Biomedical Ultrasound: Fundamentals of Imaging and Micromachined Transducers

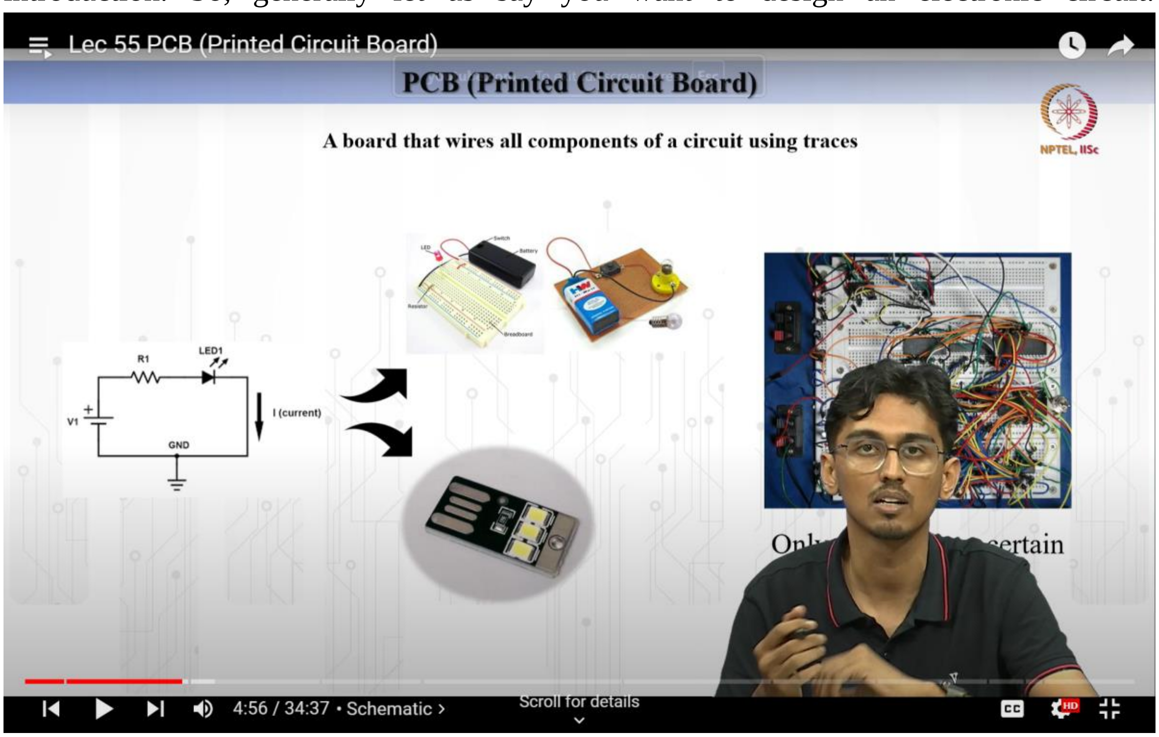
Prof. Karla P. Mercado-Shekhar, Prof. Himanshu Shekhar, Prof. Hardik Jeetendra Pandya

IIT Gandhinagar, IISc Bangalore

Lecture: 55

PCB (Printed Circuit Board)

Greetings today in this session we will be learning in general about printed circuit boards so as electronics engineer or in general if you are looking to design any circuits that can faithfully do the intended functions that you want to, a printed circuit board is a standard way to do that. So, whatever circuits that you have either be it a prototype or a final product or if it is an interfacing circuitry for a sensor, printed circuit board is basically as the name suggests it prints the circuits using copper traces on substrate such as FR4. And this gives you much more reliability and control over the signals and the power that you want in the several parts of your printed circuit board. Now, let us go in detail of what is a printed circuit board, how you can design it, what are the ways or what are the considerations that you need before designing a printed circuit board and so on. Generally, now before going into printed circuit board I want to give you a brief introduction. So, generally let us say you want to design an electronic circuit.



So, the first thing that you do is go ahead and draw the circuit sometimes it is called the schematic. And generally, you would need to know the properties of all the components that you use in a circuit, and you should be able to determine how the current flows or how the voltage potential reacts with the multiple components that are present. So, in this case I have kept a simple electronic circuit of an LED. It is a circuit that most of you will be aware of.

So, for an LED to glow what are the general components that you need. So, here I have a power source to illuminate the LED. And I must make sure that this power source conforms to the requirements of the LED. It should not be too high so that the LED heats up and fuses or it should not be too low also otherwise the illumination never occurs. Now, that is one thing we consider and then we include some form of resistance to control the current that needs to go into LED.

As we all know any light emitting source can heat up and whenever any element heats up in an electronic circuit the current goes out of control. In order to limit the current, we have to have some form of limiting factor in resistor, in this case a resistor is used. And, then we know that a fixed current will flow across the circuit because since the R 1 and LED 1 are in series we know that a fixed current is flowing and then of course, we need a reference plane for us in our case it will be the ground. Now, no need to worry about the schematic, but just in I just wanted to understand this circuit so that you can appreciate the rest of the circuits. Now, the same circuit after you design will be populated with the physical components on a breadboard or a perf board like I have shown here.

