Engineering Graphics and Design
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Week 10: Part Modelling 2
Lecture 3
Example 7

Welcome back to week 10 of Part Modeling 2. In this lecture we will be looking at one more example where we will cover splines and use of other tools in the inventor software.

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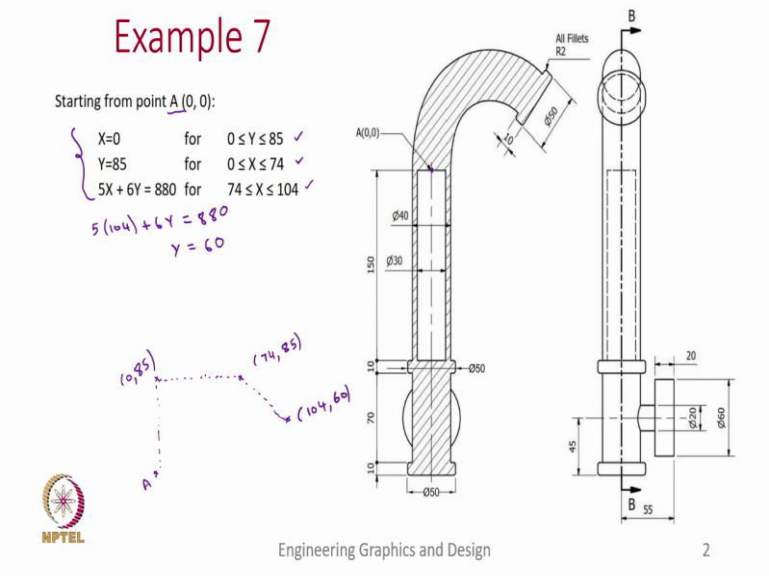

So, here is the example we have. Let us look at the features which are available on this example. Then we will be in a better position to model it. As we can see we have a vertical column and at the top of the vertical column there is a curvature. So, the column is bent. And this curvature is defined using a spline the equations of the spline are shown here. So, let us see these are defined about a point A. This point A is shown to be here.

So, what we see are these equations does not represent the spline but the control vertices of the spline. So, since we have three equations we understand that there are 3 straight lines in that control vertices. So, first let us understand the first equation. The first equation says X is equal to 0 between Y varying between 0 and 85. So, let us say this is point A. The first one is vertical because the X axis is constant at 0 and the Y axis ends at 0, 85.

Now let us look at the equation of the line for the second segment. So, in the second segment the Y is constant at 85 and X varies from 0 to 74. So, this is the end of the second segment let us say at the end you will have X as 74 and Y as 85. So, this will be the first segment this will

be the second segment and the last segment is varying between X of 74 to 104. So, let us plug in this X = 104 to get the endpoint into this equation.

So, which means 5 times 104+ 6 Y = 880. We will see that for these the Y will come out to be 60. So, X is 104 and Y is 60. So, let us say somewhere here. This will be our last segment and the endpoints are 104 and 60. So, this will help us to develop that spline curve the path is defined by the spline and the cross section here we know is a circle. In addition to this column and this spline, we also have towards the end this handle. We also have at the bottom this handle so, which we will also do once we are done with the vertical column with the curvature of the spline. So, let us get into the software and get started.

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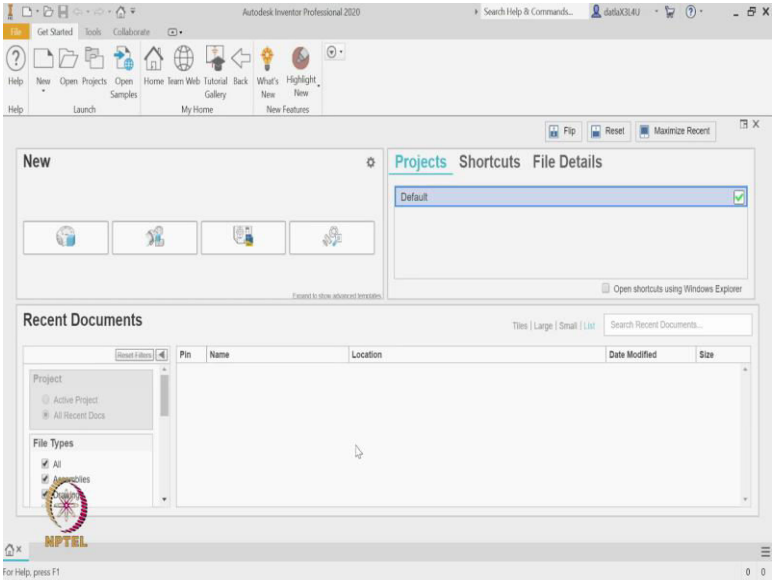

