Computer Integrated Manufacturing
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Lecture 24
Laboratory Demonstration, Computer Aided Design (part 2 of 2)

Hello, welcome back to the course. We are discussing Computer Aided Design in this week and also we will discuss Computer Aided Manufacturing in the forth coming lectures in this week itself. We have discussed something about the Solidworks software. Solidworks is one of the very common software used by so many people, by students, researchers in actual practice.

We have discussed just the part programming in the Solidworks in the previous lecture. So, in this lecture I will like to discuss the next 2 modules in the Solidworks that is assembly and then drawings, how do we create drawings from these Solidworks, from the components for the assemblies that we have made.

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So, this is the software platform here, we have discussed regarding the part programming. So, let us try to see how do we assemble now?

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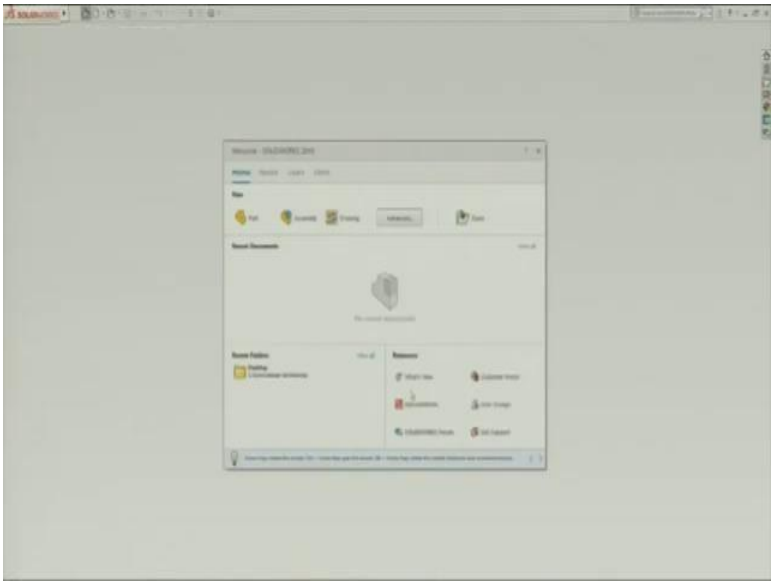

So, starting with this is the basic window, so let me click on home once again. So, this is the basic window of the Solidworks for the part.

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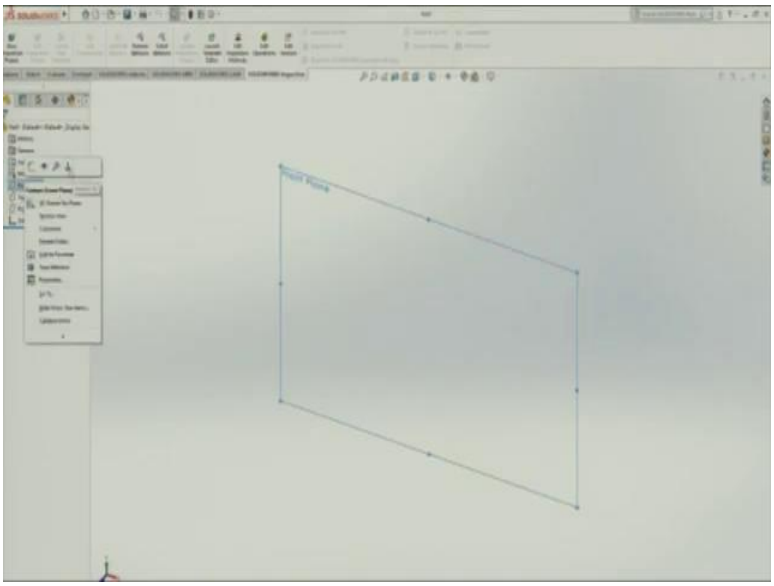

And we know that we can select multiple plains here. So, I will just, what I will do I have just selected part again and I am going to make a complete assembly. Which assembly I am going to make I will just show you.

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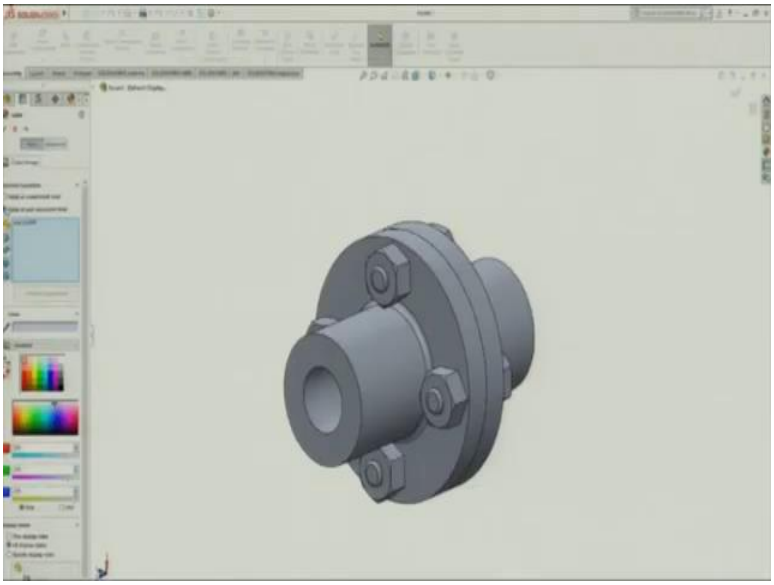

This is the assembly that we are going to make. What is this? This is a flange coupling in which we have 2 flanges and it is coupled using nut and bolt assembly.

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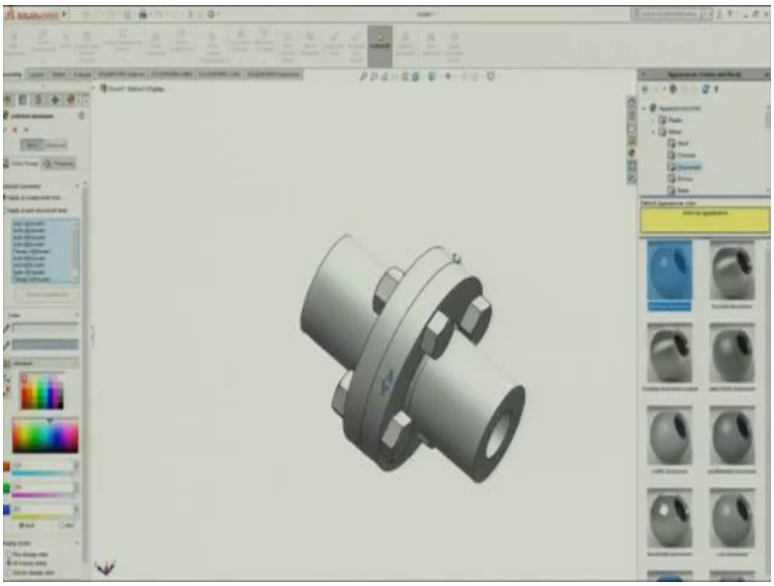

We will make each thing here nut, bolt we will see how to color them and we will see how to see the different views of this. This I will start from scratch, the previous video what we had? We had just crude information on what are the commands, what are the major operations in solidworks. In this we will make this coupling. So, I have just fixed some dimension for the shaft, for the flanges and for the nuts and bolts. So, I will move accordingly, so let me start from the part.

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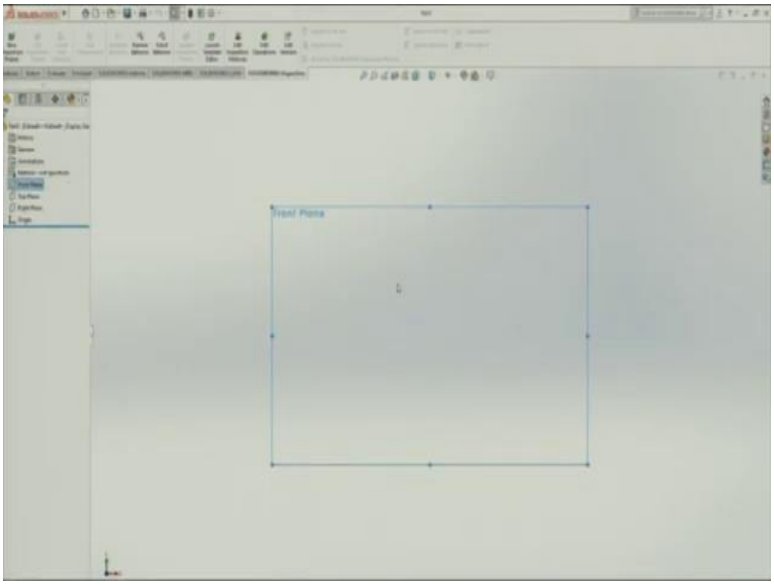

So, I am coming to part here, so it is opening now. So, basic window starting how we have seen that in the previous lecture. So, this is platform now I need to see that at a plain again immediately I will make components for the flange coupling. Flange coupling is used to connect 2 shafts which are concentric. So, we need 2 flanges which are identical so let me consider the shaft dia as 20mm. So, I will do accordingly, so 20mm dia shaft has to be fixed using this flange coupling.

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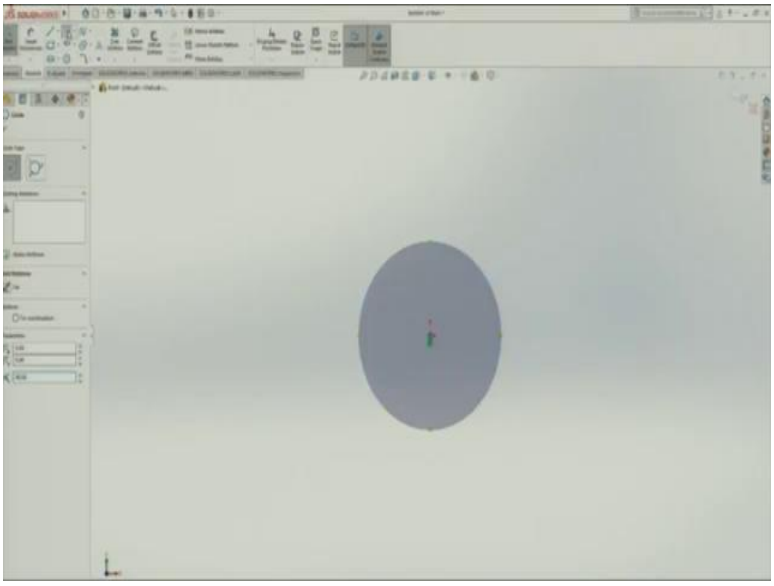

So, for making the flange first I will start form the circle, this is a circle, the radius of the circle first I will put the outer radius I will put radius as 40 because the dia is 80 millimeters here.

