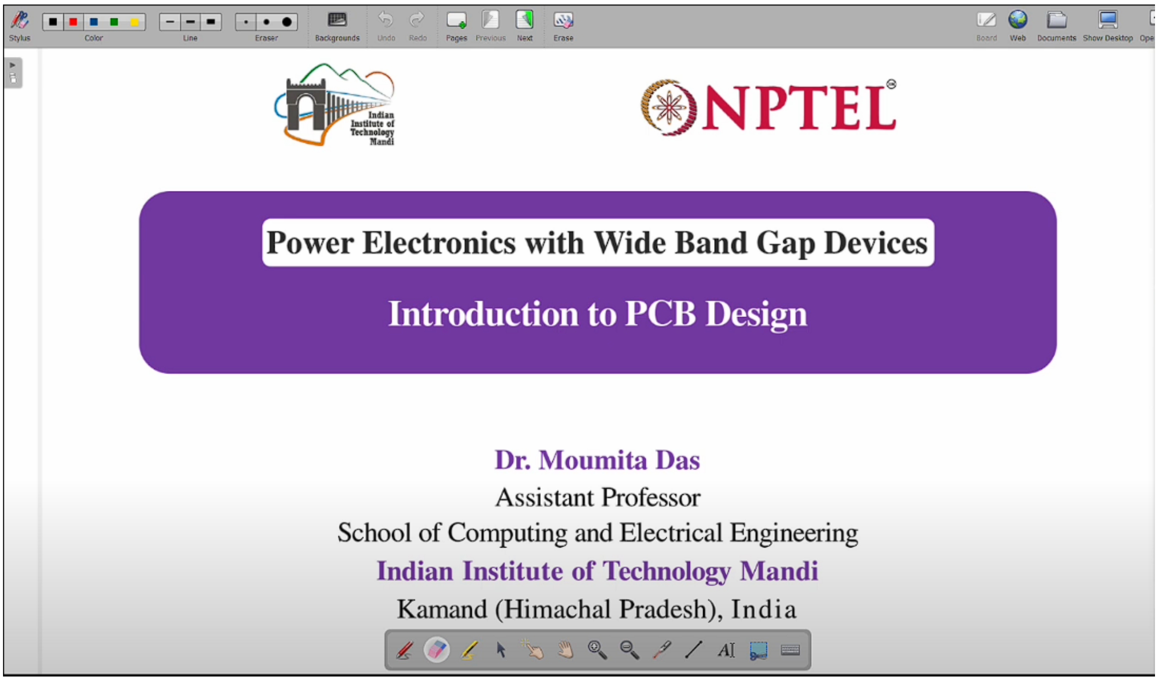
Power Electronics with Wide Bandgap Devices
Dr. Moumita das
School of Computing and Electrical Engineering
Indian Institute of Technology, Mandi

Lecture-32
Introduction to PCB Design

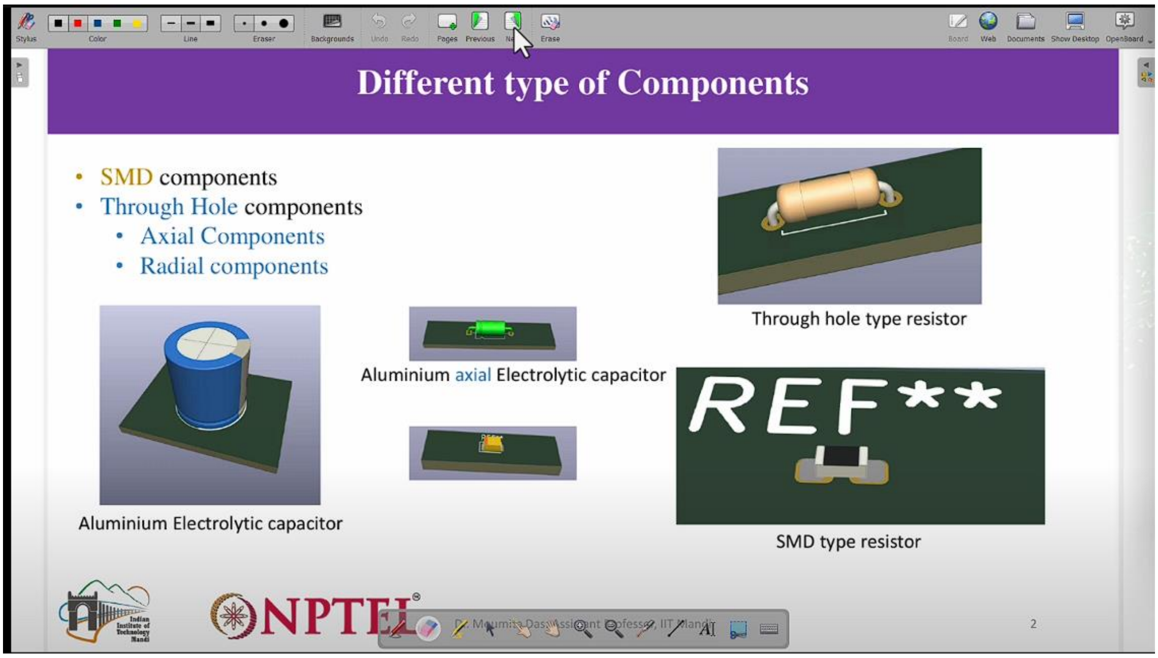
Introduction to PCB Design

Refer Slide 0:22



Welcome to the course on power electronics with wide band gap devices. Today I am going to discuss about designing of the PCB. As I have discussed in the last lecture the importance of PCB for improving the power density. So the layout designing is very important for the power density improvement, right. So, let's see how this PCB is playing important role and how to design a PCB using any particular software.

