to the surface mount footprint pattern, so that this information is carried nicely into the layout section.

In the case of a Zener diode, the symbol pin count with footprint pad count does not match, let us say. In that case, you edit, for example, if pin number 2 is missing, then you can edit this symbol, and create an additional pin, and name it in a different library that you can keep it with yourself and utilize it whenever you want.

This is the picture of a Zener diode, SOT 23, for example, this is the SMT footprint. he pad sizes are well defined in a footprint libraries. Check that, before you utilize it.

(Refer Slide Time: 37:15)

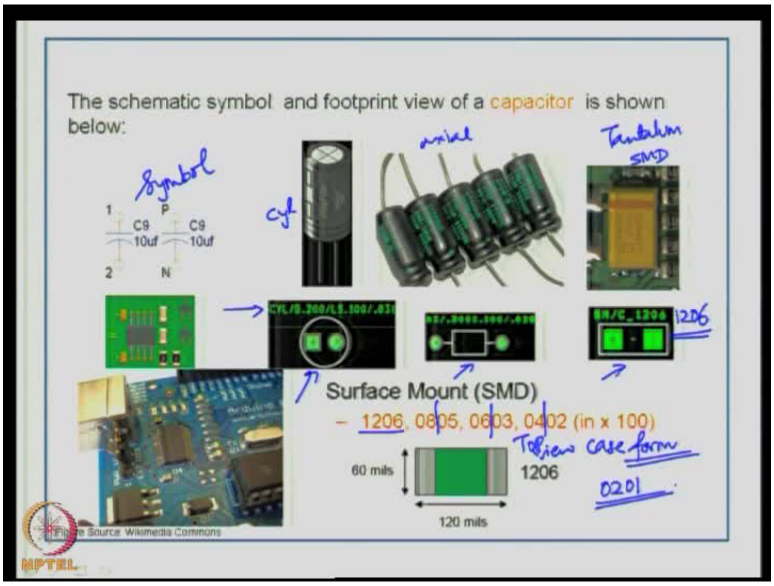


In the case of transistors, the schematic symbols are represented in 3 different forms, with pin names emitter, base and collector or C, B, E simply; or they can be represented with numbers like 1, 2, 3 in a symbol library. Also they are available in through-hole format and SMD format; so make sure which one you are going to use and tag in the footprint library of the layout module, because the symbol can be the same, the key issue here is: are you going to use a through-hole transistor; or are you going to use a surface mount transistor.

This is the symbol information; this is a through-hole (Refer Slide time: 38:00) package information; for example, for a TO5 transistor, you can see pin number 1, 2 and 3, arranged in definite spacing and sizes, including drill diameters that needs to be used.

In the case of a surface mount, you can see, this is a surface mount of a SOT 23 package; it can be, in the footprint library, named 1, 2, 3 or it could be named as emitter, base, collector. You can choose either of them and you can see the dimensions are well defined.

(Refer Slide Time: 38:39)



The schematic symbol and footprint view of a capacitor is shown below. This is the symbol, as usual, you are aware of this, for a capacitor. The values are also given 10 microfarad; positive and negative; or it could be 1 and 2.

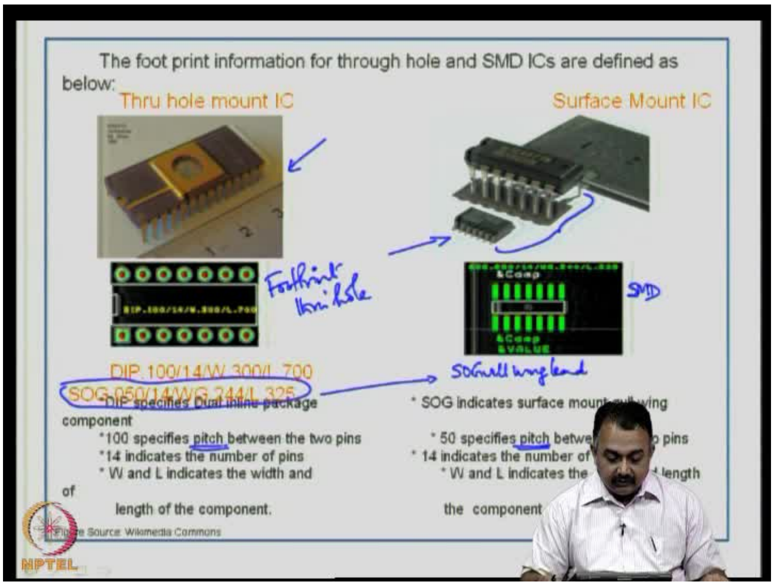
Then, you can see, these are the different axial capacitors: this is the cylindrical capacitor; then, this is a tantalum surface mount capacitor; and you can see the corresponding footprints for each of these, so, it is very essential to do this exercise in your schematic to link this with the actual footprints that you require.

