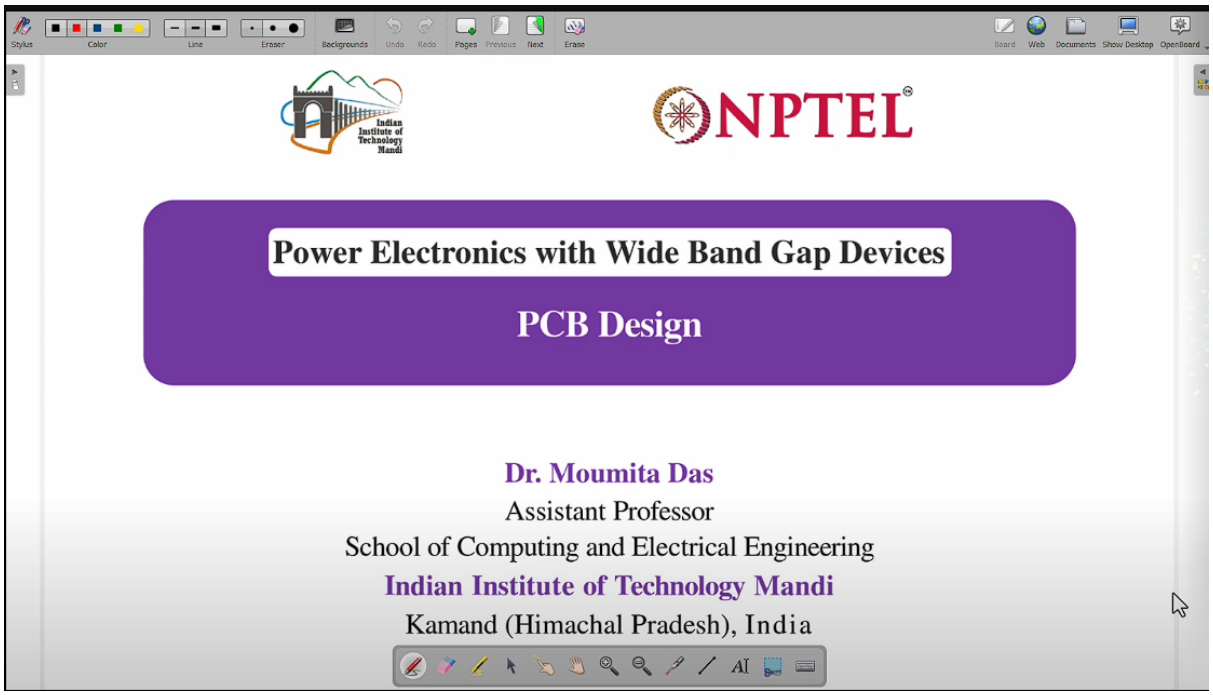
Power Electronics with Wide Bandgap Devices
Dr. Moumita das
School of Computing and Electrical Engineering
Indian Institute of Technology, Mandi

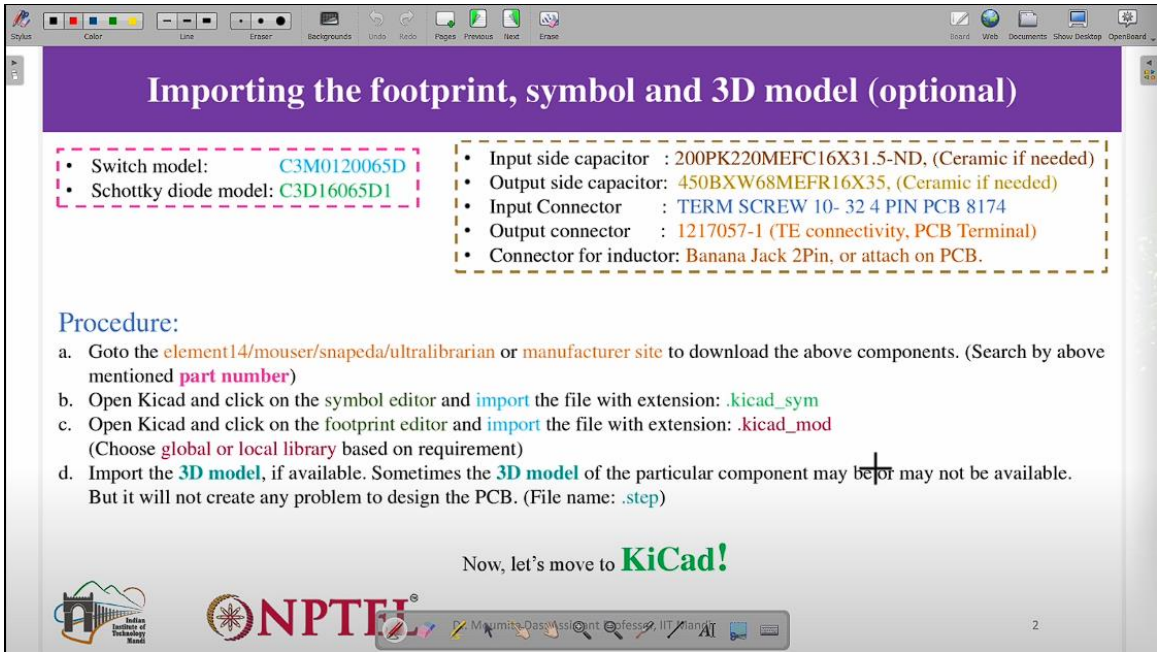
Lecture-33
PCB Design

PCB Design



Welcome to the course on power electronics with wide band gap devices. Today I am going to discuss about the PCB designing part. In the last lecture I have given introduction of the PCB design and how to install keycad software in order to design a PCB.

Reder slide (0:42)



So today we will see in details how to design a PCB by importing different component. Here you can see the switch model, diode model everything is selected based on the circuit which we have already simulated, so in the simulation of lt spice a boost converter was selected the same boost converter is used for designing the pcb in order to connect the simulation part with the pcb design part right? So the switch model remains same it is from full speed and then diode model it is also remains same and then there are like additional component which are required for the boost converter they are input side capacitors, output side capacitors, then input connector, output connector, connector for the inductor. So these are the components are additionally required.

So capacitor part anyway like it is some like part number are given. If it is available ceramic so then ceramic will be chosen. Because we need actual components, so that is why these specifications are mandatory. Then input connector, output connectors are also given.

