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 36

Building an Electronics Project

Hello and welcome to a new lecture on an interesting topic which is slightly outside the realm of embedded system design and you would recall that in one of the earlier lectures, in fact the first lecture, when I was going through the course objectives. I had mentioned the immediate course objectives for learning embedded system design.

But eventually I had said that I have a larger motive in offering this course and that is to enthuse you budding engineer to fall in love with electronics, fall in love with this idea of creating electronic circuits. And from electronic circuits to building complete electronic systems and products and this lecture deals with those aspects.

So, I am going to talk about what does it take to plan and implement an electronics project, of course today the electronics uses embedded computers, it uses programming, and so when I say an electronics project it is not without programming. An embedded systems empowers engineers like you to be able to build complete systems, because embedded systems provides the heart and soul.

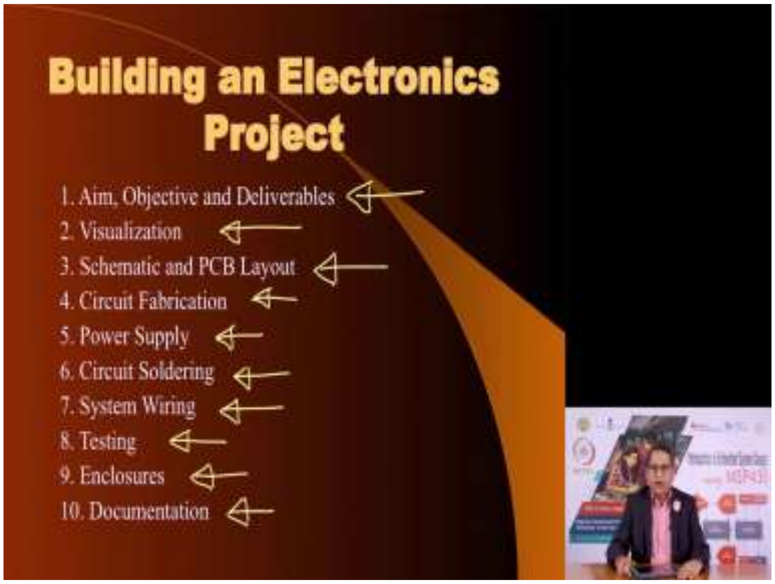
But it also requires a body, it requires a physical appearance and this project is about that, it talks of all the issues required to be considered. So, that you can imagine a requirement based on a problem that you may have perceived. How to plan for it? How to implement all the intermediaries and eventually come to a working product, and support it in various forms I am going to discuss in this lecture, all the related aspects.

So, while this is about building electronic projects, one should see it in a larger context of planning to create an entire product. So, let us go ahead and so we are going to look at all the issues that are required to be considered, when you are thinking of building an electronics project, as a student you would be interested in building some circuit, as slightly advanced student you might be thinking beyond circuit and you would be thinking in terms of system.

And if you are a graduate engineers and if not once you graduate and take up a professional job, it would be your responsibility to be part of a larger organization to build complete systems. And so this lecture is basically to sensitize you about all the aspects that one should consider from start to finish.

And so when you think of building a electronics project and I could even replace this by saying you want to build a complete project irrespective of whether it uses and the electronics circuit, or not I think this discussion will still remain relevant and complete.

(Refer Slide Time: 03:34)



So, you cannot think of building a project or pursuing a project unless you know the aims and objectives. So, the first and foremost requirement of embarking on such an activity is to know what is your role? What is it that you are looking at? And so we will see how we can think about the aim that we have at hand. How to visualize it, and how to create a list of deliverables?

So, that you fulfill your responsibilities, as part of those deliverables one of the important things that you would have to do is to visualize, what is it that has been put in your responsibility, what has been is interested upon you. And so visualize the requirement and once you visualized it in various ways that we will discuss shortly. Today no system works without electronics.

And when you talk of electronics it means creating a circuit and documenting it, building it, testing it and so the second part, the third part of this activity will be to create a schematic and implemented in some way. And one of the most common professional methods is to build a create a printed circuit board. I am going to show you various options, once you have that, then you would like to fabricate it. So, we are going to deal with various issues related to circuit fabrication.

Now, let us assume that you have fabricated the circuit, you would come to a point where you would want to power it, because without providing a source of power that electronic circuit is quite useless. So, we look at power supplies issues, we have already dealt with those in one of the previous lectures. So, I am going to revisit that topic and tell you what are the various methods of testing, once you decide about that you may want to solder the circuit so that it can take a physical form.

And a circuit is only a circuit, it has to connect to the physical world maybe your system is bigger than just a circuit, maybe it has an enclosure, maybe some wires need to be connected. So, you need to look at the possibilities of having to deal with wiring, and therefore we talked about it in the system wiring component, once you integrate all these things you would not want to put it out for others to use, or ship it unless you test it.

So, you need to think about a testing protocol, a testing mechanism and any system is incomplete without being housed in a appropriate enclosure. So, we will see what are the various options for enthuse for the electronic systems that you are building. And at the end of the day no system is complete without documentation.

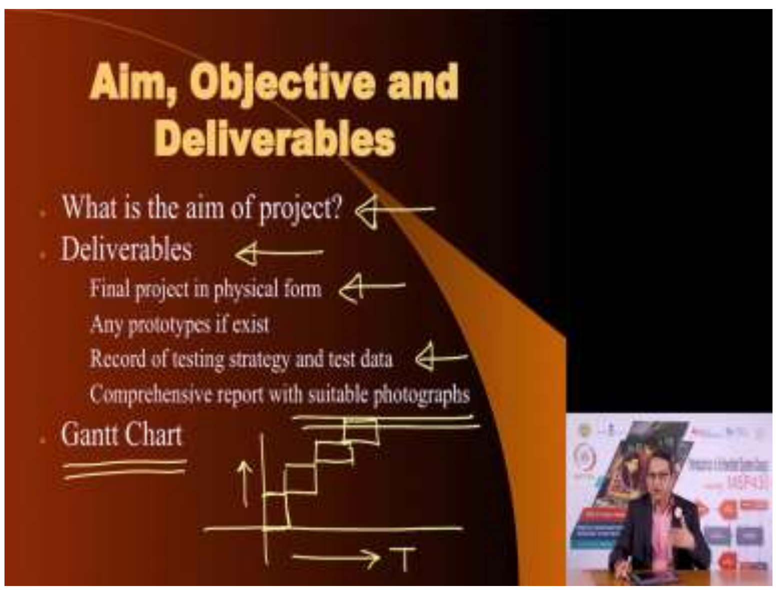
If when you buy any product it comes in a carton, or it comes in a external package and when you open it, what you find is a piece of paper, which tells how to use it. This is one part of the documentation, which is derived out of the system documentation that you as the system designer will have to embark upon.

