Product Design and Manufacturing

Prof. J. Ramkumar
Dr. Amandeep Singh Oberoi
Department of Mechanical Engineering & Design Program
Department of Industrial and Production Engineering
Indian Institute of Technology, Kanpur
National Institute of Technology, Jalandhar

Lecture – 27

Laboratory Demonstration, 3D Printing (Part 1 of 3)

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 advanced manufacturing lab at IIT Kanpur, which is an elite facility that we have here and like we have seen different electric prototyping techniques.

In this lab demonstration, we will just discuss the 3D printing. 3D printing using a specific machine that is there in the lab, the name of the machine is TECH B V 30 and our Laboratory Demonstration, the instructor is there who has made this video, and we were in the lab we develop this video and I am trying to put my voice to demonstrate it properly.

So, let us start this and go to the lab. So, right now we are in 4 I lab, that is the facility in IIT Kanpur, four I, four I is are Innovation, Incubation, Implementation and Integration. In this laboratory, we have multiple machines which are non-convention and advanced machines, we do research and also consultancy is being carried out here.

(Refer to Slide Time: 01:33)

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, and then we will set up the machine. What we will do, we will the make the whole set up, then we will see how rapid prototyping and various features, various parameters are taken in to account while manufacturing, this is additive manufacturing as I mentioned earlier.

This is a three printer TECHB V30 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 fuse deposition machining method, there are two major kinds of machines in the machine sections as we know now FDM and SLS FDM is fuse deposition machining and SLA is Stereo Lithography Apparatus. I will like to put some light, in FDM there is a big variety of colors those are available like, we can have red yellow and white, are the major color also there are certain companies who are manufacturing the colors based upon demand.

But in SLA, the color variety is not possible, there are certain differences in FDM and SLA, this is, this is only FDM machine, this has two spools, two nozzles in which we can use two colors and the multicolor of the bicolor products can be made. About FDM and SLA, FDM machine produce products with the precision, but the precision level in SLA is higher. Thus it has 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 the printer, the printer resolution is a factor of the 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 product 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 has to be removed. 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 that has to be taken out from the box where it is made.

So, there is a big difference. So, there we are more focused on FDM fused deposition method. So, this machine, that is FDM machine, deposition method of few deposition machining, this machine can use two kind of filaments; however, there are multiple filaments like ABS, then PLA, it can use the ABS PLA and ABS is a acrylonitrile butadiene styrene as we know and it is oil based plastic. It is strong study material that is widely used these days, like Lego building blocks or Lego toys are made of this ABS to quote as an example.

And the PLA, PLA is one of another materials, it is poly lactic acid it is made of organic materials specifically from the cornstarch or sugarcane, it makes the material both easier and safer to use while giving it a smoother and shinier appearance.

So, it has more aesthetically appealing products which are made of out of it. Now, this PLA thermoplastic is also more pleasant for nose, like it smells lesser like, as the sugarcane materials smells slightly sweet when heated opposed to the hash material associated with ABS, when we heat ABS the smell is quite annoying in PLA this is the not the case.

So, this means that the printing, using these parts it friendlier for the operator. And also PLA seems like a better overall choice, because it features far low melting points as well the melting point of PLA is quite lower than ABS. So, I would not move into that track now. So, let us discuss about this machine, this machine is basically using SB FDM. Technology the raw material moves from the tube and comes up to the head, there is nozzle around which heaters are there, and when we switch on the machine the heating systems gets on, when we switch off the machine the heating systems gets on there are two nozzles here, nozzle a and nozzle b you can see.

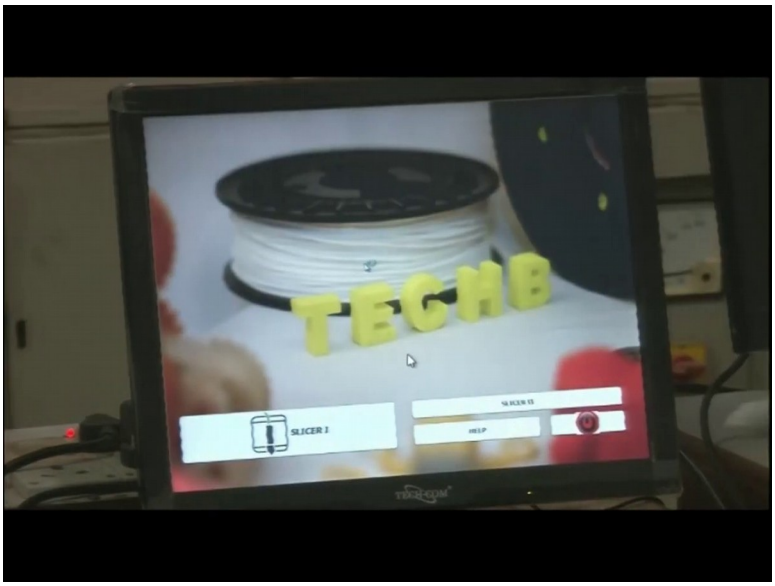
So, the temperature when you switch on the machine, the temperature rises and it goes up to 200 to 230 degree centigrade. At this temperature the raw material kept in the form of wire comes out near the nozzle, near the nozzle; it comes out through the nozzle. So, it is it coming through this tube, we 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. These are two nozzles here nozzle a and nozzle b. Why two nozzles are there? We will just explain one nozzle is for the base material, another is for support.

So, this is a very fine orifice of the nozzle, or the orifice dia of the nozzle is 0.4 mm, 0.4 mm when all these bed area maintain the soaking temperature of the bed of 50 to 65 degree, 50 to 65 degree temperature is maintained here. The temperature it heat up a little bit so, as if the temperature is it is actually the preparation of the table. It is heated to the 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 degree. So, that it can stick easily. So, that it sets easily here.

So, this will have to deposit one layer upon, another during Fabrication or Manufacturing whatever we want to fabricate here in FDM Technology the machine size is, this machine size is 1 feet by 1 feet by 1 feet. So, in x y z directions, the envelope is 1 cubic feet I can say, 1 cubic feet envelop is there. So, it is thus plug and play machine once, we have a little practice on this.

