Engineering Graphics and Design
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Week 8: Introduction to CAD
Example 1

Welcome back in this lecture we will be looking at an example that will make use of the sketching environment we discussed in the previous class. We will also introduce few features that will help us develop complex features.

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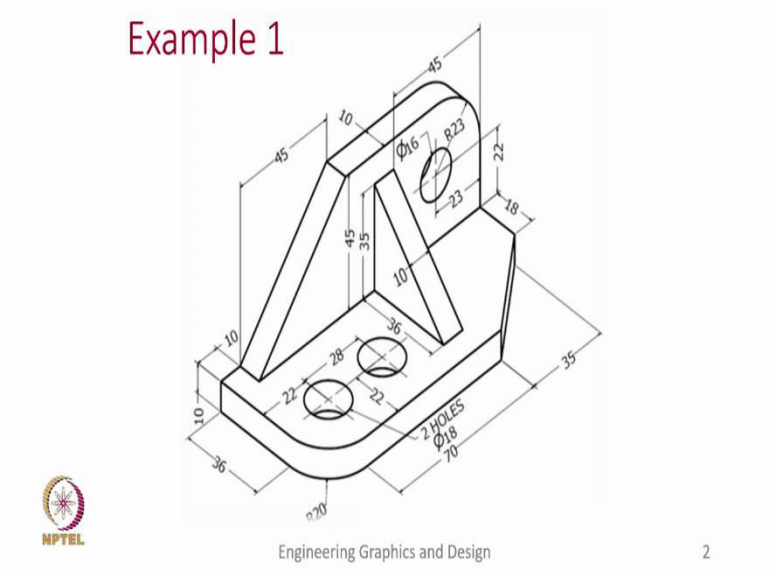

Here is a solid that we are attempting to prepare in this lecture. So, we will get started by first preparing the base of this object, and then slowly we will develop the complete object. So, to prepare the base of the object first let us see what the overall dimensions are. The overall dimensions will be 36 plus radius 20, which is 56. And the total length will be 70 plus 35, which is 105 plus the radius 20 will give us 125. So, let us move into the software and see how we get started with this object.

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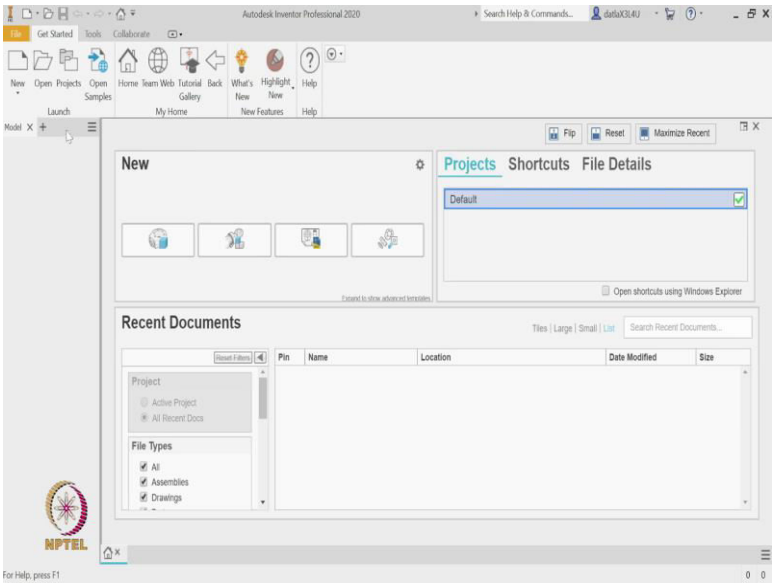

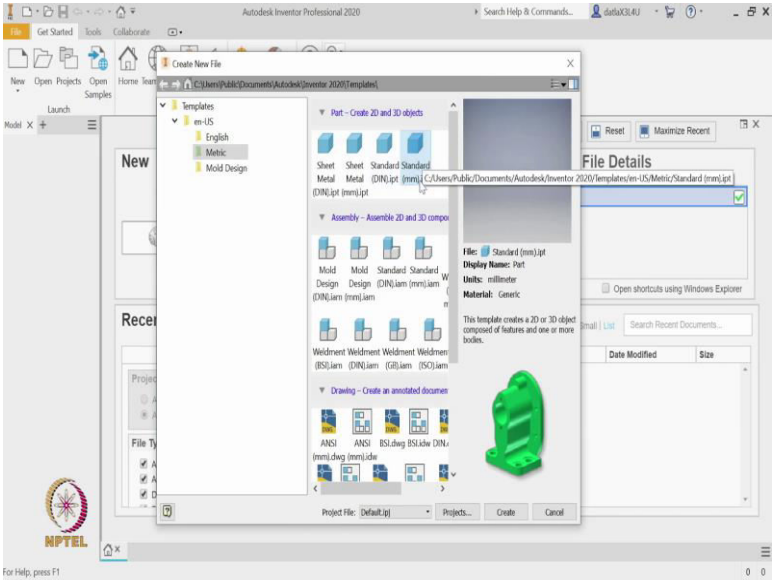

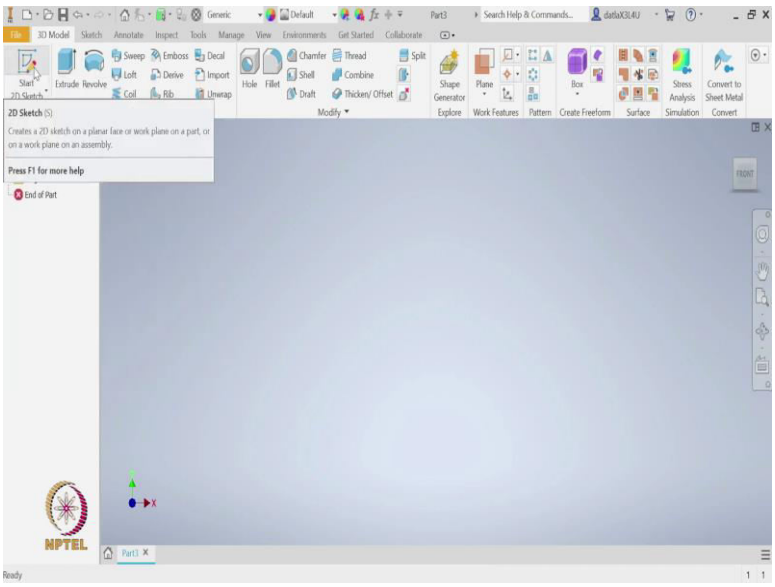

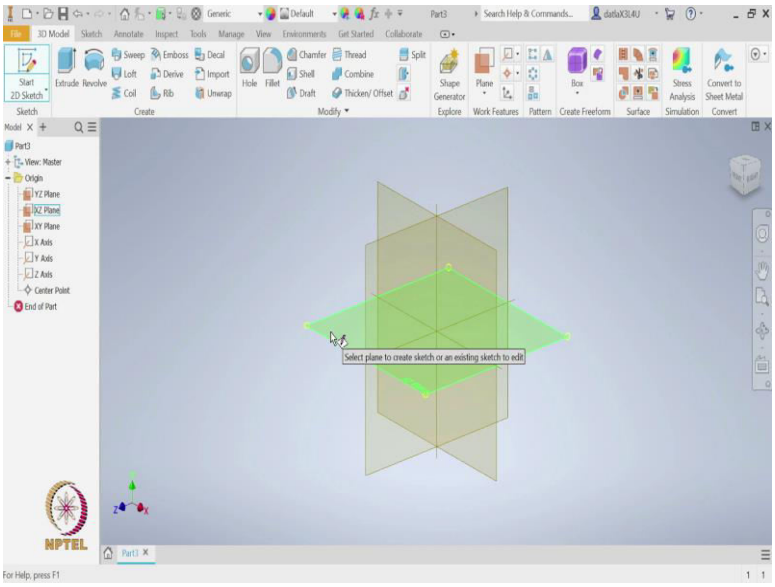

So, here is the opening screen for the software. We will click on the new. And select the part template, which is the standard mm IPT file. So, once we start the standard, so once we start the part template, we are in this part modelling environment. So, we should first select a plane. So, for that we will do this start 2D sketch.

Let us click on the it shows us two three standard planes which are the x y, y z, and the z x planes. Here are the 3 coordinates x, y and z. The centre of the coordinate system is also defined this information you can find once you expand the origin, you will see the three planes the three axes and a centre point. These are by default given by the software.

And we can either start with these or create a new planes. In this example, let us get started by selecting one of the planes. So, once I move the cursor to one of the planes it gets highlighted, we need to click on the plane. Here we are into the x z plane.

