Engineering Graphics and Design
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Week 8: Introduction to CAD
Example: Solid Model

Welcome back, we will look into another example to create a simple solid model. So, we will again review all the features we have discussed in the previous 2 lectures like the extrude feature, the hole, rib and mirror. These are the some of the features which we have already discussed. In this example we will go back and revisit those for us.

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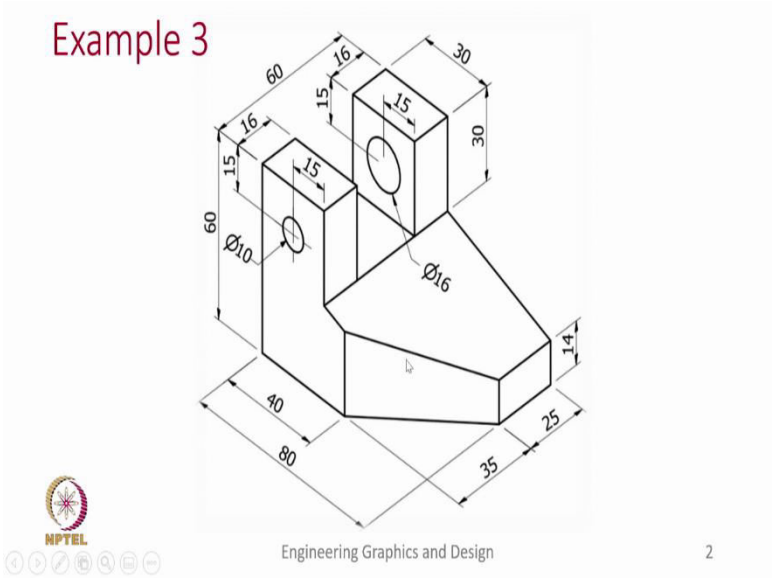

So, this is the example 3 that we will try to solve in today's lecture. So, let us go to the software before we go to the software. Let us look at what is the overall dimensions of this object, we see that this dimension is 80, the depth is 35 + 25 so, it is 60 and the height is also 60. And what are the kind of features we see? There are two holes but now see these are not symmetric.

So, the first hole is diameter 10. And the second hole is a diameter of 16. But these vertical objects are symmetrical, they both are of the width of 16. The other important feature we need to understand is there is a slope so which means we need to first start with a cuboid and then make a taper cut. So first we will make with a base of 80 by 60 and then extrude it to a height of since this is 30, we need to extrude it to a height of 30.

