Computer Integrated Manufacturing

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Lecture 23
Laboratory Demonstration, Computer Aided Design (part 1 of 2)

Good morning. Welcome back to the course on Computer Integrated Manufacturing. I am Dr. Amandeep Singh. Professor Ramkumar has already discussed a lot about the computers in manufacturing, geometric modeling, computer graphics. Then we have discussed computer aided design and computer aided manufacturing, computer aided part programming, CNC machines. This was all the theoretical concepts that we have discussed you have also seen the numerical problems on the CNC programming and also the CNC machines.

In this week I will give you a demonstration on the CAD and CAM region. What is CAD? Computer aided design you know it very well. In this lecture I like to discuss or give a short demonstration on the software of CAD. The software that I have picked is Solid Works. Solid Works is one of the softwares that is very prominently used in the market.

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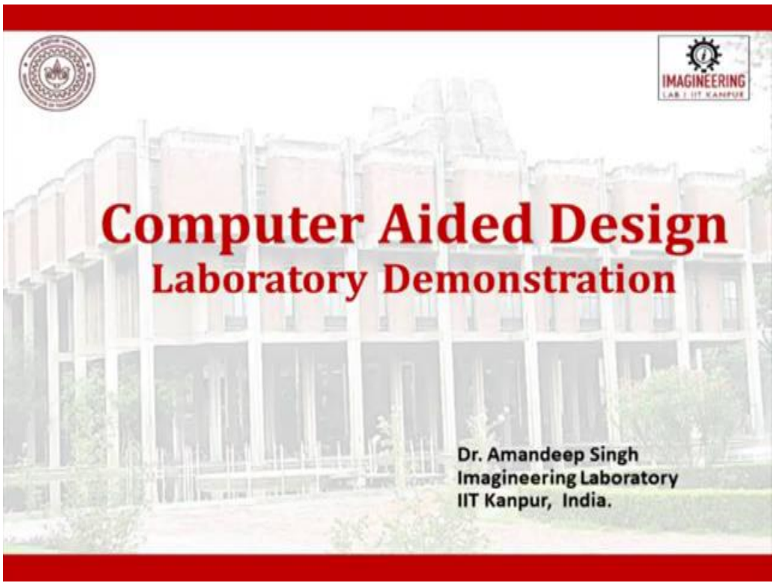

However, there are certain software's in the market such as Solid Works, Creo, CATIA, Fusion 360 then what more, AutoCAD not to forget, there are certain software's in the market. What generally is a CAD software? When I say software, software has to minimize the human effort. Software had its own evaluation platform, its own calculations, whatever we give input your software, it does some processing and gives you the output, so, it reduces the effort.

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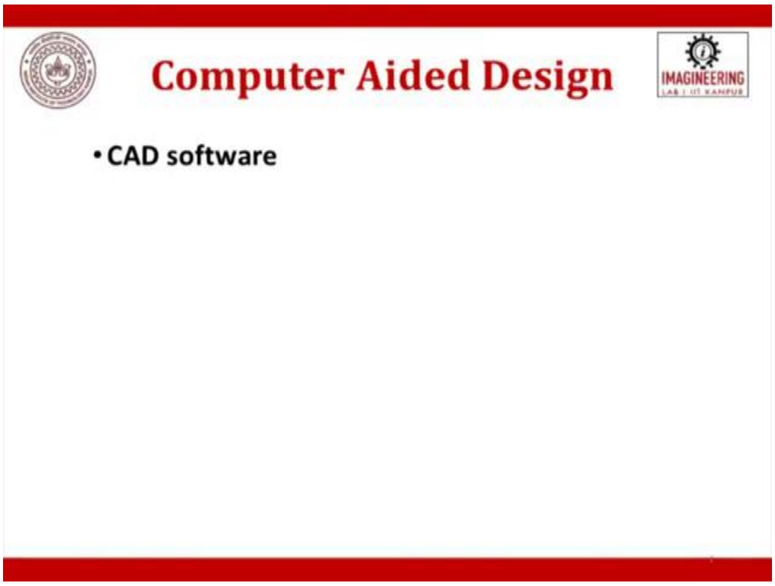

Now, what is a CAD software? CAD software is essentially a software where you can design, where can you put your design using the input devices such as keyboard and mouse, and the software that you should use when designing something to be 3d printed is entirely dependent on when you try to make.

3d printing software's and the CNC software's are different software's. 3d printing and CNC both can be done on the similar software on their specific dedicated software's for CAM, dedicated software for CAD, dedicated software for additive manufacturing, all those things are there. So, what essentially is a CAD? Computer aided design. It is the use of computers to help in the creation, modification, analysis, optimization, evaluation of a design. CAD software is used to increase the productivity as I just said.

Its use in designing in electronic systems sometimes known as electronic design automation EDA, for mechanical it is MDA Mechanical Design Automation or also CAD, CAD is some in mechanical is known as Computer Aided drafting. So, CAD software for mechanical design uses either vector based graphics to depict the objects of traditional drafting or maybe also produce raster graphics showing the overall appearance of designed objects. We have already discussed these things in the theory. Now, I like to discuss about the SOLIDWORKS software that I have picked for the demonstration in this lecture.

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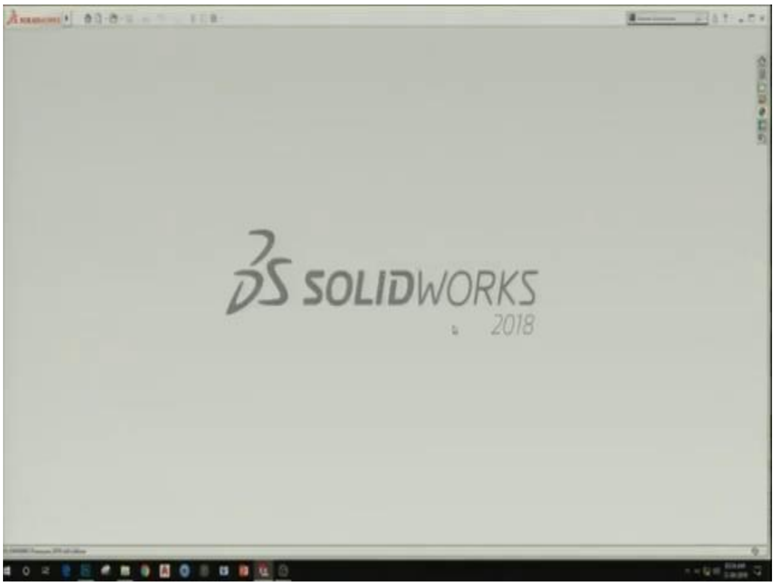

So, this is the interface for the SOLIDWORKS software when you open the system. In the SOLIDWORKS this is the interface. So, this is SOLIDWORKS 2018. So, this was started by Dassault systems, DS is for Dassault systems. So, the version is 2018. So, there are certain new commands who come in the software in the previous version those were not there. So, let us try to see how do we use this interface?

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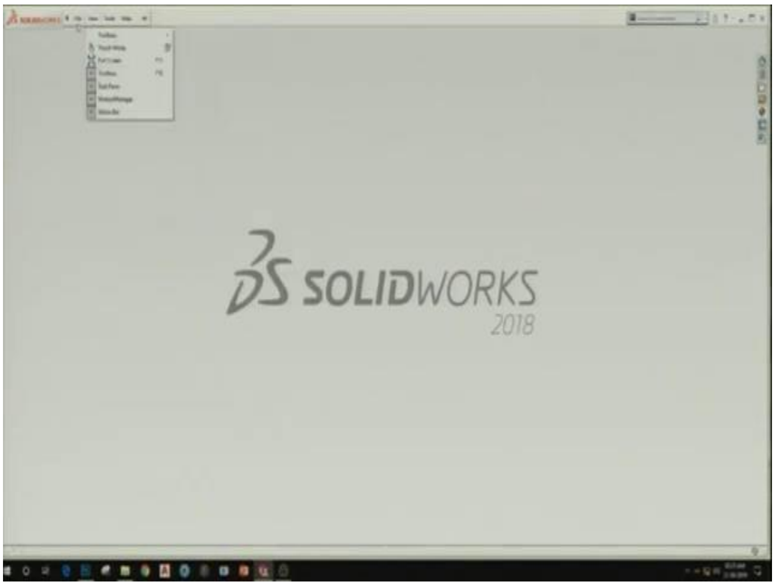

So when we click here, the file is just for saving tools. So, many tools are there.

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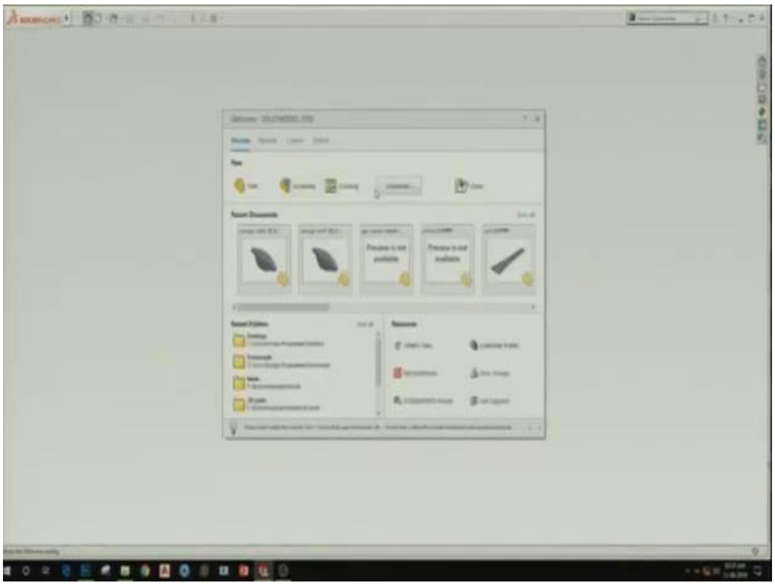

So, when I go to this home, so, it shows us the interface. So, this is part assembly and drawing. Part is basically making a single part or component like part program that we have discussed. Assembly is assembling of the parts that we have made as a component and drawing is thus producing the professional drawings for them.

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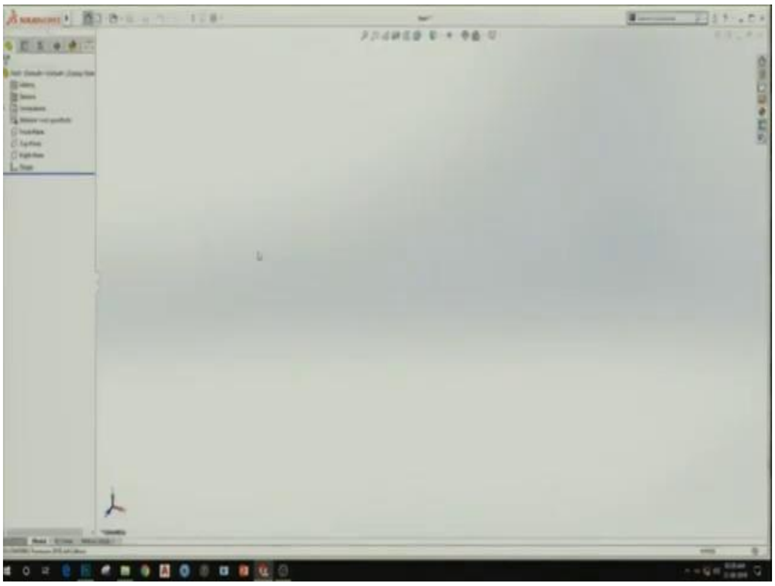

So, let me click on the part. So, when I put the part, this is the interface that is opened. So, this is a work area that we work here. And there certainly in the left side we have a navigation pane, there are different features they are said with managers here. And we have tabs here this is a controlling commands or operations that we can do here. So, these are all view icons and certain other indication icons, these are various shortcuts for the home, and so on.

So, this is a workplace or interface, we can see the bottom, bottom left there is x y and z axis in different colors, Green, Red and Blue, so this intersection of that is our region. So, this is feature manager to the site has history, sends the notations and the views. The materials all top, front and right planes, note views planes are there.

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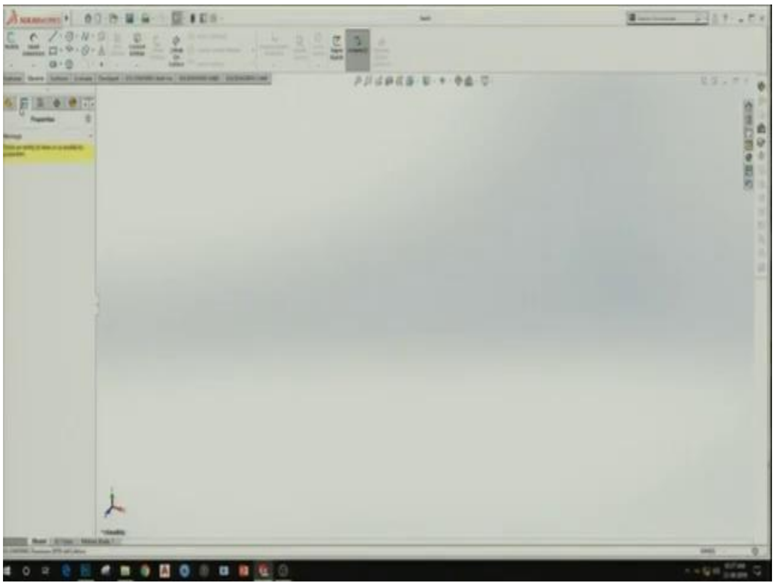

So, also we have the property manager, property manager comes into existence only when we select some of replacing, some of these sketches or some of the surfaces. So, it would be activated at that point. So, these are the only two major managers that we will consider it in this presentation. So, others we can ignore for the time being, because SOLIDWORKS is not just an hour game to learn, because SOLIDWORKS or any of the CAD software is not something that can be learned in an hour or so.

So, I will just discuss about a few important things this would be a crude presentation in which Dimensions of few important, spheres of the software will be selected to make it 2D drawing, to make a 3dD drawing, form 2D drawing and a few major commands like mirror and to give dimensions all those, a few of these things would be discussed here.

So, definitely there are a lot of videos available online by the SOLIDWORKS itself by the software development company itself. But it is all systems, you can go through that if you need to learn and the people who are already working in CAD CAM this might be a little low level lecture for them because I am going to start it from the very basic. So, these are the two major managers that will work on so. So, these are the major command as I just said.

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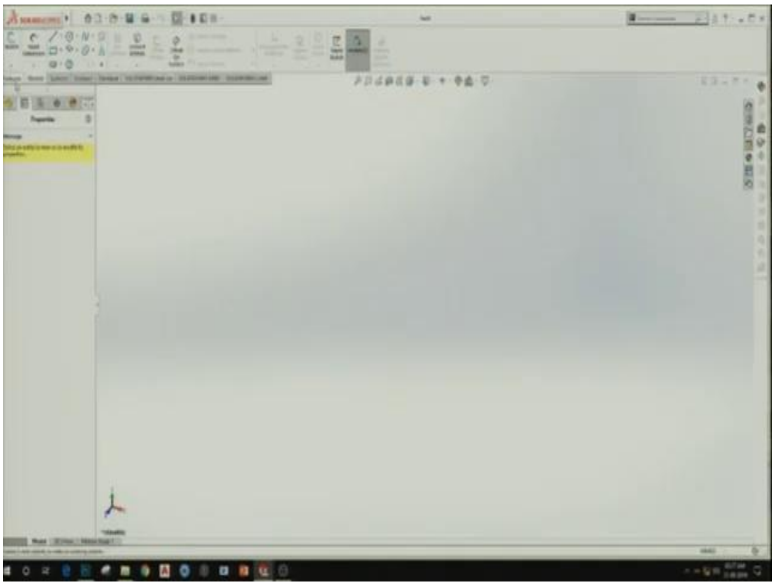

So, when I click features, there are features extruded boss. So, whatever we do in CAD model that comes here in the workplace. So, we have to fix some parameters and develop our objects here. So, this is extruded boss base, evolve boss base and so many operations are there. So, when it takes sketch it will show certain sketches here, then surfaces are there, evaluation. So, others we can just avoid as for now. So, this is a wide working area is given here to work.

So, CATIA and SOLIDWORKS are two prominent software which are being used and yes Fusion 360 is also one of the software that is available in free versions. The SOLIDWORKS is also available in the student version. Full version can also be accessed for 60 days or so, but the student version with a limited edition is also available that is completely free.

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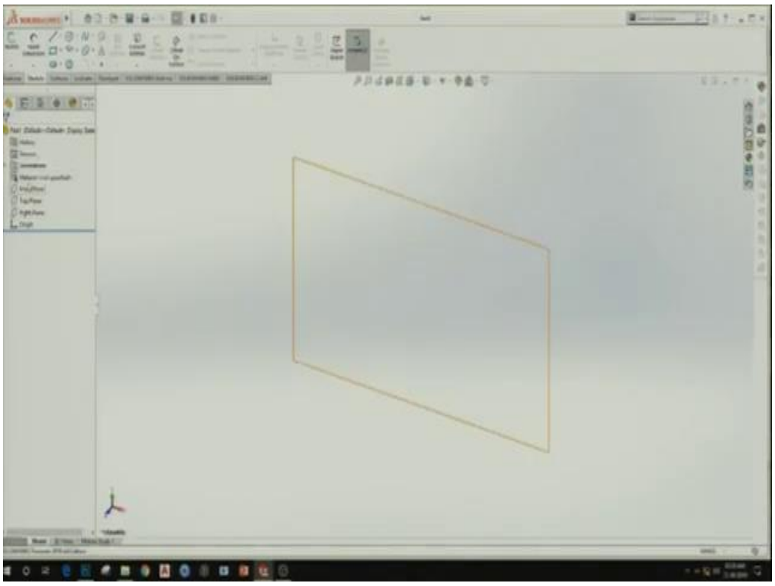

So, let me try to start from the scratch. So, we need a plane first to start, we need to decide the plane here. Then I first I have to select the plane front, top, or right plane. So, suppose if I need to draw a circle, so in which plane would I like to draw the circle? If I have to, first I have to select the plane for that.

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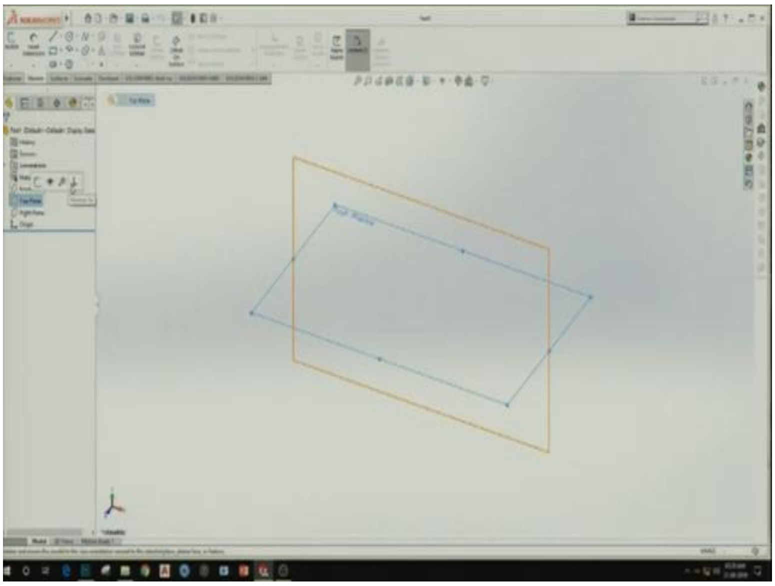

So let me select this top plane, and this is Normal To.

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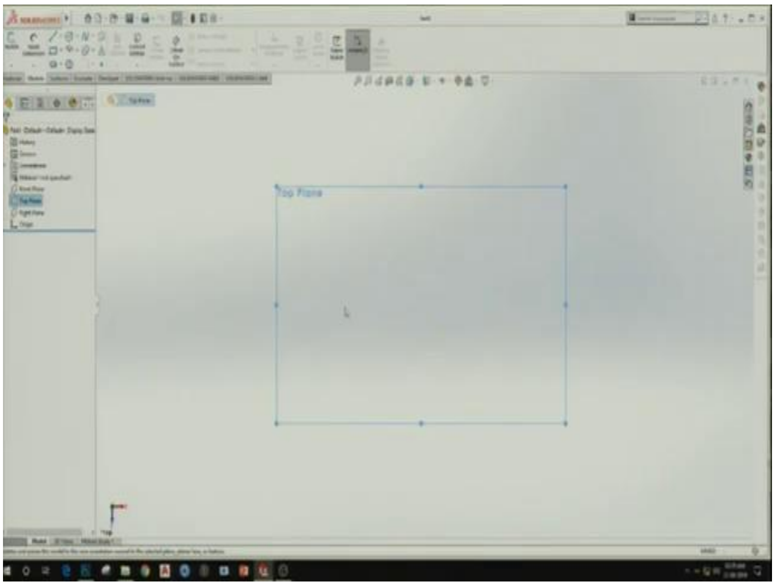

Now, if I click here Normal To, I will have a view normal to the top plane. So, also, we can rename the plane if we develop a new plane, or we would like to name the plane, something different that can be done. So, again, I will select the top plane normal to it. Then it is showing in blue color in the beginning. So, let me select circle.

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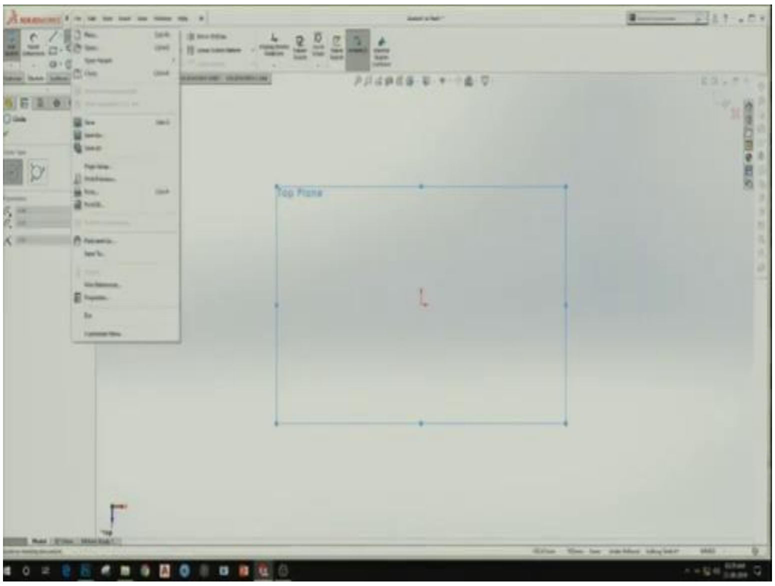

When I click circle here, you can see that the property manager is now enabled. So, many types of circle can be drawn. So, but the two major kinds that we will show here circle see these are two points center. And the second point is the radius of the circle. So, this is one of the ways and second way also in which three points can be, the three points mean the three diameter points of the circle can be taken.

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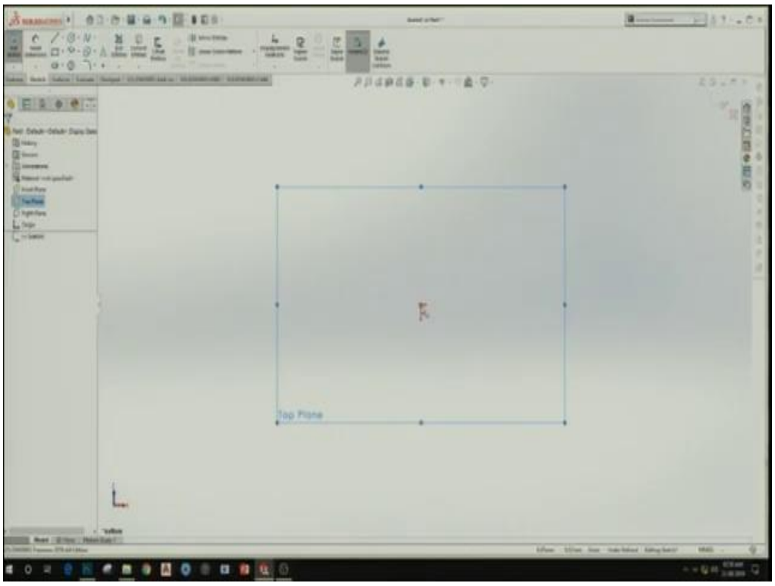

So, when we come here at the center of the plane, you will see a red dot, this is our origin. So, this is zx and this is origin.

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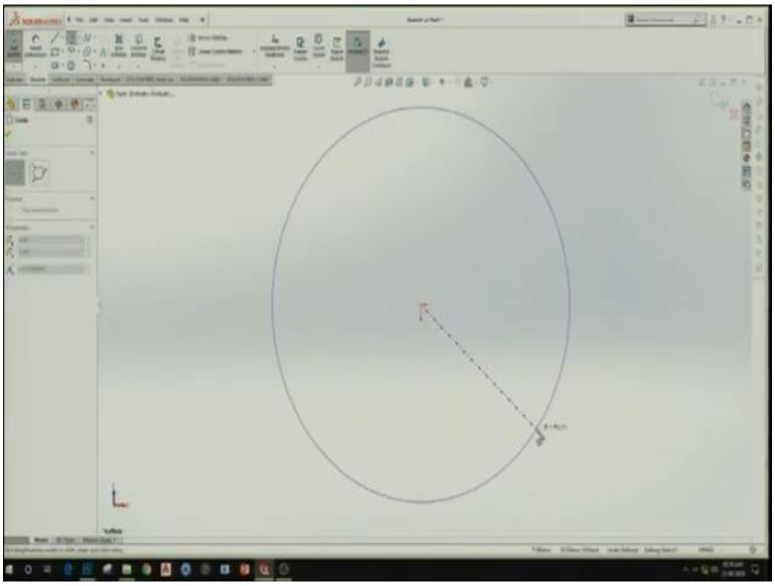

So, I will come to the region and start my circle from here. So, you can see this is free this is not locked. This is free, this r is showing here r is equal to something this is being variable. So, I have just clicked at the center and clicked and I am holding my left click of the mouse.

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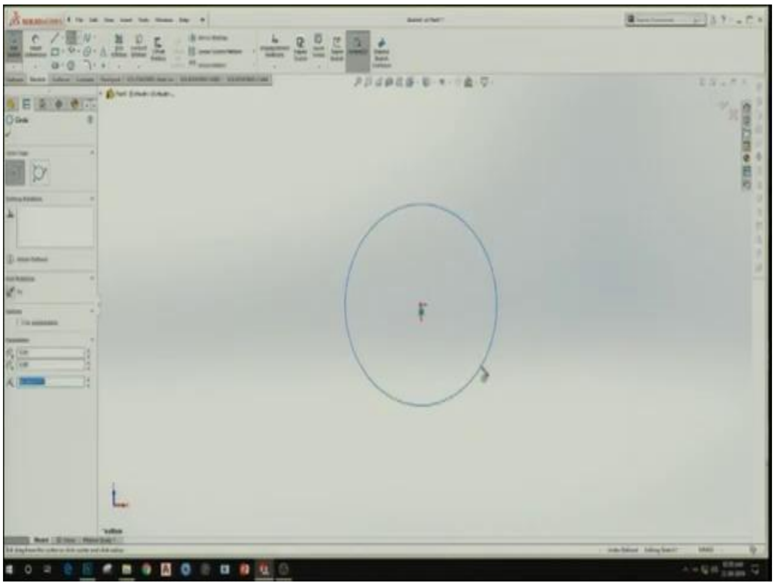

And if I leave it now here, it is still showing blue. So, in the property manager now, you can see that we can vary the radius of the circle different parameters could we put x and y coordinates, x and y coordinates this is the coordinates for the region and this is the radius of the circle.