You can see in this particular picture here, that it says it is a cylindrical capacitor; the dimensions are well given, like it says point 0.2 inches by 0.1 inches by 0.03; and in the case of a axial, again, you can see different dimensions; and in a surface mount, it says

1206, that indicates the case form; so you can see here, these are the case forms that are available for surface mount devices, where it is a resistor or a capacitor.

1206 means, it depicts the dimensions of the capacitor; that means the dimension of the capacitor - this is the top view, you can see this is the top view; the dimensions are 120 milli inches by sixty milli inches; or you can say, easy way to remember is, 0.12 inches by 0.06 inches; so this will be 0.08 inches by 0.05 inches; 0.06 inches by 0.03; 0.04 by 0.02; today, we also have 0.02 by 0.01 package; so, you can see, the dimensions are diminishing; the package sizes are reducing, in the case of surface mount devices, like resistors and capacitors; and therefore, the ability to handle these are available with a few assembly people, who have automated equipment to handle this.

(Refer Slide Time: 41:07)

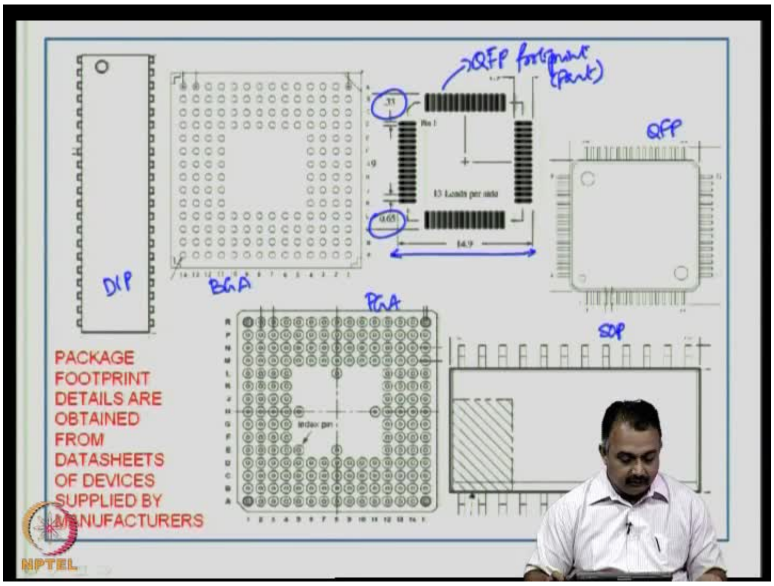


The footprint information for through-hole and surface mount device integrated circuits or active devices are seen here.

This is the picture of a through-hole mountable integrated circuit; and here you can see a surface mount; comparison between through-hole and surface mount. This is the footprint for a through-hole IC, and this is for a surface mount device. You can see this is a DIP package. The dimensions are given here: 0.1; 14 pin package; width is 0.3 by 0.7 inches; and this particular information has to come here for SMD; it basically, is a small outline, gullwing lead component; and you can see the notations are given here, the number of pins are indicated.

If you go to the footprint library, you can identify these packages and the corresponding footprints from the reference designations and the naming that are used in the library. So look for the width and the length that you require for a particular component; that is very important.

(Refer Slide Time: 43:02)