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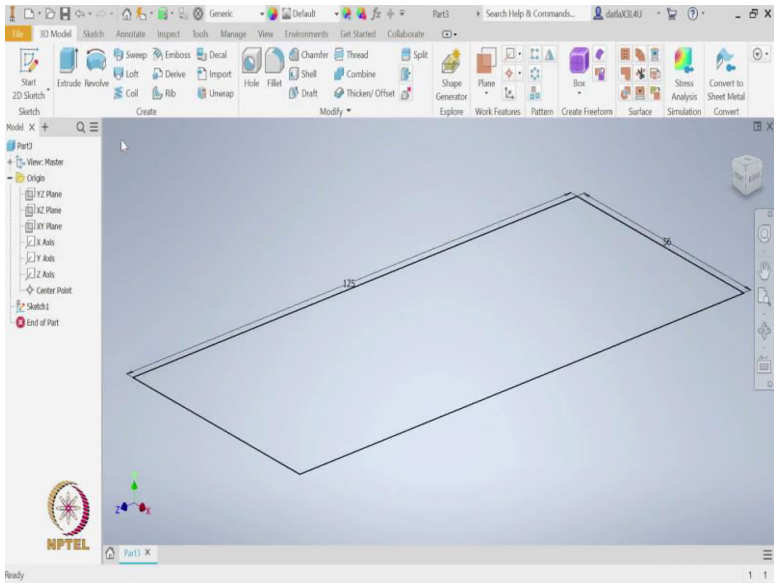
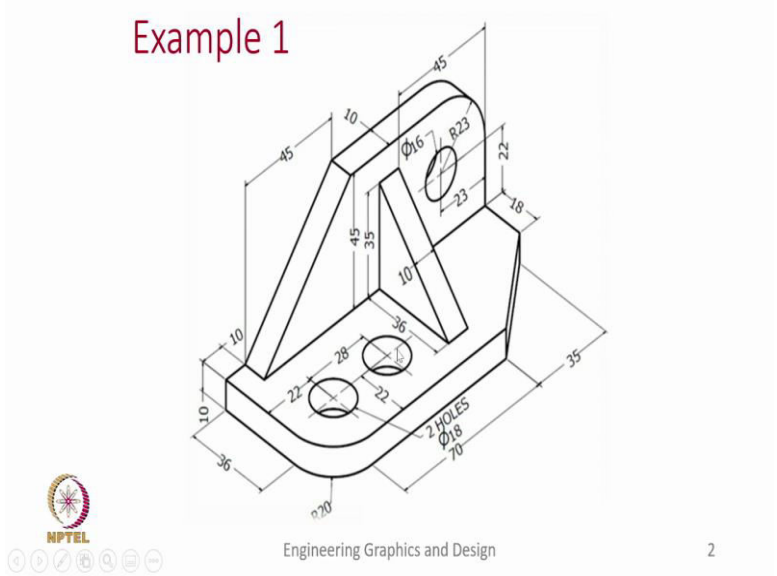
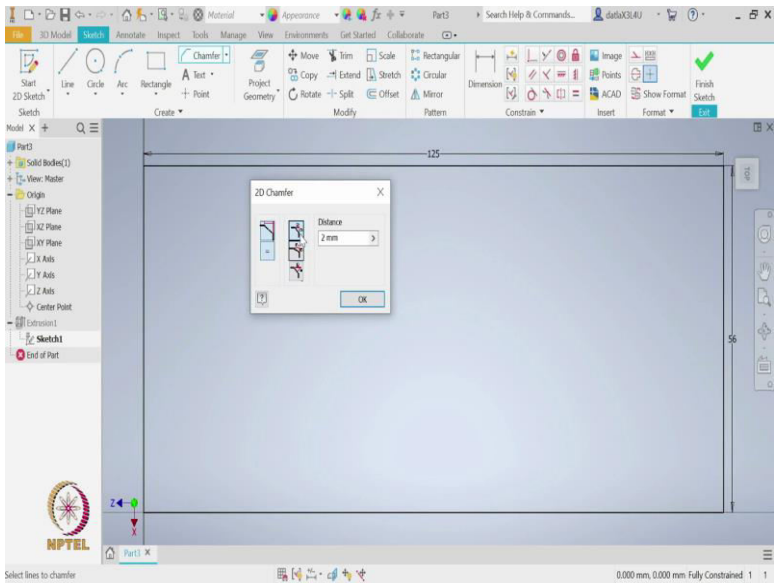
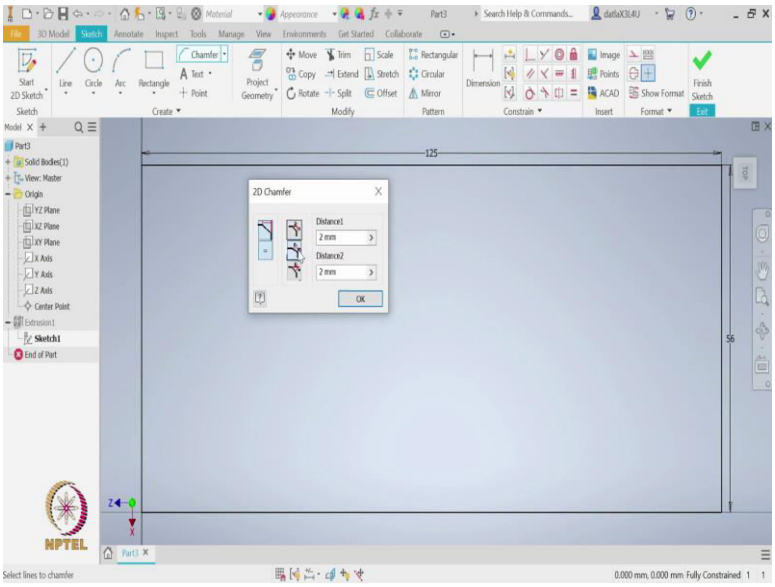
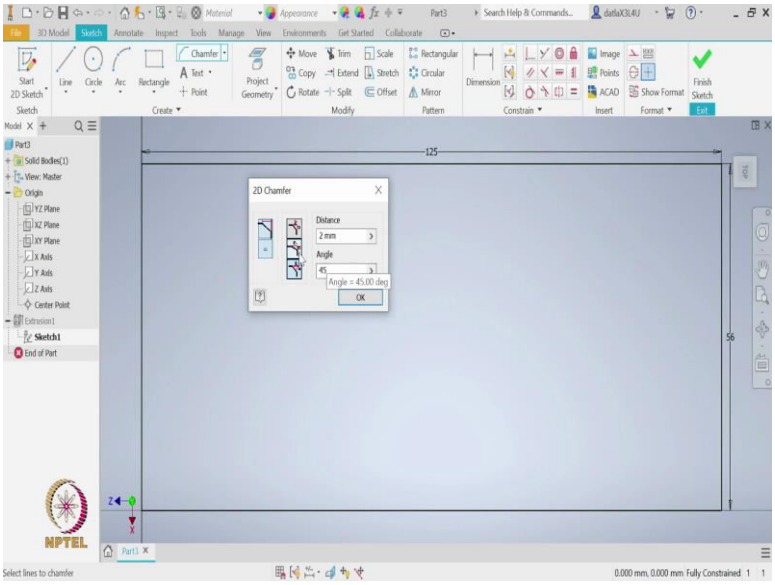
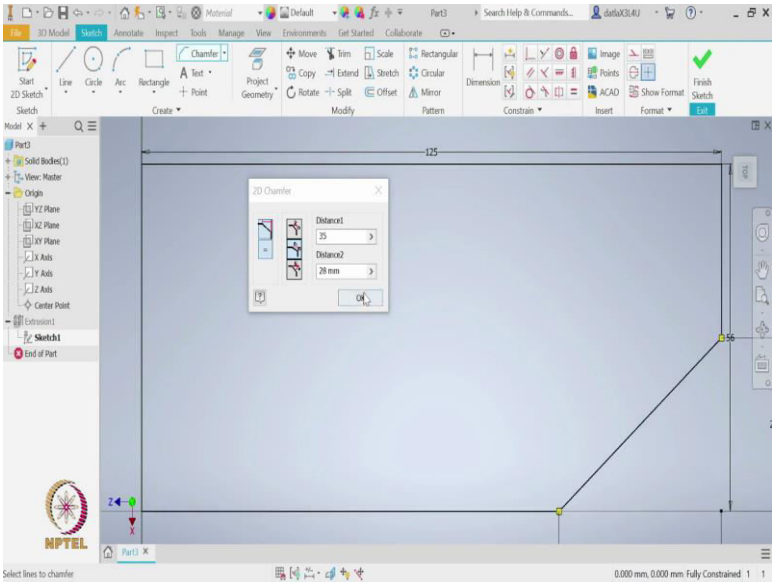