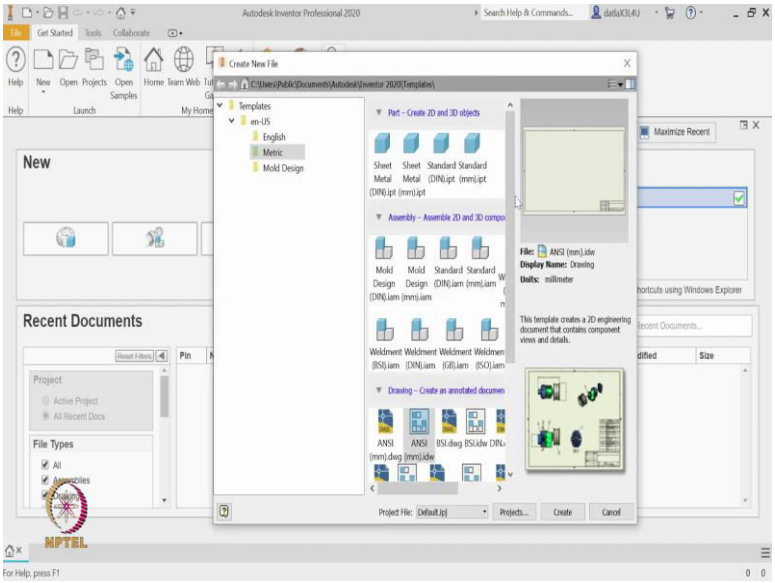

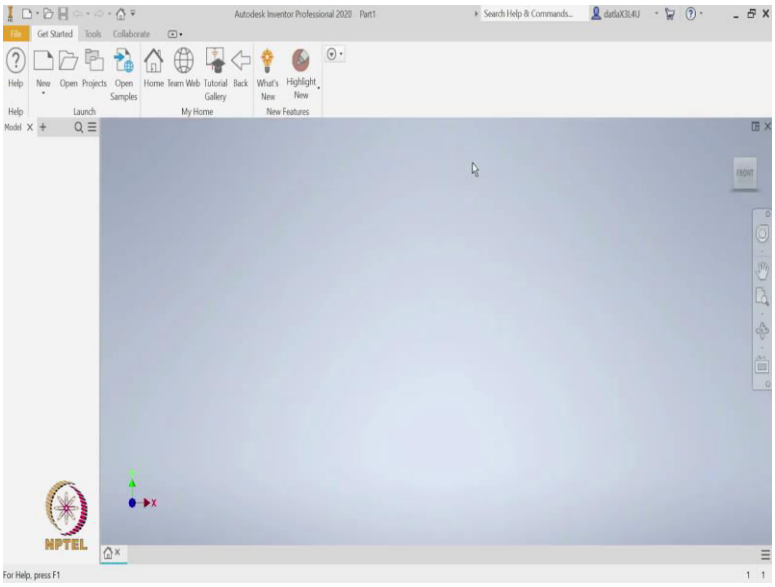

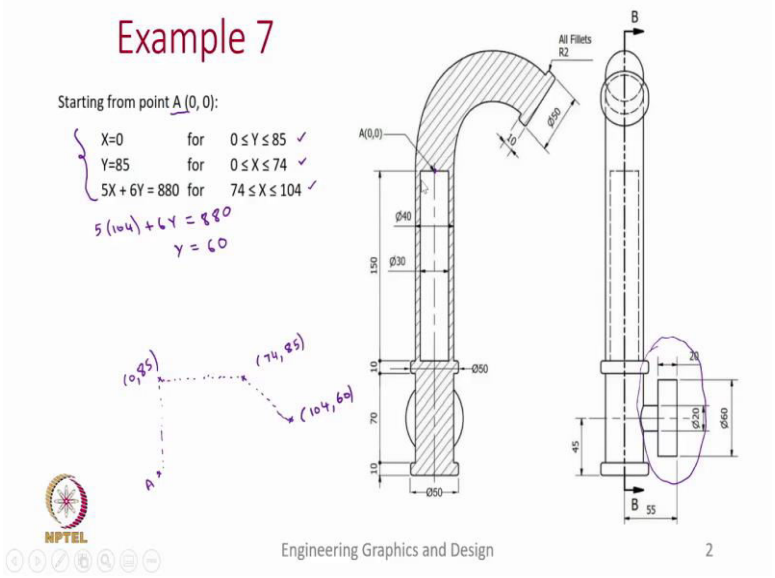

So, we are at the initial interface let us start with a part template. So, first thing what we will do is we will first create the spline and then the profile of it.

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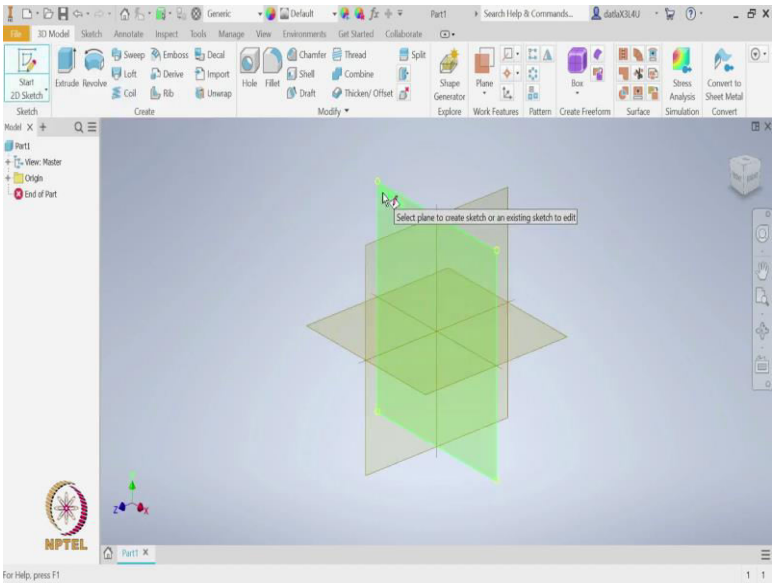

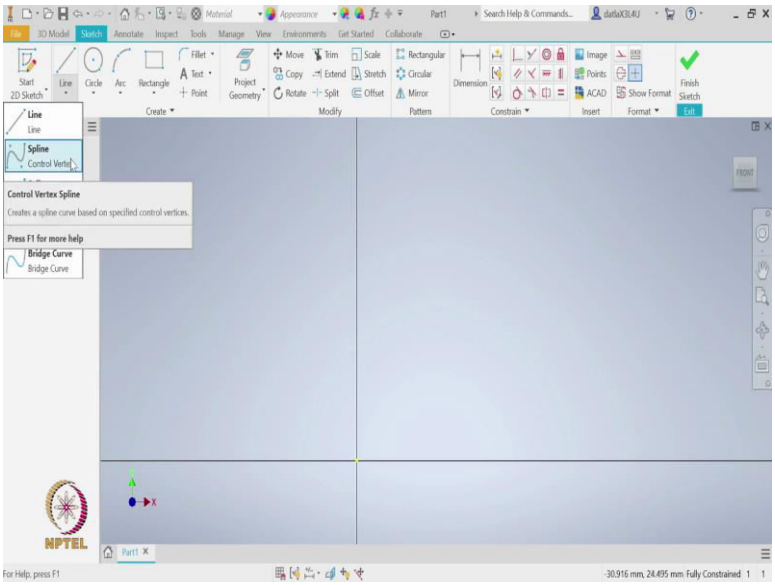

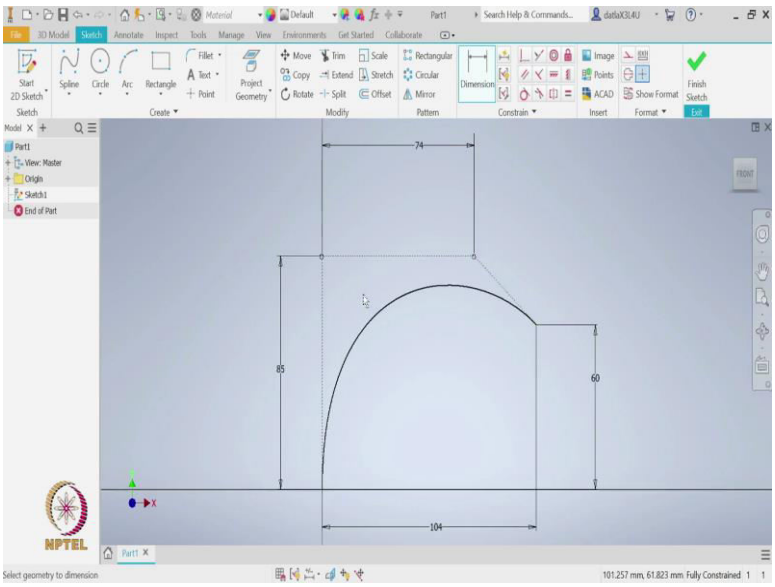

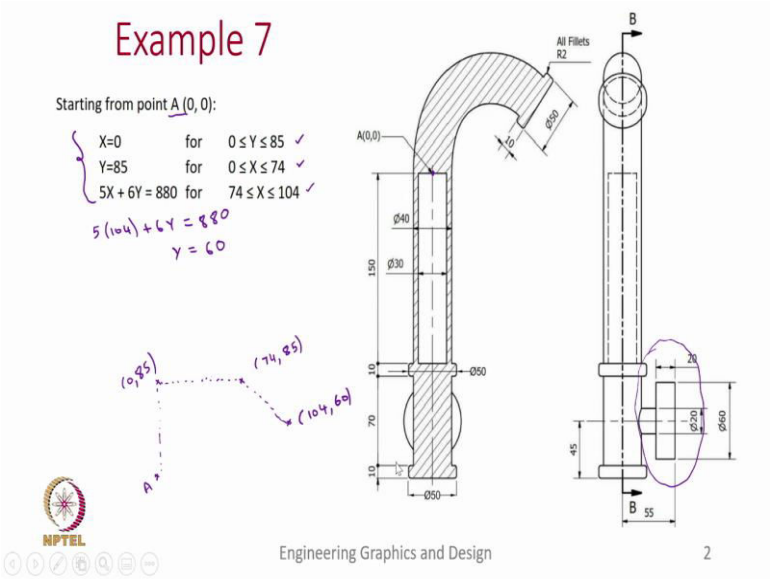

So, let us start with 2D sketch. Let us choose the XY plane and let us start to develop this spline for that we have to go to the line drop box and then the spline using the control vertex. The first point will be of the origin which is the point AA which is at 0,0. Second point we know is vertical. So, let me take a random length now and later I will dimension it.

For the second line segment we again know it is a horizontal line. So, that we will ensure that it is horizontal and then the length will be arbitrary as of now. And for the third point we will take an arbitrary point and later bother about the dimensioning. And let us click okay to finish the spline curve. Now, let us dimension it as per the equations we had in the question. So, we said the initial height is how much : 85 8 and 5.