(Refer to Slide Time: 08:14)

(Refer to Slide Time: 08:17)



Along with the hardware, we have software here that also gets on when we switch on the machine. This is the software when we switch on these it comes TECH B. So, TECH B is the machines the software is switching on this TECH B software is getting on as you can slicer 1 and slicer 2, here 2 slicers are there.

So, their CPS and controllers, which are here at the bottom surface of the machine, cps controller for this software is here. So, the 2 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

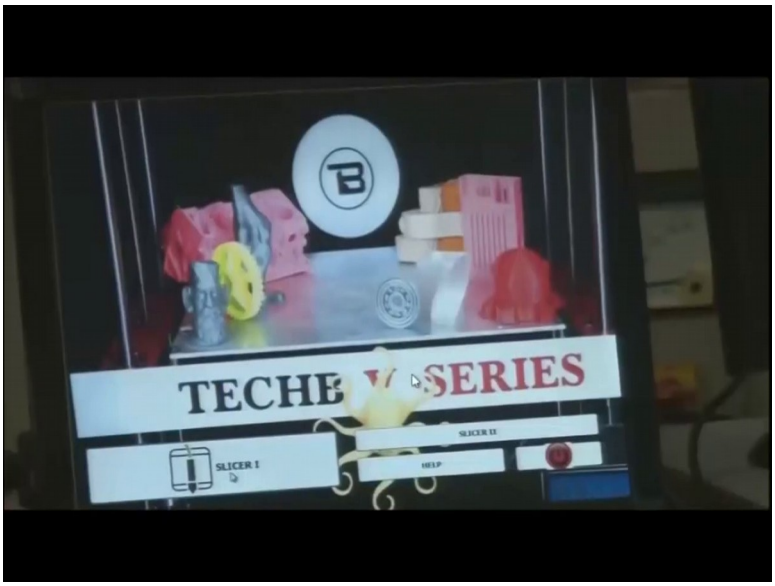
geometry where you find this taper is more than 45 degree, we need to provide support like I discussed. Then the 2 nozzles will automatically activate and that will provide the support to produce the final product, final model that we need to obtain.

So, this all feedback is given to the head with the help of software, that would 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, it is general precaution. We need to place this machine in room temperature. Air conditioners 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 hand, 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 this sharp edges and be careful in this are the safety precautions when we start the machine. Next with this machine, we can manufacture partly sharp corners any areas, any radius and the profiles with pocket, the profiles which are empty inside or is hollow.

And one of one known feature which machine is that, we can print to different color models from these two nozzles, nozzle a and nozzle b if the two different colors can be obtained like, yellow and red, different colors can be obtained and those can be produced as well.

(Refer to Slide Time: 10:32)



So, the well this computer screen we have a software called Repetier. So, we have slicer 1 and slicer 2 here. So, if the machine is the machine is with two head, with both hands (Refer to Time: 10:43) work. In this machine there is a single head, if the two heads are there, the machine though both hands can work independently you know, that is single head here; single head on the single head here, two nozzles are attached.

So, that is why only slicer one is visible here. So, the software starting on, that repetier software is starting on, that Repetier host is a specific name of the software 6.2 is a version, then we need to connect the software to hardware. Now, both the interfaced face 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 there are two fans, two fans are above 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 cannot, just a level of the bed with 4 screws, you can see this screw, 1 2 3 and 4; 4 screws are there. So, these 4 screws are have are loaded here using spring mechanism. So, when we fabricate the 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

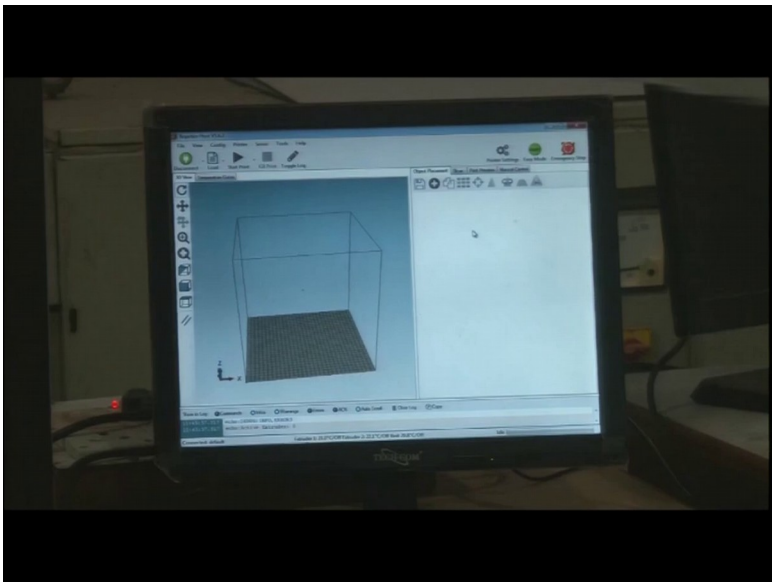
to adjust the movement in the nozzle, still you can see, there is a spring mechanism here. So, that helps to adjust the movement.

So, in this way, we will make the leveling of the bed, also when we go whenever we go for calibration, so, we use this screws to loosen and tighten and to make sure that the bed is flat, it is square with the with the adjacent surface as well. So, in this software we can deposit lay thickness in slicing from 0.1, 0.15, 8.2, the 3 levels here. Whenever we change this parameter of the position of layer height wise, then we need maintain the gap as well.

So, say let me say a fixed 1, 0.15 from the top of the bed to the tip of the nozzle. So, we need to shut this gap and lock the limit switch. The limit switch is on this side, limit switch on this side. So, we can lock the limit of the bed using this layer. So, that can move up to the specific limit only. So, bed would go up or down as per 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 we feeding here? Because, the raw material is in the form of spool, it is an open spool kept at the back of the machineries there, spool 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 a raw material that is what that I mentioned ABS or PLA, if it is moisturizes then it obstructs the proper deposition of the job here.