Refer Slide 0:52

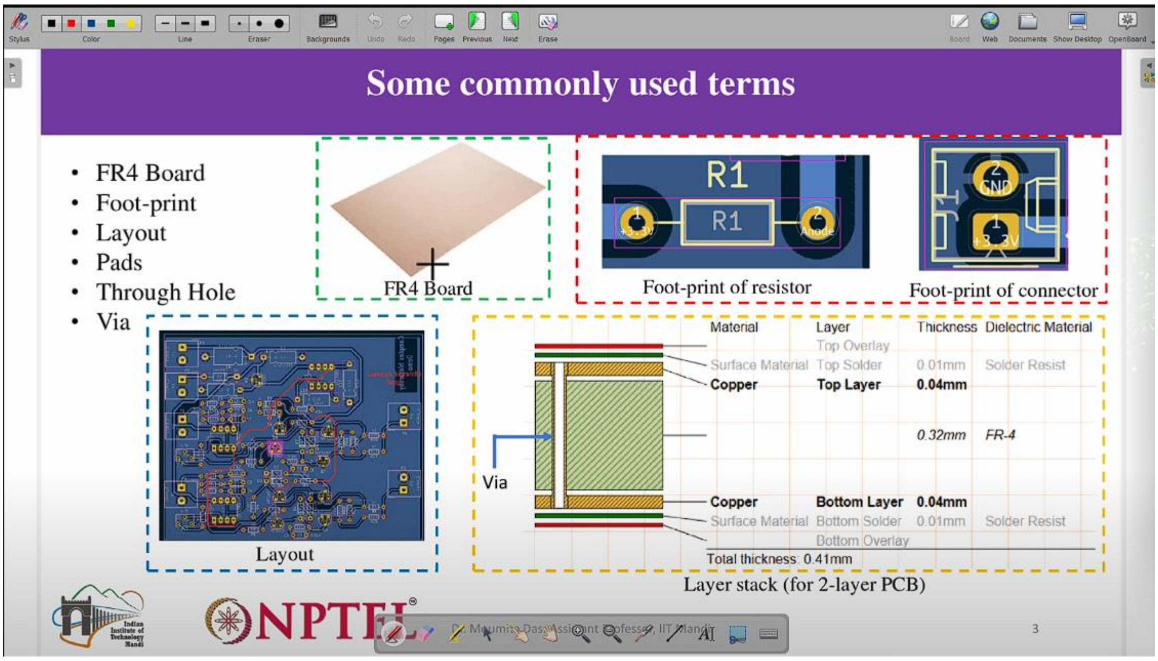


So, in this particular lecture, I am going to discuss about designing of PCB using Kicad software. So, the commonly used term in the Kicad software or any PCB design I am going to tell you now So there are actually different types of component which are available Similar to the simulation which I have discussed earlier So different term you can find in the PCB designing So first is the SMD component So PCB is printed circuit board So basically we need PCB so that we can connect different component to form a converter right so you can see here so different components are given here so smd component through hole component axial and radial component so when i say SMD component you can see here this component is basically very small you can see here so this is basically SMD component SMD type of the register you can see the part of this like this particular component is placed on a printed circuit board or the PCB so the component size is very small now on top of this there is a Through hole type of resistor. So where you can see the leads of the register is like it is visible here So this particular part or this particular part. This is the two leads of the register which connects positive and negative Point of the register. So this is actually visible but in through hole component these are not visible So that is the thing is very important.
So you remember when I was discussing about this power density or the loss calculation So, this leads play very important role for increasing inductance and resistance of that particular part. So, in order to reduce the parasitic of any of the converter, these leads play very important role. If we can remove these leads, then the parasitic part can be removed. Optimized and also like in case of power density improvement if the size will reduce down if lead is not there So again, it will help in improving the power density. So that is why for modern power converters SMD type of components are preferred rather than the through hole type of component although this through hole components are very easy to mount and Also, like it is very easy to measure anything in a through hole component.

So now you can see here so it is also having two different types axial component so aluminium axial electrolytic capacitance is shown here in this particular thing and the radial one so then you can see here this is aluminium electrolytic capacitor you see shown here so these are the different types which you can find for any component means the component which you will be selecting either it is active or passive so active components consist of different type of semiconductor switches diodes and the passive component consists of inductor capacitor resistor right so now active component also available in SMD and the through hole types and through hole also there can be different packaging again like based on the application you can select particular type of packaging and SMD it is like like for like improving power density for any specific design if you want to select SMD that is also possible now for passive components capacitor inductor register i have discussed in the last lecture so capacitor also there are different types of capacitor available Obviously, like through hole also there are different types. So, three different capacitors I have discussed in the last lecture. So, how this ESR of the capacitors changes by selecting different type of capacitor, electrolytic or titanium or the ceramic capacitor. So, based on this like obviously then different properties will also change. Same with the inductor, inductor also is having different types.

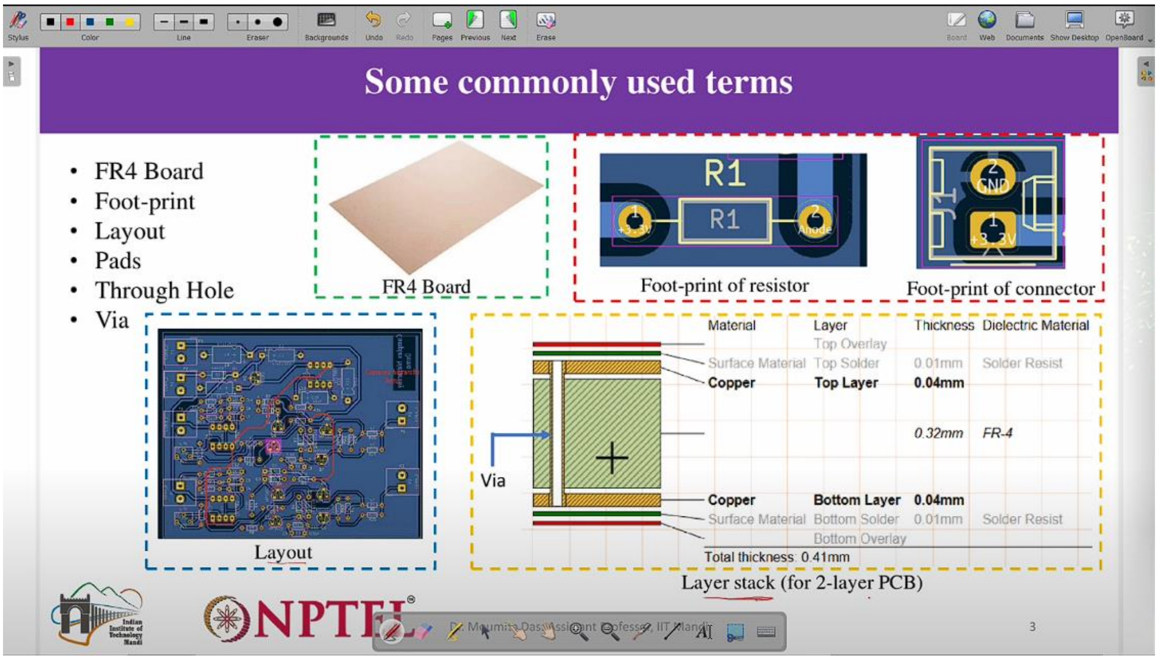
It is like It is having different core types and different sizes. So, integrated type of inductor which is like integrated in the PCB itself or it is having like very small length. So, this is like size of the inductor is very small. So, that particular inductor is preferred if we have specific requirement for high power density system, right.

Refer Slide 5:17



Now you can see here some commonly used term for any PCB design is like there is FR4 board so this board looks like this so you can see here this green highlighted portion then we get the footprint so footprint of the resistor it looks like this so basically two points one and two so it is connected so two leads will be connected in these two points if it is SMD then two points of the SMD will be connected this then the footprint of the connector you can see it looks like this in PCB when we design any converter ultimately the layout so basically converter in the PCB will have design where you can insert different component so the component connection will be there you have to connect different component in the PCB right so you can see here this PCB layout it looks something like this right now once this layout is designed then what we have to do then we can see different pads actually will be there so the pad thickness like what kind of material is there surface material so basically copper thickness and everything you can actually choose you can choose the layer like which layer the component will be placed either top layer or bottom layer what will be the thickness what will be the dielectric material what and what will be the material what will be the total thickness everything you can select so that you can select in this layout layer stack right for two layer pcb so this is fr4 so here you can see here so in this you can actually select this so now the Next is that how to design PCB.

