Engineering Graphics and Design
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Week 10: Part Modelling 2
Lecture 2
Example 6

Welcome back to week 10 of Part Modeling 2. In this lecture we will be looking at an example where we will look at using some of the tools such as sweep and shell. So, let us get to the example.

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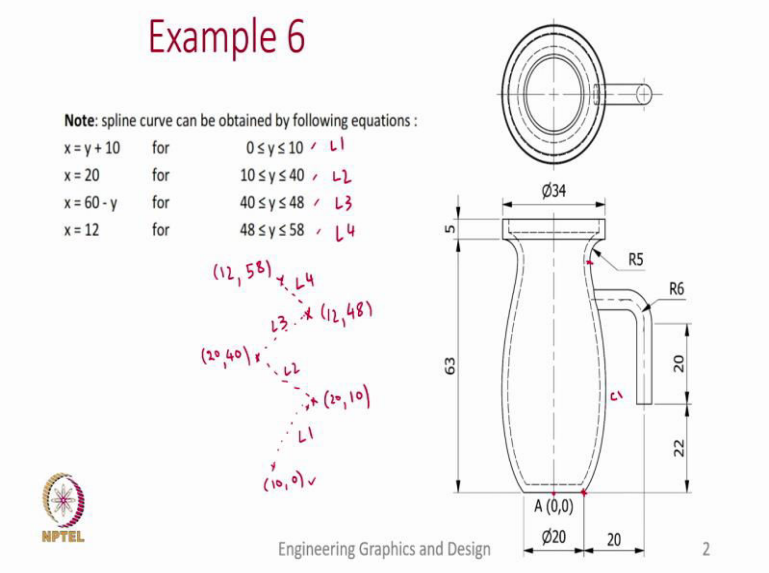

Here is a vase we are trying to create. As you can see, it has a handle to the right we need to create that as well. So, let us look into this two front view and the top view and try to understand what are the tools we may need to use to create this solid. So these dashed lines should indicate us that the interior is hollow.

So which means we will be using the shell tool. So and we also look that the outer surface is a smooth curve and that smooth curve is shown here that it is defined as a spline curve. So and these are the equations that are given for the control vertices. So these control vertices make a series of straight lines.

So the those straight lines are given by these equations. We also noticed that at the end of the smooth curve, we have affiliate of radius 5 and at the top the cross section is uniform and the height of that is 5 and the diameter of the top lip is diameter 34. So what we have seen is we

need to first create a spline using these equations and once we create that spline we also need to create this fillet and then the uniform cross section.

So once we create a profile with this spline shape, then we can use the revolve command to make a solid and then the shell command to remove the interior, that will give us the vase. At the end we will work with the handle, but before I go to the software, let me point out from where we are getting this spline.

So, let us say from point here to point here is this spline curve let us say I will say C1 as the spline call and then we will have other curves let us not talk discuss about that right now. So, in this spline curve, we are saying it is defined by four line segments 1, 2, 3, 4. So, essentially you have control points which make 1 line segment, 2 line segments, 3 line segments and the 4th line segment.

So, let us say these line segments L1, L2, L3 and L4. So, using these equations first let us figure out what are the points of the vertices then it will be easy for us to use the spline command. So, this point is given by what? So, we have known that this point is the origin (0, 0)since the diameter is given as 20, we know that this point should be x is 10 and y is 0.

What about the endpoint of line L1? So the endpoint of line L1 can be found through this particular equation, where it says x is equal to y plus 10 which means when y equal to 0, x is equal to 10. That is what we got here and when y is equal to 10. So which means we already know the y component is 10 here, then what is the x component?

It is simply y + 10, 10 + 10 is 20. Similarly, we can do that for the other points as well for let us say, for the end of line L2. Let us look at what are the x and y coordinates. So this is the end of line segment L2 where the y changes from 10 to 40. The initial point 10 we already know. So let us find out at y = 40 what is the x component?

The x component is given as 20. Now let us move to the endpoint of line L3. So again, let me write it out line 1, line 2, line 3 and line 4. So endpoint of line L3 will be y is equal to 48. So, let me first write 48 and then figure out what is the value of x? x will be 60 - y, 60- 48 is 12 and finally, endpoint of line L4.

So, the y component will be 58 and the x component is given as constant which is 12. Now that we have identified these control points, then it will be easy for us to go to the software and create this spline.

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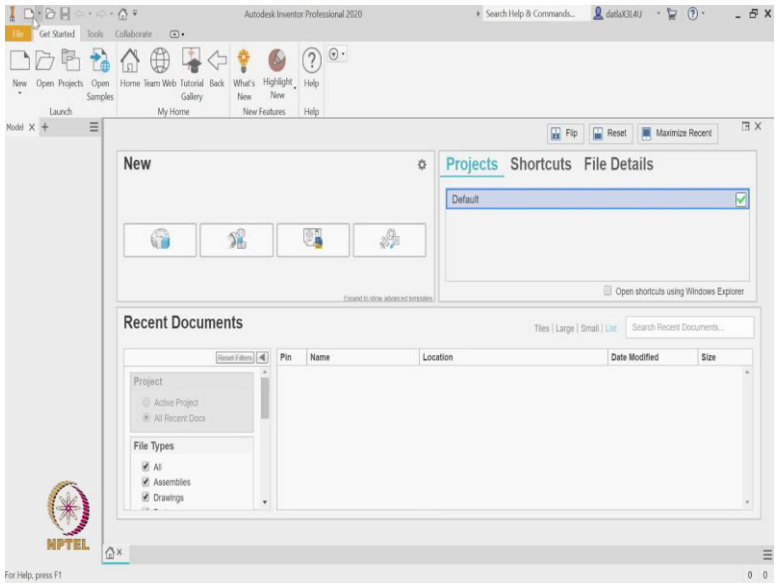

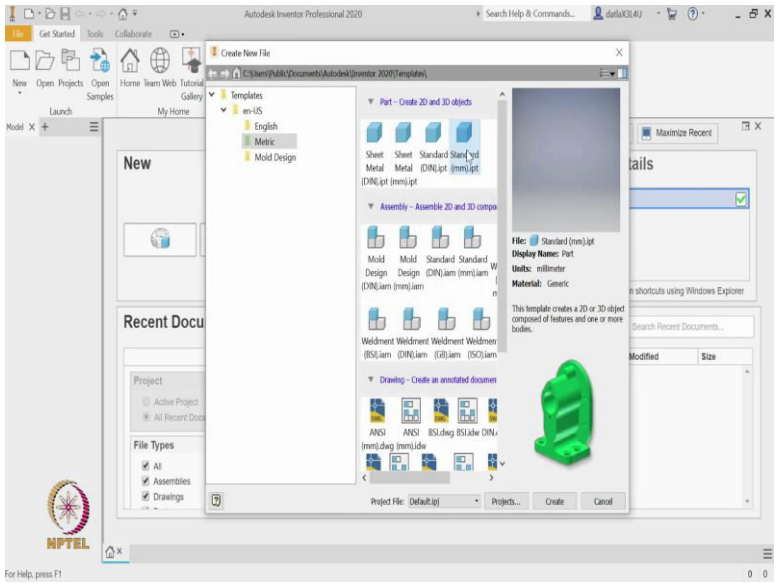

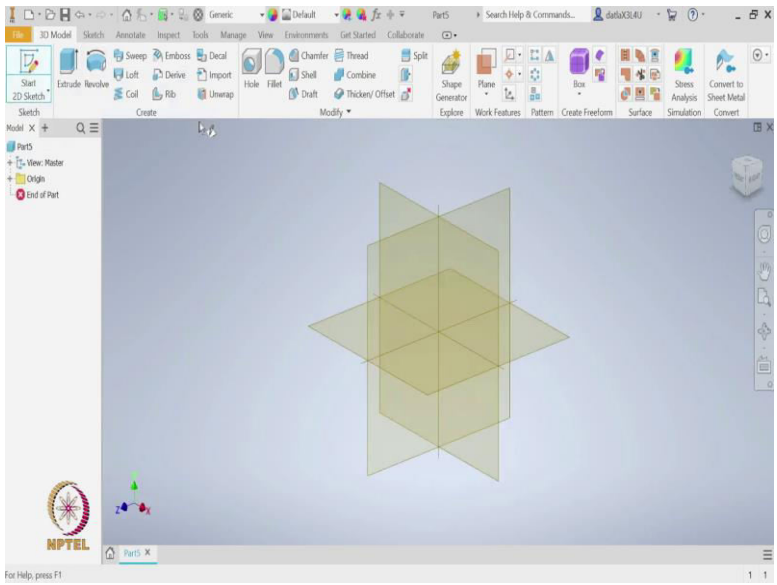

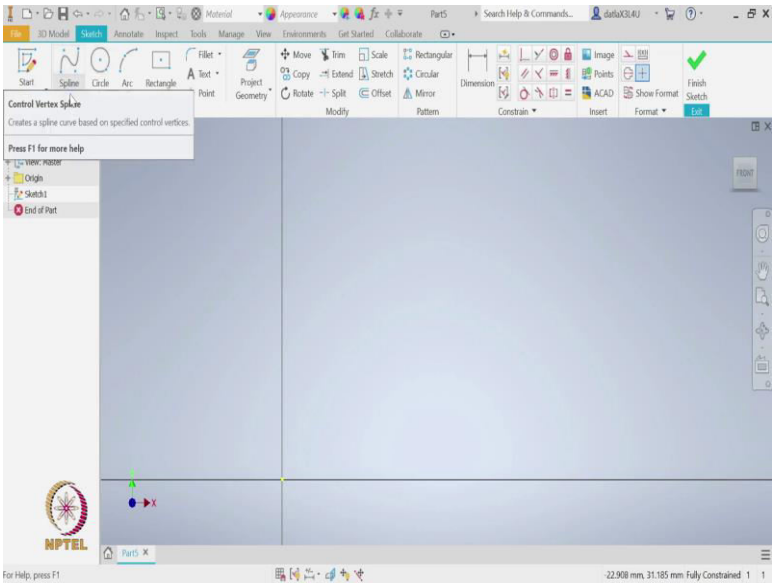

So here is the initial interface. Let us start by selecting the part template. And then let us create a 2D sketch using the X Y plane. So let us start by choosing the spline by control vertex, the curve's initial and final point will match with the control points but not the intermediate ones.

