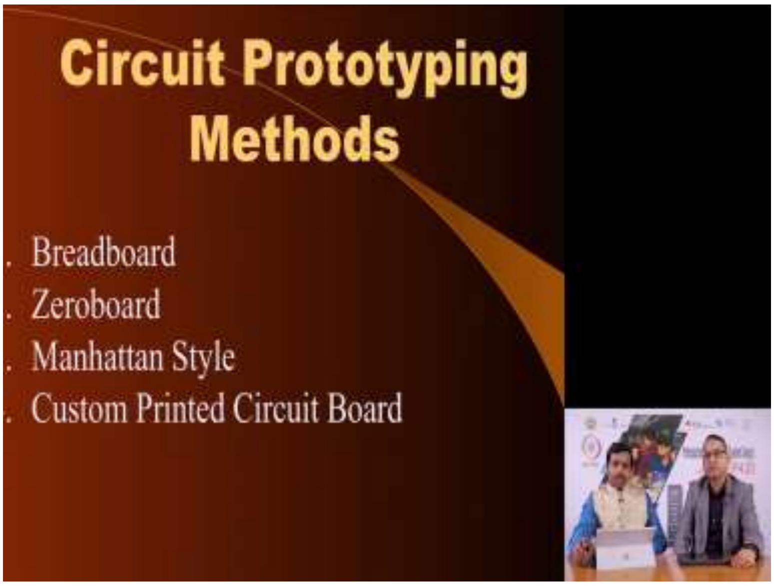
Introduction to Embedded System Design
Professor Dhananjay V. Gadre
Electronics and Communication Engineering
Netaji Subhas University of Technology
Badri Subudhi
Electrical Engineering Department
Indian Institute of Technology, Jammu
Lecture 37
Circuit Prototyping Techniques

Professor Dhananjay Gadre: Hello and welcome to a new session for this online course on Introduction to Embedded System Design. As usual, I am your instructor Dhananjay Gadre and today I am joined here by Dr. Badri Subudhi**.**

Professor Dr. Badri Subudhi: Hello everyone, myself Dr. Badri Subudhi, I am a faculty at electrical engineering department, IIT Jammu.

Professor Dhananjay Gadre: Today we are going to look at a new topic which is about circuit prototyping techniques. In an earlier lecture we covered how to design a complete system including wiring and other issues, but today we are going to look at specific part of that whole project fabrication and the specific part being how to prototype an electronic circuit, what are the various methods, which one is better than others and so on.

(Refer Slide Time 01:18)



Professor Dr. Badri Subudhi: So there are 4 type of circuit prototyping methods are there, first one is breadboard, second one is zero board, third one is Manhattan style and fourth one is custom printed circuit boards.

Professor Dhananjay Gadre: Now, you see these 4 methods have their own advantages and disadvantages. The breadboard style method is something which is often called quick and dirty method because it allows you to quickly put together a circuit that you have in mind and test it. Ofcourse the issue is that the breadboard is not very conducive for testing high speed or high frequency circuits.

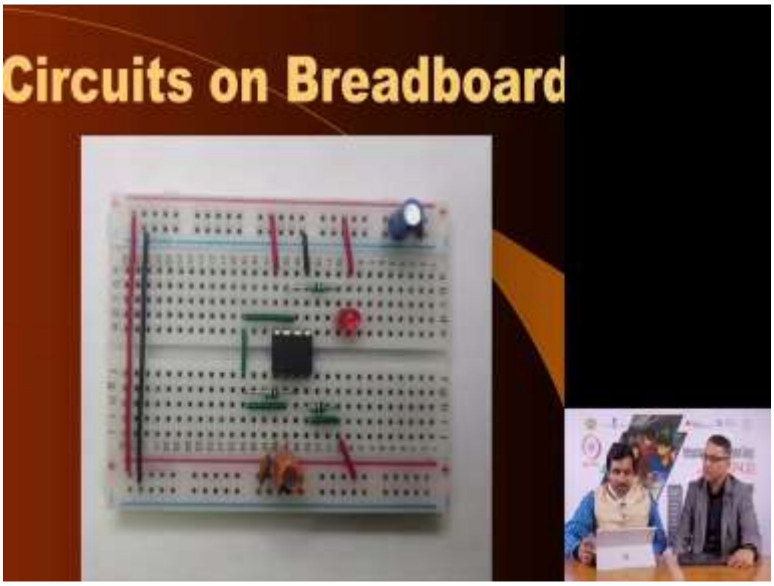
The second method is closer to the actual final professional implementation and that is called using a zero board. Zero board is nothing but a general purpose printed circuit board which has holes at regular intervals where you can insert your components and solder them. Ofcourse, this is not as close as the as having a custom printed circuit board, because you had to make do with whatever holes that are available and you have to fit your components.

It also has some disadvantages compared to the custom printed circuit board method. And that means soldering surface mount devices can be a little bit of challenge, although we will discuss how that can also be resolved, how that can also be accommodated in your zero board fabrication technique.

Another very interesting style of circuit prototyping is what is called as Manhattan style. The reason why the word Manhattan comes is because in New York in Manhattan, you have high rises, very tall buildings. And once you solder your circuit board using this technique, you will realize that with all the components rising up, it looks like Manhattan skyline.

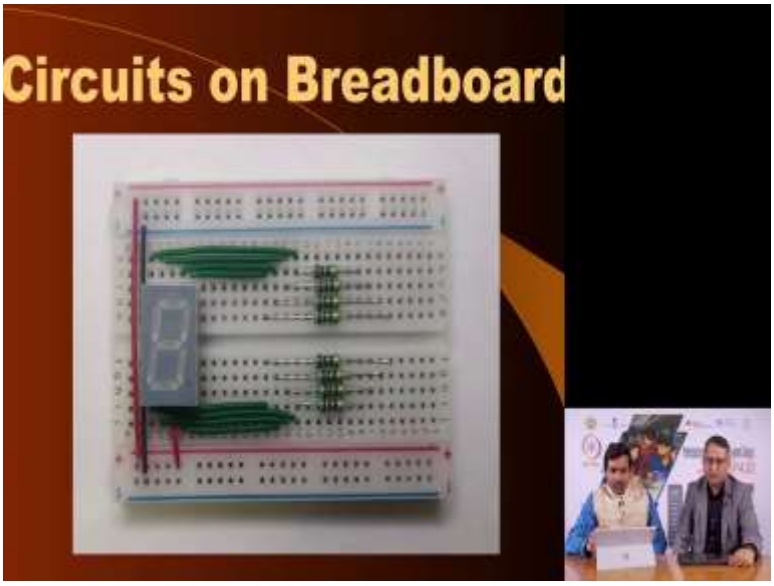
And the fourth method of course, is that you create your own custom printed circuit board, which is of course takes a lot of time to design, fabricate, solder and test. However, one must evaluate which technique is best for a given situation. Oftentimes, one starts with some breadboard testing, maybe if breadboard is not suitable for the job one may do a zero board testing and eventually if you want to do real professional job you would get into the area of creating a custom printed circuit board, so let us see all these 4 methods.

(Refer Slide Time 03:47)



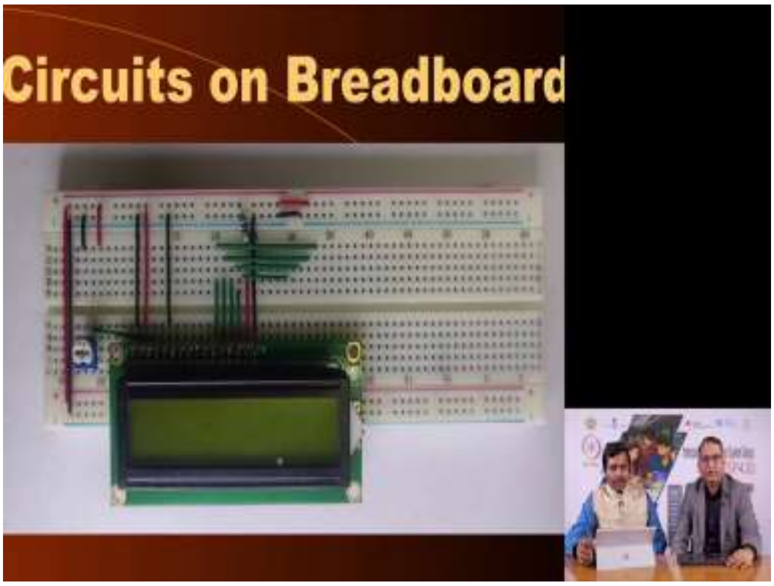
Professor Dr Badri Subudhi: So first method is circuit on breadboard. So you can see a breadboard where the horizontal line in the top are connected together. Similarly the vertical lines are connected together. So, this is a simple example of circuit when one IC is there, few single stranded wires are there, few registers are there, one LED is there and few capacitors are there.

(Refer Slide Time 04:14)



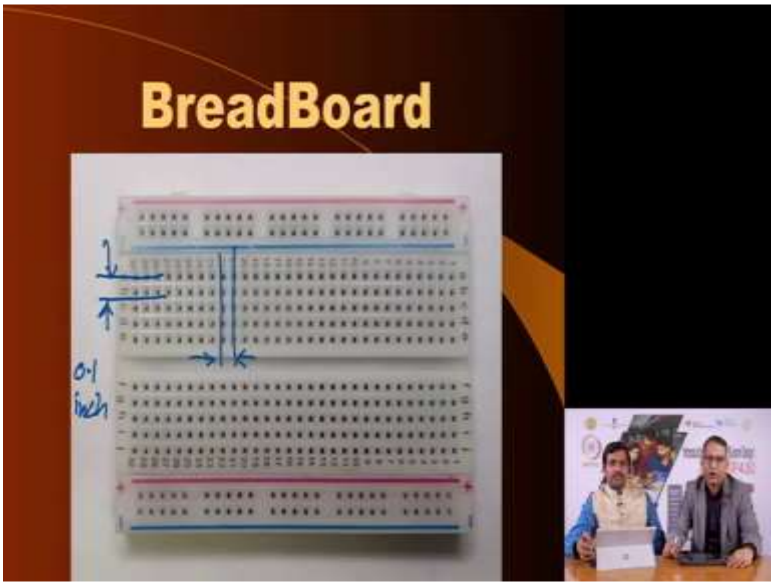
So, the second example you can see there is a 7-segment display is there, which is connected through some wires where the each register is connected with one wire and the LED 7-segment LED display is connected to these wires.

(Refer Slide Time 04:32)



Professor Dhananjay Gadre: So, the third example of a breadboard circuit is as you can see, it has a liquid crystal display LCD. And since the size of the LCD is slightly bigger than the previous two examples as you see here, therefore a larger breadboard is being used. And because it probably uses more connections, you see more number of single strand wires to make connections.

(Refer Slide Time 05:00)

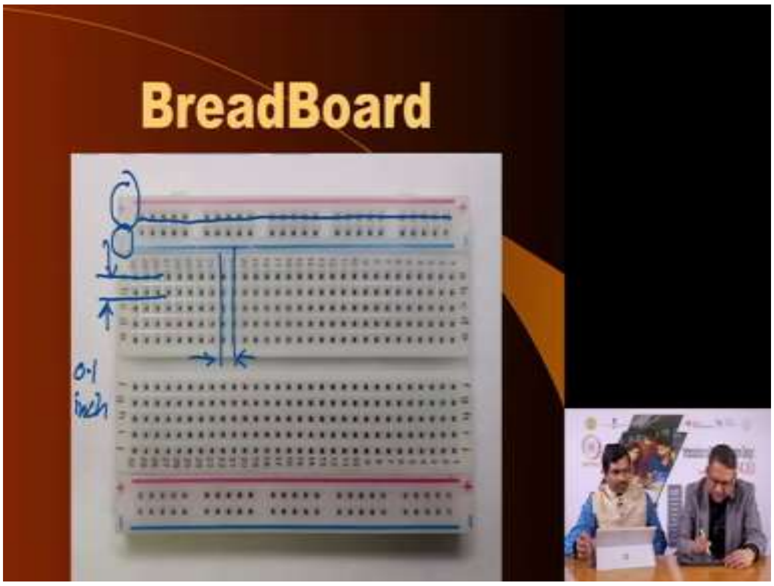


Here is an example of a breadboard, breadboard without any components. And as you see, it has a lot of holes, these holes have a certain pitch, pitch means the distance between two nearest holes. The distance between any two nearest holes here if I say this line and this line, this distance is 0.1 inch.

Similarly, if you see the holes in these lines, this is also 0.1 inch. The reason for this peculiar choice of pitch is that in the olden times all the ICs were made such that the distance between the two neighbouring pins of an IC was 0.1 inch and the distance between two rows of ICs was in multiples of 0.1 inch, usually it was 0.3 inches or 0.6 inches.

