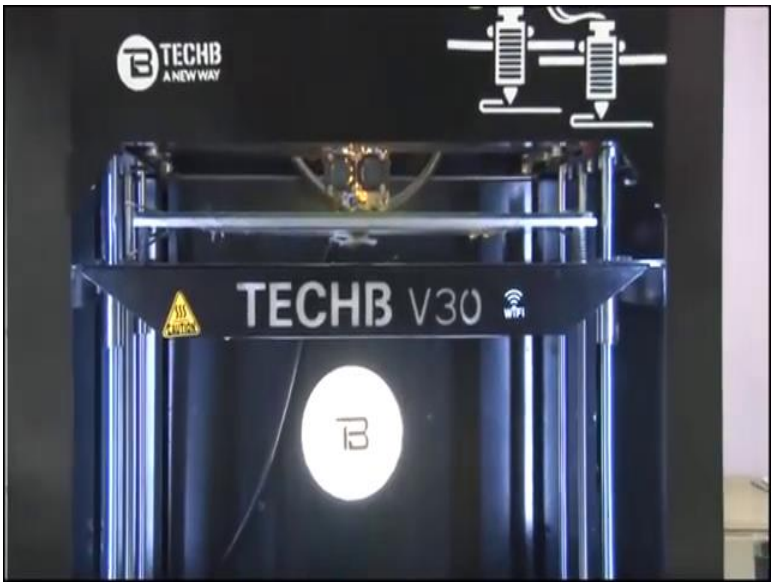
Computer Integrated Manufacturing
Professor J. Ramkumar
Department of Mechanical Engineering, IIT Kanpur
Dr. Amandeep Singh Oberoi
Imagineering Laboratory
Indian Institute of Technology, Kanpur
Lecture 40
Laboratory Demonstration
Rapid Manufacturing (Part 1 of 2)

Good morning. Welcome back to the course. In this lecture, we will take you to the lab for the laboratory demonstration the lab is 4I lab in IIT Kanpur. This is an advanced machining manufacturing lab at IIT, Kanpur which is an elite facility that we have here and like we have seen different rapid prototyping techniques.

In this lab demonstration, we will just discuss this 3D printing, 3D printing using a specific machine that is there in the lab. The name of the machine is TECHB V30. And, our laboratory demonstration the instructor is there who has made this video and we were in the lab we developed this video and I am trying to put my voice to demonstrate it properly. So, let us start this and go to the lab.

(Refer Slide Time: 01:05)



So, good morning. In this week we are trying to study the rapid prototyping and additive manufacturing. So, right now we are in 4I lab, it is a facility in IIT Kanpur, 4I, 4Is are Innovation, Incubation, Implementation, and Integration. In this laboratory we have multiple machines which are non-conventional and advanced machines we do research and, also consultancy is been carried out here.

So, at this point of time, we like to discuss about this machine TECHB V30 we will start with the demonstration of the software that is used for rapid prototyping here. Then we will set up the

machine will do make the whole setup. Then we will see how the prototyping various features, various parameters are taken into account while manufacturing, so this is additive manufacturing as I mentioned earlier.

This is a 3D printer TECHB is one of the companies in India that is making, that is manufacturing the 3D printers. So, this uses FDM technology, FDM as we have discussed, FDM is Fused Deposition Machining method. There are two major kinds of machines in the machine sections as we know now FDM and SLA.

FDM means fused deposition machining and SLA is lithographic operators, okay, I like to put some light. In FDM there is a big variety of colors that are those are available like we can have red, yellow and, white are the major colors also there are certain companies who are manufacturing the colors based upon demand. But in SLA the color variety is not possible, they are certain differences in FDM and SLA this is only FDM machine.

This has 2 spools, 2 nozzles in which we can use two colors, and the multiplier of the bicolor products can be made. Okay about FDM and SLA, FDM machine produce products with the precision but the precision level in SLA is higher, the higher resolution objects are more possible to produce in SLA because resin is there, the resolution is primarily determined by the optical spot size, either of the laser or the projector, and that is really small in SLA.

In FDM, it is a printer, the printer resolution is a factor of a nozzle size and the precision of the extruder movements that extruder we will discuss in this demonstration as well. The precision and smoothness of the printed models is also influenced by the other factors such as bonding force between the layers, is it lower or is it adequate, then the weight of the upper layers that squeeze upon the lower layers.

The number of printing problems like warping, misalignment, these printing defects might be there, shrinking, shifting of layers, all those things could be there. And there is a difference in post processing as well. In general, FDM requires no or very little post processing. Because the products are generally produced to the final shape, only the thing is that the support that is provided, the support material, or the brim or raft that we will discuss had to be moved.

In SLA post processing level is quite higher because it is made of resin and all the extra material that is there has to be removed and it needs to be taken out from the box where it is made. So there is a big difference. So, here we are more focused on FDM, fused deposition method, so this machine that is FDM machine deposition method of deposition machining. This machine can use two kinds of filaments.

However there are multiple filaments like ABS, then PLA, it can use ABS, PLA, and ABS is Acrylonitrile Butadiene Styrene as we know, and it is oil based plastic. It is strong, sturdy material, that is widely used these days like Lego building blocks, Lego toys are made of this ABS to put an example and PLA, is one of another material.

It is polylactic acid, it is made of organic material specifically from the corn starch or sugar cane. It makes the material both easier and safer to use while giving it a smoother and shiny appearance. So it has more aesthetically appealing products which are made up out of it. Now this PLA thermoplastic is also more pleasant on nose, like it smells lesser, like as the sugary material smells slightly sweet when heated opposed to the harsh materials associated with ABS.

When we heat ABS, the smell is quite annoying. In PLA this is not the case. So this means that the printing using these parts is more friendly for the operator. And also PLA seems like a better overall choice because it features follow melting points as well. The melting point of PLA is quiet lower than ABS.

So I would not move into that track now, so, let us discuss about this machine. This machine is basically using SBA FDM technology the raw material moves from the tube and comes up to the head to the nozzle around which heaters are there, and when we switch on the machine, the heating systems gets on. When we switch on the machine, the heating systems gets on there are two nozzles here, nozzle A and nozzle B we can see.

So, the temperature when you switch on the machine, the temperature rises and it goes up to 200 to 230 degrees centigrade. At this temperature the raw material kept in the form of wire comes out near the nozzle, it comes out through the nozzle. So, it is coming through this tube we will show this demonstration.

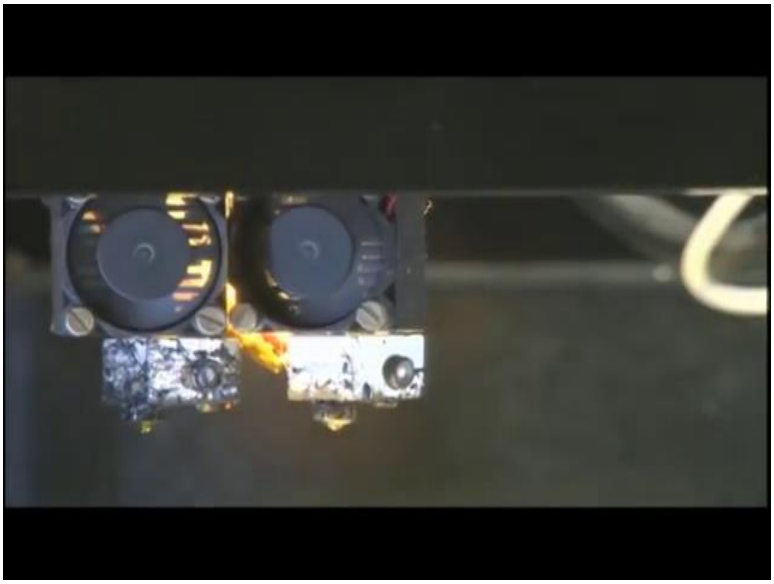
Now, this portion is known as head where the nozzles are attached. So, the material is fused and comes out from these nozzles, we have two nozzles here, nozzle A nozzle B. Why two nozzles are there? We just explained, one nozzle is for the base material, another is for support. So, this is a very fine orifice off the nozzle is 0.4 mm, 0.4mm when all this bed area maintain the soaking temperature of the bed or 50 to 65 degree 50 to 65 degree temperature is maintained here the temperature it heats up a little bit.

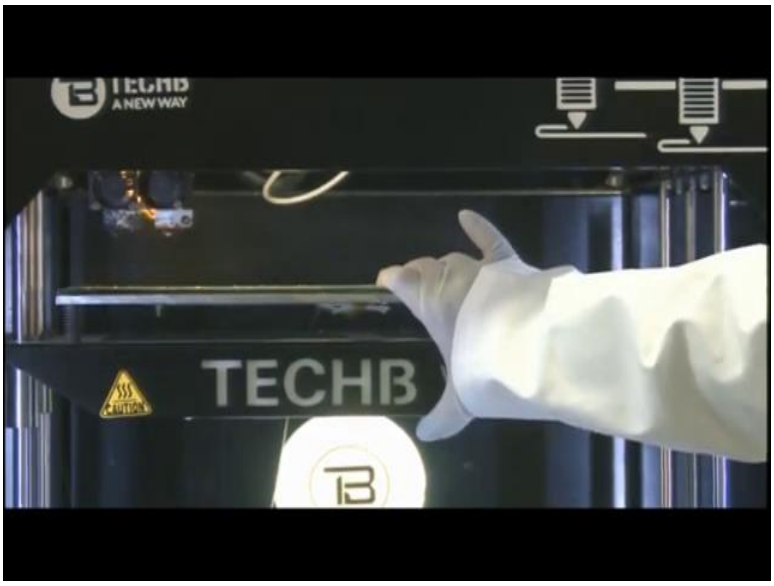
So, it is actually the preparation of the table it is fitted with some specific temperature so, that the material that is deposited here, the temperature of the material is about 200 degrees, this temperature is kept about 50 degrees so that it can stick easily so that it sets easily here. So, this will have to deposit one layer earlier upon another during fabrication, manufacturing.

Whatever we want to fabricate in FDM technology the machine size is, this machine size is 1 feet by 1 feet by feet. So in X Y Z directions, the envelope is 1 cubic feet I can say, 1 cubic feet envelope is there. So, it is just plug and play machine once we have a little practice on this.