When some product goes wrong and you send it for repair, the people at the repairing center refer to another set of document, which tells them if things go wrong, what all to test and that part of the documentation that they refer comes from the complete documentation that you as a engineer, you as a product designer will have to put together. And so we look at those aspects as well.

So, these are the what I call as 10 commandments, that you as a budding engineer and a professional engineer eventually must keep in mind when you embark upon this activity. So, let us start and look at each of these aspects. Now, there is no point in moving forward unless you know what you are doing, as they say that if you do not know where you are going, there is no need to worry about what road you take. So, you have to be sure where you want to reach.

So, that you can take the right path, and so the aim of the project that you are doing, the aim of the system that you are building must be very clear to you, maybe it is an iterative process you can start off with some assumptions, proceed on that path and then you find that some of it is not possible, you come back and revisit the initial expressions and refine them.

(Refer Slide Time: 08:05)



And so that may take some iterations, so you must come up with the aim of the project, then you must find out what is expected of you, is it just a prototype? Is it just a paper solution? Is it just a proof-of-concept?

Is it going beyond the proof of concept of building a working prototype and then from the working prototype into a you know marketable product, all these things you must consider in the second aspect of this is what is it that is expected of you from the deliverables point of view. What is the final form, physical form of the project? are you working on somebody else's work?

So, if that is the case are there existing prototypes that you could reference that you could build upon. Is there a record of testing strategy and test data, or are you expected to create it? And then once you are done with it you should be able to create a comprehensive report with suitable photographs.

Today taking photographs is a child's play all of you, all of us are equipped with mobile phones, smartphones and these smartphones offer great capability to take pictures. And you must use those, if not you can take the help of professional support to take really good documentation photographs. Now, in all this a concept called Gantt chart is very useful, a Gantt chart is nothing but a two-axis representation, where on X axis you set time, and on the Y axis you put all the activities that you have to do.

So, let us say you want to plan, so this I am going to take this much time to plan, then you say I am going to start, you know initial ideas. I am going to you know read some literature and things like that maybe it takes that time, maybe I can start prototyping, maybe while I am prototyping I can do something else. So, I can have some overlap, so a Gantt chart is nothing but a 2-axis representation of all the tasks, that need to be done in a serial, or a parallel fashion.

And each of these tasks is say take certain time. And so you can put all of that on a graph and such a representation is Gantt chart, one may must make a reasonable X, X you know a reasonable estimate of the time that it is going to take for each of these activities, maybe one can give some margin for you know some missed opportunities, or some deadlines not being followed up.

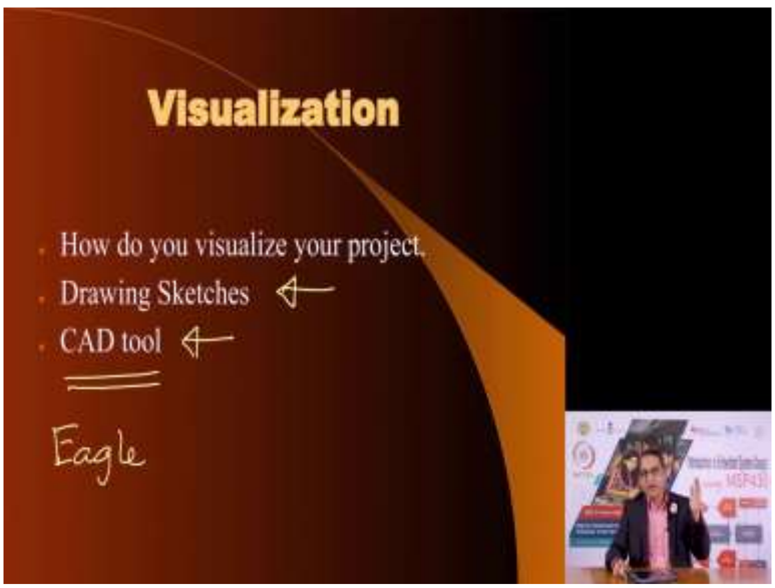
And so one should allow for that and based on these assumptions one should create a plan that this is the time I am going to take to do this. Why is it also important? Is because if we go back to my previous one of my previous lectures, where I talked of time to market, this Gantt chart will be able to help you to estimate that the activity that you have embarked upon. How much time is it going to take?

Are all the parts as I listed earlier those 10 commandments is it possible to achieve them in the time frame that has been allotted to you? If not would do you need more help, maybe more people involved in this activity would help you shrink some of the time. So, as to meet this time to deliver, time to market as one of the important criteria.

So, the Gantt chart is is a tool to sort of visualize all the elements of your project, those 10 things that I mentioned and then put them in a timely fashion, so that you finish. The next very-very important aspect of this exercise is to visualize. What is it that you have building? And this visualization usually starts in a mind, you start thinking this is what you want to do, based on either self-motivation, or as part of a job that has been given to you.

But eventually you start thinking about it, but that thought is not enough, that thought has to be translated in a form that you can work upon, that you can share with others that you can brainstorm.

(Refer Slide Time: 12:16)



And so visualizing your project is important, and one of the ways of doing that is to create sketches. One need not be a great artist to use pen and paper, or any other tool to put all these ideas that are floating in your mind onto a piece of paper, but drawing sketches is important. However poor you may be at it, and I can assure you that with practice you will be able to do a good job of it.

So, please practice using a notebook and a pen, in fact without a notebook and a pen you are not even half an engineer. So, please carry a notebook and a pen with you all the time, so that not only you can use it for drawing sketches, you can also capture the ideas that you that are coming into your mind, and onto a piece of paper for you to revisit them at a later time.

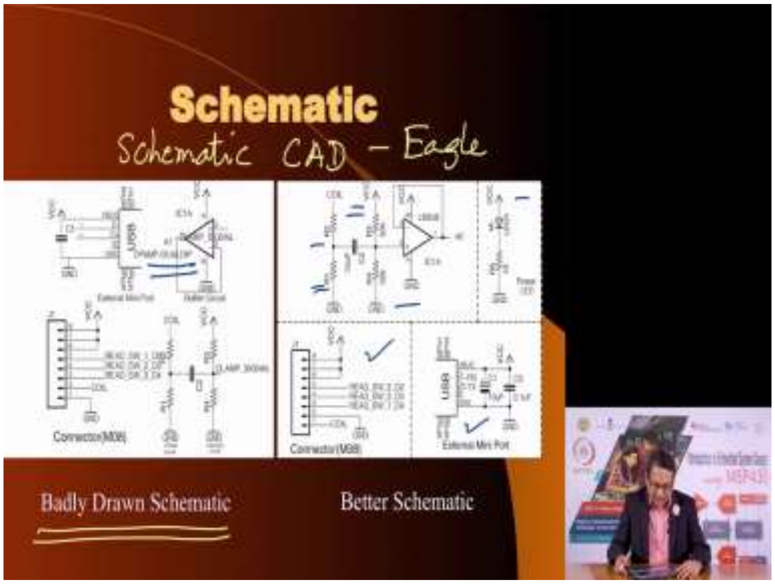
Oftentimes once we have drawn you know raw sketches using paper and pen, it may be imperative to translate them into more formal methods, which can be discussed at an engineering level using CAD tools. And there are many CAD software, for example I use Eagle CAD, which is actually a PCB schematic and PCB layout tool.