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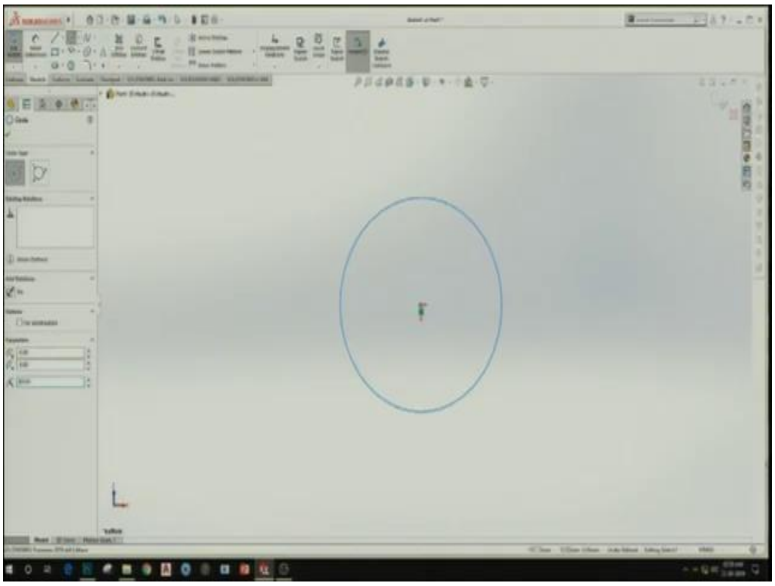

Suppose if you put 50, so, it is showing 50 here, it is not showing 50 mm radius circle. Now, what if I have to develop it 3D drawing of this circle, how to first save this? We have close dialogue here is a green tick button that is accept, accept means we save it and then exit it.

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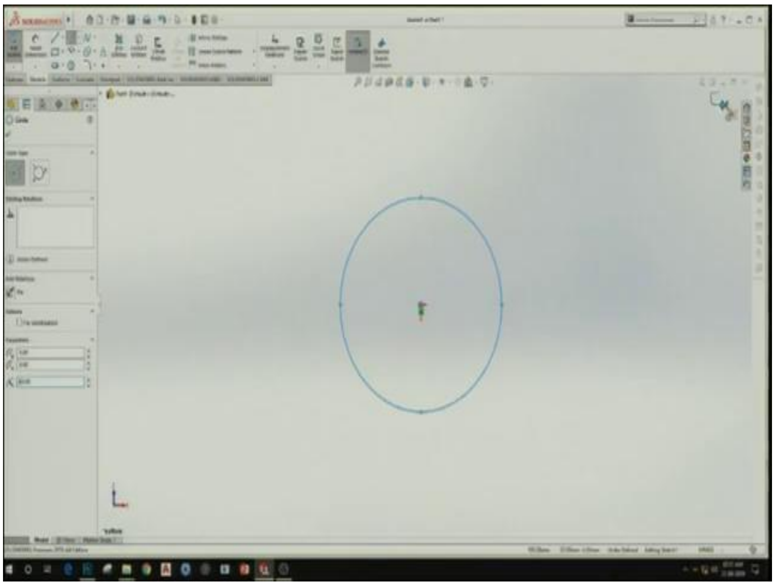

Another way is on the right side here, if I click this, it will again save and exit the present drawing.

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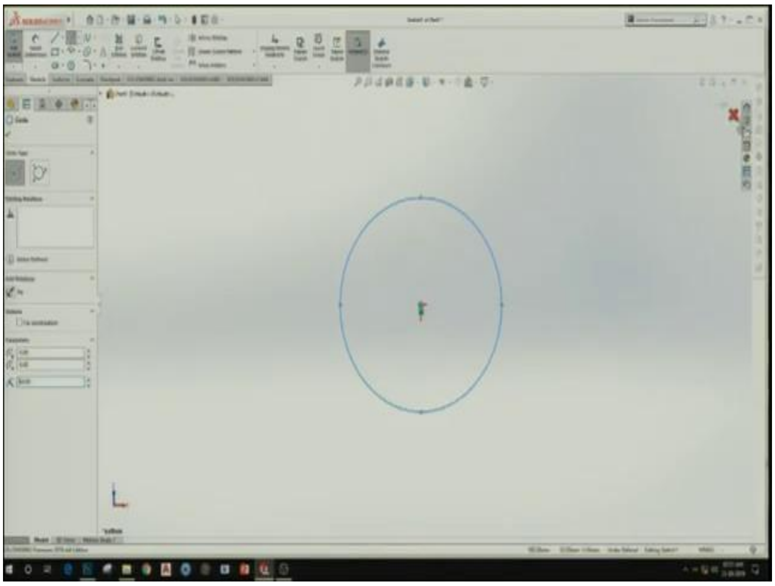

Close to that we have cross icon here which means close this would not save and clean it out. So, to save it the two ways on the left side on the right side we have to save it that is click means okay accept. Now, the circle is made it is even saved.

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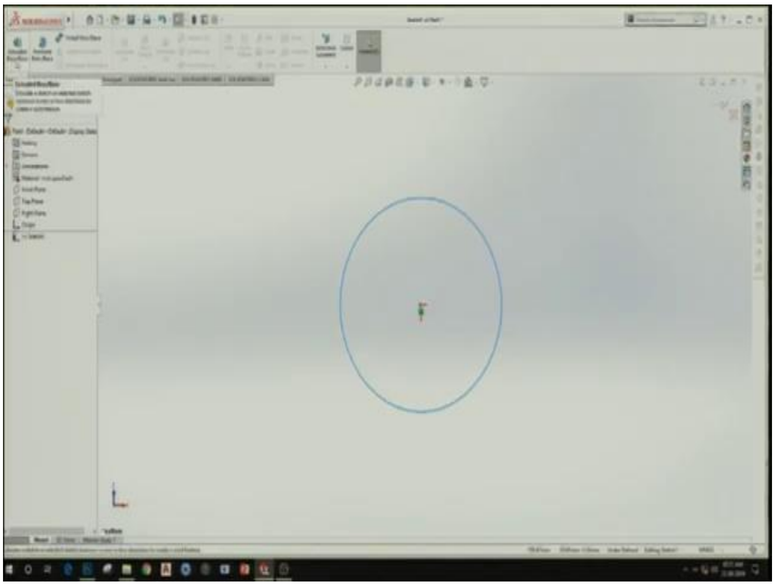

So, now come to features. So, I click here extrude boss, or base.

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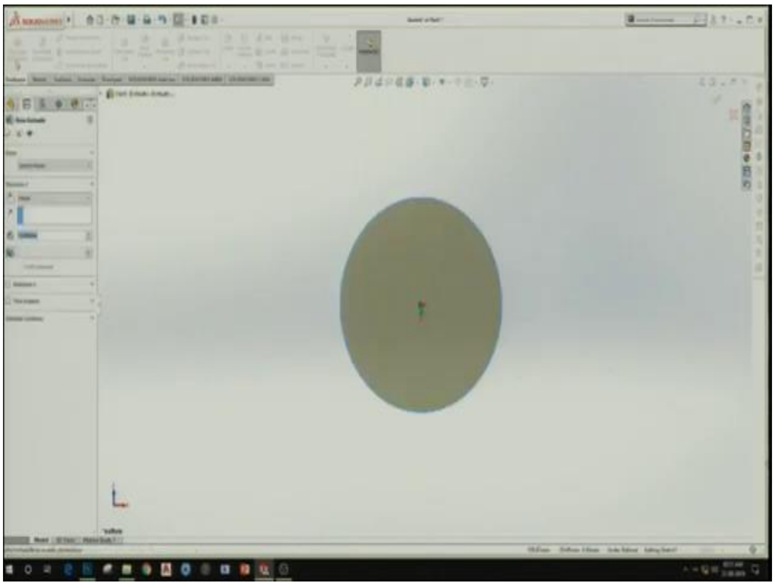

So, what does this mean I will add material to the body, but this is the top view. Extrusion command is now activated. So, it is not showing the extrusion because it is just showing the top of them.

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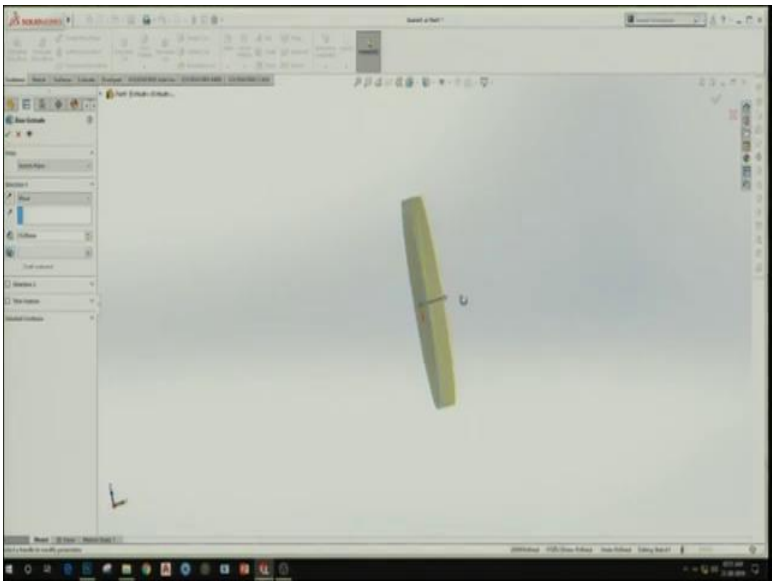

So, what I do, I click the open space and drag my mouse. Now, I can see different views, you can see. So, it is the circle is now turned to a disk. So, we can rotate object in this way. This can be done by the left click and then moving the mouse.

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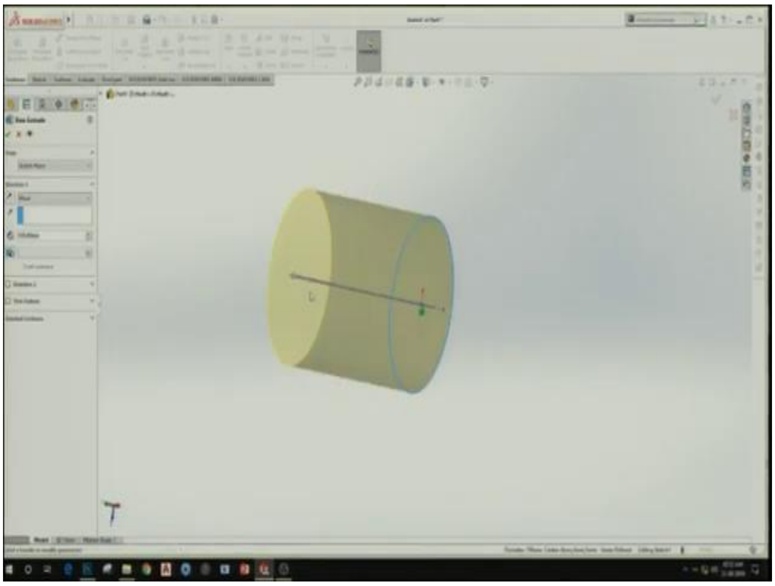

So, we can apply extrusion by either dragging this arrow, this is one ways.

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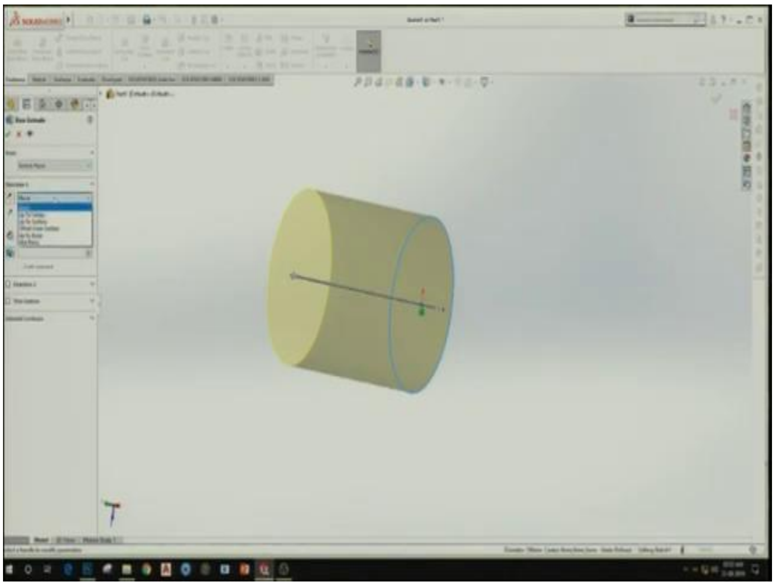

Other way is, we can just select the type of extrusion that we need to put here is it a blind or up to vertex or up to surface or offset from the surface if suppose it is some other body it will go up to the other body surface or the other body vertex. So, though this is a single body, we can just select here. So, I will select midplane. What does midplane mean? Midplane mean that it has selected the circle as the midplane and extruded in it on the both sides.

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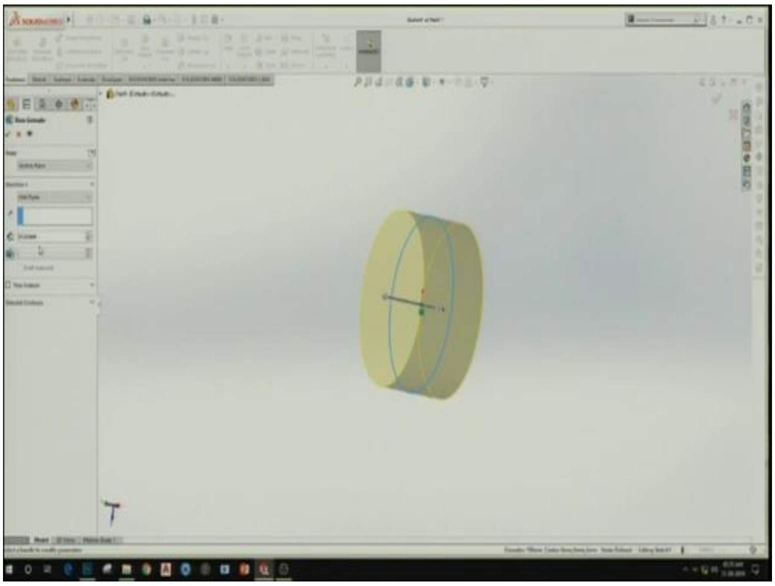

Now, it is asking the dimension. Dimension that is the extrusion length. If I call it as a cylinder, it is asking the height of the cylinder let me put 50 mm. So, this is also important to understand the units. By default those are given here are in mm. So, also we can set the units, we can change the units to inches. The different Unit systems. So, let us try to see that how do we set the units.

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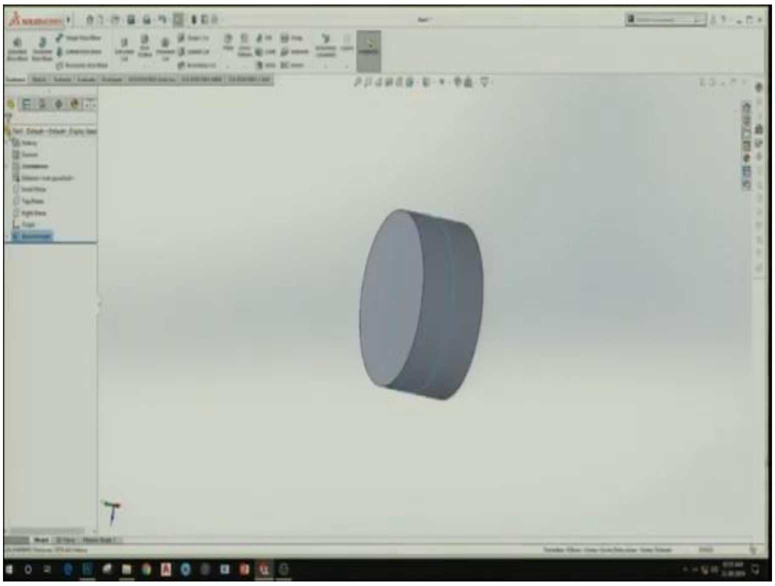

So, these are setting button at the top if I click this button now, okay first I have to save this. So, now it is saved.

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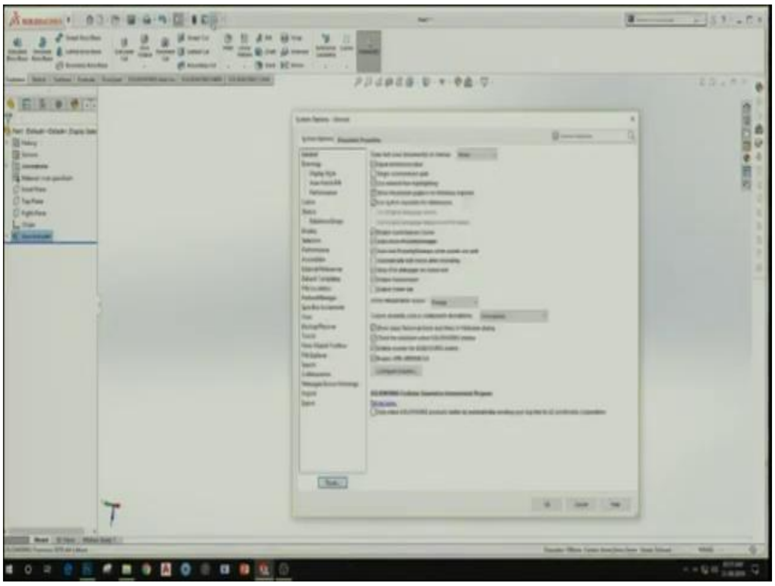

Now, if I click here.

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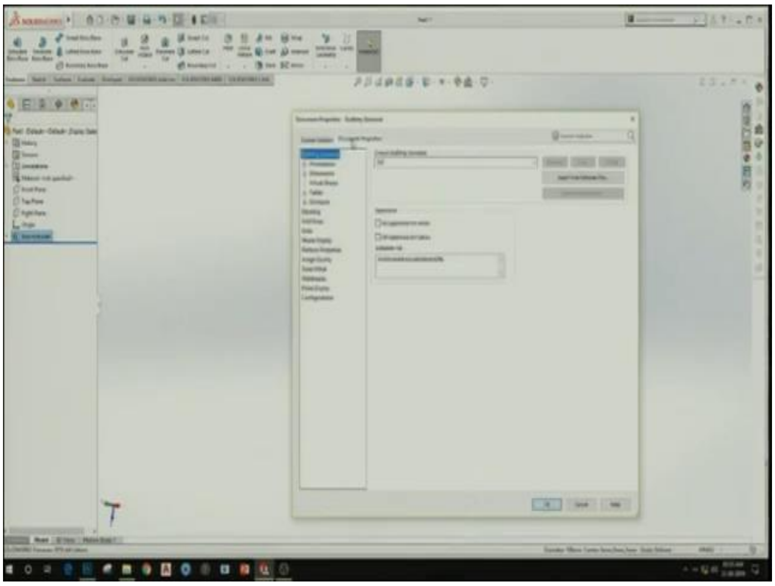

And I go to document properties.

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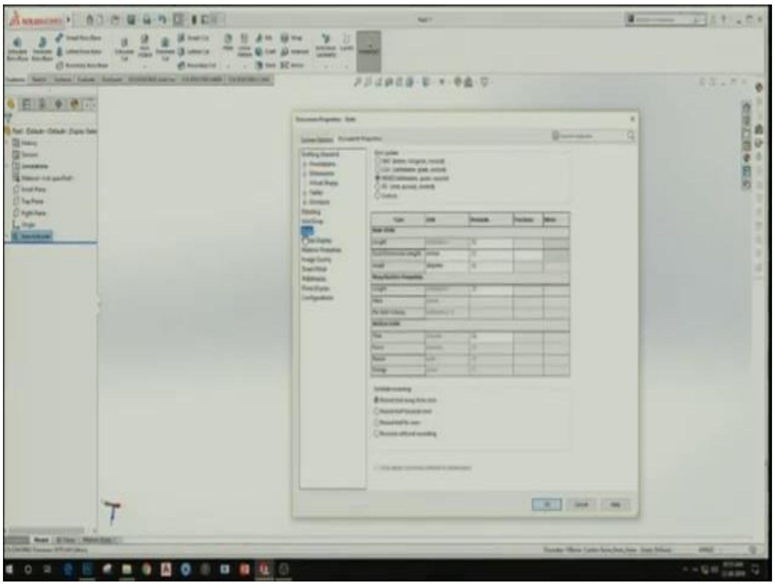

And I click units here you can see different unit systems here. It is MKS, CGS, MKS is meter kilogram seconds, CGS presents unit that is active ways. MMGS, it is the millimeter gram and second. So, we are using this. So, length is millimeters and weight in gram and time in seconds.

So, though we are not talking about the weight and time here, in the simulation, if we go for the SOLIDWORKS MBD, or if we like to simulate this, if we like to see the properties here, so, then we might have to put some weights to the material, we have to select the material or sometimes in the library, in the software library there are certain materials kept, we need to pick those materials, we can also have this test analysis, we can move the body in that case, time also comes into the play. So, those units are used there.

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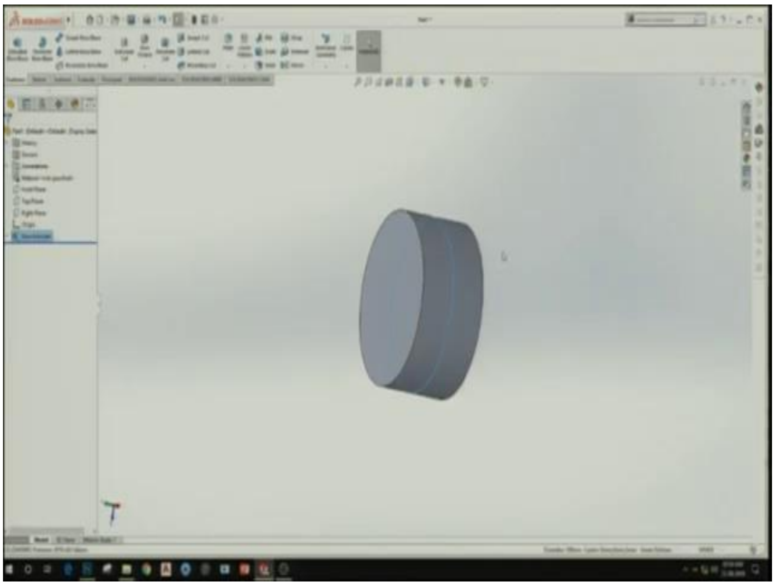

So, this is the extruded body that is made.

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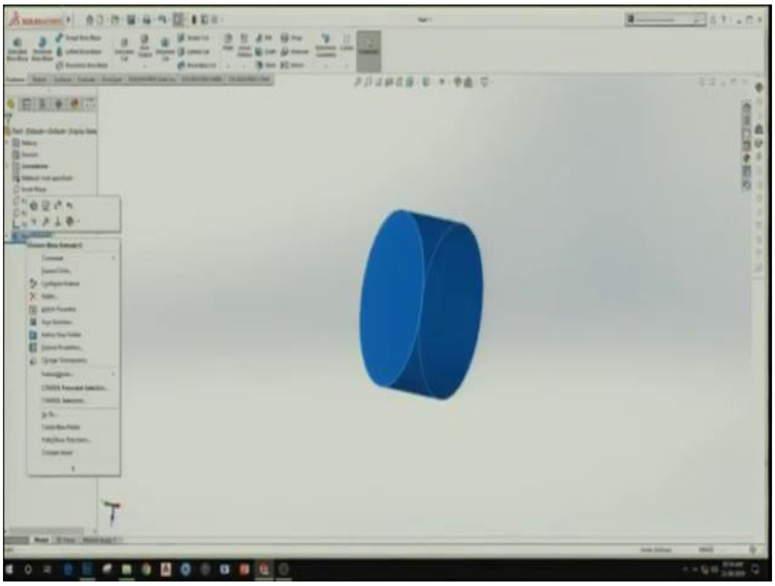

So, various operations, so, okay now I can see this operation here. Also, yes, this is an operation, operation has a sub operation is that is sketch, I can delete any of the operations while clicking delete here.

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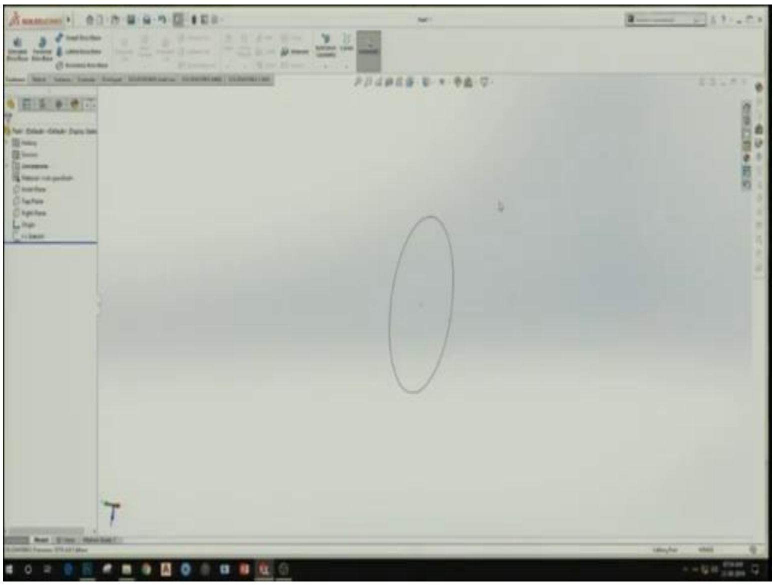

So, now extrusion is deleted but sketch remains. The sketch is circle and the extrusion is our cylinder it was what was made from the circle.

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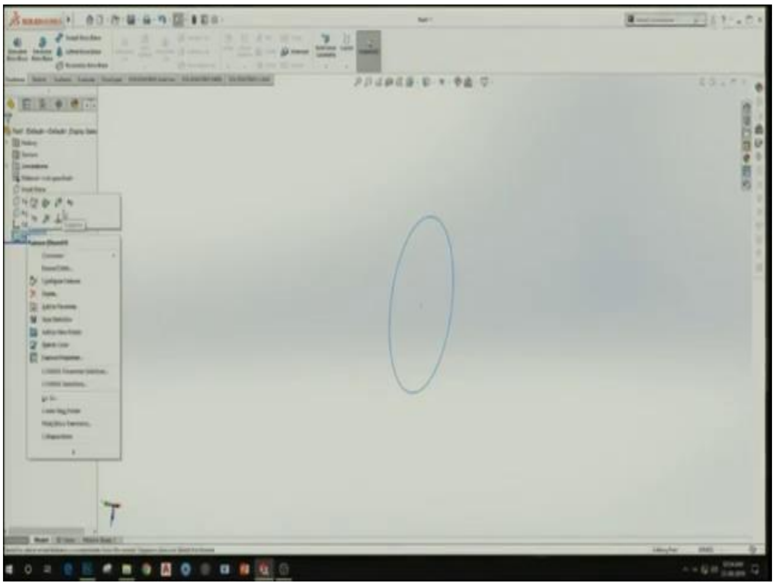

So, also we can edit sketch if I right click here or I just go there it will show the icon on edit sketch.