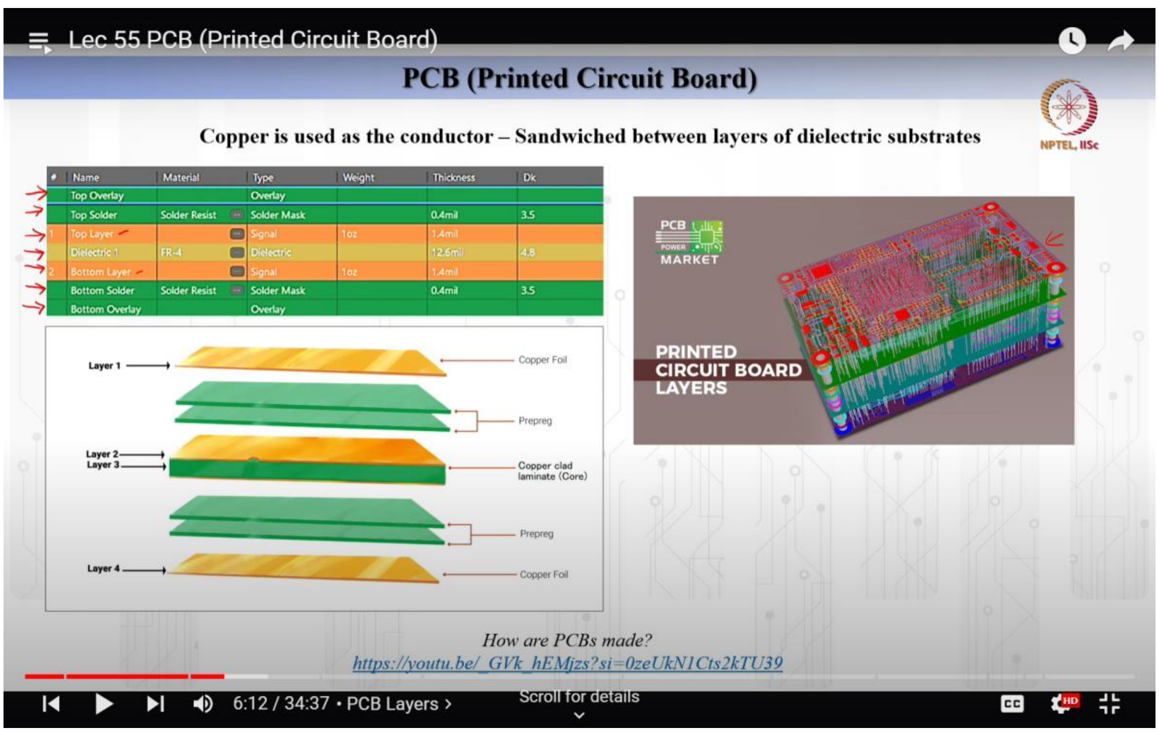
So, this is a breadboard with a battery and a switch and an LED. So, here a resistor is also included, and the same configuration is translated here. Now, when you turn on the switch which is included here in series let us say there is a switch, and you turn them on what happens the current flows and the LED glows. So, that is what is happening here. The same configuration can be soldered on to a perf board.

So, here you just plug in use it or here you have to solder the components on to each other in a perf board and use it. So, here that is the case. So, both are good for prototyping. So, why do we need a printed circuit board? So, when you have too many components like I have shown here, a printed circuit board generally works because you can see how cluttered these components are. Here all the wires are missed out, you do not know which component goes from where and you should be able to follow it up as an electrical engineer or whether you are looking to do it as a hobby.

PCB works much better in this case. This is a really good pictorial representation of what happens when you use multiple components because the connections are interlocking usually you have to connect through a component, between a component and so on. So, this is where a PCB works. So, a representation of the same circuit in a PCB is given here. Now, what does the PCB contain? What does it physically contain? So, from PCB power I have just shown you a simple representation.

This is a good representation of what is inside a PCB. So, you can see there are multiple colors represented here, those are just the colors for the different layers of the PCB. So, on the left-hand side you can see I have taken this from Altium designer which is a designer software for PCB designing. Here you can see that there are multiple layers, and it is given in a even number. So, whatever is present in the top layer the complement is automatically present in the bottom layer as well.

So, I have something called as the top overlay, then below that is the solder layer, then the actual conducting layer metallic layer is the top layer and the bottom layer which is represented in orange here and here. And then there is the insulating layer which is the dielectric material and that is usually made using FR4 substrate. You can also change the substrates that you want such as aluminum and copper etc. So, the conducting layer is usually copper because it is easy to work with as a metal, it is malleable and ductile, and you can easily form shapes and cut shapes. Now, the same layers are expanded below.



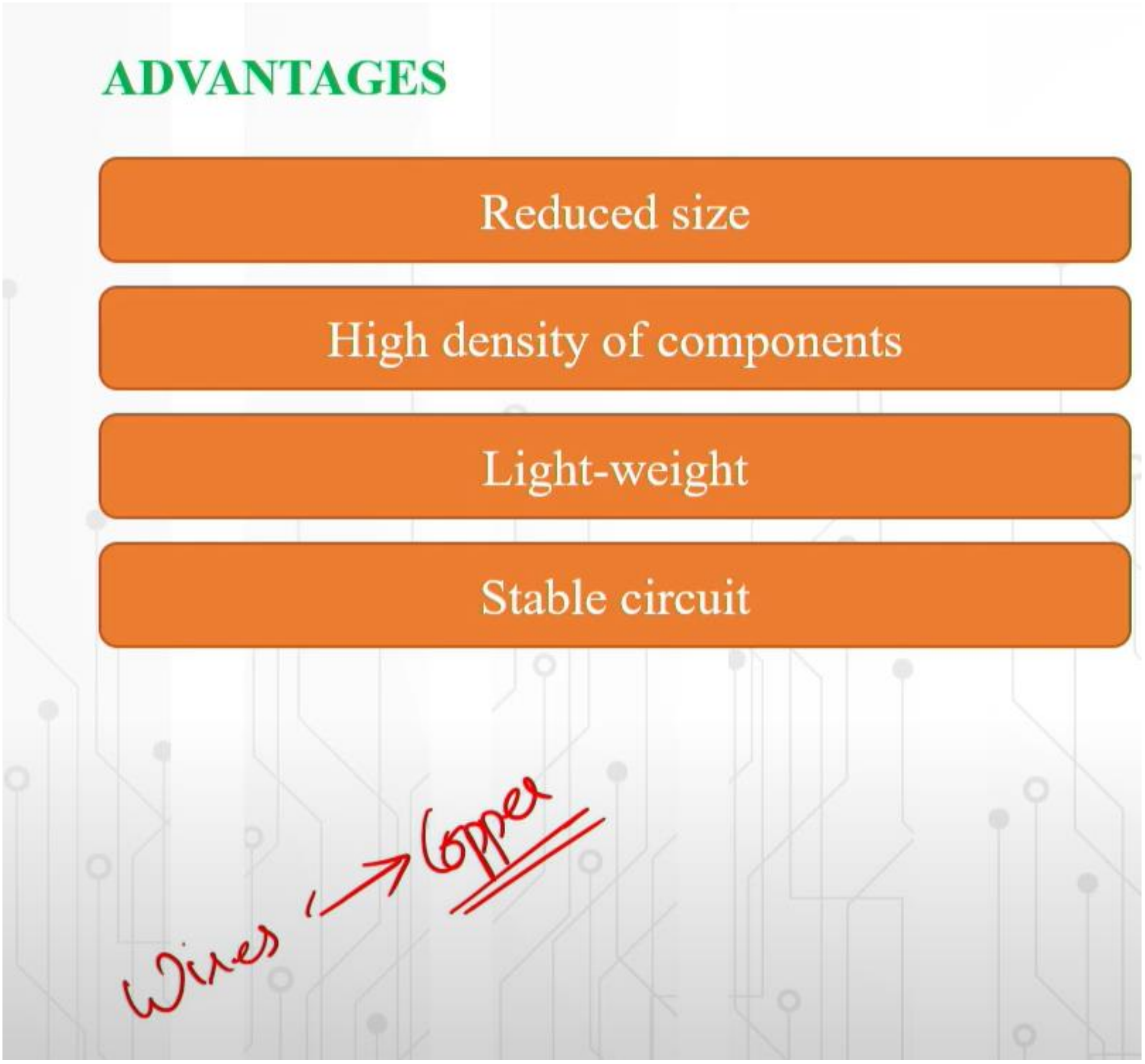
We will start from the middle. So, there is a laminate or a dielectric in the middle and it is usually made using FR4. FR4 for your reference it is similar to the fiberglass material that is usually present in boats, any water boats that you may have gone in use the same fiberglass material. Now, it is a resin which is cured and then put into thin strands and then pasted together to form a thick board. Now, above that and below that we use the copper layer.

