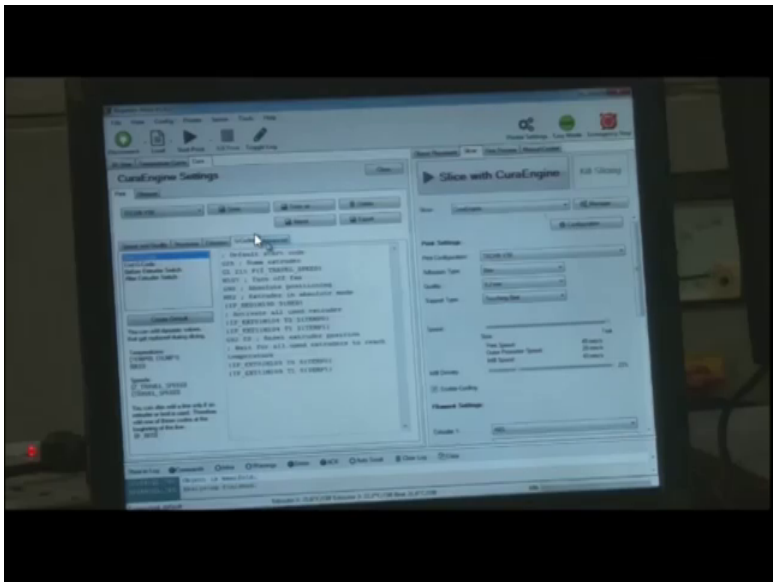
Rapid Manufacturing

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

Lecture – 28
Laboratory Demonstration, 3D Printing (Part 2 of 3)

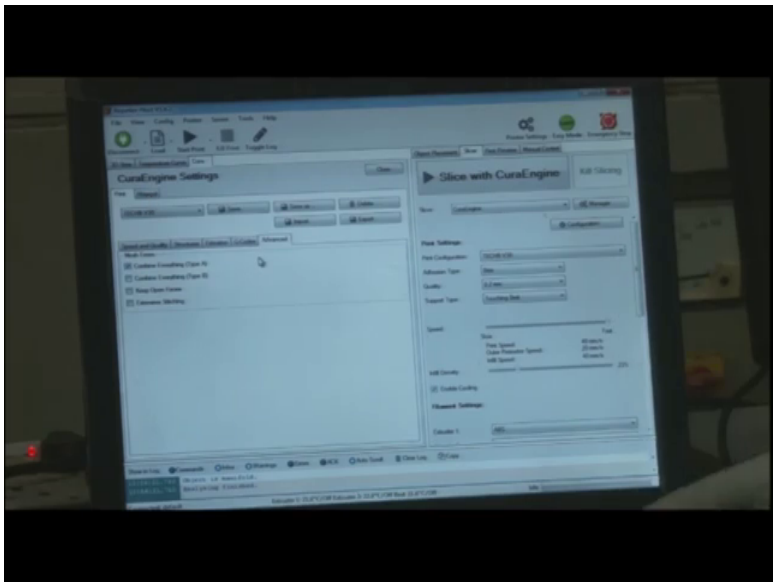
Good morning. Welcome back to the course.

(Refer Slide Time: 00:19)



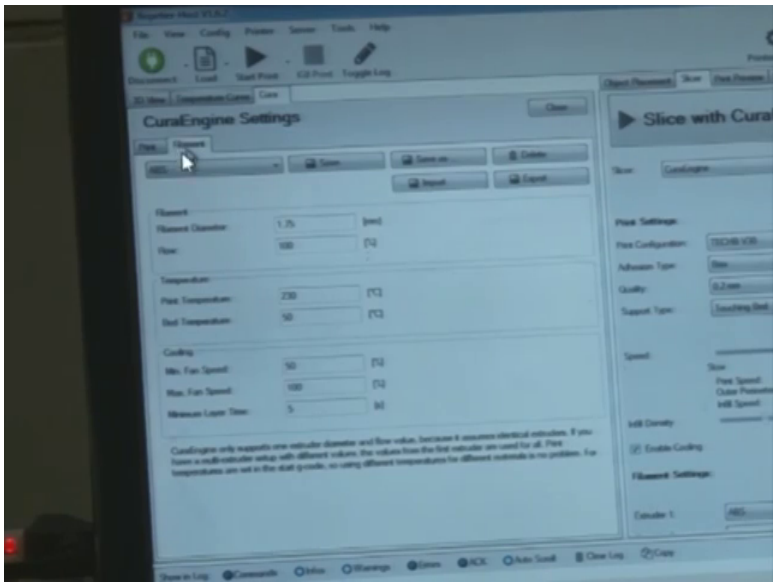
In this lecture, we will take you to the lab for the Laboratory Demonstration.