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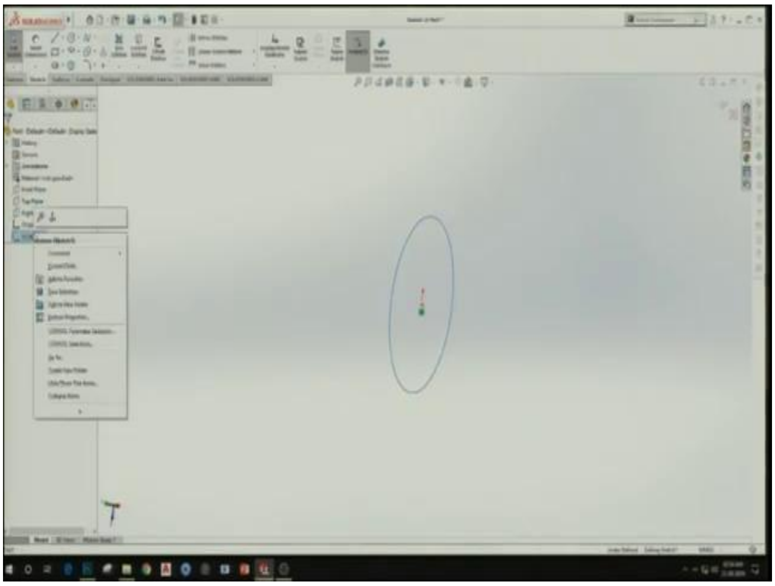

I just clicked over the edit sketch.

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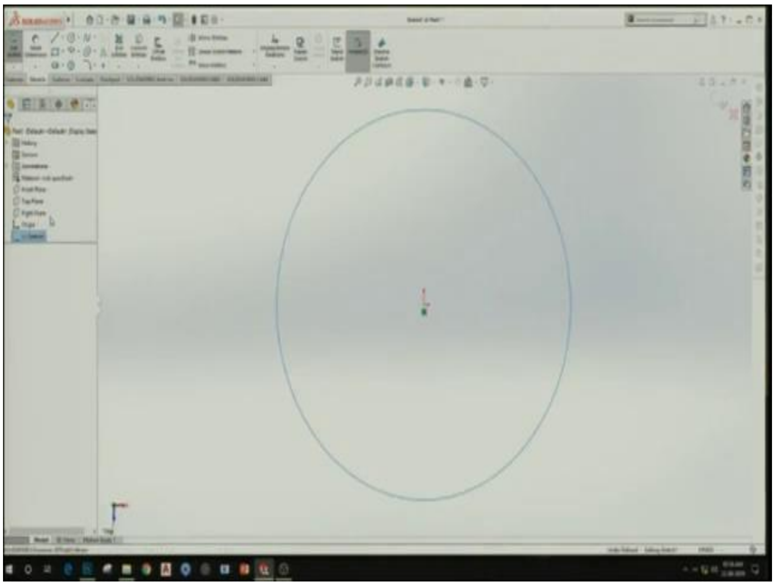

And okay and making it normal to the plane. So, I can edit this sketch, I can edit this sketch implies I can change the dimensions of the circle. So, let me change it to 40 mm now okay if I now make, like to make a different geometry, suppose if I like to make a line here.

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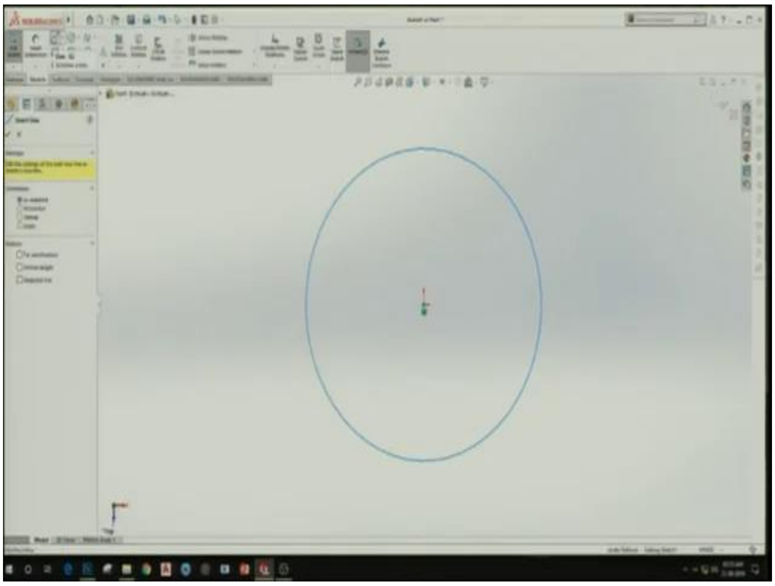

So, I will just click line. So, now it asking various options, as sketched horizontal, vertical angle. So, what kind of line do we need? So, as sketch is something that we sketch here, horizontal would be exactly horizontal based upon the axis.

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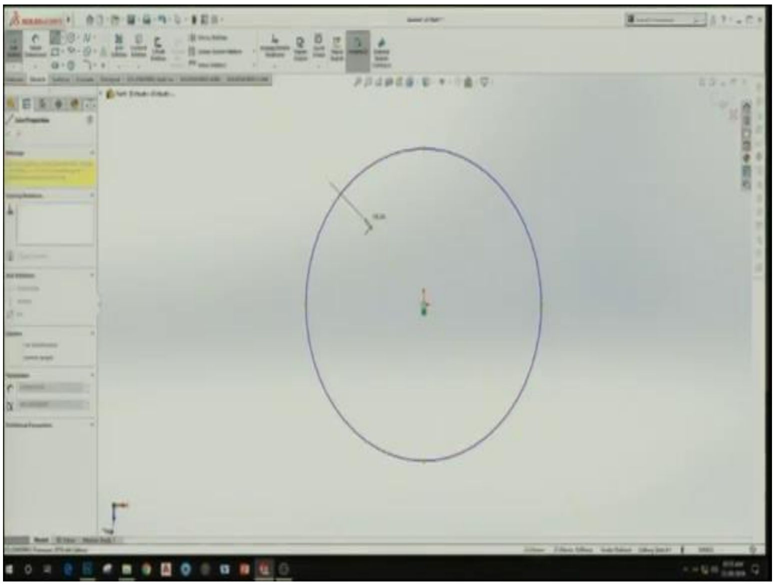

So, if I click sketched now I will just click here and leave my left mouse button here.

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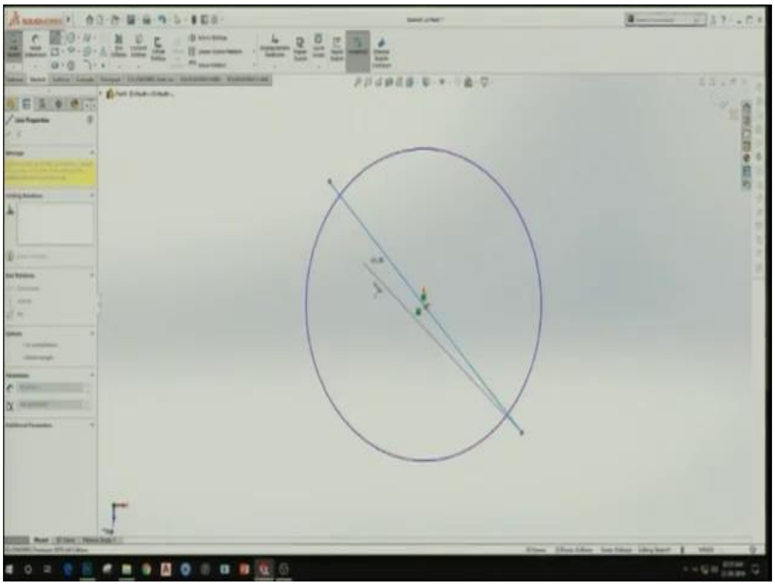

Now, a line is made, okay while working on SOLIDWORKS anytime if you press Escape, you can Escape from the command that is active now, when your mouse is active or some command is active. So, it is actually in a practice that you just keep your left hand close to the escape button because at times or multiple times, you have to keep pressing the escape button to exit the active command. So, this is a line. So, you can see the line has a little extended out from the circle. So, suppose if I like to make a semi-circle from this, how do I do?

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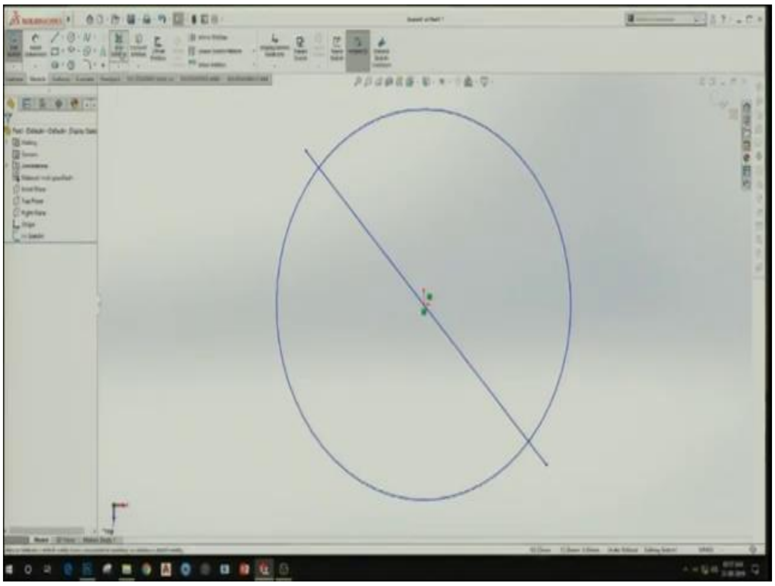

So, we can just see this trim entities, trim command here that is there in the sketches. So, there are certain options power trim, corner trim, trim away inside, trim away outside, trim to closest certain ways are here, you can see these. So, right now let me use this power trim. What does power trim mean?

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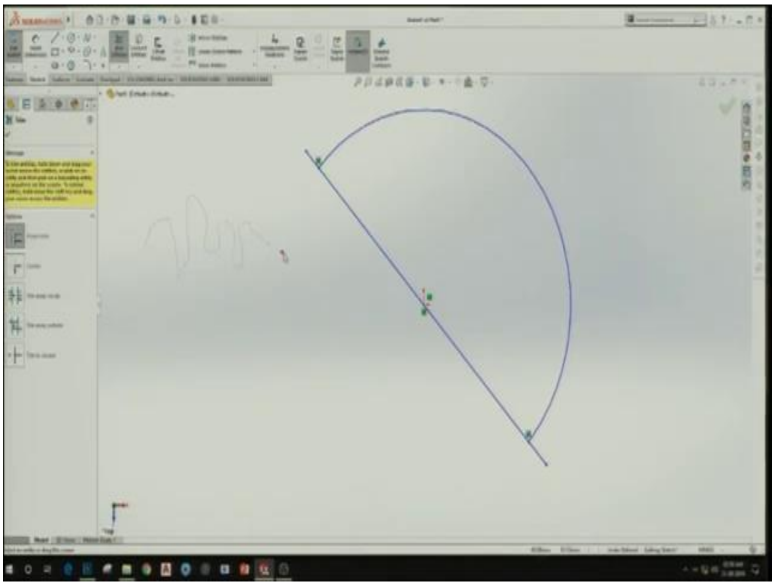

So, power trim means if I click here, I am just going free hand whenever, at where at the point if I touch anything, it just goes off. So, again this is trimmed, again this part is trimmed. So, to trim the object it will trim the sketch between any two intersections. So, by dragging the mouse we can just trim them so, accept.

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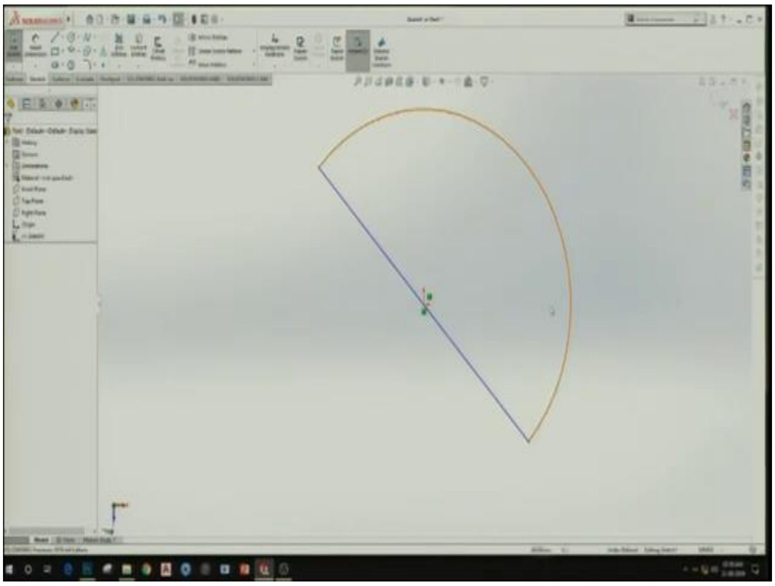

Now, we have made a semicircle, okay one of the 3D object just showed you was extrusion. Now suppose if I like to make sphere from this semicircle that is making a ball out of this how do I do?

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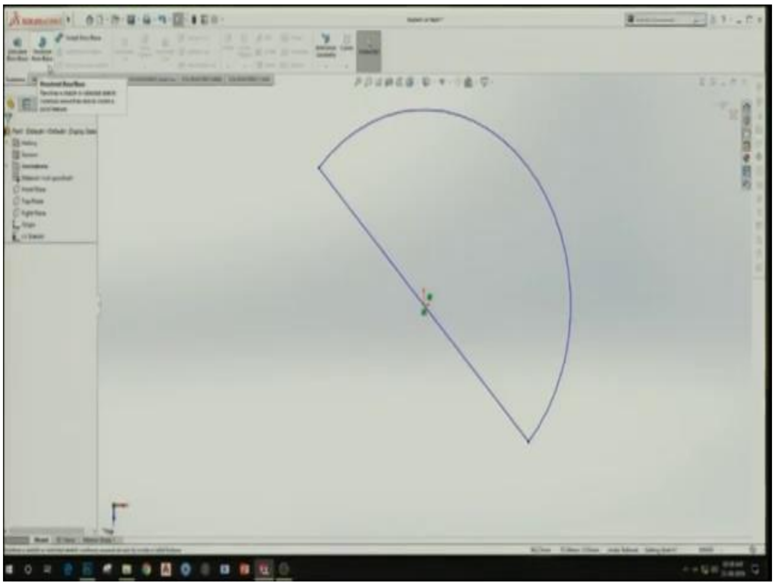

I come here this is the revolved boss or base. So, where do I need to revolve it around?

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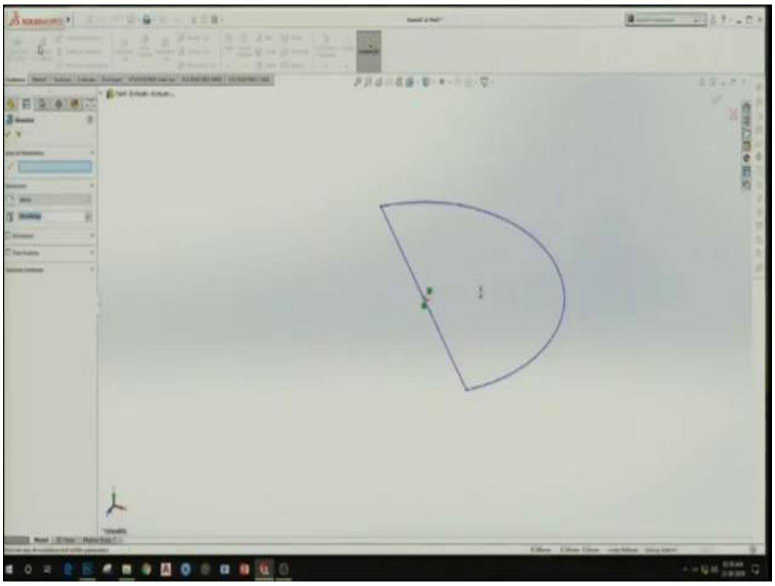

I will go to features and click revolved boss or base, it will last from where would you like to revolve ask axis of revolution.

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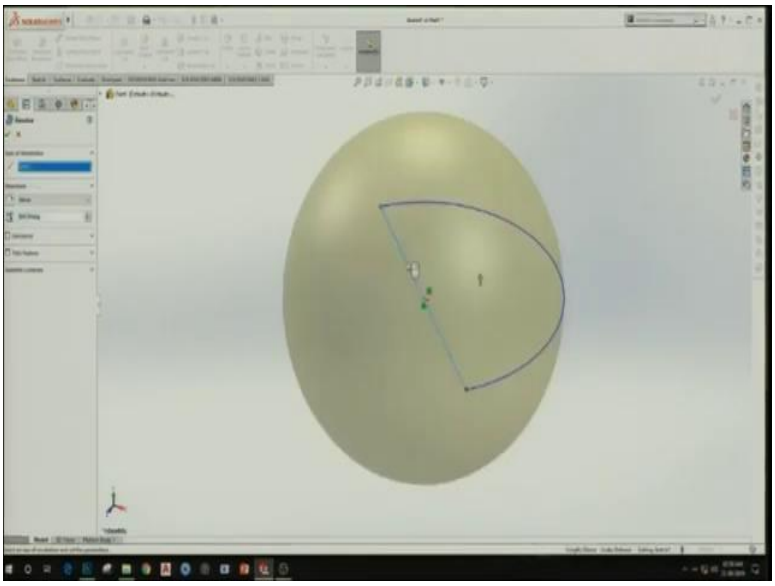

I will put this as if I just click this as exit of revolution. It has revolved it completely it, has turned into a complete ball. Why it has done so? Because there were certain options like evolve, blind evolve, up to vertex revolve, up to some surface offset from surface and so on. So, since the angle is 360 degree here, 360 degrees means it has completely evolved to 360 degree and turned into a sphere. Now, we can also revolve to maybe midplane or up to some vortex or so on.

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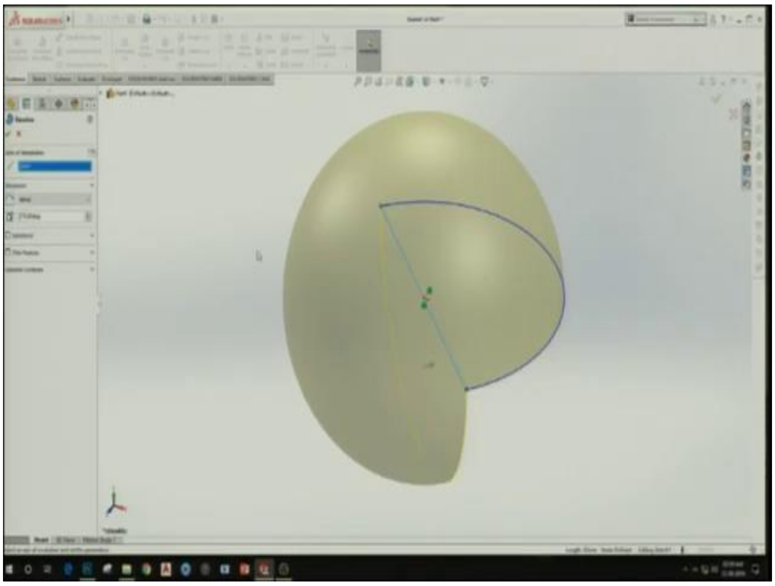

So, let me try to change this angle suppose if I like to just revolve up 270 degrees that is three quarters. Now, it is now 270 degree revolved.

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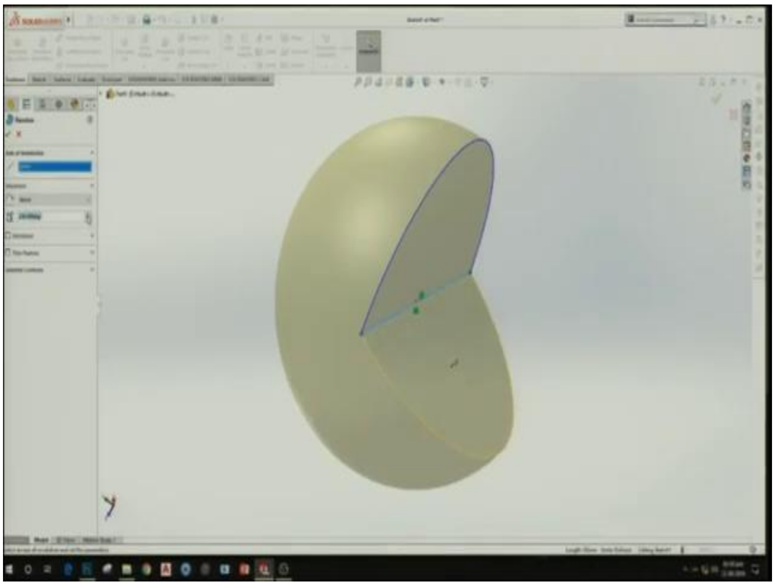

So, we can also decrease and increase while just clicking here on the arrows up and down. In case if it is the revolved angle.

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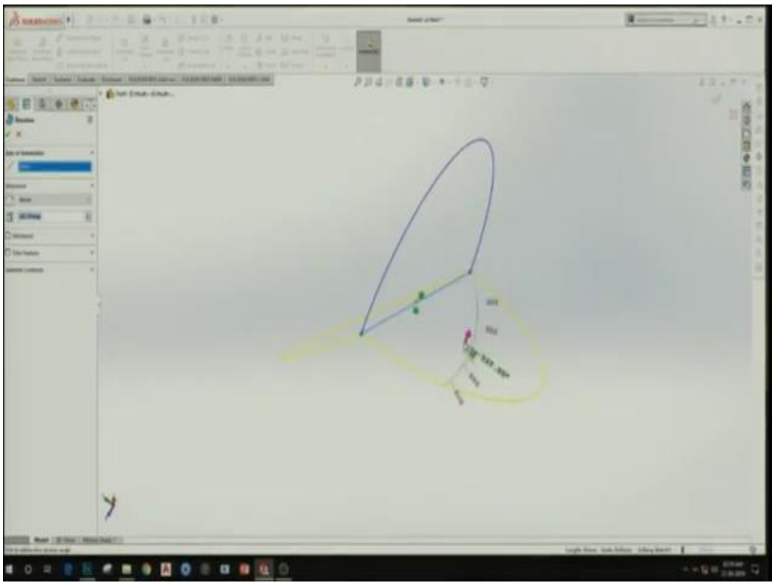

Also yes, if I click this arrow you can see if I click here it is showing 297 degrees it is increasing and decreasing whenever I move my mouse along this.

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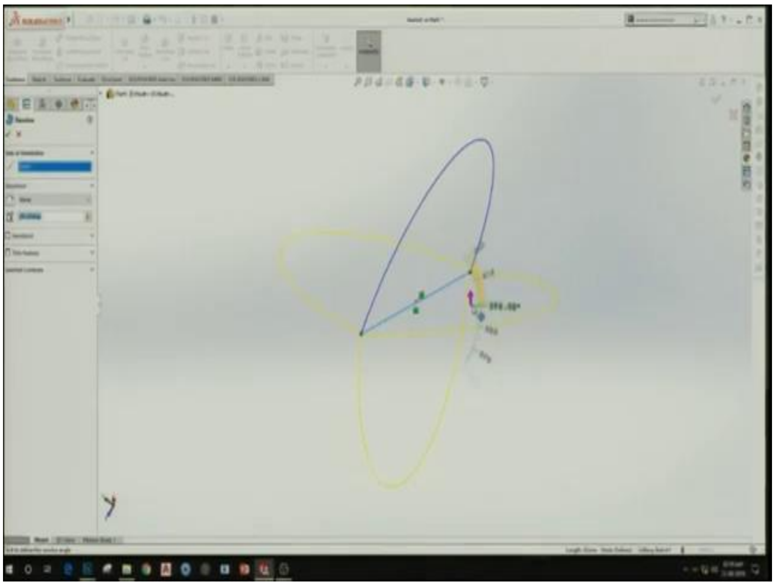

See it is showing the graduated scale as well at what angle would you like to revolve? So, this is a very common function that is used in SOLIDWORKS or any of the CAD software's.

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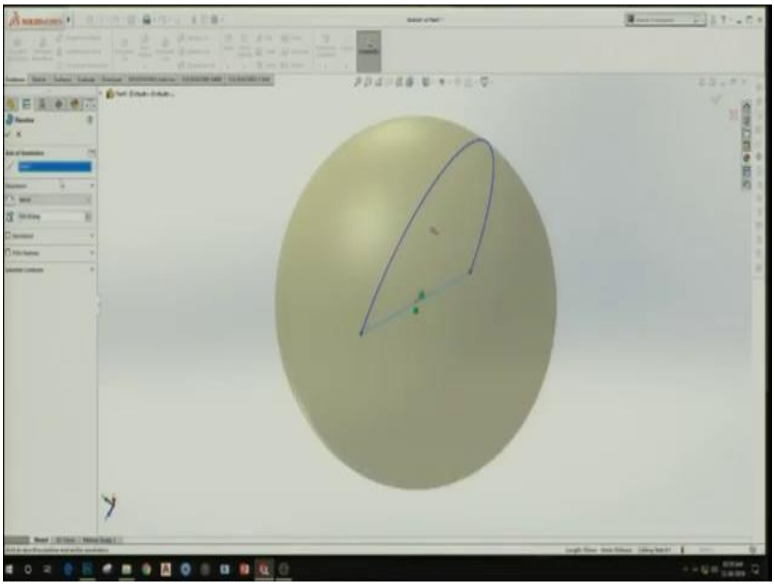

So, let me turn it 360 degree back. So, let me accept this okay.

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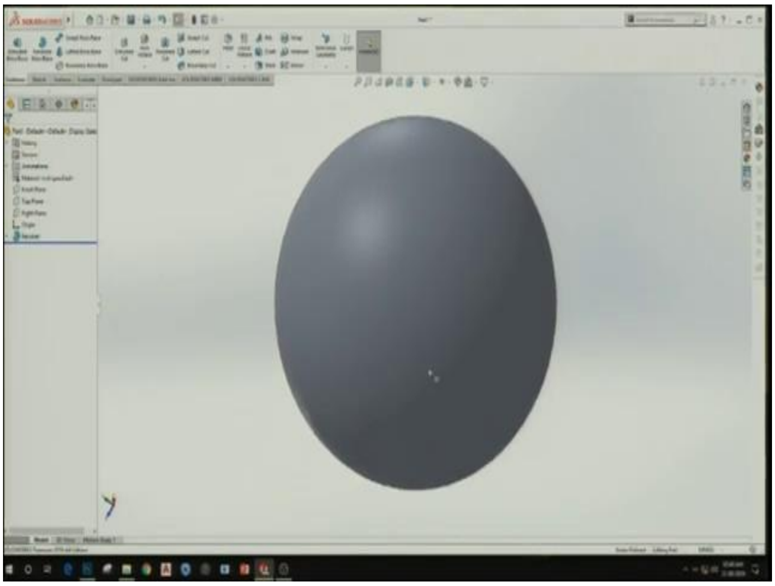

So, this is a hardball, hardball it is I had added a volume to it, hard ball means I have added a volume to it. This is a complete solid ball which is being shown here.

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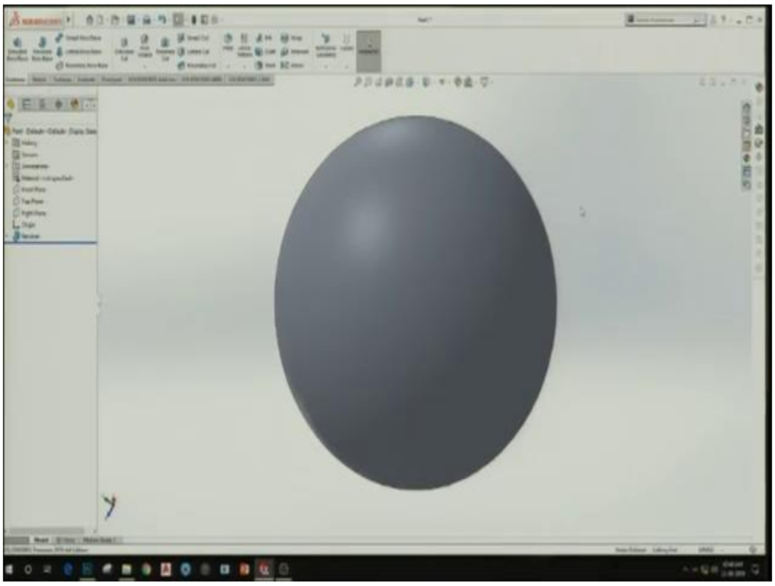

So, we applied a volume to it you can see, see I am rotating it, putting the view.

