Sensors and Actuators
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Lecture – 56
Introduction to CAD Modelling-I

Hi, welcome to this particular module, and this again experiments the component of our lecture or theory class. Today's module will be focusing on how to use a 3D printing technique. Now, 3D printing as you know has evolved several tens of folds from the last few years. And now, it has been people are exploring the, you know the beauty of 3D printing into several different applications right from automotive to medical devices.

If you recall in one of the lectures we have talked about a biochip that can be placed within a 3D printed module to understand the tissue properties I thought that why do not we also include a small part of the lab, where you will be actually shown how you can use a 3D printing module, how you can design this CAD module Suppose you want to design let us say the casing for this particular microphone, how this casing can be designed First, we have to design it.

we will show it to you in the lab how can you design this casing, I will request Sahid to discuss this 3D printer and take it over. If you have any questions again you are free to ask me to let me again remind you there would be a sudden live session, as a part of this particular program. Apart from living sessions, there are there is a forum in which you can ask any technical questions, free to use the forum in the best possible manner, you take care and I will hand over the lab to Sahid, bye.

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Today I will show you how to make an enclosure for a switch. Is a symbol switch, inside there is a light. first I want to measure the dimensions of the switch to make an enclosure. for that I will be using vernier calipers, this is a digital vernier caliper.

the width is seen that is 12.48 mm. 12.5 mm we can approximate that, we do not need that 0.01 an operation. And the height is 20 mm. the width is 12.5 and the height is 20 mm.

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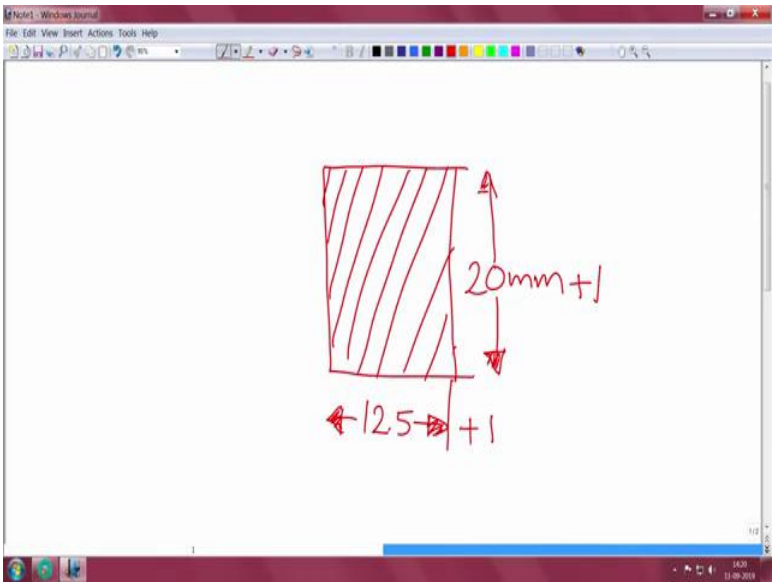

As we measured, we have a switch; the switch has a height of 20 mm and a width of 12.5 mm. this is the exact dimension of that switch. for making an enclosure, we need a hole which is greater than this dimension, this dimension 20 mm and this dimension 12.5 mm. Otherwise, the hole in the enclosure we are making and the switch dimension will be the same.

practically, there will be an overlap and the switch will not go inside that hole. we can make the whole with 20 plus 1 mm that is 21 mm height and 12.5 plus 1 mm that is 13.5 mm size. And this is the size of the hole in the enclosure in the size of the hole.

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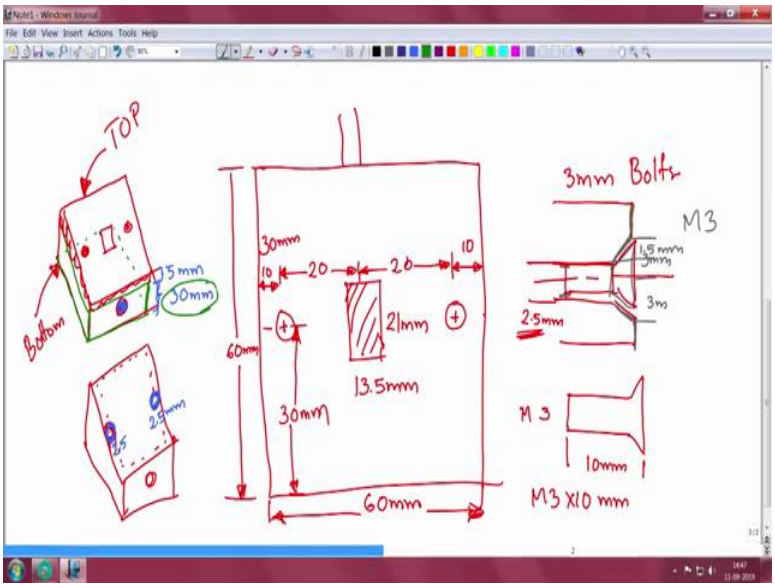