But I use it to create block diagrams, in fact many of the block diagrams that you have seen in during this online course, they have all been created by me, or my students using this great tool. It allows me to create block diagrams, to save them and then to export them in any format that could be shared on through various mechanisms.

So, please consider using an appropriate CAD tool, for converting the idea in your mind into a tangible form, which can be shared with others. This can also then be used in your documentation, that you know this is how the idea evolved from a rough sketch and then you from there you made this first iteration, and then you made another iteration, and so on and so forth.

So, this will allow you to look back at how things changed in your mind. Once you visualize what you need, eventually it is going to come down to implementing it in a circuit form. And the first you know diktat in that direction is to actually create a schematic. And again as I mentioned there are many methods you can start by creating a schematic on a piece of paper.

(Refer Slide Time: 14:39)



But again that piece of paper schematic will be of not much value, because that piece of paper schematic cannot be translated very easily in a form that can be prototyped and recreated and tested. And therefore one of the standard methods of creating schematic or capturing the schematic is using a PCB a schematic CAD.

And as I mentioned Eagle is a great software for that. It is it can be downloaded and used for free in the evaluation version and you can even buy professional versions of that, if you are a student, or involved in academia, Eagle allows you very low cost, or inexpensive, or zero cost version, which has more features than a free version.

So,, I would recommend very strongly that you consider this software and here is an example of capturing that schematic using this CAD tool. And I have basically illustrated two methods one is a badly drawn schematic. Now, again it is like an intuitive method, that normally you would say okay all my input start from my left-hand side and the signals progress through that sheet of paper towards the right.

And here if you see on the left-hand side, the components have been very badly arranged and you see the names of components are overlapping with the signals. And so it is not very readable, the important idea of behind a schematic is that you should be able to understand it and others who may be in your team should also be able to understand it. It is like coding, when you write a program, how do you comment that program? You want that the program has a long life beyond you. So, that somebody else can also read it and make sense of it.

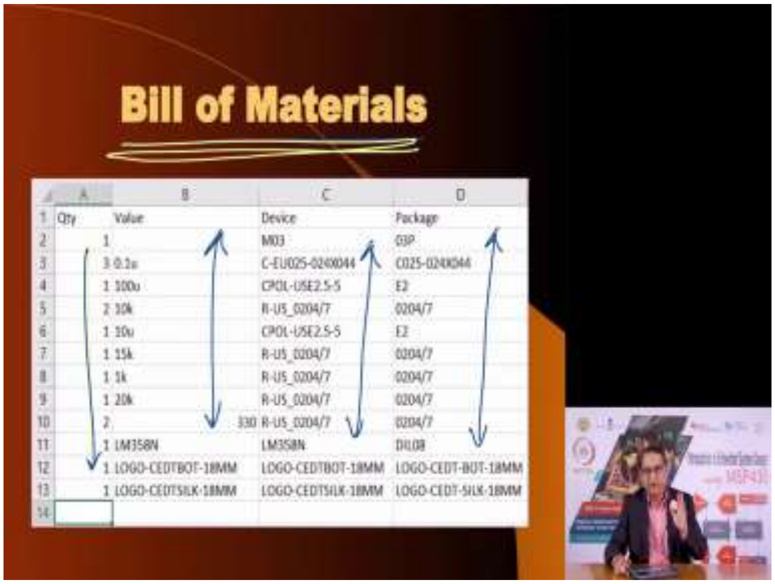
Schematic falls in the similar category that you should draw a schematic, which is easily understandable not only by you, now or a 6 months later, but also others who may not be any who may not be at all aware of it, when they first look at it. And so here is an example of the same schematic, which has been drawn in much better way.

As you see it has been labeled that this is the supply voltage here, this is the ground, this is a collector we are going to use, this is the USB interface for connecting to the system and so on and so forth. The components have been well labeled here is here are the annotations R1, R2 and so on and so forth, on the left-hand side circuit you know they are very half hazard they are merging with other components.

So, it is not a very readable circuit, so this will come by practice, you should draw these schematics using this software, or any other software that you like and ensure that you can create a schematic which is well understood. After you create that schematic, you should of course you could you know simulate it using various simulation packages, such as spice and once you have done that and you are you are confident, that your circuit is going to work, then comes a time that you find out, what are the components that you are going to use.

Because there is no point in having a schematic and then building the circuit, trying to build a circuit only to find that some of the components are missing.

(Refer Slide Time: 18:17)

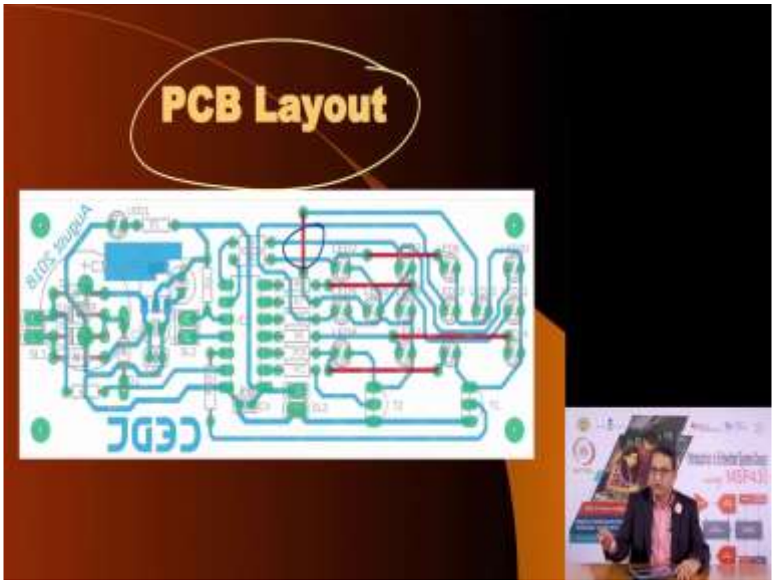


And many of these CAD tools give you this option and this facility too from this to start with the schematic and get a list of components that we call as bill of materials. This bill of material is in your control, but you get to have a look at it in a tabular form as you see here, that it tells me what is the quantity of each of these components that I am going to use that you are going to use.