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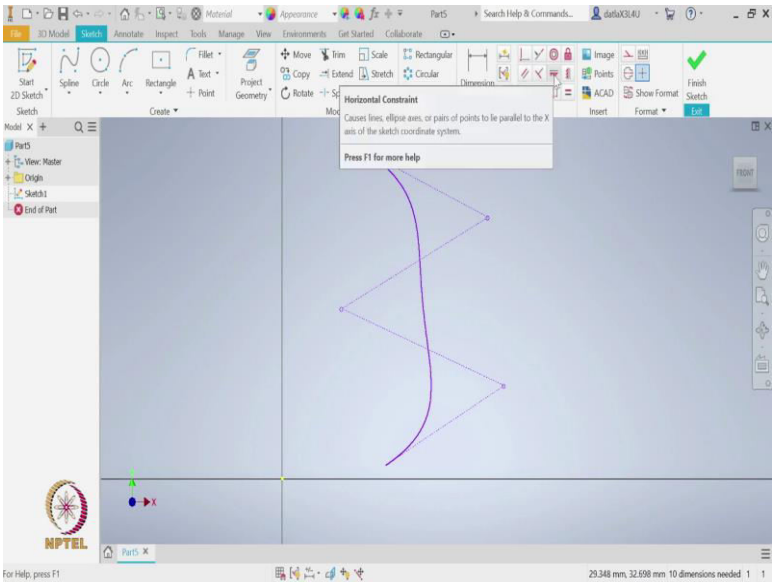

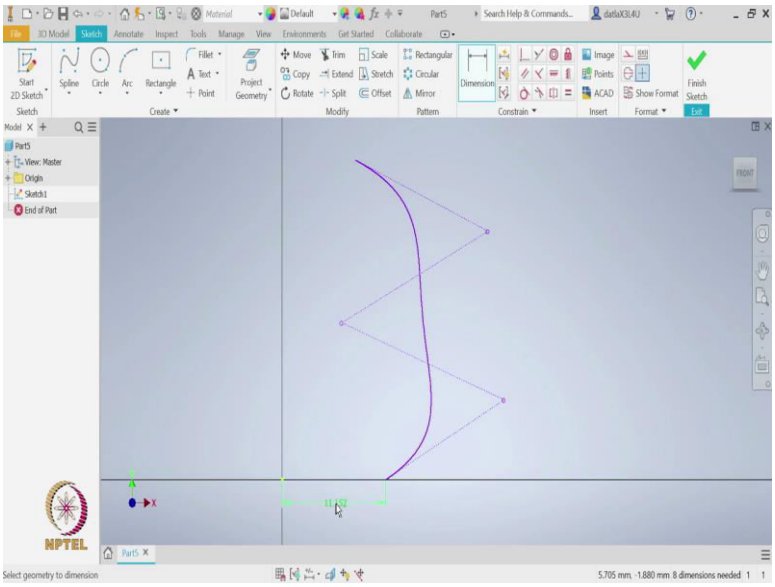
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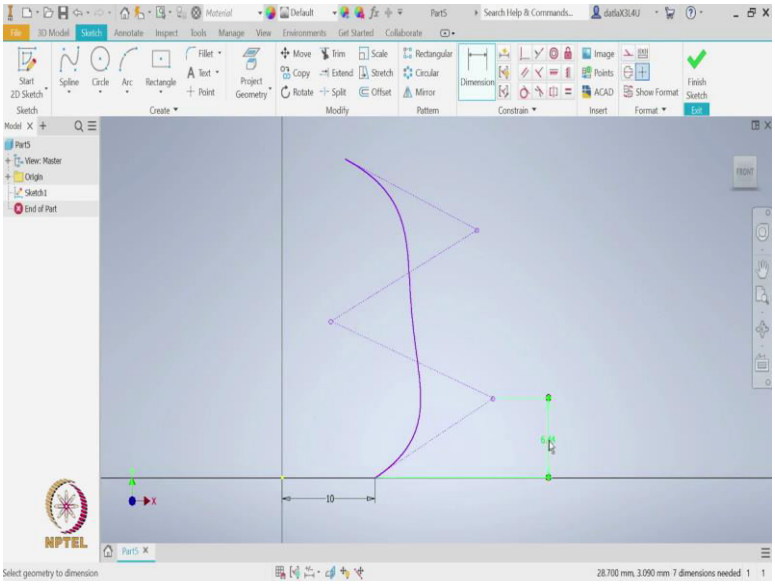
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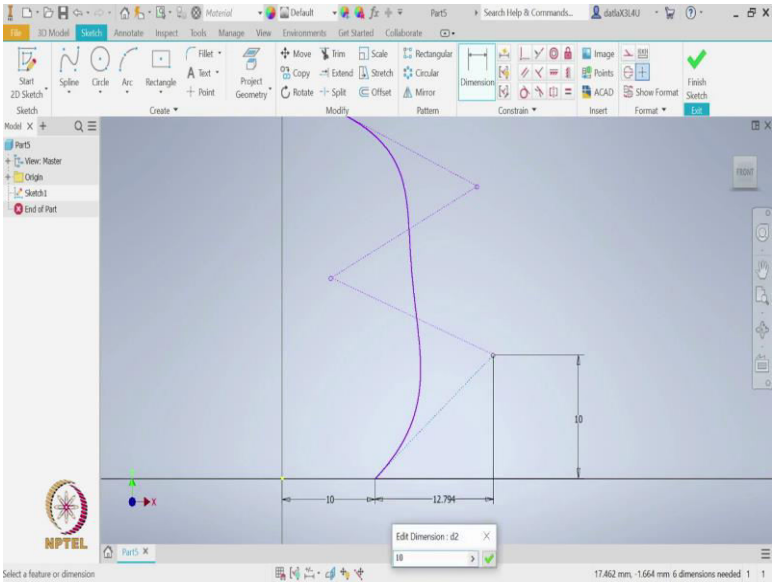
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10
12.794
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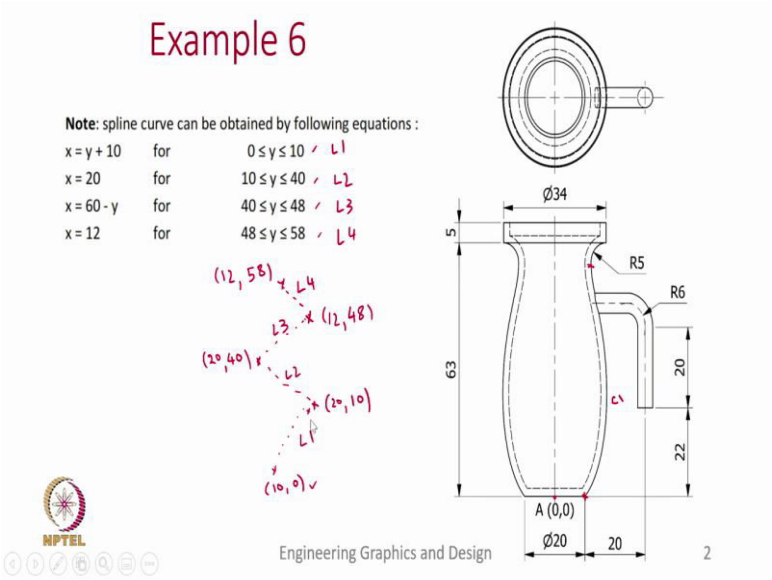

So let us start. So, this is first line, second line, third line, fourth line, I am randomly choosing these lines as of now, but later when I dimension you will see that this will be exactly what we are looking in the example. First step, let me coincide this with the X axis for that I will use this constraint of horizontal.

I will choose the initial point and the origin. Now we can see it is now on the X axis. Next I will dimension saying that from origin, the initial point is at a distance of 10. Then, let us go back and see how do we define the second point it is 20 and 10. So first, let us dimension this saying that the height is 10. And the X axis, the total is 20. But it will be 10 + 10. So this is also 10 fine. So now we have defined this line L1. Now let us go to line L2.

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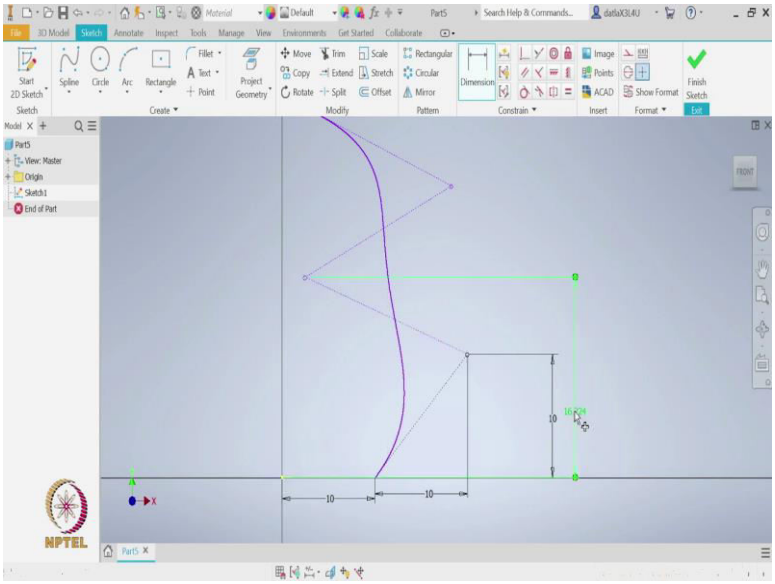

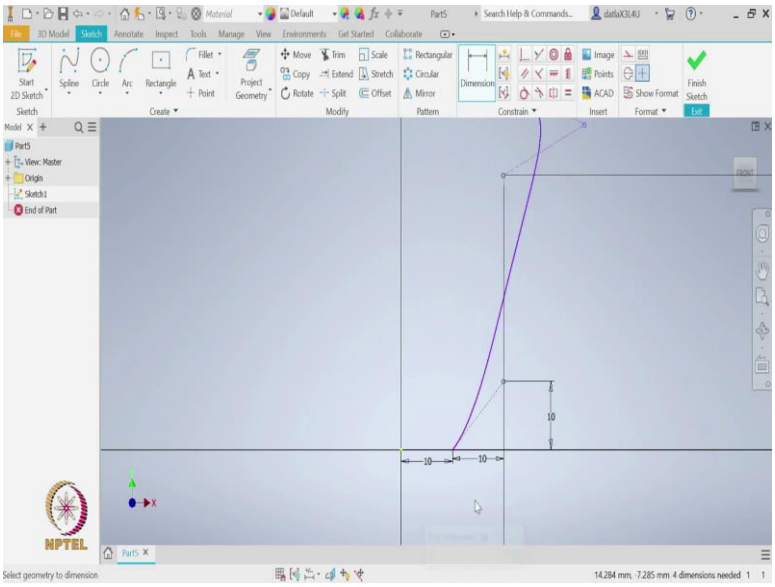

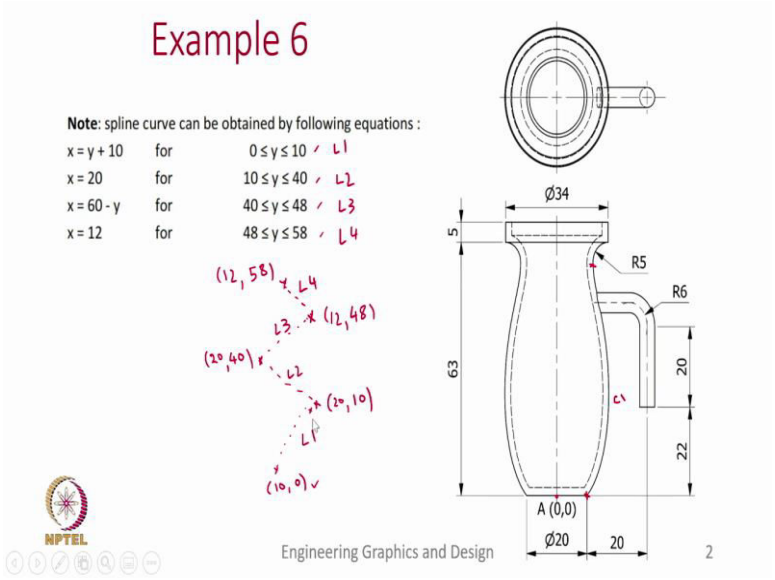

For that I will dimension this point with the origin and say the total height is 40 and the other thing we need to mention is that the x coordinate makes a length of 20. So it is essentially a vertical line. So line L2 is a vertical line fine. Now let us go back and see where is end of line L3? 12 ,14

