Sensors and Actuators
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Lecture – 57
Introduction to CAD Modelling-II

If you any questions again you are free to ask me, let me again remind you there would be a certain live session as a part of this particular program. Apart from living sessions, there are there is a forum in which you can ask any technical questions all right. So, feel free to use the forum in the best possible manner. You take care and I will hand over the lab to Sayed. Bye.

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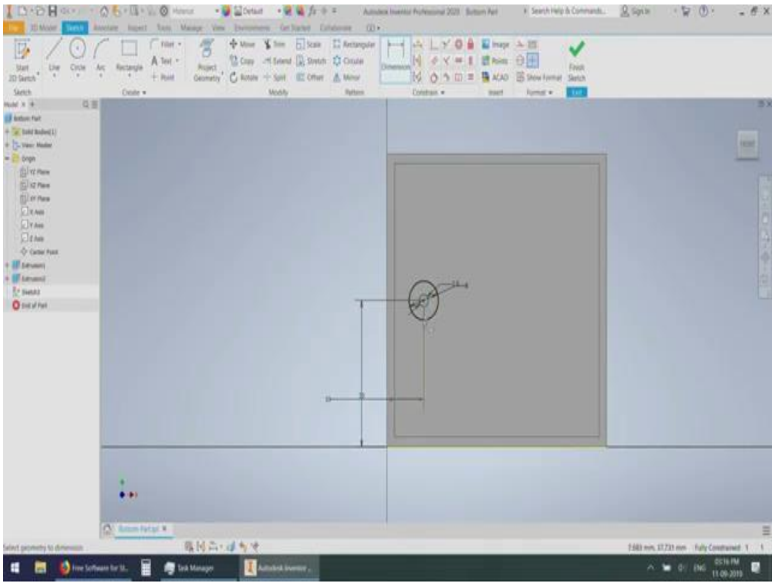

Now, we want two holes here and here. So, here the inside part inside is a hole. So, how to make two holes? We can make something solid here and here, then we can make a hole on that solid surface. So, for making a solid surface first, we have to draw a two-dimensional sketch. So, I will be drawing the sketch on the surface

Then I will make the structure as the circle. So, I will make the I want to make a hole here. So, I will draw the circle one circle here. I want a solid structure like this with this much diameter, and inside that, I need a hole here like this. Now, I want to add the dimension to the circles the hole dimension should be the bolt dimension was 3 mm. So, the hole dimension we can take it as 2.5 let us say 2.5. Now, the position of the hole such as it is ok, we will be fixing everything no three more dimensions are needed, we will be adding all those three dimensions.

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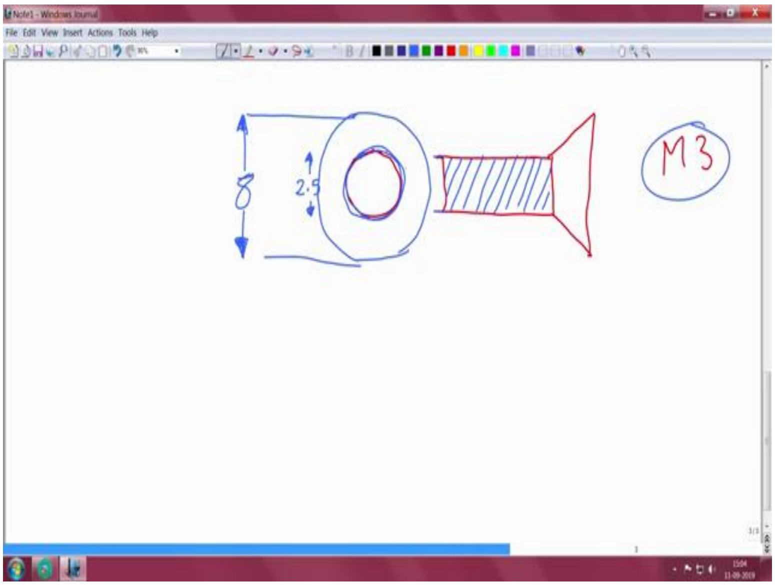

This is approximately 3 mm, it will be for a bolt this is M 3 bolts. So, this dimension will be less than 3 mm. And this is supposed this is the hole we made using a 3D printer. If we give; if we give a 3 mm diameter in CAD software and made a 3D printing, and after measuring this dimension it will be less than 3 mm that will there will be a strain gauge and that will be anyway less than 3 mm.

So, therefore this screw, there will be threads, if you want to directly screw this bolt, we need this dimension less than this dimension. So, we can give 0.5 mm less than the M 3 value. So, if we put the 2.5 mm hole and then we can sense it is a plastic material we can directly screw this while screwing this bolt we will make a thread in this hole. So, it will be a tight fit.

So, for 3 mm bolt, we will make a 2.5 mm mm hole and the solid surface should be around two times this value or more. So, let say here we can give 7 mm or 8 mm we can give, its diameter the outer solid body diameter cylindrical diameter can be 8 mm. It is recommended to go more than 2 times of this hole.

So, I will keep 8 mm here. Here the hole dimension will be 2.5 mm and the solid dimension will be 8 mm. And in this enclosure, where we need that hole that is the questing you know we can go to the old drawing now.

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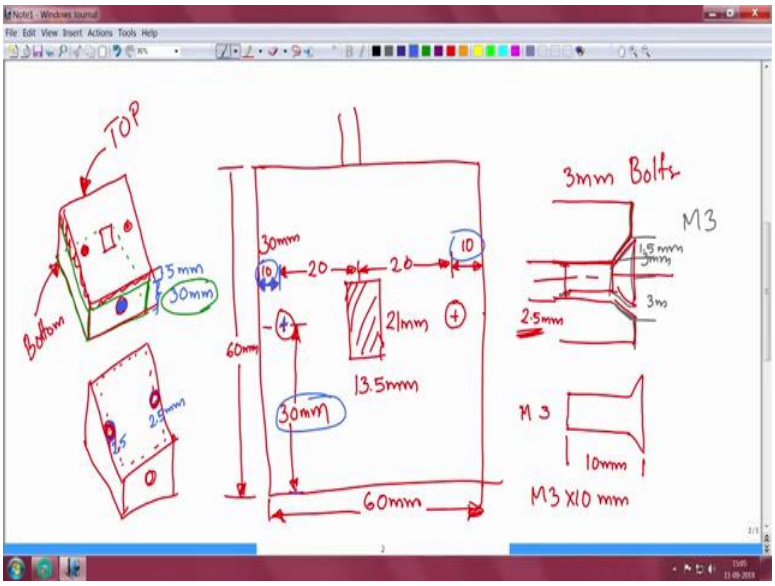

We can switch the screen nowhere see here from the center to this edge that is 8 mm we have fixed that while designing. This is 10 mm and this is also 10 mm and from the bottom line to the centerline of the circle will be 30 mm. So, we can enter this 10 mm dimension and this 30 mm dimension in the CAD software. I will click the dimension tool from the center to this edge that is 10 mm. And from here to this bottom line that is 30 mm.

So, this dimension is from the center to here this line this outer line that is 10 mm; and from the center point to this bottom line that is 30 mm. Now, we can finish the sketch. We need one more hole here also. So, I will click first I have to make a solid structure here, I will click finish sketch, and then I will go to 3D model I will click extrude.