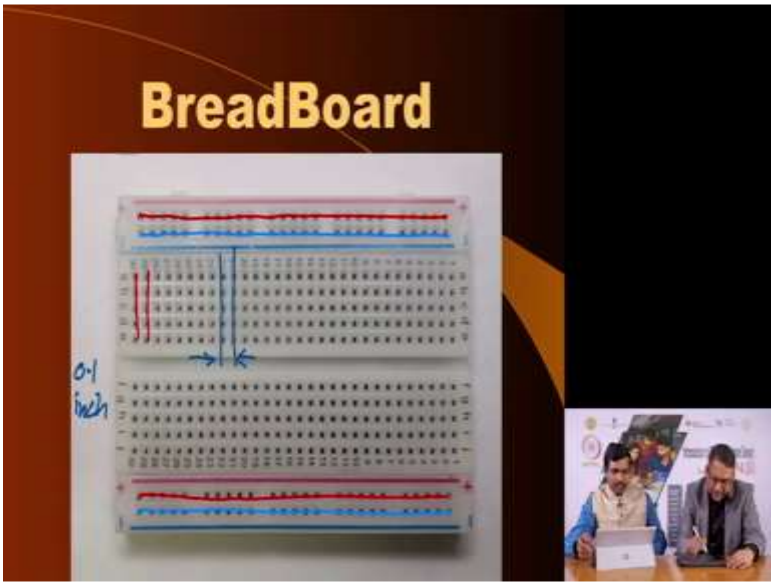
Therefore, a breadboard was designed in a way so that ICs could also be inserted, ofcourse, discrete components can be inserted at whatever pitch, the holes might be arranged that but to accommodate the integrated circuits, the breadboards were created so that the pitch of the holes is 0.1 inch.

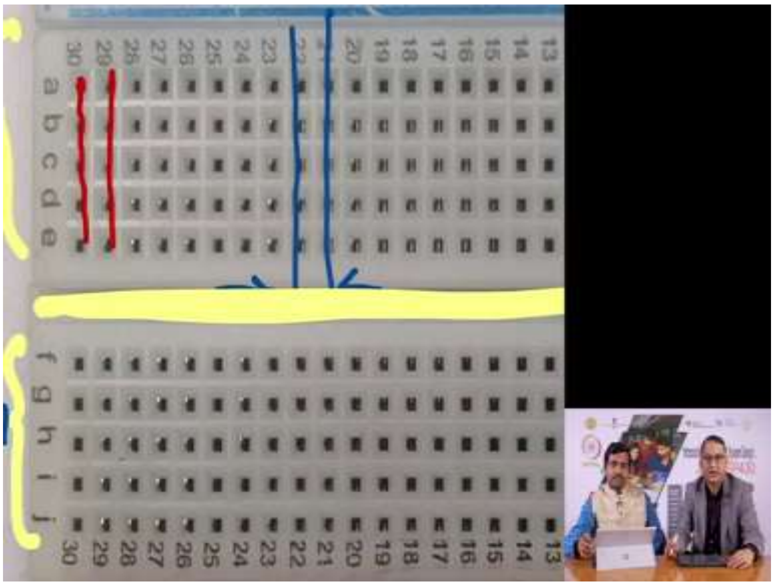
(Refer Slide Time: 06:19)

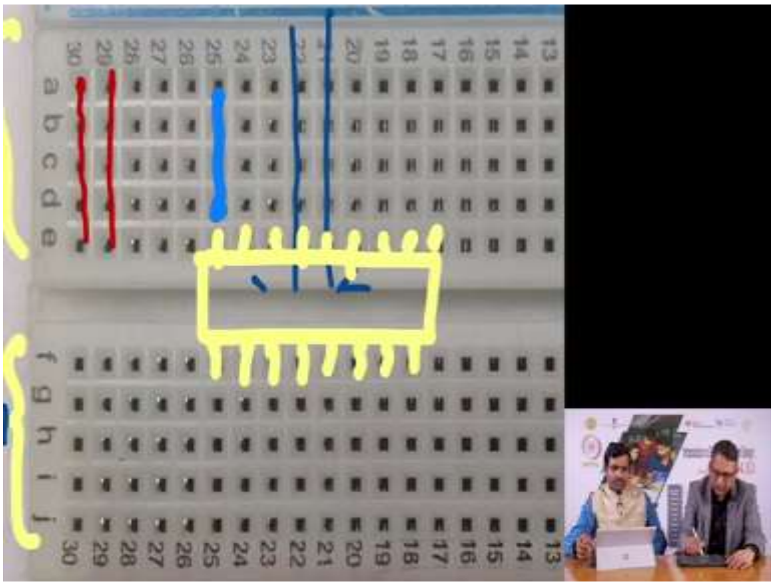


As my colleague mentioned, here, you see this is labelled plus and minus, there is nothing hard and fast about this plus and minus, it is a suggestion that you could use the top line here, which I am marking here, maybe I will use a different colour so that it is more obvious. So, let me use a red wire.

(Refer Slide Time: 06:39)







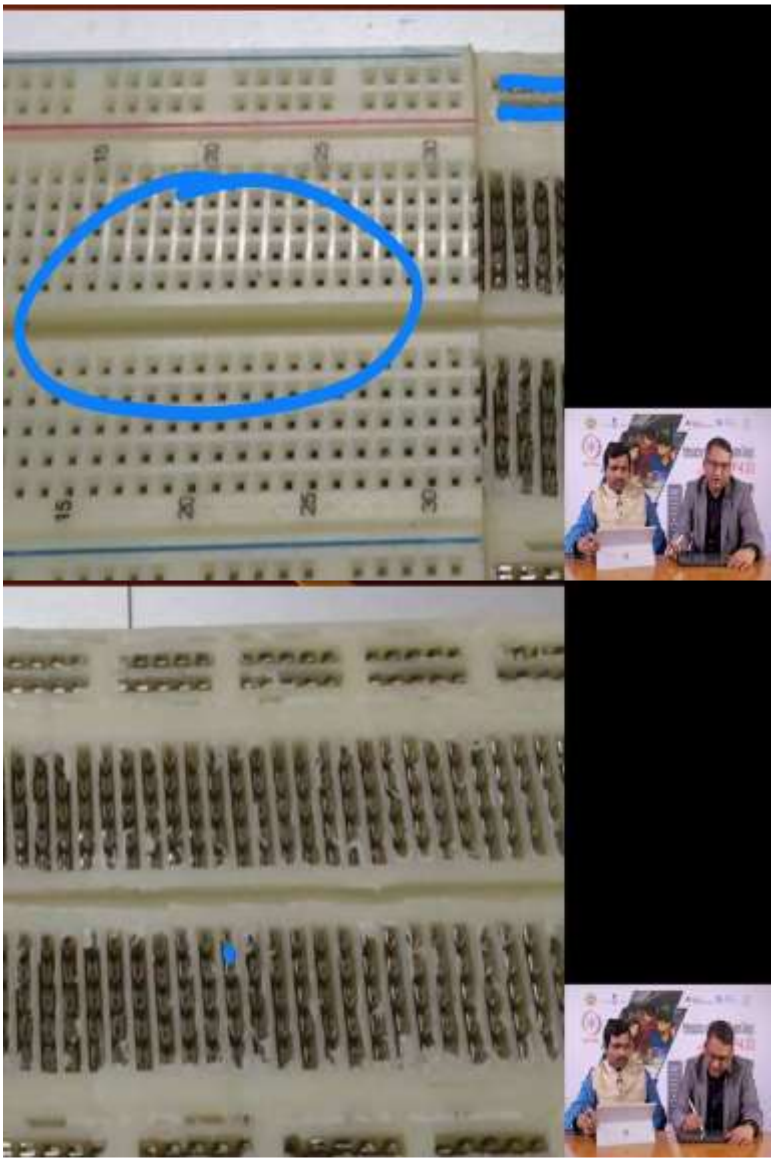
So you see this line is all connected together. Similarly, this line is connected together. And it is suggested that the red line should be plus and the blue line should be minus. It is again a suggestion you are free to utilize those interconnections in any which way you want. Similarly, the horizontal wires, horizontal holes on the bottom line are like that here you have these wires are all, these holes are all interconnected internally.

Similarly the row of line above that is also interconnected. Now here the connections are in the horizontal direction, whereas the rest of the holes they are connected in this vertical fashion which means all the holes in column 30 or 29 if I zoom this you may be able to see this, all the holes in these columns 30, 29 are all interconnected.

However, this separation big separation here if I draw it here, this is this disconnects the holes on this side and this side, they are disconnected. This allows you to put integrated circuits say somewhere say something like this. I could put a circuit here and it would occupy holes like this on both sides.

And therefore it will not short opposite pins on opposite sides of the two rows of an IC and it would allow the user for example, make connections from here and so on, similarly here and so on. So this is the general structure of a breadboard. This is a small size breadboard and you get larger size breadboards also. To discover what is underneath this top layer of plastic with holes. What we did was we took a breadboard and we milled one part of it.

(Refer Slide Time 08:45)

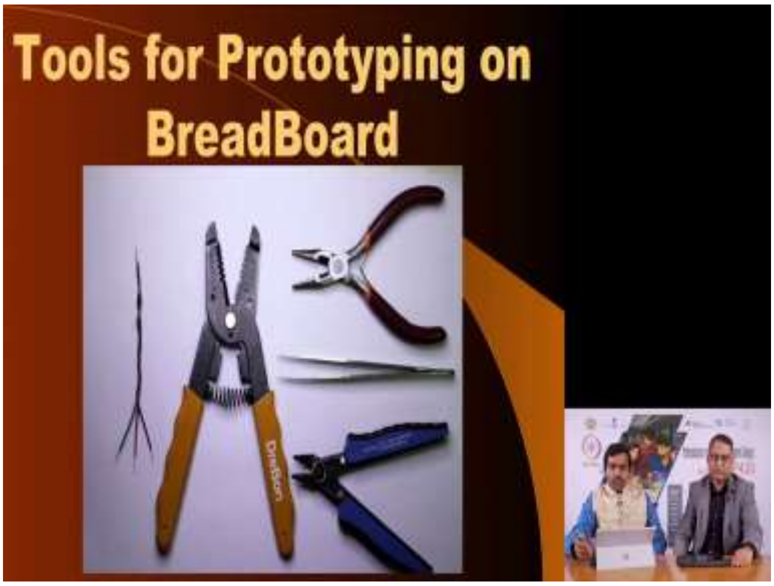


And as you see here, you see these are metal contacts. It is an entire column of contacts. Similarly, you see column of contacts here horizontally. So this is what lies underneath the holes in a plastic insulating form here underneath, this is nothing but rows of copper connections which make connections in the vertical as well as in the horizontal holes.

So this is the structure of a breadboard and this makes it very convenient to quickly prototype a circuit, you do not need to wait for any PCB inserting a component and making connection does not take much time compared to soldering and therefore this is a much preferred approach of prototyping circuits.

But because of the nature of these wires, there is a lot of mutual inductance and is not very suitable for high frequency circuits. What all is needed to prototype such a circuit?

(Refer Slide Time 09:44)



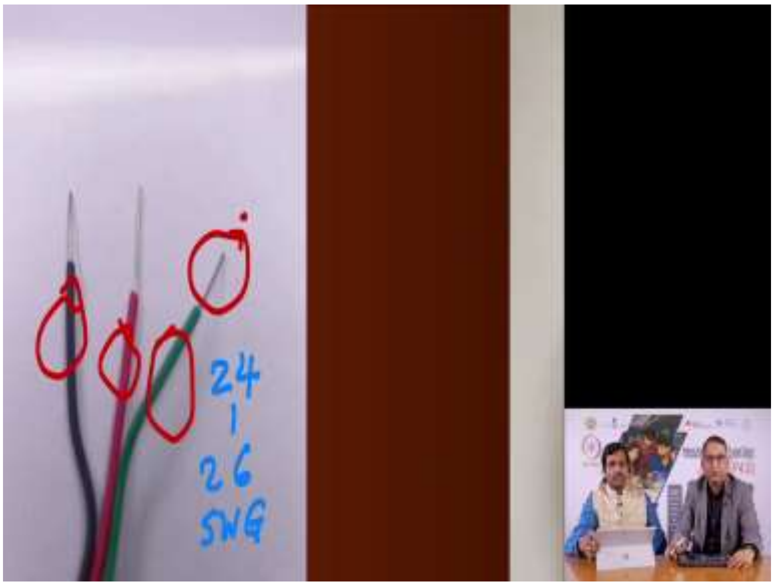
Professor Dr. Badri Subudhi: So you may need a few tools for prototyping on breadboard. So you can see few examples of prototyping breadboard tools. So first one is you can see on the screen is a couple of single stranded wires, the second one you can see is a wire clipper, the third one you can see is a nose plier.

Fourth one you can see is a pusher or a tweezer and fifth one you can see is a wire cutter. So these are the following tools you may need for prototyping a breadboard.