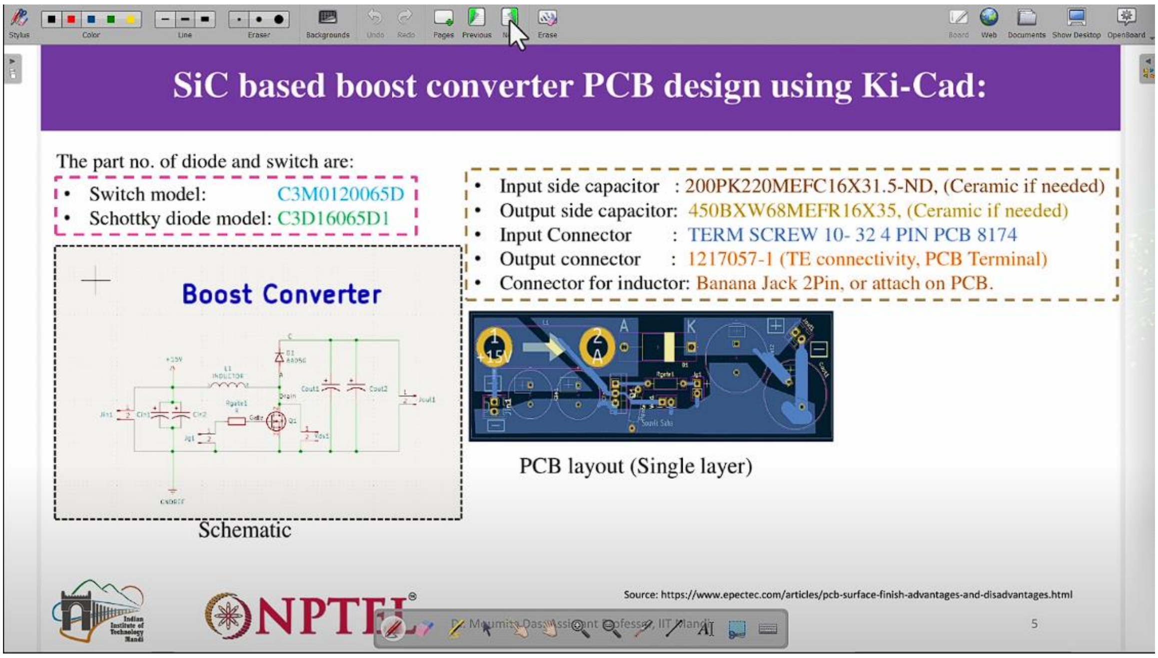
Refer Slide 7:00



So, the steps of designing PCB is that, so in KiCad software, we are going to use this particular software for discussion of PCB design. This you can easily download. So, this is like open source software. So, you can, you don't need license for this particular software. So, you can download it, use it in your PC for designing PCB.

Right. First, you, what you have to do? You have to draw the schematic diagram like the similar way which you have designed like you have used for designing any converter in LTSpice software so similar way you have to design converter schematic diagram or converter in this particular part so let's say it is boost converter once it is done then you have to find like you have to design the layout Right, so then you have to design the layout. Once the layout is designed, then Gerber file will be generated. Now, this will be sent to the PCB manufacturer because they cannot use schematic or the layout. They need this Gerber file which they use to design a PCB for you and then you can use that PCB to connect different component on the PCB board. Right. So, ultimately this will look like this.

Refer Slide 8:44



Now, In this particular lecture, I am going to discuss the same boost converter which was used for simulation in LTSpice software. If you don't remember, you can go back to the lecture and you can just see the boost converter. So, say the boost converter here, it is given the schematic of the boost converter. So it is having switch and the diode model they are same as it is used in LTSpice software.

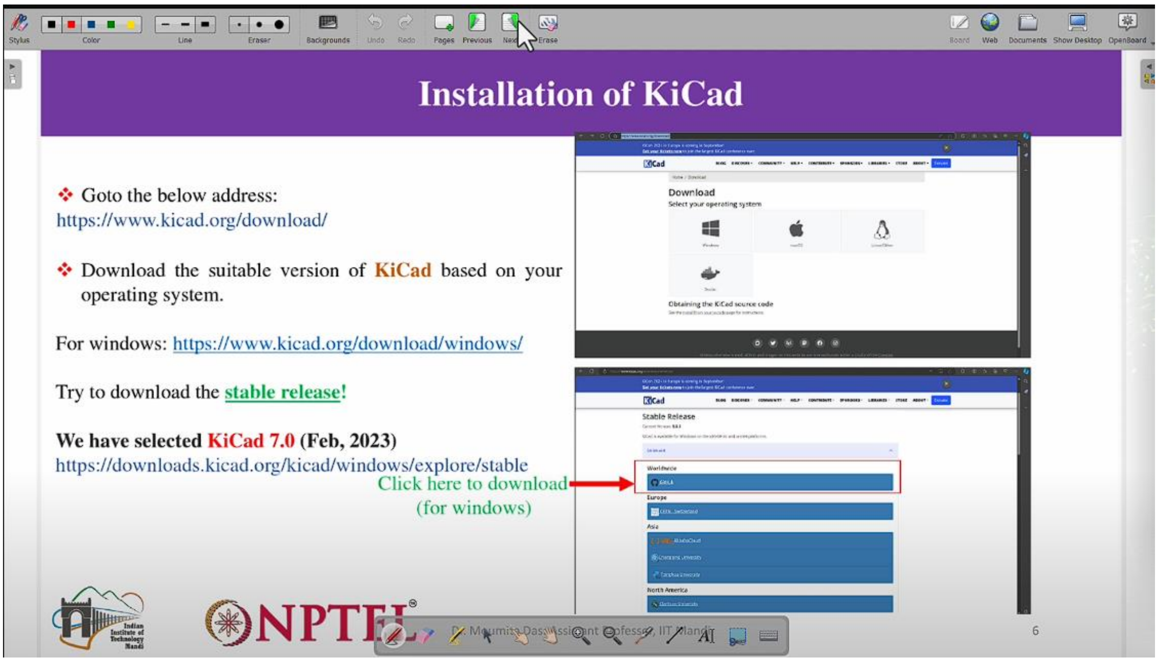
The switch and diode model is very important. LTSpice it uses actual switch and diode. Again like for designing PCB you need actual switch and diode footprint. Otherwise it will be difficult to fit that particular component on a PCB. You need actual part number, actual size, actual distance between each link.

Otherwise, this PCB designing will not be possible. So, that is why these actual models are used. So, you can see here, this is actually switch. And this is diode the same switch and diode. Now you need part number of the other components also.

In LTSpice for inductor capacitor this part number was not required. But again like if it is possible to import any particular kind of passive components which is having like LTSpice model. So then it will be much more practical kind of, it will provide you much more practical kind of output. So, here you can see there are actually two different capacitors are connected at the input. So, input side capacitor, it is selected this one.

So, you can see here this input side capacitor, the part number of the capacitor. is given you can use same capacitor for designing of the pcb if it is like if it is required to be ceramic because you know ceramics case having like less ESR so then you can choose ceramic one if you want to connect ceramic or if it is available an output side capacitor it is given here so these are the output side capacitors right now input connector it is it is this is also very important because you know you need to connect the wires and the input side if you don't have any connection it will be difficult to connect it like from the supply the source like where will be connected to the pcb input so this is connected here so you can see here so this input connector you need to provide so this is shown in the input connector similarly the output connector is also provided here so the like part number of this connector is given and connector for inductor because you know inductor is something which also we need to place like like depending upon the size of the inductor you need specific type of connector which will provide required distance between two legs of the inductor so this connector is also provided for the pcb right now this pcb layout you can see here the single layout it looks like this

