Engineering Graphics and Design
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Week 9: Part Modelling I
Revolve, Loft, Pattern

Welcome back to week 9. In this week we will be discussing about Part Modelling. In previous week we started by introducing what CAD modelling is and then we have also introduced the Autodesk Inventor Professional software. In the previous week we also started with some simple basic features of the solid modeling. In this week we will be focusing about few more features of this Part Modelling which we can use to create complex object. Essentially we will be looking at Revolve, Loft and Pattern commands or the tools in this week. So let us get started with this lecture with Revolve tool.

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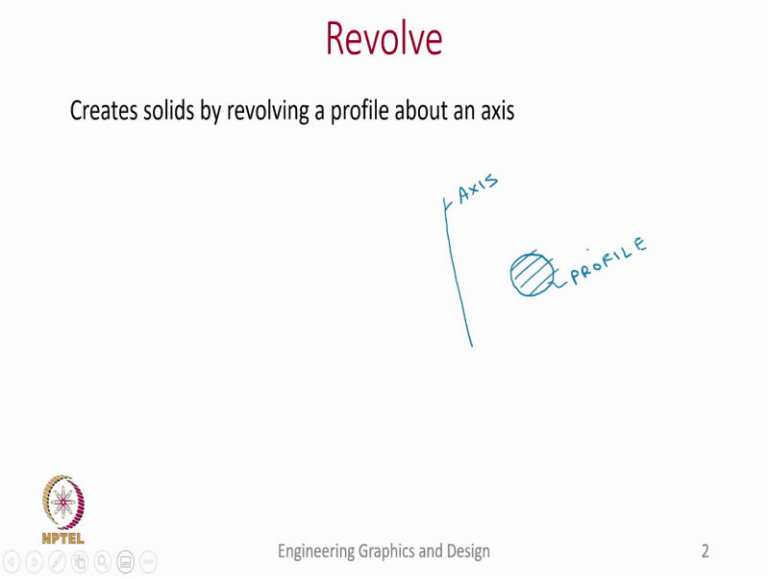

So what does Revolve tool does? It creates solids by revolving a profile about an axis. So let us say we create a circle. And say that since this is in closed area we will call this as a profile. To create a solid we need to rotate this profile about an axis. So once we rotate this profile about an axis we create a solid. So depending on the angle by which we rotate, if we rotate it by 360 or a Fill Rotation, you get shape called as the torus shape. We can also rotate it by a specified angle such that we only get a part of the complete solid.

So depending on the angle by which we rotate, if we rotate it by 360 or a Fill Rotation, you get shape called as the torus shape. We can also rotate it by a specified angle such that we only get a part of the complete solid. So all these features are available in this Revolve tool which we will look with the software now.

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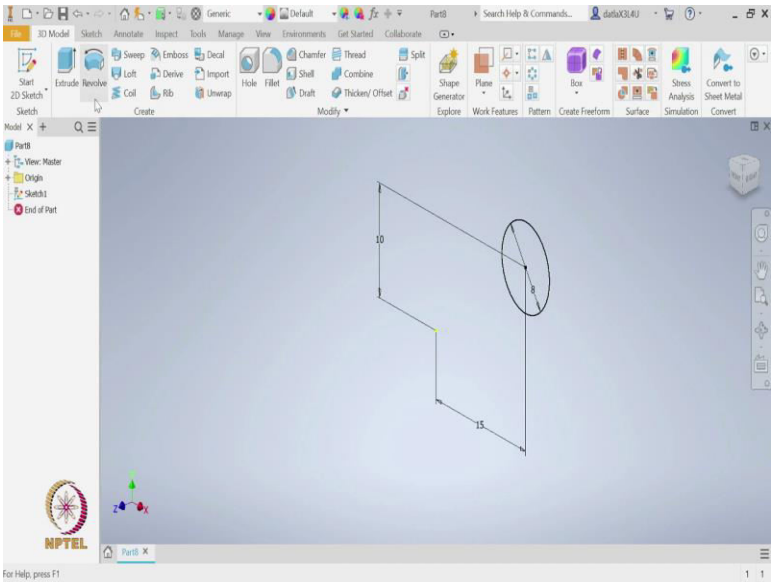

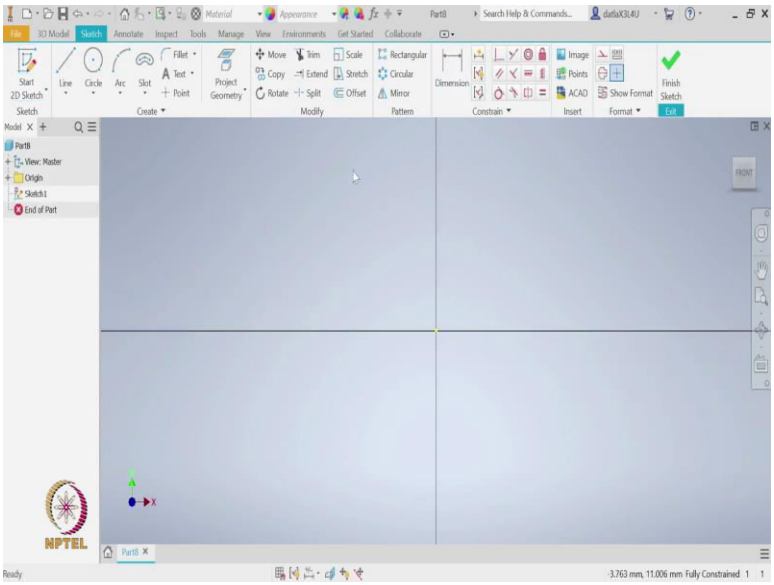

Let us start and choose the standard mm ipt template. Let us do this Start 2D Sketch and create a, select a plane. First let us create a profile. The simple thing is to create a circle with a center and a given radius, let us say of 8. So let us fully constrain it by specifying what is the location from the Origin. So let us dimension it. Center of the circle to the Origin, let us say is 15, and again

the center of the Origin to the center of the circle in the vertical dimension, let us say, 10. So now it is fully constrained. Let us finish the sketch and then choose the Revolve command.