Why do we use even number of layers? Because it makes it easy for the manufacturer to make even layer boards instead of odd layer boards. And here top and bottom we have the inner copper layers and then comes if you are using multiple layers let us say, this is a two layered PCB representation on the top and here it is a four layered PCB, see that there are four layers of copper. Now, if you are using multiple layers, we use uncured FR4 which is usually called a prepreg. Now, this prepreg is kept between the top and bottom copper layers to hide it or insulate it from the inner copper layers. Now, how is the communication or the link between the inner copper layers and the outer copper layers done? It is usually done using something called as vias.

So, vias are elements which are drilled between multiple layers, they are similar to pillars which connect one layer to the another. So, vias are present in between layers to interlink communication from one layer to another. Here, you can see this is a mounting hole on the corners, but this is a good representation of a via that goes through all the layers, the 4-layer PCB that is shown here. And these thin strands that you see, all those thin strands are via lines that goes between the top layer and the bottom layers. Now this is a simple representation of how copper is used as a conductor, and it is sandwiched between dielectric substrates inside a PCB.

And this is usually the thickness is usually measured in mils which represents 1000th of an inch. This is the imperial form of using it or the other way is to use MM. Now this is the general representation of the internal layer of the PCB. Now why do we need a PCB? Apart from the initial representation that I had here, you can see that when the number of connections increases like this, we tend to miss out or we tend to make errors when connecting multiple components and it is not feasible for upscaling the circuit. When you need to do multiple PCBs like multiple circuits like this, a PCB is generally a good way to do it.

Now, what are the advantages apart from that? So, one advantage is that the same circuit that I showed there can be made into a very small form factor. It can be made very small such that you can use it use multiple PCBs in the same area where you had that single circuit. And, then the high density of components because of the reduced size of the wires instead of wires we are replacing it with copper lines. Now, these copper lines are very small. As you saw we use mills 1000 of an inch. Since they can be very small you can imagine the density of components you can place multiple components in the same area of the prototype circuits that you do.

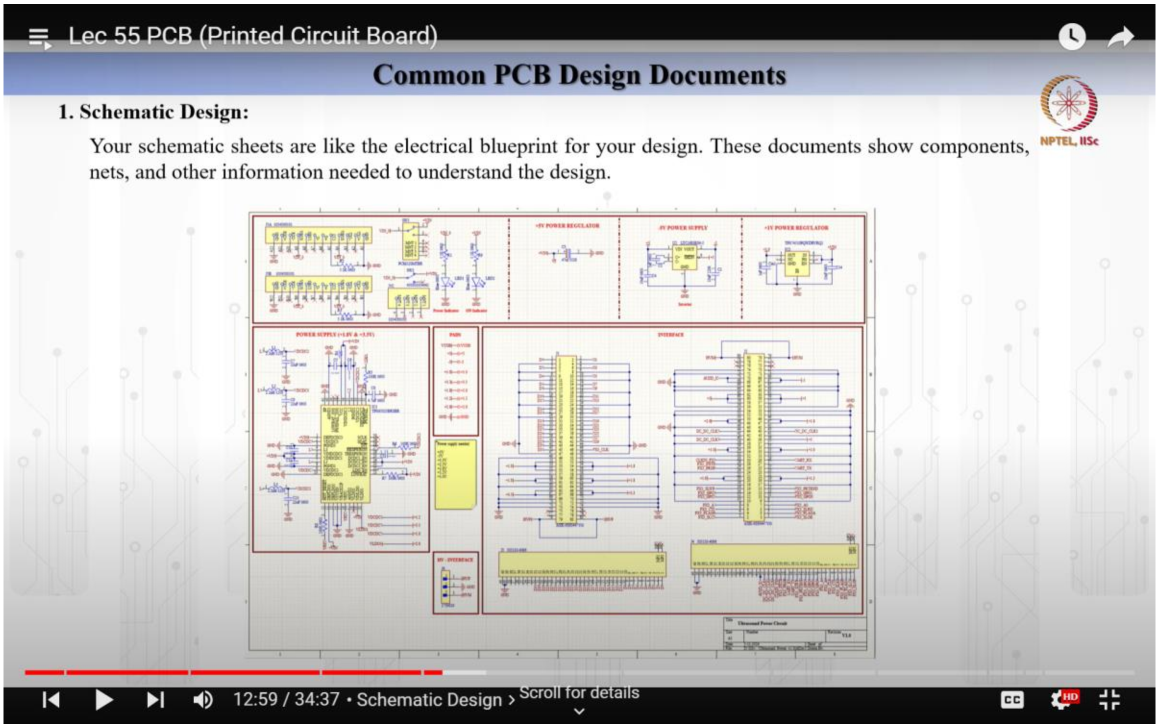


And since the wires are replaced, the weight also decreases significantly because the size is reduced the weight is again significantly reduced. This makes it a very lightweight component and it is a very stable circuit. So, when you design it properly with whatever functions that you had in mind, when you provide the intended input, you can get the expected output. So, this is a very stable circuit and can be repeatable as well. Now, there are certain disadvantages as well.