(Refer Slide Time: 08:16)


TECHB





Along with the hardware we have a software here that also gets on when you switch on the machine. This is the software when you switch on here comes TECHB. So TECHB is a machines the software switching on this TECHB software is getting as you can see slicer, slicer 1 and slicer 2, the two slices are there okay. So, the CPU and controllers which are here at the bottom surface of the machine, CPU, and controllers for this software is here.

So the two nozzles, one for model and one for support, one for model and one for support, two nozzles are there, so if any profile or any geometry we will find this taper is more than 45 degree. We need to provide support like I discussed then the 2 nozzles will automatically activate and that will provide the support to produce the final model that we need to obtain.

So, this all feedback is given to the head with the help of software that will be described later on. So, precautions, while we use this machine, are certain precautions when we try to switch on this machine the nozzle is quite hot, so, we cannot touch the nozzle with bare hands. Also, we need to place this machine is in proportion we need to place this machine in room temperature air conditioner are not recommended because humidity or moisture would hinder the quality of the product.

And we also need to avoid mishandling or prevent the machine from any rough handling and the nozzle cleaning is very important, nozzle cleaning has to be done before machining but hot nozzle should not be cleaned with hands that is very important here. So there are certain sharp edges in

the machine as well, sharp edges like in the nozzle head or certain the sharp edges are there, we need to be careful that we do not touch these sharp edges and, be careful in this is our safety precautions when we start the machine.

Next, with this machine, we can manufacture parts with sharp corners, any areas, any radius, and the profiles with pocket, the profiles which are empty inside or is hollow. And one more feature of this machine is that we can print two different color models from these two nozzles. Nozzle A and nozzle B we have two different colors can be obtained like yellow and red, different colors can be obtained and those can be produced as well.

So, we have this computer screen, we have a software called repertoire. So we have slicer 1 and slicer 2 here. So if the machine is with two heads with both hands will individually work. In this machine, there is a single head if the two heads are there, the machine though both heads can work independently.

So, you know, there is single head here, single head on the single head here, two nozzles that are attached. So, that is why only slicer one is visible here. So, the software starting on, repertoire software is starting on, repertoire host is a specific name of the software 6.2 is a version then we need to connect the software to hardware.

Now both are interfaced with each other when we start machining that is try to fabricate anything on 3D printing. We have to take care of number of things like cleaning of bed, cleaning of head nozzles, and proper cooling as well. For cooling we have to check that here two fans, two fans are just above the head that should be running.

So whenever we start the fabrication of this machine, we have to take care of few things like proper cleaning, flatness of the bed, and we can adjust a level of the bed with the four screws, you can see this screw 1, 2, 3 and 4, four screws are there. So, these four screws have been loaded here using spring mechanism.

So when we fabricate a model of big size, the height will increase, height increases and sometimes it will vibrate due to the height and gapping would vary. So, that is why the spring loaded mechanism would help to adjust the movement in a nozzle. So, you can see there is a spring mechanism here. So, that helps to adjust the movement.

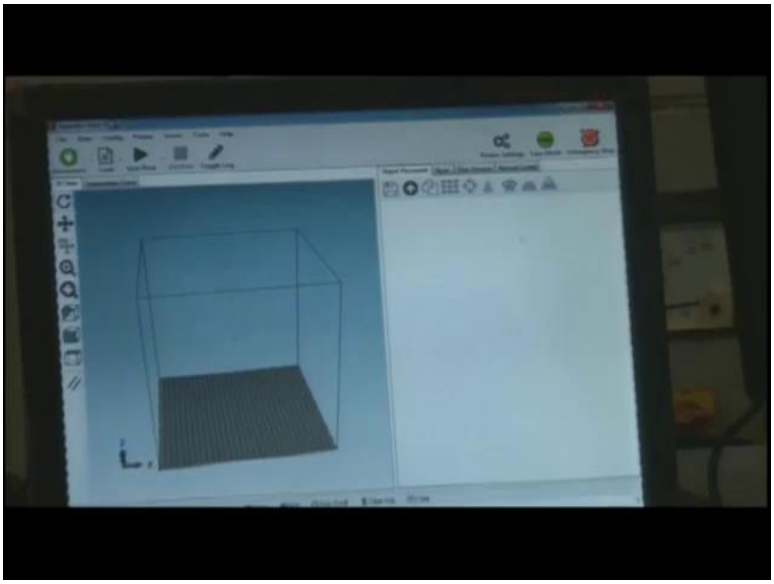
So, in this way we will make the leveling of the bed, also when we go whenever we go for calibration, so, we use these screws to loosen and tighten and to make sure that the bed is flat. It is secure with its adjacent surface as well. So in this software, we can deposit layer thickness in slicing from 0.1, 0.15, and 0.2.

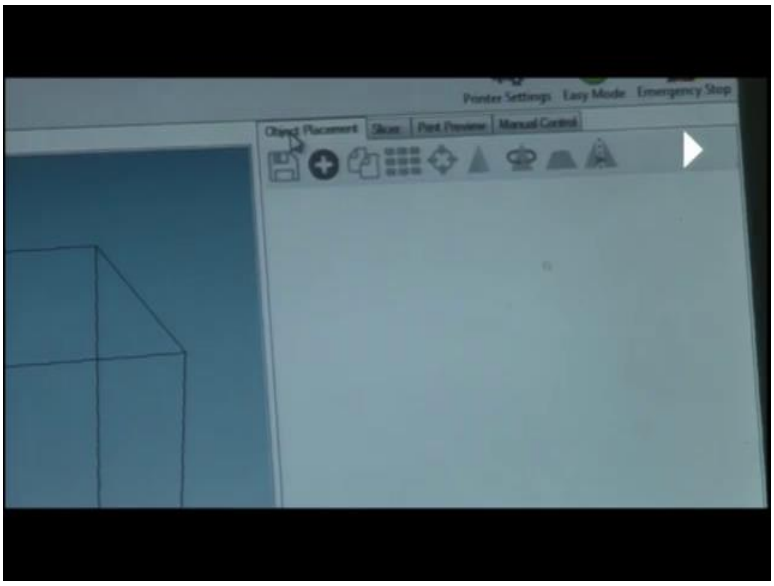
The 3 levels here, whenever we change this parameter of deposition of layer height well then we need to maintain the gap as well. So say let me say a fixed gap, 0.15 from the top of the bed to the tip of the nozzle. So we need to set this cap and lock the limits switch, the limit switch is on this side. So we can lock the limit of the bed using this layout.

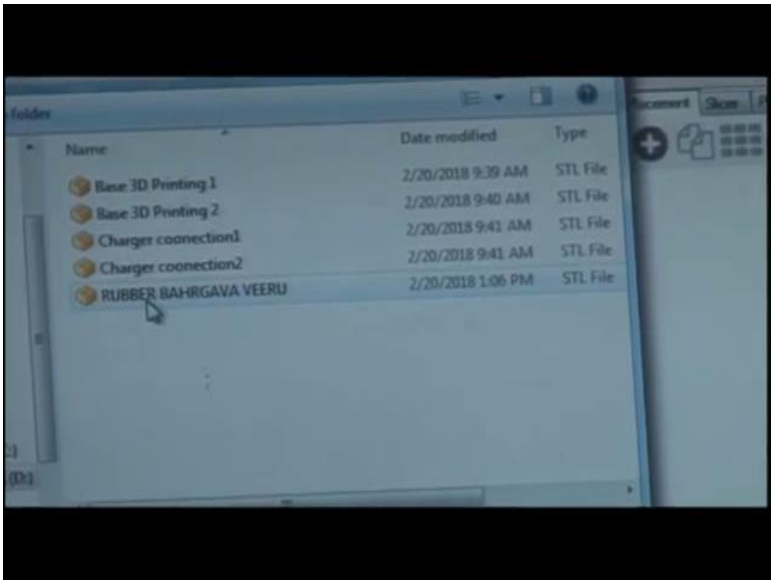
So the bed can move up to a specific limit only, the bed would go up or down as part of the locking that we make here. Also, whenever we start depositing fabrication of raw material, we have to take care of a few things like what are you feeding here, Because the raw material is in the form of a spool, it is an open spool kept at the back of the machine is there a spool we will just show you.

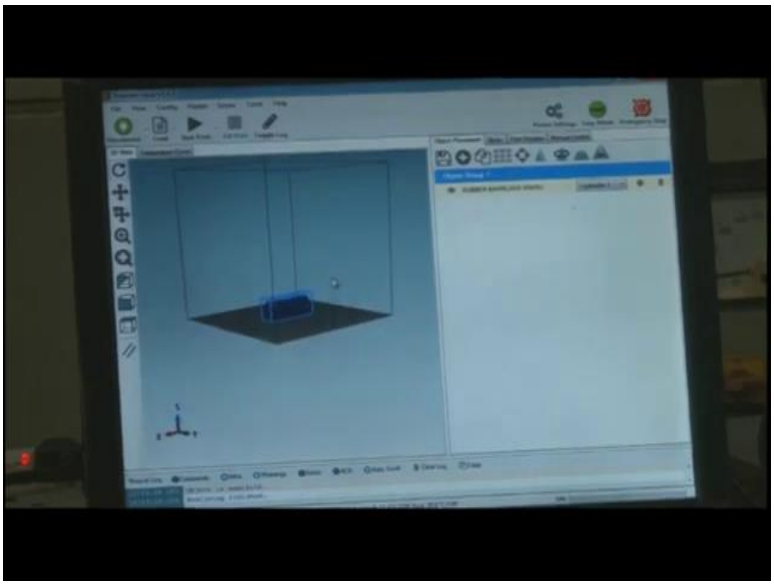
That we need to take care of it because when we keep this when we attach this material in the boxes, we have to use silica and calcium and this makes it moisture free, if our raw material that is what as I mentioned, ABS or PLA if it is moisturized then it obstructs the proper deposition of the job here.