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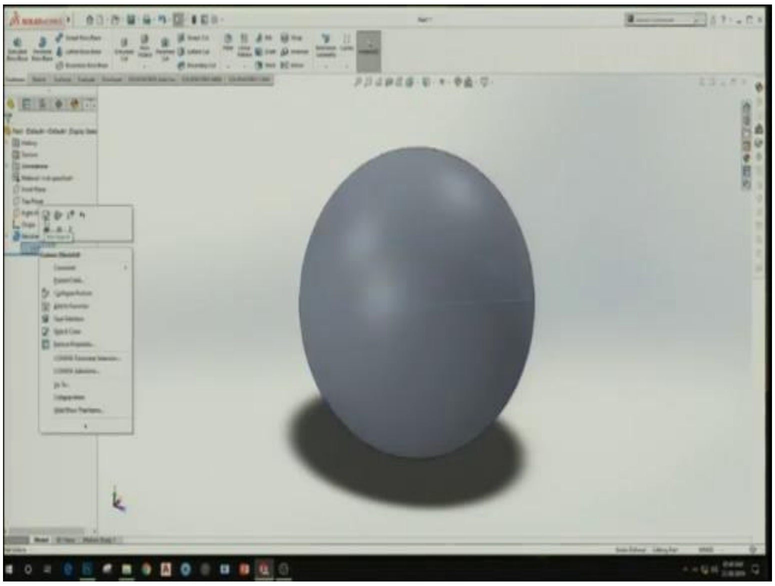

Now, suppose we want to change the radius of the ball, we can just go to the sketch only. And I will go to edit sketch, it is customize sketch, edit sketch I will just click on the semicircle, I will see this value this radius is 40. I will turn or change this radio let me say 55 mm.

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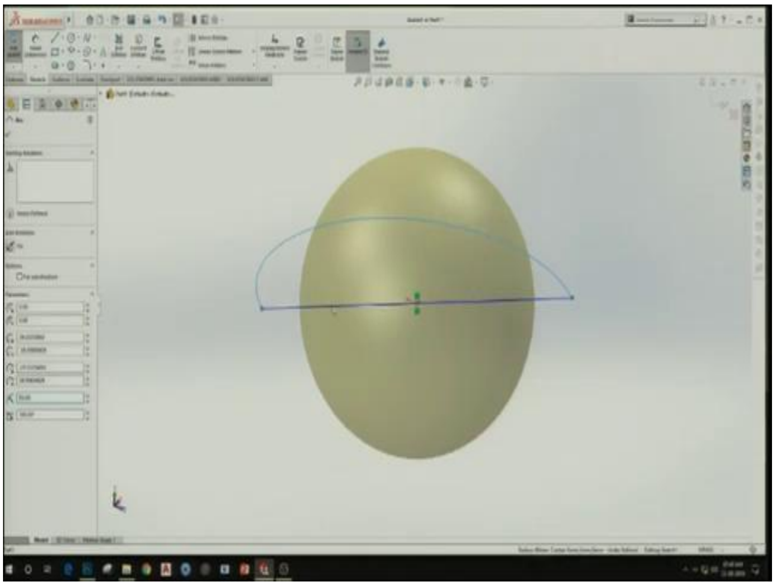

Now, the circle the sketch is changed. Now, this is changed. See, nothing has happened yet because I have just clicked the semi-circle, the radius of semi-circle is changed.

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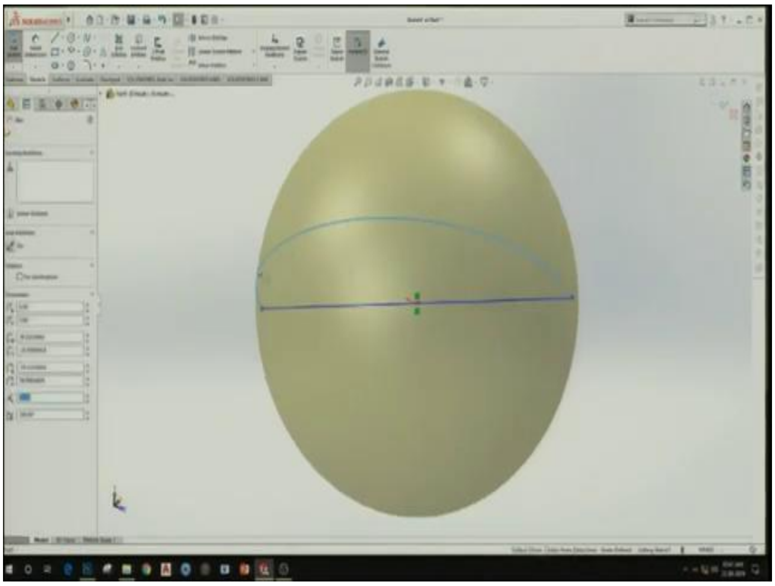

Now, if I click it again it will change the radius of the complete body here. So, it is basically a CAD software in which editing can be done very easily, so we can change the dimensions.

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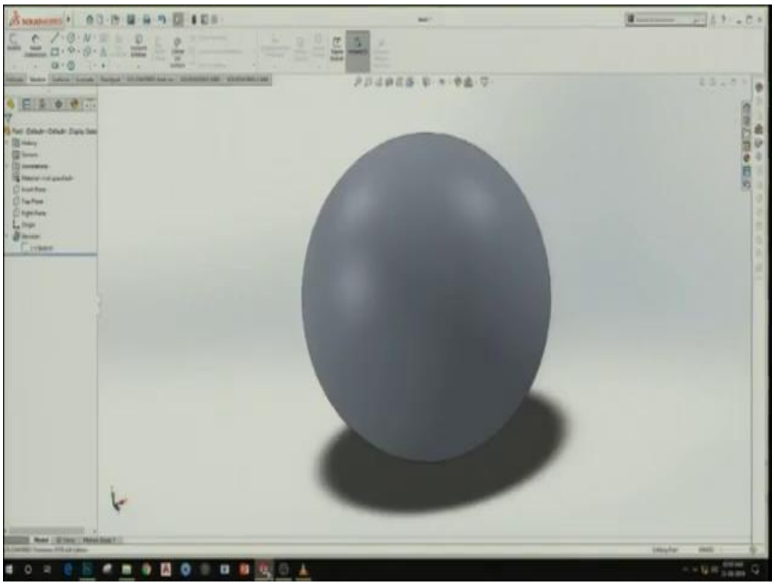

But it is important to note here that change in dimension of the one object or one part brings change to the dimensions of the all the assemblies where this part is used, if it is used directly which has to be done very carefully. One has to be mindful that what he or she is changing.

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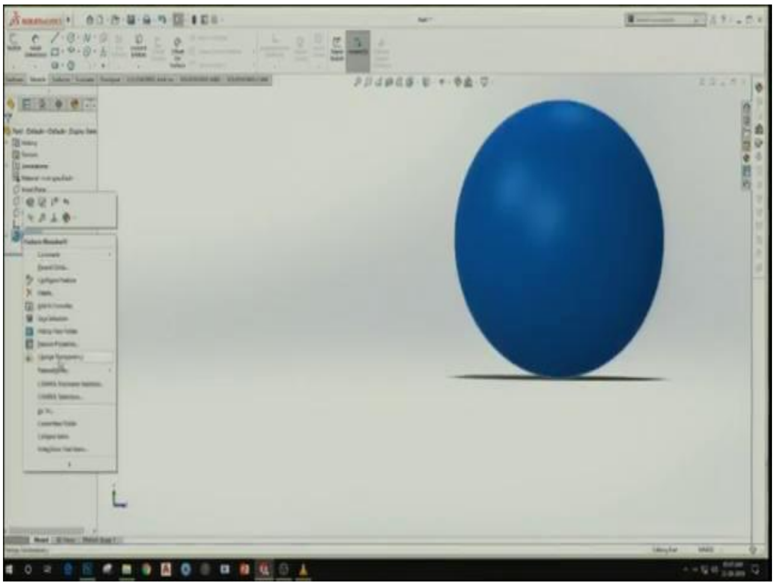

So, let us try to see some other operations. So, let me try to delete this again okay. Okay, let me try to use the same sketch and okay, let me try to extrude this now.

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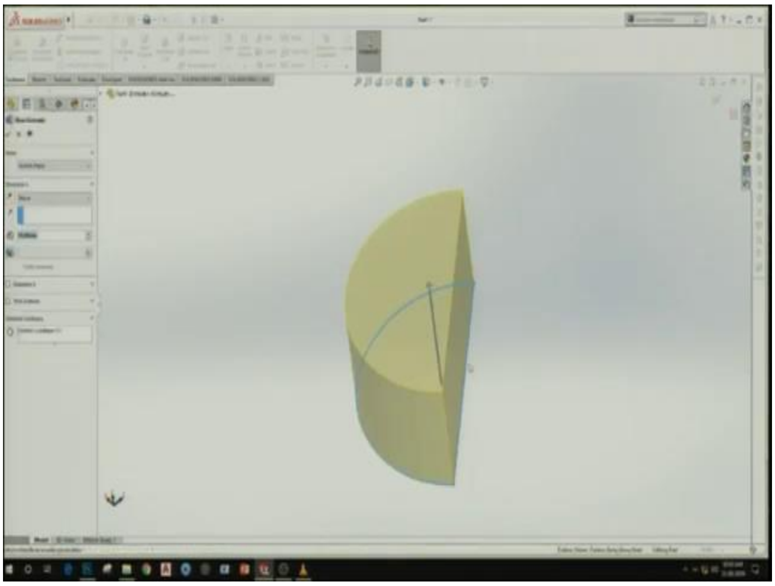

So, features, features and extrude this, okay, now it has made this extrusion.

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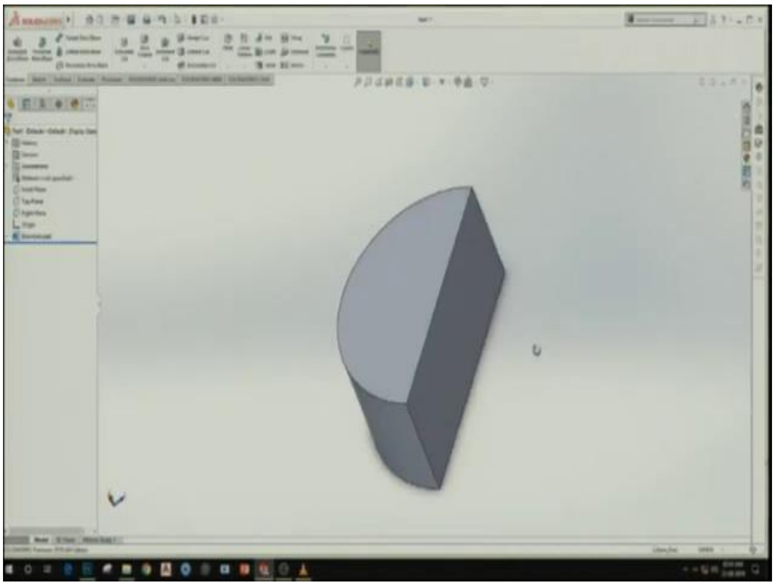

So, accepted any dimension whatever it is selected. So, suppose if I like to create two circles over a specific face of this body. So, let me select this face here. So, whenever we need to create something we need a plane.

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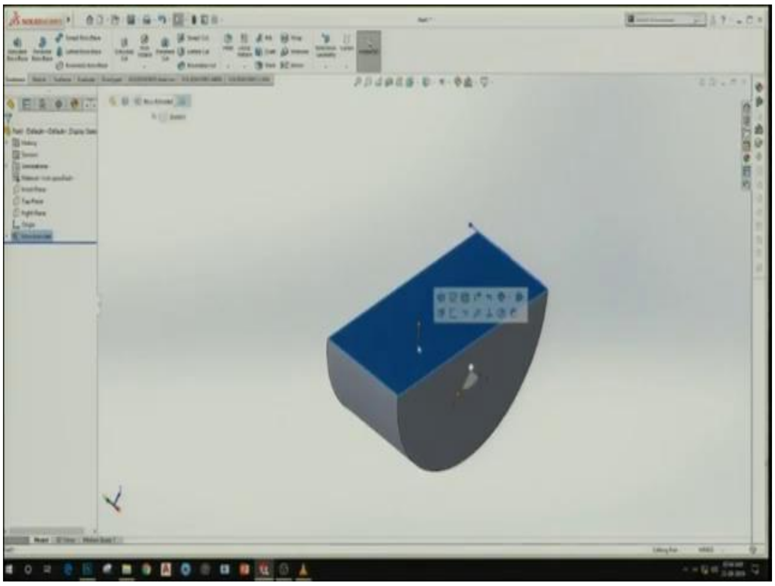

So, let me select this plane, so I will click anywhere, at this point if I select now this will be, this will be select it as my point.

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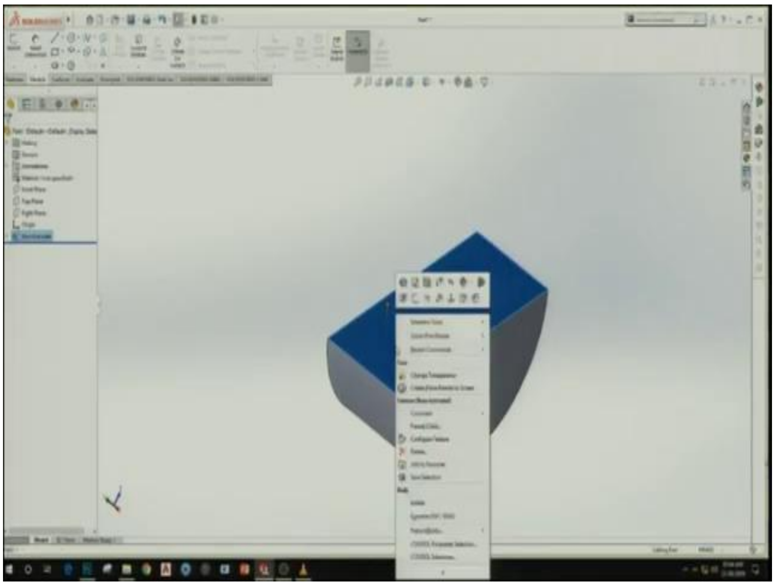

And I go to sketch and okay yes, yeah, yeah the face of plane that is the plane that was selected was not right.

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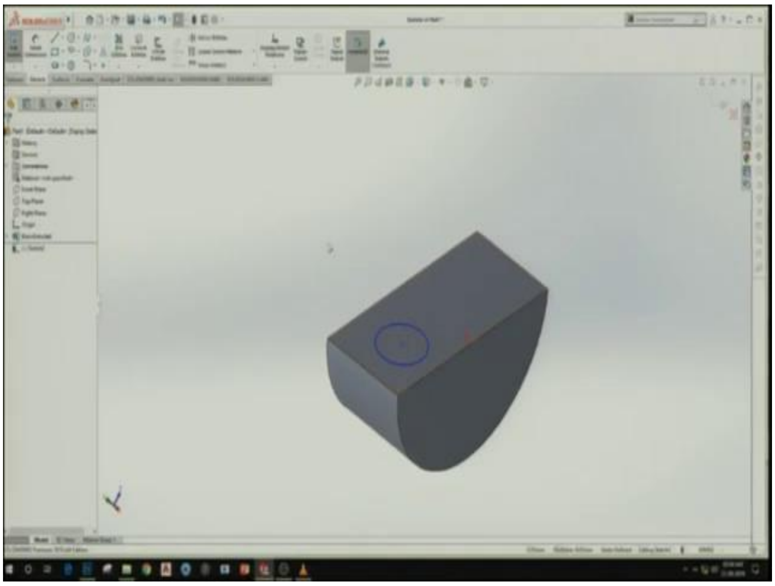

Now, I need to select the circle, like I select circle and click it here, it has become center and when I leave it here it will just make this circle, control Z is used for undo, yes, this is also an important like escape, escape will just make you escape the present active command and control Z is there in which you undo the command or the operation you just did.

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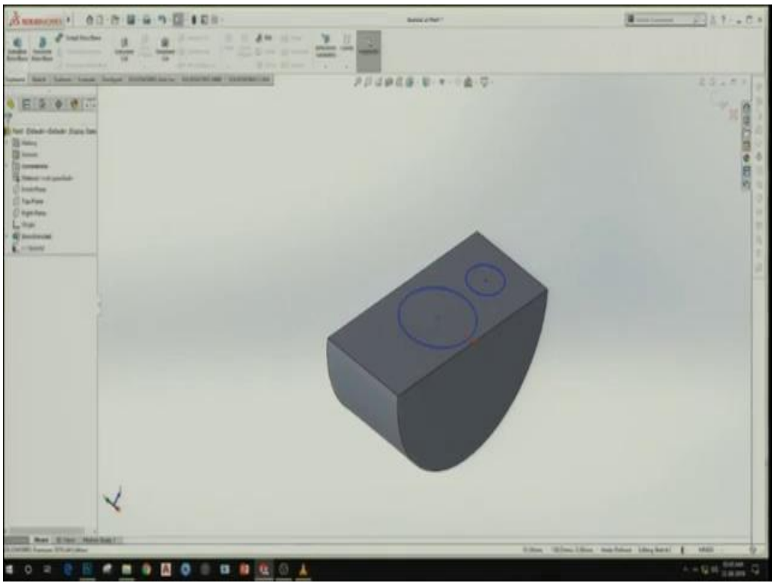

So, I will just make a circle here. So, if I need another circle, so I will put another circle here. So, at different locations, we have created a circle okay except okay now let me first change the radius, let me change the radius of this circle to 10 and this circle let me change this value to go from here I will change this value to 20.

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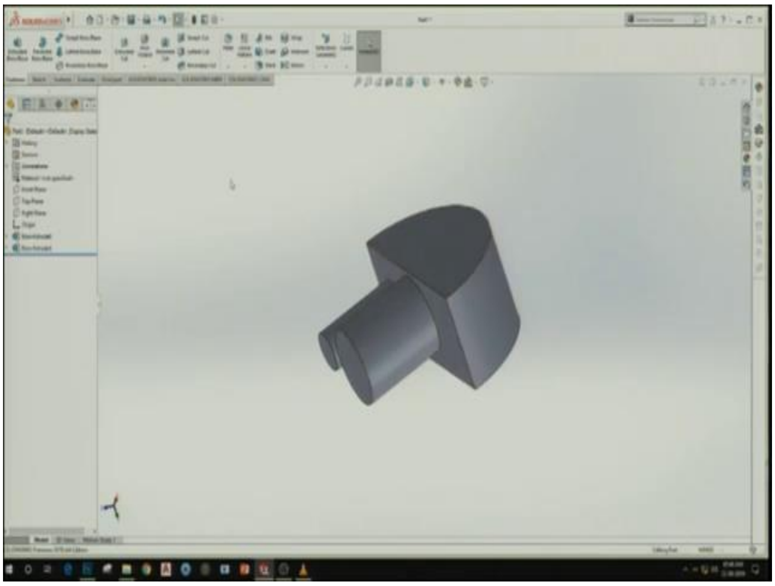

Now, both of them are selected and if I extrude both of them will extrude simultaneously. So, so it is okay 50 mm whatever we have said before it has already taken that dimension 50 mm as the extrusion length. So, this is the model that has been created just a hypothetical model not specific component or part that we want to use anywhere. So, let us try to see some of the operations or some of the commands or parameter change that we can do here.

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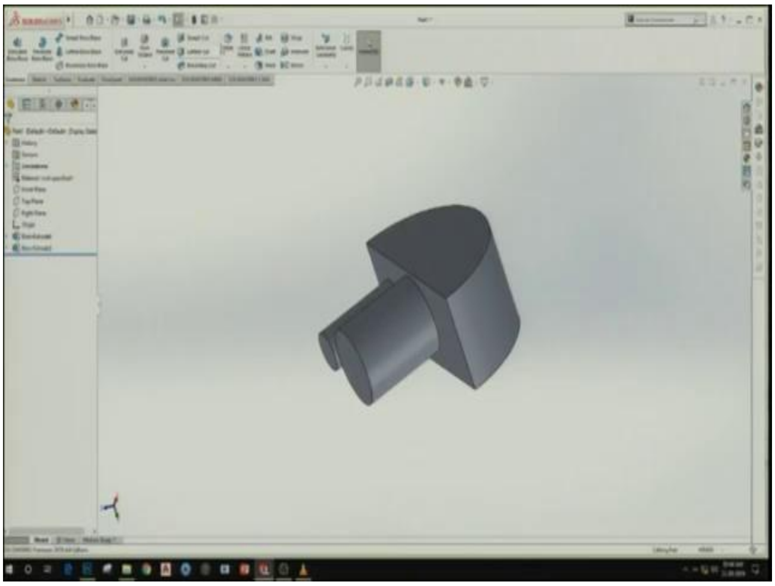

One of the things that we can do is which is very commonly used is Fillet. Okay if you click on Fillet, it will lost the edge that is filleted, so, this edge can be applied with a Fillet. So, it is asking the Fillet type coordinate variable, phase fillet, full round. So, constant size Fillet, variable size Fillet, Face Fillet and full round fillet a certain types are there and Fillet radius is given here full preview, partial preview, no preview, symmetric, and asymmetric.

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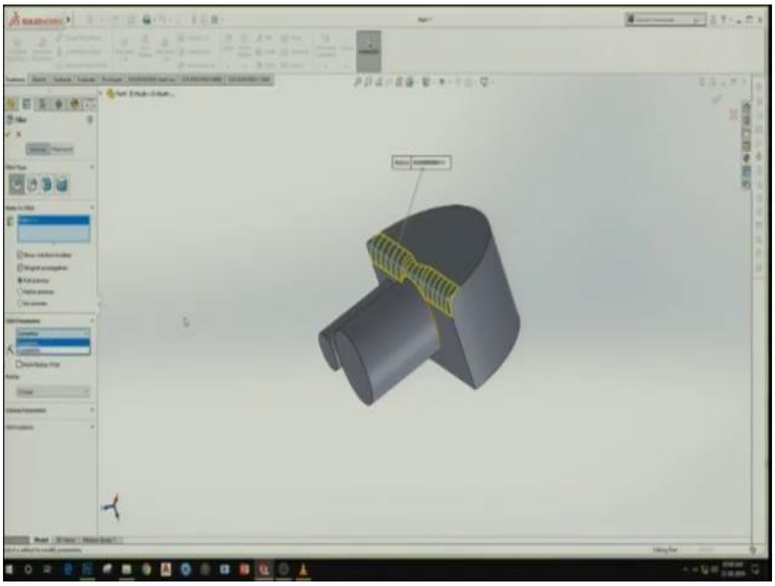

Symmetric if it is both sides like on the vertical and the horizontal side was at the 90 degree we would like to make it symmetric or is asymmetric. So, let me change this to symmetric and give it value, so 10mm seems a little big on this plus let me change to 5 mm okay this is okay somewhat okay then better. Somewhat okay then before. So, this is Fillet.

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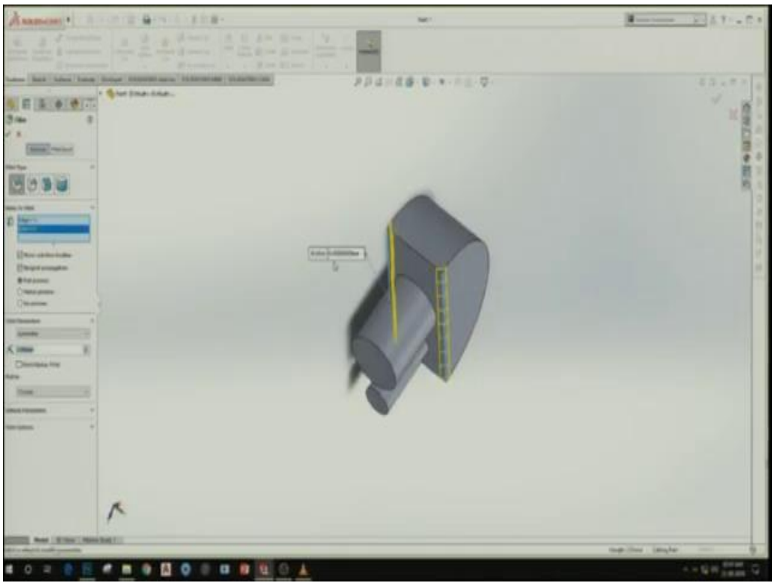

So, same Fillet we can apply on the other part directly like this. Okay change to 5mm. So, now the Fillet appears. So, if I need to apply the same command or same operation another side I will just click here because this is already active, okay, it is applied here. So, I accepted.

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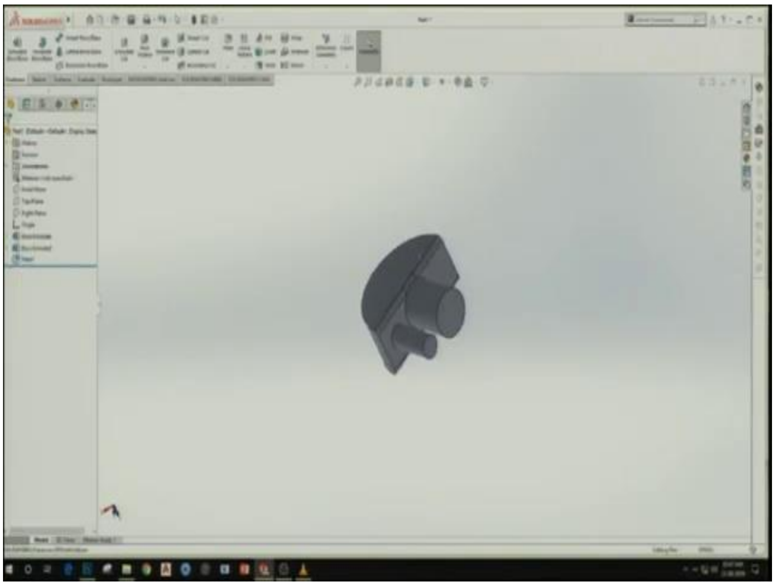

So, we can see it as smooth and surface on the edges there. So, this is Fillet command applied, Fillet operation. So, one of the close friend of Fillet is Chamfer. Fillet is a curve with a radius. Chamfer is a straight surface.

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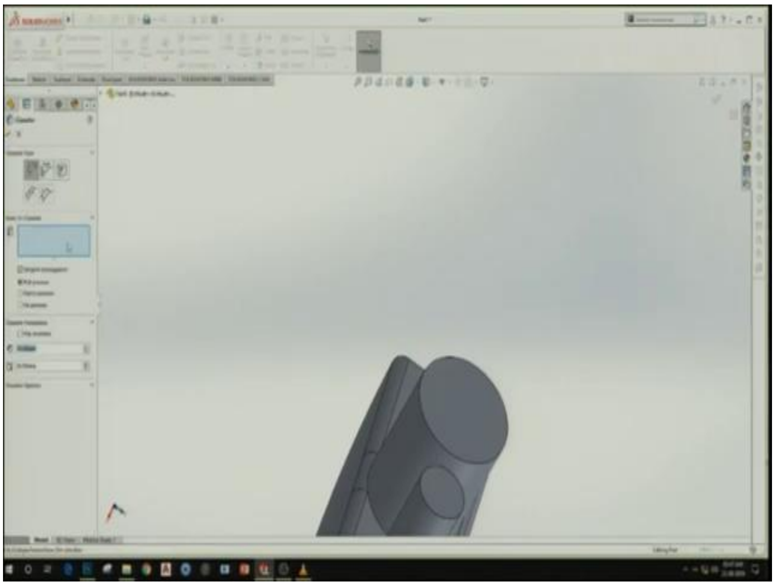

We can see different types of chamfers here. So, it is a slant height or a straight height, edge is smoothened but surface that is created is also a plane surface.