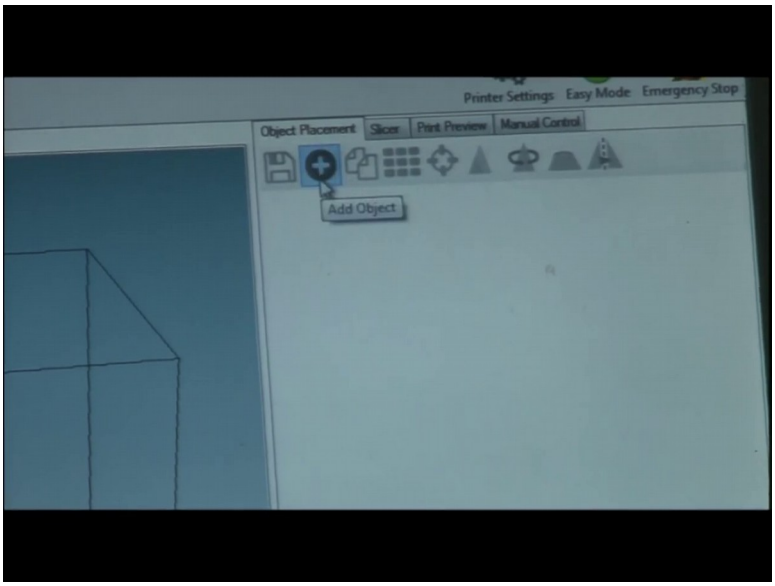
(Refer to Slide Time: 14:23)



Whenever start we input the data to the machine for fabrication. After switching on the machine you have to take care of few things or the display here. You can see this display, because the hardware that is connected to the software and displayed to this screen or the screen we have programmed, we start feeding the data to the hardware, we have to take care of number of things after switching on the machine. After switching the machine, itself gets connected to it is software.

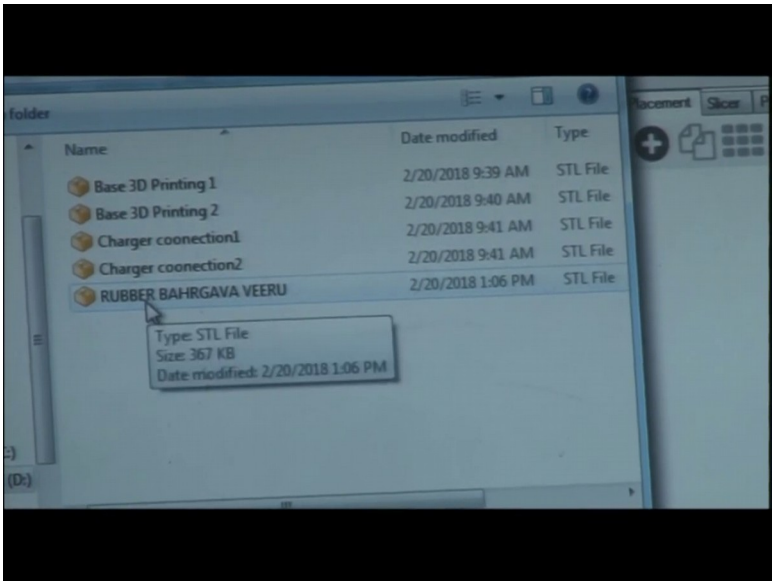
Whenever we start we have to go to the option number 1 that is object placement, which means that, what we need to print will place it here first, the place of placement of the object here.

(Refer to Slide Time: 15:03)



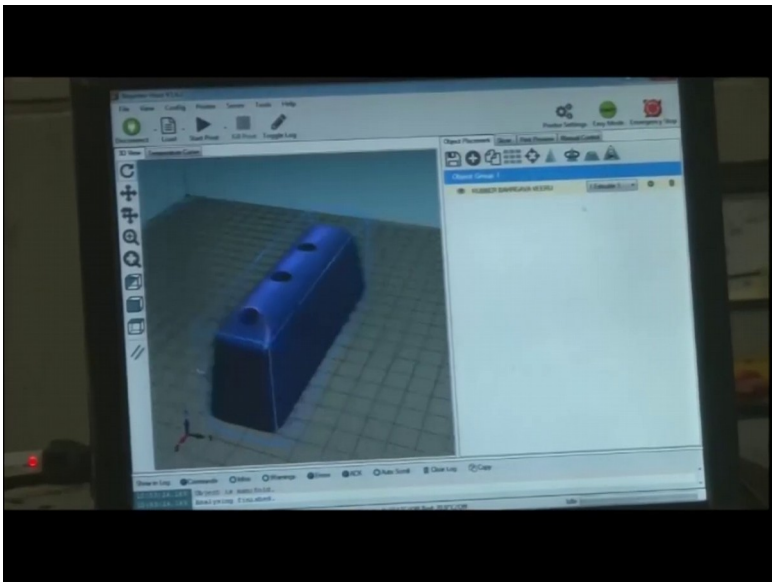
So, when we click on when we click here few things will be highlighted few icons are highlighted let us add object, this plus sign is add object. So, what we want to add, we click add object here and when we click it, it opens the drive or the any location in the computer like the cod drive where the object is kept. So, wherever we have kept the cad model for fabrication, whatever we want to fabricate; you can just go and click there or we can even generate some model.

(Refer to Slide Time: 15:35)



So, this model known as Rubber Bhargava Veeru this model is there.

(Refer to Slide Time: 15:46) .



So, we will click here. So, yes when we click here this job is there here on the envelop. What is envelop? Envelop is this where we are going to manufacture this thing. So, this is 1 cubic feet, that is 1 feet by 1 feet by 1 feet, the envelop size is this one. 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 manufactured, ok.

