Engineering Graphics and Design
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Week 8
Solid Models with Sketching

Welcome back. We are in week 8 of Intro to CAD. In the last two lectures we have discussed about introduction to the CAD in general and later we also looked at the Autodesk Inventor Professional, how that environment looks like and how we can work with it. In today's lecture we will start by learning how to create simple solids in this AutoCAD Autodesk Inventor.

We will start by learning a very common feature called as sketching which is the basis of commonly used method to create the solid models. As we previously discussed Autodesk Inventor is parametric feature-based software. So what does feature-based mean? We keep creating features which are the basic elements of the building blocks to develop your complex solid models.

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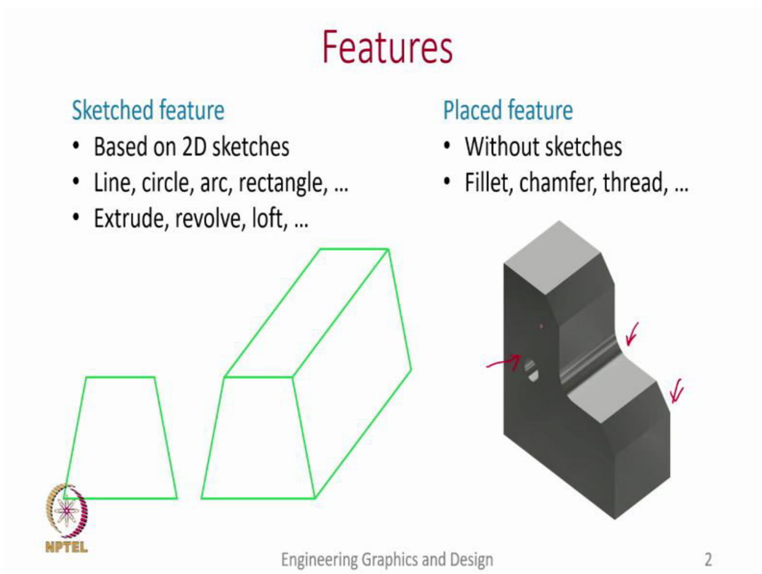

So there are two ways we develop features. The first is the sketched feature. So what you see here is a solid and to create the solid you first notice that, in one dimension the cross section remains constant. So we can start by creating a 2D sketch of that cross section like this. And then once we create this 2D sketch by using line, circles, arcs, rectangle. And there are many other features the software provides.

Once we are done by creating the sketch, what we create is a profile. So then we can use that profile to create a feature such as Extrude. The example we show here is an Extrude where we create a 2D feature and say the cross section is constant. So Extrude will develop a solid with the same cross section for a specified depth. There are other features which we can also do using sketch features such as the Revolve, Loft and we will be discussing a few more in later lectures.

Now let us look at the second method of creating features which is called the placed features. So these are features which can be created without starting with sketches. So these are examples like fillet. So in the previous lecture 2 we have shown that this is a fillet, a chamfer. To create this fillet and chamfer you don't need to start with a sketch. You can start with the existing feature.

For example here let us say, we have created a 2D section and then created an extruded solid. From that extruded solid we can directly create features such as the fillet and chamfer. And we can also create features such as the hole which is shown here as well as if we notice that there is an internal threading to the hole even that can be added as a feature to the existing solid.

So this is how we gradually start with a simple solid and then we can go to complex solids by adding few more features to it so that now it now becomes closer to a realistic object or a real world object.

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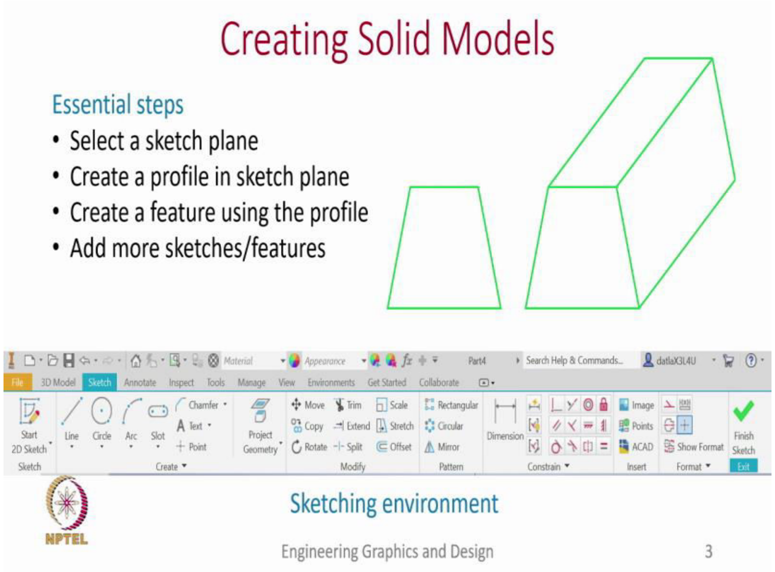

So now let us look at what are the essential steps in creating a solid. So first, as we noticed, one way of creating solids is through sketches. So let us discuss that. So before we start a sketch we need to select the plane on which we want to create this sketch. So once you select a plane then you get started by creating a profile in the sketch plane.

So for Inventor we use this Sketching Environment which I will show in couple of minutes. First we will go through a few basics in couple of minutes and then move on to the software where we will show you each and every tool available in this Sketching Environment. But essentially what we do is to create a profile.

What is a profile? A profile is a closed sketched entity. So for example a trapezium shown here is a closed entity, because once it is closed then you can represent it as an area. So once you have an area and you can use a tool like Extrude to create a solid. So we say that we create a profile in the sketched plane and then create a feature using the profile. So once we have this 2D entity called the profile we use that and use a feature tool such as Extrude to create a solid like this.

But what we have created is a simple solid. We can create more complex solids by adding more features to it. So when we start adding features, these can be sketched-based features or the placed features. So it can be a hole, a rib, a chamfer or a fillet. So if we gradually keep adding these features then it becomes more closer to the object we are designing.

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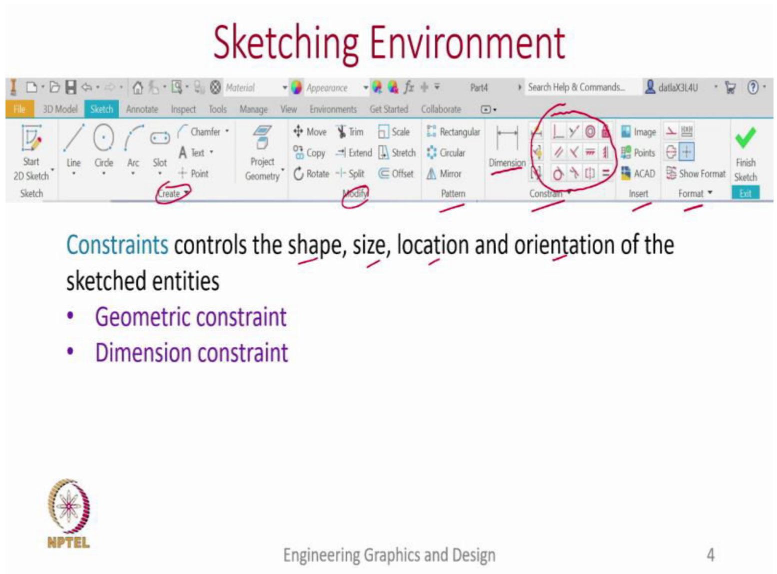

So now let us spend a couple of minutes to understand this Sketching Environment. So in the Sketching Environment you see there are multiple panels. So to start with, a Create panel, Modify, Pattern, Constrain, Insert and Format.

So we initially start with this Create panel which has several tools to help you create a 2D entity like lines, circle, arc, rectangle and we will later see actually there are lot more features available within this Create. So then you create a 2D shape which can be modified again using other commands like Move, Trim, Extend and so on and so forth. So let us say, we have created a 2D profile using these Create and Modify features.

Then later we need to constraint the object. You might be wondering why should we constrain. So just to review, the degrees of freedom, let us say, a pen here, right, it has six degrees of freedom. It has three degrees of freedom to move in the X direction Y direction or the Z direction as well as three more degrees of freedom; rotation about X axis, rotation about Y axis and rotation about Z axis.

So it has three translational degrees of freedom and three rotational degrees of freedom. So which means once you create let us say, a trapezium as in the previous slide, that trapezium is not fixed in space, because we never said it is constrained to move in the X, Y or Z direction. Z direction is constrained because at least we said this trapezium is lying on a plane.

But then again, once we have created the plane it can also, it is also free to rotate in that plane. So we need to constrain this. Then only we will have a profile which is fixed in space instead of it having a degree of freedom to move in X, Y or rotate about the Z axis. So for that we need to apply these constraints.

There are two different kinds of constraints. So before that let us see that what does constraint do? They control the shape, size, location and orientation of the sketched entity. This is very much required in these CAD softwares because we said we are using a parametric based software, which means today we will be designing a component, but later we will use this component in an assembly.

During, at that stage if you want to make changes to the component you can either make changes coming back to the component level or even in the assembly itself or at a drawing itself you can make changes to the component. And when we make those changes we want to ensure that these changes have occurred in a particular pattern which is preferable. To do that we need to constrain the object in the Sketching Environment.

So there are two kinds of constraints. One is the geometric constraint and second is the dimension constraint. So you look here all these fall under geometry constraint, and here is the dimension constraint. So with one tool we can do all different kinds of dimensions, but for constraints there are several different kinds which we will learn once we move into the software.

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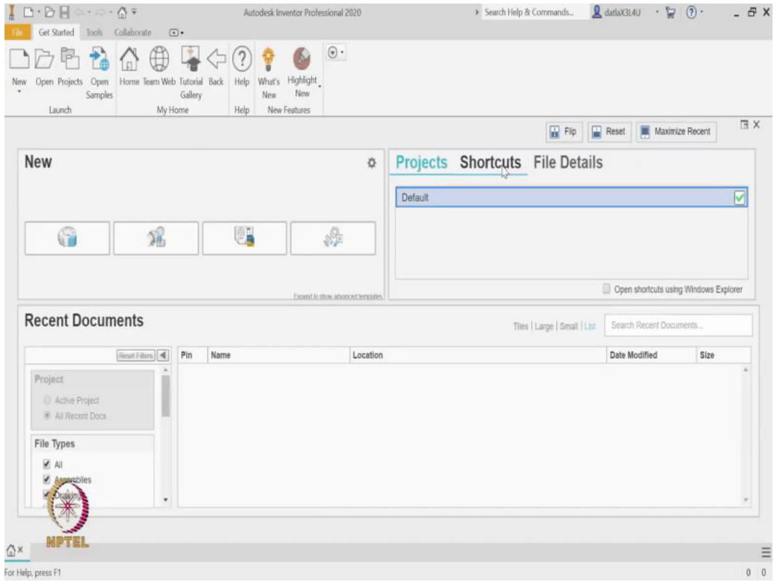

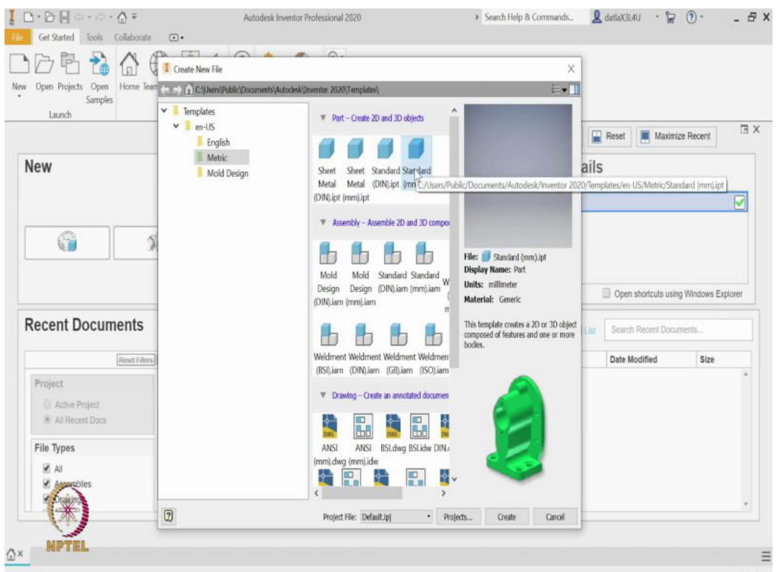

So I think, so let us now move into the software and see how this environment looks like and how we can use this environment to create simple objects. Now we are in the front panel of the Autodesk Inventor. So like what we discussed in the previous lecture we start by creating a new template. So we will be picking a part template with standard mm dot ipt file.

