Engineering Graphics and Design
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Week 10: Part Modelling 2
Lecture 1
Spline Sweep Shell

Welcome back to this course of Engineering Graphics and Design we are on week 10. In this week we will be continuing about part modeling that we have been discussing in the last one two weeks. So in this week we will be looking at a few more tools that we can use to create solid models such as the splines, sweep, coil and shell which will be followed by an example of using few of these features.

We will also discuss this week about how to create drawings from the created solid models. So that we will be looking at a different lecture, in this lecture we will look at these four commands or four tools which we will be using in the software. So, let us start with the spline tool.

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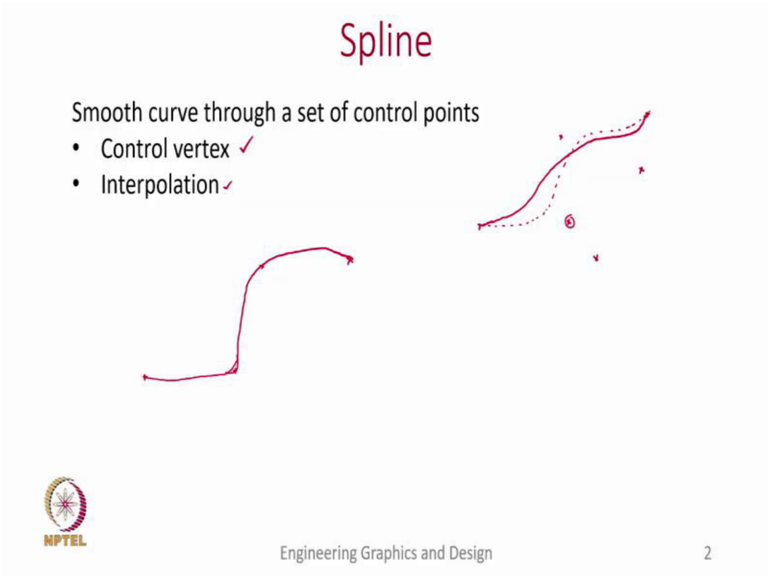

Spline tool creates smooth curves, these are not curves of known form, but mostly these are free form curves where you specify few control points and through those control points you get a smooth curve which is mostly used in most artistic designs and all. So in Inventor we can do this using two methods one is the control with vertex method and second is the interpolation method.

Essentially the difference is this. So, in the first control vertex method, you first specify the control points which we also call as control vertices. Depending on these points, we get a smooth curve which starts from the first point and ends at the last point. So it only meets the first and last control vertex, but the shape of the curve is decided by where you place the intermediate control vertices.

So for example, let us say this control vertex is moved further. So this you edit it and move a little further, then the shape of the curve will tend to move closer to the modified point. So probably the curve will look something like this. Similarly, you can move other control points to modify the shape of the curve.

The second, this is what we have discussed in now is the control vertex method. The second method is the interpolation method where once you specify the control points, you get a smooth curve passing through all of the control points. So maybe, something like this, yes, here, it should pass through all the control points.

But most importantly, what we are saying is we are looking at a smooth curve joining all these points. So at the background, the mathematics of it is mostly these are represented using polynomial curves and one can go into those equations. But for us, we are only looking at the output, we are mostly as the end user, we want to see what is the smooth curve we can have between, let us say, two parts of a design component. So let us look into the software and see how we can create these Spline curves.

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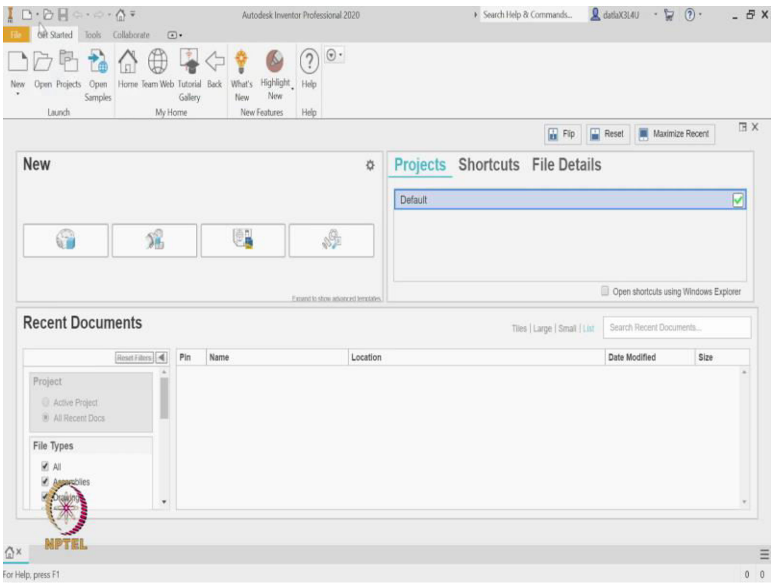

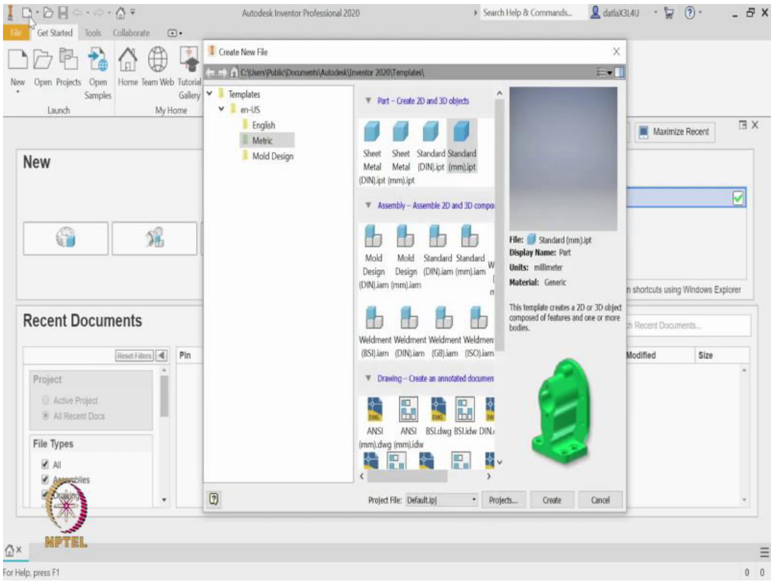
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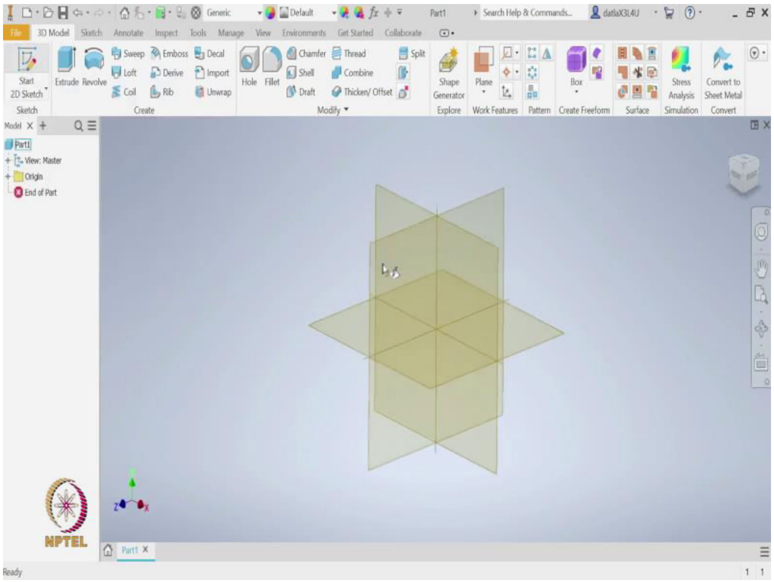
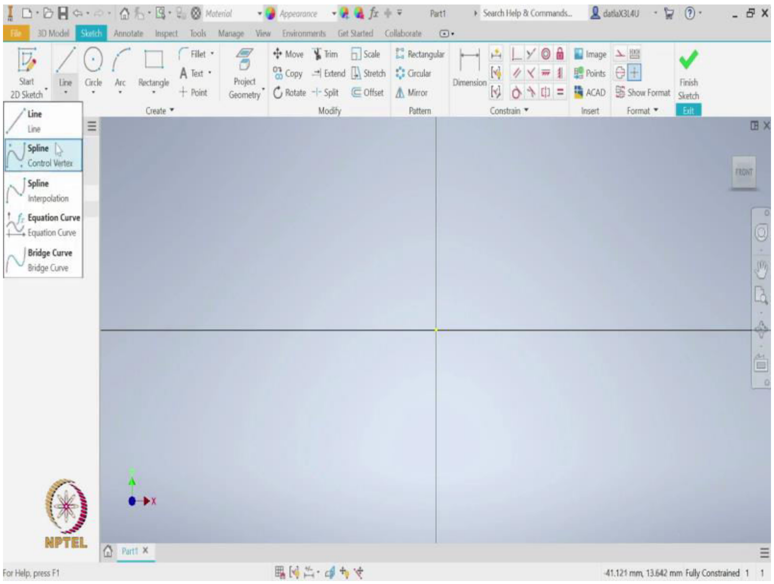

3D Model
Part1
Ready


Sketch
Line
Spline
Equation Curve
Bridge Curve
Finish Sketch
Part1
For Help, press F1

So we are at the initial interface. So first, let us start by choosing a part template. So let us first create a 2D sketch by choosing let us say, X Y plane. And once we go into this create panel and then the line tool; there are several other options in addition to the line. The second and third options are the splines, the difference is the first spline is the control vertex and the second spline is the interpolation. So let us get started with the control vertex and see how we can create curves.

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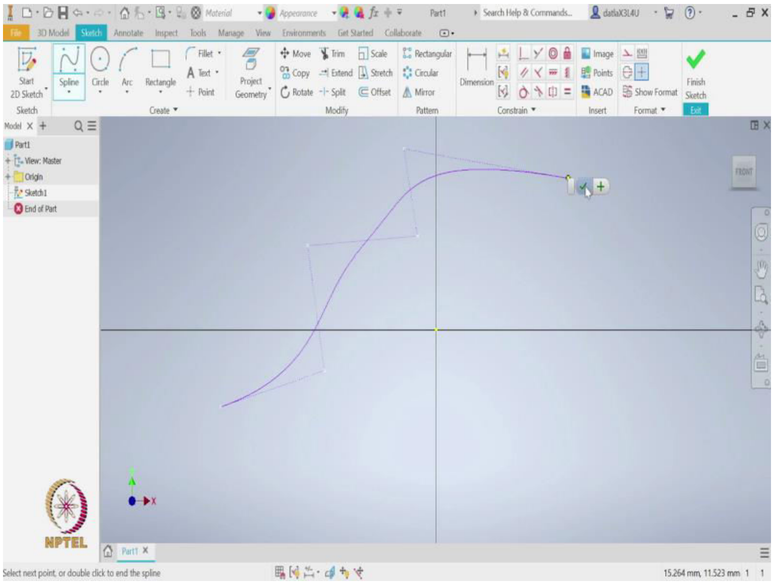