And we will be making an enclosure. suppose this is the enclosure, the top view of the enclosure here the switch will be, the switch will come. And we can take the connections through here on the side. And we can give two screws here and here. for designing that, we have to fix a dimension before going to use the CAD software that will be easy for designing. this will be 13.5 mm and this will be 21 mm, and we can use 2 bolts to screw the enclosure.

for this much small enclosure, we can use 3 mm bolts. I will be using a countersunk bolt, this is a countersunk bolt; this is a countersunk bolt, this is the head of the bolt and this is the body. this will be the surface; this will be the surface, the bolt will sit into the; bolt will sit into the surface very easily and it will give a clean look.

this is this whole dimension is 3 mm and the bolt dimension the bolt the ammeter will be little less than 3 mm. And we will be 3D printing an enclosure we have to take care of the tolerances. we can give a 0.5 mm decreased dimension to the hole; that means, the hole dimension will be 2.5 mm. that we can directly screw this bolt into the hole and everything will sit together easily.

I will be using a 3 mm bolt with 10 mm length for countersunk bolt this is 3 mm that is mentioned as M 3 bolt. We can purchase from the local market saying that we need M 3 bolt. And we are the at the same time we have to mention the length of the bolt, I will be using a 10 mm bolt, bolt with 10 mm length. I can say M 3 into 10 mm.

And now we have to fix the dimension between the switch hole and the center of these two holes, here two bolts will come. from this center to here we can say 20 mm; we can give a just 20 mm and from here to here we can just use the same dimension as 20 mm. And now this is 21 mm the height is 21 mm. this total height we have to decide. this can be a, so let us say 30 mm from here to the center of the circle is 30 mm, similarly from the center of the circle to the top portion also 30 mm.

the total height will be 60 mm and the total width here one more dimension we have to fix that here we can say 10 mm let us say 10 mm here from the center of the bolt to the end is 10 mm. Here in this side also from the center of the bolt to these heads that is 10 mm. the total width will be 10 plus 20 plus 20 plus 10 which is also 60 mm.

Now, the enclosure will be a three-dimensional object this is only the top view where the switch and the bolts will sit. the enclosure will be something like this. here the switch will come and here one bolt and another bolt. From this site or this site, there will be a hole and through this hole, we can take out the wires, we can take out the wires.

And we can make the enclosure as two parts. Suppose we are cutting through here, cutting through this plane, then the bottom piece will have, it will be like this and this will have a hole and here this will be empty. If we have some microcontroller or some other circuit for the project we can sit at into this empty space. And for screwing this we have to do one more thing, here there is a hole in this top portion; in this top portion there are holes one and two holes are there.

But to connect this top portion, this top portion and this bottom portion we will be recurring a hole here. Like I said here this 2.5 mm hole, we have to make a hole with 2.5 mm here and here this hole with 2.5 mm dimension. And since this is a countersunk bolt and this is the surface. Let us say this is the surface and this is the countersunk bolt; this dimension is let us say 3 mm the bolt size is M 3.

we can make a curved surface not a curved surface, we can make an inclined surface here and here so that the bolt will sit into the surface easily. we standard this dimension will be 3 mm and this will be also 3 mm. from here to here this will be around 1.5 mm, half of the size of the bolt.

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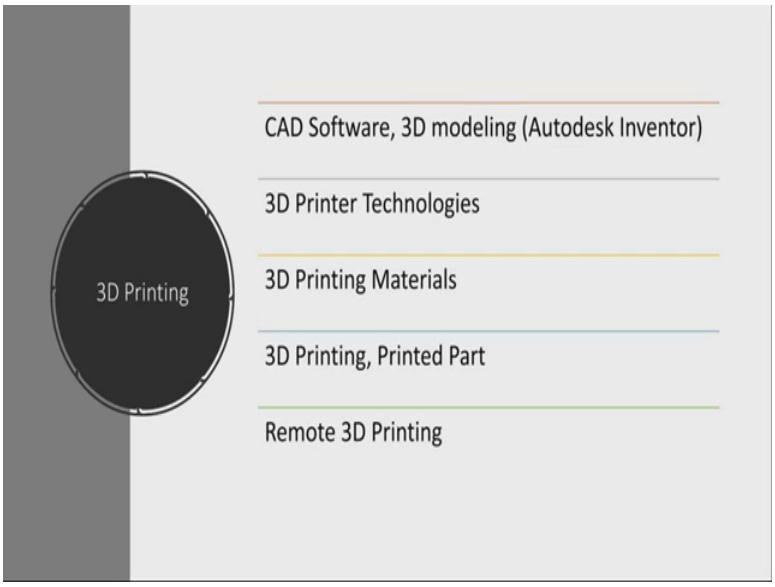

In this 3D printing session, I will go in deep with CAD software. here I will be using Autodesk inventor for demonstrating 3D modeling. And then 3D printer technologies; so that literally means how a 3D printed object is making, with what all technologies What type of 3D printers are there in the market And then the materials used in 3D printer different types of material in different types of 3D printers remote 3D printing.

normally for 3D printers small scale not industrial 3D printer, we have to sit near the 3D printer and we have to give the command for 3D printing. if we have a remote 3D printing capability, we can literally print any object from anywhere in the world.

