Engineering Drawing and Computer Graphics
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Lecture - 58
Solidworks (Contd.)

Hello everyone, welcome to our online lectures on Engineering Drawing and Computer Graphics provided by NPTEL. In today's class, we will learn how to use some of the Solidworks commands to construct 3D geometries.

In today's class mainly we will construct, a cylinder, a tube, and a hearth, how do you construct these things using Solidworks. Thereafter, we will go with assembly drawings, constructing something named a shaft and disc arrangement, and another one is a rectangular bar fit in a square hole, how to assemble these things we will see it.

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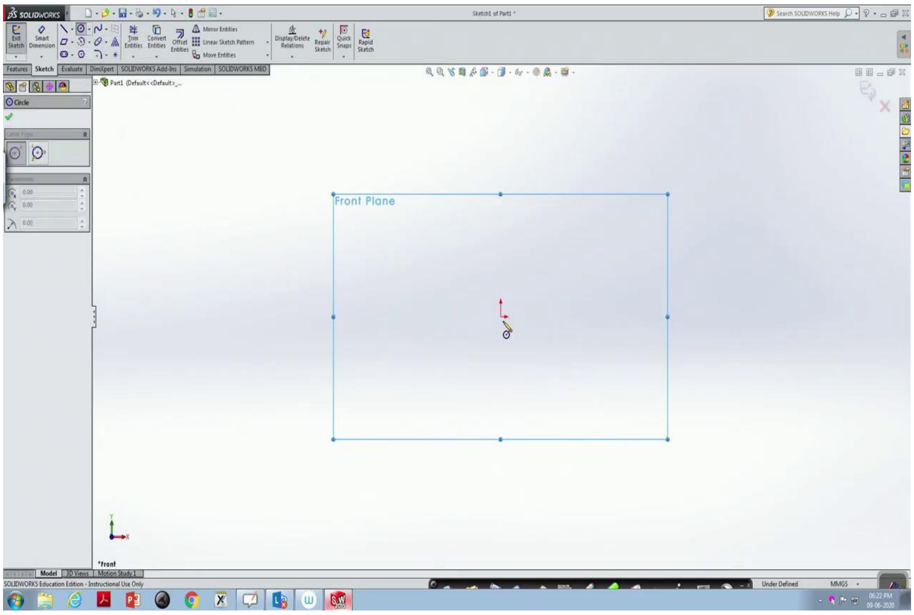

So, after double-clicking our Solidworks, opening a new part drawing we will have this white blank screen. In this, we use a revolve command to construct a three-dimensional cylinder. The first one is we always begin either with the front plane, top plane or right plane. So, front plane click, normal to the front plane we go. Cylinder, we would like to construct.

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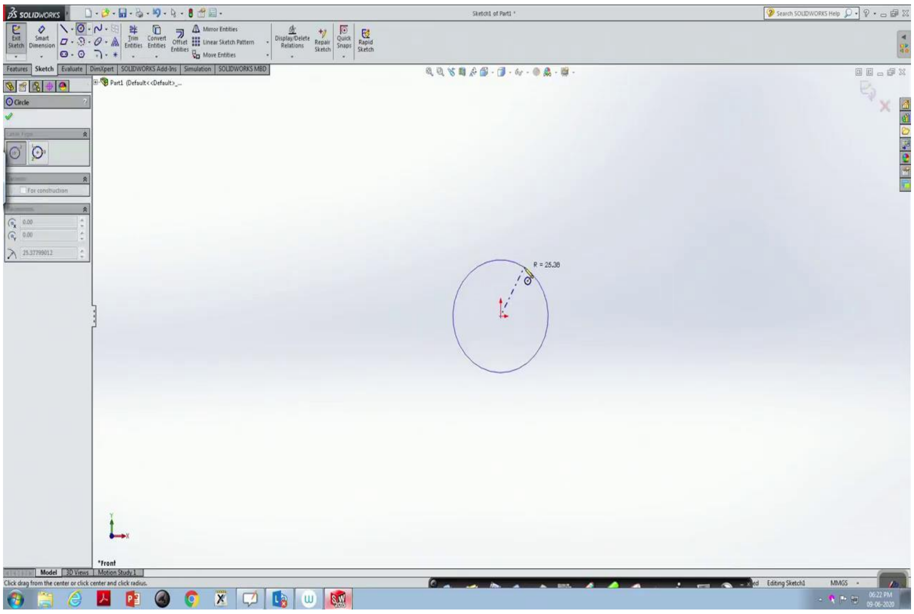

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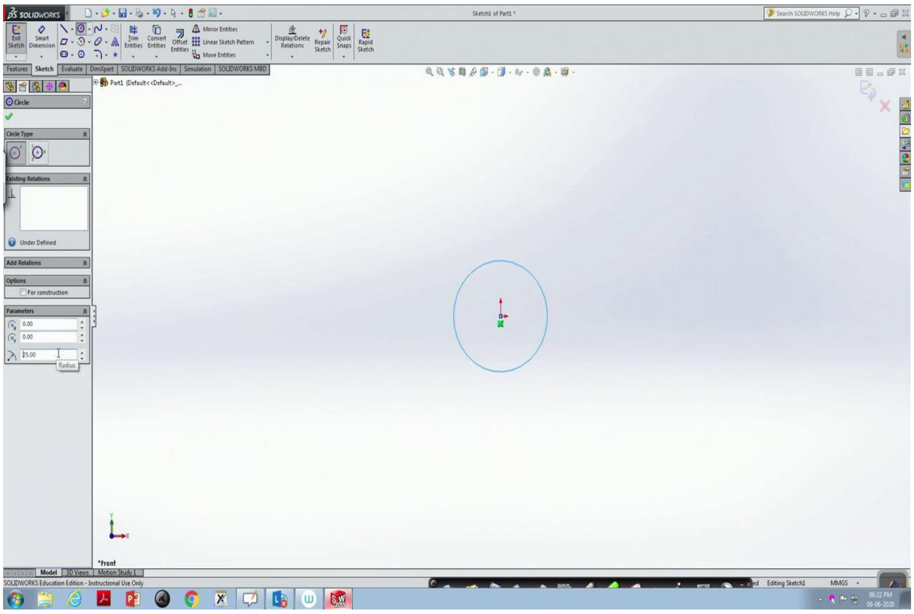

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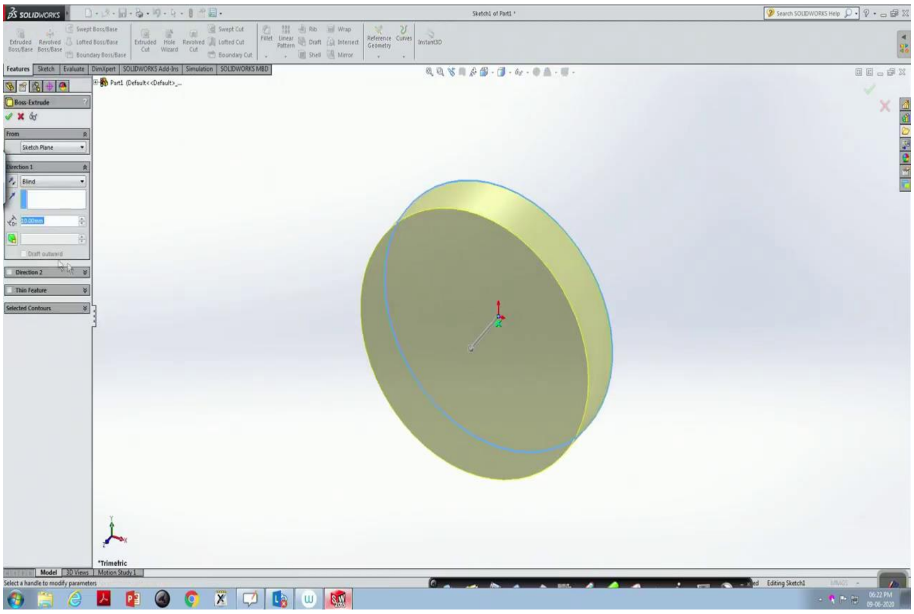

So, in the sketch mode go to a circle locate some point and change its radius to 25 units. Go to features extrude boss, go to 50 units click ok.

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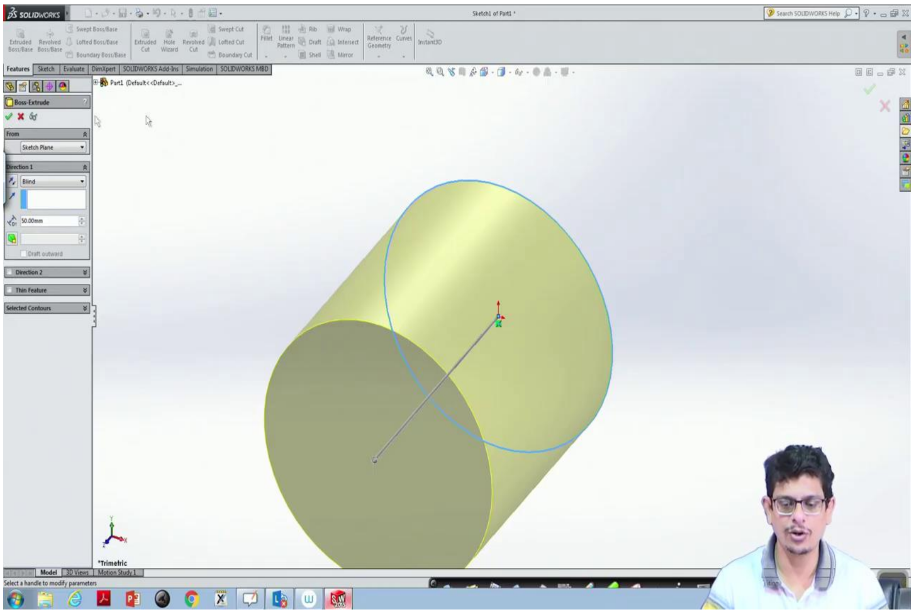

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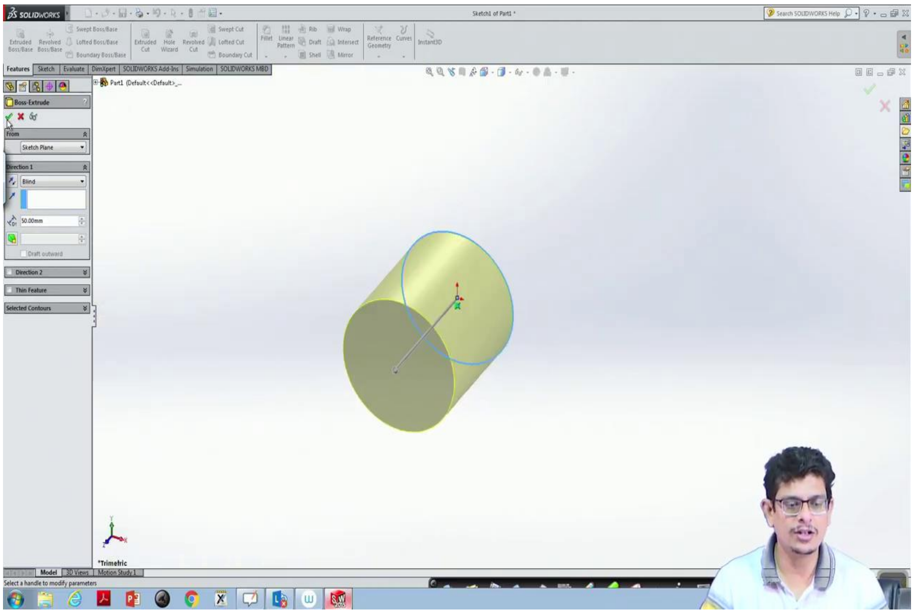

And then click ok.

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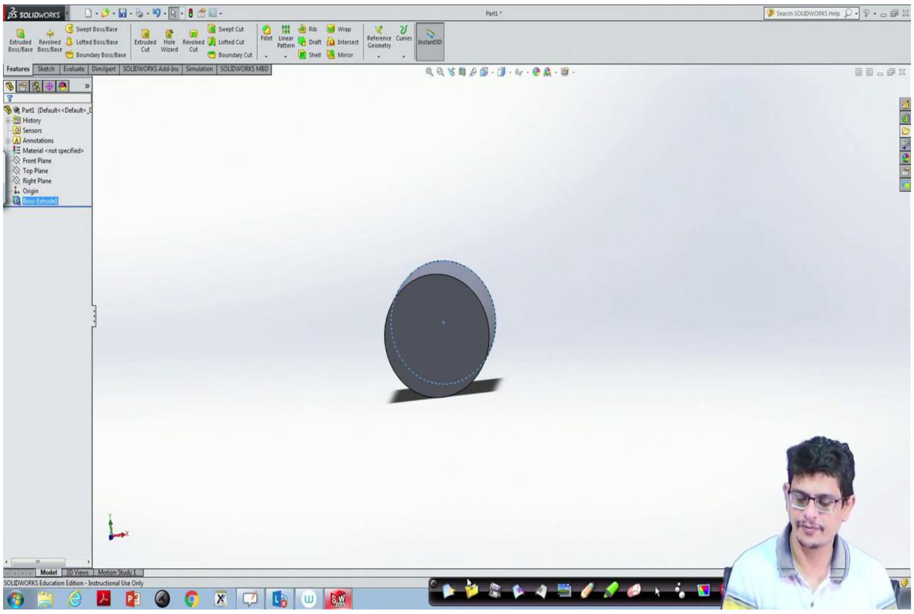

This is one way of constructing cylinders by using extrude boss base. One can also construct using a revolve command. For example, if we have, if I have a centre line, if I can construct a rectangle, rotating that rectangle by 360 degrees also, we will be in a position to construct a cylinder and that is the thing what we will see.

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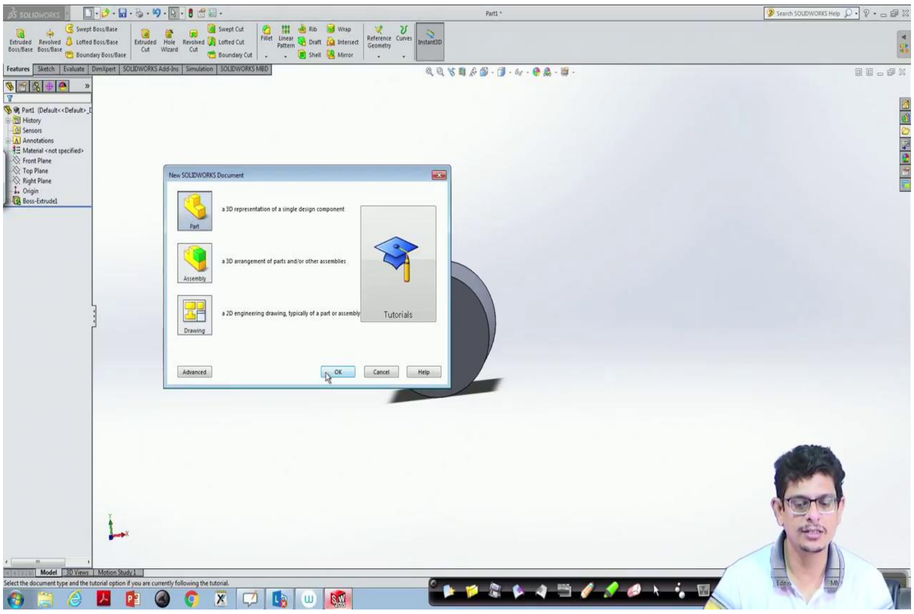

So, a new one, part ok, go to the front plane, normal to it. In the sketch mode, first of all, draw a centre line.

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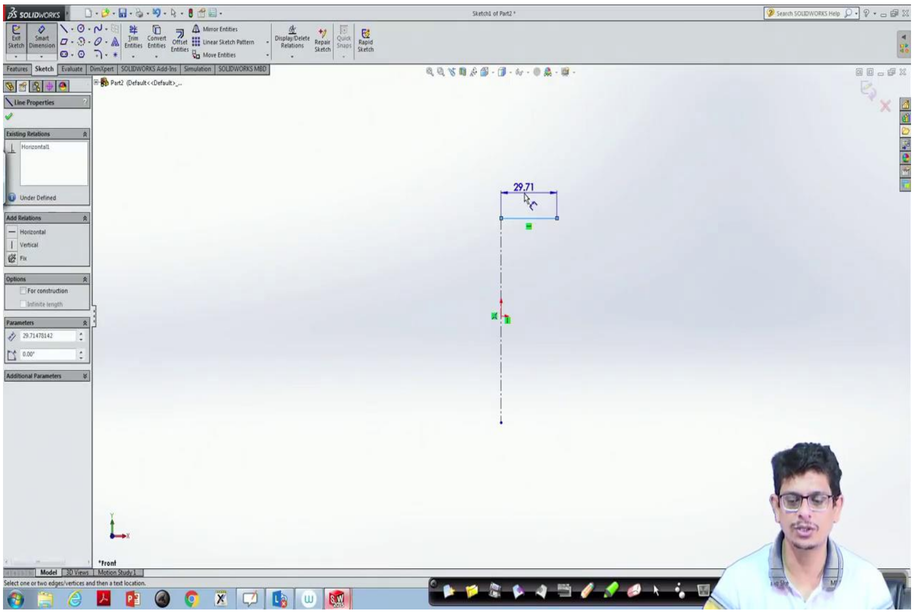

Once it is done, we would like to revolve a cylinder. So, for that purpose, I am drawing a line perhaps the using smart dimensions.

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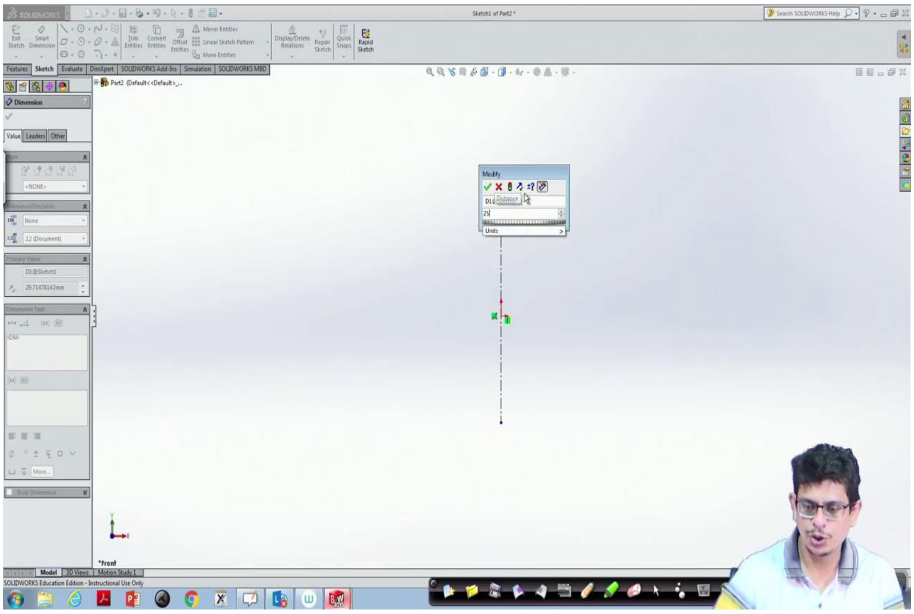

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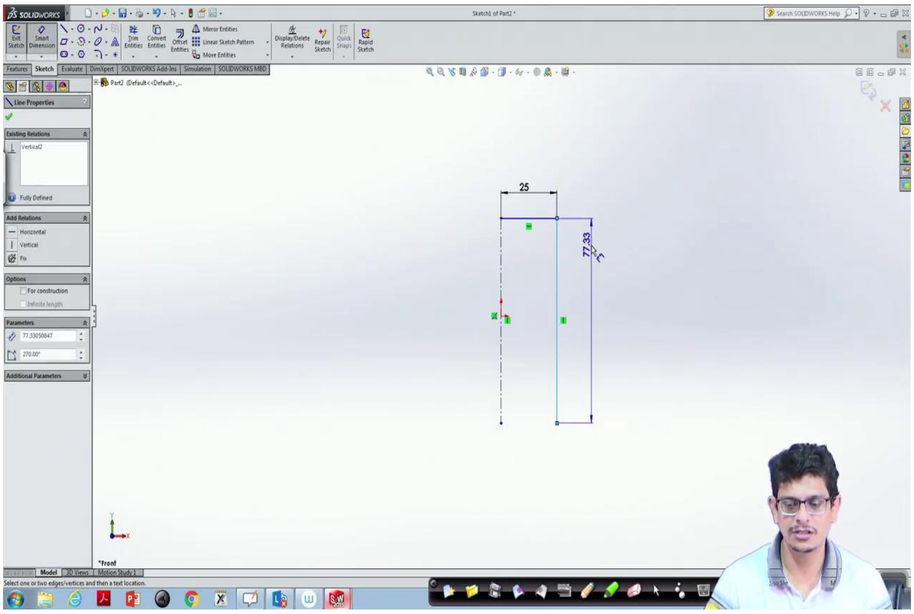

So that I will have better control on radius 25 units of length, I will draw. Then extend it from there to there. Use smart dimensions, make it 75 units, then again construct a line, goes there.

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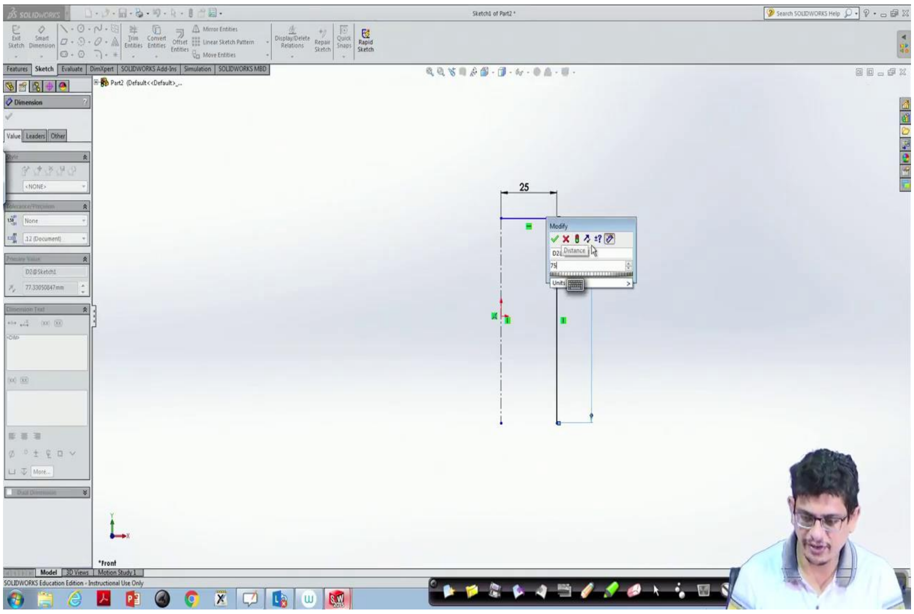

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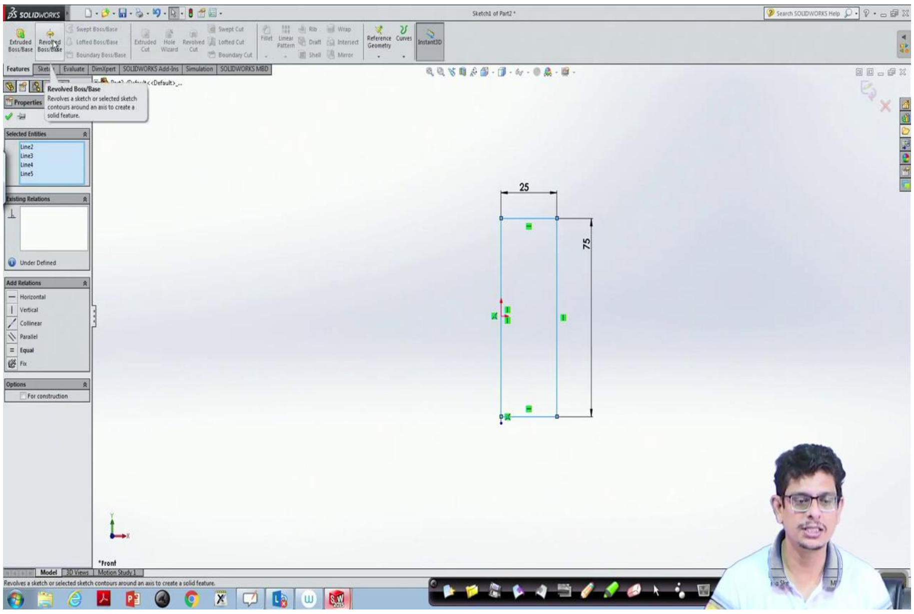

So, for revolve command, we always require closed objects. So, once it is done, use the control button, hold that line. Similarly, click the other line by keeping the control button, control hold and line.

