Engineering Graphics and Design
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Week 11: Assembly
Example 9

Welcome back. Today's lecture we will look at an example of an assembly. So, we will start from parts that are already created. So, we will bring them to the assembly environment and assemble them using constraints. At the end of the lecture, we will also extract the assembled drawing and the bill of materials. So let us look at the problem.

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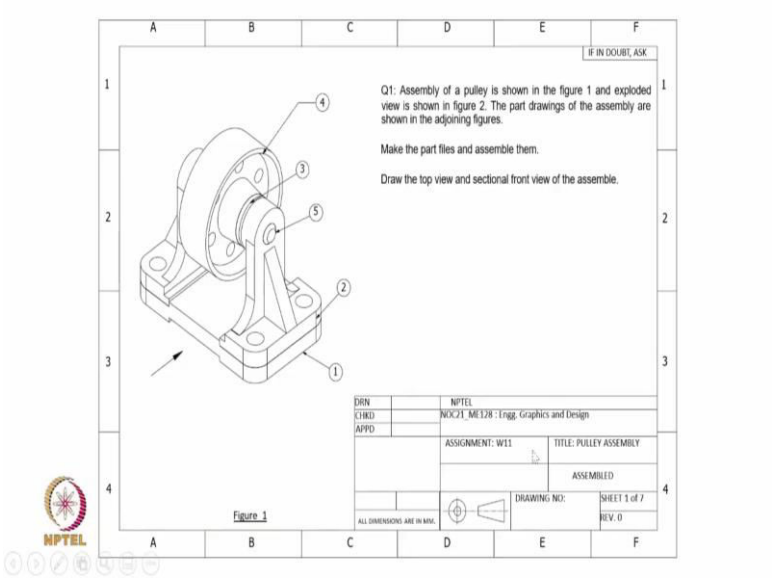

So, here is an assembly where we have five components, one is the base, two; the part two we have two components, one at the right and one at the left. And part three is the collar, again it has one at the right and one at the left. It will be much more clearer when we show the exploded view in the next slide.

But let us now focus on part three where we said these are two collars, one to the right and left. And at the centre you have this part four which is a pulley. And the thing which is holding the pulley is the shaft which is shown as part five.

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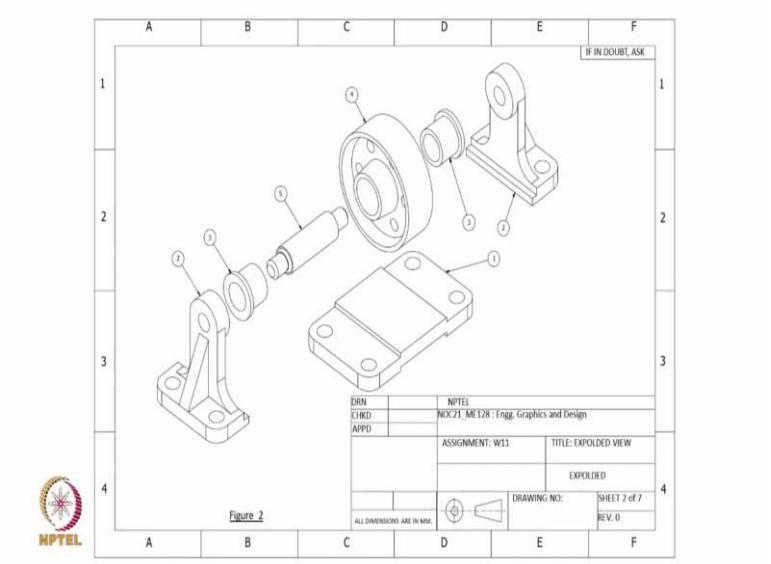

Now let us go to the exploded view where we can see all of these parts of the components more clearly. So, we have the base, to the left and right we have this component two and part three is the collar, we have one to the left and one to the right. And at the centre we have this shaft, on the shaft is sitting the pulley. Which are these parts five and four.

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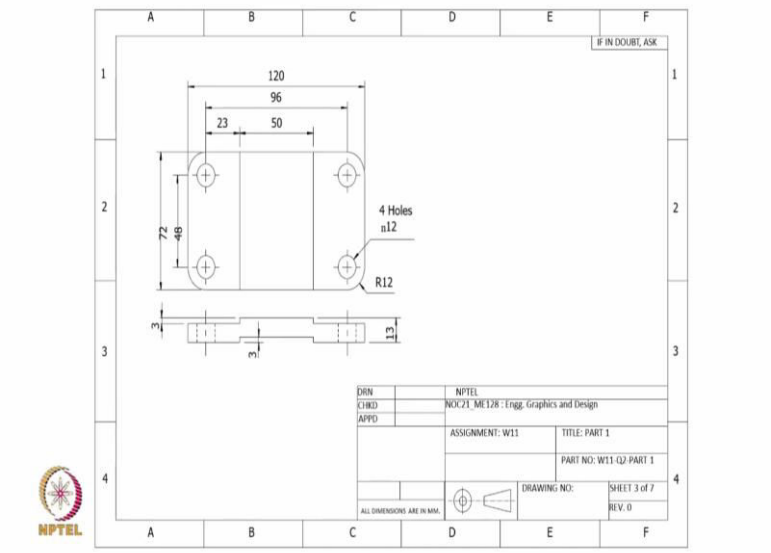

So, we have drawings of each of these parts. This is part one, the base plate.