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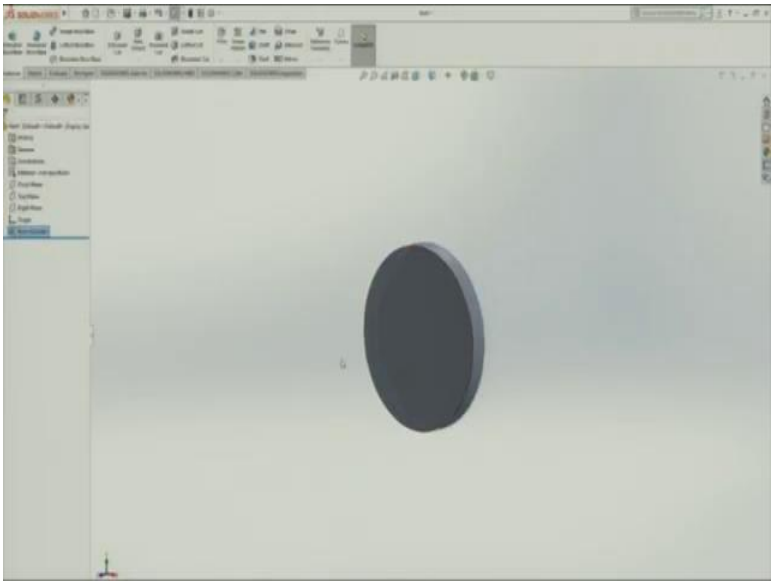

So, let me extrude that because this is just a flange. So, I accept this so now the extruded box so, I will put 10mm.

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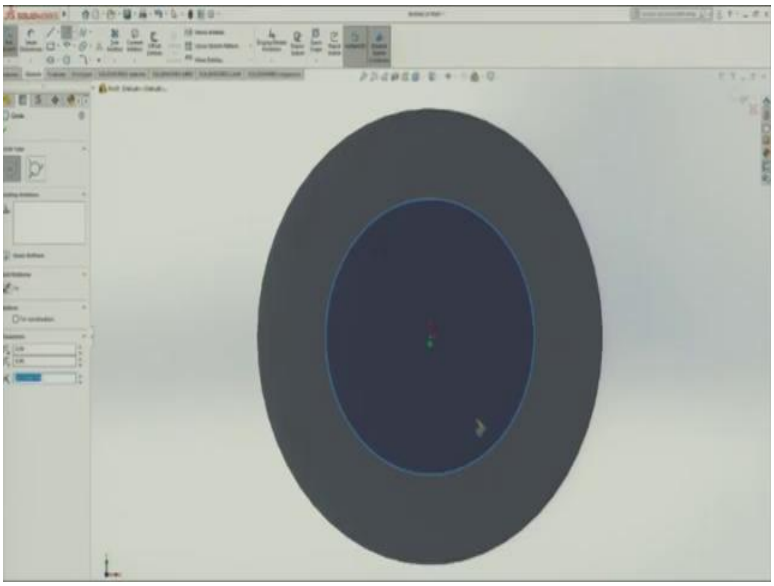

So, now this is the flange, so now I need to cut extrude here to make a hole inside it. So, we can select this and make it perpendicular we have made this phase perpendicular.

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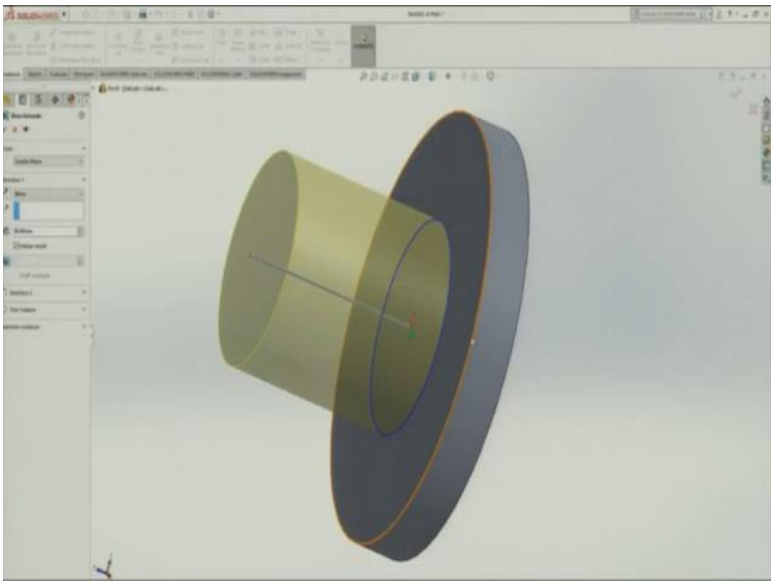

So, this extended dimension here is 40 for the half of 40 is 20 so I will put value 20 here.

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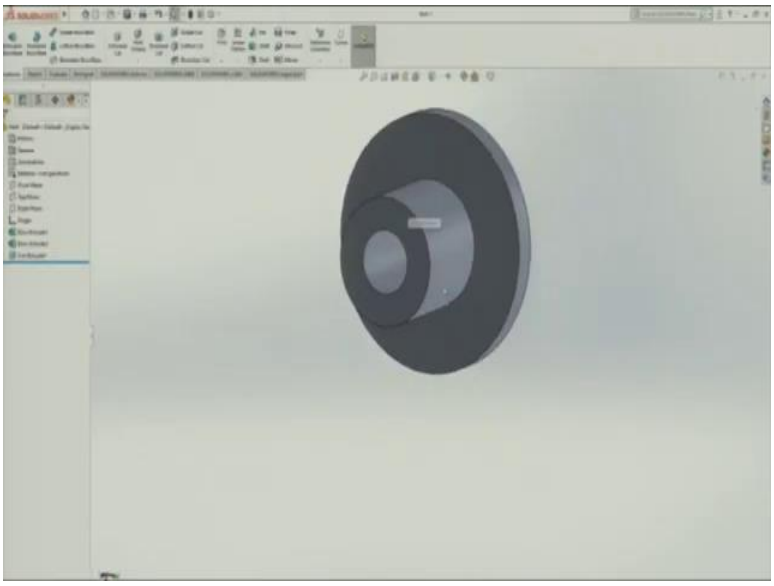

So, we need to extrude this, let me extrude up to 40mm. So, this is if extruded part the inner side so this is extruded now. So, now we not do through hole in it, so the hole has to be dealt therefore I need to know the shaft dia.

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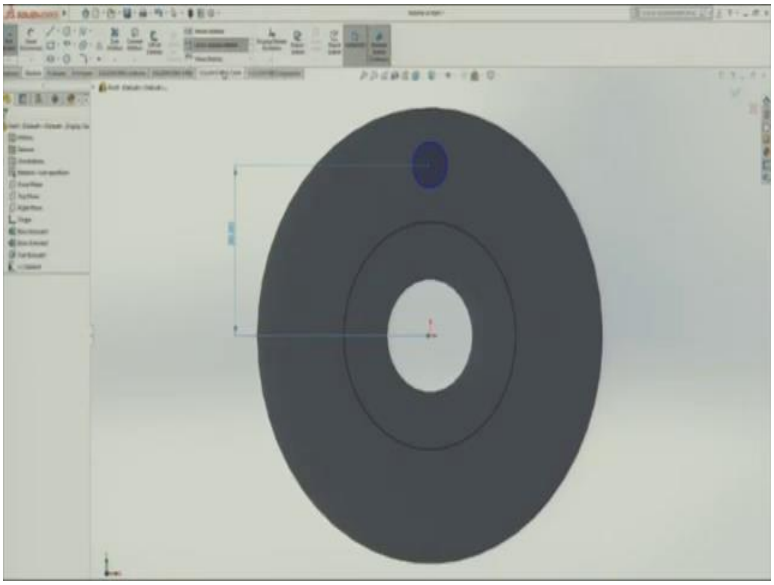

So, shaft dia let me put some circle here. So, shaft dia here is 20 millimeters, so 10 mm would be my radius. So, I will make extruded cut here. So, it has to be through so we can just extend the arrow or we can just put a through hole, both the ways could work. So, through hole that is all. Okay. Through all okay. This is shaft diameter that is made now here.

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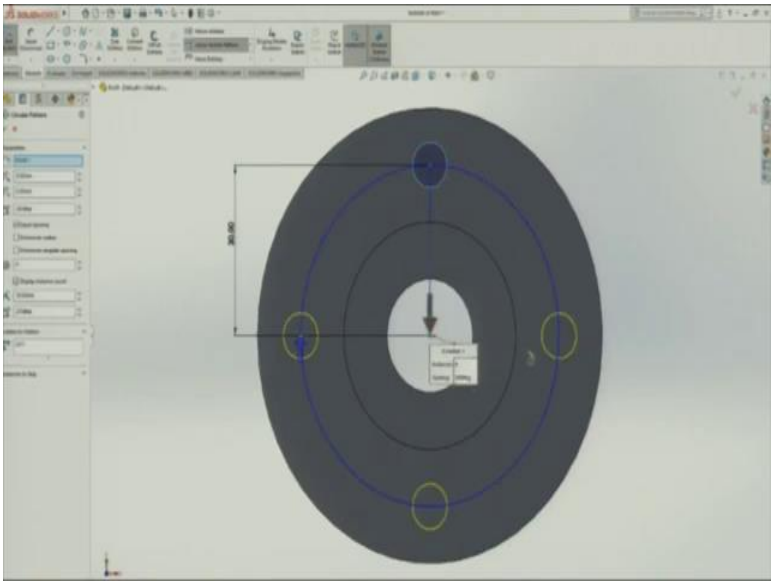

So, now we did a drill hole here for the nut and bolt system. So, nut and bolt should be perpendicular to the axis here. So, diameter for the hole is 8 mm, so before 8 mm dia I put 4mm as radius.

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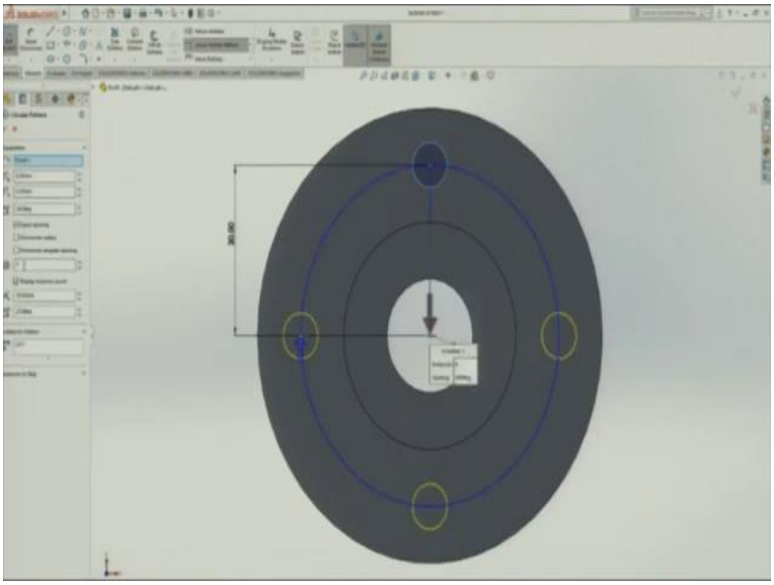

Now smart dimensioning is one of the places that I will show you. The smart dimension is to put the dimension in a proper location, so I need to put this in the centre of this. So, this is I will put value 30mm here, 30mm distance from centre. So, this is smart dimensioning, so this is exactly 30mm from the centre that is in the centre of the flanged part of the coupling. Now, I need to put 4 holes I will put linear sketch pattern here.