(Refer Slide Time: 14:27)




Printer Settings
Easy Mode
Emergency Stop
Object Placement
Slicer
Print Preview
Manual Control


Name
Date modified
Type
Base 3D Printing 1
2/20/2018 9:39 AM
STL File
Base 3D Printing 2
2/20/2018 9:40 AM
STL File
Charger coonection1
2/20/2018 9:41 AM
STL File
Charger coonection2
2/20/2018 9:41 AM
STL File
RUBBER BAHRGAVA VEERU
2/20/2018 1:06 PM
STL File



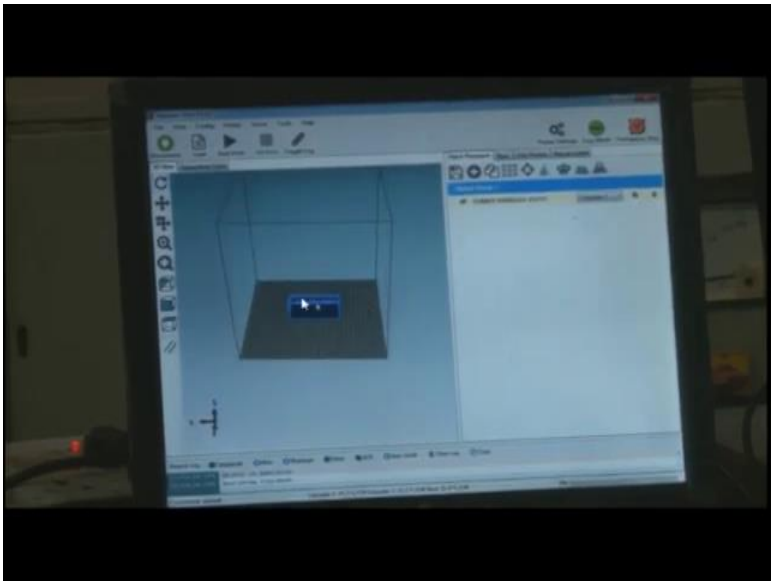
Whenever we start we input the data to the machine for fabrication. After switching on the machine, we have to take care of few things on the display here. You can see this display because the hardware that is connected to the software and displayed to this screen. On the screen, we have program we start feeding the data to the hardware we have to take care of number of things after switching on the machine. After switching on, the machine itself gets connected to its software, whenever we start we have to go to option number one that is object placement.

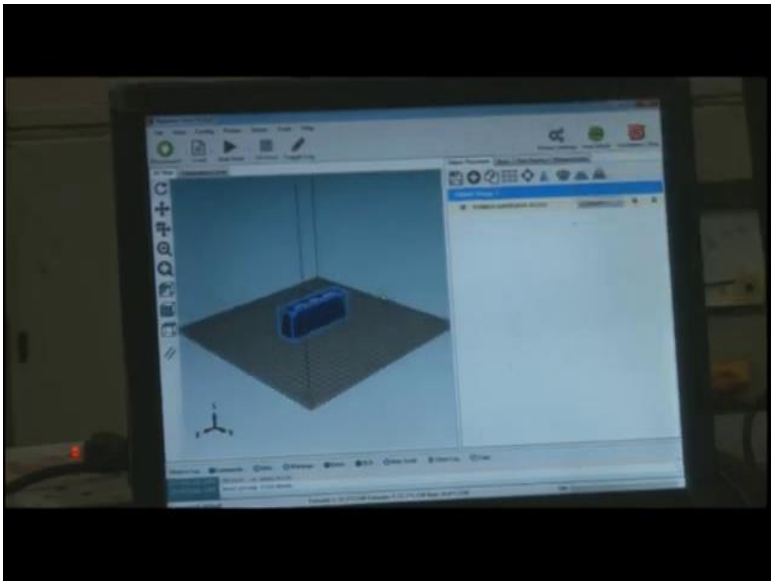
Which means that what we need to print will place it here first, the placement of the object here. So, when we click on it when we click here few things will be going highlight few icons are highlighted ad object, this plus sign is add object. So what we want to add, we click add object here. And when we click it, it opens the drive or the any location in the computer like the C or D drive where the object is kept.

So wherever we have kept a CAD model for fabrication, whatever we want to fabricate, we can just go and click there or we can even generate some model. So this model known as RUBBER BAHRGAVA VEERU this model is there, so we will click here. So yes, when we click on this job is there on the envelope, what is envelope? Envelope is this where we are going to manufacture this thing.

So, this is one cubic feet that is 1 feet by 1 feet by 1 feet. The envelope size is this one. Okay. So this area is 1 feet by 1 feet, I can say that on the area of 1 feet by 1 feet or, 1 square feet, this object would be manufacture, okay. So, actually this is our base plate where it is manufactured.

(Refer Slide Time: 16:26)



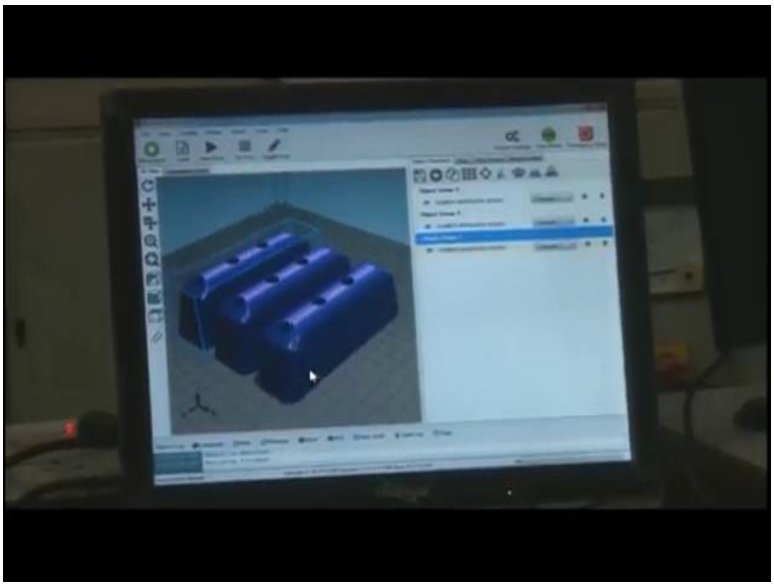


This is the envelope that is being displayed in the software. The approximate size of this model is 4 inches by 1 inch by 2 inches. So whenever we start fabricating, we need to see the program we have to program it, either a program would be generated by machines, by the software itself or we

have to manually program using G codes that we will discuss. So this job of this height of the entire job we will cut in number of slices here, number of slices, okay.

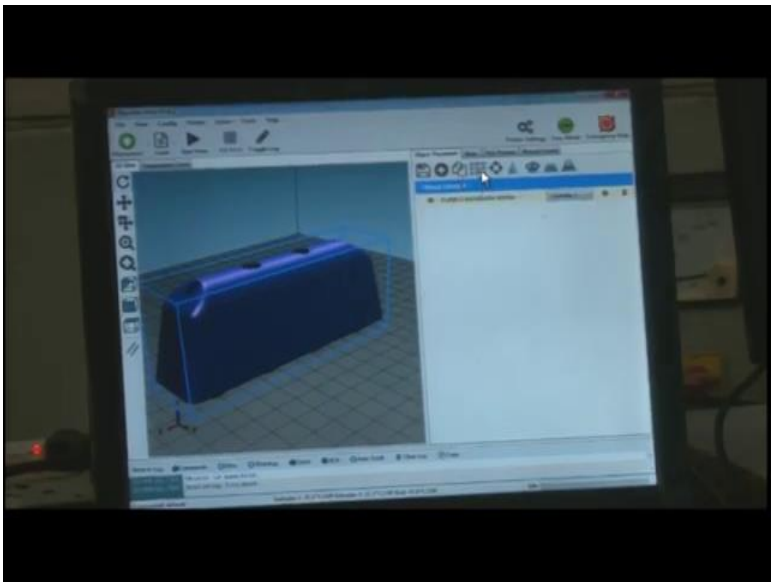
How many layers would be there that the machine would just decide? So we have to program in terms of slicing, and the machine will deposit one slice in one stroke and this will be deposited layer by layer, and finally becomes a complete job.

(Refer Slide Time: 17:21)



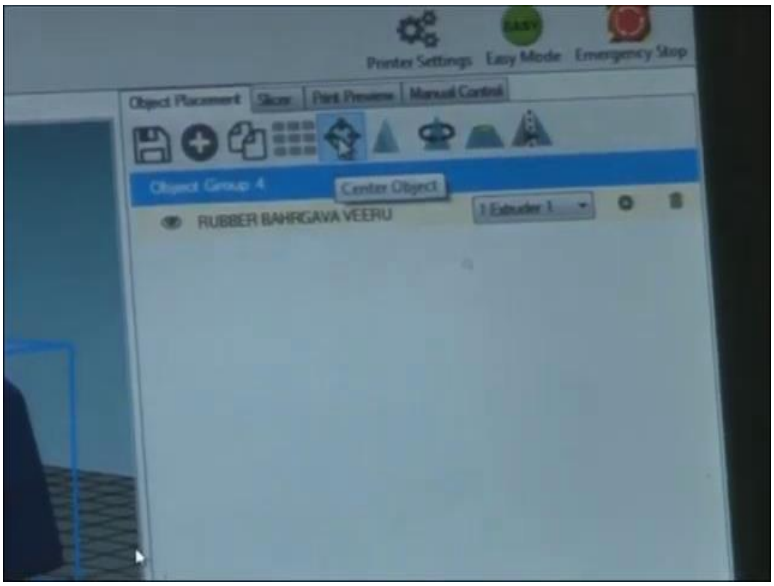
Here in object placement other icons like copy object, copy is like we can make 1 or 2 or 3 copies like I am making 2 copies, 3 copies are made of this object. So single CAD model can be printed in multiple models using this command with this copy, this is design, CAD, computer aided design. Yeah, this is auto positioning, what is auto positioning?

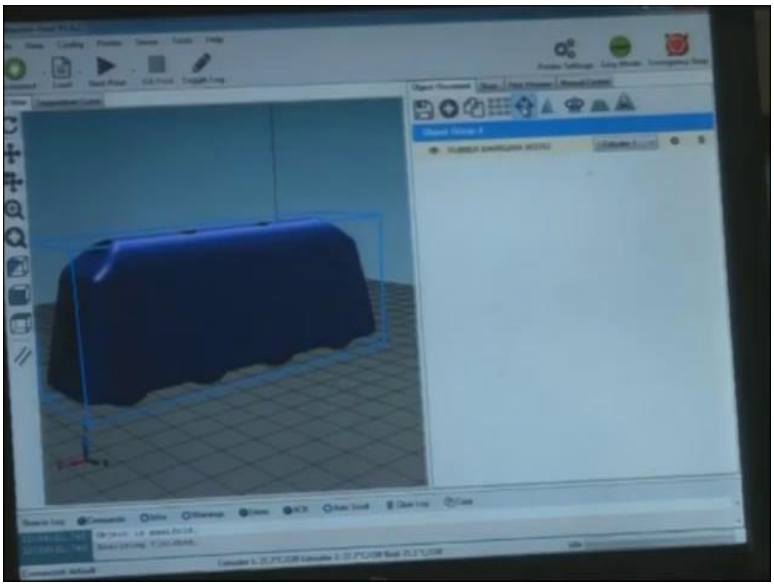
(Refer Slide Time: 17:48)



So when we need to print at the centre, auto positioning will bring the object at exact center in the envelope. So we need not to drag it like we were dragging before we need not to drag it. Next one is center object, auto positioning, auto positioning it will position the object so as the least material is used.