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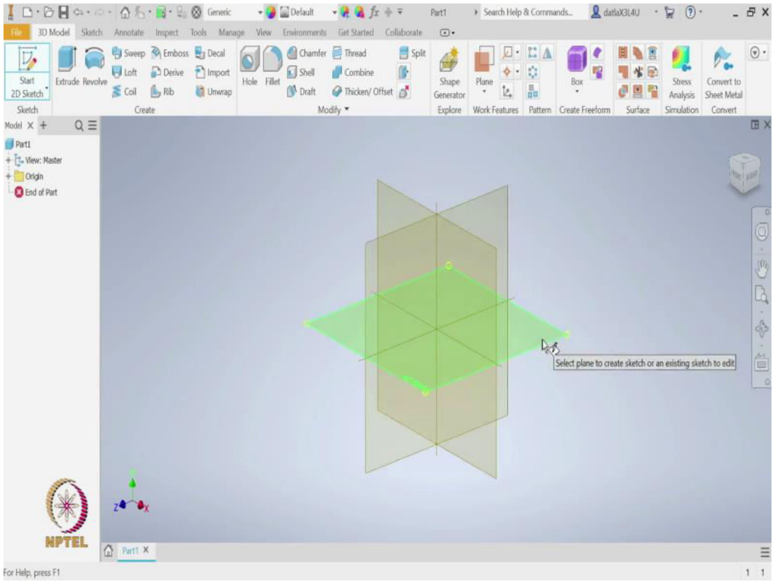

So for us we will define a coordinate system. The center will be the origin of this xyz co-ordinate system and then it shows three profiles, the standard profiles, the XY, YZ and the ZX planes. We can choose any of these. We can either choose this by moving the cursor to any of these. As you can see now I am highlighting the XY plane. Similarly I can move other things like YZ and XZ planes.

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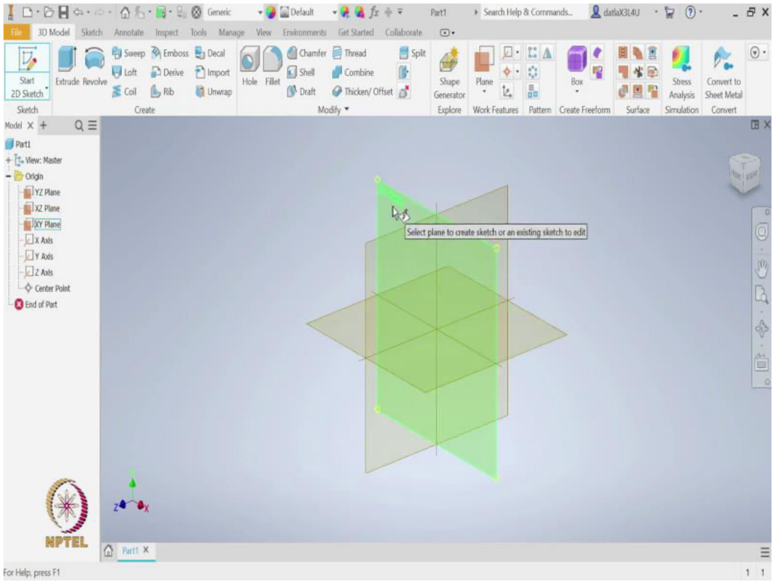

The other way of selecting this is you can use this Part Browser, and once you go into the origin here, the plus button on origin it will show you the XY plane, XZ plane and the XY plane. It will also show you the X axis, Y axis and Z axis which might be of use later, but as of now let us pick one of the planes. We will pick the XY plane.

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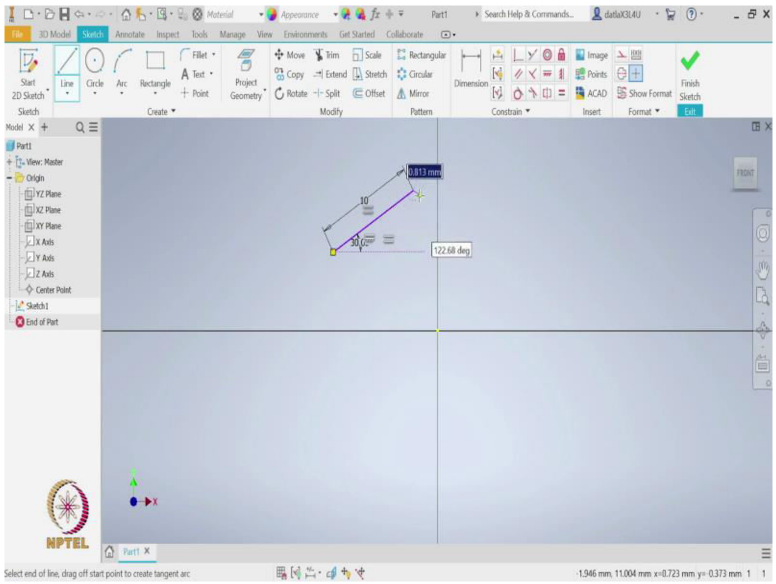

So once I click the XY plane, now it takes you into that plane. So now what we have is a 2D plane on which we can create the 2D entities, sketch entities. So how do we get started? As we have previously seen we can either start with the line, circle, arc, rectangle. The most common thing is the line. So let us get started with the line and see how we can create a straight line.

So once I click on this with the left click, then once I come into this working area you can see that wherever the cursor moves it also shows you the X and Y co-ordinates. And it’s a dynamic environment in the sense that if I move the mouse the X and Y values change. So I can click on anywhere with the left click and it creates a starting point of the straight line. And now if I move the mouse we can see now it asks for length of the line as well as the degree of the line.

So the degree says, with the X-axis what is the angle it makes. So as you can see the length of the line is highlighted. So currently it is showing 7.327. But let us say I want to create a straight line of length 10. So we just need to type in number 10 and click tab. So once you click tab it moves from the length 10 to the angle. So now we can see angle is highlighted.

Let us say I want to make a straight line at an angle of 30 degrees. So again I type 30 and again I say tab. So once I specify the length of the line and the angle of the line, now it is well-defined. The straight line is well-defined. So we again click left button to click Ok for that line.

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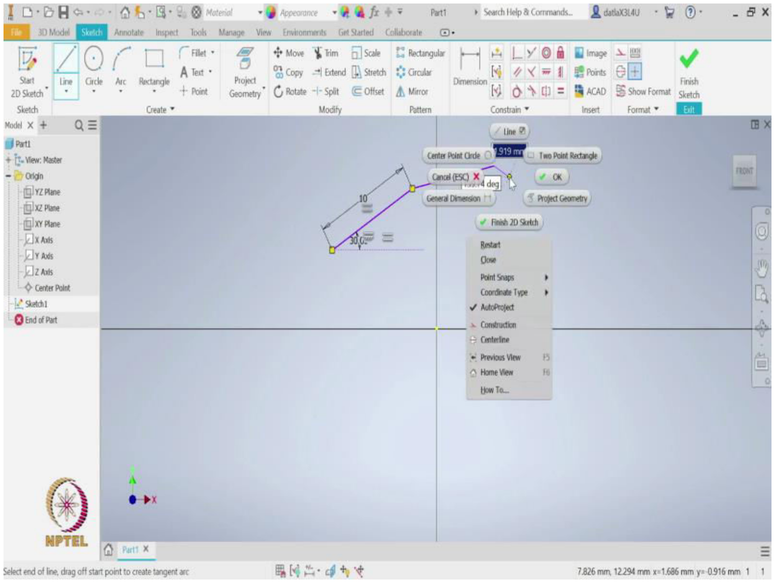

But you see that after I create the first segment of the line it is asking me if I want to create the second segment of the line. If I want to create a second segment of the line, again I need not start by clicking at the starting point. The last point in the previous segment will be the starting point for my second line segment. So I just need to click on the endpoint.

So this time I will click, create an endpoint just randomly somewhere. I am not specifying the length or the angle. So let us say, I just want to stop now saying that we only have two line segments. So there are multiple ways to stop it. Either I can right click, I can simply say Ok.

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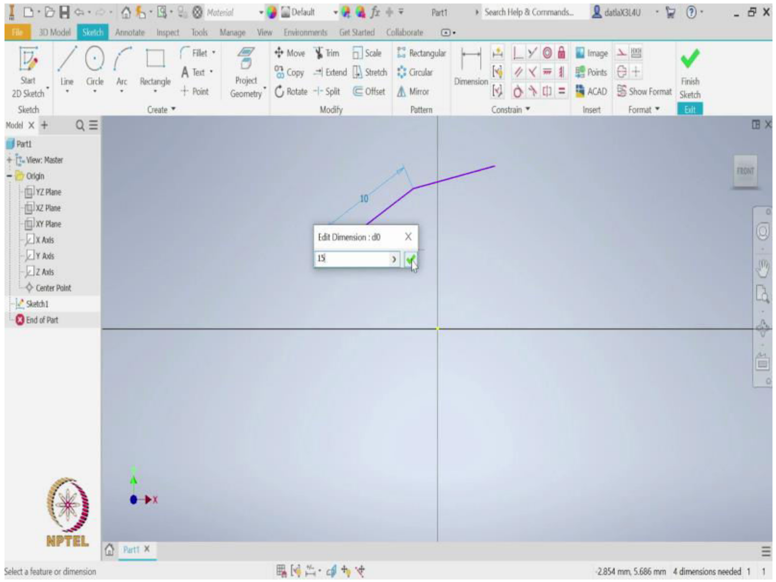

Fine, so now we have seen how to create straight lines and how to change the dimensions. So let us say, previously we have given the dimension of straight line as 10. Now let us say, I want to change the dimension to some other value. So I just need to click on the dimension 10, double click and then change the value. Let us say the value I want is now is 15. I can click Ok or the green tick here.

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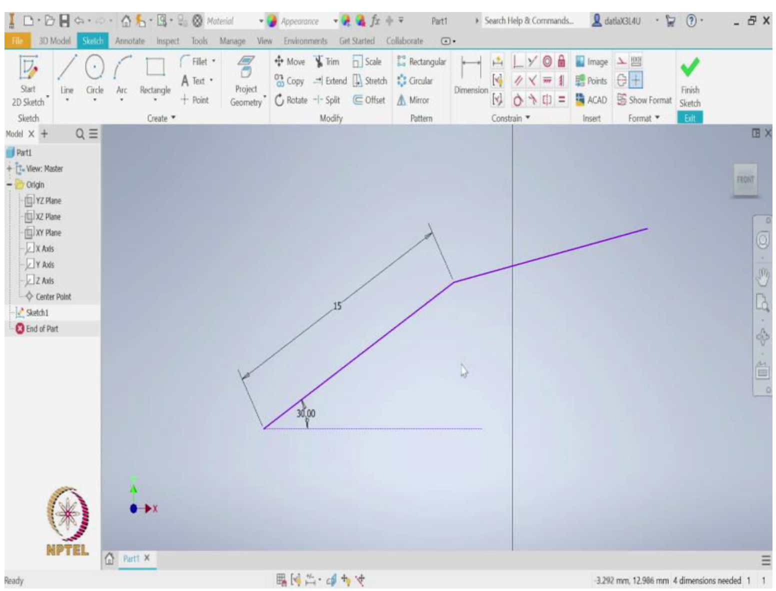

So let us fit Zoom All, and for me to move this to the center I can use the wheel on the mouse. If I click on the wheel of the mouse and drag it now this acts as a Pan button in the Navigation Bar. And now once I lift the wheel button now it stops panning.