It tells me what is the value of these components, what is that device that we are using and in physical form how does that device exist, you may want to verify and modify it, if you feel that some of these components you are not able to get, or some of these components you cannot prototype, because of the size of restrictions.

You can modify them by going back into the schematic, change the package of the components that you are using and come back and look at the bill of materials again, take a printout of this and use it as a reference that this is what you need before you take the next step. And what is the next step?

(Refer Slide Time: 19:12)

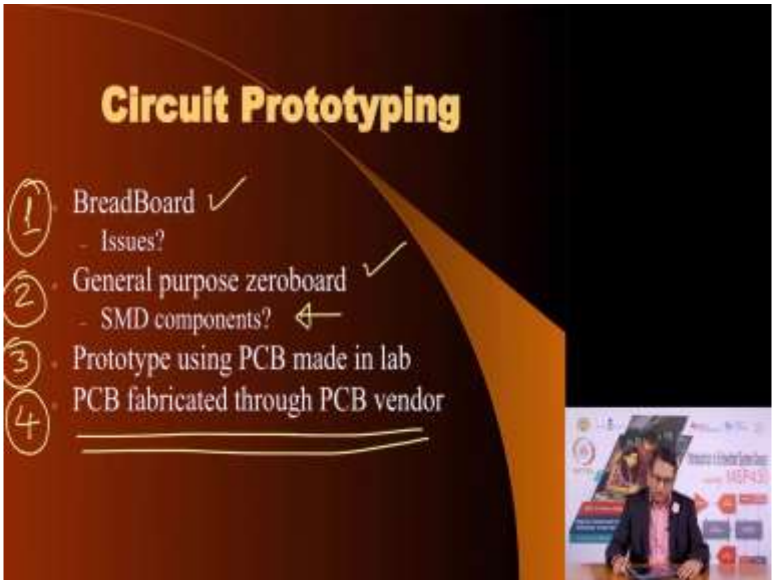


That you would want to create a PCB layout. Why PCB layout? Because that is the professional way of doing things and a printed circuit board can be fabricated in many-many ways. This is a very simple example of a circuit that I am going to eventually show you the entire working prototype. This is a dual dice that I have actually demonstrated during the first couple of lectures.

This is the layout of this, the blue color represents the copper tracks on one side of the printed circuit board and it is actually a single sided board. So, that is the only copper tracks that I have I may need some wires to jump over some components, or wires the tracks, and so they are being represented with these red wires. So, these red wires represent jumper wires, that you would actually solder on your PCB.

So, you create a PCB layout and then you take go to the next point, where is you consider wherein you consider, how are you going to prototype it, you did layout, are you going to build that PC and prototype at that level, it turns out that there are many other alternatives. Instead of coming to the PCB layout option, you could try to prototype in more simpler forms. There are issues with those approaches, but it does not hurt to consider them.

(Refer Slide Time: 20:32)



And the options are that you could use a breadboard, the nice thing about breadboard is that you can quickly put wires on the breadboard, insert components and test your circuit. The disadvantages are that the breadboards necessarily require that the components should have enough spacing and that you cannot use certain types of components such as surface mount devices, or components which are too small.

There are ways around, work around, around it work around, around this problem. But in general breadboard should be resorted when you have no other option, or you want to quickly test some circuit. There are other electrical issues that the tracks, the holes the in which you insert these wires, they are like inductors.

And so there is mutual inductance between these tracks and they may modify, they may alter the electrical characteristics of the circuit that you implement. So, you may not be very sure that if your circuit is not working for some reason is it because the circuit is faulty to begin with, or is it because the breadboard does not allow that circuit to operate at the in the region that you expect it to work. So, you must consider that.

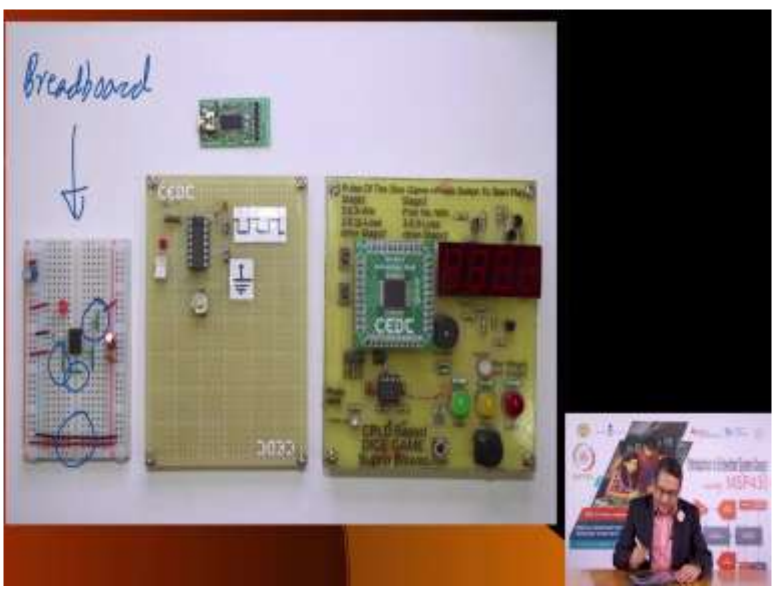
The other option is so this is the first option, option number one that you use a breadboard. The second option is a general purpose zero board meaning a circuit board, in which there are holes and under the circuit board there are pads, copper pads, on which you can solder things. We call them zero board, again this is very similar to the breadboard.

And it may have issues for incorporating SMD components and there are workarounds around that also. And then you could prototype, the third option is this is the one that you can prototype by creating a printed circuit board in the lab. And in fact we have a dedicated session on how to go about building a breadboard circuit, or how to build a general purpose zero board circuit, and then how to prototype printed circuit board in the lab.

It is a quick and dirty method of creating a at least a single sided board and with some little more effort you can even create a double sided board, that means a circuit board in which there are copper tracks on both sides, that is the way most professional circuits are, in fact real profession circuits have several layers. They are what are called as multi layered PCBs I will come to that.

And the most professional method is to you know create this layout and send it for fabrication outside to a PCB fabrication house, which will create that prototypes for you and send it to you. And then here you can order any complexity, meaning whether it is a single sided board, or multi-layer board you can you know order it. It takes more money, it takes more time, but in some, in some cases it may be the only option that you have.