So, it is actually this is our base plate where it is manufactured or this is our envelop that is being displayed in the software. The approximate size of this model is 4 inch by 1 inch by 2 inch. 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, ok. 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. Here in object placement, there are other icons like copy object, copy is like we can make 1 or 2 or 3 copies like, we are making 2 copies the 3 copies are made of this object.

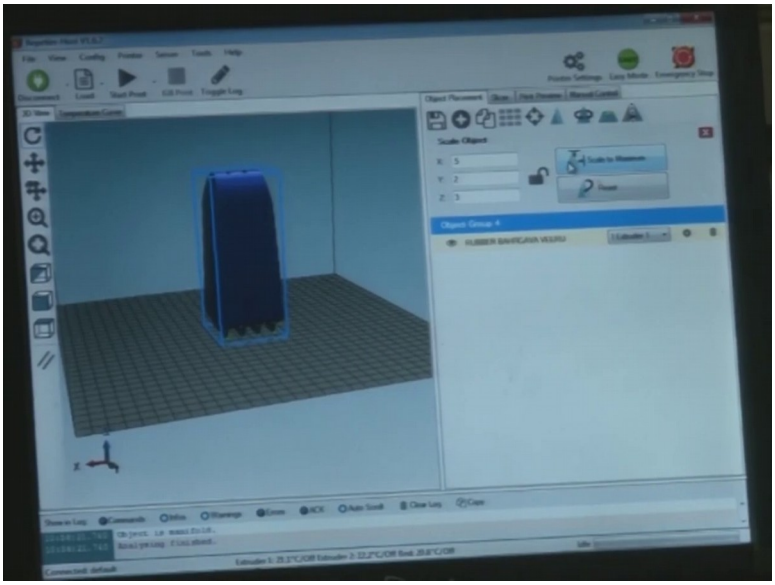
So, single cad model can be printed in multiple models using this command, we just copy, ok. This is product design cad, computer aided design; yeah, this is auto positioning. Auto positioning, what is auto positioning? So, when we need to print at the

center, auto positioning would bring the object at exact center in the envelop. So, we need not to drag it like, we were dragging before, we need not to drag it, ok, next one is center object.

Auto positioning, auto positioning, it will position the object so, as the least material is used. Center object is similar to auto positioning, in auto positioning what happens this is auto positioning, in auto positioning what happen, it place the object, so, as to the minimum material is used. So, as the material has to travel the minimum path and in auto centering object would be placed at the center of the envelop exactly at the center like, we were dragging like migh,t we have might, offset the object while dragging manually, but this located at exactly at the center

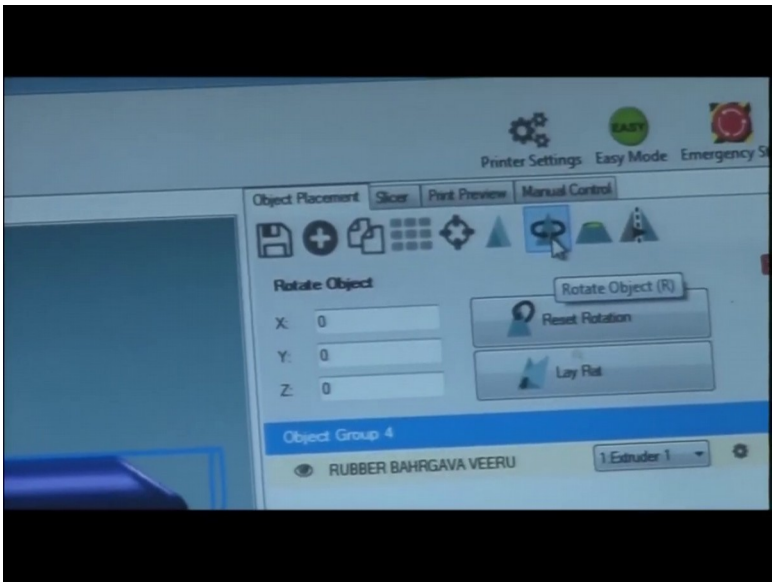
Now, yeah this is scale object. Now, here scaling means changing the size of the object, making it over size, under size or upscale and downscale are better words, you can say axis made twice, ok, x direction made twice now y is made again twice, thrice we are scaling the object. So, up scaling this one, you can see it is midpoint 5.

(Refer to Slide Time: 19:14)



So, this is scaling of the object, sometimes the cad model will draw is big and we just need to produce a prototype or just a feel model out of that, then we can just downscale it and produce. So, this is, this is a direct provision. So, that this is direct feature in the software for this one.

(Refer to Slide Time: 19:52)



So, next I can do here is rotate object. Sometime,s like I discussed like we have discussed as the placement of the object or the alignment of the object in is very important in deciding the support material that is used, what is the amount of support material that is used.

So, to properly align the job, like in x direction, we have aligned the to 90 degrees, so it is rotation. So, we can make the job in this position as well, like if the if we see that the how much material will 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.

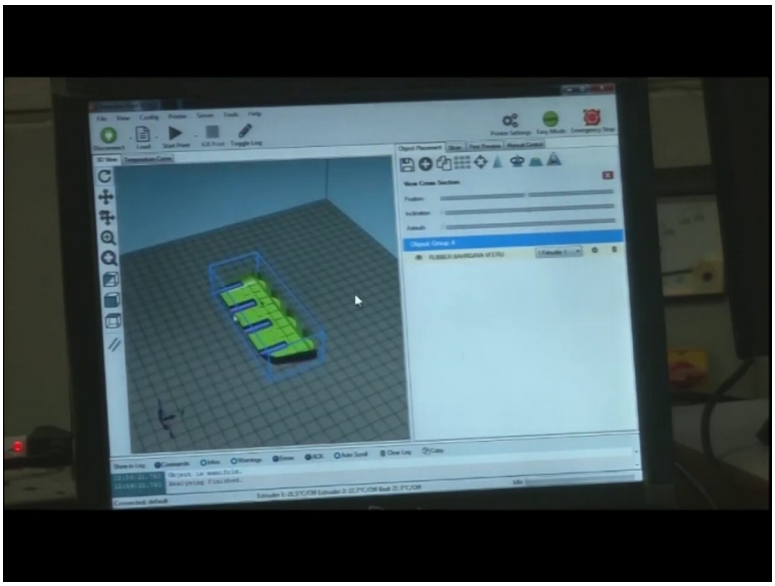
So, sometimes we have the pure flat surface. If this is flat surface it is the perfect thing, the flat surface it will deposit properly on the flat, the flat surface is exactly perfect, but you can see that we have an angle here. In this, we have an little angle here like, we need to have some support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

So, we rotate and try to keep the cad, cad is this model, computer design model, we have I will use keep on using the word cad for this model, this is cad is kept in proper alignment. So, as to minimize the material used, the Y is kept 0, X 90, Z 90. So, if Y is kept again 90.