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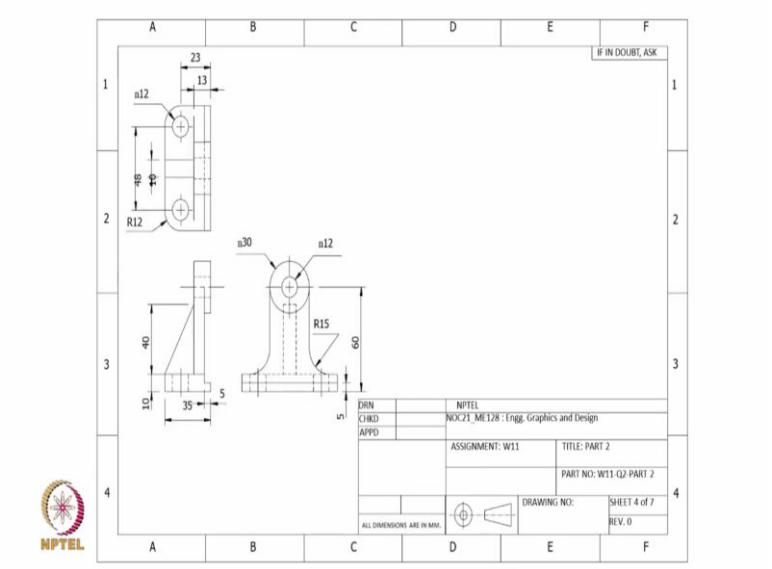

Next is part two, the ones which are sitting to the left and right which hold the shaft.

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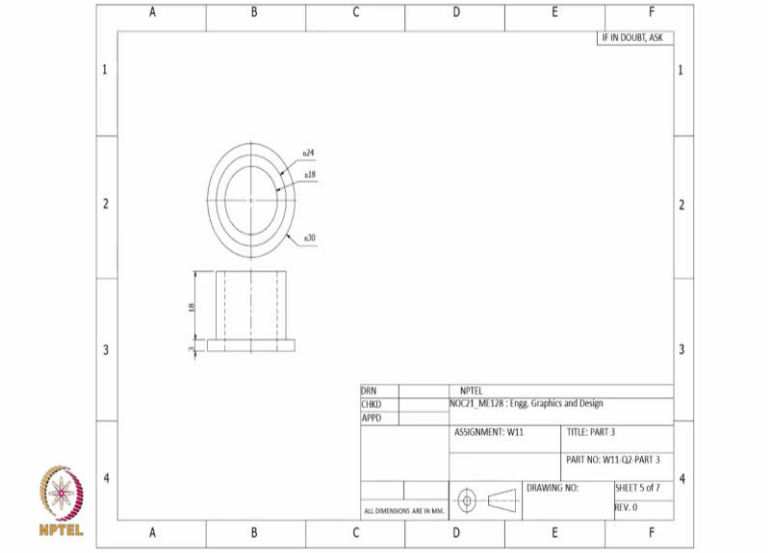

Next is part three which is the collar. Again, we have two quantities, one to left and right.

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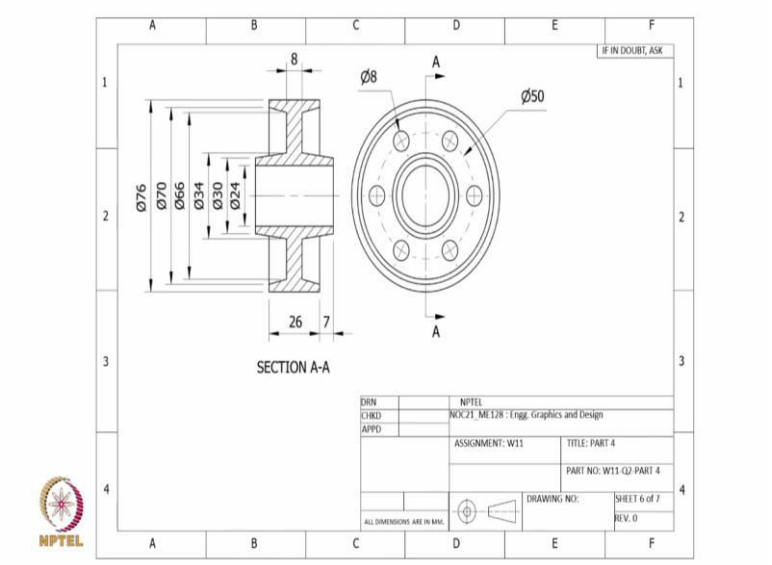

Part four which is the pulley.