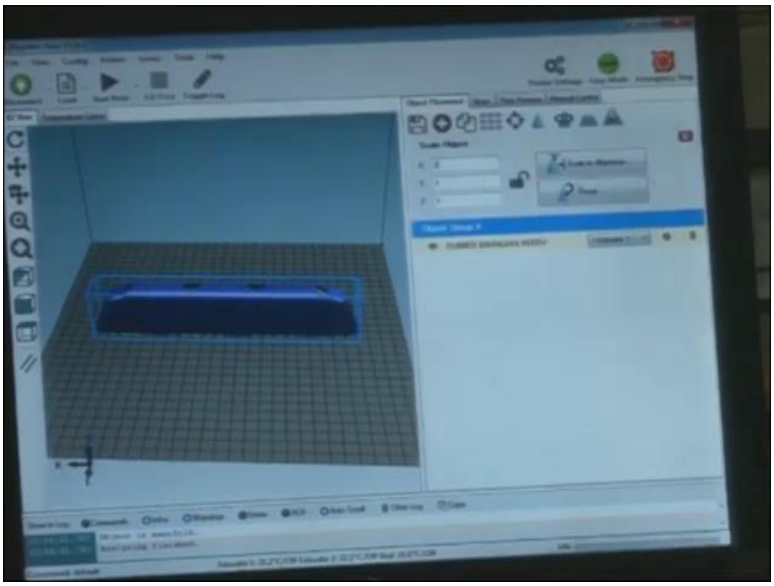
(Refer Slide Time: 18:18)

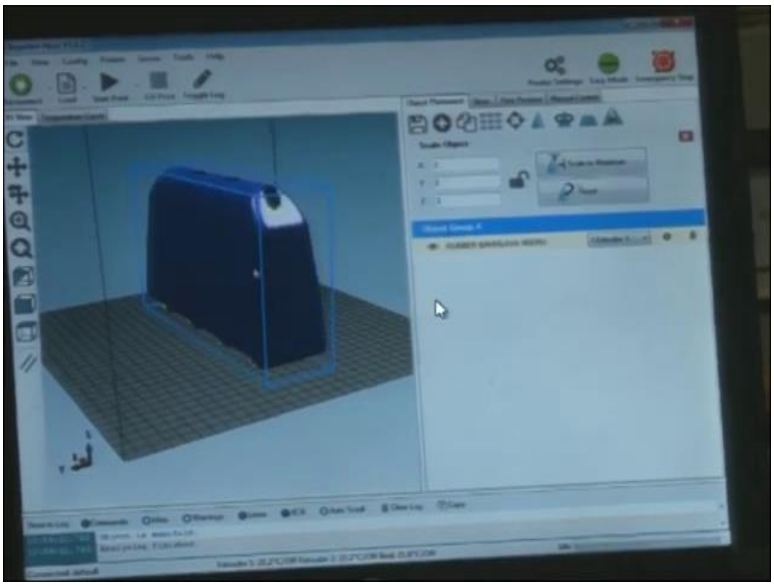


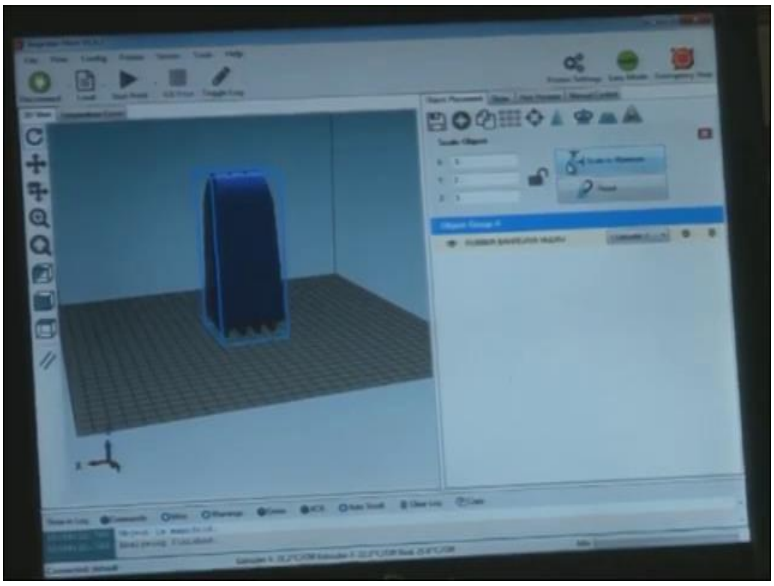


Center object is similar to auto positioning and auto positioning what happens, this is auto positioning. In auto positioning what happens, it places the object so as a minimum material is used. So, as the material has to travel the minimum path and in auto centering objects would be placed at the center of the envelope exactly at the center like we were dragging like might have offset the object while dragging manually, but this will be created exactly in the center. Now, yeah, this is scale object.

(Refer Slide Time: 18:53)



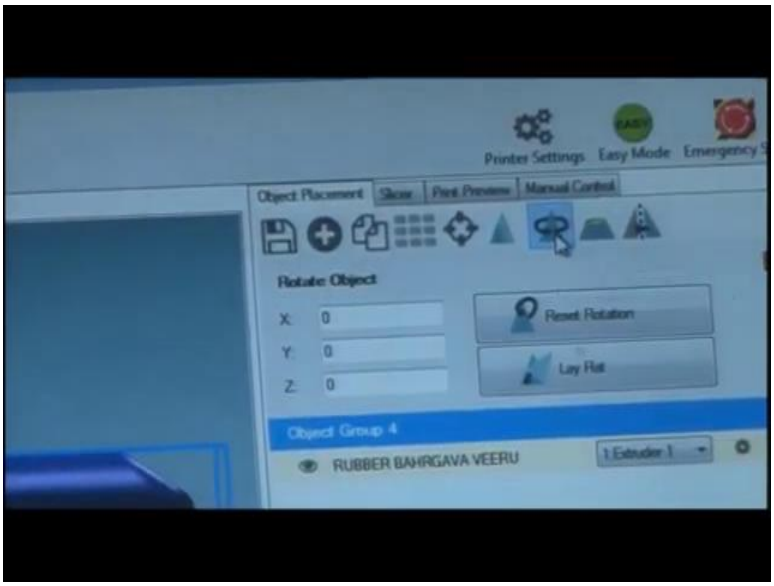


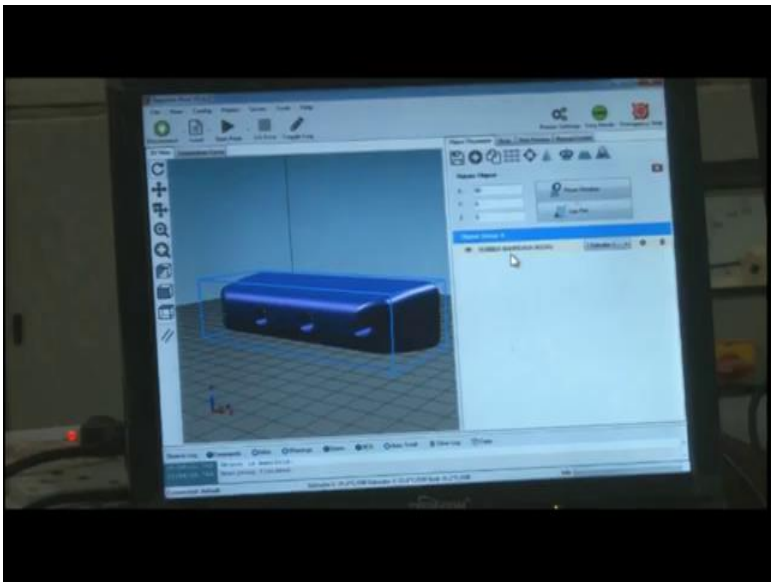


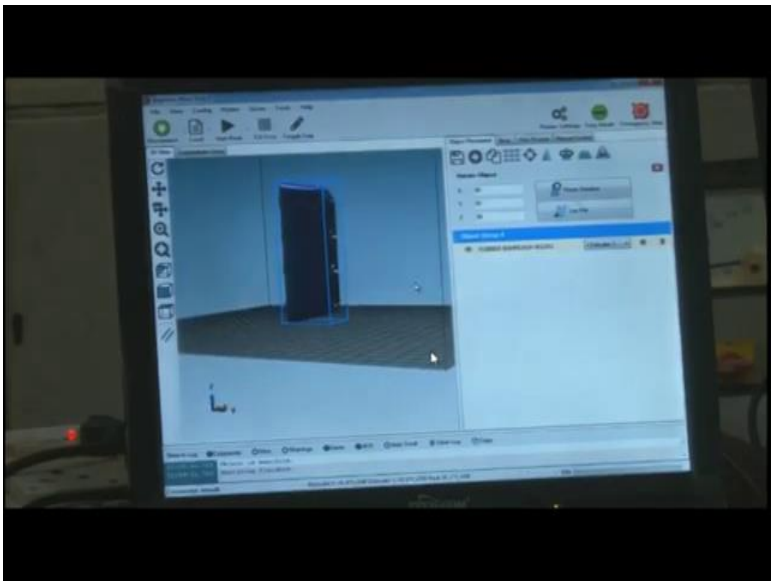
Now, here scaling means changing the size of the object, making it oversized, undersize or, upscale and downscale a better word. You can say x is made twice, x-direction made twice now, y is made again twice, thrice we are scaling the object so upscaling this one. We can see it is midpoint 5. So, this is scaling of the object, sometimes the CAD model we draw is big and we just need to produce a prototype or just a full model out of that, then we can just downscale it and produce.