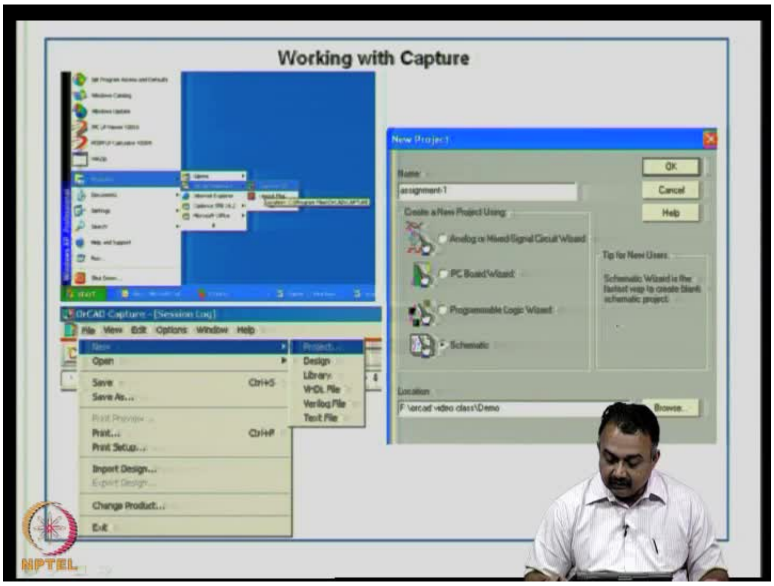


The other thing is the pitch; what is a pitch you want to use for this particular package? Here, it indicates fifty mills; in this dip package it is it is indicated as 100 mills. So, carefully look into the library information for each of these devices. This is the overall view of various mechanical details that you will see in your data sheet; and which needs to be well interpreted and invoked for utilizing the current footprint. For example, this is a QFP footprint or part; you can call it as a part; that is available in any CAD package.

It basically is a 52 pin QFP 13 leads per side; and the distance between each pin is well indicated; for example, here it says it is a 0.65 mm pitch QFP; and the thickness of each pad is also indicated here, 0.33; and the total package outline dimensions are also indicated.

These things, if you want to edit or if you want to create a new part, you can utilize this information and store it. This is for a dip package; this is a pin grid array; this is a ball grid array; this is again a QFP package; and this is again a DIP package or a small outline package, as the case may be. Package footprint details are obtained from data sheets of devices, supplied by the manufacturers. Most of them are available in the library, otherwise you have to create one with the basic information that I have showed here.

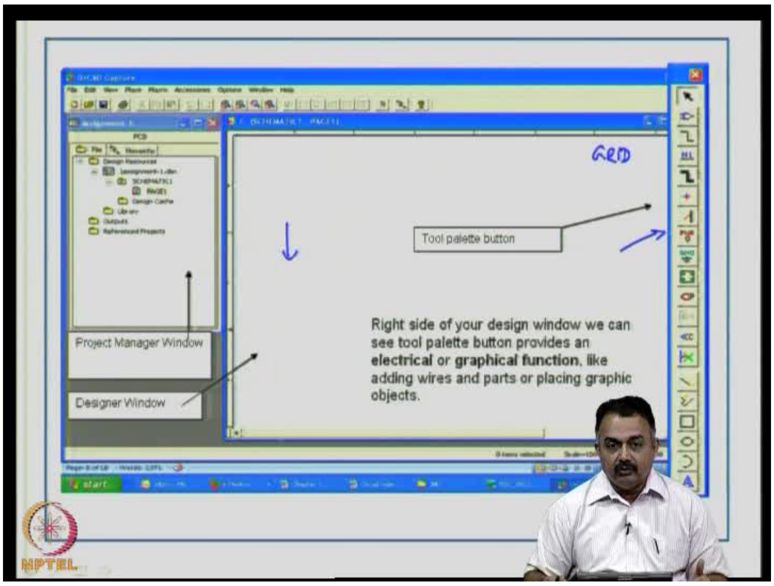
(Refer Slide Time: 44:43)



If you work with this tool, let us see how we move quickly into creating a nice schematic capture. In an OrCAD capture, you have to invoke the capture tool; using this particular screenshot that you are seeing here, you can easily maneuver yourself into the CAD program.

You can open a new project. Whenever you work with a design, start a new project, because a project will have various subsections; a project will have 2 or 3 designs embedded; you may have to link these designs at a later point of time; so, it is very advantageous to create a complete project where you can link. Otherwise the electrical information may not be available, if they are 2 different files. Then you can name it as a schematic and store this information save it as a new file.