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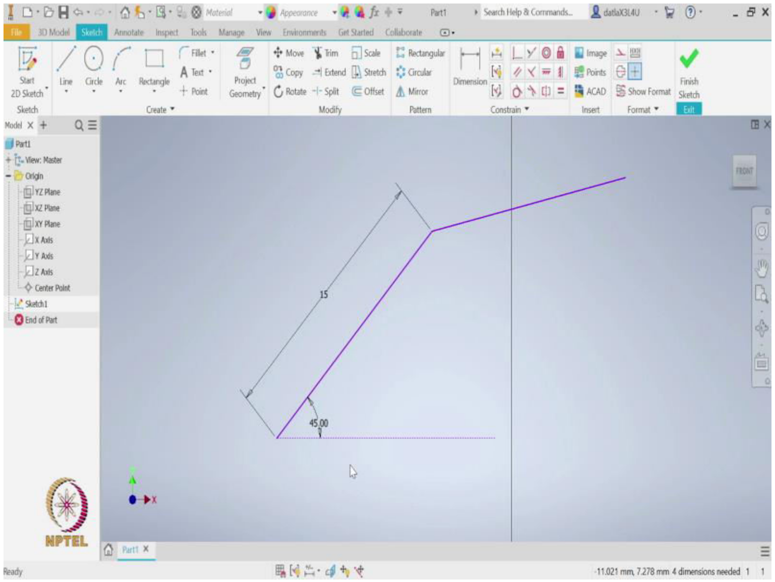

Similarly I can change this angle to, let us say, I double click on 30. First highlight, then it changes color to green and then double click. So now let us say, I want to make it 45 degrees. So this way you can create a straight line as well as edit the straight line.

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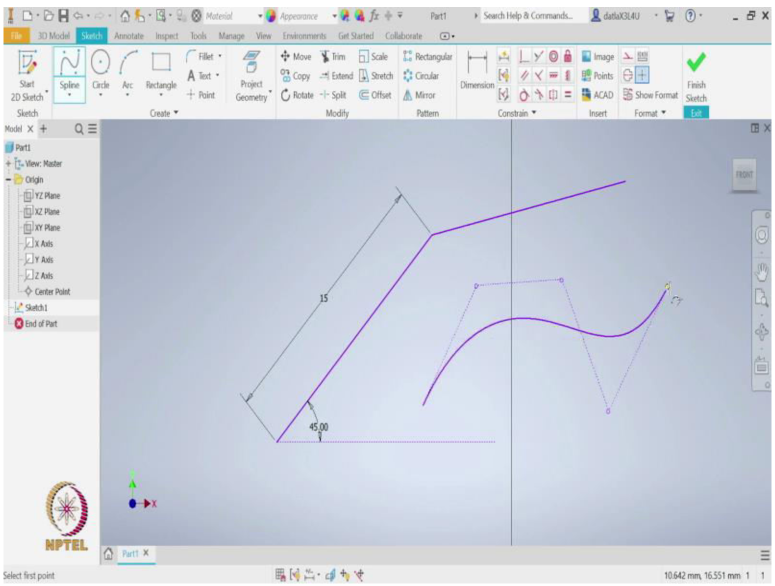

So now let us see what are the other options I have. So I if I click this small triangle it brings a drop down menu which shows Line, Spline, Spline with Interpolation, Equation Curve and a Bridge Curve. So I can, for example I can use this Spline curve to create a curve. I just need to select points. So starting point, second point, third point, fourth, fifth and let us say, this is the last point. For last point I can right click and simply say Create. With that it has created a smooth spline curve.

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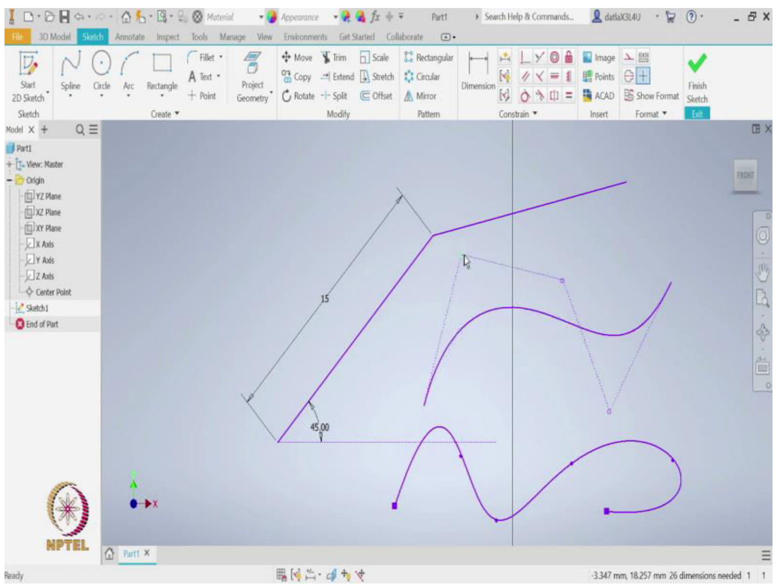

And there is an another option where you can either create spline using control vertex or the interpolation. Let look at this Spline with Interpolation and see what is the difference. So point 1,2, 3, 4, 5, 6 and I can click this tick mark to say that I am done with the creating a spline. So the difference between the first spline and the second spline is the second spline followed all the points that I have specified; but whereas the first spline, it only followed the initial and the final point but the rest of the points only help me to decide the shape of the curve.

So, for example, I can click on any of this specified point and move it. So if I move this point respectively the curve too changes.

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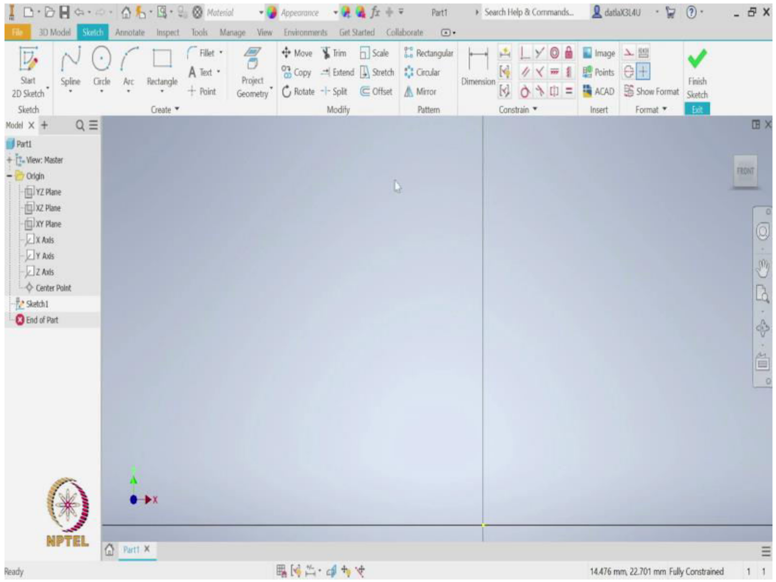

So let us say, I want to delete these entities. One way of deleting it is… Let us first say Zoom All. You can simply create a triangle by using the left click of the mouse, select all these entities and then simply press the Delete button. So it will simply delete all these entities.

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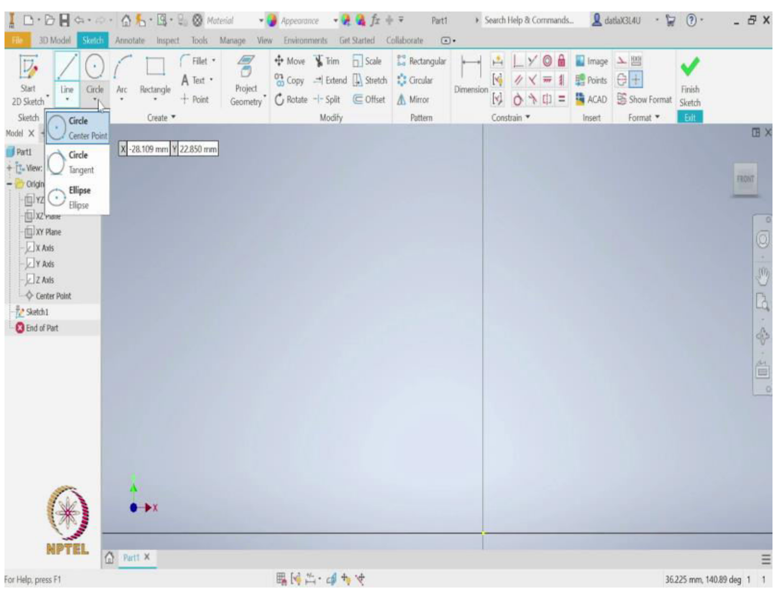

So what we have done till now is to look at what are the options we have in the line form. Similarly let us now look into what options we have in creating a circle. So we have three methods. One is circle with the Center Point, two is the Tangent and lastly the Ellipse.

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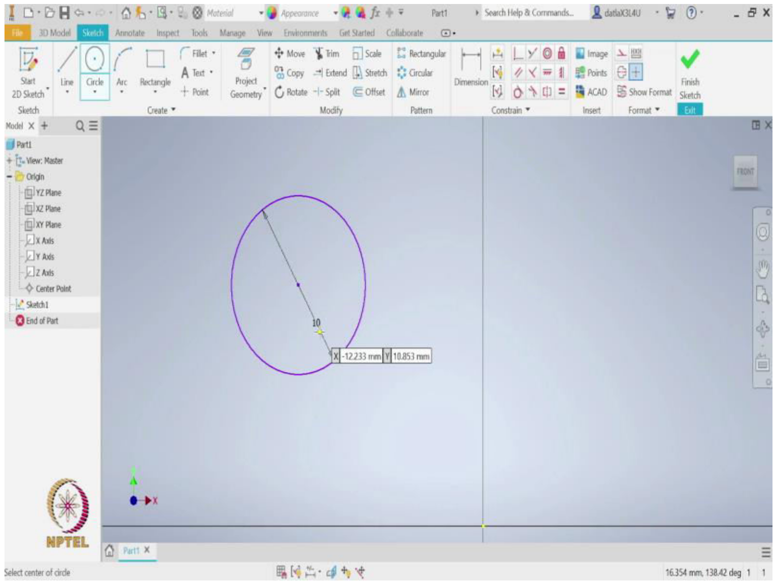

So let us start with the first one which is circle with Center Point. So it asks for the center of the circle. Once I click it then it is asking for diameter of this circle. I can simply say the diameter is 10 and click enter. So it has created a circle with the specified center and the diameter of 10.