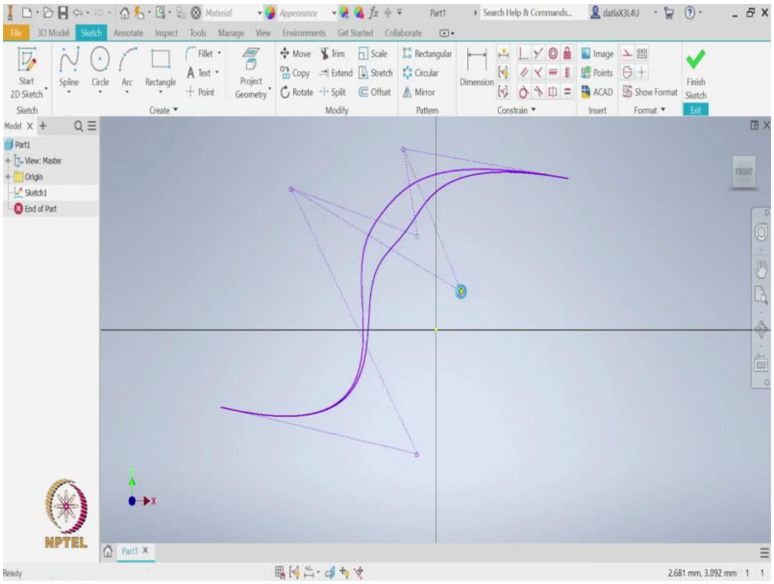

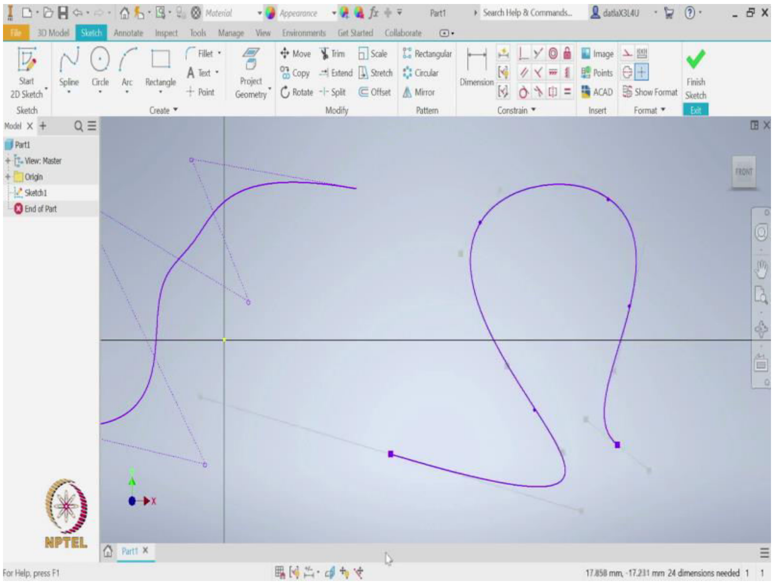

So we can start by choosing those control points. So left click to start the first control point. And then keep continuing creating those control points or control vertices using number of clicks. So, when we are done with the control points, then we can click this okay and then we get a curve. So after you create a spline curve, you can always modify by simply going into one of these control vertex and moving it.

So previously, as I mentioned, if you are moving a control point the curve tends to move with the control point and tries to go closer to the point. So like this, you can control the shape of the curve. So let us now look into the second category of these spline curves, which is the interpolation spline.

So let us choose this spline by interpolation. Here we said that, we can choose a number of control points and the curve passes through all of these control vertices. So here is point 1, point 2, point 3, point 4, point 5 and let us say this is the n point and then let us click okay. So now we can see the curve which is coming up, the spline curve which we now created is passing through all of these control points. So these are the two kinds of splines we can create. And later we will use these splines to create solids. But Let us now move back and look at the second command.

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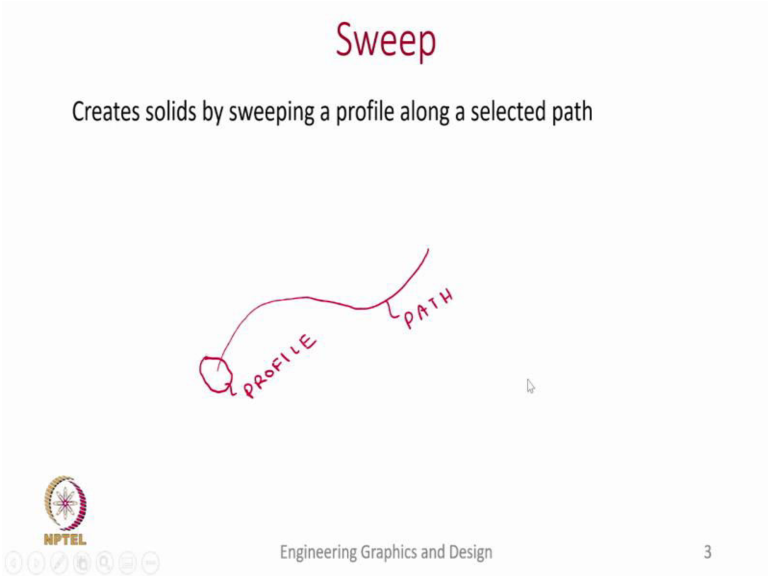

The second tool we will be discussing is the sweep tool. Essentially, this is again used to create solids by sweeping a profile along a selected path, you can create a let us say circular profile. And then depending on if you create a path let us call this the path curve and this is the profile. So once you create a profile, the same cross section can be maintained all throughout this path and then it creates a solid. So let us move to the software to see how we can use the sweep.

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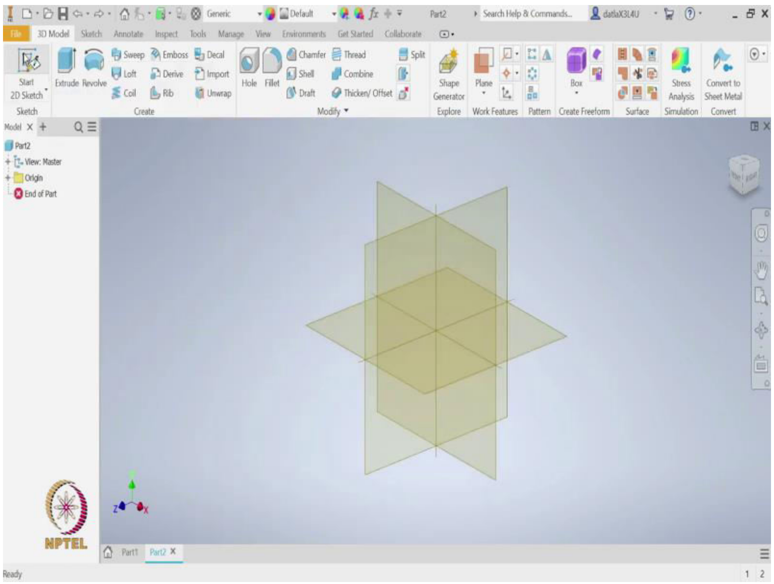
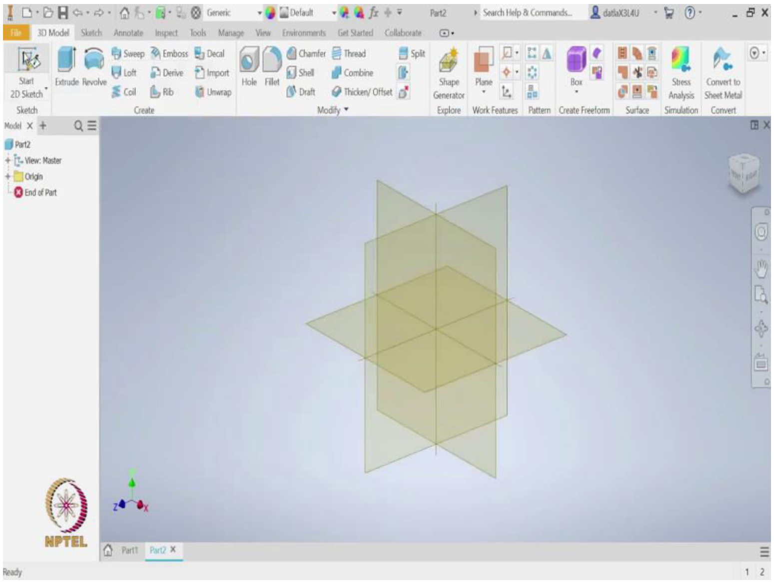
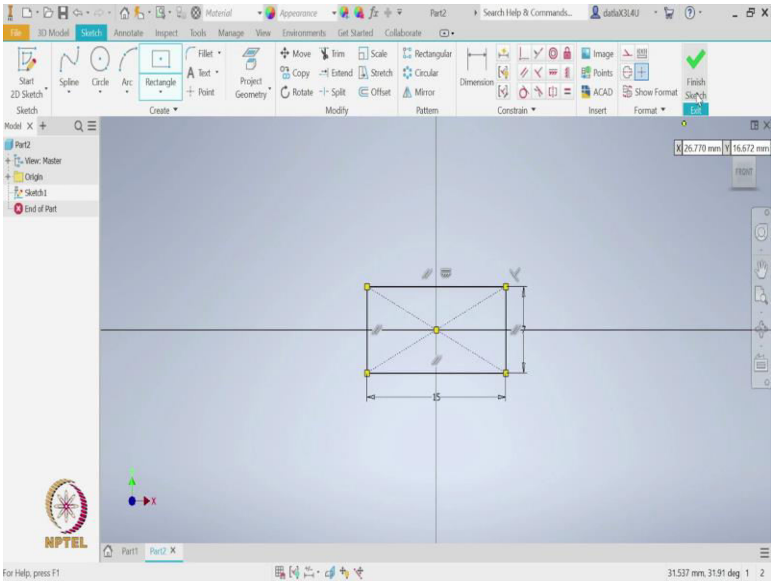
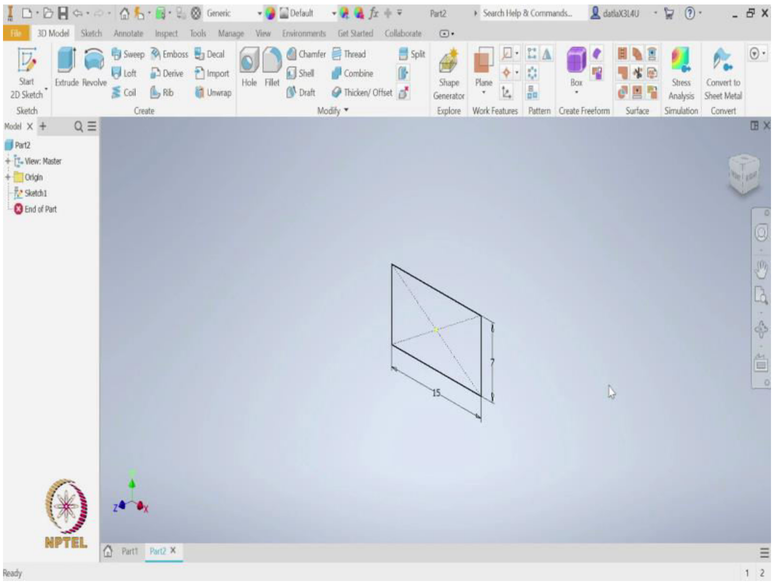



Part2
3D Model
Sketch
Create
Modify
Explore
Work Features
Pattern
Create Freeform
Surface
Simulation
Convert
View: Master
Origin
End of Part
NPTEL
Ready


Part2
Sketch
Rectangle
Create
Modify
Pattern
Constrain
Insert
Format
Finish Sketch
View: Master
Origin
Sketch1
End of Part
15
NPTEL
For Help, press F1


Part2
3D Model
Create
Modify
View: Master
Origin
Sketch1
End of Part
15
NPTEL
Ready

We will start with a starting a 2 D sketch. First let us create a profile. Let us say we will choose the X Y plane and then create a let us say rectangle using two point center. So I will set an origin as the center and then create a rectangle of 15 by let us say 7. So let us finish this sketch we are done with the profile. So what is now remaining is we need to create a path.