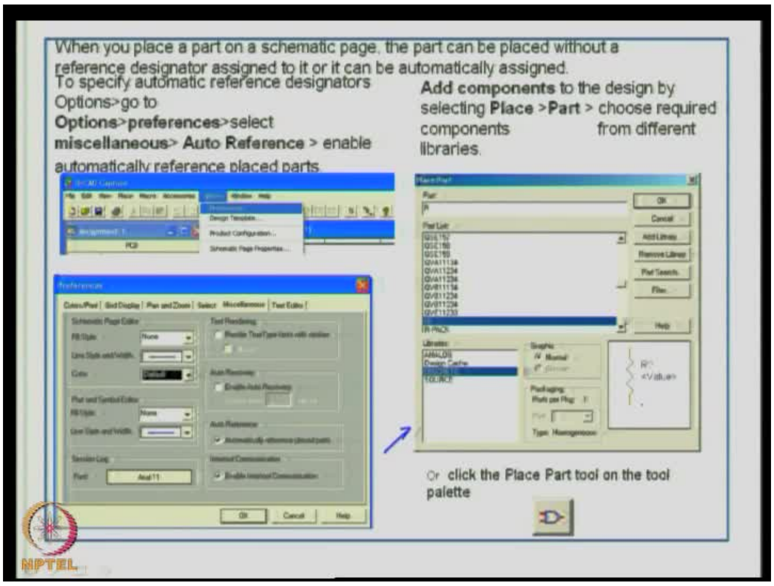
(Refer Slide Time: 45:47)



Once you invoke the schematic, you will see a grid here and on the right you will see a palette, which basically contains all the tools; for example, it contains component symbol information, wiring tool, netlist tool, routing tool, a power - ground information insertion, symbols, text editing, various shapes, keep off distances to be drawn, various nonelectrical symbols that you need to create and so on

So, you have on the left, a project manager window, in terms of files; page sequences; page numbering - you can have page number 1 of the schematic page number 2 of the schematic up to 5, 6 and so on, which you can link, page number 1 electrical information will be the input for page number 2, so, that is the advantage.

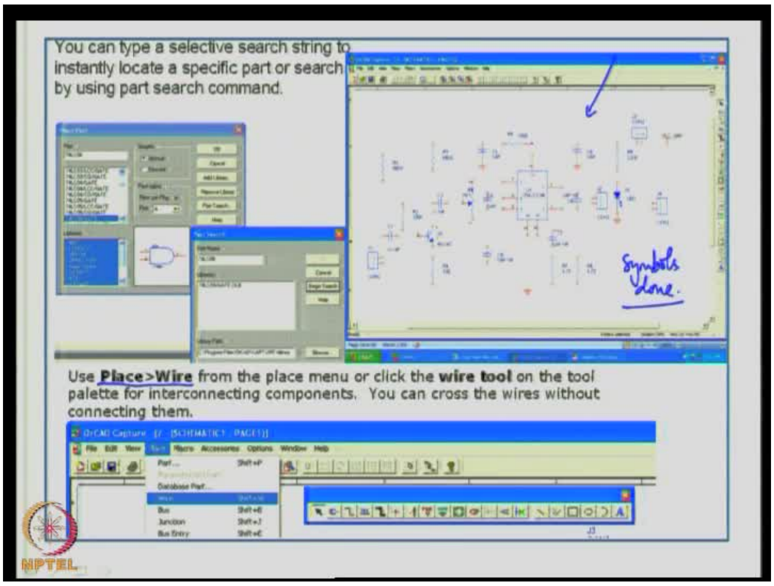
(Refer Slide Time: 47:07)



This is the designer window here, where you will start entering your symbols and do the wiring. When you place a part on a schematic page, the part can be placed without the reference designator assigned to it or it can be automatically assigned. As I said earlier, when you add a symbol, immediately you want to reference it to a part. So normally, you can do auto-referencing by choosing the right command here, by saying look for or suggest an auto-numbering or naming sequence, for the corresponding symbol, in the part-library or you can choose it by yourself, manually, to make sure