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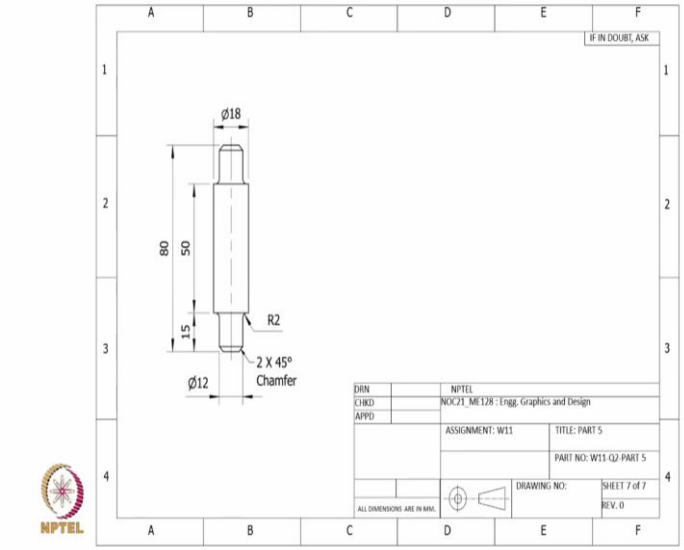

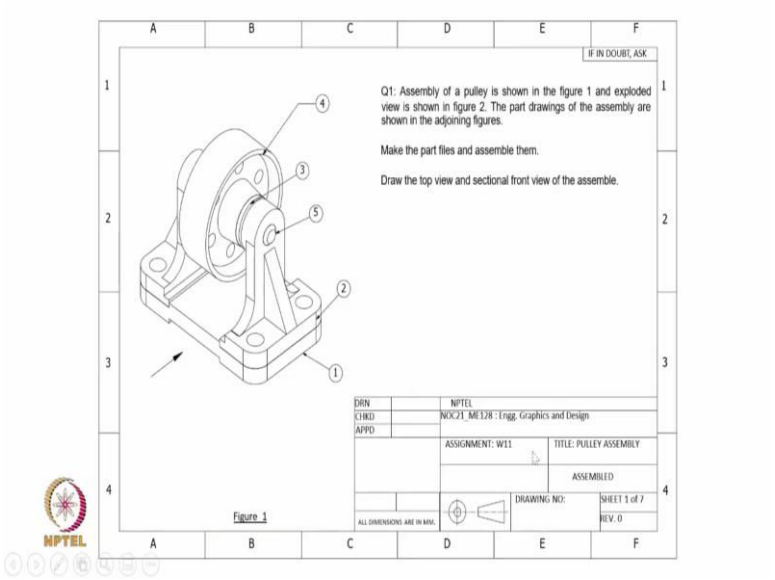

And lastly part five is the shaft which is sitting on the supports at left and right. So, the reason I am showing all these parts is, for me I have already created these parts so I am starting directly into the assembly.

But if you are planning to replicate this example, I want you to go back in the video, pause it, the each of these slides of the parts and create those parts before we get started with the assembly. So, going back to the assembly. So, we said there are five parts. We will go into the AutoCAD inventor and see how to get started with this problem.

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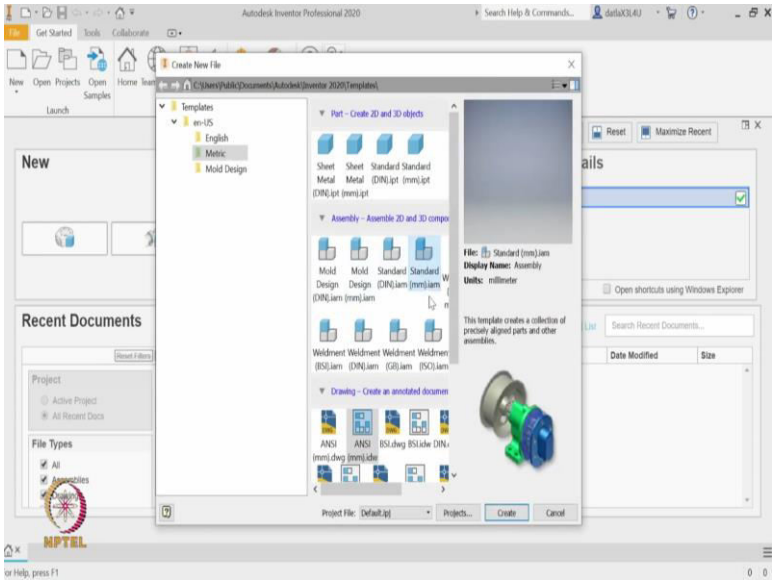

Here is the initial interface of the inventor. Let me open assembly template. Here is the assembly. We will go with the standard mm.iam. Double click on it will take us to the assembly environment.

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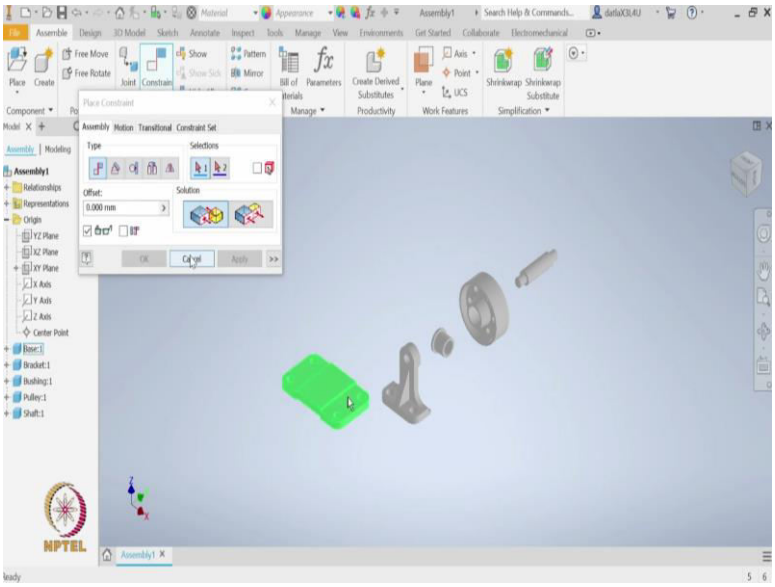

So, once we are in the environment, we need to place the parts. We are following this bottom-up approach where we have already created the part models. We just need to bring it to this space by selecting them. So, I will take the base, bracket, bushing, pulley and the shaft. Let us open it. So, a single click will give me all these five components.