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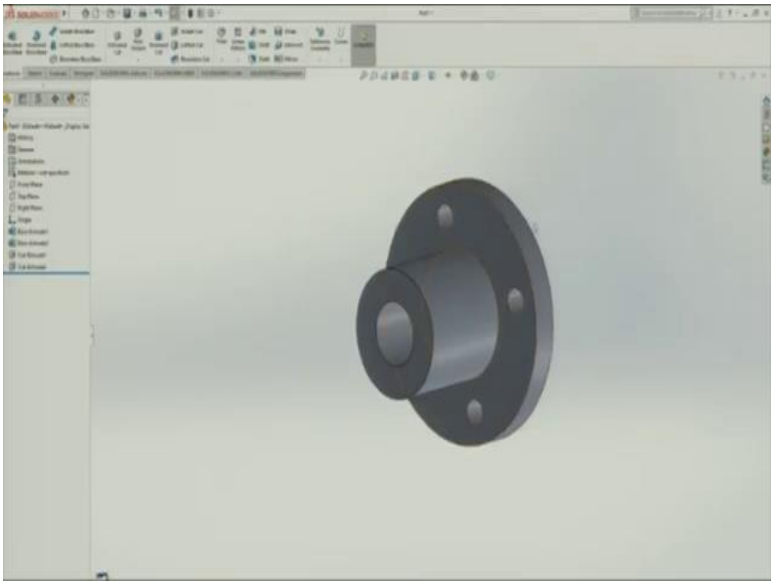

So, simply linear sketch pattern I can put just 4 is already selected, okay. 4 copies, all the dimensions are the same. It is already 360 degree, total number of holes are four. So, each separated

by 90 degree. So, I will just accept this. Now, this hole is made, now again extruded cut command to make a through hole.

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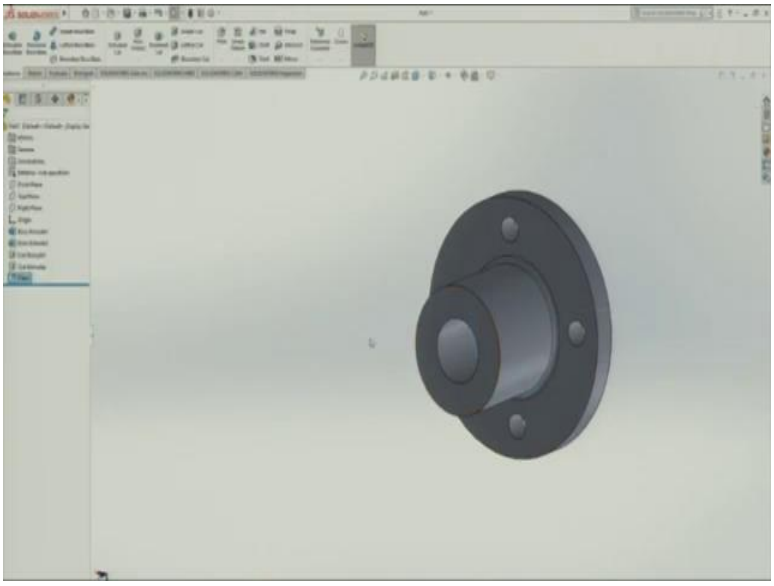

So, extruded cut all the 4 hole are selected in one go and extruded cut command is applied. Now we have good holes.

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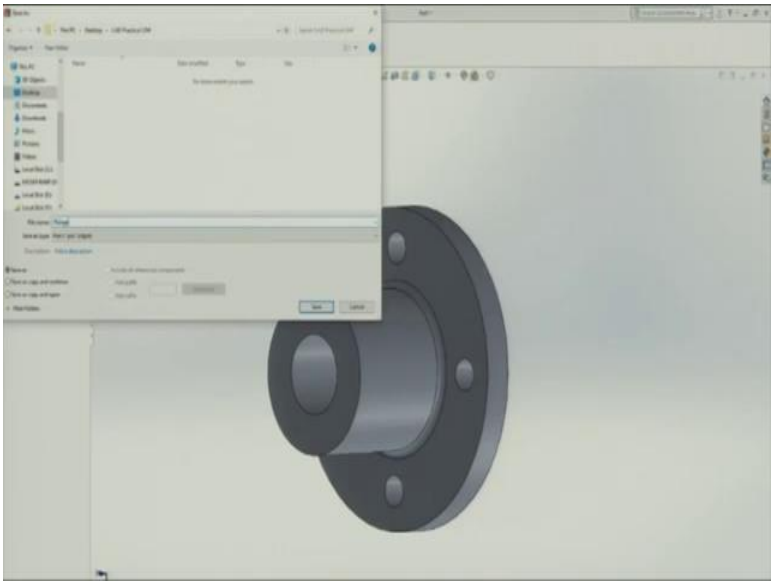

Now, this is the flange that is ready, now it is almost ready but only things that we need to take care of now is the fillet part to make the corner smooth, the edges smooth here. So, constant size,

various size fillet again, the fillet that we discussed before are here. They filled on fillet so let me make it constant size fillet here.

So, the radius for the fillet here is I will put 5mm and I can visualize that how does this 5mm looks like, if it looks large we can decrease it or we can exchange accordingly. Okay, I will put 2mm, 2mm looks better. So, 2mm, press control + S for saving it. I will name the part as flange. This is flange, this is one of the components. So, this is ready. Now, again one more part I need to create bolt and nut.

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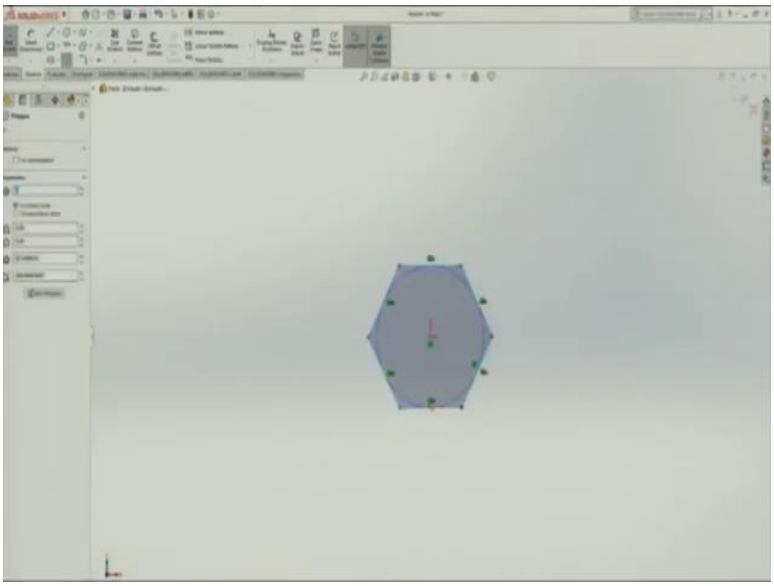

Let me first select bolt to be made here. I will select a plane, front plane here, I will select circle, okay. For bolt I think let me start from the head only, the head for the bolt is hexagonal, let me have hexagonal bolt, so a hexagon here.

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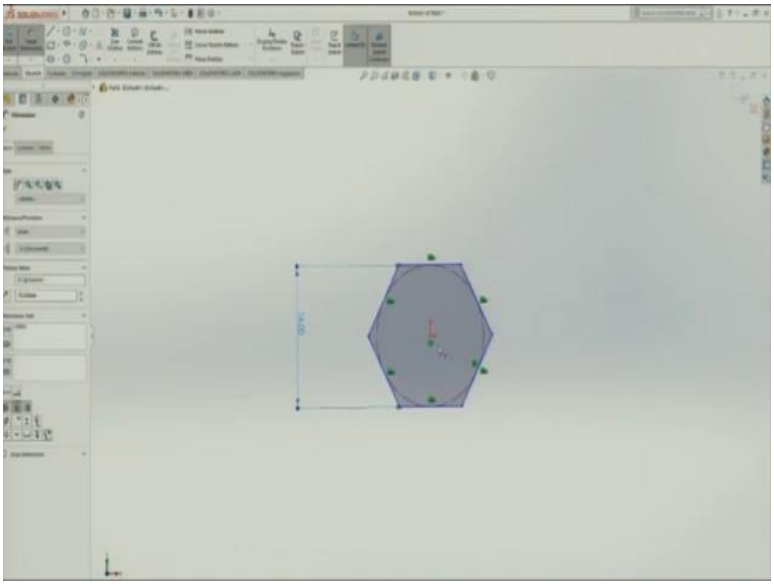

So, a random size is selected let me put the dimensions here. So, this is width across the edges is actually 1.5D plus 3mm so my D here is 8mm, 8mm hole is there on the flange I am going to put some value here. So, that value 16 is selected, also I am making a little, also I am making a little changes I am selecting my own dimension, however the width across corners has to be 2D, I am putting 2D for widths across the edges.

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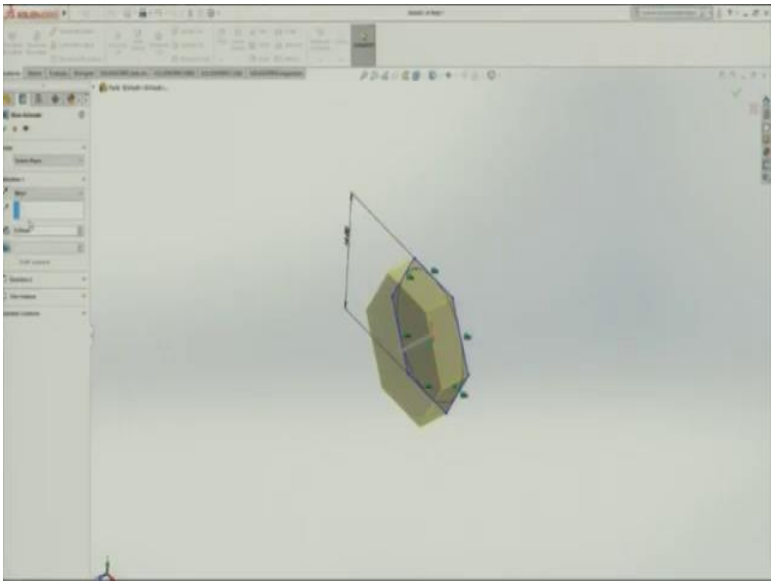

So, this is a customized bolt that I am making here not a standard bolt. Then I can extrude this hexagon. Let me make the head of the bolt as 5mm thick so this bolt is there, bolt head is there.

So, we can just increase and decrease from here itself, let me make it 5. 5 looked a little smaller so let me change it to some value, may be 7, 7 looks okay.

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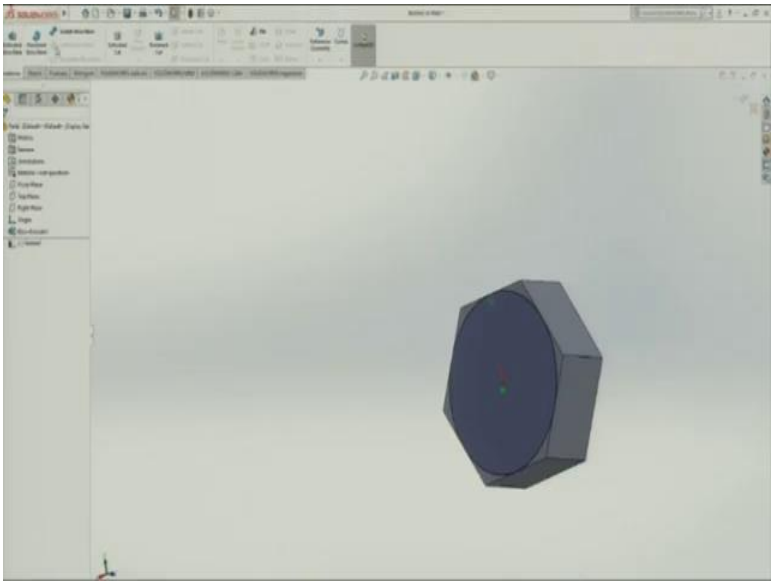

So, this looks like the head of the bolt, top of the bolt head is to be still made, now I will select a circle, I am trying to align it with the edges of the hexagon here. So, now here also I need to select them simultaneously, I select this and this control.