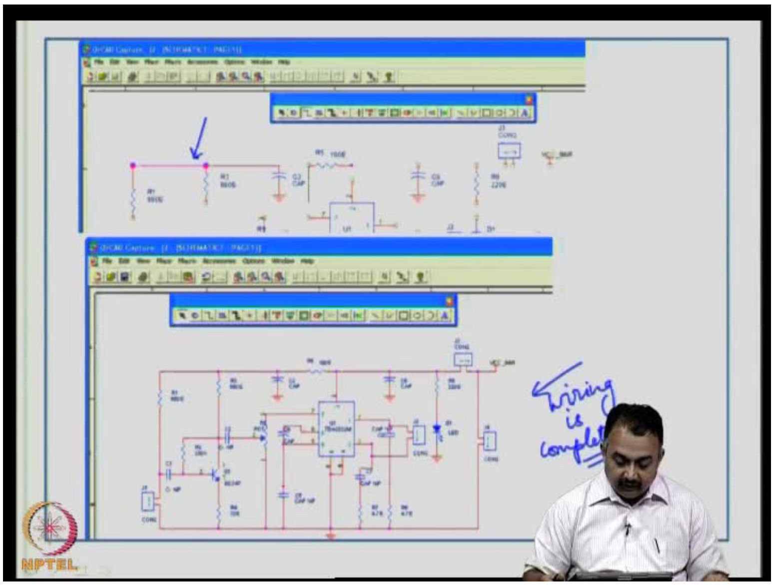
If you look at this particular capture here, you will see, in the part called resistor, you will have a discrete resistor, normal graphic symbol, you will get; and once you click OK, this will go and get placed on the screen.

(Refer Slide Time: 48:16)



You can do a selective search string, to instantly locate a specific part or search by using a part search command. The best thing is, before you forget, if you bring a particular processor IEEE symbol, immediately attach it to a part; and as you can see here, in this particular screen capture, you have seen that various devices have been brought; they have been place around the active device, in a particular fashion; and the criteria for placing the other devices around the microprocessor or the active device will be that, the length of the interconnections to adjoining part has to be a deciding factor. Because ultimately, in a schematic or your final CAD layout, you will see that the interconnection length will be kept to a minimum.

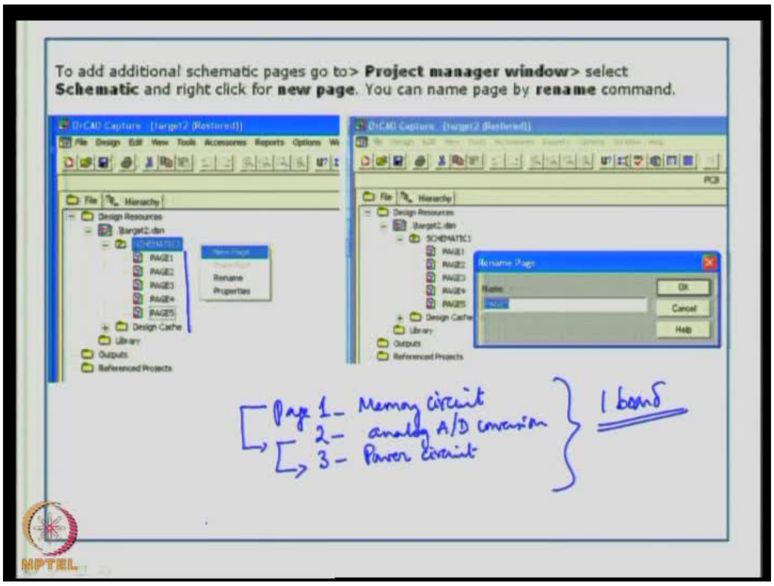
(Refer Slide Time: 49:34)



Now, you can start using the command called place wire, which is, you are going to begin to interconnect here; so, this is symbols, done; now you have to start placing the wire; so, go to the place wire tool in your schematic capture and start placing the wires, as you can see here; different components are being connected, according to the circuit diagram, and here you can see, the wiring is complete.

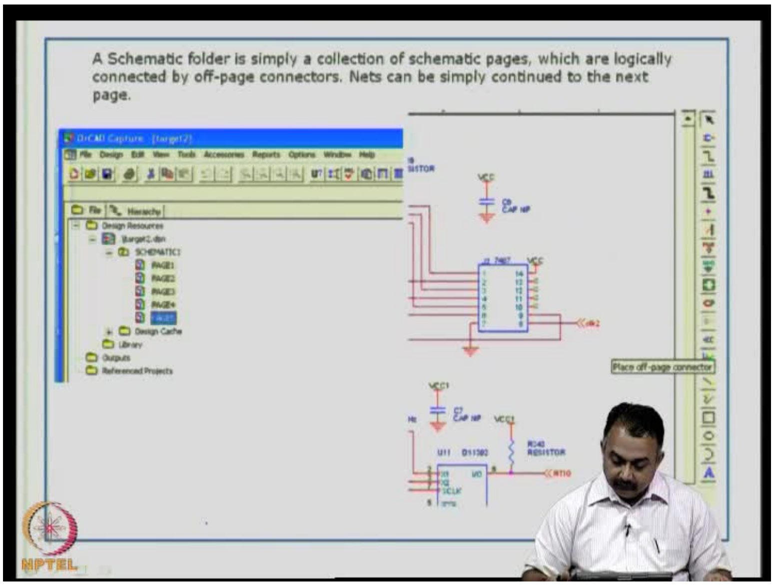
We can see, wiring is complete and again, mind you, this is a schematic; so you will have basic point to point interconnections and you can cross over a wire with or without electrical interconnections, which it will prompt you for; so, make sure that your schematic layout is well done.

