Engineering Graphics and Design
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Week 9: Part Modelling 1
Example 5

Welcome back to week 9 of Part Modelling 1. In the last two lectures we have discussed about three tools which are used for solid modelling which are the Revolve, Loft and the Pattern. In the previous example 4 which we solved in the previous lecture we showed some of these tools. In today's lecture where we will solve the example 5, we will again use those tools, so that you can see what are the different options in these tools we can use to create complex objects.

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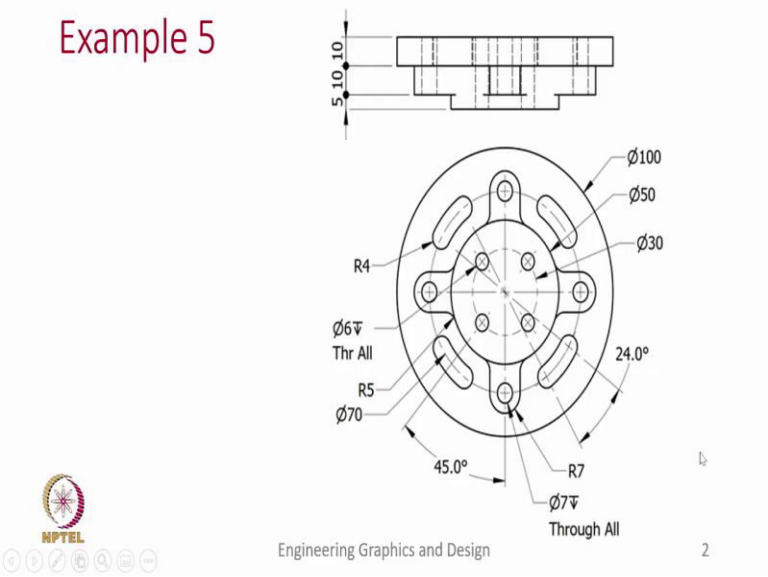

So let us get to the problem. Here you see two views of the object we are creating. This is the front view and this is the top view. So let us try to understand by looking at the different features of this problem. So if you look at the top view and the front view one should understand that there is a base plate which looks like a cylinder of diameter 100 and height 10.

On that base plate we have this slot cut. And on top of this base plate we also have this feature as an extrusion. And at the center there is another cylinder which protrudes for a height of 15 and it has a diameter given as 50. It is given here. And there are other features like the small circle, the cut and this feature, which if we notice closely they repeat. They repeat four times, which means

we can use the circular pattern because they are rotated by an axis at the center here. So we will use the circular pattern to repeat these or duplicate these by four times.

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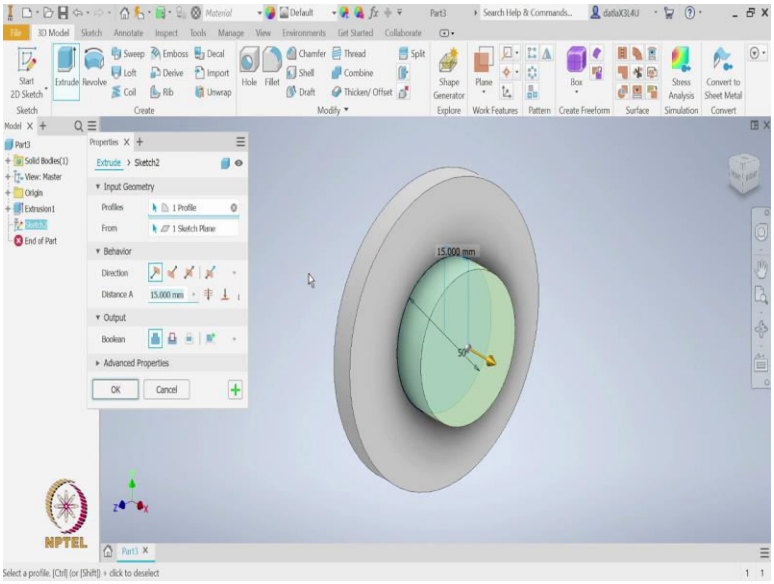