Now, there is a command here, there is a relation here, tangent, if I click tangent it will bring the circle close to the edges and make the edges as tangent to the circle now you can see here, now we

what we can do? Because the corners of the head are filleted, it is better I would say fillet would not apply well here. It is actually drafted here. So, draft command I did not discussed, draft command is something like making an edge slant. For instance this is completely rectangular body.

So, if I need to draft it I can put some angle here. So, this body might become something like this. This is draft command. If I say this line, this is the draft command the body would have, suppose if, and let me say this thickness 30mm, if I need to put only 2 half, 20mm this side and 30mm this side, it will have some taper in it, it is draft command. This could be applied to circular surfaces as well.

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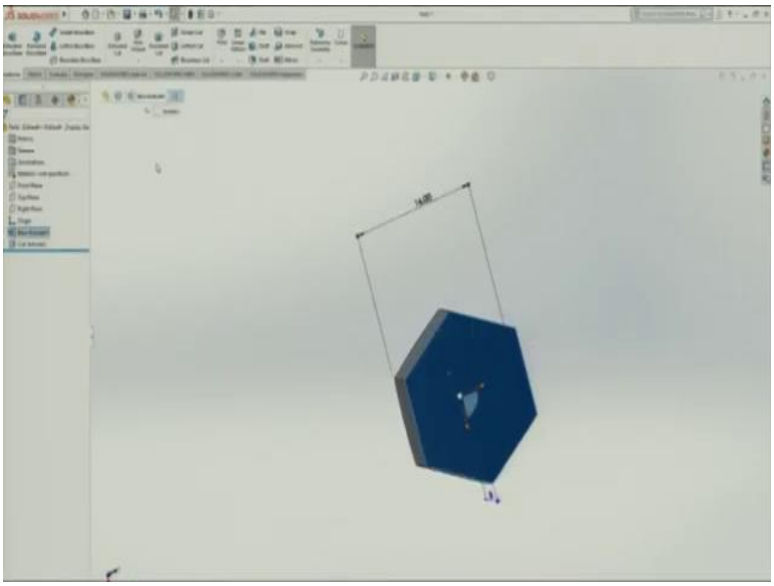

That draft operation I will just apply here. So, how do we do this? So, I just need to flip to the side and I will try to make a draft here. So, you see 60 degree draft is made. The corners are now a little are brought to a 60 degree angle that are smoothen to the top surface of the bolt. So, next I am trying to make the threaded part of the bolt I will not make threads but just as a presentation. So, the hole size is 8mm, so I need to put I have selected this circle of 8mm size. So, let me say okay 80mm bolt. I have made it a little 80mm is too big, 60mm, 60mm will be big as well. So, I will might change it when I go to the assembly.

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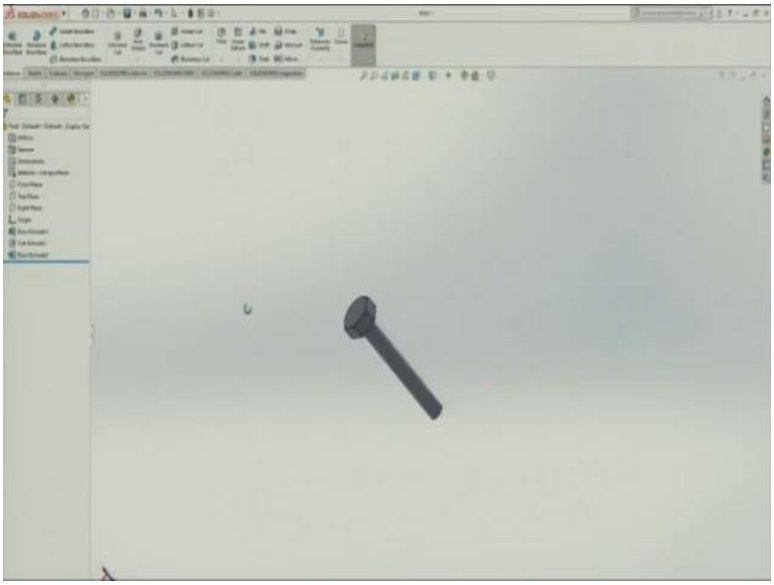

I will see I show you that whilst assembly goes well we can change the component dimensions or component properties and that would changes all the components. So, I can put some fillet here, around the circumference of this, the extruded part.

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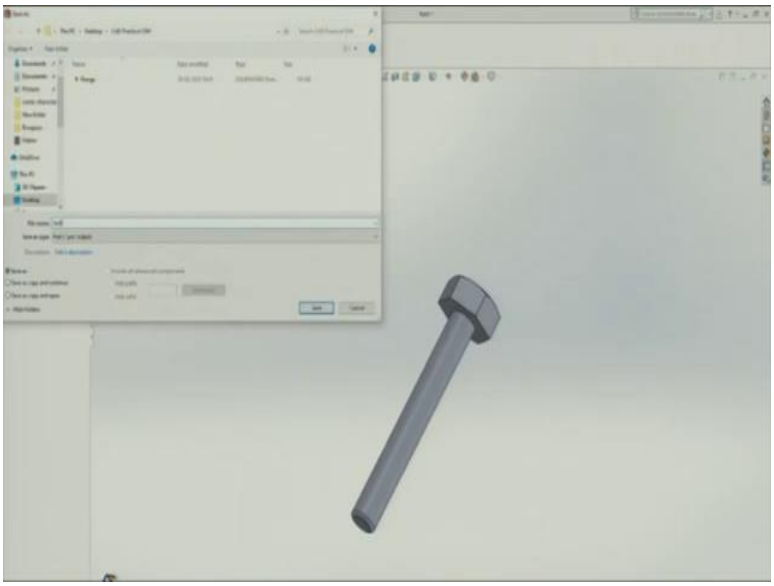

So, I will name this part as bolt. Bolt is saved now. So, similarly, I will select a new part, new part I am going to make is a nut.

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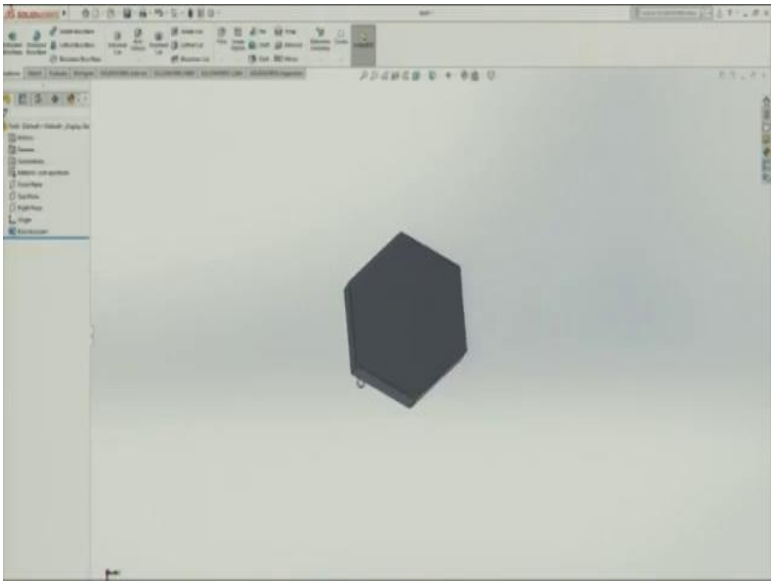

So, sketch a plane, normal 2x2 plane, then circle, okay hexagon first. So, again same size 60. Then this is 2D, this is 60, this is 60, twice dimension, the customized bolt selected. Okay, applied. So, let me extrude it, this is extruded to size 10 is little larger 5 would be lower, I can put 8mm is okay.

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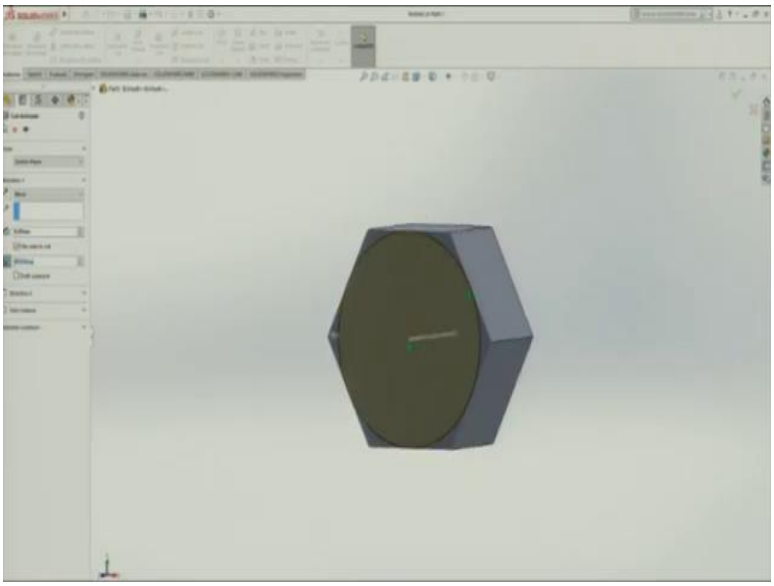

So, this is the, I would say base for my nut, I need to now put again the similar operations, I put circle here because I need to put some draft over it. So, again this smart dimension, this to this. So, better I do it in a way to make it tangent, then this, circle and hexagon, tangent that is now. So,

again a 60 degree draft has to be put into the extrude cut, the flip side to cut 60 degrees. It is, actually draft is being applied here using extrude cut.

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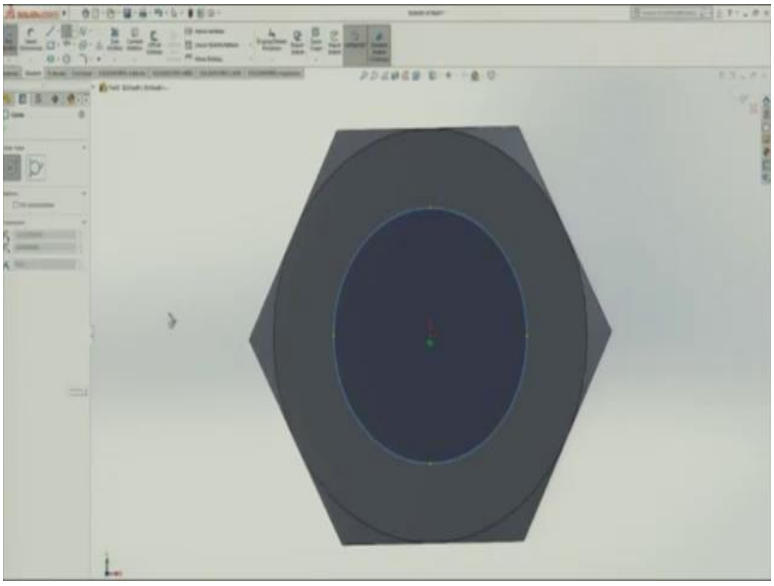

So, now this feature is almost study okay in this, this side I need to put hole now here. This hole size is 8mm dia and 4mm radius, I need to cut extrude this as well.