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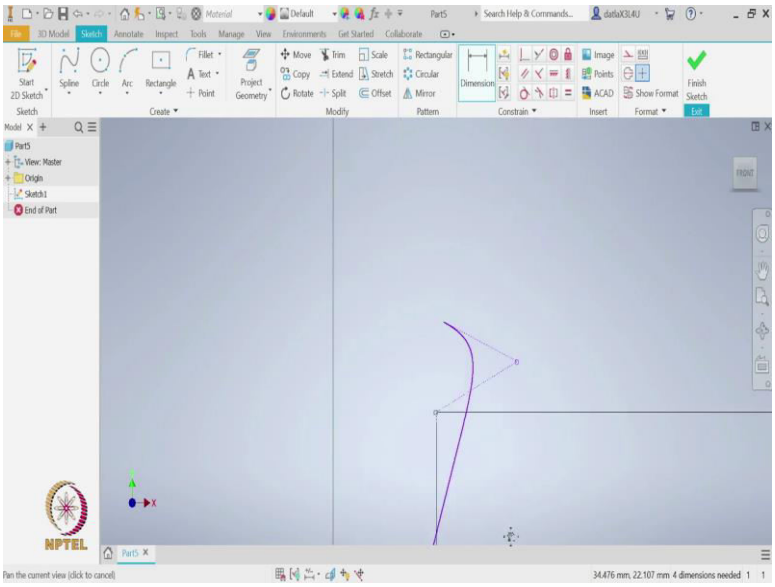

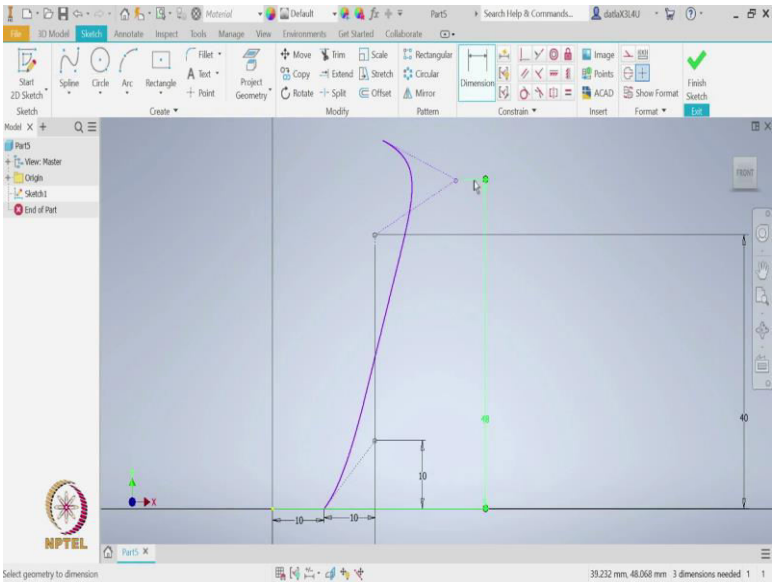

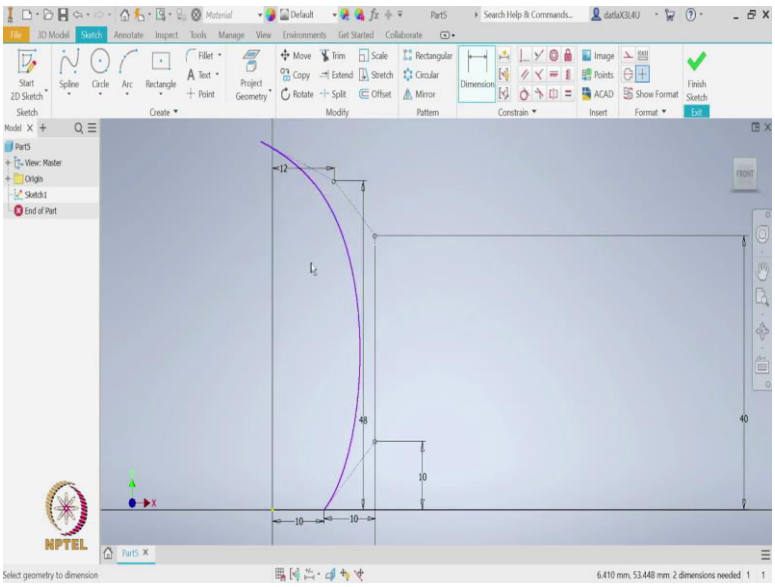

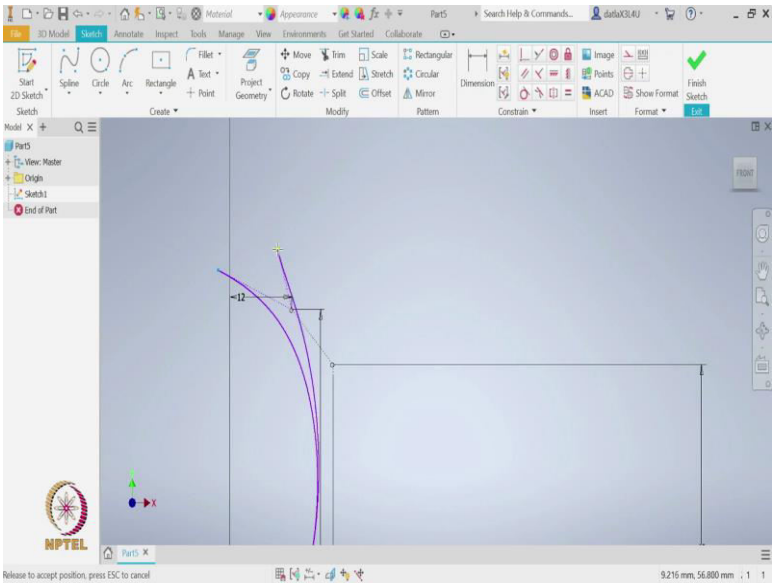
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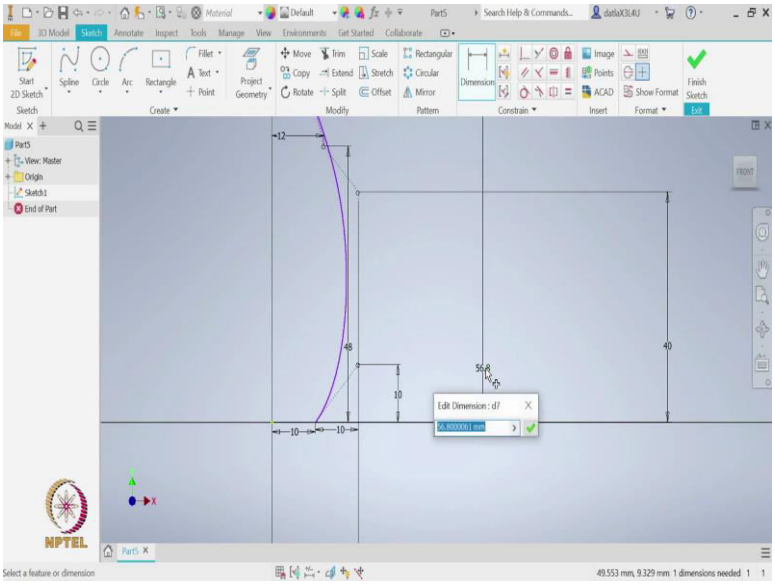
9.216 mm, 56.800 mm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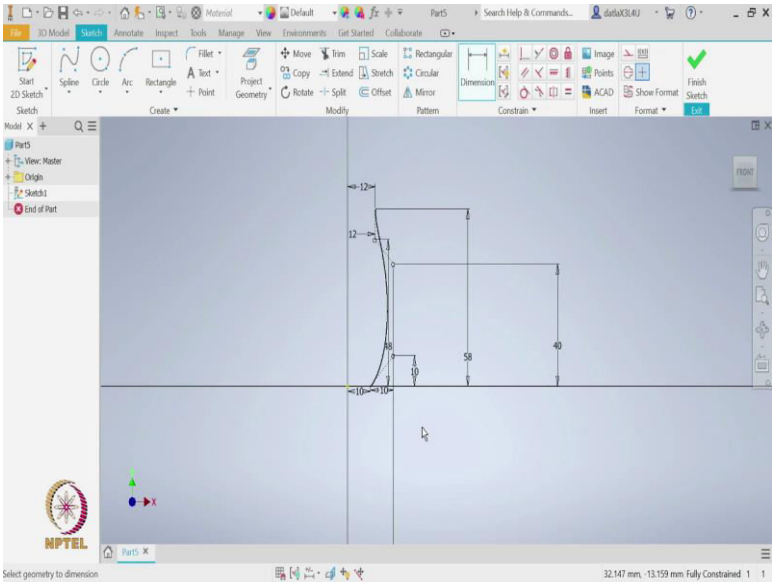
Select a feature or dimension
49.553 mm, 9.329 mm 1 dimensions needed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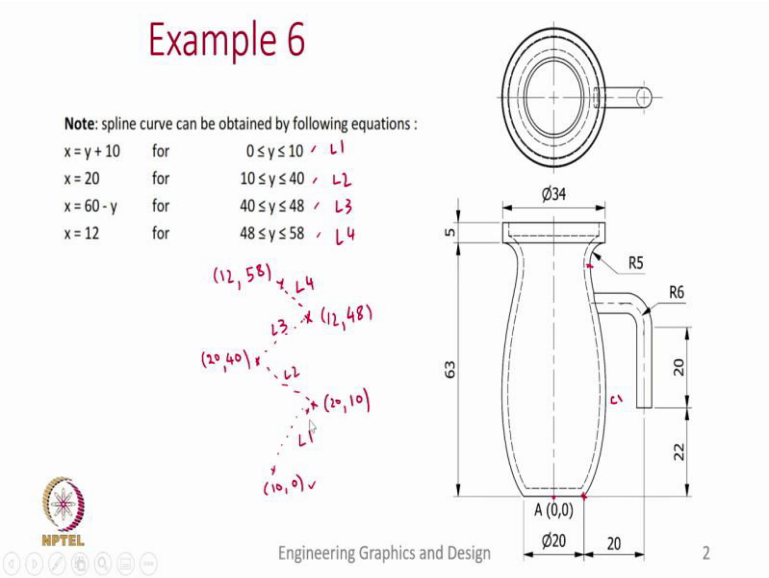
32.147 mm, -13.159 mm Fully Constrained



So first, let us define these 48, 48 and with the X axis, it makes a dimension of 12. So for the last point, let me first move the point closer. Now, I am moving the last point. I am just trying to bring it closer, but we will go back and see what are the coordinates of it.

So we see that the last point coordinates are 12,58. So first, let us dimension the y and say it is with the origin making 58. And this endpoint with the origin makes 12 units in the X axis. So with this we have completed the spline curve. So now you can also see that it is completely constrained and we can finish the sketch but before we finish the sketch, let us create the complete profile by making it a closed curve.

Until now what we are done is only creating this curve C1, which is spline. But we are yet to do this fillet and uniform cross section vertical thing. So now let us create those. Before I create a fillet, first I will create a sharp corner later I can apply the fillet command.