Refer Slide 12:05

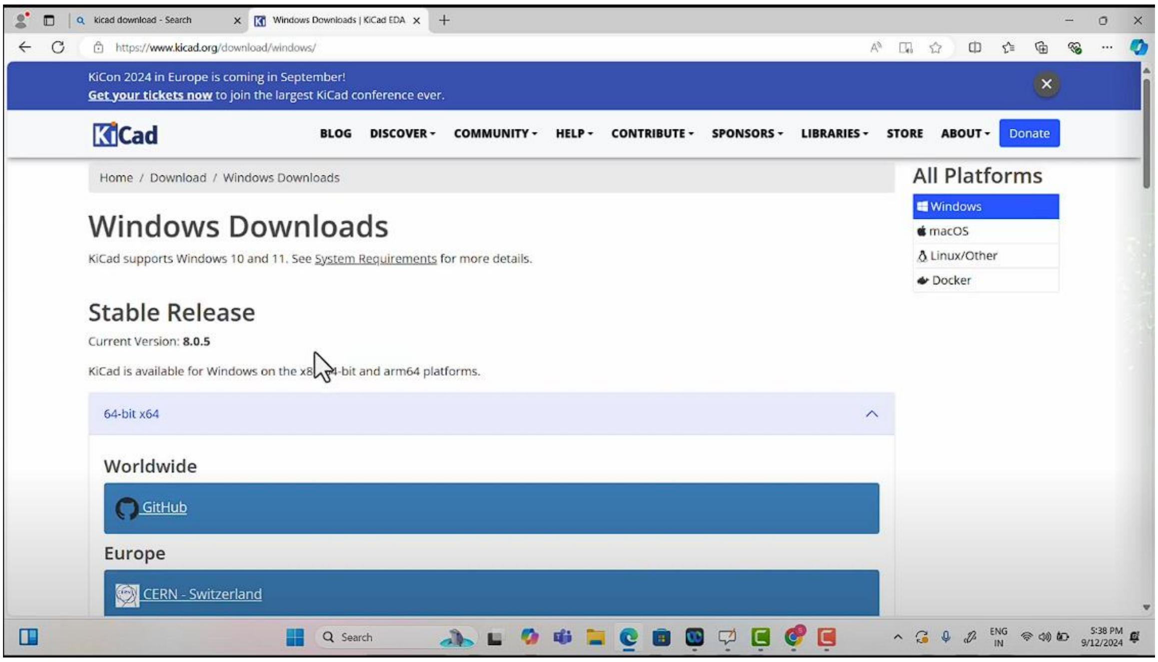


so now let's just see how to use this kiCad software for designing pcb in order to download this what you have to do you can you need to go to this address kicad address so it is www.kicad .org download so the it will look like this this particular window will be visible to you and from there you can actually select particular type of operating system you have if it is windows you select the first one if it is like other operating system then you select that particular operating system for downloading this kicad Then download the suitable version of the kicad based on the operating system.

Okay. Then now for Windows, it will be like this as we are using the Windows operating system. So that is why we have used this particular address. Now try to download the stable release. Now this keycat we have selected. So this is stable release.

Now KiCad we have selected it is 7.0 released February 2023 Now you will be able to see this particular address in your address bar So then click in this particular position to download download the KiCad software Right. See this is how you should be you can search in Google

Refer Slide 13:25

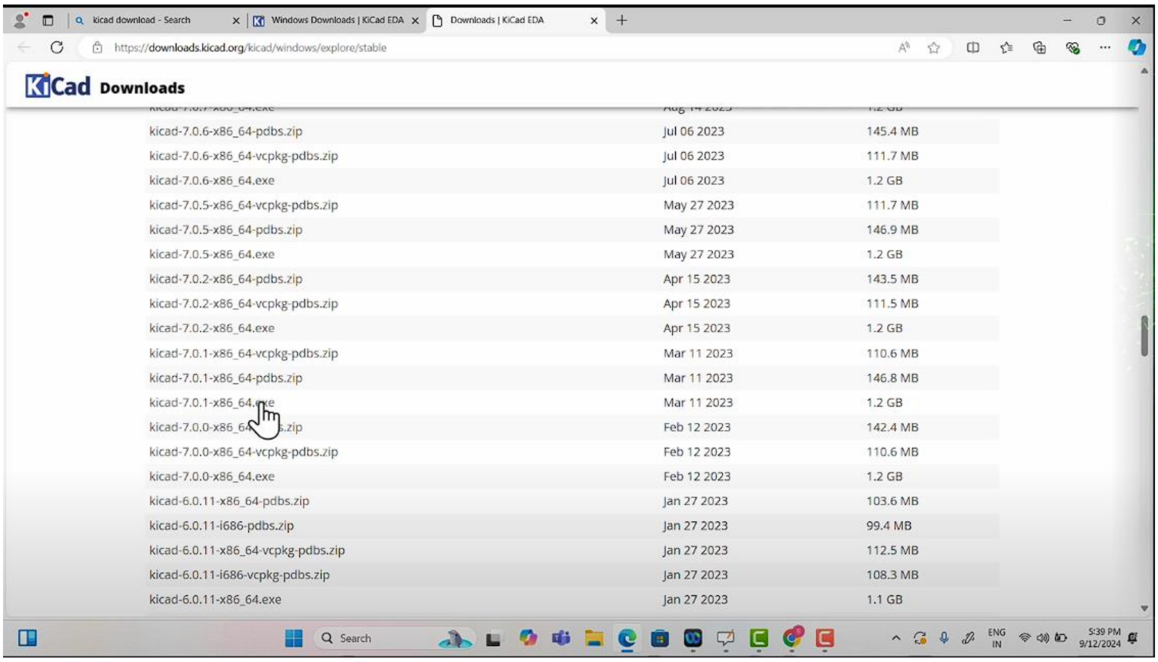


KiCad download then this particular window you can just go and windows download we have entered so we are like in this particular page we are in now we need to select this particular part so then you can see then this particular link you need to click Once you click this, so there are like different version of this kicad. So, we are using particular version for some time. So, we areFind anything you

focusing to download that particular version.

You can use any version, that is not a problem, it is up to you. Procedure remains same, right.

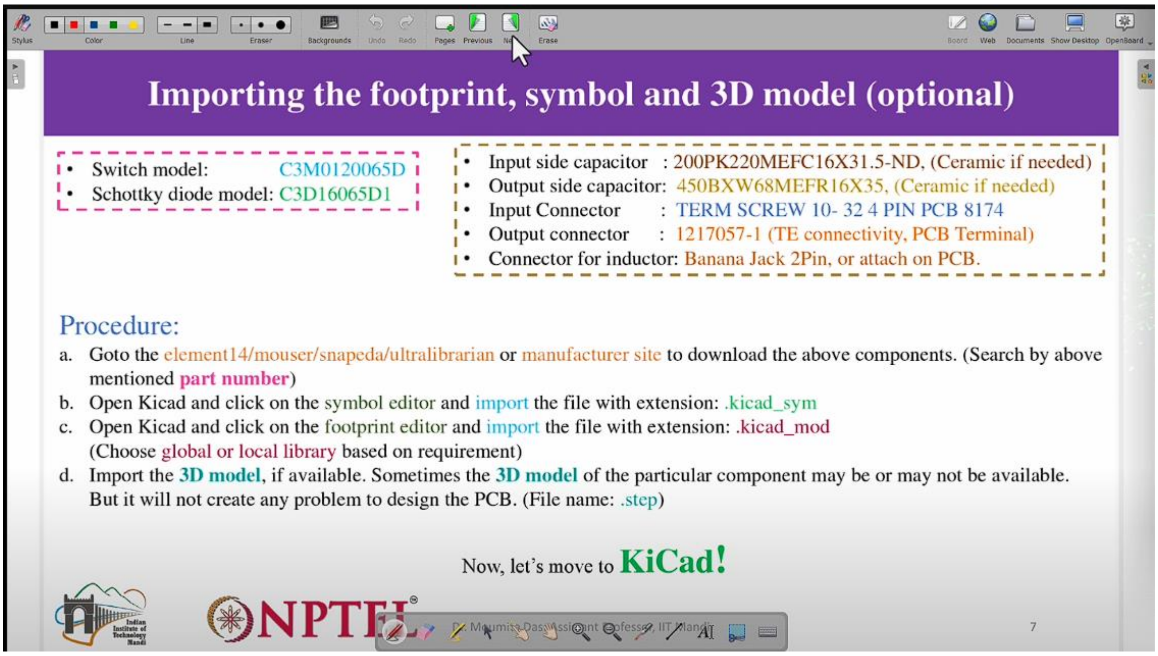
Refer Slide 14:26



So, this particular version we have downloaded, right. So this .exe file, so kicad particular number and then .