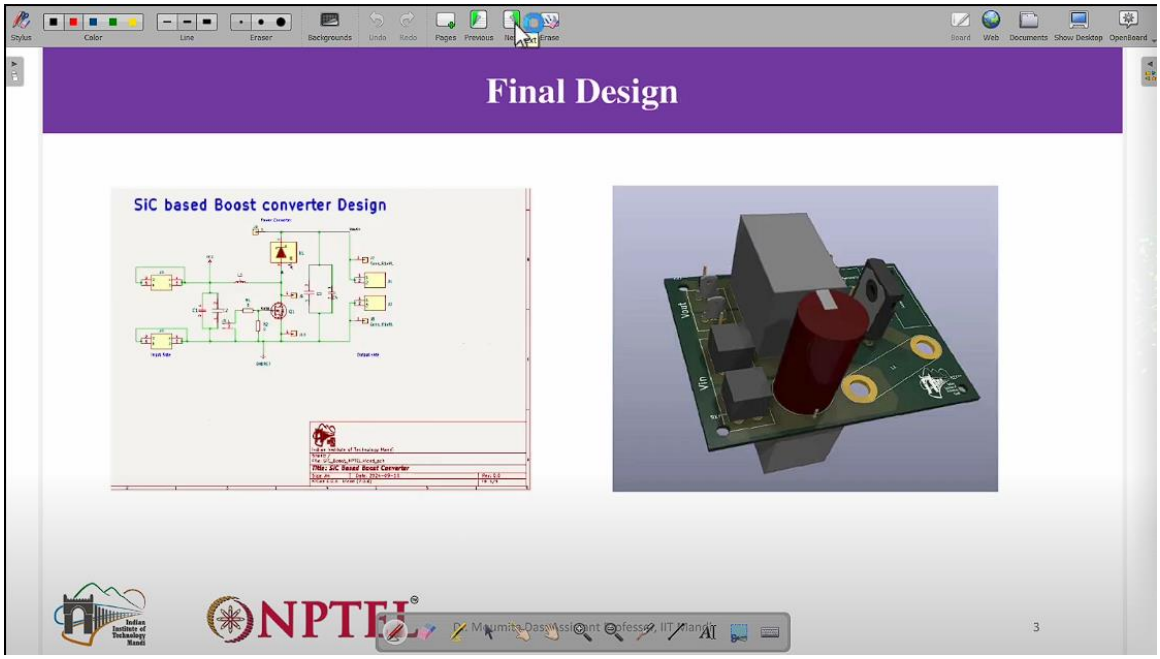
So, like what type of connector we need to use, and for inductor also there is a connector so in that place inductor will be connected since the inductor is connected externally we need the connector part in the PCB, so later that external inductor will be connected if it is like kind of a inductor which is mounted on PCB, so then different kind of design will be required, so in this discussion we are considering inductor which is connected externally right? So you can see here the procedure to import different component that is given here so we need to go to element 14 or mouser or anywhere like the component is available in order to download the components so that component can be connected switch diode if the connector is available in the library then we don't need to download it and the capacitor anything. So, then the procedure is that open the keycad, click on the symbol editor and import the file with extension keycad underscore symbol and then open the keycad and click on the footprint editor and import the file with extension keycad sorry keycad underscore mod. So, choose global or local library based on requirement right? Then after that if the 3D model is available then import the 3D model. So, this will give the idea of the system how it will be once the PCB design and the all

the components connection is done right. So, that is why this 3D model if it is available this will give you this complete idea of the about the system.

And also you can actually measure the power density from this PCB design part also. Because you know like once you have the 3D model, from there you can actually calculate complete volume of the system. So, length, height, width, everything. Because you know 3D model will give you the actual volume of the different component which are connected in the PCB itself, right? So, sometimes this 3D model of particular component may be or may not be available. If it is not available, it doesn't matter.

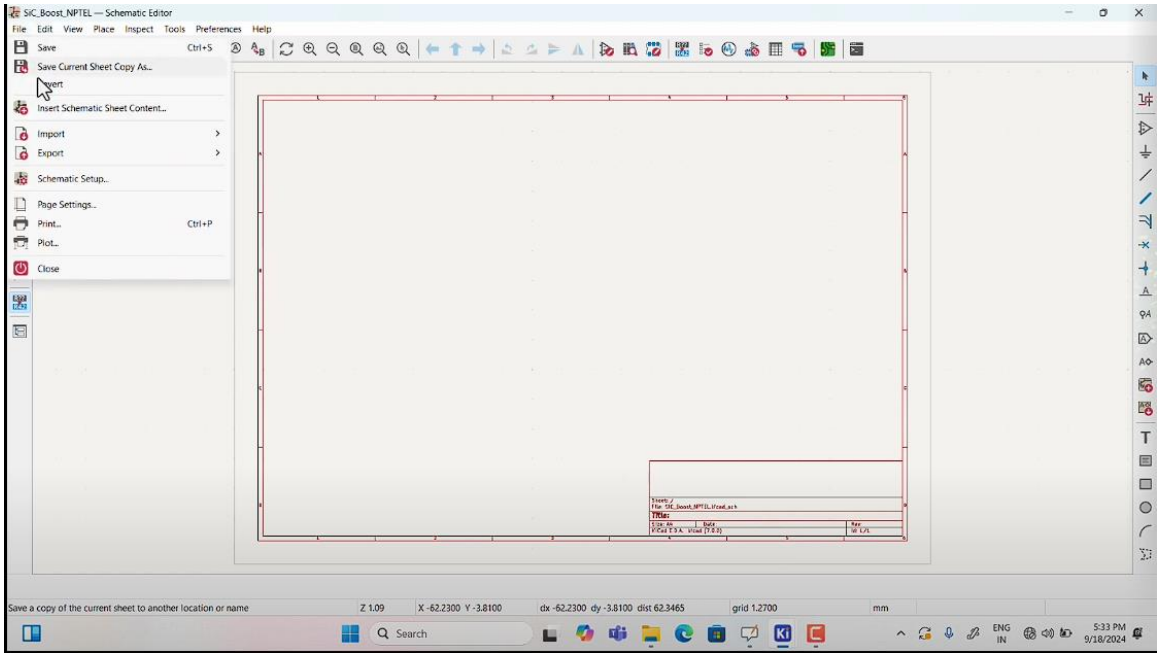
It is not a big problem. But if it is available, then it is good to import it. But it will not create any problem to design the PCB. okay?

Reder slide (4:23)