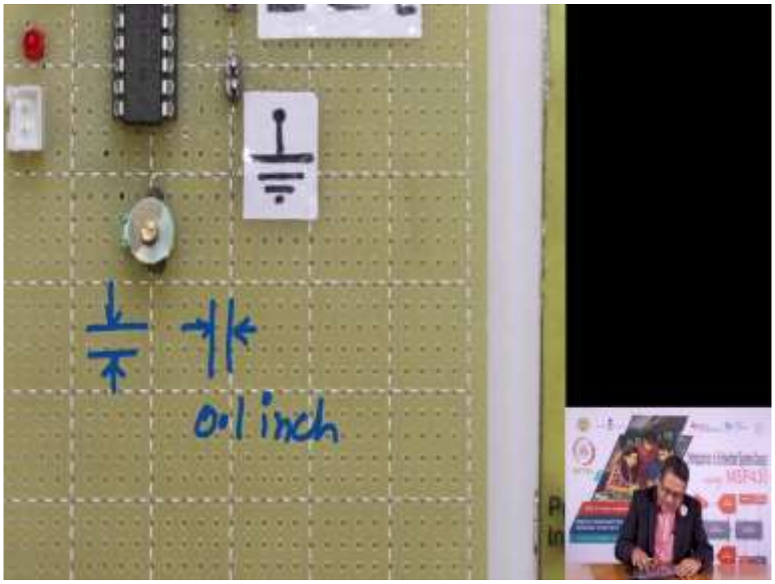
(Refer Slide Time: 23:26)



I am going to show you the 3 methods here, one is how to prototype it in the lab, and we have seen these during our MSP430 lunch box experiments. You see here are these wires which are being used to make connections here green and red, and it is good to use multicolored wires. So, that you can reserve certain colors for certain signals, for example we use red and black wires when we are dealing with power supply and current, and maybe a green wire to indicate other signals.

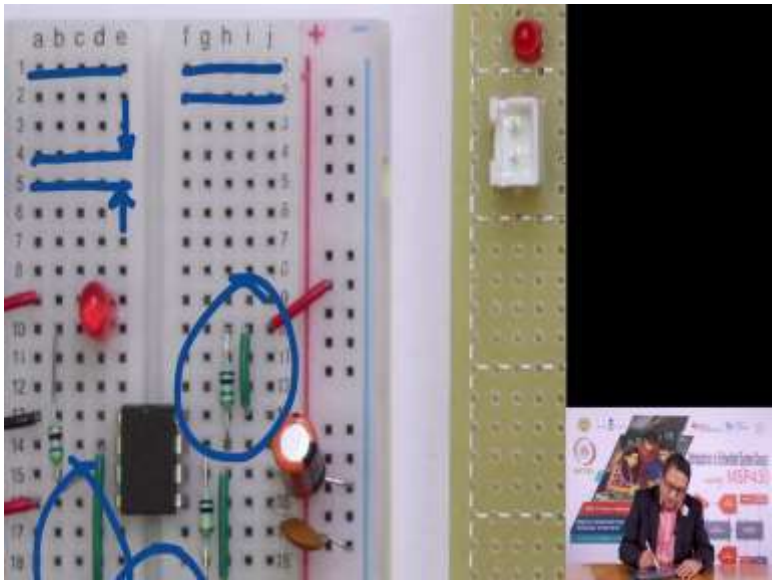
And it is good to layout your circuit comfortably, you should not try to you know pack your circuits very close together. Because they may create short circuits, which may you know reflectance circuit not working, but maybe the circuit is working it is the fabrication which is creating problems. So, make sure that your circuit is well you know there is good spacing. So, this is one method using the breadboard.

(Refer Slide Time: 24:37)



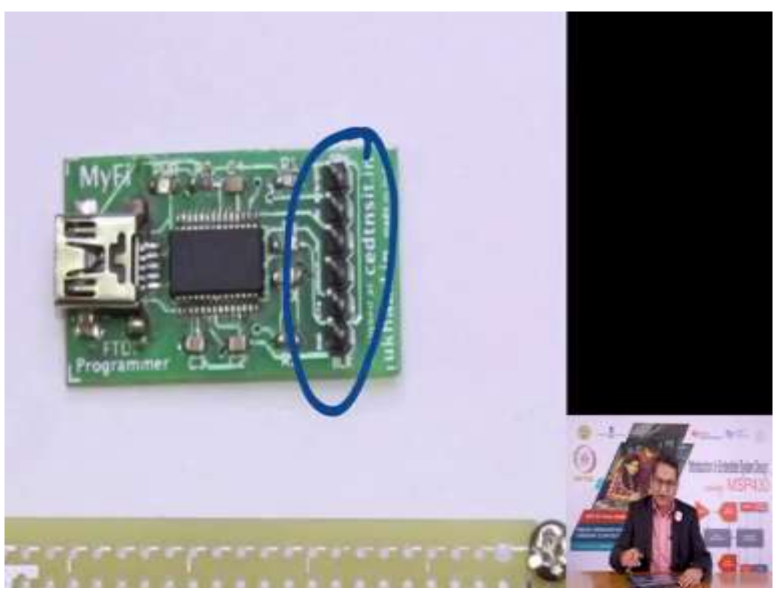
The second method is this is the zero board, and as you see if I zoom this up you see there are holes in both the direction and the spacing of these holes is 0.1 inch in either direction, either here, or in this direction. This spacing is uniform, this is the same spacing that you would get even on a breadboard.

(Refer Slide Time: 24:54)



Here you see, these are all connected together, this is connected together and this separation between these two tracks is 0.1 inch. And similarly you have this and this and so on and so each of these holes are 0.1 inch apart. The nice thing about this is this is closer to the final prototype and as I mentioned in you in some cases you may have SMD components and it may not be possible to solder them on a zero board and certainly not possible on the breadboard.

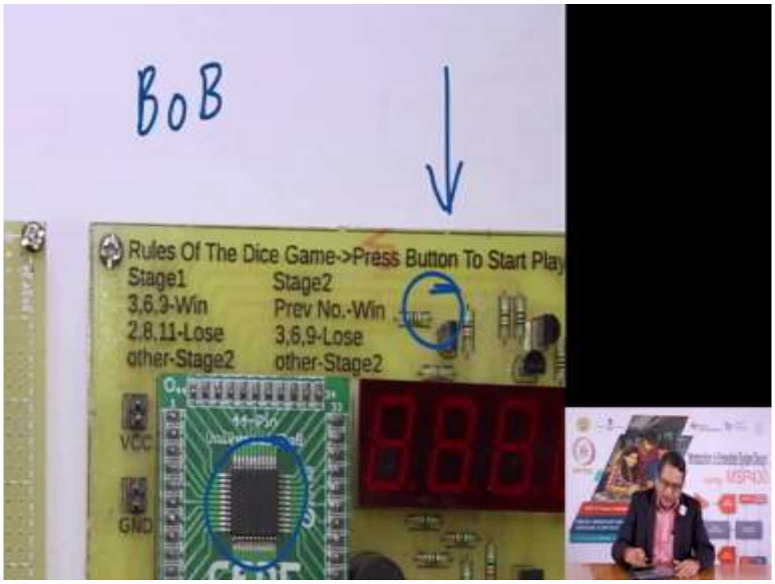
(Refer Slide Time: 25:35)



And so we use this approach called creating a circuit, which is able to where you are able to solder a SMD component and bring the rest of the connections on pins which you can connect to the rest of your circuit. I will this discuss these in more detail and then here is a example of a prototype of a PCB which has been created in my lab.