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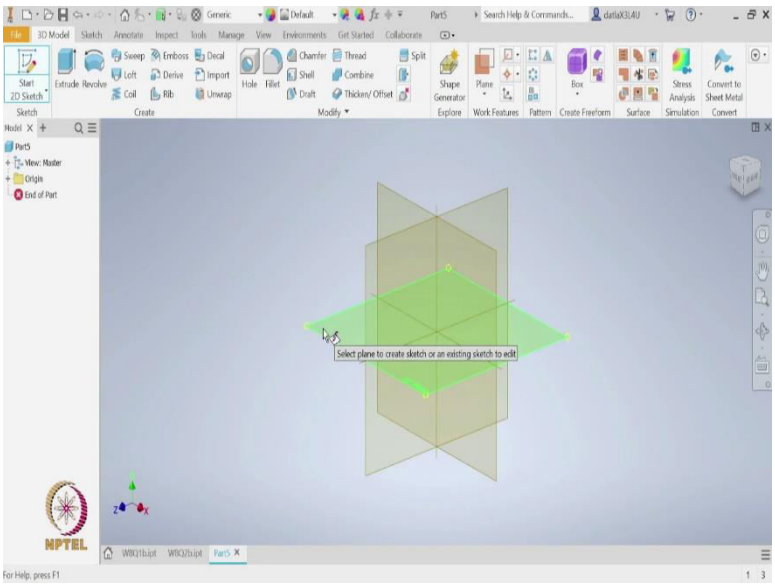
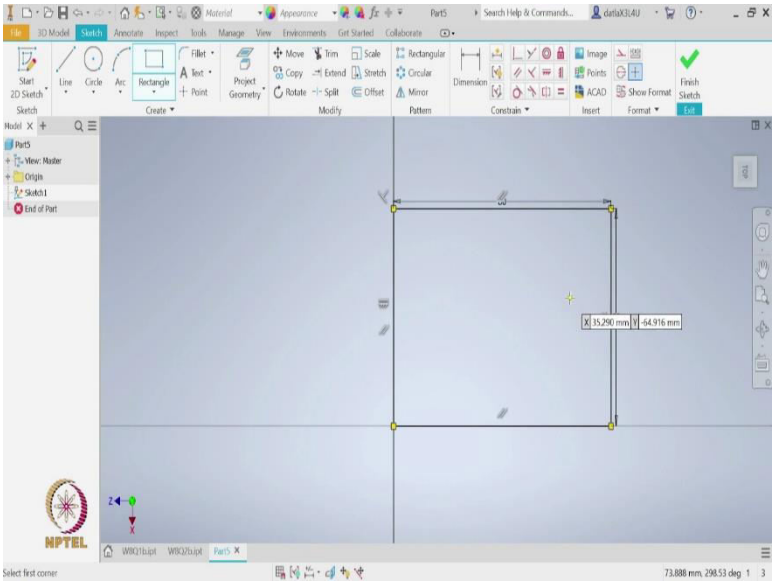
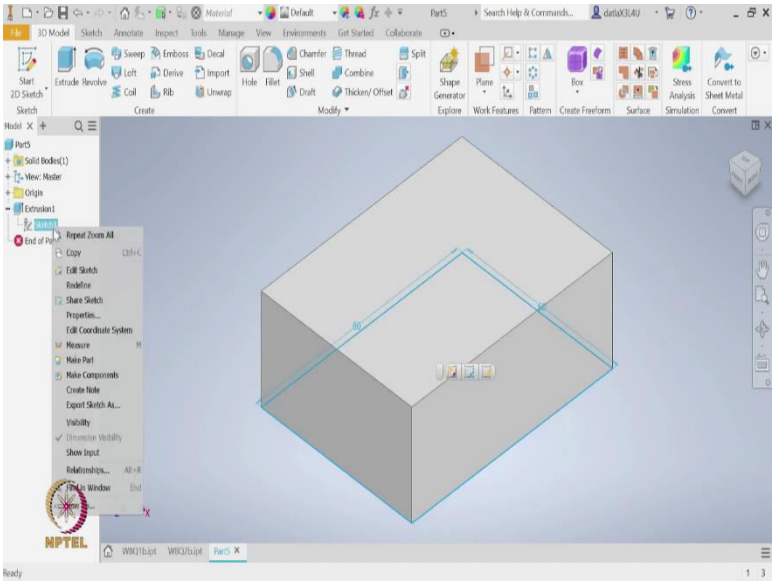
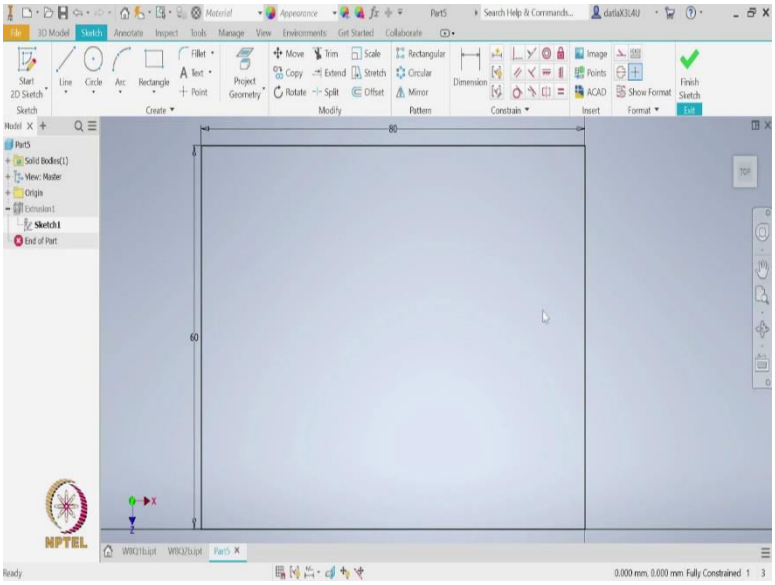
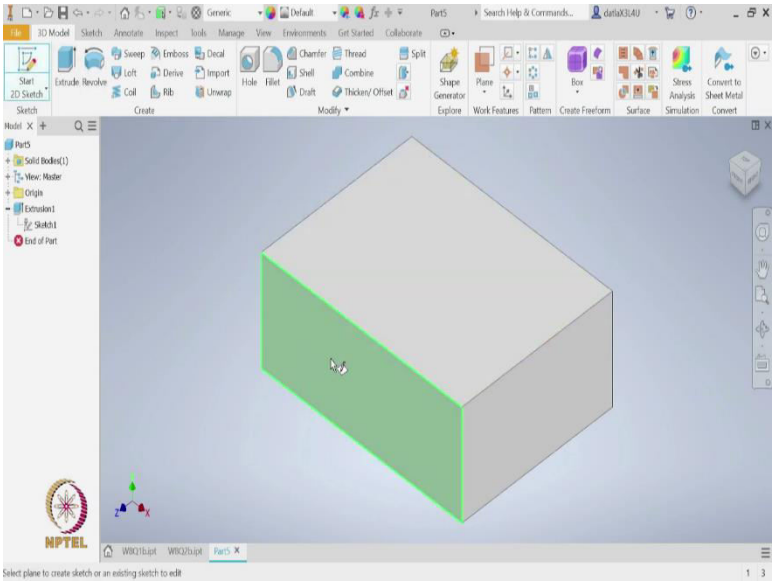
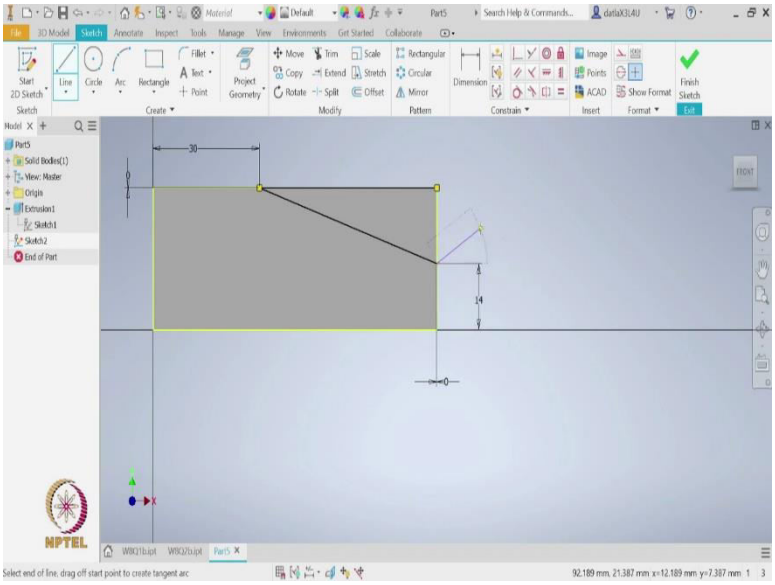