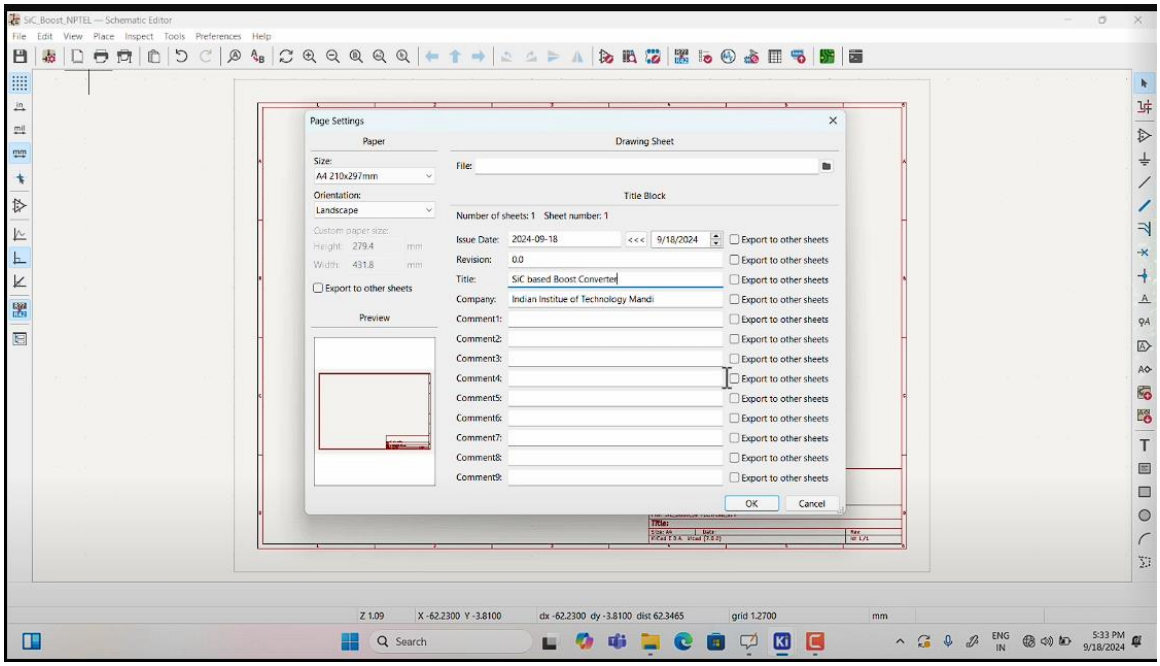
now this is like the final design so let's see the import part first then we will see the final design how it looks like so you open the schematic editor here.

Reder slide (4:37)



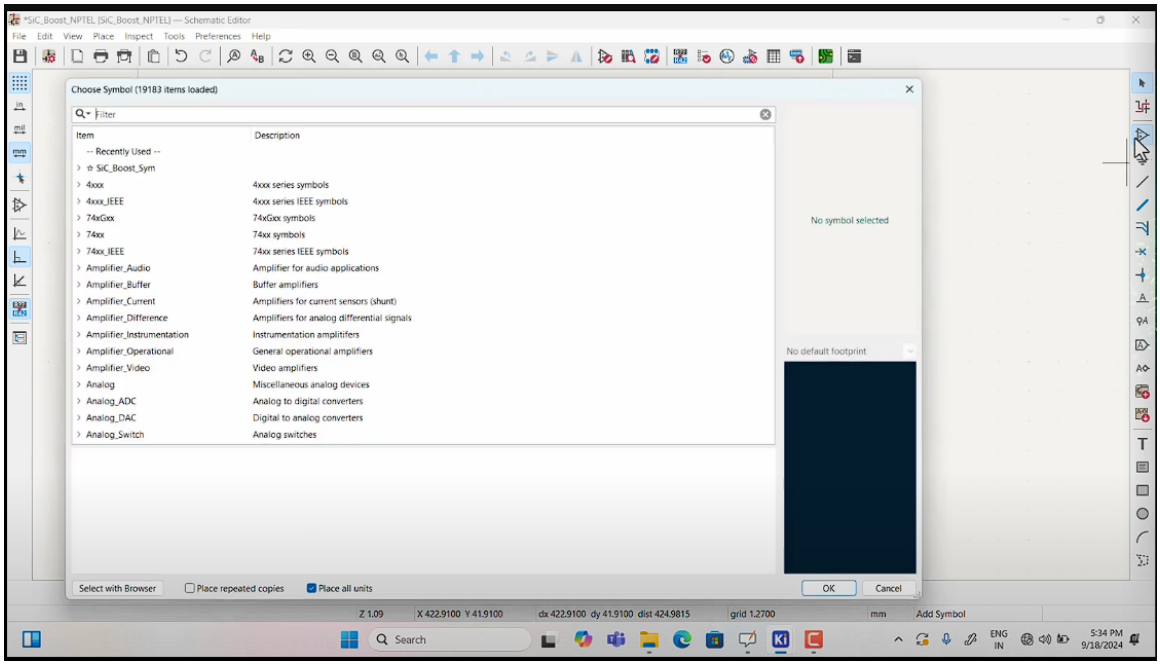
okay? Then here you can see the page setting so you can just write different input which you want to provide on your pcb. This will help you to identify the PCB.

Reder slide (5:08)



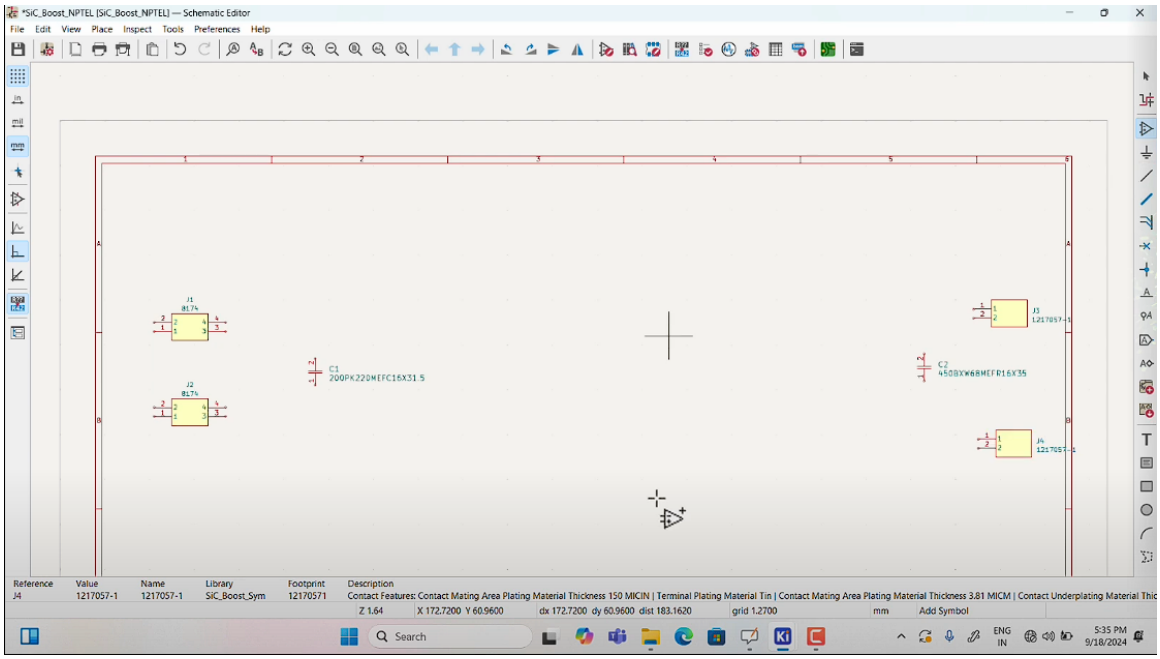
If you have like different PCBs then it will give you to identify what kind of PCB is this one. Right? Or later on if someone wants to use it they will have information about this PCB. So this is the thing is given here. So this is silicon carbide based boost converter then title is given and then file name is also given.

Reder slide (5:34)



This is for library. This will open the library part. Then here you can see silicon carbide symbol. So these are the different component which are required for this boost converter. Right? So these are already there in this.