So, this is a direct provision for that, this is a direct feature in the software for this one. So, next icon here is rotate object.

(Refer Slide Time: 19:48)





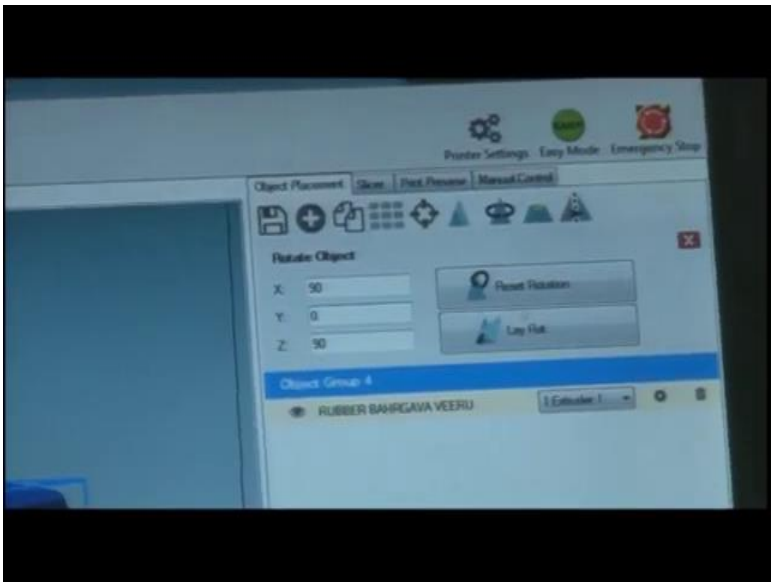


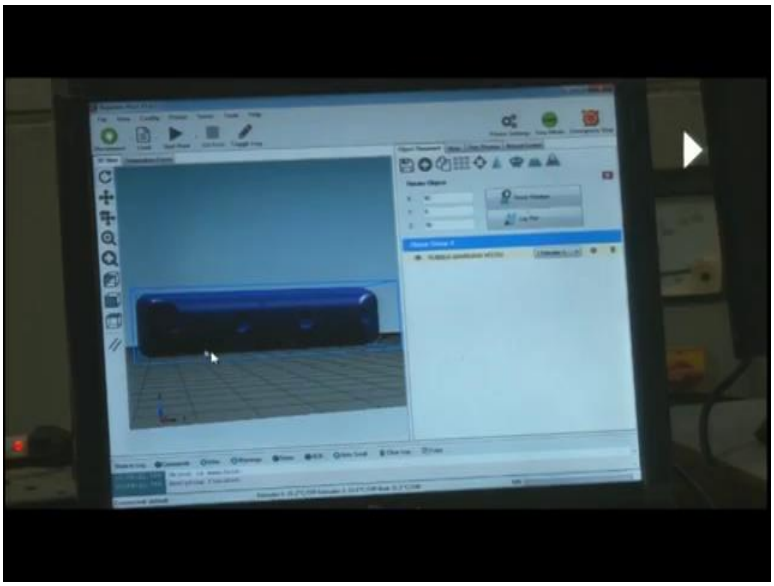
Sometimes, like I discussed like we have discussed that the placement of the object or the alignment of the object is very important in deciding the support material that is used, and the amount of support material that is used.

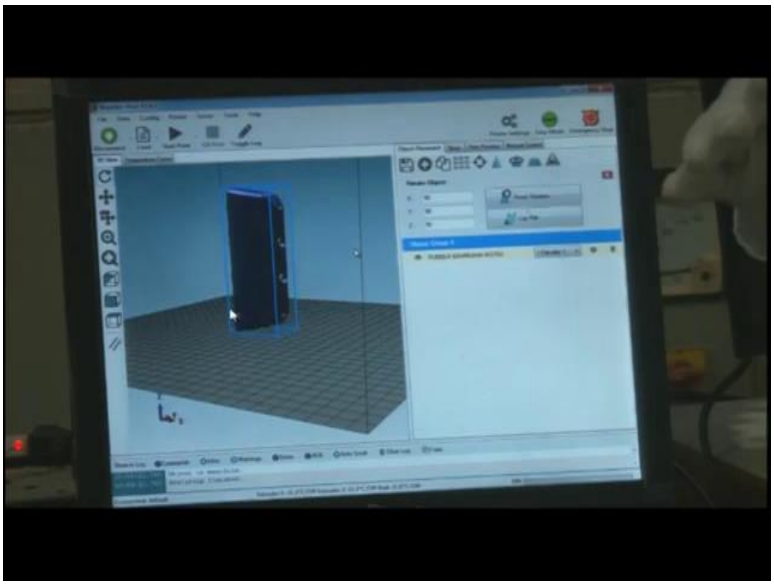
So, to properly align the job like in x-direction, we have aligned it to 90 degrees. So, this is rotation. So, we can make the job in this position as well like if we see that how much material would be used lesser material will be used. So, support material that would be consumed would be lesser, can be reduced by deciding, or by selecting the specific alignment here.

So, sometimes we have the pure flat surface, if the flat surface it is the perfect thing the flat surface you will deposit properly on the flat surfaces deposition is exactly perfect, but you can see that we have an angle here. In this, we have a little angle here, like we need to have some support here. So, for that, we need to see what should be the proper alignment so as this support is minimum.

(Refer Slide Time: 21:02)





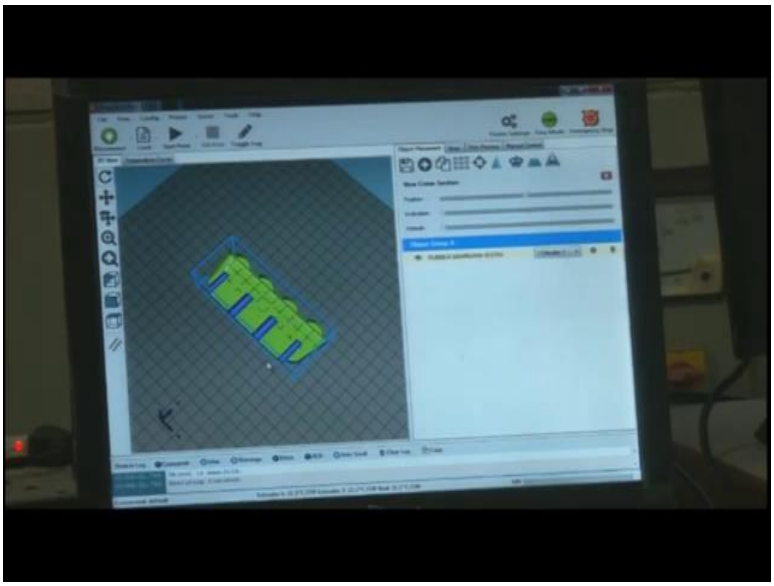


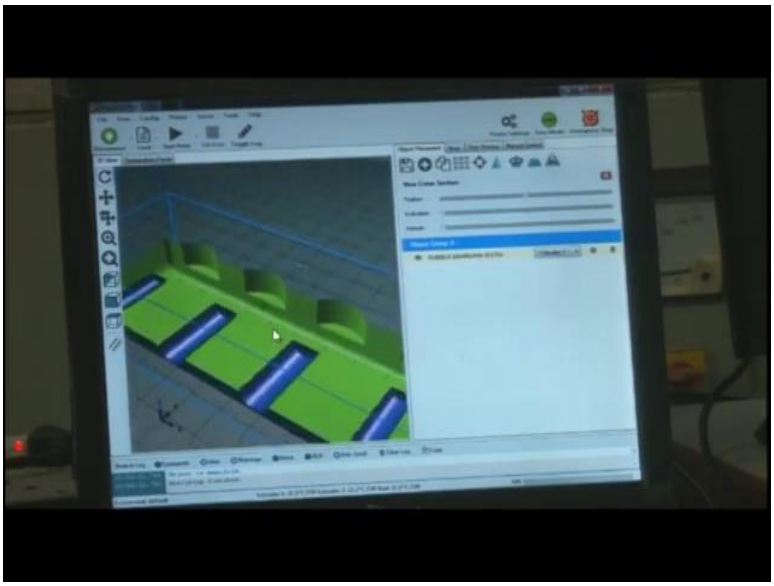
So, we rotate and try to keep the CAD, CAD is meant about this model computer aided design model we have I will use keep on using the word CAD for this model, this CAD is kept in proper alignment so as to minimize material use the y is kept 0, x 90, z 90. So, if y is kept again 90, so the height increases, what is the drawback when we increase height?

When we increase height, because it has been affecting layer by layer, this is very important to note, if we increase the height, the total time would increase. So, the height has to be optimized, it has to be minimum. You know, this many number of layers are to be deposited, so the total time would increase, so the total time, total economy, total cost of the product will also increase because, in a product cost function, the machining time is one of its factor.

So, this is not a very good position to manufacture here. So, we can opt to select the in-flat position, and we orient the job in proper place and position. So, next is view cross section.

(Refer Slide Time: 22:36)





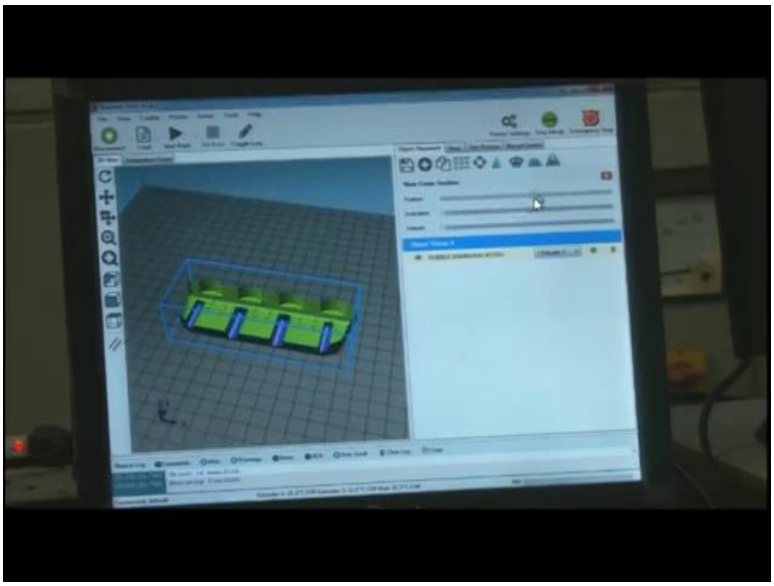
The icon is view cross section, sometimes we need to see what is, what is in the inside, inside portion of the job. So, when we need to see the inside of the job for example, we have called this CAD model from the customer, a customer has just made this model brought this model for us. So, we are not aware that what is the profile inside? Are there any holes, some pockets, some counter sinkholes?