(Refer Slide Time: 00:21)



These are in a tab here that is advanced tab. Advanced tab is again used for manufactures like combine everything combine type A and type B kinds of the materials here. Then the second tab is filament after print we have filament.

(Refer Slide Time: 00:39)



Here, we have the feeding speed. Filament the filament diameter the general diameters size is which are available in the market are 1.75 mm and 2.85 mm. This filament that we have here that we are using to fabricate and to demonstrate it is 1.75 mm. The flow is kept 100 percent.

So, there is a particular property of the material the filament that is then what is length of the filament, what is diameter, what is the flow, what is the temperature, equivalent temperature. All these properties are there for the specific material for the specific kind of spool that we get from the market. So, those are sometimes given with the material, then when we purchase.

So, this is the filament. So, it is in the form of a wire again the diameter is 1.75 mm and when we see this flow is 100 percent; that means, whatever input we give whatever speed, it is feed input we give it will run 100 percent in that speed only. So, whatever we are going to input. So, it will work in that according to that only.

Now, next we can talk about temperature. To what temperature it will deposit at the bed? What would your temperature of the bed? Print temperature is 230 degrees and bed temperature is 50 degrees. So, it is kept at least 50 degrees in general. 230 degrees is a temperature we will discuss about this and well.

(Refer Slide Time: 02:36)

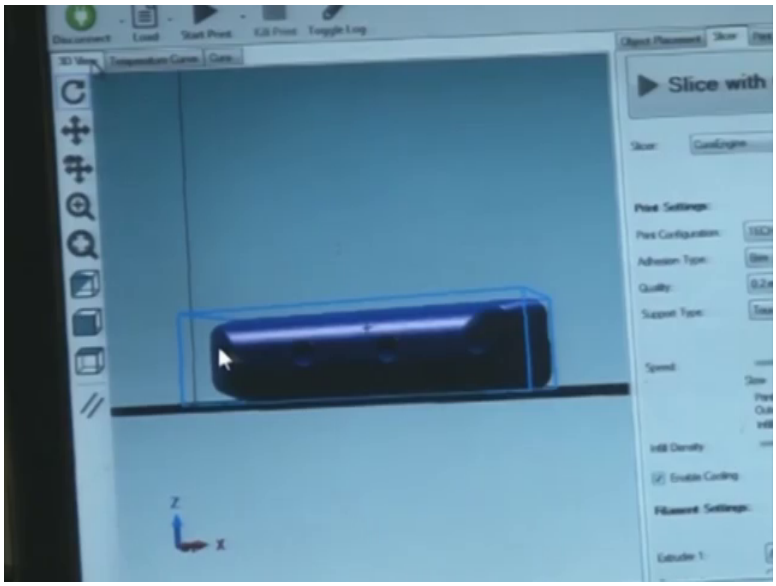
This is the one more thing that we mentioned earlier; there are 2 fans; fan 1 and fan 2 here. The 2 fans here fan 1 and 2 after fuses it becomes viscous and for the long time, it will de shape the CAD model, if it is not properly controlled with a certain temperature.

So, after depositing, this fans are responsible for the cooling and solidification of the layers. So, that is why it is mentioned in the software cooling. So, minimum fan speed

maximum fan speed; there are two fan speeds here. Minimum fan speed is kept at 50 percent, maximum fan speed is kept at 100 percent and minimum layer time is 5 seconds. It takes 5 seconds to cool the last layer of what was deposited earlier. So, in 5 seconds till it will take care of cooling, 5 seconds is time set in that.