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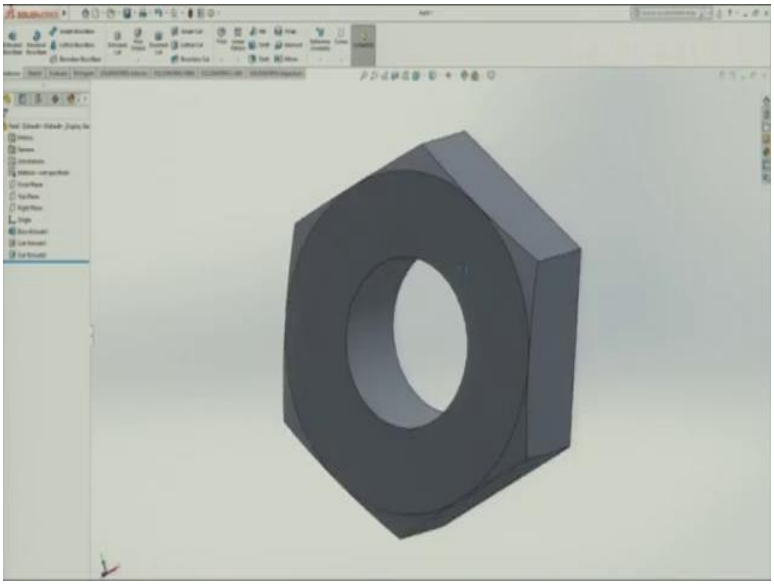

So, I am not focusing on threads at all. I have just made a hole that is to be fitted here.

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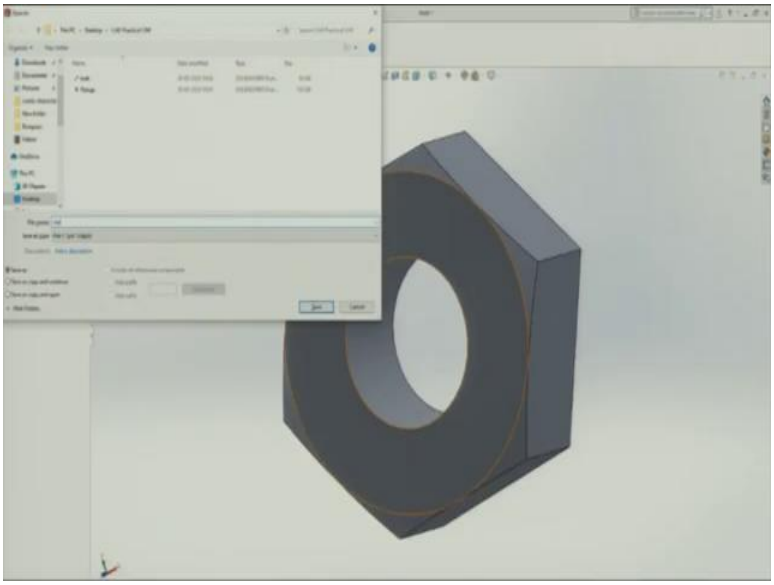

So, I will save this part as nut. Now, the three components are ready. Now, I am going to assembly now.

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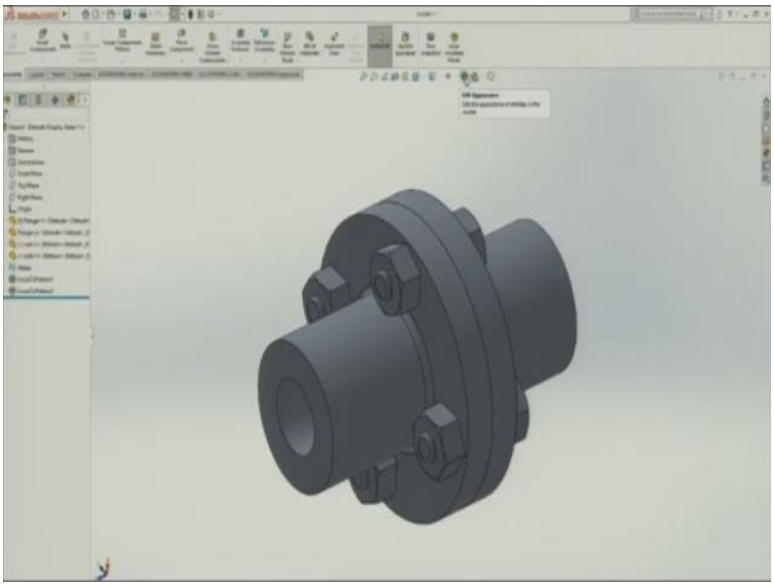

So, this is the final assembly that I need to make. In this we can see we need two flanges, four nuts and four bolts. So, to make them let me start with my assembly part. So, I have made nut, bolt and flange.

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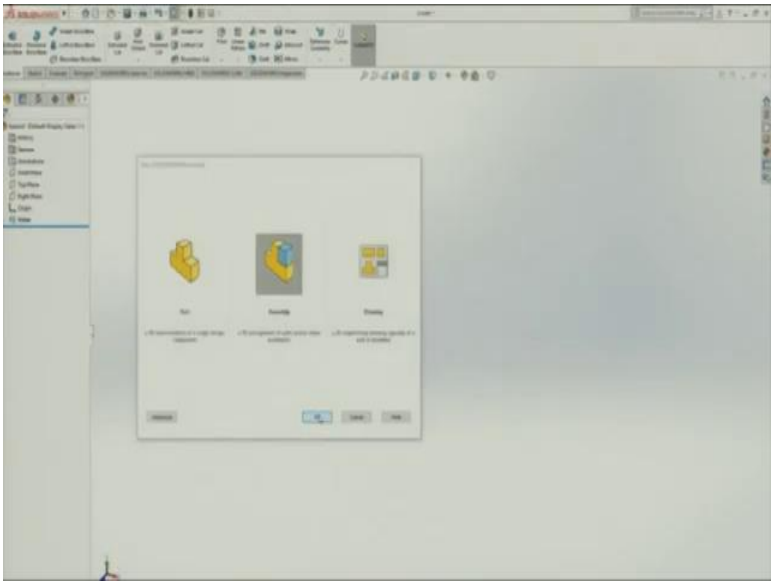

So, now I will again open a new file, I will open assembly here, its empty okay, when I click assembly it is showing the components those are there. Flange is one of the components here, the three components here, we can also browse a component from here, three components already we have made putting one document, putting one folder.

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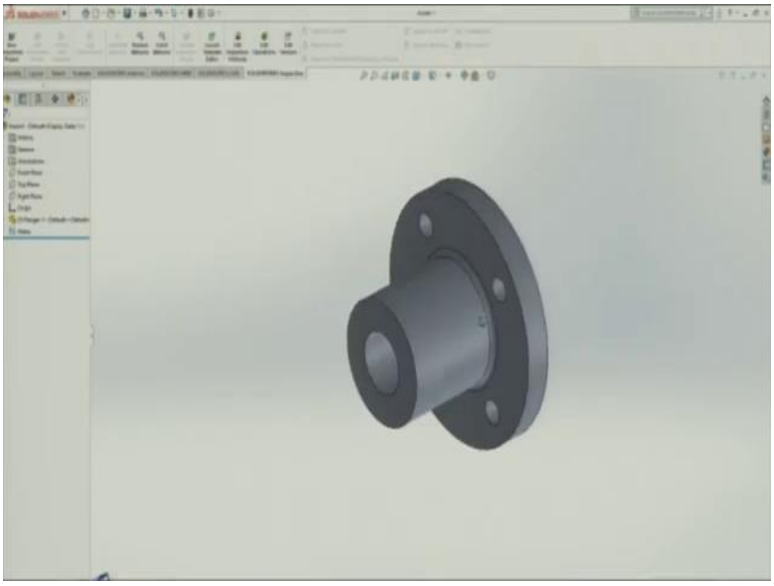

If I select flange from here. So, as we need two flanges, so we know that and we do need to assemble them as well. So, this is the flange that we have made.

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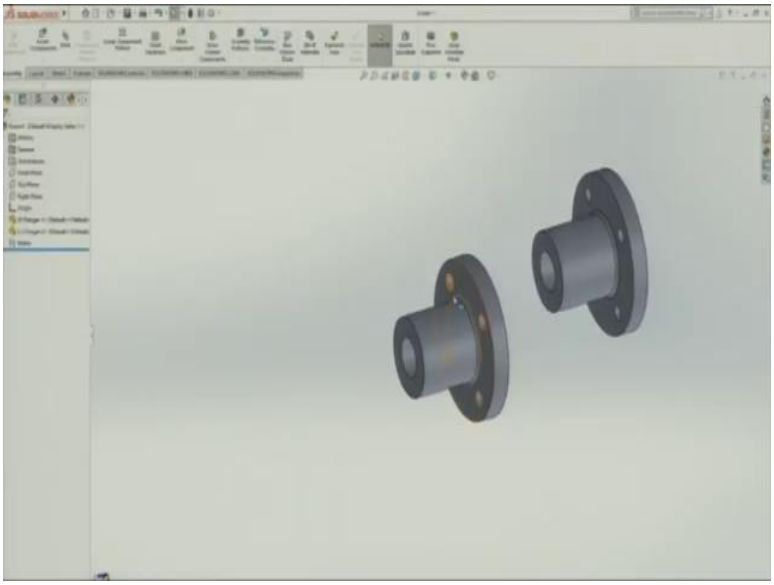

So, the second flange can again be inserted while again opening the part list, and clicking on them, assembly. So, I will, again insert a component from here, again a flange. Okay the flange has come now

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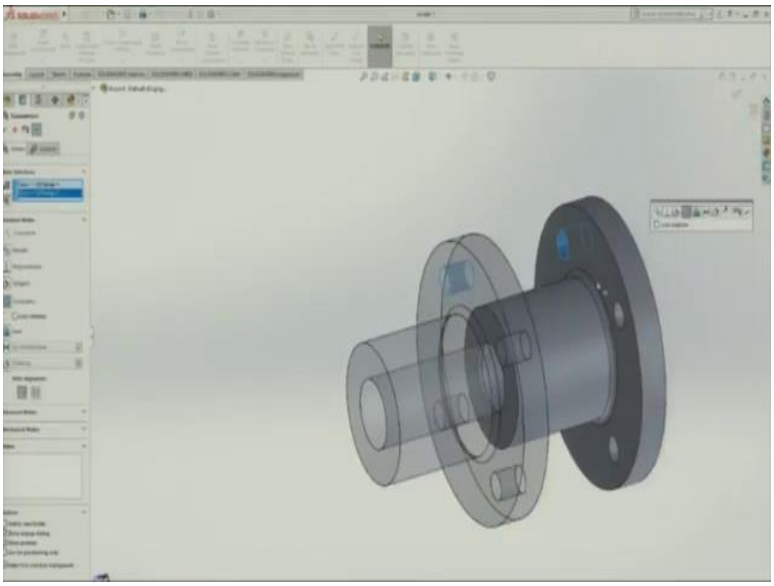

Now, these components to be mated and one of the flanges to be flipped over. So, as we know that when we have to mating the hole has to be coincided, so I will just mate the holes, so I will mate this hole with this hole. Okay this is mated again please see.