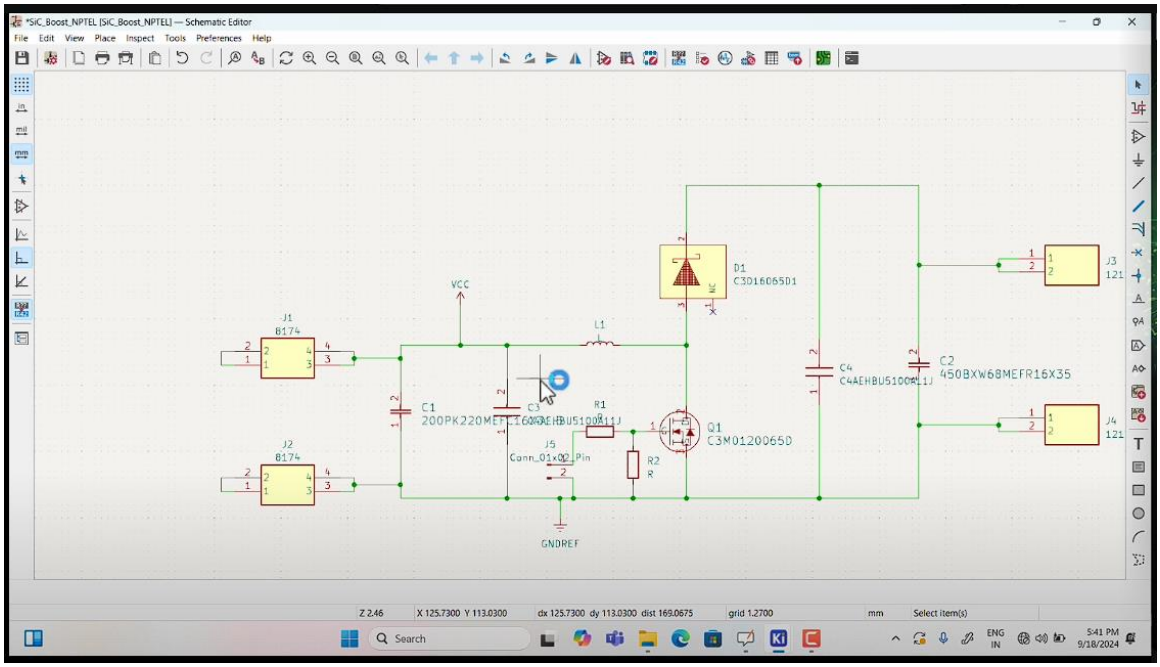
Reder slide (6:43)



Okay. So just pick this any of this. The capacitor one so this is the capacitor so you can just select the capacitor this is uh so there are actually two different capacitors one for input another for output so you can select from here this is for the output. So there are two different capacitors are connected for input as well as for output. One is for low frequency another is for high

frequency. So values of two different capacitors are different.

Reder slide (13:05)



So this is the connection which you can see directly in the like conventional boost converter so this input this j1 and j2 which this is shown here so these are for the input connector you know like if you want to connect the power supply at the input of the boost converter so you need to connect the positive and negative terminal from a power supply right so now this power supply can be placed like far from this boost converter PCB. So then you need to provide proper connector so that it can be connected through wires. So for that this J1 and J2 connectors are given here. Now sometimes it takes some time like some time to just search because you know there are like different components present in this library. So, this is the inductor part, so like which is chosen from the library.

So, this like later it will be like replaced with a connector. So, right now like for connection purpose this is given as the inductor, right. So, this is like input boost converter is connected after the capacitor part, input capacitor part. So input capacitors are connected because if there is some disturbances or noises from the input source, so like that can be filtered out. For that purpose these capacitors are connected.

If you have like any supply which is like next to just PCB or in the PCB like some supply is there, so then you don't have to connect these two different capacitors. But it is always good to connect this filter so that noise can be minimized. The input connection is connected across the input capacitor. Similar way this is the connection for output capacitor. C4 and C2 are the output capacitor.

So this positive terminal is connected to the diode. And then negative terminal is connected to the source of the switch. And these two terminals now will be connected to output connector.

So you can just keep it such a way that it will be easier for you to understand different part of this particular converter or the system which you want to design. So it will be better to start with a simple converter.

Then the understanding will be proper. After that you can go for any complicated kind of converter. So the whole system schematic is now complete. So this is the boost converter. One thing is that one more thing here you need to connect.

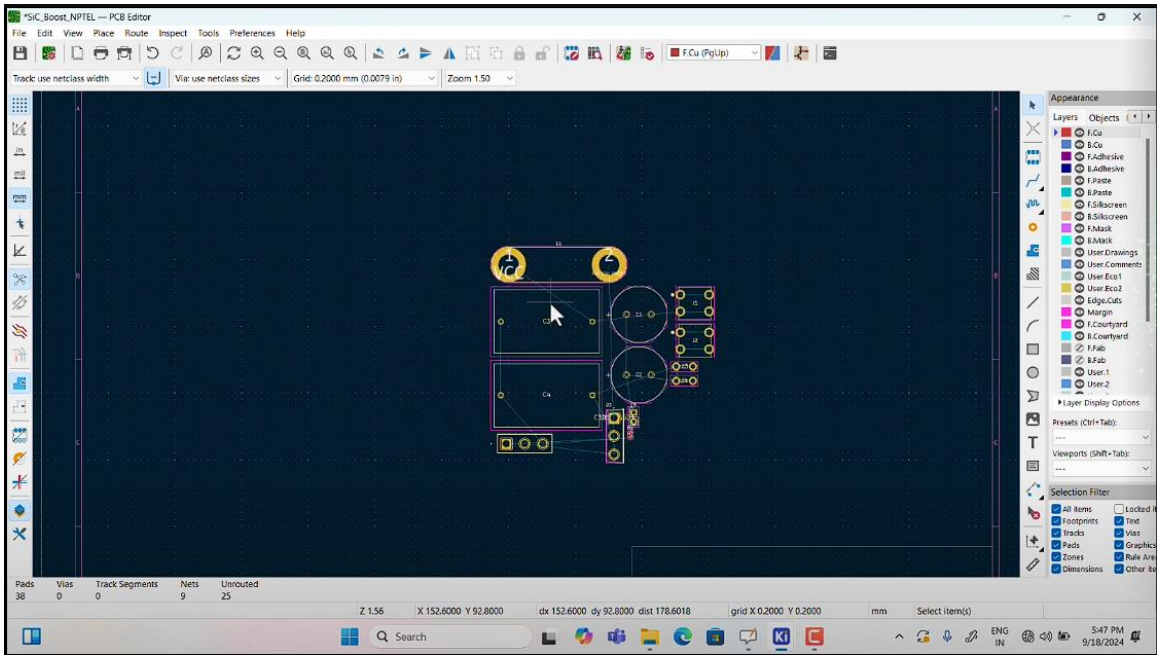
The ground reference so sometime the ground reference is the negative terminal sometime. It may not be the negative terminal, so in that time you have to connect it separately wherever the ground connection will be as i have discussed during the EMI discussion the ground actual ground is different so here you have to connect the positive and the ground terminal okay? If there is a connection where no connection is given then you have to put cross otherwise like it will be difficult to generate layout from this schematic, so If there is any open connection then it will give error. So if you don't want to connect any like connection which probably you will be connecting separately through wire or anything you can either connect connector or maybe you can just put this no connection as it is shown here in the cross, so this is with respect to the gate driver part which probably which i have already discussed earlier but like it is not like, it is also shown during the simulation. So in the boost converter diagram it is not shown. So this part you have to connect here.