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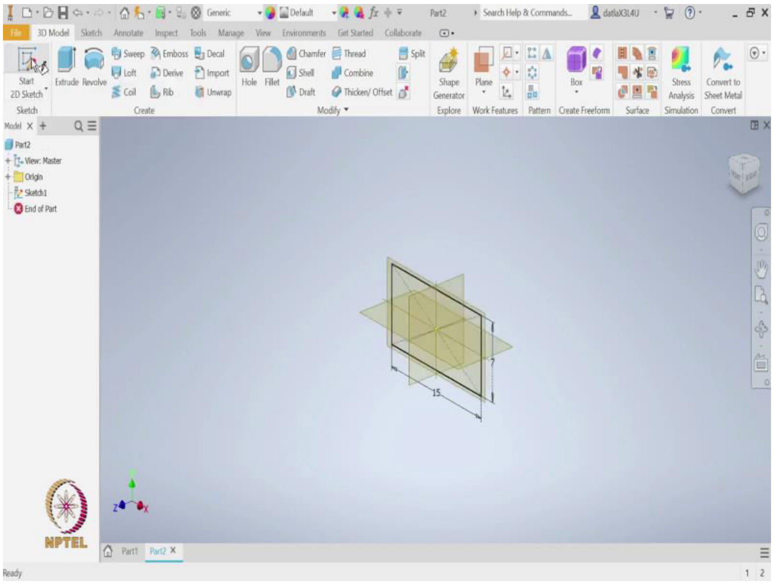

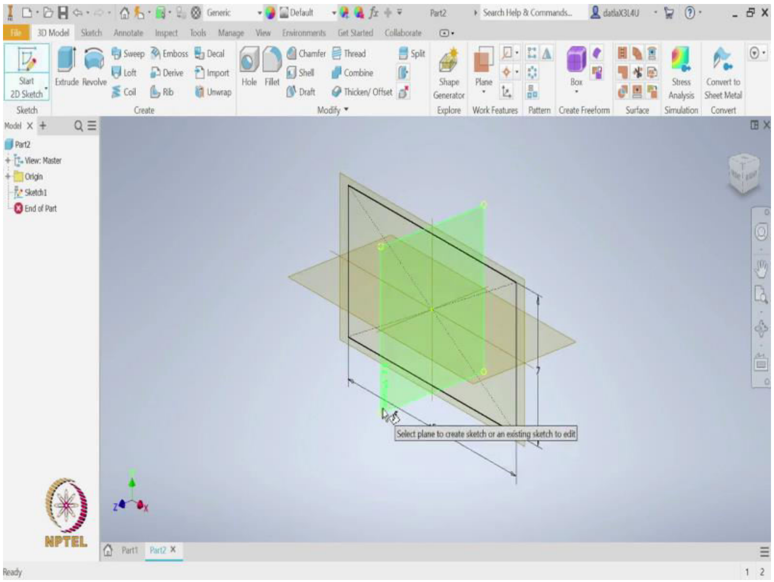

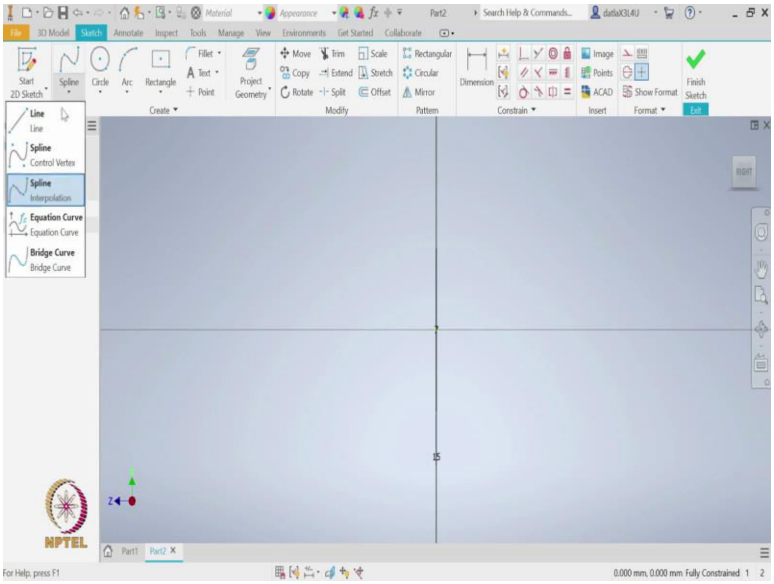

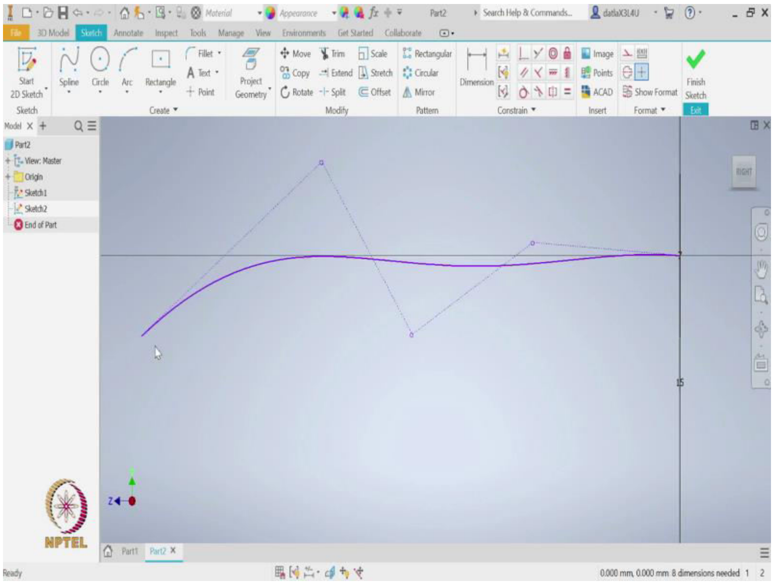

For that let us again start with 2D sketch. And then maybe I can start with this plane. And now let us create a spline with let us say control vertex. So we will start with the origin which is the center of the profile we already created and then create a curve. So let us complete this spline and finish the sketch so that we have defined the spline.

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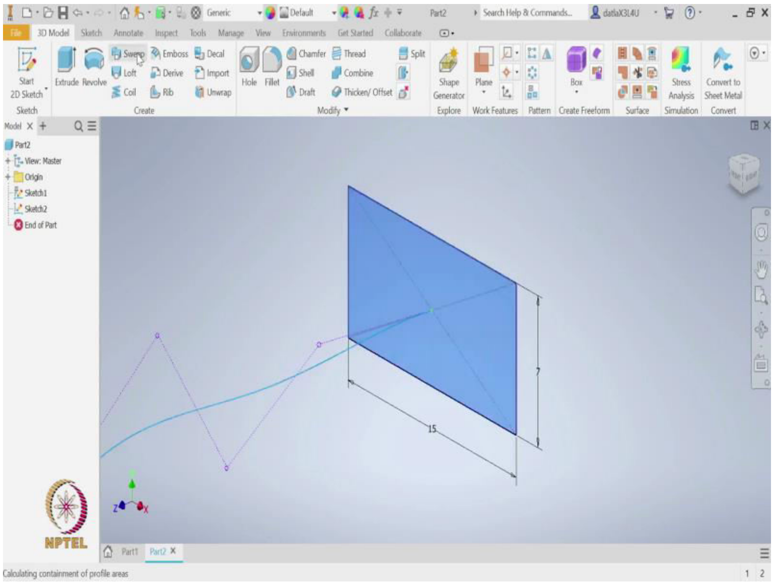

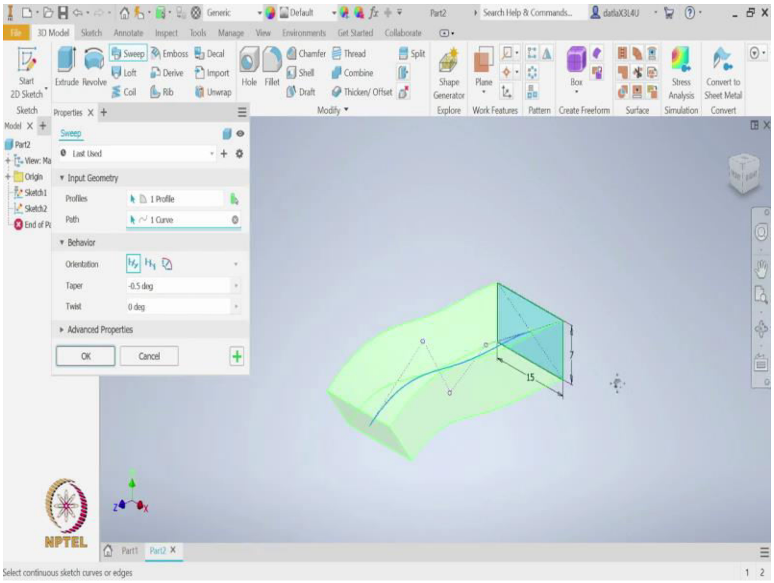

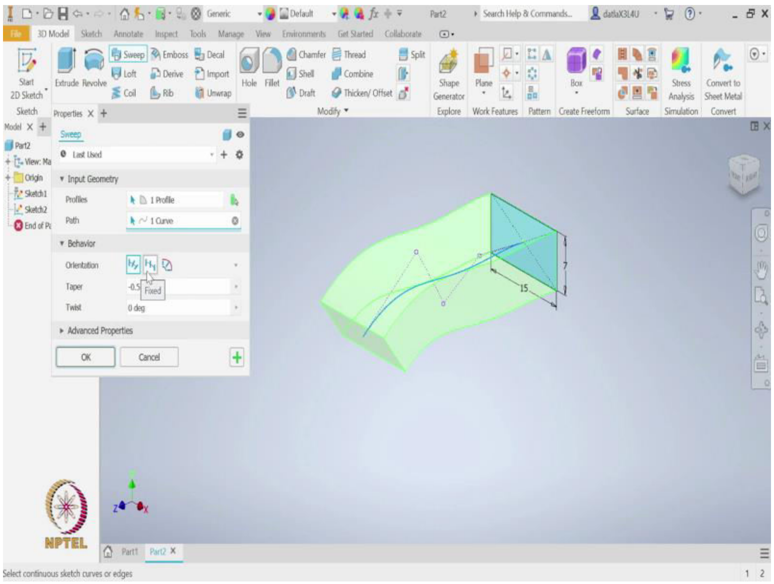

So now I can go into the sweep command here. So when I select sweep, since we have already created a profile and a curve it has by default chosen those but let us say we will delete those so that we can manually select what profile you want.

So let us start with the profile. We will say this is the profile we are choosing and for the curve, we will choose the spline that we have created. Let me zoom in. So for the selected profile and the selected path, this is the solid we are creating. But before I click okay let us look at what are the other options we have the other options we have are first is the orientation.