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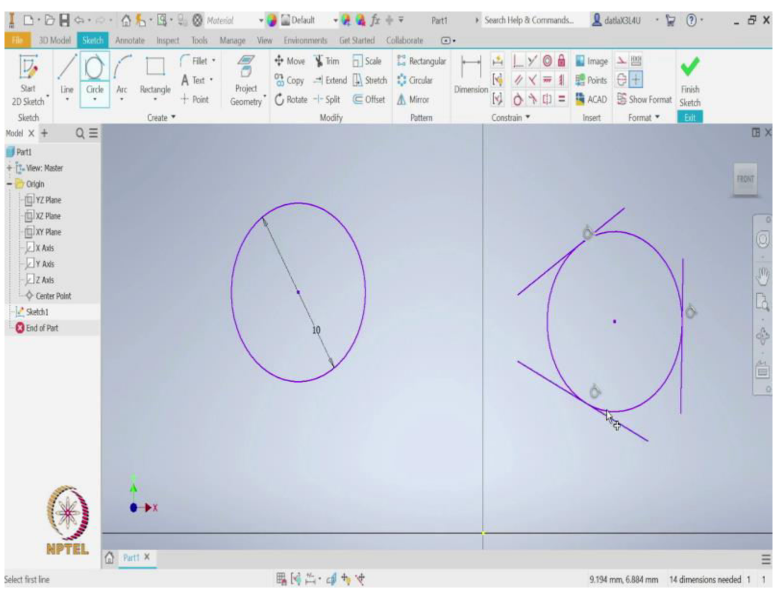

Similarly we can use the second method but before we use the second method we need to make the three straight lines and ask to fit in the circle between those three straight lines. So let me go back to the line again. So this is the first line. I click, right click and say Ok. Again another straight line, again right click and Ok. And lastly one more straight line. So once I have these three straight lines I can create a circle which is tangential to all these straight lines. So I click line 1, line 2 and line 3. So as you can see once I created these three lines it created a circle.

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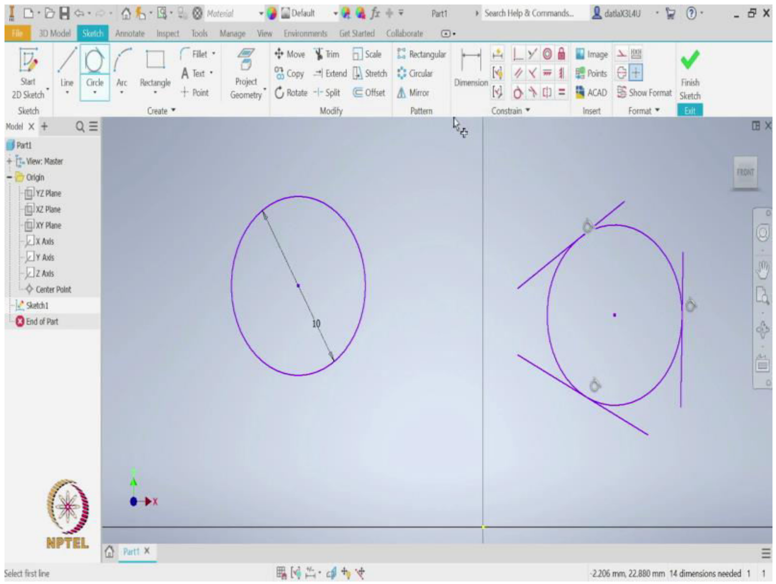

You will notice that when it created a circle it shows some symbols. These symbols are actually constraints. So, for example this symbol signifies tangential constraints. It says the circle is tangent to the line here. Similarly we have one more tangential constraint with the line 2 and another tangential constraint with line 3. If you look at this closely we have this constraint, tangential constraint here too in the Constraints option.

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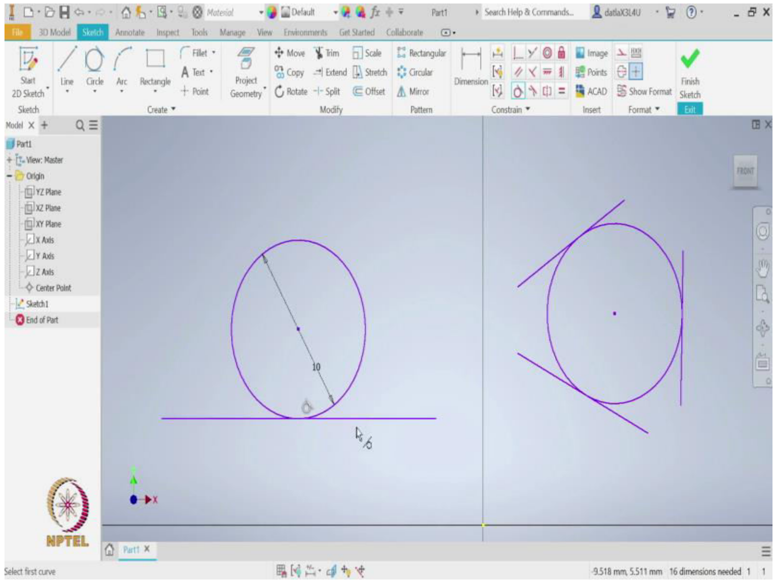

So let us say, I want to use the constraint. First let us create a straight line. And now let us use this tangential constraint. So I just need to click this straight line and this circle. Now what it is doing? It is moving the circle in a way it is tangential to the straight line which I selected first.

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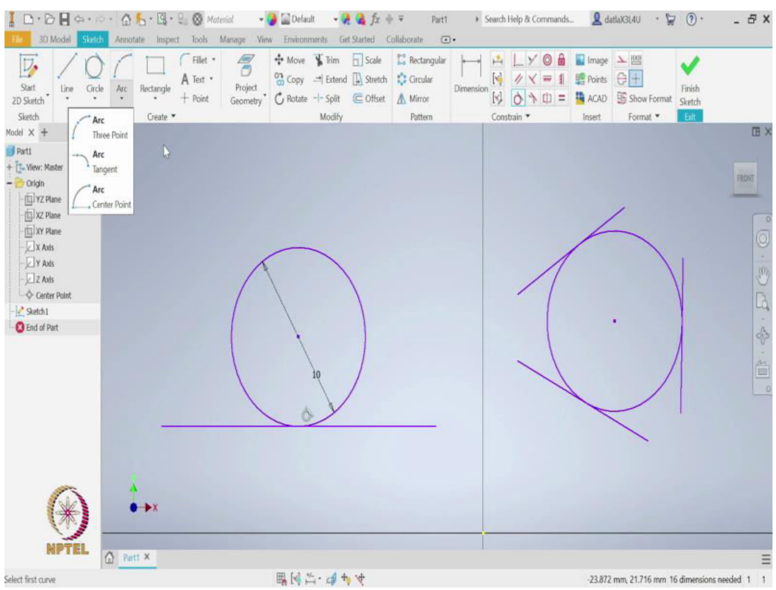

Similarly you can also, one can also create ellipse. But now let us move into arcs. So again in arcs there are Three Point Arc, Tangent Arcs and Central Point.

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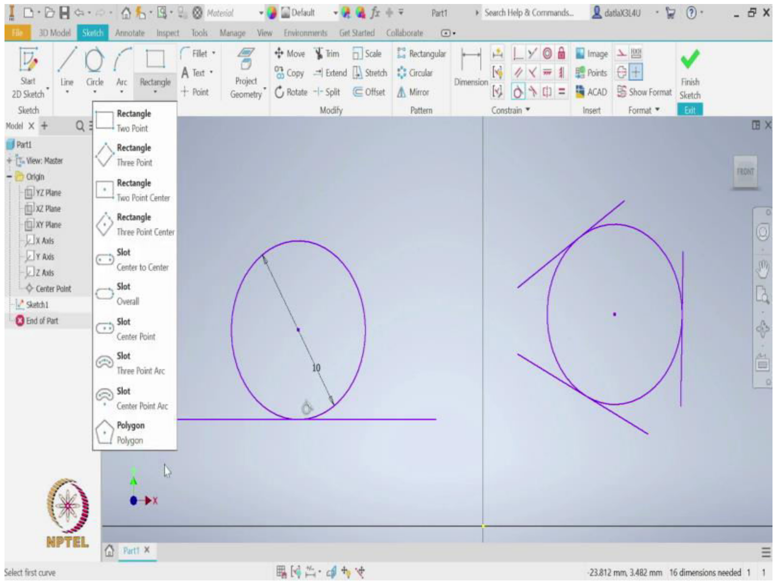

So another common feature which we will be using quite often is a rectangle. So as you can see there are several options in rectangle starting with the basic thing of creating a rectangle using Two Points, using Three Points, rectangle using Two Points and Center, rectangle using Three Point Center, a Slot, Overall Slot, Polygons. We can create shapes as if highlighted, it will say I can create polygons of at most 120 sides. Of course you may not need that many sides but at least, depending on the requirement you can choose between 3 to 120.

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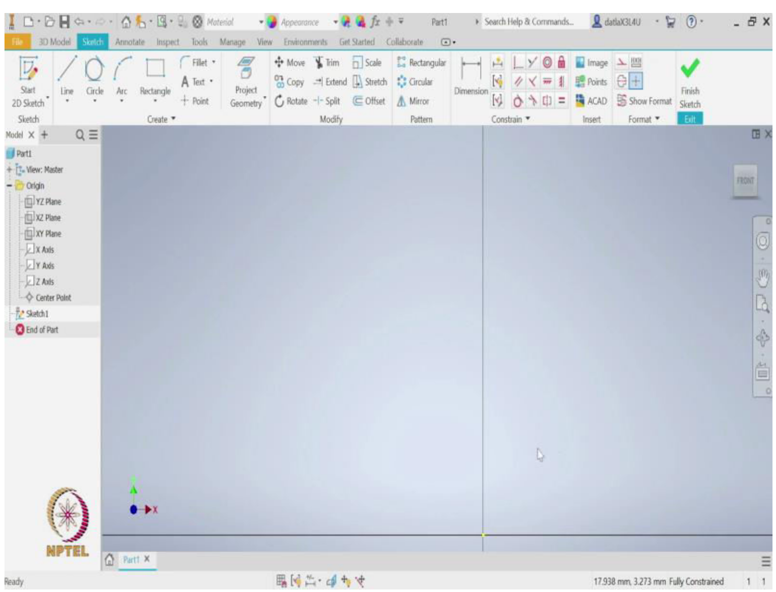

So let us start using some of them. Let me delete this. So I just need to select all these using this box and then press the Del, Delete button.

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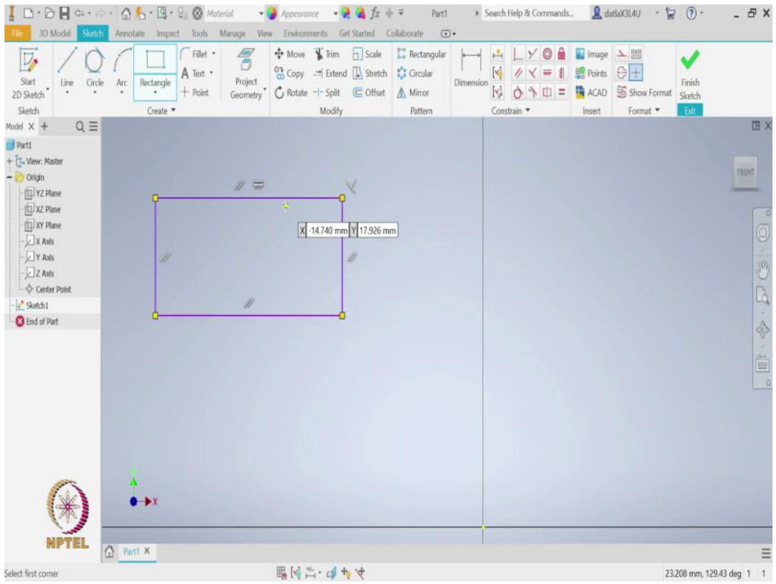

So now let us start with the easy way to create a rectangle. So you first select one corner and then the other corner. So if I click it you can see there have many constraints with which it has created. First it said there are parallel lines. The left one is parallel to the right one, and two more set of parallel lines, the top line and the bottom line.

It also has other constraints like perpendicular. It says this top line is perpendicular to the right side. And similarly there is one another constraint saying that the top line is horizontal. So once you specify all these constraints then we can fix the shape of this to be a rectangle.