Sometime if you have like complicated driver you can use different board for that. Now this L is selected. Now footprint you can just select for this particular L. So what kind of, because you know this L is something which will be connecting externally. So the connector you can just select.

So this is the kind of connector which is selected for this particular PCB. For j1 j2 this also footprint it is given so this is also like considered like different footprint if you want to select different one the capacitor also you can check the footprint how it is like 3D you can just check whether you want this kind of footprint or separate footprint you can just check and you can change it, Right by selecting the different component here like see the reference number values are given and the footprint here you can just press on this so then it will open like a different footprint, so there you can just select whatever footprint is suitable for this board or like whatever you want to connect particularly for this particular converter, okay? Similar thing is actually done for all the components. You can see here so this is like three different pins are given for these three in so this is for the like switch switch will have three pins gate drain source so that is visible there similar thing this. This is like also like selected like three pins for the diode. So one pin is no connection so that is already given here.

Only two pins you need to connect. so it will be always better to check all the footprint before going into the PCB layout in the schematic only you can check everything and you can check if anything is missed so then that footprint also you can assign you can assign that footprint by selecting like suitable footprint from the library Okay, now all the like assignment like assigning of footprint for all the components it is complete now.

Reder slide (18:02)



Okay, next step what we need to do we need to update PCB from schematic. Okay, that once you select that it will take some time for the updating like different footprint in the PCB board itself. So you can see here so like if it is having any warning or error then you need to go back and check in the schematic itself. If it is having zero error or warning then you can just go ahead to design that PCB port.

Reder slide (18:39)

So see these are the different footprint which was chosen from the library for different component. So now you need to Use different footprint and place it in different places to make the PCB board. So, these are the different point is given. So, which will reflect here directly.

It will help you to just sort out all this different component in the PCB. Once you have like more complicated kind of converter which has more components then this will be much more difficult to sort out from this different like layout like different uh pins so basically you will have more components then you have to actually sort out all the components and you have to connect it so it will take long time so that's why if you give more information in the schematic itself it will easy it will be much easier for you to connect those So, you can if you do not remember this J1, J2. So, these are the connectors. So, same you can just go back and check in the schematic port itself or you can just keep side by side to check whether this J1, J2 with

respect to different connector which are placed input or the output. Similarly, the capacitors also you can check.

Anytime if you want to see the 3D plots how it looks like so you can just check that 3D plots. So this once you bring close to any component right this is showing red means what you cannot place it on top of it. So if it is like placing on top of it then there will be like some connection problem right. So you have to check the connection if you are placing anything on top of it. So this connection so like in the front in the top or bottom layer it should not overlap each other.

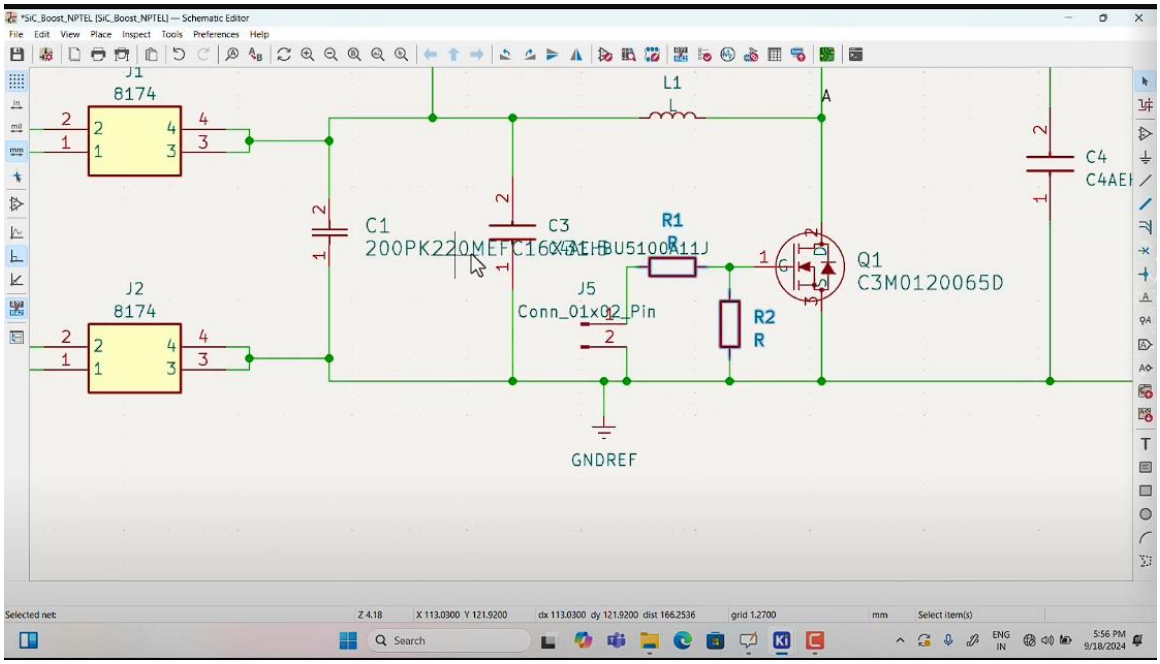
Component may have layout or the circle which may be bit like more area than that of the paints. So that is why you need to check these paints are not like overlapping with each other. That you can check after putting all the components. So this is where you can just check whether you want to keep it in the front or like top layer or select what component you want to bring front and what component you want to keep in the back also. You can choose the top and bottom layer also, if it is smd kind of device then that also you can do right now in this particular converter pcb we haven't selected any smd kind of component so only through whole component was selected for this so this is something is smd so see is so this um R1 and R2, they are basically SMD.

Other parameters are not SMD component. So like the main converter doesn't have SMD component. Only the driver part is having SMD. You can also choose SMDs for the devices, basically main component, different devices or the capacitors which are connected in the power board itself.

That you can also choose SMD devices. But at this moment we haven't selected. It will be easier to connect through hole component rather than SMD component. Resistance will not make much difference because you know like connection may not be that difficult. So this like the way it is connected like you can just keep straight line, horizontal, vertical or it may be connected in different angle.

That you can choose. I mean this is done so that the space can be optimized. Right? That is why this kind of like connections are kept. this is the entire board size you can check here so you can also measure using scale how much this is like basically length and height of this width height everything like whatever you want to choose basically x and y axis you can just see the how much area it is having right If you have like any application there probably you have given certain area within which you need to fit this PCB.

