Engineering Graphics and Design
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Week 8: Introduction to CAD
Example 2

Welcome back in the previous lecture, we have used couple of features like extrusion, fillet, chamfer, hole. In today's lecture, we will use those as well as few more new features such as the rib. We will show this using an example which is shown here. So, this is example 2.

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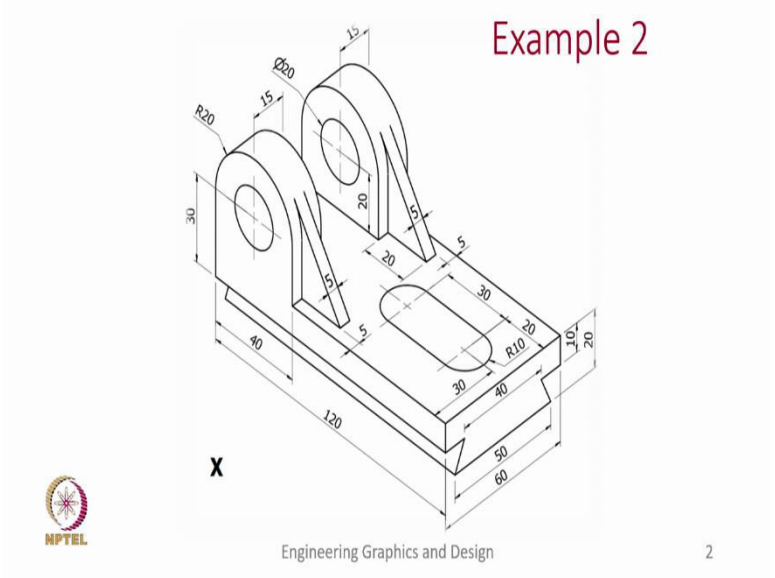

Where you have a base plate where this part or this feature needs to be cut out and there is a slot which needs to be cut off and there are two vertical members which are symmetric on either side of this base. And even in this vertical features we see there is a fillet, there is a hole and here is the rib that we are talking about. Ribs are thin features which are given to improve the stiffness of the feature.

So, for example here, this rib will give some rigidity or stiffness to this particular member. So, even if there is load, it can sustain the load with less deflections. So, let us get started.

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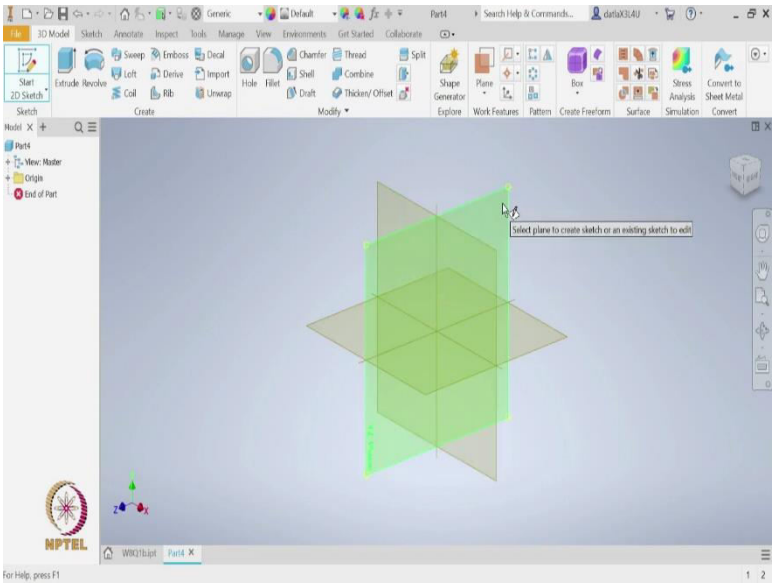

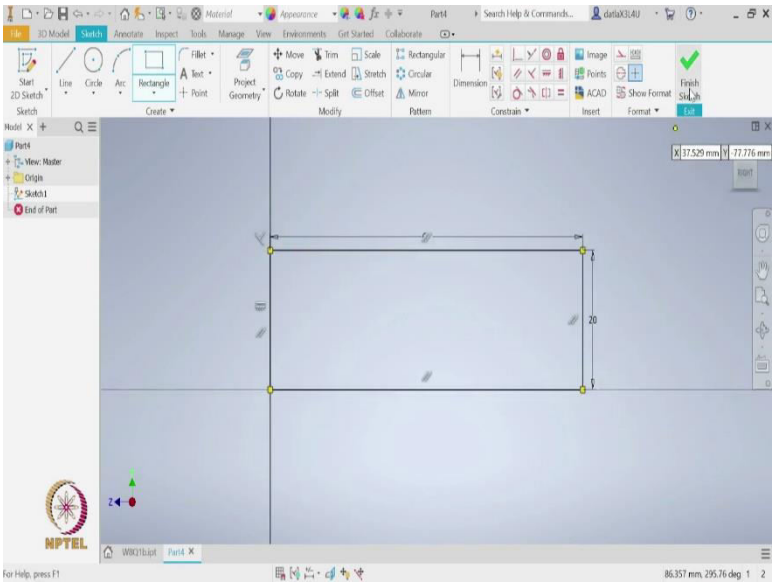

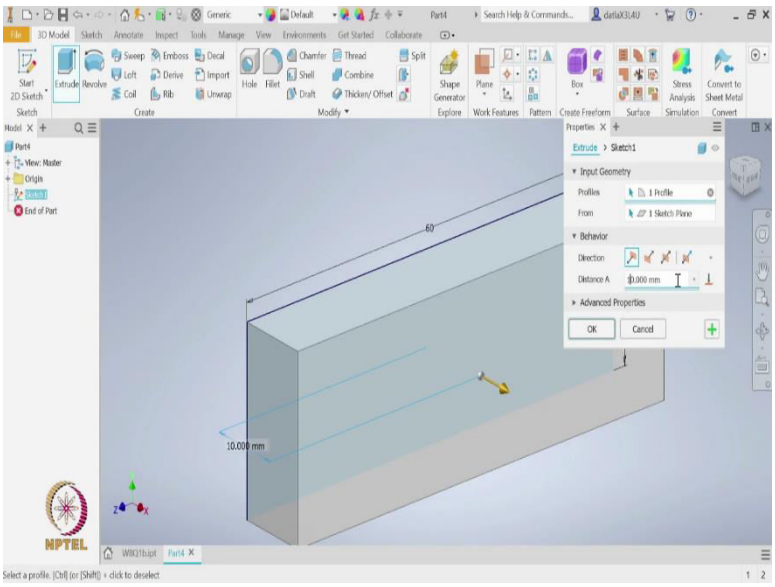

Let us look into the software. We as always, we will start with start 2D sketch. And since we are planning to start with the base, let us pick this plane and what are the dimensions of this section? We have this length is 60 and the total height is 20. So, we let us do 60 by 20 rectangle, again we will use the origin and rectangle.

So, 20 is the height and width is 60. So, let us do select all. So, with this we are, we can finish the sketch, select all and then do the extrude. So, what is the dimension of this extrusion 120. So, let us specify 120 here and click OK.