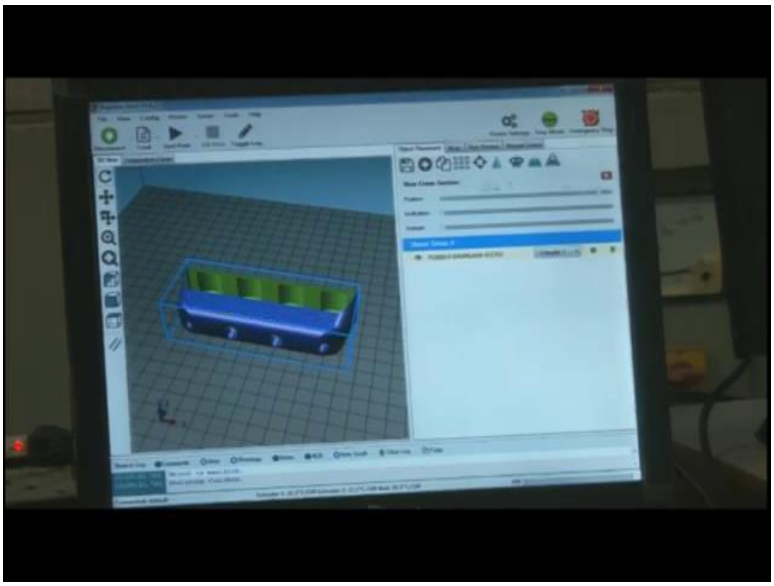
So, if the customer is not able to describe it properly the manufacturer or the operator has to be very careful it is a duty of the operator to be conversant with what is going to manufacture. Then

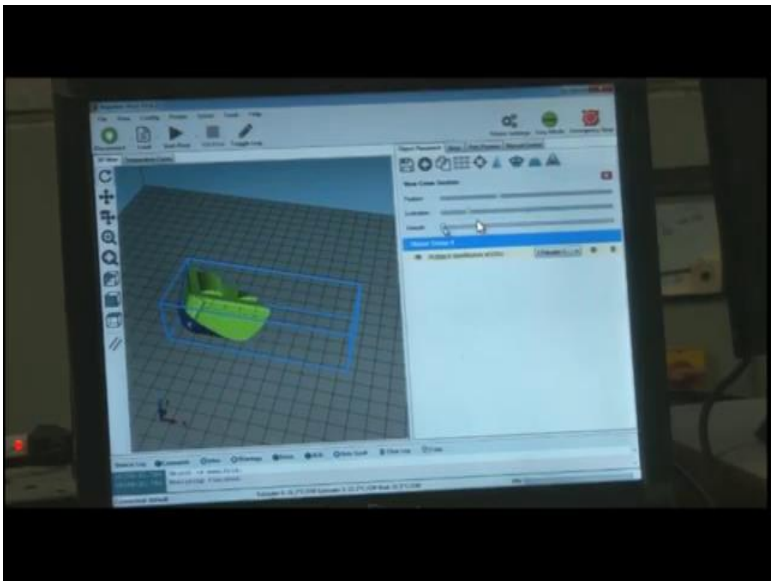
the manufacturer has to take care of everything. We use this icon to check it in the section view, you can see, so blue is upper surface, blue color is upper surface, and green color is inside what is inside.

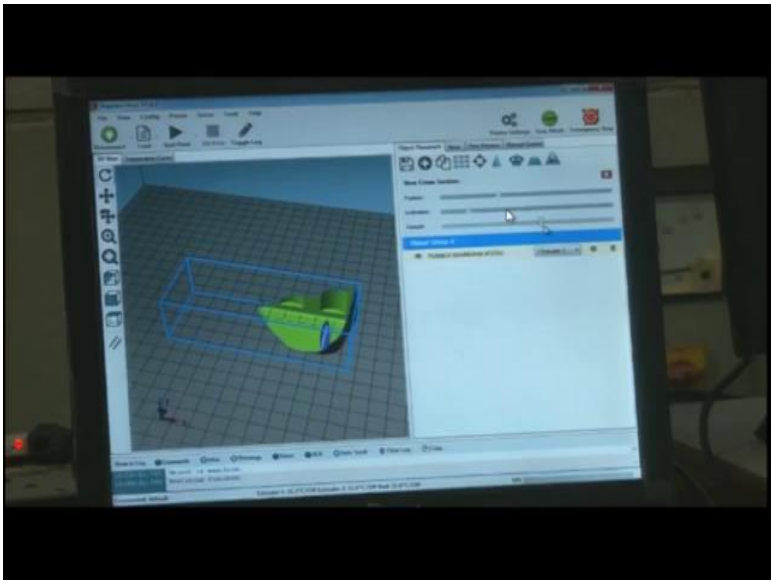
So, this is the color difference, color coding is here. So blue is the upper surface. Inside it is a solid model it is not hollow from inside. This is the inside model.

(Refer Slide Time: 23:53)









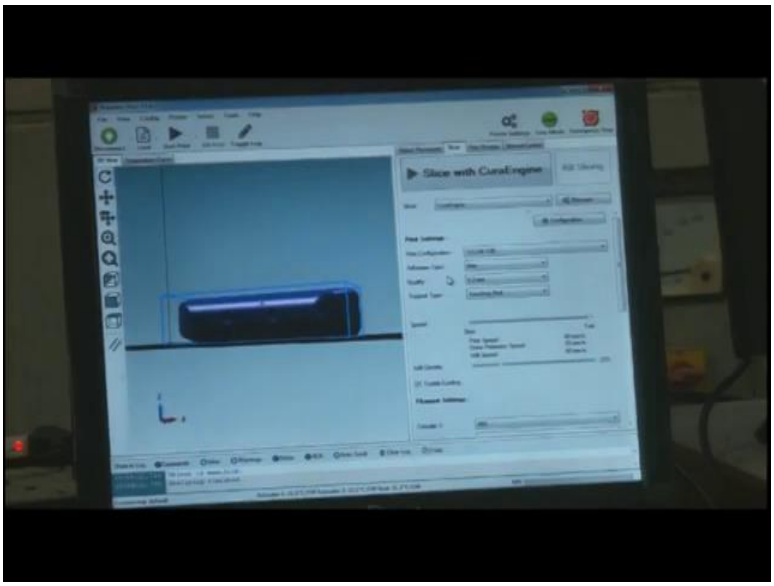
So we can just in view cross section, tab, we have this bar position, we are checking the position from here. So, major purpose of this is to identify whether the model is solid or hollow. If it is hollow, then we have to be careful about the thickness, what has to be thickness of the surface, the thickness of the shell has to be according to the requirement, it has to be according to rigidity what are our requirements we need, so, this is again his inclination, inclination at some angle also, you can see the angle is changing this 0 angle, this angle is about 90 degree here.

So, this is azimuth, so you can rotate it from the bottom here, we can see another viewpoint from this. So, in this way using these deep three tabs, we can see each detail, for instance, there are

multiple features, deep features or the intricate features in this job, we can see that what is there inside the job, provided the CAD model has all those features. So, this is the actual part of now.

Now, the last one is very common thing, mirror object. This is mirror object is we can change from left to right or right to left. So, now, what we identified that the placing of the CAD has to be proper only then we can go to less level it is slicing, okay. So, next tab is slicing.

(Refer Slide Time: 25:42)

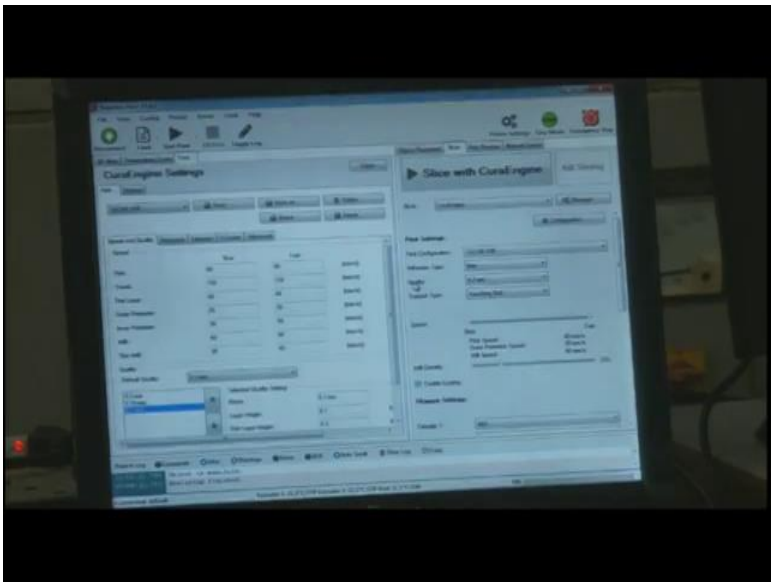


So, this is I can see the most important tab because we are going on discuss this slicing here. So, slicing with Cura engine. So, Cura engine is the software that is used for slicing here. So slicing the time used in V30, this machine, V30 is Cura engine. So, it cares the quality of the product that we need to manufacture, it will discuss everything like thickness, wall thickness, quality, support, then volume, time.

So the word engine means it is the heart, word Cura Engine, I just can say it is the heart or lifeline of the software. So, this whatever we need to fabricate would be majorly decided by this portion only. Now, we have manager here. Manager tab when we click the manager tab, this opens, this window pops up. A manager is for the advanced version. This is, for instance, this manager is here.

This is for the advancement sense, some manufacture is there some big manufacturer is here I need to he need to calibrate the machine. He need to go to the advanced settings then you can use this manager we will not use this.