This is called curing of the metal. This is called curing of the metal when we do cure. Cura word is also derived from this curing.

(Refer Slide Time: 03:47)



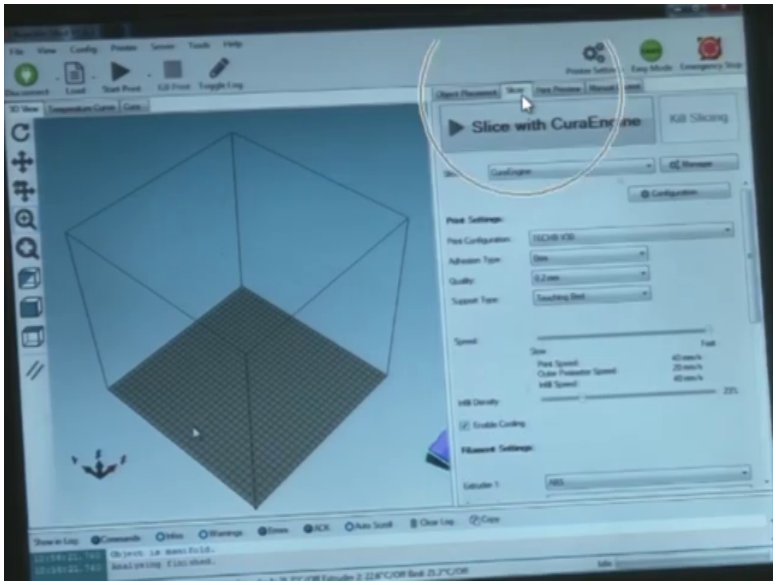
So, this Cura you know, we have now the broad undertaking of all the Cura software.

(Refer Slide Time: 03:55)

So, this is temperature curve. Temperature curve we can see the temperature or of the output of the bed it is from 0 to 100 if we say. The output is the output, it is coming actual temperature here what is the output coming and the output of the extruder, then in it can just show the curves when we actual do machine, when the machine is running.

This is 3D view this is temperature curve and this is Cura. Now, this is in 3D view we can zoom in and zoom out this is zoom out, this is zoom in, this is moving up and down zooming out.

(Refer Slide Time: 04:36)



Then isometric, then front view, isometric view and other axonometric views that we can see here this is top view, this is front view ok. This is now asymmetric view. So, we are still working in this tabs slicer; we are still working in slicer only.

So, as we discussed before, the slicer is the most important tab what I feel in this software when we need to decide most of the parameters are decided here only. So, print preview is just the preview and we can do manual control is when we need to control the machine manually. When we do not trust the G-code or this not the question of trust, it is about when we need to make some modifications according to our requirements. So, as we can machine it in lesser time or we can induce some new features then manual machining control, but whenever we do something manual then manual machining control can be done.

So, we have discussed object placement in slicer. So, we have discussed like this is a copy command you know the placement and mirror command, rotation command; all these commands whatever we have used for the proper alignment. Now, in slicer we have a few things like Adhesion Type. What is the adhesion type? Adhesion type here that is given is brim. Brim configuration, there is a part of the kind of the machine model that is put here the adhesion type is brim.

What is adhesion type? Now, here we provide a little bit more material. Here we use a little more area to deposit the first layer. For the very first layer, it will be bigger in size of from the CAD model what we have whatever we have obtained and what we actually going to print this layer the first layer would be bigger. So, but it has to be only first layer the adhesion type is the first layer. First layer where has it is the brim or it is raft we can.

After this first layer, the second layer would follow the CAD model ok. So, we can have 2 major kinds of skirts along our model; like this is the model, the skirts along this can be built this skirt can be brim or raft. So, this way this play very important role because this layer, we will stick to the machine. It will stick with the machine this table top; this hotplate, if they stick here. So, if we do not provide any extra material to the periphery of the job may stick on the bed and we may lose some features on the base of the job.

So, if I we can use this feature only when we find that more material has to be added that this job is necessary when more area is there. So, if we say none, it would not have any brim or raft or it might decide brim. So, if we that if we are very much conversion that this specific type of job would need a brim or raft, brim or raft or both the kinds of skirts.