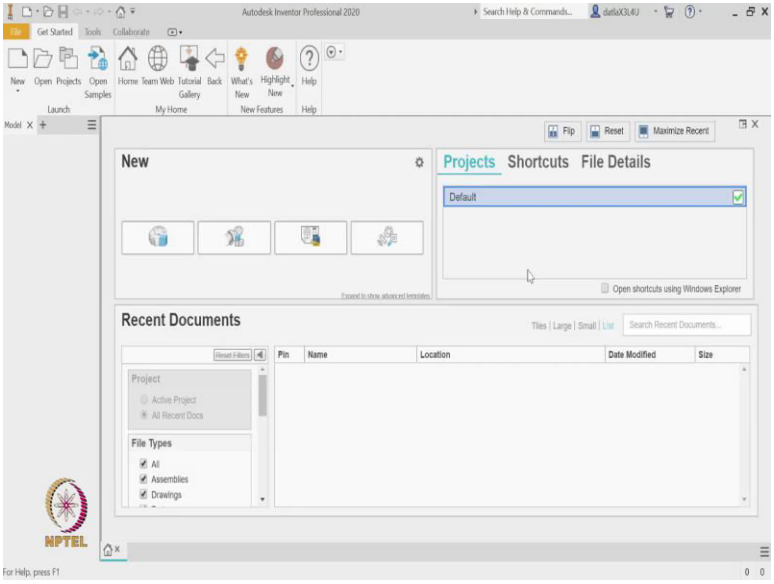

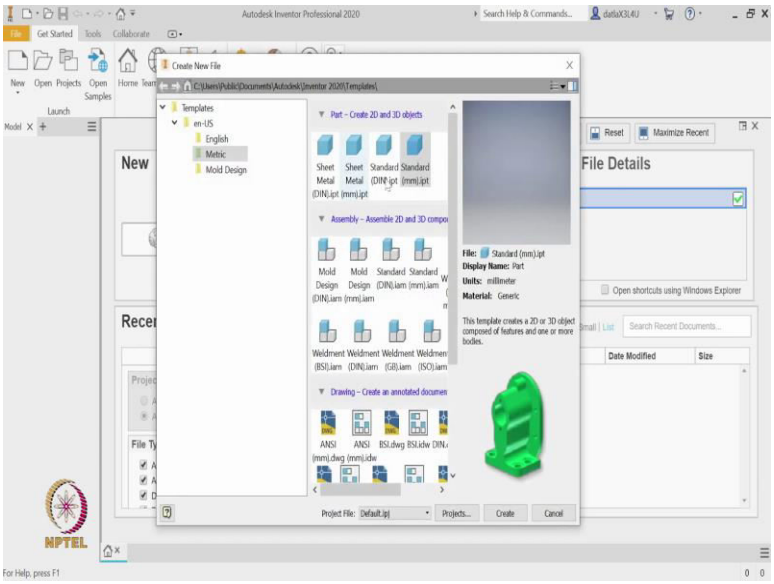

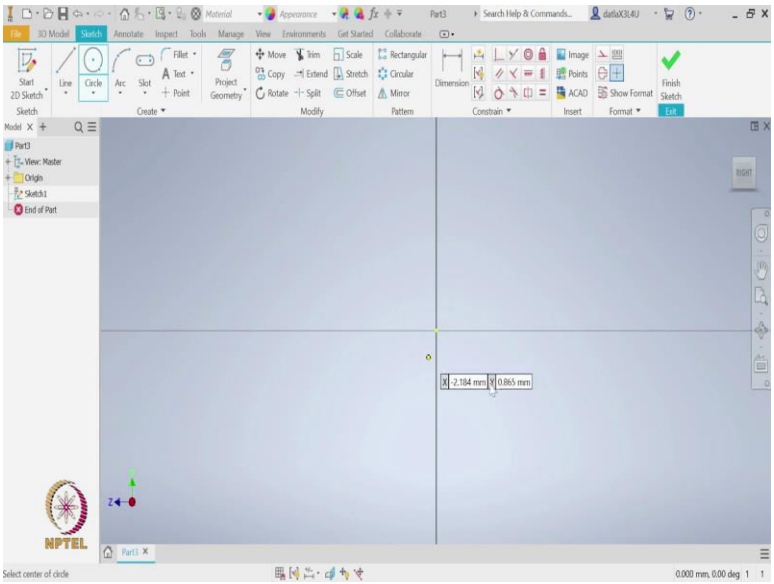

So, let us get started with the software. We are at the Initial Interface. Let us select the part template. And let us get started by creating a 2D Sketch. So let us create this circle of radius 100. Finish the sketch and extrude for a height of 10.