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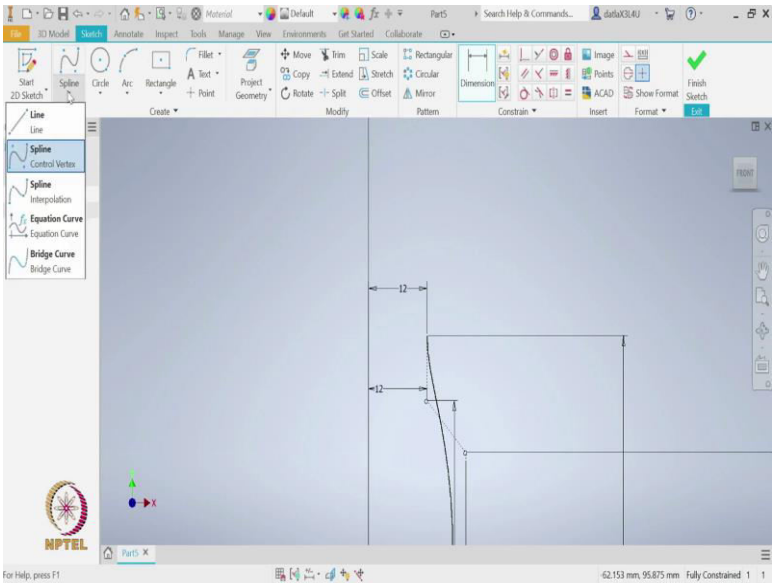

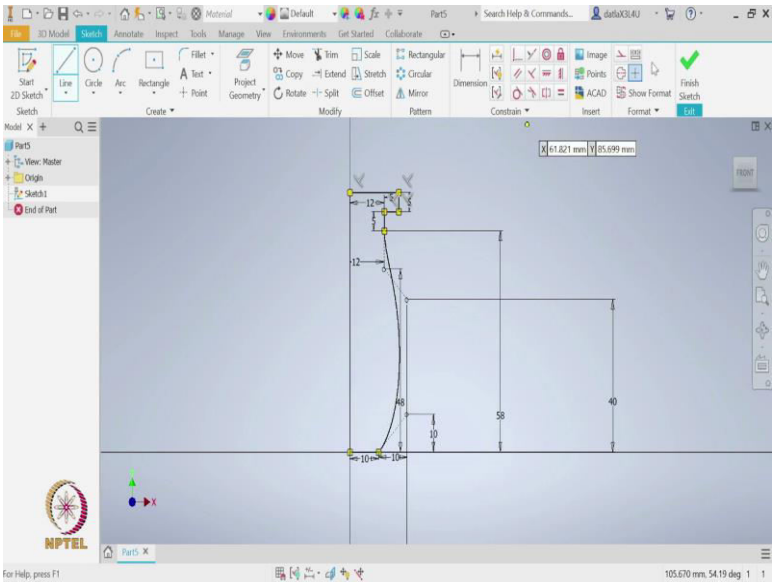

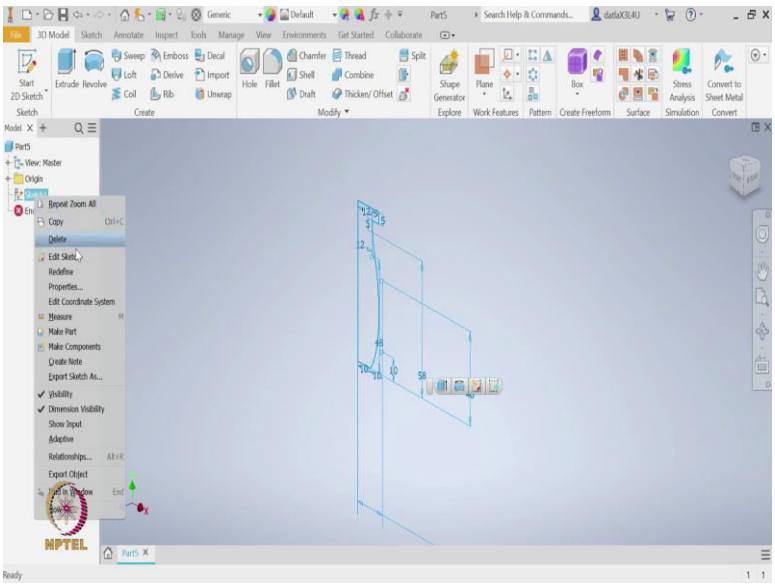

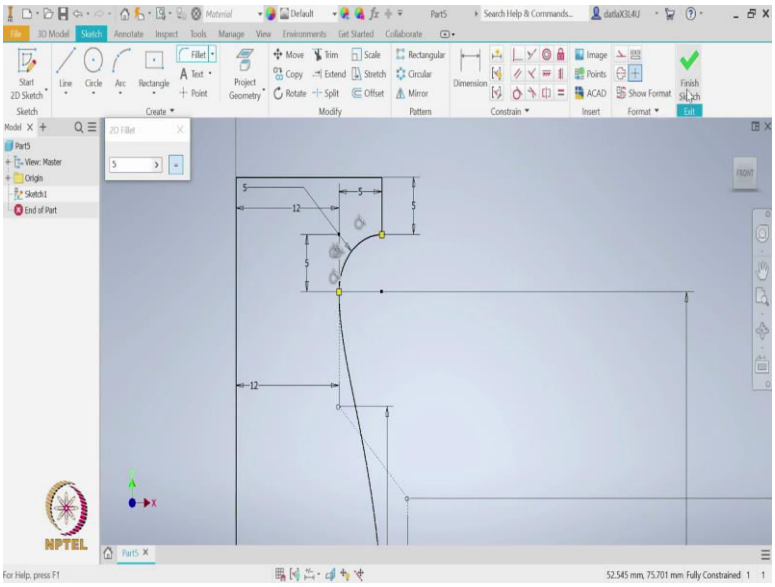

So for this let us zoom in and use the line command and start doing it. So I have to go vertical of 5 units and then horizontal 5 units. This will be done with the fillet. But regarding the uniform cross section again I need to go up by 5 units and the upper lip diameter we have seen is 34 which means the radius is 17. So let us specify the 17 and also let us close this curve by this vertical line and then a horizontal line.

So let us finish the sketch now. And now we can use the, before I go to the revolve, let me edit the sketch because we forgot to do the fillet. So edit sketch. So I can select the fillet tool, specify the radius is 5 and then select the two edges. So now it has created the fillet, I can finish the sketch now.

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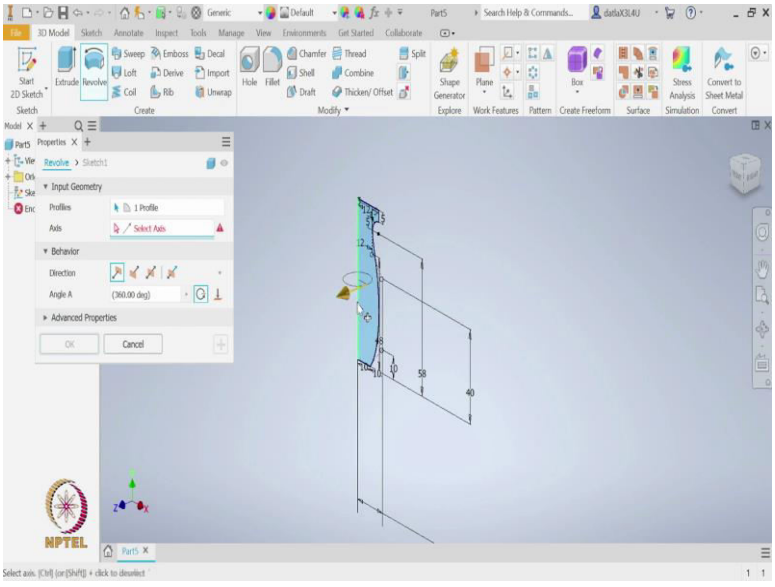
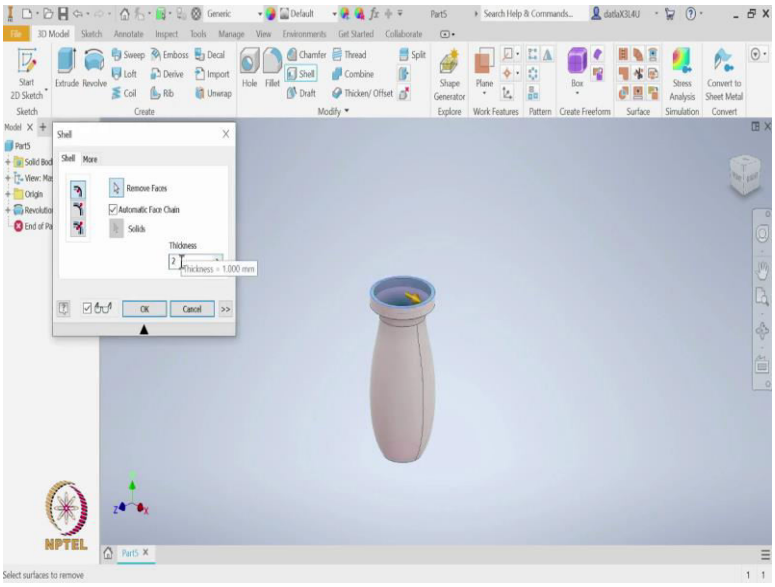
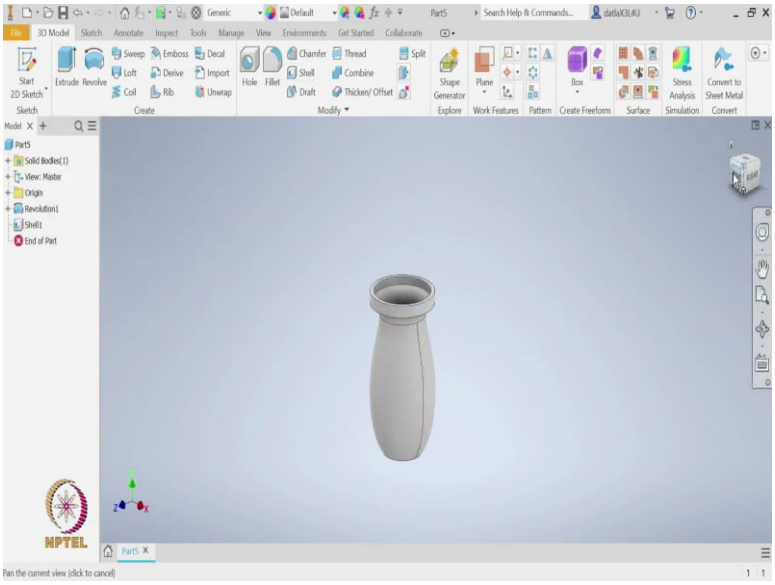


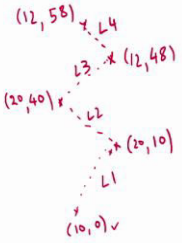


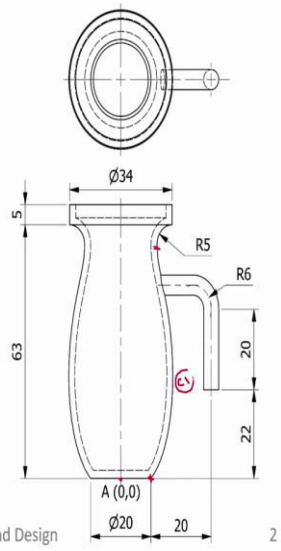


Example 6

Note: spline curve can be obtained by following equations :

x = y + 10	for	0 ≤ y ≤ 10	L1
x = 20	for	10 ≤ y ≤ 40	L2
x = 60 - y	for	40 ≤ y ≤ 58	L3
x = 12	for	48 ≤ y ≤ 58	L4





And then select the revolve tool where it by default it has selected the profile that we have created. And for the axis let us give the axis as the Y axis. So now we have completed this vase as a complete solid. The next step is to remove the interior material so that we can create this cavity with a specified thickness.

I noticed that we have not given any dimension to the thickness, but let us assume it to be 2 millimeters. So to create the shell, we select the shell from the Modify panel. And then let us remove the top face and specify the thickness to be 2 and click okay. So with this we are done with creating the vase, at least we have created the cavity and we have, the shape is already captured.

So what is now remaining is to create the handle. So now let us go back and see how do we approach to create this handle? So one way it is we can use the sweep command for which we need to know what is the profile. So probably it is not specified on this slide. But the cross section is a circular circle of diameter 5.

So we will take that as a cross section and then use the sweep command which goes initially vertical and then this again, another curvature of r is equal to 6 and then the horizontal line. So we need to create both the profile and the path.