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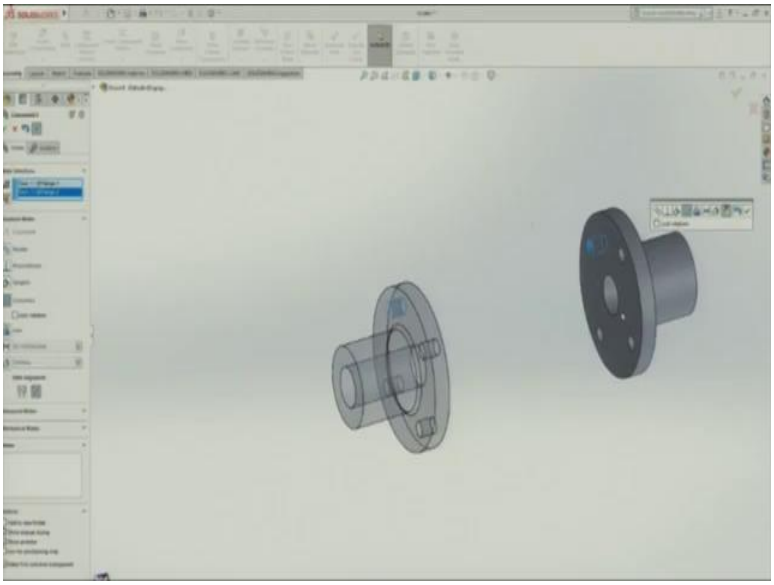

Again I am going to mate command, I am making this hole concentric with this hole. So, this has gone concentric now.

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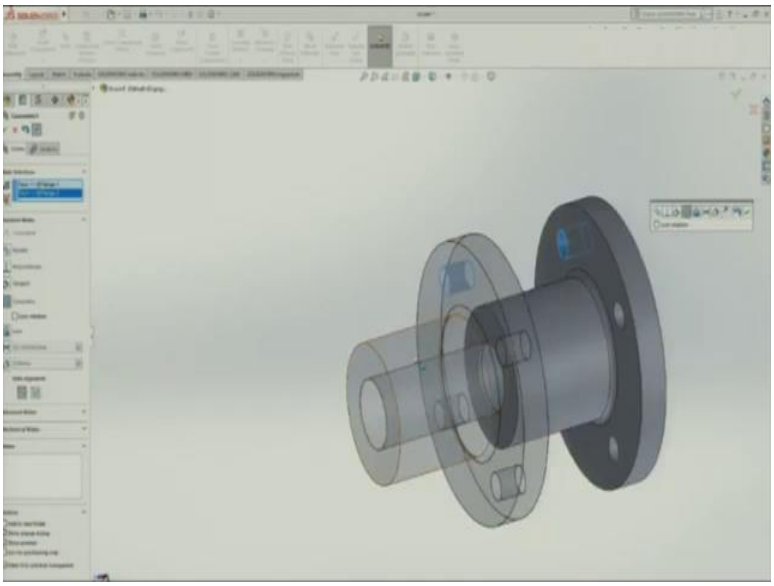

So, I have flipped it over, see this certain options here. See there are certain options here. I need to flip it over, to flip it over see lock rotation. I have flipped it over. Now the holes are, holes axis are coincided and they are now concentric circles.

So, concentric, they are coincident, parallel, perpendicular, tangent, concentric, you can lock rotation, so there are certain ways. So, on this circle is concentric, this circle we need to make concentric. Now, this is accepted.

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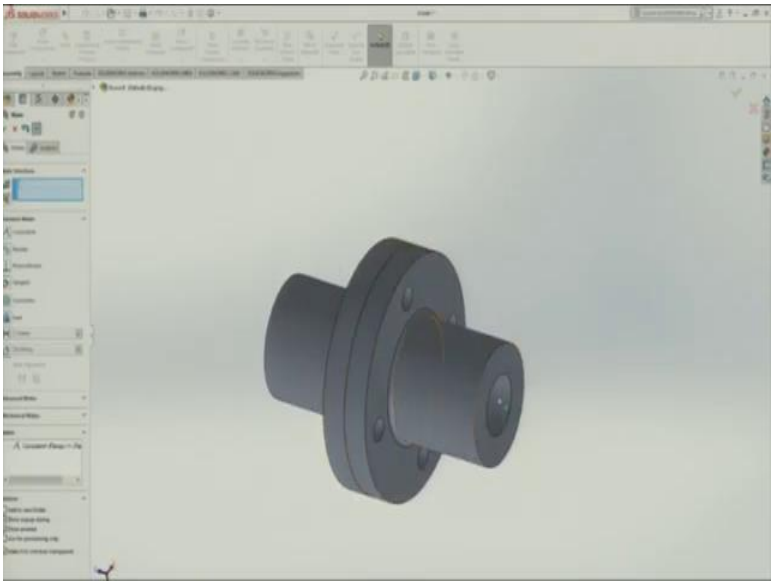

Again this phase has to be mated with this phase when I say this has to be mate, it has to come closer. So, in this case, now these are in the same direction now I have to stick the faces of the flanges, this face is the face of the other flange, I will again put mate and put select this surface is to be mated with this surface. So, the mate has happened. So, now the flange setup is ready. So, it is mated now I need to insert components again, I need to insert nut and bolt.

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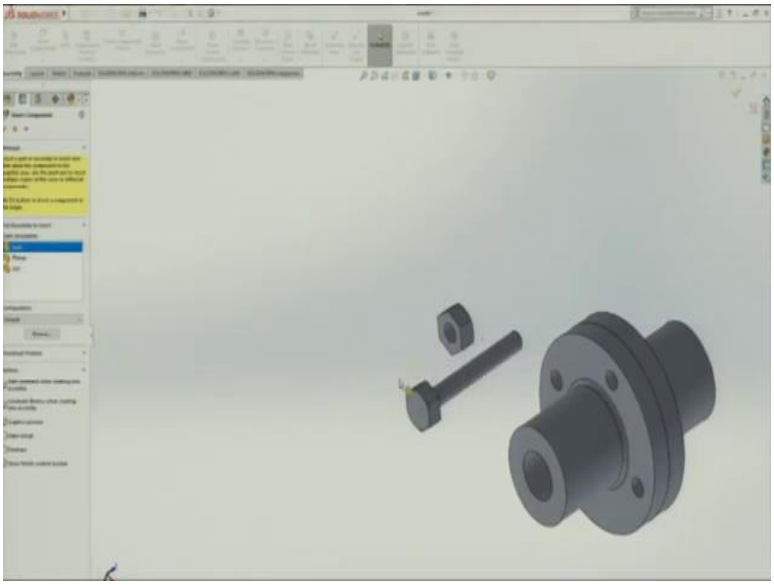

So, I will insert nut here and I will insert bolt here, insert component bolt, so this bolt looks a little larger, the length of the bolt is larger, so now while working as well in this assembly we can change the dimensions for the bolt as well.

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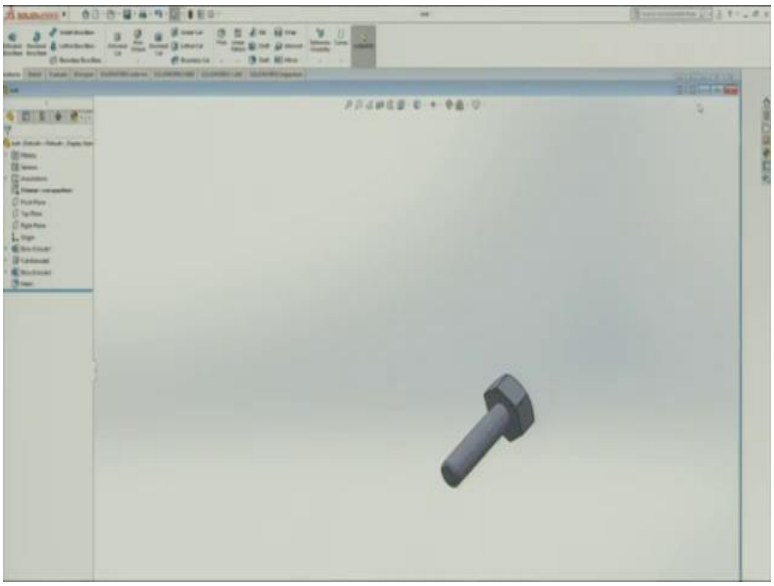

So, these components are here, this is the component bolt, I will change this dimension, so this dimension, I am just making it may be close to 30mm, this should be okay. So, again I am opening the assembly part, in the assembly okay now it looks too good. So, again mate, we have to mate this circle, this hole with this shaft, this is broad concentric, till now it is just mid concentric, axis,

has come into one line, second meet command is between the surfaces. So, we have to coincident the faces, this face and this face, yes now it is going in. This is known as mating.

All this is done, now that nut has to be put on other side, the similar command is used here, I will just put the circle and the okay, yes now this is interesting, now where has nut gone? It is mated, this is important to note since we cannot see the nut here, so we need to bring wire mesh, or need to make something transparent to see the nut, so we can hide the flanges or we can make them transparent.

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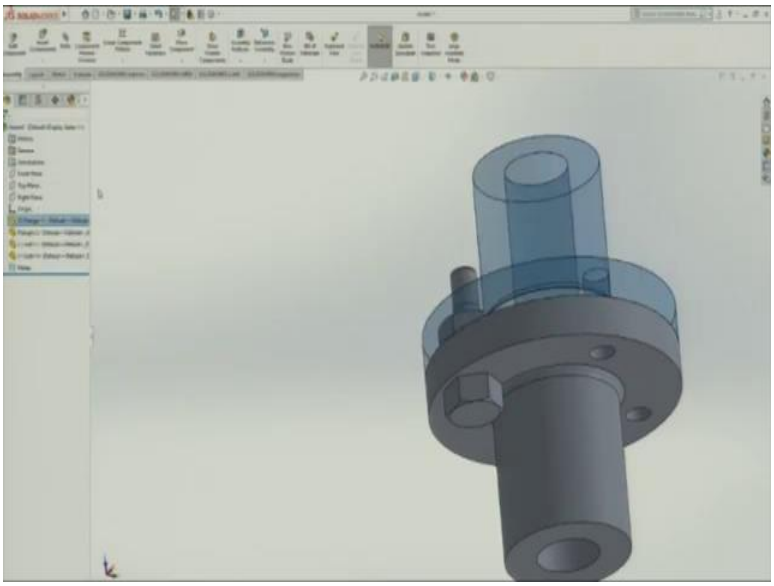

I will go to flange here, flange, flange properties, change transparency, if I go to change transparency here, it is change their transparency, now we can see the nut inside. For the second flange as well I will change the transparency. Now, the transparency has been changed, it is just showing the edges here. So, we can see the nut, because nothing is still locked, we can easily move the nut, okay.

So, we can move it, it was not locked, it was not mated, the surface were not mate. Again met, this thing, surface mate with this now it is ready. Now, this assembly with one of the nuts and bolts is ready. This has now one nut and bolt and the flanges. Now, I need to create three copies of these to it four nuts and bolts.

One of the ways is to insert components separately from the parts components that we already we have like we did it in flange. The another way is to make the pattern, because we know there is the 4 nuts and bolts in the 360 degree rotation here, so that same thing we can apply here.