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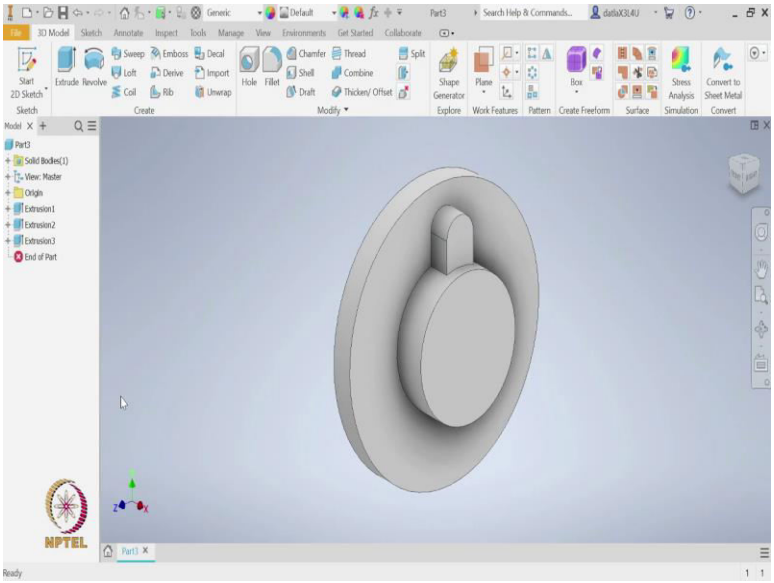

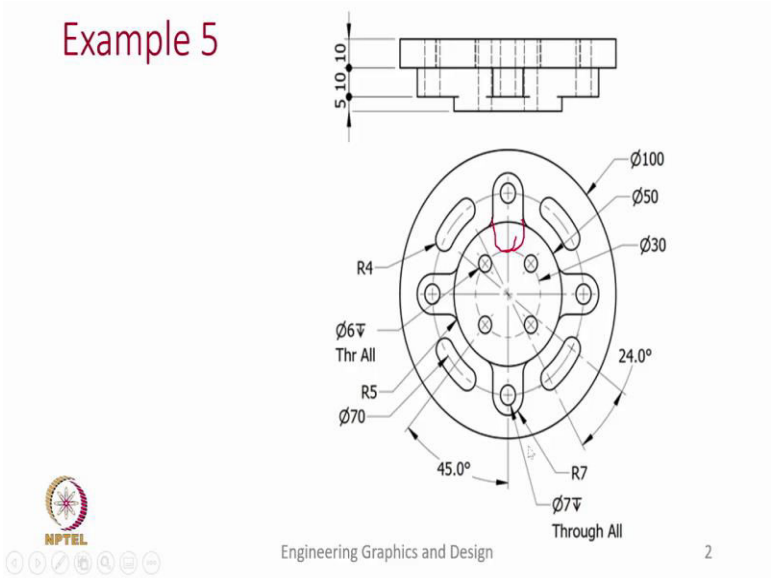

So, next we will create another cylinder on the top face. We will use this circle to create a circle of diameter 50. Finish the sketch and extrude by a length of 15. Now we will create this feature here. There is one way of creating this feature by either creating a rectangle and making a fillet of this radius.

The other way is I can use the slot which is available in the dropdown menu of the rectangle in Sketch. So what we are trying to do is we will say that this is a continuous slot. So though we create a complete slot, the one which I am showing in red does not matter because it will be as a part of the 50 diameter cylinder which is already modeled.

So let us create that slot. Start 2D Sketch on the top face of the base plate. And then I will be using this Slot by Two Centers. So let me select Center 1 and a vertical line then say Central 2. And then I have to specify these dimensions.

Now let us look what is this dimension. We have said this radius is 7. So we should specify this as 14 and click Enter. So let me dimension to constraint this. So this first center to the origin is at a distance of 35, and this origin, let us align it with the Y axis by specifying this dimension to be 0. And let us finish the sketch. And Extrude. So this time we will extrude by 10. That is what is given in the problem.