(Refer Slide Time: 50:22)



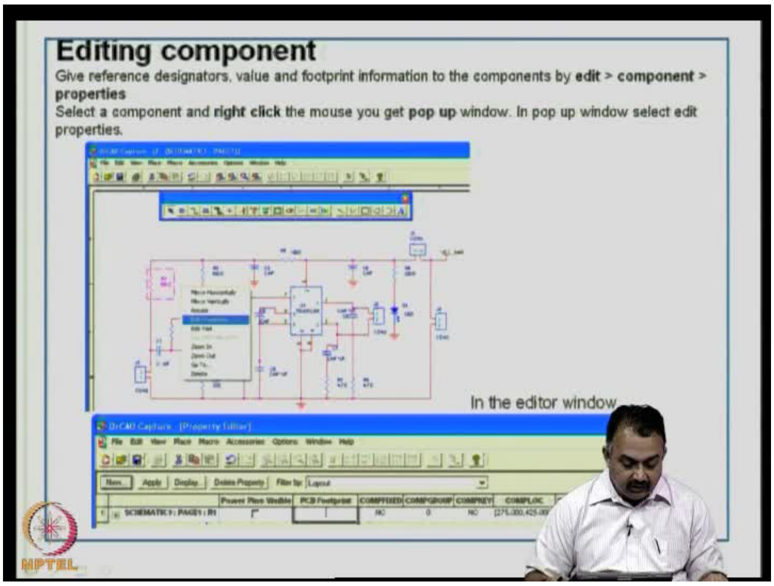
You can add additional schematic pages, by asking for or going into the appropriate menu and creating various pages - Page 1, page 2, 3, 4, 5 and so on. As I said earlier, it is better to do in different pages. For example, your page 1 can be a section of your board, which is a memory circuit, let us say; page 2 can be a typical analogue A to D conversion; page 3 can be a capture, that is simply some kind of a power circuit or power supply circuit and so on. But, this is typically one board, but for ease of editing; for ease of understanding he circuit, you can split it into different pages; you can always link them; because, the input of page 1 can always go to page 2; page 2 can go to page 3 and so on. The netlist can always be referred to.

(Refer Slide Time: 51:43)



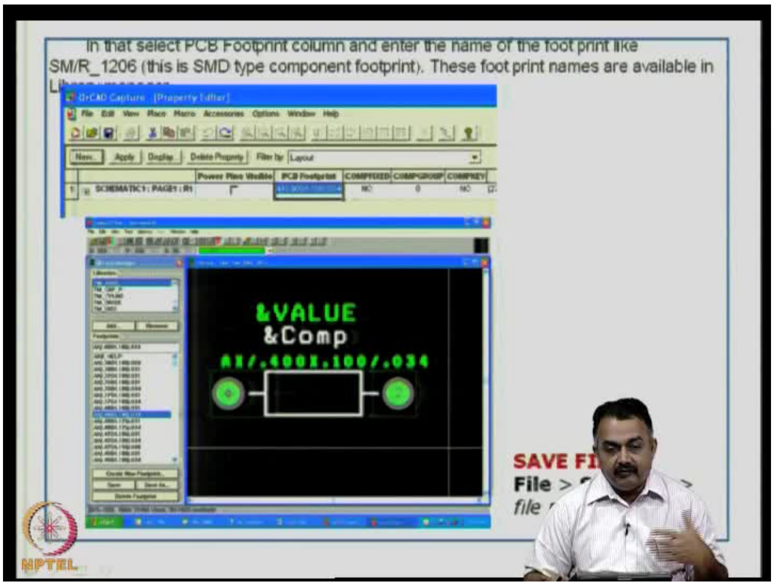
A schematic folder therefore, is a simply a collection of schematic pages, which are logically connected by off-page connectors. You can imagine that there is a connector between pages; but actually, it is not a connector; electrically, it is the information that is connected; the electrical points are connected; nets can be simply continued to the next page; so, with some experience, you can become a master of creating a schematic with different pages; initially, you may find it very difficult to do it or link it, but after some experience it becomes very easy; and obviously, the software also enables you to link them very nicely.