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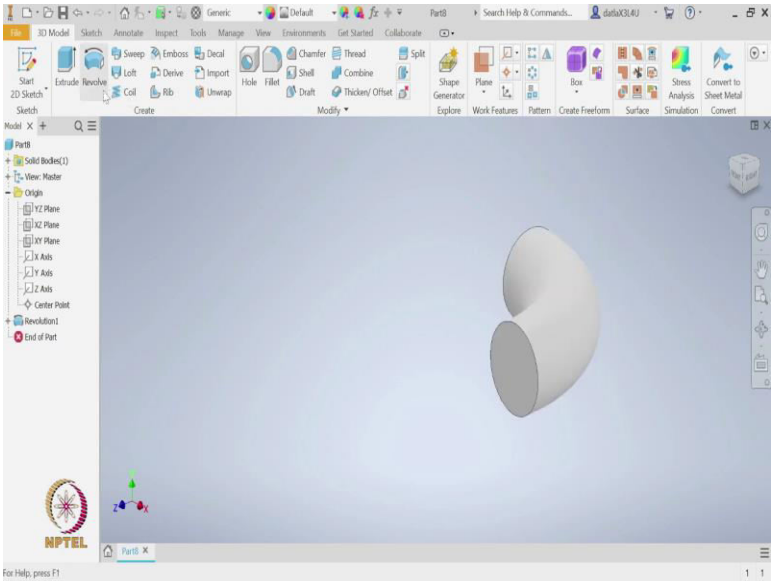

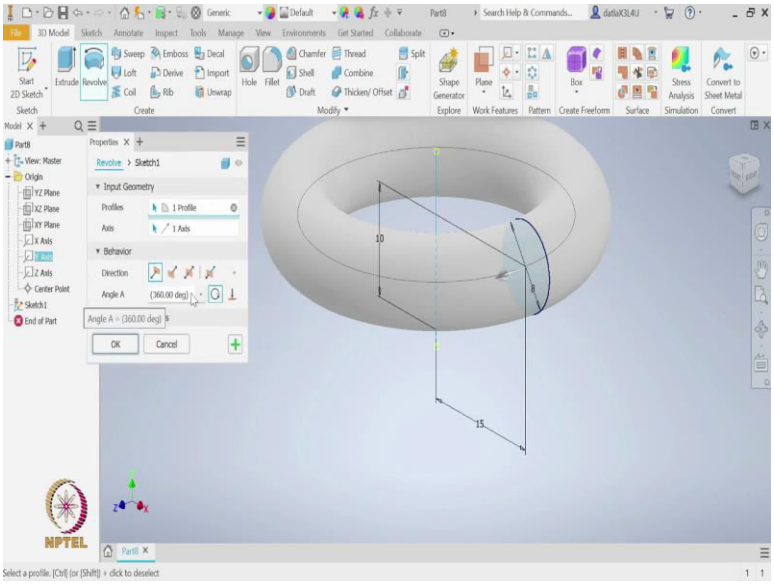

So when we go to the Revolve command it automatically choose the profile. Now what it is asking is the axis. So since the axis is not here we can go to the origin and choose which axis we want. We can either choose the X axis or the Y axis. So let me choose the Y axis. The moment I choose this it has created a solid. The default options it is taking are, that the angle is 360. But now let us say we only want half of it, so 180 degrees. So as you can see it has started from the profile in the left direction and rotated by 180 degree.

Of course I can change the angle to any other angle, let's say 90. But now let us say, instead of going to the left you want to rotate in the other direction. So here we can click the Direction. So as you can see I can toggle between any of these directions. Or of course we can go in both directions by equal amount or in both directions by different angles. So in one direction it is 90 the other direction it is 45. But let us stick to this single direction of 90 degrees. So this is how we create a solid using the Revolve command.

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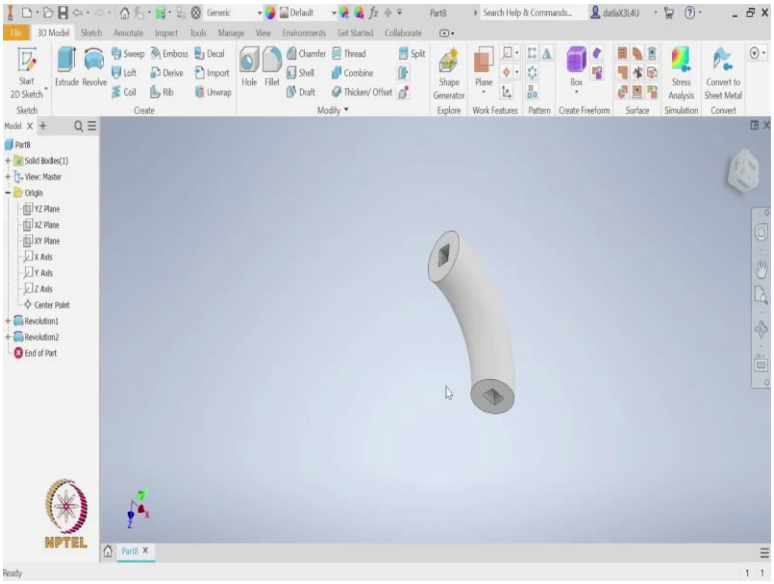

So in addition to creating solids we can also create cavity. Now let us look at how to do that. So first, again we will create a sketch on this face here. Let us say we will now select a rectangle. So I can finish the sketch. I know that this is in blue color but just for practice I need not make it fully constrained. I can still go ahead and create solids using this sketch.

So let me finish the sketch and then again use the Revolve command. In this dialog box we see there are Boolean operations. By default it is selecting the Join. But we can also ask it to do cut. So once I select the cut it is asking the axis about which I want to make the cut. So we will again select the Y axis. By default it has picked A direction and is doing a cut.

But what you can do is we have two options here. It will either do a Full Cut which means it is doing the 360 degree rotation. Or you can select a To option where you can specify it to cut until it reaches a particular face. Now let us pick the other face of it. And ask it to do the cut. So we

see that it is confused because the direction is opposite. So instead of making a cut by going from this face within the material, it is going out.