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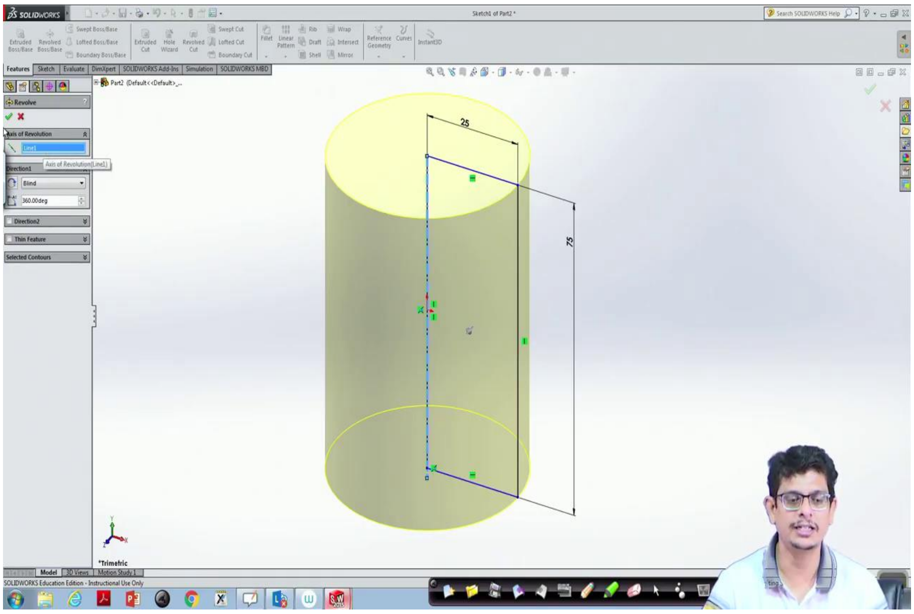

Now, go to features revolve boss base around this axis we would like to have which is automatically selected as line 1 because the first line what we have constructed is that centre line. So, it is automatically selected. If that is not the case we go click that line then it will be selected there. Then click ok.

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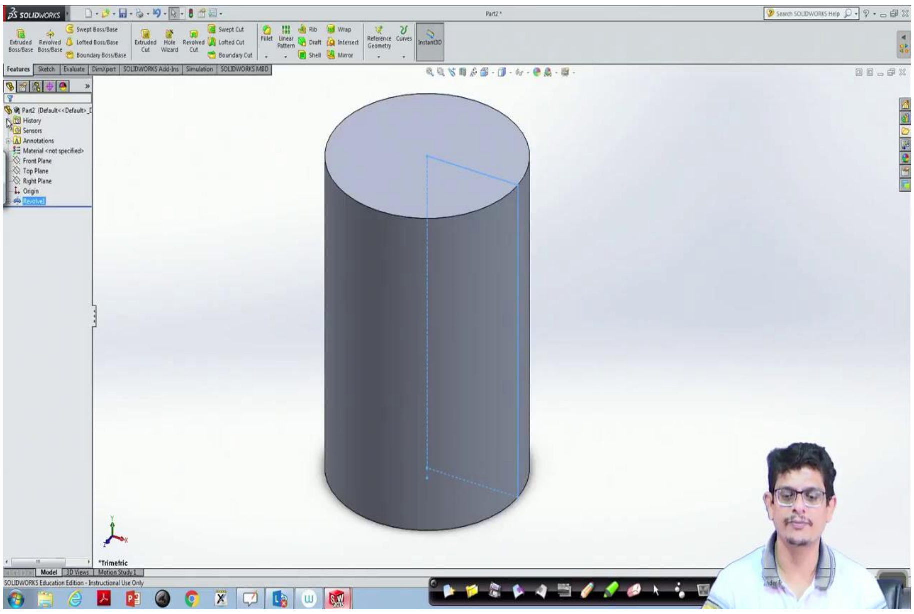

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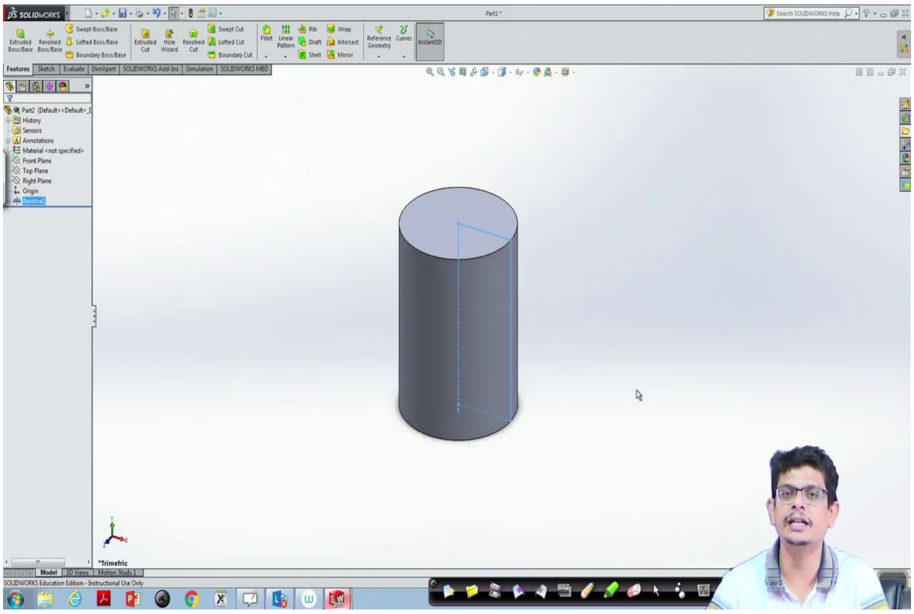

This is another way of constructing a cylinder. Now, using this revolves, we can construct axis-symmetric geometries. Let us begin it in this way. Now, a new one, click part ok, now go to the front plane, now we will construct an axis-symmetric object, line.

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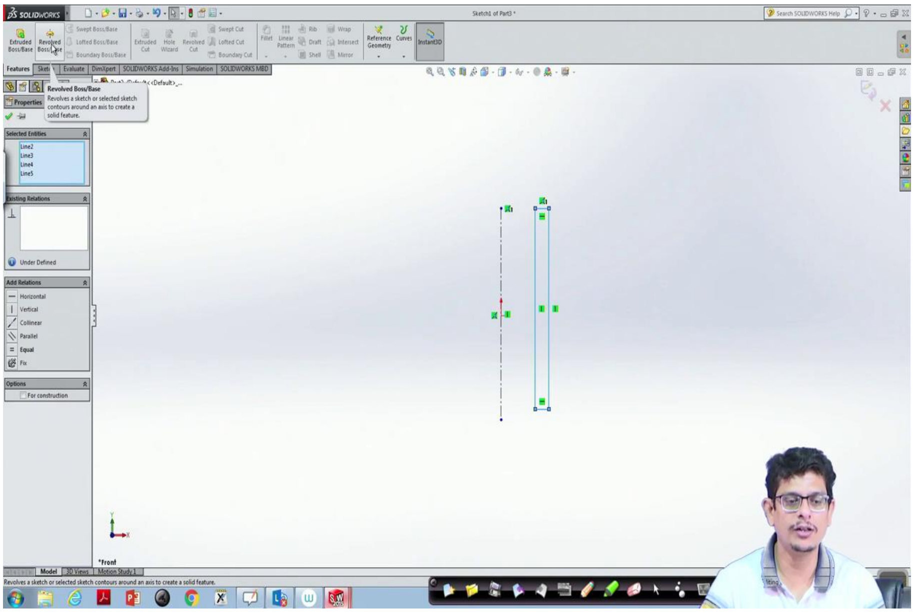

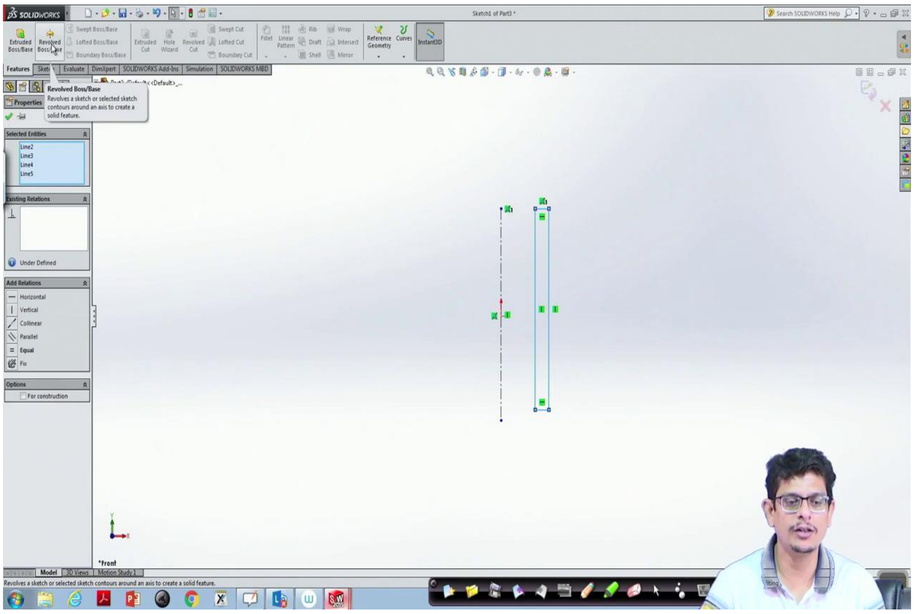

First of all a centre line, this is always the first line ok. Now, construct another line, because we would like to have a hollow cylinder tube construct that. So, a centre line we have constructed, and a rectangular patch. Now, use control button, click, select this one also.

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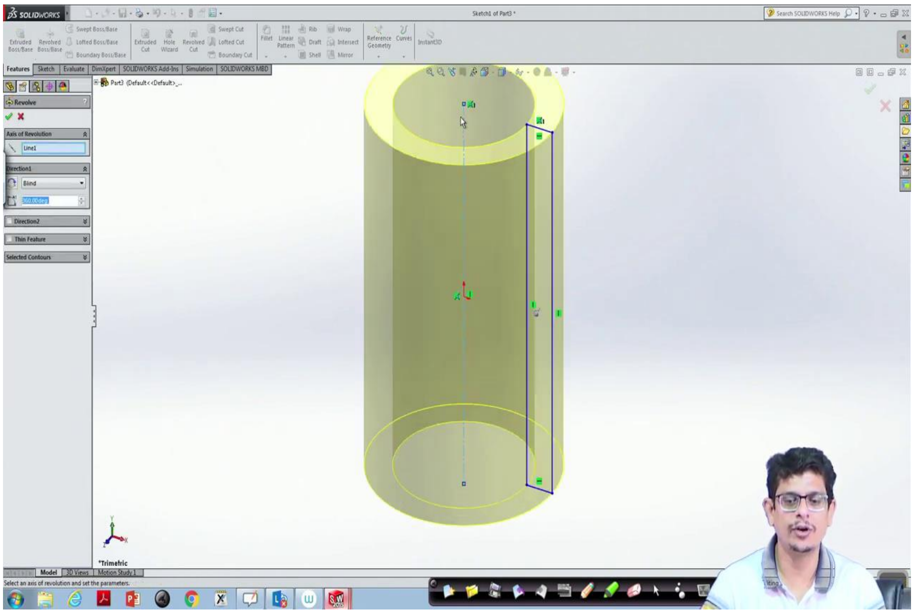

Now, go to features, click revolve boss base, above that centre line, now click ok. So, we have that hollow cylinder.

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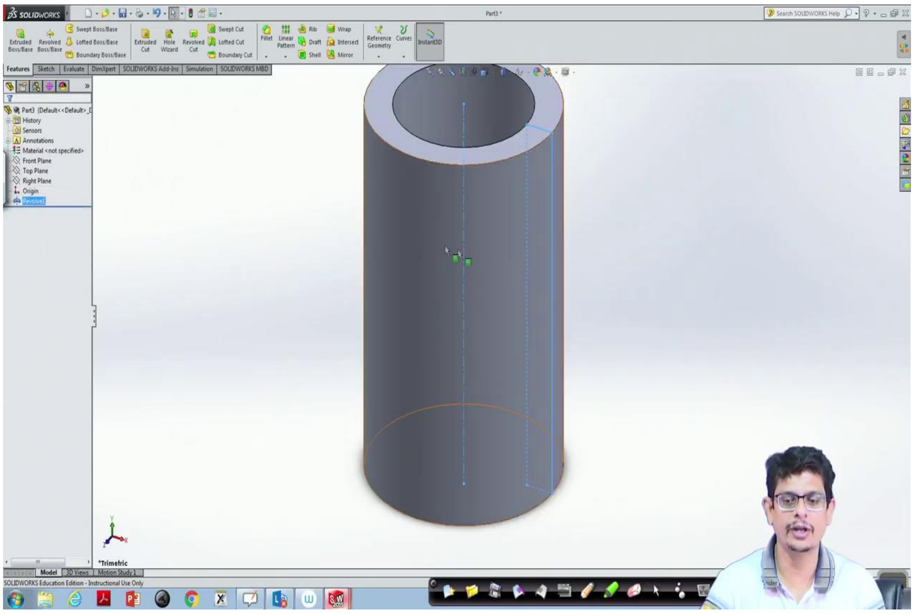

If we are interested in the stepped cylinder what we have to do?

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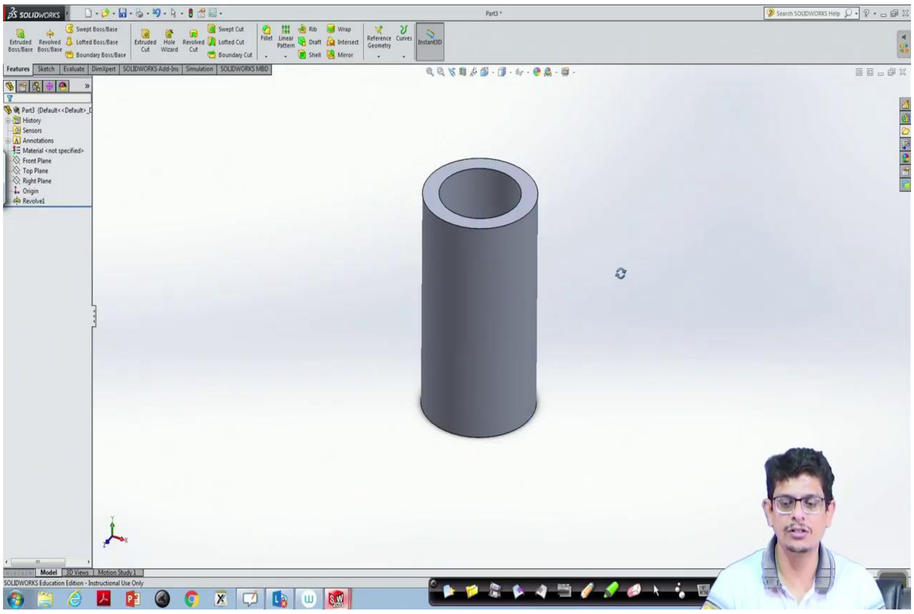

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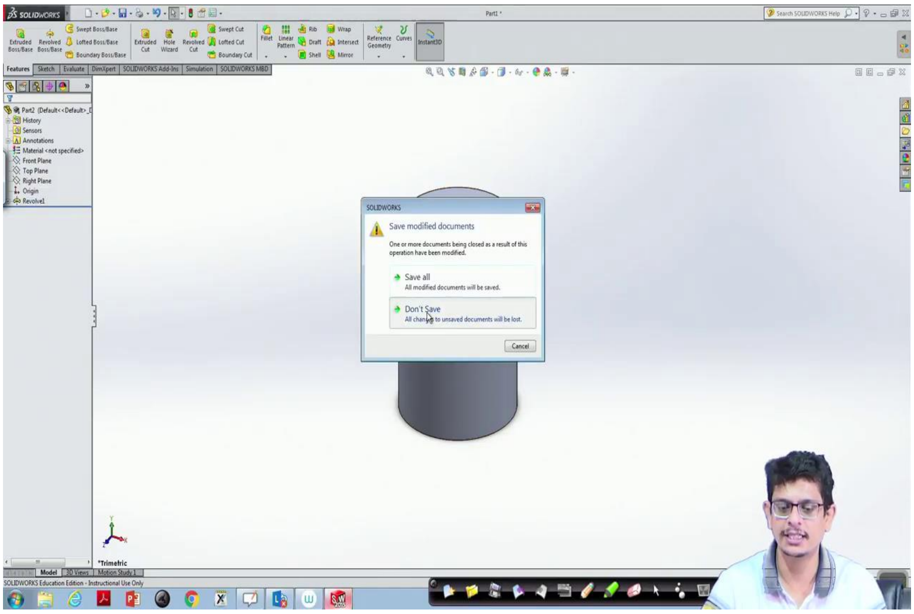

If you are not interested in saving geometries, do not save, click ok, do not save. Now, open a new one. We would like to construct a stepped cylinder using revolve.

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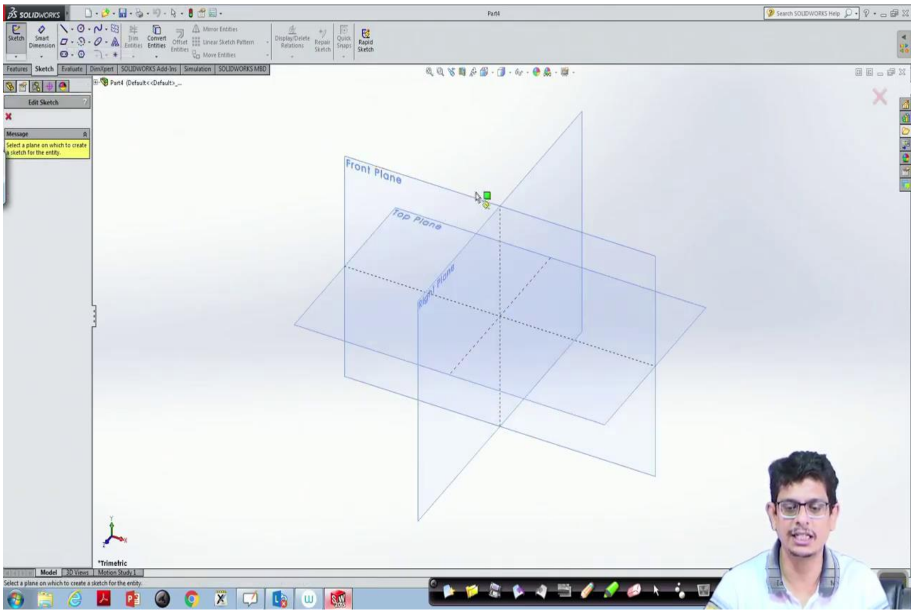

So, click part, ok, go to sketch. The first step is always the centre line on a front plane, normal to it.

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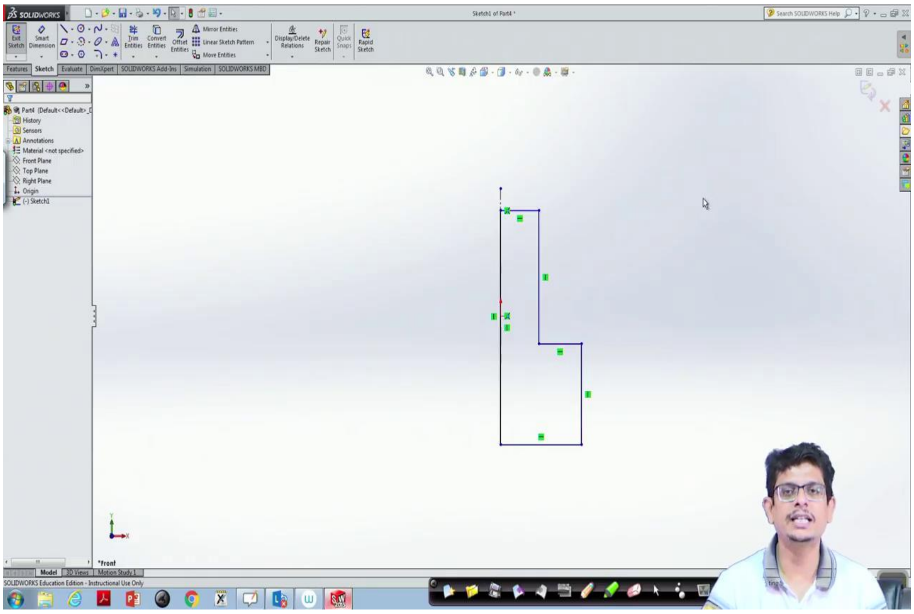

From there to there, draw it is a stepped cylinder that means, we have to construct a line goes down, from there again goes down, click ok. Then from there, join this one.

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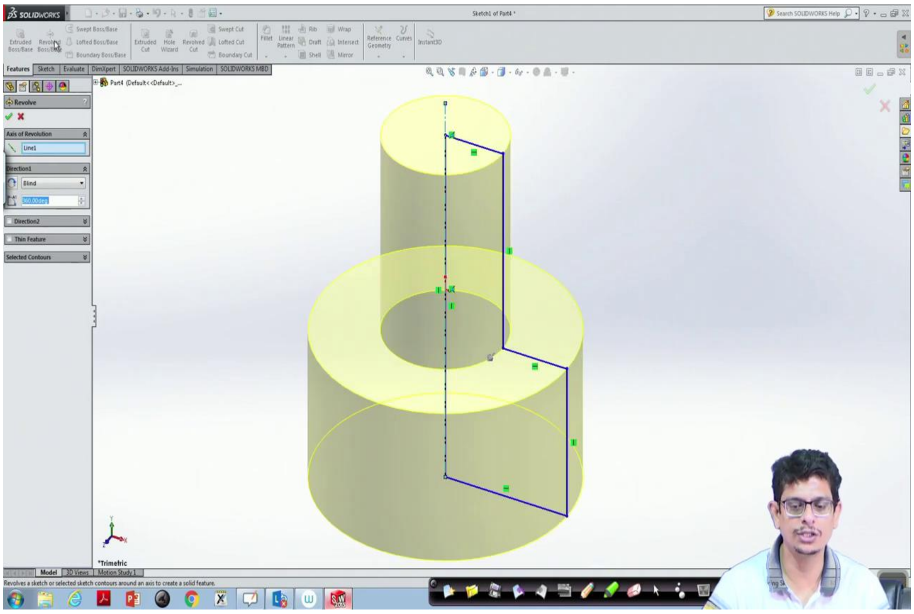

So, this entire object is done.