But since we are only looking at one component each as of now, I will right click and say ok. So, first thing I will do is first with the base plate, let me align it with the others. Which means I will make it in the x-y plane. To do that I will go constraint, select the bottom face of this base and for the second I will do a mate using the x-y plane of the origin. So, let me apply the constraint.

So, as with all assemblies we need to make sure one of them is grounded. So, I will select this base component, right click on it and make it grounded. So, which means all the degrees of freedom for the base are constrained.

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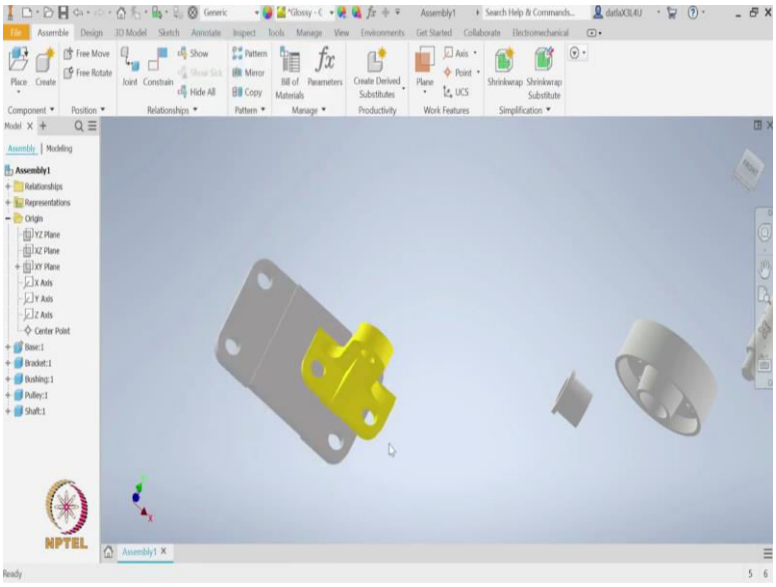

Now, let us move to the second part which is this support. So, first I will start by creating a constraint of mate between this surface of the base and the bottom of the support. So, I will tilt it and select this bottom face and apply the constraint.

Since both are in the same colour it would be helpful if we change the colour of this supports, so I will change it to let us say glossy. So let me zoom in and see how do we constraint the others such that they sit on each other. So, these surfaces are parallel to each other.

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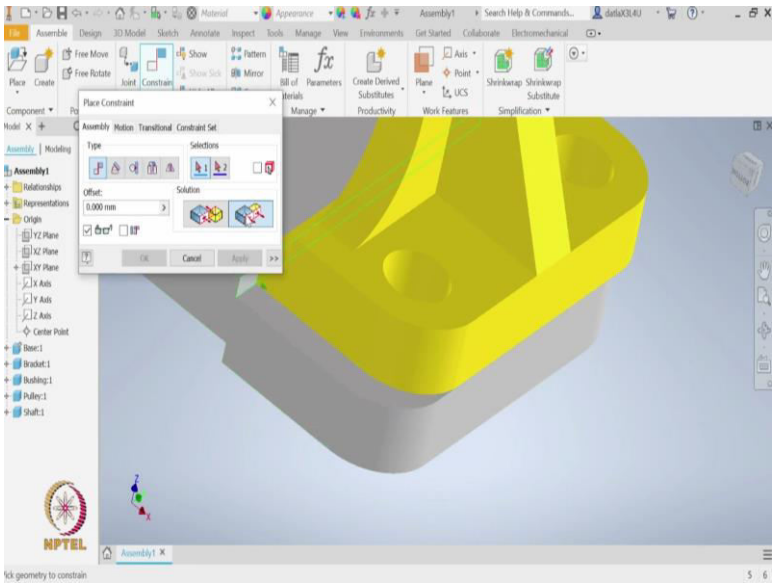

So, first thing I can do is, so maybe I will start with constraint choose flush and select this surface and this surface. So let me apply this constraint. Later if I zoom, I see that this surface

and this surface are not in the single plane. Again, I will use the flush, select surface one, surface two and now they are flushed. Let me apply the constraint.