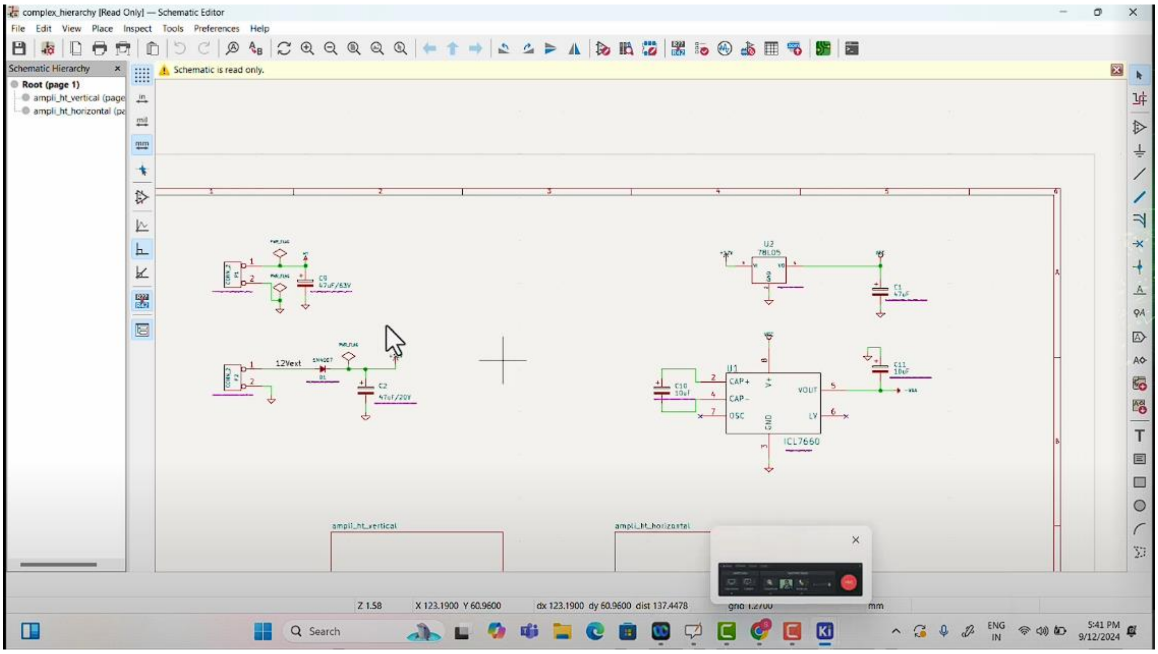
exe file you need to download. Okay. Okay.

Refer Slide 14:46



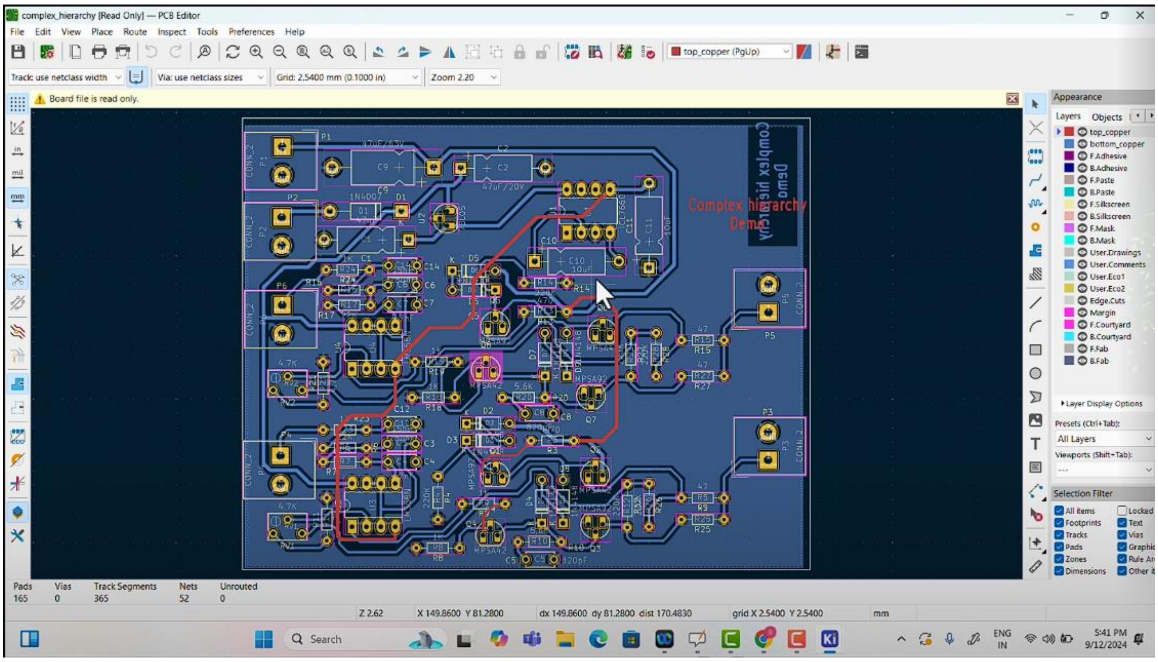
So now it is, like after that it will be installed in your PC. right so let's just open this keycat so that we can actually see after downloading it will have this logo like this ki so then let's open this and see so when you open you will have this different file so what you have to do first first you have to create a new project or you can open a demo project

Refer Slide 15:58



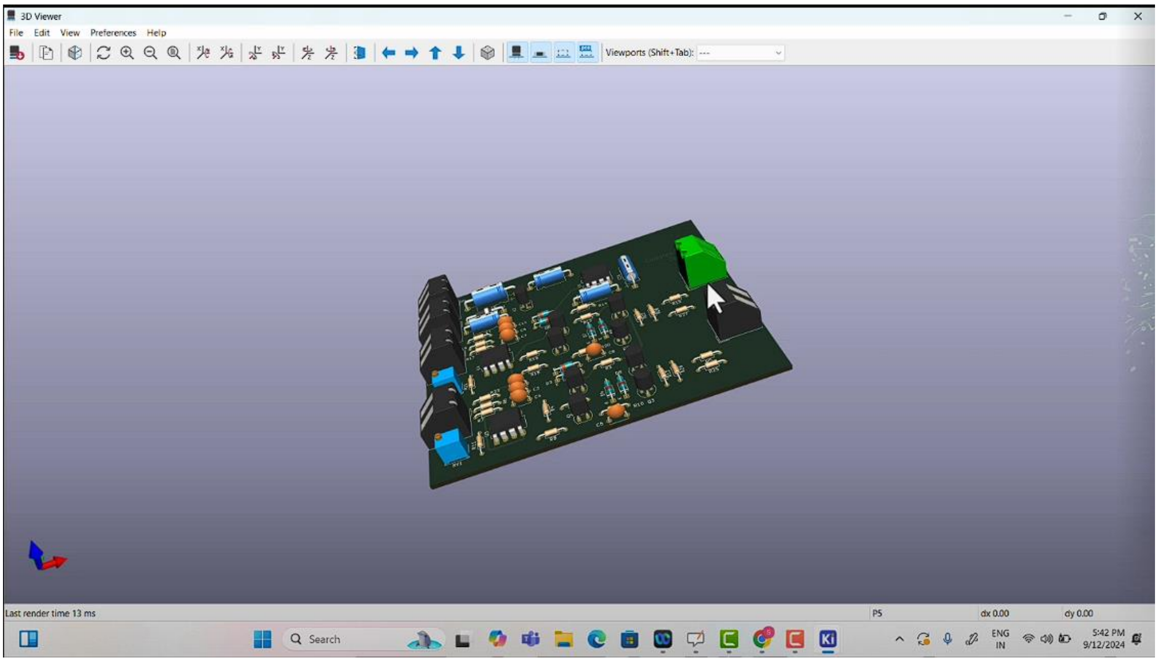
so if you open a demo project or create a new project so then you can just see so like if you want to select a demo project then how it will look like so then like demo project are given so here schematic of the demo project it is like this right so this schematic since it is demo project it is not editable so like once you have this then what you have to do in the procedure schematic is there then you have to generate the layout It is already there, so that is why we are just opening to see the layout.