So now let us flip the direction. So once I flip the direction I already specify it should make a cut and till it reaches the second face. So now we can see it started from the Face 1 and made the cut all the way till the Face 2 using this Revolve command. So now let us click Ok and finish the cut. So now we see that we can use this Revolve cut tool to both create as well as create objects, new objects or solids as well as make cuts.

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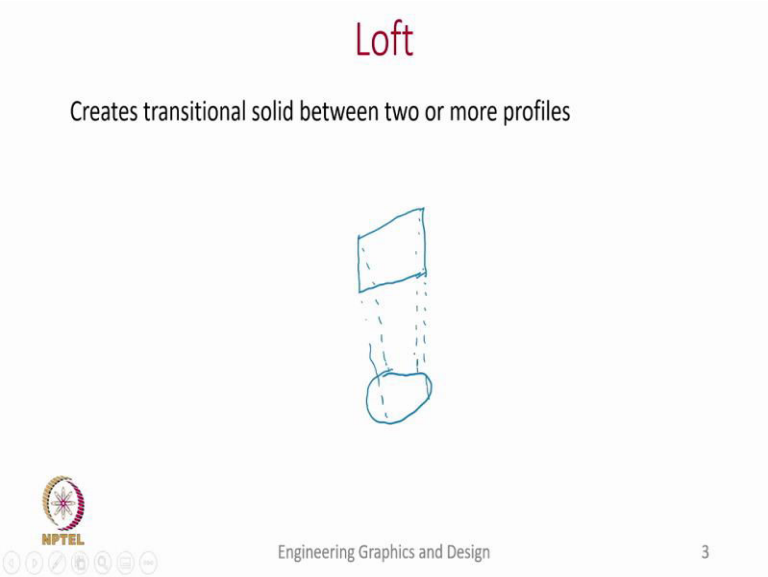

Now let us move into the second tool we will be discussing today which is the Loft. What does Loft do? It creates transitional solid between two or more profiles. Let us you have a profile of a circle, and then at the top you have a profile of a square or a rectangle. Now you can make a solid by connecting these two features. So let us say it will connect something like this and make a solid. It will be much clear once we move into the software.

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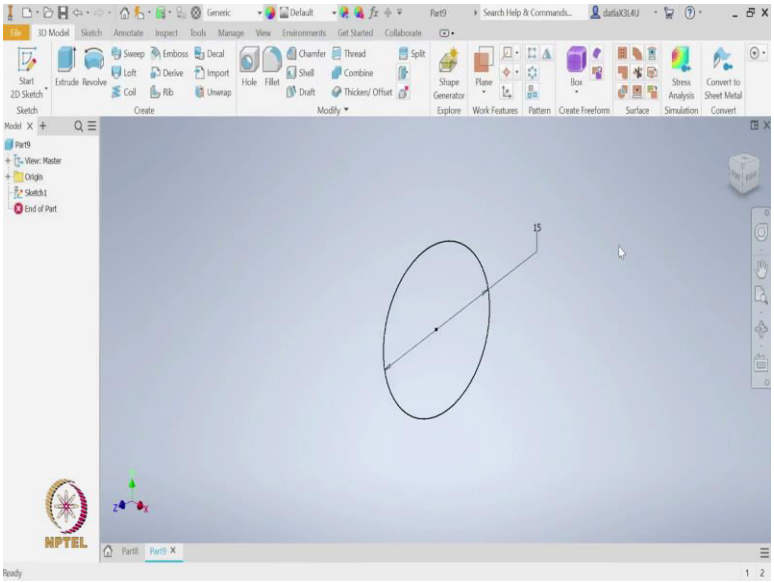

So let us again start with a new template and start to create sketch. So let us say we will first create a Profile 1 which is simply a circle. Of course I can dimension to fully constraint, otherwise also for the purpose of the example it is fine. But let me fully constraint and finish the sketch.

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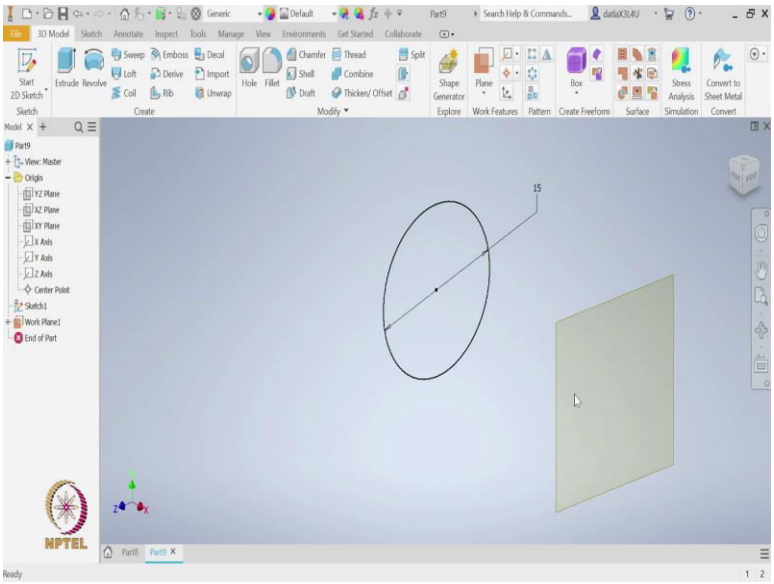

Now for me to create a second profile let me create a plane so that I can sketch on that plane. So I will use the Offset from Plane command. And here since I initially created in the YZ plane. I will pick that YZ plane and then pick a distance of, lets say 30. So what I am doing? I am now

creating a new plane which is at an offset of 30 mm. So the reason why we are creating this offset plane is, for me to draw a sketch I need to specify the plane. For me to specify the plane I have to create one more additional plane.