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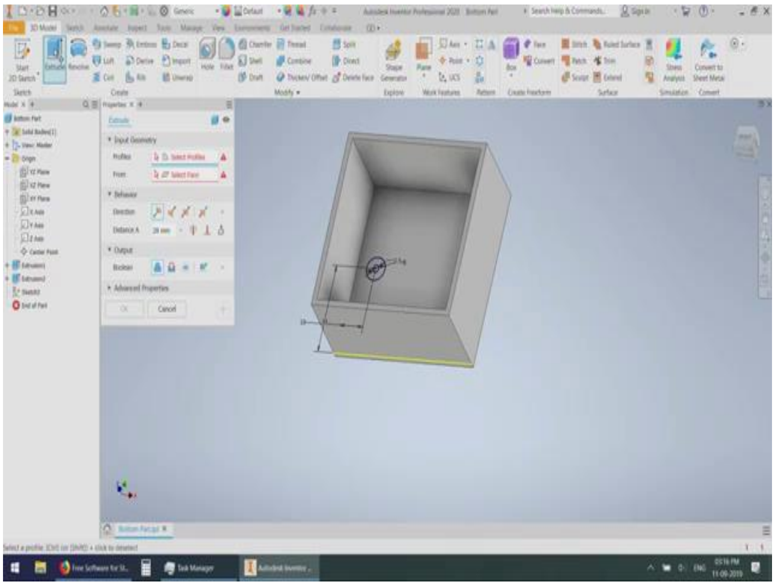

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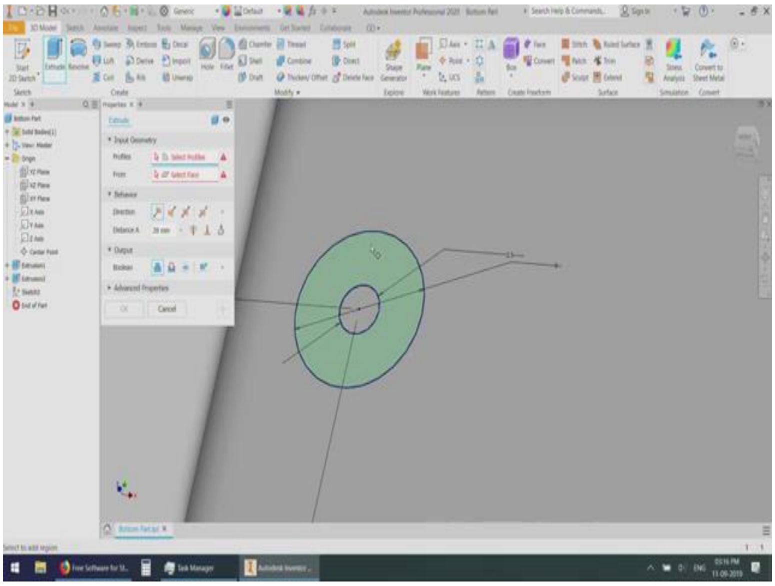

So, for extruding, I want a hole here and a solid body here. So, I will click in a space between this circle and this circle.

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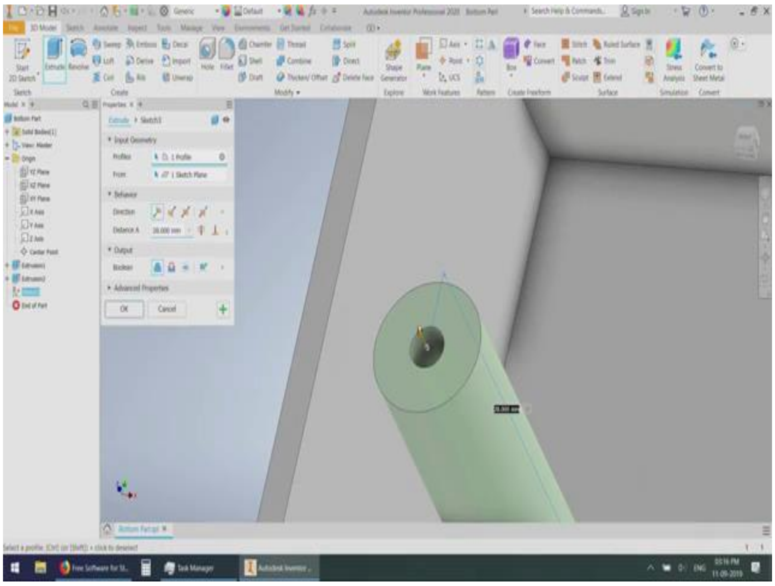

So, here see it came as a hole here in the inside there is a hole, and the outside there is a solid body. So, the height is 28 mm this height from here to here also 28 mm. So, this will be fine. I will click.

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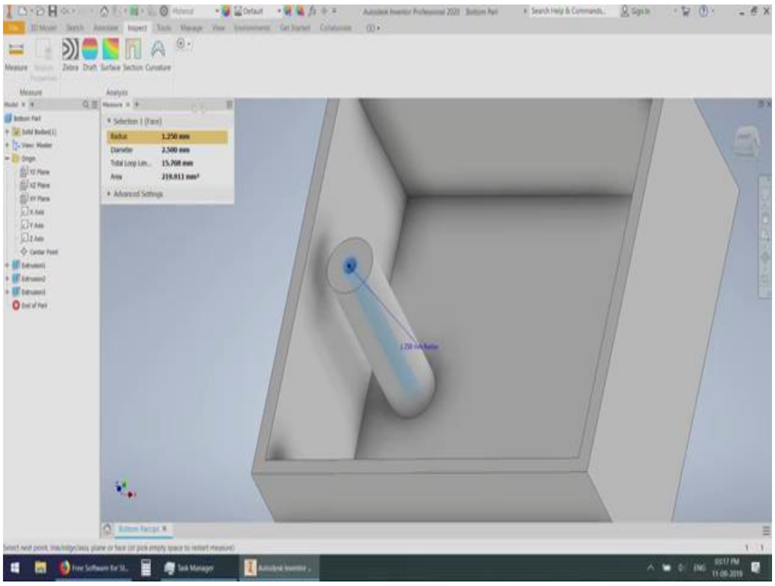

So, here see if you rotate there is a solid body with a hole, the hole dimension is 2.5 mm. We can measure by going inspect tool and measure the diameter is 2.5 mm, and the radius is 1.25 mm.

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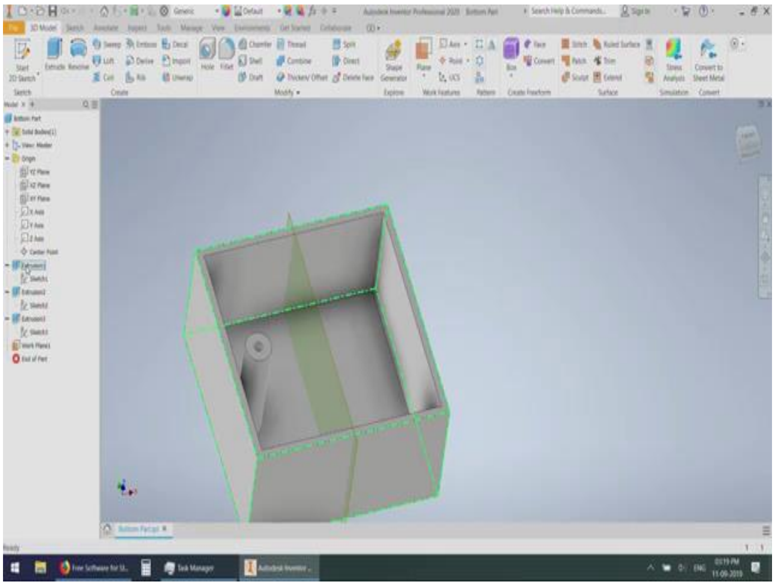

Now, as I said I want to make a plane in the center of this. So, for making that I will go to the 3D model.

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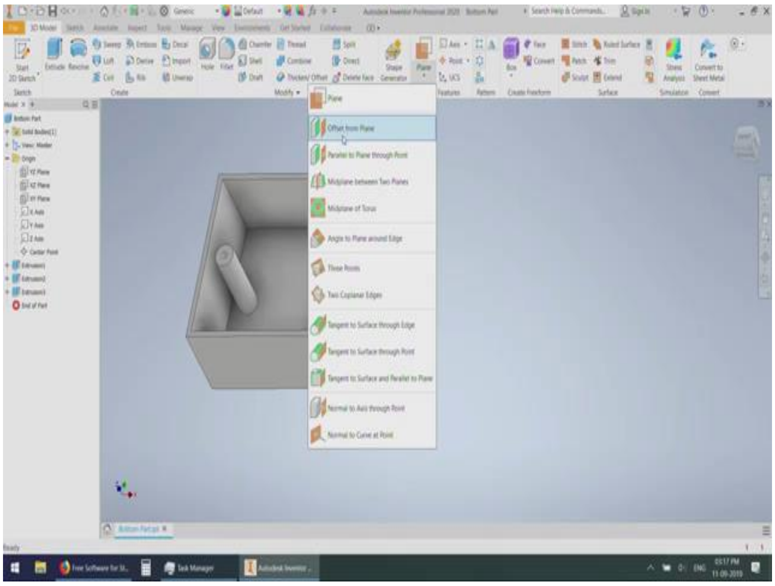

And I will click this downside arrow. And here we can see there are a lot of options to make planes. The first option is offset from a plane, that means, from one surface we need another plane at a particular distance, we can select we can choose this option. I will be choosing this one, this midplane between two planes that will be applicable in my case. So, I will select this plane, and then I will select this plane.