Refer Slide 16:22



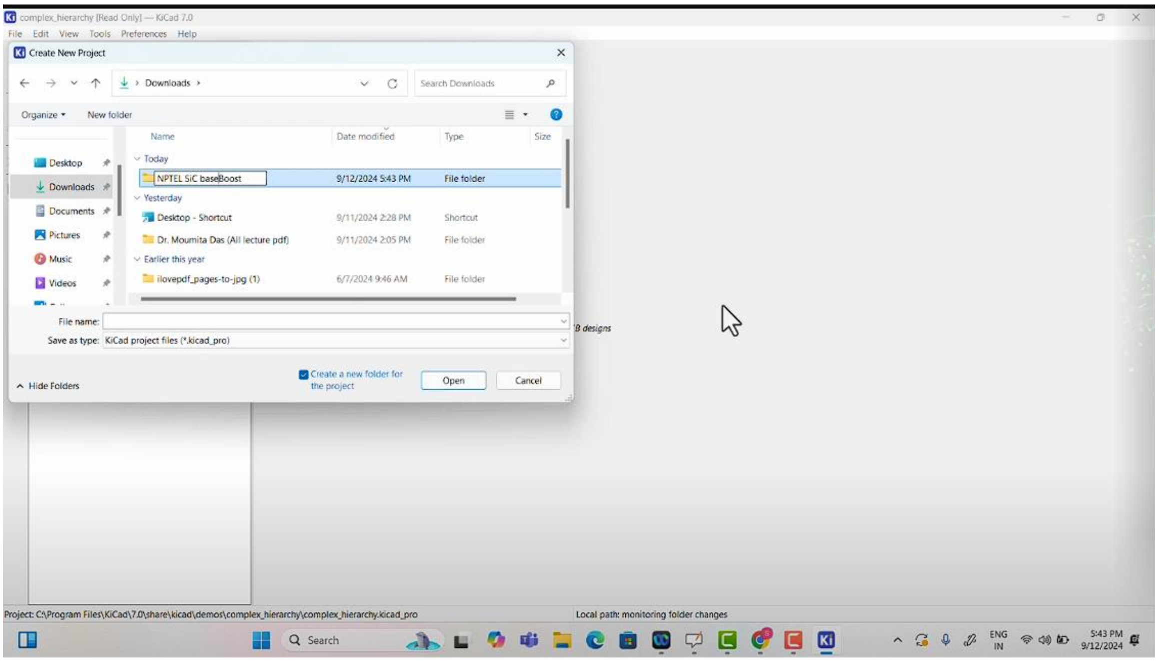
See, this is the layout. This is how the layout looks like. So, different connectors are visible in the yellow color, right? So, connection between different connectors, depending upon the top or bottom layer, so it is either in blue color or in red color. So, like it is two layer PCB, so either top or bottom layer connections are there. So, you have to see where like it is like the wires are connected. It should not cross each other in the same layer or it should not cross any like connector which is placed. So, then what will happen? It will be shorted. So, you have to avoid that. We will see that in details. So, this is how it will look like. So, after that what you have to do? You have to generate the Garber file. So, the Garber file will be generated and this is how the final circuit should look like. So, once the PCB you get and you insert all the component then it will look like this.

Refer Slide 17:23



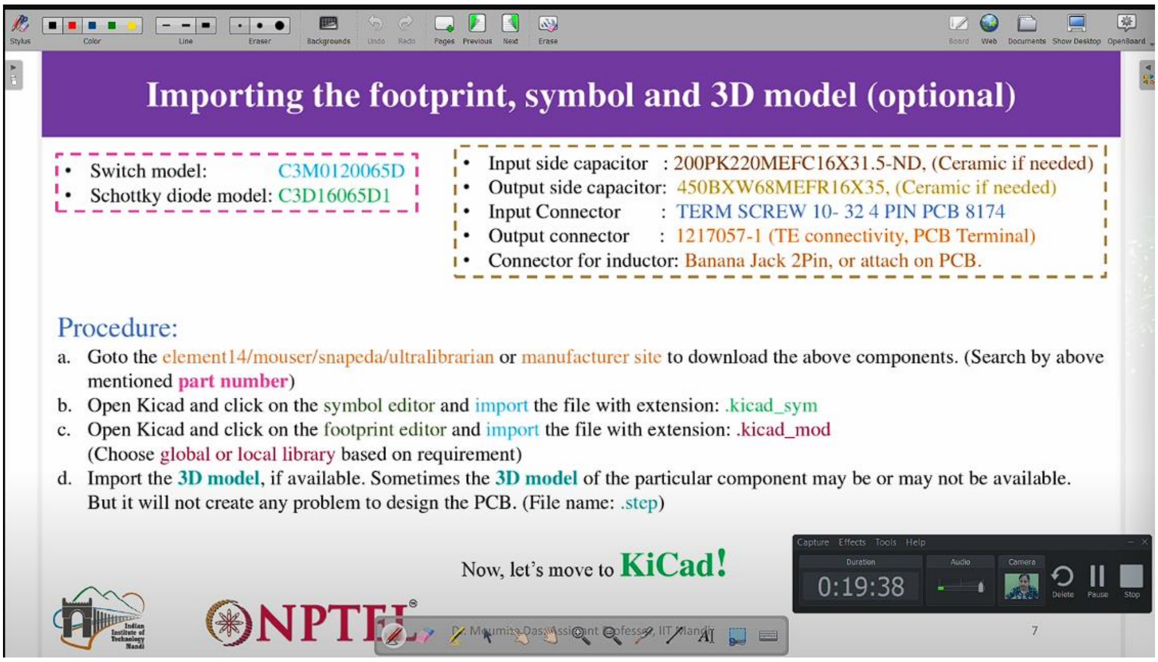
you can also see the 3D part of this how it should be looking and how close or whether the components whatever you are fitting is it possible to fit in that particular layout or you need to provide more space so these are the different connector you can see so this is the different connector and the points are given so this is like this is potentiometer different resistances this is different ceramic capacitances so all these things are connected here okay So, this will be used to generate the Gerber file and it will be sent to the PCB manufacturer.

Refer Slide 18:24



Now, let's see how we can design same boost converter using this Kicad software. Now, you know the procedure. Let's see how to design that. So, you need to select the new project. Let's select the new project. Then you have to What you have to do? You have to provide name of this particular project. If you have new folder, you can select new folder so that you can keep everything in that particular folder.