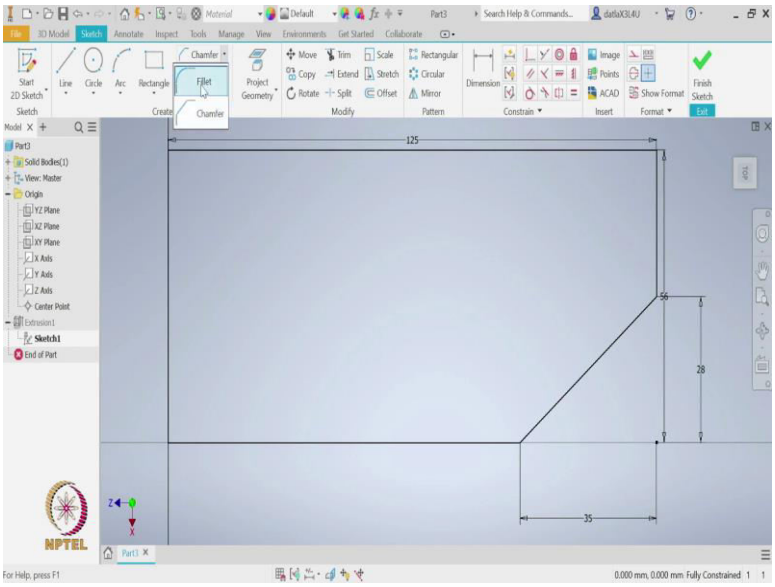


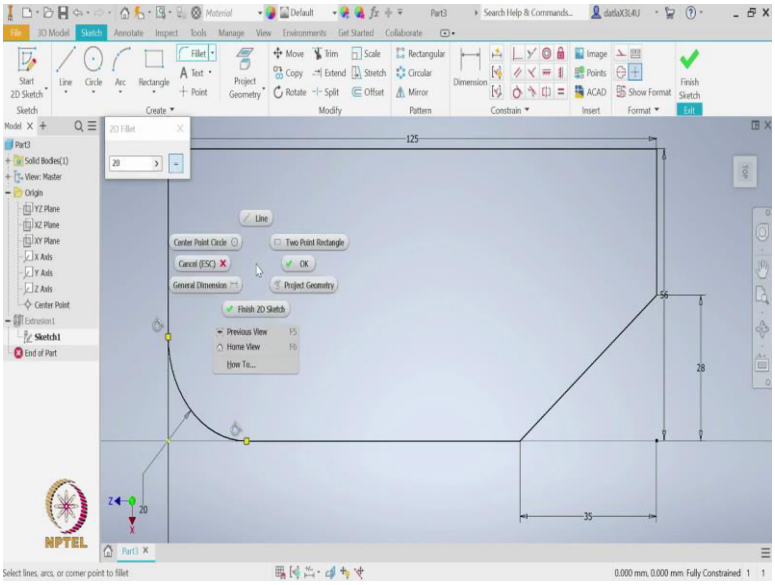


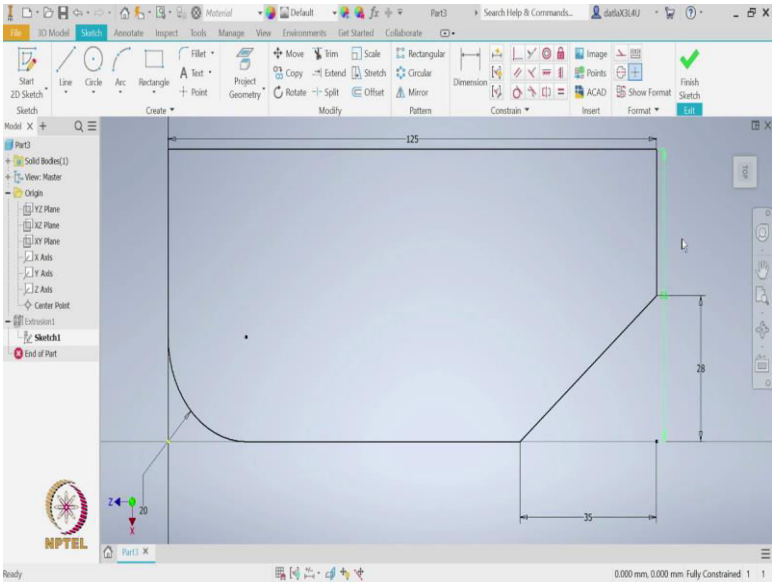

2D Chamfer
Distance1
2 mm
Distance2
2 mm
OK
125
56
Select lines to chamfer
0.000 mm, 0.000 mm Fully Constrained 1 1


2D Chamfer
Distance
2 mm
Angle
45
Angle = 45.00 deg
OK
125
56
Select lines to chamfer
0.000 mm, 0.000 mm Fully Constrained 1 1


2D Chamfer
Distance1
35
Distance2
28 mm
OK
125
56
Select lines to chamfer
0.000 mm, 0.000 mm Fully Constrained 1 1


Chamfer
Fillet
Chamfer
125
56
28
35
NPTEL
For Help, press F1
0.000 mm, 0.000 mm Fully Constrained 1 1


2D Fillet
20
Line
Center Point Circle
Two Point Rectangle
Cancel (ESC)
OK
General Dimension
Project Geometry
Finish 2D Sketch
Previous View
Home View
How To...
125
56
28
35
NPTEL
Select lines, arcs, or corner point to fillet
0.000 mm, 0.000 mm Fully Constrained 1 1


125
28
35
NPTEL
Ready
0.000 mm, 0.000 mm Fully Constrained 1 1

So, let us create a rectangle with the dimensions of 56 by 125. So, I will select the origin as one corner of the rectangle and the two dimensions are 56 tab 125. And so, select all will bring the complete sketch. So, let us finish this sketch. Select all and then extrude. So, let us see what is the height of this? It is 10.

But this creates an overall base. But if we look at the object again, we see there are several features like there is a fillet at this corner. There is a chamfer at this corner, and there are two holes on the base. So, let us start creating this. So, one way to create this is to either work at the sketch level or to work at the feature level.

We will show both of it but let us get started by editing the sketch. So, I will go back to the sketch which we have created previously. And right click on it and say edit sketch. So, it will take me to the sketch which we have created previously, which is the rectangle of 56 by 125. So, let us first work with this chamfer the dimensions of the chamfer are 35 and if you calculate it will be 28 because 28 + 18 + 10 will give us the total 56.

So, to activate the chamfer in the fillet there is another option of chamfer. So, when we say chamfer there are 3 options it gives, it says whether it is equal chamfer, which is the first option. Second option is if you have unequal chamfer like the way we have in our example and third is you can also define a chamfer by distance and angle saying that it is a 45 degree or 30-degree chamfers.

So, for us the option 2 which is distance 1 and 2 is more relevant. So, let us say distance one is 35 and the distance two is 28 and let us now select the corner we are looking at. So, we select the two edges of the corner this will prepare the chamfer so let us again select all to see these dimensions of 35 and 28. Secondly what we will do is we will now create this fillet.

The radius of the fillet is 20. So, again, let us select the fillet option here it asks for the radius of the fillet let us give it as 20 and then we need to specify which corner we want to fill, we want the corner is specified by these two sides of this rectangle. So, if we say OK, it will create the fillet. So, let us finish the sketch. And then since we are already extruded it before, and we have just updated the sketch.

Now, even the extrusion is updated. But it now has this fillet and chamfer. For us to now create these two holes, there are two ways I will show. So, first we will use by editing the sketch and second by directly a feature of hole. So, first let us get started with editing the sketch.