In orientation as of now we are saying follow path. But there are two other options saying the fixed and the guide. So now let us look at what happens when we say it is a fixed. So now what is happening is the profile is maintaining the same orientation throughout the solid. So let us say initially it is parallel to the X Y plane the same parallelism will continue throughout the sweep but in the first option of which is called a follow path.

What it is doing is, let us say initially you have the profile perpendicular to the curve, the same perpendicularity will continue throughout the curve. So that is what you can see in this option. But now let us look into the other two options we have. One is taper and twist.

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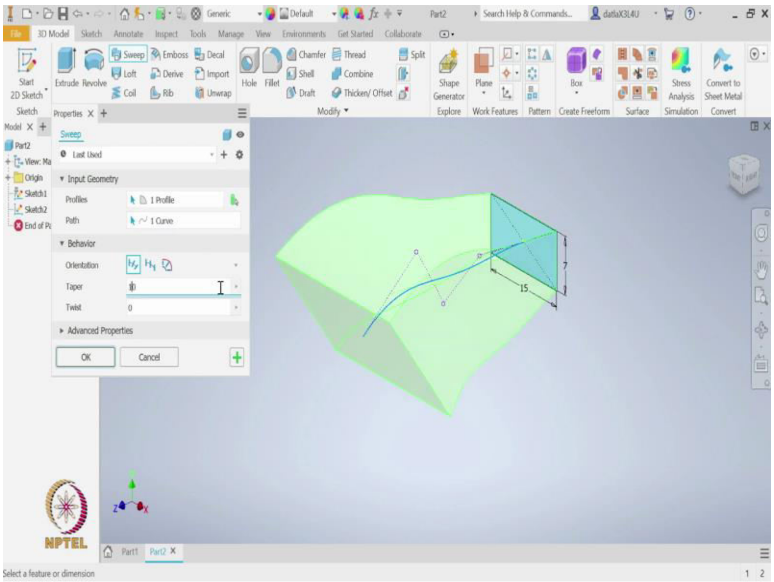

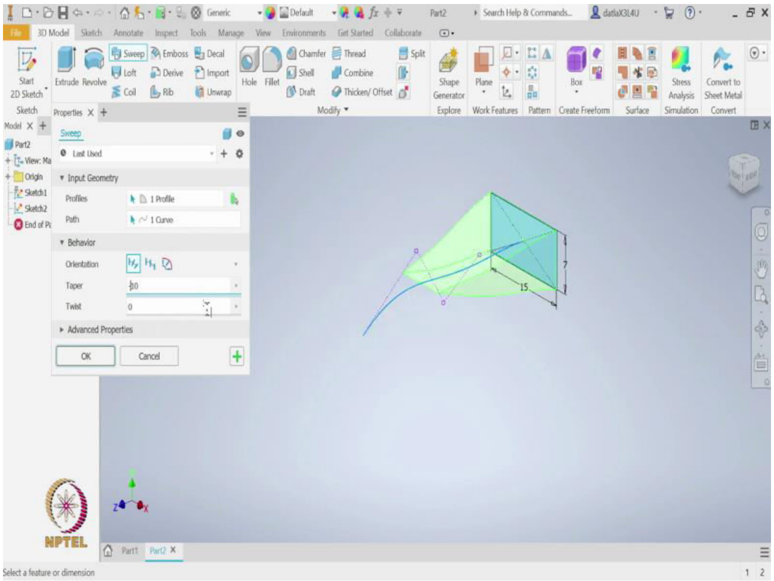

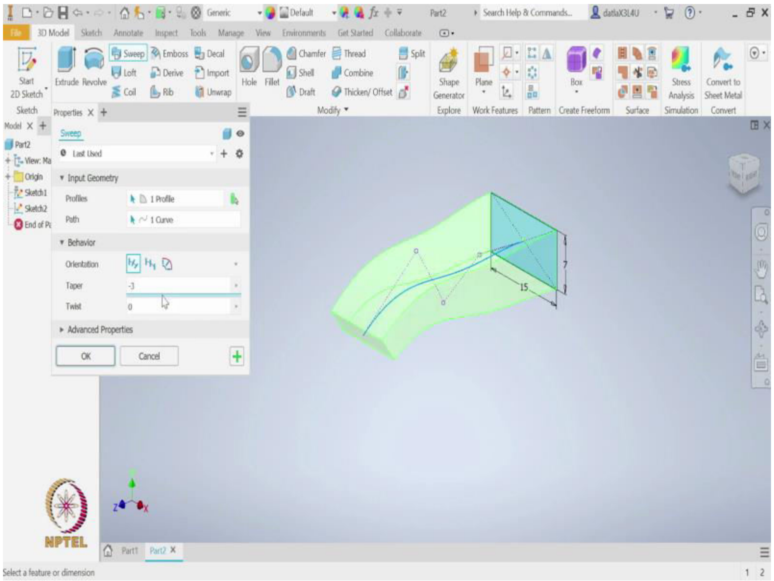

Taper, let us change it to 10 and you can see what happens. So essentially, what it is doing is while it sweeps from the initial profile to the final profile, gradually it increases the taper by an angle of 10. Of course, it is an increasing taper, but I can give a minus sign to show that it is decreasing. Sometimes you need to be careful with the value we give, for example here 10 degrees is too much, because it is not able to reach the end of the path. So let us say only three degree, then we can see that starting from the initial profile, the cross section gradually decreases.

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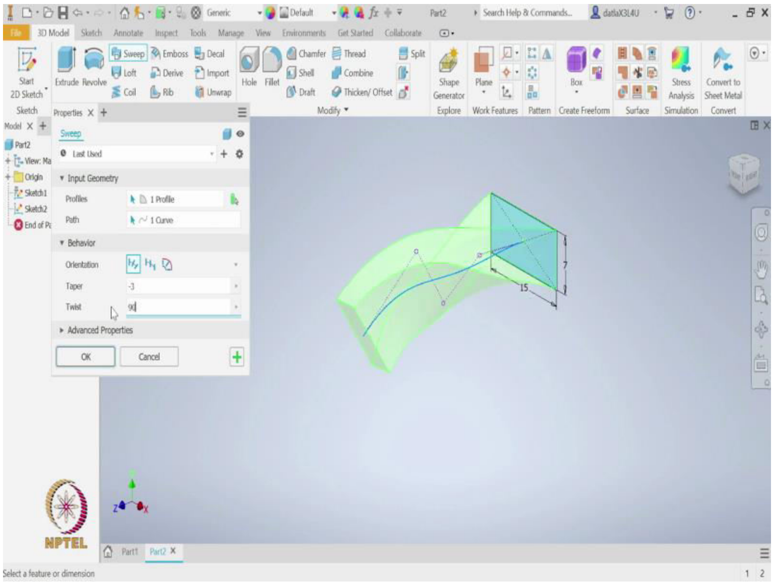

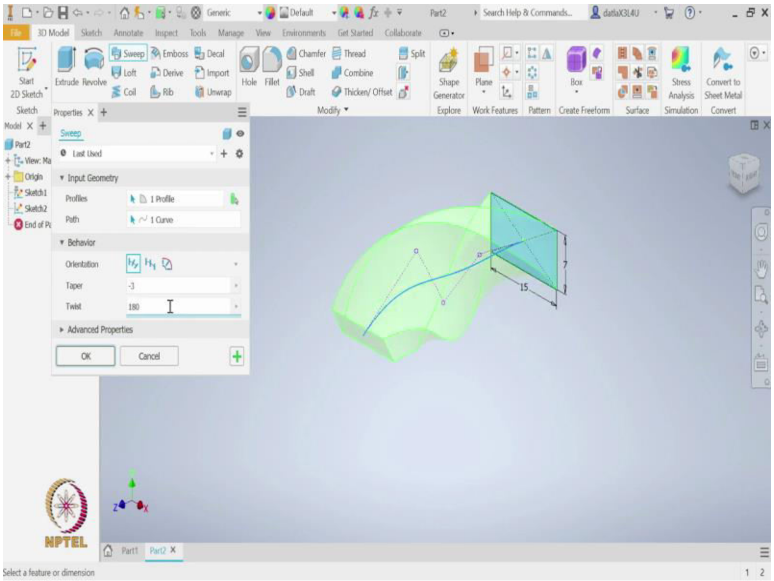

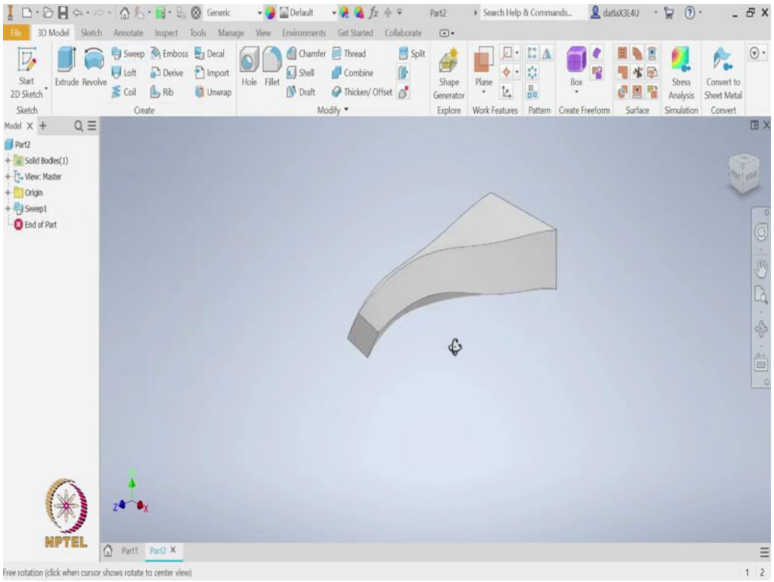

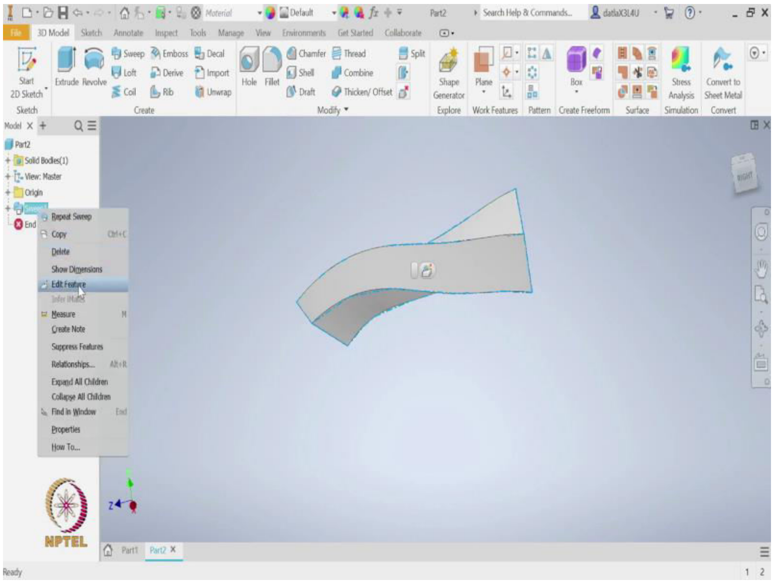

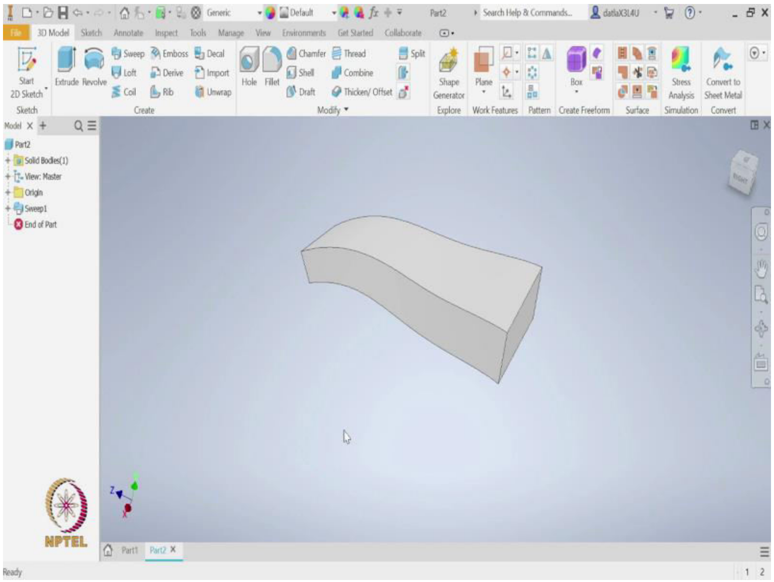

So there is another option of twist, let us give a value and see what happens. So let us say the twist is 90 degrees. So now you can clearly see that it is almost like taking a pen and saying that initial profile and final profile instead of they are parallel, you twist them.