Now, go to features, revolve boss base, it is revolving around this centre one that is the thing what we can see the axis of revolution as this line one. If we are interested in about 360 degrees, we go with 360 degrees. Now, we are interested only 270 degrees, we click 270, ok, then click ok.

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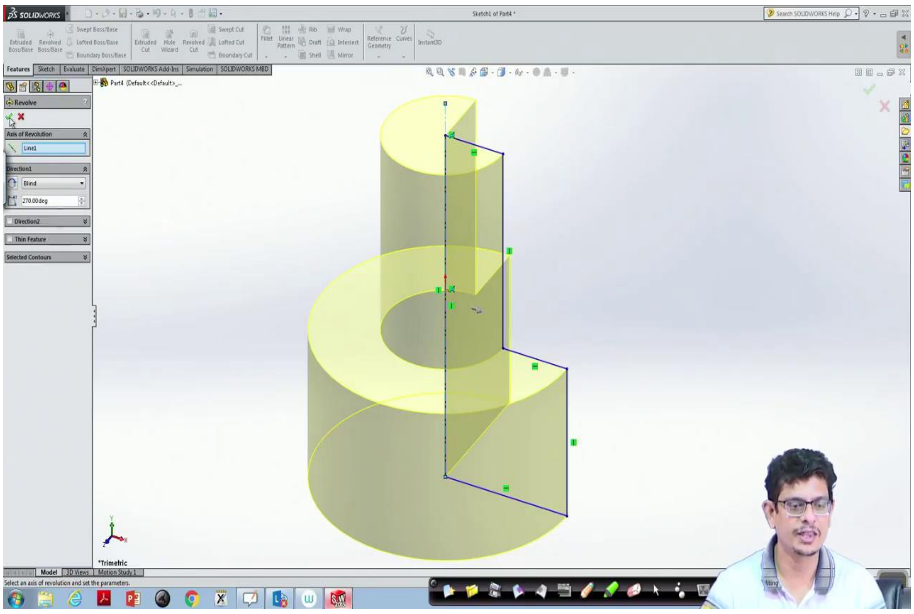

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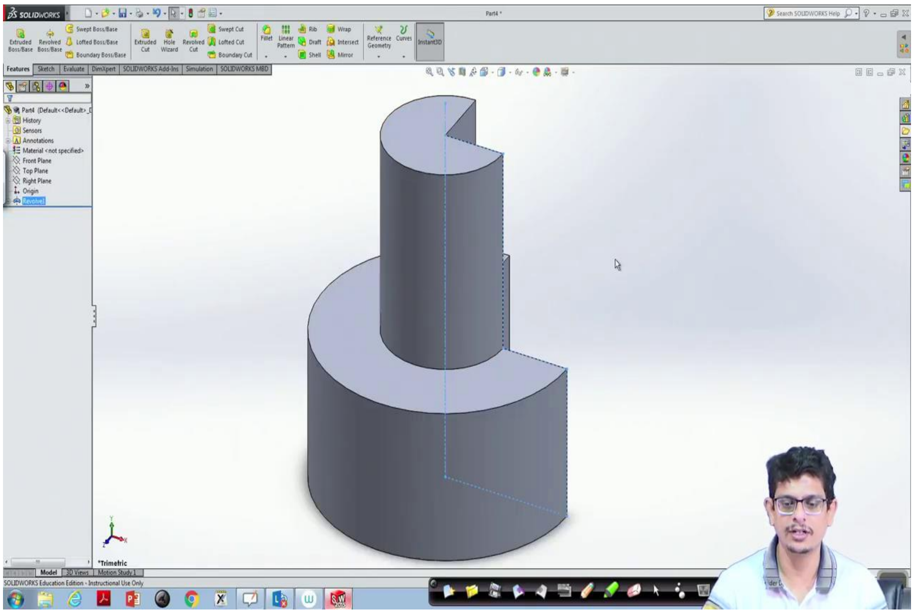

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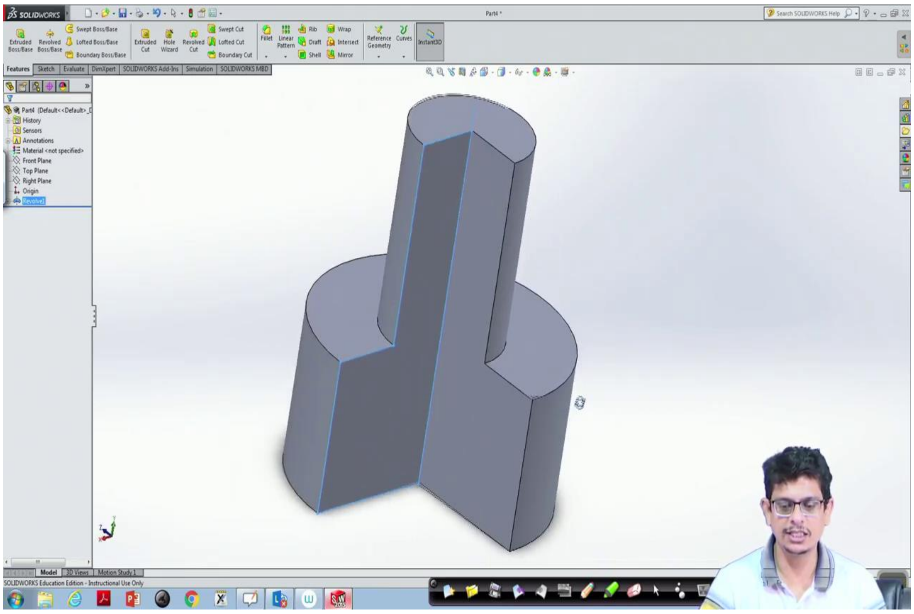

So, when we revolve because it is 270 degrees thing we see this object. Now, let us look at the isometric view. To visualize isometric view, what we have to do, go here, click that one.

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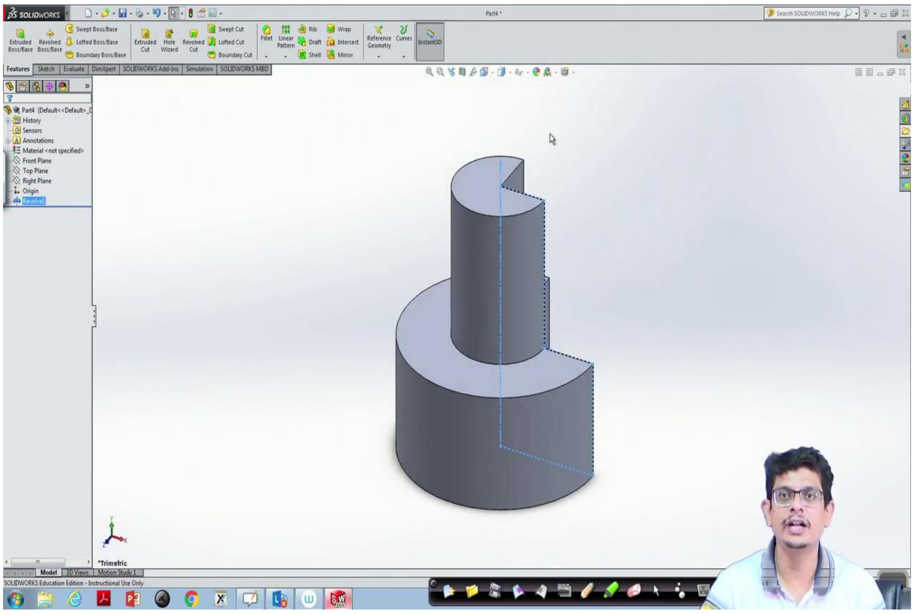

Any other view like diametric view, trimetric view, it will be in that way. Now, if we are interested in see different orthographic views, here we have different things something like single view, two views horizontal, two views vertical, and four views. Let us click this four view.

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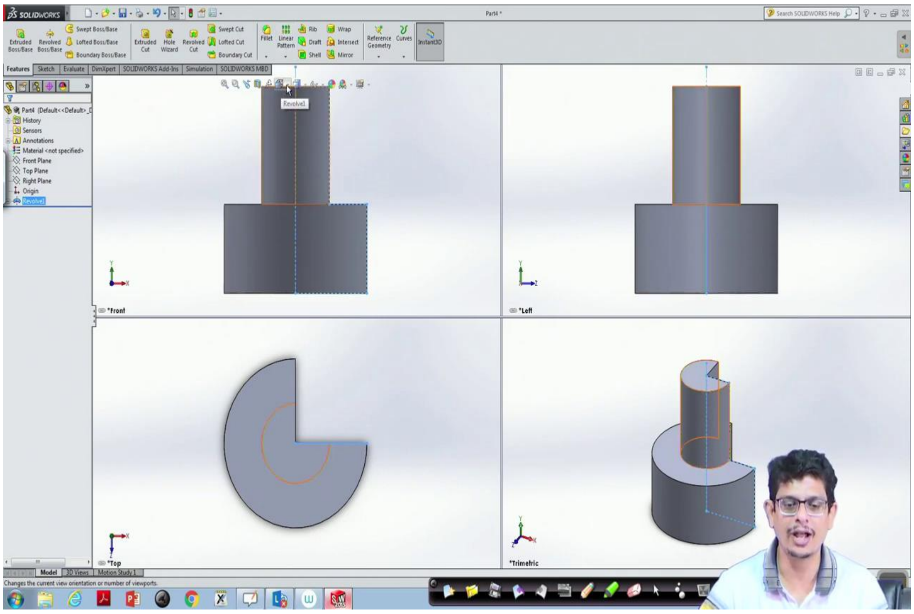

What we can see is something like the front view here, top view here, from the left side from the right side to the left side if you are seeing the left side view what we are seeing there, and whatever the trimetric view. So, all the pictures at a time we can get as different views. Now, if we want to go back, what we have to do, go there click the single one.

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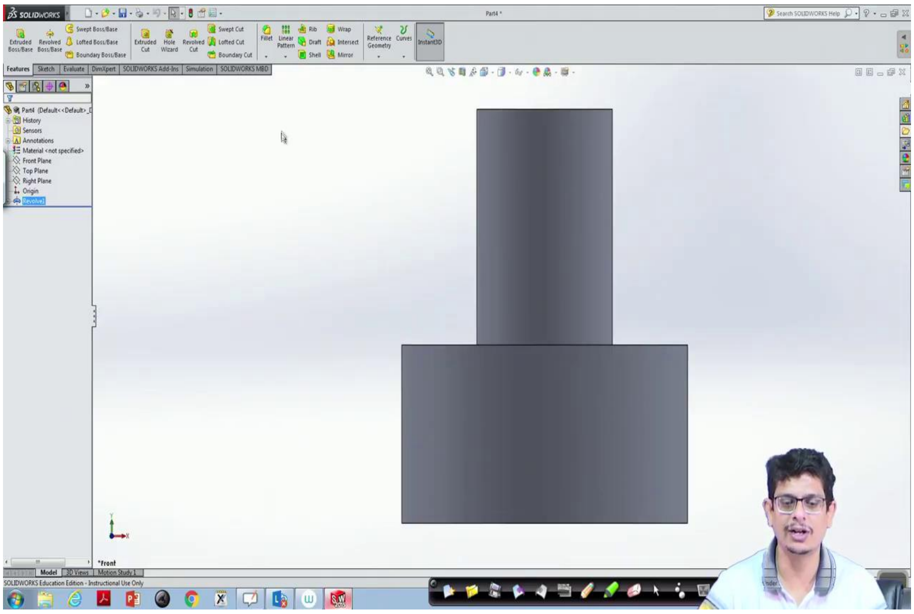

In that single one also, we are more interested in isometric view, then click that. This is the way a revolved thing we will construct. Now, a stepped hollow cylinder if we are going to construct, the procedure is going to new file part ok.

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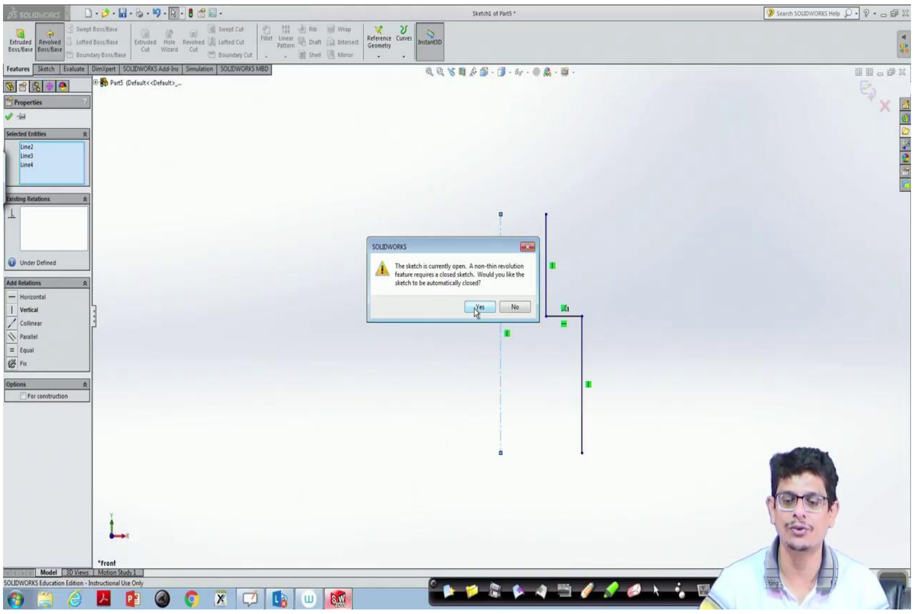

Now, sketch a centre on a plane front plane, go there. now click a centre line. It is supposed to be hollow. So, what I will do is click one, I would not construct a complete sketch, but something like a line only we will construct see what will happen. So, it is not a complete sketch, only lines I have constructed, not a closed one.

Now, what I will do? I will select this entire one, go to features, try to revolve boss base, then it says because it is not a closed sketch, the sketch is currently open. A non-thin revolution feature requires a closed sketch. If you want to give a solid visualization and so on, you have to give the thickness. Would you like to sketch to be automatically close? Let us try what will happen.

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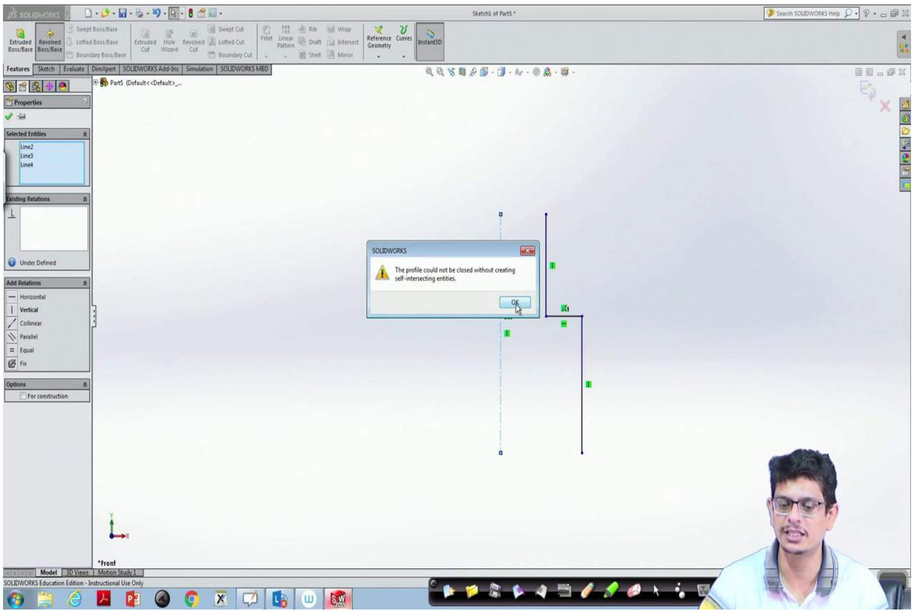

If I click yes the profile could not be close without creating self-intersecting ok.

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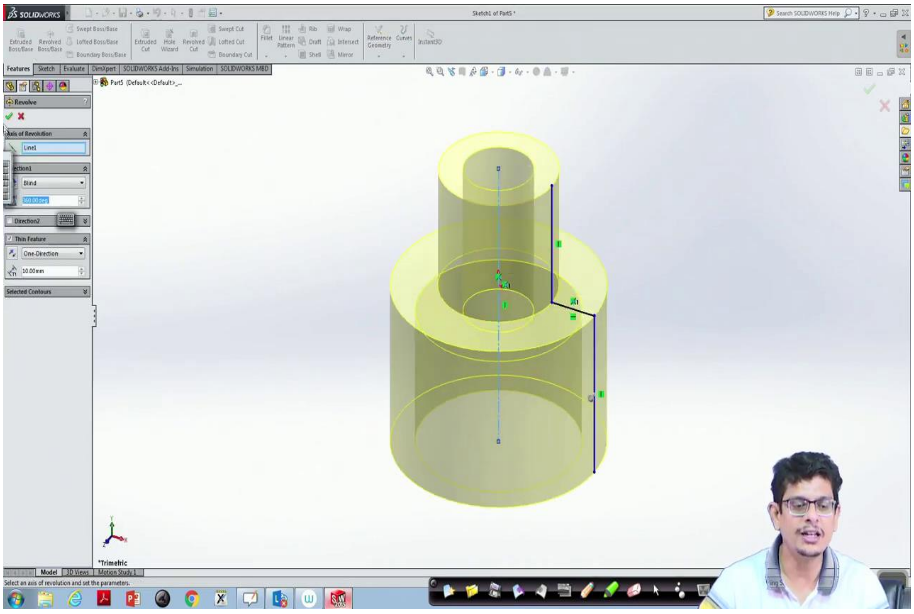

Then it picks automatic dimensions where 10 mm you can give it something like 0.5 mm also.

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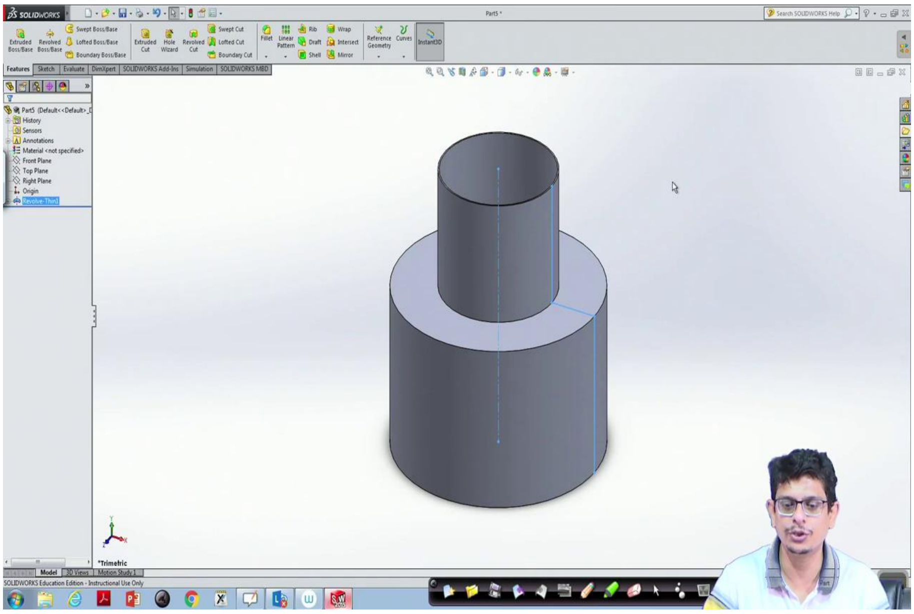

A very thin shell one will be in a position to construct something like a shoe.

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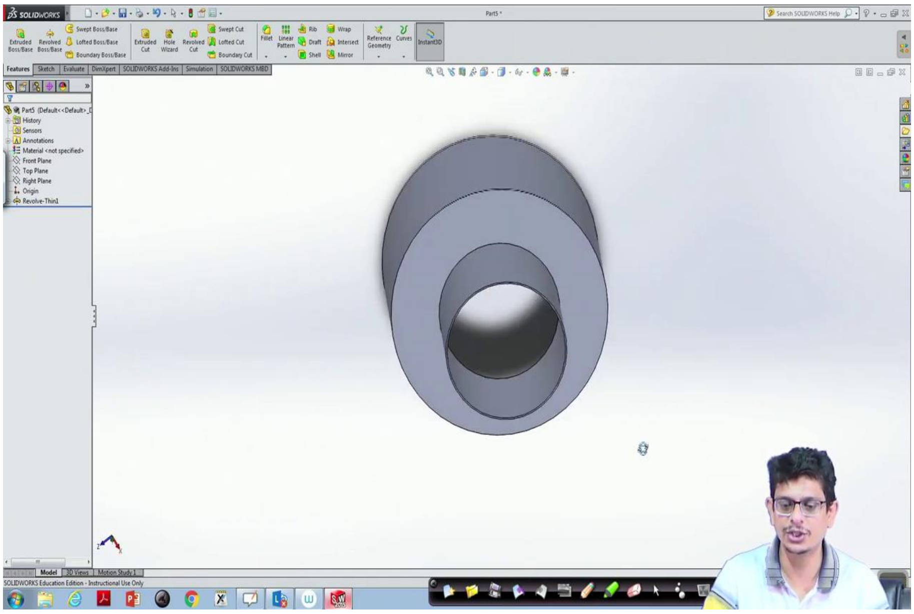

So, if you are not giving any closed entities, it tries to have this back end program which automatically assumes certain dimensions try to construct this. So, this kind of ducts where the thickness is very small compared to other dimensions can also be possible.

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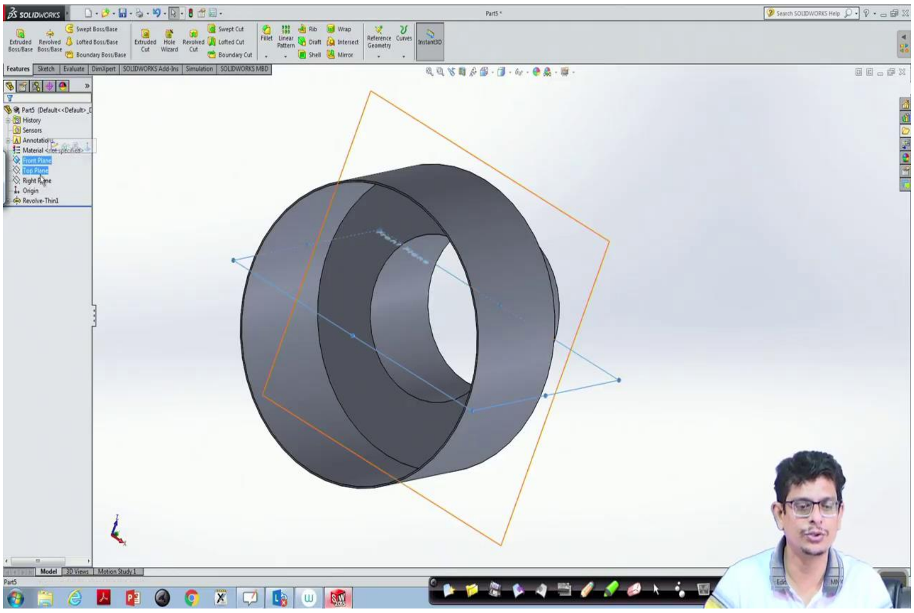

So, if you want to close this one something like a cane, water cane kind of thing, what we have to do is perhaps pick this circle. On that, again extrude the portion and fill it. So, let us try that. So, let us go to the top plane. Especially normal to this plane, let us go there. Now, go to sketch, we have to carefully select this thing ok.