(Refer Slide Time: 10:21)

Professor Dhananjay Gadre: Let me explain the types of wires. It is very important that we use only single strand wires, because by the way wires come in two types; one is what is called as a multi strand wire which means tiny strands of copper wire are bunched together so they are all shorts. Apart from multi-strand wires you also get what are called a single strand wires which means that inside an insulation like here you see this is green insulation, this is red insulation and this is black insulation, you have a single strand of copper wire.

(Refer Slide Time: 10:57)



This may not appear like copper and that is because it is covered with tin that protects it from oxidation. So these are the kind of wires that we recommend you to use. Why? Because it is very easy to insert it, if you have instead of a single strand wire, if you have a multi strand wire, then it will not have the physical strength to be inserted in those holes and it will not go in those holes.

So it is very important that one uses single strand wires, you can get an entire roll of single strand wires with different colours of plastic insulation. Here it is a black insulation and you can have a bunch of that. What is important to note is what is the size of, what is the gauge of these wires and we recommend that you use something like 24 to 26 swg, this will go nicely into the breadboard holes and it will also make good contacts.

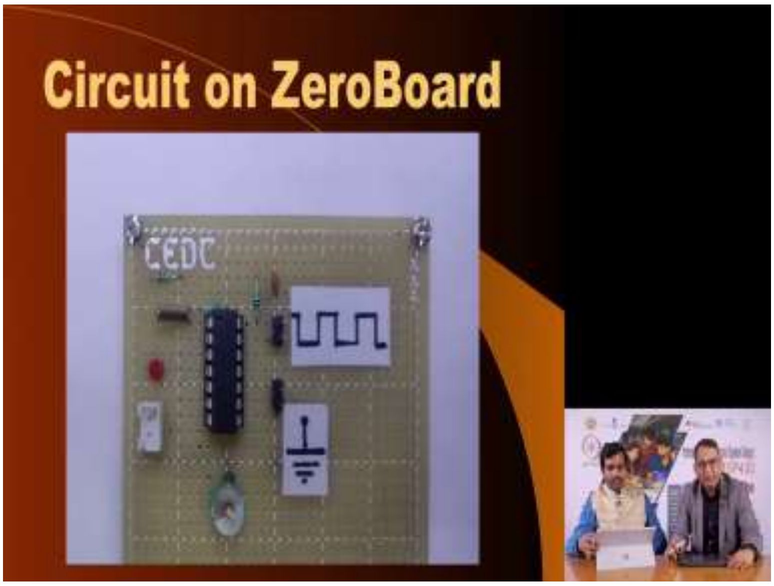
If your wire is too thick, it will not go, you will have to force the wire inside. If it is too thin it may go in but may not make solid electrical connection with the wires within the breadboard and therefore thin wires are not recommended thick wires may have physical difficulty of insertion so we do not recommend those kind of wires.

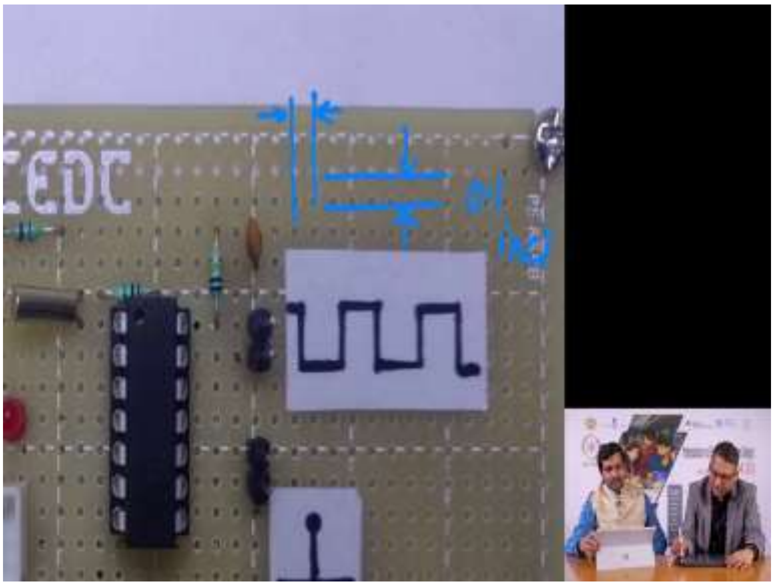
(Refer Slide Time: 12:16)

So, what we what you should have is access to single strand wires of such a dimension such gauge and with different insulations. On insulation let me also make a point that the typical insulation that you would get in the market is plastic insulation.

However, apart from plastic insulation, you also get such insulated wires with Teflon insulation and it is not recommended to use Teflon insulation for bread boarding purpose because removing a Teflon insulation is a very-very tough and tricky job and you need a very specialized wire removing tool and we do not recommend using that for breadboard purposes, using a normal plastic insulation copper wire of 24 to 26 standard wire gauge is quite suitable.

(Refer Slide Time 13:09)



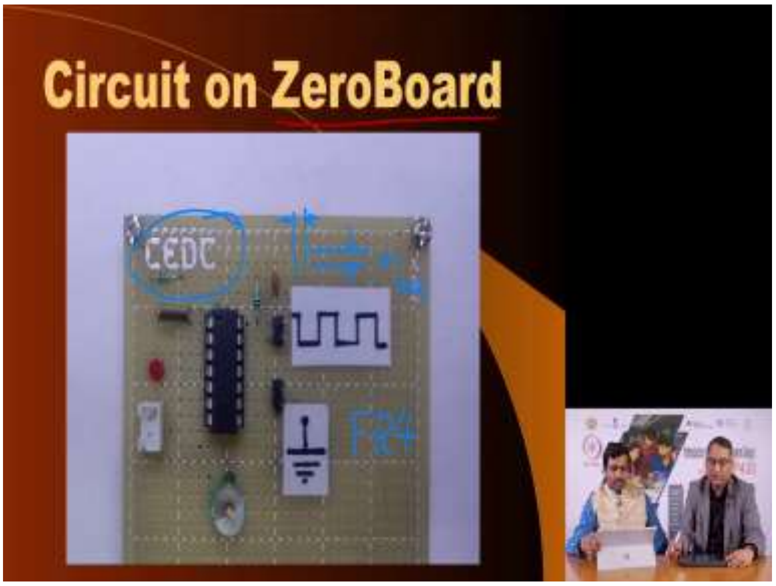


Here is a second example. So breadboard as we mentioned is a good way in fact you have seen the use of breadboard circuits in many of our experiments in the past. Here is a second example of how to prototype a board and that we call a zero board. This is a name we have chosen and it is commonly used.

There are other names you could call this a general purpose printed circuit board because it is for general applications you can use this PCB for any kind of circuit and again here it has similar feature and characteristics that the holes here the pitch of the holes whether in the horizontal or vertical direction, here is constant of 0.1 inch.

Why? Again because it allows easy insertion of ICs and other components. And the difficulty is that it is difficult to use it with surface mount devices and I will show an alternative to that.

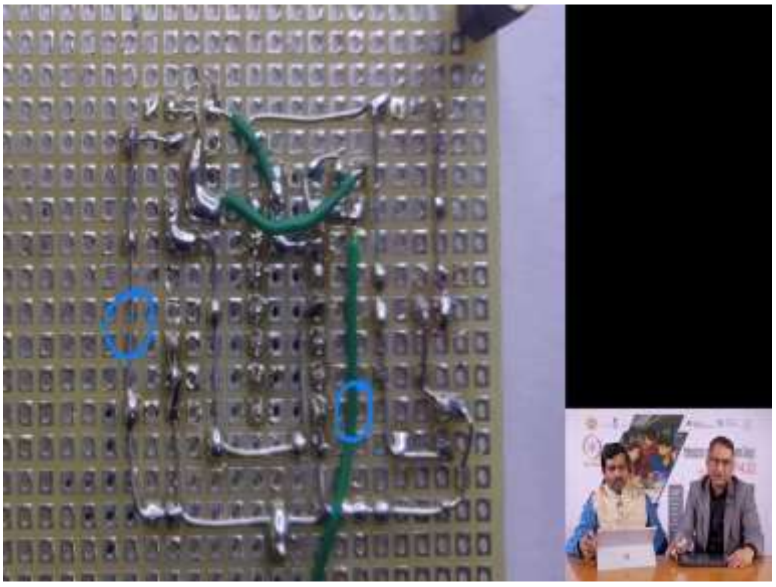
(Refer Slide Time: 14:14)



Here you see a PCB, a zero board that we ordered in bulk, which is why it also has a name of my lab, but you can get it in the market very easily of various dimensions. The material of the PCB is very important, this is called FR4 stands for fire resistant type 4 substrate that is the insulating material on which copper tracks or copper pads in this case are pasted.

You can insert components from this side and they are soldered on the other side. So, this is the top side of the zero board or general purpose board.

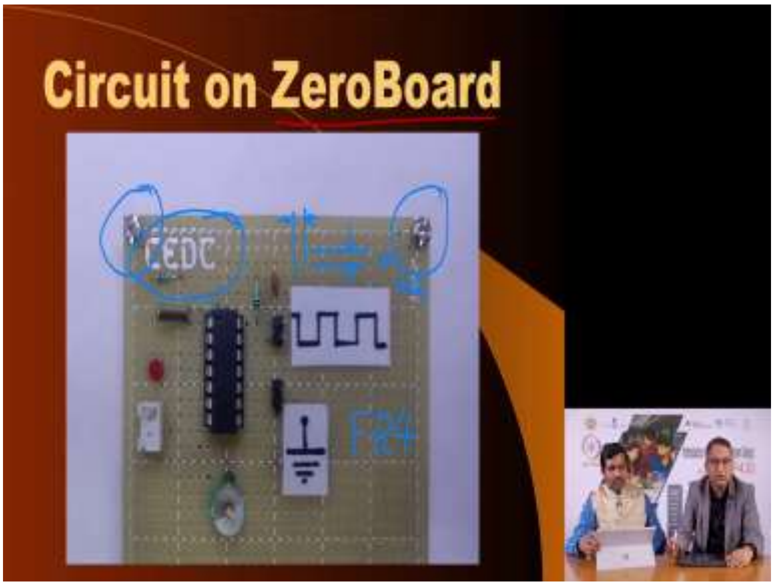
(Refer Slide Time: 14:58)



In the next slide you see here is the bottom side of the same circuit. As you see here, it consists of these are the copper pads. They do not appear like copper as I mentioned copper if left without any coating will lead to oxidation and therefore usually it is coated with tin. Tin does not oxidize as easily as copper and it also helps in soldering. As you see, we have used a couple of types of wires, you already recognize this wire, this is the same plastic insulation wire, single stranded plastic insulation wire.

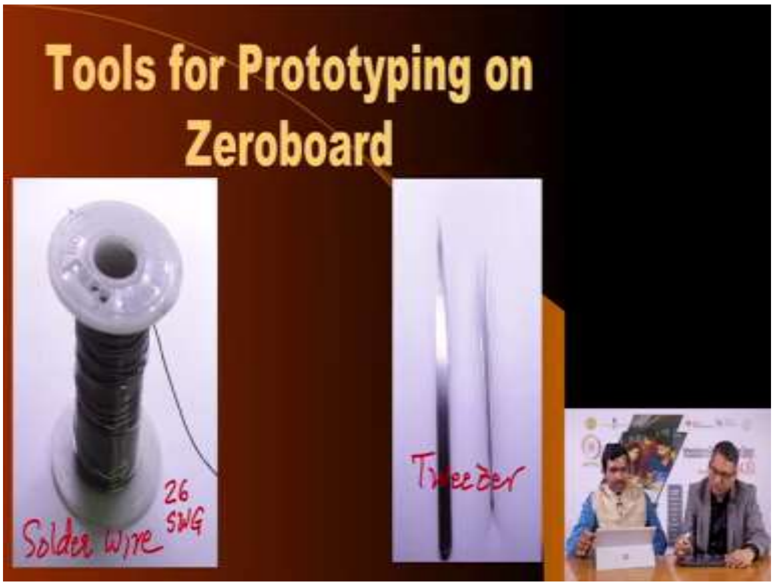
But when we know that the wiring is not going to be creating any shorts with anything else then you can also use this is called hook-up wire. This also is in the same, it is a copper wire with tin coating and this also is available in different dimensions. As I mentioned earlier, we use a standard size of 24 to 26 swg for such purpose.

(Refer Slide Time: 16:01)



And here you see the circuit has been soldered. And you see this is very important, these are all mounting holes, because at the end of the day this PCB will rest on some surface and if the surface is conductive, then it might short the connections on the bottom side therefore, as you see here, these are the mounting holes mounting screws, which we have, which allows us to have the PCB little bit (up) away from the surface. So, you should plan your circuit to be soldered in a way. And then this is a good method for prototyping and testing a small circuit.