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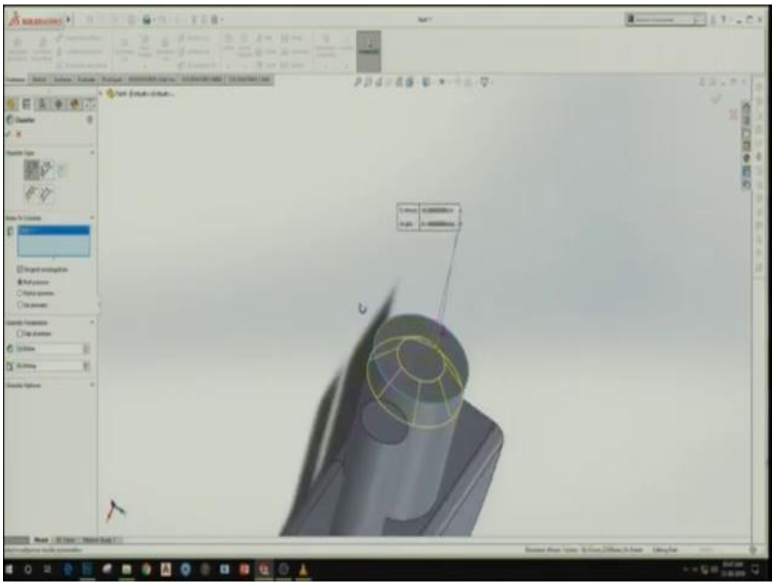

So, let me try to apply Chamfer here. So, it is showing Chamfer of distance 10mm into 45 degree. So, let me try to change this length also to 5mm. So, when I click here, so I will click right button.

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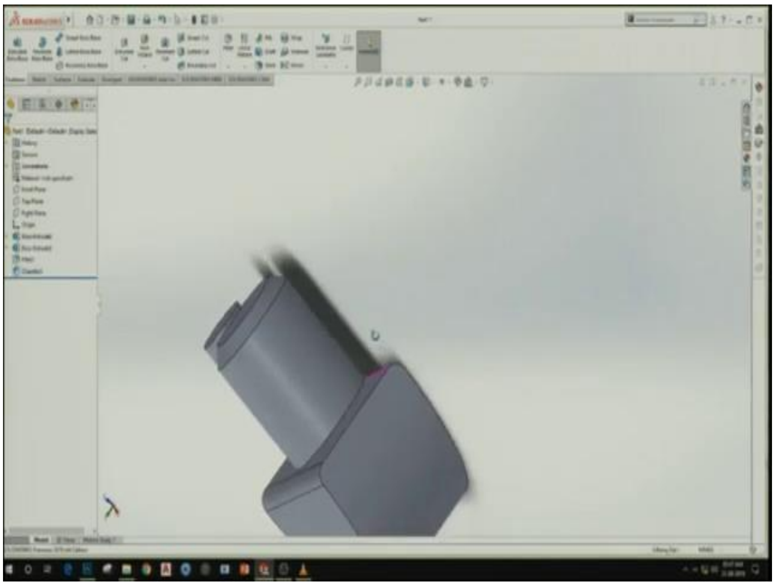

Now, we can see the chamfer is applied here.

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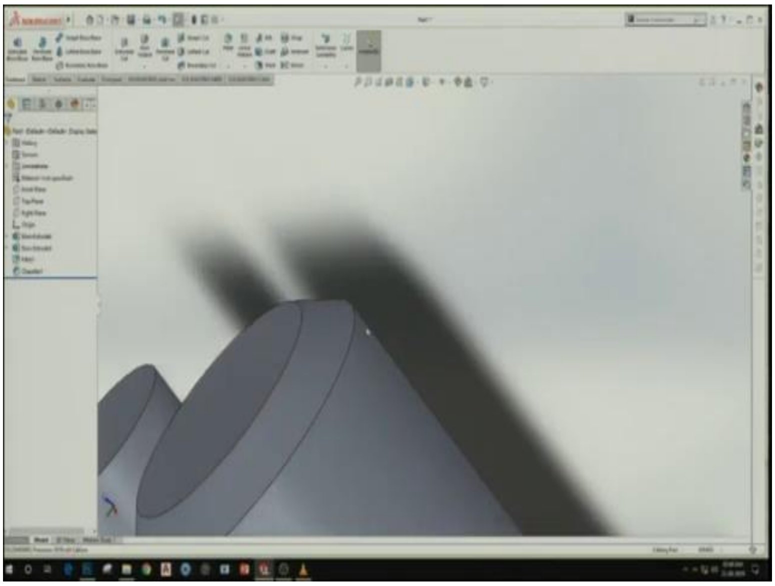

Say exactly 45 degree and 5mm.

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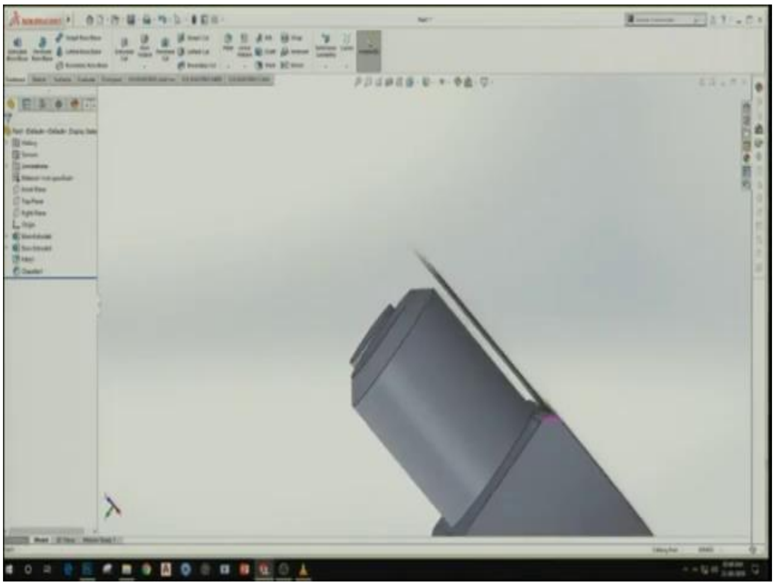

So, this is Fillet.

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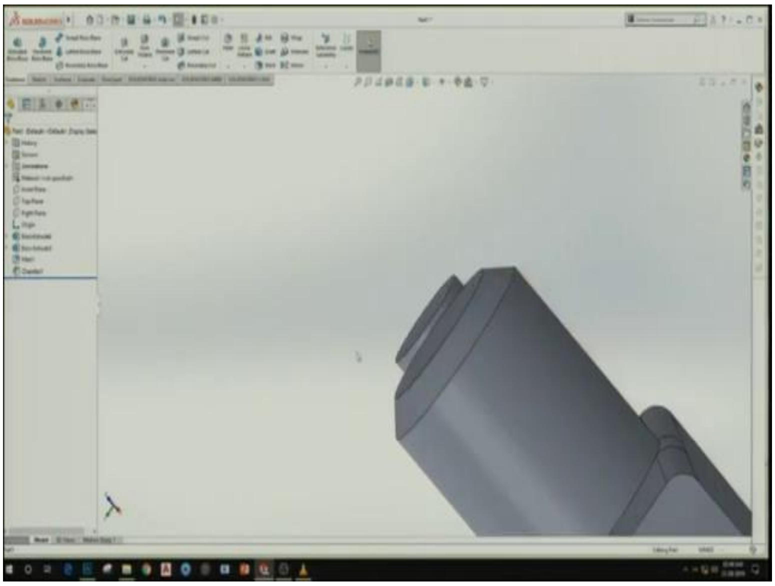

This is curved and this is straight, the other one is straight that is Chamfer. So, in machines actually we cannot all the time use these sharp or edges it is generally like this because someone has to hold that hands out there. There are certain curves are there, certain angle are there, so this is very important thing when we design physical components. So, let me just try to see another command. So, let me try to delete this and try to put some other operation now here.

So, I am just trying to pick some random operations and try to give you just small information on how SOLIDWORKS work. So, let me pick one more sketch here. Okay for the sketch, I have to pick a plane.

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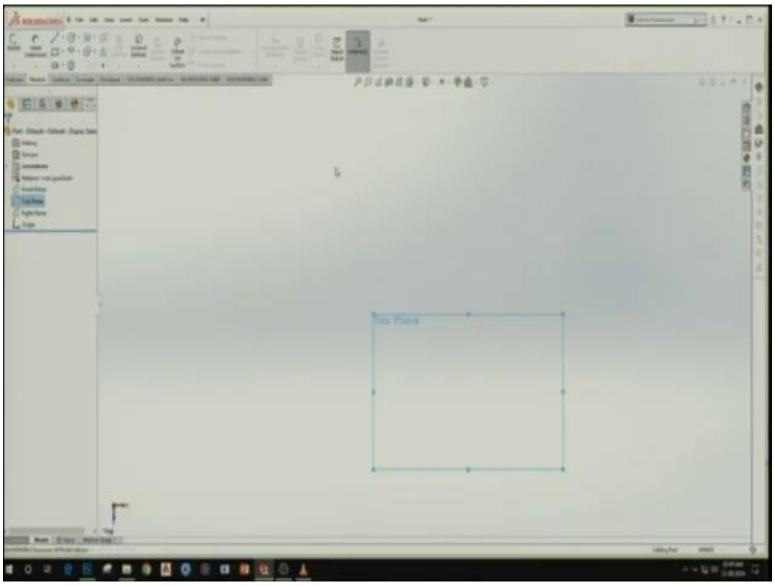

So, let me try to see the normal to the plane, normal to the tape plane. Yes, this is an important to note that how do we pan that we click the control button and then we move it, it can pan and the central scroll up and down can just make it zoom in and zoom out and by the simple left click it can show different views.

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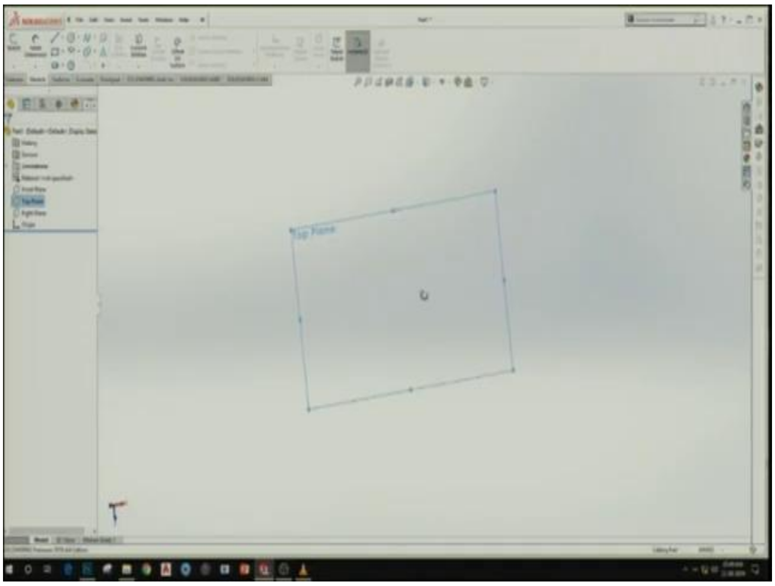

It can just rotate it like this, this is just a left click. So, just we need a normal view. I will just select normal view, top lane normal view, normal to the top. Now, let me start with the rectangle again. So, corner rectangle, center rectangle, three point certain ways of drawing the rectangle as it again a corner rectangle. So, when we click corner rectangle it will last the points, one two points.

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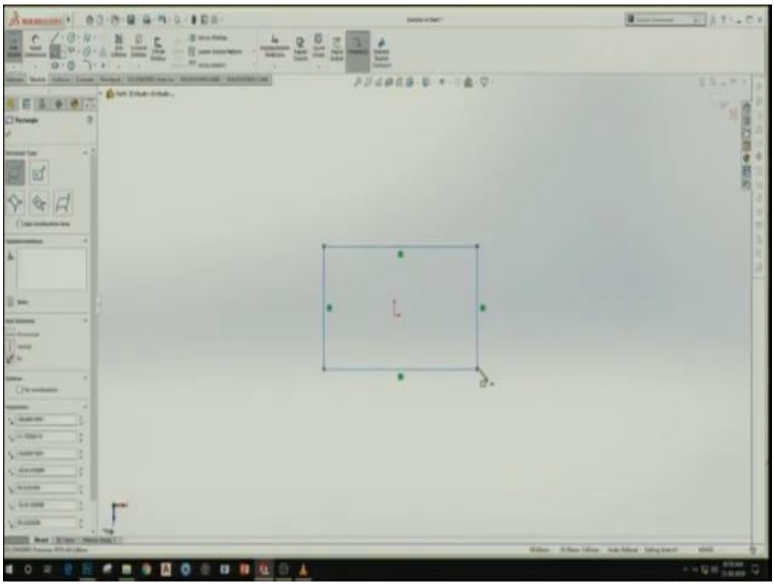

So, one point and hold and leave it here, second point. So, this is the first rectangle it is created. So, it shows all blue. So, the blue line here show that is not constant, the dimensions are not yet fixed that is we can just change the dimensions.

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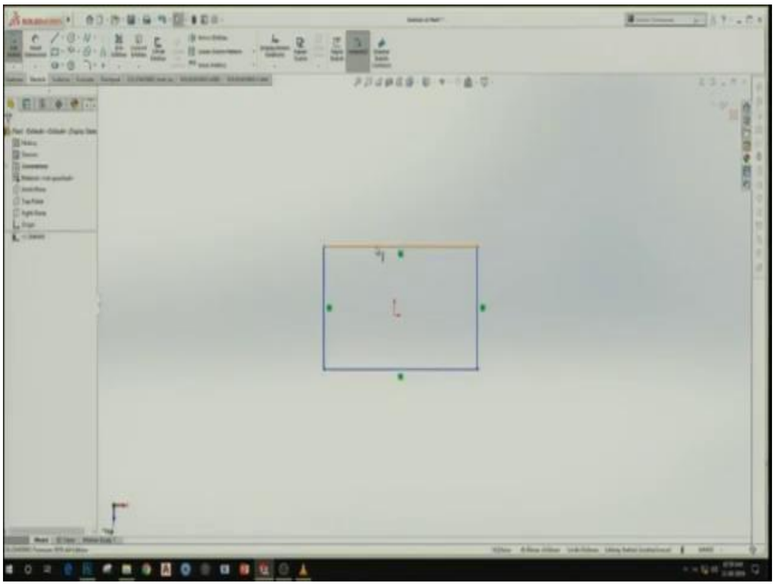

So, suppose if I click here and I try to move it, so if I change the dimension here, I make it 100 we can see this is vertical axis okay this small green squares what are they show that okay it is showing these that these two horizontal lines and the these two, this is vertical, this vertical, there is a horizontal to the axis, this a vertical to the axis. So, let me give some dimension here, let us make it 50.

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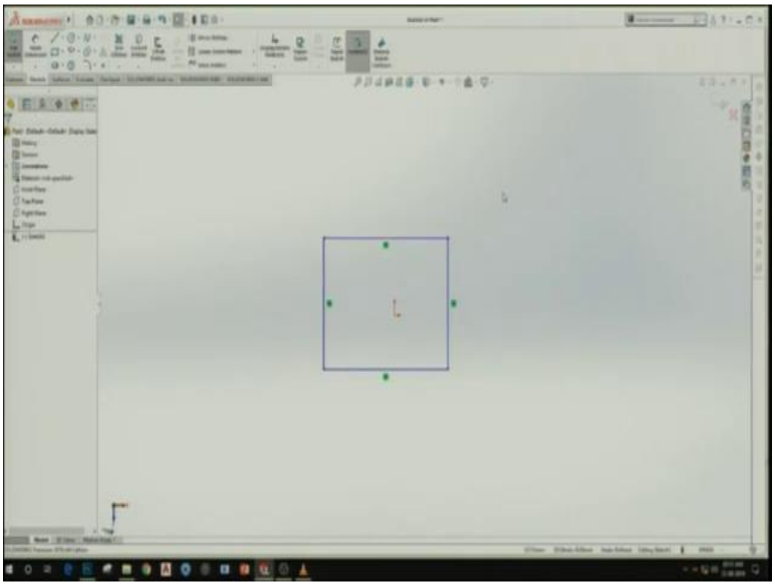

So, now it is fixed, so, let me try to put some more feature over it.

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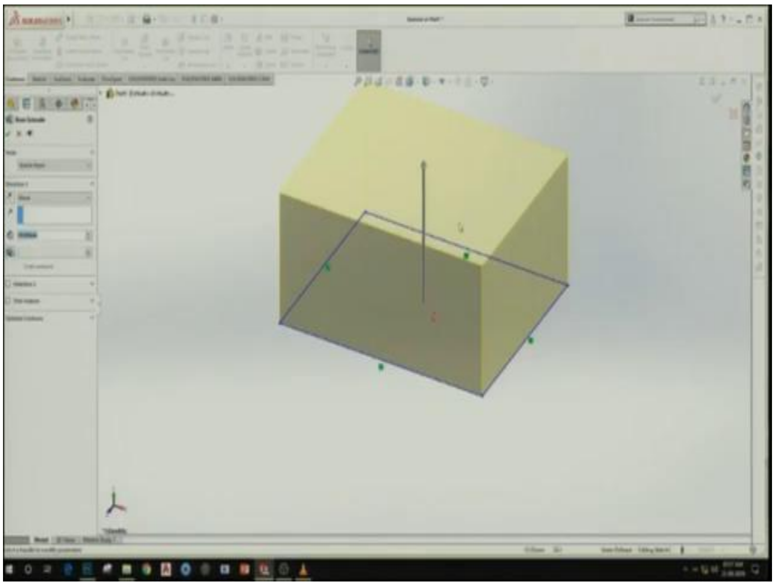

Let me try to first extrude this, okay this is extruded here. So, let me give some dimension here 50 mm. So, let me try to put some other, okay first accept it.

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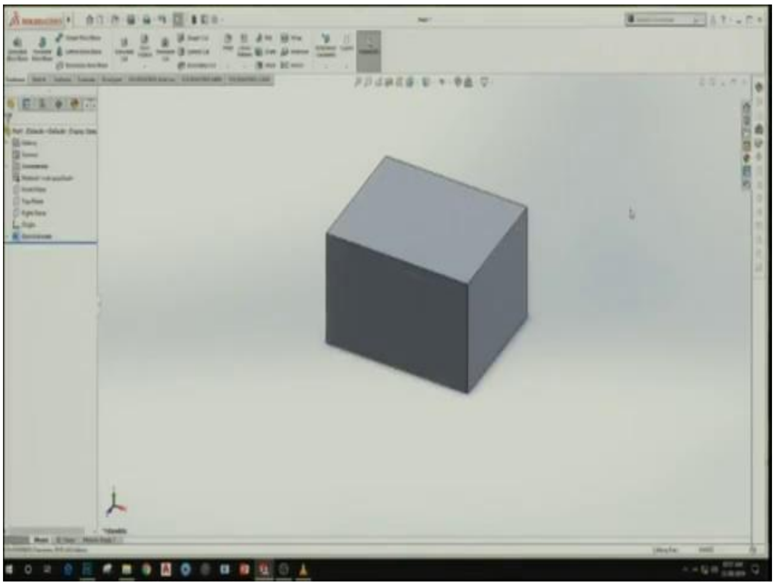

First accept it then I need to put some other feature over it. So, let me try to do an extrusion. And opposite extrusion that is extrusion cut. What does that do?

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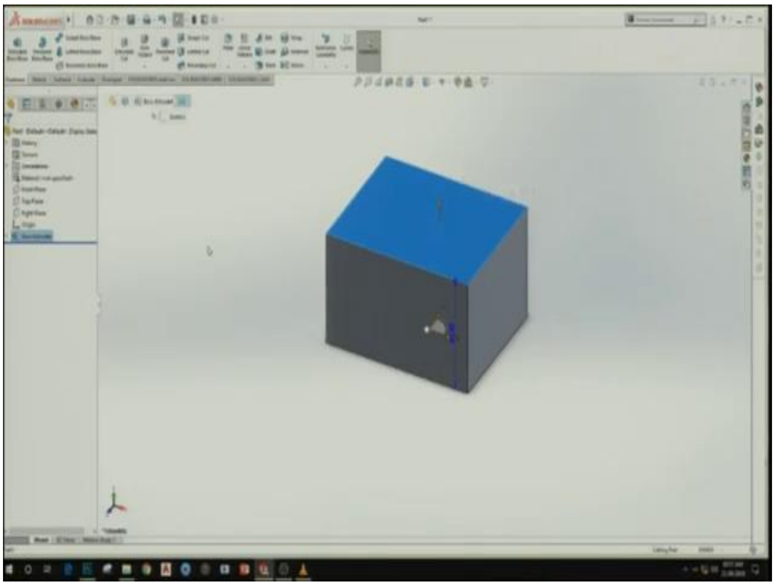

That makes a hole blind or through on the surface, this is a normal view of the face that we are going to select.

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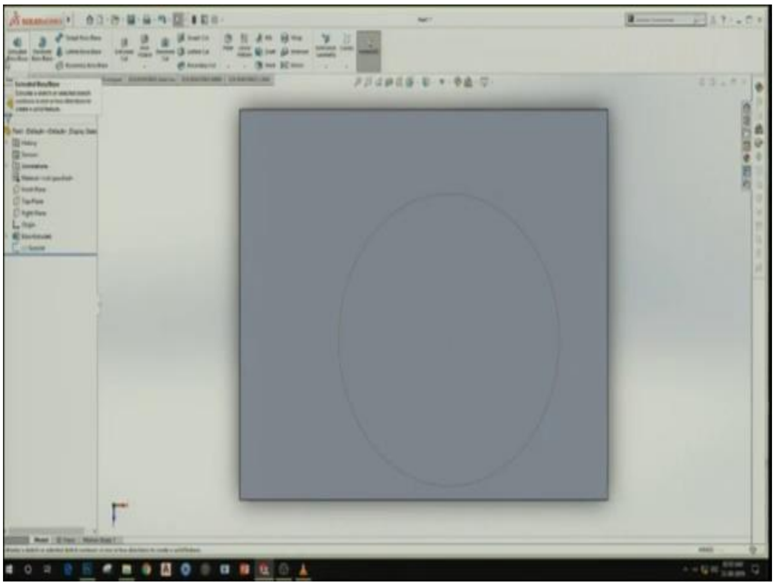

First we need to put the circle along which will cut the hole. I am just creating a random circle here. So, this is, okay if suppose if I need to align this to an origin I will change this to 0 and 0 very simple. Now, the circle has by itself come to the origin of my workspace here. So, let me try to give the radius as 30 mm. So, from the origin 30 mm radius and parallel to the plane.

Now, circle is in the black colour that is, it is fully constant it is not variable, it is completely locked now, that is the black color that indicates. Blue color indicates that it can be varied it is not constant when it is made constant the dimensions are fixed, then that color turns to black.

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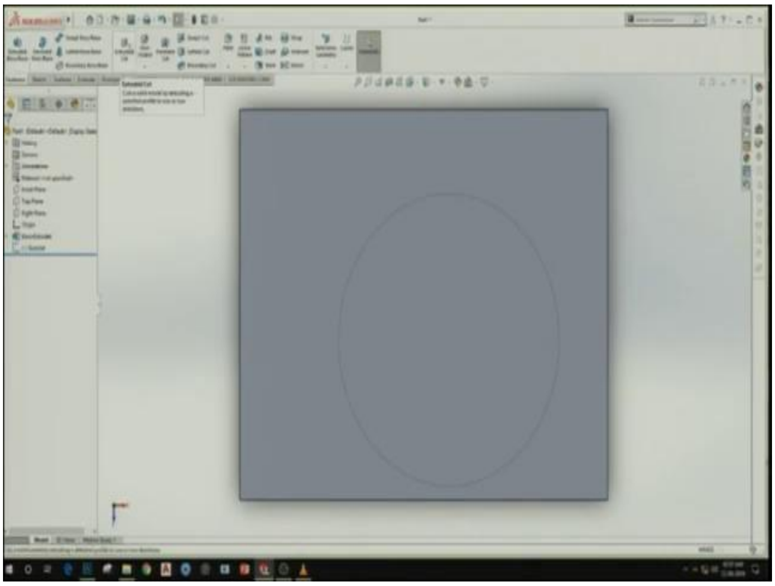

So, now, let me try to extrude cut I put on a suit and click on the surface had to be cut. So, let me try to rotate it.

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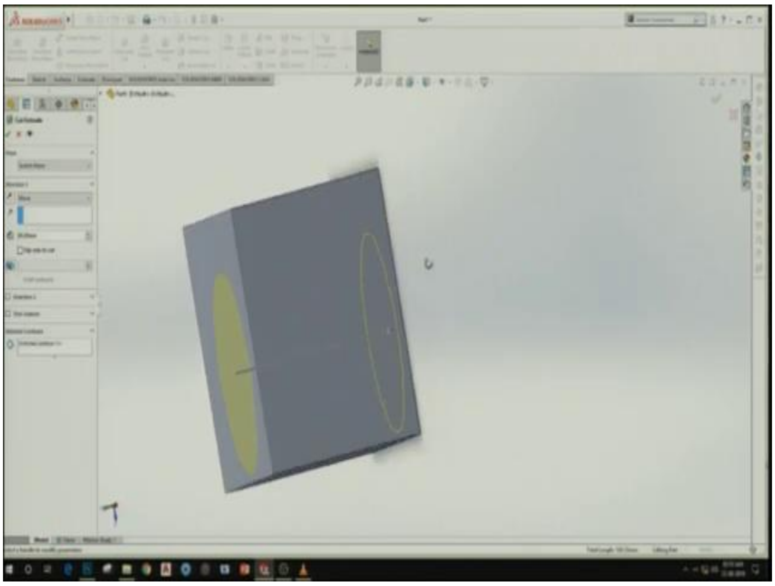

So, on the top of it will not be seen in the other way we can see that it has cut it. So, what are the kinds of the extrude cut those can be made. So, it is blind through both up to line, up to vertex, up to surface then offset from the surface and so on. Let me try to make this a blind hole of 60 mm, okay 60mm is a blind hole that is selected here. So, I will accept it.

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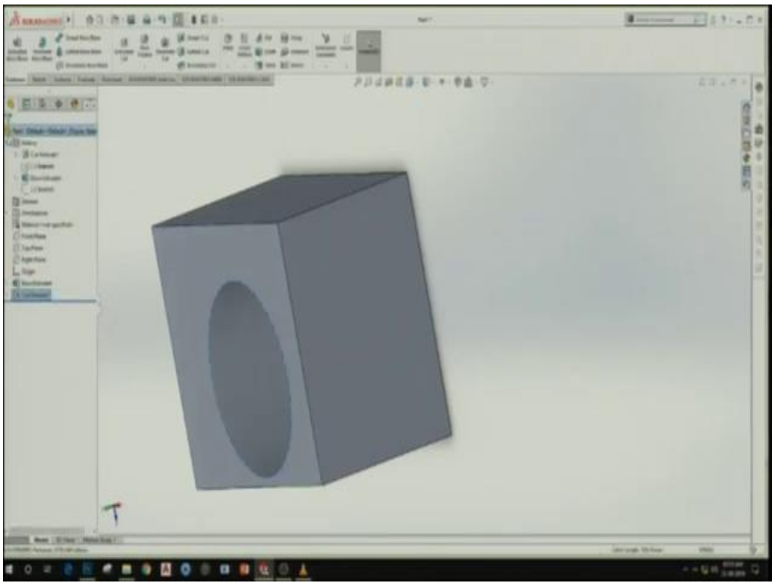

So, though 60 mm is larger than the width of the rectangle or width of the cube that is there to this blind hole itself has turned to a through hole, because the length for the hole that is given a 60 mm here. So, I am rotating it. So, this is extruded cut. So, there are certain whole operations like whole wizard, adventure whole, we are not going to any complex operations that you might not be able to understand at this point of time or if you are an amateur or if you are a very beginner. So, let me turn it back to the normal view and try to make another sketch over it. So, let me try to make a polygon.