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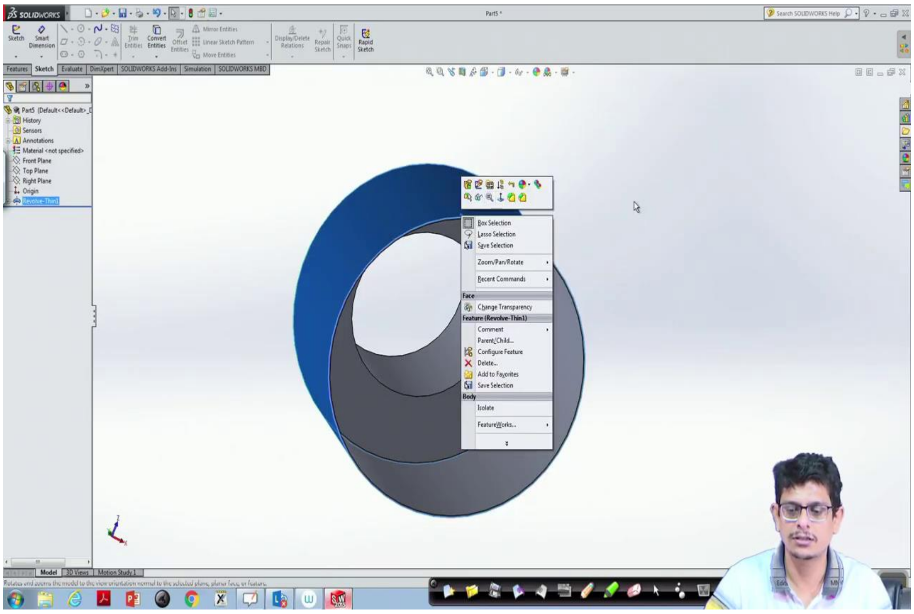

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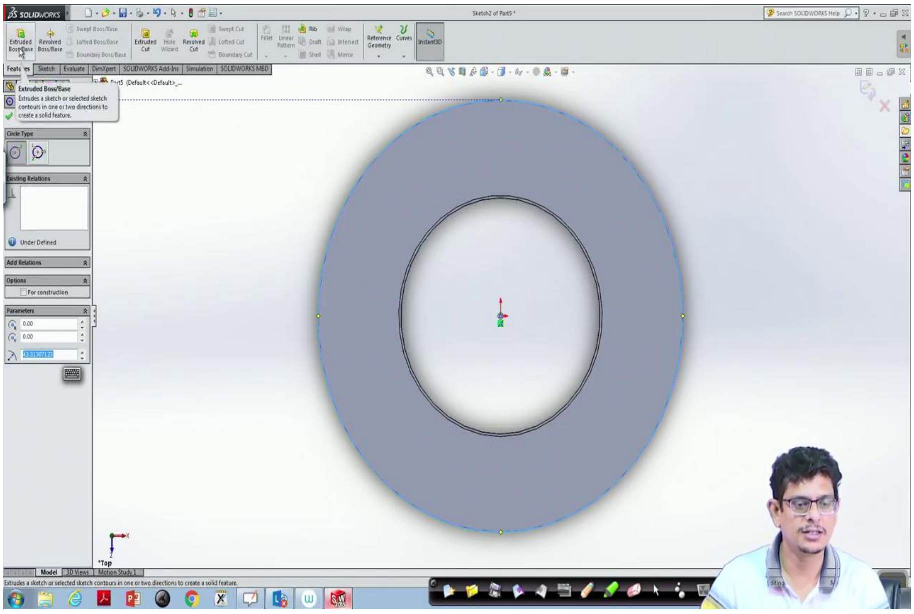

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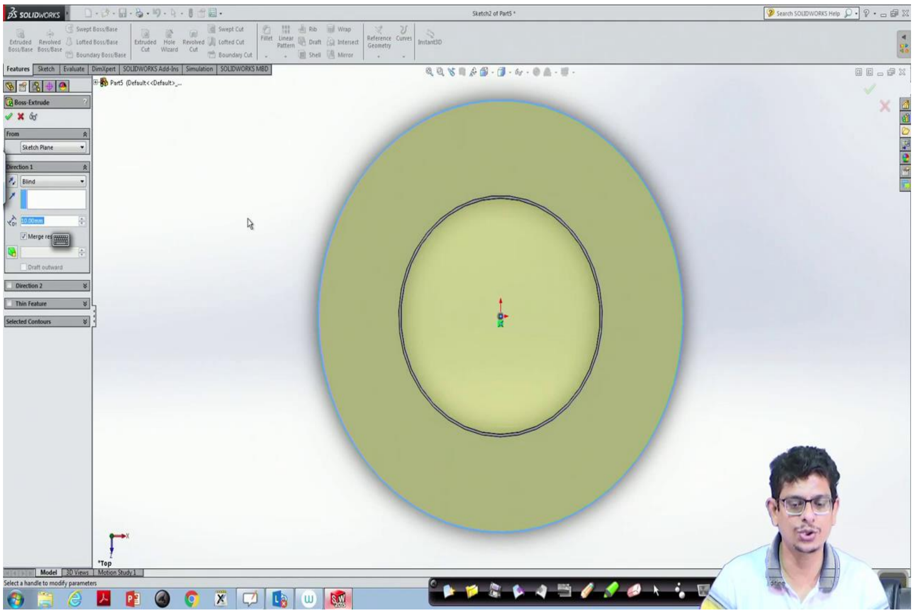

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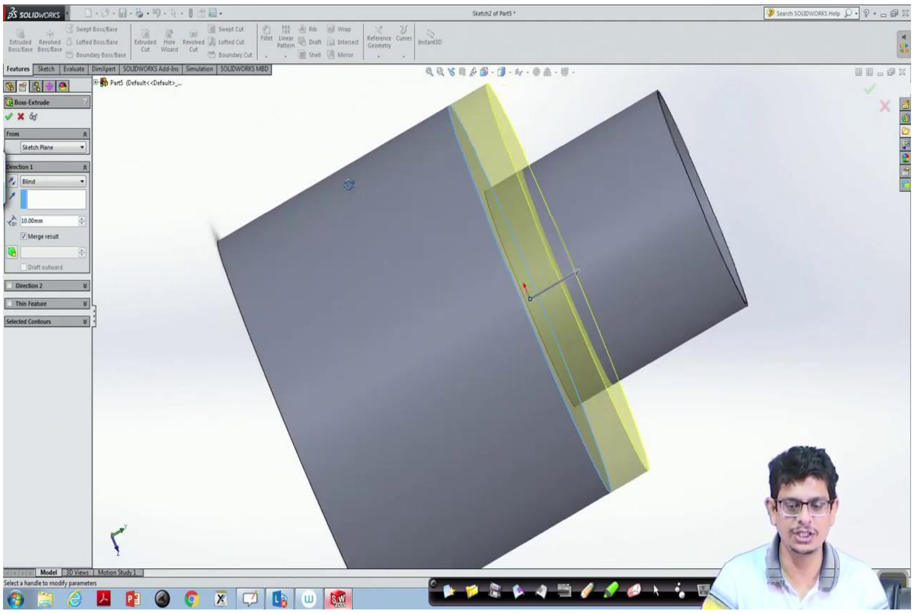

Now, we draw a circle. Now, let us see the same dimensions. Once it is done, we go with extrude boss base, see in which direction we would like to have. So, it should be in the reverse direction, reverse direction.

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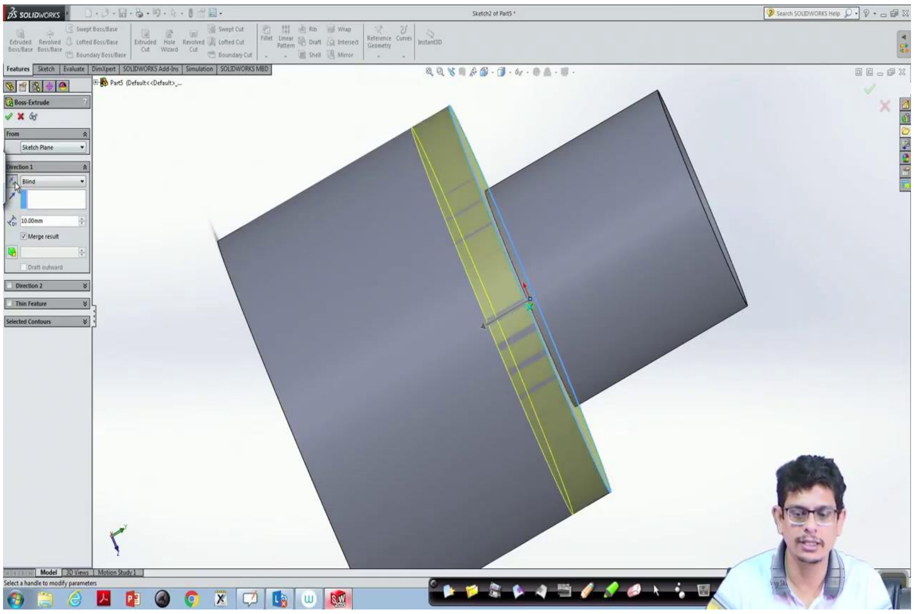

And what we would like to have is close the object something like some 50 mm, maybe something like 70 mm, click ok. So, the solid object will be filled.

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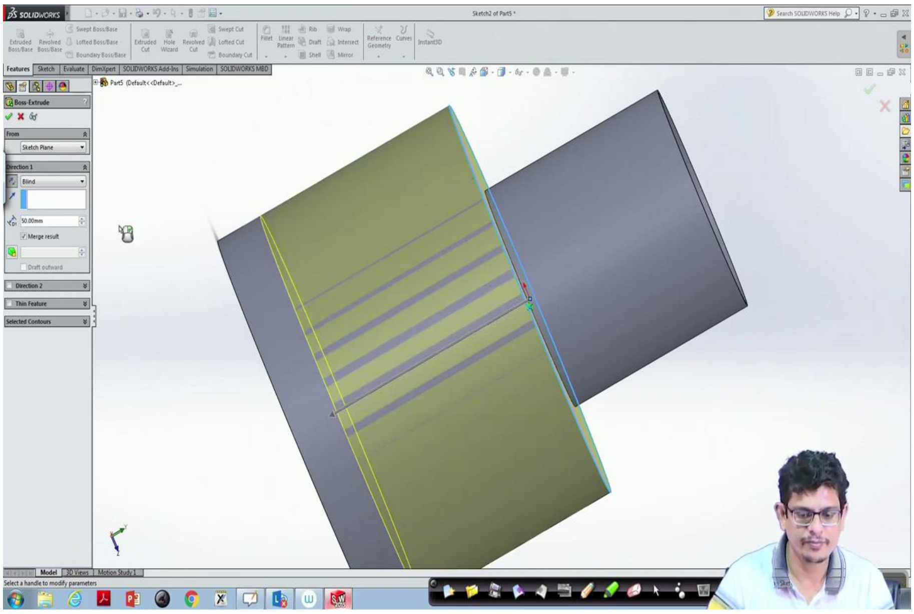

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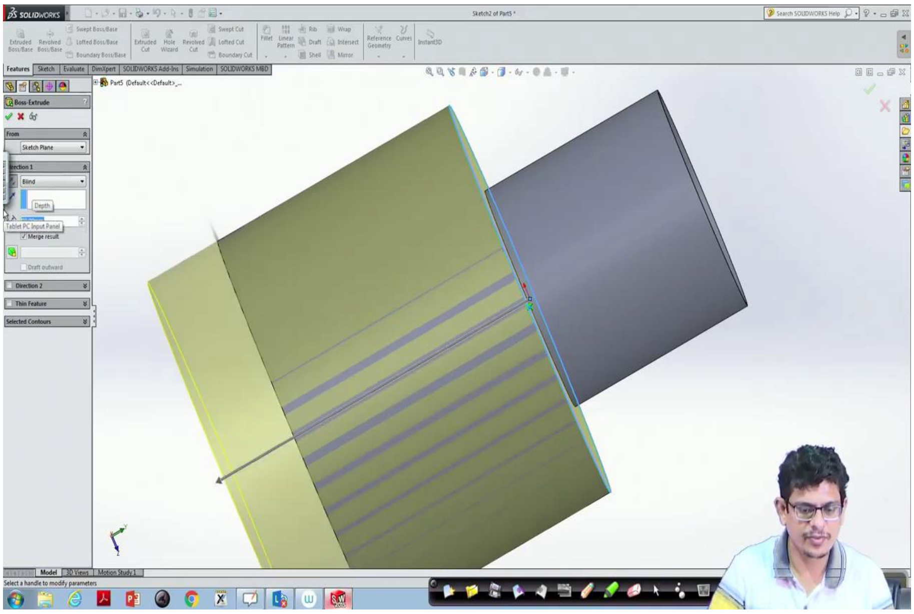

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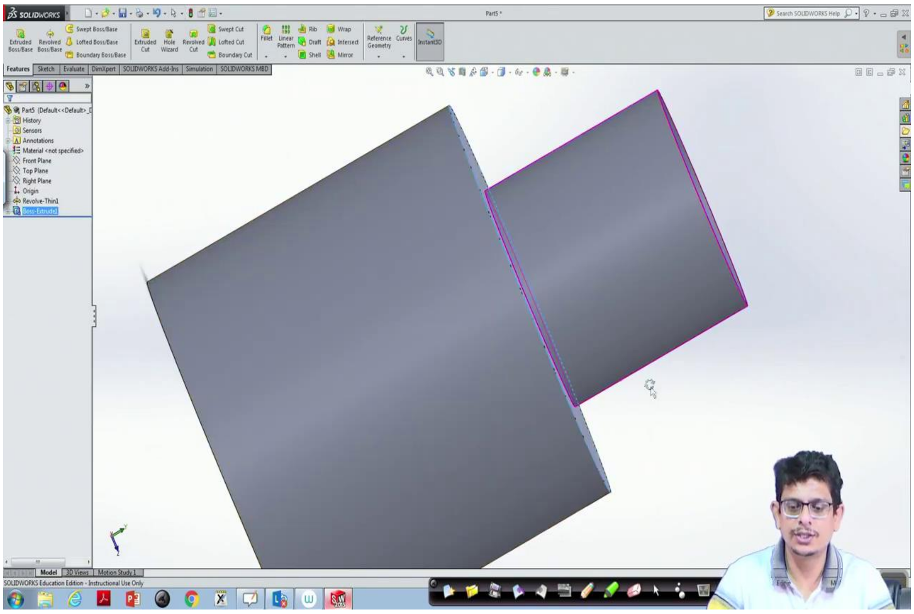

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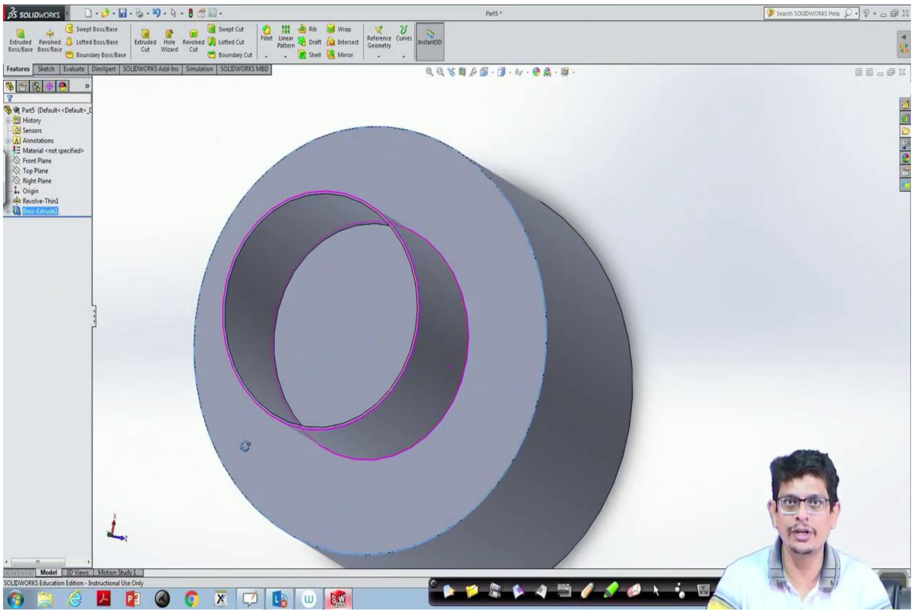

Now, if I would like to construct only hollow cane, what we have to do is, first create a circle give something like a cylindrical portion up to the certain level of thickness. After that, construct a rectangular one and rotate it. Let us do that with another example. So, what we are going to do is a close cylinder if we want to construct in a stepped way cane, cane style. Let us try that a sketch. On the front plane, normal to it.

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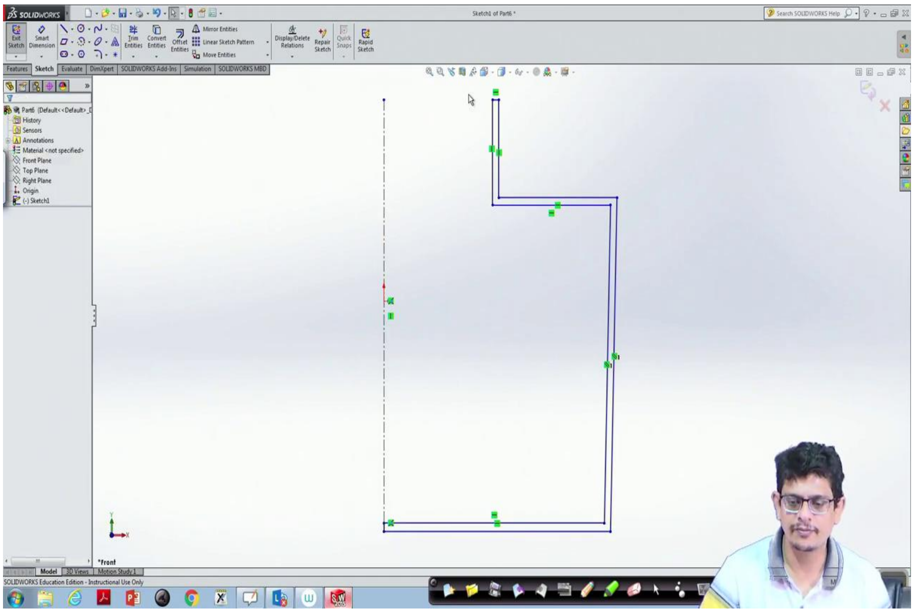

A centre line, we would like to have a cane style kind of thing. So, what we will do is pick a line from somewhere of certain mouth, it goes there down the end. Now, it is not a closed one. So, control, pick this one, this one, this one and also this one.

Go to features, revolve boss base because thickness we did not mention. It is not a closed one, 360 degrees, oh something has happened wrong. Go to the front plane. Now, a small thickness let us give it. In this case, offset is the best option to go with.

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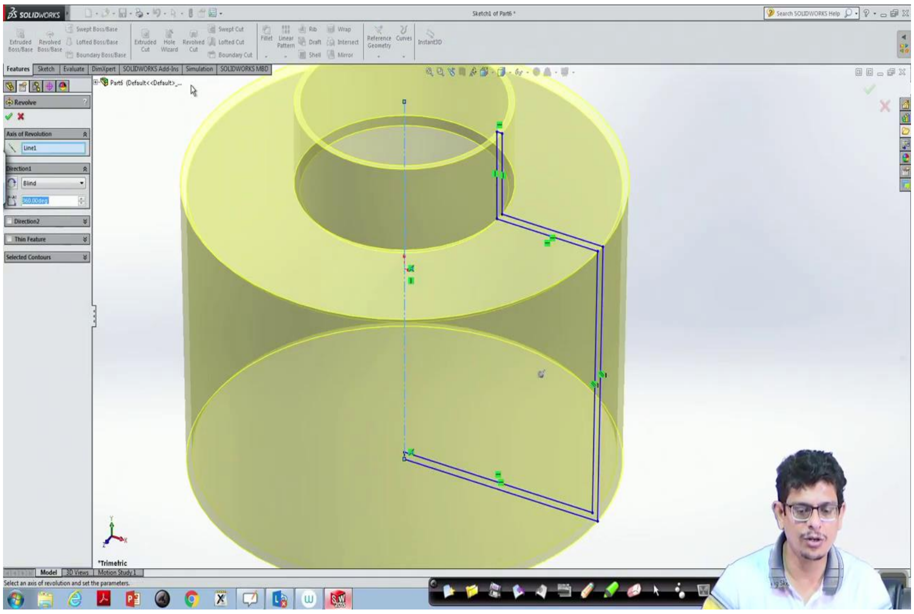

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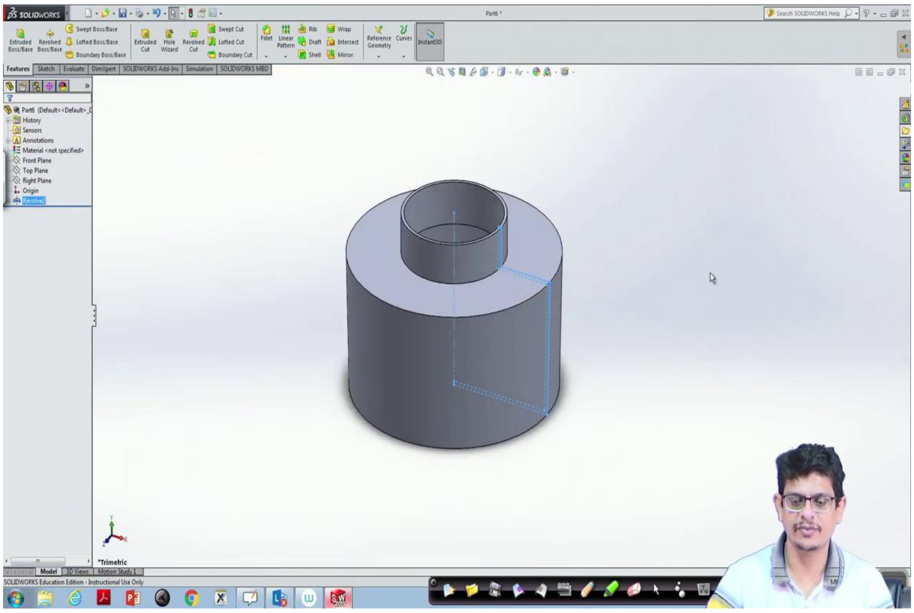

So, now, it is a complete one. Select this one, go to features, revolve boss base, click ok.