In this example we are given that this twist angle is 90 degrees. So you can see that the final profile is oriented at 90 degrees to the initial profile. Of course, we can keep changing this value so that it will be more obvious what we are trying to do. So if the twist is too much, since the sweep we have taken is of not relatively large length, it looks more distorted.

Probably we should have lesser than 90 degrees to show it a more uniform or a gradual twist. I hope now you are able to see the twist. Let us say we do not want to have the twist, we can always go back to the sweep command, right click on it and select the Edit feature. Now we get back this dialog box where I can change let us say the twist is something like 15 degrees, which is a gradual twist.

So this is one way we can create sweep solids. So remember, when I created this path, I have created a plane so that my curve is lying in a 2D plane. But let us say you want to create let us say like a spring where the path, spring is a solid, where you again can use the sweep because the cross section remains constant. And the path here is not a 2D curve, but instead a 3D curve. So let us now see how to create those kind of 3D curves which we can define it as the path.

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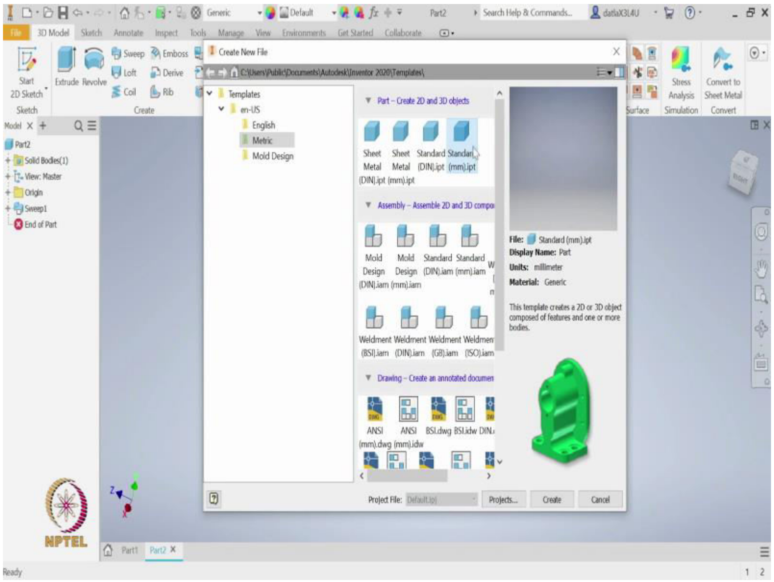

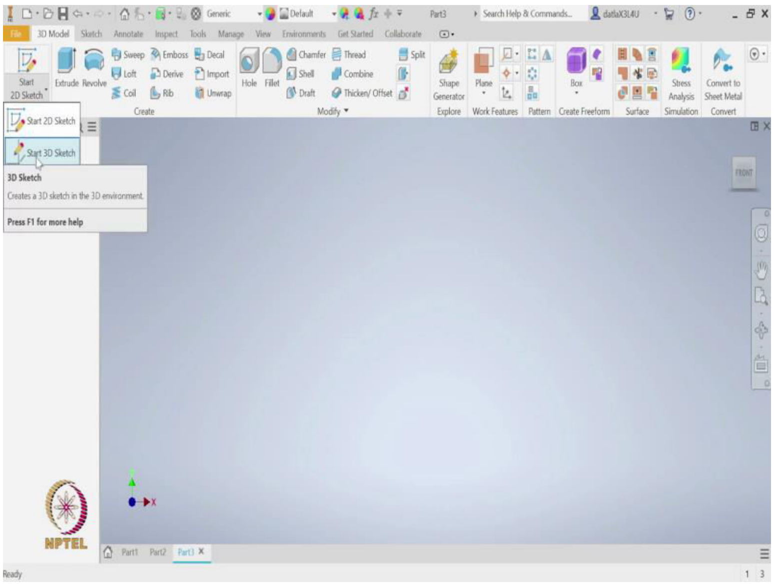

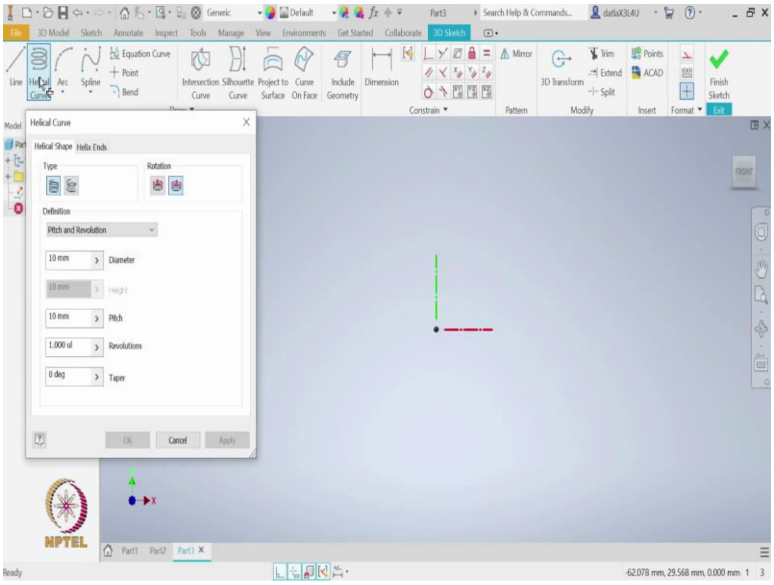

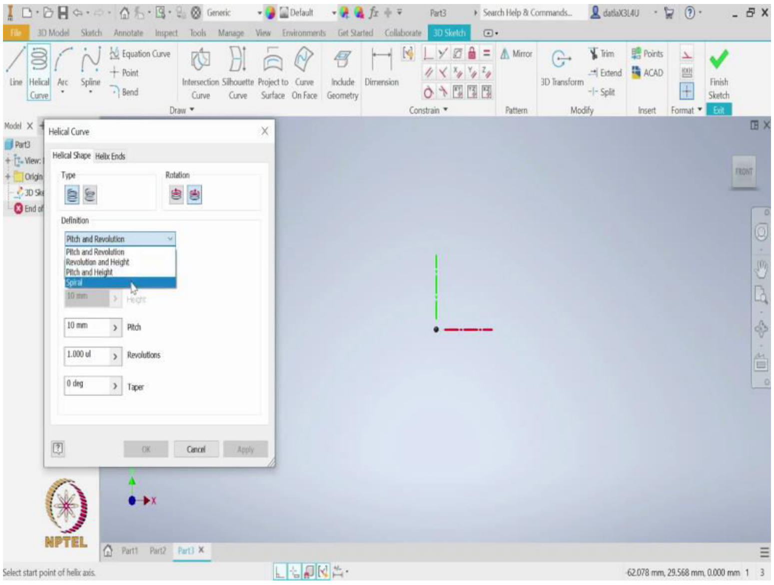

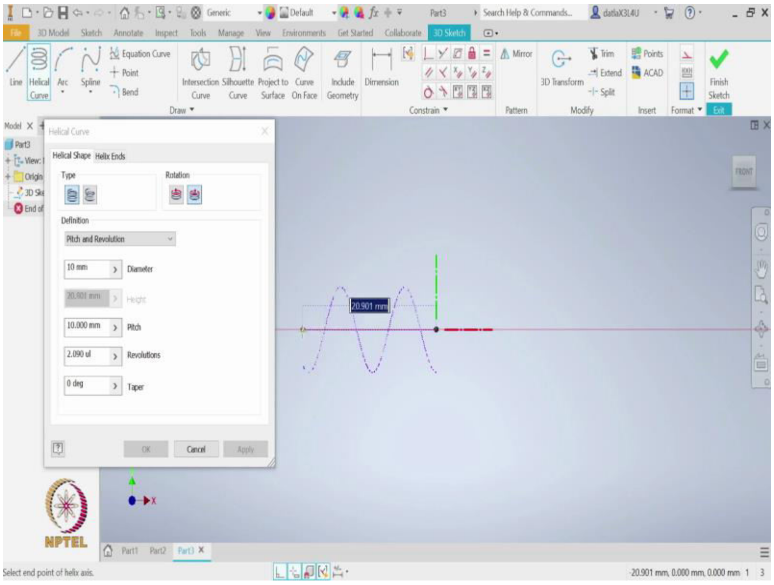

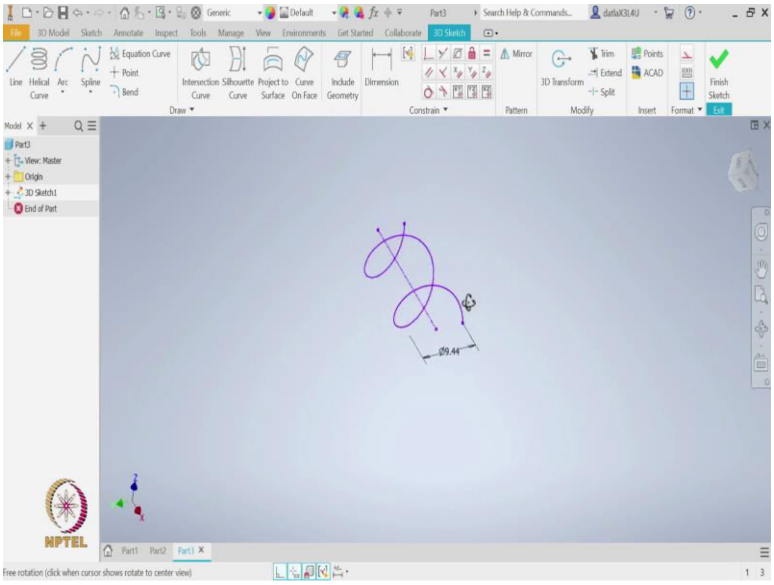

So for this let us again start with the new template. This time to start creating a sketch instead of selecting the 2D sketch, we click this triangle. So it shows that we can start with a 3D sketch. So when I go to the 3D sketch, it gives me again a bunch of options how to create 3D curves in this will only focus on this helical curve. So let me select this helical curve. So when I select this, there is a drop box coming up with several options. So first, let us focus on this one where it says constant helical curve.