(Refer Slide Time: 16:40)



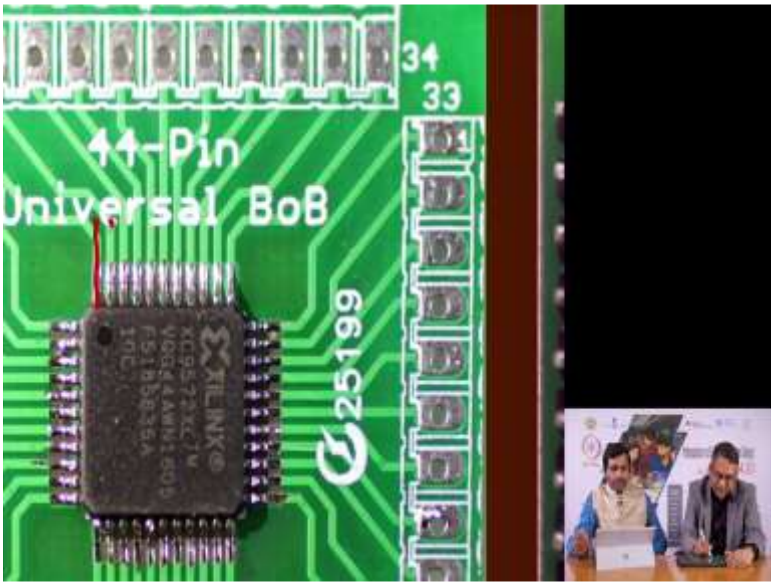
Professor Dr. Badri Subudhi: So as like, we have used different tools for breadboard, there are some tools we may need for prototyping zero board also, so part of these components is soldering wires, so you can see in the left side of the display, so this is the soldiering. So these wires are made up of 60 percent tin and 40 percent lead and right side you can see these are forceps of different size or tweezer of different sizes.

(Refer Slide Time: 17:19)

Similarly, you may need a wire cutter so as previously we have used a wire cutter similarly, you can use the same type of wire cutter here also. In spite of that you may need a soldering station. So, you can see the example in the right side this is a whaler soldering station, but it is not necessary to use a soldering station in place of soldering station, you can use a soldering iron which is easily available in the market.

Professor Dhananjay Gadre: Now, as I mentioned earlier, often times, you may need to use a surface mount device. And the issue with surface mount device is the distance between the pins of an IC or a component, the distance has been reduced significantly. And the other problem is that there is no uniformity, there is no standard like you have your normal DIP, the distance between pins is 0.1 inch, but surface mount devices have a variety of pitch.

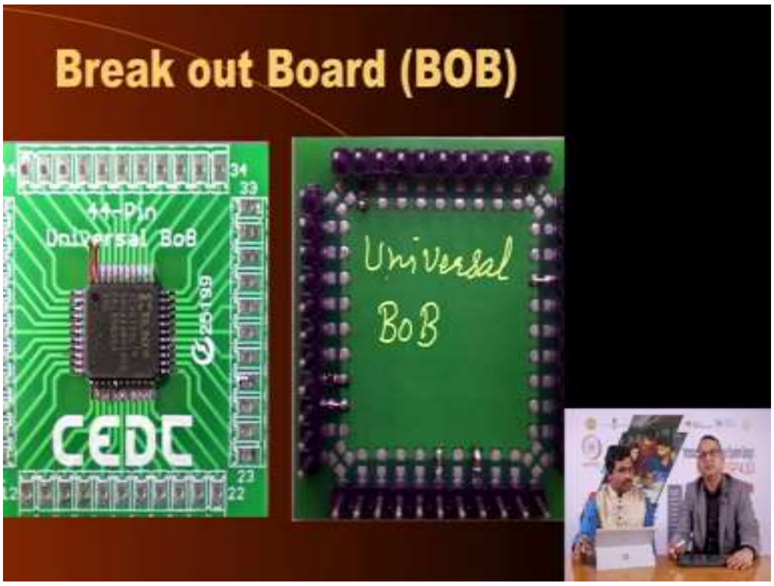
(Refer Slide Time: 18:21)

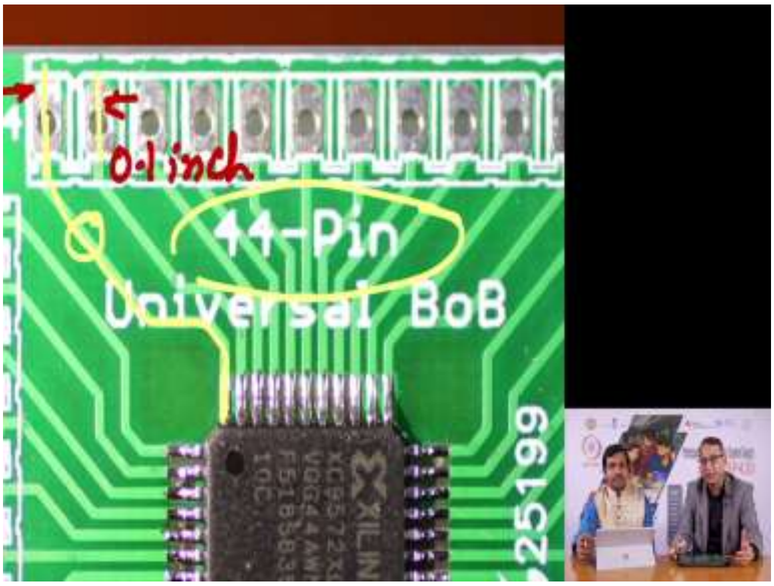


In this case, the distance between these two wires is relatively large for SMD. And there are other SMDs where the pitch of the pins is even narrower. And if that is the situation and in fact increasingly, the integrated circuits that are available these days are being fabricated are being made available only in SMD versions.

And therefore it raises a very important question and a doubt as to how could we prototype using the zero board technique involving circuit components which are primarily surface mount devices. So the solution is to use what is called as a breakout board. In this example, we have a breakout board in fact, we call it a universal breakout board.

(Refer Slide Time: 19:08)





I will explain briefly why we call it universal BOB. The reason is, you see each IC would have its own characteristics, some pin would be connected, would be used for connecting the supply voltage, some maybe input pins, some maybe output pins, and if you have certain pins which are ground and on SMD IC oftentimes multiple pins are available for connecting to the ground.

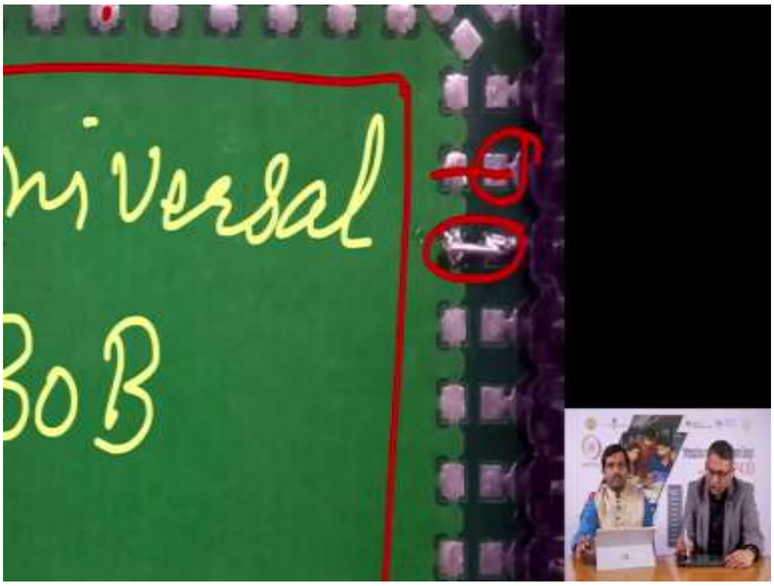
So if you dedicate a breakout board for a given IC, then you would be stuck to using that breakout board only for that IC. So we wanted to create something of a universal breakout board meaning for a given size of the IC. Irrespective of the type of the IC, I should be able to use the same breakout board and here is an example. This is a 44 pin universal breakout board.

Here it does not matter, in this case, this is a Xilinx CPLD which we used and we will use for displaying or demonstrating the single purpose computer during that lecture. And you see we have soldered it here, but we can solder other types of ICs as long as they are available in 44 pin TQFP footprint. Now, how do you make a universal breakout board?

Now, you see from each of the pads, you have a copper wire as you see here, this is a copper wire. This is from this pin this copper wire is going here sorry it is going to the next pin, if I erase this, so you see from this wire, this pad, the wire copper track is going here.

These holes are placed at a pitch of 0.1 inch which as we mentioned earlier, the zero board allows you to insert components which have a pitch of 0.1 inch so, this solves that problem. Now, the second problem how do we make this universal?

(Refer Slide Time: 21:20)

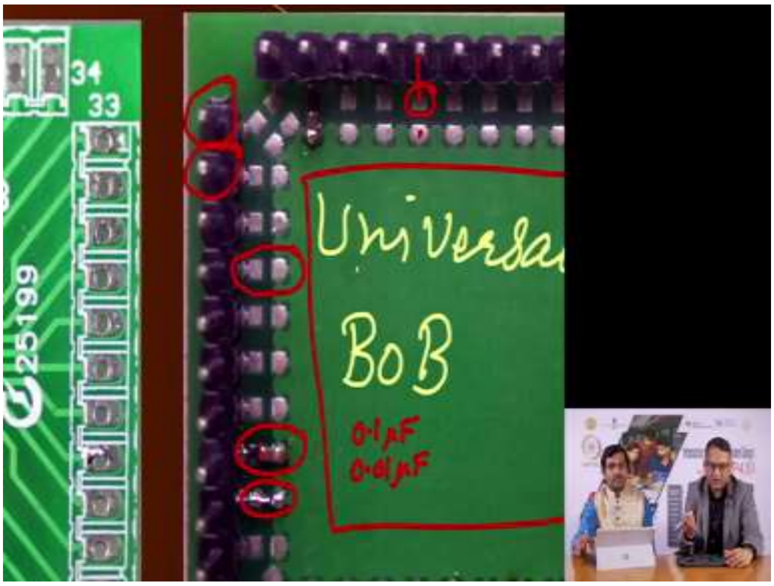


For that we must look underneath the other side of this PCB, you see all these pins have two pads, this is a pad here connected to this pin and here is a pad which is connected to an entire what we call as the ground plane that is this entire green portion is actually copper. Now, if we decide, if we know that this particular IC has say this pin to be grounded, then we will simply short it with a wire as you see here.

You see this wire shorts this pin to the ground plane, any other wire you see here? Here you see this is shorted together. So, all the ground pins of that IC can be shorted without being committed to a particular type of IC, then some of the pins will be power supply pins and as is well known that whenever you have a power supply pin in an electronic circuit, you must connect a decoupling capacitor.

Typical value of the decoupling capacitor is 0.1 micro farad or it can also be point 0.01 micro farad. And here what we see is the neighbouring pin is actually a Vcc that is a supply voltage pin and we have soldered a 0.1 micro farad ceramic SMD capacitor.

(Refer Slide Time: 22:43)

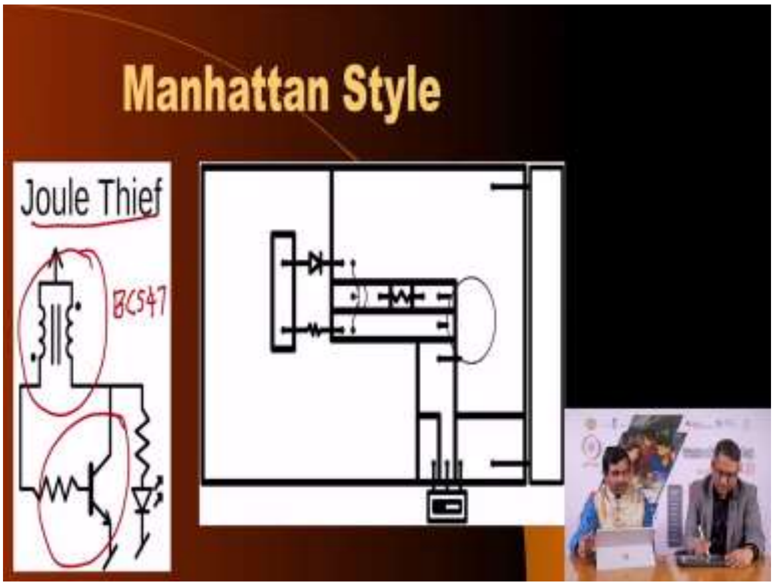


So, what these pads serve the purpose of either connecting a particular pin to ground plane and therefore connecting all the ground pins together or if that particular pin is a Vcc supply pin then it allows us to connect a capacitor between the supply pin and the ground pin, rest of the pins which are neither supply pins nor ground pins, they are left unconnected.

And these are jumper pins, which are soldered on the breakout board so that this can be inserted in the zero board. So, this technique allows one to solder or utilize surface mount device components even in the prototyping technique using the zero board or the general purpose board.

So, this is a very important facility that using a breakout board you can mix surface mount device components as well as normal leaded components as they are called in a mixed design and yet still be able to prototype using the general purpose PCB method.