So, if you had noticed we print the PCBs and once you print the PCBs, it is very tough to make multiple alterations. If you have to make major changes to the connection, let us say a wire was connected to different pins and you have to replace it. It is easy to do in a prototype, you just plug it out and replace it, but in a PCB, it is very tough to do that. So, redesigning the altered circuit is extremely difficult and repairing is very hard. Let us say you have to remove a trace, you have to scratch through the surface, remove the traces and then resolder something using a thin copper wire.

So, that takes a lot of time when the circuit is very complex and dense especially some of the integrated circuits come with very dense pins it is very tough to manually repair it. And the process of making a PCB is even though it is getting better the some of the processes like the etching which uses chemicals are very harmful for the environment. And as we are increasingly growing in the semiconductor industry, as we continue to use these methods, it is going to degrade the environmental conditions as well. So, these are some of the disadvantages. So, when we do not need to actually use a PCB, we should prefer to go with the prototyping. Now in a PCB generally, there are certain let us say certain documents that you need to create.

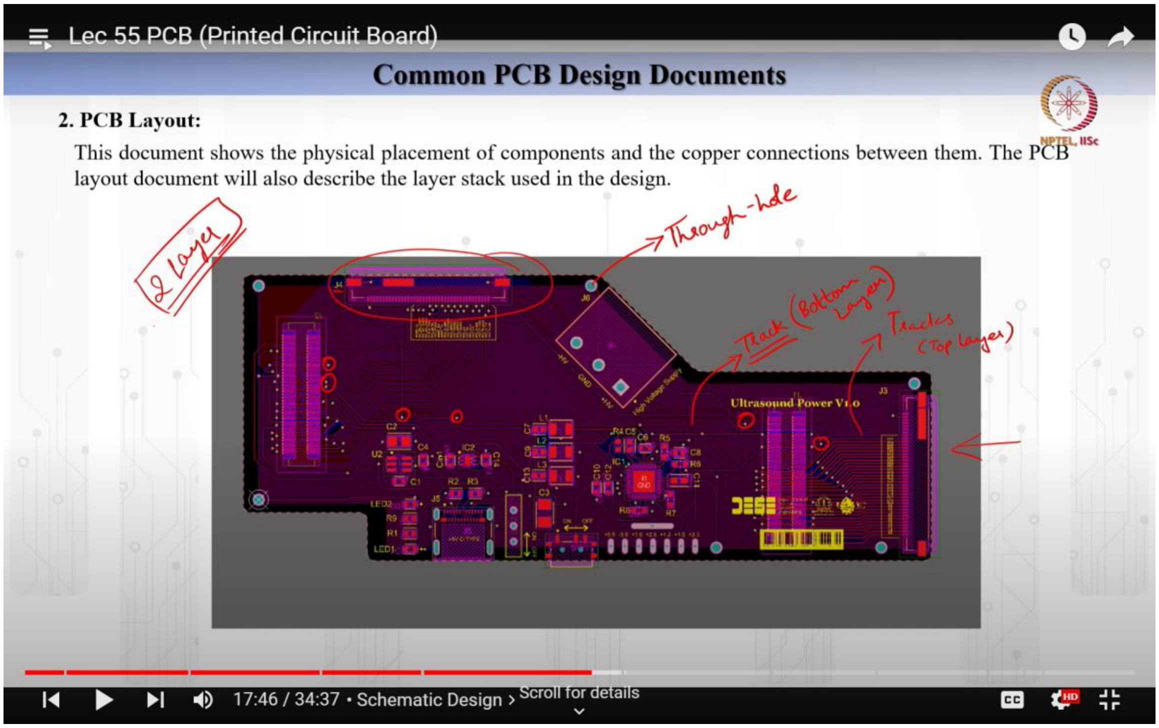
These are the general documents present in all design software. So, let us say you have chosen a software that you want to use to design a PCB, you will have to go through these steps or these documents in order to make a complete PCB. So, what are these documents? So, first is the schematic document or the schematic design. So, this schematic design is similar to an blueprint for your building. So, before you build a building you make the drawings architectural drawings in your blueprint.



So, this is the electrical equivalent of it. So, you draw your designs and connections between multiple components in your schematics and then you translate them into a physical PCB. Now, these documents what do they contain? So, they contain all the schematic for the components. So, for example, I have a simple connector here if you see. So, the connector is represented as a square and all the connections that can be taken out are given a simple line on the left and right side here.

So, what I do I interface other components through these lines it is similar to connecting your components on a breadboard and wiring them together. So, instead of wires we use something called nets in order to make this schematic simple and easy to read and in order to not make them very cluttered we use something called nets. So, when we use the same net between two wires. So, let us say I use ground here. I use the same ground here; it means that these two are actually connected by a wire.

But instead of using wires here, here and here, instead of making them cluttered, we prefer to use nets. Nets make it simpler, and it makes it easy to read for anyone. Now, this is the schematic which gives an overall idea of the design that you are intending to do. Then comes the PCB layout where in the previous slide I showed you a simple connector. So, the same connector is given as the actual footprint actual with the actual dimensions which you can place it on a PCB and connected.



So, the same connector is shown here in with the designated J4. So, this is a simple example that I have taken from our work here which was for a power delivery circuit for ultrasound system. So, this PCB is the translated PCB of this particular schematic. So, this schematic is translated into this particular PCB. Now, you can see that the shape of the PCB is not square like a breadboard or a puff board, you can change the shape according to the system that you are designing.