And this length for the second segment is 74. And lastly for the endpoint 104 and 60 with the origin. X dimension is 104 and Y dimension is 60. Now we are done with this spline portion. So, instead of stopping at the spline portion, what I will do is I will also continue with an vertical line because I see that the cross section remains constant not just in the spline but all the way down till let us say this 150 + 10 + 70 because the last 10 is at a different diameter. That we will do it later. But at least, this 150 + 10 is 160 + 70 is 230. So, for these 230 let me draw a line So, that the path is vertical, I can use the sweep command to capture that solid.

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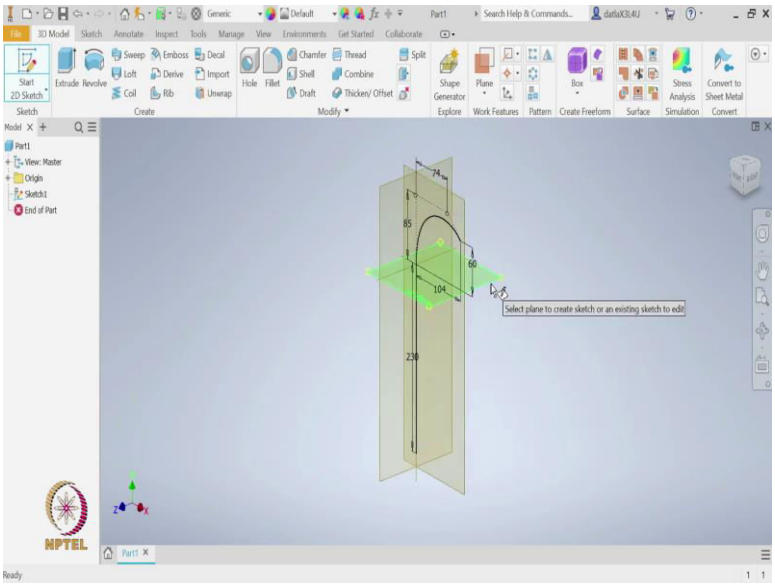

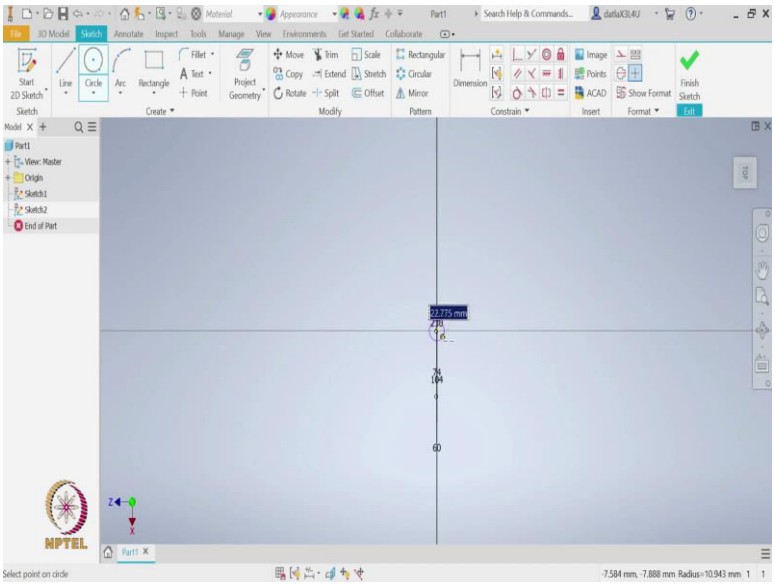

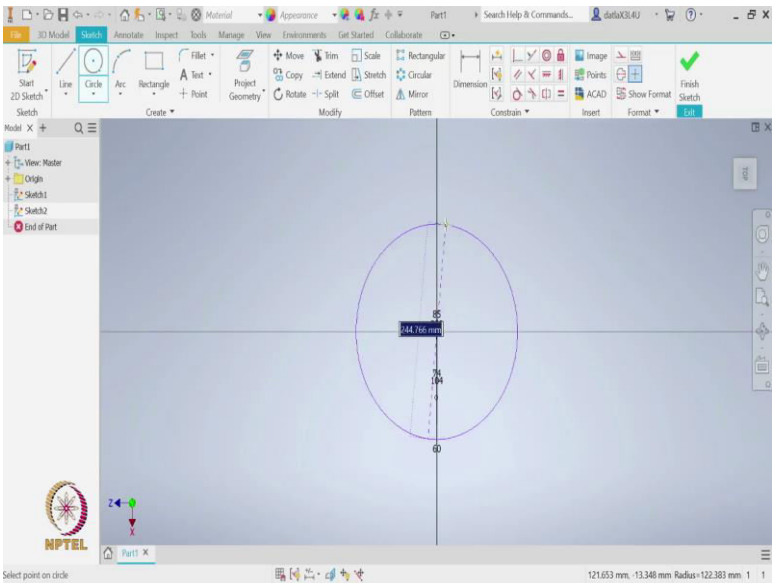

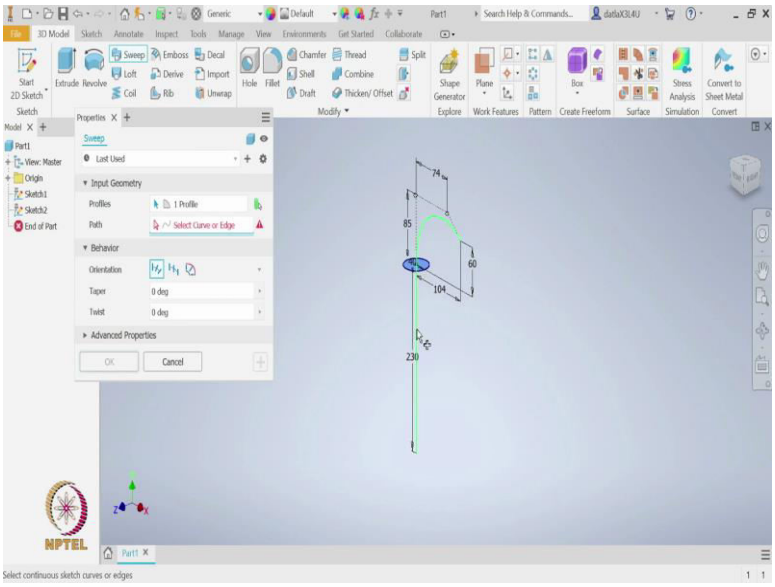

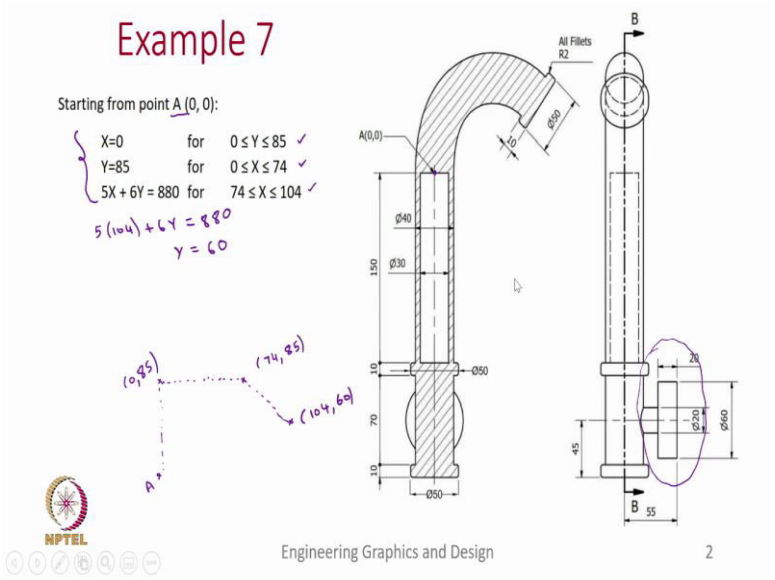

Now I will draw a straight line starting at the origin. And vertically going downwards by length of 230. Let us finish the sketch. Now what we have is the path. Let us also create the profile so, that we can use the sweep comment later. So, again I will go to start 2D sketch, let us choose this plane for the path. For the cross section, we know that the cross section is a circle. I will start with the origin and the radius.