So, when should we use raft or when we use a brim what are advantages that they with each other that in that we can say a raft would allow for better adhesion for the whole print, as raft attaches the printing surface and the print attaches to the raft. Like if this is a print and this is the print that we are need to produce; I need to produce this model. If there is first layer that is throughout this throughout the base at the whole base that is raft, brim would only at the outer periphery.

So, this brim and raft can vary we can see that whether raft is required or brim is required depending upon the material that we are producing. So, brim does not help a lot with layer adhesion it as it is only one layer; raft can even be more than one layers. Because normally when we use raft, it needs a nice looking the first layer like in general like we

are talking about the first layer already adhesion type, but raft can be even more than one layer brim is only one layer always.

Rafts are primarily used with ABS material to help with whapping and bed adhesion. They can also be used to help stabilize models with small footprints or to create its strong foundation on which to build the upper layers of your part. The raft a lot of work is being carried out lot of research is being is carried out in raft and brim design like to decide how many number of years optimizations is happening in this. So, 100 of different machines are produced to minimize because raft or brim whatever we have raft and brim would be an extra material that is wasted at the end. So, this wastage has to be minimum, but the quality of the product should not be compromised. So, we can decide between two these two ok.

About brim; what is brim? Brim as I said it is just at the outskirts of the job. So, brim are brims are often used to hold down the edges of your part which can prevent whapping and that it also helps in adhesion with the bed. It is actually known to the manufacturer that the quality of the product vary with using brim or not. So, in general brim command is referred into that like to reduce the wastage of the material.

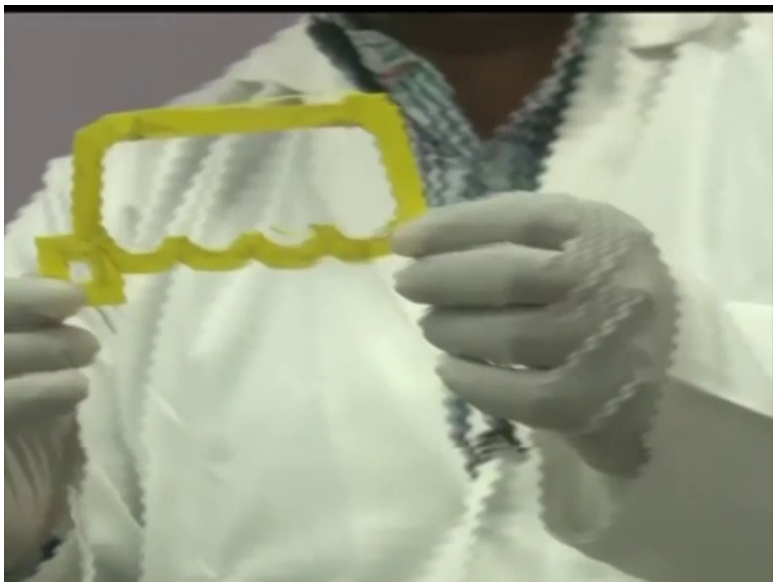
So, the second layer onwards would follow the CAD geometry. In this specific model, the brim and raft are only the first layer as I mentioned before, but in certain machines raft can be more than one layers as I mentioned. So, we have just kept as kept it as none that is we are not getting any brim or raft here. So, whatever we do whatever we dimension we give, it will just start manufacturing the component without depositing any other layer here.

So, like you can also see brim none disables this function; brim as a single layer high area around your object which can easily be cut off afterwards. Brim is required if objects whap at the edges if the edges are more important. If the edges are more important like this edge, this edge, this edge, edges are more brim is used.

Raft adds this is the help that is given by the software; raft adds a thick raster below the object and a thin interface between this and the object which can be removed after printing. This is required if you have a bumpy bed or such a small object that would not stick well during the print. So, I think this makes it very clear.

So, normally we do not use this feature brim or raft we were just kept anywhere, we can use this feature only when we find that our job, whatever we are going to print is perfectly flat at one end that is rectangular, cuboidal or cubical or at the cross section or at the bottom of the job, then we can use this.