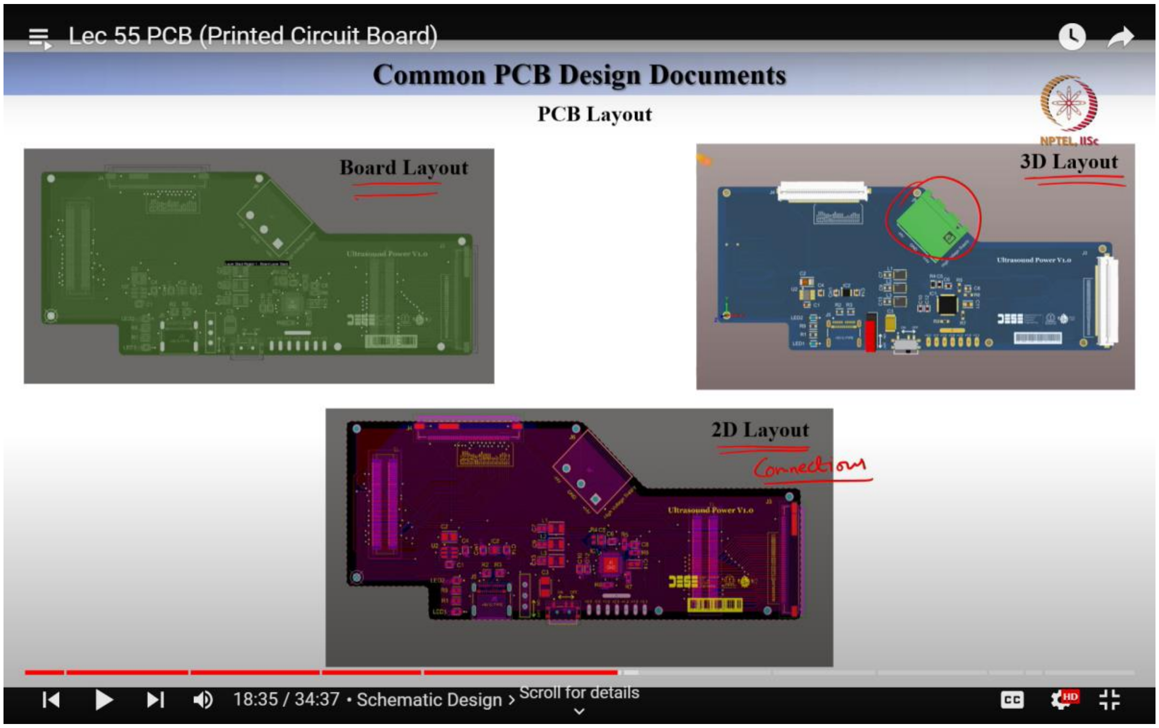
So, usually people work with mechanical engineers to determine the formations of your casing or the system and then determine the PCB dimensions accordingly. In a multidisciplinary system, in a team with multidisciplinary personnel, this is really useful. Now, here we make physical connections. Unlike nets, here we have to physically connect the components together using something called tracks. So, these are called tracks, the individual lines that you see on the right in simple red colored lines, these are called tracks.

All the items colored in red are actually in the top layer in this particular design software and all the items colored in blue which you may or may not be able to see are in the bottom layer. So, this is actually a two layered PCB. Since the number of components are very less, I have used two-layer PCB and can see that two layer PCB and you can see all the components populated here. All the components colored with light blue are multilayered or through hole components, which means there is a hole through and through there is no physical PCB present there. So, these can be used as mounting holes whereas, here if you see these are the small holes present which is not visible.

But these are the wires that interconnect my top layer with my bottom layer. So, if I have to jump through certain components, I go to the bottom layer and come back up in the top layer here and then connect the pins that I need. So, this document is used for the physical placement of components and the copper connections are given using tracks physically and then the interconnections are made. So, this will describe the layer stack that you use as well. So, the number of layers that you use has to be determined in this document not in the schematic.

So, finally, when the vendor with whom you are going to manufacture asks for your design, we will be generating files using this PCB layout and then giving it to the vendor. So, this layout is the most crucial component of PCB design. So, PCB layout the PCB layout document again can be of multiple types. The previous in the previous slide I showed you the 2D layout where we actually perform the connections between the multiple components using the copper tracks. Whereas, when you are working with 3D mechanical engineers or 3D modeling, you can use the 3D layout and the board layout.

So, this enables you to determine the shape and the dimensions of the board to check whether the components height matches with the spacing that you gave in your system and so on. So, this is also crucially useful and in some design software's you get these options as Now, components according to the document that you are working on are termed differently. So, physically we call any electrical item a component. For example, this is a resistor, it may not look like it, this is a for those who do not know this is a surface mounted device, there are two types one is the through hole and the other is the surface mounted device.



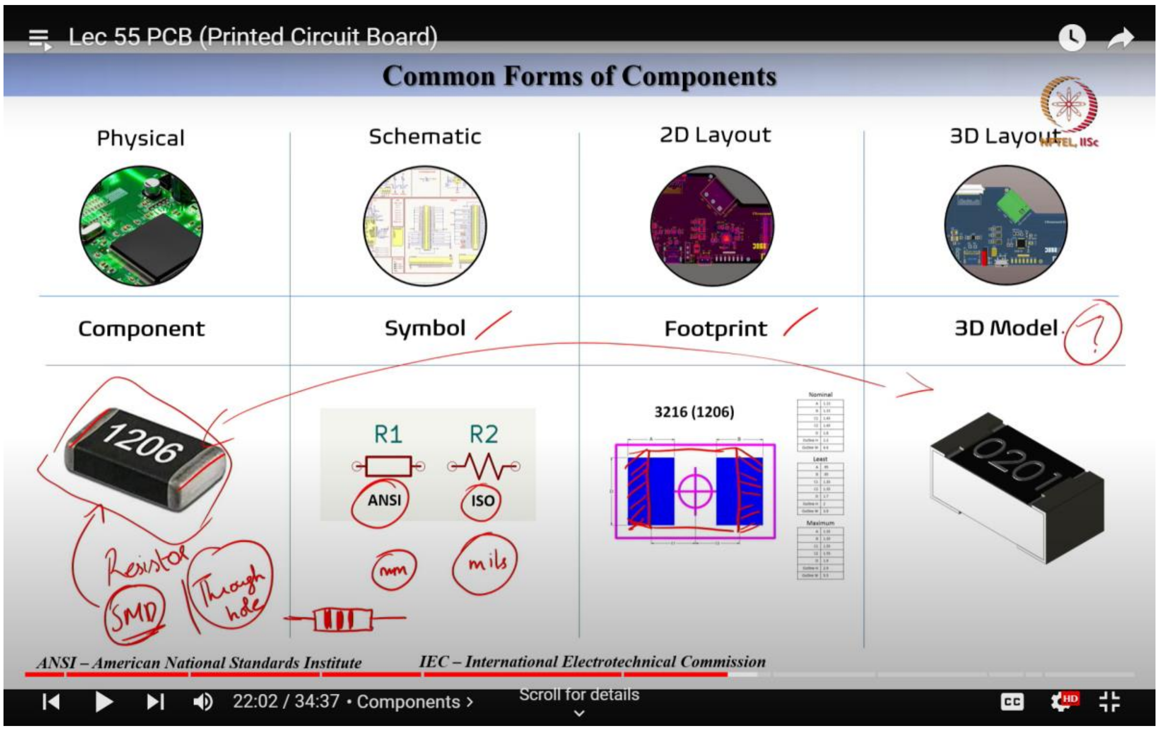
This is a surface mounted device. It has two electrical contacts which has to be soldered on to a PCB. This is what you generally find in a PCB. The other type of resistor that you may have seen through hole looks something like this with wires out and then there are multiple bands in the resistor this is the through-hole resistor. Now, in order to reduce the size and the form factor of the PCB we generally prefer SMD resistors. Now, in schematics in physical form this is called a component whereas, in schematic this is called a symbol.