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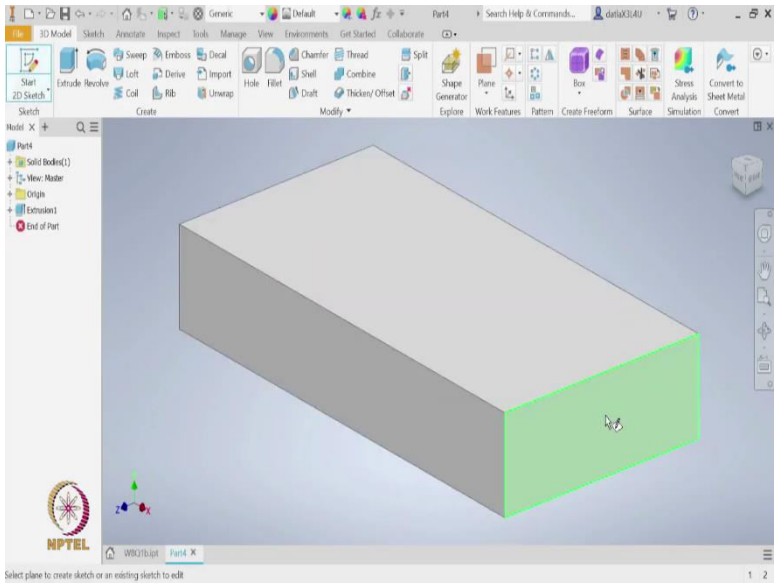
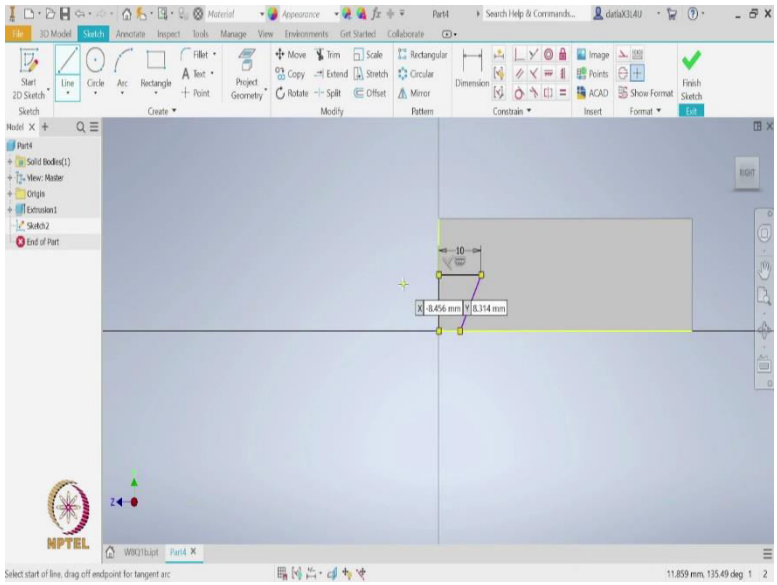
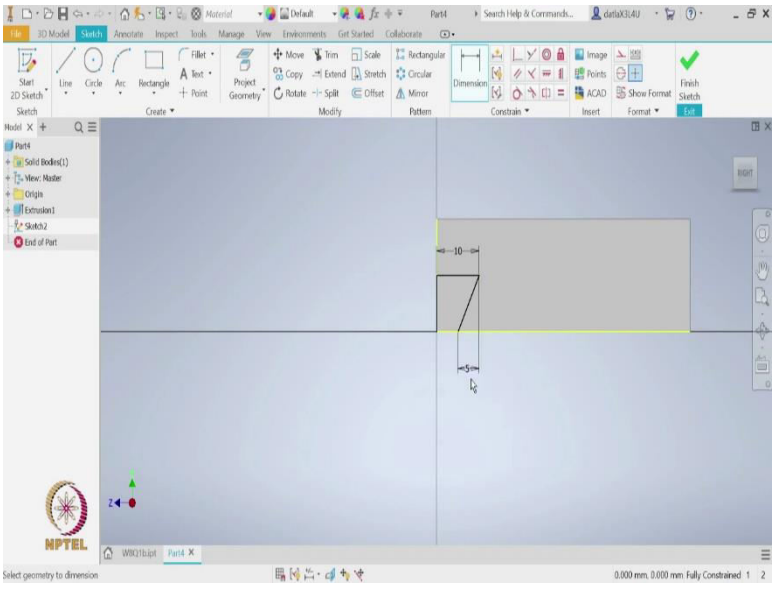
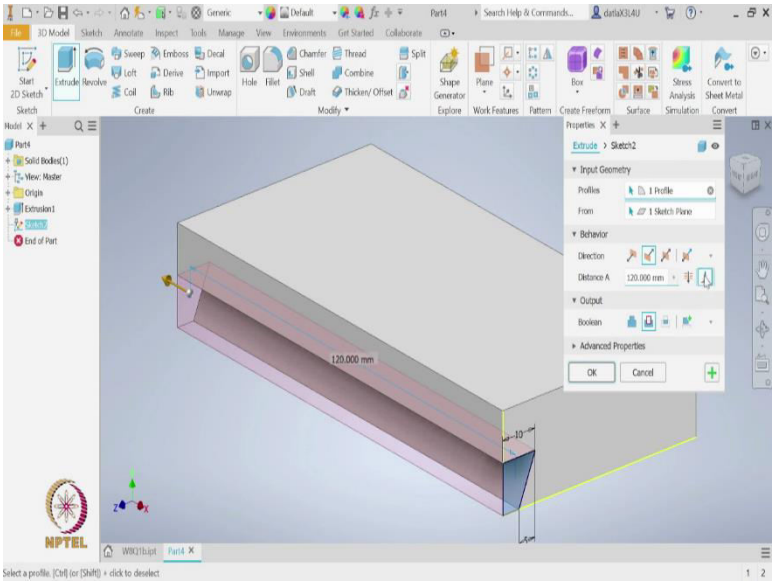





10
5
Select geometry to dimension
0.000 mm, 0.000 mm: Fully Constrained


120.000 mm
10
Properties
Extrude > Sketch2
Input Geometry
Profiles
1 Profile
From
1 Sketch Plane
Behavior
Direction
Distance A
120.000 mm
Output
Boolean
Advanced Properties
OK
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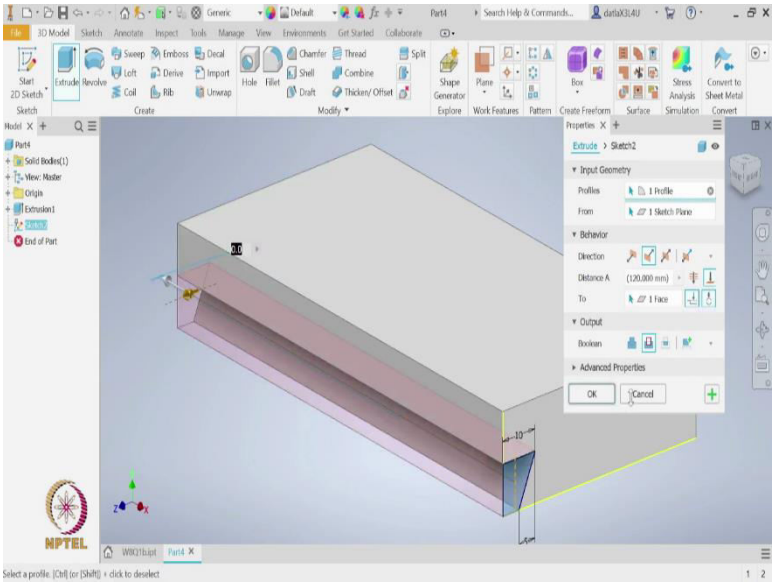
Select a profile. [Ctrl] (or [Shift]) + click to deselect


10
Properties
Extrude > Sketch2
Input Geometry
Profiles
1 Profile
From
1 Sketch Plane
Behavior
Direction
Distance A
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To
1 Face
Output
Boolean
Advanced Properties
OK
Cancel
Select a profile. [Ctrl] (or [Shift]) + click to deselect

So, the next thing we will do is how do we make this cut here to make a cut here we will select this front face and on that face we will make this section so, we will again start 2D sketch, we will select this front face and then let us start making a multi-line.

So, we need to start at the mid height. So, if you notice this cursor is at the centre is yellow but if I keep moving up at the midpoint it turns to green. So, let us select the and then let us make a horizontal line we know this particular distance is 10. So, let us make this horizontal line of 10 units and then we will make another inclined line which will touch the base and then the origin and we will close it up.

So, it says one more dimension is needed to complete the constraint we need to specify this inclined line what are the dimensions? So, if we look at the object, we see the, this distance is 10 and the bottom distance will be 60- 50 by 2 so, it will be 5. So, let us specify again with dimension this horizontal distance is 5. So, already given as 5 but otherwise we can specify.

So, after that we see that it is fully constrained. Now we can complete the sketch and then use the extrude command or the extrude tool to make the cut. So, we use the extrude and select the Boolean operation cut. Then again of course we can select the direction of the cut but the software has already given the right direction. And we can choose either through all our to feature.

Through all anyway works for this problem. But if you still want to use To you need to specify the rear face and let us click OK. So, once we are done with creating a cut to one side the other side instead of redoing the whole process, we can use a new feature called as mirror. But before we create these mirror image, we need to have a mirror plane, the plane over which we define the symmetry.