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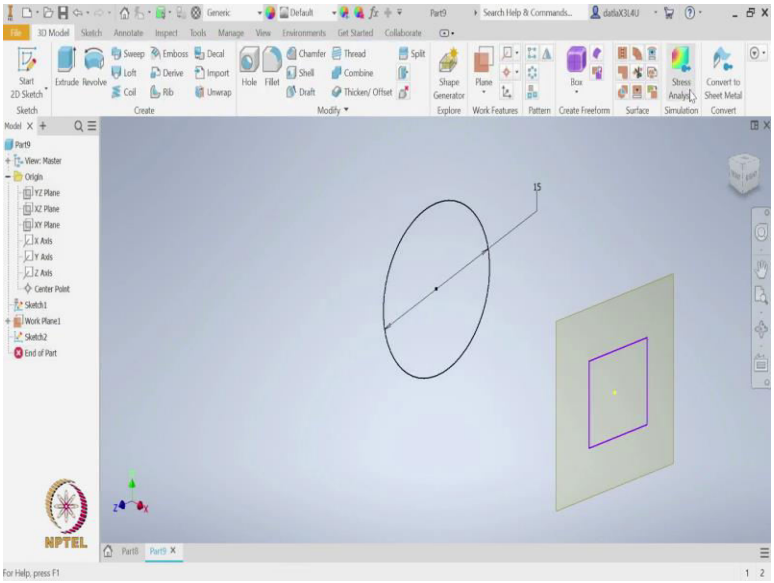

Now let us start creating the Sketch on this plane. This time let us do a rectangle, let us say like this, and this time I will not fully constrained but of course you can always do the fully constrain. And let us finish the sketch.

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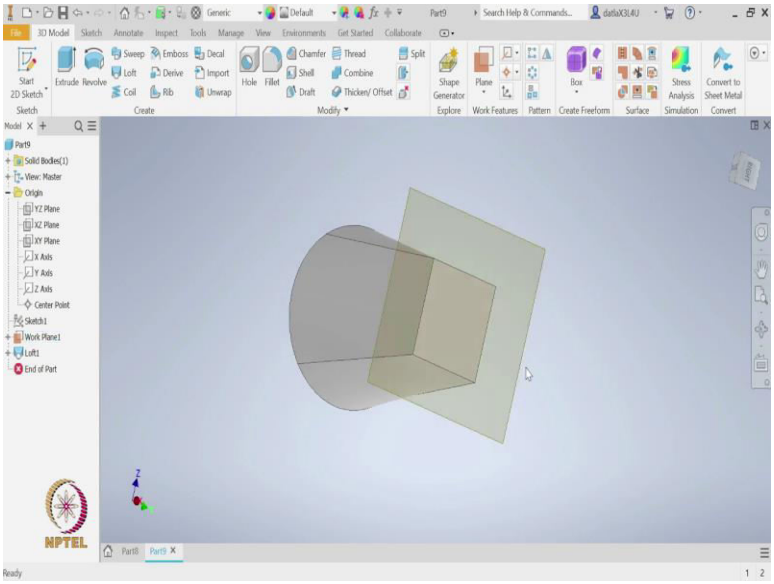

Now what we have done is we have created Sketch 1 which is this circle, and Sketch 2 which is this rectangle. Now we can start using this Loft command. So once I go to this Loft command which is in the Create panel it first asks me to select the sketches. So let me select Sketch 1 and then Sketch 2. So now it is showing me a preview of the solid. It is creating the transitional solid between the Profile 1 to Profile 2. So let us click Ok and see how it looks like. So now we can see that it has created a solid between the 2 profiles we have selected.

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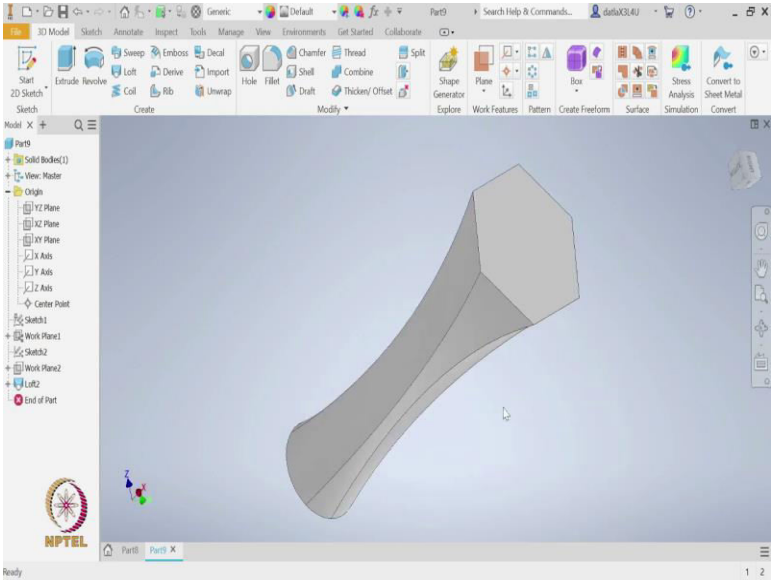

So we can use the same Loft command not just for two profiles but for multiple profiles. So it can be 3 or more. So quickly let us see if we create one more profile how things change. So we will again create one more plane which is an offset to this second plane. And let us say the distance is something like 40. Now let us create a sketch on this new plane and this time maybe another polygon. So currently it is saying a six-sided polygon which is fine with us. Done. Let us finish the sketch and see how this new profile sketch we have created.

We will first delete the initial Loft. So right click and Delete. It asks me whether to also delete the consumed sketches. I will uncheck it because we want to retain those sketches. And now let us restart with the Loft. So when I select the Loft tool it asks to select the sketches. So I will do Sketch 1, Sketch 2 and Sketch 3. So here is the solid which it is creating. So if I click Ok, now you can see. Let me turn the visibility of the planes to uncheck it so that it will be even more clear. So this is the solid we have created now using the Loft command.