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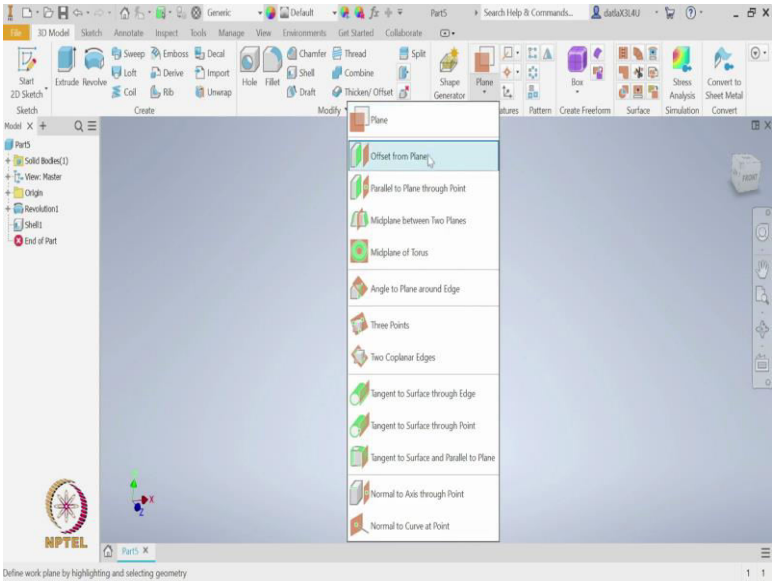

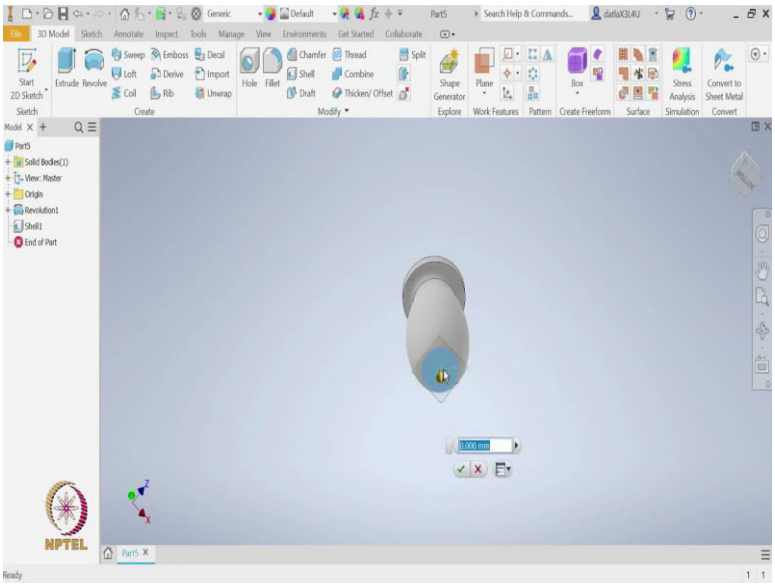
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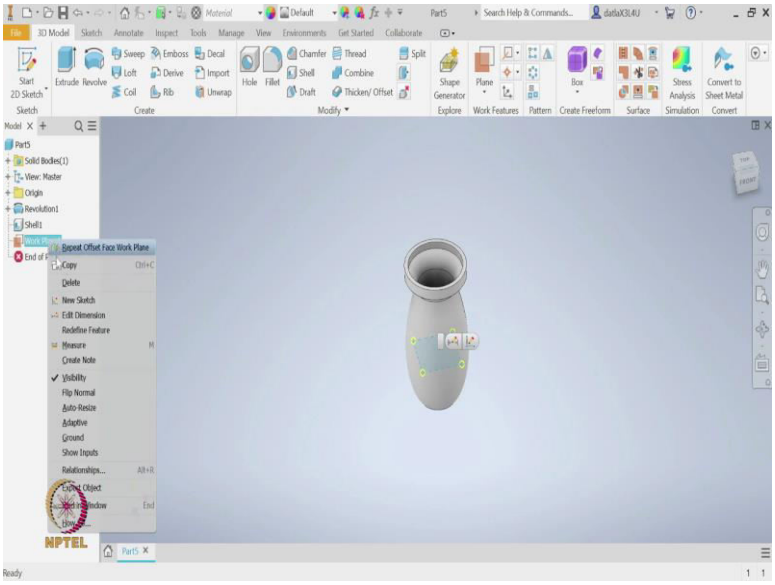
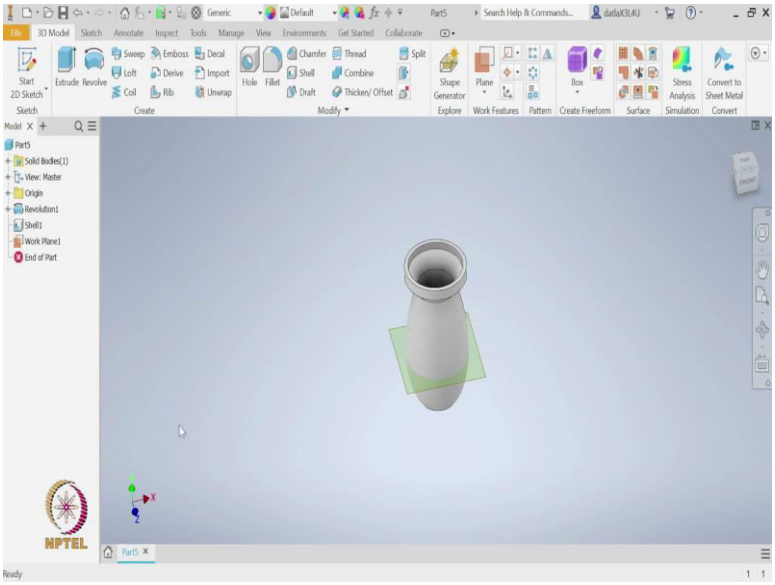
View: Master
Origin
Revolution1
Shell1
End of Part
1.000 mm
Ready


Repeat Offset Face Work Plane
Copy
Delete
New Sketch
Edit Dimension
Redefine Feature
Measure
Create Note
Visibility
Flip Normal
Auto-Resize
Adaptive
Ground
Show Inputs
Relationships...
Ready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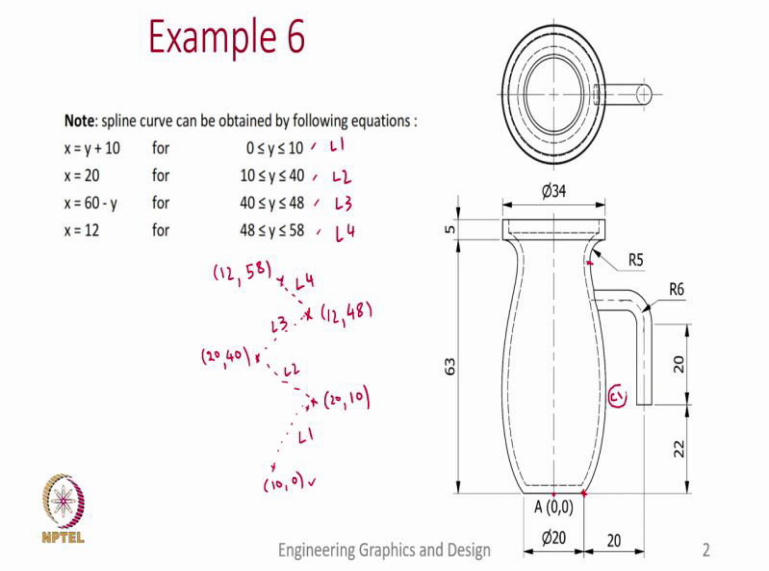
Solid Bodies(1)
View: Master
Origin
Revolution1
Shell1
Work Plane1
End of Part
Ready



So first, let us get started with the profile. But we noticed that the profile, the circle needs to be created at a plane which is parallel to the base. So let us first create a plane using the offset from plane option, so let us select the bottom base and then specify the dimension. Let us look at the example it says that it is at a height of 22.

So, it has created the sketch but we are not able to see it because the size of the sketch it is showing is too small compared to the dimensions of the vase we are creating. So for that, what we can do is we can go to the work plane, right click on it and say auto resize. So it has changed the dimensions of the plane just for the display purposes so that now we can see both the plane and the solid that we created. But now let us go to 2D sketch to create the profile for the handle.

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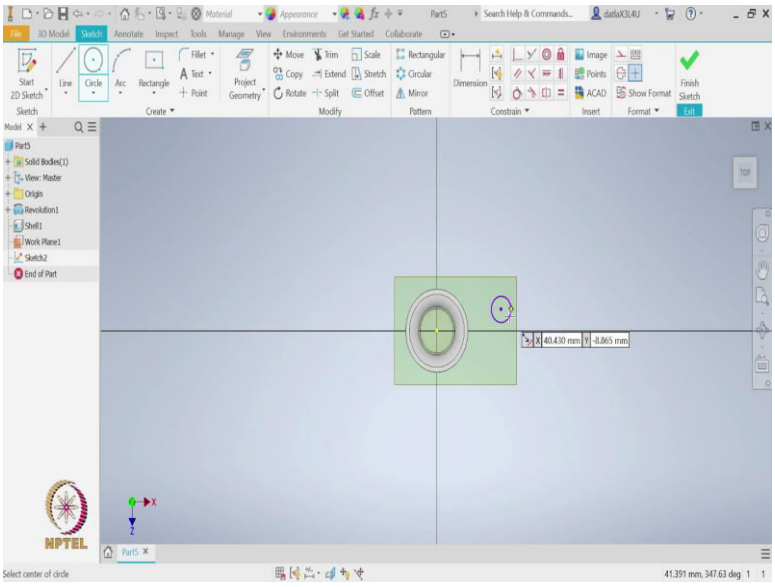

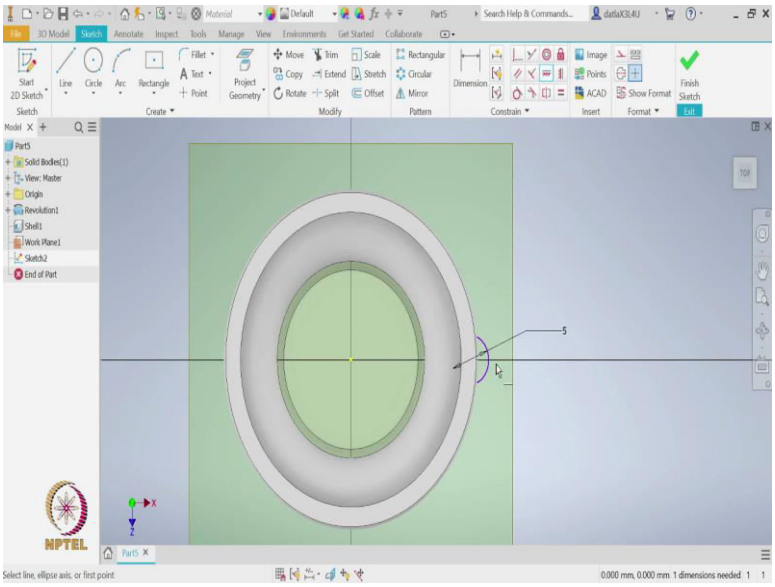
5
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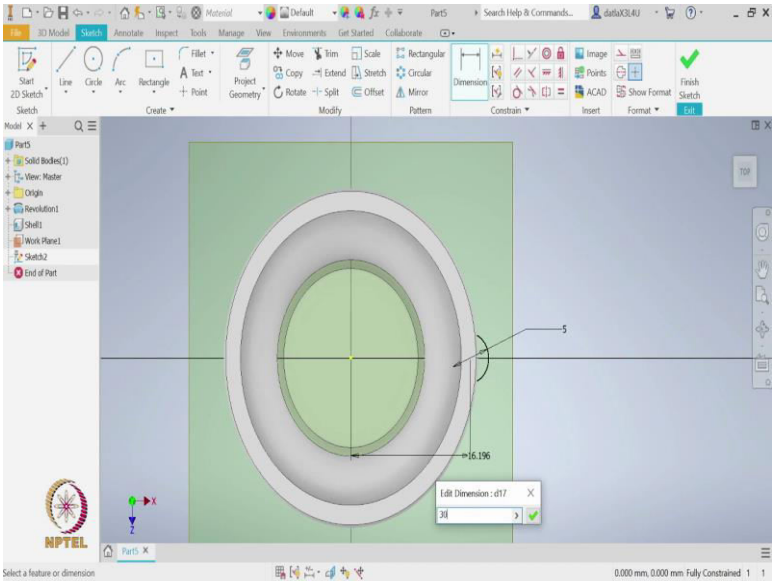
0.000 mm, 0.000 mm 1 dimensions needed