This is a CPLD base based dice game, which actually I am going to go through in detail in a lecture to be titled you know designing single board, single purpose computers.

(Refer Slide Time: 26:08)



They are being as you see here see this is the point I was making, this I see is far too you know it has far too closely spaced components, closely spaced pins to be prototyped on this or the earlier method. It is also difficult to solder this on a lab based PCB here. So, what we did was we created a, we ordered professional PCB service to create what we call as BOB.

We call it BOB, BOB stands for breakout board, this breakout board allows us to solder, solder commonly used SMD components. And bring out connections on the edges of the PCB, where the spacing is comfortably possible to be used on a PCB, or on the breadboard or on the zero board.

Because the spacing is all 0.1 inch apart, which is the standard for zero board and breadboard. And then the same thing we can put on our own PCB and this is the result of such a exercise. And we are now dealing with the power supply issues for the prototype.

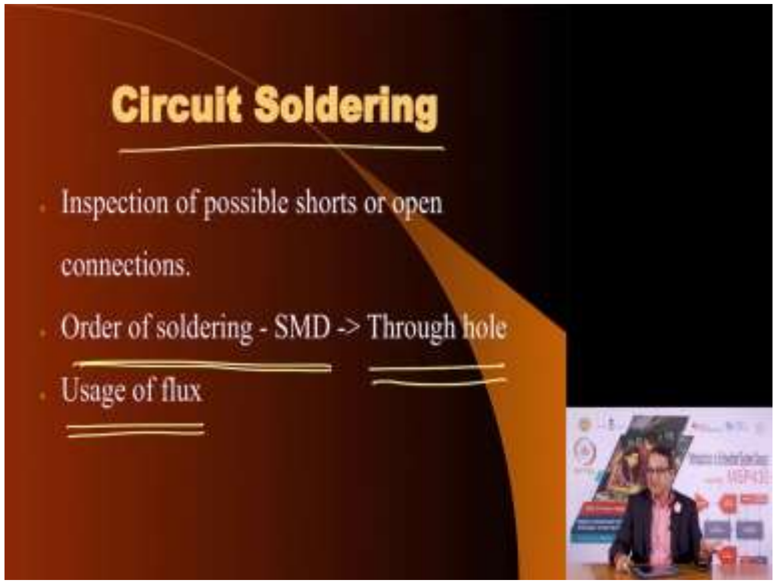
(Refer Slide Time: 27:48)

Now, in the lab you are going to prototype, you must have access to a general purpose power supply, a power supply which satisfies the need for your powering requirement, and you could have what they call as the bench top power supply. But these days you can have adaptors just like your adapters for powering your charging your cell phone. And there are other adapters which will give you varied voltages say 5 volts, or 9 volts, or 12 volts.

These are all SMPS based circuit adaptor, a power supply adapters, you must have access to that, you must be able to measure voltage and current and therefore a digital multimeter is of great help. It may not oftentimes be able to measure current. So, you must find out some alternative method maybe attach a series resistance, low enough resistance.

So that you can measure the voltage drop across it across it to estimate the current that your system is consuming and yes eventually your system must have its own power supply, because you cannot ship a product without a power supply, you must integrate a power supply but to begin with you need not worry about that the testing can happen with the commonly available power supply sources.

(Refer Slide Time: 28:54)



Now, having considered that you would want to solder your circuit, whether it is for the zero board or for your lab made prototype, or for the final version, this does not apply to the prototyping technique using a breadboard. But as I mentioned use that approach only for the very simple circuits and for complex circuits you either take other methods.

And so you would want to solder and it is very important that once you solder you are able to ensure that no wires have shorted or connections which ought to be soldered are not left open. And so you should use appropriate tools for doing that maybe a magnifying glass, these days you get glasses that you can wear and they have LED lights with the magnifying lenses in those glasses.

So, that you can inspect if you can afford it, if you can afford even more you get microscopes with where you can zoom in to a part of the circuit board just to inspect everything in detail you must do that. The ordering of circuit board is very critical, because your circuit may be consisting of SMD components which are really tiny circuit elements, and they occupy they are very close to the PCB.

And so if you solder something which has height then reaching this PCB to solder circuit elements, which are close to the PCB may be difficult to achieve. And so we always recommend that you start by soldering your SMD components, or assembly resistors and ICS which are providing a very low profile.

And once you do that then you can go to soldering through hole components, one should avoid using a flux, flux is an important component. It is a necessary evil sometimes your components may be oxidized and corroded and the flux allows you to remove that oxidized the surface clean it up. So, that it is ready to be soldered, but leaving flux on your PCB is a recipe for disaster.

So, it is very important that if you have used flux you should be able to clean it up. And incidentally the conventional soldering wires that you have they may look like a wire, but they are actually hollow and they oftentimes contain some flux.

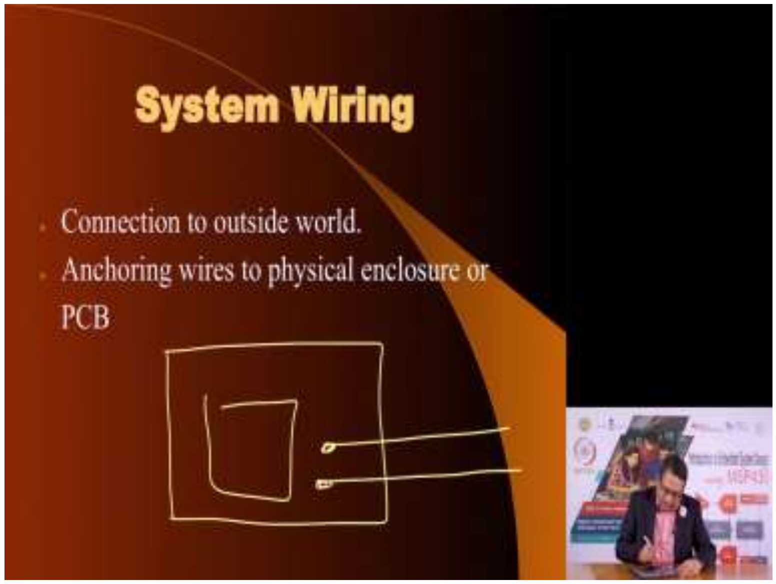
So, that when you melt the solder wire on the components the flux comes into picture and helps in cleaning, which is why when you solder something you see fumes coming out, these fumes are not from anywhere else, but is it is because you, you are melting flux and flux as I mentioned is necessary evil.