(Refer Slide Time: 23:58)



This next style as I mentioned earlier is called the Manhattan style of prototyping. Now, here is an example, here is a simple circuit that we want to prototype, this is popularly called Joule thief, because it is actually a sort of a DC to DC converter. This is basically an oscillator with these two inductors which are coupled with each other and general purpose NPN transistor such as BC547 and the supply voltage here is nothing but a simple (I am going to erase all this).

(Refer Slide Time: 24:40)

So the supply voltage for such a circuit is a single 1.5 volt battery, I am going to ground it to show 1.5 volt AA or AAA battery you can use, and this is a white LED, this LED is a white LED in this circuit. Now, as we have discussed it earlier white LED cannot be turned on or cannot be controlled with a 1.5 volt supply voltage, you need much higher voltage closer to 3-3.5 volts to be able to turn on a white LED.

But this circuit allows you to do exactly that by using sort of a DC to DC very cheap DC to DC converter configuration, which uses one transistor and these two inductors, they form an oscillator. And with that you get roughly 8 to 10 volts in spiked form, meaning it is not a continuous 10 volts, the output is more like something like this.

And this peak could be 5 to 10 volts and this is sufficient to drive a white LED. This sort of circuit is called Joule thief because it sort of robs or rather extracts the voltage from a small 1.5 volt battery and yet drives a white LED, this is a popular circuit you will find on the internet very commonly.

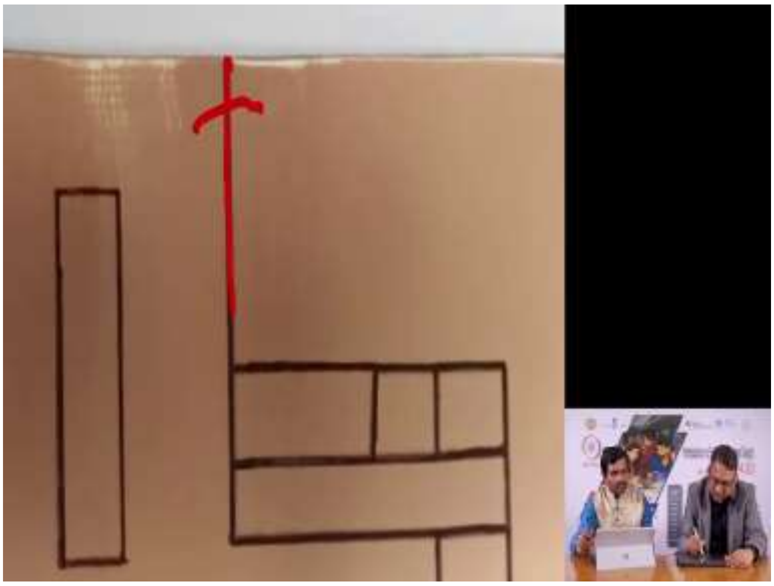
Now, we wanted to show how such a circuit can be fabricated using this Manhattan style circuit board. So, for that, we first created a layout meaning given this circuit, how would our components be arranged and here you see, this would be our, this dimension is our printed circuit board.

Actually it is not the PCB, it is a raw copper clad board, meaning the board that you would get in the market with the substrate of various sorts pasted with very thin sheet of copper, typically the thickness of the copper that is available that is pasted on substrates is roughly 35 micron thick. And this is the raw form of the copper clad board that you would get.

These tracks that you see these black things is basically used to cut and separate various segments of the copper to form small islands. And once these islands are formed then you can solder components as listed here you see a diode, this is actually the LED, this is a resistor this one, here is another resistor which is this base resistor. This is the transistor here in between and what else we have?

We have an inductor, this inductor is here and on-off switch is not shown here, but this is the on-off switch and you need a 1.5 volt battery, this is the battery holder, these are the two connections as you see the plus and minus are separated because the copper layer has been removed from these two parts so let us see how it looks like.

(Refer Slide Time: 27:40)

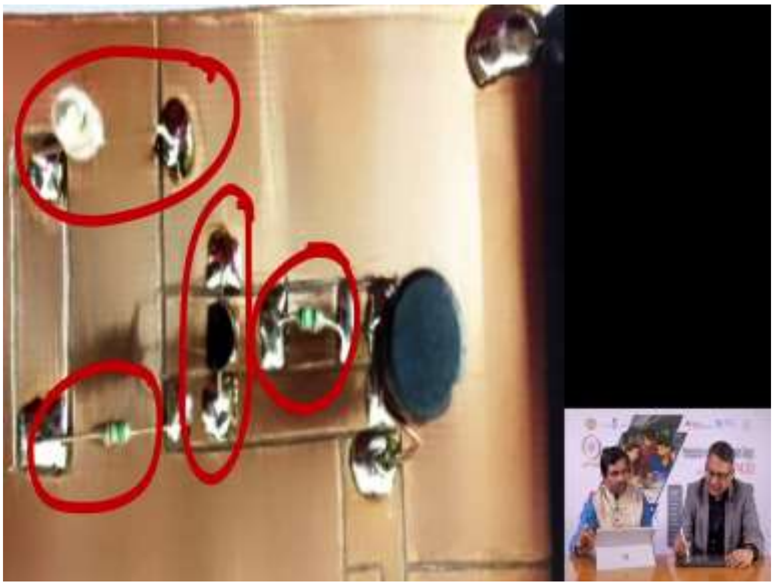


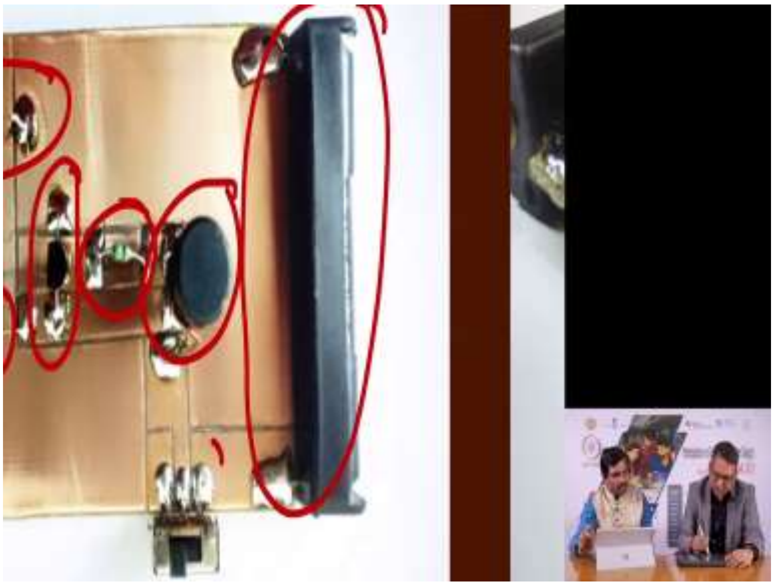
Once you have laid it out, you have to have some practice of laying out layout on piece of paper and then you cut a raw copper clad board of suitable dimension and then using a file, a thin file or you can even use the sharp edge of a knife or sharp edge of forceps to remove the copper. You see this, there is no copper here and so, islands have been created.

(Refer Slide Time: 28:15)

And then once you have this is another view of the same PCB, different islands of copper have been created which means now instead of a contiguous layer of copper it is partitioned in small islands and now you can solder your components.

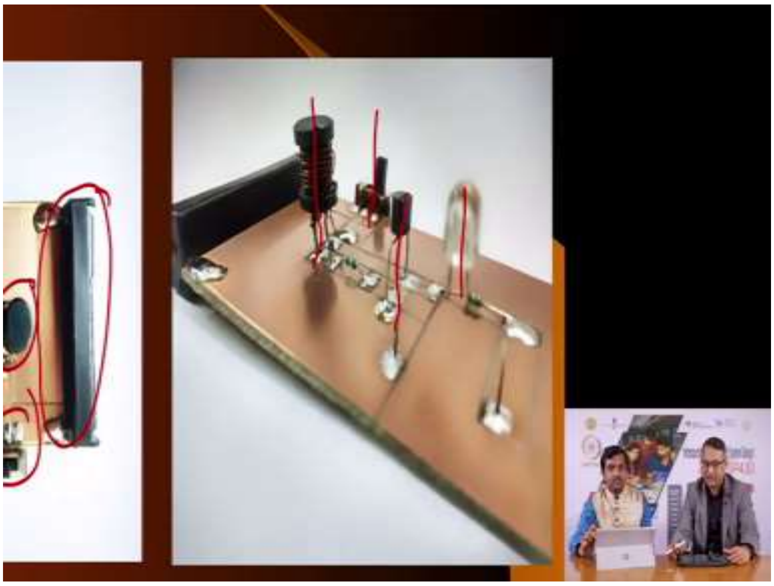
(Refer Slide Time: 28:31)





And this is what a soldered Manhattan style PCB looks like, here is the white LED, here is the resistor in series with the LED, here is our BC 547 transistor, here is the base resistor for the transistor, here is an inductor. And you see a battery holder has been soldered here and an on off switch is soldered here.

(Refer Slide Time: 28:55)



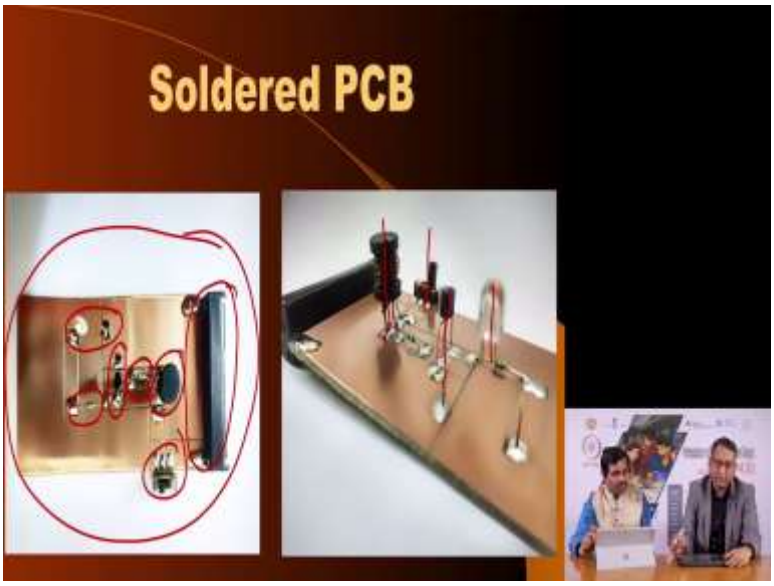
Here is another view. Now you see as you see here, you see these components are rising up in the air and this is a very simple circuit, but imagine that there are 50-100 components like this you will suddenly see these components as tall buildings. And so this is the reason why this technique of prototyping a circuit is called Manhattan style, this is very popular whenever you use high frequency circuits.

(Refer Slide Time: 29:23)

Now, in this particular case we used a single sided board meaning that copper was available only on one side of the PCB, the other side was blank. But use also get such PCBs of the type called double sided PCBs which means, you have copper on both sides. So, you could utilize one side force soldering your circuit and the other side of the copper could be connected to ground to create what is called as ground plane.

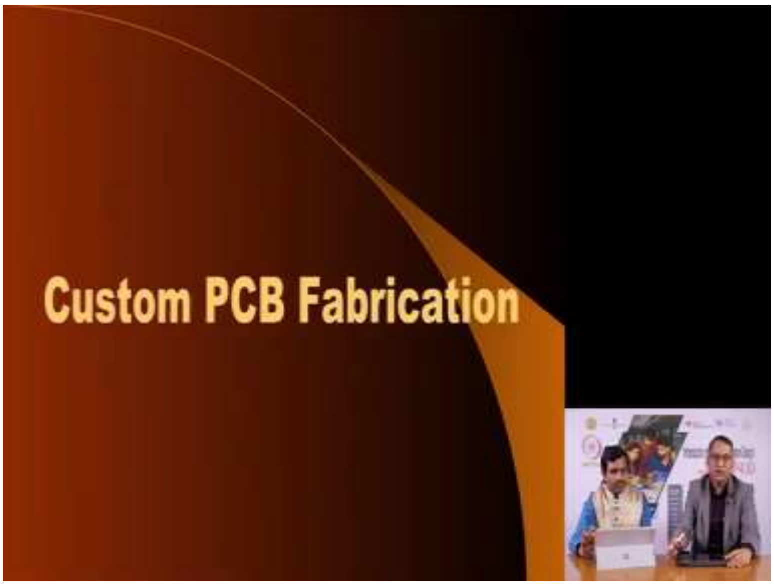
And having ground plane is very helpful whenever you are testing and prototyping high frequency circuits. So, in fact this is very popular with the ham radio community who build radio circuits of very high frequency this is a very popular method, quick method of creating your own circuit implementation. And in this what all you need apart from soldiering wire and soldering iron, you need a cutter for cutting the components to appropriate size.