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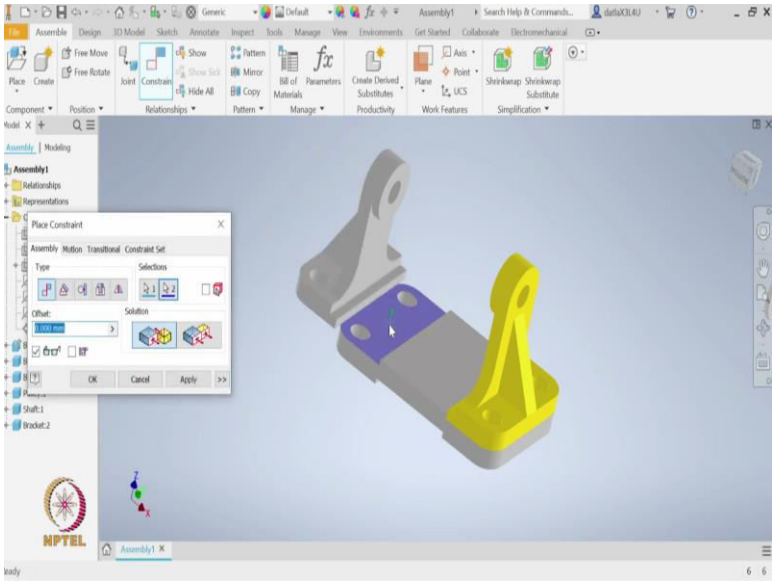

So, now what we have to do is we need to create this left support. For that I will do copy paste. So, I will select this part, control C, control V. So, it has placed it somewhere here. So, how do I constraint it?

First what I will do is, I will make sure this hole axis and the axis of the second support. both of them are collinear. For that I need to go to constraint, again with the mate. This time I will zoom in and select the axis of the right support.

Zoom out and select the axis of the left support. Let us see how it is. Now we see both of them are aligned in the same direction but in the problem, we have that they should be aligning in opposite directions. So, we will go with these opposite directions.

So, I know this surface is coming up but later we can take care of it by mating this surface with the base. But as of now let us apply this constraint and let us say we have this degree of freedom as well as it is free to rotate. We can constraint it by using a mate between this bottom face of the support and top face of the base. We will apply this constraint

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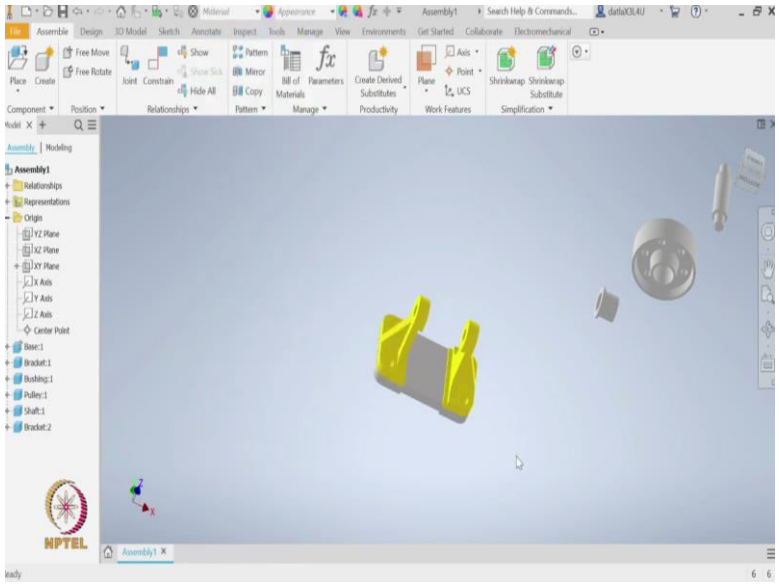

And lastly, we need to apply flush one more time so that they both are nicely aligned. So, for this I will choose flush. This surface on support and this surface on the base. Now we see that even the left support is perfectly aligned with the base.

We will apply this constraint and we are done with placing both the left and the right support. For consistency let us also change this colour to yellow, such that we have enough contrast to differentiate between different parts.

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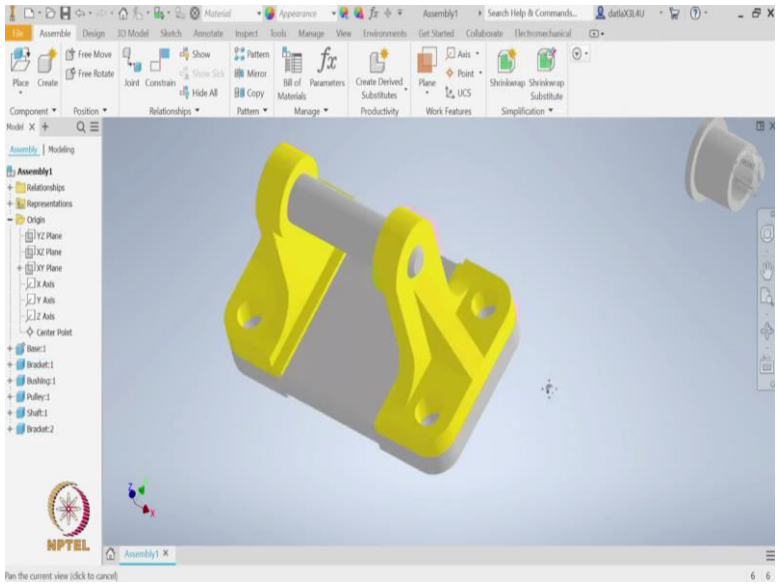

Now what we will do is now let us place this shaft in these holes of the support. So, for that again let me first drag this shaft close by. Let us zoom in and we use constraint of aligning

the axis. So, let me first choose axis of the right support and then axis of the shaft. And now we made sure that both of them are coincident and let us apply this support.