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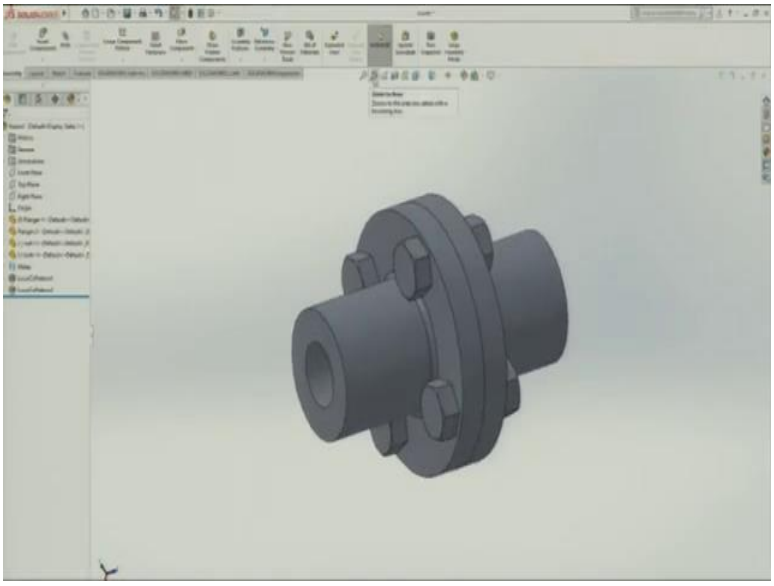

We will make the copies of copies that those are there in the parameters, we need to just rotate it we will make 4 copies of this pattern, spacing I will put 90 degree and did we along what I will put, equal spacing in 360 degree it is better, so yes 360 degree equal spacing. Now, bolts are copied or a pattern of bolt is created. I will accept it, I did not select nut along, similarly we can do the nut, we could have done it together.

So, again a circular pattern for the nut. We can delete this pattern, this nut, again this is the parameter, face of flange 4 copies, yes, it is okay. Click here and select. Now, this coupling is complete. I did not considered the threading part, I did not considered the key, there is actually in a coupling to fix the shaft, there is also a key in the shaft that is also there with the coupling so that the shaft is locked with the coupling with the flange. So, those I have not putting it right now.

So, just consider that this is the assembly, just a basic introduction to how does assembly happens in solidworks. So, let us now try to see the different views here. So, this is zoom to fit, effective this it will fit it best to the screen, zoom to area I can just select, the whole area I can select, a small area I can zoom it, suppose if I need to check this area, this will zoom it to this area, then let us check next is previous, it was the zoom to fit.

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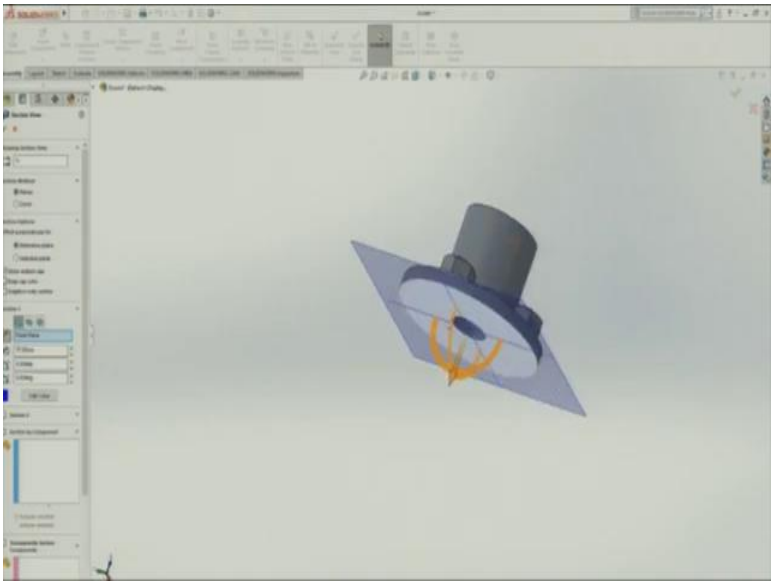

So, this section view, this we could section at any plane that we need to see, also we can rotate this, to see the section at any angle if we needed, we will show this section along this plane, I can just make it back and forth, and also rotate this to see this section view. So, this is just how do we like to have the output of our file. This is the section view, so this is 3 views that we need to see, which view we would like to see there.

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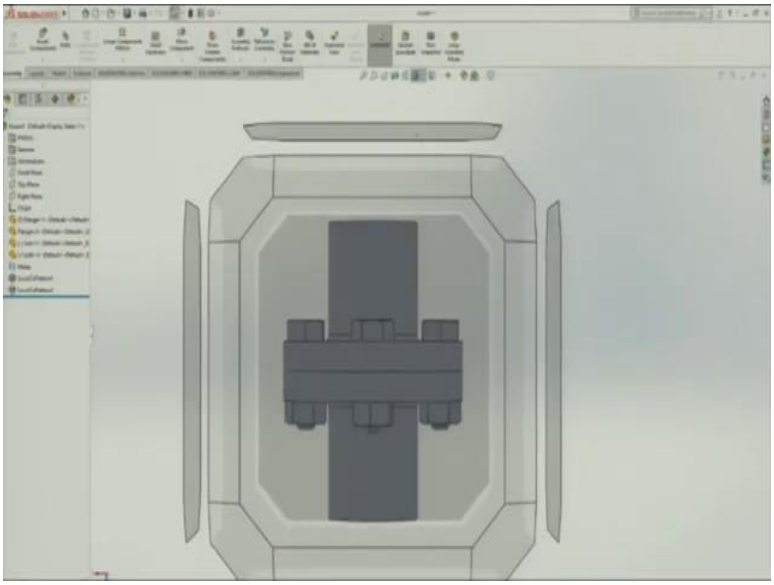

Suppose we need to see this, we can click this so this box appears, so suppose if we want to see this we can easily get this. See these options, it is the shortcut of getting the views. You can see, so isometric view is here. We directly see from here, you can see any face here. So, many views here.

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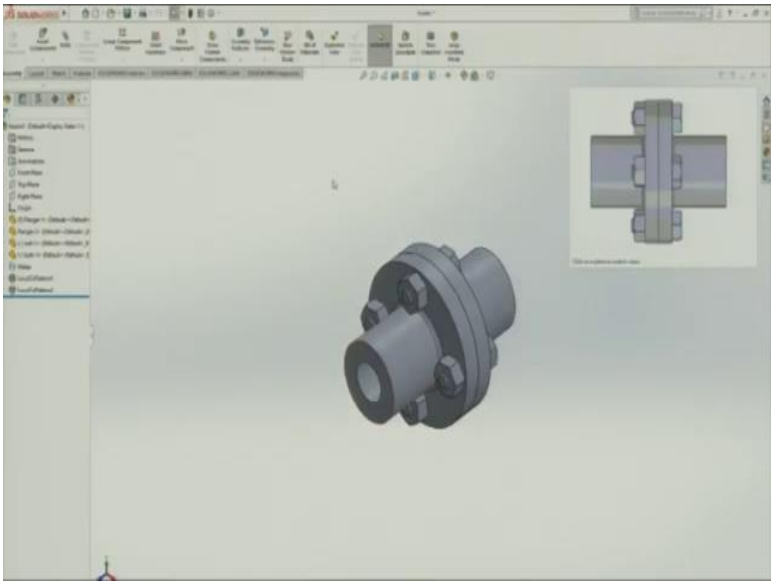

So, this generally viewed in isometric, this is isometric view in which we have all the views, top, right and left, so also we can change the edges. We can change the views solidity I would say, now

I have removed all the edges. So, as edges are disappeared from here, also we can see only hidden lens removed and only the edges.

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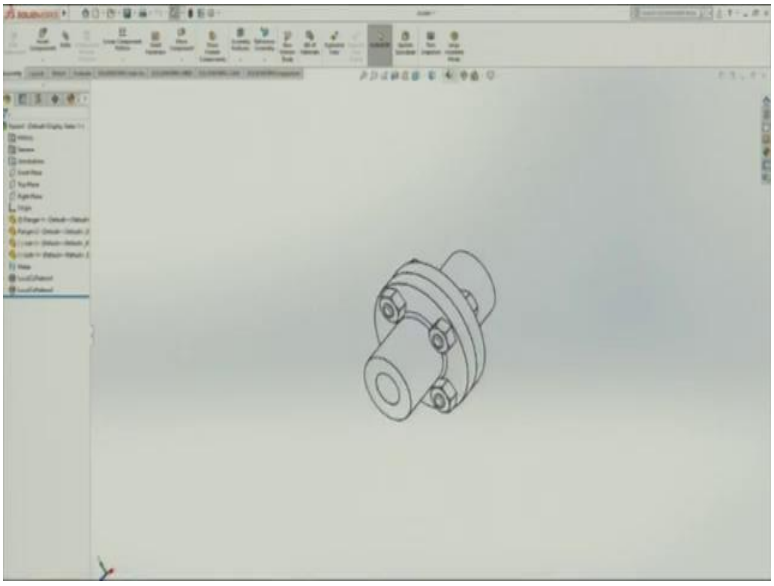

So, this is the one of the views that is prominently used in this industry. Yes we can show the hidden lines, you can see the hidden lines at the back. Hidden lines the dotted lines there are shown, being visualize here.

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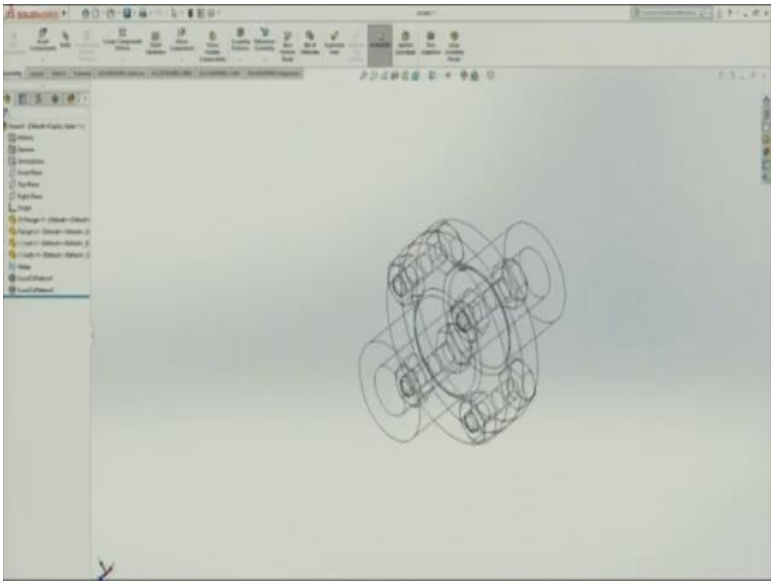

Also this is mesh, this is wireframe, wireframe is little difficult to understand, generally it is not shown. But sometime contingent requirement just to see whether what are the various internal components, internal lines, or drawings, dimensions, sometimes also shown.

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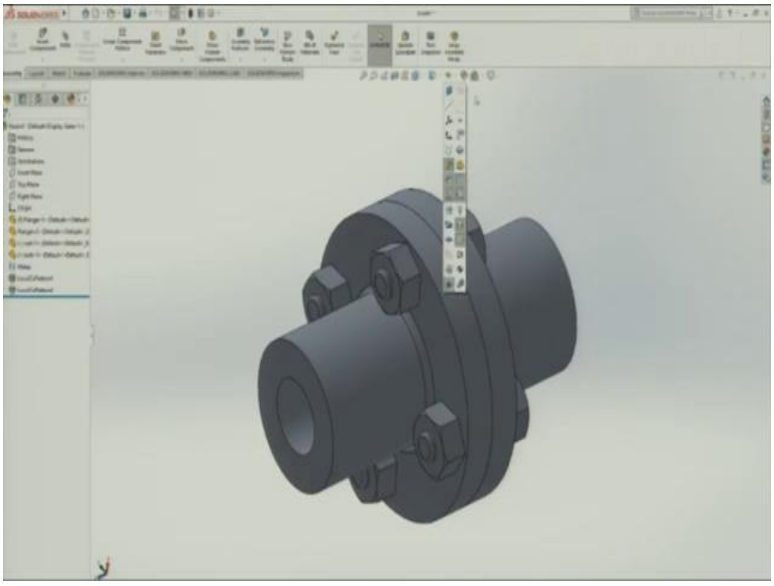

Also yeah these are also the shortcuts that we had been using. The view plane angle coordinate system, view planes, the various shortcuts that we have been using here as well. The shortcuts are there directly, list here. So, these are also a few views we can put grid at the back, we can view coordinate systems, we can view axis, view planes, then view lights as well, we can see shadows and lights. So, this is part design options were different, for assembly options are different.