(Refer Slide Time 30:32)



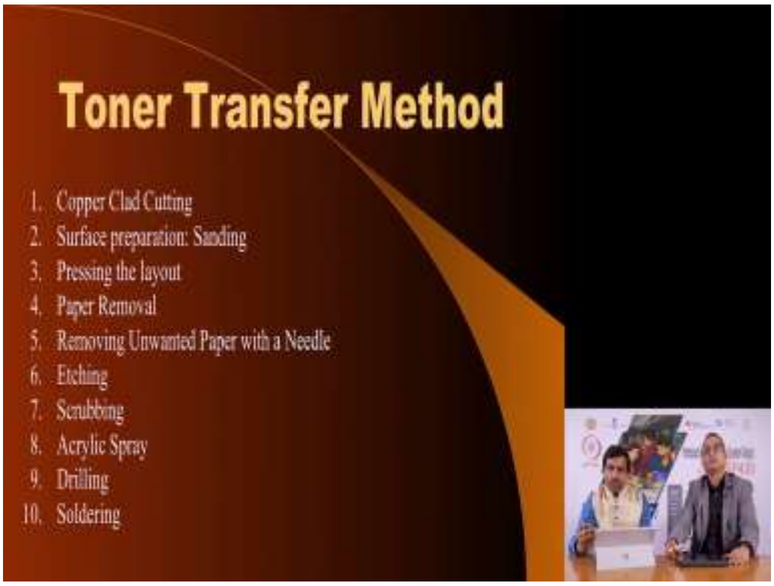
And ofcourse, you need a file what is called as a watchmaker's file, which is a very thin file for removing unwanted copper to create these islands, apart from that, you need a hacksaw to cut a PCB of appropriate dimension. Other than that, no specialized tools are required for prototyping a circuit using this Manhattan style PCB.

(Refer Slide Time: 30:56)



At this point, we now come to the most common and most comprehensive method of creating professional and custom made circuits and that is the using custom PCB fabrication techniques.

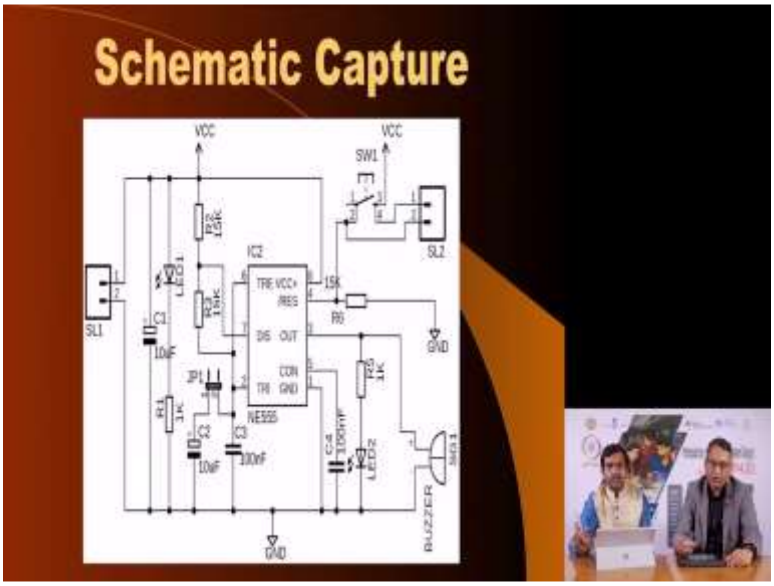
(Refer Slide Time 31:00)



Professor Dr. Badri Subudhi: So first of them, you have to use a toner transfer method, so in toner transfer method, initially copper clad cutting will be there. So, after that sandpaper can be used for sanding the surface for surface preparation then a layout will be placed so it is called as placing the layout.

After that paper will be removed so paper removal, then removing unwanted paper with a needle, then etching then scrubbing then acrylic spray then drilling and soldering. So, these 10 stages are mainly required for toner transfer method.

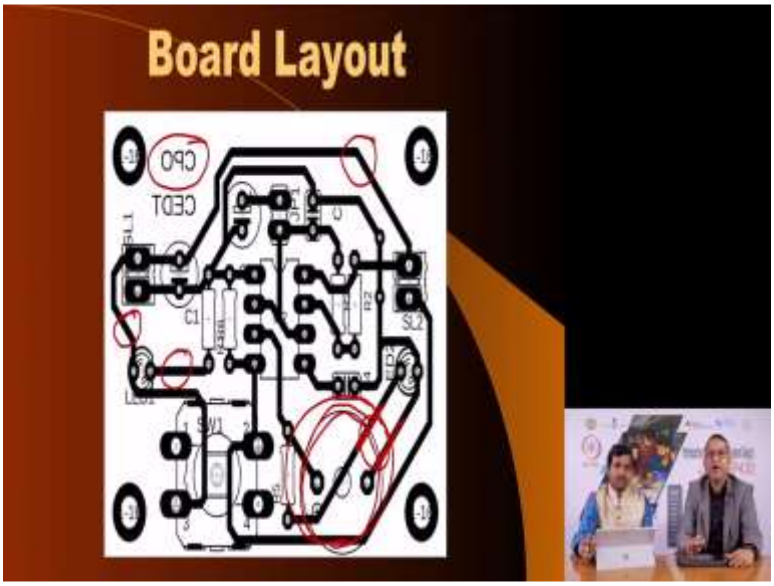
(Refer Slide Time: 31:35)



Professor Dhananjay Gadre: So, let us start with the first step. Now to be able to create a circuit board a custom circuit board you definitely need a schematic. Here is an example of a schematic, circuit schematic and as I mentioned in the very first lecture, that it would be helpful if you knew how to use some schematic capture and PCB layout software.

We recommend using Eagle CAD. And this schematic was drawn using Eagle CAD, it allows you to create a schematic and from the schematic it allows you to create a layout, I will explain both of them here. Here is a circuit diagram, which is basically a oscillator using a 555, as you see here a 555 IC, it has a bunch of components. It is an A-stable multi-vibrator and you create that layout, you create the schematic, from the schematic you create a layout.

(Refer Slide Time: 32:32)



Here is an example of the board layout. Now, you see many of the things you see here, it appears to be mirror image. And it is not by mistake, it is by design and desire that once you create a layout, you can print it out on a piece of paper and you can choose to have the true version or the mirrored version.

And for the toner transfer method, we have to have the mirror version which I will explain briefly shortly why a mirror version is required. Also, this is a board layout which means this is what it would look like on the screen of your laptop or desktop PC that you are using to create this layout.

The thick black tracks, these are going to be our copper tracks which will connect various components as you know mentioned or as listed here or as designed here. So, they will translate into this, ofcourse this is not done automatically, you have to place the components that are shown in the schematic diagram onto your board and then you have to make the connections.

You can choose to use the auto routing feature of the software. But in this case this was done manually. Once you have laid out your components and you have interconnected them then you see since this is just a view on the laptop or the screen, you see it shows where all which components are going to be placed.

The actual layout will not have this information, you do not want because otherwise what will happen is this track here and this track here are going to get shorted because of the outline here which is off a buzzer. So, this is only a view that you see on the screen of the PC, but this is important, once you have this then you have to print it out.

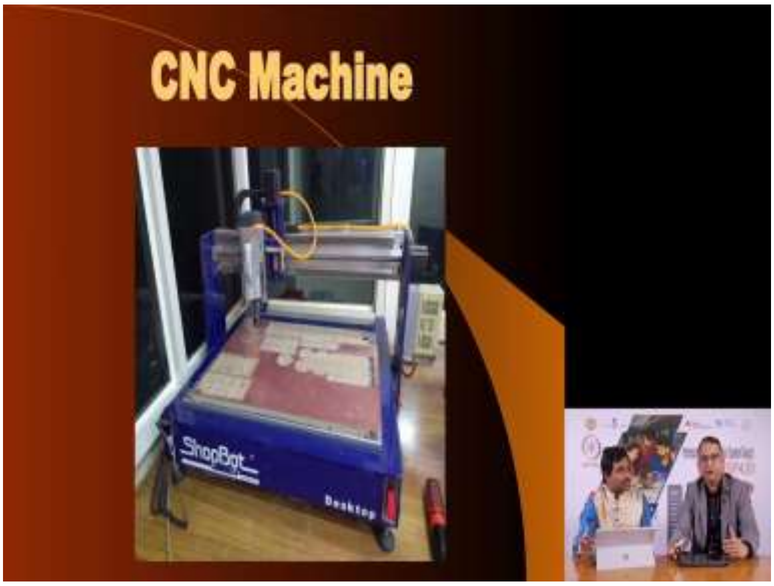
(Refer Slide Time: 34:27)

Now, to prepare for the process, you need a copper clad board and you had to cut that copper clad board to be slightly bigger or equal to the dimension, this is the dimension of my PCB that I wish to fabricate and therefore you must make a appropriately sized, you must cut a appropriately sized piece of copper clad board from a larger stock that you may have.

For that you can use a hacksaw, you can mount the raw copper clad board onto your table in a vice and then using a hacksaw you can cut it. Or if you have access to a Computer Numerical Control machine called as CNC machine then you can program it to cut it off exactly the same dimension as you want.

You can also have that CNC machine drill holes, especially the mounting holes or other holes also and this method is very good if you want to fabricate several, say 20 PCBs you want to make, using the hacksaw method will be manually taxing and may take more effort. But if you do not have a CNC machine then it certainly can be used. So you first cut a copper clad board of appropriate size.

(Refer Slide Time: 35:46)

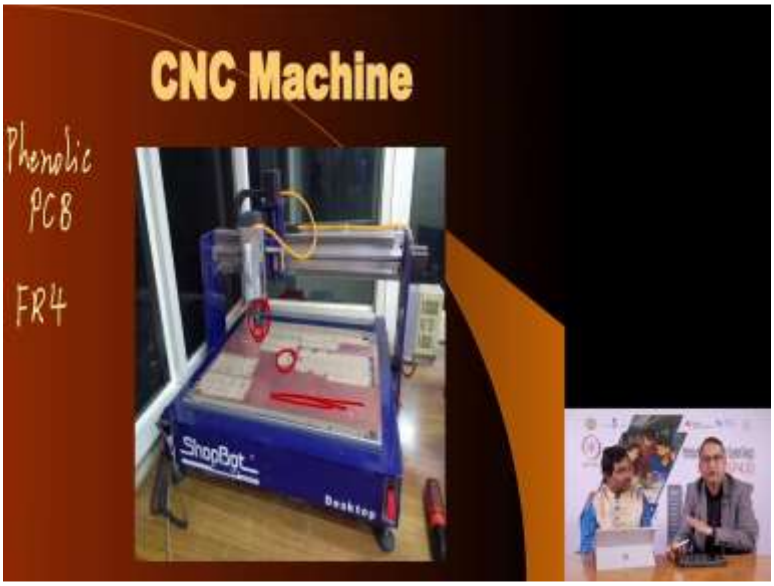


This is by the way, an example or view of a CNC machine that I have in my lab and we use it often to cut as you see here. This is the raw big size copper clad board that has been mounted on the table of the CNC machine and here is the spindle onto which a cutting bit has been installed and it can cut in whatever arbitrary shapes as you see, circular PCBs have been cut so on and so forth.

Anyway, once you have the raw copper clad board and you cut it to size, the surface of the copper clad board will often be dirty and may have some scratches. And having making a PCB on such a dirty surface will not result in a good quality PCB. And therefore, the first important step to follow is to smoothen the surface and to remove all residue of oil or grease or dirt that may have accumulated.

Even oxidized copper would not allow a good PCB to be made and therefore, it is strongly recommended that you first prepare this surface of the raw PCB, and for that the best method is to use a sand paper. Before that, let me also mention that the PCB itself is of various types, you get very cheap PCBs.

(Refer Slide Time: 37:12)



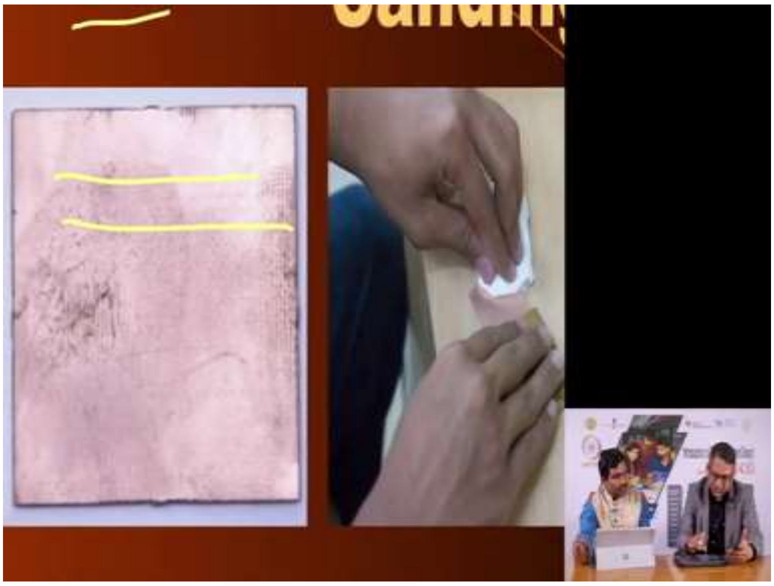
You may have seen brown coloured PCBs, they are made out of material called phenolic material phenolic PCB, and I we do not recommend using such a type of PCB, the good quality sturdy PCB is called FR4 and this is made out of fibre glass. This is the substrate, this is a non-conductive surface on which a thin layer of copper has been pasted. And this is an example of a FR4 type of raw PCB.