Now let us go back and see where the diameter of this is given. Here it is given that the diameter is 40. So, let us specify that 40 here. And let us finish the sketch. Now we are good to use the sweep command. So, by default, it has chosen the profile. Now let us define the path. And we are fine with what it is showing as a preview. And let us click okay with it.

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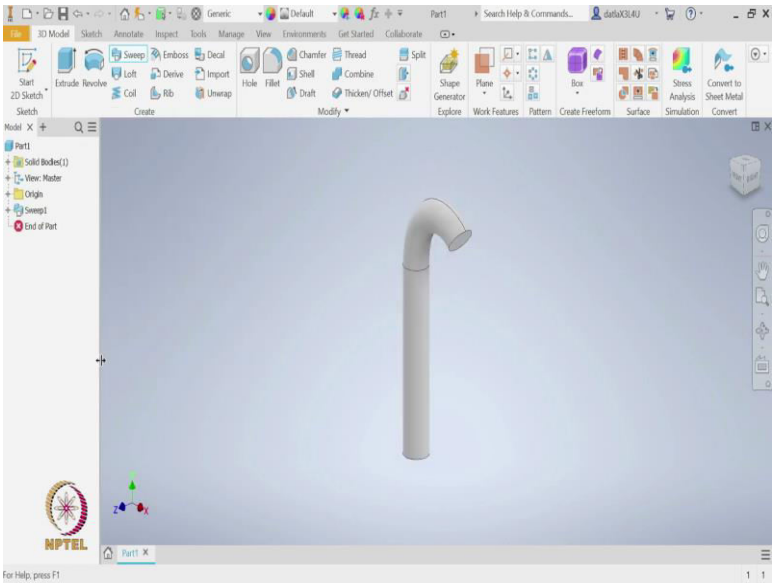

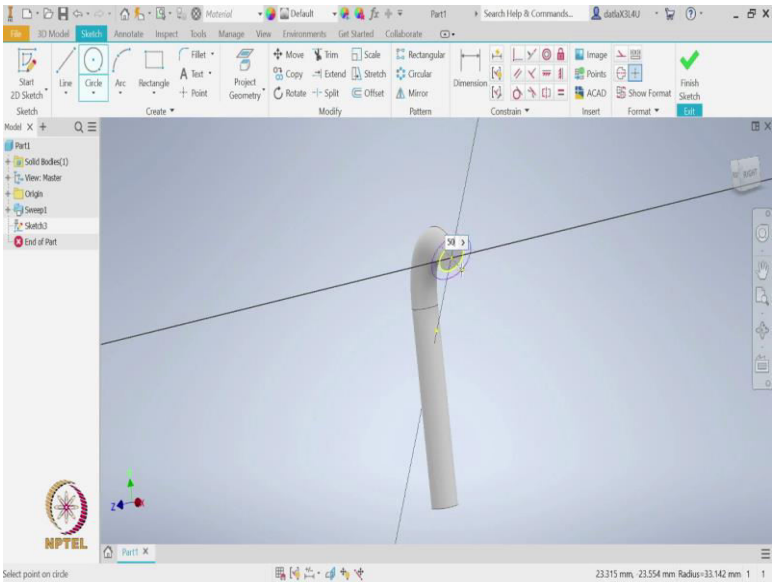

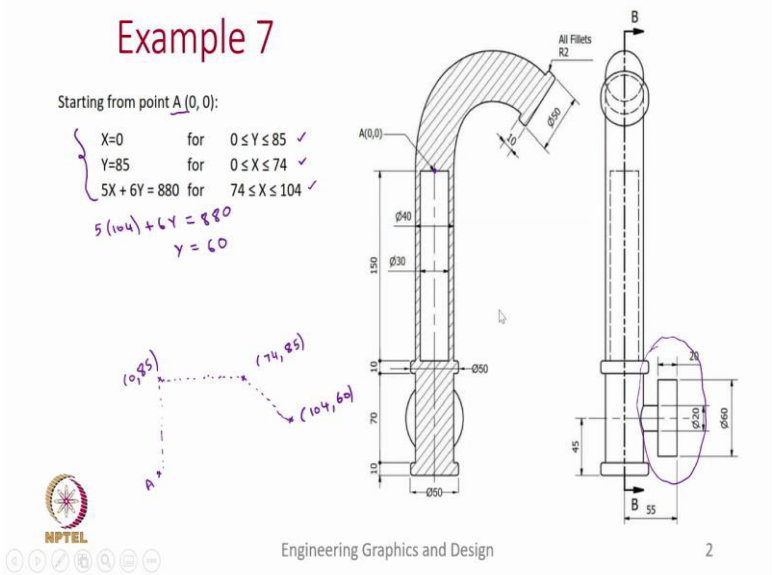

So, now let us go back to the question and see what are the other things we need to do. So, one is at this end of the spline, we need to create again a circular cross section but this time with a diameter of 50 and a length of 10. For that we will go to the extrude and select this plane. Let us now create a circle I see that the center point is not there. So, let us first project the geometry to get the center point.

So, I will project the circle it which will bring both the circle and the center of the circle. But for as we to draw the circle I just need the center point. Now it is turning to green saying that there is a point at the center and this diameter is 50.

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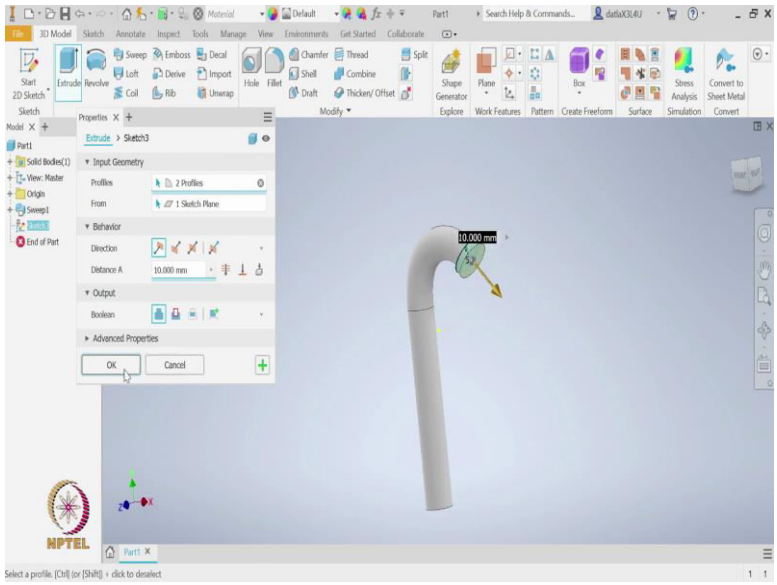