Refer Slide 19:24

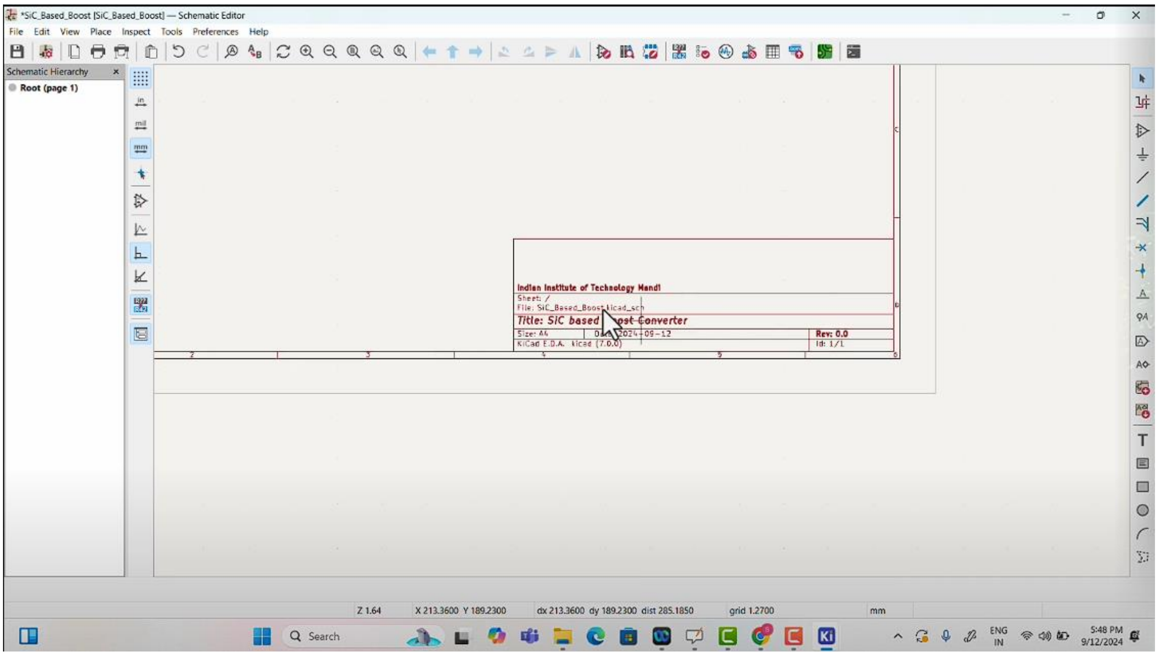


Let's say we are considering the silicon carbide based boost converter for designing PCB. So, we have given that particular name silicon carbide based boost converter. Now, two different files are saved. Okay, so this kicad file or this file, so the schematic file you need to select and then schematic here, right now nothing is there. Now let's go to this particular window and see here basically you have to select this different model.

Switch model you have to select this one. Diode model you have to select this one as I told you in the previous slide. Capacitor models, input connector, output connector and connector for the inductor. So these are the seven different component you need to select for designing the boost converter. Now, in case if any component is not available, then you need to import that particular component.

Means, let us say we are considering here silicon carbide device. If we do not have that particular part number in the PCB software, in this kicad software, then we can easily download PCB PCB actually legs or so basically the file will be there particular for particular component you can just download and import it here in that particular software to use it for the designing of the PCB okay so this is the procedure procedure is that you have to go so here we are considering so like element 14 mouser so basically library or the manufacturer side whatever manufacturer will be there to download the above components if it is not there in the software so then download it search by above mentioned part number so you can just provide the part number and then you can download open kicad and click on the symbol editor that I'll show you now and import the file with extension dot kicad underscore symbol right open kicad and click on the footprint editor and import the file without extension dot kicad and underscore mod right choose global or local library based on the requirement import the 3d model if available if 3d model is available that is okay sometimes the 3d model of the particular component may be or may not be available so that time it is not a problem means like it if it is available so then you can see the final pcb how it will look like if it is not available then also it is fine because anyway like this footprint you have so you can just directly place it on the pcb and then you can connect the component later you will not be able to see virtually how it will look like but you will have the pcb but it will not create any problem to design the pcb file name dot step now let's move to the kicad so you can see here this PCB designing so you can see so first is the so you can you have to actually actually select different like date and everything you can just put it so that it will be easier for you to refer this particular schematic later so this will be printed on the pcb if you want it to be printed

Refer Slide 23:05



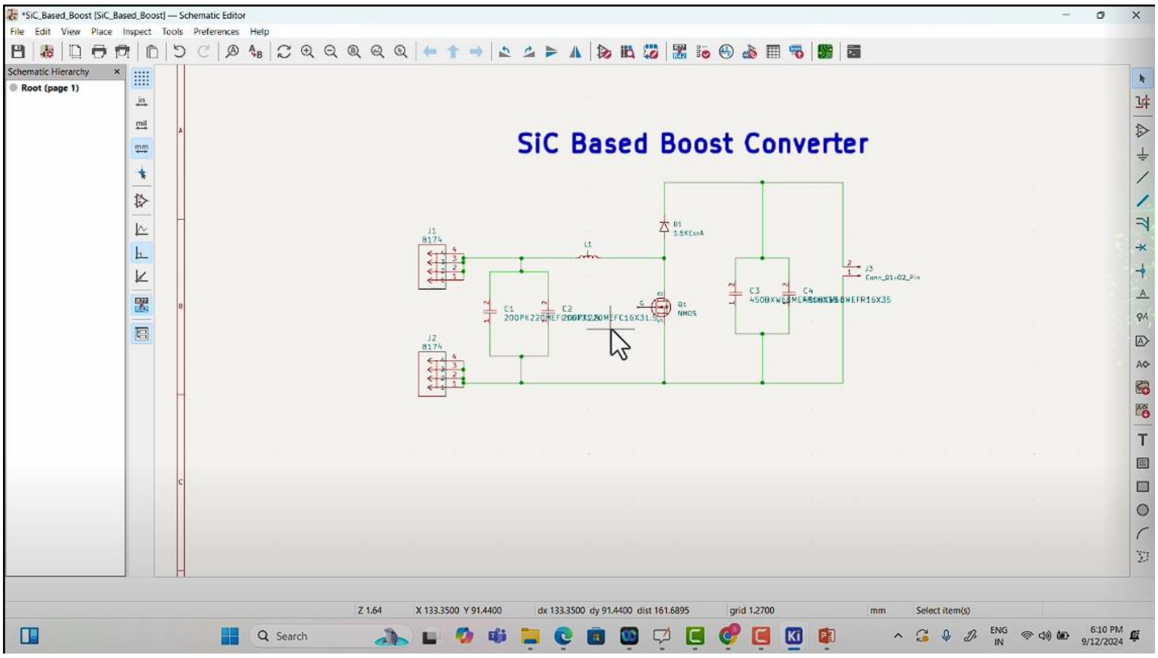
so then i mean like if there are like many pcb sometimes what happens it is very difficult to understand which pcb is this so then this will ultimately help so you can see here so you can provide all the details the date and everything size and particular time and where exactly it is developed so all this thing information you can provide So now you can see this different part number. Okay, so now we will search any particular component in the website and then download that installed in the PCB. Okay, let's see this particular component which is already given here. You see this particular component capacitor is already given.