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in the CAD software 3D model, I will be using Autodesk inventor software. In Autodesk inventor software, the student can sign up and can have can get a free license for using the software for 3 years for that you need an email address and after signing up you have to download that and install it. I will be showing a basic step for signing up and download. after that, how to operate that model in symbol and I will show how to make an enclosure.

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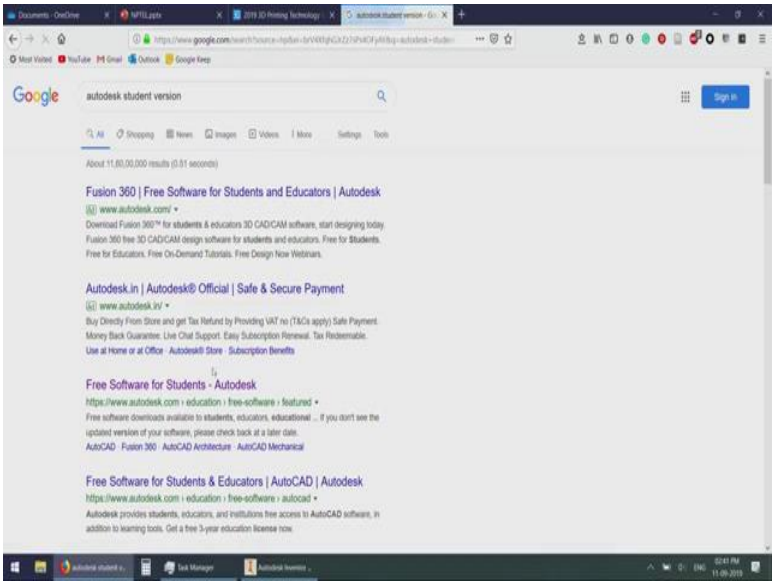

first, we can go to the Autodesk website. Autodesk student version if we search for that there is a Free Software for Students link.

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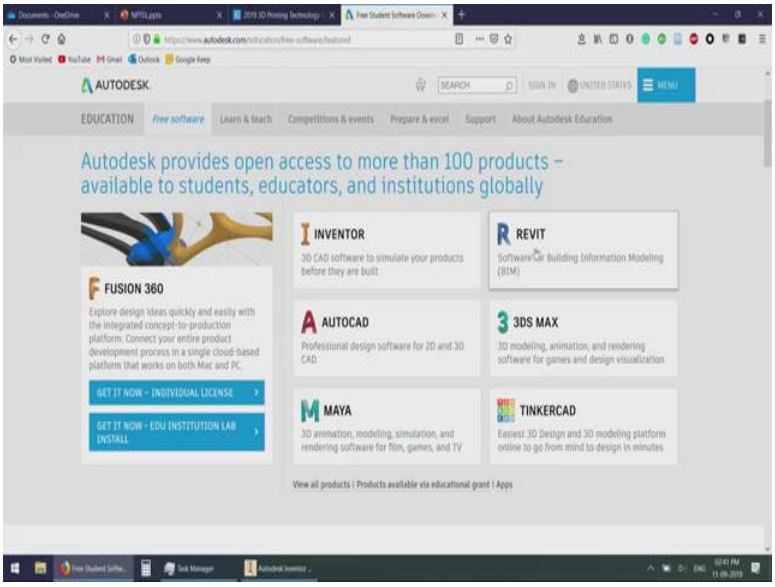

Here there is a couple of software Autodesk Inventor, Autodesk Revit, AutoCAD, Maya, 3DS MAX and Tinker CAD. I will be using Autodesk inventor for our purpose for making a 3D model.

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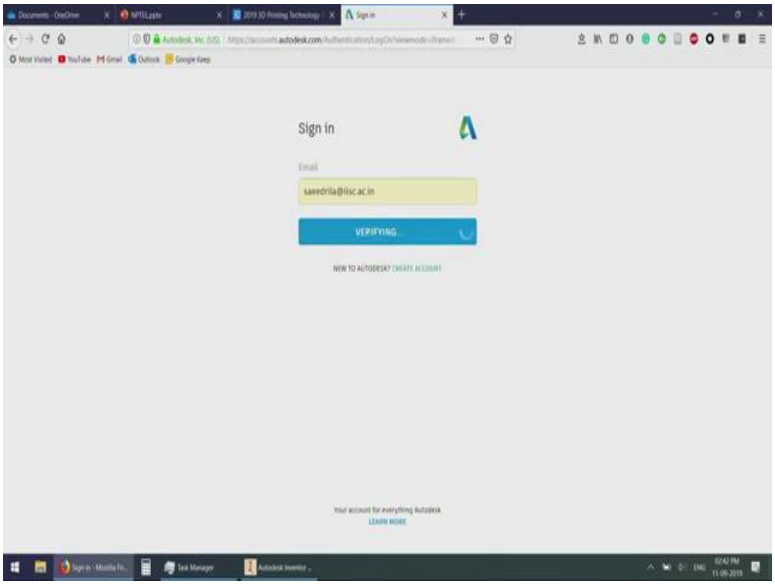

here click on the sign-in, I already made a student's account; making a student account is very easy and you only need an email address, then you have to verify that email address.