(Refer Slide Time: 12:05)



Now, are going to demonstrate this, but we are going to clarify you that this one thing this is brim. So, this is brim that is taken off from the job that is prefabricated. So, this is the model that would be developed. So, this is the pocket; this extra material is brim, this extra material is brim ok. This whole material is brim.

So, this is taken off from the for the that was manufactured before. So, this will stick with the bed properly and this enlarged area. This is the first layer, this will provide a good holding with the bed. So, we are talking about an selection brim here in the software. So, when the nozzle had starts printing, it actually it is the actual model that we want to print, but it will deposit a little extra material to produce.

(Refer Slide Time: 13:00)

This first here this is actually the brim, this is the model that we have. This is the model that is produced. So, it will deposit the material the little more material. So, this is this material was produced in this way this. So, this extra material is brim, also we have more extra material here those are supports that we will just try to show you how when we remove them.

We can remove brim by hand very easily, manually and we can get the finished final product. If the CAD model is flat it is flat, it is ok; if it is not flat; if it had some inclination as I said, what would happen? How would the machine take care of the printing? It would provide some support. Now, the printing would take a place.

Now, we can see this profile that is created here. It is a mesh; this is not a solid profile. This is just a mesh that is created because it has support some material. Now, this support we are trying to remove this then. We are trying to remove this support, this is the both can be removed by hand. So, this is a support material. In this case, only one spool is used and both the model and the support materials is post spool that is where the color is also the color of the spool is; obviously, one that is yellow only. So, it is done by a single spool here.

So, we can even have the support material that is a little cheaper because the support material is not the final function that we need to present that we need to provide. So, this is the potential or the beauty of FDM technology, that if there is any inclination or any

pocket or any profile inside this, the machine will program on its own and deposit the material. Again this is what we mentioned earlier the quality that is at what quality we want to print that is 0.1 mm, 0.2 mm or 1.0 0.15 mm support type again that selection is there.

Now, this is printed this product is printed with the quality 0.2. So, they are fine they are a little lines. So, these are fine lines we can say fine lines or fine and course are subjective terms, but according to this machine these are coarse lines 0.2 mm. So, if we make with if we make job is 0.1 mm; these lines would be a little finer. So, the smoothness will be better.

So, we can select as we mentioned before that 0.1, 1.0 0.15 and 0.2 mm of quality there are 0.2 mm per slice what is the thickness of the slice that we are selecting. So, again the term quality is here like we said quality 0.2 we need to select the quality according to the kind of the finish we require and also as I said the quality and speed are countering each other. If the quality would be higher the speed would be lesser because 0.1 mm and 0.2 mm; obviously, the speed would be 50 percent of 0.2 if I use 0.1 because 50 percent of the layer 50 percent size is been deposited at each layer at each.

The support type as I said as we said we use the word support that is if there is a any inclination or any pocket any profile we need support the support type we can again play in 3 modes – none, touching bed or everywhere touching bed means it would just like to touch the bed. For instance if this material is here and this is my bed, it would just like to touch the bed. So, touching the bed it would just apply support wherever it need to touch the bed just touching bed.

So, which we need by higher quality we can even use this option everywhere otherwise no support. Also you know it is given none disable support, touching bed is the most common setting it will only create support where the support structure will touch the beds. So, everywhere creates support creates even on the top of the parts of the model, even on the top even if it is not touching the bed, it will create support. So, if we see know that our CAD model is good enough and is flat we need not to have any support we can choose none.

So, if I am not aware or we do not have any information about that whatever the CAD model would be from inside or it what their different intricate features so, in case of

complicated very highly complicated CAD models. The if the face is there is no flat shape or shape anywhere, then we need to set the commands to just this command just this option to everywhere. So, we were talking about the supports type and the support type is set to everywhere.

(Refer Slide Time: 18:35)

This is one of the jobs that is a kind of complicated. So, you can see that number their multiple shapes here. So, in this CAD model; in this CAD model if you see, in this CAD model we use everywhere command because it is profiled everywhere like just not touching the bed. For instance this is the model as I like I just said it is touching the bed, if there is another model for instance another thing is in fractured here, this portion the portion in between here is not touching the bed, but it would provide support here as well because it has to produce this thing as well.