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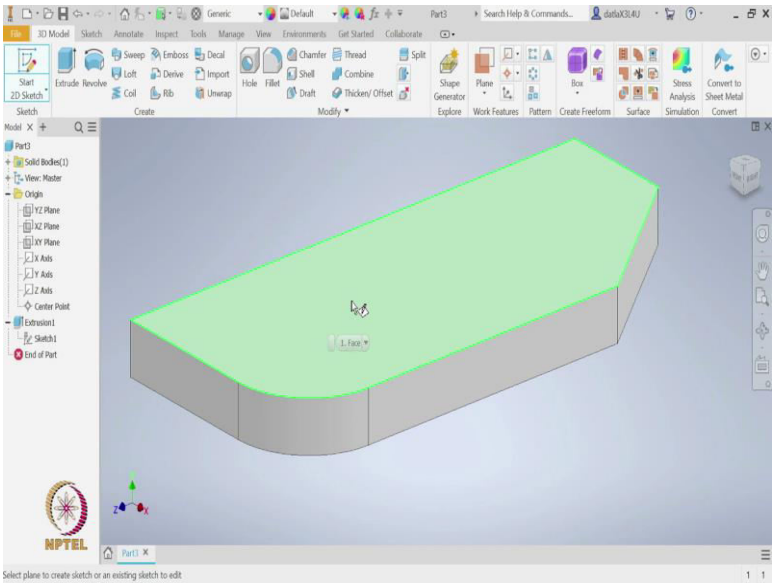

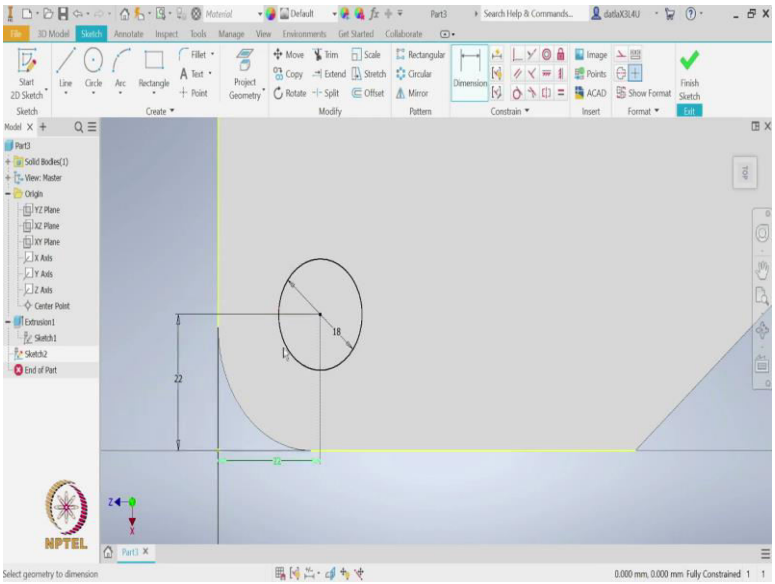

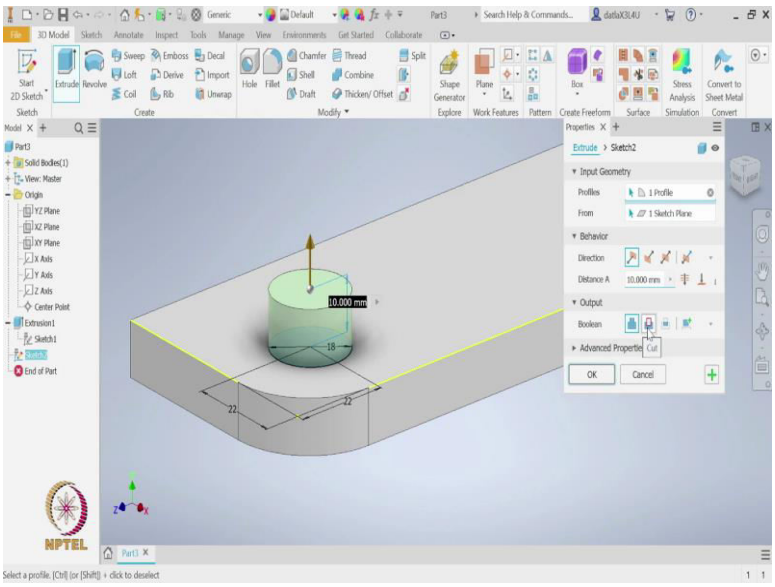

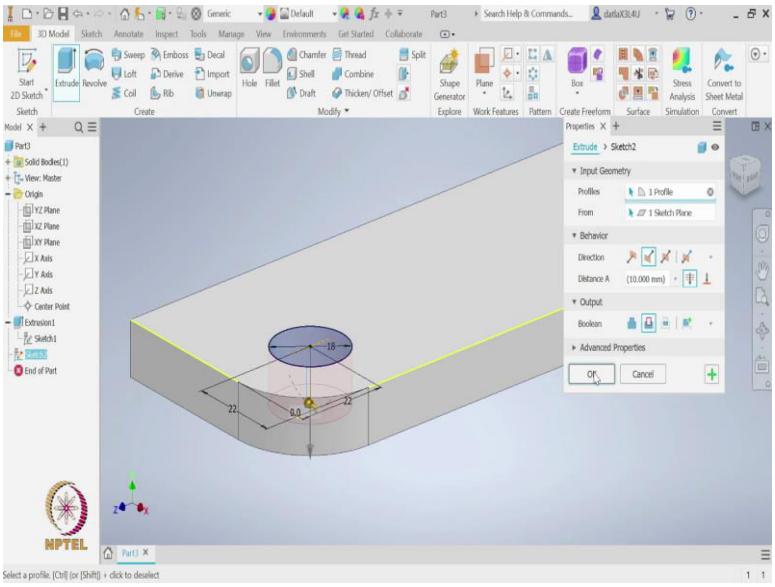

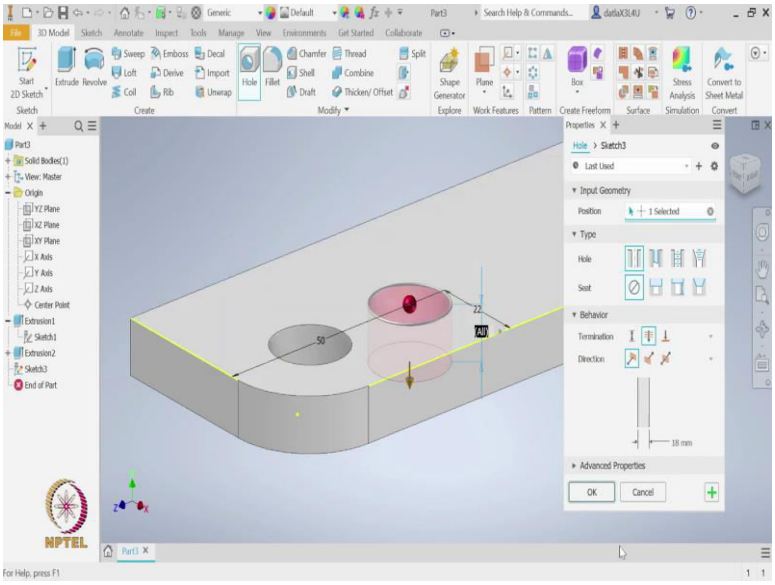

So, we will say we want to create a 2D sketch for that we need to select a plane. Let us pick the top face as the selected plane. So, I will highlight it and then click on it. So, once I click on it, I need to create a circle of diameter 18. I need to specify the exact location of the centre for the circle so let us do it with the dimensions. So, centre of the circle to this edge is 22 and centre of circle to this edge is again 22. So, let us say finish sketch and we can again use the extrude command once we use the extrude command.

In these Boolean operations we need to select the second option which is cut so once I select the cut, the software automatically realizes the material is in the built edge. But you can always control the direction of the cut by flipping up or down in this case let us leave it at down and then we can say it is through all cut in that cash fashion you need not specify what is the depth because the cut will go until it reaches the opposite face. So, let us click OK.

The second way we will create a hole is by using this feature called hole which is in the modify panel. So, let us pick this hole tool and first we need to specify the position or the plane on which we want to create the hole. So, let us again pick the top face and let us specify the diameter of the hole which is 18.

And then we need to specify the location of the hole for that this has a smart feature where you simply need to select one of the edges and then it will ask you the distance between the hole and that feature. In our case it is 22 and let us again select this edge so that we can specify what is the distance between the centre of hole and this edge.

If we go back to the figure we see it is 22 plus 28 which is 50. So, let us specify the and finally, you can also use specify the termination either the depth or the through all in our case it is through all. So, finally, we are able to compete the bottom phase. Now let us go back to the figure and see what are the other things we need to do.