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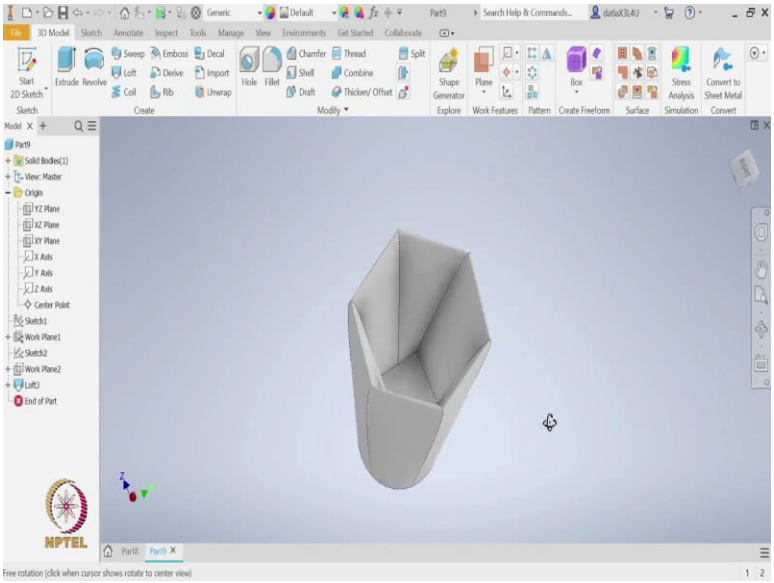

I need to stress that when you select this Loft command it is very important that the sequence of this Sketch 1, Sketch 2 and Sketch 3 is important. So previously we started with Sketch 1, Sketch 2, Sketch 3. Now let us see what happens if we use a different sequence. So let me delete this Loft. I will retain the sketches.

So we will restart with the Loft and this time we will start with Sketch 1, then let us say Sketch 3 and lastly this intermediate Sketch 2. If I select Ok you will see that the kind of solid it has created is very different. So from the bottom sketch to the top sketch it created a solid. But from the top sketch to the intermediate sketch it created a cavity, because to maintain the sequence we have enforced it to select.

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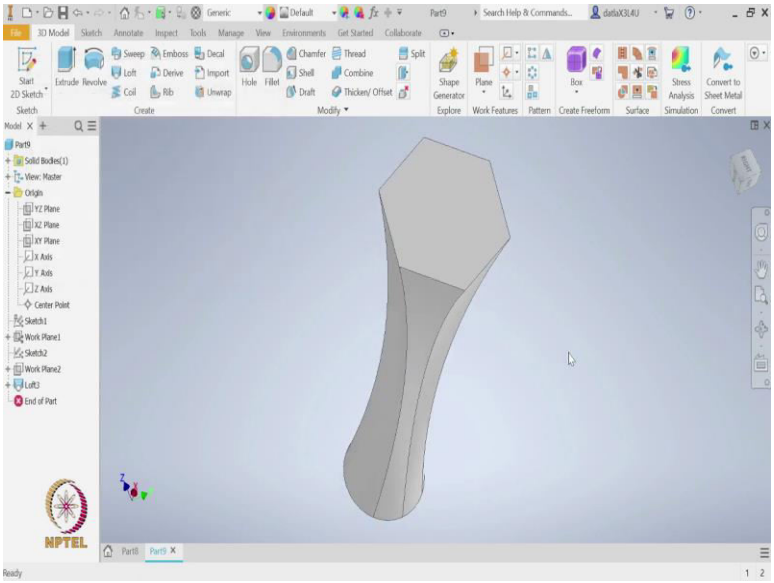

So if this is exactly what you need it is Ok. But let us say if this is not what you want to do but you want to select them in a proper sequence then you can go to the Loft command, and then go to this edit feature and then drag this sketch to up. So the moment I drag it has corrected it by saying that the sequence is Sketch 1, Sketch 2 and then Sketch 3. So I think now we are clear about how to use the Loft command.

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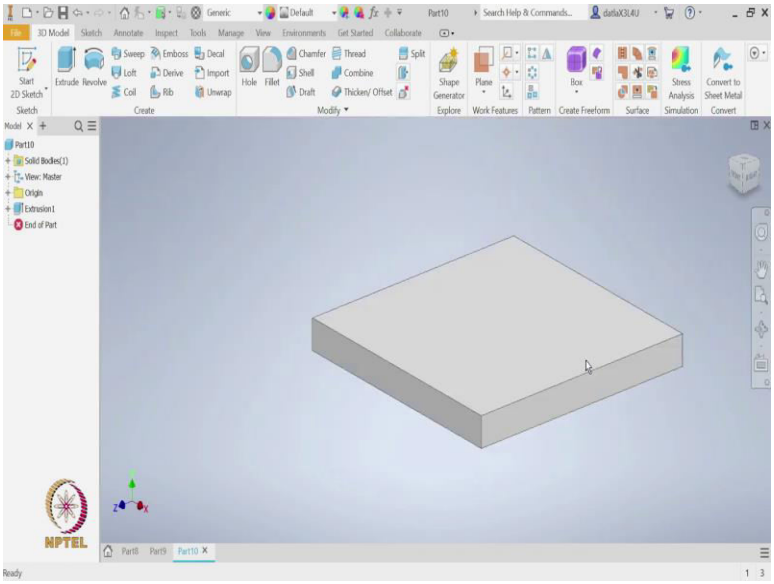

Now that we have got a good idea about the Loft command let us now move on into the Pattern command. So if you look at the Ribbon there is a Pattern panel here. In the Pattern panel there

are different options we can do patterning. What is patterning? Patterning is if you want to create the same copy of a feature multiple times, then instead of laboriously doing one at a time, let us say if it is 10 or 100 times, instead of doing it multiple times we can create them using this Pattern command in one go.

So, as you can see there are multiple options. First is the Rectangular Pattern. It creates duplicate features of solids and arranges them in rows and columns. So, second is the Circular Pattern where again it creates duplicate features, solids or bodies and arranges them on an Arc or Circular Pattern. And lastly we have this Sketch Pattern. We will discuss about this where you can specify what is the location of each of these duplicate locations.