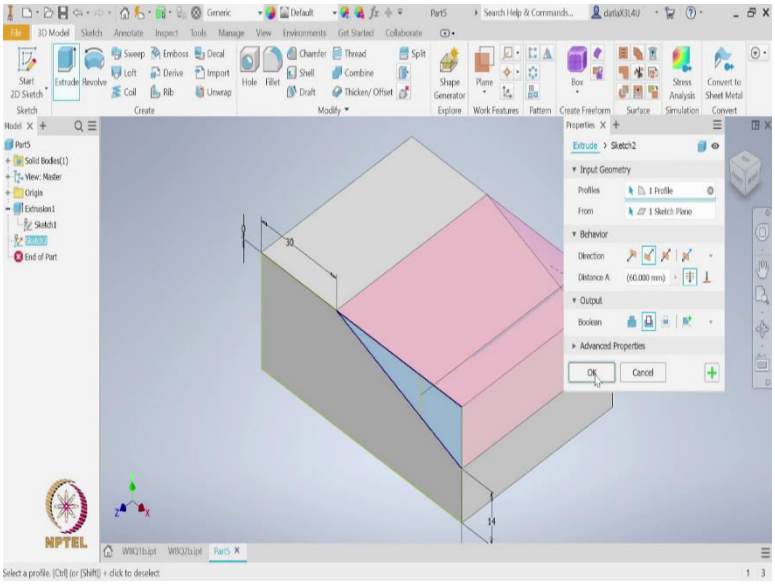


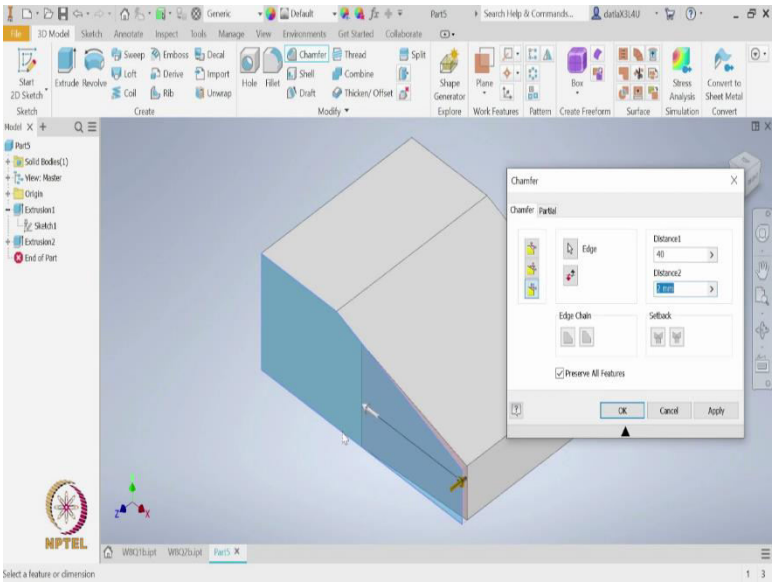

Repeat Zoom All
Copy
Edit Sketch
Redefine
Share Sketch
Properties...
Edit Coordinate System
Measure
Make Part
Make Components
Create Note
Export Sketch As...
Visibility
Show Input
Relationships...
80