So, here we can see there is a new plane. Here on this left side, we can see my object name is a bottom part, I have saved the file. This is part three this is the origin and the planes we have this are the axis’s we have and this is the center point here let us see. This is the first function I have used that is to extrude. This was the sketch I have made with 60 mm length and 60 mm height. This is the next function I have used extrusion, extrusion. And the sketch was this here we can see in blue color.

Then the square length and height were 56 mm, and the dimension between this line, ah, the outer surface and the line here was 2 mm, similarly, 2 mm to make a 2 mm wall thickness. And next is the extrusion of a solid body and a hole together. This was the sketch we have made for that. So, this is a very useful option in the CAD software (Refer Time: 12:51). Most of the in all the software, there would be an option like there will be something like this. This is called a tree. So, this is the server part and this is the (Refer Time: 13:03). So, here the plane we have created.

And if we want to suppose if we want to change this dimension of the box, or I want to place this hole somewhere here like 2 mm; 2 mm, I want to change from here to here, then I can directly go to this tree. And I can right-click here and click edit sketch, and then I can change this value. And similarly, the updated object will come. So, now we will be mirroring this feature with respect to this plane I have created. So, for mirroring in this 3D model, I will select here the in a pattern, there is a mirror.

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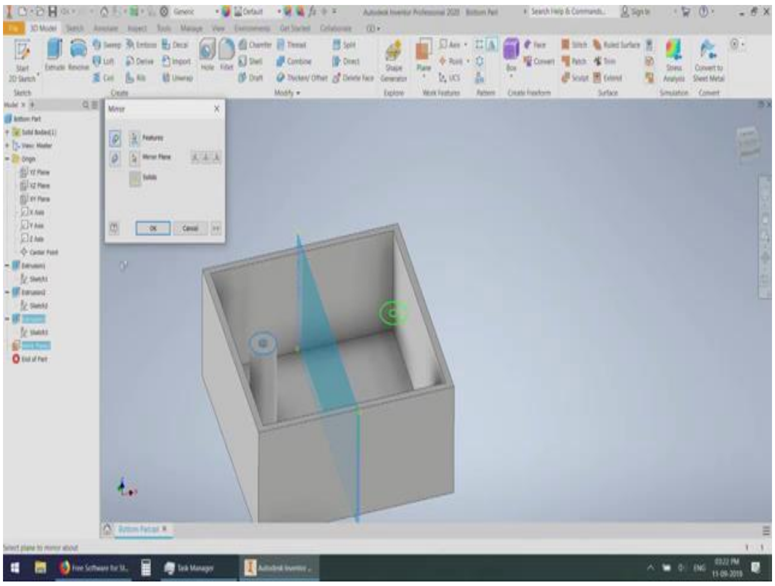

So, when I click the mirror feature, the mirror option first I have to click the feature I want to mirror. So, I will click this feature here. I can select the feature here or from the tree. Now, I will select from the tree this is the feature I want to mirror. And now I will click here the mirror plane. I will click here; I will click here or I can click here to select the mirror plane, I want to mirror this feature to mirror (Refer Time: 15:10).

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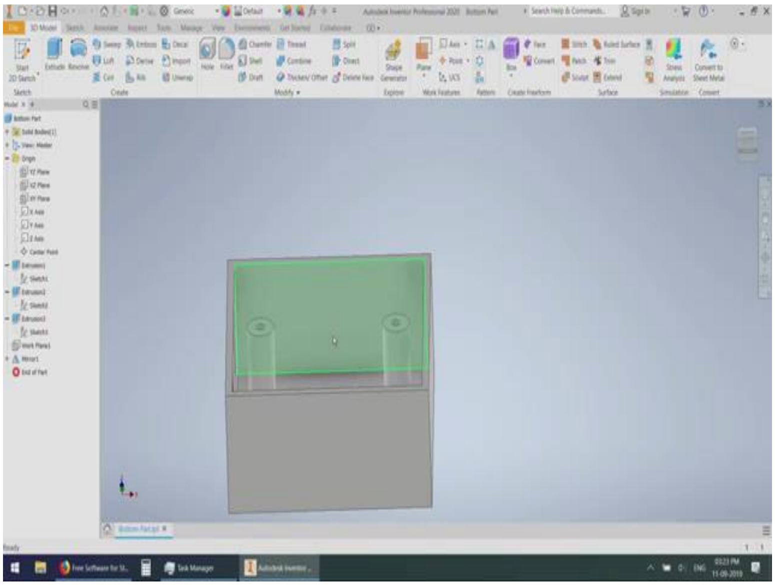

Now, when I click here and press ok it is the new feature is created, the new mirrored feature is created. Now, I do not need this plane. I cannot delete it because I made some changes to the 3D object using this plane. I can decrease the visibility of this plane. Here on the plane click the right button on the mouse. And in the visibility, this by clicking here will vanish. Now, this is the bottom part I have created. And one more thing is we want to make a hole here as we discussed. So, for making that hole, I want to make a sketch here, then I want to subtract some material from this object.

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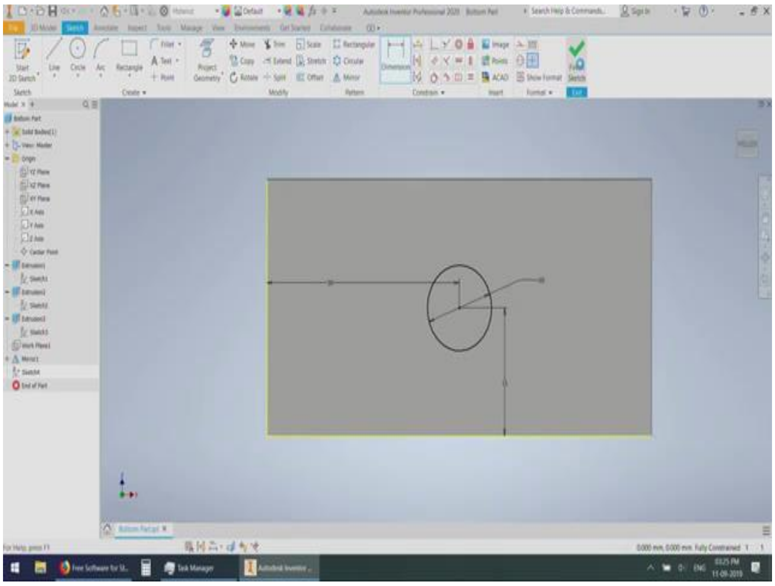

So, I will click on create a sketch and then I will click on the circle. And now we have to give dimensions. The circle dimension maybe a 10 mm, 1 centimeter. I want to take some wires from that switch. So, I need a hole with a 10 mm diameter. Now, I will enter 10 mm here.

And now here if you see we need two more dimensions to constrain this sketch. I will click on the dimension again, and from here to here now it is 3.633, I will enter 30 mm, this total length was 60 mm. So, I want to place this hole in the center, so I will enter 30 mm. From the center of the hole to the bottom line currently it is 19 mm; the height of this box this bottom part is 30 mm. So, I will enter half of that that is 15 mm. Now, the color of the circle has changed to black. Now, here we can see it is fully constrained.