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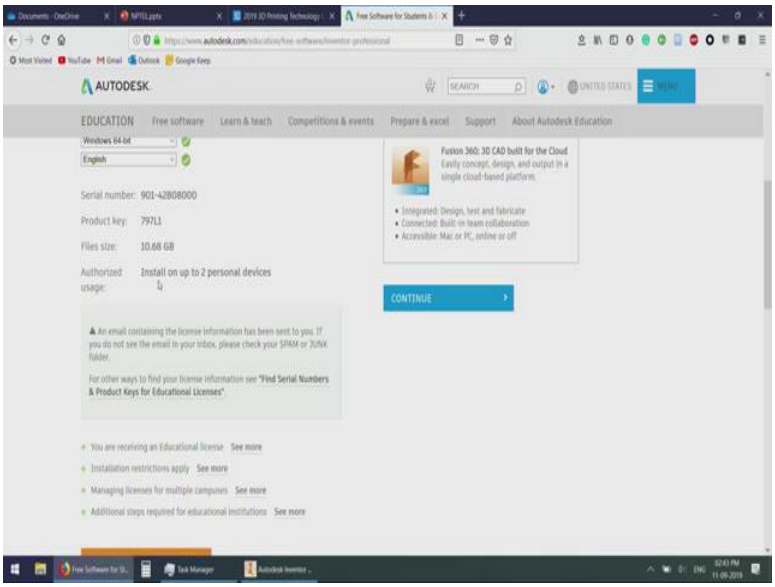

here I can select this inventor software and they will ask which software do you want Which version do you want Here to download now I will select or inventor professional 2020 and the software I am using is Windows 64 bit, and I will choose the English language. here you can see the serial number and the product key.

The download size will be around 10.68 GB. if you have an internet connection with good bandwidth, you can directly go to the install now option otherwise you have you can download it and you can use this serial number and product key to register your software. You can only install up to or two personal devices with this educational Autodesk inventor software license.

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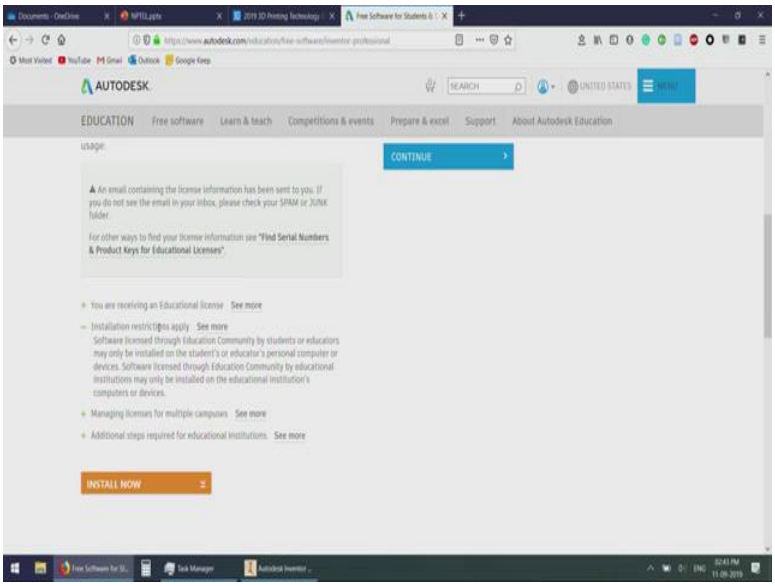

Here see there are some restrictions apply since Autodesk is giving you free software, a free software license. You can only install this on your personal computer.

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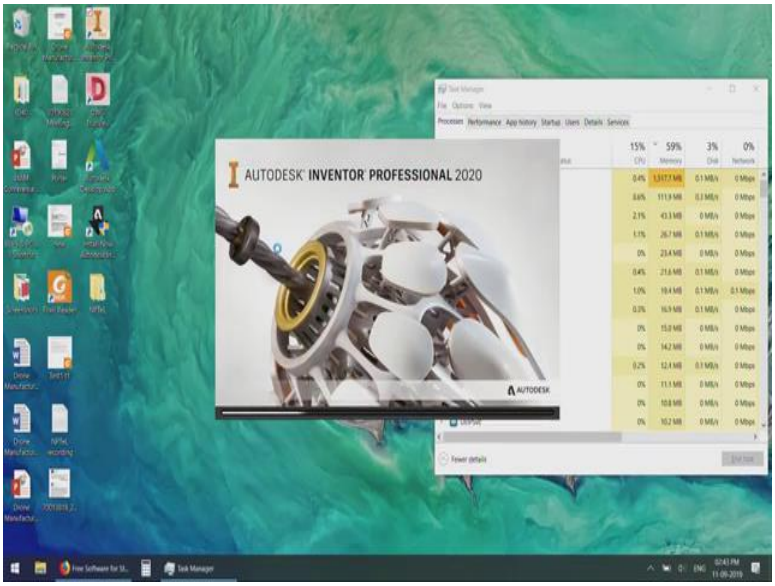