(Refer Slide Time: 52:29)



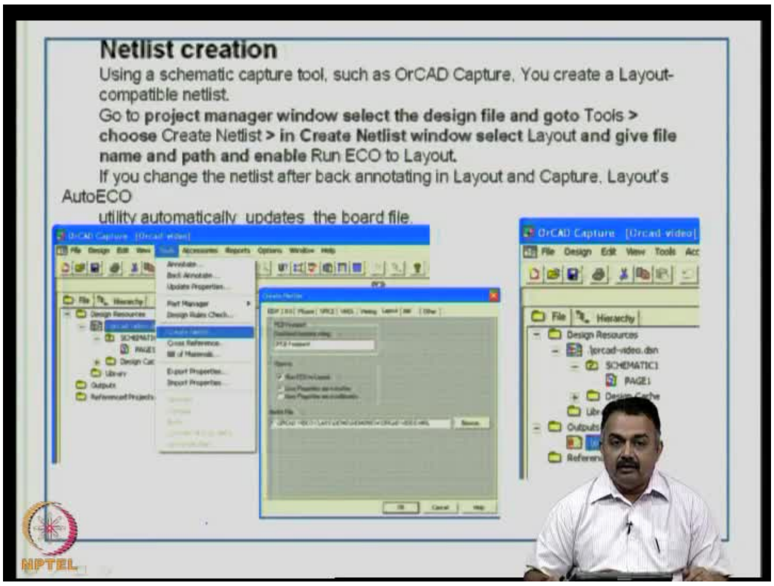
Editing component - when do you require this? The reference designators, value and footprint information to the components can be given by invoking the edit manager or the edit library tool in your particular software, going to the particular component and looking at the properties.

(Refer Slide Time: 53:11)



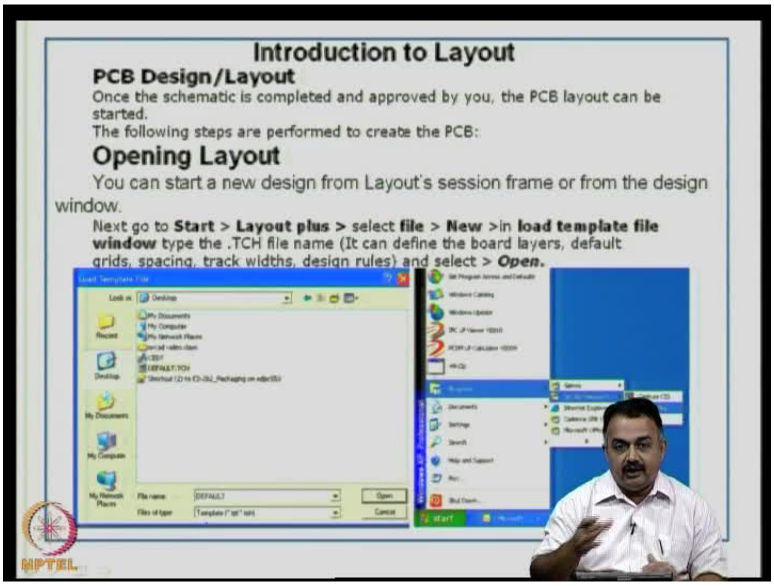
Select the component and look at the properties; and then edit whatever pin information is required, the numbering sequence that is required and so on; and then save it as a new symbol or a part. The same thing can be done with the part library; in the PCB footprint column, enter the name of the footprint; let us say, 1206 SMD resistor; and then you can go there and edit the required information; and save it as a new part.