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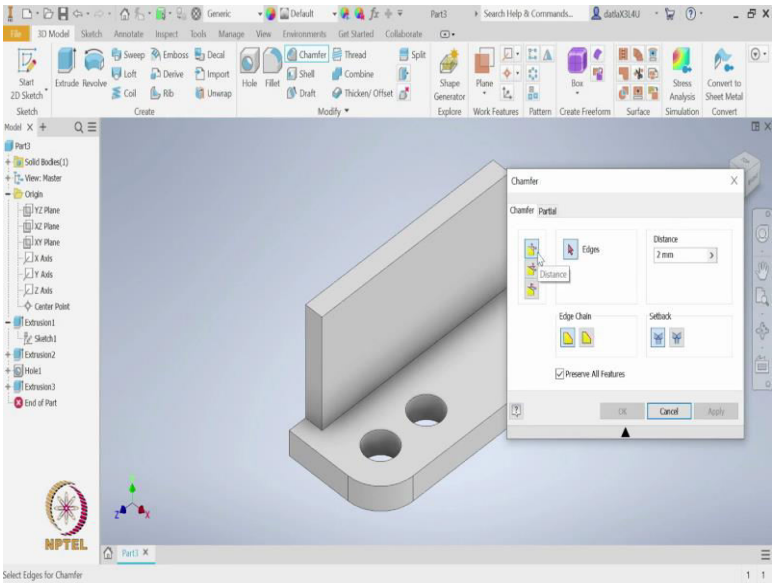
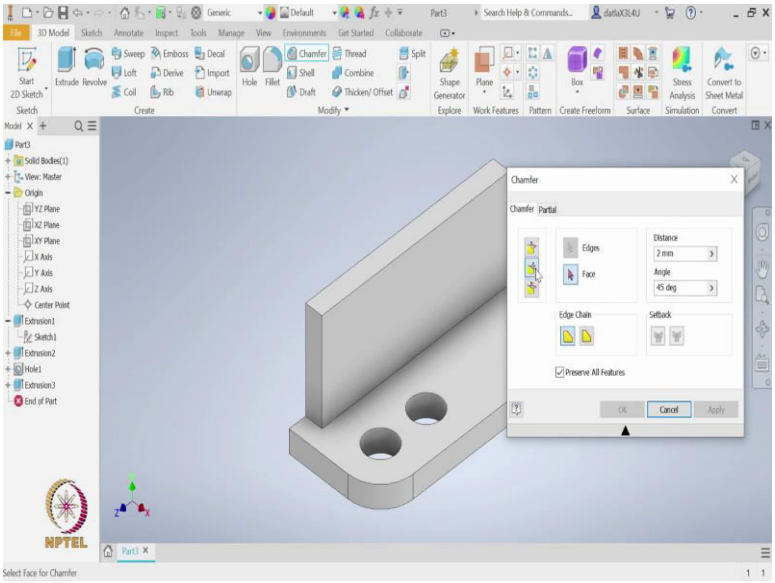
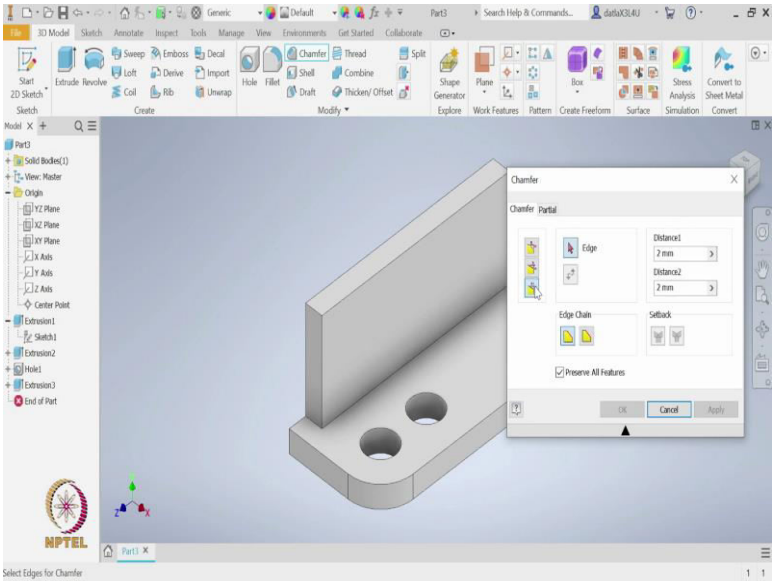
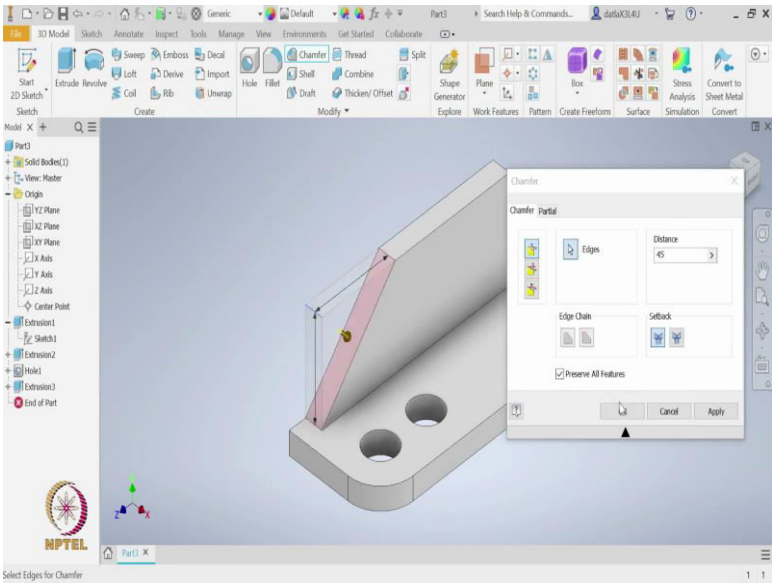
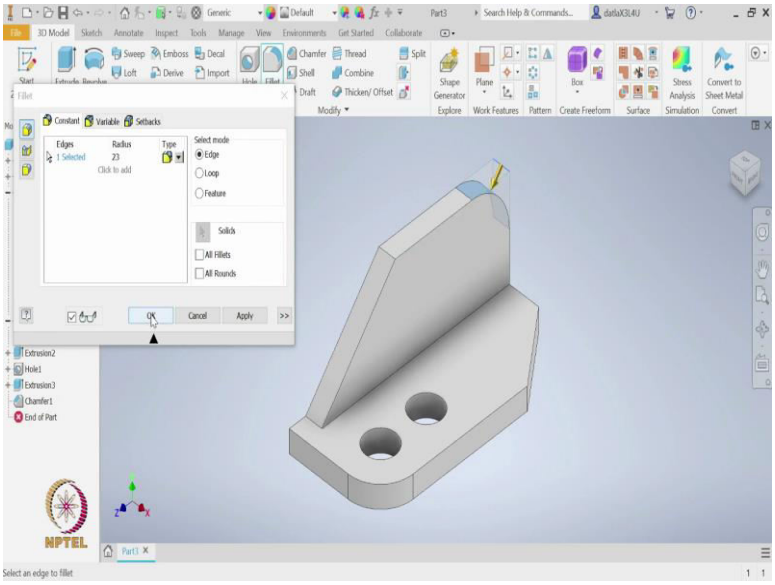
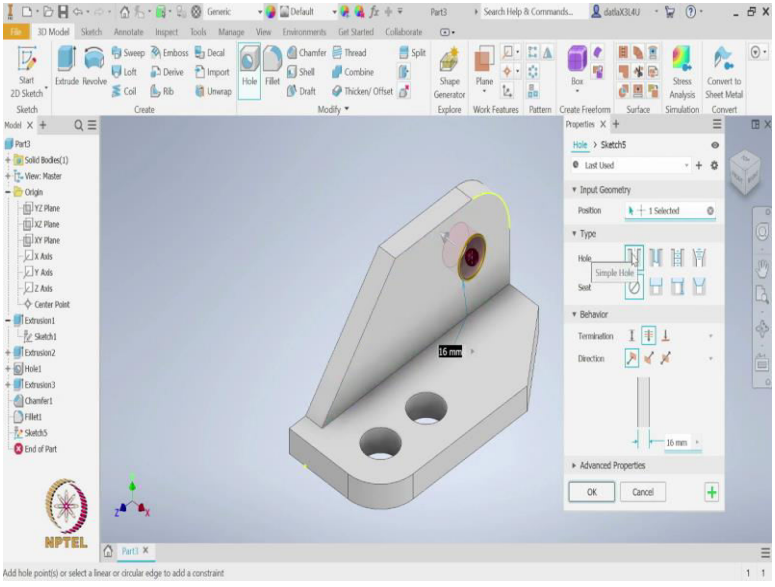







Chamfer
Chamfer Partial
Edges
Distance
45
Edge Chain
Setback
Preserve All Features
Cancel
Apply
Select Edges for Chamfer


Fillet
Constant
Variable
Setbacks
Edges
Radius
Type
1 Selected
23
Click to add
Select mode
Edge
Loop
Feature
Solids
All Fillets
All Rounds
OK
Cancel
Apply
Select an edge to fillet


Properties
Hole
Sketch5
Last Used
Input Geometry
Position
1 Selected
Type
Simple Hole
Behavior
Termination
Direction
16 mm
Advanced Properties
OK
Cancel
Add hole point(s) or select a linear or circular edge to add a constraint

So, the next let us try to build this feature which is this vertical part. We let us see the overall dimensions of it lengthwise 125 - 10. So, it will be 115 and height wise. Since it is a chamfer of 45 by 45. The height is also given here as 45. So, let us create a sketching plane and let us select the this surface let me keep it in this fashion so that I can create that wall.