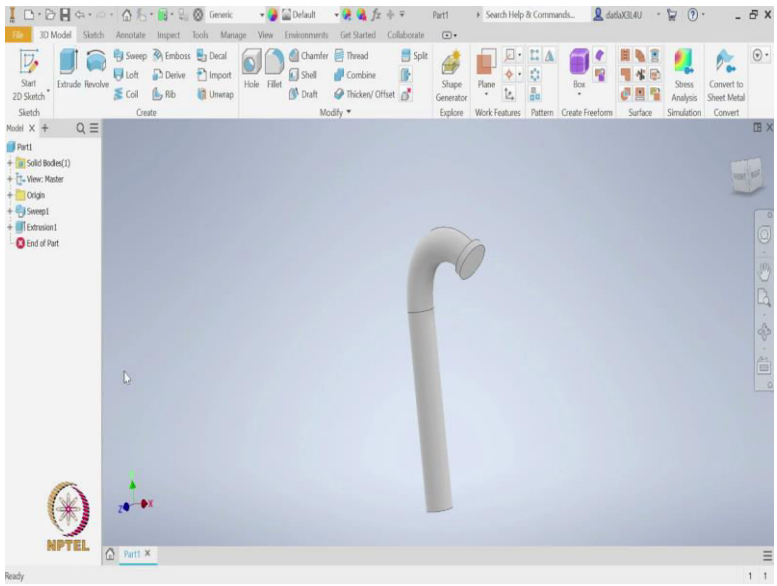

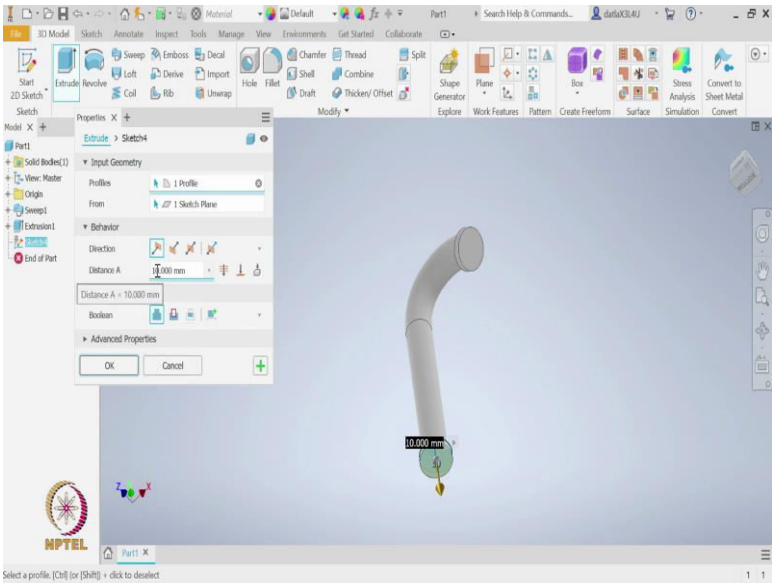

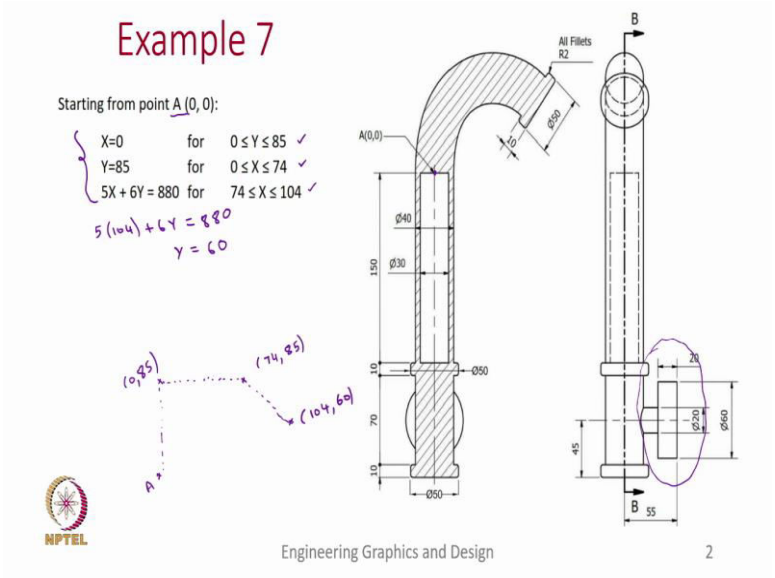

Let us finish the sketch. Let us choose the profile. The distance by default is 10. And the direction is also what we wanted. So, let me check the direction. So, it is protruding out. That is what we want exactly. So, let us click ok to finish the extrusion. So, similarly, we will do the same art at the bottom end. So, let us again go to extrusion, select this face and let me see if there is a center point. Yes. The diameter is 50.

Finish sketch and extrusion. So, again the it has taken the previous extrusion distance of 10 which is fine. Let us click okay. Let us look back and see what are the other features we need to add on now. So, one we need to add on is one more extrusion of this diameter 50 and second is this cavity first let us finish with this extrusion of 50 then we will come back to this cavity.

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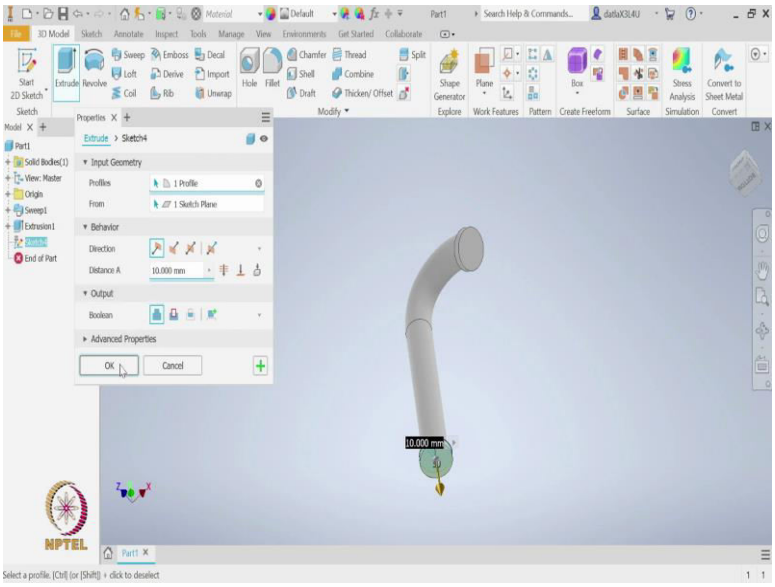