So, the height increases, what is the draw back when we increase height. When we increase height, because it is manufacturing 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 minimum you know, these many number of layers have to be deposited. So, the total time would increase total time, we need total economy, total cost of the product will also increase, because in product cost function the machining time is one of it is factor.

So, this is not a very good position to manufacture here. So, we can opt to select in flat position and we orient the job in proper place and position. So, next is view cross section, this icon is view cross section. Sometimes, we need to see what is in the 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; the customer has just made this model brought this model for us. So, we denote we are not aware that what is the profile inside the, are there any holes, some pockets, some counter, sink holes?

(Refer to Slide Time: 23:05)



So, if the customer is not able to describe it properly; the manufacturer or the operator has to be very careful. It is duty of the operator to be conversant with what he 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 the upper surface, blue color is the

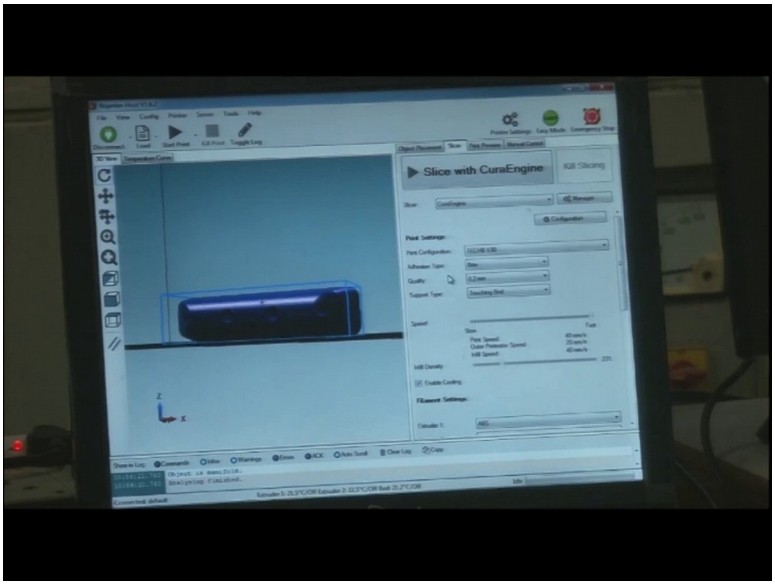
upper surface, and green color is inside, what is inside. So, this is the color difference color coding is here.

So, blue is the upper surface inside, it is a solid model, it is in hole and is not hollow from inside; this is the inside model. So, we can just to in view cross section tab, we have this bar position, ok, 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?

Here, the thickness of the shell has to be according to the requirement; it has to be according to rigidity whatever requirements we need. So, this is again is inclination; inclination, at some angle also you can see the angle is changing, this is 0 angle, this angle is about 90 degree here. So, this is azimuth. So, we can rotate it from the bottom here, we can see another view point from this.

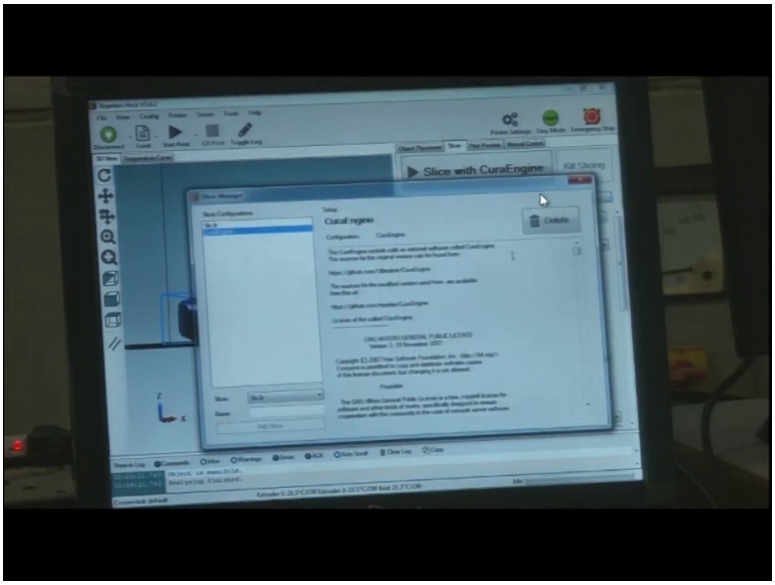
So, in this way using these three tabs, we can see each detail for instance they are a multiple features deep features or the intricate features in this job, we can see that what is there inside the job. So, why did the cad model have all those features? Now, this is the actual part now. Now, the last one is very common thing, mirror object; this is mirror object. This 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, ok.