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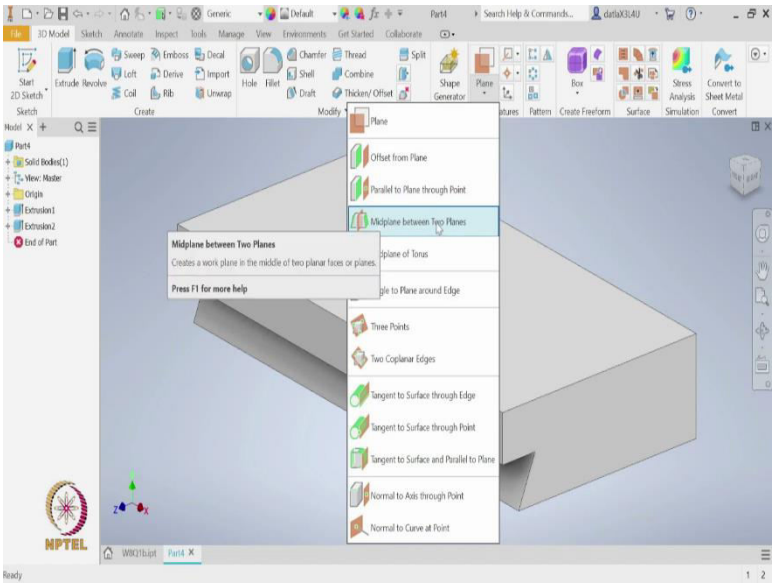

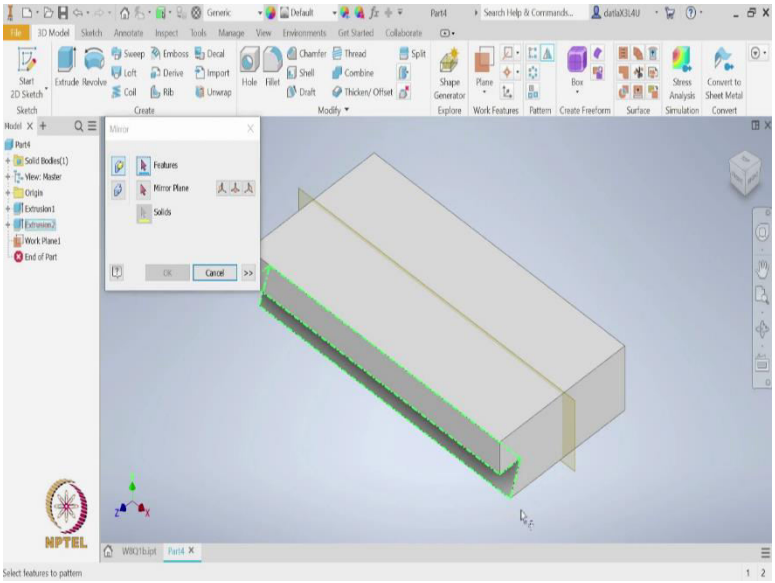

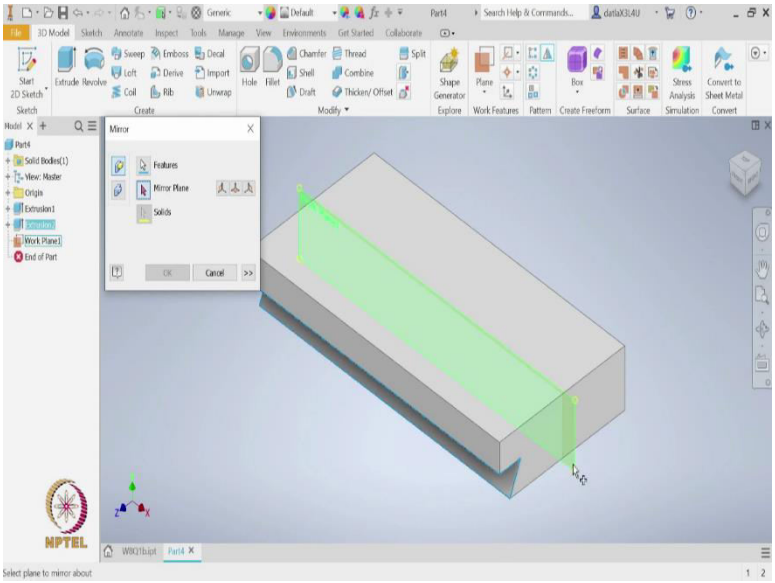

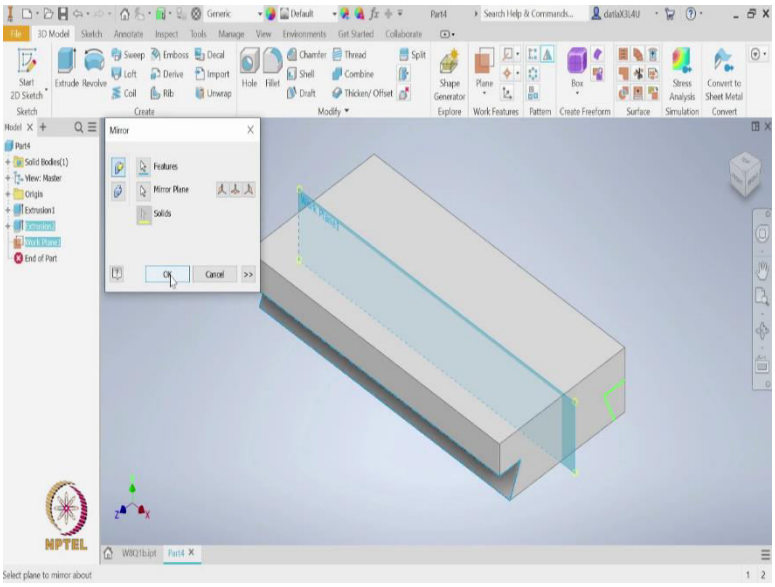
Mirror
Features
Mirror Plane
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OK
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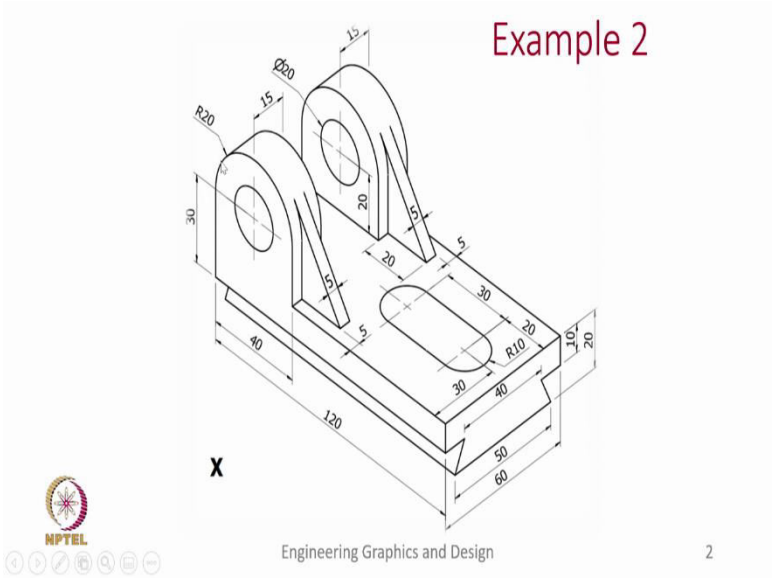
Select plane to mirror about