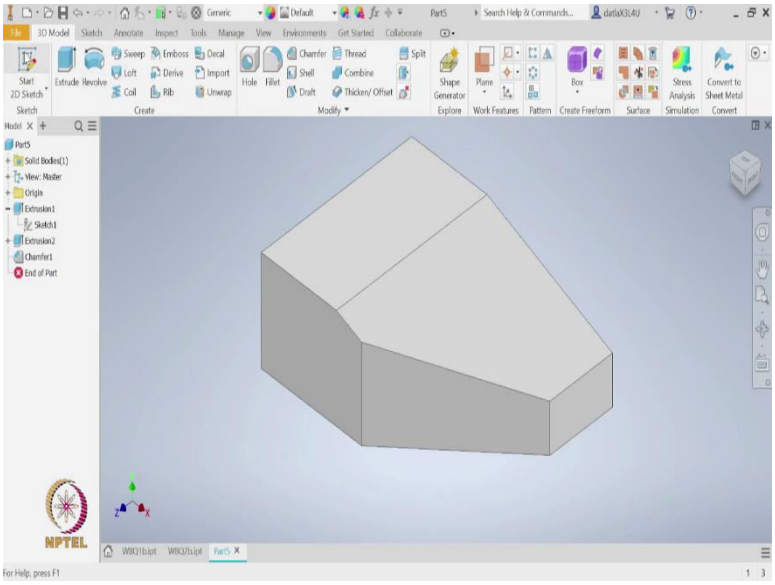

80
60




Start 2D Sketch
Line
Circle
Arc
Rectangle
Fillet
Text
Point
Project Geometry
Move
Copy
Rotate
Trim
Extend
Split
Scale
Stretch
Offset
Rectangular
Circular
Mirror
Dimension
Image
Points
ACAD
Show Format
Finish Sketch
Sketch
Create
Modify
Pattern
Constrain
Insert
Format
Exit
Model
Part5
Solid Bodies(1)
View: Master
Origin
Extrusion1
Sketch1
Sketch2
End of Part
30
14
0
FRONT
NPTEL
Select end of line, drag off start point to create tangent arc
92.189 mm, 21.387 mm x=12.189 mm y=7.387 mm


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
Create Freeform
Stress Analysis
Convert to Sheet Metal
Explore
Work Features
Pattern
Surface
Simulation
Convert
Properties
Extrude > Sketch2
Input Geometry
Profiles
1 Profile
From
1 Sketch Plane
Behavior
Direction
Distance A
(40.000 mm)
Output
Boolean
Advanced Properties
OK
Cancel
30
14
NPTEL
Select a profile. [Ctrl] (or [Shift]) + click to deselect


Chamfer
Partial
Edge
Distance1
40
Distance2
Edge Chain
Setback
Preserve All Features
OK
Cancel
Apply
NPTEL
Select a feature or dimension



So, let us get started with the software. Now we are in the initial opening interface of the software. So, let us start by start 2D sketch and we will pick the x z plane. In the x z plane, we said the dimensions are 80 by 60. First let us specify that using a rectangle. We will again select the origins which makes it convenient. So, the dimensions are at base 80 by 60 tab 80 and select all.

So, now it is fully constrained we can finish the sketch and extrude, the extrude height we have already seen as 30. So let us specify that and select all. So, let me reorient the object by so we will go to the sketch, edit the sketch by saying that this is 60 and the other is 80, finish sketch. So, now we will pick this left face on which we will create a taper. So again, start 2D sketch on this face.

And to create a taper we need to draw a straight line. So, since this distance is it given? Yes. So, we know that this distance is 30. So, we should start from 50. So, let us first draw a random line and then we can dimension it. We can dimension saying that from this point to this left edge is 30. And let us make it coincident with the top face. And the other end. Let us again coincide with this edge by making this distance as 0.

And lastly, we see that at the bottom of the taper we have 14 so that is what we will specify. So with this, we have completely constraint this line but for us to make a cut we need an area so let us draw two more lines. So, let us finish this sketch and then use extrude, select this feature and in the Boolean we will mentioned that it is a cut. We will say it is a through all cut.

So, now we have created this taper so to create this chamfer let us again use the chamfer option here, we need to specify the edge and then it is asking whether it is a symmetrical or asymmetric. In our case it is asymmetric which means we need to specify both directions. So, again it asking to set the edge and the 2 directions. So, the first direction is 40 and direction 2 is 35. So, there we have created a chamfer.

So, the next step is to create these two vertical things since both of them are symmetrical we will first create the left vertical part and what are the dimensions of it? The depth is 16 and the height is 30.