(Refer to Slide Time: 25:35)



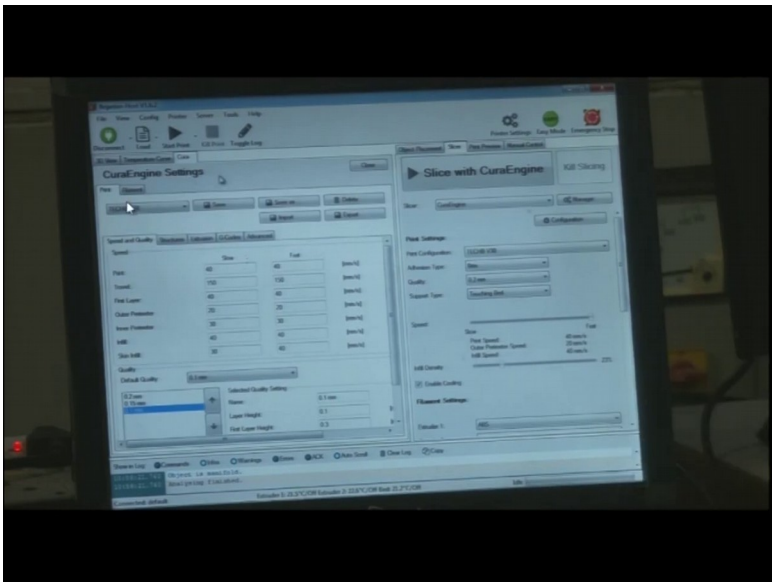
So, next tab is slicing. So, this is I can see the most important tab, because we are go on discuss this slicing here. So, slicing with CuraEngine so, CuraEngine is the software that is used for slicing here. So, first thing the time used in V 30 this machine, V 30 is CuraEngine. So, it cares the quality of the product that we need to manufacture. It will discuss everything like, thickness, wall thickness, quality, and support, then volume, time. So, the word engine means it is the heart, word CuraEngine 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.

(Refer to Slide Time: 26:28)



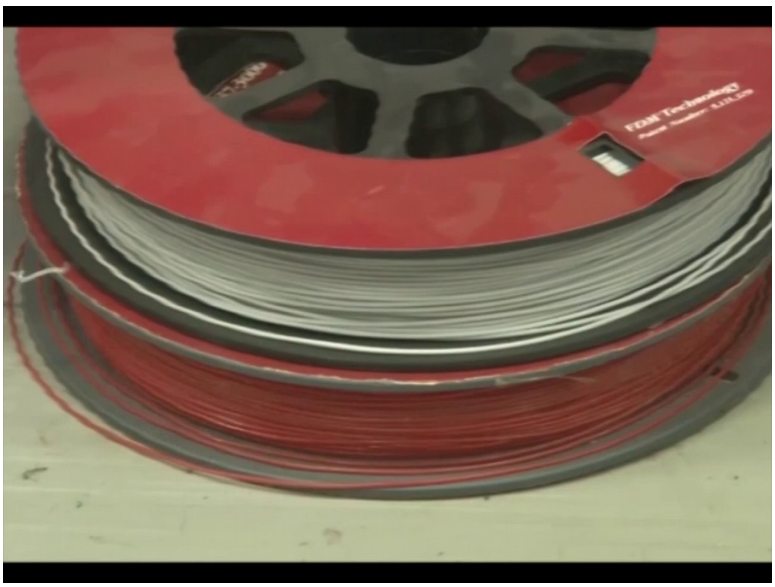
Now, we have manager here. Manager tab, when we click the manager tab, this opens; this window pops up. Manger is for the advanced version, this is for instance this manager is here. This is for the advance, for instance some manufacture is there some big manufacture is there and the need to he need to calibrate the machine, then need to go to the advanced settings, then he can use this manger we would not use this next one is configuration.

(Refer to Slide Time: 27:00)



When, we click the configuration, what we can see? we have this CuraEngine settings here. So, that two tabs here filament very important here and print.

(Refer to Slide Time: 27:19)



What is filament? Filament is the 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 color ABS filament, this is rolled on spool.

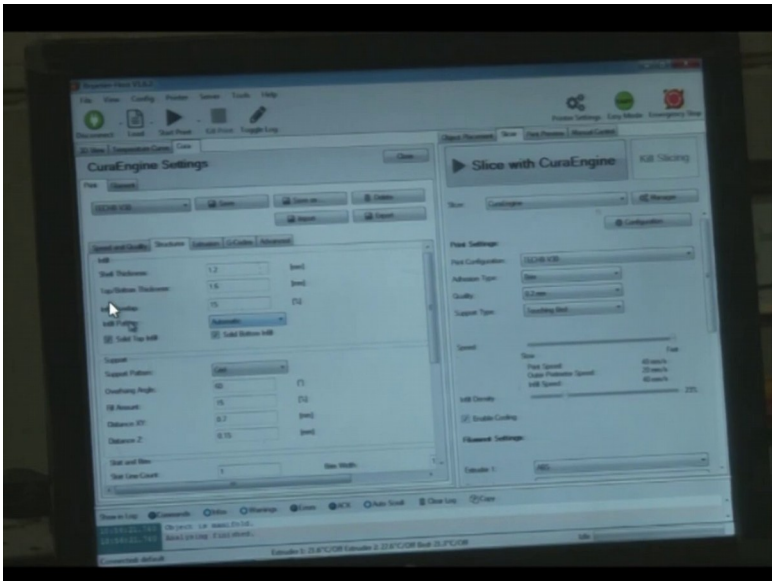
So, it can be white, it can be red, you can say red and white. So, white is the most common color,that is used in FDM technologies nowadays. So, white and yellow actually white and yellow are the most common colors that is used in FDM technologies.

So, this is 3D, 3 D Cura, 3D view, in Cura in the sense we have 3D view, 3D view temperature carbon Cura, temperature carbon. We can just see when we actually do machining. So, nozzle at the nozzle heaters are there heater supposed to heat. So, to fuse the raw material heaters are heated. So, it the orifice it, from the orifice the material would come and this temperature would be shown in the window.

So, in the Cura, we have these settings. So, we can talk about print or filament here. This is print, this is filament. So, very first thing is speed and quality. Speed and quality, speed and quality are two interrelated terms, I can say the opposite terms in the 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 deciding these different parameters, in speed and quality print travel I will just discuss this, also the automatic setting is 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 to because that would increase the cost, if we lose the material that will also induce some cost.