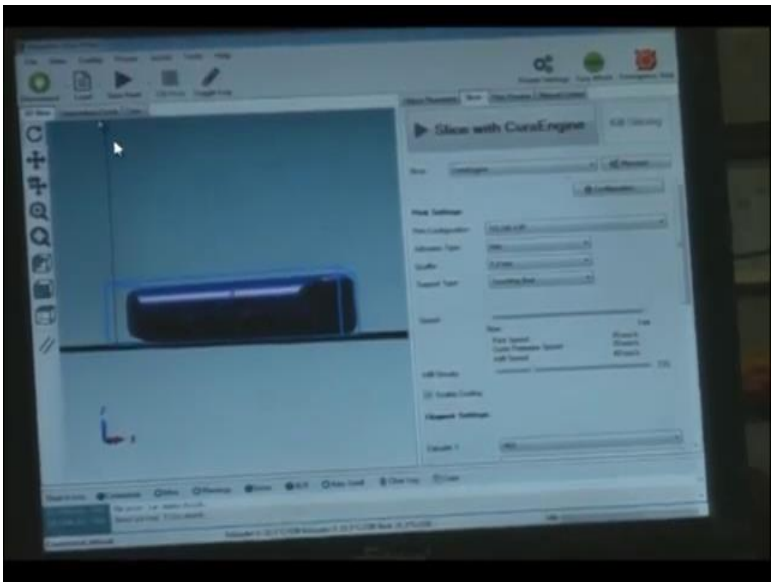
(Refer Slide Time: 27:04)

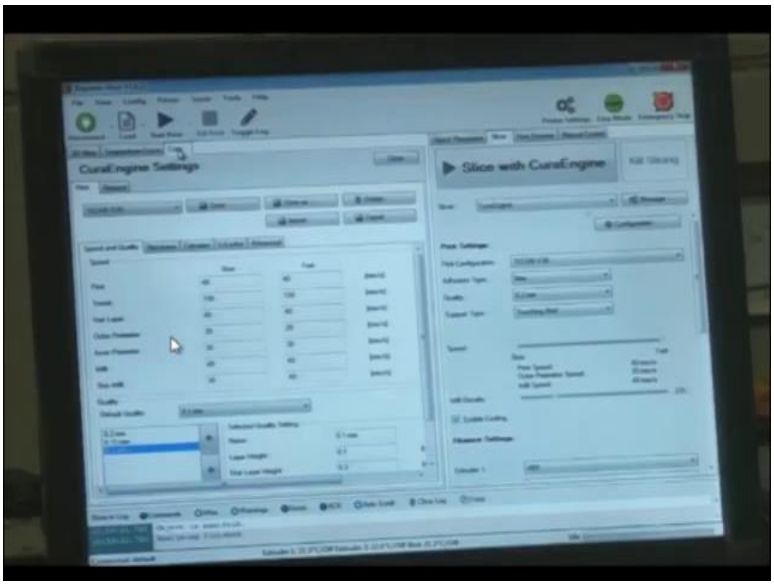


Next one is configuration. When we click the configuration what we can see, we have this Cura engine settings here. So, the two tabs here, filament, very important here, and print what is filament? Filament is a raw material that we are using for printing like I mentioned ABS or PLA that is there in the filament form, this is the raw material, this is filament. So, this is white in color. ABS filament, this is rolled on a spool, so it can be white, it can be red, you can see red and white.

So white is the most common color that is used in FM technology nowadays. So, white and yellow, actually white and yellow are the most common colors that are using FM technology.

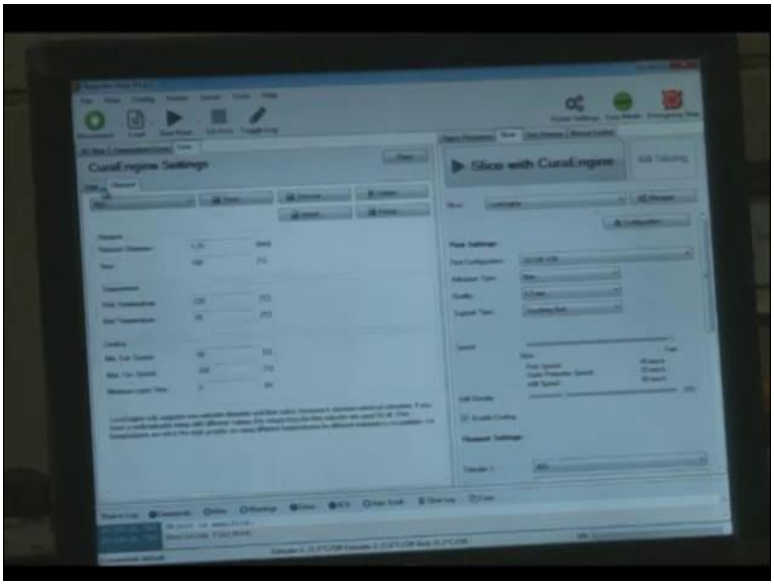
(Refer Slide Time: 28:02)

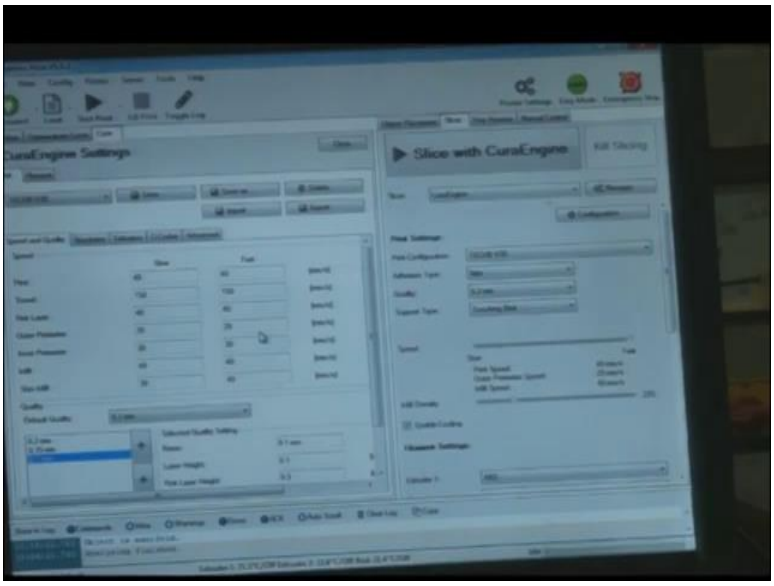


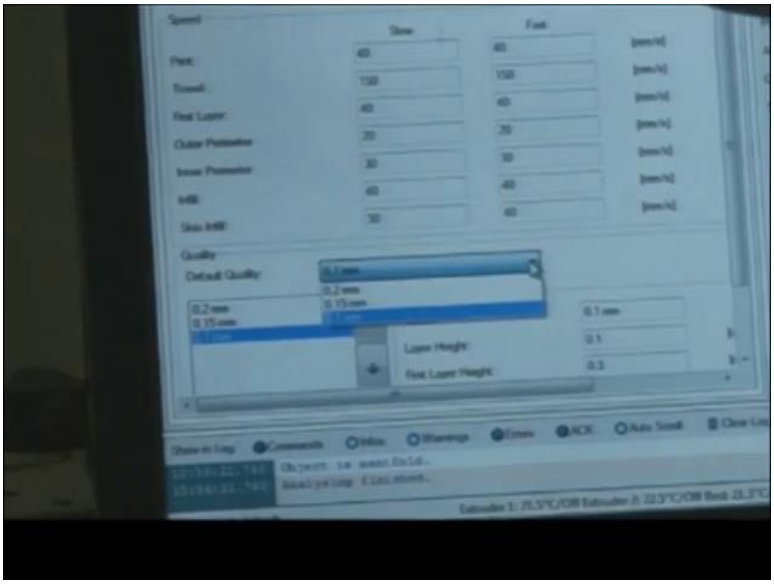


So this is 3D Cura, 3D view includes understanding via 3D view, temperature curve in Cura. Temperature curve we can just see when we actually do machining. So nozzle, at the nozzle heaters are there, heaters supposed to heat so to fuse the raw materials, heaters are heated. So, the orifice, at from the orifice, the material would come and this temperature would be shown in the window.

(Refer Slide Time: 28:35)







So, in the Cura, we have these settings. So we can talk about print or filament here. This is print this is filament. The very first thing is speed and quality, speed and quality, speed and quality are two interrelated terms I can say the opposite terms in this specific machine because if the speed is higher, the quality would be a little lower. And, if the quality has to be good, the speed cannot be very high. So speed and quality like it has to be optimized. So that is why this term speed and quality is there.

Like while designing these different parameters in speed and quality print travel, I will just discuss this. Also, the automatic setting is all there in the software as well. So, speed and quality are

interrelated. So, we do not want our waste the material, we do not want to lose time, because that will increase the cost if we lose a material that will also induce some cost.

Next is structure. The structures we have shell thickness, top bottom thickness, infill overlap, infill pattern, so we will discuss this one by one. So let us come to speed and quality first, speed and quality. So we have print, travel, first layer, outer parameter, inner parameter, infill, skin infill, what are these, let us try to discuss these one by one.

So, what we have, we can have the limits here for the any of these parameters any of these manufacturing parameters I can say okay, we will print the slowest speed and the fastest speed. So, at slow it is kept 40. So, our purpose is that the material that we drop here should stick with the base plate, if it does not stick then there is a complete wastage of the material in even the part might not build properly.

So, the travel speed of the nozzle has to be there. So, it is kept slow and fast, it is same that is a fixed PVS 40 millimeters per second for printing it is 40 and 40, for travel, travel means when it is not printing when it is not productive, it is moving just ideally from one place to another, if the speed is higher it is 150 and 150. 150 means, both 150 slow, and 150 fast means the speed is fixed to 150 millimeters per second.

So, you can see the first layer is kept 40 and 40 and outer parameter is 20 and 20. So, this is an important thing to discuss here. So, if the part is hollow from inside then we have to be very careful about the outer shell. So, it is mandatory or it is I can say compulsory that the outer surface becomes really smooth and shiny and up to the size, up to size, up to mark and the dimension of the profile should be accurate.

So, inside filling can go fast, but outside software's that the periphery of the jobs is to be of high quality. So, good finish, we always take care of the outer perimeter speed that is kept lower that is 20 and 20, okay 20 millimeters per second, 20 millimeters per second for slow and fast both. For filling it is 40 and 40 again. So, that is not very significant.