Later we will come back to the other options. So in this constant helical curve again, by definition, we can say there are four different options it is giving, where you can define the pitch and revolutions to define the helix or the revolution and height or the pitch and height and lastly, the spiral. But let us start with the first option pitch and revolution.

So first, let us specify the starting point. So let us say it is at the origin and now it is asking us what is the height? So maybe let us give it as 20. Now it is asking me the diameter, diameter of the helical coil. So let us say it is diameter of 20. Now let us create, so let me rotate this curve to show that it is a 3D curve.

Now I think all of us can see that it is a 3D curve. Essentially we define this 3D curve by the total height of this curve and the diameter of the curve. Of course we can always go back and change this, modify these dimensions, let us say it is 10.

Let us finish the sketch and for us to create a solid using the sweep, we are done with creating the path but now we need to create the profile. So how do we create the profile? For that we again we need to create it with a by selecting a sketch and drawing the profile. Let me create a plane by using a different option now.

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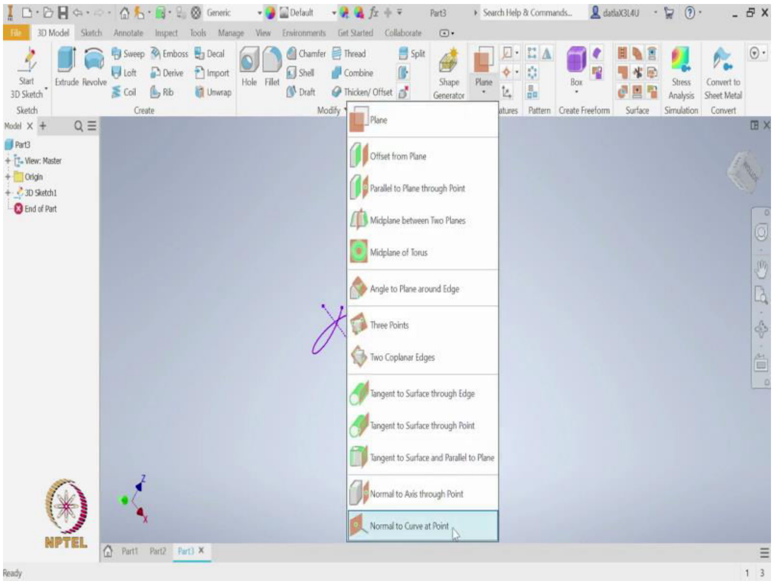

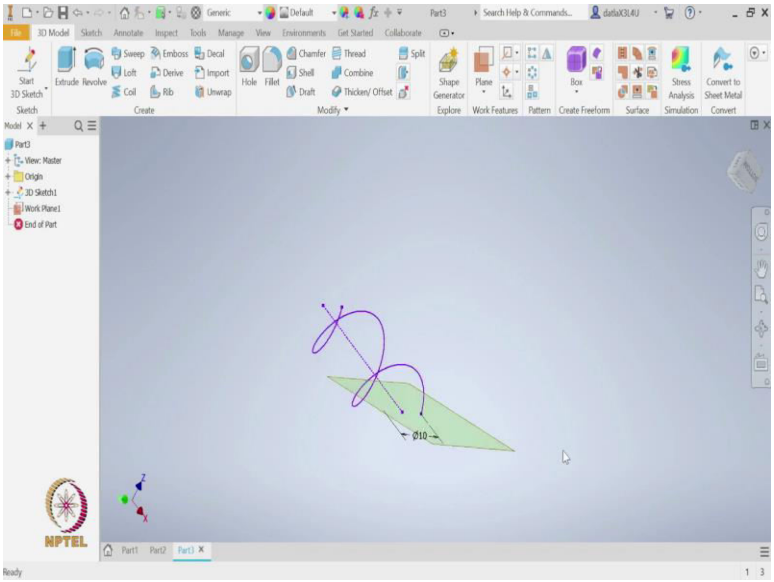

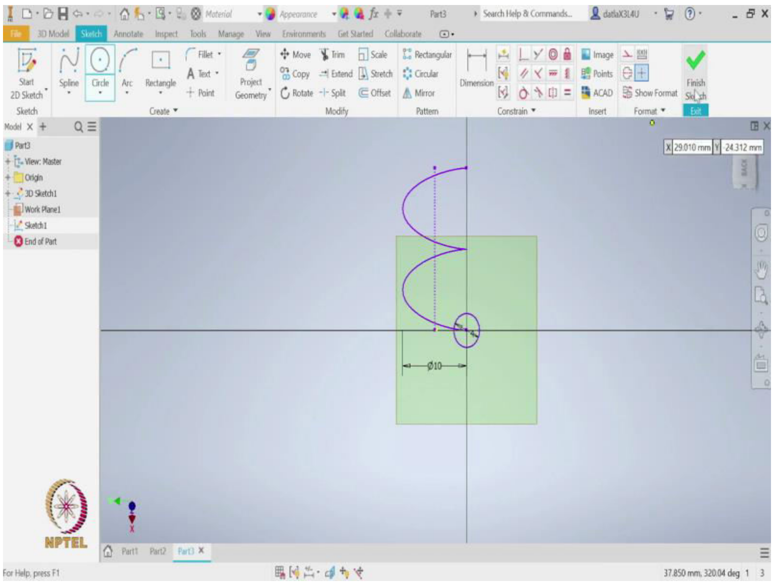

So in the last we have to create a plane using normal to curve at point. So essentially what it is saying is if you have a curve, you can choose a point on the curve and it will create a plane which is perpendicular to the curve. So let us choose this and choose the let us say the starting point of the curve, so it has created a plane. So now let us go to 2D sketch, select this new newly created plane and on that plane I can now create a profile. So now let us create a circle of let us say diameter 4 and finish the sketch.

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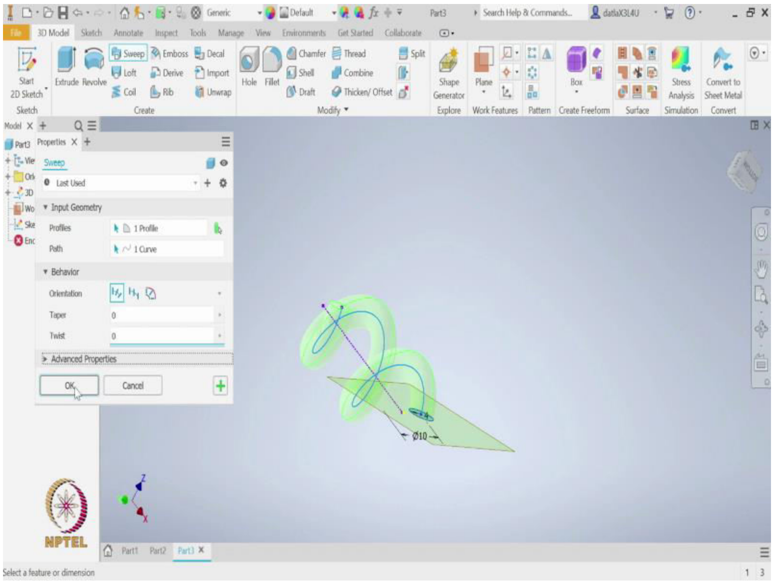

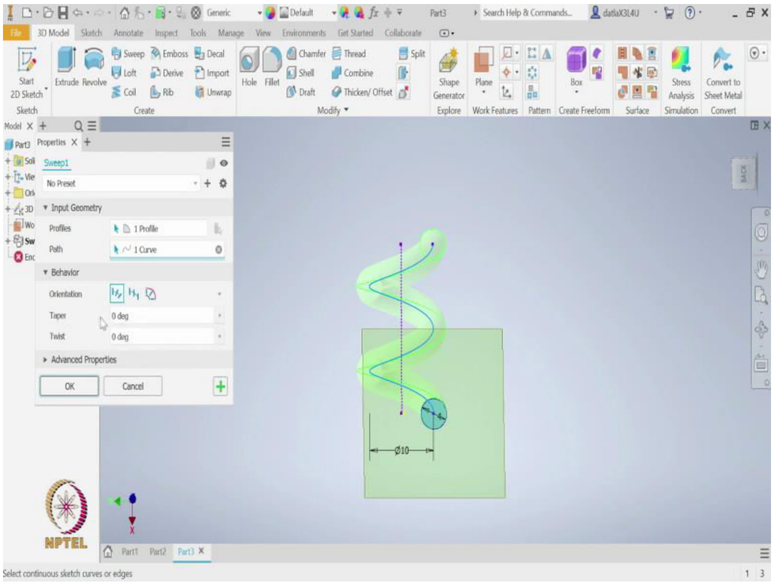

Now I can use the sweep command, it has already chosen the profile now I have to select the path which is the helical curve that we have created and click okay. So of course there are many other options similar to the previous sweep where I can choose the angle, so as of now let me choose the taper angle as 0 and the twist angle is also 0.

So this has created a sweep solid which is close to the spring which we have been discussing. Of course if you look at the sweep, there are other options as well. Like what we discussed we can again modify the taper or the twist but in addition to this, we can go back to the 3D sketch.

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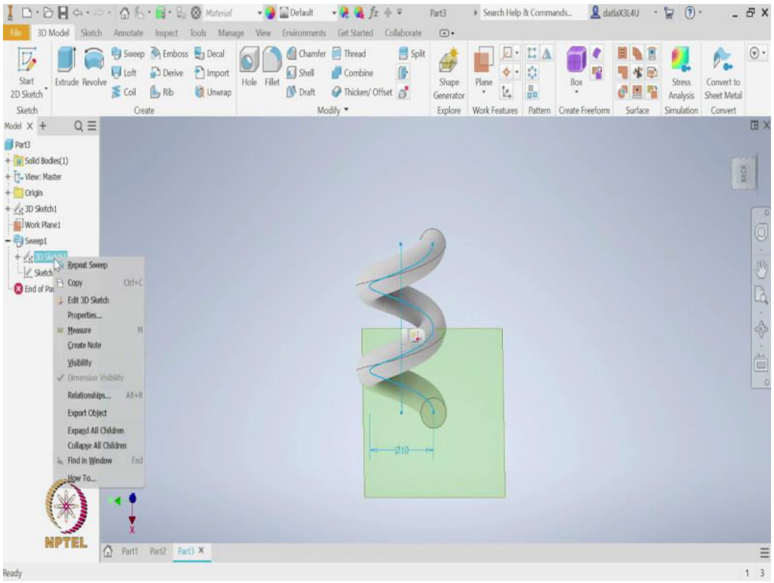