Example 2
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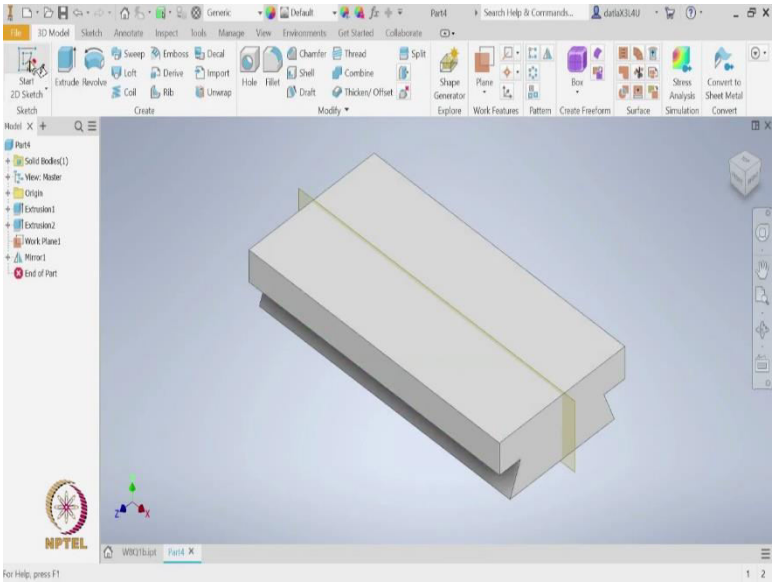
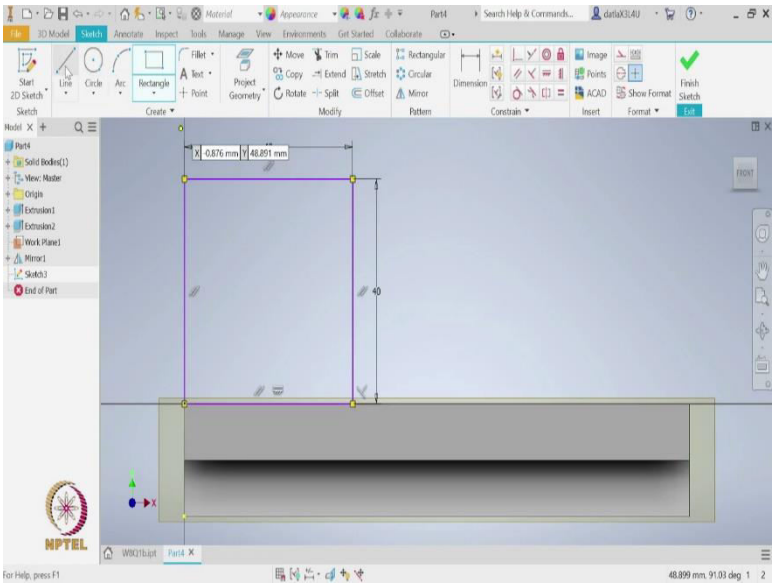
15
R20
30
20
5
40
120
R10
10
50
60
X
Engineering Graphics and Design
2


For Help, press F1


X -0.876 mm Y 48.891 mm
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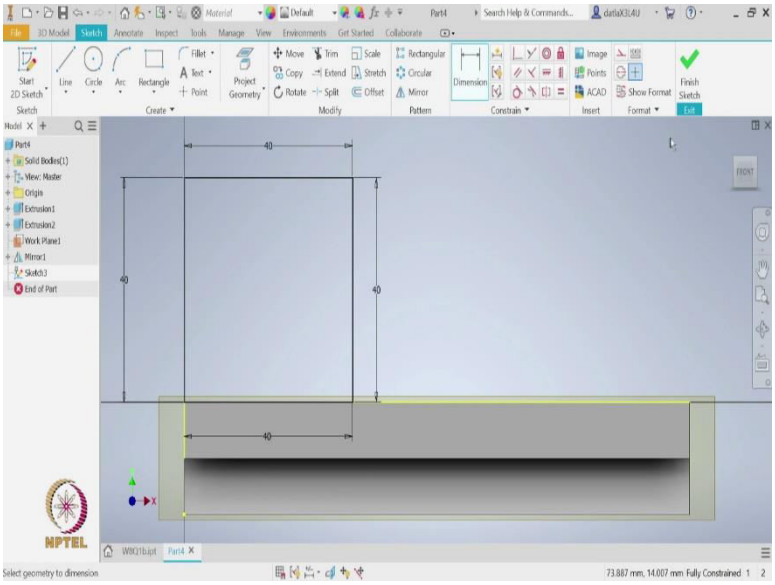
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FRONT
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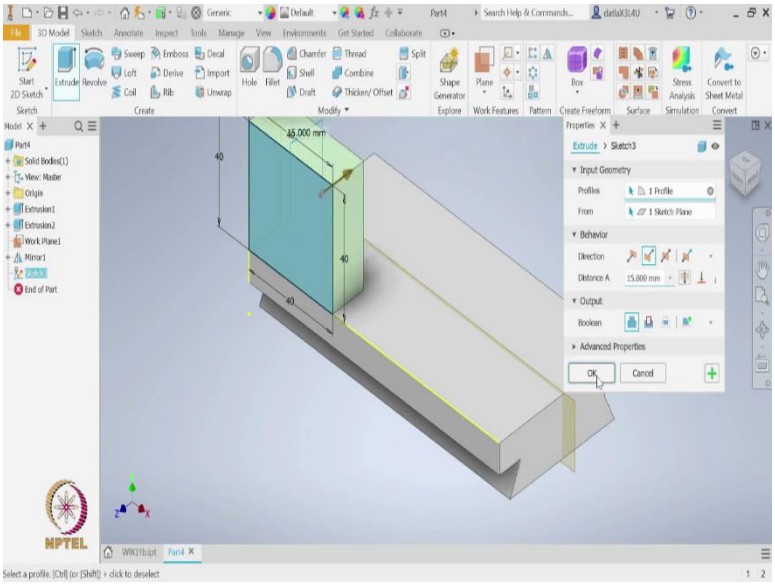
Select geometry to dimension
73.887 mm, 14.007 mm Fully Constrained


45.000 mm
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Extrude > Sketch3
Input Geometry
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1 Sketch Plane
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OK
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NPTEL
Select a profile. [Ctrl] (or [Shift]) + click to deselect

For that let us first create a mirror plane. Let us create a plane using this option of mid plane between two planes. So, we will take the left plane and rotate the object and select the right so, the moment we do this, a new plane is created. So, now let us over here is the mirror option or the mirror feature. So, once we click the mirror feature the dialog box opens up. The first thing to do is we need to select the feature we want to mirror.

So, if you go close to the cut it gets highlighted and we select it. Next you need to select which is your mirror plane. So, we already created a mirror plane and let us select it so, once you select it with green lights it is already showing you where it will make the mirror cut. So, we need to simply click ok because we are satisfied with what it is suggested.

So, after this we are done with the symmetrical cut let us see what is the next step to create the object. Now let us create this vertical member and what are the dimensions of this vertical member we know this radius is 20 so, the total length will be 40 and the height will again since this 10 is already specified maybe this is 20 + 20, 40. So, it will be a 40 by 40 square we need to create. So, let us start with 2D sketch and select this front face.

Once we have selected this sketch plane let us create a rectangle with this corner and specify the dimension says 40 tab 40 and hit enter. So, we see that it is still in blue colour which means there is something which is missing probably this point is not exactly coincident so, let us dimension it one more time to make it completely constraint. So, we use the dimension feature, the top face and this bottom face will say it is 40 and similarly the right face and this face let us say this dimension is also 40.

Now it is completely constrained. Let us finish the sketch and do the extrusion. So, when we extrude we need to do it in the opposite direction and the extrusion depth is 15 say OK. So, the first thing we can do is create the fillet and then the hole.

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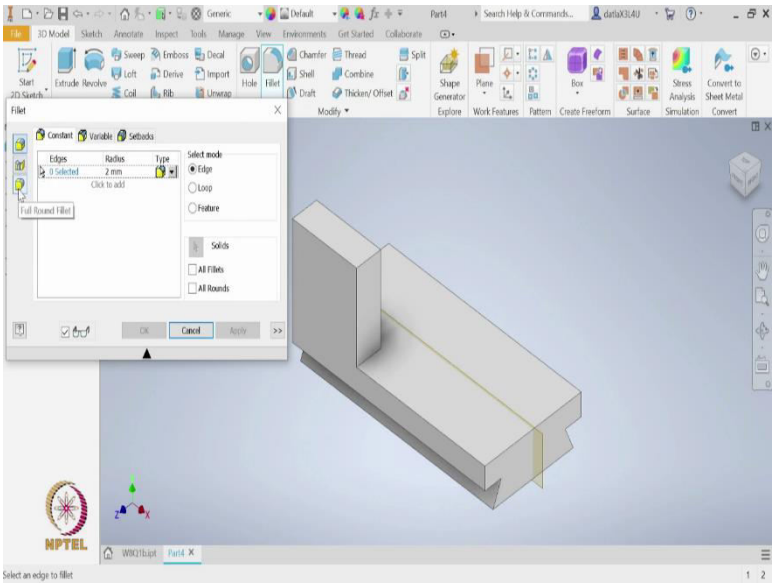