So, we have applied this collinear constraint. Now we see that if I cancel this, now I can see the shaft is free to translate in this direction. For me to restrict it if I go back to the question, I see that the shaft is perfectly at the centre.

One way of applying that constraint is, so maybe after studying the drawing I see that this circle is collinear, it falls on this plane of the support. So, the way I apply it is, go to again constraint, select just the circle not the surface. I select this edge as well as this edge. Now, I see that both of these edges that we selected fall on the same plane, apply. So, now we are done with constraining the base, the left and right supports and the shafts.

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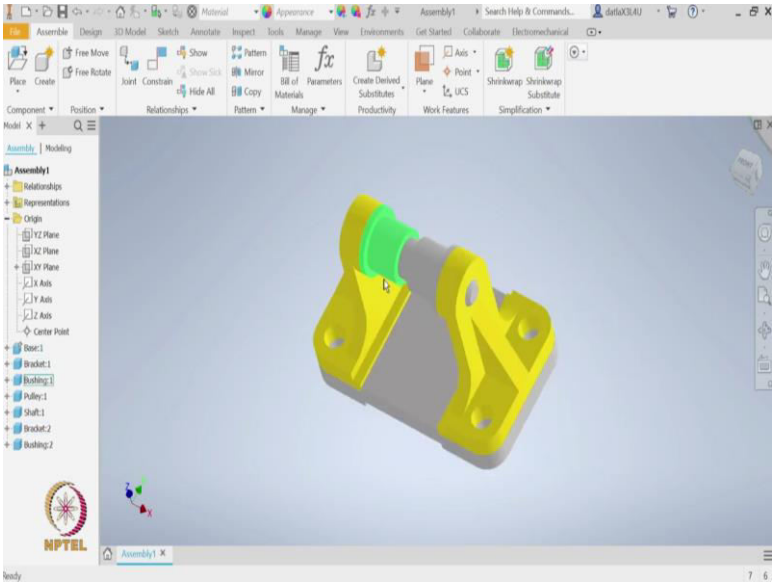

So, now let us get to this collar. First, we can align this collar with the axises like we are doing with the rest. So, let me select the axis of the collar and the axis of the shaft. Now, let me apply the constraint. Next again going back to the drawing, we notice that this collar should be touching the left and right supports.

In this case it should go and touch the left support. For that we will again apply a constraint where we go with the mate and again only select the edge from the collar and the edge from this part of the support. And let us apply the constraint. Next, we also know that there are two of these collars. So, let us copy paste this control C, control V. Now we got the second collar.

To apply the constraint first we will apply the axial constraints. So, let us choose the axis of the collar and then the axis of the shaft. This time I know that the collar should be in the other

orientation. For that I can go with the opposed solution. So, now I see that the axis of the collar is at 180 degrees with respect to initial collar. So, let us apply this constraint.

But now we notice that this collar should be matching with the face of this support. So, for that I will select only the edge of the collar and then the edge of this support and then mate each other. Let us apply the constraint. With this we are done by applying constraints to the base, left and right support and left and right collars.

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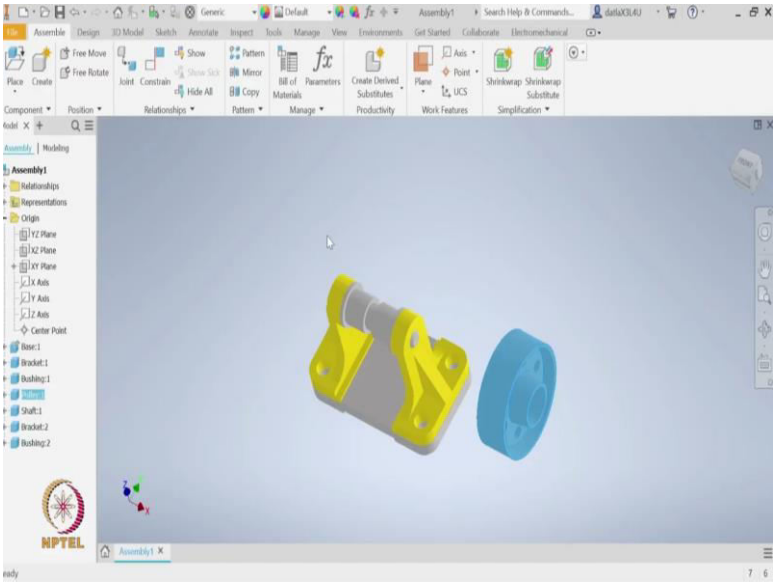