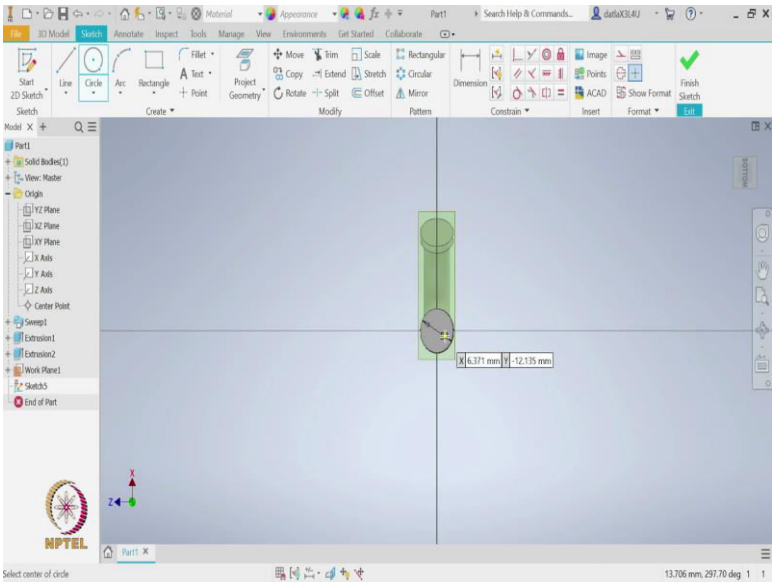

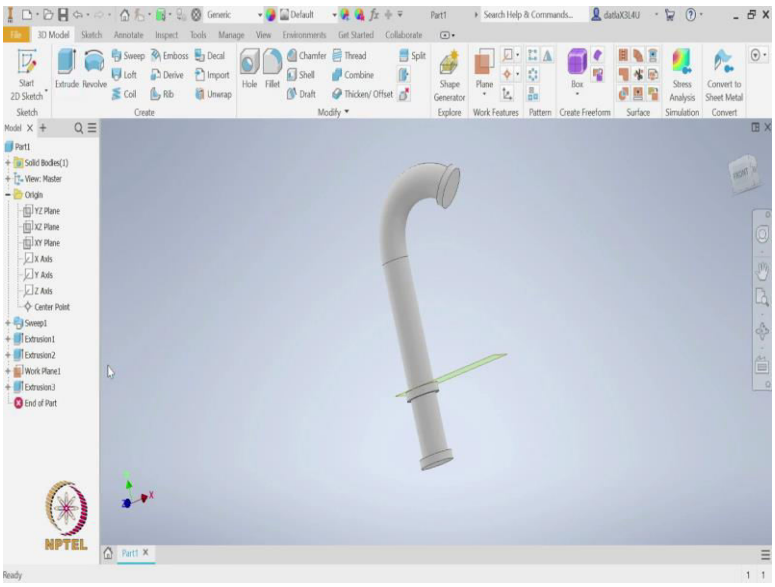

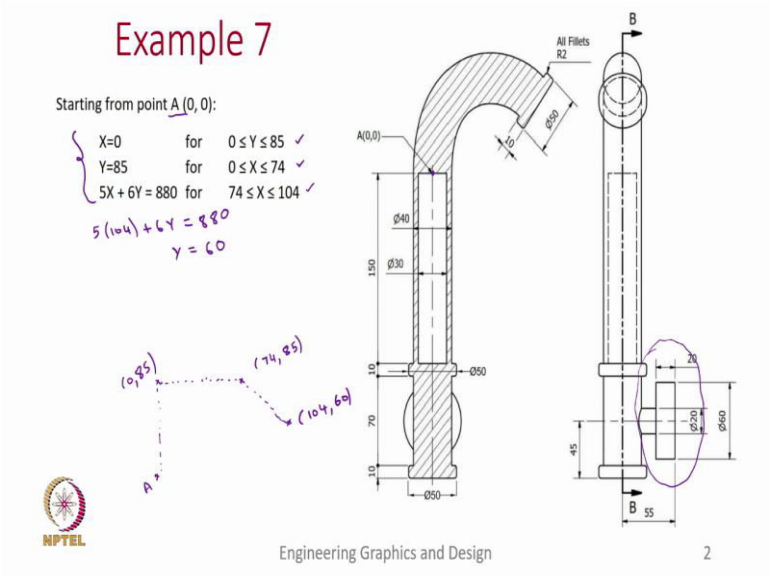

So, I need to create a plane at this distance of 150 So, let us first create that. Plane offset from the plane and we will choose the plane to be from the origin as X Z plane. So, the distance we should be going downwards by minus 150. So, now again let us create a sketch select this new plane and then specify the draw a circle of diameter of 50, finish the sketch and extrude.

So, we want to get this in the opposite direction.So, let me flip the direction and then extrusion distance of 10 is fine let us click okay. So, we have got this diameter 50 extrusion here. The next thing to do is this cavity for this I can use again the extrusion tool. But this time I will use to the Boolean operation of cut.

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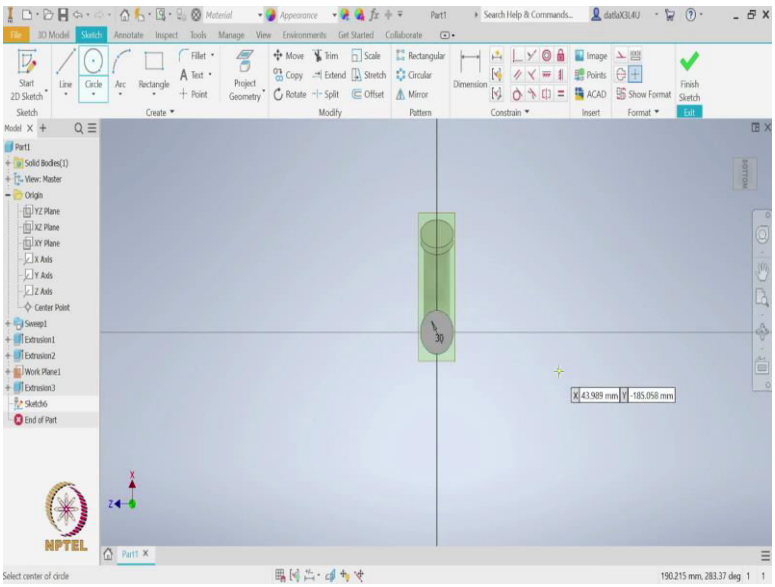

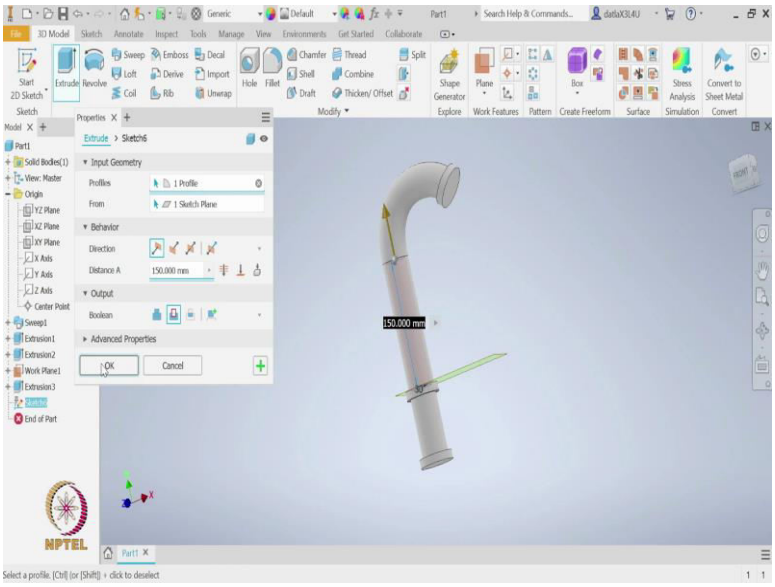

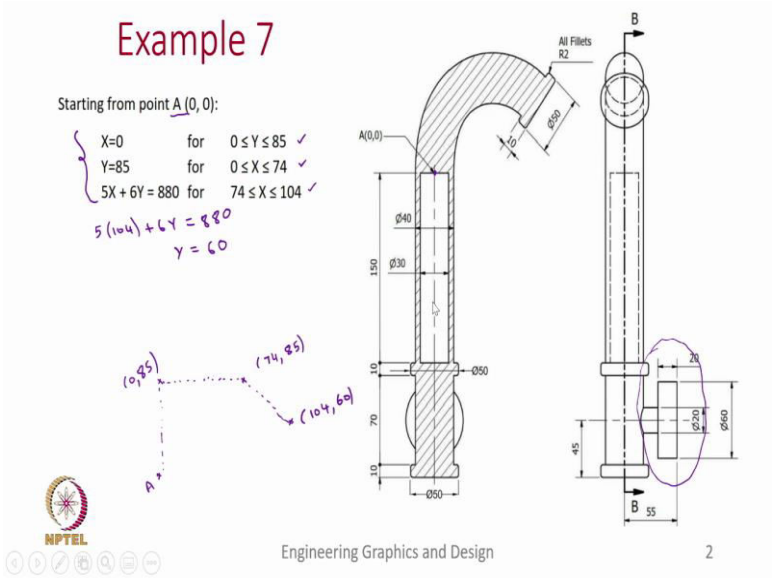

So, let me again start a sketch on this plane. So, let me start by drawing a circle of diameter 13, finish sketch and extrude.So, this time I need to make sure that I am selecting the right sketch. So, the last sketch we have done is sketch 6 fine extrusion distance will be 150 and remember to use the Boolean of cut. So, essentially what we are doing is we are removing material within this tube.

So, let us finish this extrusion of a cut. And then we are done with this vertical column with these extrusion side either end and at the middle as well as the cavity. Now what is remaining is this handle which is a circular cylinder and then there is one connecting between the initial column in the handle. For this we need to select a plane which is perpendicular to this section.