5
16.196
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30
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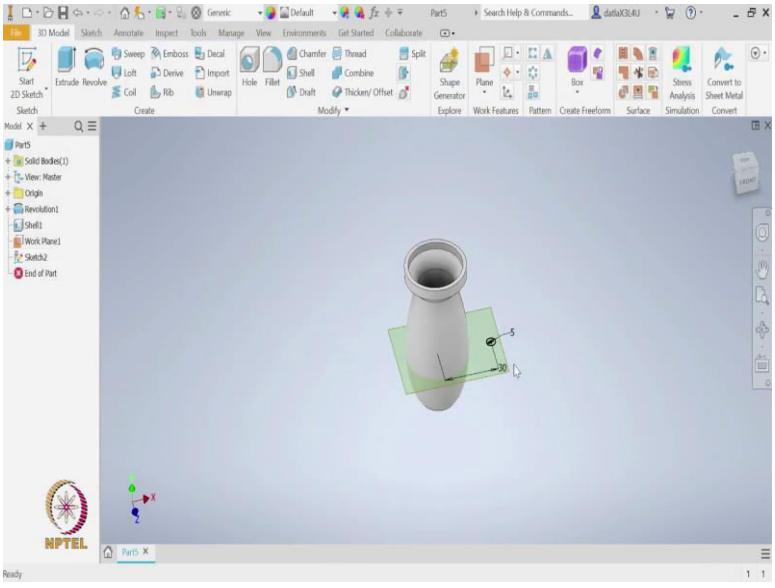
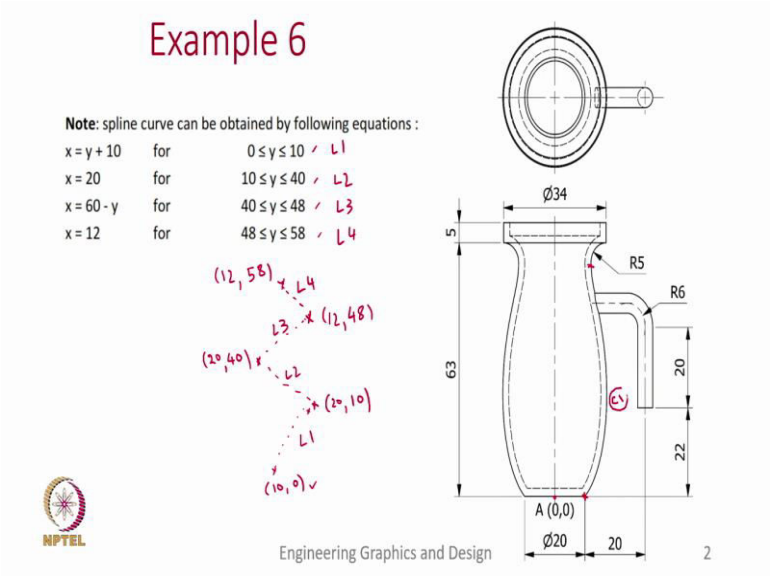
0.000 mm, 0.000 mm Fully Constrained


5
30
Ready



So let us create a circle of diameter 5 and now let us position Horizontal. Let us choose the center of the circle and choose the origin. So, that now it has positioned it at the center or the on the horizontal axis. And now let us dimension saying that the center of the circle from the origin is at a distance of let us look at the example. Here it is given the distance is 20 and then the diameter of 20 so it will be 10 + 20 30. So let us finish the sketch because we are done with the profile.

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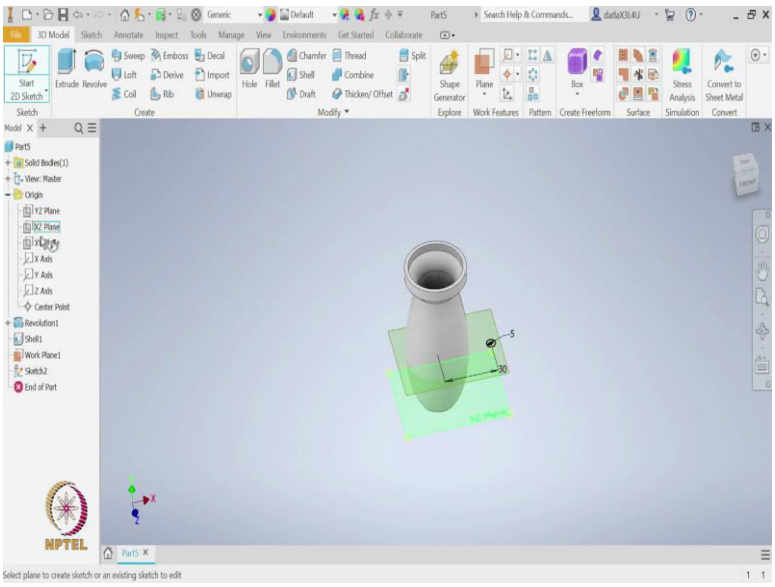
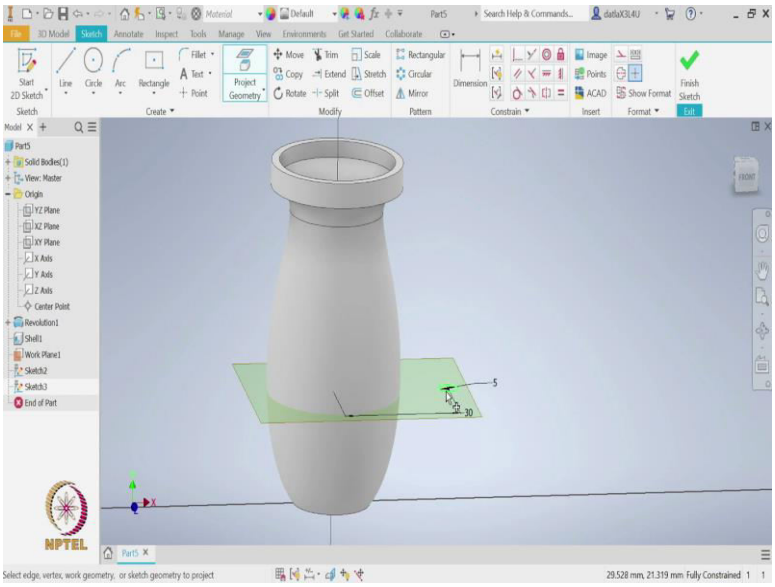
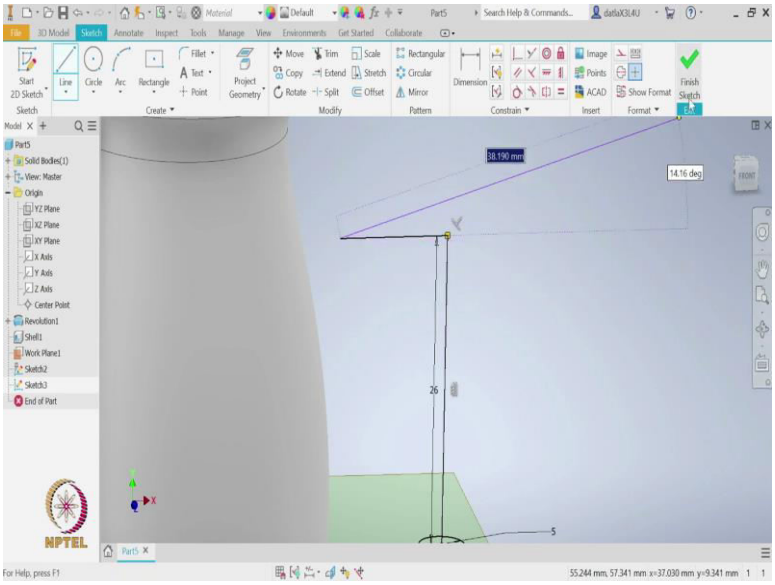


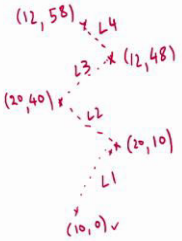


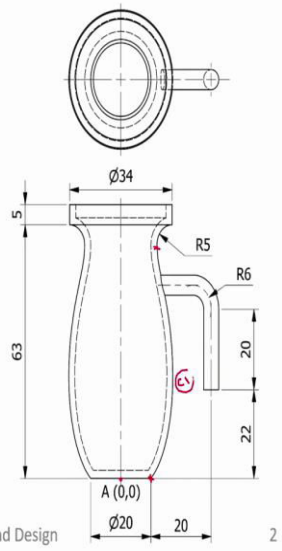


Example 6

Note: spline curve can be obtained by following equations :

x = y + 10	for	0 ≤ y ≤ 10
x = 20	for	10 ≤ y ≤ 40
x = 60 - y	for	40 ≤ y ≤ 48
x = 12	for	48 ≤ y ≤ 58



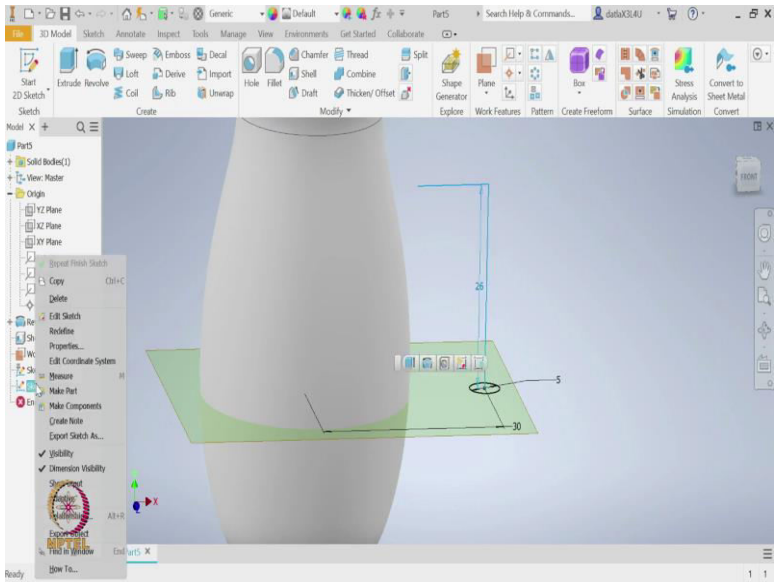


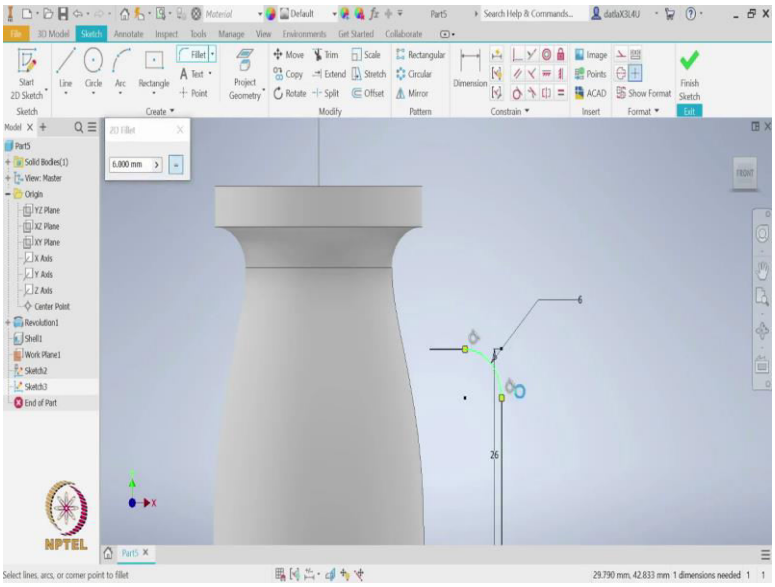
Next we need to create the path and for the path we need to create it on a plane, which is normal to the profile we have created, for that let us go to 2D sketch. Let us now pick the X Y, I can go to origin and pick the X Y.