So, let us start again with this rectangle let us select the corner of it and give the dimensions as the length we said is 115 and height is 45. So, we see that it is fully constrained because we started with a corner and given the dimensions of the rectangle so let us finish the sketch and now let us extrude so when we extrude what is the dimension 10. So, we see that it is in a wrong direction so let us flip it and the dimension 10 is already taken.

So, let us click OK and finish the extrusion. Now we see in this vertical part, there are two features we need to add one is the chamfer to the left and fillet to the right there is a third feature which is the hole so let us go back and see how to do those. So, we will start with this filet this time instead of creating fillets and chamfers through sections we will do it through features so that option is available here.

So, first let us start with the chamfer for the chamfer again it is asking three options whether it is equal distance, or does it have, do you want to specify a distance and an angle or both distances. So, let us specify the click the first option because it is single distance on both sides which is 45. Now I need to click the edge where we want to have the fillet, sorry chamfer.

So, this is a preview that looks OK, so let us press OK. So, once the chamfer is done now we can go ahead and prepare the fillet so again we will be using the feature option of fillet again we need to specify what is the radius of the fillet? Let us look into the problem it is given as R23. So, let us specify this radius as 23 and then we need to select the edge where we want the fillet so let us select this edge and there you go we get the fillet by clicking OK.

So, we have created the chamfer as well as the fillet using the features. Previously we were using by modifying the sketch to create chamfer and fillet. But now using the features directly we can create this chamfer and fillet and lastly we need to create a hole on this vertical part. So, I have selected the hole it is asking for the face on which I need the I need to create the hole so let us select this and let us see what is the diameter of the hole?

It is 16. So, let us specify it as 16 and we noticed that in the problem this whole it is concentrated with the fillet we have here. So, for these we have a dynamic feature or a smart

feature in the software, we just need to select the fillet and once you select the fillet it will understand the hole needs to be concentric with the curve or the fillet.

So, with this, I mean of course we can choose different options of termination in this example, through all is good and even when you create a hole there are other options like what we are selecting is a simple hole, but one can do a clearance hole and tapped hole, taper tapped. So, essentially you can create some threaded portions to create that tapered hole, tapped hole.

And we can also think about if we have a nut sitting on this hole, you can go for counterbore or spotface or the countersink, depending on the mechanical component you are having here. But in this example it is a simple hole. So, let us take this, select this option and click OK. So, coming back to the example we see that we are almost done. Last feature which is remaining is this triangular thing here.

Usually, we call it as the rib. But in this lecture, I will create it as though as an extrusion object. In the next lecture we will talk about how to use the rib feature in the software. But let us stick to the extrusion. But let us think for a moment, how will you create this triangular extrusion. For these, first, we need to select a plane on which we can create a triangle, then we can simply extrude.

So, how do we select a plane? We know that we have to create this triangle on a plane, which is parallel to either this face here or this face here. And what is the distance here it is specified as 45. So, let us try to do that in the software.

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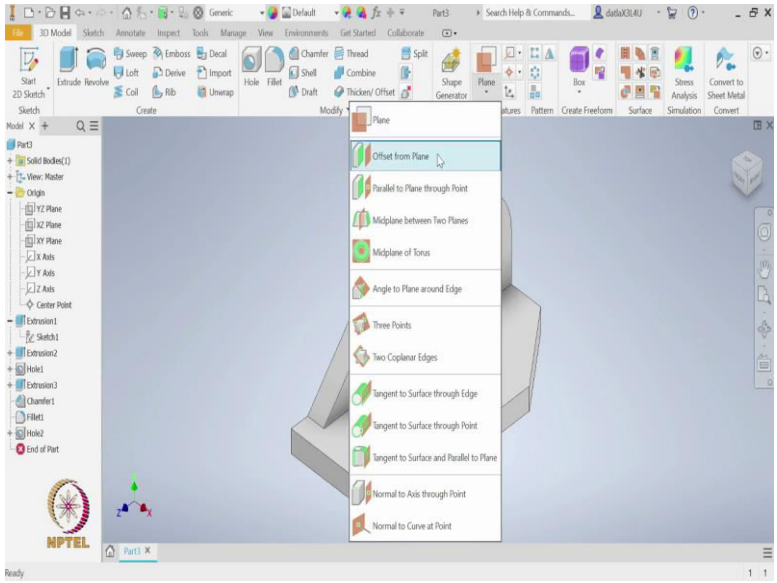
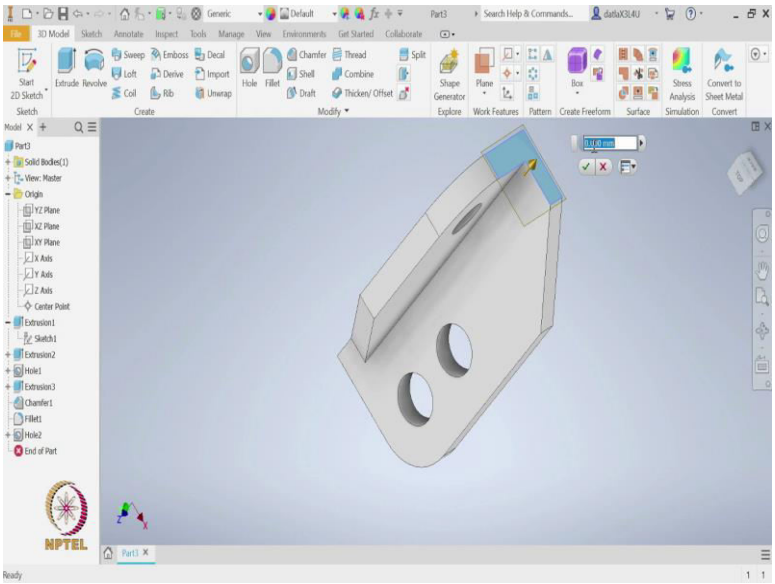
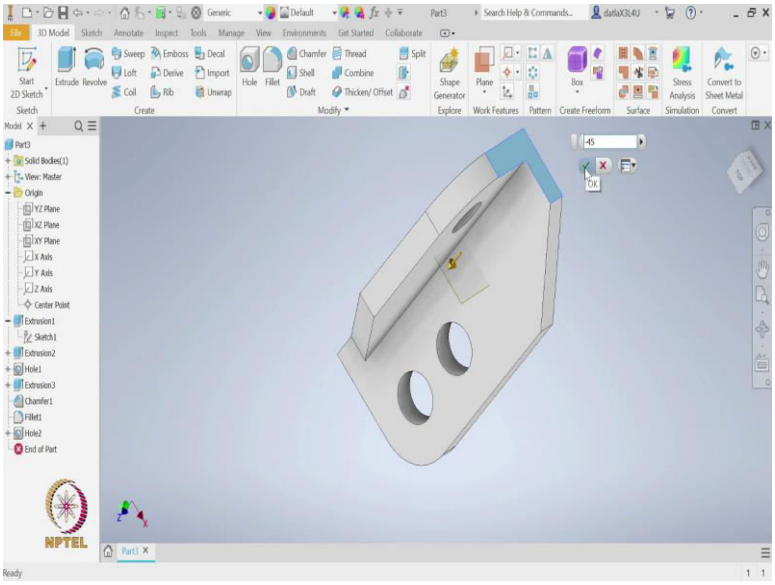
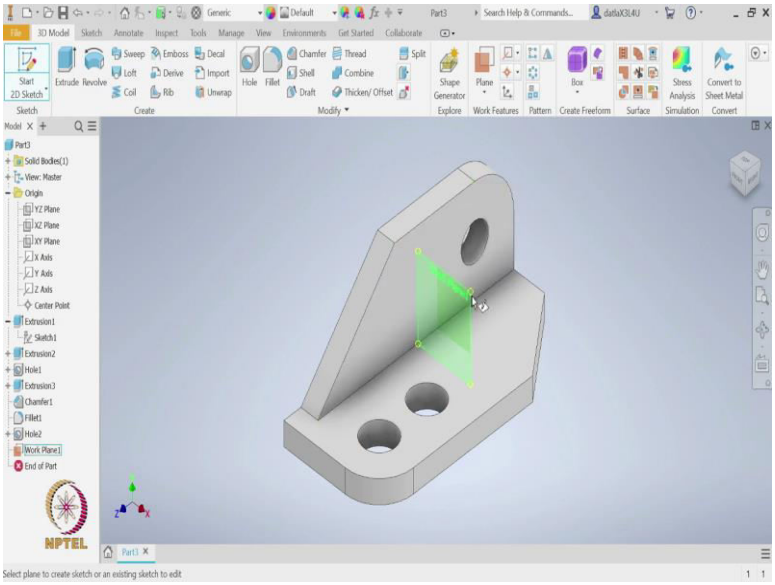
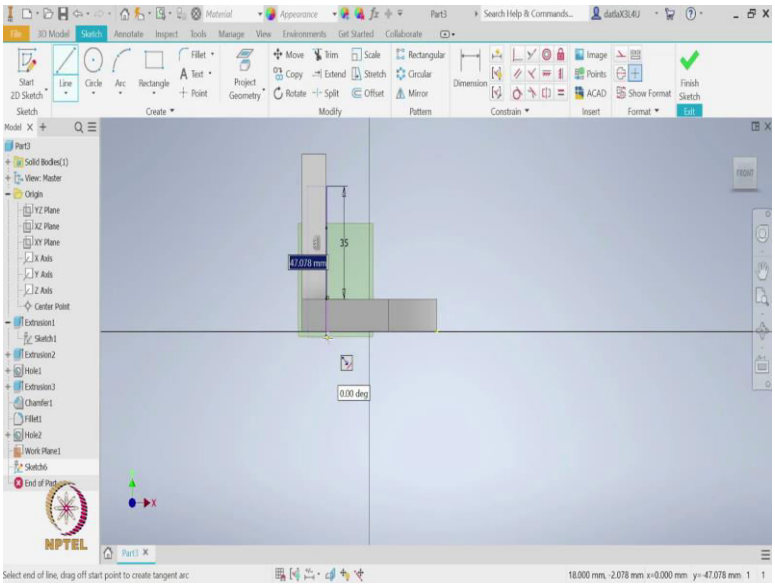
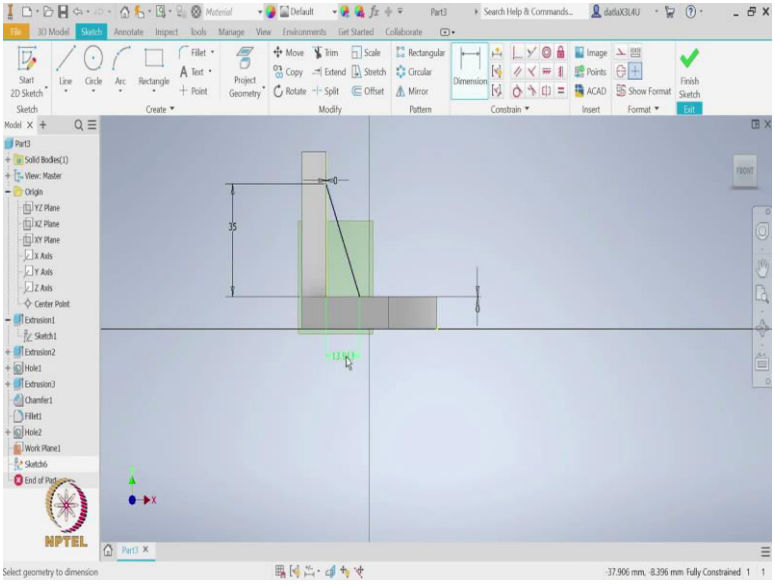