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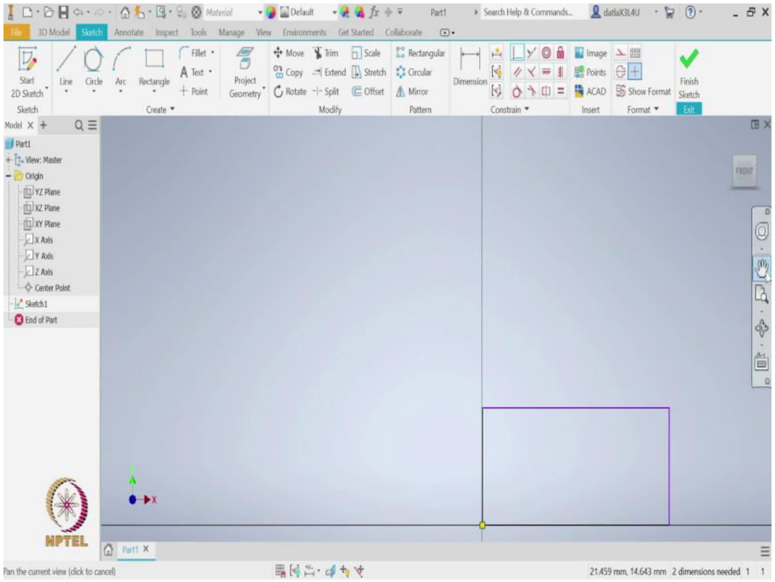

But as you can notice we can still see that the rectangle is drawn in blue color. And for us to completely constrain this rectangle, once we eventually do that, the color of this rectangle will change to black. But now let us try to attempt and see what we need to do to make this rectangle completely constrained.

As of now what is missing is we are yet to specify what is the exact location of it. Though we know the size of the rectangle and the shape of the rectangle we do not know the exact location in the plane we have selected. So to do that, one way to do it is you can start with this Coincident Constraint. A constraint points to other geometry in 2D and 3D spaces.

So first I will click this Coincident Constraint and then say, I am selecting this bottom left corner of the rectangle. So I selected a point. I want that to be coincident with the origin. So if I move close to this yellow point it changes to green. So if I select this origin, now this rectangle the bottom left corner is coincident with the origin of this XY plane that I have chosen.

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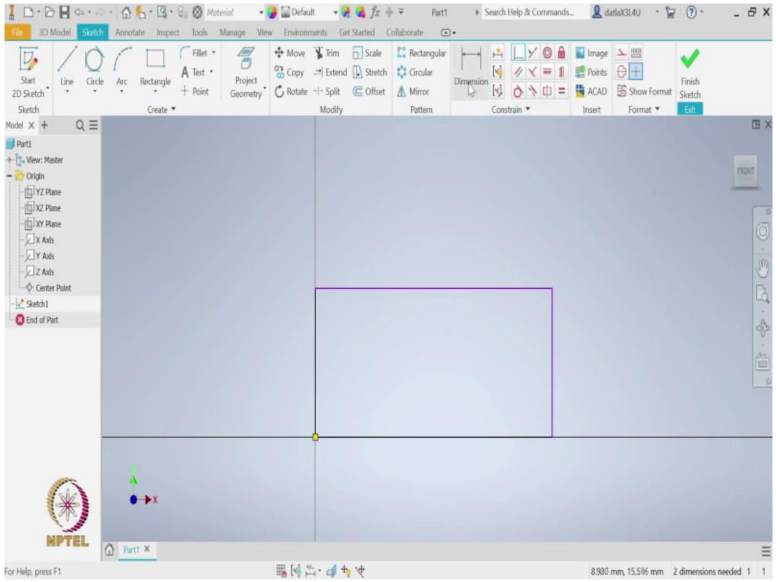

So now again let's do Zoom All. So probably this is a bit high. So we can also use the wheel in the mouse to zoom in and zoom out. We can see that two of those lines of the rectangle have turned black but there are two others which are still blue. The reason is we still did not define what is the dimension. Whatever dimensions it has taken now is automatically generated by the software but we need to specify to make it exact.

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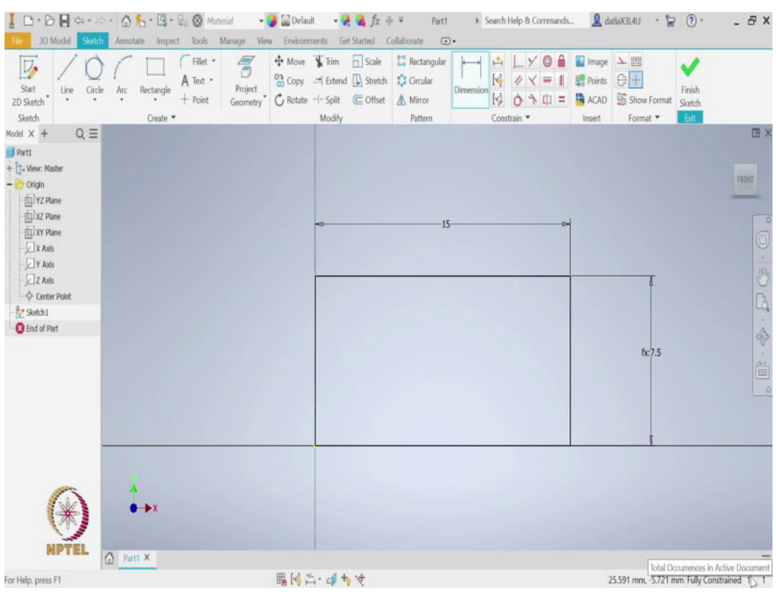

So now let us use this Dimension tool and click this top line. So once you click there it ask for the dimensions So we can double click on this to say the dimension is, let us say 15. Enter. So notice that in the top of this dialog box it gives a variable for this rectangle saying it is D3. Please remember this. We will use it to define the height of the rectangle.

And now let us again use the same Dimension tool to dimension this right side line. Double click and here let us use the previous variable D3. D3 we know is the width of this rectangle. So we say D3 divided by 2. So we are saying that the height of this rectangle is half of the width of this rectangle. So as you can see here, now it mentions that it is a function and the function has a resultant value of 7.5, because we say this function is D3 by 2. D3 is nothing but the width of this rectangle.

Now you see that the rectangle is completely black which means we have fully constrained. One way to know it is fully constrained is of course by seeing that the colors changed to black. And if you look at the bottom bar, in the bottom right it says Fully Constrained. Here this also indicates that we have completely constrained this rectangle. We have constrained the shape to be rectangle. The size is constrained by 15 and 7.5. And the location is also constrained because we moved the bottom left corner and coincided that with origin of this XY plane.

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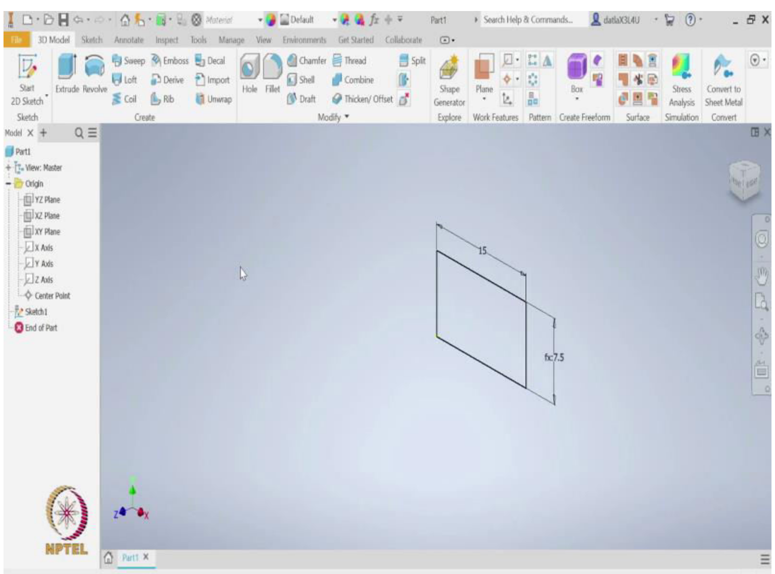

So with this we can say we are done with the sketch and let us say, we can, we need to click this Finish Sketch to say that we are done with the sketch and we have created a profile. So if you now look into this Part Browser you will see that there is a sketch F1. If you click on it, it will highlight the sketch we have already created.

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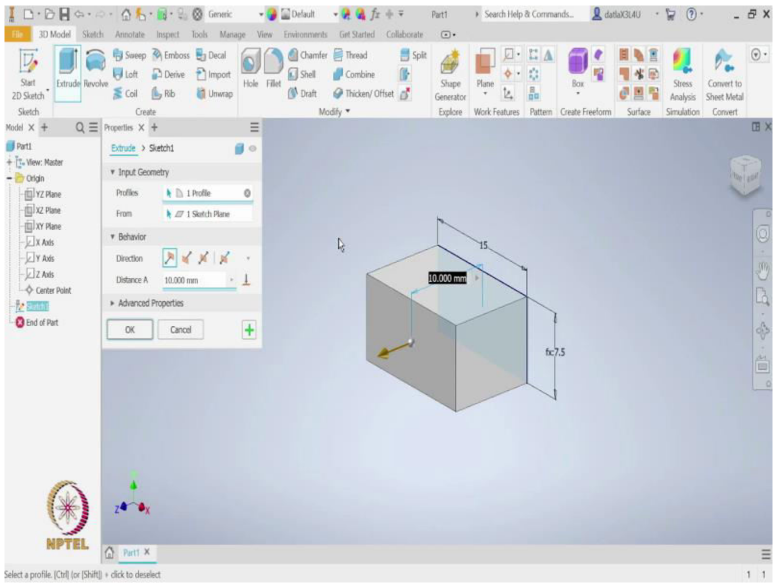

Now let us quickly see how to create a solid from this sketch. One quick way is we can, let us say, use the Extrude command. So once I click the Extrude command it automatically selects the sketch, because there is only one sketch it knows which one to pick. Otherwise if there are multiple sketches we can go into this Input Geometry.

We can first specify what is the profile and from which sketch plane are we taking that profile from. But in this case since the software is dynamic and smart enough it can pick up the interface without we selecting it manually. But then Behavior has several options. So, for example, let us start with the direction. This is the default direction. It is saying that it in this direction it will do the extrusion.

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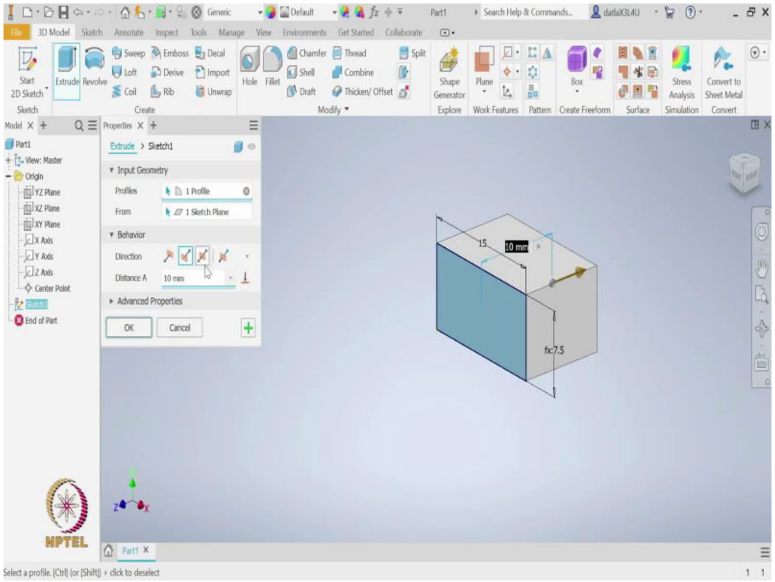

But let us now look at the second option. Now you can see the extrusion direction is reversed.