So, the same resistor is defined as a symbol as I have shown here and there are multiple formats representing it. Just like we use MILS in the imperial system and MM in the metric system, we use IEC format as well as ANSI format. ANSI is from the American National Standards Institute; IEC is from the International Electrotechnical Commission. So, you can use anyone except that there are minor changes to the way it is represented, that is all.

Now in a 2D layout, it is called a footprint. Now the component is represented as a symbol in schematic, it is represented as a footprint in a 2D layout, and the dimensions are just the translation of the real life models. So, the dimensions remain the same, so that when you give it to a PCB vendor, they can make the pads for soldering the resistors just like you intended. So, usually we use components when it is translated to the footprint, we just use the pads or the places where we need to solder the component. We do not actually represent the component itself. But just the footprint of the component as the name suggests, we just represent the places where they need to be soldered.

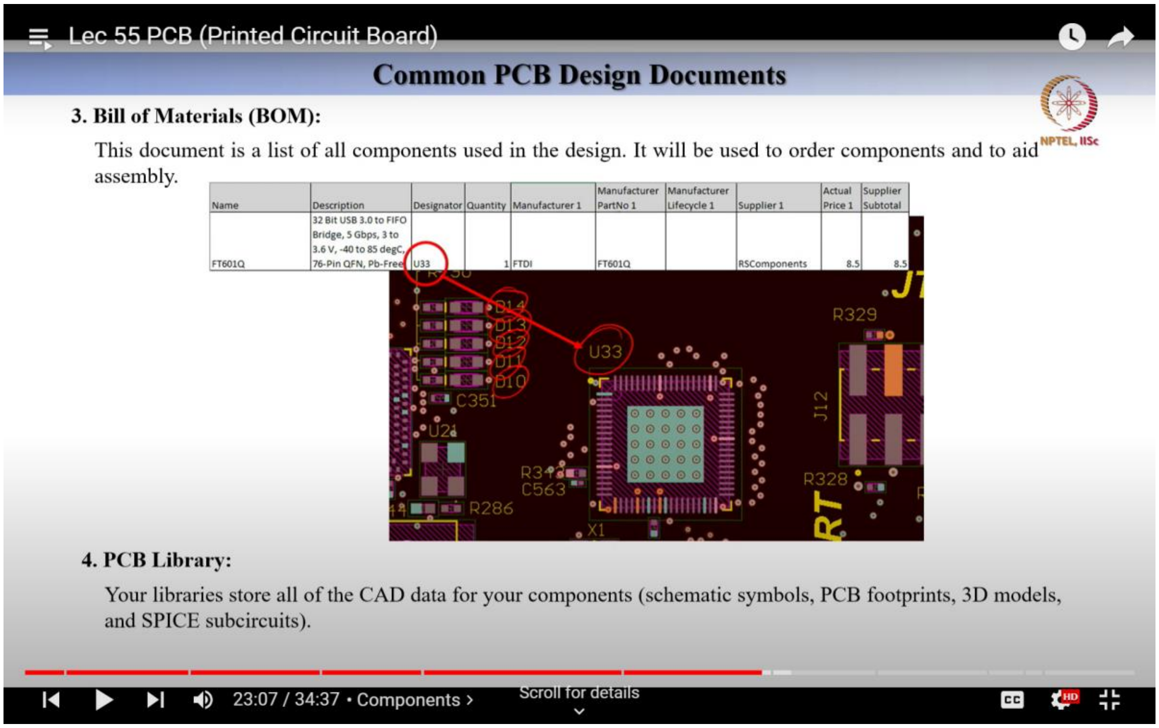
For example, this will be just more than the size of the resistor. So, when you place a resistor and when you solder them, you can attach them to the PCB here. So, this is the actual resistor on top of the pads where you will solder them. So, the pads where you solder them will be determined by the footprint of the component and the 3D layout. The actual 3D layout of the original component will be represented. So, these are the different forms of components that we have in any PCB designing software.

So, the mandatory forms of component that you need to have when PCB designing are the symbols and footprint, 3D model is optional depending on your use case.



After now that we have discussed the PCB design documents, schematic document, there are other documents as well such as the bill of materials. When you have a very huge PCB design, and you need to place the order for the components that you have used in the PCB design. You can use this particular document called the bill of material. So, it contains the list of all the components that you have used, the unique designators here these are the designators which are unique for each component will also be represented.

In some cases, the manufacturer and the price may also be represented depending on the software that you use. So, you get a complete description which you can directly give to a vendor, or you can place the order easily in case of a huge or a bulk component usage. Then comes the PCB library, so how do you get all the documents that I showed previously, the symbol, the footprint and 3D model, how do you get that? So, you get that from a PCB library which stores all your CAD data, the computer aided design data for your components such as the symbols, footprints and 3D models and in some cases you may be able to do simulations. So, in those cases you can even get these PI model sub circuits, if you want to do simulations. So, these are the general documents that you need to look for in a PCB design.