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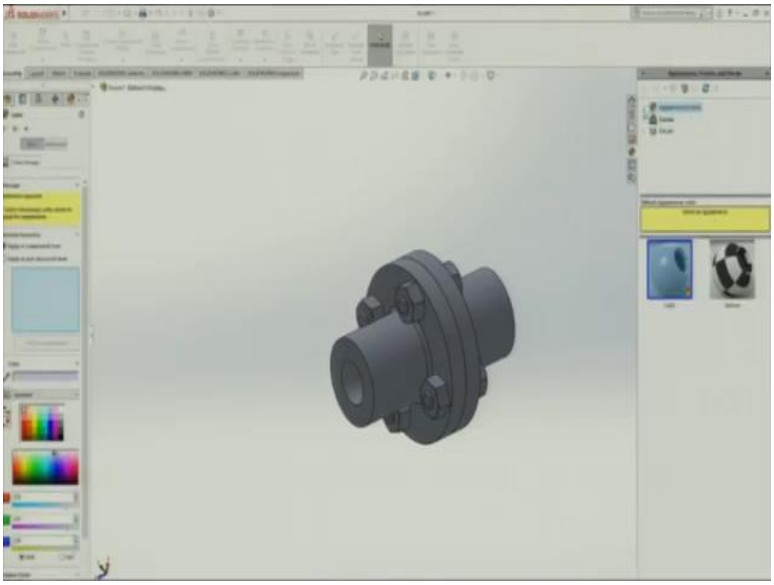

What test is this? This is regarding edit appearance. Edit appearance, this is just a mode appearances, as we know that material of the components I can just change the appearance accordingly suppose our component is made of mild steel, aluminum, magnesium, copper, gold, these all metals have different colors.

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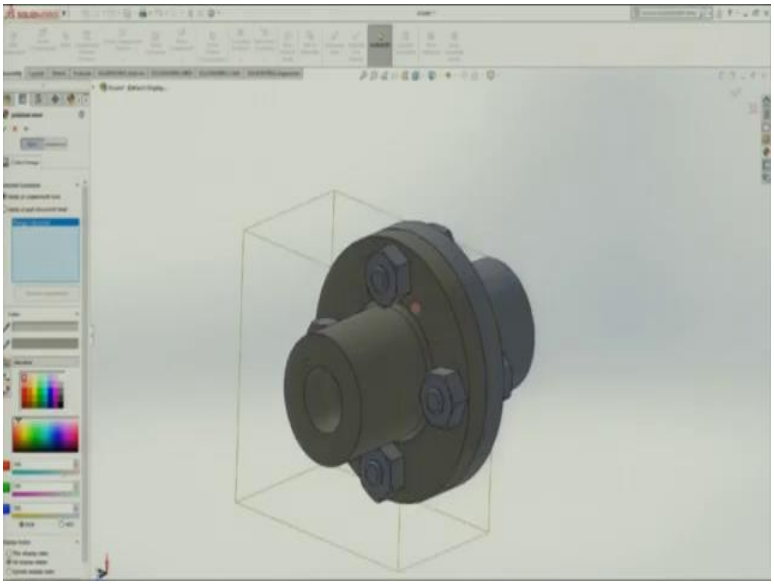

For steel, it is like polish steel, if it is polish steel, it will show it like a polish steel, okay this is a polish steel. That is showing here. So, also we can change the chrome, the chromatic value here, this is just appearance, the RGB value.

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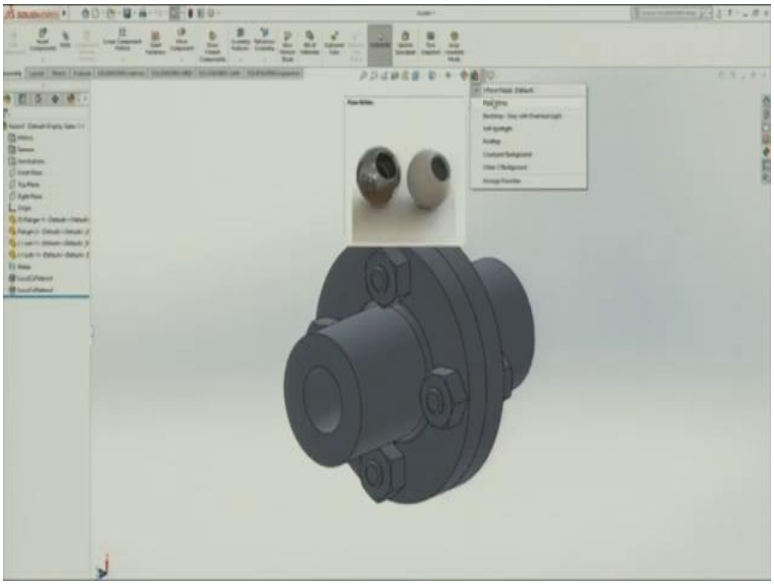

So, also we can change the background like plain white, backdrops, soft spot like roof top then country courtyard, urban background, so these options are also there. You can just keep trying these commands separately and keep learning how those things happen, if you wish to work in CAD.

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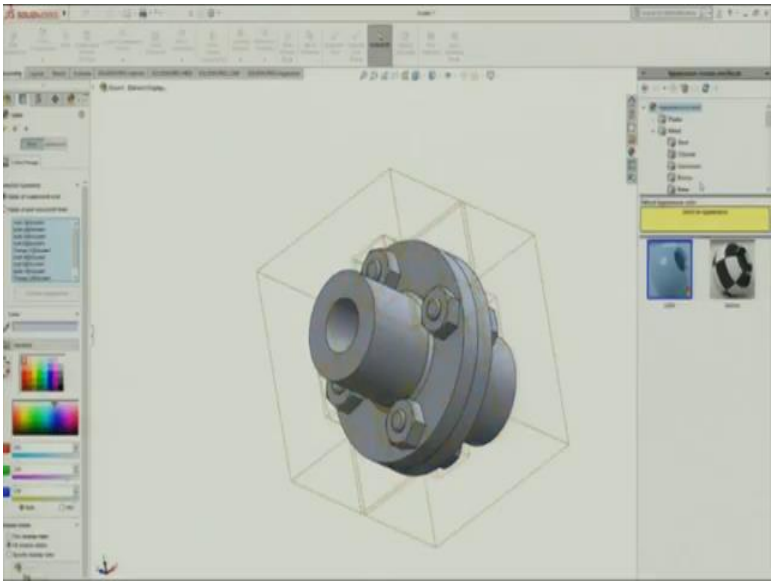

So, this whole assembly is now selected and I am selecting polished aluminum, it is showing the appearance like polished aluminum surface, magnesium, okay let me change the color, gold.

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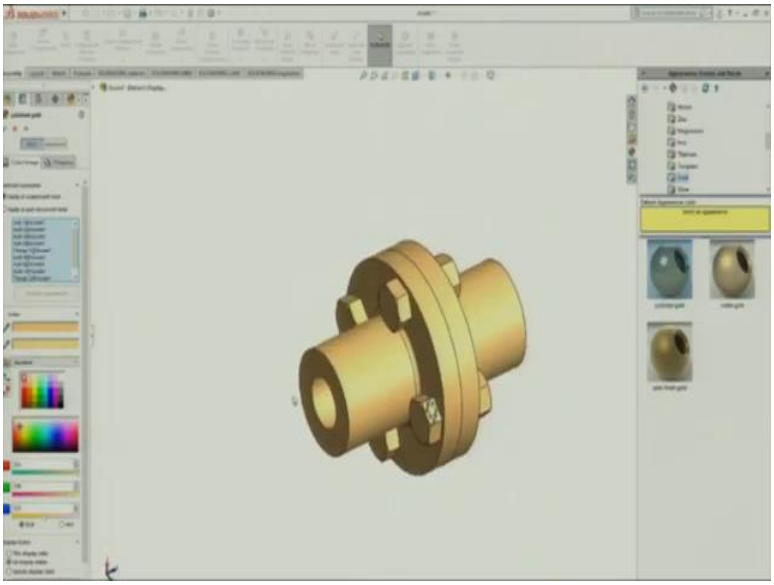

Now, this is gold. It is showing the gold color here. These are certain ways to represent or to show different kind of the components we can skip coloring them.

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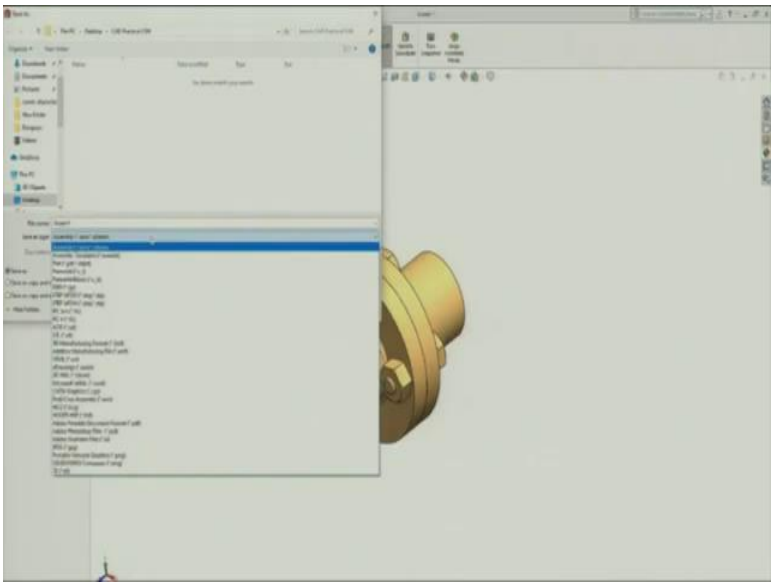

Okay let me first save it. Assembly format in the assembly format, we have the assembly format which is ASM, then we have the IGS, this is IGS, when we change one computer to other, STL is, Standard Tessellation, so this is used for 3D printing, IGES is graphical exchange format, CATIA, PRO even different softwares are there, Adobe Photoshop, JPEG images can be made.

So, Solidworks provide us a great variety of storing facilities, storing options here. I will just name this assembly as flange assembly and save it. So, this is our assembly that is ready now. For the appearance, for instance, there are certain complex components, different components in which we have I would say different assemblies are there, where the components are there, we need to see, so this can be use, this coloring pattern can be used when we have a big assembly components and different parts those are meeting we can show the components of part with different colors for instance.

If we have different kind of bolts, different sizes of bolts we can show all the bolts of 5mm or so with the green color, all the bolts of 10mm with a light green color the components which are made of different materials steel, copper, like grey color for steel, then a little brownish color for copper that can be shown.

So, these all options are already there but the major part in computer aided design is engineering softwares as I keep telling, software GIGO, garbage in garbage out, it is all intelligence or understanding of the operator or the CAD designer who is putting the data in, the proper data, if you put in will get the right output. Also we can do certain simulation on softwares, we can do stress analysis, thermal analysis, before taking the part to the actual manufacturing those things are also possible. So, we will continue our lecture on computer integral manufacturing, see you in the next lecture. Thank you.