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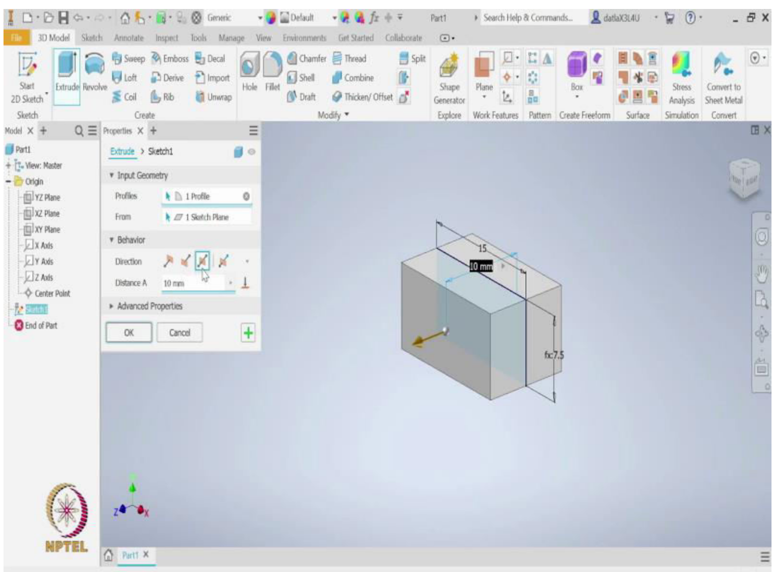

And the third option is you can choose both front direction and the back direction for extrusion. Then you need to specify what is the total length or the total depth of this object. This third option is symmetric which means, in the positive direction and the negative direction, it will it take the same dimension.

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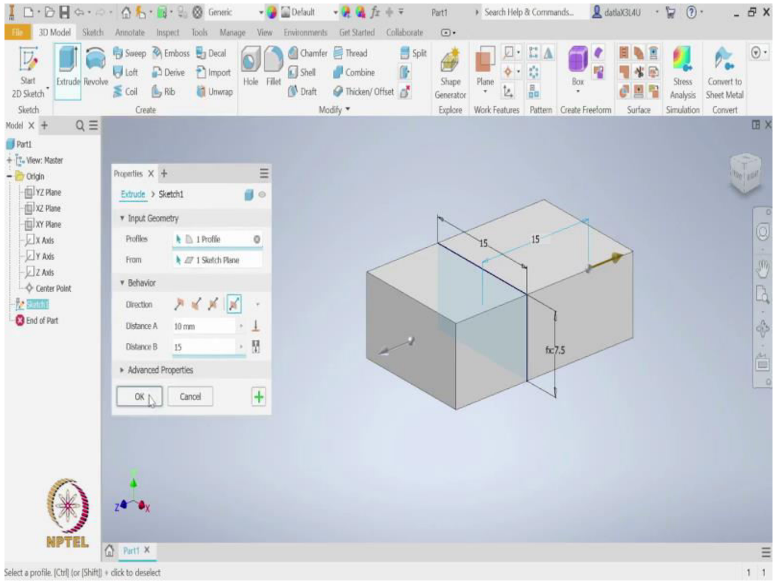

But now let us say, you select asymmetric. Asymmetric will ask you for distance A in direction 1 and distance B in direction 2. So as of now it is suggesting 10 and 5 but we can always change it, let us say, the second direction we will make it 15. You can see it is showing you the preview of what happens if you choose these specific directions for the extrusion command. Finally if we are happy with what is to be done with this extrusion dialog box we can click Ok.

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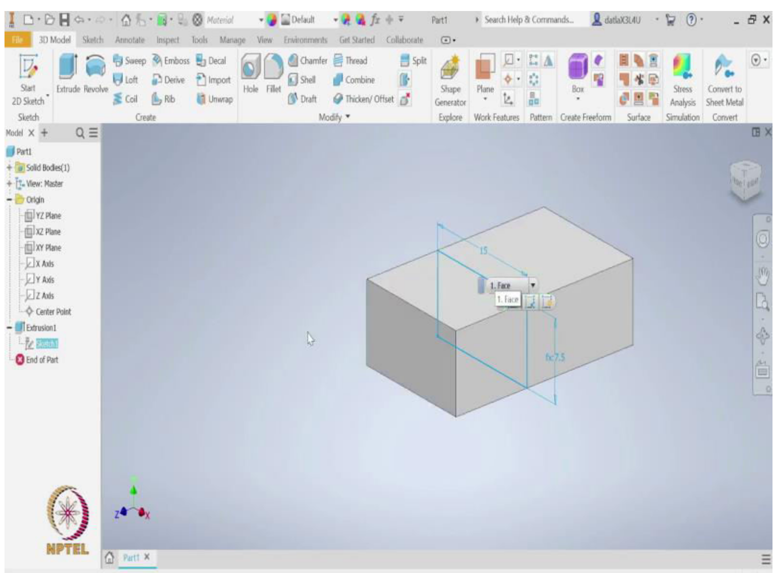

And then we get a solid by using the sketch plane or the profile we have created. So if you again now look into this Part Browser you will see it has created one more entity called Extrusion. If we click on this plus it will say this extrusion is created using the Sketch1. So if I click on Sketch1 it shows the sketch with which we created this extrusion solid.

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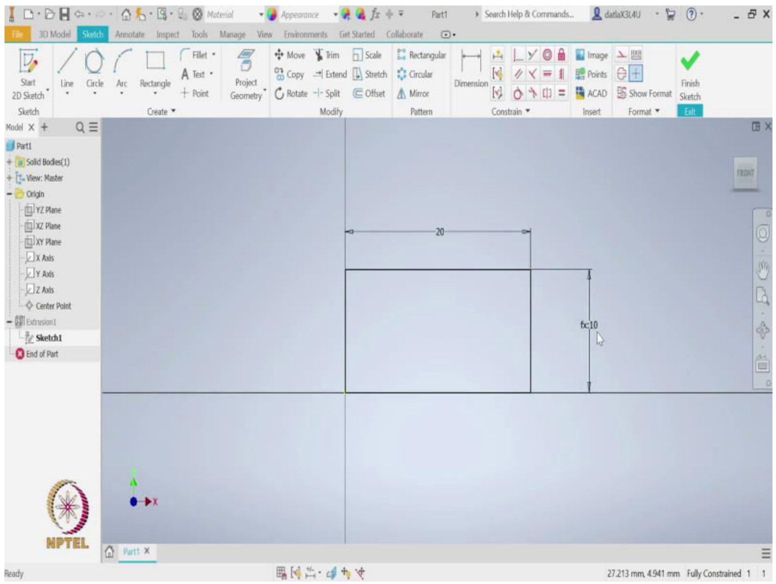

Let us say, in future you want to come back and change the dimensions of this solid. There are multiple ways to change this solid. One is you can go to the sketch, right click on it and then click on the Edit Sketch option. So once you click on the Edit Sketch option you are again back to the sketching environment where you can make any changes.

Now let us say, we want to change the width from 15 to 20. Now you can see the width is changed to 20. Similarly the height is again changed to 10 because, if you remember, we said the height is half of the width.

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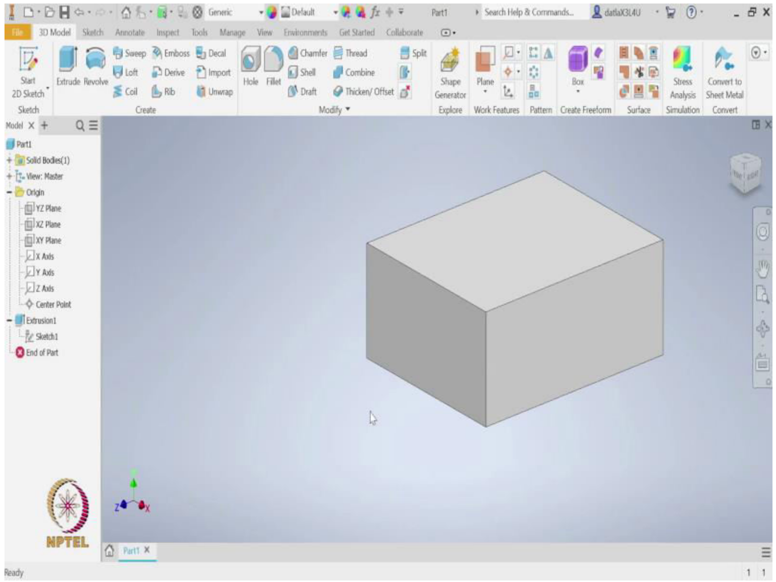

Now if I say Finish Sketch now we will see that it will automatically update the sketch as well as the extrusion because the sketch is defining how this extrusion is done. So it will apply the same changes we made to the sketch to the extrusion. So this is one way of modifying this object.

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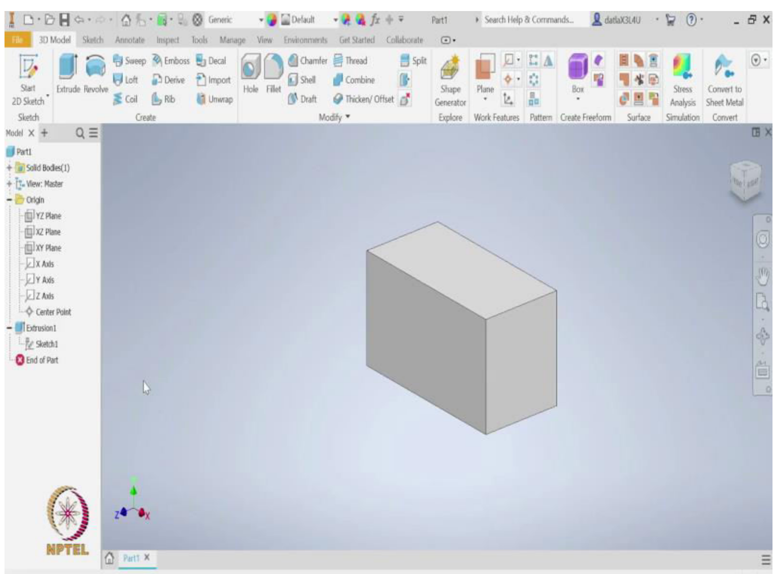

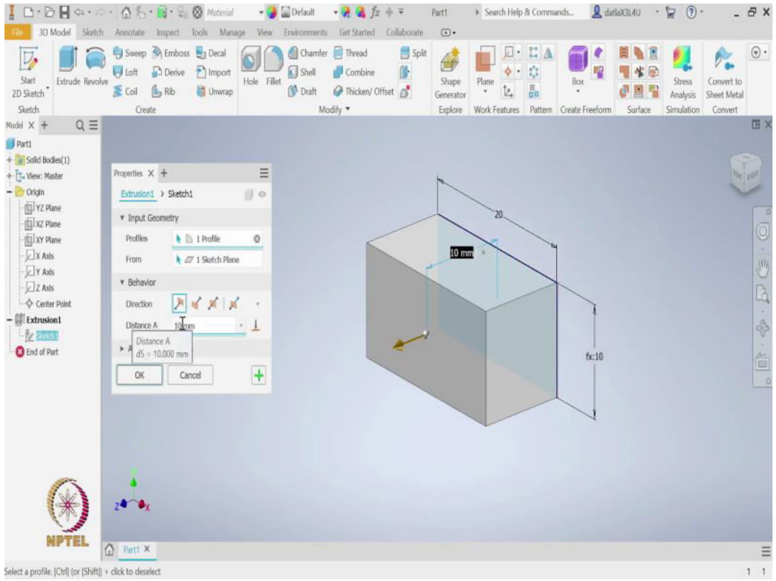

The other way of modifying is you can go to these, extrusion features, select it, then right click on it and then, say we have two options. Either we can edit the Sketch or edit the feature. Now let us say I want to edit the feature. Once I go to Edit the Feature again I come up with this dialog box. It gives me options how to change the extrusion.

So let us say, instead of using this asymmetric extrusion I will select this default Behavior and keep this distance to 10, and click Ok. So, as you can see now it has modified the depth of this extrusion.