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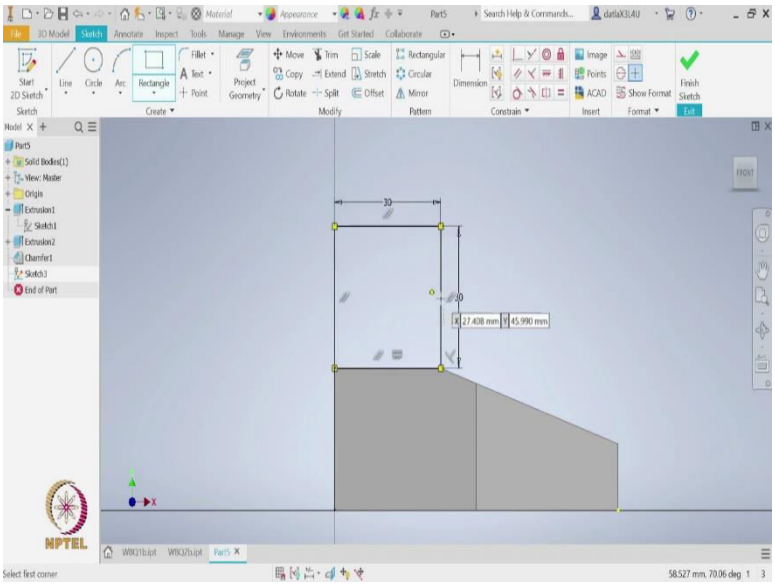
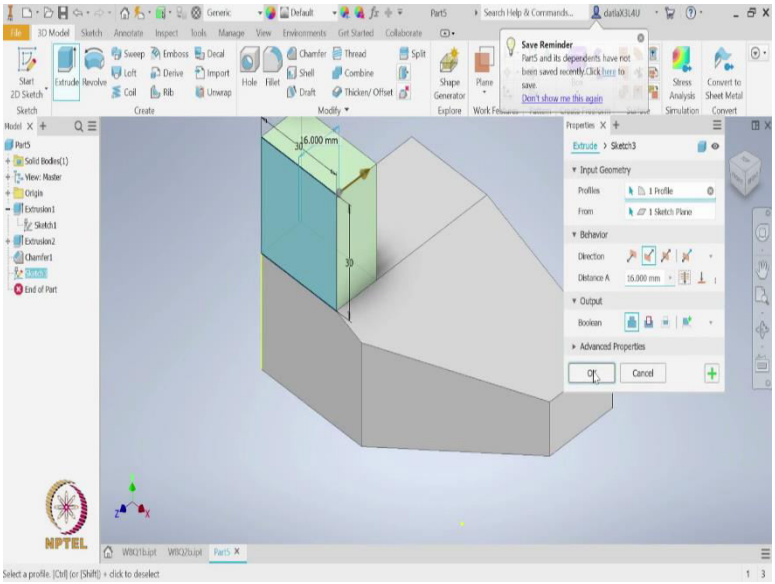