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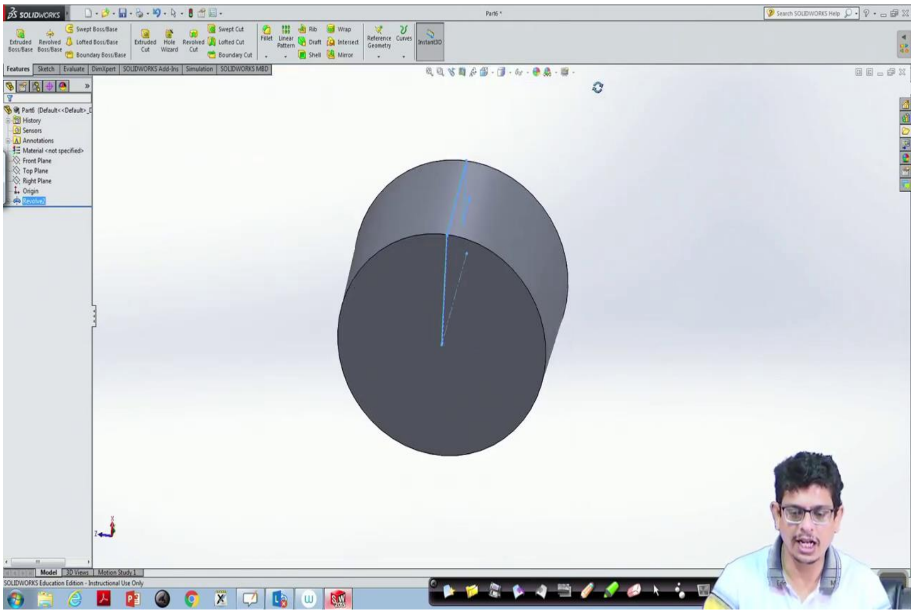

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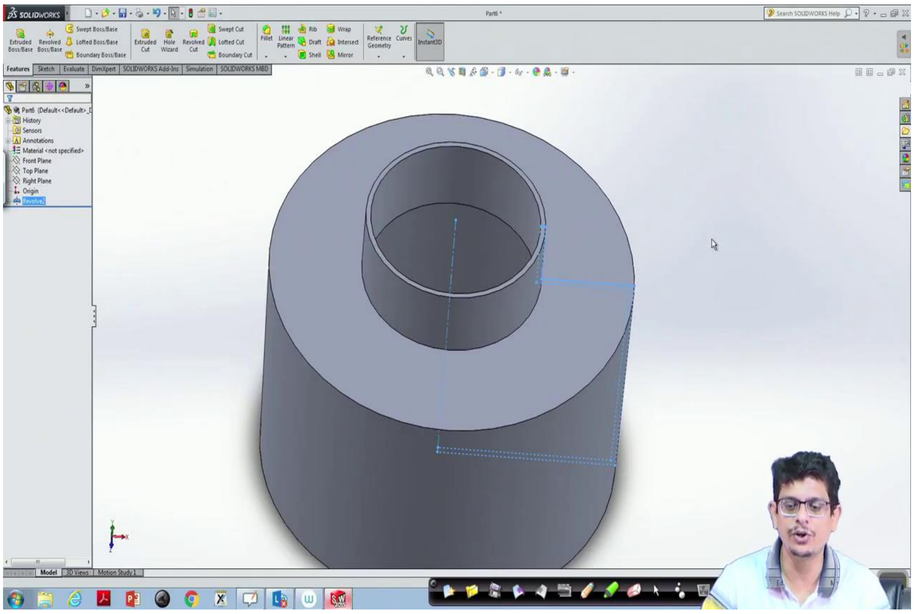

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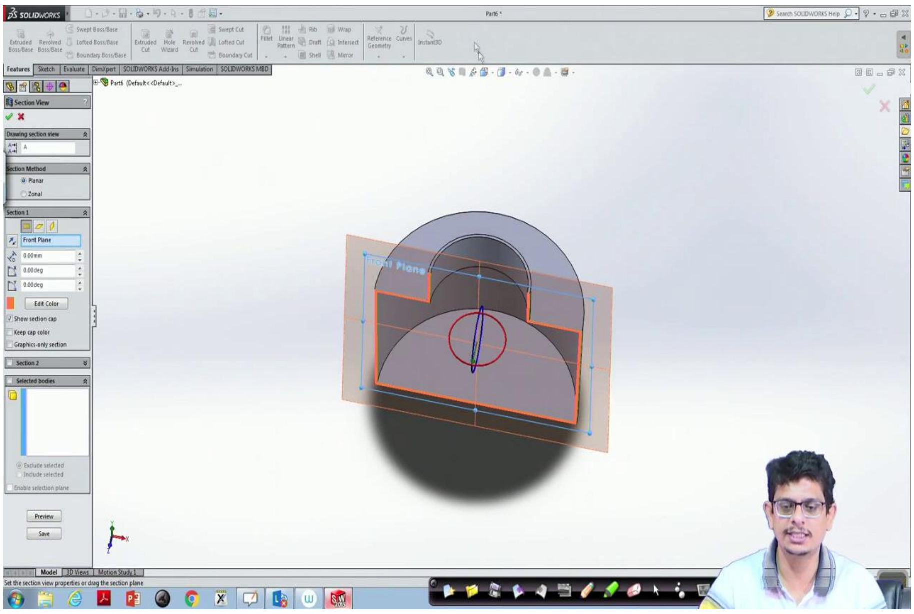

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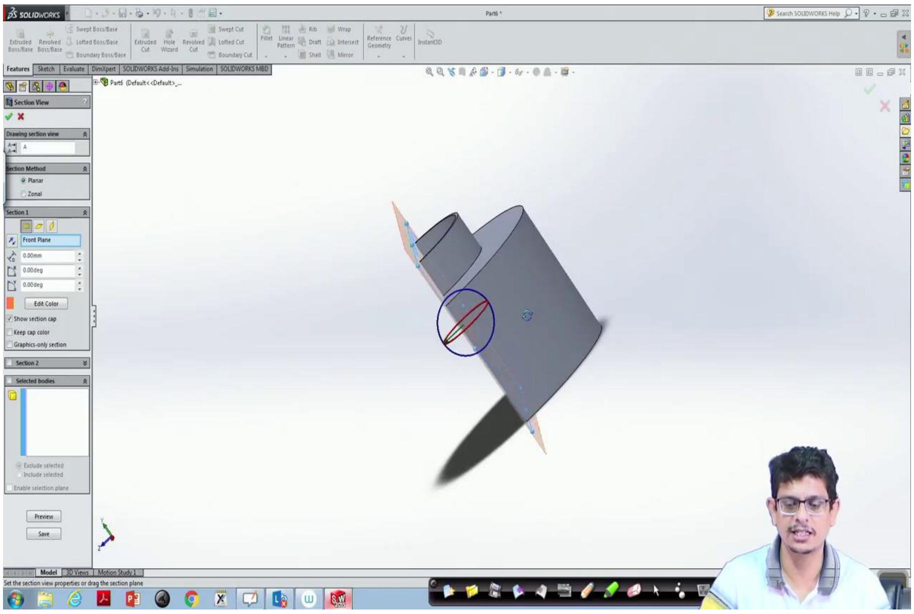

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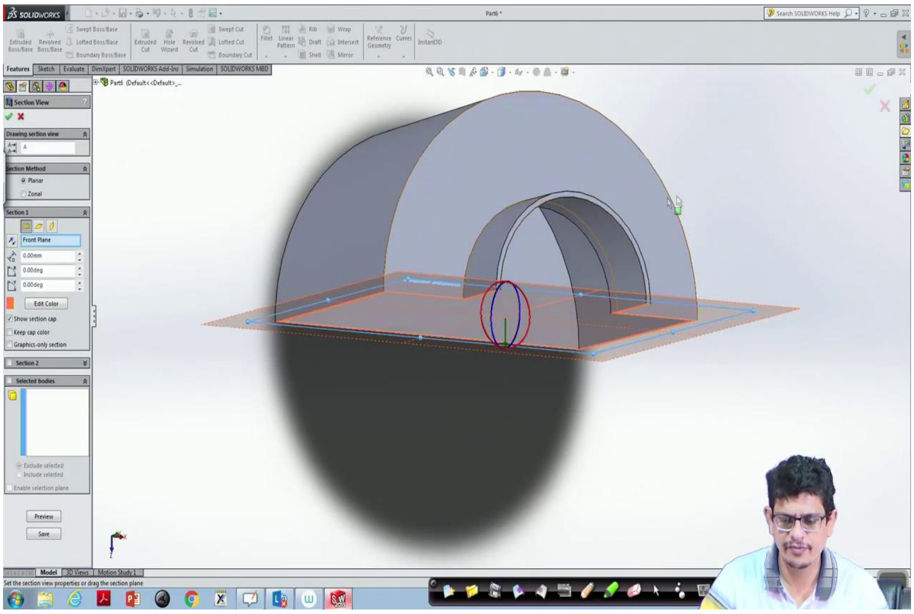

So, if you are seeing on the other side, it is close; inside it is a hollow one. Now, let us look at a cut sectional view of this entire cane. You see the cut sectional view - a hollow cane. This is the way you construct the objects.

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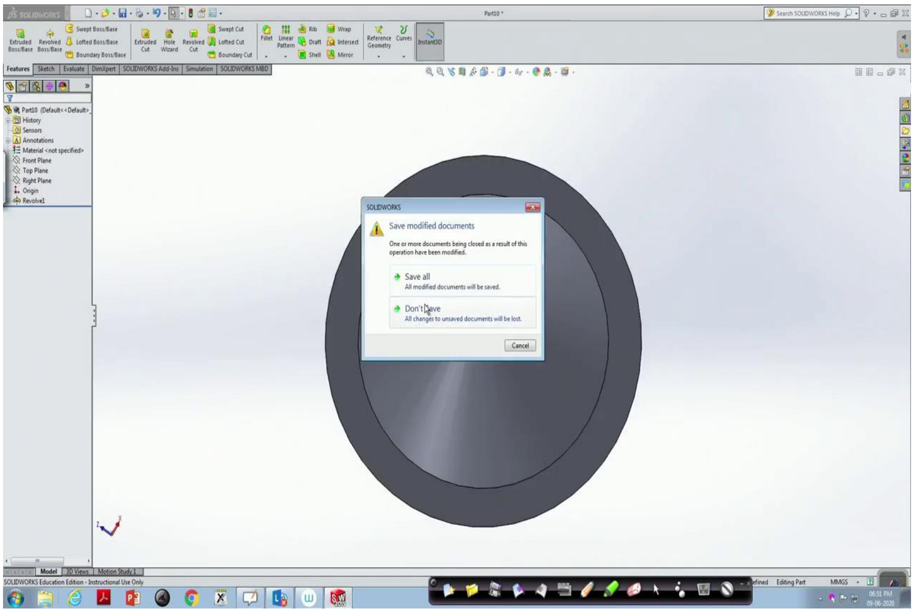

If I would like to construct a converging-diverging kind of nozzles, typically in engineering, we see nozzles, these nozzles always are to accelerate the flow from one location to other location if you want to increase the speed, you use these nozzles. So, how to construct it a typical rough sketch, we are going to see.

The typical nozzles have one particular kind of functional relations, they can be converging, they can be diverging, they can be both converging-diverging kind of nozzles. If you are seeing rocket engines on the backside wherever the exhaust gases are coming out such kind of thing what we call these nozzles.

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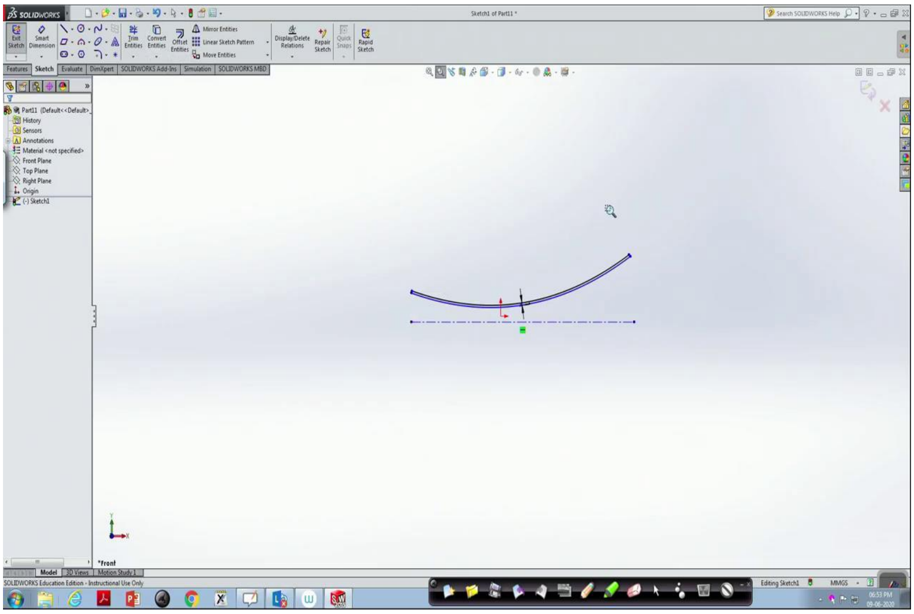

And one of them a simple way of constructing these nozzles is first to draw a centre line, then use a spline, the easiest construction what we are going to do that portion is converging, and that portion we are having diverging. So, this one I would like to revolve around that with a hollow gap.

So, for that purpose, if we would like to have a hollow gap, what I have to do, pick this one. First construct an offset not in this direction, but the reverse direction. We do not require that thickness maybe 1 mm thickness is good enough like a tube, then click ok.

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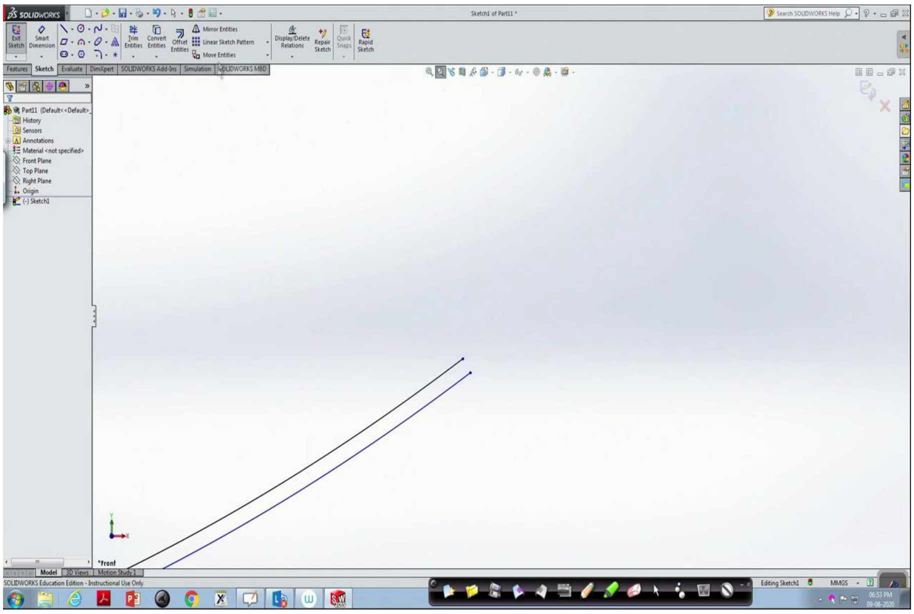

Now, zoom into this portion, add lines, perhaps we would like to have a very vertical line from there to there, done.

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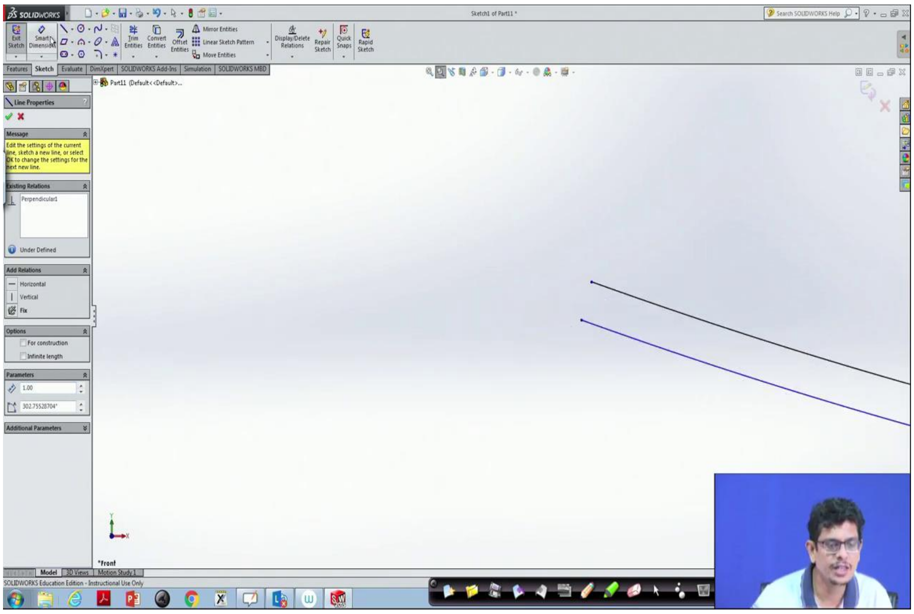

Similarly, zoom into this portion, join bylines, there to there. Now, we have a complete close picture.

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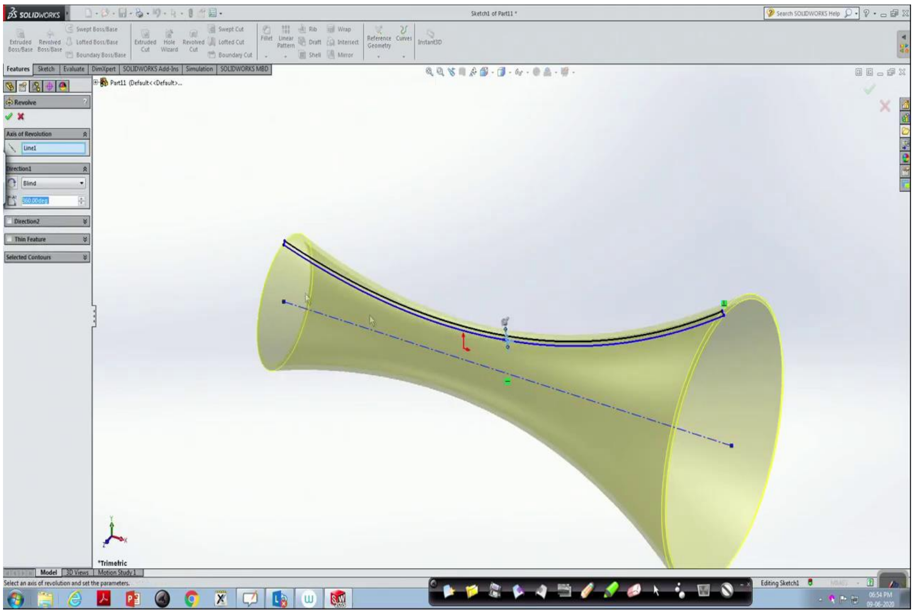

Now, press escape > select this entire thing. Go to features, revolve around that.

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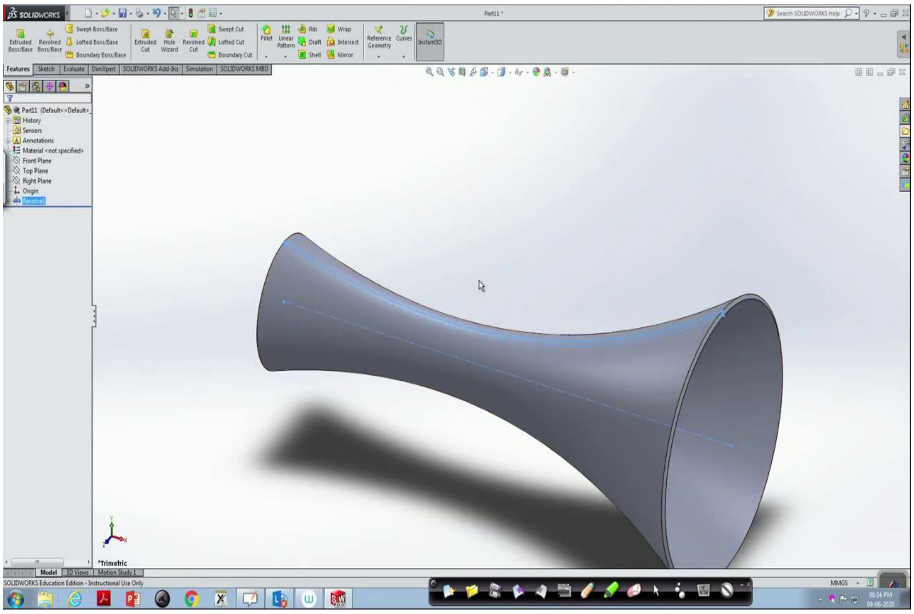

We have this particular shape.

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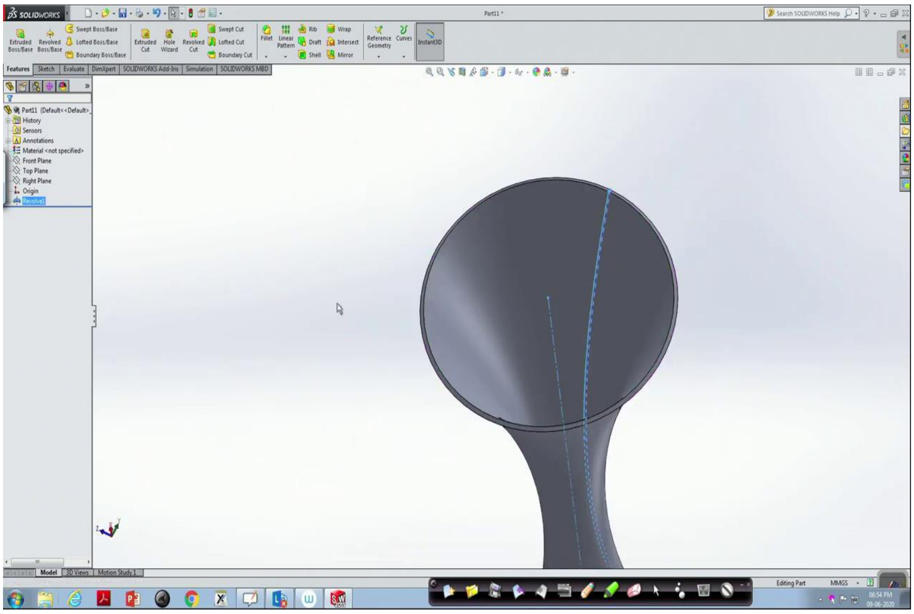

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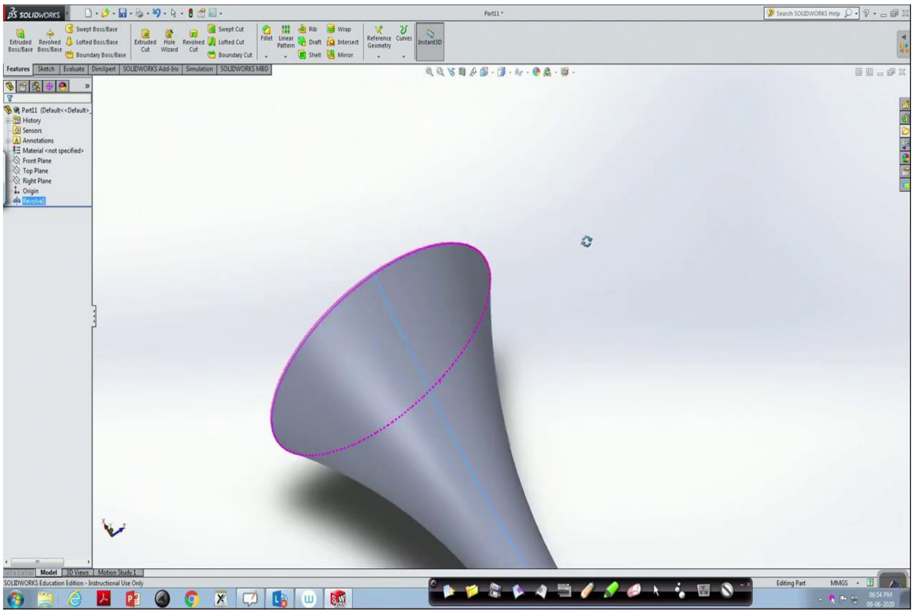

Based on the engineering designs and numbers, the technical drawing one has to construct it. Let us look at different views of this nozzle.