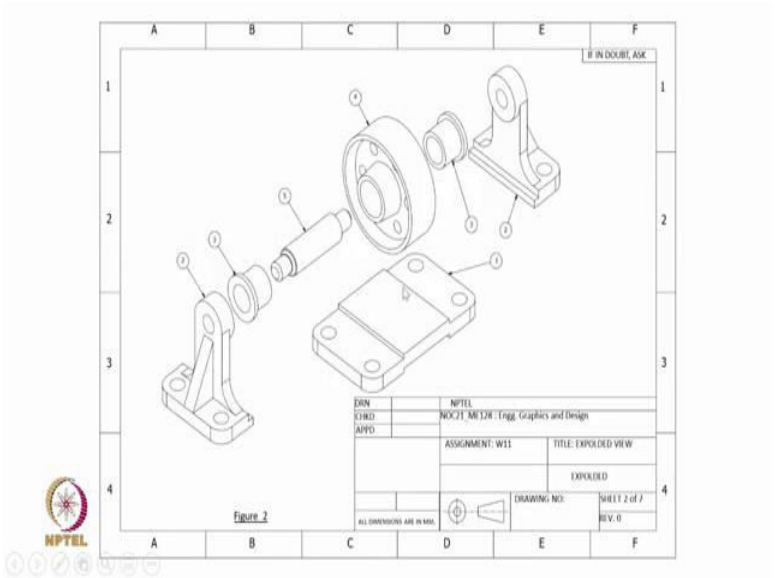

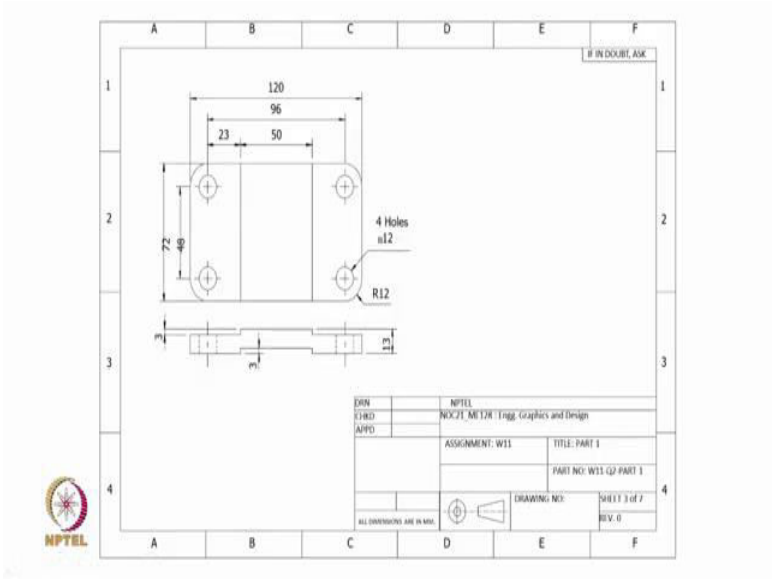

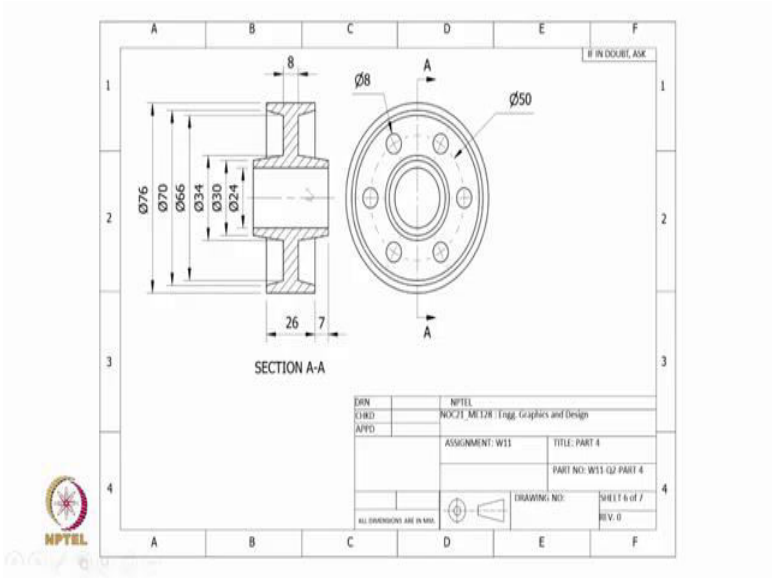

The last thing remaining is this wheel. So let me bring the wheel close to the assembly and then let us see. So, again with the similar to the rest of the constraints, here too we will first start with the axis of this pulley and align it with the axis of the shaft. And let us apply this.

So, now we see that this, there is one degree of freedom, this translation degrees of freedom which we need to constraint. To apply that let us study the problem one more time. If you look at here, we see that this pulley should be falling in the middle of the base plate.

So, let us go to the base plate drawing which will give us clear idea. The length of the base plate is 120, so the centre plain is at a distance of 60 from the edge. But let us also study the pulley. Here is the pulley. We see that the total width of the pulley is this 26 + 7 + 7 on each side, so in total it is 40. So, which means the total width is 40 and the half width is 20.

So, we will use this half width 20 as a reference dimension to place this pulley with respect to the base.

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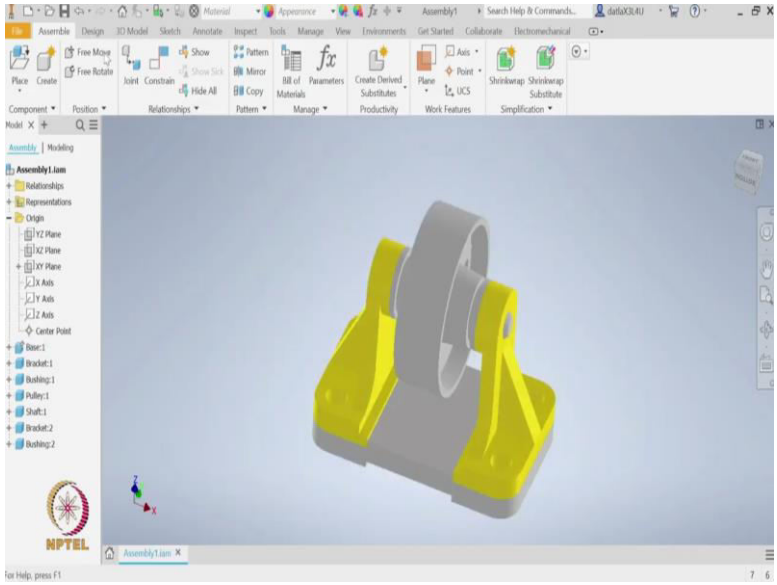