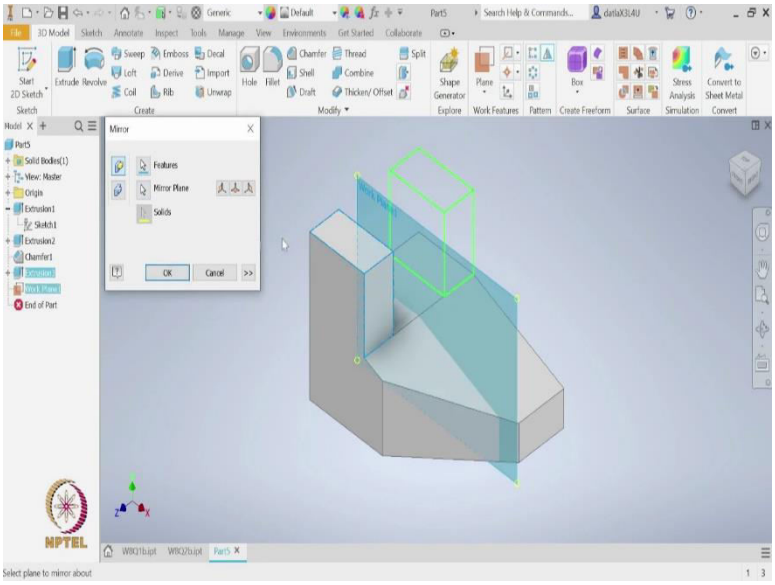
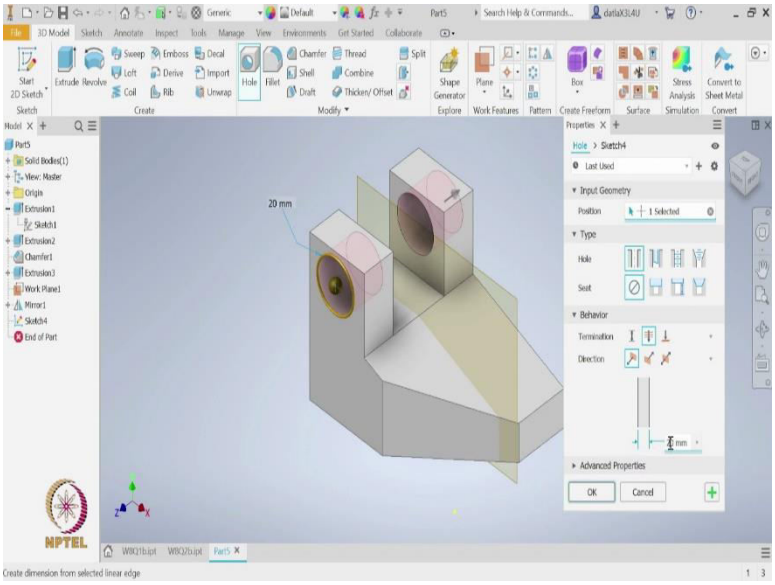
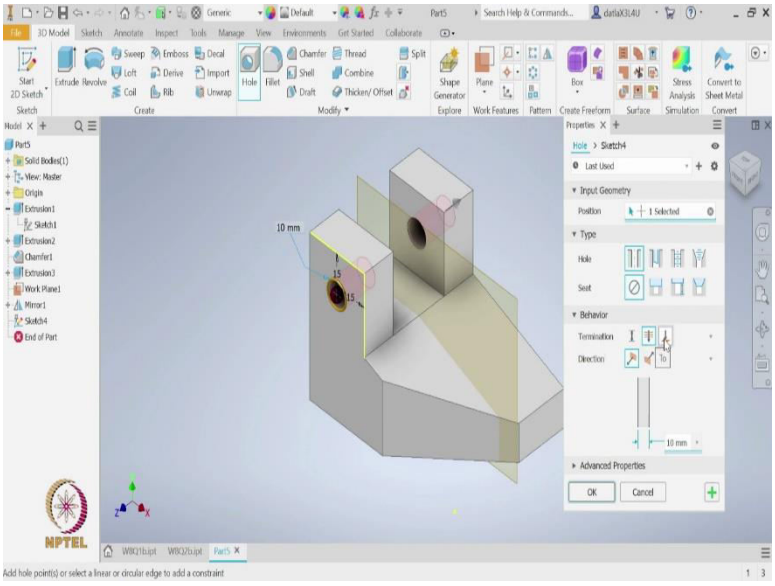
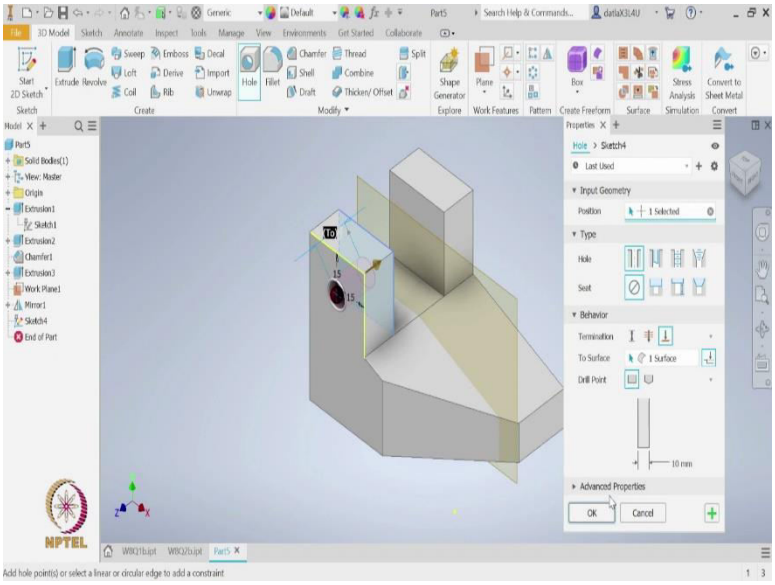