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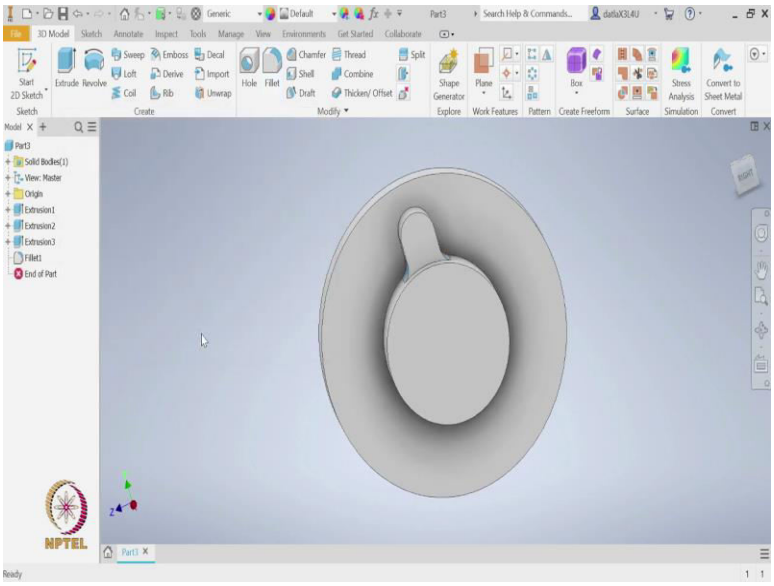

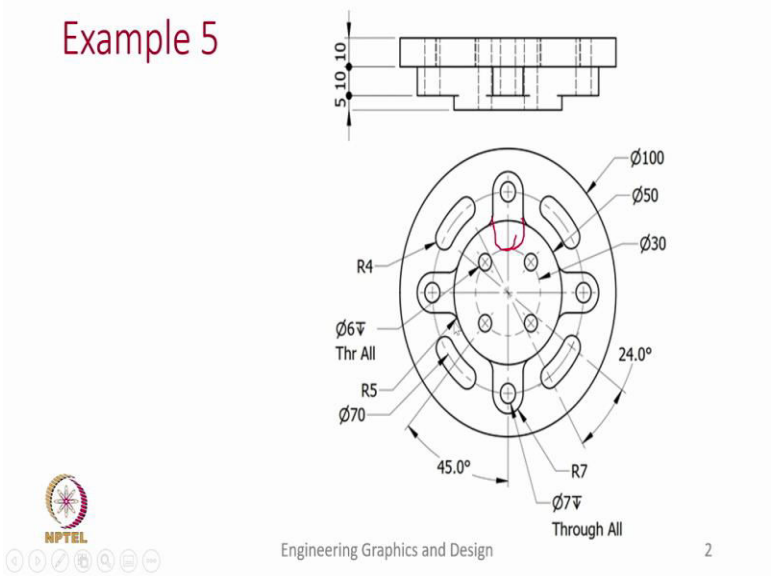

So now if we look at the problem we see that there are two fillets at the bottom of this extrusion, and the dimension of this fillet is mentioned here as radius 5. So let us use the Fillet tool, mention the radius as 5 and then select the edge where we want to make the fillet. This is the Fillet 1. Let me rotate and select the second edge for the second fillet. So now we are able to create both the fillets by selecting Ok.

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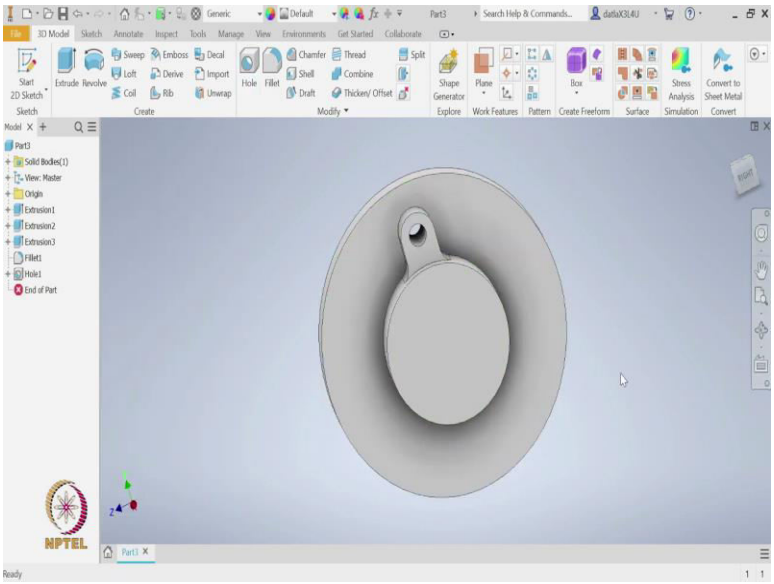

Next we notice that there is a central cut on this feature. The diameter of the small hole is given here. It is diameter 7 and the cut is Through All. So we will use the Hole tool. Select the top face. The diameter is 7 as mentioned here, diameter 7 and Through All. So let us select the termination

as Through All. But let us make it concentric with this circular arc by selecting it. Now it is concentric to complete the Cut feature.