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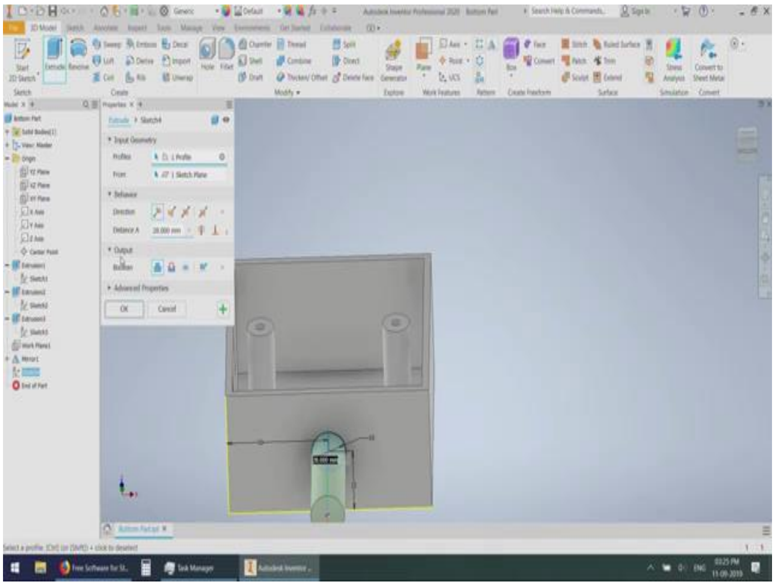

So, I will click the finished sketch. and I will use the same extrude feature; I in the output I will change to cut.

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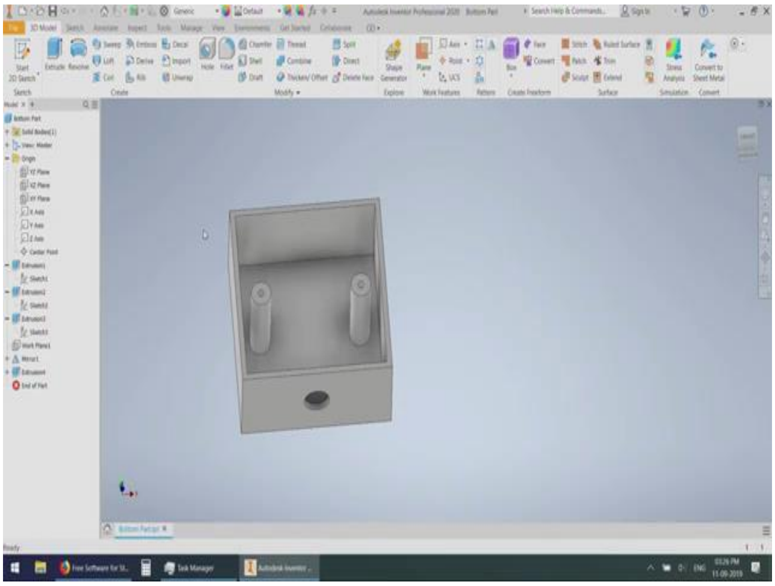

I will click ok. Now, you can see there is a hole. In the enclosure bottom part, there is one hole to take the wires out, there are two holes to insert the screws. This is one part of our enclosure.

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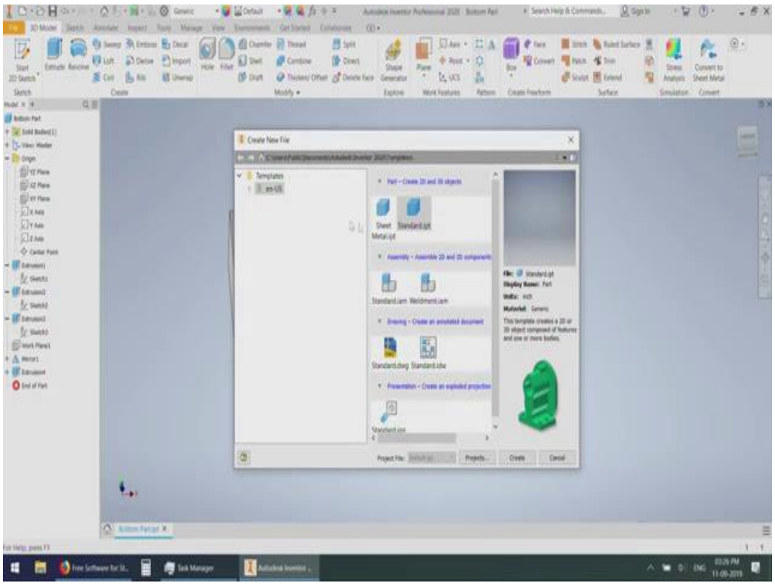

So, now I will make the top part also. For making that we have to create another part. So, I will click the same new standard part file, standard IPT creates is the new path from the tools.

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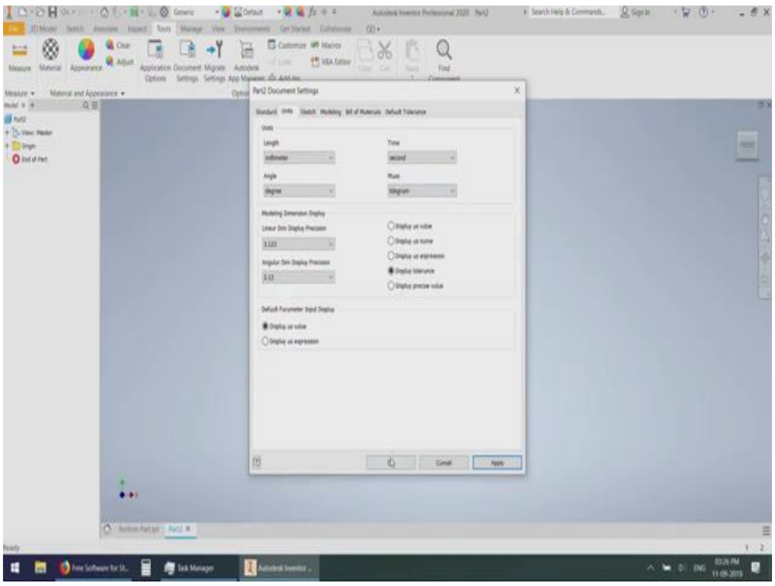

In the document settings, I will change the unit to millimeter and kilogram oh click ok.

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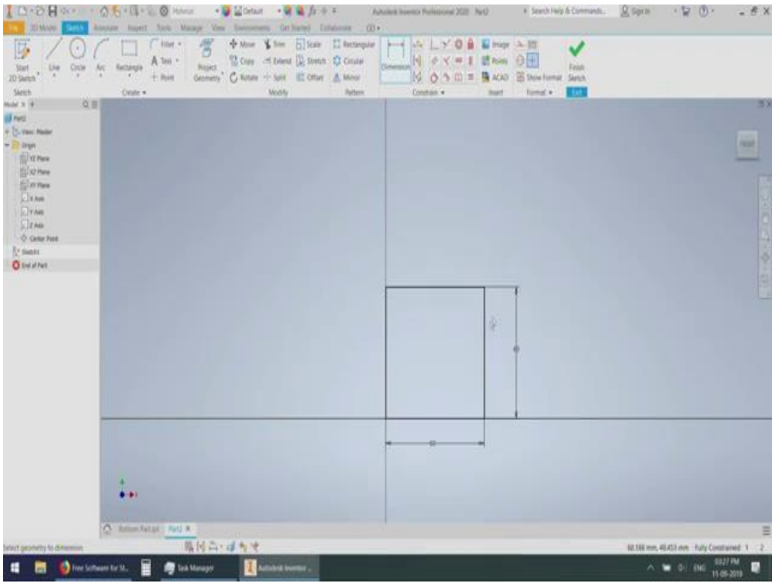

Then I will go to the origin of the x y plane, I will create a sketch. I will be creating a rectangle. And it will have a provision for two bolts, and there will be a rectangular hole for placing the switch. Now, the dimension I will enter 60 mm and 60 mm as we discussed. Now, I will finish the sketch.