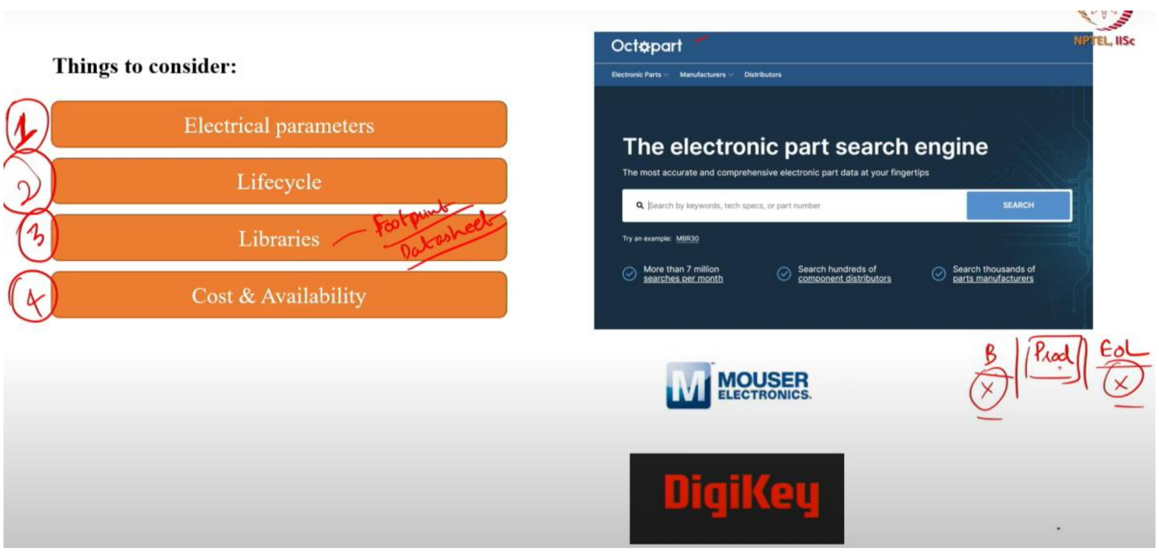
One more thing to consider before doing a PCB design is that when doing a schematic, you will have to connect different components together, but when you are using a component there can be multiple components with the same functions. How do you select the component that you need? So, here are some things to consider based on my experience so far. So, the first thing that you need to consider are the electrical parameters. So, this is something that you set or your client sets for you. So, the electrical parameters cannot be changed because that is your major requirement it should be able to handle a certain current, it should be able to it should consume less than the set amount of power so on and so forth.

Now, and it should follow a specific conformal function that you have set. Now, once the electrical parameters are set what else can you look for. So, whenever an industry standard PCB is made, they look for these parameters such as life cycle. So, component can be in its beginning of life cycle for example, a new resistor that has just been introduced to the market. Or it could be in the production stage where it is produced in bulk quantities, and it will be in production for 5 years let us say.

Or it could be at the end-of-life cycle. So, it could be in the beginning, or it could be in production, or it could be in end of life. So, depending on what you select beginning product may have some bugs which may not be desirable and end of life if it is not used for a prototype if needs to be ordered in bulk may not be suitable as well. So, you need to search for the one that is most suitable for you, in most cases it will be the components in production. So, depending on your cost and availability and the life cycle you can choose your user own cup of coffee.

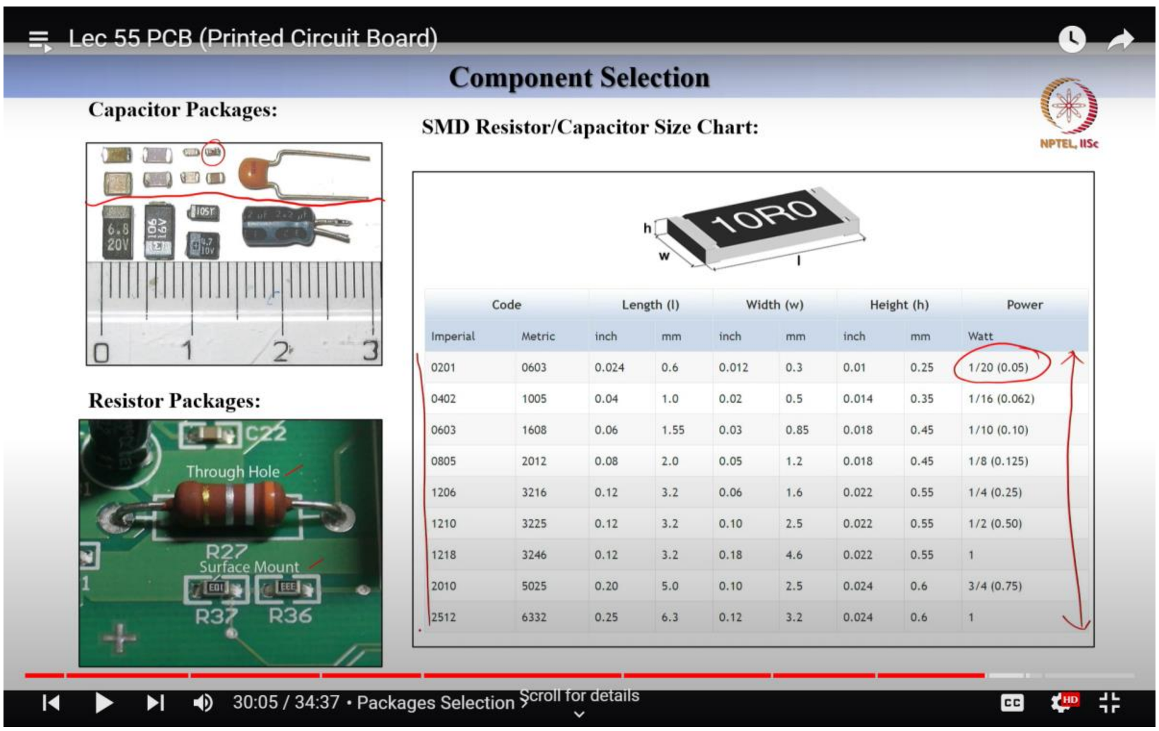
So, that is the second one then comes the library. So, you if you have if you have chosen a component which is widely used you may get the libraries readily otherwise you may have to create the libraries by hand and most of the software's allow you to create the libraries by hand you just have to look for the footprint which is the crucial most crucial one from the data sheet. So, any components data sheet will have the footprint for any component. So, you can use that to create your own libraries. So, that is the third main crucial factor. Now, at the end as I have already mentioned some of the components if it has been the beginning of life end of life may have a very cheap cost compared to the ones in production or if it is a very rare component the cost will be very high.

So, depending on the cost and how it is available if it is readily available, or you have to place a lead order. So, depending on those factors you can again filter your components. So, you can use the resources available online such as Octopod, Mouser or DigiKey or any other resources that you can find in order to filter the components that you need.



And in component selection depending on the form factor of your PCB the surface mounted components or the through hole components can be selected. So, there are different packages, it is called packages and there are different packages available for each component.