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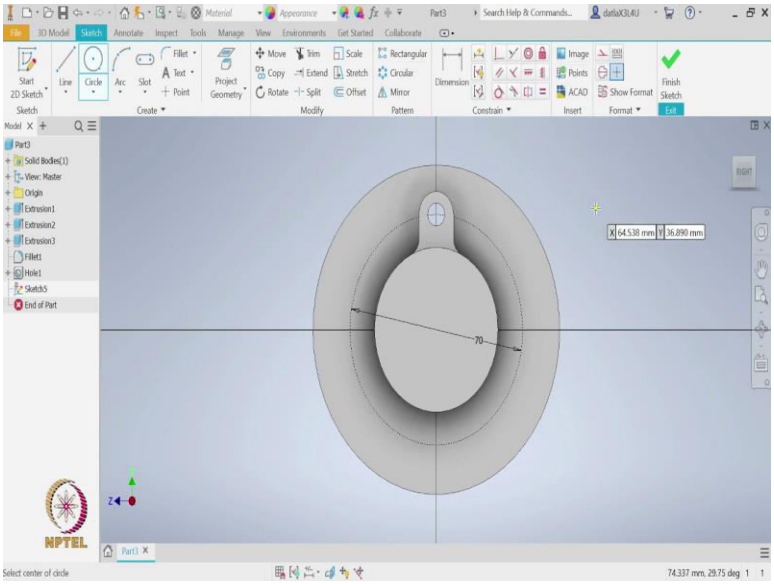

So now we are done with this extrusion feature. Now let us focus on how to get this slot as a cut. So for this again we will create one more Sketch. So let us go to Start 2D Sketch. Select this base and on this base, even before I start I will be using this command on the Slot which is Three Point Arc.

But before I do that arc, let me first understand on which circle is that arc. So it is on a circle which has a pitch circle diameter of 70. So it will be helpful for me if I make a construction circle. It is not the regular circle where we draw a arc or a circle, but it is a construction which means it is only useful to draw other features.

So let us see how to do that. So if you see there is this Format panel. In this Format panel, I go and click the Construction. So when I click on it, it is highlighted. Once it is highlighted and I select a Circle, basically the circle which I am about to draw will be a construction circle. So let us select the center and specify the diameter of this Construction Circle to be 70. So now you see the construction circle that it has created is with these dashed lines.

But this is what we need to remember very carefully. After we are done with the construction lines we need to again click it, otherwise it remains selected. So next feature whatever we are

able to sketch will again be construction. Since we do not want that I will click on this Construction one more time to deselect it.

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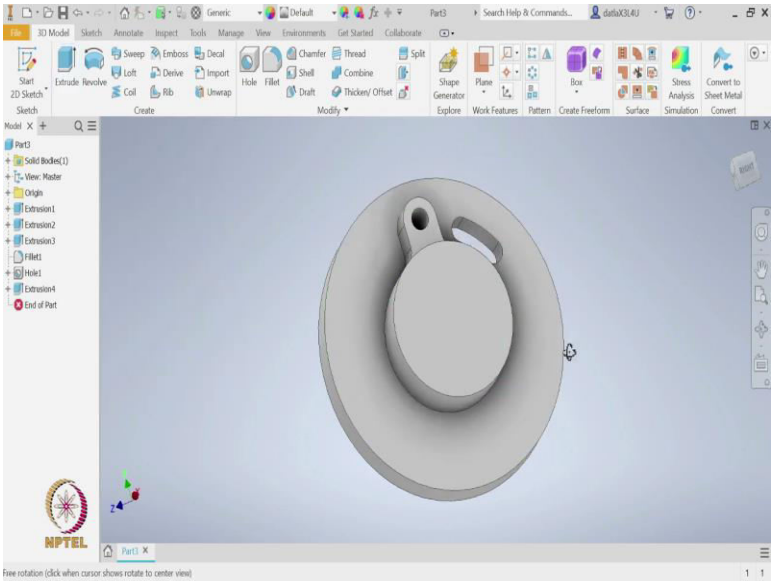