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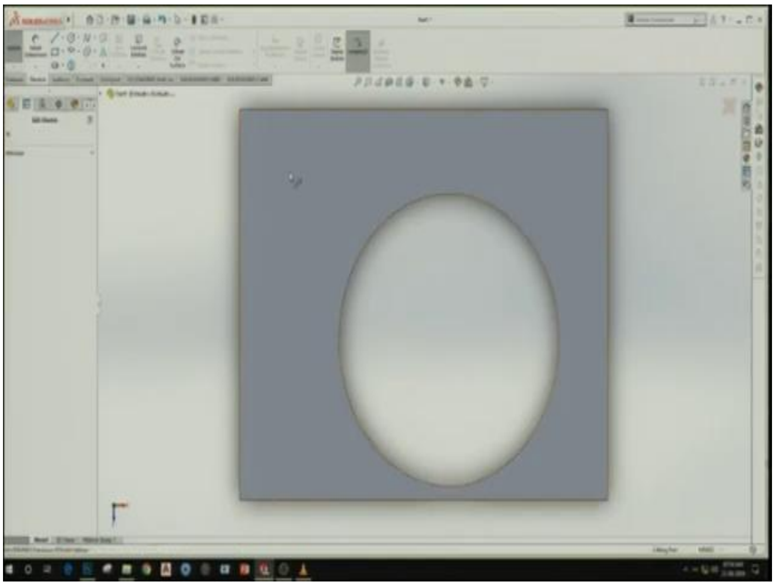

So, this polygon, click center and leave it here. So, this is a hexagon here. So, we can change the polygon in two ways either we change the radius of the circle that is inscribed in the polygon or we change the length of the edge here. So, in polygon, generally the length of the edge is known. So, we have just fixed something here.

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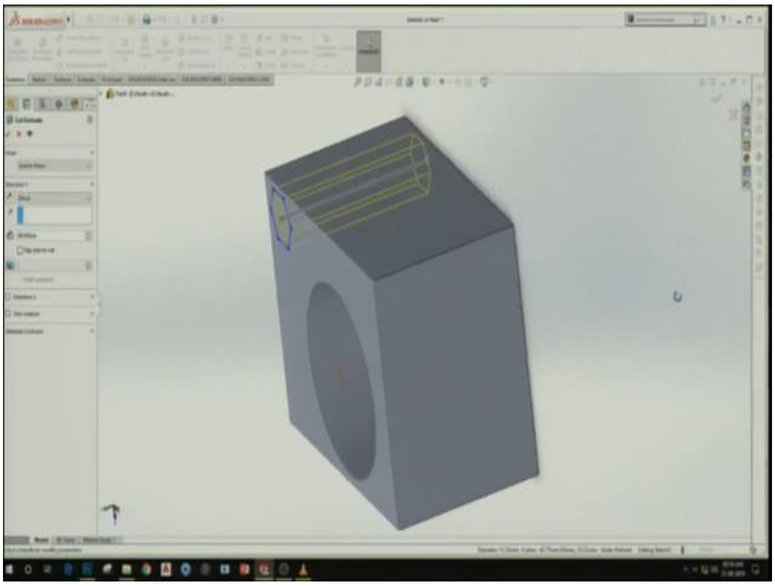

So, we are going to change the same extrusion we want to apply same extrusion here, so it is extrusion cut here, so this is already blind, blind means up to certain limit. So, let me try to change its length. Now, it is 40 mm blind through all would be obviously through all, so up to mid plane is one of the options. Okay, blind up to let me say 30. Okay, up to mid would also, would actually up to mid would be up to 25mm and a blind up to 30 mm is hole or up to 30 mm here.

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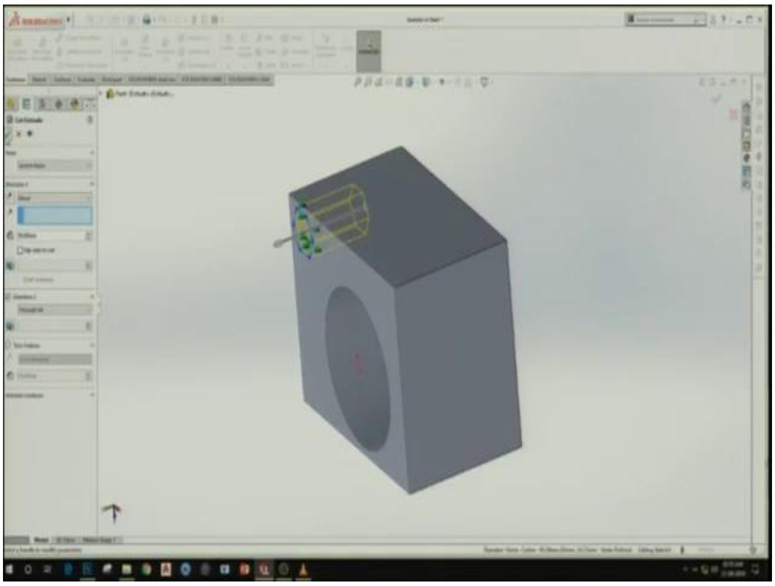

Would accept, we can see this slot here, we call it a slot or a blind hole, hexagonal hole. It is a cavity. Okay, one of the things I like to show here suppose if we need to sweep this object from one plane, so what is sweep? So, let me try to pick the plane along with a sweep has to be done.

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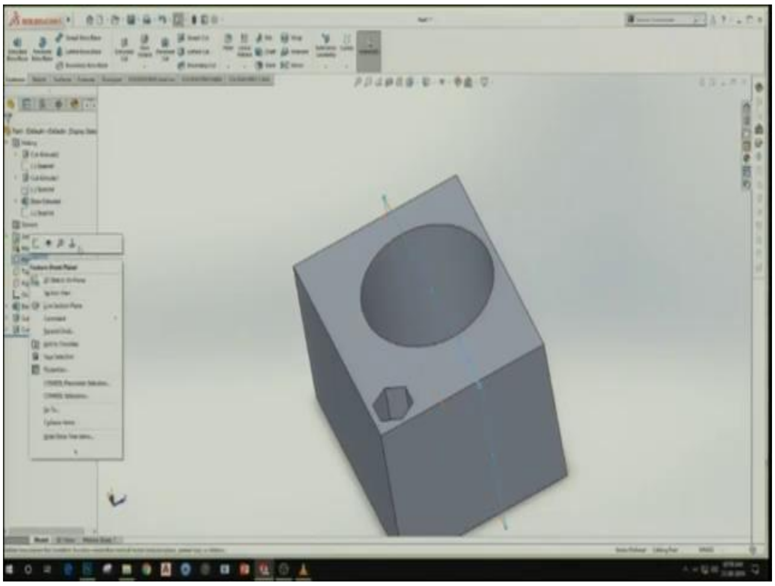

So, this is front plane I have selected okay, along this plane I like to sweep it. Not along this plane, like at the side of this plain. I will create a line, just I am creating a line, I am selecting sketch that is line.

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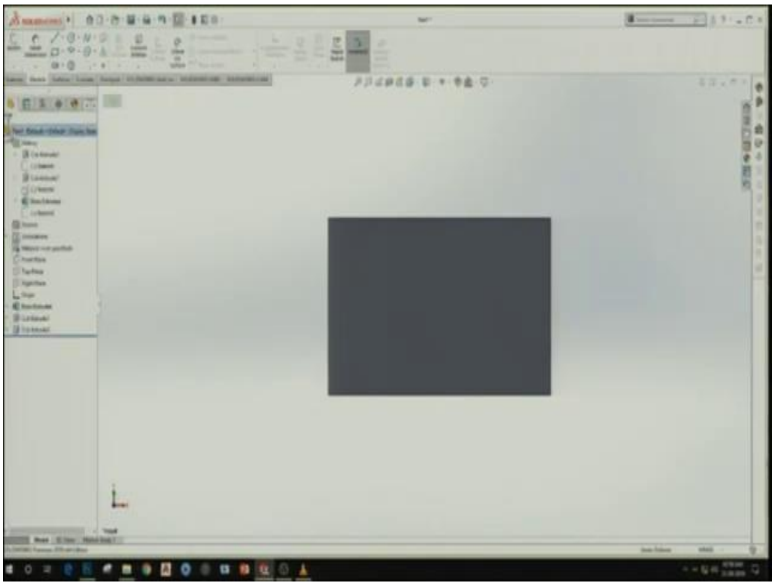

Okay, select. Okay, click on the line, line.

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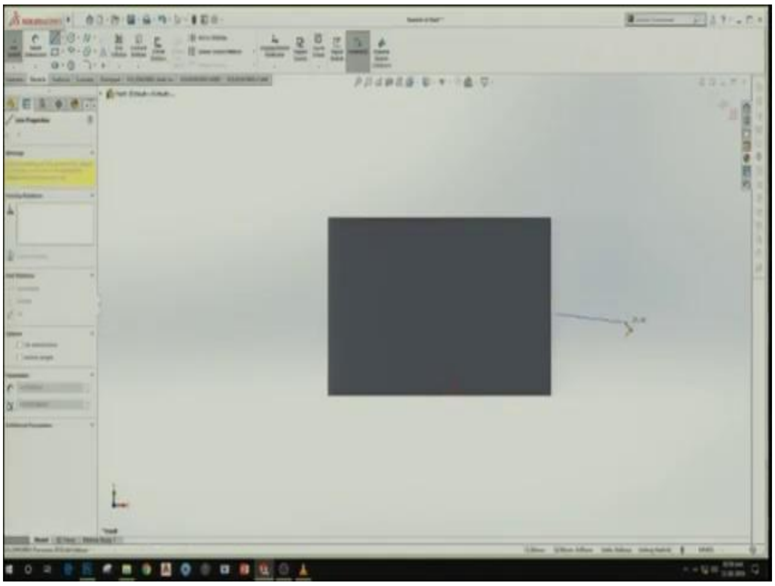

Then at any point in the space I click then I just made a random profile escape. Okay. So, how to combine? How to touch this line to the object? One way is we can just drag it there. Okay, okay I will suggest some other way. See to make an object or to make a drawing in SOLIDWORKS there is certain ways, there are number of methods and number of ways in which you approach the specific design, in which you make the specific drawing or the model. There are number of features that can be used.

It is suggested I would definitely like to suggest you to use multiple ways to design the CAD model you can choose different features you can use different, like if we now make different holes, which are similar to each other, you can just use pattern command, you can use layer command different ways to do that. So, more features you know, more handy would the software to you. So, it is suggested to use different commands, different operations to make the similar kind of drawing.

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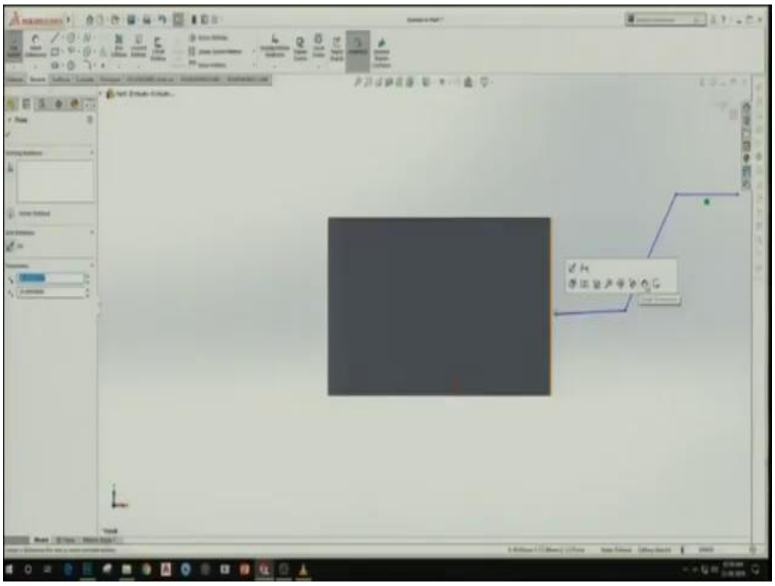

So, in this case what I am doing, I am trying to make a line for the profile to be connected. So, what I do? I will see this dimension, this distance between these two, this is 2.29. I will make this distance as 0. Okay, this is made 0, now it that means it has touched the body.

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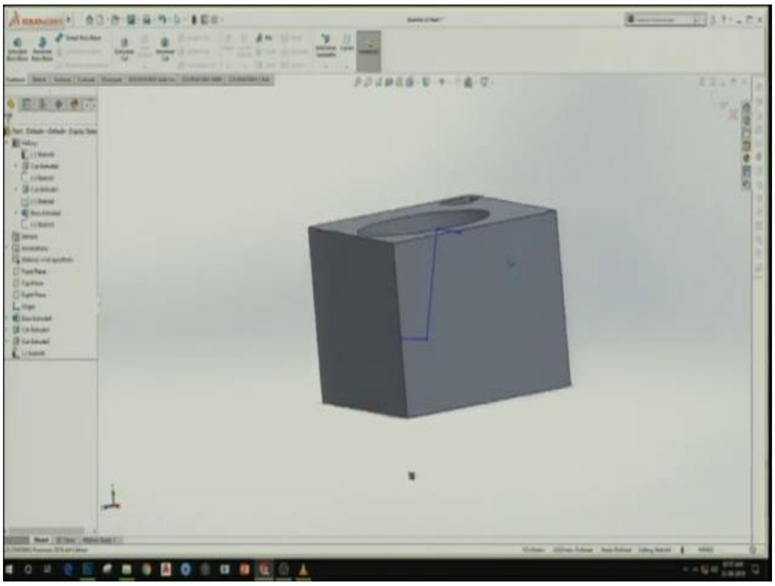

So, now I need to sweep this? What is sweep? So a swept boss, okay if I see there you can see here that there is some profile that is test with that.

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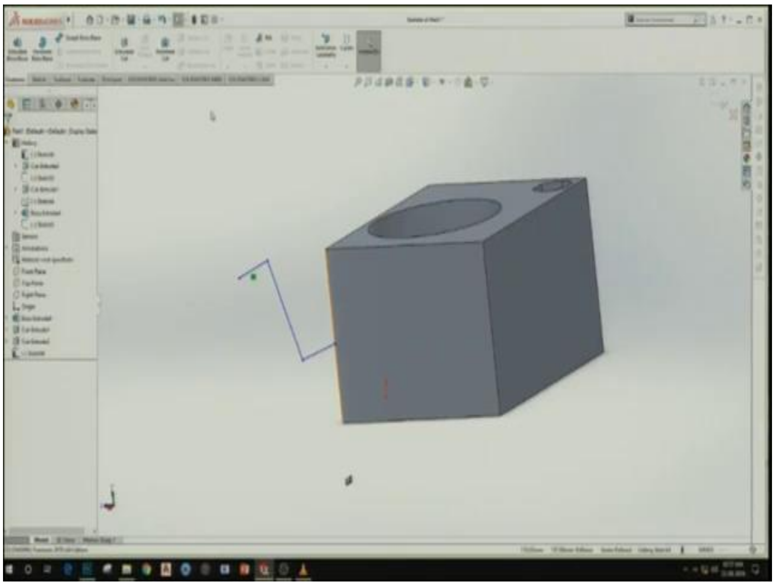

Now, so there is a green icon here that shows that it is not yet fixed, it is okay.

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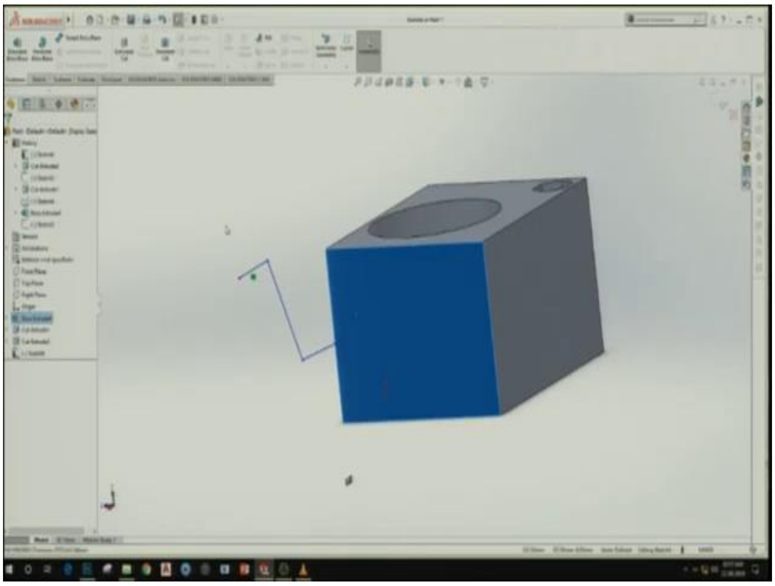

Now, come to features and let me select this profile all these in one go.

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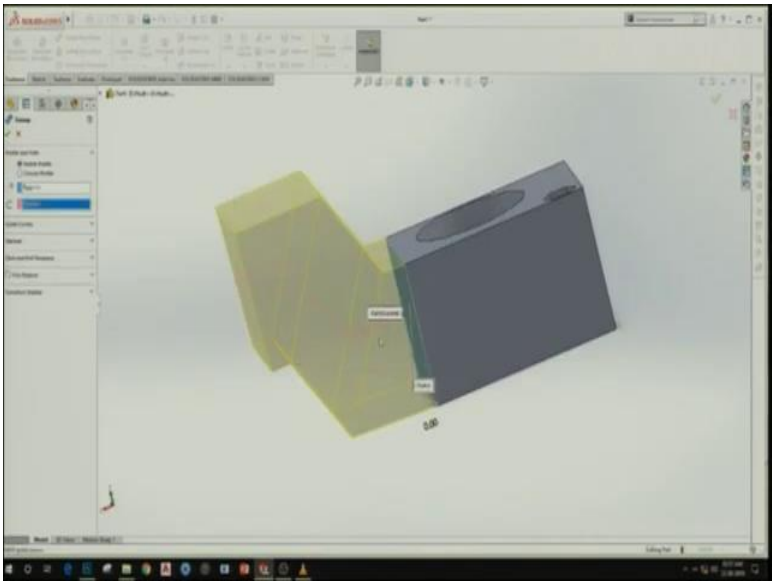

I need to sweep this face along this line, okay. See this is how sweep happens. The whole face is swept along this line. This is how this happens. This is accepted.

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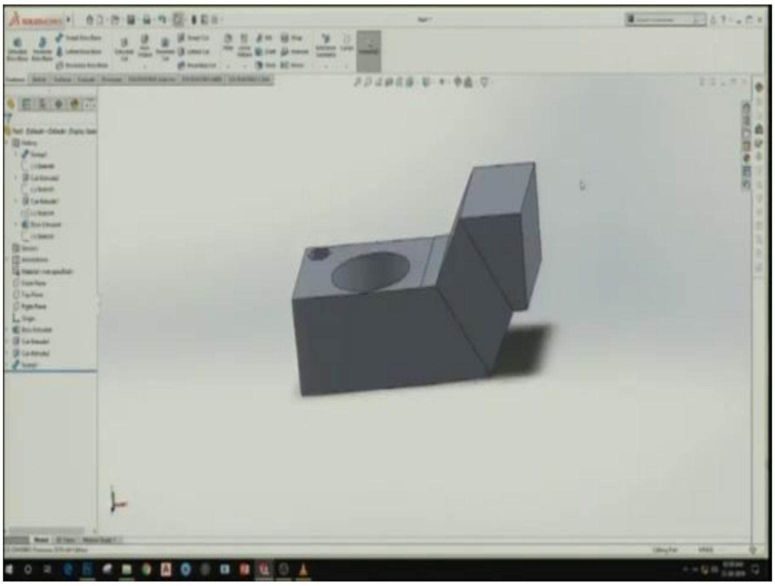

So, we have got this body here. This is a sweep operation and this is also an important operation to learn.

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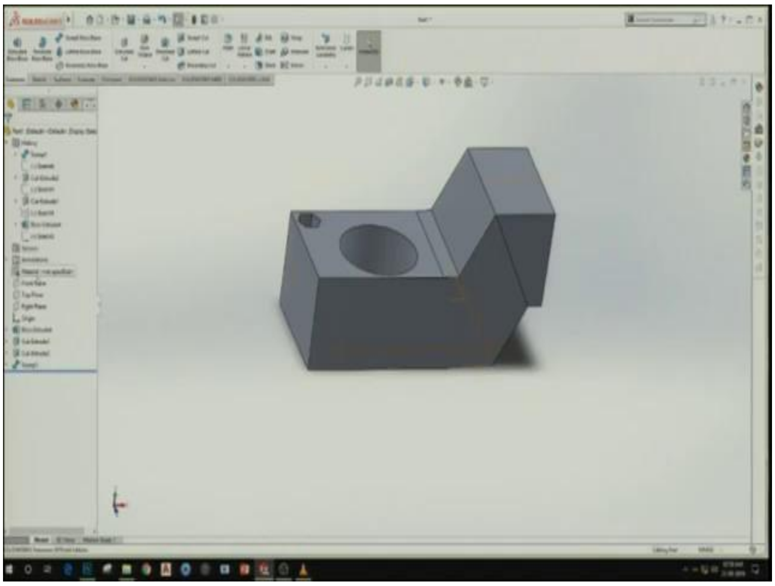

So, another operation like sweep is Loft, okay, I am leaving it for you, I would like you to install SOLIDWORKS on your computers, if you have systems, if you might not be specifically having, I would suggest you to work in groups, and to use your friends computers, or maybe in your Institute's, you can use different computers.

Try to install the SOLIDWORKS and try to work on the different features. And if you have any questions, we would be happy to answer them. If you need any commands and how does this happiness in SOLIDWORKS he can see just videos. So, this you can definitely practice in your institute.

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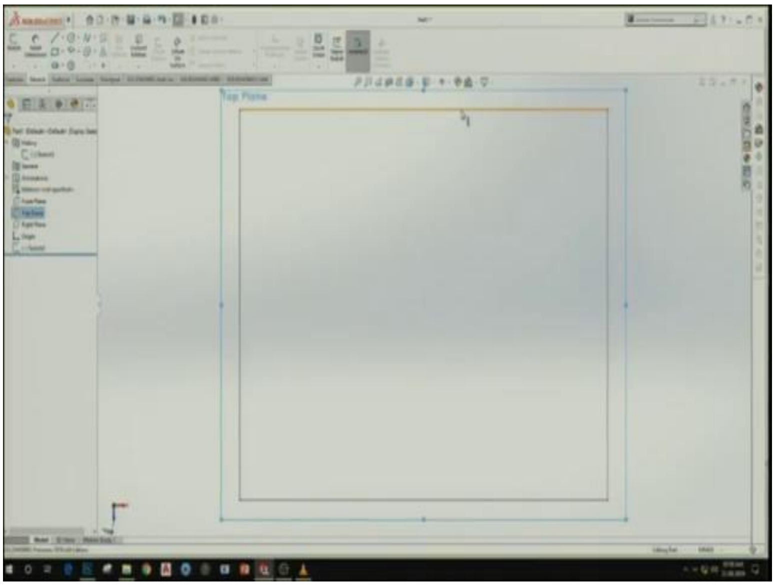

So, next what I will do, I will pick another sketch here. So, I will try to pick a plane. This top lane is picked here. So, this already we have a rectangle of the top plane. So, let me try to first delete this rectangle again or we can edit this rectangle.

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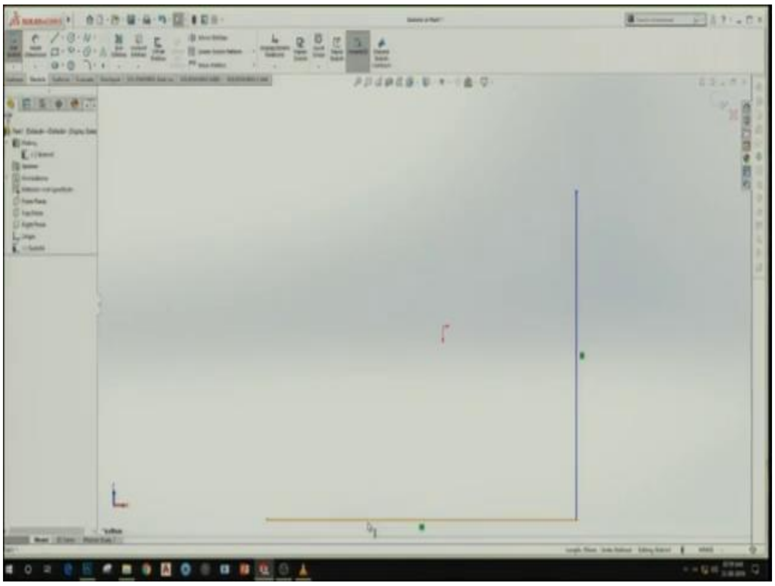

Okay let me, okay separately deleting the different lines that is also one of the ways. So, I am showing you different ways to delete in normal.

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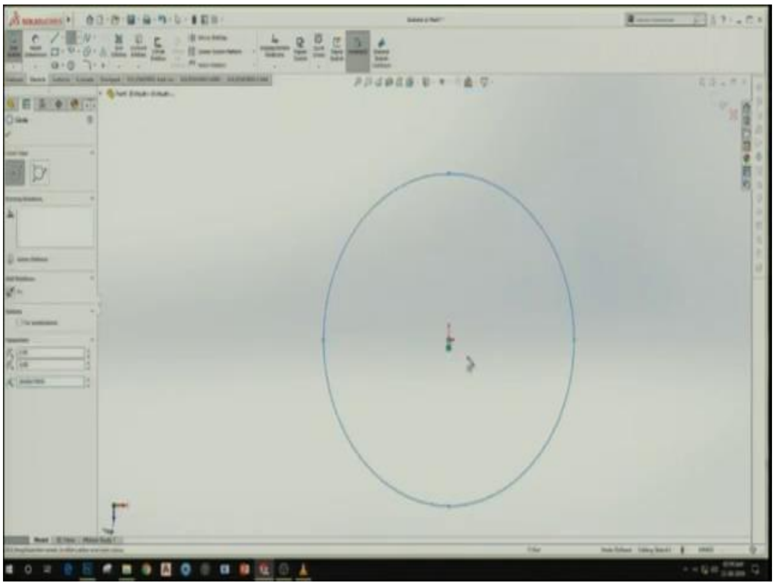

So, let me say if I need to create a pattern as it just said, if you need to create different bolts, different nuts of similar size, then you can create a pattern for them as well.

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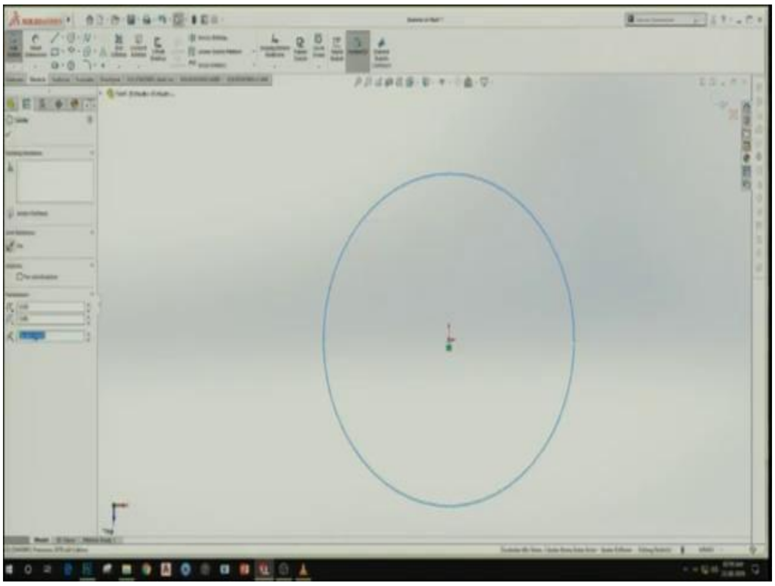

So, let me try to put some radius here, okay let me put 90 as a radius for the circle and accept this.