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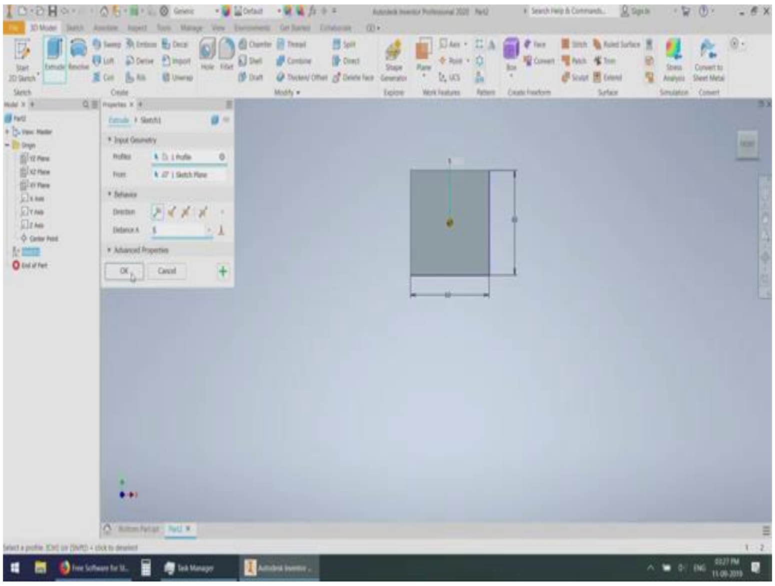

And I will extrude as we discuss the thickness of the top part was 5 mm, I will enter 5 mm here.

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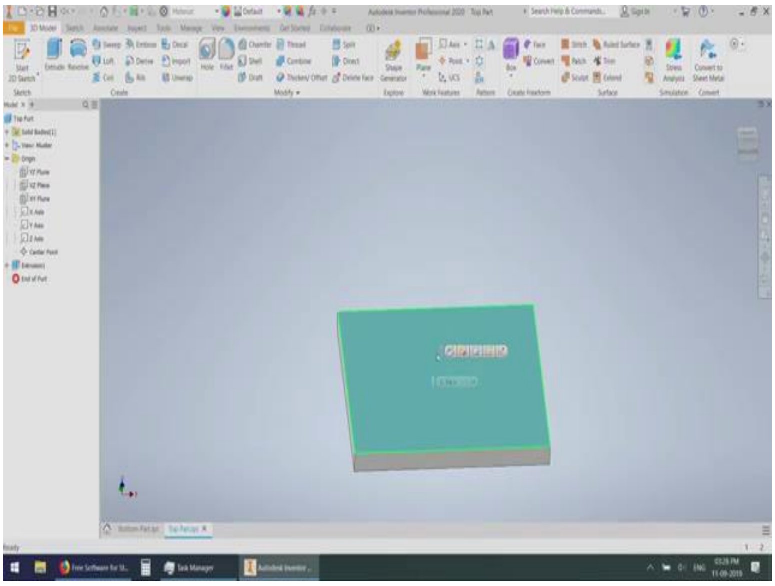

See this is the solid part we have created.

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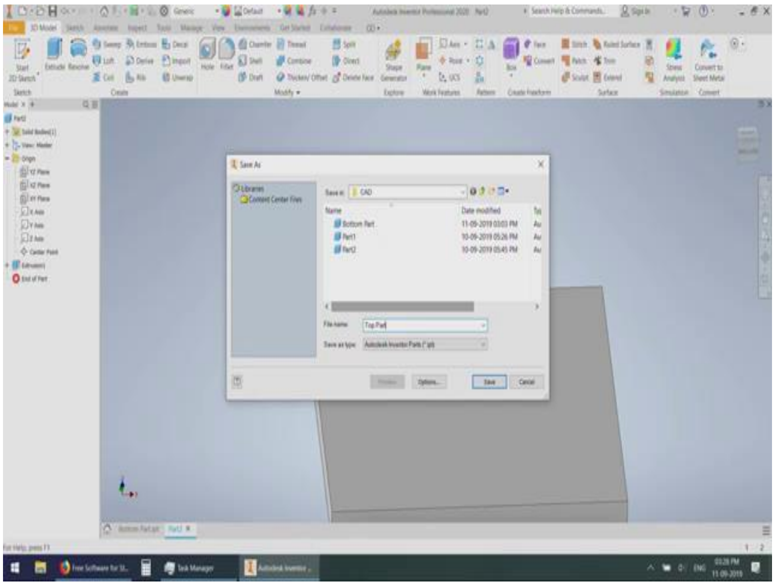

And I will save this file and give the name top part. I want to make a rectangular hole in the center, and two holes in one on the left side, and another on the right side. So, for making that, I want to make a sketch on the surface. I will click on this to create a surface. If you just click with the left button of the mouse, these options will show.

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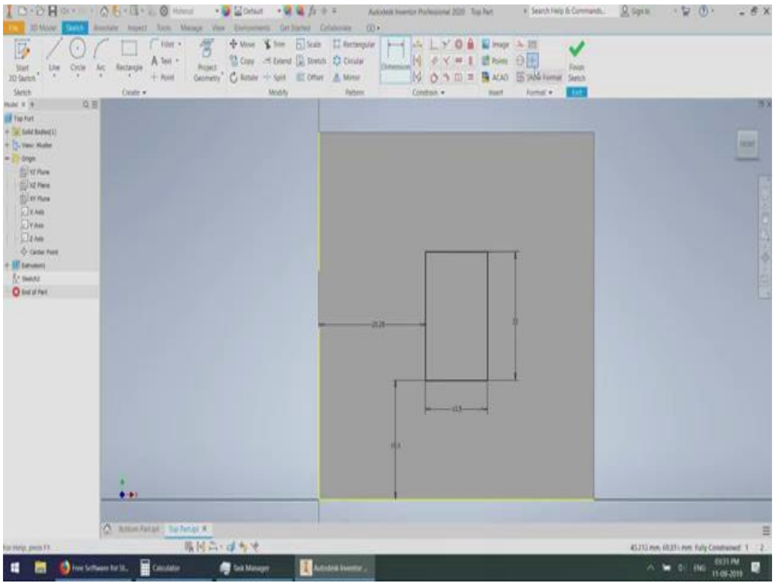

Here you can just click the create sketch and zoom in or zoom out appropriately. Now, I will create a rectangle here. As we discussed the width of the rectangle was 13.5 mm, and the height of the rectangle was 21 mm. See this rectangle is not in the center. So, we have to move it to the center. For moving we can use these dimensions tool, from this side to this side we can calculate the total length was 60 mm and this length is 13.5 mm.

So, 60 minus 13.5 by 2 23.25, from here to this line that should be 23.25; from here to this line that should be 23.25. Horizontally, now it is in the center now we have to align the vertical part also. So, the height of the square is 60 mm. Here 60 minus 21 by 2 that is 19.5. From this line to this outer line can be 19.5 or we can fix the dimension between this line and this line, from here to here that is 19.5. I will click on the finished sketch.

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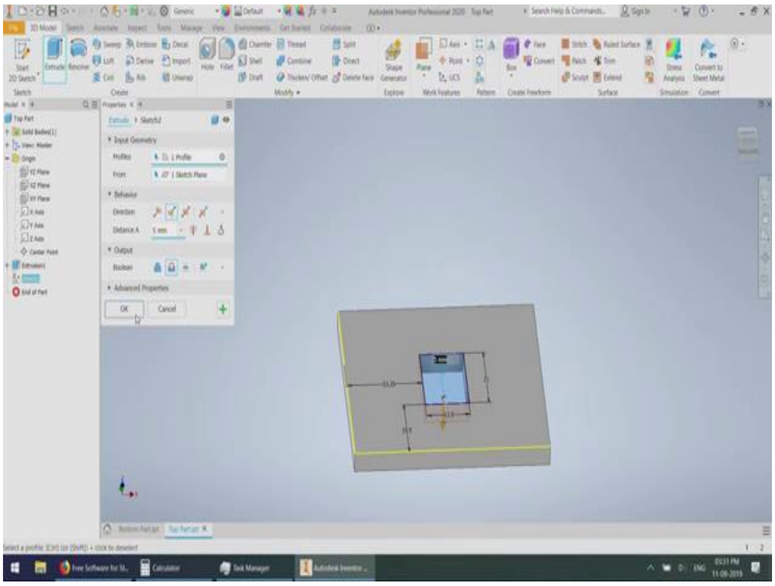

Then I will use the same extrude feature. And here I want to select this cut in the output.