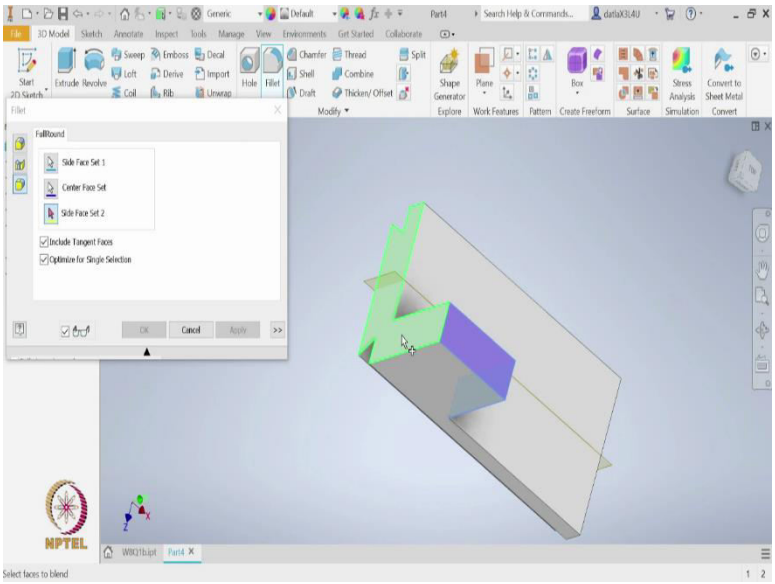

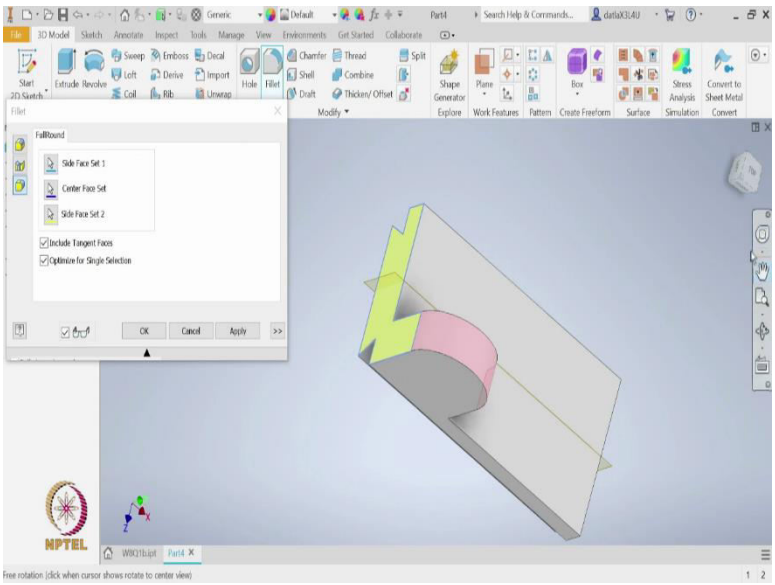

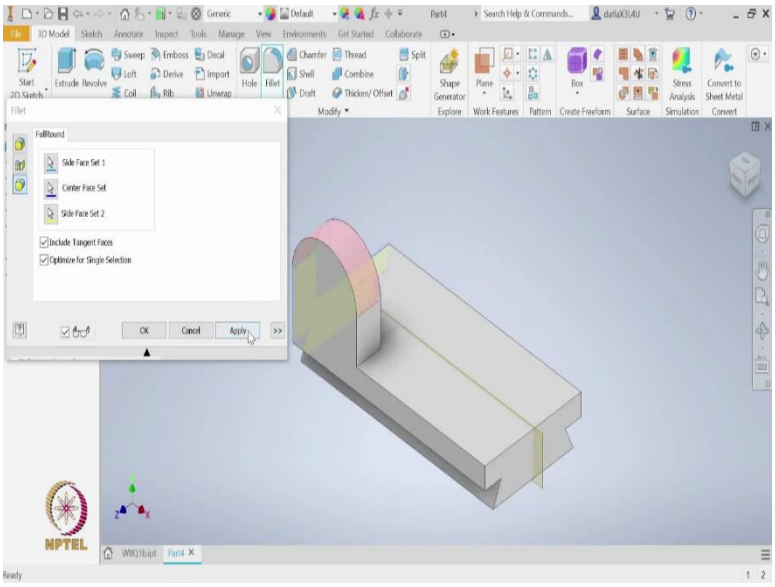

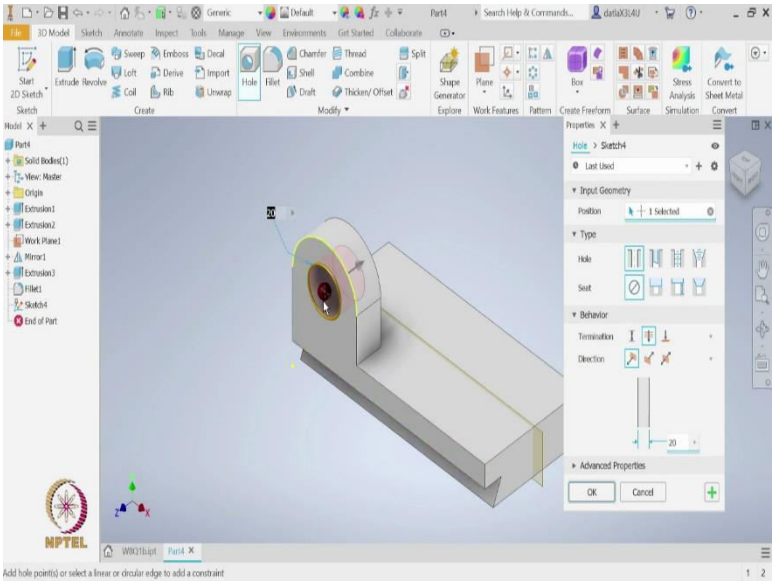

So, to create the fillet we will use the feature option feature tool fillet and once we select the fillet, previously I did not show that there are other options to create fillet but now let us see there is a third option which says a full round fillet. So, once I select this full round fillet it asks you to select side face at one central face and side face set.

So, this is first face, second face and to select the third face I need to rotate shift and the mouse wheel should help you to rotate. Now let us select the third face and it recognizes the radius based on the two parallel faces and creates a round for you. So, let us reorient it and since we are satisfied with it let us click OK and now let us create the hole the radius of the hole in both vertical features is the same the diameter is given as 20.

So, we will use the hole feature create the face on which we want to start with the hole and the diameter 20. Let us plug in here. And since we have smart features we can specify that

this hole is concentric to the fillet by selecting the edge of the fillet. Once I selected it, recognize the centre of the hole, and we can click OK.

So, what is the other feature which is remaining? The other feature which is remaining is now to create this rib. To create the rib let us create one more plane on which we can specify the rib. And what is the location of this plane? This is again at half width of this 15.

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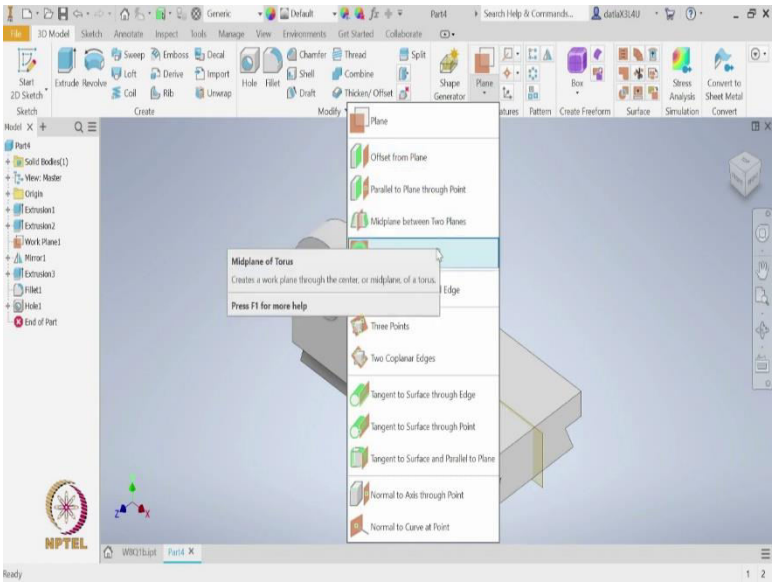