So, now let us again start with a new template. So I will create a basic feature. We will go to the Start 2D sketch, select a sketch plane and then, let us say I will create a rectangle. I will start from the Origin, and let us say these dimensions are 100 by 100. Select all. Now let me finish the sketch. And extrude it for a height of 10.

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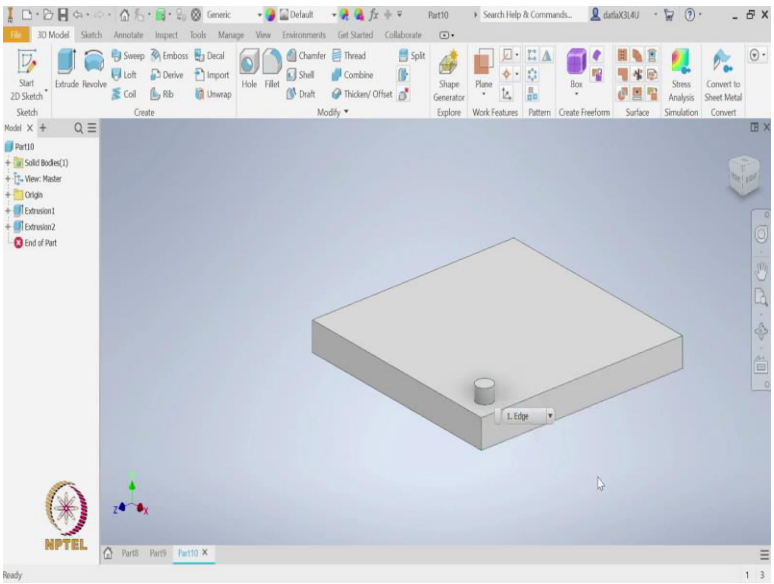

On the top feature I will create, so again I will create few buttons. So let me start with one button by going to 2D Sketch, selecting the top face and here, let us say we will again take a circle, select a center and specify the radius to be, let us say 8. Now also let us dimension it such that the center of the circle from the Origin is 10 in both directions; in the horizontal it is 10, vertical as well let us make it 10. So let us finish the sketch and extrude this portion. Let us give the

height half of the initial height, so 5. So let us say this is the feature we have just created and we need to make multiple copies of this.

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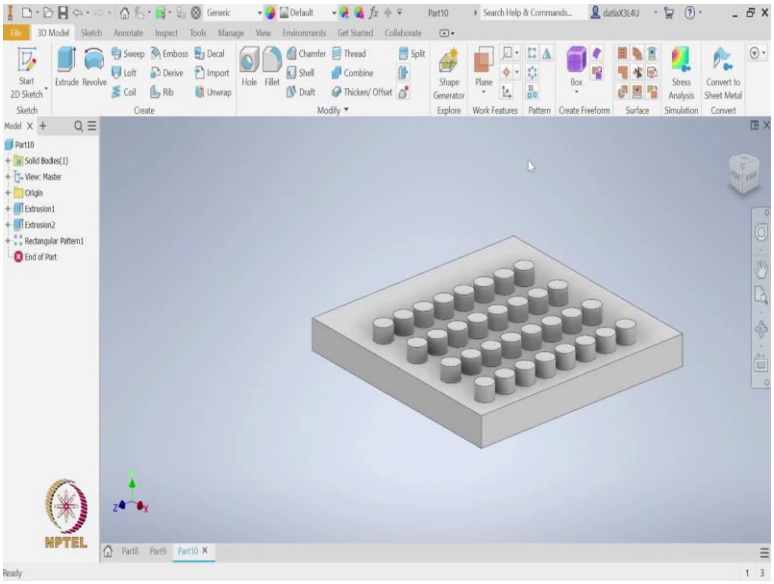

For this we can use this Pattern command. Let us start with the Rectangular Pattern. So the moment I select it, by default it is asking me to select the feature. So let us select this button we have just created. And now it is asking what are the Direction 1 and Direction 2. So it is essentially asking you to what is the direction of the rows and what is the direction of the columns.

So, let us start with Direction 1 and say it is, we can specify an edge for the direction. So the moment I select in the edge it is picking some default values like it will do 2 duplicates at a spacing of 10. But now let us say this is 8 duplicates we want. So currently what it is doing is it is mentioning the spacing to be 10 which means the space between each of these features is 10.

But there are other options. We can say the distance between the first and last can be any number. As of now it is taking as 10 but let us say it is 80. So what it does is the distance between the first feature and the last feature, that distance it is taking as 80. We can also specify the curve length. So this patterning we can also do it on a curve then we can specify what is the length of the curve. But let us pick with spacing and say that the distance between the two features is 10. I can click Ok. And then we get these multiple buttons, eight of them in this direction.

But now, let us say we can again go back to the Rectangle Pattern feature and Edit Feature. So previous we have only selected one direction. But now let us select the second direction as well. If I select the second direction I just need to mention that direction, which I can mention using an edge by selecting an edge like this here.

So by default, as you can see, it is going in this direction which moves the features out of the base plate. I can flip that direction using this button. So here I can flip the direction, and then again I can specify the number of duplicates I need. Let us say it is 4. I can leave the spacing as 10 or maybe I can specify it as 20. And let us say Ok. So now you can see in direction 1 we have 8 features and in the direction 2 We have 4 features. So in total 8 by 4, 32 features we have created using one single feature which is this Rectangular Pattern.