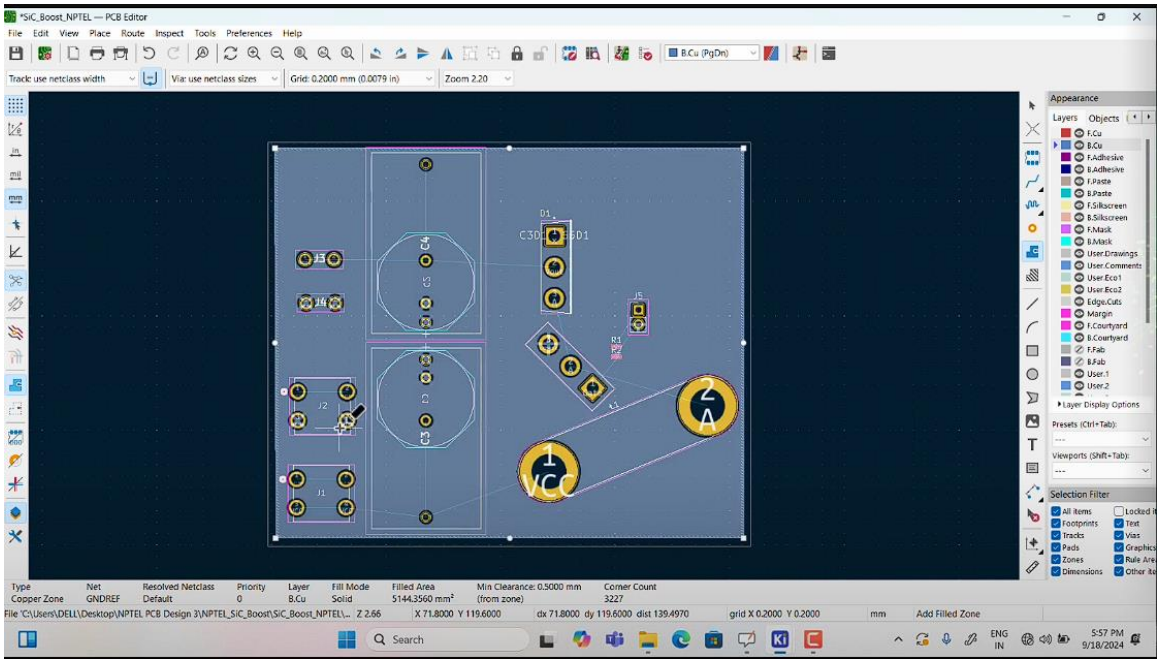
Reder slide (27:13)



Then it is very important to choose this scale or choose this area whatever is given and based on that you can just keep all the component. In case if you forgot to add anything or any point you want to just update, you can go back and schematic and update the PCB board.

So it will, without moving the different component placed in the PCB, it will just update that particular part, that particular node.

Reder slide (28:07)



So these are the ground references you can check here. So basically all the ground references are now visible. So this is where gate, ground reference, different points are connected. Those are actually visible here clearly. so this is B out so basically input so B out part is visible so

you can check here like different points which you have assigned here so basically where the B out will be connected where ground reference all these points are now visible so you can actually move this also separately this is with respect to Vout so here the output of the converter can be seen or it can be connected to the load what you want to connect so there this connection can be taken out from that Vout similar way you can just assign name for Vin So here input from the power supply will be given in these two connectors. So this is J2 and J1. This you can also check from the schematic. So input is given here. This is now connection of different point right. So you can see these are actually going to be like this wire connection between different component. You have to this now wire this wire is selected from here right. Now this connection just connecting. Now you can change this width and everything of this particular wire. so you can just draw this like wherever the like once you put the like one point once you select any point here let's say this one point in this register is selected so it will show you that where exactly this connection is Now let's say this R2 connection will be drawn.

So now select any point in this R2. Okay. Now select any point in this R2 point. So then it will be able to give you exactly where this R2 is connected. See this is actually showing where it is connected so this is if you want to connect it to different point where it is not connected this connection will not be done. So all this when it is actually selected this particular point it will show entire portion is for Vout. So, this entire thing and like all the points which are under this particular shaded portion.

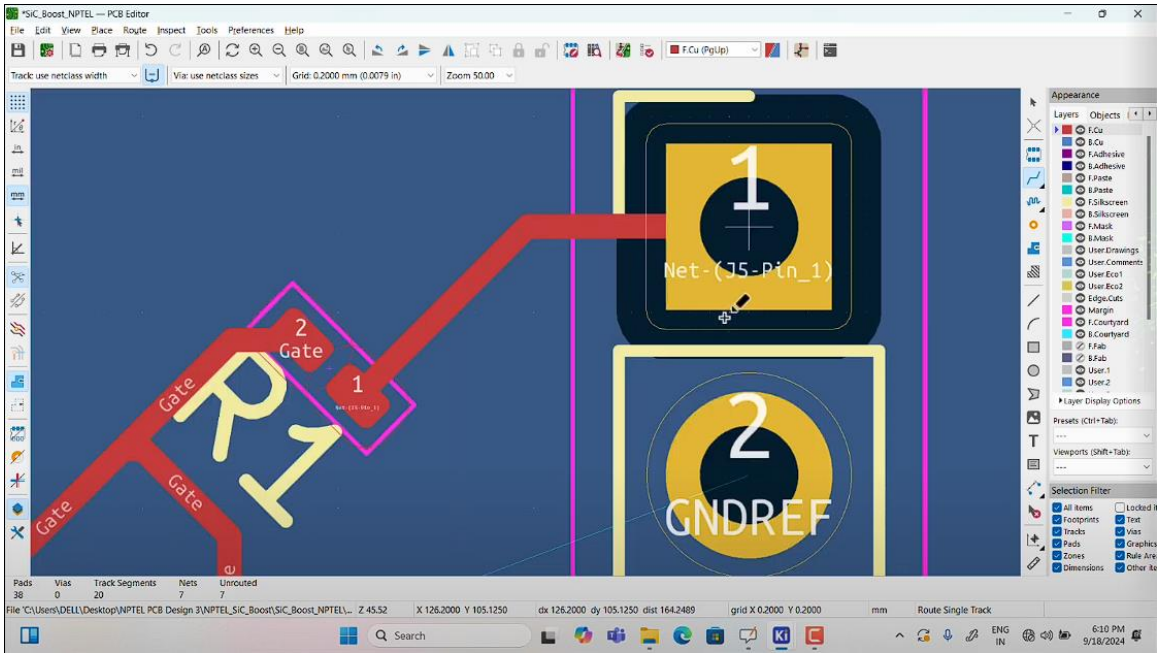
So, their Vout is connected. Right. Similar way the ground reference it is connected in this red particular portion. Here all the points related to ground it is connected. The entire shaded portion. You can just use simple wear or you can just show in the shaded portion.