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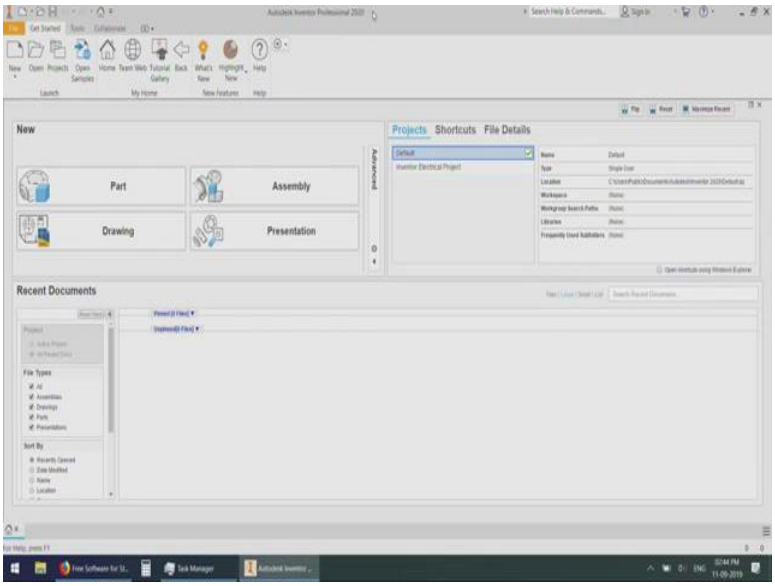

this is the Autodesk inventor professional 2020 window. Here we can make parts using the software or combine different parts and constrained to make an assembly and we can also make drawing for giving part inspiration to the manufacturer. I will be making a part I will be making two-part to make the enclosure.

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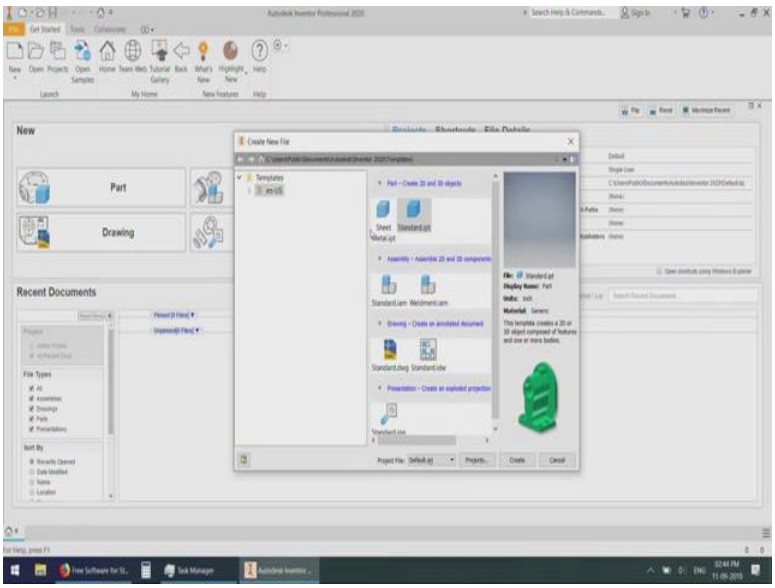

here in the new; menu new here I will be selecting standard, standard part here this is a standard part. If we want to make sheet metal parts it is easy to you is you can use this

sheet metal per option and then create. Now, I will select another part that is a standard part, click on create.

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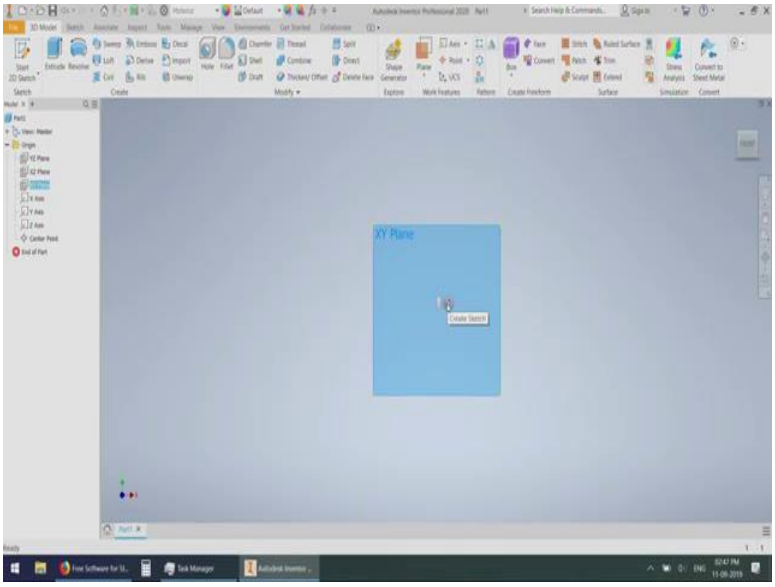

there are some basics to know before you are going to do any, you are going to make any object in 3D printers in CAD software. We will be dealing with 3D three-dimensional objects we have to think about how we are we will be making that Suppose if we want to make if we want to print a square on a piece of paper we will probably open the word and insert an object a square object.