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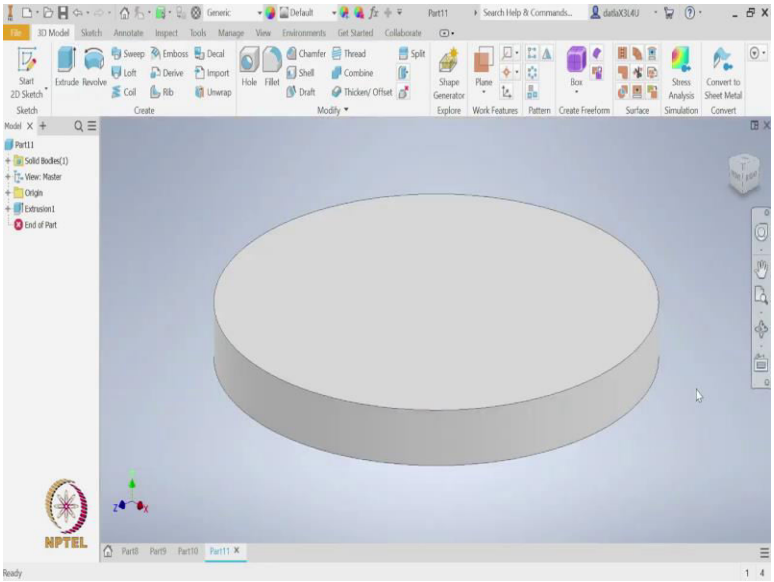

Now let us look at how to use the second option which is the Circular Pattern. For that let me again create one more new template. And let us start to create a sketch by selecting any of these planes. I will start with a circle, center as the Origin and the diameter as, let us say 100. So let us finish the sketch. Extrude for a height of 10. Let us say this is the plate we have created.

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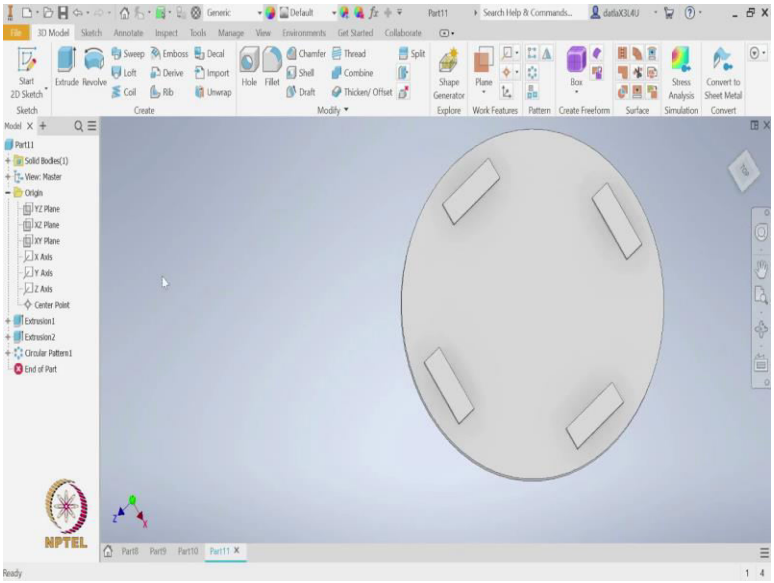

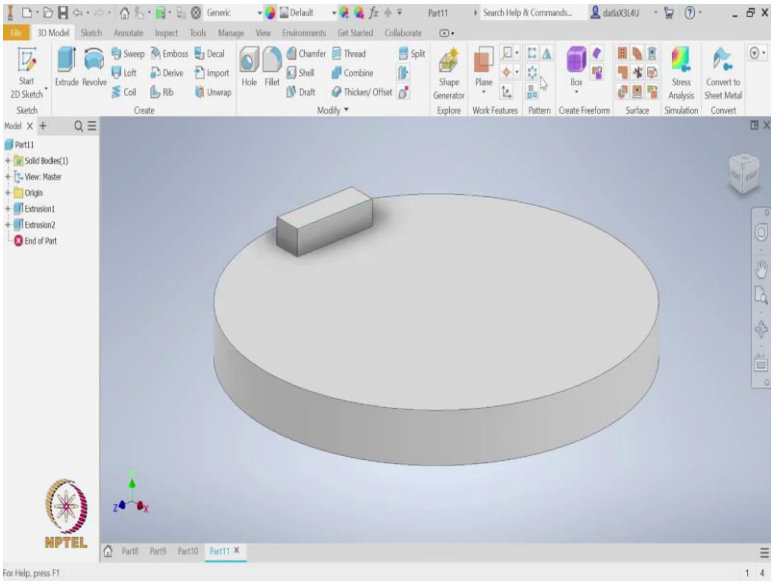

On this let us create a feature which we will duplicate, this time using the Circular Pattern command. So I will again use Create 2D sketch on the top face. This time I will pick a rectangle, let us say and have it like this. So again I can specify the dimensions to completely constrain it, but for the purpose of the example we can let us go ahead without completely constraining. So I will finish the sketch and then use the extrude feature with a height of 5. Click Ok. So we have created one feature on top of this base plate.

And now let us use this Circular Pattern to show what options we have when creating the Circular Pattern. So once we click it, it asks you to select the feature. You will select this

rectangular extrusion. And then let us, it ask you to select the rotational axis. So for the rotational axis, there are multiple ways you can select the rotational axis. You can go to the Origin and select.

In this case which is the rotational axis? Let us say Y axis. So once I select Y axis, by default it is asking how many rotations I want to do, so 6. And what is the angle? So I can change from 6 to 4 and the angle, let us say is 360 degrees. This is one way of selecting the rotational axis.

But, let us say there are other ways to select the rotational axis. So if I select this cylindrical feature of the base plate then it picks the axis of the cylindrical feature. So let me click this. Again it has picked the same Y axis because Y axis is the axis of rotation for this cylindrical surface. First let us click Ok to see how the solid looks like. So now we have created 4 duplicates.