Right. The entire red portion here it is V out. All the pins belong to this are connected together. So that is why you can like. Keep it properly so that you can actually give particular area for that particular Vout or the ground reference. It will reduce the noises in the PCB. So either you can just connect similar to the wire connection for different resistances or you can just put this shaded portion for power.

But it is always advisable to put this input and output like ground and the sources in the shaded way. It will always give you better result. So this ground difference it is outside. So we need to take it inside this red portion. So you have to check all the ground difference where it is.

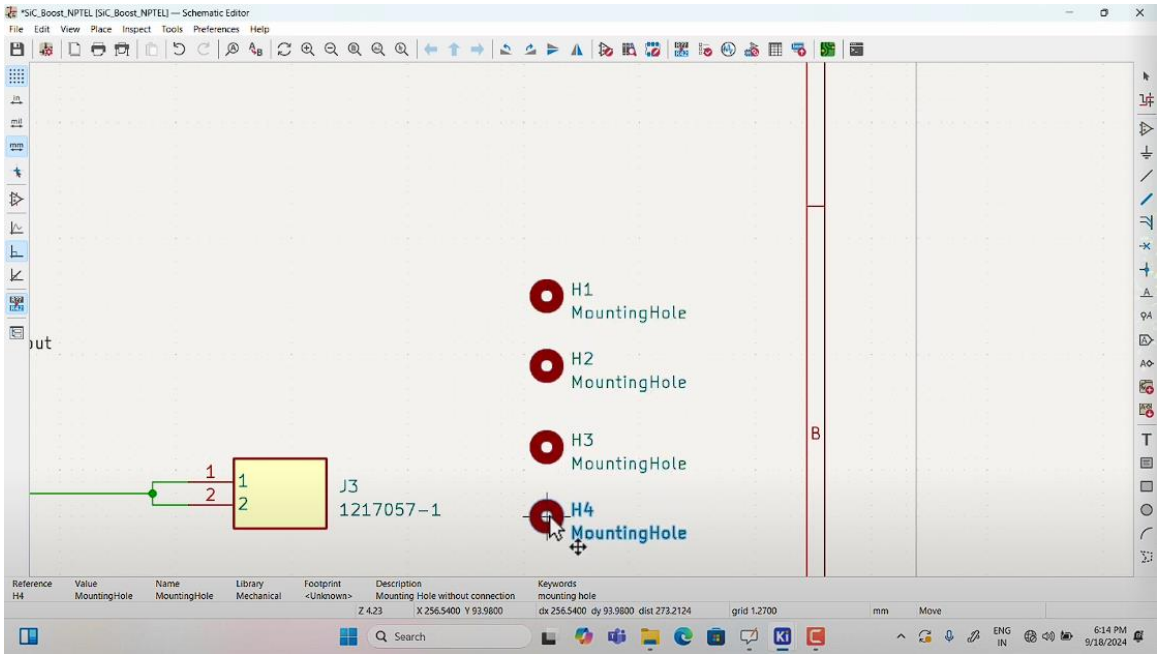
So for power you need to provide thick line because the current will be more and for like controller part or the gate part you need to you can provide thin line because the power level or the current is much less. See this blue lines you can see thin line it is actually showing the connection between like particular pins. You have to make sure that you are keeping it in such a place that it is not overlapping with any other pins, will also highlight you can see this highlighted part which are generally visible like by choosing particular pin connection it will not connect to any other pin so this one see this is visible with respect to one.

Reder slide (41:06)



So just by choosing this you can actually give this different area. You can just provide what you are giving it for. This is different pin connection you can again modify based on your requirement so there is like different footprint is given you can choose that or you can modify as per the requirement, you see now it is visible like this after modification so this is for the diode because you know like if it is very thin then there may be a chance of like malfunctioning after soldering due to like connection problem but if the this thickness is more the pad connection here here so then it will be much easier to connect the component and also the problem of malfunction can reduce down.

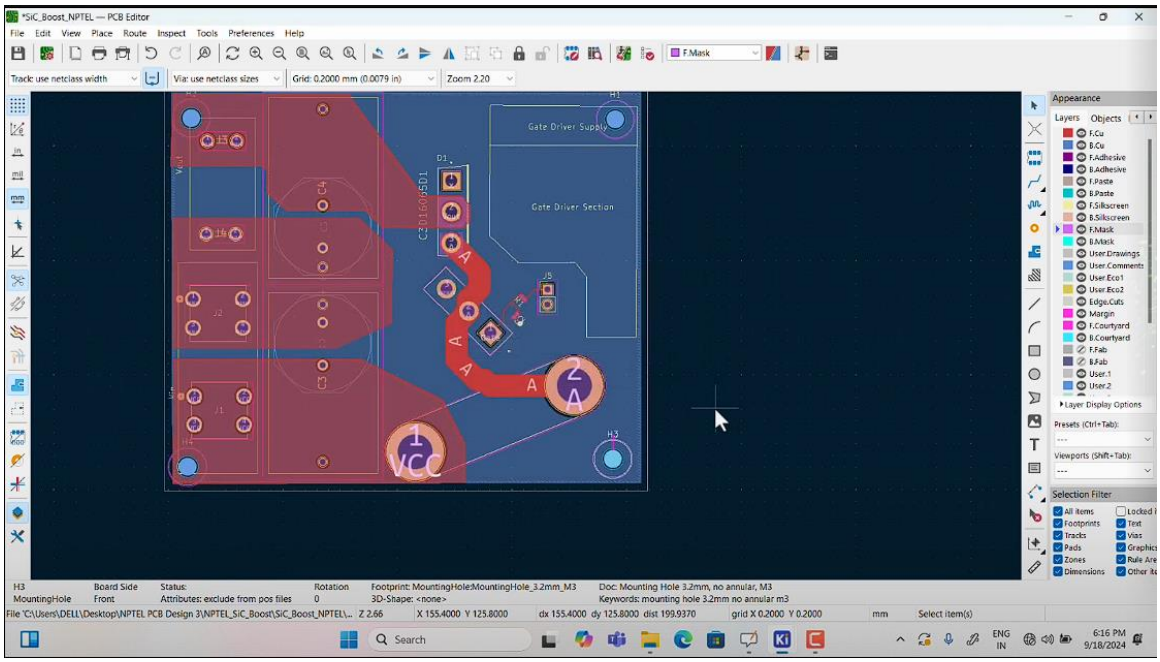
Reder slide (45:57)



Anything you want to connect separately mounting hole or anything that you can just connect,