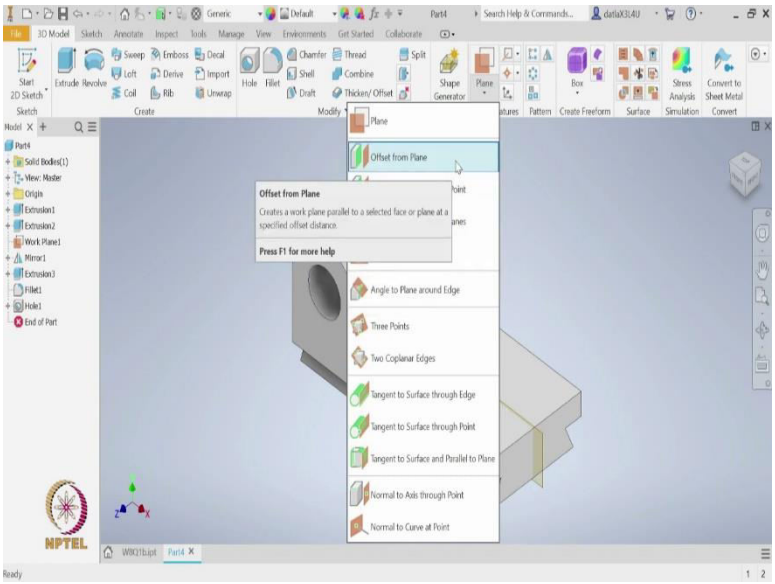

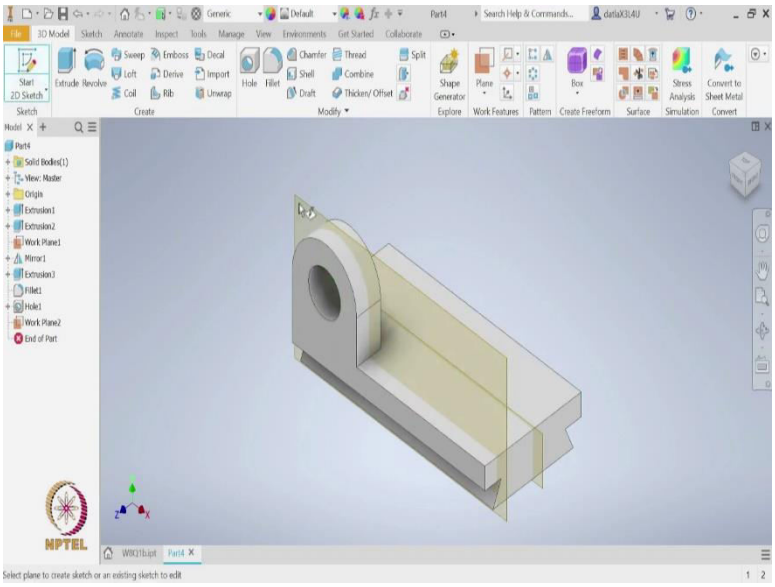

So, let us create another plane using this option of you can either use the offset from one plane, or simply say mid of two planes. So, I will select mid of this plane and rear plane. So, it has created me the centre plane. So, now let us create a 2D sketch on this centre plane. Let us bring it to the centre.

And now for us to create the rib, we need to specify this line and when we specify this line, if you look carefully, we see that it is tangent to this fillet area. So, first, let us draw a line which corresponds to this diagonal, and then we can specify whether it is tangential. And what are the dimensions of this fillet? One dimension we know is 20 is this length from the side of the vertical object to this end of the rib.

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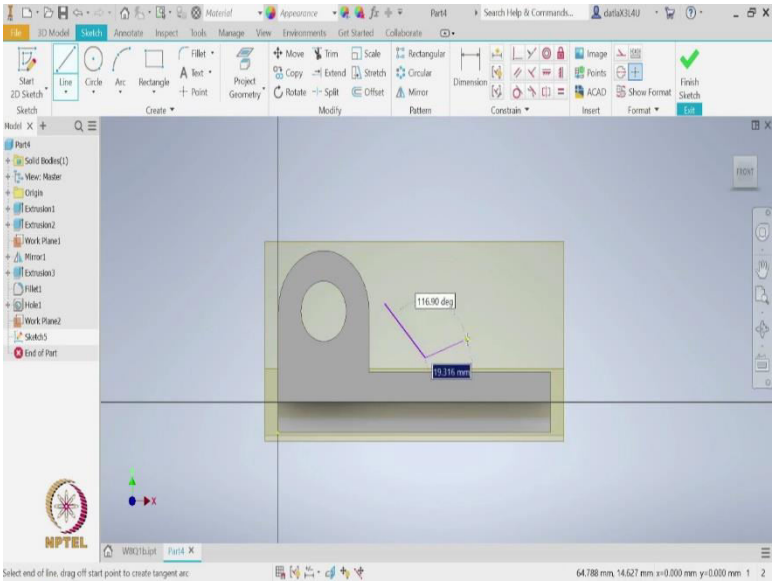
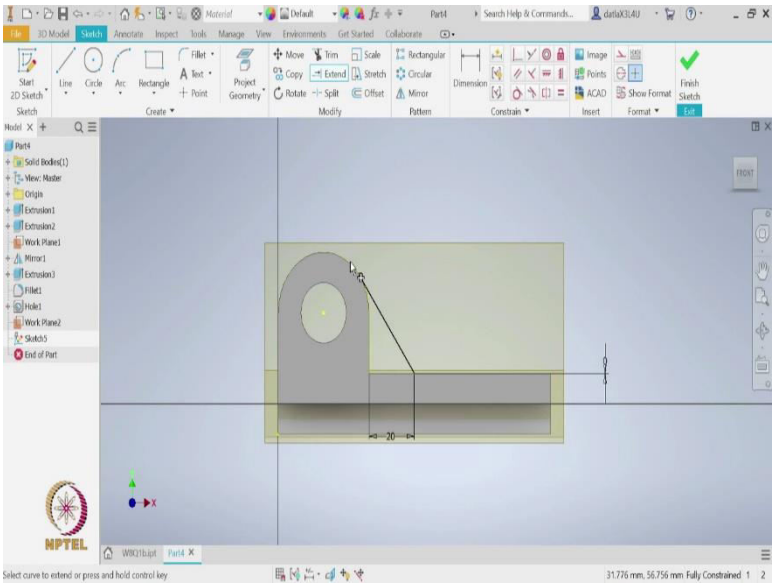
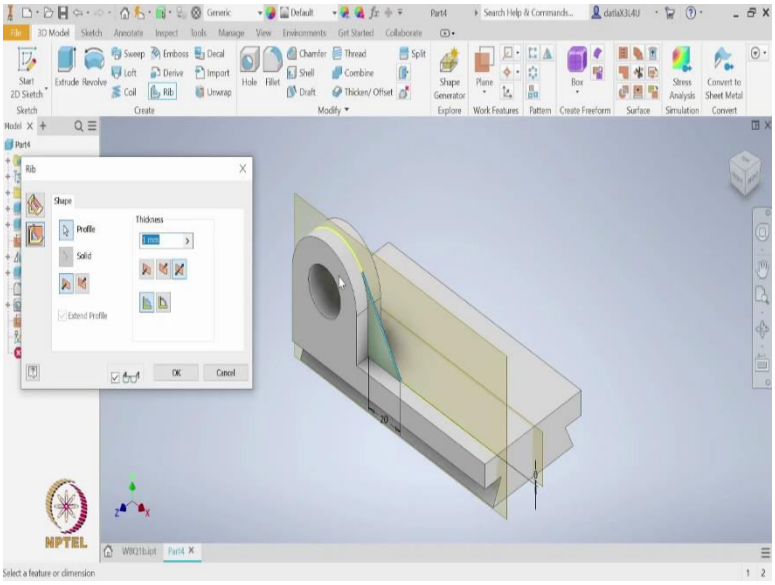
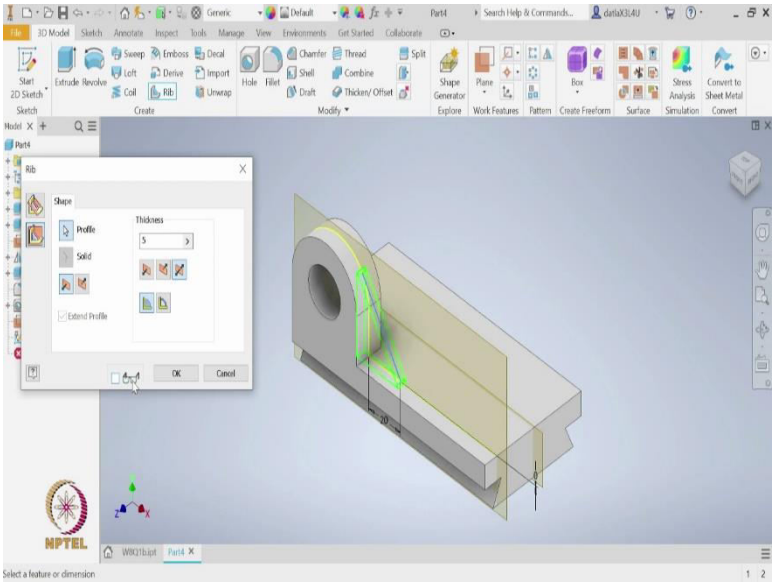