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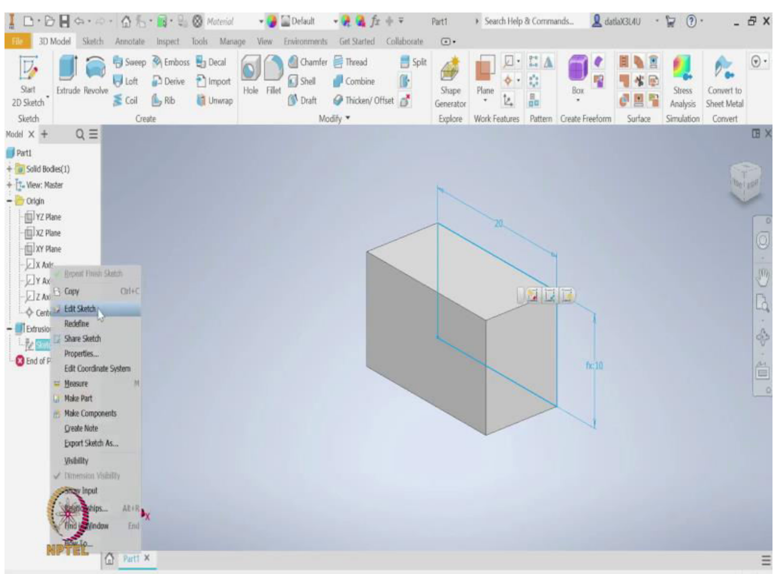

What we have quickly have seen till now is how to use the Sketching Environment, especially we have started with that rectangle and using rectangle and extrusion features we have created a simple object. Now let us get back to the rectangle because there are other features we can use with the rectangle. So how do we get back to the sketch? We can either create a new sketch or go to the previous sketch. For the previous sketch we said we will select, say Sketch1 and right click, and go to the Edit Sketch.

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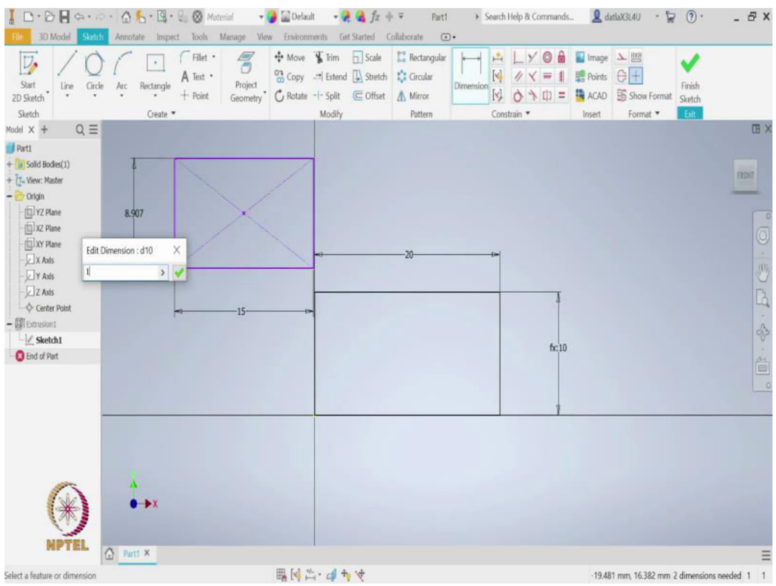

So now let us look at this sketch and see what are the other options we have. So there is another way to create a rectangle using the three points, and there is an another way of creating rectangle using point and center. So let us try this Point and Center first. So we are creating one point on the rectangle and then saying what is the corner of the rectangle.

So first we clicked at the center of the rectangle and then to one corner of the rectangle. I can also specify the dimensions or simply click on it. But once I create a rectangle I can now change the dimensions using this dimension constraint. So the width currently is 12. I can change it to 15. And similarly I can use the height to be 10, let us say.

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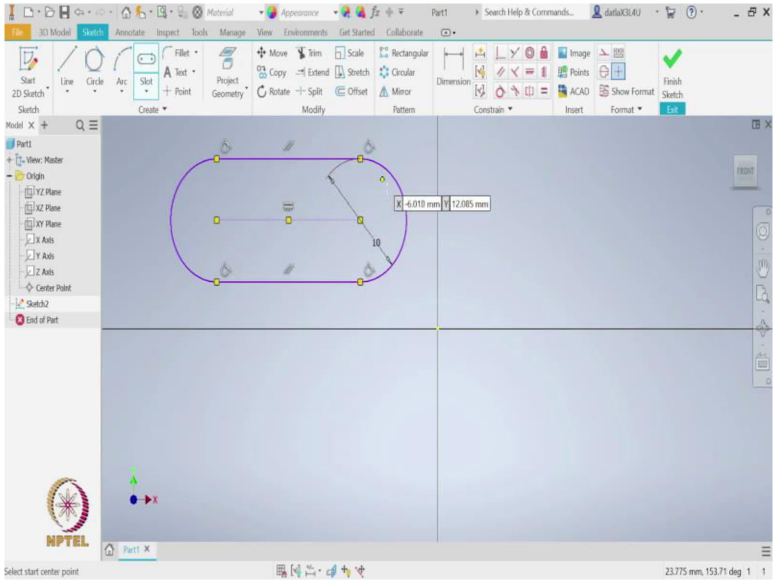

So we will again start with sketching. We have these standard planes which are available. Again let me check the XY plane. And now let us look into the other features we have in this rectangle. So we have something called a Slot which means you can specify two centers and then the size of the slot. So let me give it as 10. So with this it has created a slot with a specified shape.

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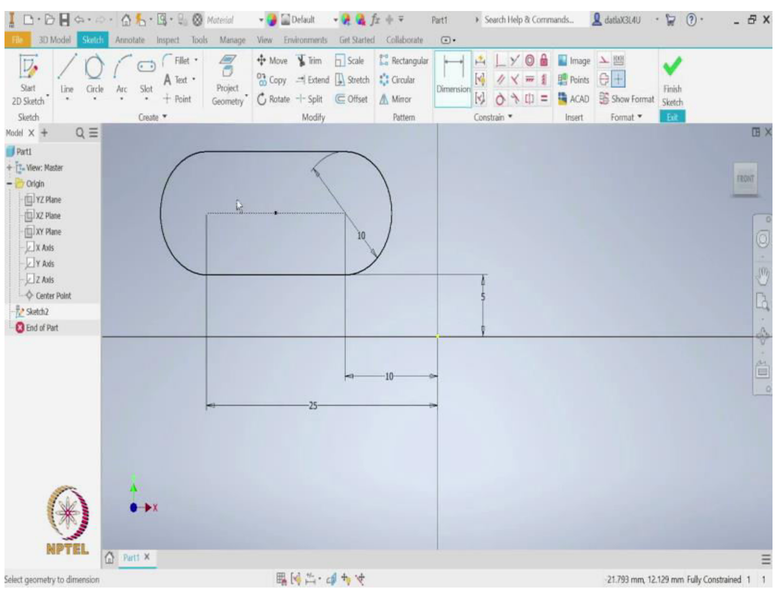

Now let us say it is not fully constrained. As you can see because it shows in the blue line. And if you see in the bottom right it shows you still need three dimensions before you can fully constrain this slot. So how do we constrain? One way it is through dimensions saying that this bottom with the origin is 5 distance. Similarly one end of the circle, I can say, is at a distance of 25 and the other center, let us say, is at the distance of 10.

With these three dimensions we have completely constrained this slot. And we can now use this slot to finish the sketch and then create a solid. But that we will look at it later.

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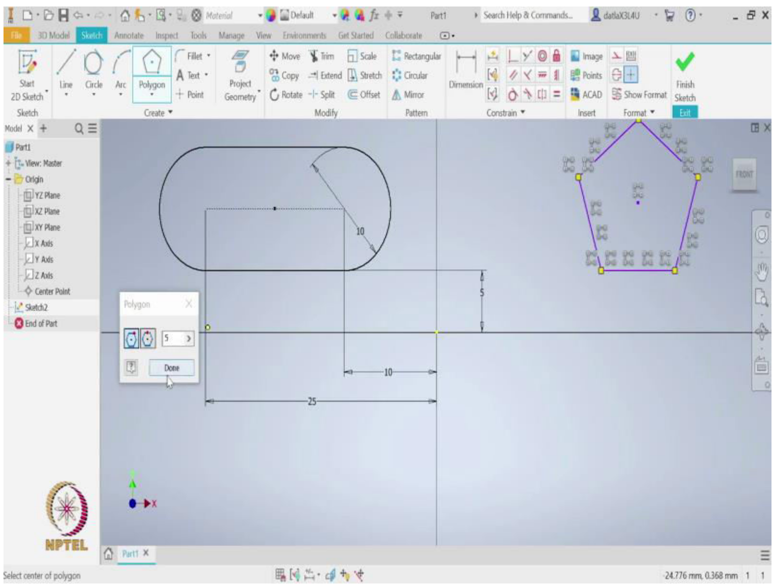

Let us look at what are the other options we have. So let us look into this polygon. So it asks you like how many sides of the polygon we need. Let us say, it is suggesting 6 but let us say, it is 5-sided polygon is what we need. So we need to click at one point which will be the center of the polygon, and then click at one of the corners of the polygon to create the polygon. So it has automatically given several constraints such that it is, you get the shape to be a regular polygon of 5 sides. We can hit Done.

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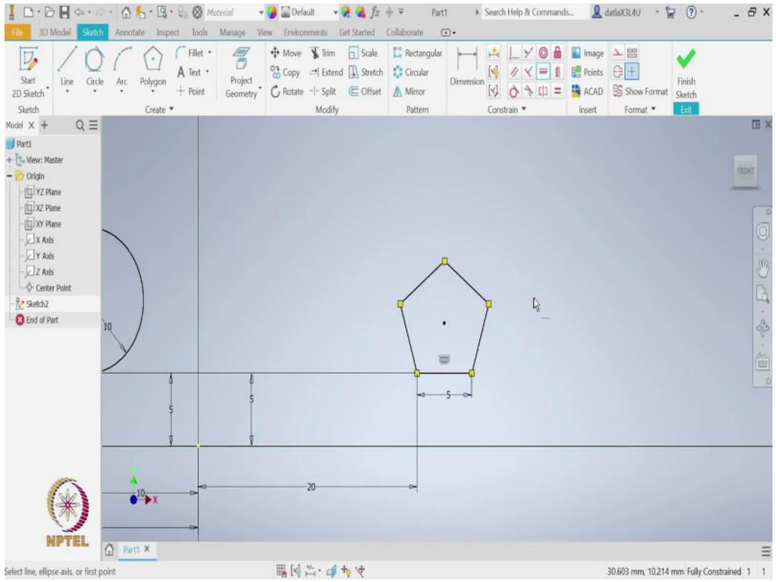

But let us see how many dimensions are remaining. So it needs 4 dimensions to fully constrain this polygon. So for this pentagon let us say, we dimension it by saying the length of this line or the side of it is 5. We can dimension this corner to be of so and so.

Lastly it asked for one more dimension. So let us say, what are the dimensions? Did we specify that this line is horizontal? Let us specify that, saying that horizontal constraint for the bottom edge. Do with that it we have completely constrained this pentagon and which means it has create a profile which is fully constrained which can later be used to create a feature.