So now this component will just open in the website. so this is the component we needed so this you can go to the website DGKey now this window will open ok now this model you can just click so this model so now this model will be there and there different symbol footprint 3d model everything is given for this particular capacitor so now you need to download download format also you need to select so this step you need to select first step and then kicad file you have to select like version 5 6 whatever it is it is 6 plus we are in 7 so this particular version we are selecting so now download this so it will be downloaded if you have any other component similar way you can just search you can select particular version for kicad now this this is downloaded so it is in zip file so we need to open this and the kicad file footprint will be there so this footprint then what we need to do we can actually put it in the same folder right under components so that it will be easier for us to import so you just give the name of that particular component so that it will be easier for you to refer now symbol editor you need to select So here you need to add new file So you need to select from the folder where this component is available So then once you select this, this will appear like this This is the symbol, so that is why symbol is appearing like this Particular capacitor symbol, in the red color you can see here So once you save this symbol, it will be saved in the library similarly you can also install other components so this is the output capacitor so you can see here this output capacitor is actually through hole component so you can see how it is looking like so like different footprint and everything similar way you can again like save the component ok now all the components are installed right so then we have to actually see this footprint we have to load all this footprint in the libraries so that you can use it right so this loading is going on now since it is uploaded it is basically downloaded now you need to import it in the keycad software if it is already imported then you don't have to worry about all these things since it is importing now later on you can use it use the same thing for other converter designing so the library new library you need to get so this is under named project then importing from the different like places where it is already kept so this is going on so see this is like same thing you have to do for all the components you have to select different components and then you have to like this is connected then you have to keep it in the library using the same procedure right so again you have to this is input connector so now you have to select output capacitor So this is the output capacitor footprint. Now this output capacitor is not, now input capacitor is important. Okay, so these components are now imported. Now we have to go to schematic editor to design the schematic diagram of the boost converter.

Now boost converter all the components are given. So now we have to design the boost converter. So, the components which were placed already I have showed you in the PPT. So, the same component you have to just select from the library and then you can connect it similar way as you have drawn circuit in the LTSpice software. So two input and two output capacitors were placed.

So exact like part number of the output and input capacitors are given. Exact part number only you need to select. Otherwise while like making the layout it will have different types of layout like

Refer Slide 38:04



Whatever you have chosen for output and input, that will not be the same. So, that's the exact part number you need to select. The same component, like for LT spice, you can use the same component.

And then you can just assign some values to make it work as that particular capacitor. But it is not the same thing here. You have to select the particular part number. So you can just select all the components from the library. If it is there in the library, you can just select it.

If it is not there, import and then select. So we have already this boost converter which is already designed. We are just showing you the procedure. how to select different components from the library to form the converter later we will use like the converter which has the defined components in the PPT so there that is what we are going to use so see this is the connection so you have to connect it this way otherwise you know the circuit connection will not be there you have to provide the connection. This will provide the actual connection. It is like actual wires are connected between two components.

So, these connections you have to provide in this schematic diagram. So, the way it is given in the boost converter, the similar way you have to provide all the connections. So, this is the connector which was downloaded and imported. So, this same connector is placed here. So, like two connectors are connected at the input, right.

This is like input capacitor positive and negative are connected in two different connectors. Similarly output also you need to connect connector. So output connector is also there. So, like all the points should be connected. This if anything is there, any extra wire is there, there is no connection.

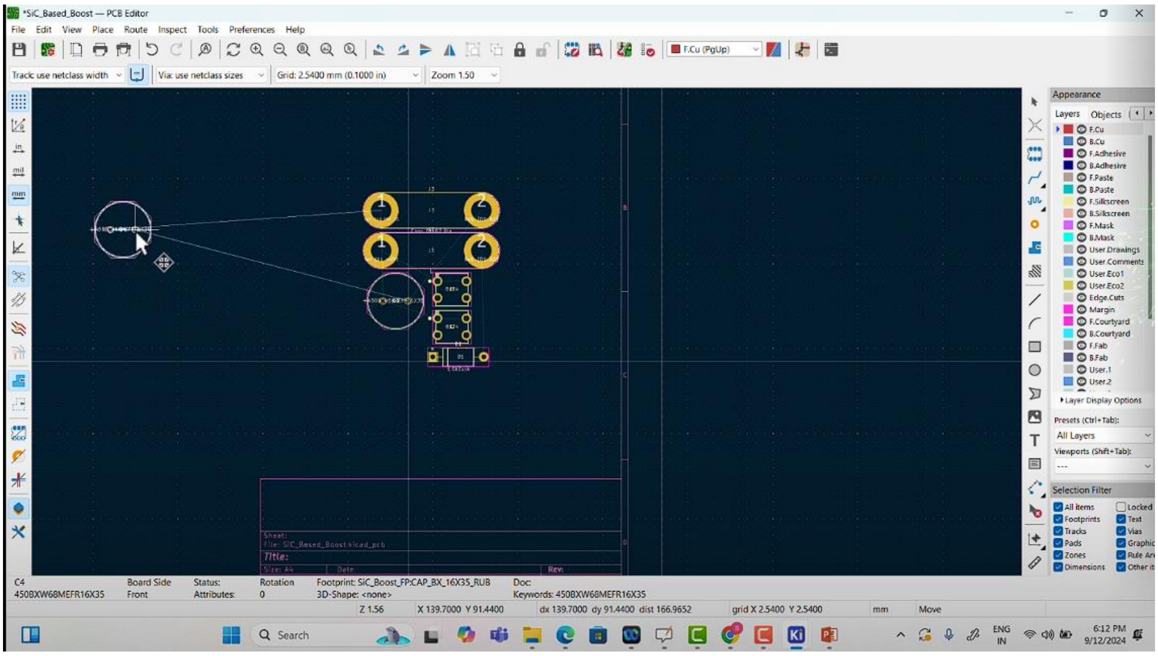
So, then it will not give you proper layout. You have to connect all the points properly. So, that while getting the layout from the schematic diagram, it will give you proper kind of connection, right. You can just provide the name of the particular converter here in the schematic board itself. Okay, now boost converter schematic design is complete.

Right. Now the second step, we have to go to the second step. So this is the gate part. So this is like we are keeping it empty at this moment. We are just designing the power converter part.

So gate part can be connected later. You can just change this differently. So this is like inductor also like how it should be. So inductor was selected. Now you can choose the actual footprint of the inductor.

How you want it to be. So you can select like whatever the inductor footprint you want it to be. So that you can just select directly from there. So this input output connector everything you if you have imported then it is fine. If you want to select it from the library after putting all the component you can just select if that component is applicable. is footprint see it is here it is blank right you can just select it from here now after that you have to update this

Refer Slide 40:37



So all the components which are connected so first you have to see whether the footprint is defined or not the same way which was shown.

Now once you are updating it in PCB so what will happen you will just be able to see the footprint and then the name will be given like whether it is an input footprint is with respect to input connector capacitor MOSFET diode or output connector so everything will be given the name will be given so accordingly you have to place it in the different places in the boost converter how it want it to be how the design should look like so accordingly you have to place right so see here all the components you have to just see that whether footprint is defined or not otherwise you have to define first so all the components you can just check like where it is like if you don't remember you can check the name and then you can just select the pcb board to put it in the proper place You can just see the footprint. If you don't remember then you can just see it how it looks like. If you want to select different footprint then again you can go back to that previous schematic and then you can update it. but okay so this is about today the whole complete thing now the layout designing will be shown in the next lecture.

So, this part you can see this is the schematic part is complete. Now, how to go from schematic to final diagonal that will be discussed in the next lecture. So, these are the all references. Thank you.