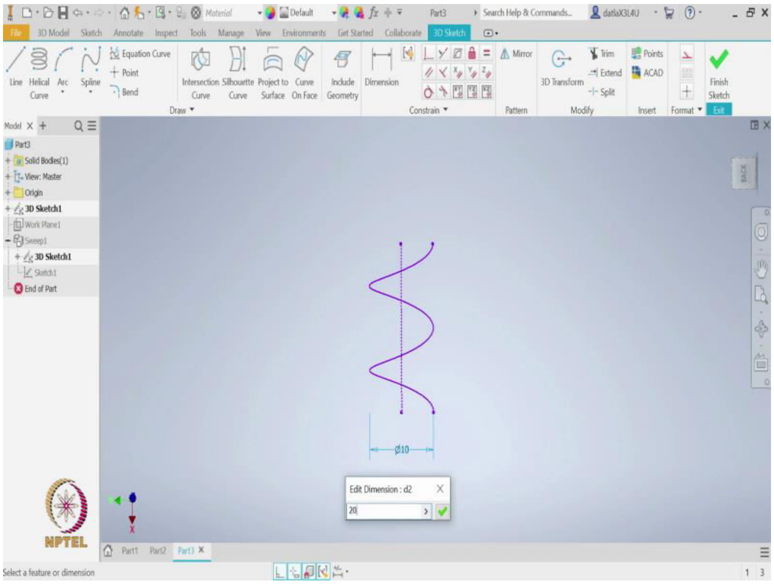

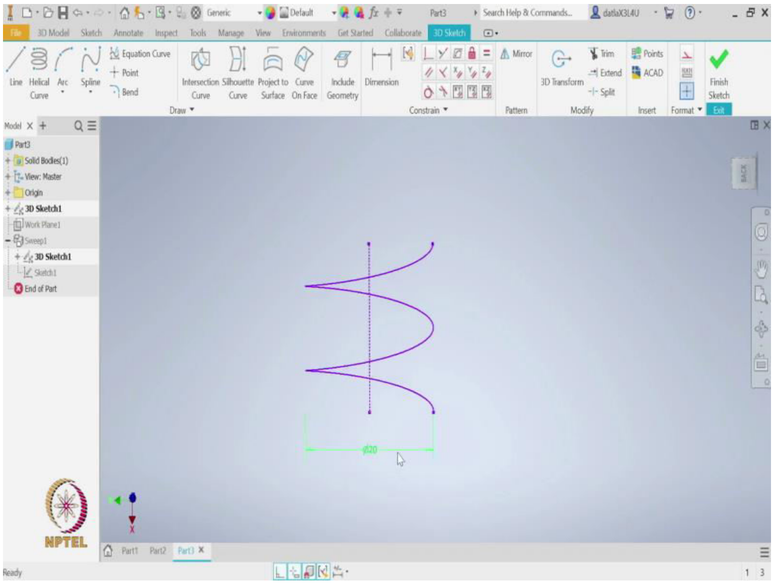

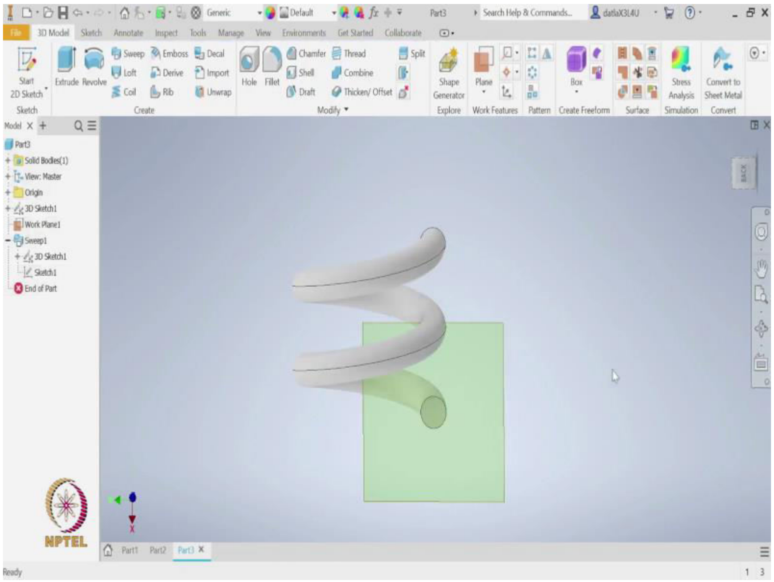

Let us say we will edit the 3D sketch and when we edit the 3D sketch we can look at how to modify the helix, let us say this diameter, let us make it a little bigger 20, finish the sketch and it should give us the modified spring using the sweep command.

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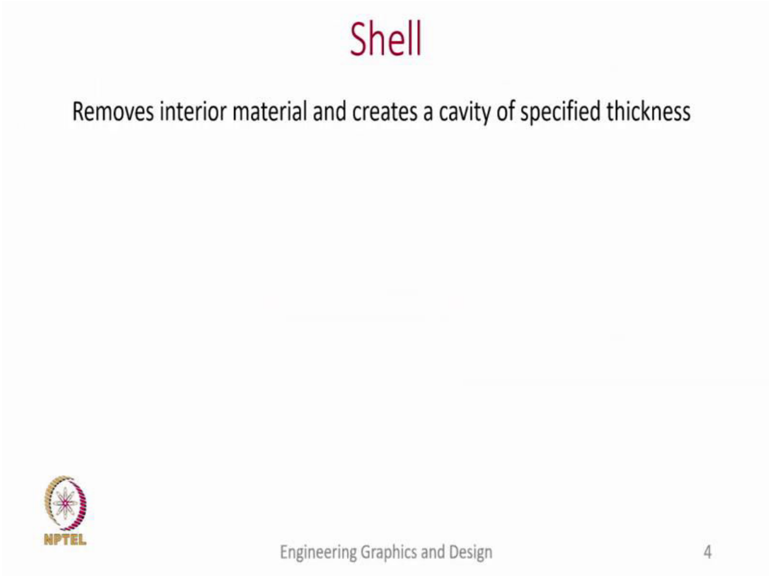

So now let us move into the next topic which is shell command. So, what does a shell mean? It is like a hollow cavity. So that also can be created using our part modeling module of the Inventor software. Using this command, you can remove the interior material from a solid which we have created and then create a solid which is of either uniform thicknesses or which is takes a specified thickness.

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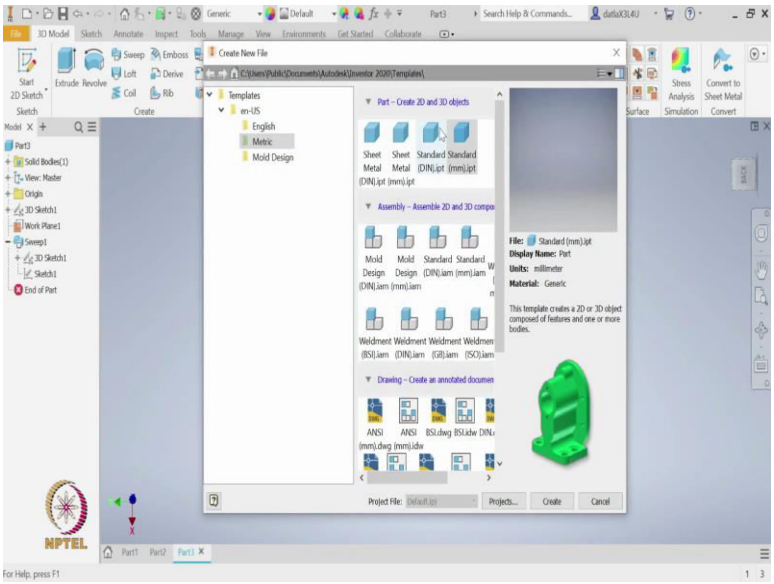
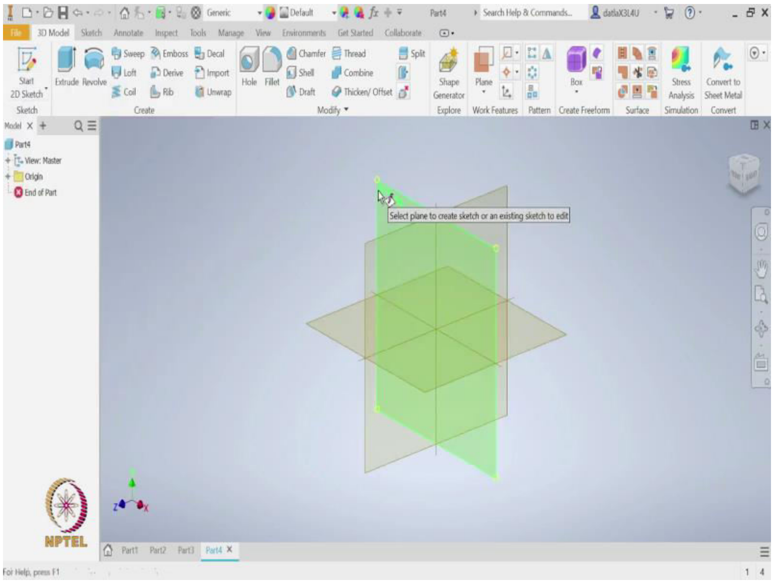
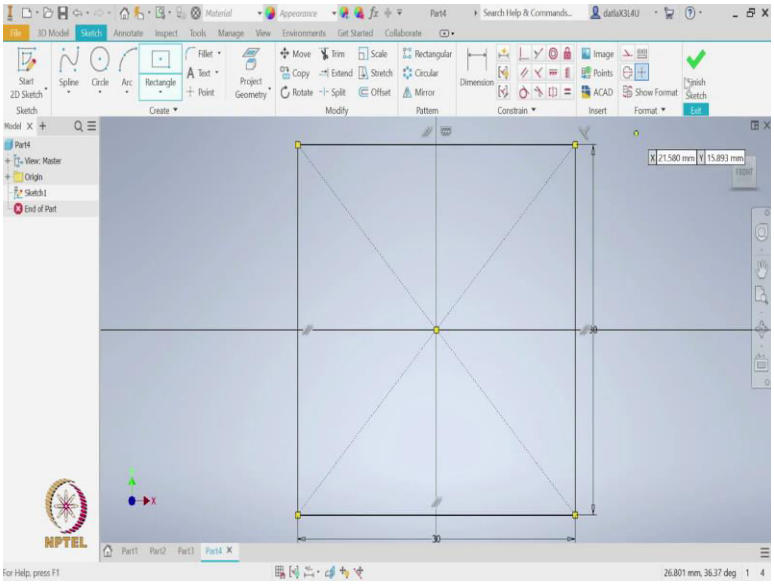
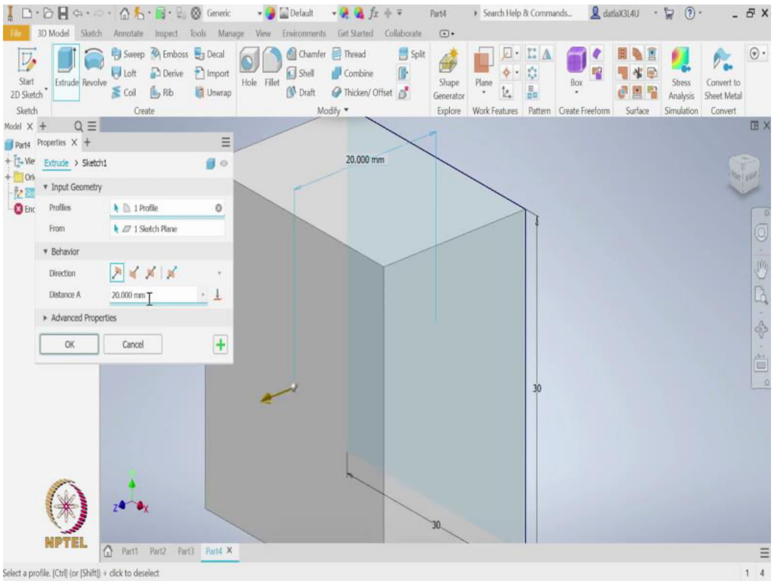



Select plane to create sketch or an existing sketch to edit
For Help, press F1


For Help, press F1


Properties
Extrude
Sketch1
Input Geometry
Profiles
From
Behavior
Direction
Distance A
Advanced Properties
OK
Cancel
20.000 mm
Select a profile. [Ctrl] (or [Shift]) + click to deselect

So now let us look into this software. Let us start with a new template. So let us create a simple solid like a cuboid. For example, we will start with 2D sketch, X Y plane, and then let us create a rectangle, dimensions of 30 by 30. Finish. Now I can use extrude and let us say here it is, it will extrude by 20 millimeters. So what we quickly have done is created a cuboid of 30 by 30 by 20. So now let us say we want to make it like a shell command.