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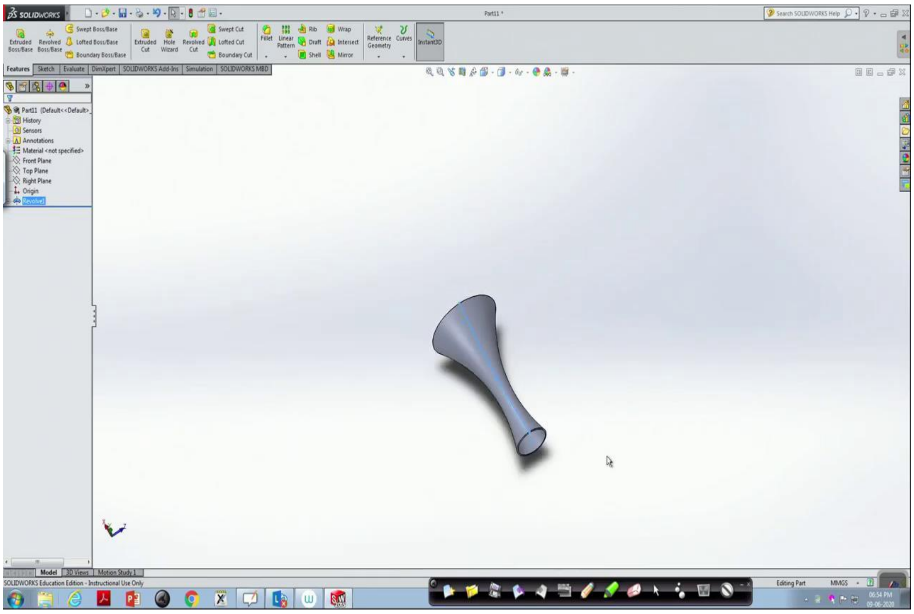

So, for that purpose, what we have to do first of all keep it in isometric view. Then go there look at these different views.

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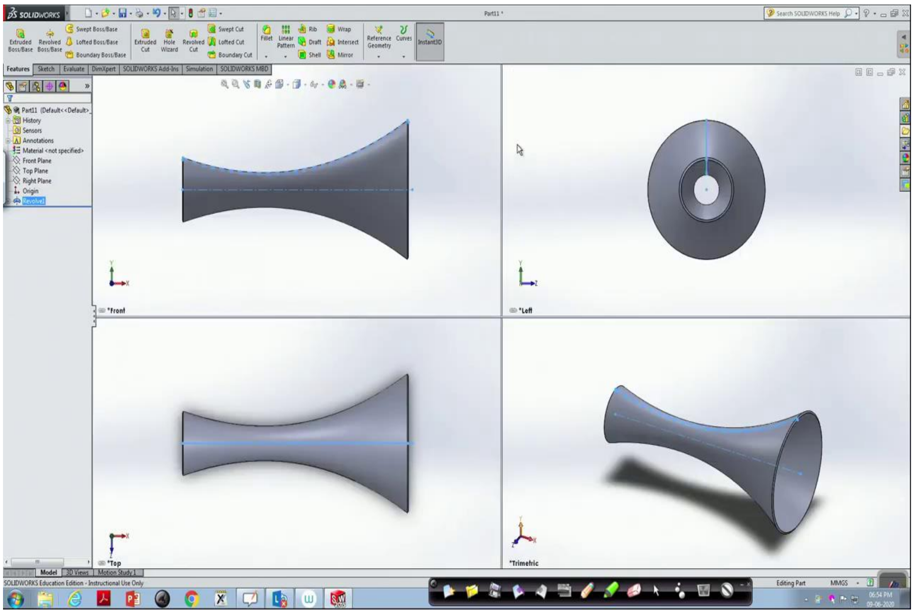

So, something like front view, top view, side view and isometric view we will get.

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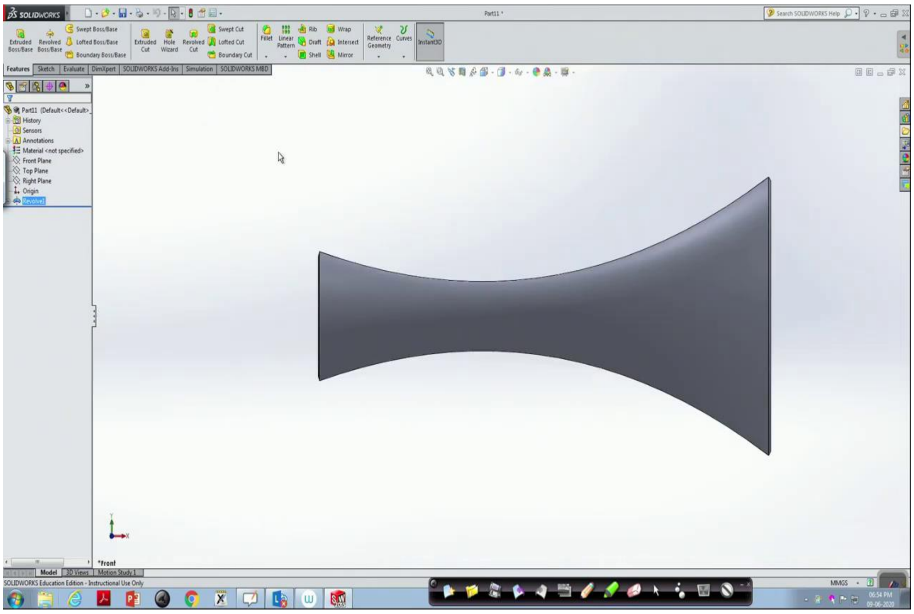

Now, we will go with assembly drawings. So, for example, if we have to do assembly drawings first of all what we have to do is construct different parts, and then join them together.

For example, I have something like a cylinder on which there is a circular disc I would like to mount it and it is supposed to be fixed at one end. How to do that? So, for that purpose, what we do is as usual first we have to construct different parts, one is cylinder shaft, the other one is a circular disc. And we would like to mate them together.

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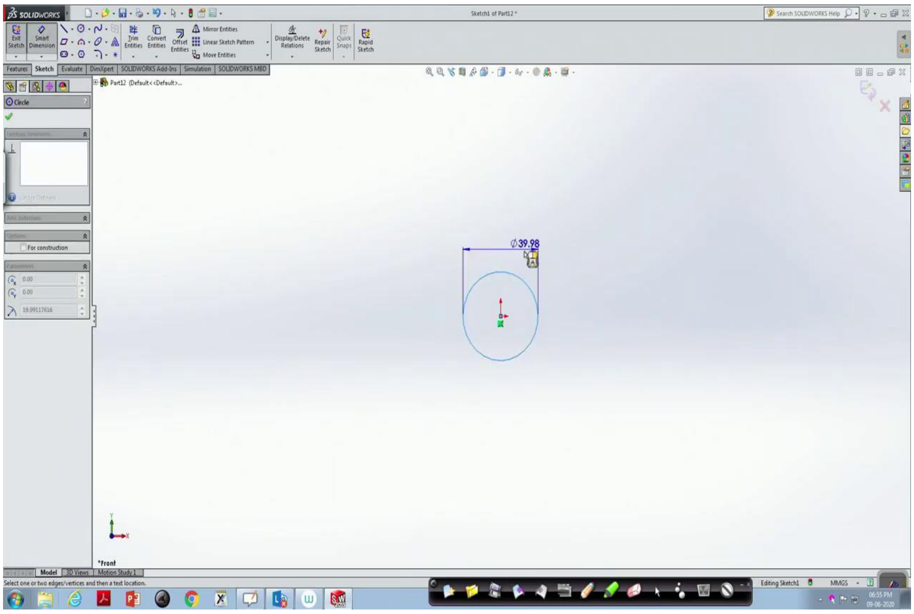

So, to do that, the first one is go to sketch, frontal plane. Construct a cylinder.

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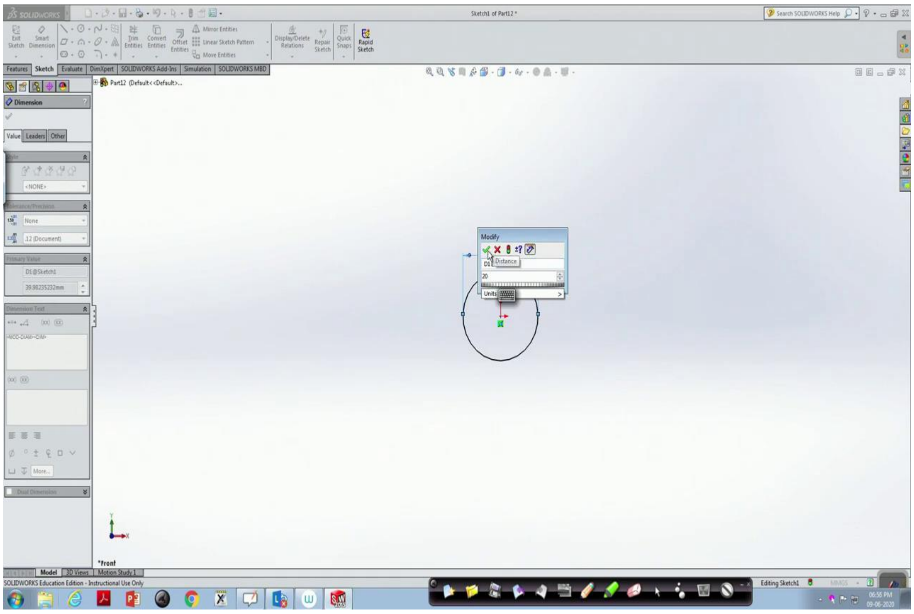

First, we are going to construct a circle. This circle dimension supposed to be correct 20 mm in diameter. Go to features, extrude boss base. It may be some 100 mm or 100 units, then click ok.

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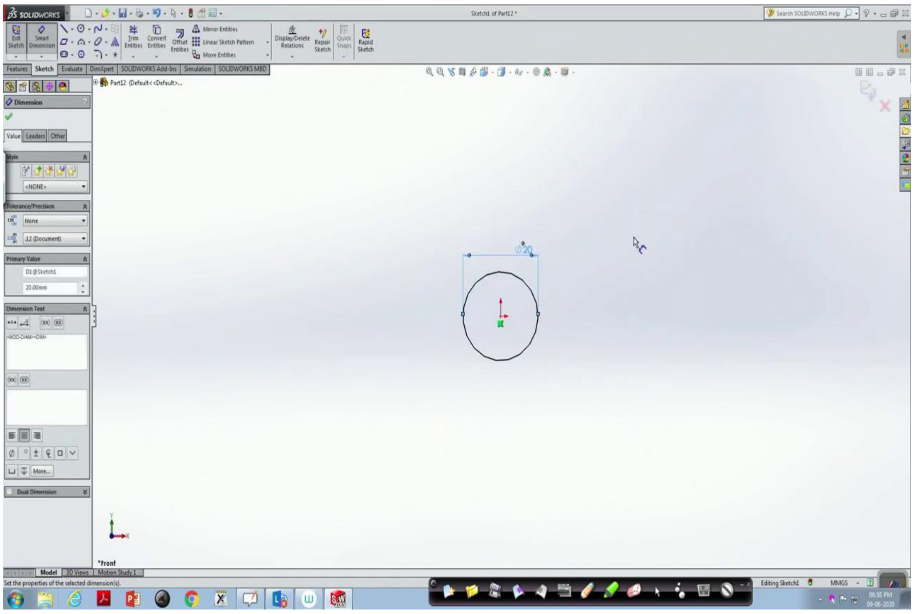

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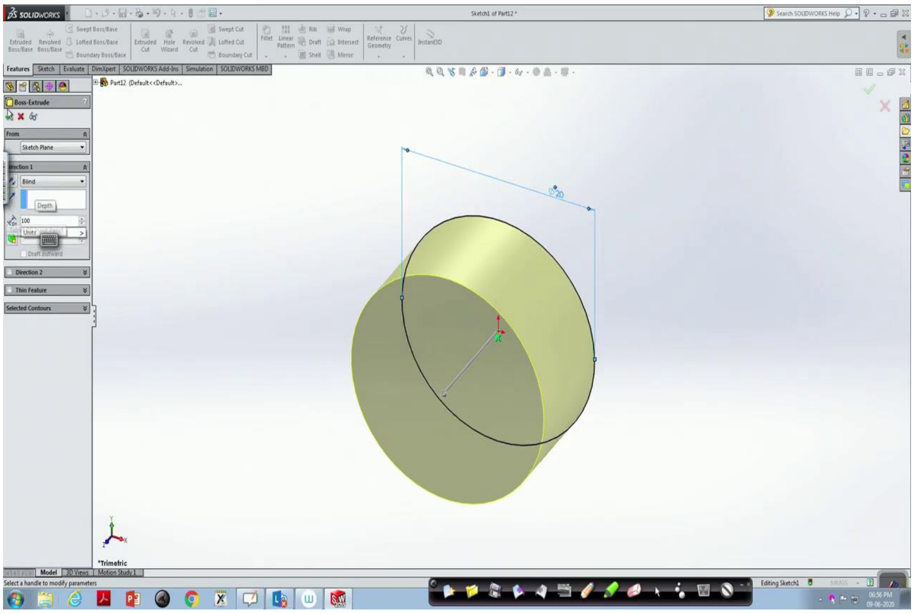

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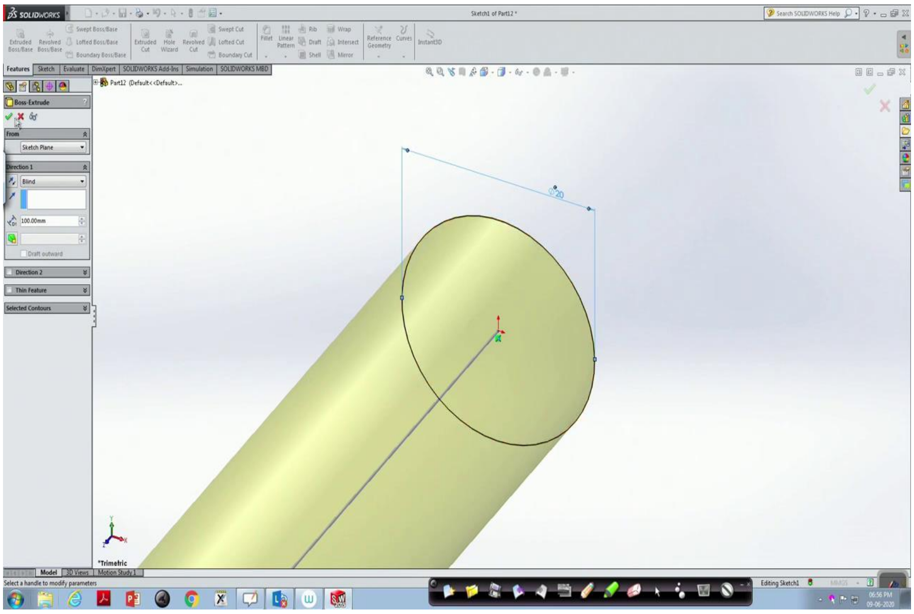

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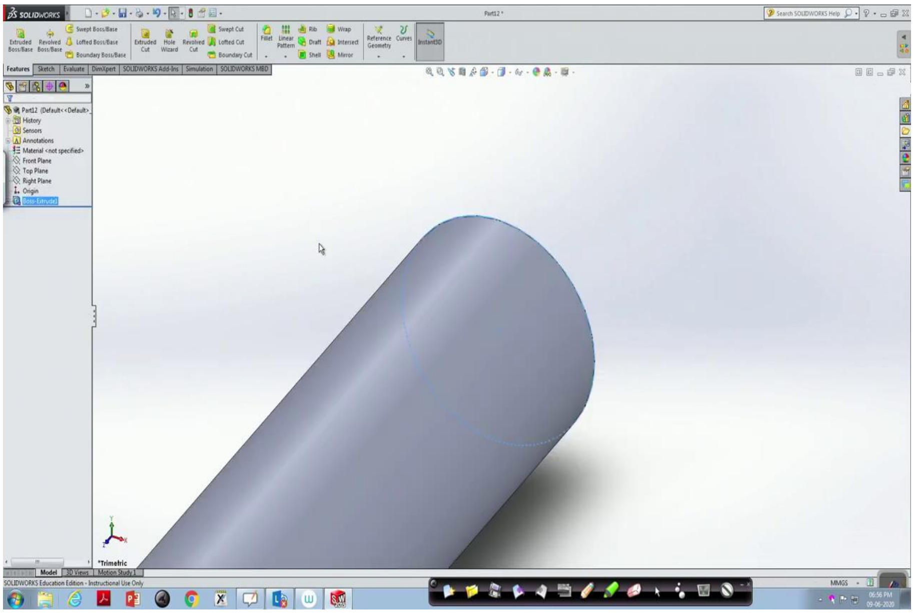

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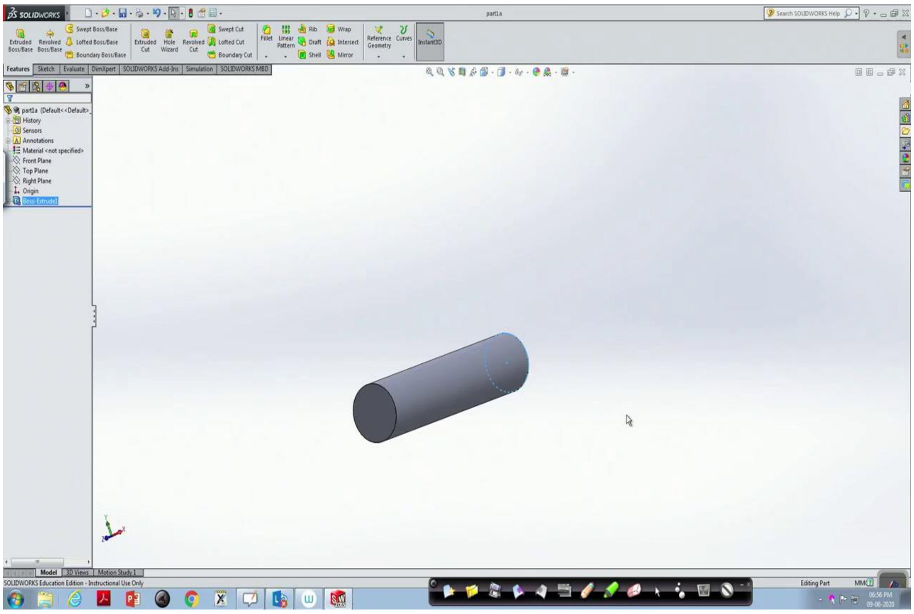

So, we have this cylinder. Now, what we have to do save this one as an object.

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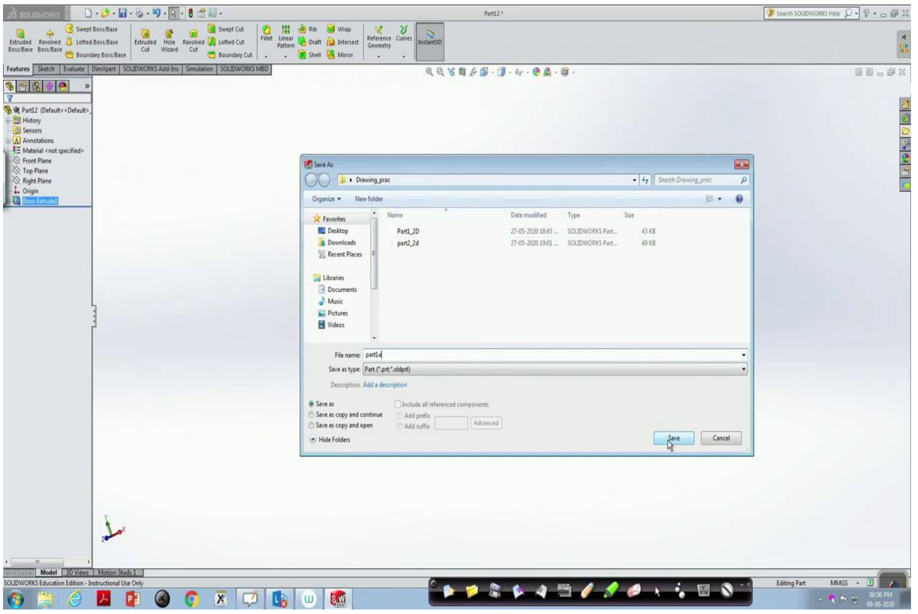

So, go there save as give something like part1a, save. So, the cylinder is done. Now, open a new one. Again part drawing, click ok, go to the frontal plane, we would like to have a circular disc.

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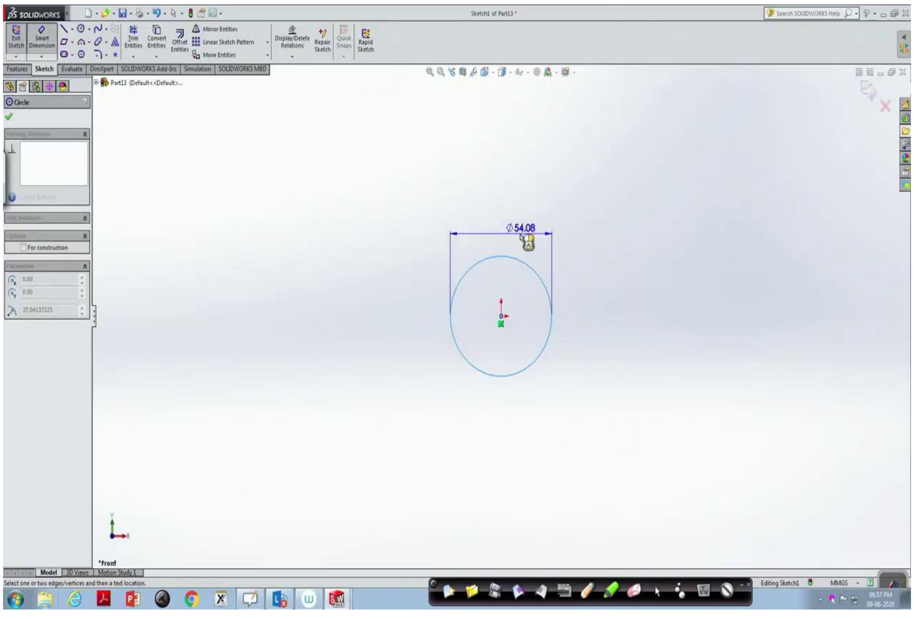

So, circle, go there, construct. This circle circular disc what we are going to construct, it is going to mount on the cylinder. The outside diameter of this cylinder is 20 mm. In that case, our disc what we are going to construct inside of that portion supposed to be 20 mm.