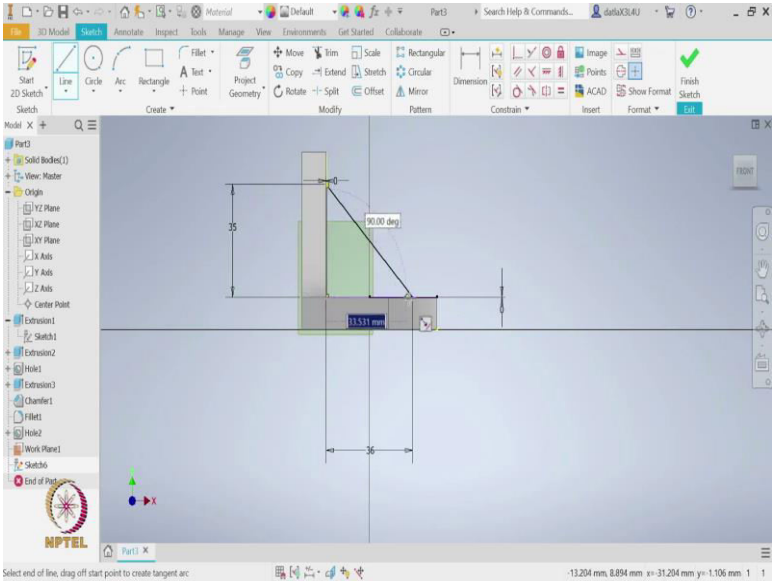

Start 2D Sketch
Extrude
Revolve
Sweep
Loft
Coil
Emboss
Derive
Rib
Decal
Import
Unwrap
Hole
Fillet
Chamfer
Shell
Draft
Thread
Combine
Thicken/ Offset
Split
Shape Generator
Plane
Box
Stress Analysis
Convert to Sheet Metal
Part3
Solid Bodies(1)
View: Master
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Extrusion1
Sketch1
Extrusion2
Hole1
Extrusion3
Chamfer1
Fillet1
Hole2
End of Part
NPTEL
Ready

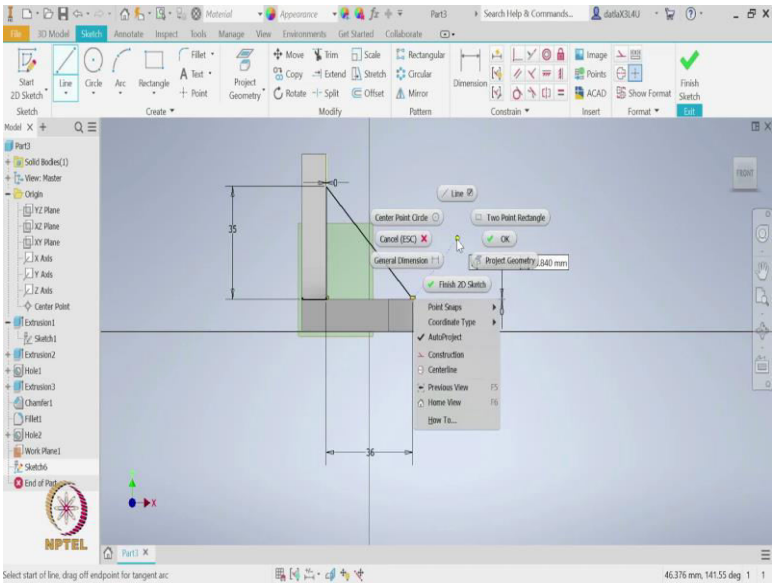

Start 2D Sketch
Extrude
Revolve
Sweep
Loft
Coil
Emboss
Derive
Rib
Decal
Import
Unwrap
Hole
Fillet
Chamfer
Shell
Draft
Thread
Combine
Thicken/ Offset
Split
Shape Generator
Plane
Box
Stress Analysis
Convert to Sheet Metal
-45
OK
Part3
Solid Bodies(1)
View: Master
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Extrusion1
Sketch1
Extrusion2
Hole1
Extrusion3
Chamfer1
Fillet1
Hole2
End of Part
NPTEL
Ready

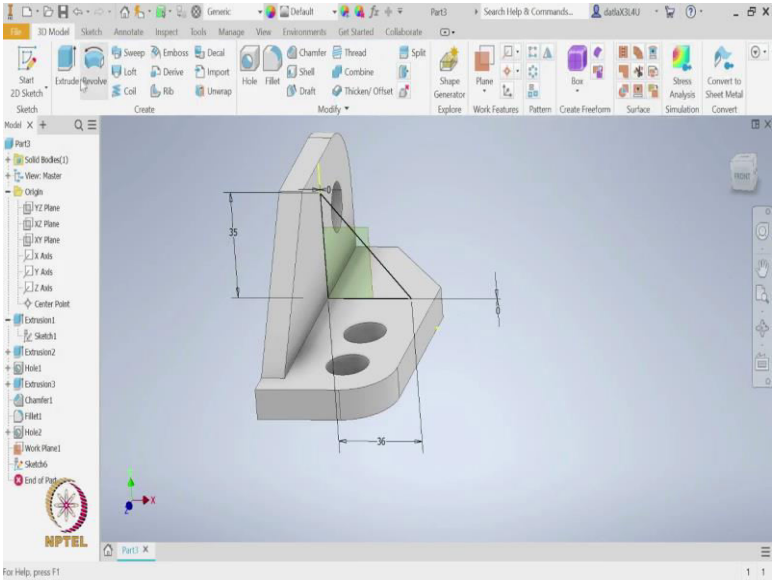

Start 2D Sketch
Extrude
Revolve
Sweep
Loft
Coil
Emboss
Derive
Rib
Decal
Import
Unwrap
Hole
Fillet
Chamfer
Shell
Draft
Thread
Combine
Thicken/ Offset
Split
Shape Generator
Plane
Box
Stress Analysis
Convert to Sheet Metal
Part3
Solid Bodies(1)
View: Master
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Extrusion1
Sketch1
Extrusion2
Hole1
Extrusion3
Chamfer1
Fillet1
Hole2
Work Plane1
End of Part
NPTEL
Select plane to create sketch or an existing sketch to edit