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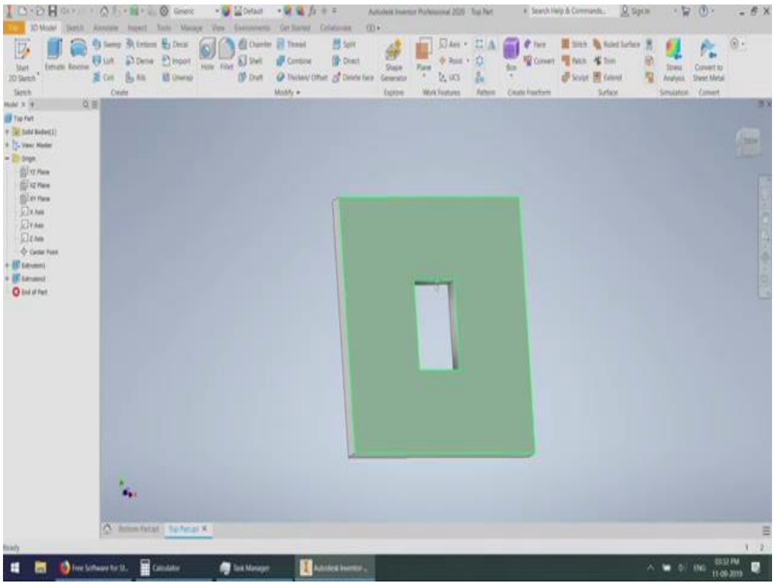

Now, see we have the top part with a hole for placing the switch now I need one hole here and one hole here. So, as previous as we did previously, I will create one hole here on the left side, then I will create one hole here on the right side by mirroring this hole with a plane.

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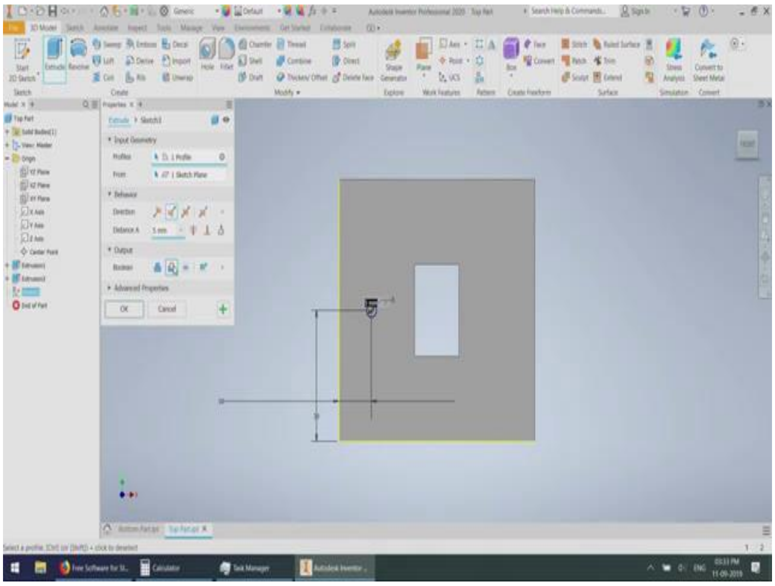

I will first create a sketch here. In the dimension of hole, I will choose 3 mm; the dimension of the hole in the bottom part was 2.5 mm. Now, here I am choosing 3 mm. The reason is we will be screwing a bolt through this hole. So, the bolt does not have to; do not have to bind to this part, it should catch the bottom part.

So, here from this hole to the bolt, there should be some gap that is why I am giving the 3 mm hole; more than 3 mm will be too large for the 3 mm bolt. So, I will give just 3 mm. And from the hole center to this line was as we discussed that is 10 mm, and from the center to the bottom line that is 30 mm. I will enter those two values. And I will click this finished sketch I will use the extrude feature, and here is the output I will select the cut option and I will click ok.

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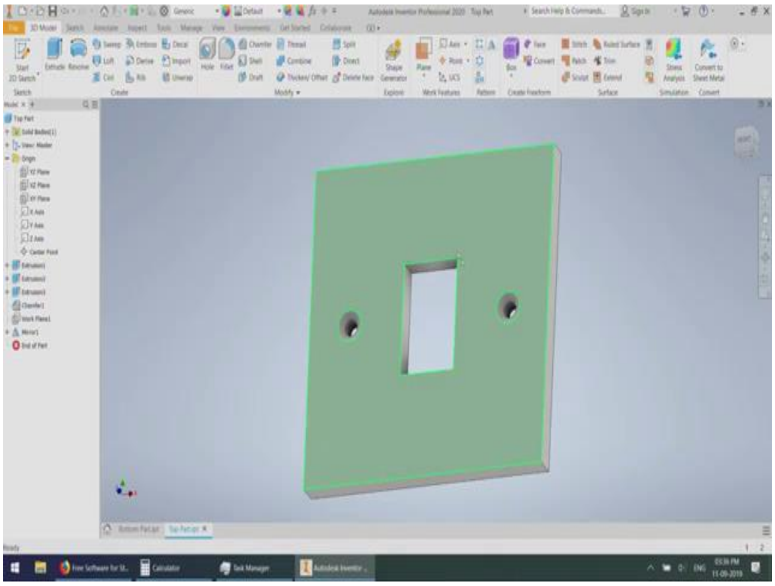

Now, you can see I have created a hole on the left side. I will be using a countersunk bolt.

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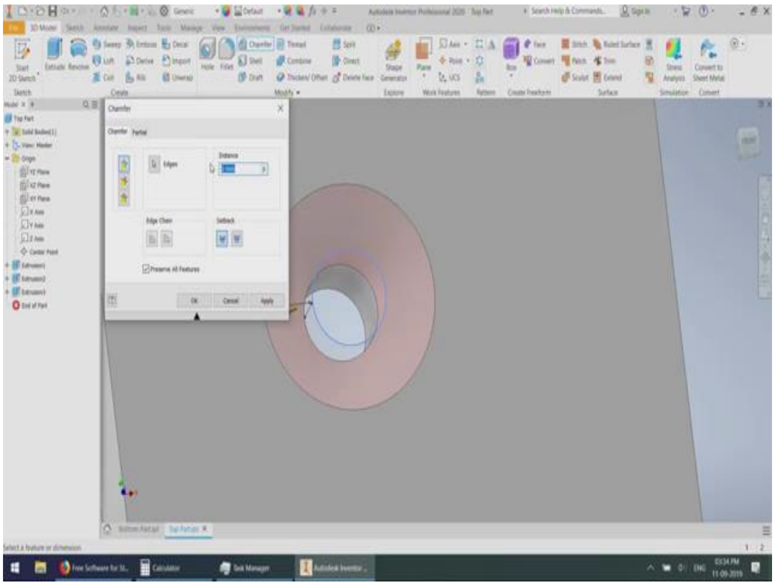

So, I need a structure here for inserting the screw inserting the bolt from this surface. So, here I will use the camphor tool. And I will select this is the circle I want to make camphor from. So, the distance will be normally the distance will be half of the size of the bolt. So, we were we will we are using 3 mm bolt, I will enter 1.5 mm value here. So, see we have created a hole for a countersunk bolt. Now, I will mirror this hole and this camphor about a mirror in the center. So, I will create a mirror midplane between both; this surface and this surface. So, this is the plane I have created.

So, for mirroring, I will use the mirror feature. And I will select I can select this hole from here and the camphor here, or I can choose from here in this tree. I selected at the extrusion and the camphor. I will select the mirror plane to select the mirror plane, and I will select it. Here we have this word a top for with one hole for the switch, and two-hole for two bolts. I do not need this plane anymore. So, I will disable the visibility of the plane here right-click on the work plane, the plane I have created. And I will select this visibility to disable the visibility.