Select curve to extend or press and hold control key
31.776 mm, 56.756 mm Fully Constrained


Rib
Shape
Profile
Solid
Thickness
1 mm
Extend Profile
OK
Cancel
Select a feature or dimension


Rib
Shape
Profile
Solid
Thickness
5
Extend Profile
OK
Cancel
Select a feature or dimension

So, first, let us draw a straight line. We will draw some random straight line and then click OK. Now let us specify the dimensions with dimensions. So, once I click the dimension, I can go to this end point and say that with this edge it is at a distance of 0.

And again, the same point with this edge makes a distance of 20 that is given in the question. And now we need to ensure that this line is tangential to the fillet. So, how do we do that with a constraint what we have here, tangent constraint first we will select the line and then this edge so, suddenly, immediately it has changed the angle such that it is tangential to the fillet, but we see that we need to extend this line.

So, let us again select this extend feature here, select the line and it automatically extends the line until it touches the fillet. So, now let us do the finished sketch. And then to create the rib, we have this option here in the create panel we have this rib tool. So, let us select the tool and once we select the rib tool, the first thing is we need to select this line we have create and once we selected the line we need to specify whether we need the rib normal to the sketch plane or parallel to the sketch plane.

In our case our rib is on the sketch plane so, let us select the second option which is parallel to sketch plane. So, after that these directions I can toggle and see which one gives me the right answer. I see that the first direction gives me the correct representation of the rib, but I need to specify the thickness, specify the, thickness is given as 5 units. So, let us specify that and since we are this looks okay we can click OK.

There is one more option which is the enable or disable the feature preview. So, here I am showing you the feature preview you can toggle because sometimes it is confusing to know whether it is creating the right one or not. so, once you toggle you will have a clear idea and it will give you a preview of what you are intending. So, let us click OK since we are fine with the feature it is showing us.

So, once we are done with the fillet hole and the rib, now we noticed that these are symmetrical there is symmetrical about the middle plane. So, in this problem since we are already created the middle plane, we can again go back and use this mirror feature.

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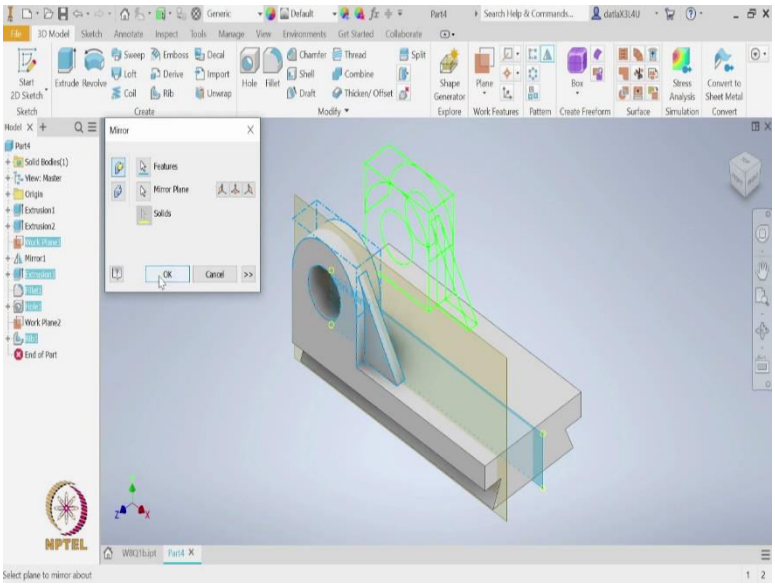
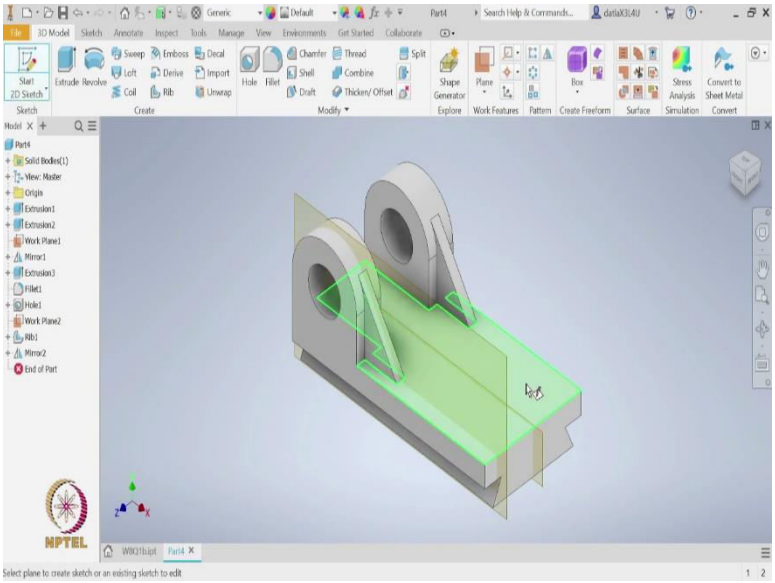
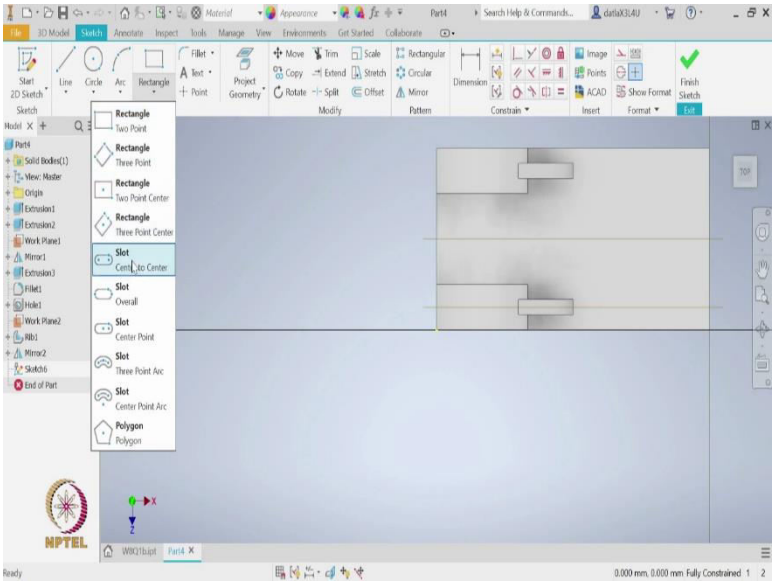
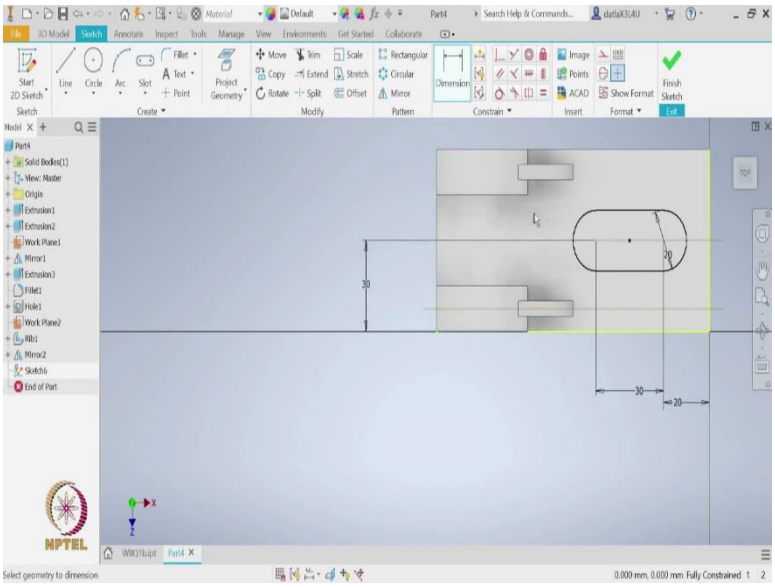
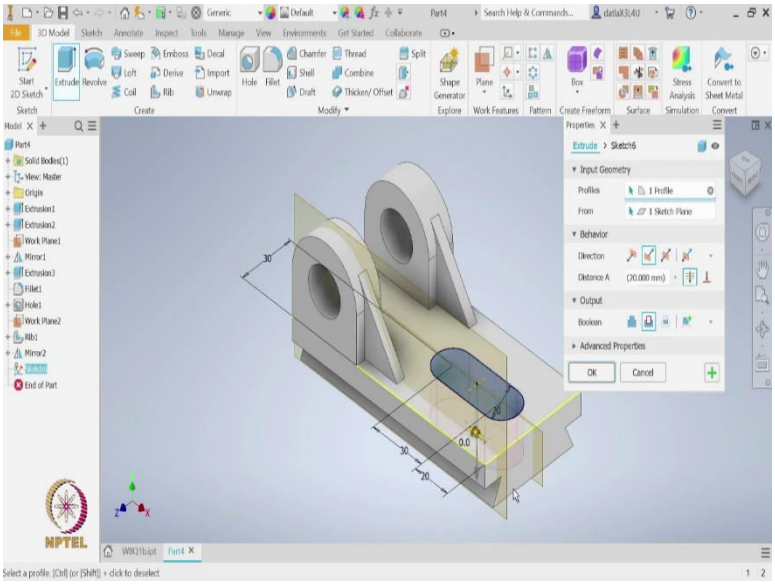
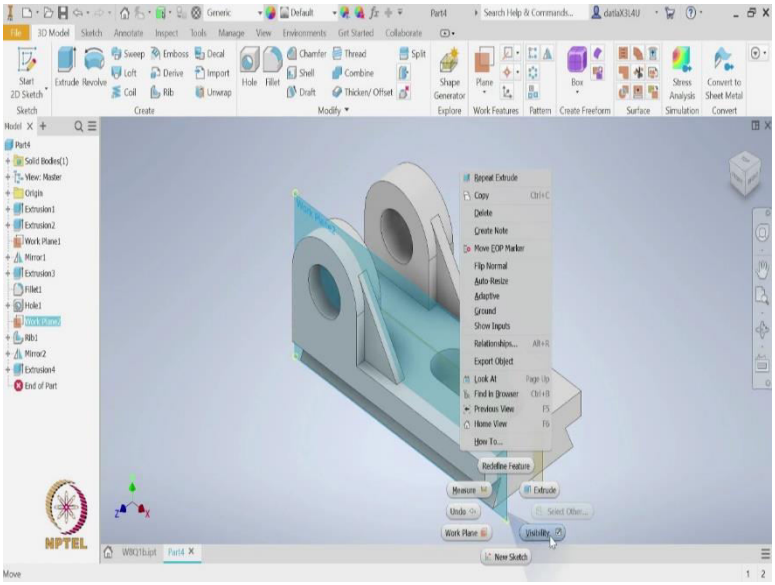