And we can just draw it on the words word software and we can print it. But here this is a three-dimensional modeling software we will be having so many planes, three planes we have. These are the XY plane XZ plane and YZ plane. If we are if we just want to make a printout of as square we do not have to care about which plane it is, there is only one plane. Here we have an XY plane, XZ plane, and YZ plane.

The basic idea is, we can draw a two-dimensional sketch on any plane and then we can do some operation to extort. Suppose if you want to make a cube, we can draw a square in a plane in supposing let us say in XY plane and then we can extrude that to make a cube. here I selected the XY plane and then I am clicking here in the create sketch.

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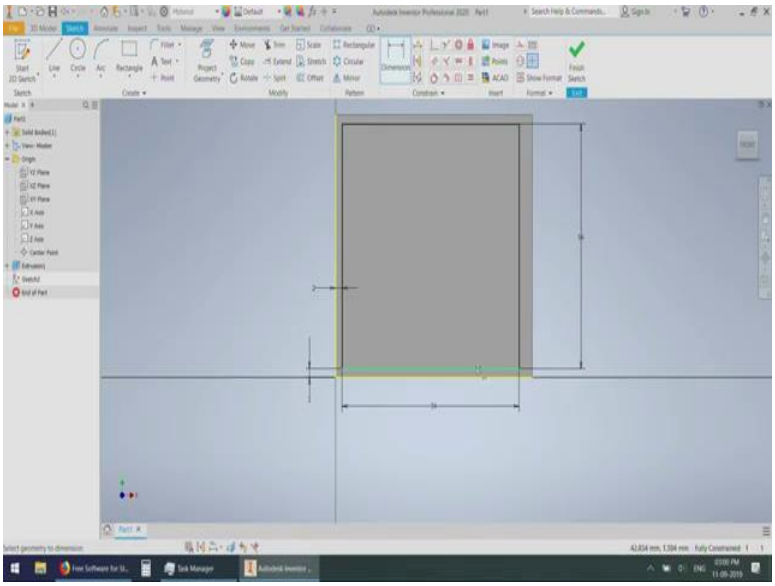

there are some options to create a line, to create a circle, to create an arc, to create a rectangle. I will choose the rectangle tool and nowhere when I am moving the cursor you can see the X and there is in that box. There are a dimension and inch, then Y in the next square there is a 0.185 inch.

hereby default in Autodesk inventor, the standard dimension will be an inch, now we have to change that we will be using mm. we have to change that for that we can just go to the tools and then document settings here units.

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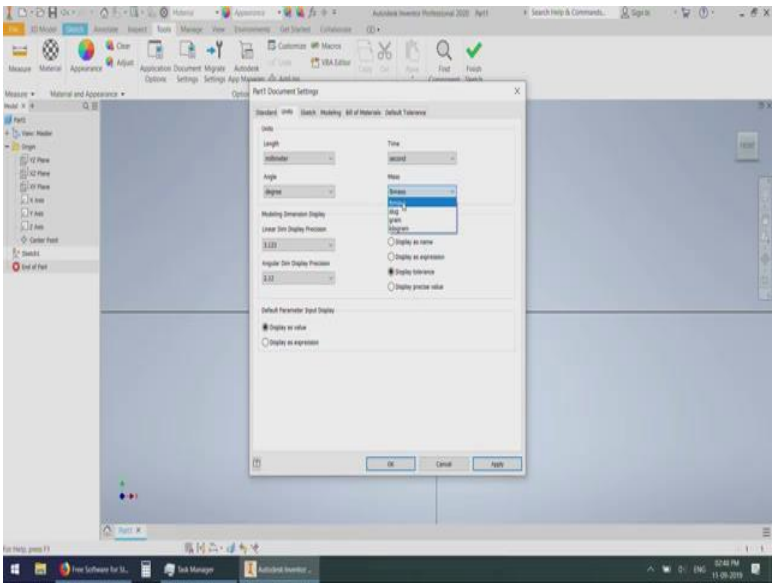

I will change the length to millimeter from inch to millimeter I will change, for the mass, I will change from pound to kilogram, I will just click ok, now it is ok. I will again come to the rectangle tool, now you can see X in the next square in the next rectangle 4.341 mm.