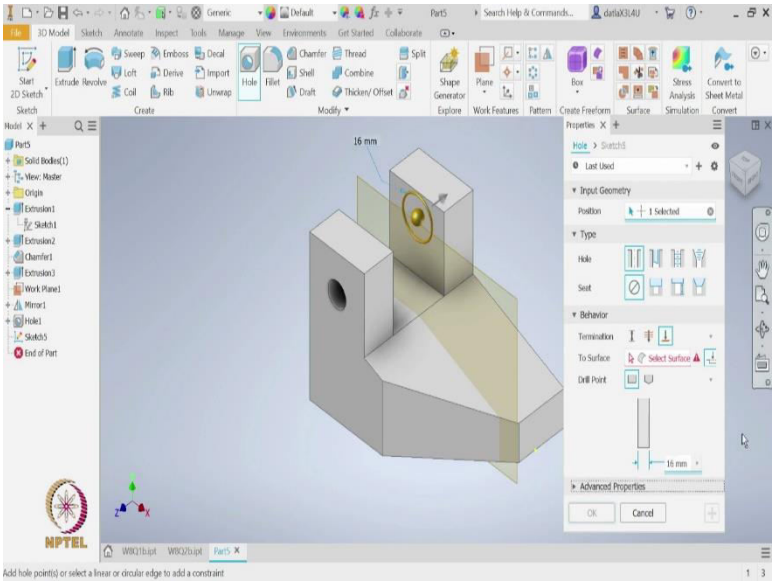


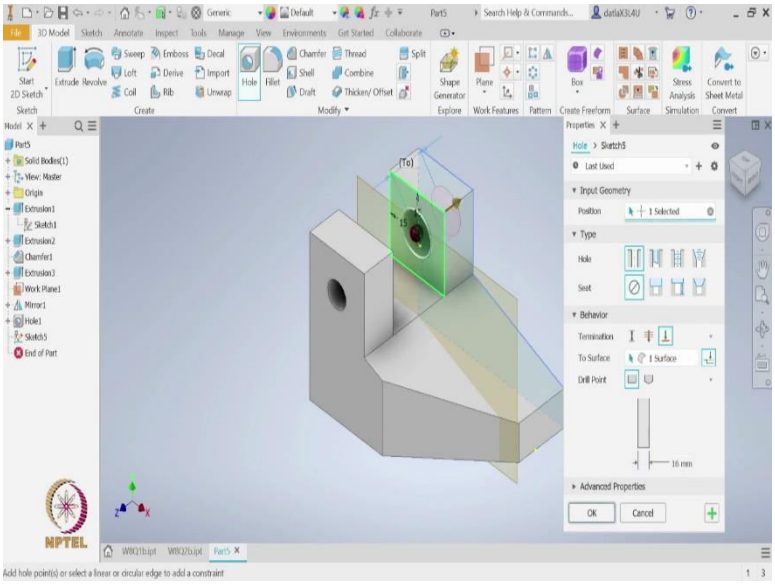

Mirror
Features
Mirror Plane
Solids
OK
Cancel
Select plane to mirror about

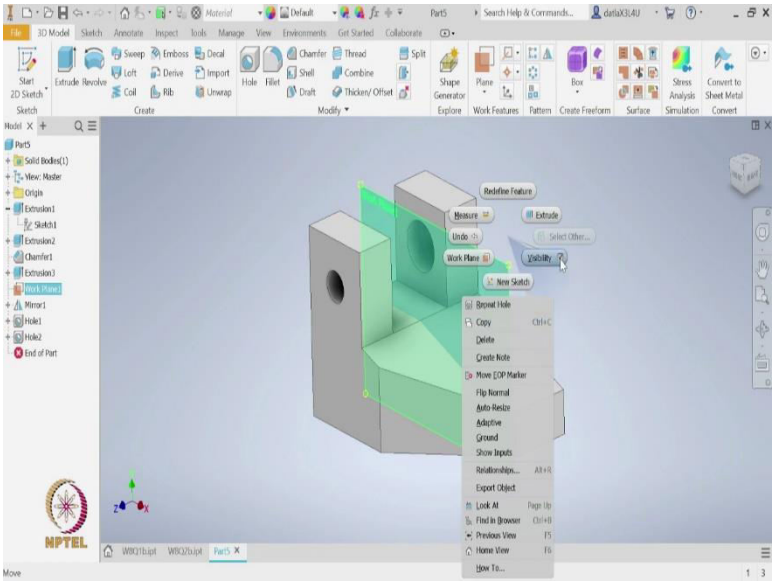

20 mm
Hole
Sketch4
Input Geometry
Type
Behavior
OK
Cancel
Create dimension from selected linear edge

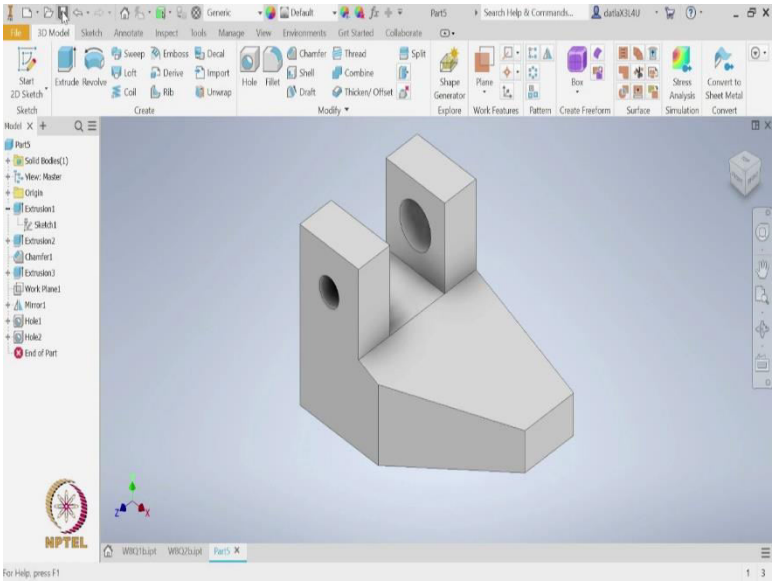

10 mm
15
15
Hole
Sketch4
Input Geometry
Type
Behavior
OK
Cancel
Add hole point(s) or select a linear or circular edge to add a constraint


To
15
15
Hole > Sketch4
Input Geometry
Position
1 Selected
Type
Hole
Seat
Behavior
Termination
To Surface
1 Surface
Drill Point
10 mm
Advanced Properties
OK
Cancel
Add hole point(s) or select a linear or circular edge to add a constraint