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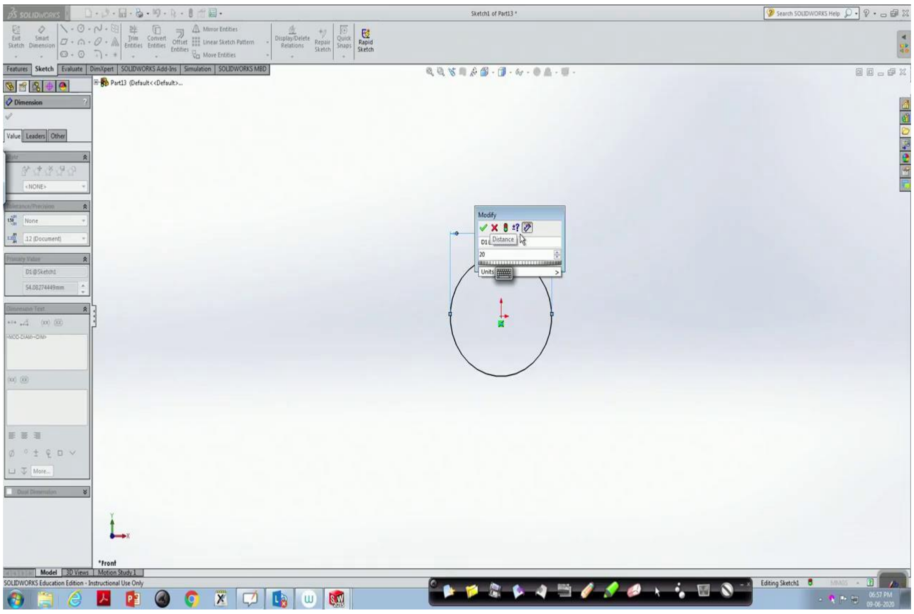

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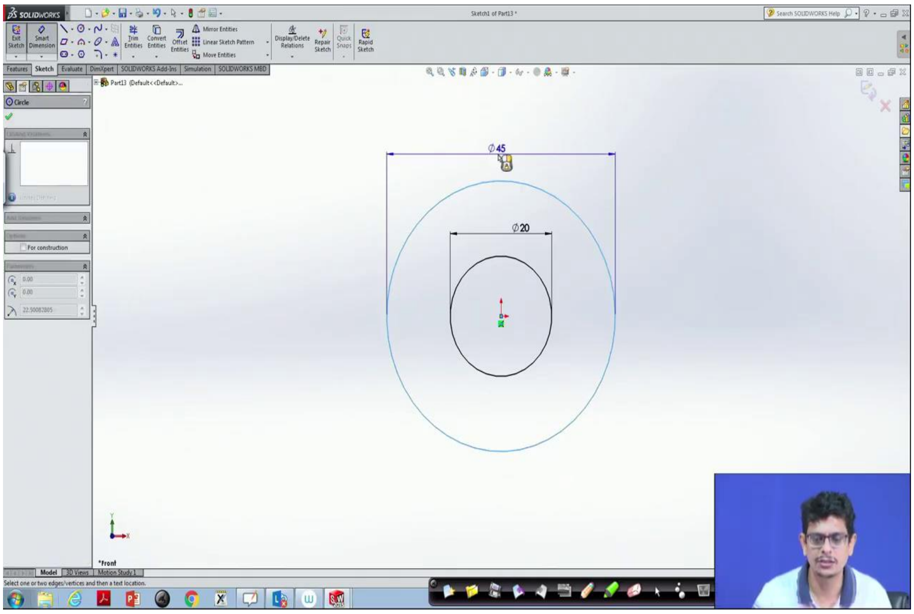

So, let us go smart dimensions pick this one, inside supposed to be 20 mm, done. But it is a circular disc. So, a circular disc means again we pick a circle, go there, maybe it might be of smart dimensions 45 units, done.

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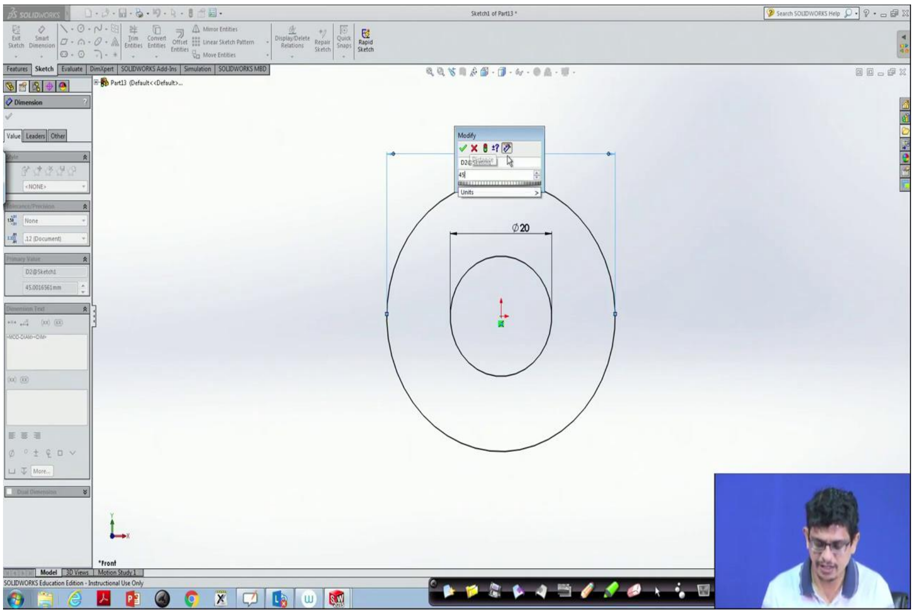

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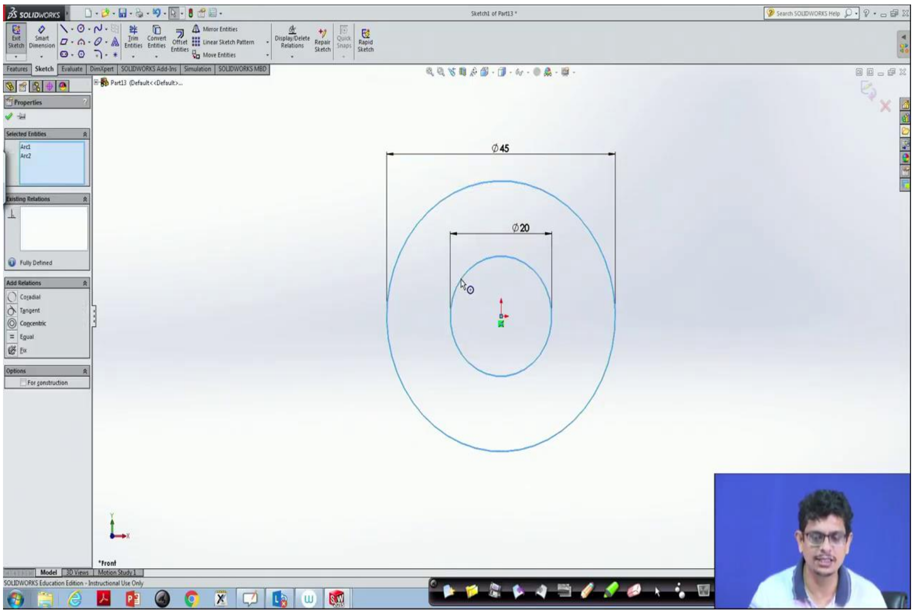

Now, we would like to make a circular disc. So, go to features, select these two things by control object.

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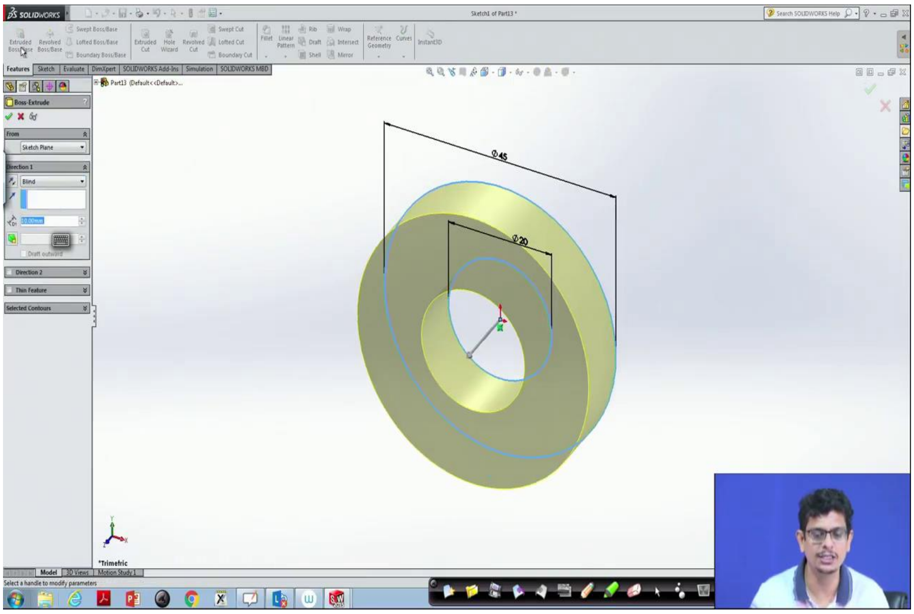

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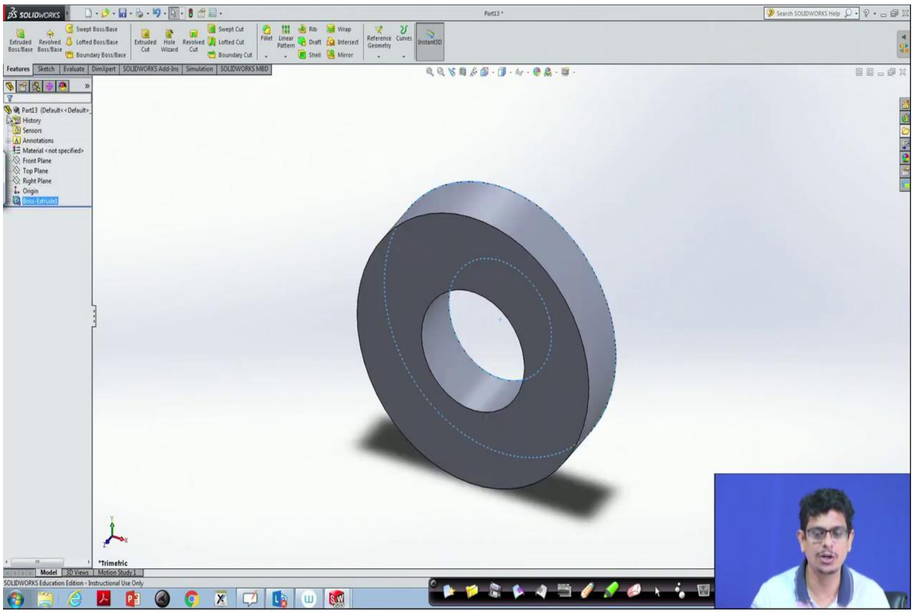

Go to extrude boss base, we have a cylindrical disc done.

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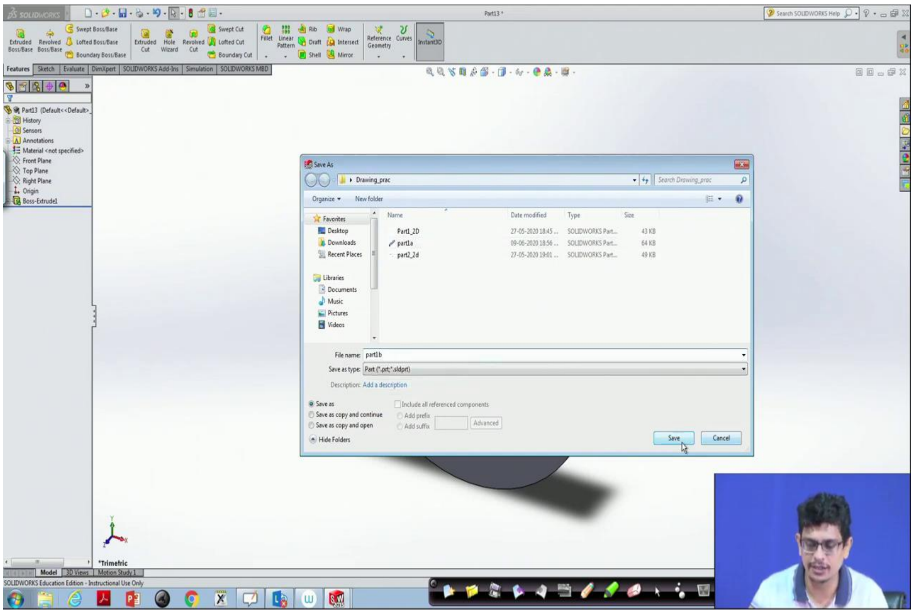

Now, save it as. Save as part1b at the same location. So, we have constructed two parts. What we have to do? Now, we would like to make these two parts, so that one single assembly one can get. For that purpose, what we have to do, click this new. Now, go for assembly, because parts we have already constructed. Now, we are going to construct an assembly, click ok. Now, when you do that either it will automatically select the parts whatever we have constructed in a specific directory.

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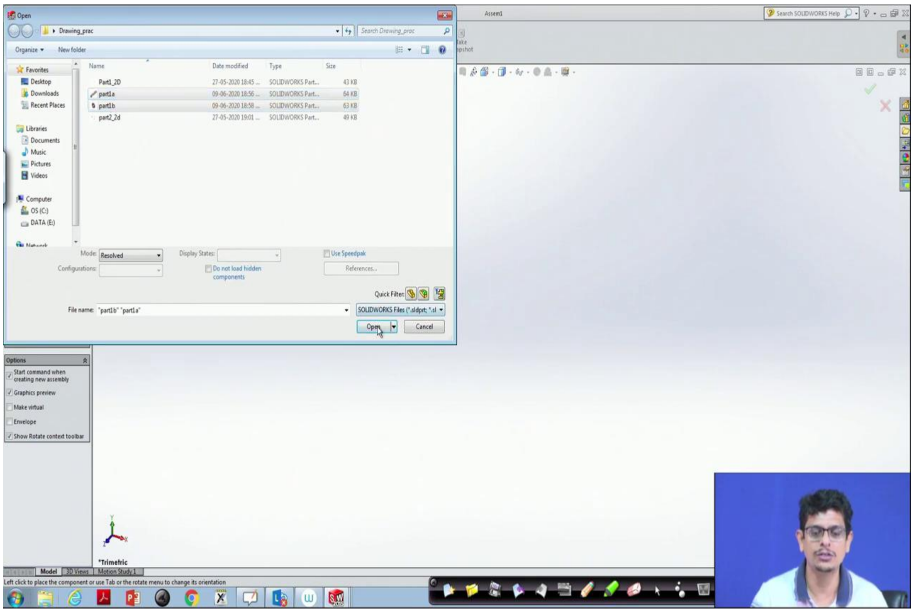

Otherwise, we have to click browse, pick those parts which we are interested in part 1 control part 2, open.

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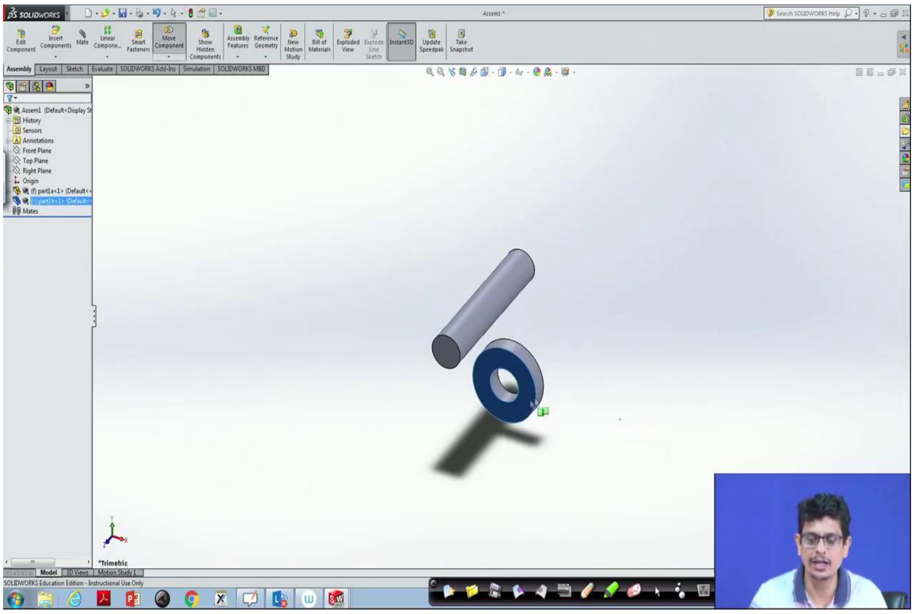

Then it will automatically show. Now, leave them by click 1, similarly, leave them by click 2. So, both the parts are brought together in a single assembly drawing. Now, this one has to be mounted on that. For that purpose, what we have to do? These two things, you can move it whatever the desired location you would like to have in that way.

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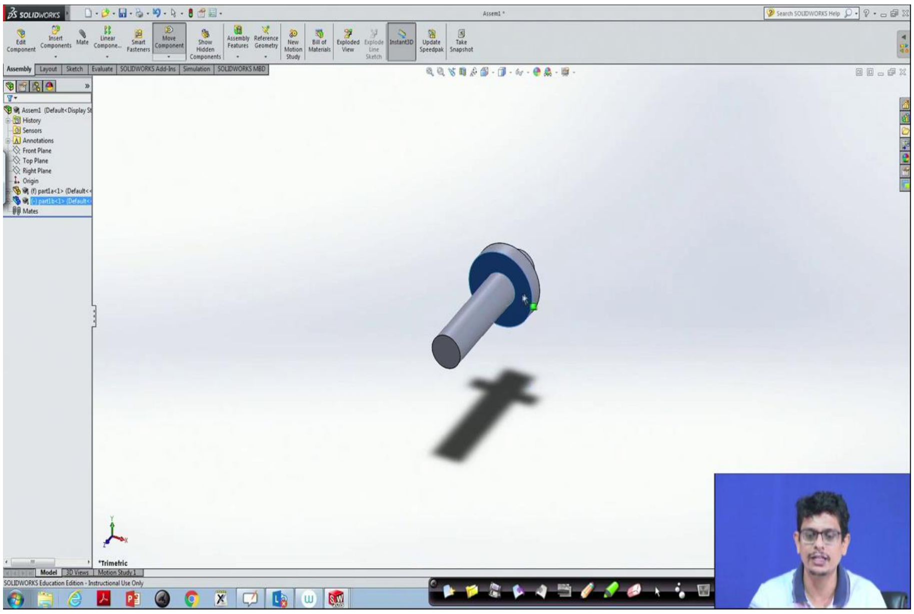

But these are not fixed on the cylinder, they are freely moving. For assembly drawing, what we have to do is mate these objects. For that purpose, what we have to do, there is something like mate command, click this one.

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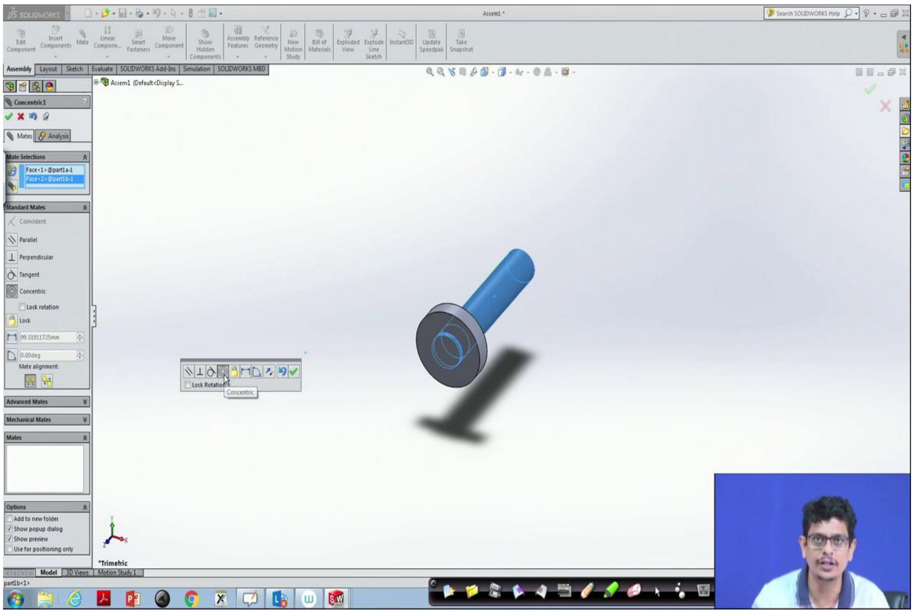

Now, this surface and internal surface, they are supposed to meet each other. For that purpose, what we have to do? Here you can see concentric mate. So, click that and ok.

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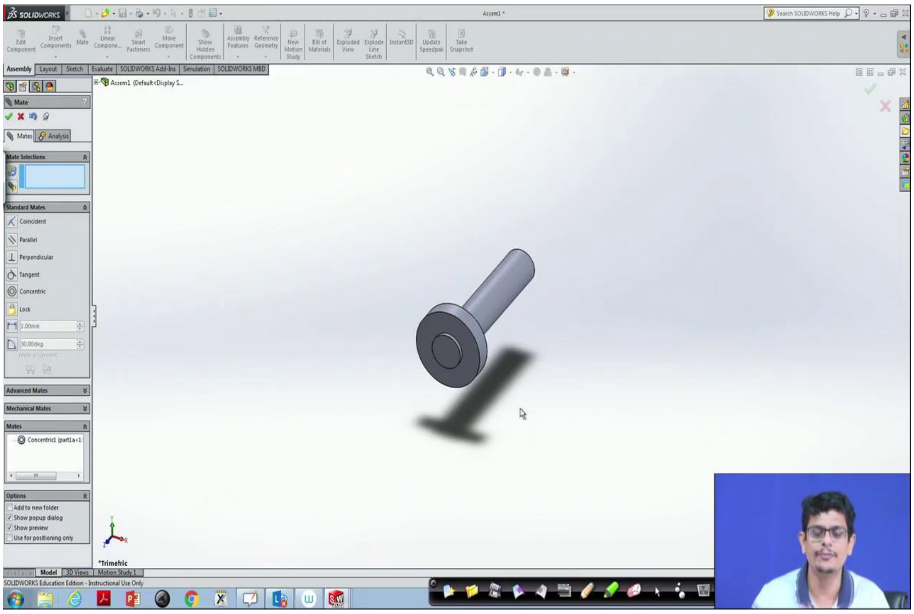

Once you are done, this object can move concentrically out of that only, but you cannot move this circular disc away parallel to this in this direction. So, even if you are trying to move that one, this one goes only in the concentric way, it cannot go anywhere.

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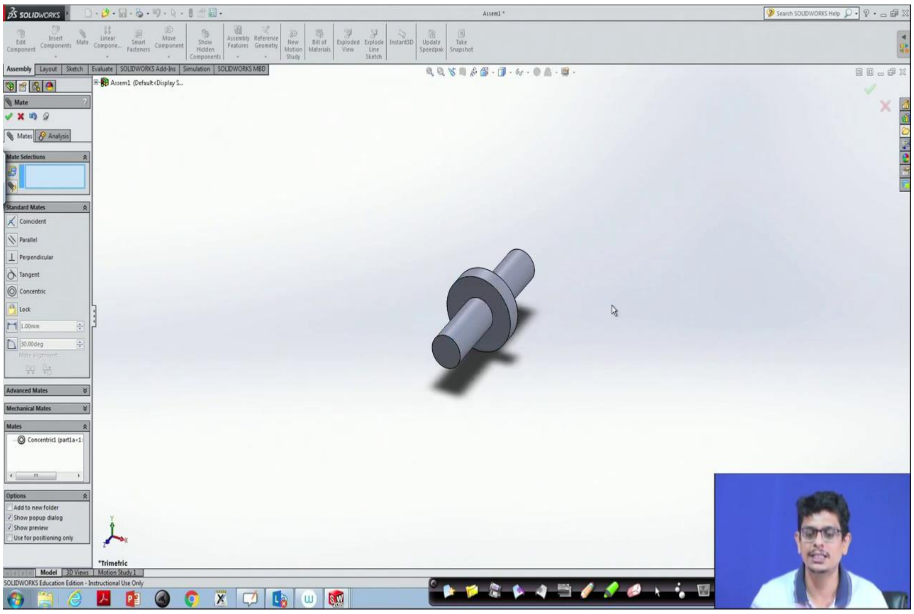

Now, we want to fix this entire thing, lock it from here to one of the ends. For example, I want to keep it to this end. To do that what we have to do, after clicking ok, now again click mate, command picks one of the surfaces.