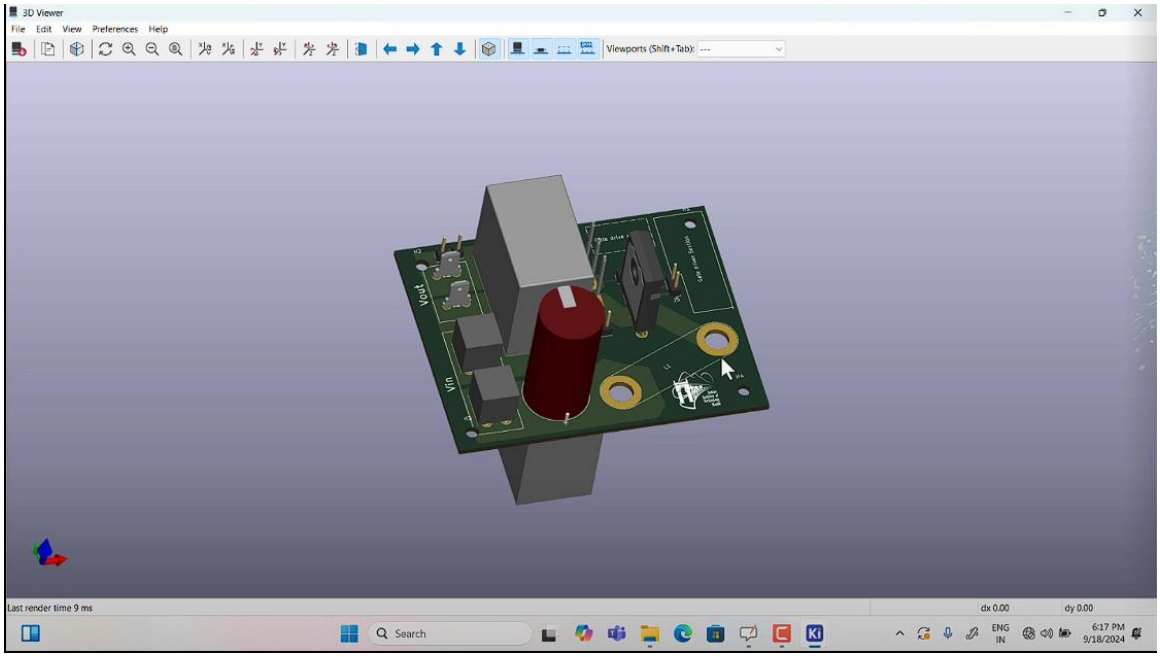
these are the six connected here, so this is kept for gate driver supply like this this different actually not for supply or anything so this is like for like pcb uh like board itself like if you want to connector like put some if you have to put some height so then you need to provide like screw for that so that, for that you can put this you may need it or may not need it but it will be always useful to have this kind of holder point in the pcb.

Reder slide (47:09)



Okay, now it is complete. Okay, so all the pins are connected properly. So everything is now done with respect to the schematic diagram. Now what you can do, you can just check this PCB.

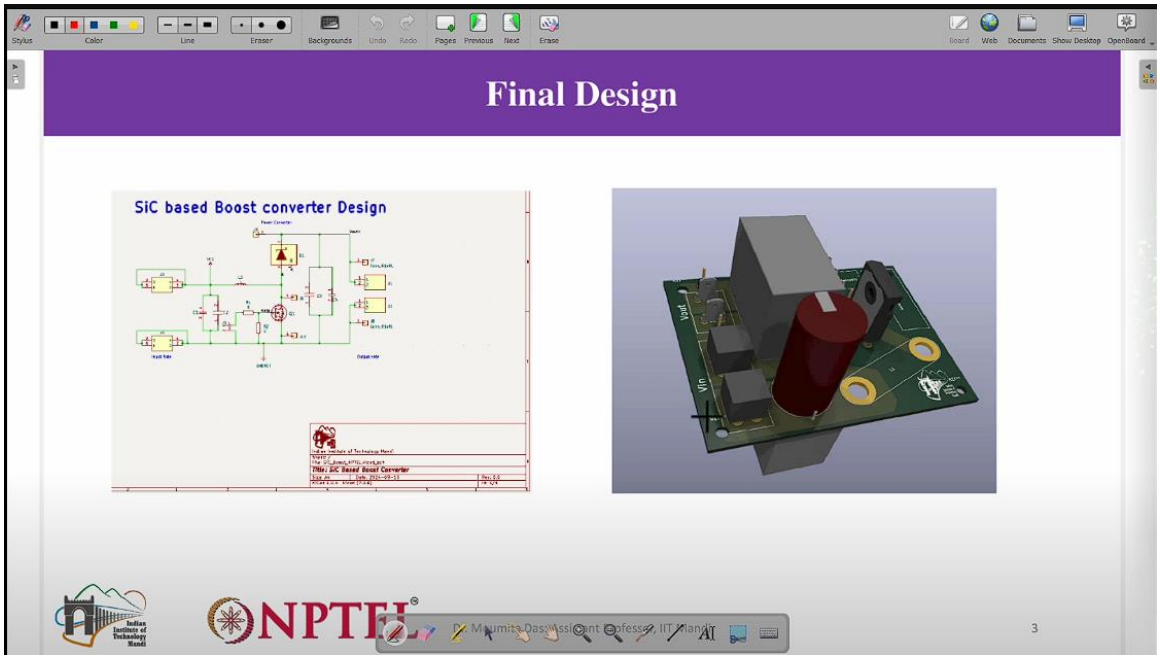
Reder slide (48:26)



So this is the 3D diagram of this PCB. You can check this how it will look like after placing different component. On the PCB board itself. So different component 3d Diagram is available. So that is why you can see this is like actual switch Diode so sometime it is it is on top like sometimes some of the components are at bottom.

So you, so this also you can just move and check like what are the components are placed at in the bottom so then this is how it will look like and from here like 3d point of view you can just measure this like what will be the volume of this particular converter, so you can just put scale of this particular thing or you can just rotate all the points and see that different components are after connecting how it should be looking like right? This different point you can just see and you can just measure this different point to check what will be the volume of this. Okay, so now this is the entire PCB.

Reder slide (50:08)



So now you can see here so this is the schematic and this is the final design which is like which you have like got after the designing layout of this. Now what you need to do you need to in order to like send it to the PCB manufacturer you need to provide the Gerber file.

So that how can you generate it? So that you can just see now. So once you have this, you can check if everything is fine, all are fine. Then you can go to the next step. This plot and then here in the plot format you can just get this gerber file so this gerber file you can just run a drc right so run drc so everything if it is like proper so if it is warning also then it is fine no problem some errors are showing here So, you need to check this footprint like where these errors are shown. So, see this is like overlapping.

So, that is why it is showing the error. So, if it is proper then it is fine. Or if it is not proper then you need to change this size little bit. Like this points you can just move little bit like towards outside. So, that it should not overlap with actual power line.

So, then you can just plot again run drc. Now check whether this errors are there, so this error is zero the warning is not a problem warning may have there like if there is a error then you cannot provide that gerber file sorry gerber file so now this you can just close this.

Reder slide (52:45)

It is done. So here you can just check this Gerber viewer. All these things you need to select. Then it will be like this. This is what the PCB manufacturer can see. And this is what you need to send to the PCB manufacturer. Right. Okay this is the final pcb you will be getting it once pcb manufacturer send you back. So like they will do drilling and everything so based on the your design and you will get the pcb okay so this is the final design you need to then you can refer all these references uh in order to like design the pcb. That's it for today. Thank you