So, in the same way, inner parameter is 30 and 30, in the parameters 30 and 30 and the infill if you have if your job your part is solid, if a part is solid, the infill the inner part whatever they filled in

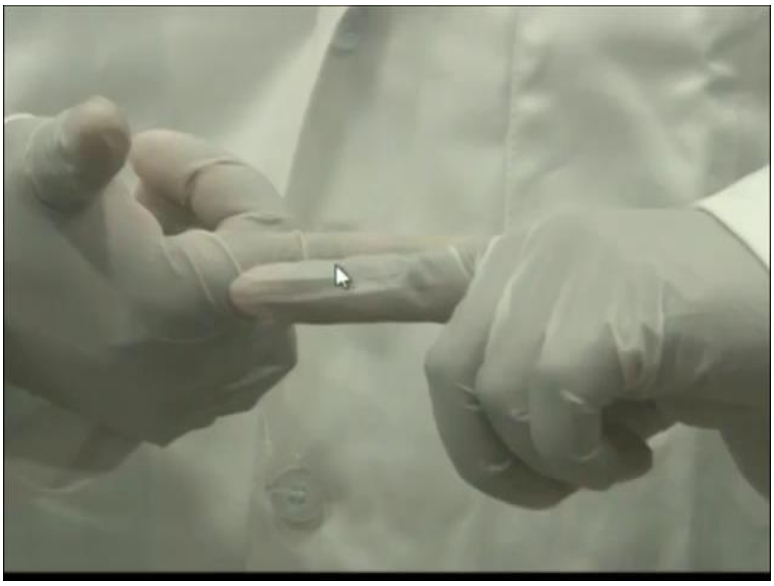
is called infill it is 40 and 40 Okay, another parameter here is skin infill, skin infill is kept 30 and 40 millimeters per second.

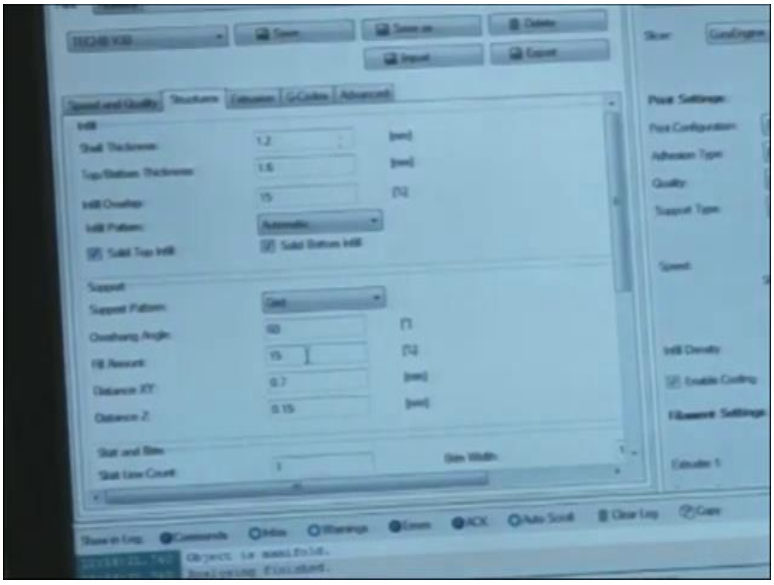
So, this is quality and speed, this is speed and this is quality speed is also related to quality in one way. It has mentioned another term quality here. In this tab, so what is the quality that is the quality of the layers that we are going to deposit here, 0.1 is very thin is very fine. So, it is highest quality, 0.15 is a little higher and the lowest quality is 0.2.

So, if the layer thickness is 0.1 the job would come very smooth and shiny, if it is 0.2 it would be the worst among these, and 0.15 is in between. So, 0.2 means that it consumes less amount of time, but the quality goes down. So, in this way, sometimes we find that our dimension here or our profile shape is important.

So, quality factor is not very important that only the profile shape is there then we go now we can go for rapid printing that is 0.2 mm, sometimes we feel that the size and dimension as well as quality that we need to keep as to be higher. So, we can keep 0.5, 0.1 mm layer thickness. After setting all we go for the next thing, that is called a structure, A structure we have shell thickness it is set as 0.12 top and bottom is 1.6 mm. Shell thickness is 1.2 mm, top bottom thickness is 1.6 mm. So, infill overlap is 15 percent.

(Refer Slide Time: 34:34)





So, what is basically infill overlap, when one bead is deposited over the other bead will overlap. So, if it does not overlap it cannot stick with the first layer. So, the first layer would stick the first layer we stick with a second layer.

For instance, this layer thickness is 0.1, the second layer thickness is again 0.1. So, there will be some overlap. So, this overlap how much overlap we need to we can keep here. So, what will happen, it will displace the nozzle if this overlap is not here this is 0.1 and this is 0.1 the nozzle will touch this one and it will, it might displace the nozzle it might the path may distort.

So, some overlap should be there. So, as it has to be some gap here for the nozzle to working, okay some gap at the surface has to be there. So, for that, this overlap limit is given. So this overlap limit can be 15 percent or 20 percent. So, this is kind of an overlap, the second layer would not be exactly on the top of the first layer.

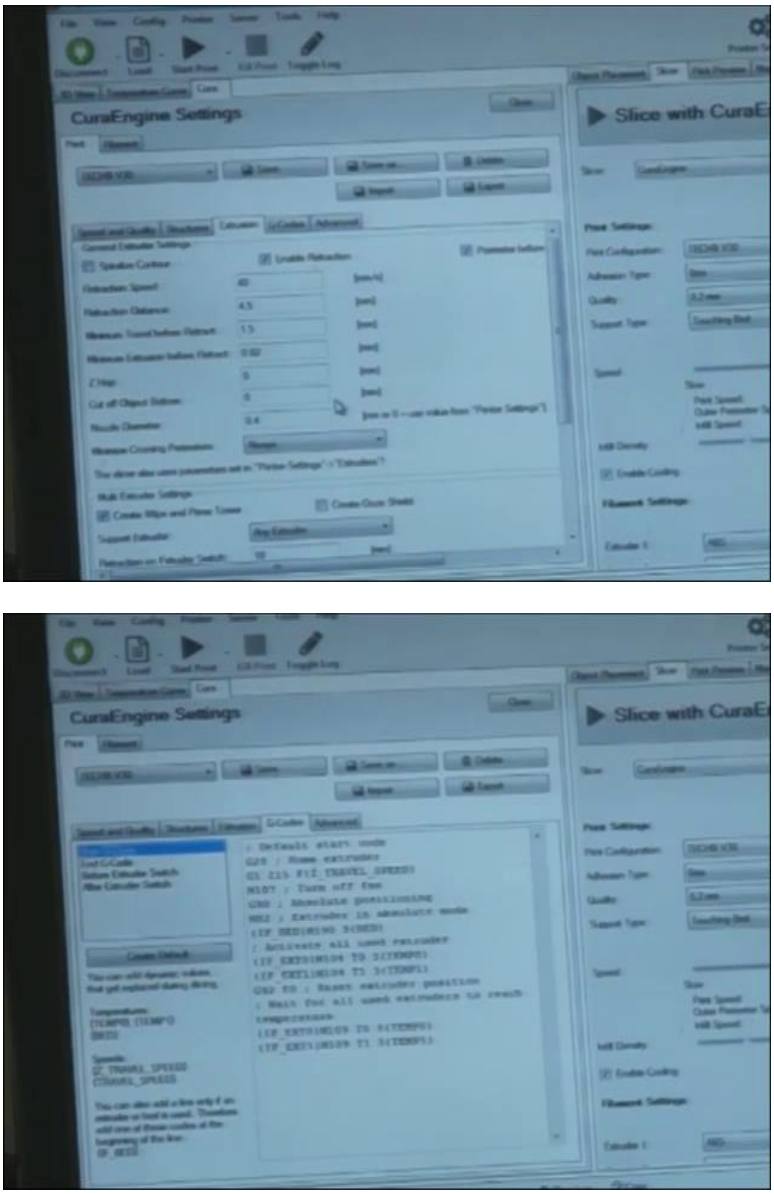
It will be a little overlap in this case, as you can just see like the fingers we are just try to demonstrate that 50 percent of overlap can be seen here but it is one five, 15 percent of exact overlap would be there if we put this input in the machine. So, this bead is deposited in this way.

So, sticking the layer would not be proper if this overlap is not there and the proper overlap that the fill amount of 15 percent then this overhang angle as well here. So overhang angle is 60 degree here.

So, overhang angle here is for the machinery, for this machinery if we are going to deposit better say deposit or print or manufacture the job if it is solid, this is no issue at all. If it is a conical shape, an angle is up to 60 degree, As 60 degree means if the angle is 60 degree like this if it is about 60 degree, the thing would happen it just deposit slice by slice, but if angle is more than 60 degree here, it has to have supports here like it had to have supports here to support the material.

So, this over an angle that we can decide, that we can decide over an angle has to be fixed that is fixed at 60 degrees here.

(Refer Slide Time: 36:58)



So, this over an angle is 60 degrees and next, what we have here. So, we need to see that is there anything overhang in the CAD model? In the CAD model if there is an overhang is there then we need to put this angle.

So, if anything is overhung then we need to provide the support and only then we can proceed further. So, then overhung angle has to be decided. Next is extrusion. So, this is extrusion of the filament from the spool to the nozzle. So, this is retraction speed, retraction speed is 40 millimeter

per second retraction distance is 4.5 mm, there is basically a roller at the back of the machine that is responsible for bringing the filament to the nozzle.

So, there are two rollers and a filament passes through them. The filament moves through them to tube to go to the head, so this retraction speed or feeding of the raw materials is 40 millimeters per second. So, as per speed in printing whatever the speed of printing is there that speed will be kept here in extrusion, now retraction distance is 4.5 mm and number of considerations are there, but these are not very important.

Because when we are going to servicing we go to service overhaul the machine these things are set by the G code because when we go for overhauling or service of the machines, these things are set by the manufacturer or the provider themselves. Next one is G code. So, we are talking about slicing here, okay, this slicing with Cura engine, in slicer in the slicer and first, we had object fill up placement.

When we have slicer second tab were, in slicer multiple options here now, which is G code. So, slicing is nothing but depositing the layers, but how to control the movement of the nozzle in the x, y or z direction, that has to be taken care by a program and this programming system is known as G code programming system if you know CNC programming G code and N code might be constant.

In this programming, system is known as G code and G code and M codes are both there, these are mostly used for CNC programming like milling, turning and other machines same code is used herein 3D printing for the movement of the head over the bed for depositing the material. We will meet in the next lecture we will discuss further about the course.

Thank you.