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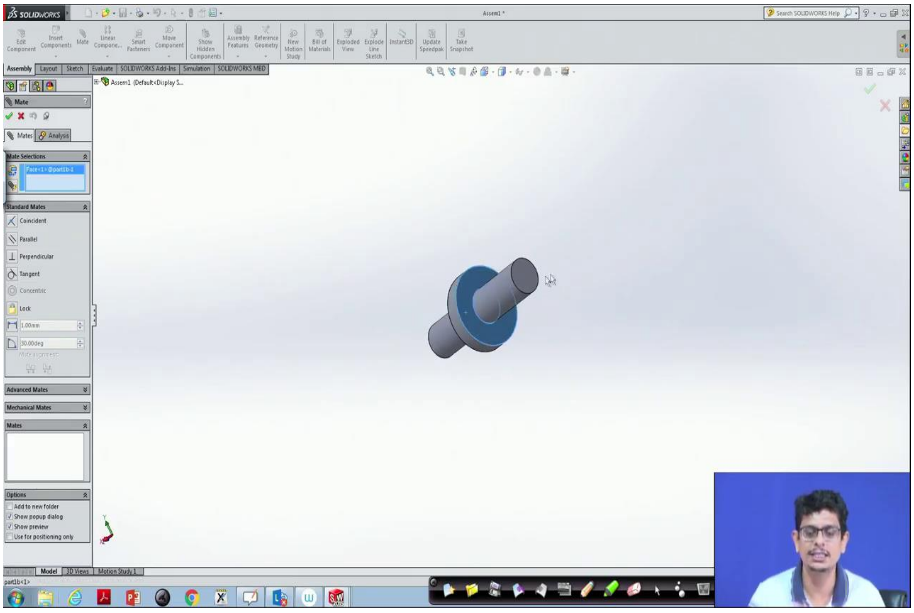

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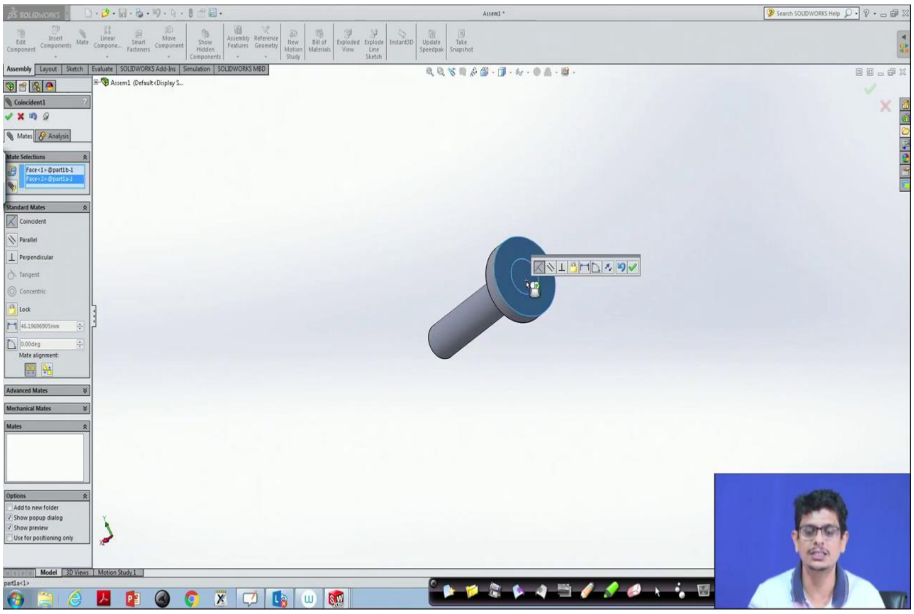

So, for that purpose, we can rotate this. This surface and this surface control, by holding this control. Let us do it once again. So, this is the object. Now, this surface whatever this on the circular disc, and this surface supposed to be together. To do that what we have to do click mate, pick this surface. Once you select the surface it will be highlighted, then go to this surface, click. These two things are coincident. So, once this coincident is done, click ok. Now, you have that object.

Now, even if you want to move it, they would not move out of that object. Now, if you click ok, it will be assembled. So, I cannot move it. Now, I do not want to lock it here, but I want to lock it on the other side. For that purpose, what we have to do? Let us just redo that for that. First of all, I am undoing it so that this object can be easily moving. What I would like to do is this surface and this surface supposed to be coincident.

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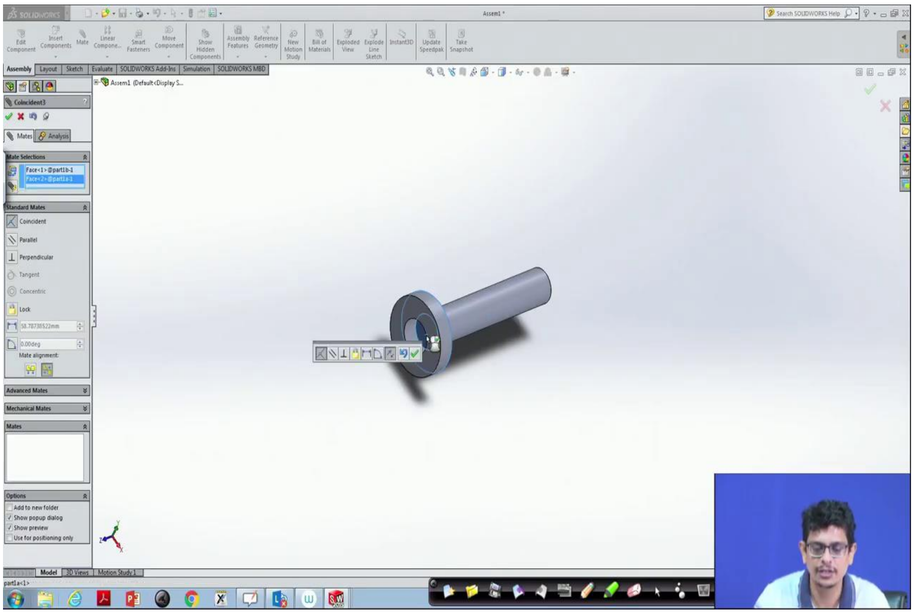

For that purpose what we have to do, click ok the surface. Let us undo. So, click mate, pick that surface, control, pick this surface, then ok. This is the way we can create that object.

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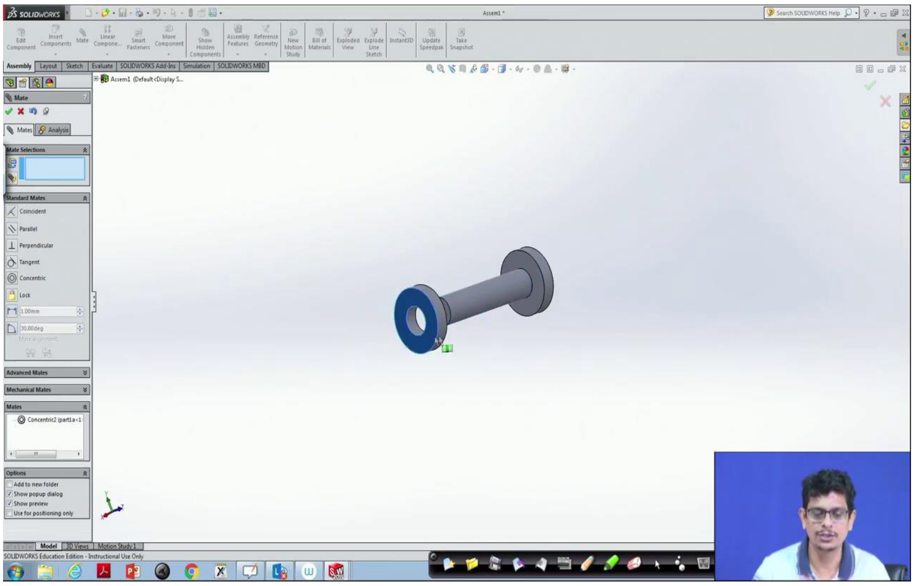

Now, we would like to have one more disc here. We do not have to reconstruct one more thing. For that purpose, what we can do is, we can again insert one more component perhaps part b done. So, now, I would like to mate these surfaces what we have to do, click ok mate, this surface and internal of this surface we would like to mate. So, this one can move on this object.

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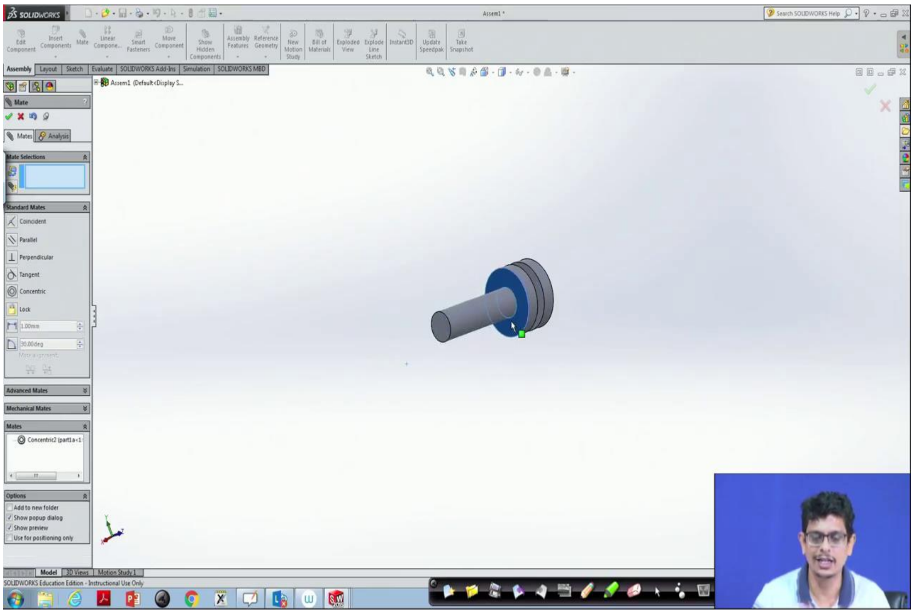

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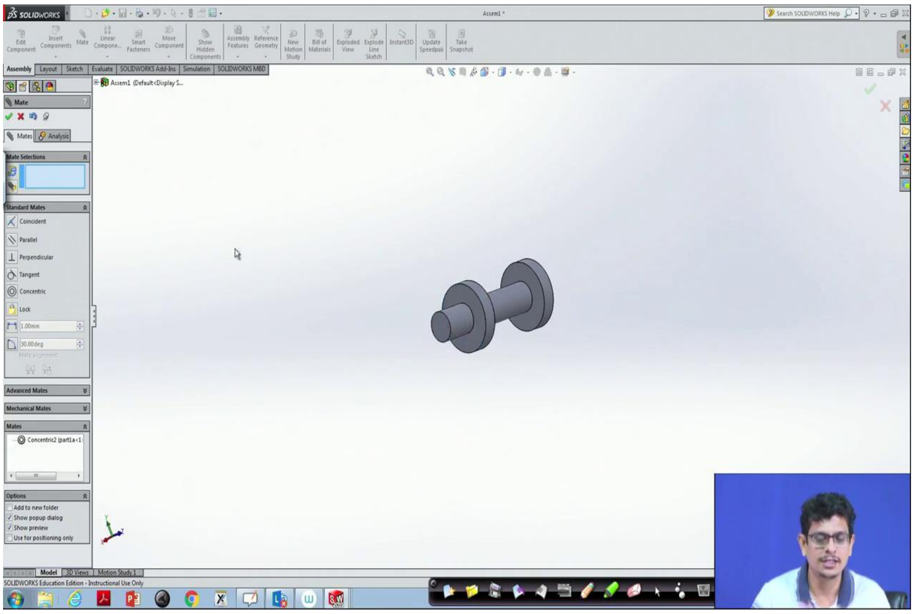

Now, what I want to do is this one supposed to be locked on that. For that purpose, what we are going to do? Mate this surface and this surface, coincident. This is the way we create that assembly drawing.

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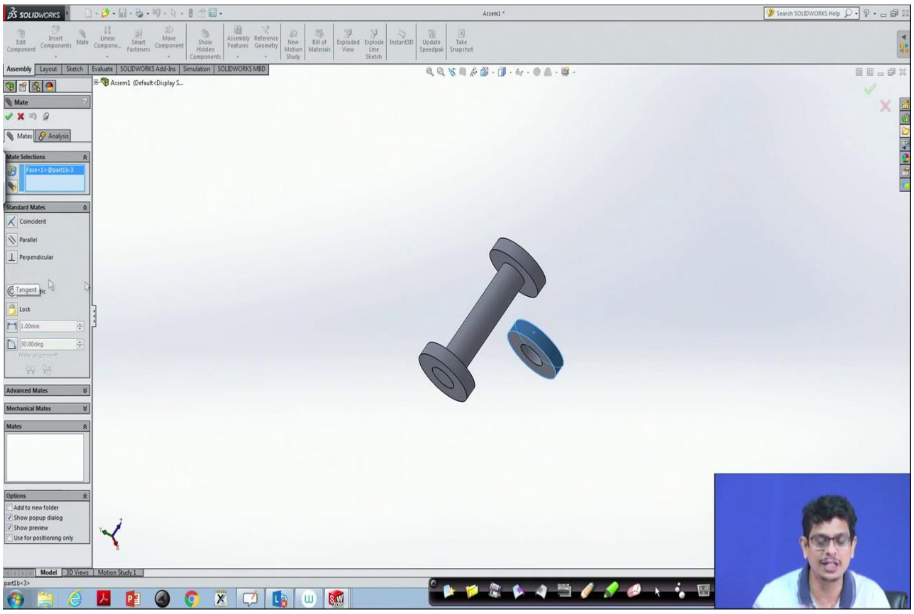

Now, I would like to have something like a tangent to this surface, rolling on that surface only I would like to have. For that purpose, what we have to do? Perhaps again insert part b pick this one. So, this is the object what we have. Now, I would like to mate this surface tangent to something like a cylinder.

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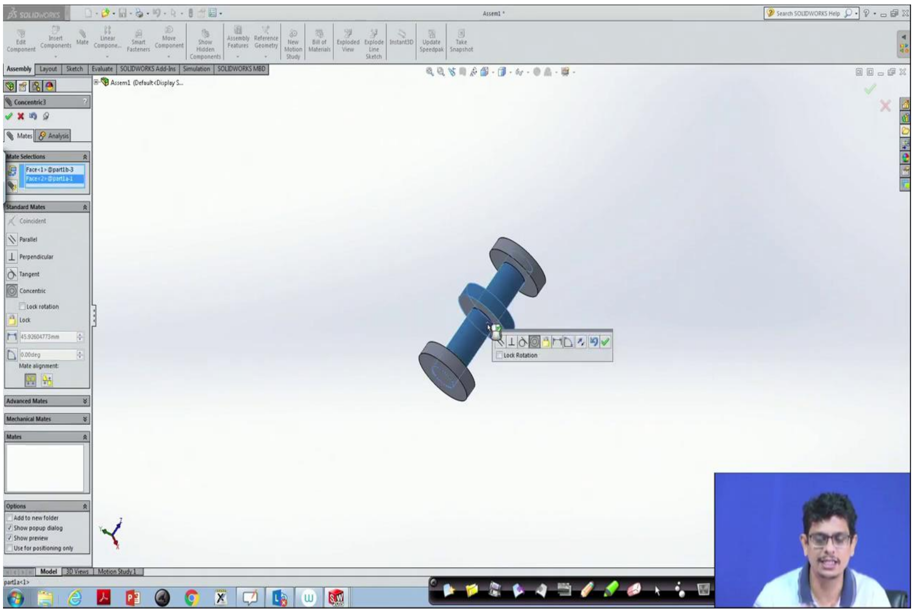

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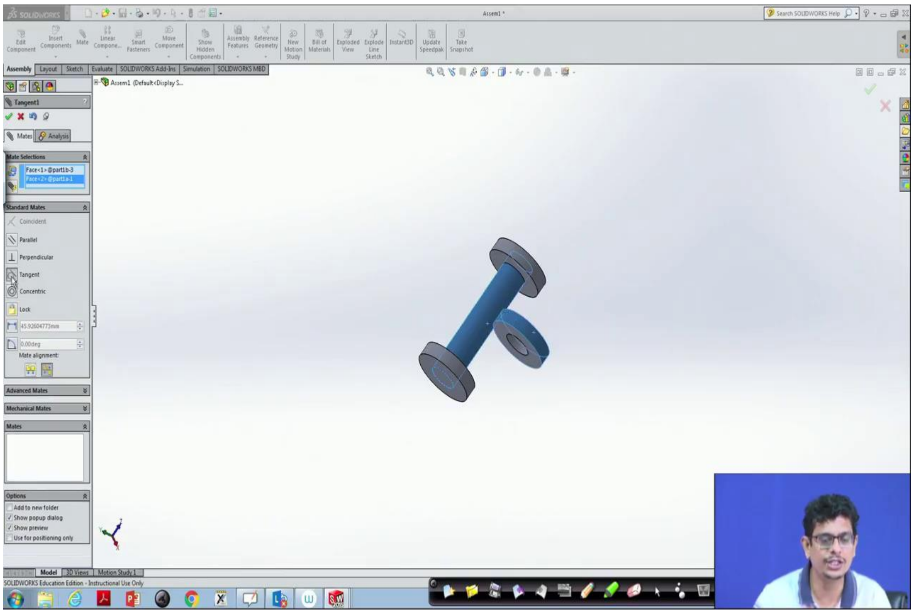

This is the cylinder, but tangent only I would like to have to click ok. So, this object always is I can move anywhere, but it always is tangent to that surface. Whatever I do it always be tangent to that surface we will have it.

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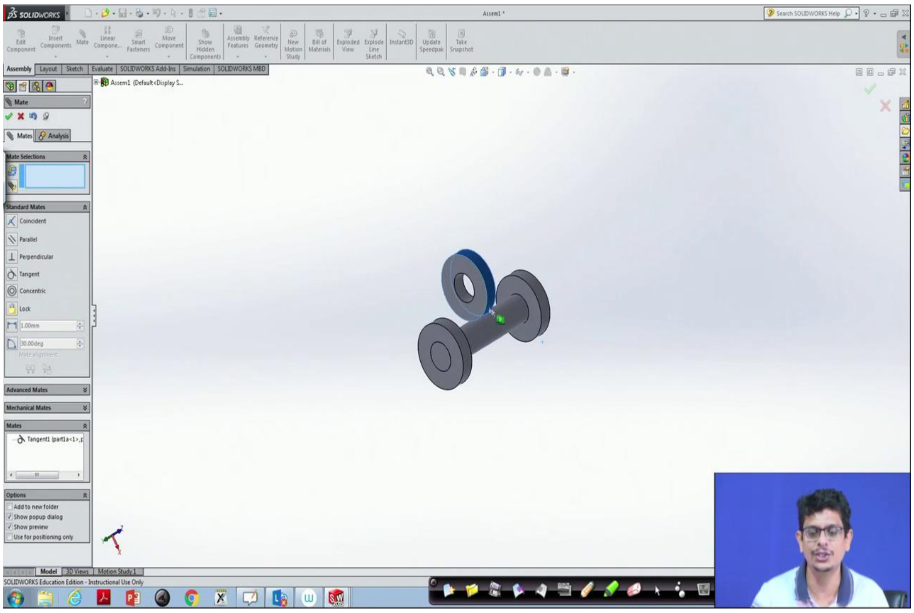

Now, I want to lock this surface with this surface. They should touch each other. For that purpose, what we have to do?

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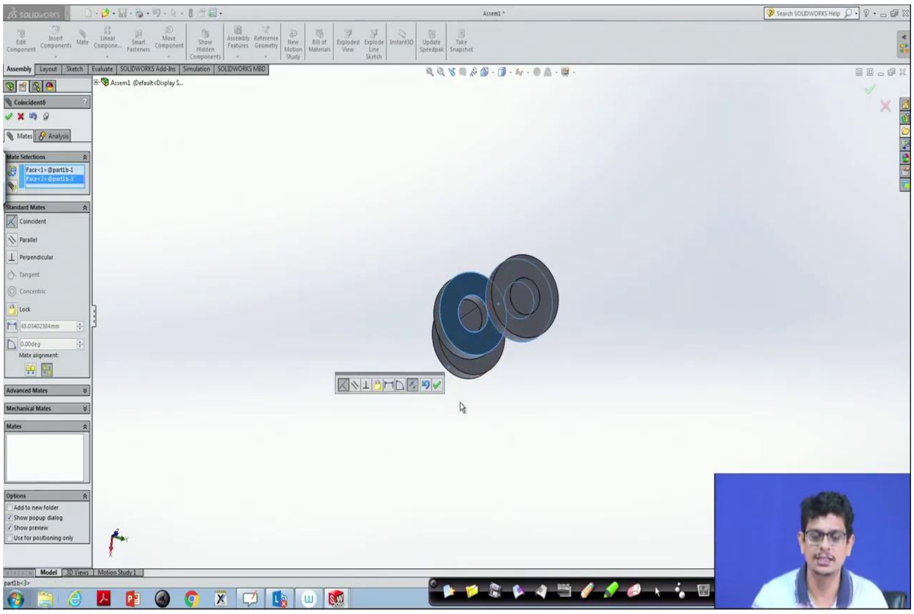

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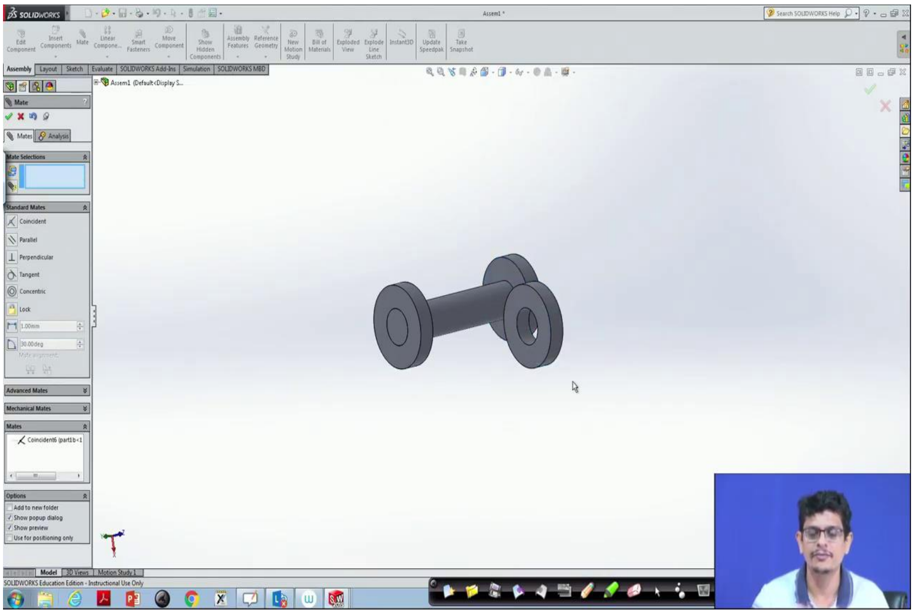

Once after clicking that mate this surface and this surface. You see they are coincident kind of surface that is the reason it picks this one. Now, click ok. Once it is done, if you want to move this object, it turns that surface and tangential to that surface only it rotates. This is the way we construct a simple assembly drawing.

In the next class, we will know about how to make pipes, hollow pipes, how to make threads over these pipes.

Thank you.