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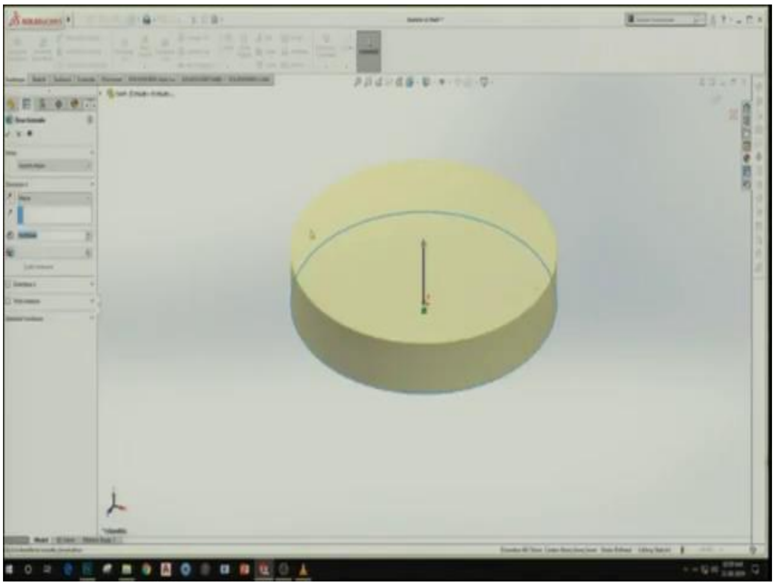

So, then extrusion.

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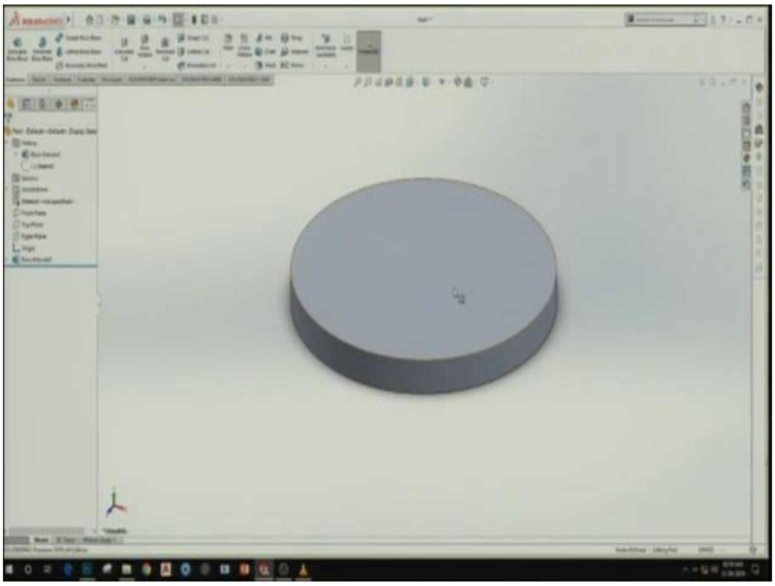

So let me put extrusion value as 20. So, it has turned to a disk now. So, this is a disk of 20 mm height and 19mm radius. So, let me try to put various pins over it. So, how does that happen?

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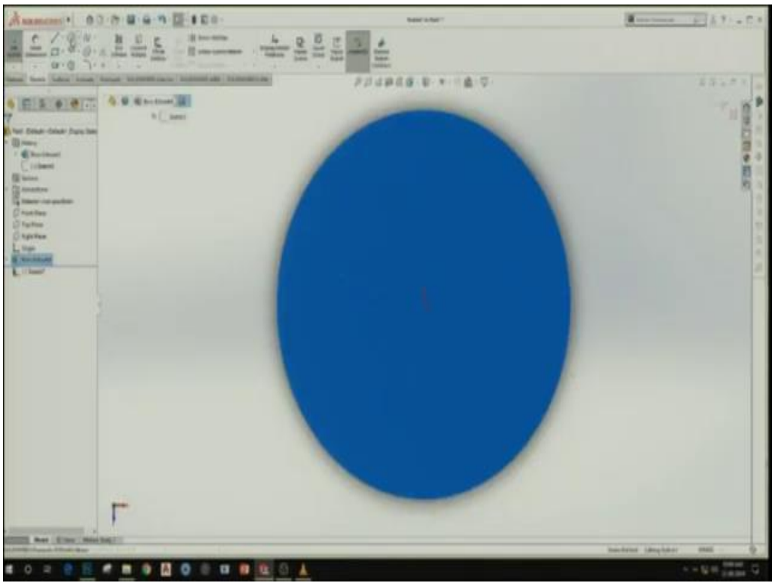

So, let me try to put one circle over it. So, we can see a blue dotted line here. What does this show? This shows the vertical axis, this shows the horizontal axis.

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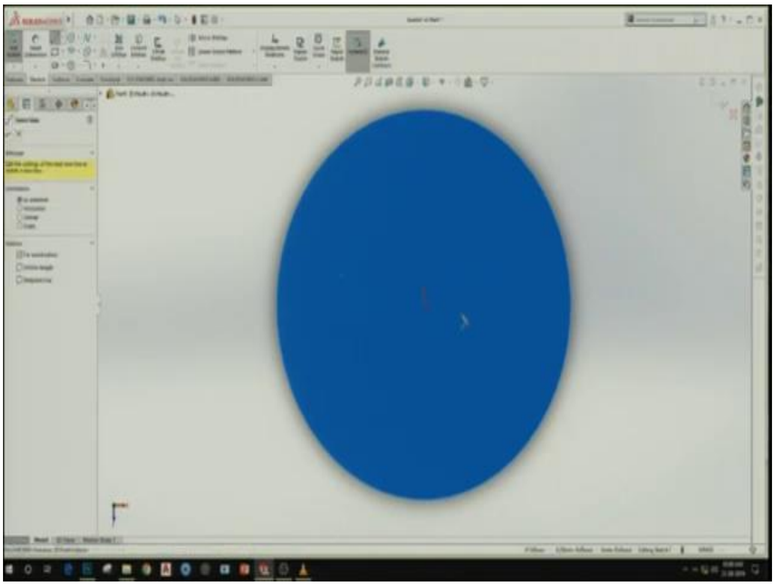

This actually top plain you can see at the bottom left that it is showing zx, it is zx view because it is a top view here.

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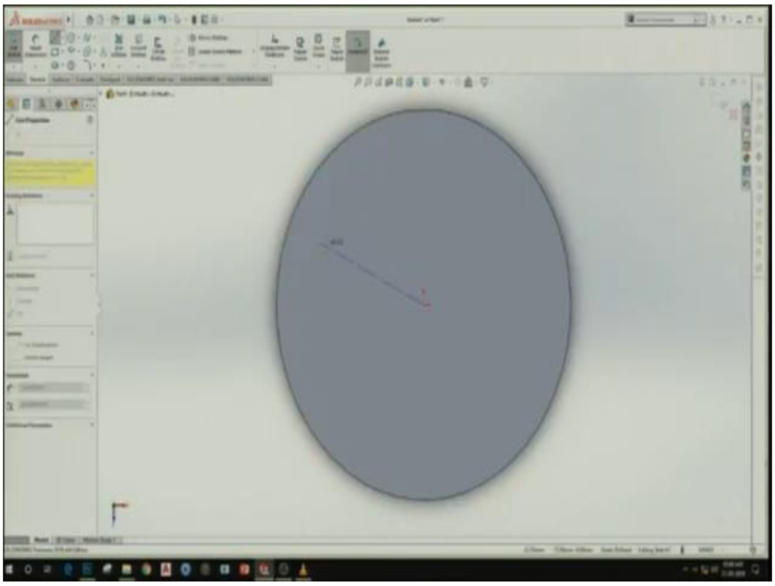

So, this is a origin so, this is a center line, I am just putting this central line here that let us, let us try to see again, how do we put the center line. This is the vertical horizontal line. Align Center line, center line here.

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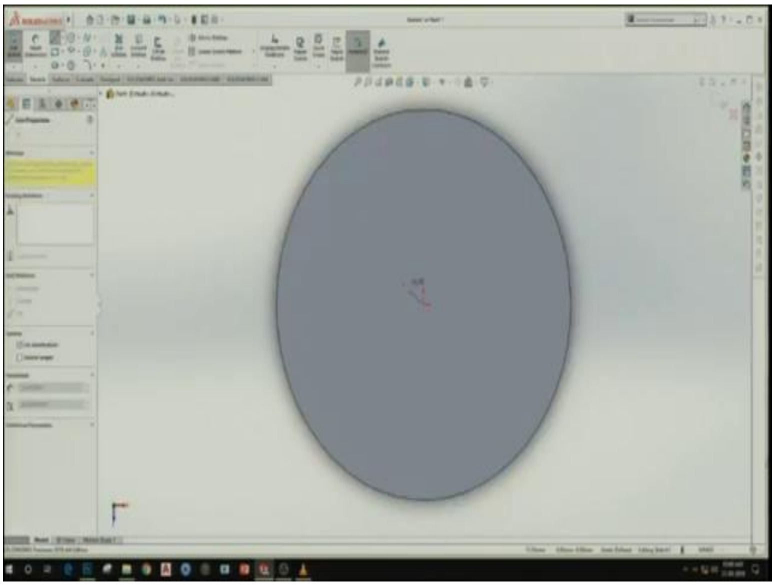

I will click here and try to put this center line to this extent here, but this is just for the reference a central line is there, okay you might be know the rotation for the central line. So, how to draw the central line you have seen.

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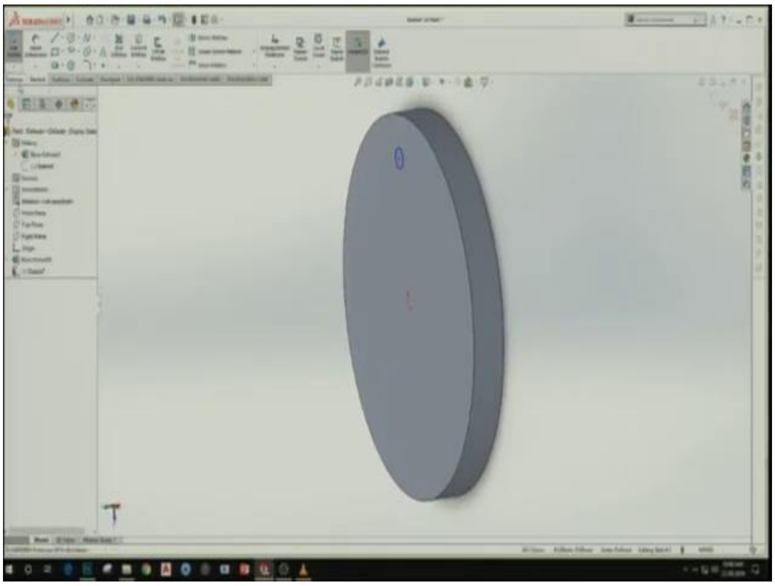

So, let me draw a circle here. So, I draw a circle of some radius let me see radius 5, okay, accepted. So, you can see a circle is here.

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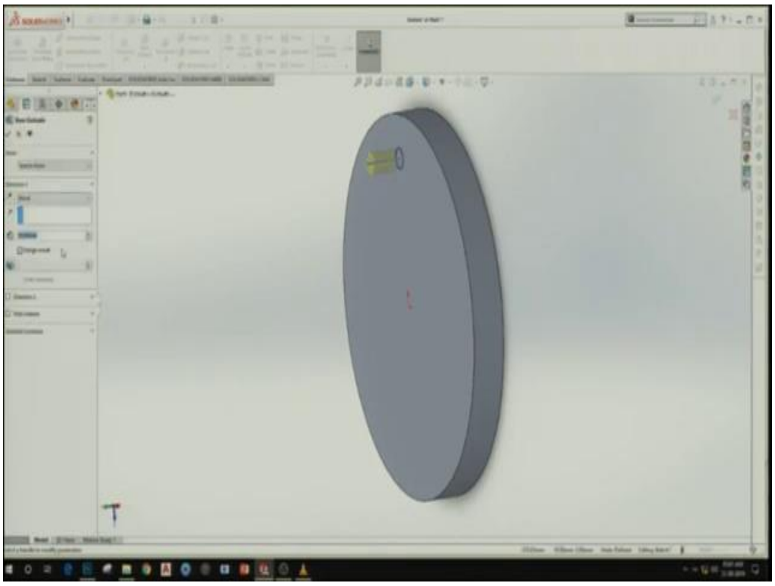

Now, you need to extrude this circle okay this is extruded, okay let any random length whatever it has selected. Now, we need this pattern in the whole disk distributed evenly, we can select the number of pins here.

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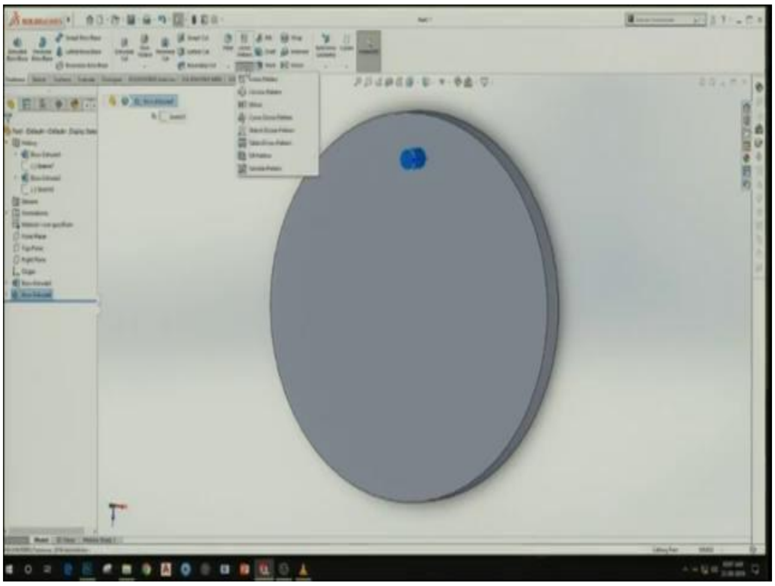

So, I will click here on the linear pattern I will select circular pattern from here it will ask instance spacing or equal spacing. So, I will put equal spacing then how many numbers do you need all along 360 degree how many number let me say, if I say 8 I put the number 8 here and it will ask me direction that clockwise or anti clockwise? It is clockwise is selected here 8 numbers symmetrically put there. So, x y z direction it is asking, okay everything is correct okay done.

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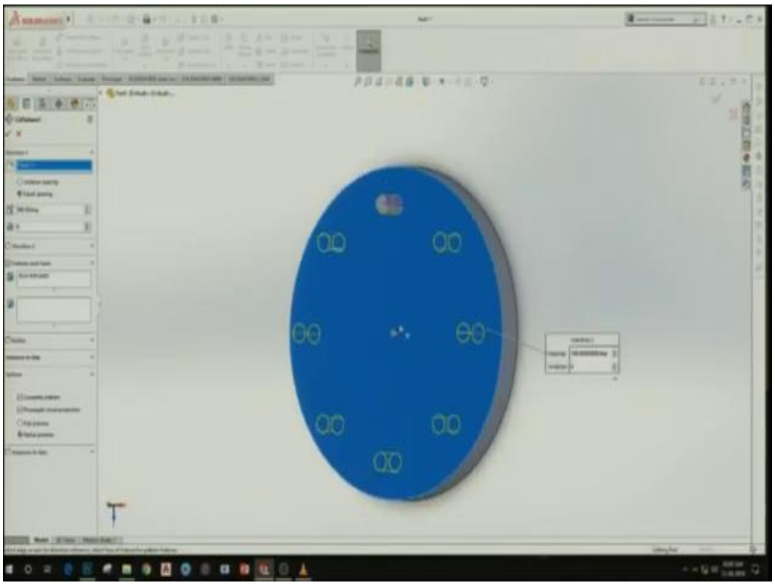

So, when you click the body it has created pins here, 8 pins I will not say pins, it has produced 8 repetitions all along the circle, 60 degree.

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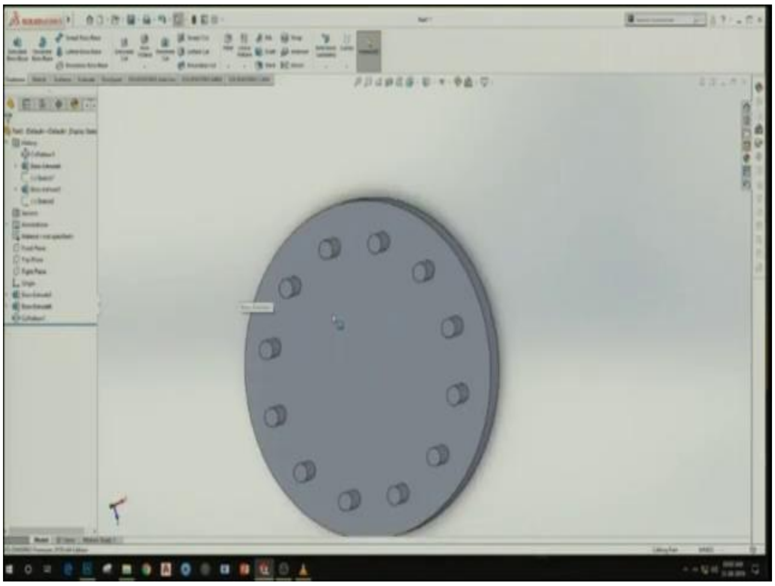

If I turn the number to 10, 11, 12 so on, it will show the equal number of the bodies or the objects those who have created here. So, this is pattern, the similar pattern in extruded cut can also be produced. In that suppose, so okay let me pick rectangle now here.

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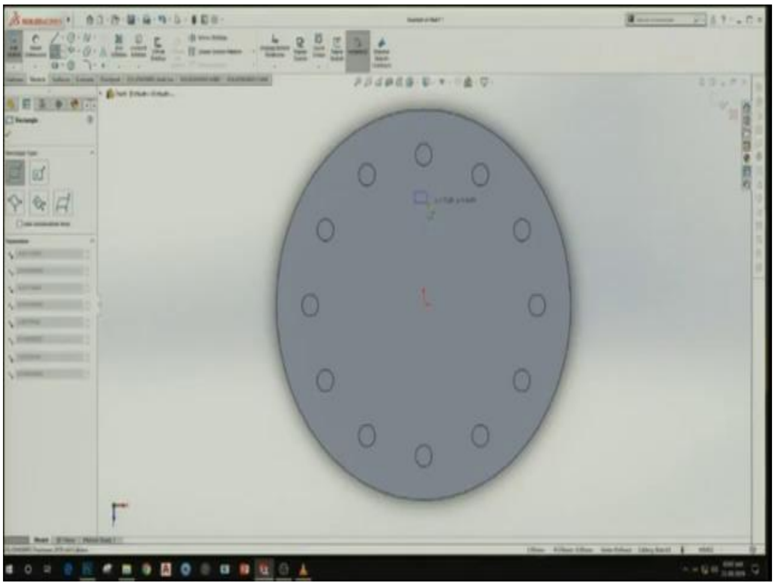

So, I will make a rectangle here at the center. So, then select, the sketch is selected I will put extrusion cut it is, it will cut to extrusion rectangle select. So, let me try to put one more pattern here.

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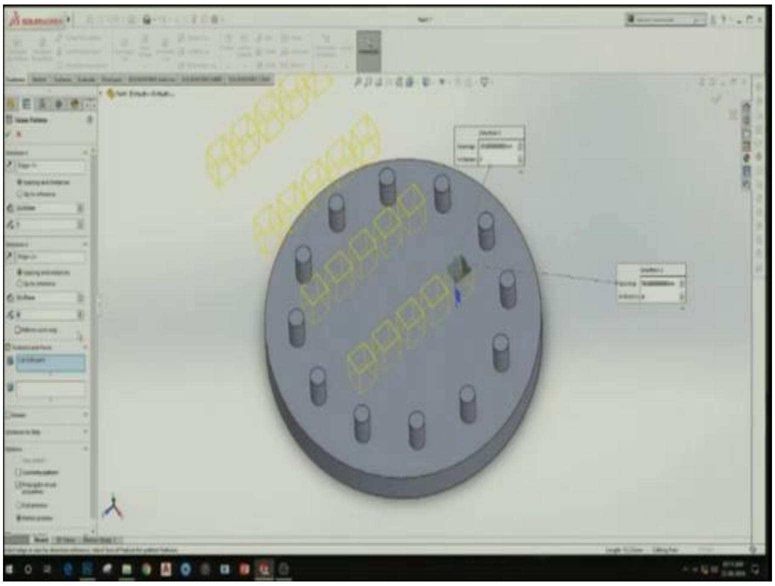

For this pattern let me try to select the linear one now. This is selective, the cut extrude is selected and go to linear pattern which is in which direction we would like to do pattern on this direction the number is 5. Also in the both directions can also be done if you select both the directions and if I say one other direction as well, the second direction it is extrusion one, I turn this number to 2, 3, 4 the distance is also there at each 50 mm it is producing the pattern. So, 4.

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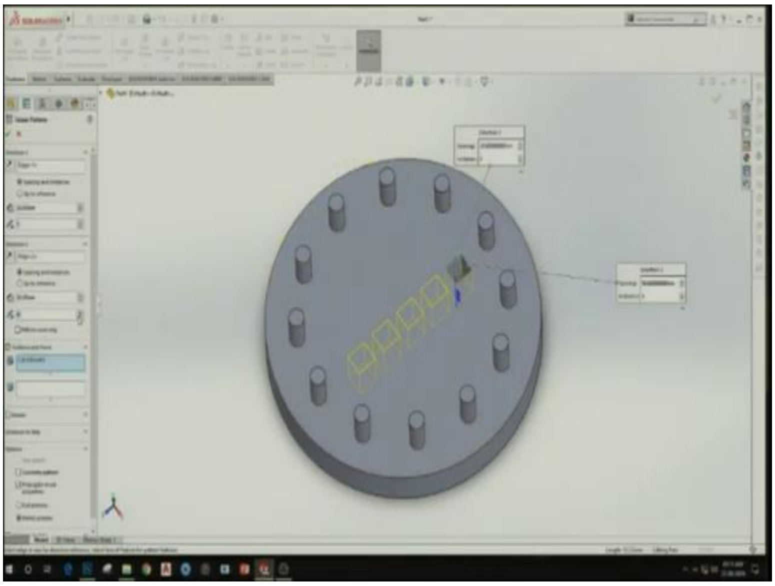

So, let me turn this back to one direction only and select.

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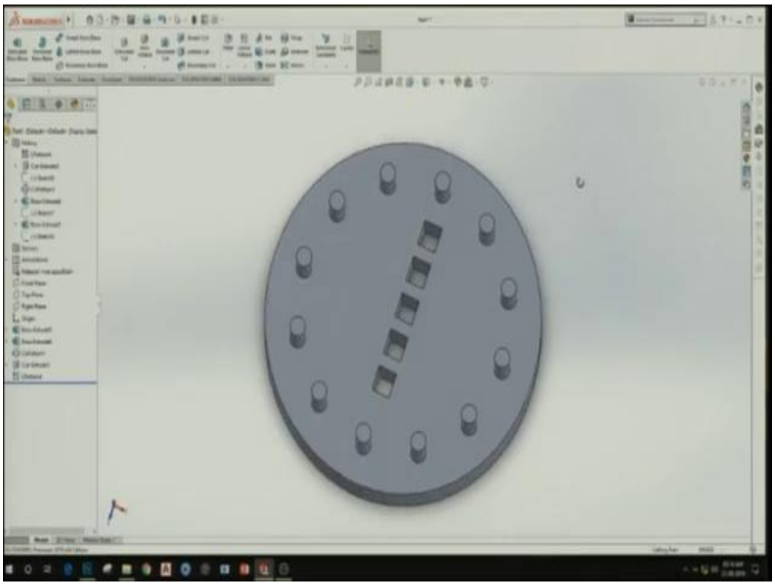

So, it has made the cut extrude here. Cut extruded pattern here or extruded cut pattern, so this is how do we use pattern command. Also yeah, one of the options is mirror option.

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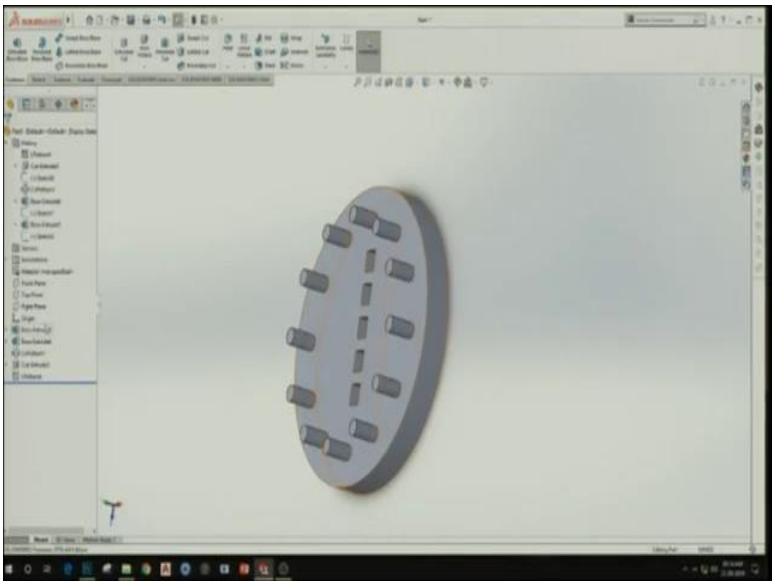

So, first we need to select what we would like to mirror. For mirror there is a command here the option there mirror. Okay for mirror let me select a different body.

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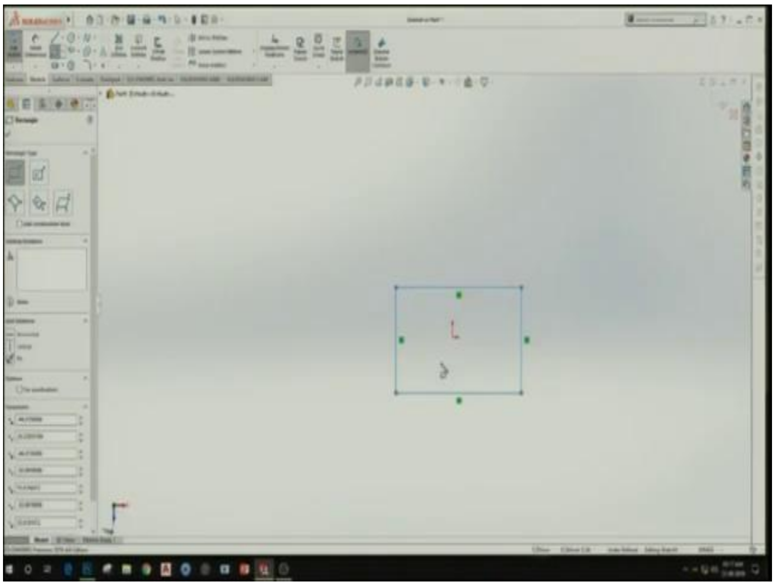

So, I will select a plain, let me select a top plane. Again from the start I am doing. Just I will select, select. I will select sketch here. So, select for the mirror command. Let me select rectangle again. I will make this rectangle okay.