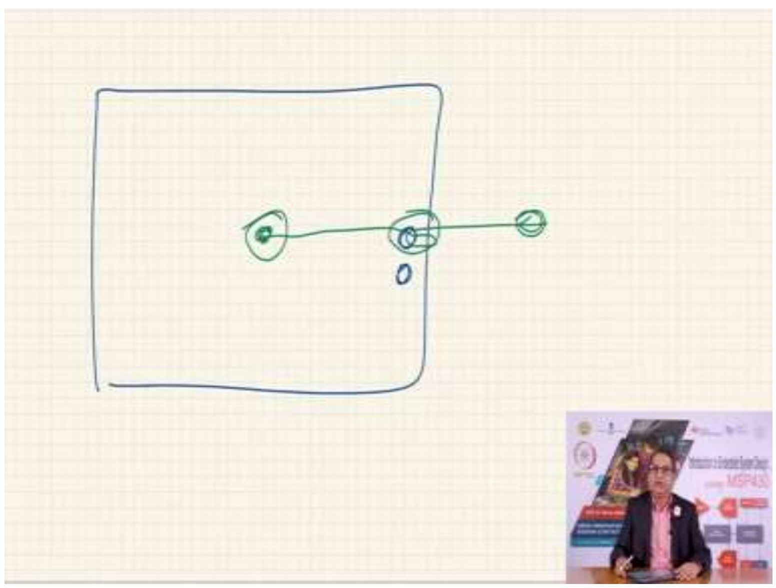
It will help you in soldering well, but you should not leave flux on your circuit board, you must use some mechanism to clean it up, one good method is to use various sorts of cleaning agents and the properties are based on alcohol, various types of alcohols, using a brush and things like that you must remove all traces of that flux. This will ensure that your circuit works correctly, if other things are in place.

Once you have soldered your circuit you must perhaps connect it to the rest of the circuit boards or rest of the intrusion and so on. So, you must find out what is the connection to the outside world and how are you going to take these wires, which connect your circuit board to the outside world.

Now, very simple physics comes into picture, that if you just connect a wire and bring it out and somebody pulls that wire, it may break the connections, or it may you know short some connections on your circuit board. So, you must anchor these wires before they go out of the enclosure.

(Refer Slide Time: 32:18)



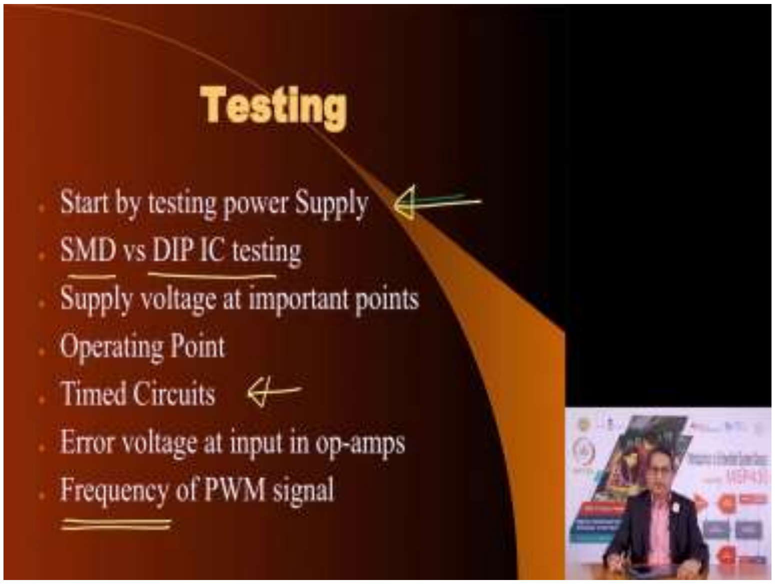


And one way to do that could be that suppose this is your circuit board and here is some component and you are going to solder a wire and bring it out here like this. So, instead of just bringing out like this, you should method of doing it is as follows, that you take a PCB, this is your PCB and maybe it drill a hole here, maybe two holes and when you are taking a wire let me use another color.

So, let us say you solder a wire here and you take it, put it through this and then take it out. And so you anchor this wire in this hole. So, that when you if somehow this wire gets pulled here the force will not be transmitted and conveyed to the solder joint, it will be stopped at this anchor.

And so your circuit will remain safe. And this is a good method of doing that you can open many professional you know products and you see that is how they are doing it. And you must follow that technique.

(Refer Slide Time: 33:58)



Once you have soldered your circuit, you have connected it to external wires, or wires to other of your system which maybe other circuit boards, it is time to test. And one good method is to not use your actual PCB, or system power supply, but to apply power supply from other sources without inserting any IC, you have soldered most of the components that you cannot do without soldering.

But if there are any ICS which need to be inserted in sockets, you can start by not inserting those IC and just applying power. And then use a multimeter to test the voltages at various points and in that it is very helpful that while planning you allow for certain test points. These test points can be you know jumper wires, or they could be you know pins that you solder and the purpose is so that you can probe them, this is very important.

You should also consider how are you going to you know look at SMD versus DIP IC testing. If you are using a DIP IC you can remove it. But if it is a SMD IC that you are going to be using in the circuit board, you cannot do without soldering it. So, that is a compromise that you will have to do, you must measure the supply voltages at various points, what is the operating point voltages of the circuit? If you are using any circuits which have a timing relationship, that is they provide certain timing parameters, you must be able to test it.

And you should be able to find out is are they producing the right frequency waveforms, right frequency, correct frequency, correct time period by connecting a oscilloscope and testing it. And if you are using in your op-amps, these op-amps have what are called as error voltages, you should be able to estimate that before you insert them in the in your circuit.

If you are using any pulse width modulation signals you should be able to measure the frequency of those PWM signals, because the frequency of the PWM signal is very important. These are some suggestions that we have compiled together based on our not too much, but quite a bit of experience that you should consider while testing.