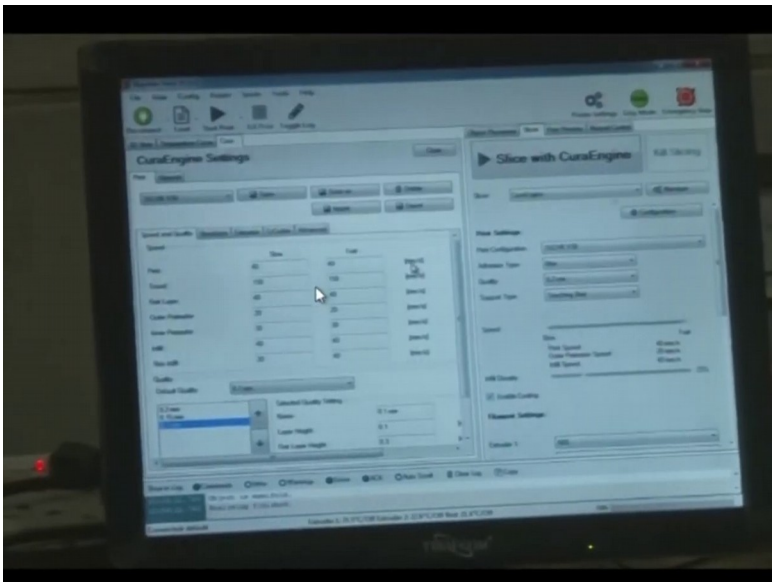
(Refer to Slide Time: 29:40)



Next is structure. In 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, skill 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 parameter I can say, print this lowest speed and the fastest speed. So, thus at slow it is kept 40. So, a purpose is that, the material that we drop here should stick with a base plate with this. If it does not stick, then there is a complete wastage of the material and even the path now might not be built properly.

(Refer to Slide Time: 30:43)



So, the travel speed of the nozzle has to be dazed. So, it is kept slow and fast it is same that is the fixed speed is 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 high it is 150 and 150. 150 means both 150 slow and 150 fast means the speed is fixed to 150 millimeters per second.

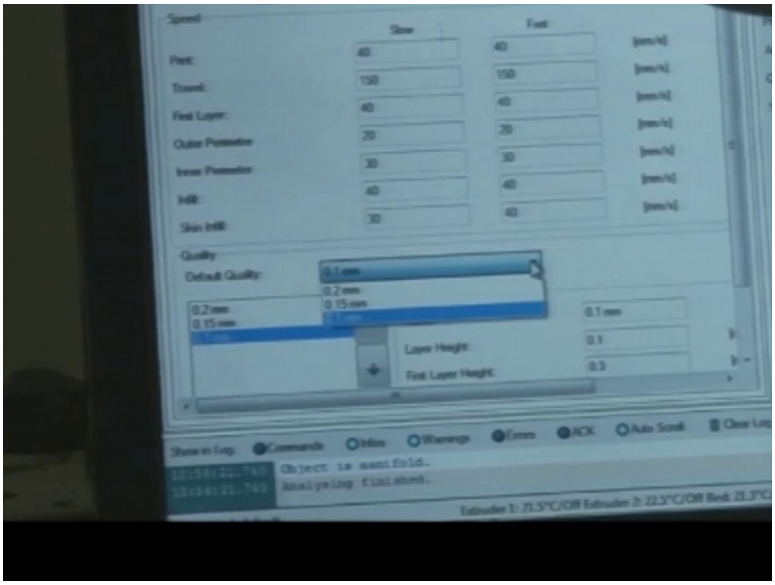
So, you can see the first layer is kept 40 and 40 and outer parameter is 20 and 20. So, this is an important thing to discuss here. So, if the part is hollow from inside, then we have to be very careful about the outer shell. So, it is mandatory or it is I can say, compulsory that the outer surface becomes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

So, inside filling can go fast, but outside surface that is the periphery of the jobs is to be of high quality. So, good finish we always take care of the outer parameter speed, that is kept lower, that is 20 and 20, 20 millimeters per second 20 millimeters per second for slow and fast both, for feeling it is 40 and 40 again.

So, that is not very significant. So, in the same way, inner parameter is 30 and 30, inner parameter is 30 and 30. And the infield if your job is a part is solid; if a part is solid the infill the inner part, whatever the field in is called infill, that is 40 and 40 ok.

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, here in this tab.

(Refer to Slide Time: 32:52)



So, what is the quality that is the quality of the layers that we are going to 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 in 0.15 is in is in between. So, 0.2 mm means that it consumes less amount of time, but the qualities goes down.

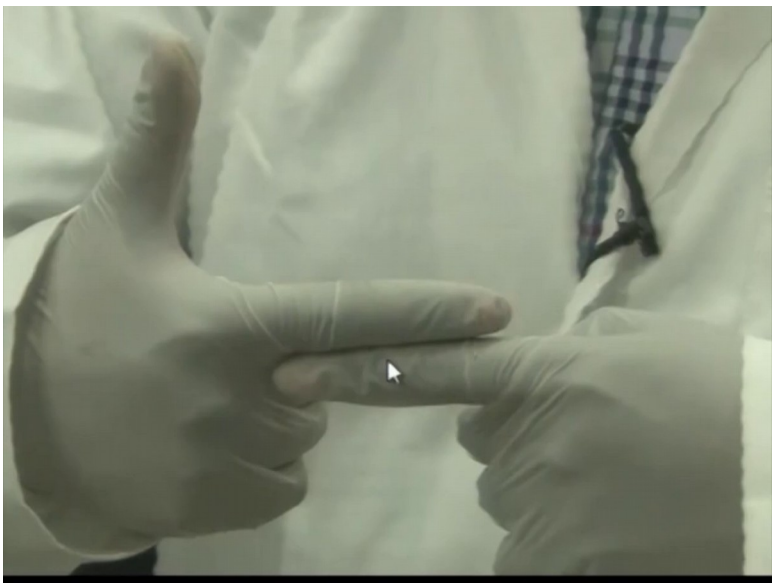
So, in this way, sometimes we find that our dimension here or our profile shape is important. So, quality factor is not very important that only the profile shape is there,

then we go we can go for (Refer to Time: 33:45) painting that is 0.2 mm can be there. Sometimes we feel that the size and dimension as well as quality that we need to keep has 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 as structure, ok. 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. So, what is basically infill overlap, when one bead is deposited over the other bead, it will overlap. So, it cannot if it does not overlap, it cannot stick with the first layer.