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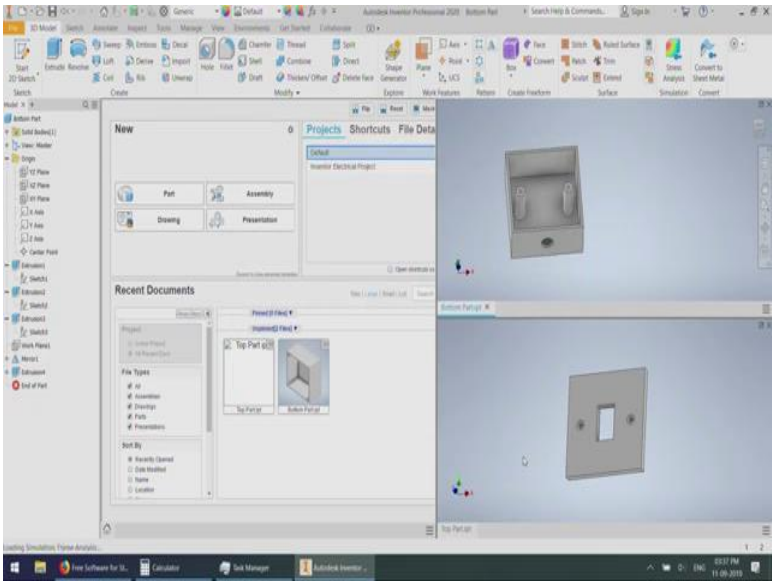

This is the top part I have created. I will save this file. And this was the bottom part I have created. Now, I will make an assembly and make this as a total enclosure for the switch.

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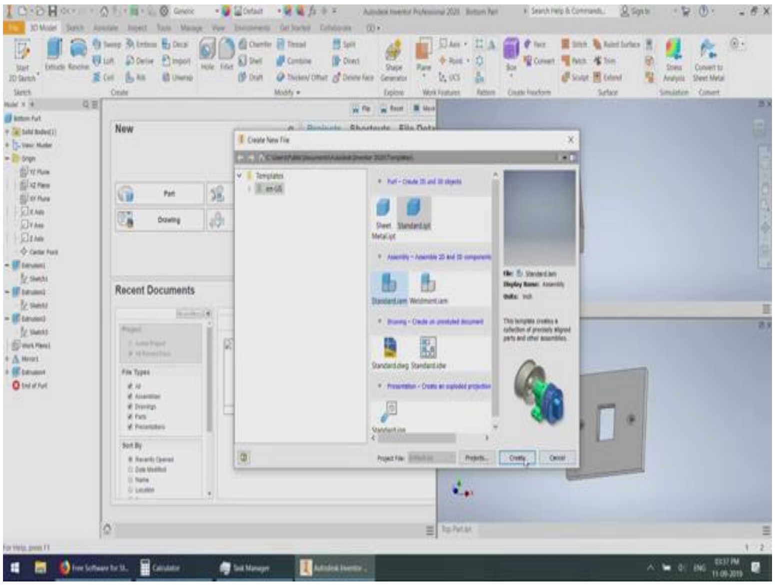

Here in the new, I will from this new I will select a standard I A M a standard part.

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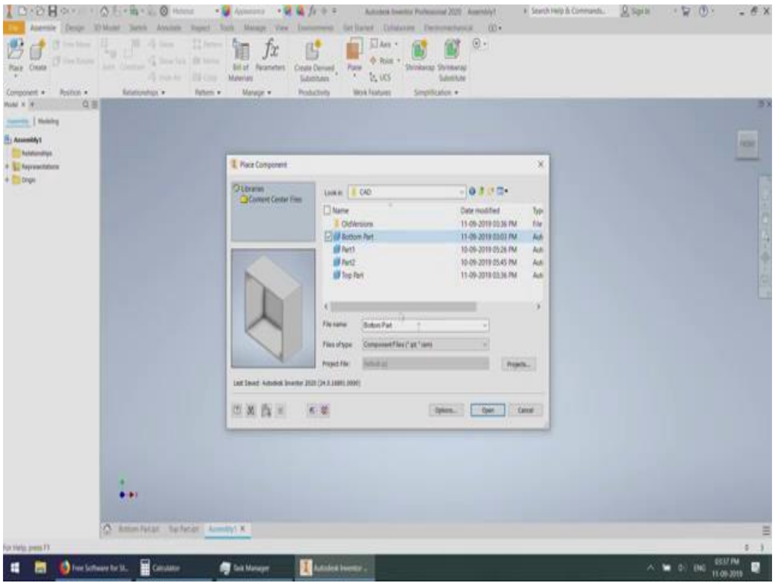

I will click on create this is the new assembly file. I will maximize this document window. And nowhere this is the assembly file. I want to place the bottom part and the top part of this value. For that I will use this place with this place button, this is the bottom part.

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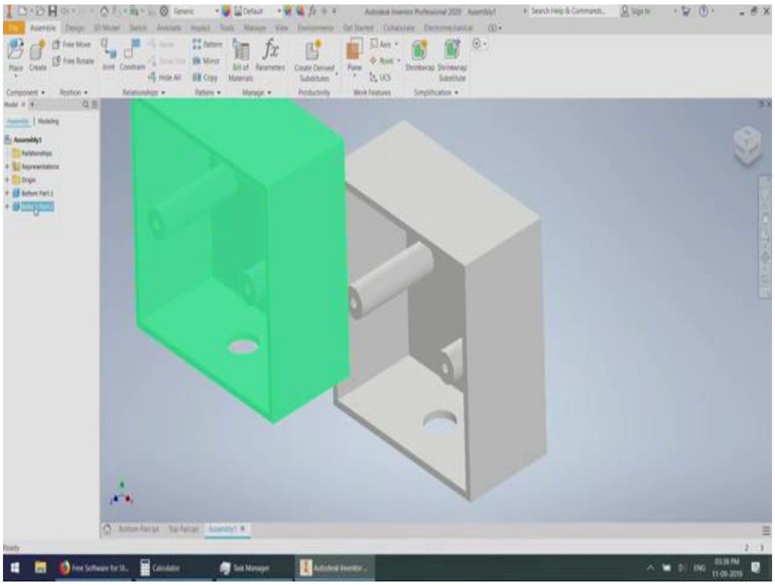

This is the bottom part I have created accidentally make two bottom parts here. I just click escape, and I click this bottom part two in this tree and press delete.

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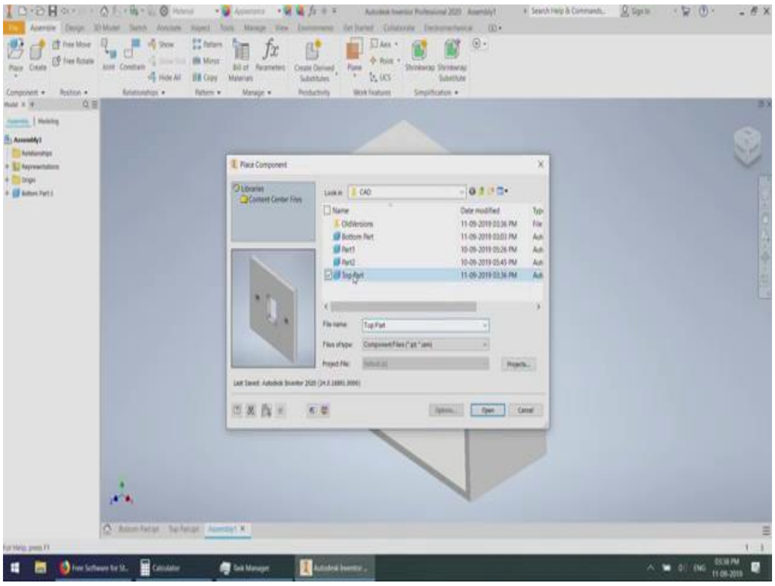

I have to place the top part here, I click the top part and I will click open.