So, in case of touch bed, it will provide support here only in case of everywhere we will provide support here as well which is the which is complicated we use the option everywhere. Now, the software is responsible to fill the support wherever it is needed. So, it is the art and skill both. You know in manufacturing as we know the CAD, the Computer Aided Design there different softwares are there, but whatever the software that is just software might be quite intelligent, but it is all the all designs come from here it has come from the designer. So, this is the always a an art associated in designing than besides the skill as well.

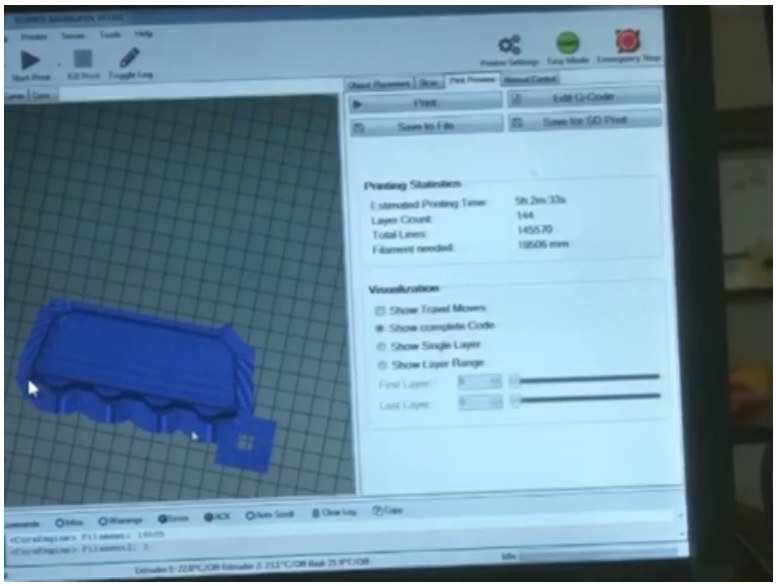
So, this is a, we are talking about the slicer and we were talking about the support tab and speed. As we discussed the speed print speed as it is fixed to 40 millimeters per second, outer parameter speed is fixed to 20 millimeters per second and infill speed is again 40 millimeter per second; we can make it fast and slow using this tab as well. So, again print speed is 40 millimeters per second, outer parameter speed is 20 millimeters per second infill speed is 40 millimeters.

So, there is an as a check box here enable cooling. So, if we enable this check box, it activates cooling fan when while printing. This can be significantly improve the print quality especially for the PLA in bridging and overhangs. It is specifically mentioned for PLA. If we enable cooling the cap fans would run when we print.

So, there are two nozzles here ok; extruder nozzle, extruder 1 and extruder 2 ok. Extruder 1 and extruder 2 two nozzles are there. So, accordingly we have two nozzles here; extruder 1 and extruder 2. We can put the material what is the kind of material we have put it ABS for extruder 1 and ABS for extruder 2. So, we can just input the kind of the material that we actually have. So, as the as we said we cannot manufacture multi color jobs with the, this machine because we have only two nozzles capacity as two colors can be produced.

So, we are just trying to tell you that how slicing would be taken care. So, how do we do slicing? Slicing is just one click away. Now, this big button here Slice with CuraEngine after making all these settings in slicer this really big fit button, the slicing is just one click away. When we click this button slicing starts, now the slicing starts; you can see that it has made a brim by itself ok, as it might have found it important. The machine is quite intelligent to do this; the software is quite intelligent to decide this thing.

(Refer Slide Time: 22:37)



So, this brim is you can see. This cure is here. Why this cure is here? This cure is for holding. We need to hold the job and take it off. It hold the brim; hold the brim layer and take it off ok. This scale is for the holding and the other portion this outer portion is this outer portion is the brim because it has to it would like to save the edges or the make the edges proper brim is only at the edges it had been rough, it would be along all the surface complete surface of the job ok.

So, this slicing now it is showing the properties. Now, it is showing estimated printing time estimated printing time is 5 hours 2 minutes 33 seconds. So, it is giving the data this is print view after slicer it has to went to the next tab here that is print preview ok. In print preview, it is giving the information that this kind of this is the time that would be taken 5 hours 2 minutes 33 seconds total layer counts would be 144, total lines would be 1455701, 145000 lines would be drawn. And filament that would be needed would be the length of the filament that would be needed would be 19500 mm.