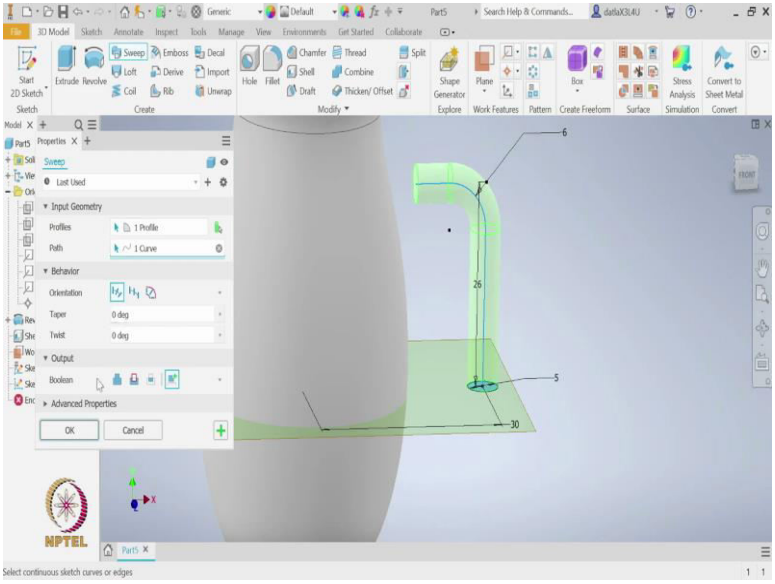
Now you can see that the X Y plane is normal to the profile we have chosen. So before I start to create the path, I need to start with the center of the circle for that let me start with Project Geometry I will project this geometry and the center of that circle. So now it has created a point from which I can draw this vertical line.

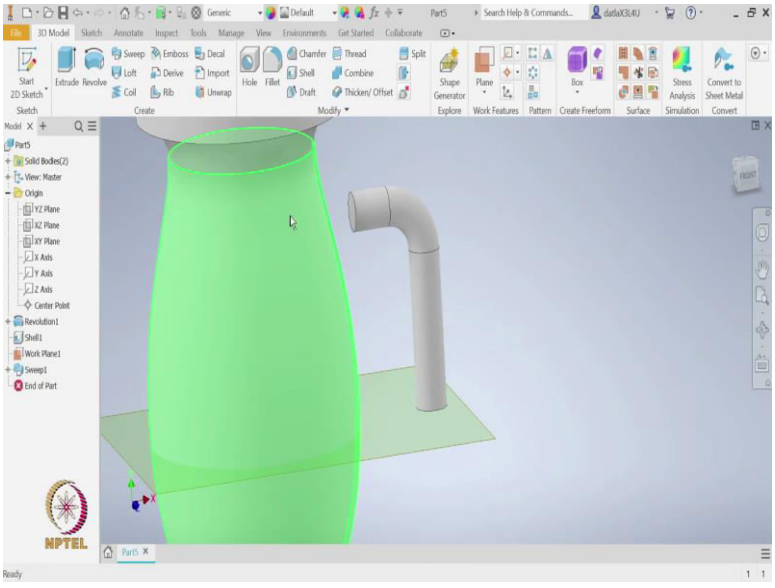
Let us look at the dimensions, first it is 20 and then this radius of 6, so let us go by 20 + 6, 26 and then go left. So this vertical line at 90 degrees is 26 length and then another perpendicular line. Let us leave it as a arbitrary length such that it is not penetrating the object for now. Later we will extend this handle such that it touches the vase. So this is some arbitrary length and finish the sketch. So we see that we have created the profile and this path, we need to make this fillet here.

(Refer Slide Time: 17:28)




Part5
Solid Bodies(1)
View: Master
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Revolution1
Shell1
Work Plane1
Sketch2
Sketch3
End of Part
2D Fillet
6.000 mm
6
26
FRONT
NPTEL
Select lines, arcs, or corner point to fillet
29.790 mm, 42.833 mm 1 dimensions needed

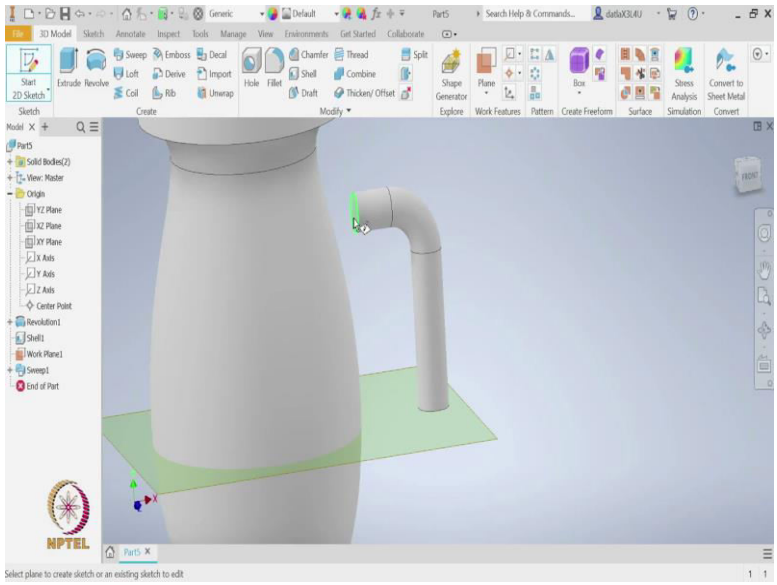

Part5 Properties
Sweep
Last Used
Input Geometry
Profiles
1 Profile
Path
1 Curve
Behavior
Orientation
Taper
0 deg
Twist
0 deg
Output
Boolean
Advanced Properties
OK
Cancel
6
26
5
30
FRONT
NPTEL
Select continuous sketch curves or edges

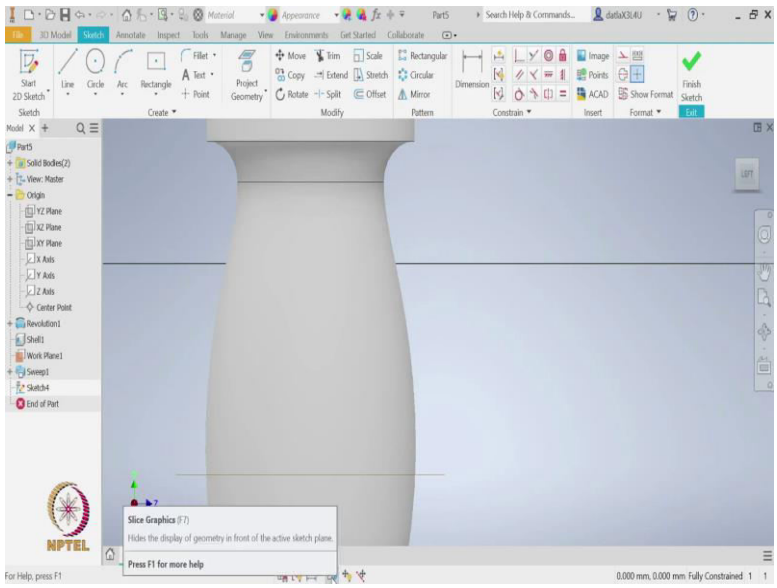

Part5
Solid Bodies(2)
View: Master
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Revolution1
Shell1
Work Plane1
Sweep1
End of Part
FRONT
NPTEL
Ready

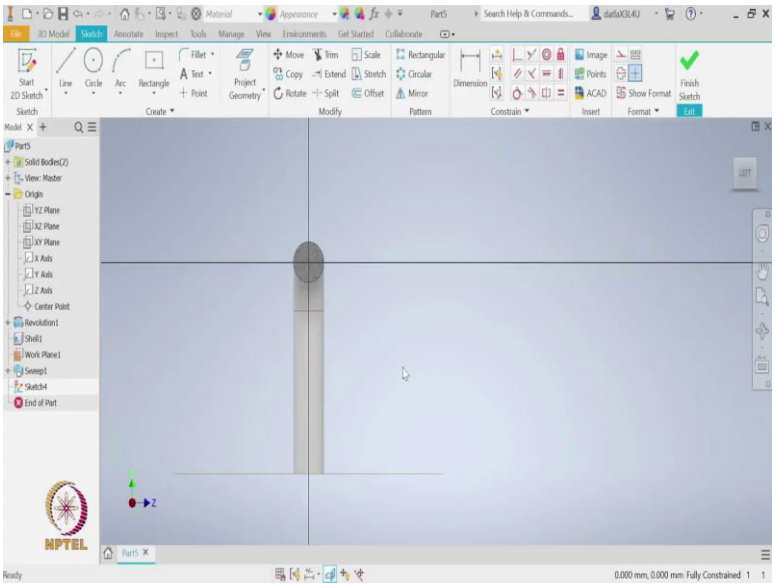
So let us use the, let us go back to the sketch, edit the sketch and choose fillet between these two lines. So the radius needs to be edited to 6 millimeter and the second edge. Now we have created this fillet and we can finish the sketch where we now have both the profile and the path.

Let us go to choose the sweep tool. It has already by default chosen the profile now let us choose the path. And let us leave the orientation which is follow the path and click okay. So now you see we have created a part of the handle. But what is pending is this handle should go and touch the vase.