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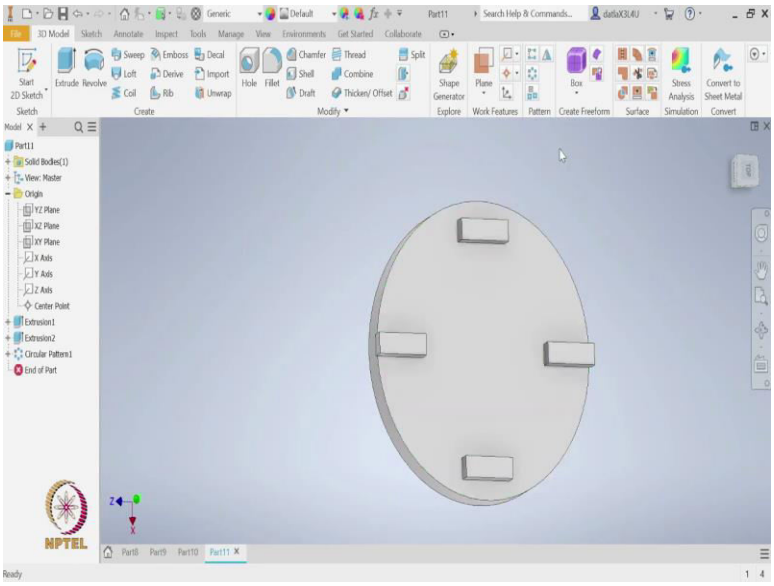

Let me now go to the Edit Feature of the Circular Pattern and see what happens if I change this orientation. So the Orientation has two options. One is Rotational and second is the Fixed. In the Rotational, essentially when this second duplicate comes, the first feature is rotated. The orientation changes.

But in the second option, if I say it is Fixed, so let us say if this rectangle, the length is along the X axis, the same X axis is will be maintained for all the duplicates. But, let us say if it is Rotation, so the orientation of the object keeps changing as we go to the four corners of this base plate.

So let us fix for the moment. We see that one of the parts is coming out. But let us look how it will create. So we see that there is an extrusion happening here, but it was able, the software was able to handle it. But essentially it is able to create the Circular Pattern using the Fixed orientation.

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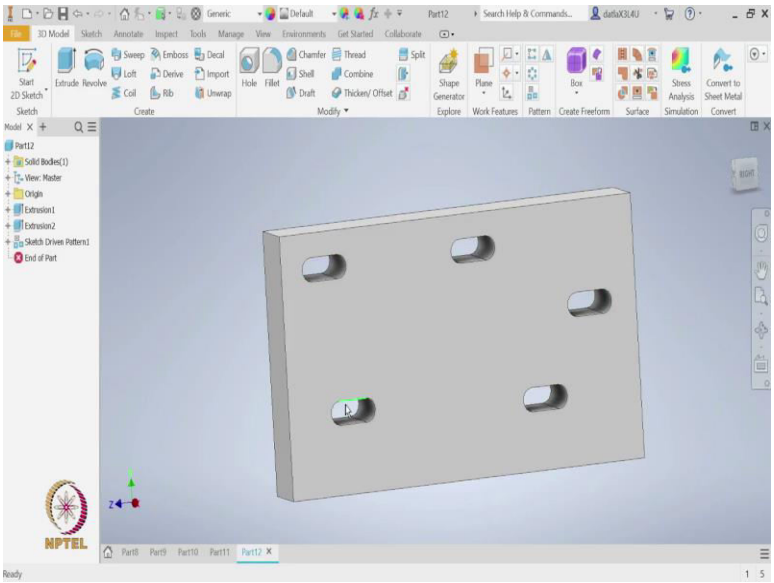

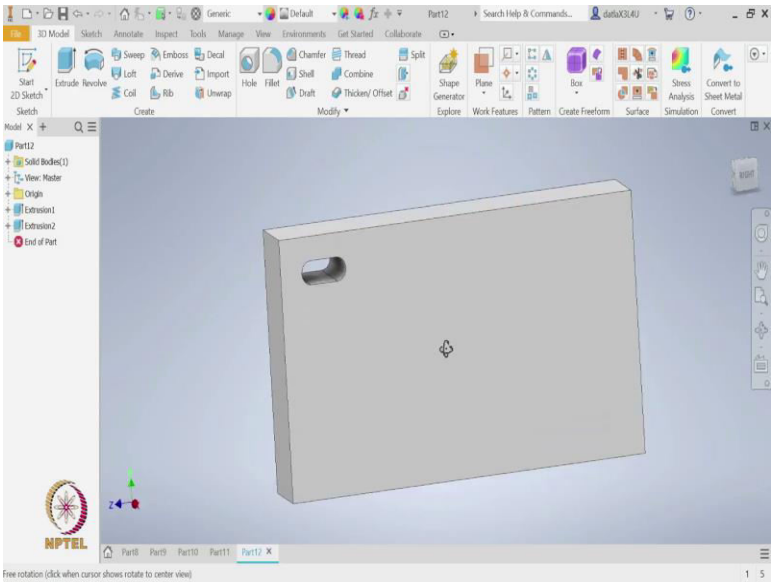

So after we covered the Rectangular Pattern and the Circular Pattern, now let us look at Sketch Driven. So again let us start with a new template. Let us create a sketch. Let us say, a rectangle, Finish Sketch, Extrude.

So now let us create a feature on the top face. It could be anything. So let us say, a slot. So let us say it is Finish Sketch. And then Extrude, I will select the Boolean operation Cut. So this is the feature 1 which we want to duplicate.

But this time when we want to duplicate, let us say I want to use this Sketch Driven which means the locations I have to specify it through control points. so for that what I will do is, first start a 2D sketch, let us say the top face. And on this top face I will create some points. So let me select this Point feature and randomly say I want one more slot here, here, here, and here. So let us say Finish Sketch.

This time when we go to this Sketch Driven pattern first it will ask us to select the feature. so let us select this feature. I can also select the feature from this browser. So this is the Extrusion feature where we made the cut. So after I selected, it has already selected the four points I have created by default. So if I click Ok it has duplicated this Cut feature at the specified four points I have given.

Like this we can use this Sketch Driven pattern where we can specify, maybe it is not in rows and columns or rotation, but here we can specify four different points where we want to replicate or duplicate this feature and we can use this option. So in this Pattern panel there is also Mirror but that we have covered it in the previous lecture.

So to summarize, what we have discussed in this lecture is we have discussed three different tools of the software. We started with the Revolve tool to create solids of revolution. And then we also discussed about the Loft command where we can take two or three profiles to create a transitional solid.

And lastly we discussed about the different ways we can do patterning where it duplicates a feature either a rectangular pattern, circular pattern or a sketch-based pattern. With this we will end this lecture. In the next lecture we will be solving few examples where we use these tools that we have discussed in the lecture.