(Refer Slide Time: 37:47)

Once you have cut it, then you start scrubbing it, you start using a sandpaper, again sandpaper is available in various types. The grid that is the size of the sand particles which are stuck on the sandpaper is very important because we do not want to scratch the surface, we want to smoothen it.

Therefore it is very important that you choose a fine pitch fine grade sandpaper. In our lab we use 320 grade sandpaper, you can choose any grade above 320, 400 so on and so forth. If you choose 100 grade sandpaper it will leave visible marks on the PCB so we do not recommend that.

(Refer Slide Time: 38:35)



And here is an example, here is the raw copper clad board as you can see, it is very dirty and then you use a sandpaper to clean it and once you have rubbed it, you see it is much more smoother and uniform and devoid of any oxide. So these are the three stages of sanding, sandpapering the PCB. Once you have you have this prepared copper clad board, you are ready for the next stage. What is the next stage?

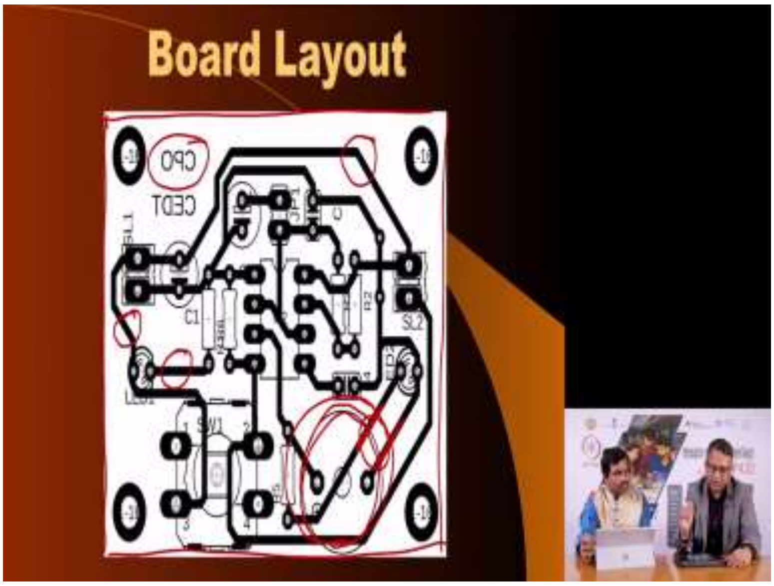
(Refer Slide Time: 39:02)

Then you take your layout, now you see this, you print the layout on a photo paper, you do not print it on a normal paper, what is recommended is a photo paper. Photo paper is available these days on various internet sites as well as regular stationers, this photo paper is the same paper that is used to print photographs.

Because these days people can print photograph using home printers inkjet printer, so similar photo paper is used. This photo paper is used on a laser printer, you cannot use the inkjet printer for printing the layout of the PCB that you want to make. So two important things one of photo paper, two a laser printer. This is very-very important.

Now once you print the layout on a photo paper, you cut it slightly bigger than the dimension that you want. Now you see this printout is different from the layout that we saw earlier let me show it to you.

(Refer Slide Time: 40:02)



Because here, the references of the components and their names and where they are going to be placed they have been removed, while printing we make sure that we want to print only the layout as well as any text on the copper side of the board that you may want as you see here, I have the holes, the components, this is where IC will go, this is where a switch will go and so on.

And I also have this printed here, this is the name of my lab and this is a CPO that stands for code practice oscillator, you can print other relevant information as to who made it, when it was made left up to you. So once you print it out, this is the outside limit layout of the PCB and these are the four mounting holes.

Now what you do is, you flip this and place it on the copper side of the cut PCB that you just prepared, which you use the sandpaper to shine. You flip this layout, the print side, place it on the copper side of the PCB.

(Refer Slide Time: 41:12)

And then use a normal regular home iron for ironing clothes, we use a similar iron about 1000 watts and we firmly press the hot iron on the on the paper. So, that the idea is that the print which is there the laser print gets stuck to the copper. So, you may need some practice but roughly five minutes of intense pressing is required so that the entire paper gets stuck to the PCB.

Here you see the pressing is in progress and after it has been pressed, this is what it looks like. You see, because of the heat you see a little bit of blackening here. This blackening is nothing to worry about, it is basically the rubbing action of the iron which appears to have made the paper black, no issues. Make sure that you do not hurt yourself in this process. Once you have pressed the board, you are ready for the next stage, what do you do in the next stage?

Now this paper was a necessary evil, we do not want the paper, what we want is the layout and we want the layout to be transferred on the copper surface of the PCB.

(Refer Slide Time: 42:19)

And so we will remove this paper and that is very easy. All you do is dunk this PCB with the paper stuck on the surface into cold water in a plate or some in a little container and gently, gently, it is like you are giving a massage to the paper stuck on the PCB. And the idea is basically the paper will get dissolved or will get wet and you can gently ease it off. And once you do that, this is what you will see.

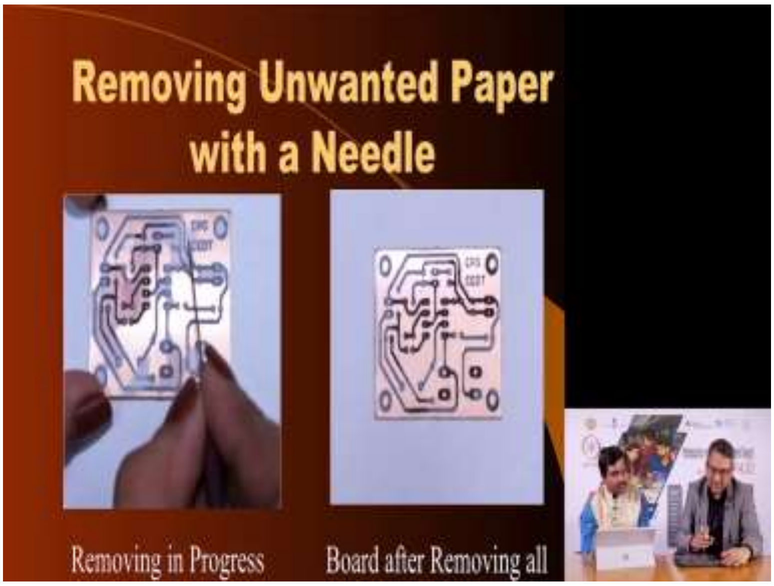
(Refer Slide Time: 42:51)

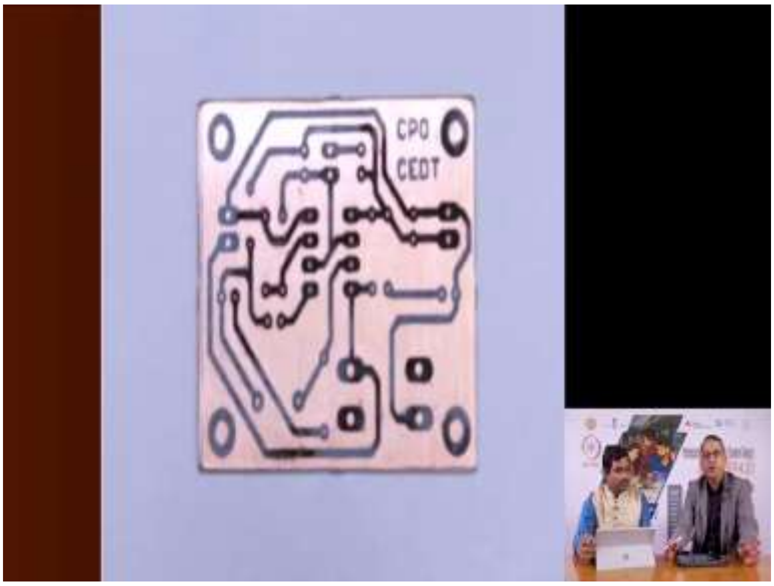
Now you see, you remember I mentioned that we need to print the mirror image of the layout so that when we press it automatically mirrors the mirror and we get the original. So you see, everything you see here is in the correct orientation that we want. This is you can it is readable English, on the layout, it did not appear to be readable and that is the test to see whether this is the correct printer you have made or not.

And now, you see the copper, you see all these tracks which have been which is nothing but the laser print which has been transferred on the PCB. You also see there are white patches here, this hole is not open, that is nothing but paper which has not been removed from the copper.

And so the next stage will involve removing this paper so that you are left with only these print tracks, holes and copper and nothing else, there should not be any leftover paper on the PCB that that is what we need to do in the next stage.

(Refer Slide Time: 44:00)





And here we are going to use a small needle to manually remove what appears to be paper. And once you do that, this is what your PCB will look like. Now, it is ready to be etched, what you have achieved is, you have achieved the print which was from the laser printer on the photo paper to be transferred on to the copper surface of a PCB.

This print will resist any acidic reaction from chemical that we are going to use. And so, it will protect the copper underneath the black tracks or black print that you see the copper underneath will be protected. For that we need to put it in a solution, in a chemical which will react with copper.

(Refer Slide Time: 44:52)

And the most common form of etching process is to use ferric chloride. Ferric chloride may be available in liquid form or you may by crystals and using distilled water, you can create ferric chloride solution. If you have access to a chemical laboratory in your college, they may offer you the best advice.

Always use a plastic container for etching because you cannot use any metal because the ferric chloride will react with the metal. So here we see, dust bin actually is being used for etching the PCB. We also have a rubber glove. This is an important you do not want to interact you do not want the ferric chloride to come in contact with your skin.

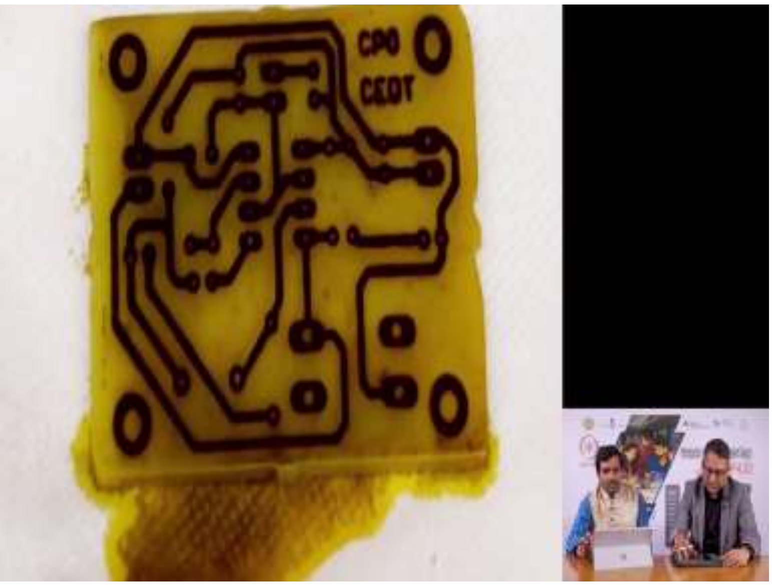
Although it is not very harmful you can always wash it off, but if it can be avoided, why not. So you see copper glove is used to handle the PCB as you put it in the etching solution.

(Refer Slide Time: 45:50)

Once you have put it in the etching solution, you have to vigorously agitate the container and this basically will accelerate the reaction. Another method to accelerate the reaction is to increase the temperature of the etching solution and you could use a heater or something like that, but one has to be very careful not to melt the container.

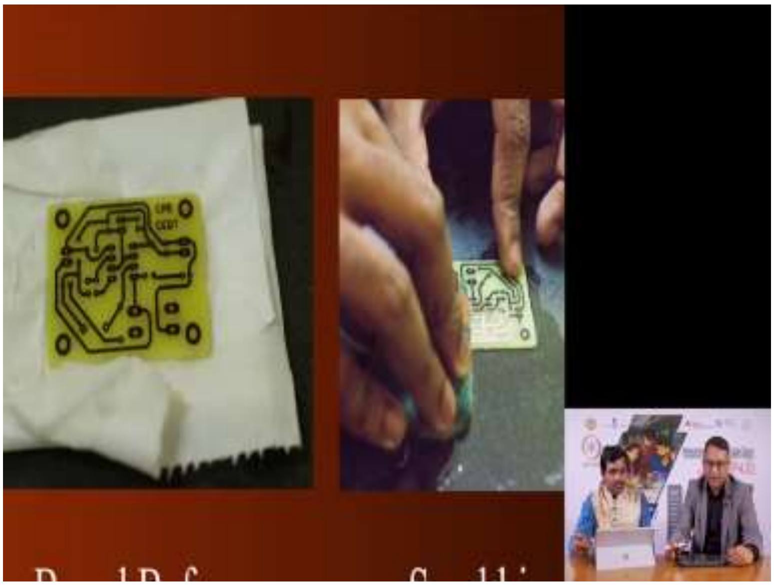
Even if you do not heat it up, it will take about 15-20 minutes of vigorous shaking the container and this is what once you take the PCB out of the solution, you see the copper is gone. Only black tracks are left and the substrate of the PCB, which is the fibre glass material is left.