So, this information is displayed here. So, in this print preview once we start click this button print a we can show, we can say see how show travel moves, how the layers would move. Now, this sky blue lines indicate that we will show that it will print material in this way. We could go with the single layer also and show layer range, show layer range ok. This is the first layer, how it would print? Look it is it shows, it is showing travel moves here showing travel moves, how it will travel when we will actual

do will also try to show you how it moves and it is showing 3 layer range. Now, we can just span through the window show compete code.

Now, there is a button here or not button there is a radio button here. Among these three we can select one Show complete Code, slow Show Single Layer, Show Layer Range. So, Single Layer we are just shown you, Show Layer Range is also shown now; Show complete Code, this is the complete code complete; code would be like this the material will come from this side ok. It will start machining, it will start producing actually machining when I say machining in general when we think of what we think of machining is machining is subtraction, here machining is addition material addition happens from here this is produced.

Now, let us click Print. Before we actually do print we have to also prepare the bed, prepare the machine and all those things the machine has to be set up properly before we actually start because once we click this it will start printing. Now, we have made certain settings, we have made certain inputs here and some settings are there the CAD model everything is there we can save this to a file; using these thing save to file ok.

Also if we have deep information or deep knowledge about the G codes about the C and C programming we can add or subtract or added the features using Edit G-code. XYZ movements can be controlled using G-code programming system. So, this is G-code editor ok. After Print Preview we also have G-code editor here. So, we change the CAD profile that what we need to print without printing without using the CAD model.

Now, the last tab is the last tab is manual control. The main manual control what happens we can control the machine manually you can see X, Y direction here, X and Y direction can be controlled manually, Z direction can be controlled manually and also this extruder can be controlled manually. In manual control, it is very important that it starts from beginning if this individual we are doing manual control it starts from beginning and it ends at finishing as well.

So, like movement of the nozzle in manual control we can move the axis manually, here we can play with a setting changings and we can change the temperature as well here. So, before printing we also need to set a temperature the set this is known as soaking temperature ok. The nozzle temperature that was kept 230 degrees so we have to activate.

Now, this controls the extruder temperature typically the temperatures for PLA is from 180 to 220 degrees and for ABS it is it a little more than that. So, we are using synchronous extruder for both printing; printing that is the actual model and for support. So, we are activating this. So, this red cross means it is not activated. This that cross is taken off means that line is taken off that is it activated.

Now, extruder 1 extruder it is now starting started it has now started heating might be once we have activated the temperature 200 degrees. It will now heat up to this temperature 200 degree temperature actual temperature we can see here when it is heating temperature curve here, when we will open here.

So, bed temperature bed temperature is also activated now bed is also heating, it has started heating. So, the bed temperature again for PLA the bed temperature is 55 degrees and the fan is also switched on manually. Now, there is another way if we go to Print button here, when we click Print all these manual controls that we have done manually these will be activated or enabled automatically because bed has to be heated, the nozzle has to be heated, fan has to be kept on, all the settings that we made before would be activated when by just clicking of this button. So, this is the automated mode.

So, the use of manual control is that we can display the temperature, we can alter here the temperatures. But, in the automated control we cannot alter once we fixed. So, we can increase or decrease the temperature. See we are increasing the temperature of extruder it was 200 we are increasing it to 230 ok, we increase it to 230. So, sometimes we also play with the atmosphere temperature. In the summers the temperature room temperature is generally here in Kanpur is from 37 to 43 in average. So, it is quite hot in extreme summers and in winters it even come downs tool 5 degrees or 6 degrees.

So, depending upon that the temperature is also set. If a temperature room temperature is lower this temperature is also kept accordingly. It is winters here at this point of time when we did this demonstrations. So, the temperature is kept a little high. So, because the room temperature is quite low here so, the filament temperature is made a little higher otherwise it what should be kept 200 degrees here.

We will meet in the next lecture. We will discuss further about the course.

Thank you.