(Refer Slide Time: 18:23)



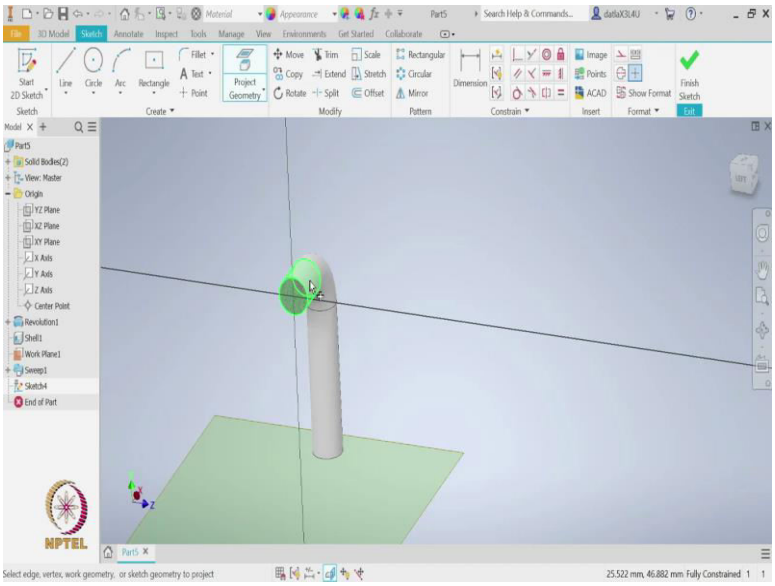


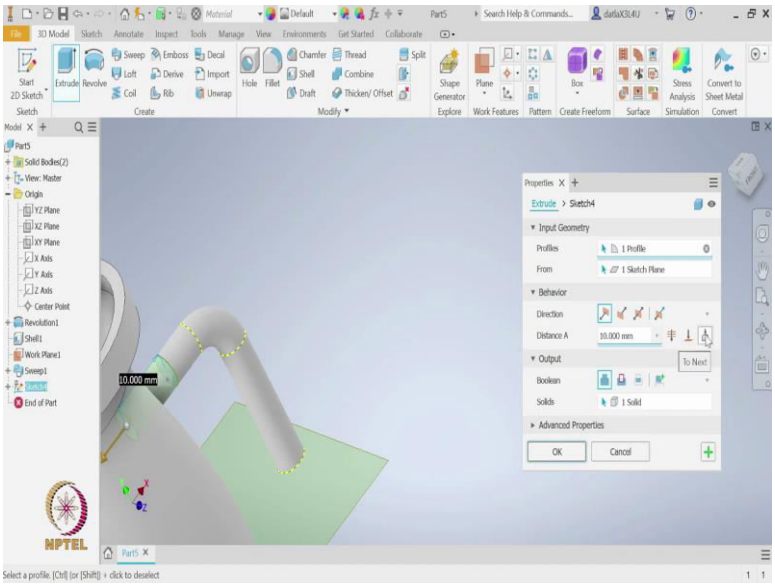


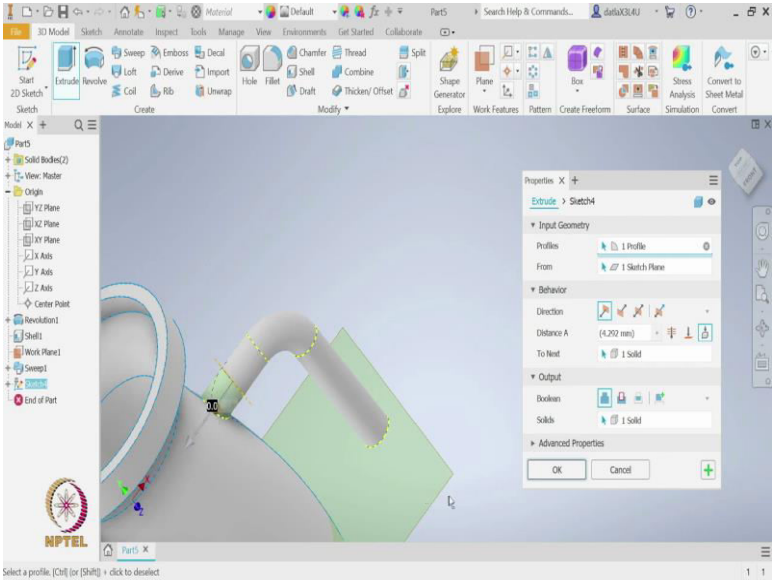
So for that let us do the extrude; extrude tool and I will select this end of the handle as the face of the plane where I want to sketch. Now you see it has gone to the sketch, but the vase is blocking our view.

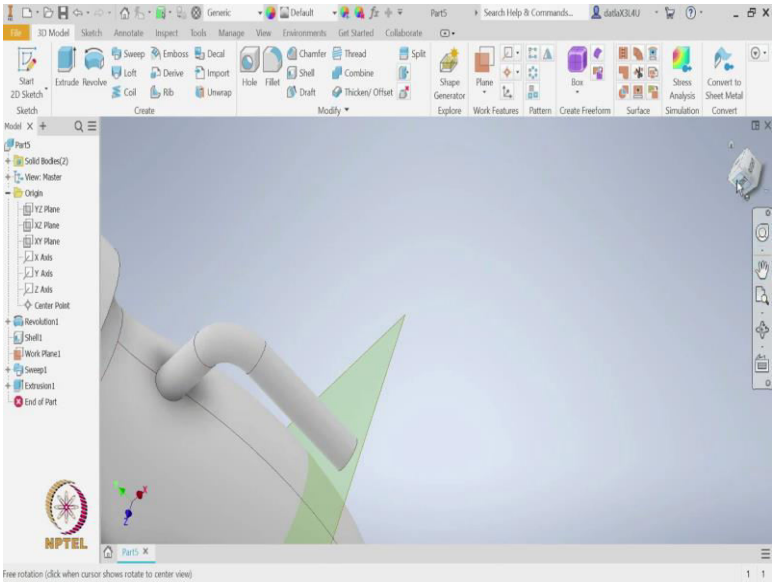
The one way to remove that is to use this, which is slice graphics. So once I do slice graphics, it removes all the solids which are in between you and the plane that we selected. Of course I can toggle it back to bring back the solid but let us again use these tools such that we can focus on the plane what we want to create.

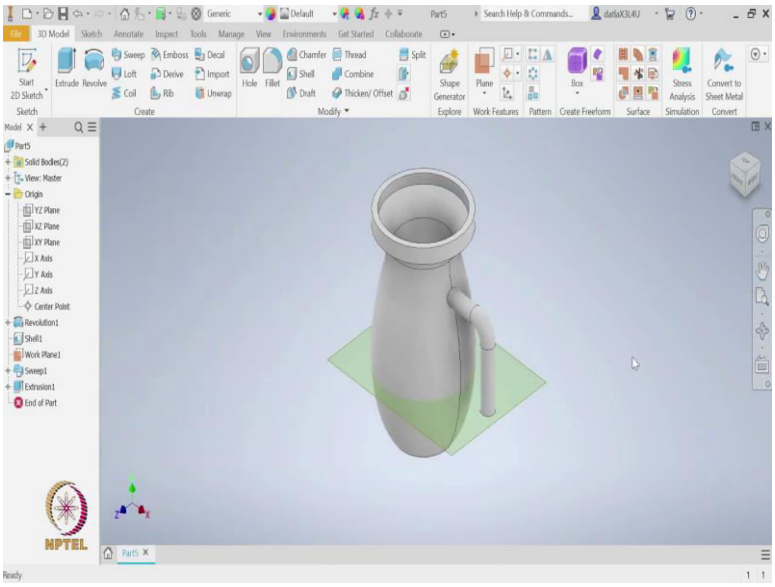
(Refer Slide Time: 19:07)

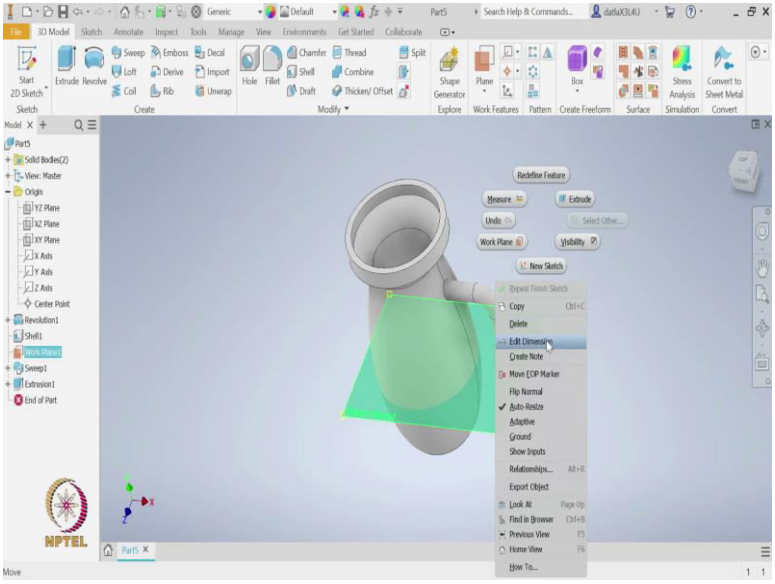


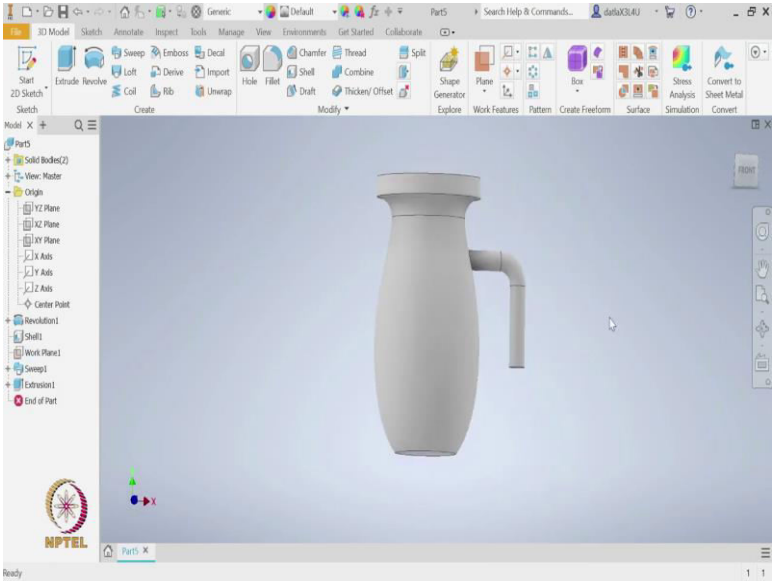

Extrude > Sketch4
Input Geometry
Profiles
1 Profile
From
1 Sketch Plane
Behavior
Direction
Distance A
10.000 mm
Output
To Next
Boolean
Solids
1 Solid
Advanced Properties
OK
Cancel
10.000 mm
Select a profile. [Ctrl] (or [Shift]) + click to deselect


Extrude > Sketch4
Input Geometry
Profiles
1 Profile
From
1 Sketch Plane
Behavior
Direction
Distance A
(4.292 mm)
To Next
1 Solid
Output
Boolean
Solids
1 Solid
Advanced Properties
OK
Cancel
Select a profile. [Ctrl] (or [Shift]) + click to deselect


Free rotation (click when cursor shows rotate to center view)


Part5
Solid Bodies(2)
View: Master
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Revolution1
Shell1
Work Plane1
Sweep1
Extrusion1
End of Part
NPTEL
Ready


Redefine Feature
Measure
Extrude
Undo
Work Plane
Visibility
New Sketch
Copy
Delete
Edit Dimension
Create Note
Move EOP Marker
Flip Normal
Auto-Resize
Adaptive
Ground
Show Inputs
Relationships...
Export Object
Look At
Find in Browser
Previous View
Home View
How To...
NPTEL
Move


Part5
Solid Bodies(2)
View: Master
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Revolution1
Shell1
Work Plane1
Sweep1
Extrusion1
End of Part
NPTEL
Ready

So what I will do is first project geometry, I will project this circular, cylindrical surface so that on the plane it will come as a circle. So essentially what I am trying to do is draw a circle of radius 5. That is what I have done using the project geometry. Then I can finish the sketch. Now since we started with the extrude, it is already trying to create this extrude tool. By default, it has chosen the distance but now we can either choose through all or to where I can say to and it can go until it touches this solid of the vase.

But now I will use this To Next. What To Next does is, let us first delete this solid so that now we can select it until with solid it has to go. So if we select this vase, now it is extruding from this circle until it touches the solid. If I click okay now you will see it has created this extension where it touches the vase and the shape of the vase is also matching.

So here is the final product that we are asked to do. So of course, I can select this plane and make it uncheck the visibility so that it looks close to the problem that we are asked for. So with this we complete this lecture we will meet in the next lecture. Thanks for your attention.