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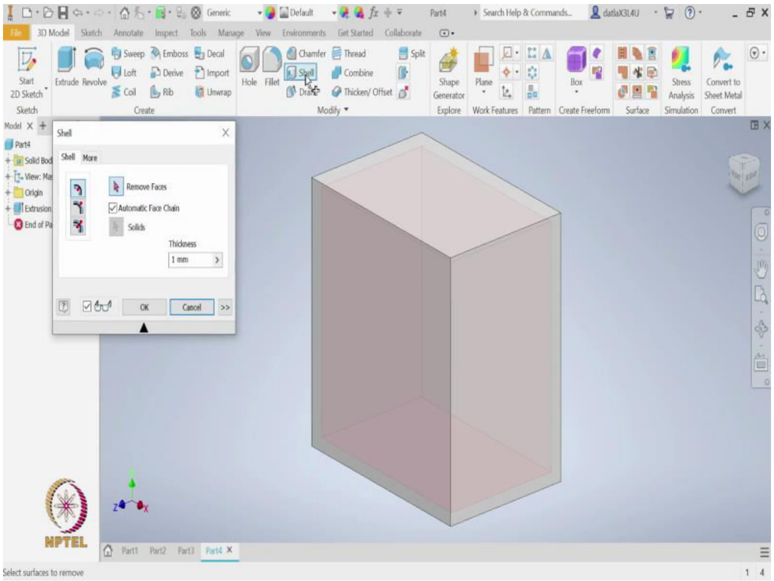
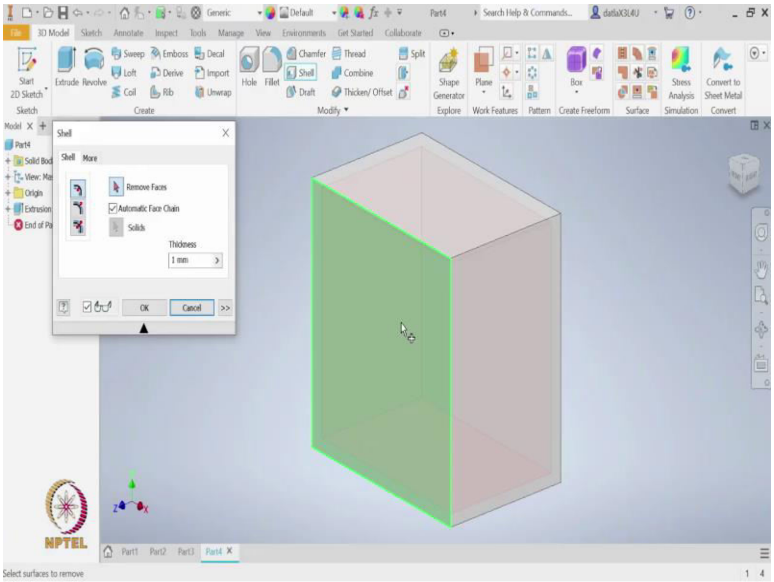
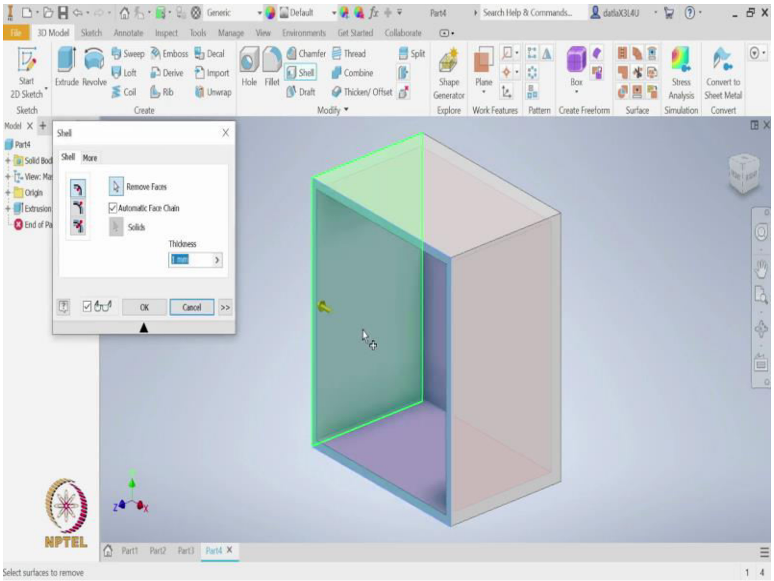
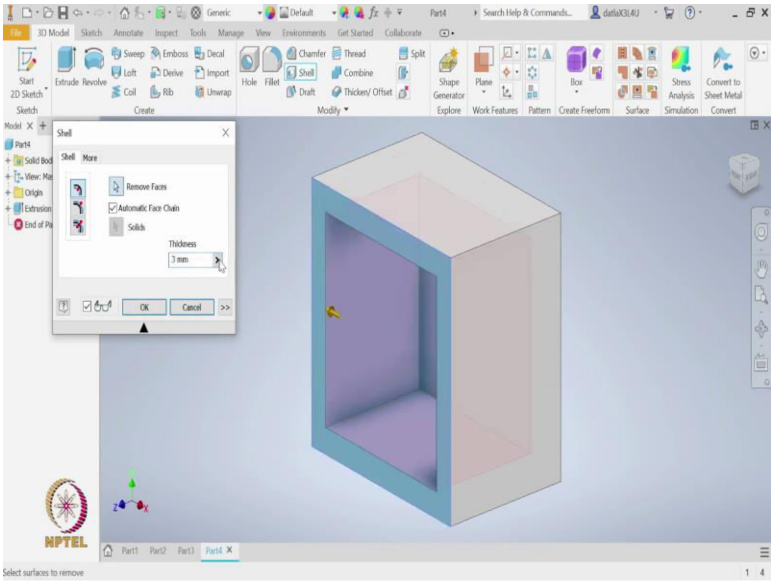
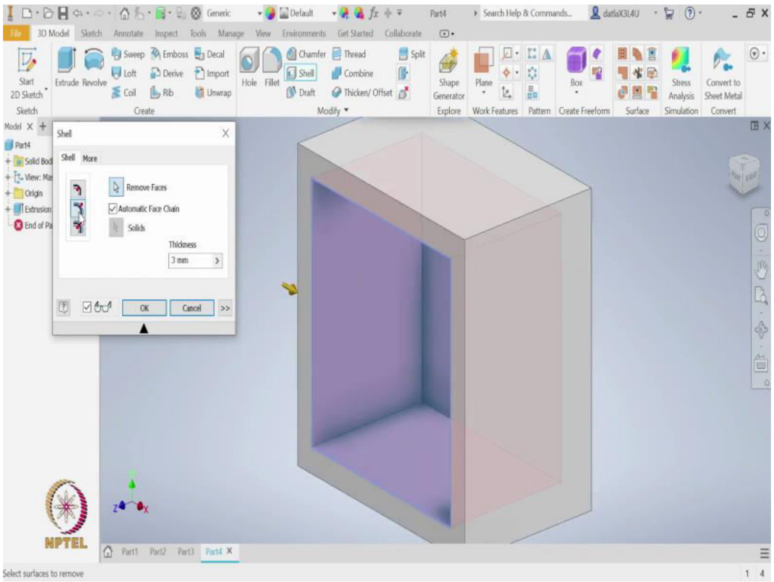
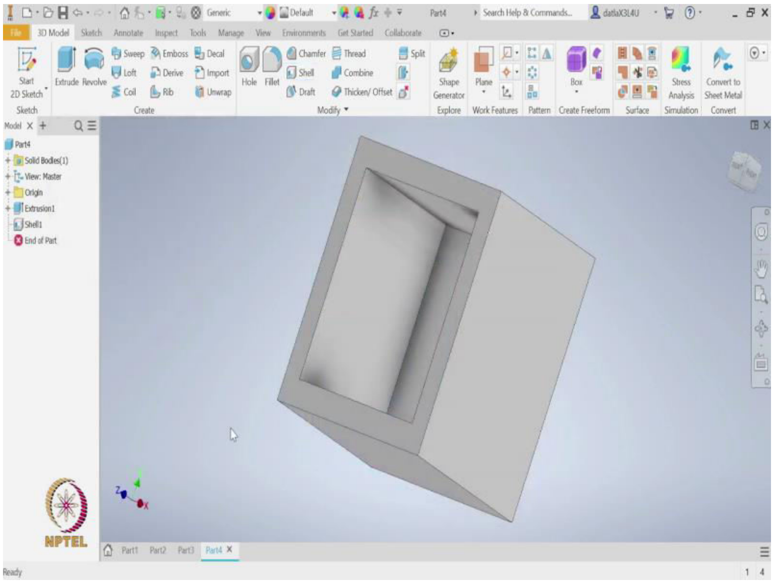





Shell
Remove Faces
Automatic Face Chain
Solids
Thickness
OK
Cancel
Select surfaces to remove


Shell
Remove Faces
Automatic Face Chain
Solids
Thickness
3 mm
OK
Cancel
Select surfaces to remove


Shell
Remove Faces
Automatic Face Chain
Solids
Thickness
3 mm
OK
Cancel
Select surfaces to remove



So we can find this shell command in the Modify panel. So we select the shell tool. So the first thing it is asking is if you want to remove any faces. So usually what happens is, if you do not remove any faces we started with 6 faces for this cuboid so you will retain with the outer cuboid and there is a cavity which is created inside.

So if that is what we want it is fine but let us say in some applications where you want to remove one of the faces in this example let us remove the front face, you have a box where with the lid removed. So we can specify the thickness of this box. By default, it is saying to 1 millimeter let us make it like 3 millimeters a thicker one.

And there are few more options in the shell. By default, it has chosen this Inside. What it means by Inside is, we have created a solid and the thickness will be within this created solid. But let us say the solid what you have created should be your interior of the shell, then we should choose the other option which is Outside.

So it will create material outside the existing cuboid. So now as you can see, since you are putting material on the exterior of the cuboid, the dimensions are much bigger versus the initial one where it is going inwards for the material. And the last option is of course, you can go in both directions by an equal amount and you need to specify the thickness of it.

So here is the box with a thickness of 3 that we have created using the shell command. So let us summarize what we have discussed in this lecture. Essentially, we have looked into few tools of the part modeling such as creating splines and using these splines to create solids using the sweep to where you need to give provide a profile and a path.

Lastly, we also looked into the shell command where we can remove the interior material and create cavities of specified thicknesses. So with this we will conclude this lecture and in the next lecture we will look at an example where we use some of these features to create a solid. Thank you for your attention.