16 mm
Hole > Sketch5
Input Geometry
Position
1 Selected
Type
Hole
Seat
Behavior
Termination
To Surface
Select Surface
Drill Point
16 mm
Advanced Properties
OK
Cancel
Add hole point(s) or select a linear or circular edge to add a constraint


(To)
15
Hole > Sketch5
Input Geometry
Position
1 Selected
Type
Hole
Seat
Behavior
Termination
To Surface
1 Surface
Drill Point
16 mm
Advanced Properties
OK
Cancel
Add hole point(s) or select a linear or circular edge to add a constraint


Redefine Feature
Measure
Extrude
Undo
Work Plane
Visibility
New Sketch
Repeat Hole
Copy
Delete
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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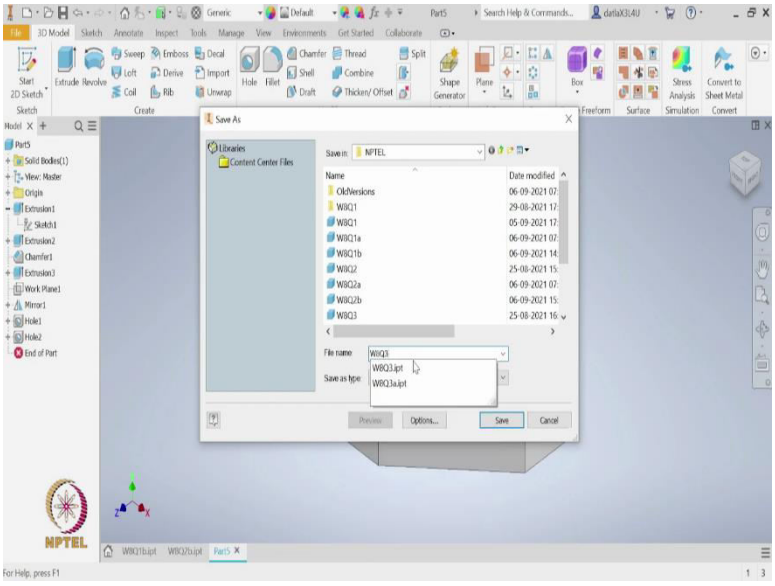
Save As
Libraries
Content Center Files
Save in: NPTEL
Name
Date modified
OldVersions
W8Q1
W8Q1a
W8Q1b
W8Q2
W8Q2a
W8Q2b
W8Q3
File name: W8Q3
Save as type: W8Q3.ipt
W8Q3a.ipt
Options...
Save
Cancel
NPTEL

So again, start 2D sketch on this face. Let us create a rectangle let us take this corner, we said it is 30 by 30. So, it is completely constrained so we can finish the sketch and extrude. So, let us flip the direction and extrusion is for 16 millimetres. So, let us select the OK button to complete the extrusion.

So, since we said it is symmetric, these 2 vertical blocks are symmetric, let us do that. For that we should first create the plane which is symmetric. So first let us create a midplane between the left face and the right face. So, it is created. Now I need to select this mirror feature I need to first select the feature I want to mirror, which is this vertical thing and second the mirror plane. So, it is showing me a preview which is what I intend.

So, let us click OK to complete the mirroring. So, the last step remaining is to create these 2 holes. The first one is of diameter 10 and the second is of diameter 16 and the locations are 15 and 15. So, let us use this hole tool let us select the left face the diameter we said is 10 and we need to specify the dimension so with this edge it makes a distance of 15 and with the top edge it makes it another distance of 15.

So now since we have specified the diameter of the hole and location of the center of this hole, we need to specify where to terminate. This time we do not want to use that through all because what it is doing is it is creating hole on both these vertical objects and the vertical parts. So instead, we will say we can either specify the distance or simply say To. We will set it this way face of the first vertical part.

So, we have a hole only on the first vertical part and termination. This is only for drill bits and all we have these 180 degrees but otherwise it is a what is this called? A flat termination. So, now let us do the same thing for the hole on the second vertical face. We will first select the diameter which is 16 I believe, yes, and the centre is at a distance of 15 with the vertical edge and is what and then we need to specify where what is the termination of this hole.

So, we again use this To and select the range. Now we select the rear edge to complete the hole OK should be done. With this we complete the object we can rotate and see that the two holes are cut as per the requirement, as per the question we have. And again of course now we see this additional middle plane but if you don't want to see we can right click and add the visibility we can uncheck it.

So, with this we now complete this object. Let us review what are the different features we have created while creating this object. We have created the extrusion command, chamfer. Again, mirror command and the hole command. So, after we are done with the object, we can save it by simply clicking this Save button and then it will ask you what is the name you want to give. For example, here I am saying the week 8 question 3.

So, by that you can save your object as well. We will meet again in the next week, where we will then discuss more about some advanced features of creating more complex objects. Thank you for your attention.