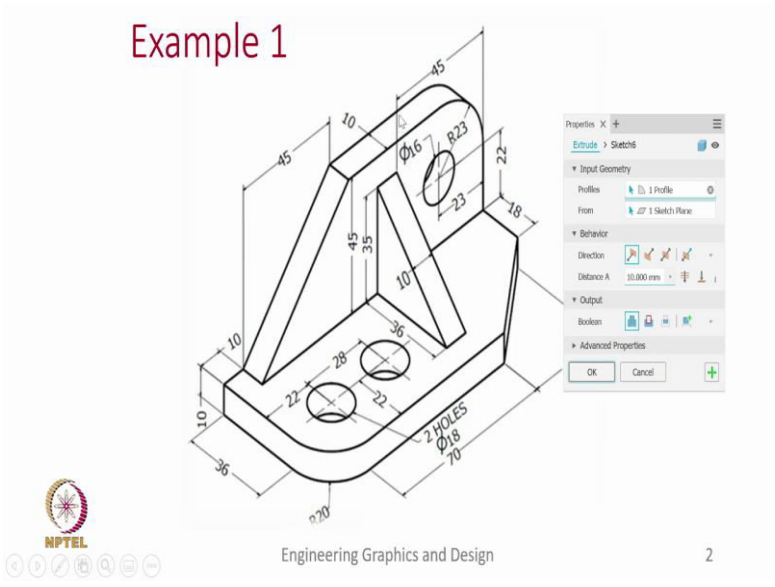

35
47.078 mm
0.00 deg
Select end of line, drag off start point to create tangent arc
18.000 mm, -2.078 mm x=0.000 mm y=-47.078 mm 1 1

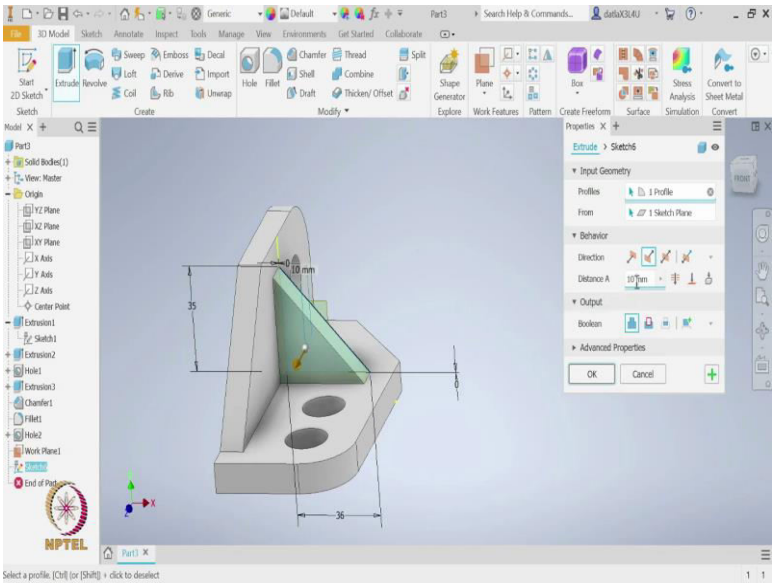

35
Select geometry to dimension
-37.906 mm, -8.396 mm Fully Constrained 1 1

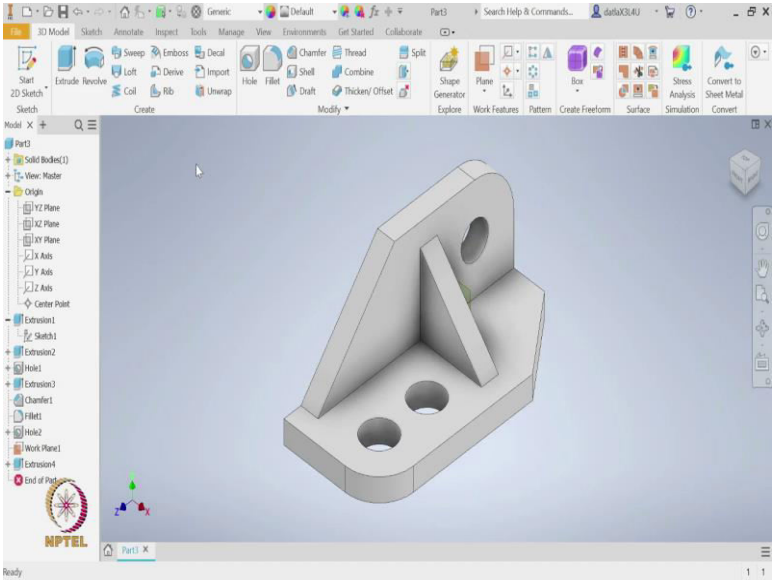

35
90.00 deg
33.531 mm
36
Select end of line, drag off start point to create tangent arc
-13.204 mm, 8.894 mm x=-31.204 mm y=-1.106 mm 1 1


Line
Center Point Circle
Two Point Rectangle
Cancel (ESC)
OK
General Dimension
Project Geometry
Finish 2D Sketch
Point Snaps
Coordinate Type
AutoProject
Construction
Centerline
Previous View
Home View
How To...
35
36


35
36


Example 1
45
10
45
Φ16
R23
22
23
18
45
35
10
36
28
22
22
10
10
36
2 HOLES
Φ18
70
20
Properties
Extrude > Sketch6
Input Geometry
Profiles
1 Profile
From
1 Sketch Plane
Behavior
Direction
Distance A
10.000 mm
Output
Boolean
Advanced Properties
OK
Cancel





So, when we go to this plane option, there are many options to create a plane, we will be using the offset from plane, which means we need to select an object. Sorry, select a face. Here I will be selecting this face. And then it asks what is the dimension of the offset, and the direction currently the direction is showing in an opposite direction.

So, instead we wanted in towards the object. So, instead of giving a positive value, if we give a negative value, it should do the job. So, minus 45 it is showing, showing me a preview that it will create a plane somewhere here since I am satisfied with it, I will select this, but so now we have created this plane on which I can create a sketch.

So, let us again go to the start 2D sketch and select this plane. Now I need to create a triangle. So, let me use this line feature. So, I will start at this corner. Now you can see once you go close it changes the yellow dot changes to green. When it changes to green, it is snapping on

to the already existing corner. So, I will create a vertical line of what is the dimension of this line 35.

So, let us start by creating this triangle using the line feature here. So, we will start with this corner and draw a vertical line and then an inclined line. So, let me so let us get started by creating that triangle. First, let us draw the inclined line randomly, extend this line until it touches the vertical edge and the horizontal edge that we can do through dimensions.

So, if I select the endpoint and this, I have to specify that this dimension is 0 and similarly, continuing with the dimension command, I can pick the endpoint of this line and this edge and mention that this height is how much? 35. Similarly, I will do the same with the other end of the straight line. So, this and this edge I will say the distance is 0.

This and the vertical edge I will specify this distance as 36. Since it is a single line, I need to create 2 more lines to complete this triangle and make an area. So, we have created the vertical line and let us again create the horizontal line between this point and this point let us select OK. Now, we have created 3 lines required to create this triangle.

So, let us say finish sketch and then go to extrude. So, now let us see whether the extrude direction is what we wanted. So, if you go back to the figure we see we have already created a plane at 45. So, this triangle should be extruded in this direction. So, which means I should select the other direction and mentioned that the distance is 10.

So, with this we complete this problem. Now you can see it completely matches with the object we have started to attempt for. So, to conclude, from this example, what we have done is we have used couple of features which are very useful. First, we have used the extrude feature which is already covered in the last class, but there are some new features we have used to create this object like the features of the hole, fillet, chamfer.

And we have also used in sketches, how to create this, fillet and chamfer as well as the holes. So, with this, let us conclude this lecture.