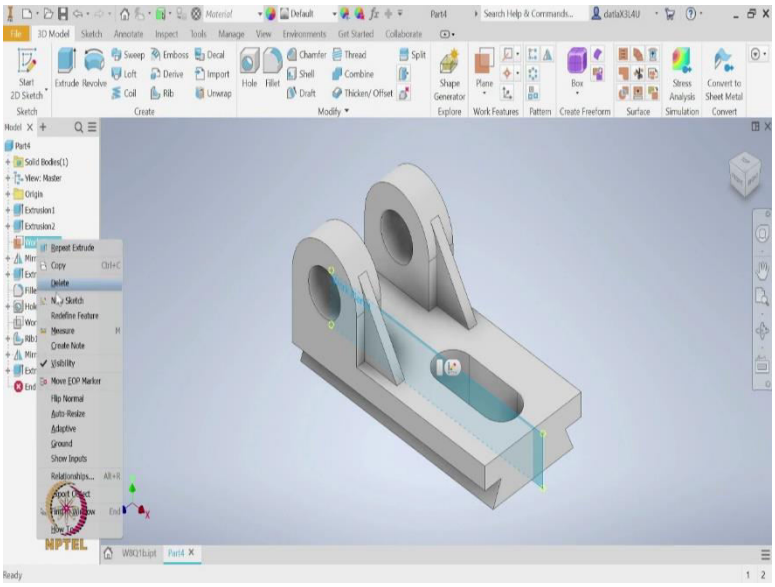


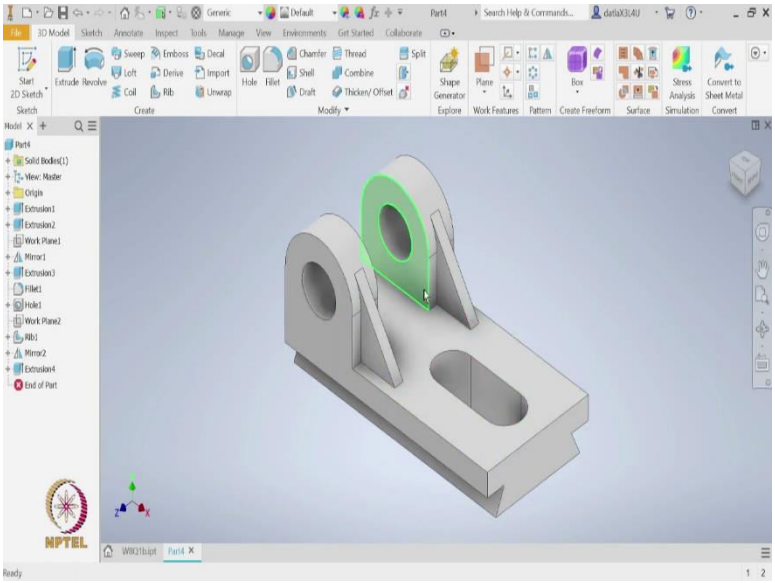



Select geometry to dimension
0.000 mm, 0.000 mm Fully Constrained


Select a profile. [Ctrl] (or [Shift]) + click to deselect


Move





So, in the mirror feature first we need to identify what are the features you want to mirror. So, the which will be? So, you can start here as the select extrusion, then the fillet, then the hole and then the rib. So, all of these are selected these are the 4 features we selected and next we go to selecting the mirror plane. Mirror plane was already created previously so, I select that and now it is showing you a preview of how it looks like since the preview looks ok to us, let us select OK and then finish this mirror.

So, now we are done with creating the vertical features too. The last remaining feature is the hole on the base that we can quickly do because once we go to the start 2D sketch, I need to select the top face of the base and then if you go to this drop down menu of the rectangle, there is one option called slot. So, you need to click the 2 centres and those dimensions.

These dimensions are given as radius of 10. So, here I need to mention it as 20 and here it says like it is not completely constrained. So, let us first work on constraining it. So, again going to dimension let us say the first hole is from this edge at a distance of 20 and the centre of the second hole is at a distance of 30 from the first centre and then let us also specify what is the distance from the bottom edge since it is symmetric it should be 30.

Now it is completely constrained I can complete the sketch and then use the extrude command with the Boolean operation of cut. To make this complete cut. Here we will specify that it is a through cut. So, that it goes all the way to the bottom and then click OK. So, one option is I mean we are done with the problem and this is the final object.

But if you think that these profile planes are coming in not letting you see the complete final object you can always enable the visibility. So, visibility is now on will make it off. Similarly, you can do the same thing by first selecting this plane, right click and then visibility we will uncheck it. So, now we see the final object that we have started to work.

So, to summarize, what we have learned in this lecture is of course, we have created this object and, on the way, we have learned some new comments like using mirror, creating a plane between two faces, as well as creating the slots are the new features we have studied in this lecture. So, thank you for your attention.

I suggest that you practice this with your software and then get comfortable with it. There is no better way to learn the software than to practice. Thank you for your attention and we will meet again in the next lecture.