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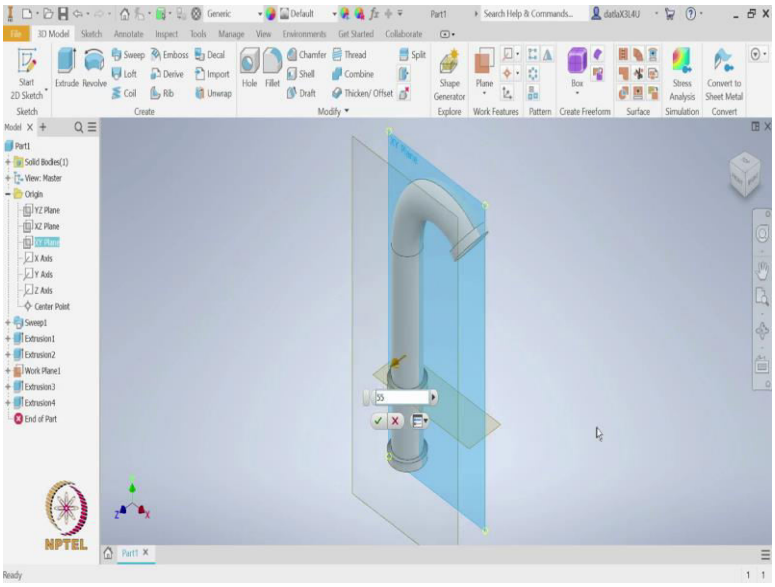

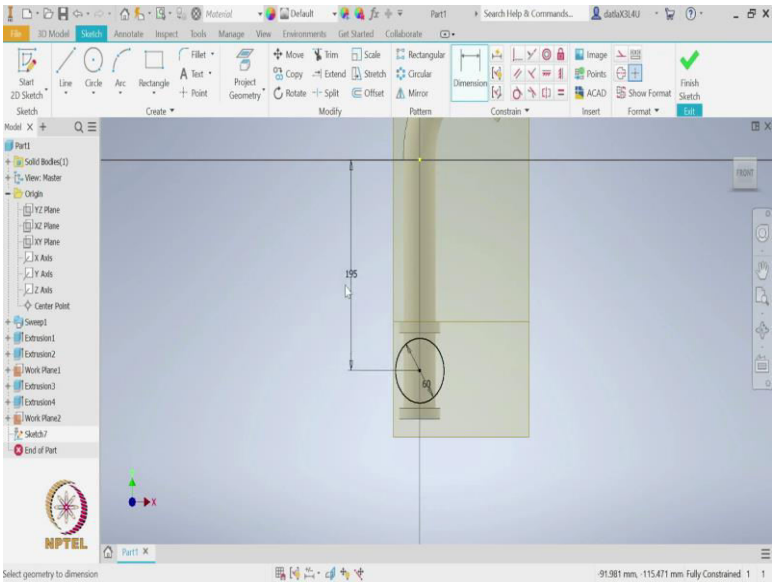

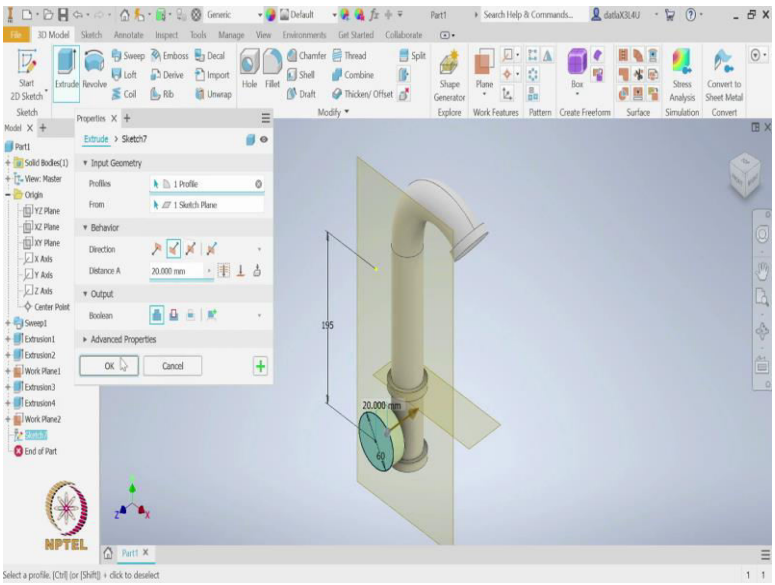

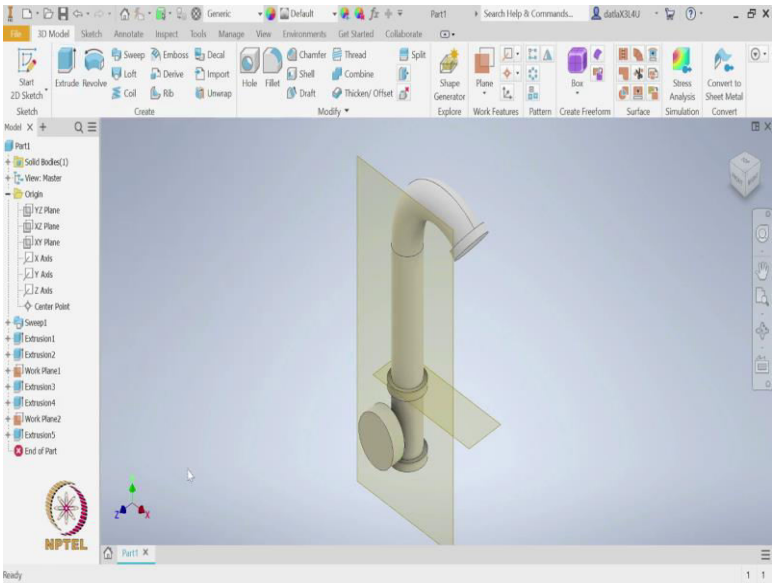

So, let us create an offset plane for X Y plane by a distance of 55. And now let us create a sketch on this new plane. First we will create a circle. So, let me first draw a random circle and then align it later. So, the circle diameter is 60. So, first let us bring this on to the Y axis. And then we will position it at a distance of so and so from the origin.

So, let us calculate it, from here it is 110, 160 and half of it is 35 160 +35 will give us 195. So, distance between the center of the circle to the origin is 195. So, let us finish the sketch and go to the extrude. Let us flip the direction and give the extrusion distance as 20 and click okay.

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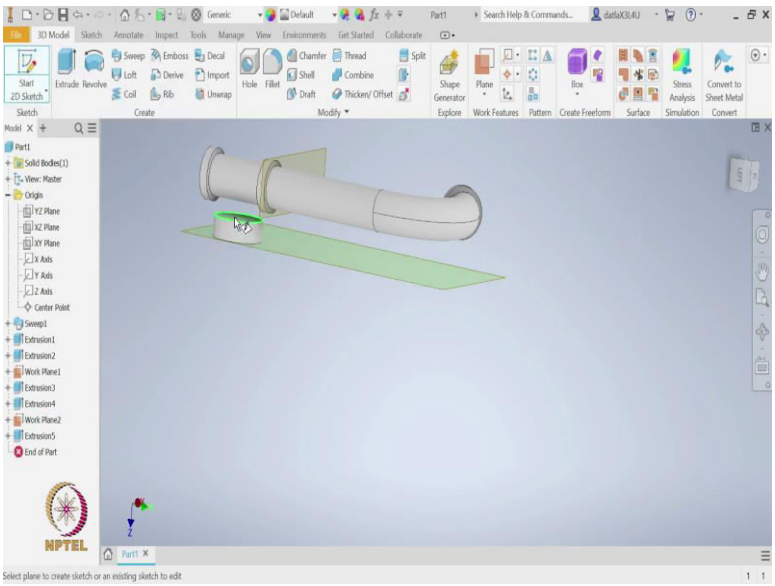