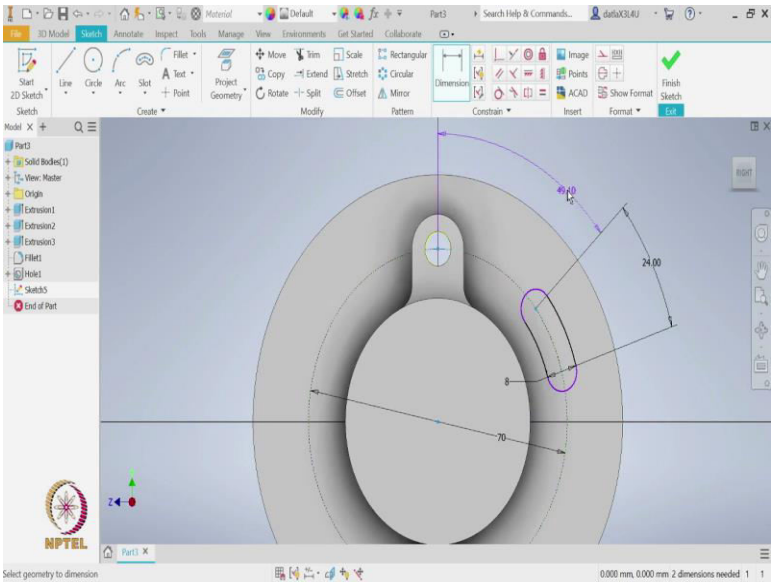

Now we are ready to do the curved slot using three points. So this is the Three Point Arc Slot. For this I need to select; so the first point I need to make sure it is falling on the construction circle. The second line also should fall on the circle. So the moment you move the cursor close to the construction circle it changes color from dashed black line to solid blue line. So these are the points 1 and 2. And the third point should be in between this first point and the second point. Pay attention, because if this is selected randomly, again you will not get as the intended slot.

Now, we need to specify what is the width of it or the radius of the circular arc. Let us look at the problem where it is mentioned. Here it is mentioned that it is a radius of 4. So let us dimension it accordingly. So we need to specify 8 here and click, press Escape or Enter.

So the first thing I need to do is to make sure that this arc is properly dimensioned and it is in a proper location. First let us see how do we mention the length of this slot? That is defined by this angle 24.

So let us give a dimension of this angle 24. For this you go to the Dimension tool and select Three Points. So until now whenever we said linear dimensions we selected Two Points but when you select Three Points it will give you the angle between those three points.

So let us select the first center of the slot, then the origin, and then the second center of the slot. Then I can specify that this angle is 24. As you can see this is at an angle of 45. This slot is at an angle of 45. How do I specify that? The center of the slot is an angle of 45 but the total angle of the slot is 24. So what angle will it make with this line? So let us say between, and this center. So that will make an angle of 45 - 24 by 2 which is 33.

So now let us see how to give this angle of 33. So before doing that we need to first create a point on this vertical line. How do we create that? For that let us use the project geometry. So we will project this circle. The moment I do that you will notice that it has created a center point at this circle on the sketch plane. So that I can use it to dimension using the three points. So if I go to the dimension, first I select this center point and then the Origin and then the first center of the slot. Now I can specify that this angle is 33.

By this I am able to ensure that the location of the slot is at an angle of 45 which is given in the question. Fine. What do I do with this? I finish the sketch; make a cut using the Extrude, extrusion. So I need to specify it is a cut in a Boolean space and then it is a Through All cut.

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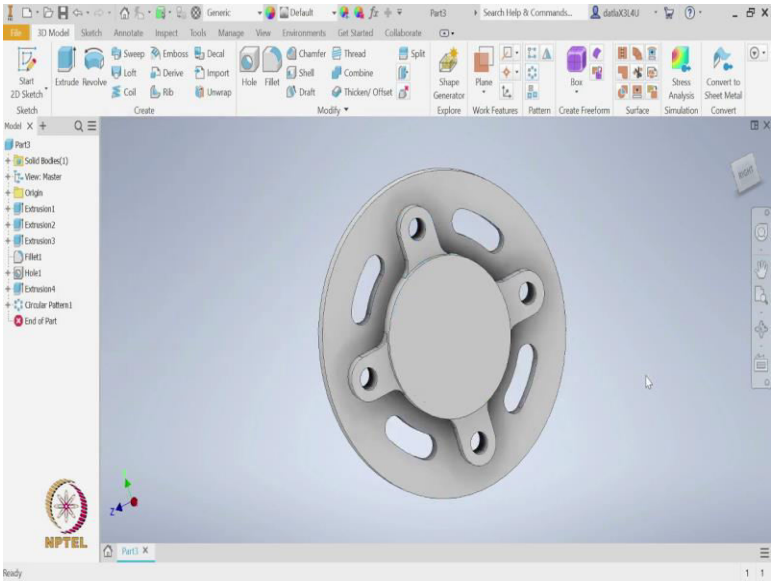