(Refer Slide Time: 53:32)



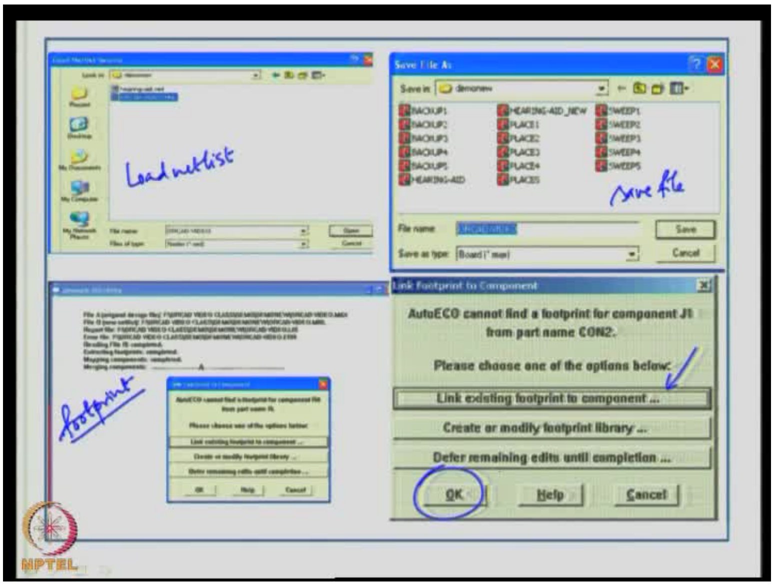
Once the schematic is ready, with all your editing done, you can now transfer this information to create a netlist. Netlist can be created, as I said, from any schematic capture, in any tool and in fact, if you have a low end software, your netlist from this tool can be exported to a high end CAD tool, for other packages like work like layout, thermal analysis, EMI, some kind of a simulation and so on. So, create a netlist, basically using the nets that you have generated by placing the wires. So, accordingly, you can create your own file name, which needs to be inputted later.

(Refer Slide Time: 54:23)



So finally, the netlist is ready. The netlist, as I said, is a very essential tool for understanding your electrical work. Now, you can go into the layout section, where you create the PCB design; so, once the schematic is completed and approved by you, the PCB layout can be started; so, you create a new file name from the layout module and again, here, you can create typical projects; layout names, typical of the Here again, you cannot create different pages, it has to be a single layout page, because it is a culmination of all the schematic work that you have done; and here you are going to define a single board size, but it can have different layers, vertical layers: layer 1, 2, 3, 4 and so on.

(Refer Slide Time: 55:17)

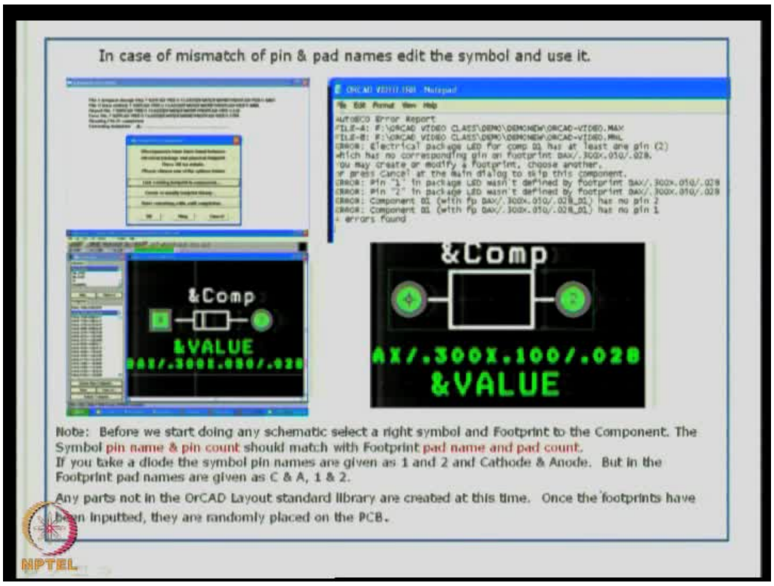


So, what this slide essentially shows you is some of the capture from the process:

Here, loading the netlist, which you have generated

Then saving it as a new filename, for the layout package; then here, you have to tag - some packages will ask whether the footprint information that you have referred to in the schematic, can it be loaded? So you can say yes; in some cases there may be missing footprints, which you can refer to in the error file or go back to the schematic can change with a new footprint information, if it is missing; so here, for example, link existing footprint to component, say OK; and then now, you are ready for bringing the package information.

(Refer Slide Time: 56:12)



So, in case there is a mismatch between pin and pad names, you edit the symbol and use it. Before we start doing any schematic, select the right symbol and footprint to the component; the symbol pin name and pin count should match with the footprint pad name and pad count; so, this, I have explained earlier. Any parts that are not available right now, needs to be reverted back and then looked at in the schematic.

So, we will continue with this process, from the layout to the routing and technology file creation in the next class.