So, going back to the problem. So let us apply constraint, zoom in. From this edge of base we will specify that this face of the pulley is at a distance of 40. So, let us apply 40. Maybe we need to apply - 40 because it should go to the other direction.

So, once we apply, we see that this problem now matches exactly with the question we were given. So, we can rotate and see that now we have all the components in the right orientation and the position. So, let us save this file. So, let us name it as assembly 1. So, we have saved this assembly.

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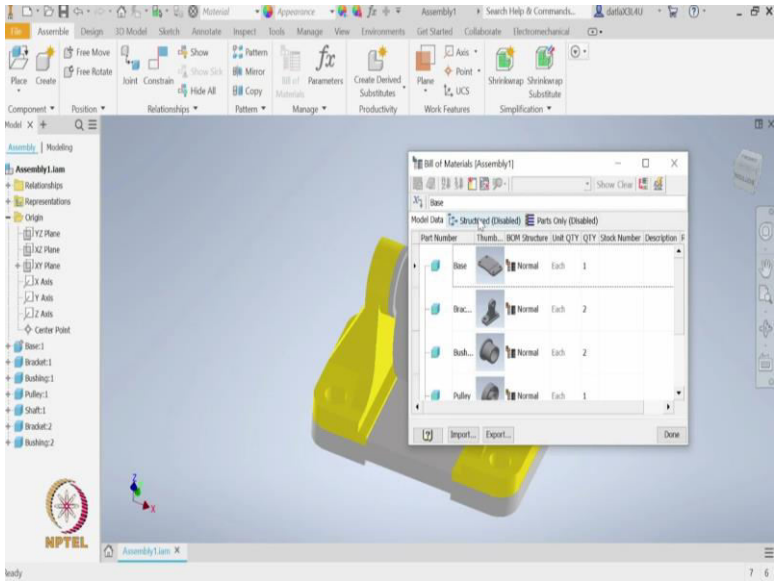

So, let me also go and check this bill of materials. We see that now it is showing all the five components. It also shows that the quantity of this bracket and the bush are twice. So, let us go to the structure, right click and enable the bill of material view. Similarly for the part only, let us enable the bill of material view and let us say done. Let us save it one more time. So, let go back to the question and see that in the question we were asked to draw the top view of this assembly as well as the sectional front view of the assembly.

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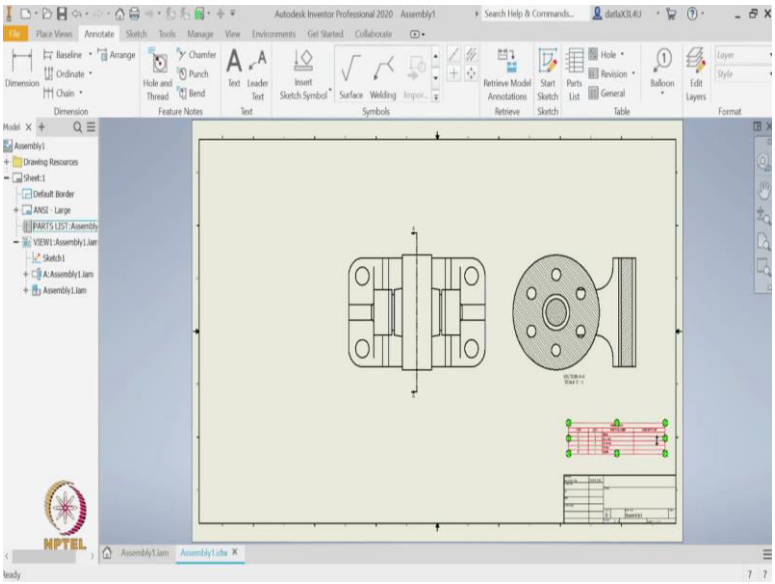

So, for that we will first open a drawing template. So, we will go with this ANS(mm).idw and start by clicking on this base. So, since my assembly is already open by default it has open otherwise, we know that we can browse it and open the assembly. So, I see that maybe bigger scale would be better and let us place it at the centre, okay.

So, now we got the view of the assembly. For us to create the section, let us first select the base and draw the section line, right click and say continue. So, now depending on from which direction you want a view. If you are viewing from left or right, we can place the section view. And lastly once we are done with this, let us save this assembly, save this drawing. And we can also place the bill of materials by choosing this part list.

I will choose this base, click okay. And then it is asking where to place the bill of materials. We will place it here. So, let us zoom in and see that it has all the five parts and the quantities for the brackets and bushing it is showing as two because for those we have copy pasted.

So, with this we have completed creating the assembly by defining the constraints as well as extracted the sectional and the top views along with the bill of materials. With this we conclude this lecture. Thanks for your attention.