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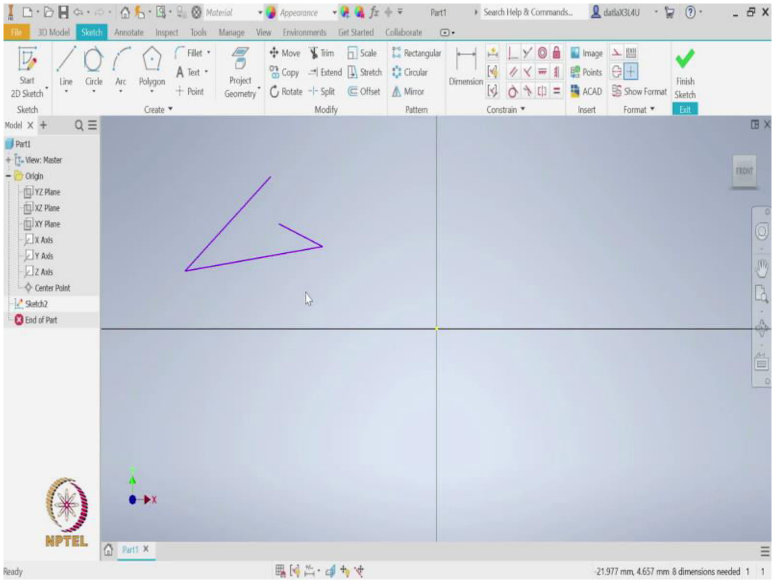

So what we have been looking till now is how to create 2D entities in the sketches using line, circle and rectangle or polygons. Similarly we can, we have also used constraints to completely constrain the profile so that in future whenever we make changes using the parametric capability of this software we get to create a neat object.

So there are other ways to modify the created geometry. Here we have several commands we have in the Modify like Trim, Extend and Split. Let us quickly look at them. So I can start with a straight line saying that this is one straight line; 2 3 and 4 points. So I will click. I will select Escape button to say that I am done with creating this multi-segmented lines.

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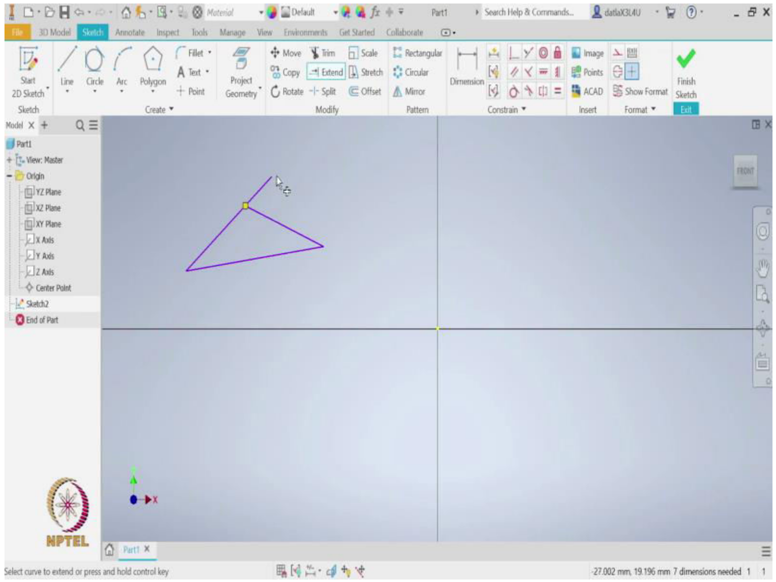

So now let us use this Extend button. So once I clicked this Extend and come to this third line it suggests that it will extend it until it reaches the first line segment. So let us select. So this is how we extend a line until it reaches another entity. In this case it is the first line segment.

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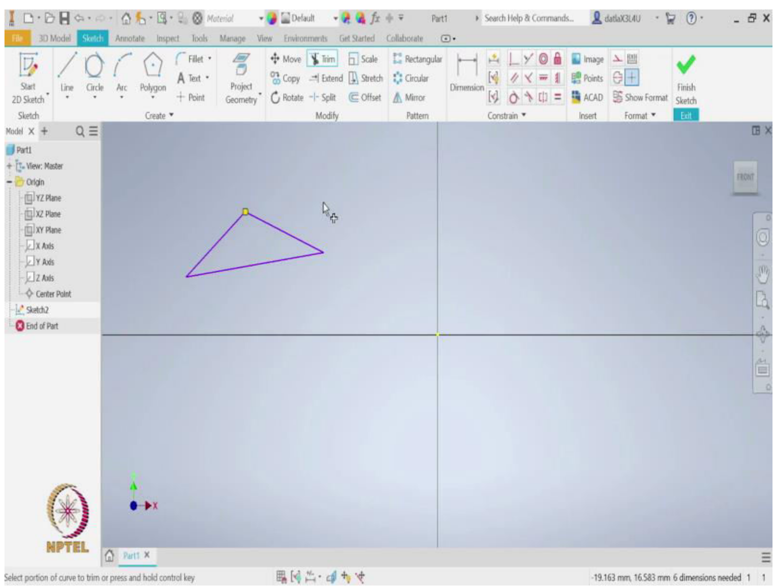

But now let us say this hanging straight line needs to be deleted. For that we can use this Trim command. So once I select this Trim command and move over this first segment of the line and select it then it understands that there is an extra segment which needs to be removed and it has trimmed the straight line.

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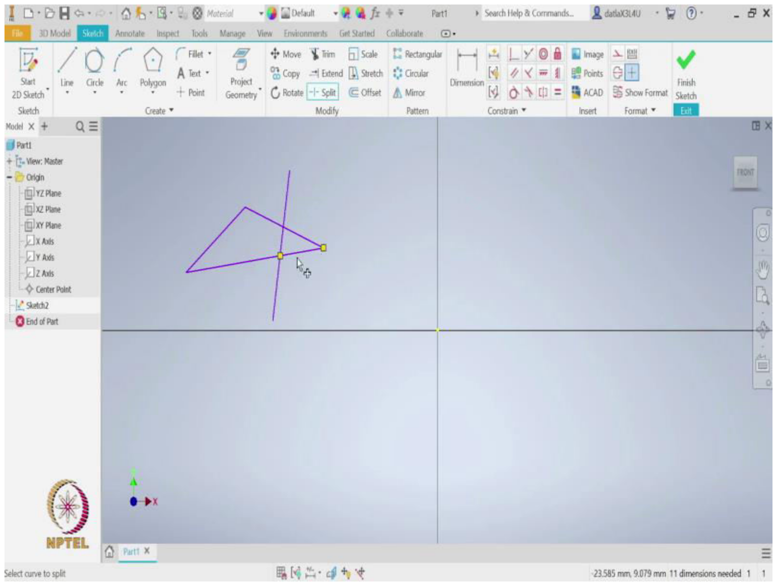

Similarly let us say, now we want to split a straight line. So for that let me again create a line here and now you use this Split command. So we will use this Split command to split this part of the triangle. So now it has split this straight line into two parts.

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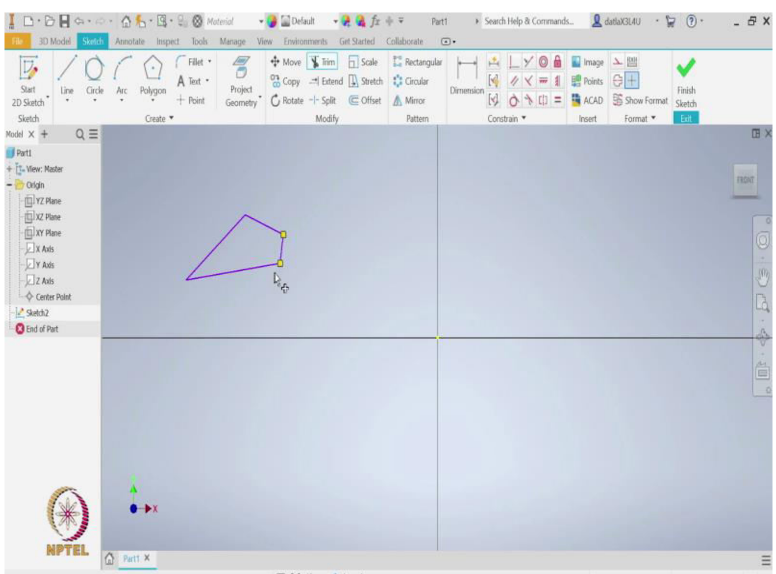

So now I can use this Trim command to remove this part. Similarly I can remove this part of the straight line, and again these parts too. So with this now I am able to create a closed loop which can be used to create solids.

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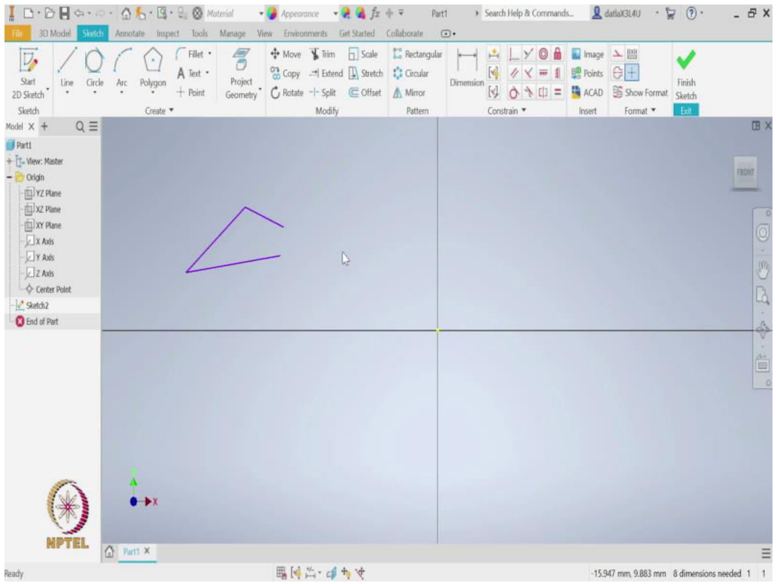

But one might have a question like what happens if we do not close the loop. Let us say, for example we select this line and delete it. What we now have is a open loop.

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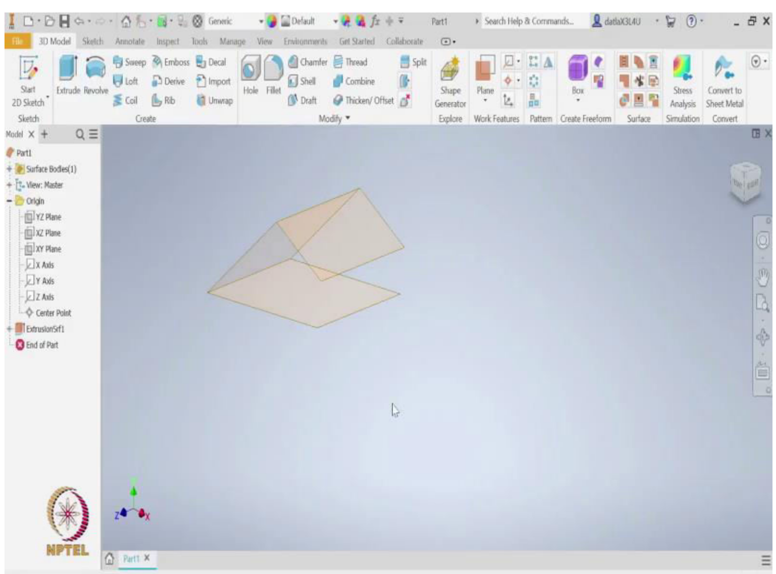

Let us say, I finished the sketch and then select Extrude and select this straight line. So now what we see is it since it is not a closed loop it is not an area in the sketch. It is simply a series of lines. What happens if you extrude a series of lines? It will create a surface instead of a solid, because what extrude does, it increases the dimension by 1.

If you start with a curve the Extrude will bring you a surface like what it is trying to do it here. But if you start with an area from the sketch plane then it will make that area into a solid. That we have been seeing in the previous examples. With this let us end this lecture. Essentially in this lecture what we have seen is how to use the sketching environment to create profiles.

And we also talked about how to use the Extrusion command or the Extrusion tool to create simple objects. In the next lecture we will be talking more about other features to create and other tools which are available to create solid models such as the Revolve, Rib and other features. So, with this, thank you for your attention.