So now let us go back and see if we are done. So what we see is this small hole is remaining. But that we will do last because it is in a different plane altogether. But now let use the Circular Pattern and see that we have four duplicates to be created. So that is what we will try.

So let us select this Circular Pattern and then let us start selecting the features. So we will start with this Extrusion 3, Fillets, Hole and this Extrusion Cut. And now let us select the rotation axis. For the rotation axis we can go to the Origin and select this X axis. Or I can also select this circular face because that matches with the X axis. I need to specify that I need only four duplicates in the total angle of 360 and click Ok, fine.

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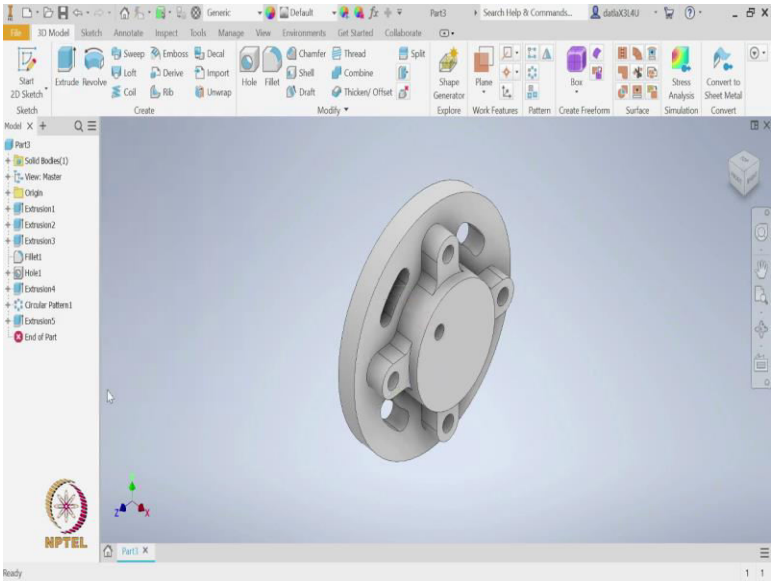