So, in terms of capacitor you can see on the right it is a through hole component, these are through hole, this is through hole and on the left, these are all SMD components surface mounted devices. Now, depending on your requirements you may have to use through hole or in some cases or in some cases you can just get away with an SMT component. And in capacitors also you can see that they have ceramic capacitors which is on the top and then electrolytic capacitors which is on the bottom. So, depending on the function you can go for either one. So, here it is represented with respect to a scale this is a general metric scale that we use in centimeters and you can see that the size goes as low as 2 mm 2 millimeter ah, but the disadvantage of going very low is that sometimes the disadvantage of going very low is that sometimes the power rating is very low for some capacitors.



So, depending on the power rating we can choose the capacitors that we need. Similar to the capacitor packages, we also have resistor packages. So, here I have represented a through hole package and surface mount package, you can see the difference in the sizes that they have. And a general way to find the size of the resistor or capacitor is by going through the imperial or the metric code, generally manufacture represented by the imperial code because Both metric and imperial contains the same code.

So, you may get confused with 0603 of imperial with 0603 of metric. What this represents is the length cross width of the component in inches when you are using the imperial code. Let us say you want 0603 capacitor it is 6 mils cross 3 mils in length and width respectively.

So, it is as simple as that. In 0805 it is 0.08 inches cross 0.05 inches. So, it is representation of length and width when you are using the imperial code, and it is much easier to use the imperial code because it is the direct relation to the length and width of the resistor or the capacitor. Now, as you can see here 0 to 01 is the minimal size that we have we even have sizes lower than that, but if you see here as we go smaller in size the amount of power that can be that this device can handle is also very low. So, depending on your depending on your power rating and the form factor you have to reach middle ground and choose one of these capacitors or resistors. So, these are widely used in terms of passive devices. In terms of other active devices, you can go with the different packaging of integrated circuits.

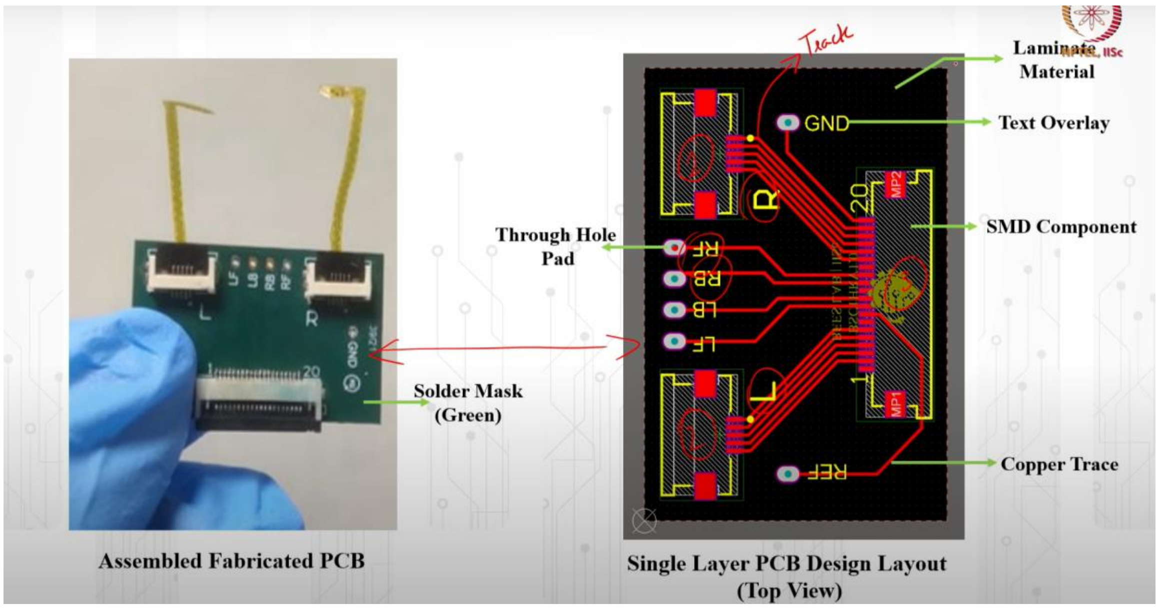
So, here you can see quad flat package, QoP package, you can see multiple other packages, for transistors we have SOT packages, for ICs we have SOIC packages. and then and so on. So, these are selected by the customer according to the ease of connection with your PCB and generally the same component will be provided in multiple packages for ease of use. And we also have with respect to the metric scale the different sizes of some components, transistors, diodes etcetera and some ICs. So, it is a small outline transistor here, small outline transistor with the bigger size, then small outline IC and so on and so forth.

Then once you have selected the components and you know the circuit that you want to make, you can translate the schematic to a PCB design using one of these software. So, these are some of the softwares available for electronic design automation, Easy Eda is one free software, but it is web based. Then there is SkyCAD with this open source, there is Altium, R-CAD and Eagle and so on. So, this on the right is made using Altium and I have shown the schematic on the top and the PCB layout on the bottom. So once you make the PCB you send the Gerber files to the manufacturer and you get a PCB something like something that looks like this.

So, this is just the interfacing PCB that we made which is assembled with the components here on the top and the bottom. in order to interface the sensors that we have fabricated. So, this is a neural acquisition sensor that can be placed on the scalp of the brain or on the skull in order to receive ENG or ECG or ECOG signals. Now, here I have shown a blown-up diagram of the PCB layout of the same physical PCB on the left. So, in this PCB layout you can see the PCB only has one layer the top layer it is a single layered PCB, and these red lines are the tracks which represent the copper lines connecting the different components that are present.

And these are the three connectors that we have used flat connectors which you can see here as well. Now, the things in yellow are actually drawn on the top overlay layer it is also called the silkscreen layer which actually gives you an idea of where your components have to be placed. What are these through holes for? If you want to represent something you can use them for the ease of the user. Now, there are also through hole pads given here which means there is a hole through and through there is no PCB physical PCB present there and we use it to connect some reference and ground lines. So, similarly for the ground and reference we have given some through lines and the text overlays also shown and this black area represents the laminate material or the PCB substrate in our case it is the FR4 substrate.

So, this is the general idea of what a PCB is, how we can translate your circuits to your PCB schematic and then the layout and then 3D model, finally you can send the Gerber and get the physical PCB.



So, you would have gotten a general idea of what a PCB consists of. and how you can go about selecting your components for the PCB and how you can translate your circuits for efficiently obtaining the signals. And I hope that this will be useful for you to grow your interest in PCB designing. If you feel any difficulties or if you have any queries, please feel free to reach out to us. And as always, thank you for watching and see you next time.