(Refer to Slide Time: 34:39)



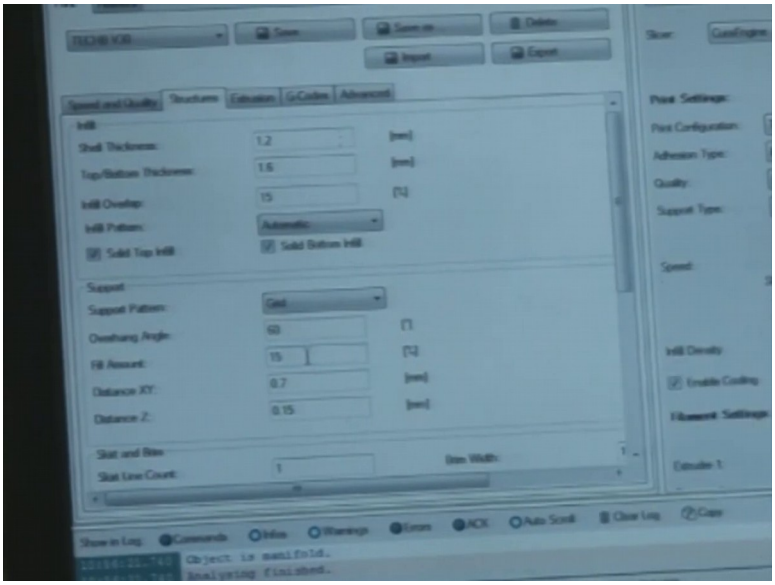
So, the first layer would stick, the first layer will stick with the second layer, for instance this layer thickness is 0.1, the second layer thickness is again 0.1. So, there would be some overlap. So, this overlap, how much overlap, we need to we can keep here. So, what will happen it will displays the nozzle, if this overlap is not here this is 0.1. And this is point one the nozzle will touch this one and it will it might displace the nozzle it might the part may distort.

So, some overlap should be there. So, as there has to be some gap here for the nozzle to work in, ok, 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 little overlapping. In this case, yes you can just see like the fingers we are just trying to try and demonstrate that, 50 percent of, 50 percent of overlap can be seen here, but it is 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 to the layer would not be proper, if this overlap is not there. And the proper overlap that the fill amount of 15 percent, then there is an overhang angle as well here.

(Refer to Slide Time: 36:08)



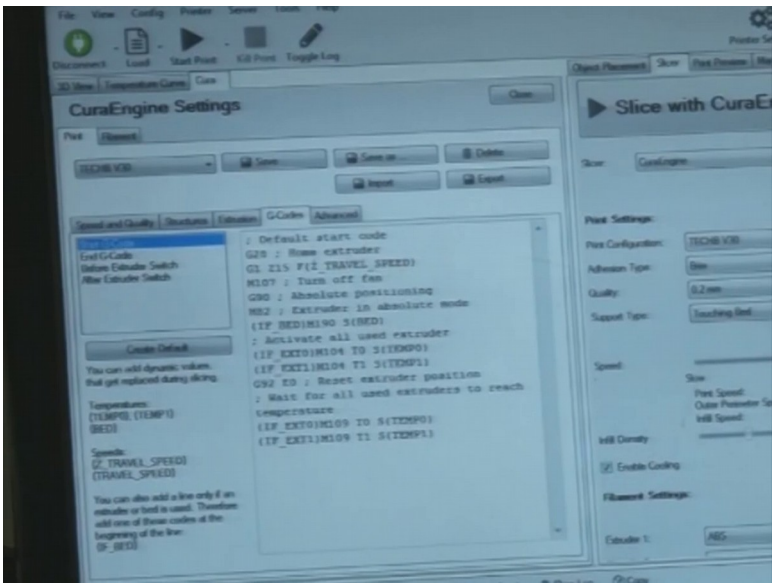
So, over an angle is 60 degree here. So, over an angle here will be for the machinery, for this machinery. If, we are going to deposit, I will better say deposited print or manufacture the job. If it is solid, there is no issue at all. If, it is a conical shape and angle is up to 60 degree.

That 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, slice, slice, but if angle is more than 60 degree here, it has to have supports here like it have to have supports here to support the material. So, this overhang angle that we can decide, ok, that we can decide overhang angle has to be fixed, that is fixed at 60 degrees here. So, this overhang angle is 60 degrees and next what we have here. So, we need to see that is there anything overhangs in the cad model, in the cad model if there is an overhang is there, then we need to put this angle.

So, if that anything is overhang, then we need to provide this support and only then we can proceed further. So, then over an angle has to be sided. Next is extrusion. So, this is extrusion of the filament from the spool to the nozzle. So, this is a retraction speed, retraction speed is 40 mm 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, that two rollers and the filament passes through them, the filament moves through them. So, tube to go to the head. So, this retraction speed or feeding of the raw material is 40 millimeters per second. So, as power speed was printing, whatever the speed of printing is there that speed would be kept here ok, in the 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 are go to service or overhauling the machine, these things are set by the G code.

(Refer to Slide Time: 38:28)



Because, when we go for overhauling or service of the machines, these things are set by the manufacture or the provider themselves. Next one is G code. So, we are talking about slicing here ok, this slicing with CuraEngine in slicer. In the slicer, first we had object filament placement, then we have slicer second time here.

In slicer they have multiple options here, now this 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 we know C and C programming G codes and M codes might be conversant.

In this programming system is known as G code and, G code and M codes are both there, these are mostly used for C and C programming milling turning and other machinings same code is used here in 3D turning 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.