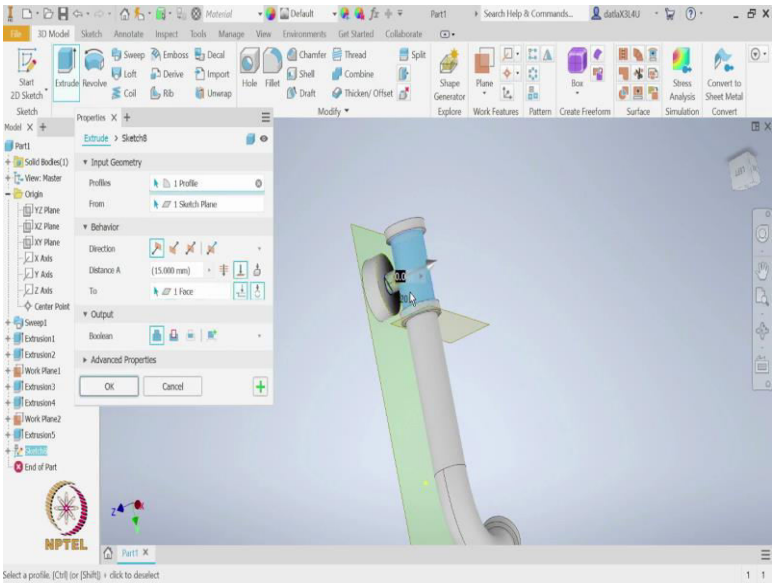
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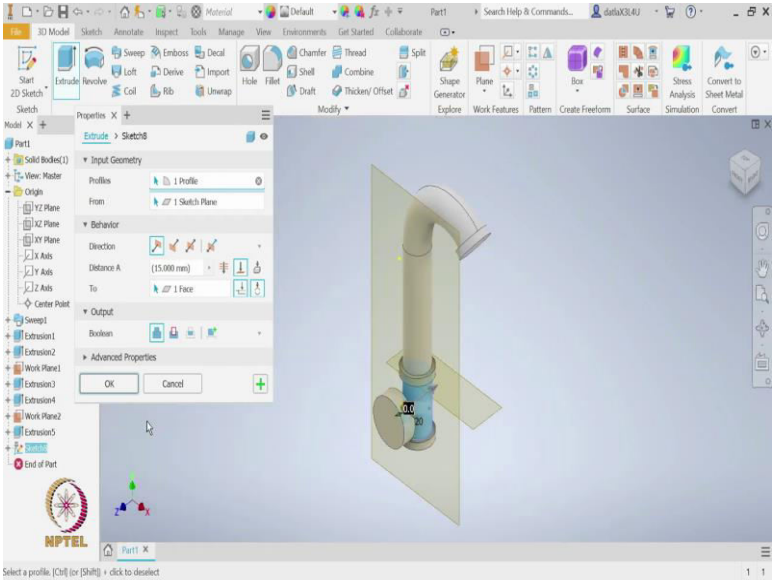
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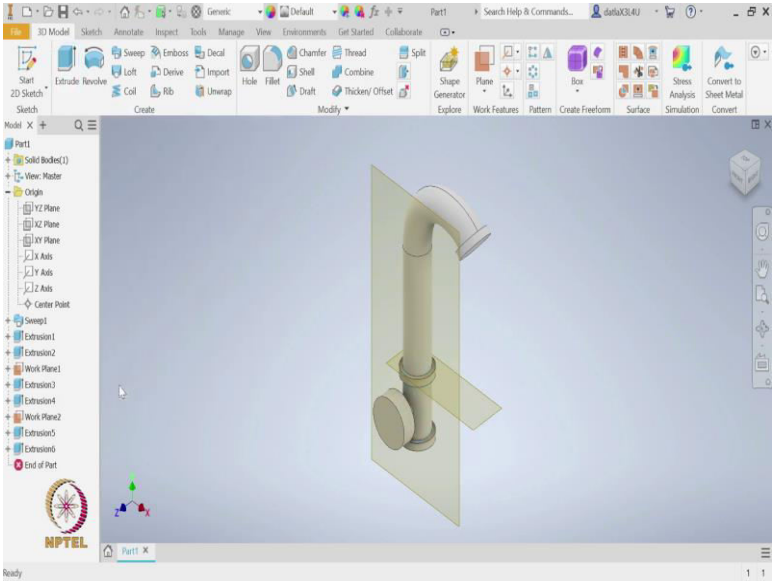
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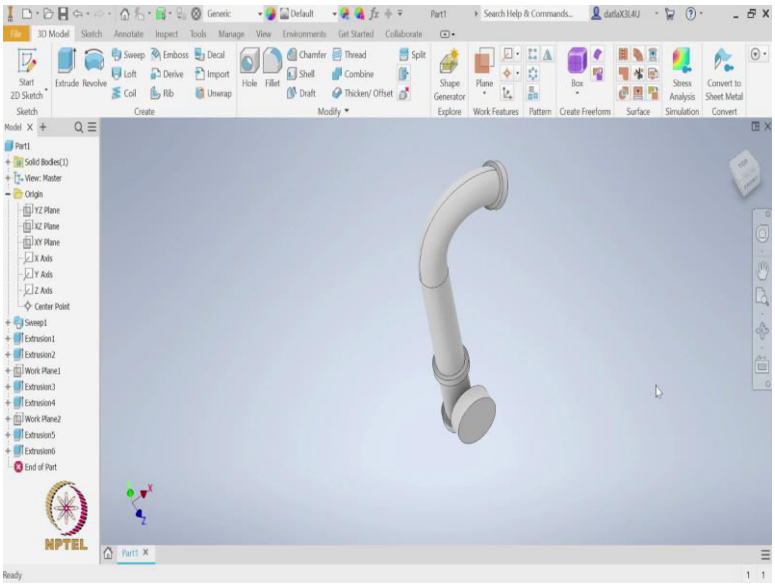

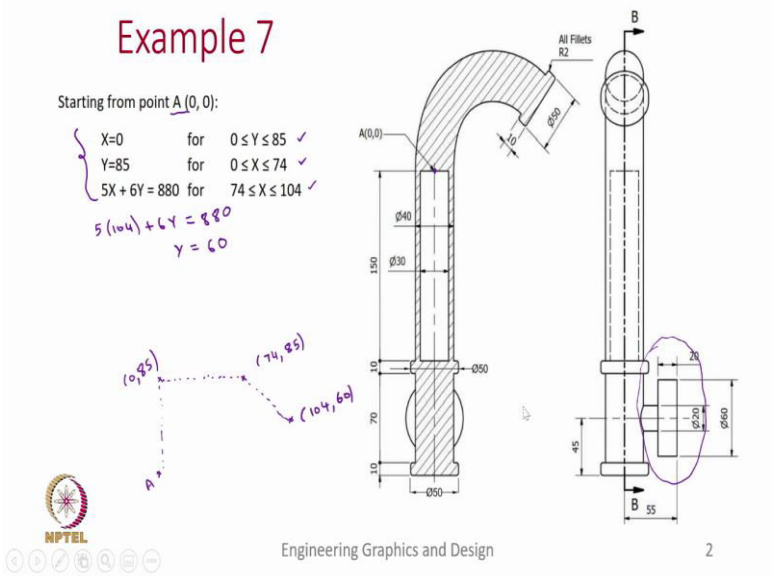

So, we have created the circular cylinder but now we need to connect the circular cylinder to the vertical column. For that what we will do again choose extrude and select this end of this circular cylinder. So, now let us choose this slice graphic so that now we can get rid of this vertical column for the purpose of drawing this sketch. First let me project the geometry then I will have the center of the circle properly.

And let us look at the diameter of this column this is 20 mm. So, circle of I will choose the center and the diameter is 20. Let us finish the sketch choose the profile the last sketch this time we will choose the To and specify it is this part. So, if we zoom in we will see that this has went and met with the circular column.

So, let us zoom in and see are we done. I need to click okay to finish that extrusion with this we are done with the problem we are given it let me uncheck the visibility of these planes to

make it clear. Now we see the solid model what we are asked with. With this let us conclude the lecture. Thanks for your attention.