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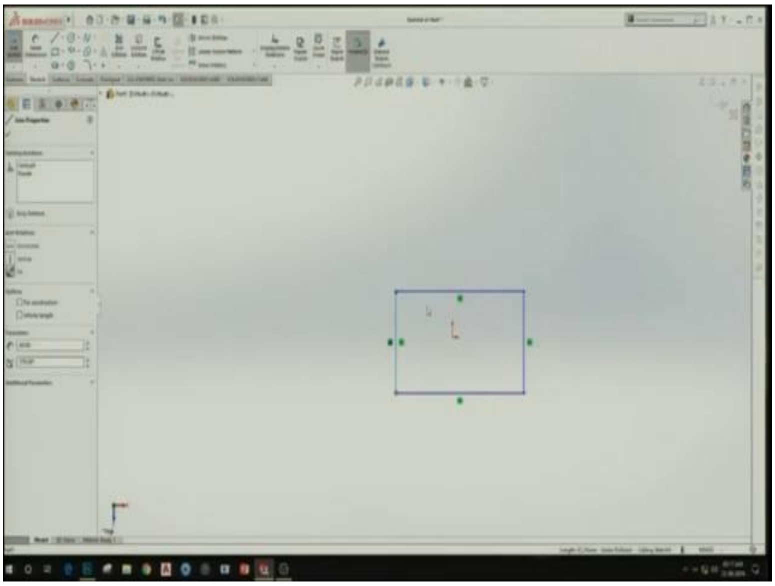

Switching the dimension as you like 100. This is again say 60, some random dimensions. Okay, there is a fixed pattern. If I fixed this now it will not change now.

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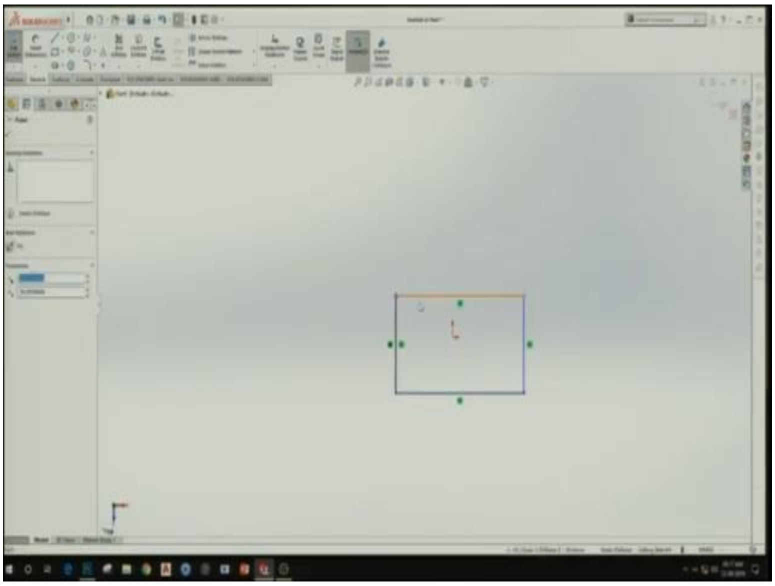

We can easily change this. If we do not want to change this we can use fixed command here okay.

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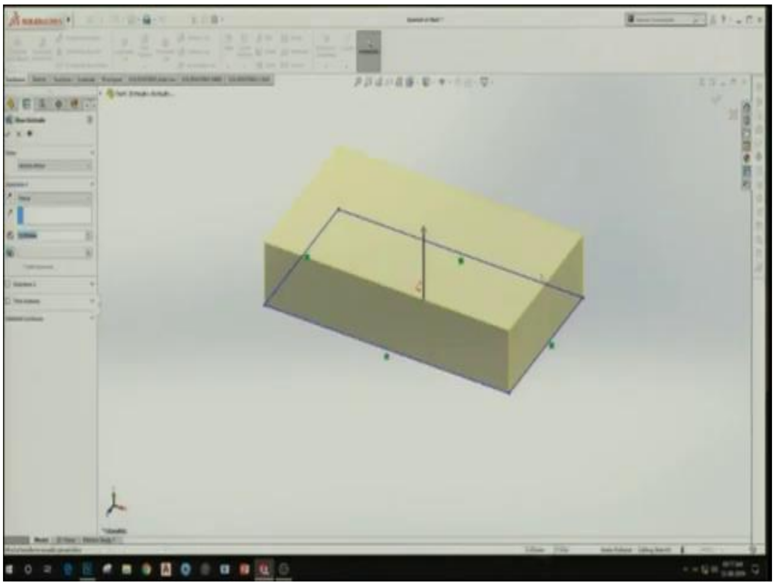

So, let me try to extrude this. So, other operation mirror I was talking about.

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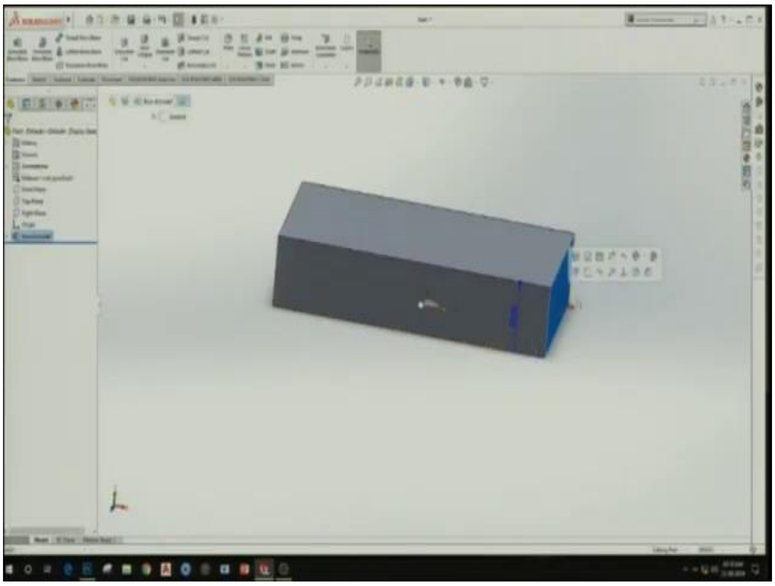

So, I have made a cut extrude for you here. And I like to just mirror this. But this is a box kind of a shape that has made. So, I will just select the whole body here. So, I will go to mirror command.

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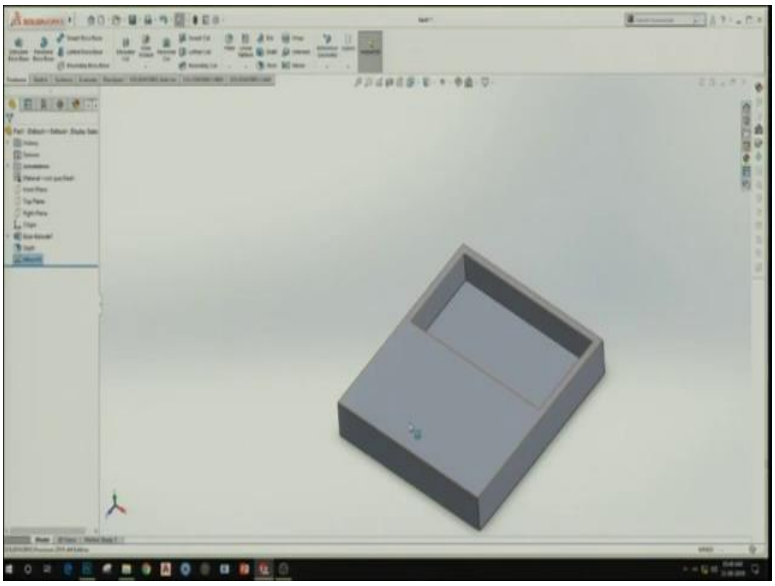

And in a mirror command, you will see mirror face and period, it is asking mirror face. If I select this face, it will mirror it. So, when I say mirror, by mirror means yeah, it has not come very perfect. Why? I will just let you know. What does mirror means?

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Mirror means your right hand is the mirror of your left hand. Your right arm is mirror of middle of your left arm. So, when I say pattern, it will just produce this, this, this, this, similar kinds of shapes. If it's a mirror, it will just mirror it. It will put a mirror here and then mirror it over. So, how does mirror work? Let us see.

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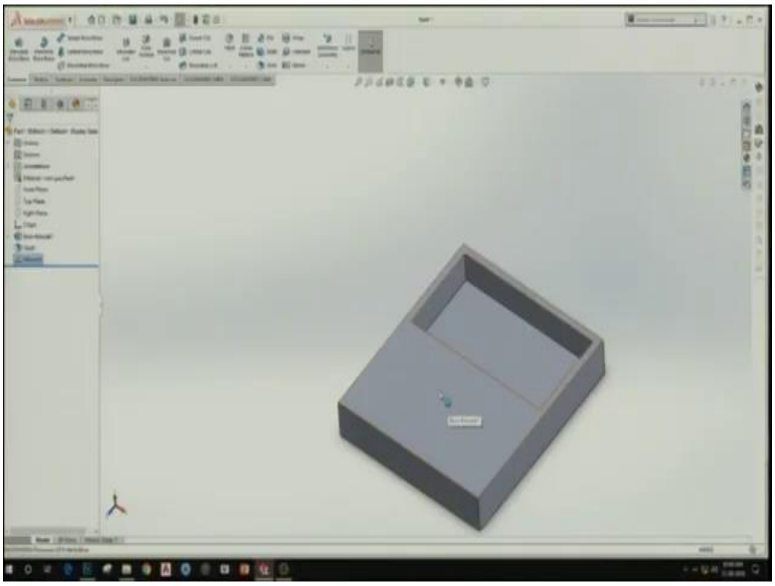

So, it did not come here properly. So, what was missed let us see.

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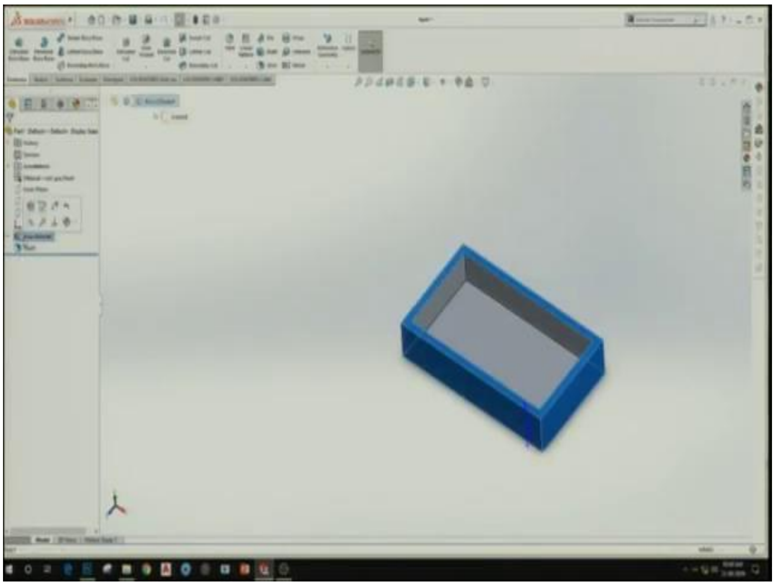

So, because the whole geometry was not selected.

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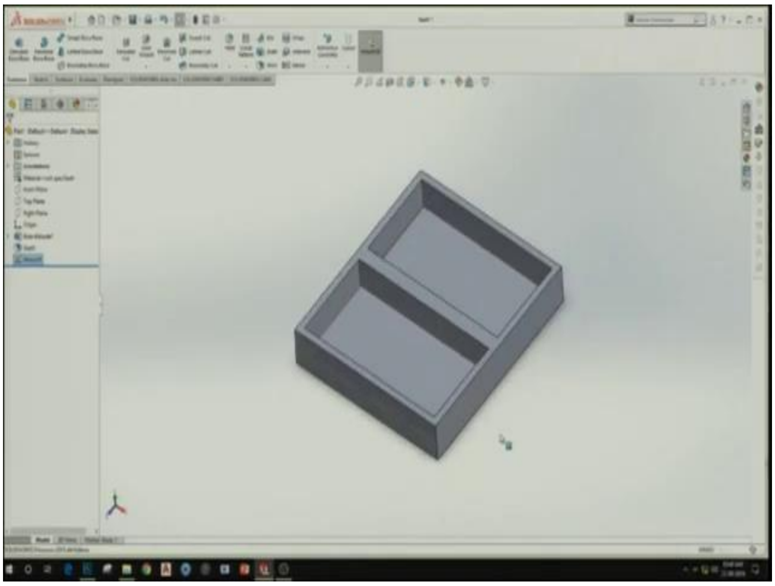

To select the whole geometry, we need to check this here geometric pattern. So, if say geometry pattern yes the whole geometric pattern is definitely it has mirrored it. So, this is mirror command. There are a number of other commands, a number of SOLIDWORKS is full of commands if I keep working on them, you can select okay.

So, many commands that you mirror on the other side, so many similar commands are there. If I keep working on them it will definitely take a whole day. Even in one, one cannot expert. The only thing that makes you expert in SOLIDWORKS is practice. So, if we practice more, you will keep learning the things try to pick some examples or some problems those are there in the industry or you develop your own problems, you can definitely keep learning how to use SOLIDWORKS software.

Okay, one thing is important that is the holes, the types of holes. So, let me select another command one of the important command. So, one operation is that I like to show at last here is different kinds of holes. There are different kinds of holes here.

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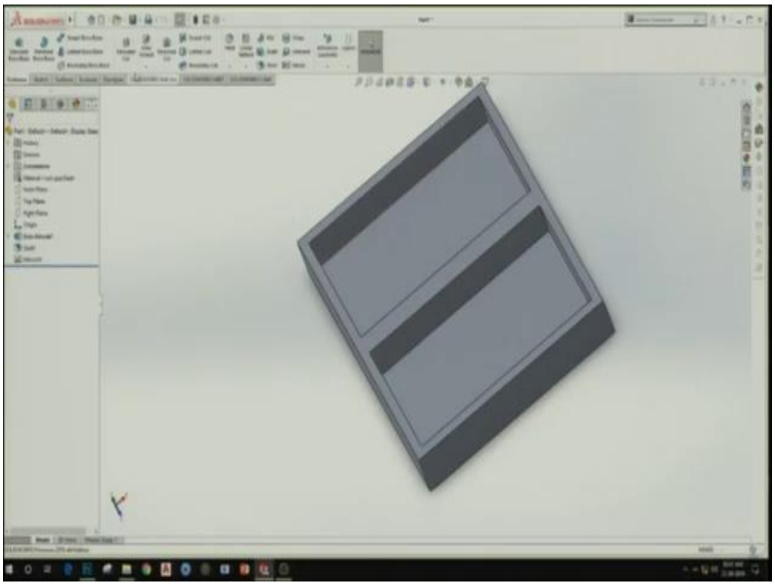

So, let me select this plane where you like to make the hole. So, this is counter borehole, counter sinkhole, this is a pin hole, blind hole, this is straight up holes. So, many holes are there. We can select a specific kind of hole and just apply it here. So, if I select apply it here, it has some dimensions. So, there are certain dimensions, certain standard dimensions are there in the library that is asking Okay, 11mm, 12mm, dia of the head of the hole and what is the angle of the counter bore that you have selected?

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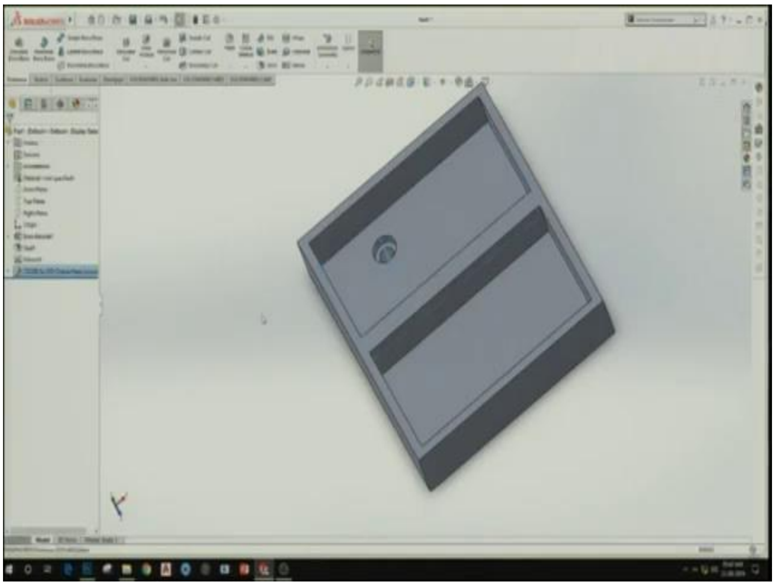

If I selected here it has made a hole that is a counter bore. The hole than a boring over it, it is generally used to but just the space for the screw or the bolts that we use over it. So, in this case it would be screw. For bolting, we have to have a spanner. So, this is also important thing. So, similarly there are certain things that can be done. So, let us see the various features at the center of the workplace.

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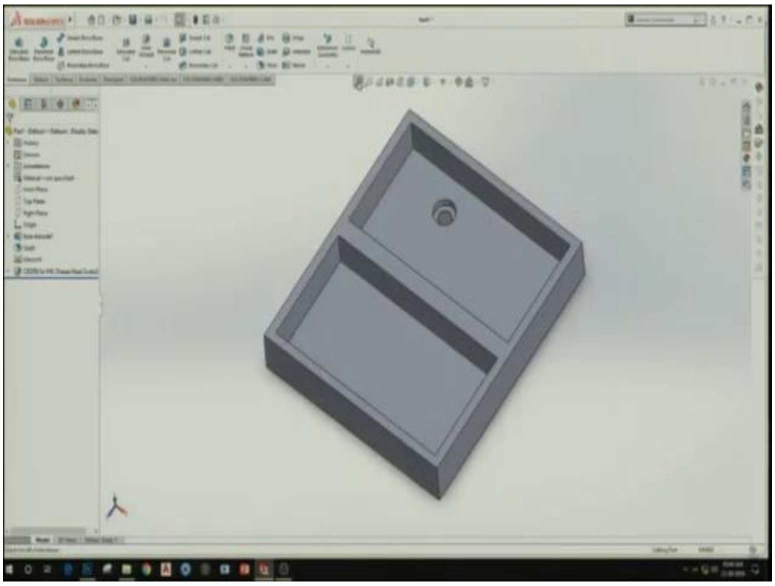

This is zoom. Zoom to fit, this just zooms in and zoom out. So, even drawing we are just immersed in the drawing we have (())(50:47)- if I just click this button it will just zoom it to fit it proper in the space, it is zoom to fit.

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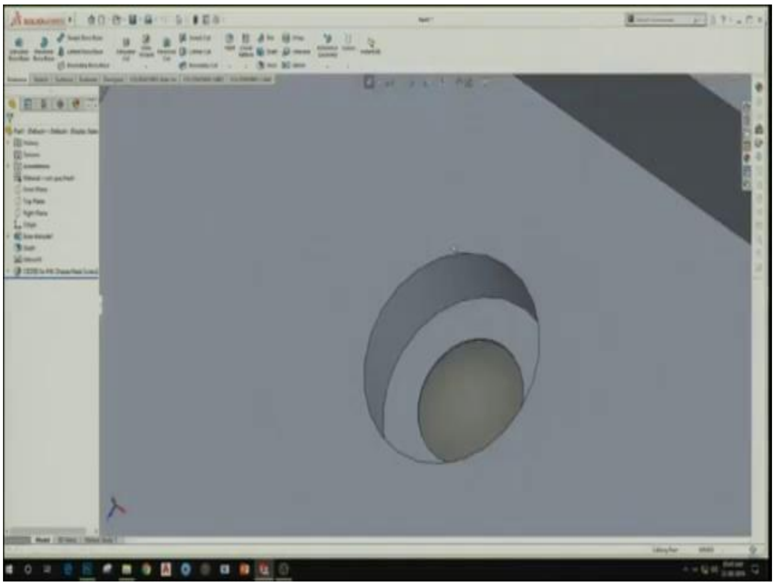

So, this is zoom the area, suppose if I select this area, it will zoom this area only.

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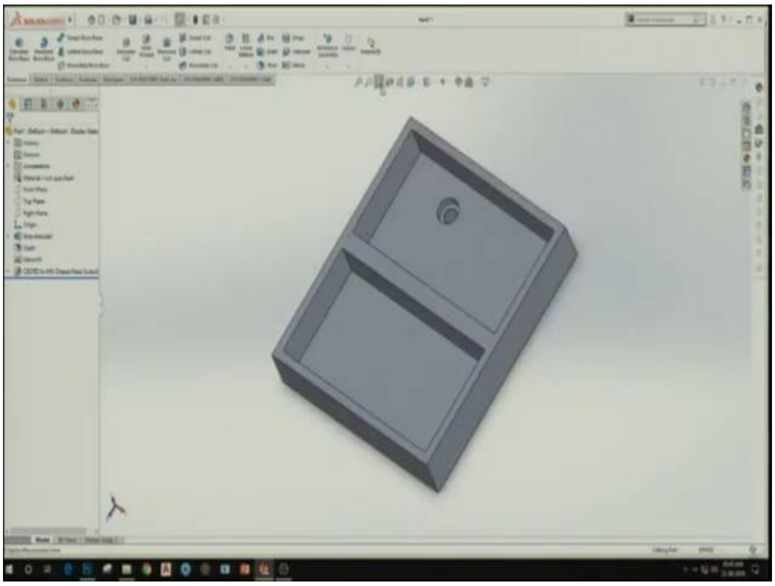

So, this is the go back to the previous view.

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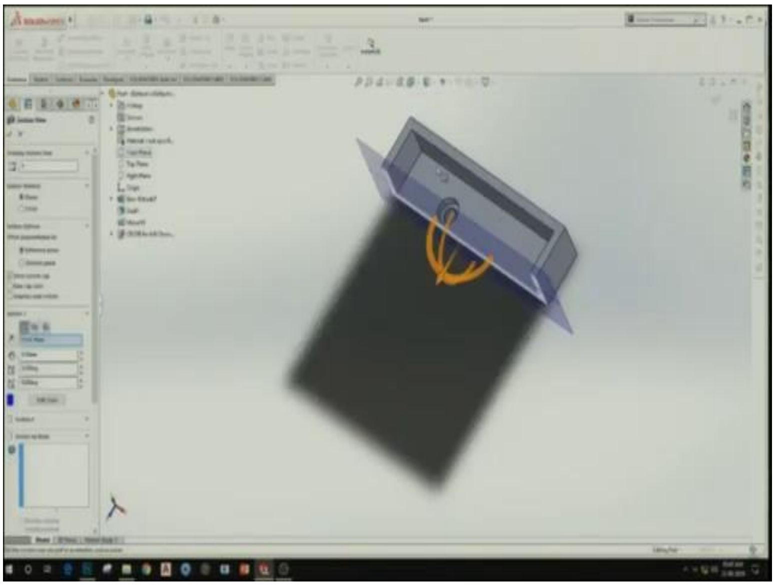

And this is Section View. Section view is the wherever we move this plane, it will just show this section of the plane. So, this is section view. So, similarly, there are different kinds of views are there. So, this is the general things those we can do in the SOLIDWORKS.

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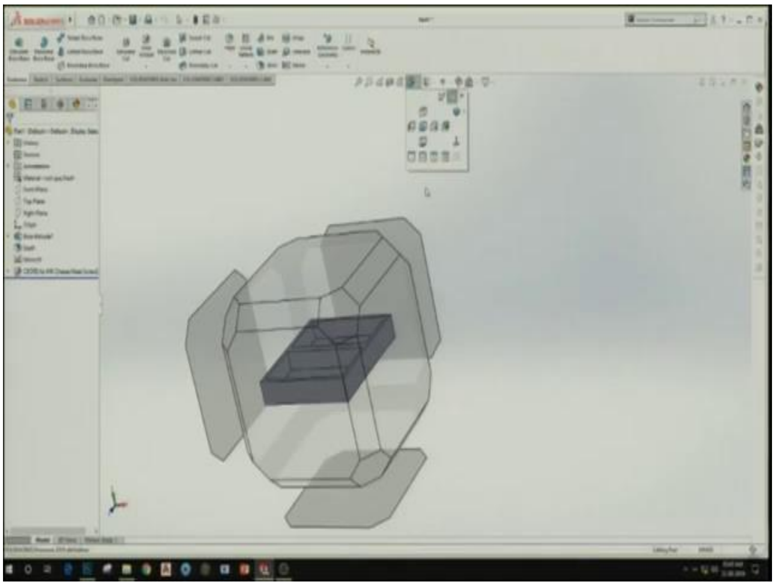

Okay, this is orientation. So, in this we can see a different orientation it is showing here, x, y and z planes.

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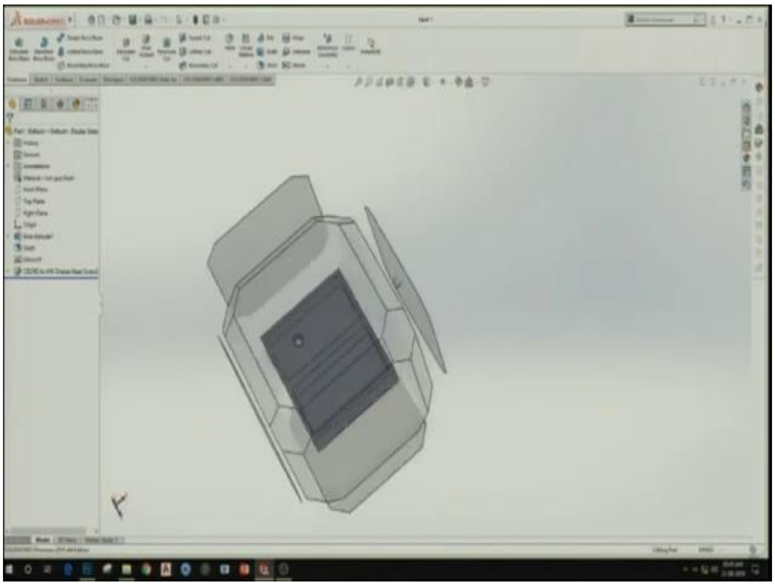

Different rotations here. So, which orientation we would like to see. So, there are certain shortcuts there is keyboard shortcuts as well there for the SOLIDWORKS. Definitely, you can also learn those shortcuts as well. In the beginning it is easy to use mouse because the mouse makes your life quite easy and you can just keep on clicking and keep on changing whatever dimensions you like. You just use the number keys on the keyboard to put a dimensions.

So, if you learn the keyboard shortcuts suppose there are certain software because people generally pick one or two software's where will they like to work in. And they will keep on mastering themselves in that software and keep on producing the models or the components or assemblies animations on them. So, if the software that you choose is SOLIDWORKS, you can also learn the shortcut keys on the keyboard because that also makes the process very quicker.

So, this was just an introductory lecture on SOLIDWORKS. I like to stop here because the complete information SOLIDWORKS will be too long and I will definitely take some more part that is the assembly and the drawing part in the SOLIDWORKS and like you to introduce to the component based and assembly based software. That introduction would happen and after that, we will go to the CAM part computer aided manufacturing.

I like to show you the computer aided manufacturing software, introduction to the very common software those are there in the market, a few of them the top six or seven software I will just discuss a few features of them. Then I will show you the Power Mill software. Power

Mill is one of the CAM software's that can help us to generate the NC code numerical control code for the CAD drawing that we have there.

And also then we will go to the laboratory to have demonstration and to see how does actually CNC machining happens and what are the various intricacies those are to be taken care of when we actually work on the machine. So, let us meet in the next lecture. Thank you.