(Refer Slide Time: 35:40)

Now, suppose you are fortunate enough to have a working circuit, the power supply is working and everything is working fine, what would be the next step to obviously put it in some form of enclosure. So, that it can take the next step of looking like a professional product, and there are many options you can use a ready-made enclosure.

I will show you some examples of that, you can even use PCB, which consists of copper on one side and the non-conductive substrate on the other, you can actually solder them together to create an enclosure, again this is good for prototyping. These days with the advent of 3D printing mechanisms, you can actually 3D print enclosure for your prototyping ofcourse you cannot 3D print mass numbers, because 3D printing is a slow process.

But it could it is very good for prototyping, once you are happy with your prototype, you can make sure that large numbers could be already using conventional manufacturing methods. So, 3D printer could be very useful for prototyping you could use paper or cardboard, if you want to take that approach.

This is good and then traditional materials like wood, or plastic could also be used to create enclosures, you could use existing boxes. If you do not have ready-made enclosures these days, there are lots of sizes of boxes that we use for you know carrying food, you could consider using those. And once you have put your circuit inside these boxes, whichever approach you must put identifier marks.

So, that you know if there is a terminal here, if there is a wire entering at this point, what is this wire for? So, that if there is a jack maybe a connector that you have to connect, you should know what that connector is for. So, that you do not by mistake insert a wrong connector, so putting identifier marks on your enclosure is very important.

(Refer Slide Time: 37:47)

Here are some examples on your left is a circuit which has been created in my lab using a CNC machine. This is a sort of plastic and this was one of the projects that I showed you during the demonstration, which is a LED hourglass, the box has been milled out of thick stock of plastic and it has been snap fitted together to create the enclosure.

This is 3D printed enclosure for a color mixer, in the center is actually underneath this is RGB LED, as I showed you again and this white thing is not 3D printed, actually this is a table tennis ball, a ping-pong ball which has been cut in half. So, as to diffuse the LED light underneath it, and this has been created using a 3D printer that we have in our lab.

(Refer Slide Time: 38:43)

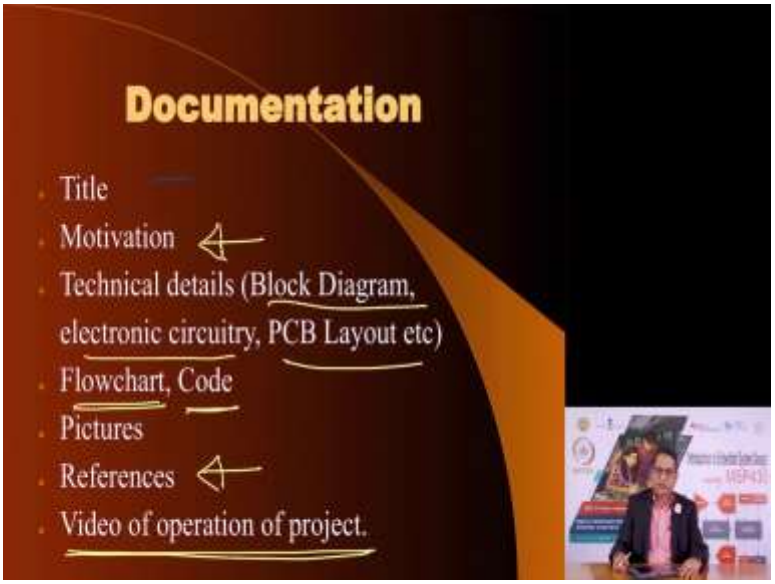
What are other options? You can also create acrylic, you can use acrylic boards and you can cut them using a you know hacksaw, or if you have access to a CNC machine as you see here it has been nicely cut, you see these edges are designed specifically and they have been put together. This is the lab made PCB here and it is sandwiched over top and bottom acrylic sheet, which has been drilled holed.

This is the PC, this is the speaker here and so to allow for the sound to come out of the speaker, here are holes that have been drilled into it. And this example here is a ready-made, this is a ready-made enclosure and you get these enclosures of various shapes and sizes and dimensions and you could have a variety of them available at hand.

So, that you can select a particular enclosure for your application and at the last point I mentioned that you must you know mark your enclosure, this is what has been done these two are voltage controls. So, as you see it has been listed that this will vary the voltage in the fine form and this is the course control, this is the LED for on-off and this is the title of this project that this is a precision voltage regulator for a particular application.

So, this is a good approach of prototyping of course this may not be the final form of your product, but the prototype must be well made in this form. So, that this can act as a reference for the actual product.

(Refer Slide Time: 40:22)



At the end of it in your after your prototyping or while you are prototyping, a documentation is one of the most important elements of any system design. Because this documentation will be a guide to various levels of you know debugging, or trying to fix things, or to create you know papers out of it as in you want to publish something, or you want to share with the rest of the world.

This documentation in great detail you must you know create it and the documentation must follow certain process, procedure, that it must have a title it should also have why did you do this, the motivation, why are you documenting it? What are the technical details of this project in terms of block diagrams, in terms of circuit diagrams, PCB layout? If it is using some embedded controller or computer then what is the kind of flow chart that was used to create the system, may be a record of the code.

And pictures and pictures not just in the final form, but as the project was being was evolving at various stages you should take pictures. And what are the reference that he used was it everything from your own thought process, or did you use any books, or did you use any you know internet websites, all those must be in incorporated in the reference.

This would be helpful to go back and revisit if some things do not work, then you can go and look at those references to find out what was the thought process at the time that you looked at those references. And it would help these days it is very much possible to be able to create a video record of the project in operation. This you could again share it on social media on YouTube, or you could save it archive it for future reference.

(Refer Slide Time: 42:16)

Here is an example of a project built from scratch and you see here the PCB, this side is the this is the dual dice PCB, it has been fabricated in the lab. The other side here, this was fabricated in the lab, but it is basically nothing but a couple of electrolytic capacitors for storing the voltage. And at the center here this is the Faraday based energy converter from mechanical energy into electrical energy.

And this is basically an acrylic tube, the ends of these are sealed with the hot glue ofcourse this is a prototype, and this is not something that you would ship out as a product. But for prototyping it is good. So, that the magnets that you are enclosing do not fly off as you shake it. So, you must have access to all these prototyping tools and techniques. So, that you can create a prototype that works reliably and you know passes before it can converge into a product, if that is your idea.

I thank you for listening to me and I hope all these 10 points that I discussed here would be of help to you, as you take may take great strides from being a student to professional engineers. I hope that you use these techniques in your work, various levels of prototyping as simple circuits going on to projects and products maybe you participate in various competitions, where it may be expected of you to build prototypes.

I hope these points you would consider and use whenever such a such an opportunity arises. I thank you again and I wish you all the best in using these skills and experiencing them in real life thank you.