I will start from origin, it is easier to start from the origin then from somewhere here it will be easier. If you have some complex part or if you want to start from somewhere here also that is also possible, now I will start from the origin this is the origin. I will click here and I will go here and I will click here also.

this is the rectangle I have drawn. I have previously we have designed enclosure with dimension 60 mm height and 60 mm width. I will be giving the dimensions for giving dimensions I will click here for dimension and I will click here, this is the width. I will click here then the dimension value will come currently it is 24.786. I will click here now it will ask; now it is asking how much should be there. I will just enter 60, 60 mm then I click ok.

There are some basic moves here for zooming in and zooming out we can use the scrawled on the mouse like this. And if you want to move this sketch or if you want to move an object we can click the roll button on the mouse and we can move it. Now, we have to fix this dimension.

here if you see there is one there is something written as one dimension needed. To constrain this sketch, we have to define two-dimensions for a rectangle, two lines will be parallel and these two lines will be parallel. And we need two-dimensions to constrain that rectangle. I will be giving dimension to this also. this one also 60, I will enter 60 now this is fully constraint see here, this is fully constrained.

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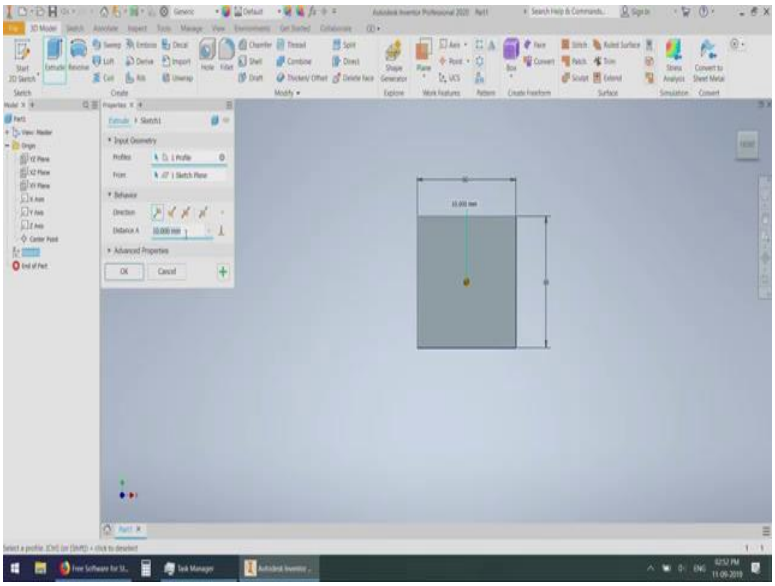

And now I will finish this sketch, I will zoom out it will be and this is this 2D drawing I made. I can change the position of the dimension, I will extrude from this sketch I made 60 mm length and 60 mm height, I will extrude from this. if I press the extrude button, it will ask how much height should I need

here we can write there are a lot of options here these are the default option. I will only change the distance; that means, the height I want. here we have two-dimensions; one the length 60 mm and 60 mm height and one more thing are we have to fix this height also; so here this height; so this much height, the bottom part and this much height the top part.

let us say we can give 30 mm height for the bottom part and say 10 mm, 10 or we can give 5 mm that is required that is only required. let us say 5 mm for the top part. I will be using this 30 mm. I drew this much, I draw this rectangle and I need this much height in this software. I will enter this 30 mm height, I will enter 30 mm height.

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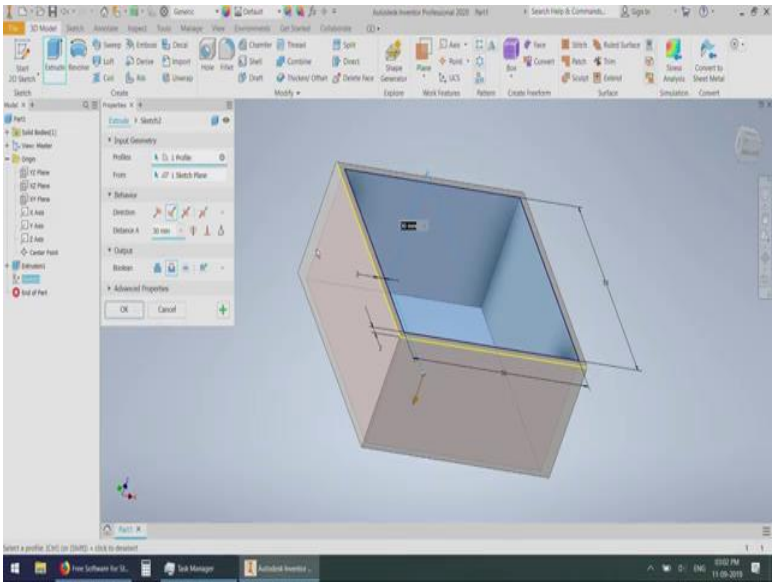

Now, we can only see a rectangle we do not know it is a 3D object or not. for viewing this as a 3D object, we have to rotate this object. For easy rotation, one option is we can use this option we can click here and drag it. Now, here we can see this is the 3D object. There is another shortcut for rotating, we can hold the shift key and hold the scroll button and we can rotate like this.