(Refer Slide Time: 46:36)



Here is the here is what it looks like. And it has just been placed on a tissue paper so that additional ferric chloride is absorbed.

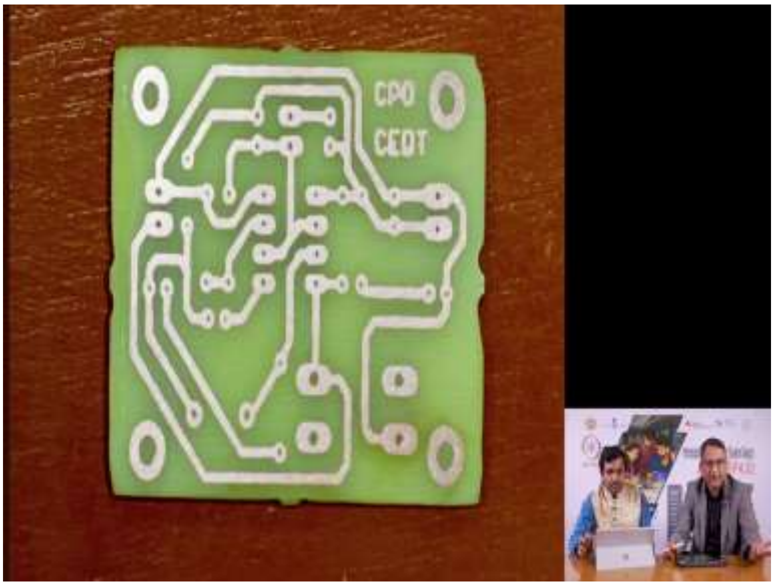
(Refer Slide Time: 46:49)



Now you need to go to the next stage which will involve washing and scrubbing the PCB. So it should be thoroughly rinsed in flowing water and so you could use soap also to clean it. And then what you do is, you use any method to scrub and remove this print that you got from the laser printer, you have to remove that.

You could use a sandpaper but use it gently, you do not want to sand off the copper tracks, you just need to remove the print.

(Refer Slide Time: 47:22)



Once you do that, here is your beautiful looking PCB, and you can match it with your layout to ensure that no tracks are cut. For that you may use some inspection mechanism like magnifying glass or something like that. Make sure that no tracks are broken. And now you are ready to drill holes into this. Once you drill holes, this is what you this.

Okay, before you drill holes, you need to now as I mentioned, copper will oxidize quite quickly. And if you do not protect the copper from oxidation, in a day or two, it is suddenly start looking brown in colour.

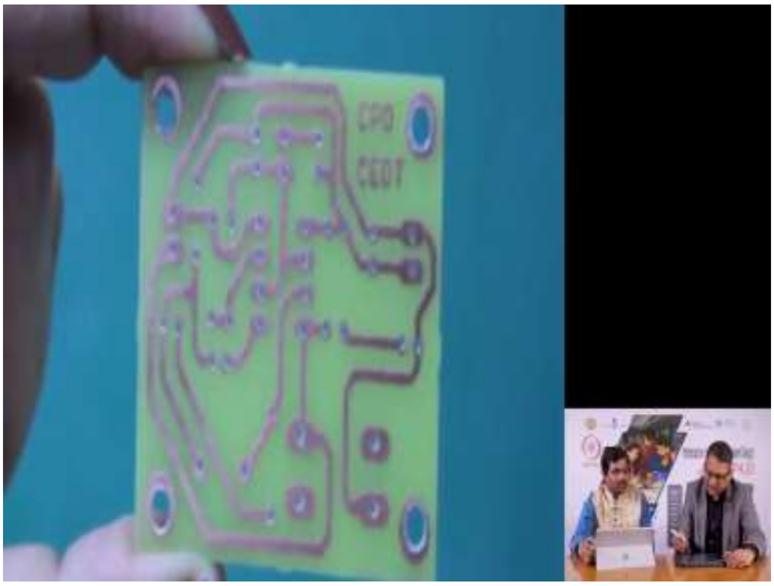
(Refer Slide Time: 48:21)

And so what we do is we use an acrylic spray. It is called conformal coating, this is available in many electronics markets, we spray a very thin layer of this acrylic material onto the PCB and in about half an hour's time it dries off and you are left with a beautiful PCB which has been coated with the acrylic material and it protects the copper tracks, now you are ready to drill holes.

(Refer Slide Time: 48:31)

The first thing you must drill is the mounting holes which are usually we use a 3 mm drilling bit for that. Once you have drilled the mounting holes, you can use a thinner drilling bit. In this case we choose either 0.7 mm or sometimes for thicker components that is components with thicker leads, we use 1.0 mm drill bit to drill holes.

(Refer Slide Time: 49:10)



Once they are drilled, this is what the completed PCB looks like and as you can see the copper, the paper underneath the PCB. And you can see that they have been neatly drilled. Now you are ready to go to the next stage which is to refer to the layout, refer to a bill of material which will tell you what should be the value of various components that you must solder. And so you must bring all these things together.

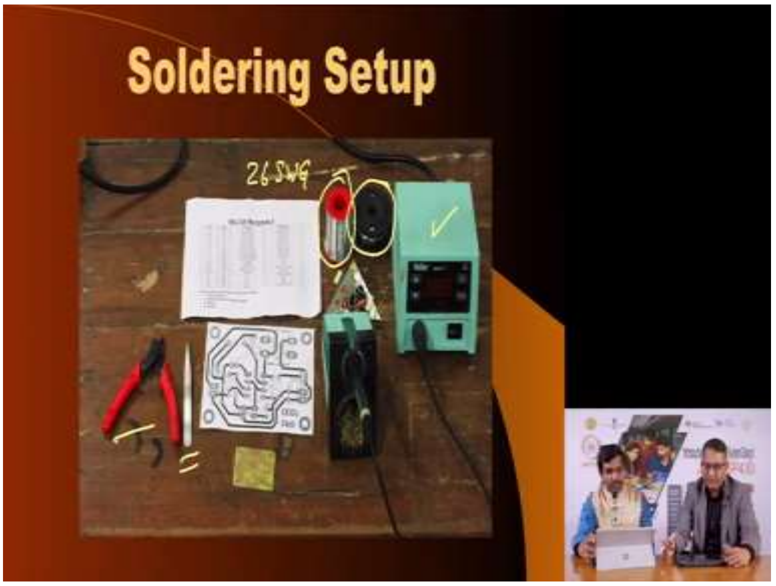
(Refer Slide Time: 49:39)

Here it is being shown in the soldering setup. So not only do I have the soldering iron with me, I have the other material which is very important while you are sorting because you do not want to insert the wrong component in some hole, only to have to de-solder it. So it is better to take preventive measures, make sure that you are sure that this is the component to be soldered in given holes.

(Refer Slide Time: 50:02)

And so we have a layout printed a print layout, we also have a bill of material which tells me which component, what is the value, where it is to go and so on. Once we have that then the rest of the components, rest of the implements and tools are very similar to what we used in the past.

(Refer Slide Time: 50:17)



Here is a cutter, 3once we insert the component in the holes, we have soldered it we need to cut off the unwanted length of the wires for that we need this cutter. This tweezer will be used to bend the components nicely so that they go right into the holes where they are supposed to go so tweezer is meant for that.

On this side we have the soldering station here. This is the soldier wire, spool of soldier wire. We choose the thin size of the diameter of the soldier wire, it is often you may have to find it in the market. We use a 26 swg for that again for the solder wire.

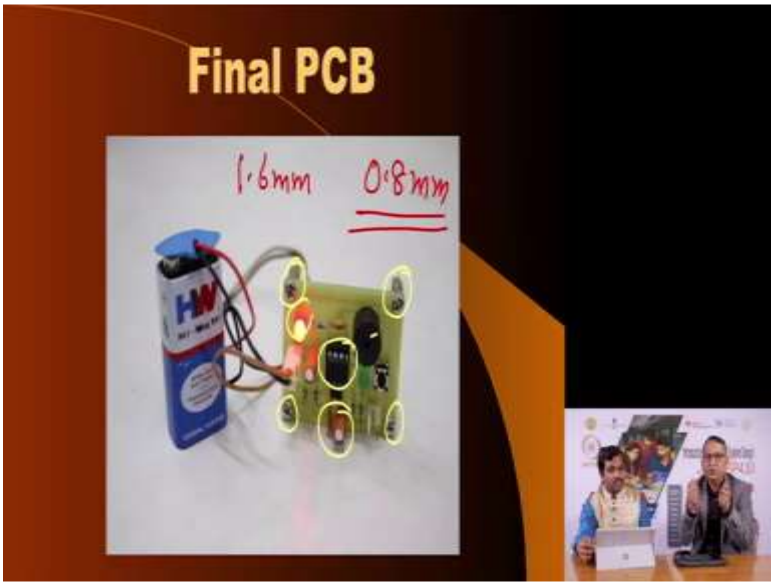
By the way, the solar wire is not solid, it is hollow, it contains flux and the proof of that is when you melt a solder wire on a solder iron or a solder station you see vapours and those vapours are because of the chemical flux. This chemical flux helps to cleanse the surface when you are soldering any electronic components, it cleanses at nano, micro level which you cannot clean otherwise.

And here you also have a spool of what we call as jumper wire or we call as hook-up wire, which is a conductive wire, there is no insulation on it, it is tinned copper wire again, 26 or 24 swg, this would be used to create jumpers because we are making a single sided board. So you need all these things on your table. Your table needs to be very clean and once you have that, then you insert components.

(Refer Slide Time: 51:57)

The way to insert soldering components is that you must hold our components with smallest or lower profile first, which means you solder registers and SMD components first, then you can solder IC bases or LEDs. And as the height of the components increase, you should solder them later. Once you have done that, you can snip off extra leads of the components with this cutter as you see here.

(Refer Slide Time: 52:25)



And once you do that, you are left with a beautiful PCB where all the components have been inserted rightly as you see here, the resistors have been nicely bent at 90 degrees so that they are easily inserted in the given holes. Here is a LED, it is powered with a battery, here is a buzzer this is the 555 IC, here are two capacitors one of them is a electrolytic capacitor, the other is a ceramic capacitor.

And these four mounting holes with screws there allows you to put it on any surface without a danger of making unwanted connections because the surface might be conductive. Now, this is an example of a custom made PCB, which is very good for prototyping. Ofcourse, if somebody says we want to have 10000 pieces of this circuit, it would be very difficult to do all these manually.

What you would do at that point is the layout that you created, instead of making the piece of yourself you can order this PCB to be made by professional PCB fabrication companies and they would give you a large volume in small amount of time. You could choose to solder it manually or even the soldering process can be done in the automatic way.

This is an example of a single sided board meaning that the copper tracks are only on one side, but using this technique in a lab, you can create a double sided board also for that you need isntead, by the way I forgot to mention the thickness of typical copper clad board that you get in the market is 1.6 mm, but you also get a 0.8 mm thick PCB.

And when you want to make a double sided board, we choose to use 2.8 mm thick PCBs. Each PCB will make one side of the layout, and then we align them and glue them together and put wires through holes so that the two PCBs join up and make like a 1.6 mm PCB. So it is possible to prototype a double sided layout also in the lab.

If your circuit is complex enough to merit a multi-layer board, beyond single sided you have double sided, you also have 4-sided, 4-layered, it is not 4-sided, 4-layered or 8-layered or 6-layered, 20-layered PCBs also you can have, for that it will be difficult to prototype in the lab and the best way is to send out your layout to a PCB fabrication facility which has capability to fabricate PCBs of the type of the number of layers that you wish.

Once you do that, you get the PCB back then you can solder manually as I mentioned. In the activities related to this course, in the introductory kind of projects that you may want to prototype, single sided as well as double sided PCBs might be sufficient for a large number of cases.

Ofcourse, I am not saying that only single and double sided boards would suffice. In some cases, you may need multi-layer PCBs, where the only option you may have is to get even a prototype PCB fabricated outside. But you must be aware of all these methods of testing a circuit before committing for a final version. And with this lecture, we hope that we have introduced you to all the common methods of prototyping and we hope that you would use these techniques to chart your own project whenever you are ready with an idea, and you want to actually build it in a way that a professional would build.

And as we discussed in an earlier lecture that PCB is only one part of a system, you would need an enclosure, you would need if there are multiple PCBs you will need to wire them together. I have already covered that in a previous lecture,

And I hope using both these lectures together, you will be able to plan and go onto the path of creating a professional looking PCB, which will give you a good confidence that your final project will work successfully. I hope you find our lecture here very useful. Thank you for listening to us and we will see you soon in a new lecture. Bye, bye, Thank you.