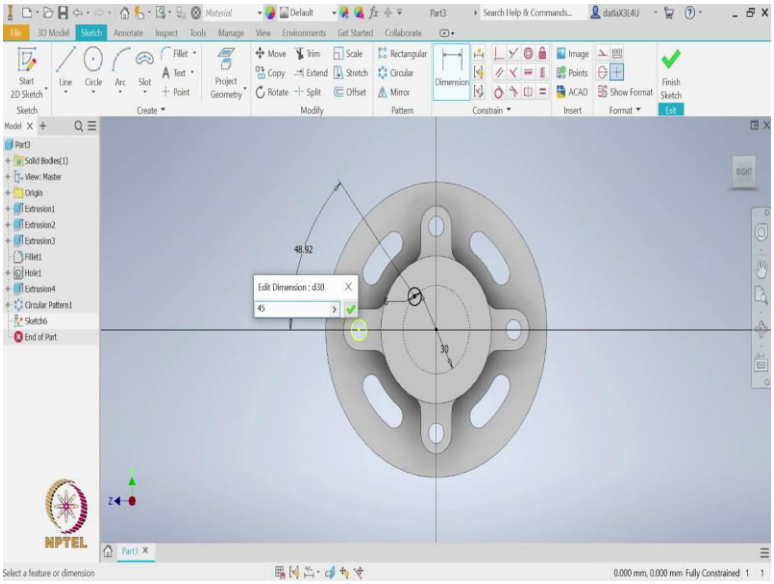

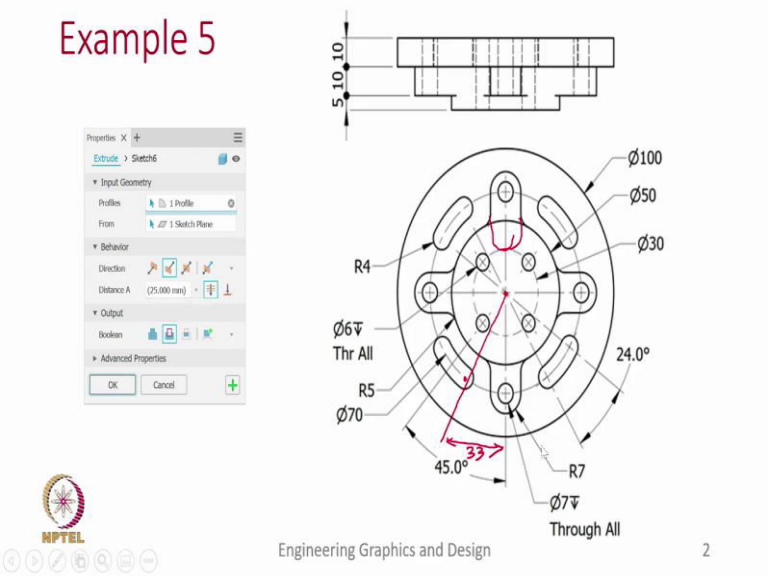

Lastly what is remaining is to create these four holes. Again I need to make sure that these holes are located at an angle of 45. So let us see how we do that. So let us create a sketch on the top face since we noticed that these small holes are on a pitch circle diameter of diameter 30. First let us draw the construction circle so that then we can specify the center of this hole on the construction circle.

So to do this first we select the Construction feature and then draw this circle at a diameter of 30 from Origin. I need to remember to uncheck this Construction so that now when I create a, use the, again the Circle feature to draw a hole I need to make sure the center of the hole falls on the construction circle so that the distance from the center is maintained as 15. And this diameter we have seen that it is diameter 6 Through hole. So let us specify 6, Enter.

Now we need to specify that it is an angle of 45. So the location needs to be precise. For that let us use this Dimension. Select the center of the hole, Origin and, we noticed that we do not have a line. So first let us use the Project Geometry to create a point. So now let us again go back to the Dimension. Let us specify the angle using the Three Point method, first on the center of the hole, Origin and this point we have on the horizontal axis. So this angle we said is 45.

Let us complete the sketch and make a cut using the Extrusion tool. So let us select the feature, specify that it is a Boolean cut and also mention that it is a Through All, because we see here it is mentioned it is, it is a diameter 6 hole which is Through All. So that is what we selected. And click Ok.

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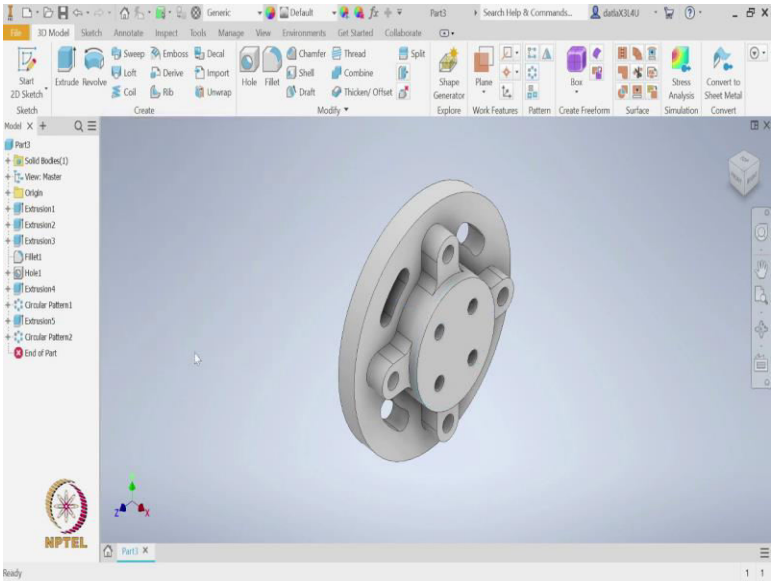

Now we need to again make a circular pattern. Select this Extrusion and now specify the axis. We will specify that it is, the axis is the axis of the circular surface. I need to ensure that there are only four duplicates and click Ok.

So with this we are able to complete the problem. So this makes sure that we have all the features which are given in the question, which are these small circular holes, the slots, and this protrusion with fillets and a central hole. With this we conclude today's lecture. Thank you for your attention.