Then if you want the front view, we can just click here the front you will see the front view. Here this is a block, from this block I have I want to subtract some material and then we will have some space to feel the PCB or something we are making. for subtracting some material on how to subtract, I have to tell the software how to subtract. for that previously we have we draw a rectangle and then we extruded to make a 3D object.

Now I will select this from this surface I will draw another rectangle and I will subtract some materials from here. to draw a rectangle here on the surface, I will select this create sketch option. From here in this sketch toolbar, I will use the rectangle and from here to here, I will draw a rectangle.

Now, I have to give dimension to the rectangle I will choose the dimension tool and I will click here. And it will ask how much should be the dimension of this dimension of the width of the outer rectangle is 60 mm. we have to think about how much width how

much thickness the wall should have; now it will be an enclosure, so how much wall we need.

let us say we need 2 mm wall thickness, then here 2 mm is required here another 2 mm is required 2 plus 2, 4 mm. from the 60 mm width you have to subtract the 4 mm width 4 mm wall thickness. that will be 54 mm I will enter 54 mm value then I will enter the same value here al here I need 2 mm, here also I need 2 mm.

Here we can see this dimension is not equal to this. there is some problem. The thing is when we draw the rectangle we did not start from 2 mm from the surface and this is also a node 2 mm. we have to give that dimensional now we have to start to we need 2 mm between this line and this line. Here also when you see two dimensions are needed for the full constraint of this sketch.

I will just click dimension and click this line and this line. now it is 1.881 mm I will click here and I will enter 2 mm, I will surfaces 2, I will enter 2 also if this is 2 mm and this is 2 mm this should be 56 mm, should not be 54 it should be 6 56 mm. We can always change the dimension just by clicking the value.

here it will be 2 mm from this will be 2 mm see here and from here to here that is 56 mm; 56 plus 2 now this 58 then another 2 mm that will result in 60 mm, the height also is the same here 2 mm here 56 mm and then 2 mm. I will finish this sketch and I will use this extrude option, for that.

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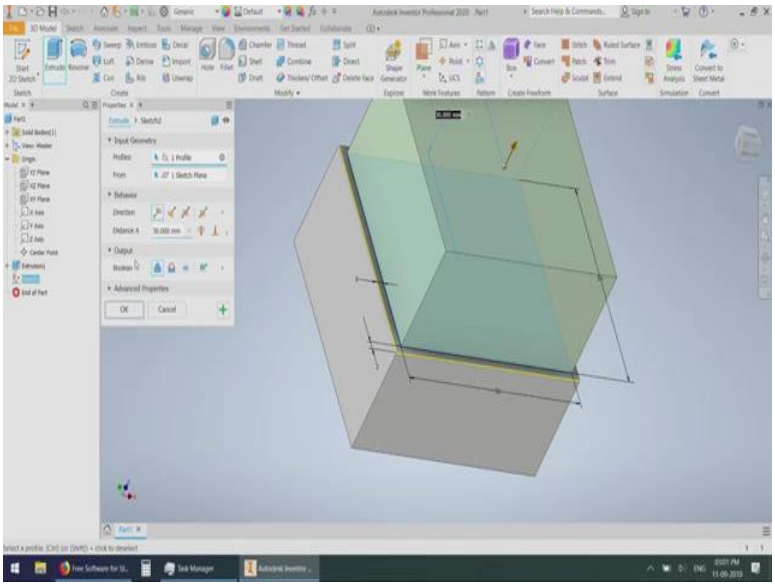

Here there is we have to change one default, change one option from default. Now, this is where you can see the output this is a function for Boolean join. That means if you are if you want to make another block, another cube on top of the basic cube you can select the joint.

Now I will be selecting this cut option, then here see be the final object. we are seeing a preview of that till now we did not click ok. this is a temporary view of the final thing. the total height was 30 mm and we need we want to make an enclosure. the bottom should not be hollow, the bottom should not be the hole.

here we can say we can change the dimension and I will enter 28 mm. that 2 mm wall thickness will be there in the bottom side al even I click here we got a box kind of thing like this, see this. Now, I will save this temporary file. here I will change the name to the Bottom Part.

The outer dimension is 60 mm and the wall thickness is 2 mm here, 2 mm here, 2 mm here, 2 mm here and here this thickness from here to here this also 2 mm. To measure that in the software, we can use this from the inspect toolbar there is a measure option. We can select two surfaces or two lines to check the dimension between them.

If we want to check this dimension, I can select this surface and the surface or I can select this line and this line. If I press only this line, it will show the distance of the

length of this line that is 56 mm; I will select this line and then this line. it is showing 2 mm, the distance between these two lines this line and this line is 2 mm.

if I select this surface, it will show it will automatically show the surface area we do not need that. Here it will show the surface area perimeter and this kind of thing. I will select the surface and then this surface. it is showing the distance, that is the minimum distance between these two planes this plane and this plane that is 2 mm.