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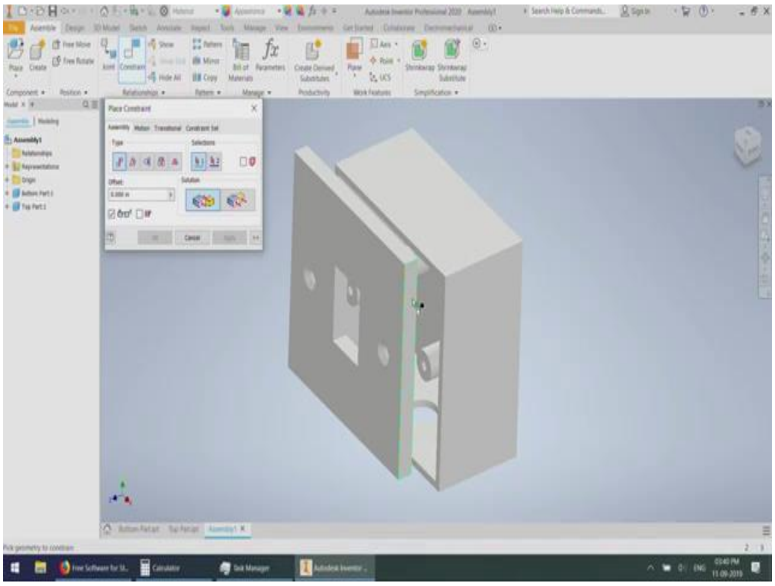

So, let me place it here, and I will press escape, here see this is the enclosure we will be making, and I will place the top part on the bottom part at the exact place. For that, as we use in this sketch, we have some options for constraining the parts constraining the dimension between parts.

So, to place these two objects to the bottom part and top place in place, practically this surface, this surface, and this surface should be in a plane for the enclosure. And this surface and this surface should be in the same plane, and then this surface and this surface should be in touch with each other.

So, I will click on the constraint. Here there is a flush constraint, the flush assembly and here there is a mating assembly. I will use this flush assembly. Here it is showing two parts place near each other. This is how we need this and this surface and this and this surface. So, I will just click here on the surface, then I will click on the surface.

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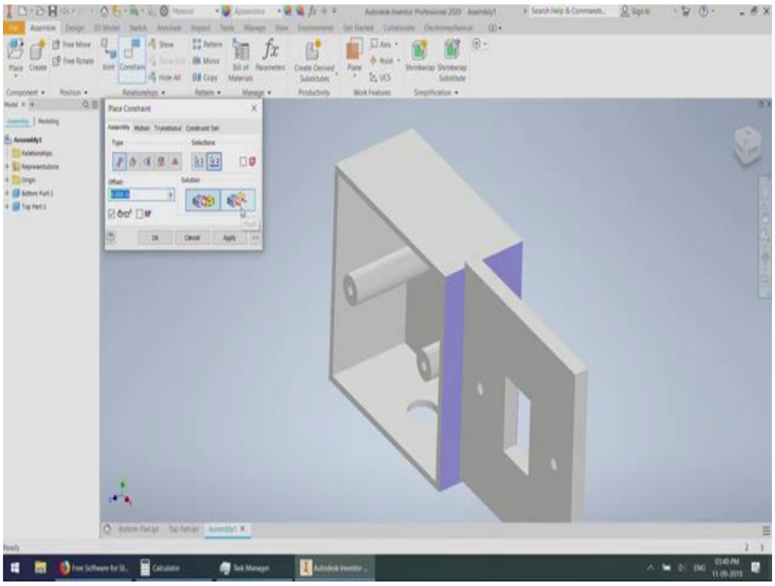

Now, I jump to the mate option I will click here in the flush option. Now, this is the way I wanted, but the top part came inside the bottom part, it is ok. I can move. Now, I want to constrain this surface and this surface in the same flush joint, I click ok. Now, I want this small surface and this surface to be in touch with each other.

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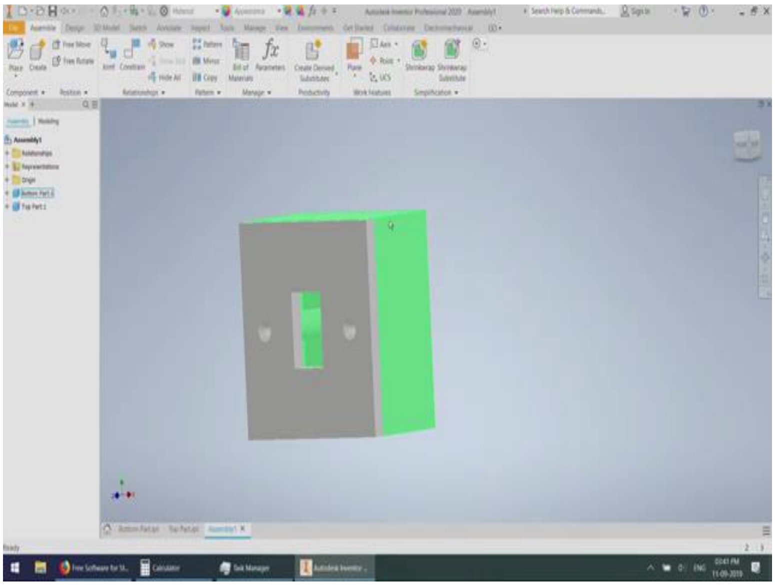

I will click on the constraint here; it is in the mate option I will click on the surface, and then found the surface; I click ok. Here see how our enclosure is ready. So, this is just looking like a square enclosure. So, for good aesthetics, we can give a fillet to these edges that mean, we can make these edges a little bit curvy. For that, before doing that I will save this file.

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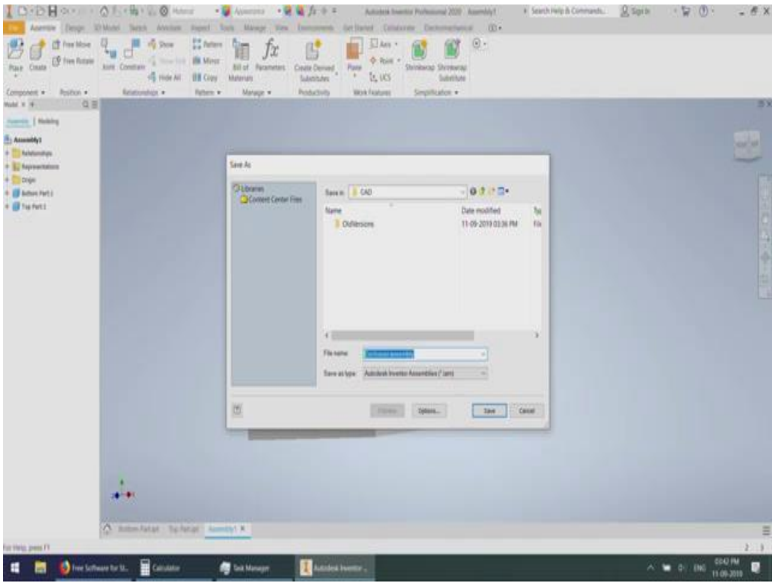

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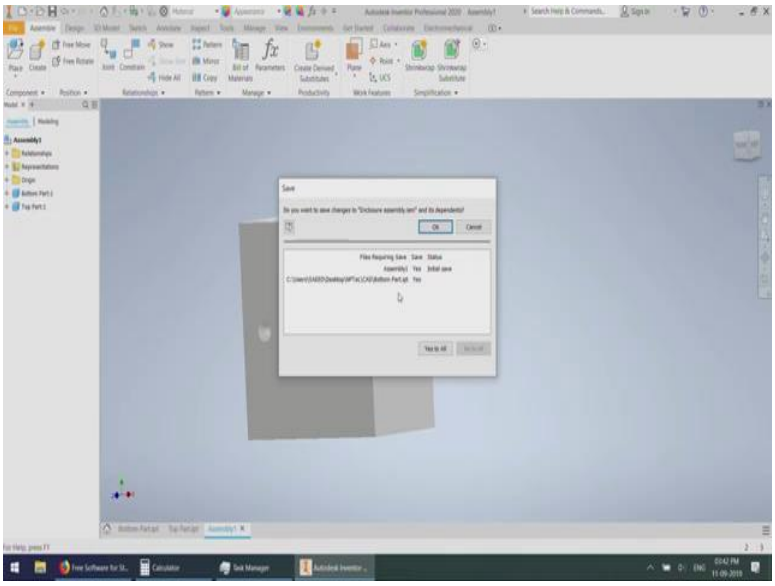

It is CAD, enclo enclosure assembly.

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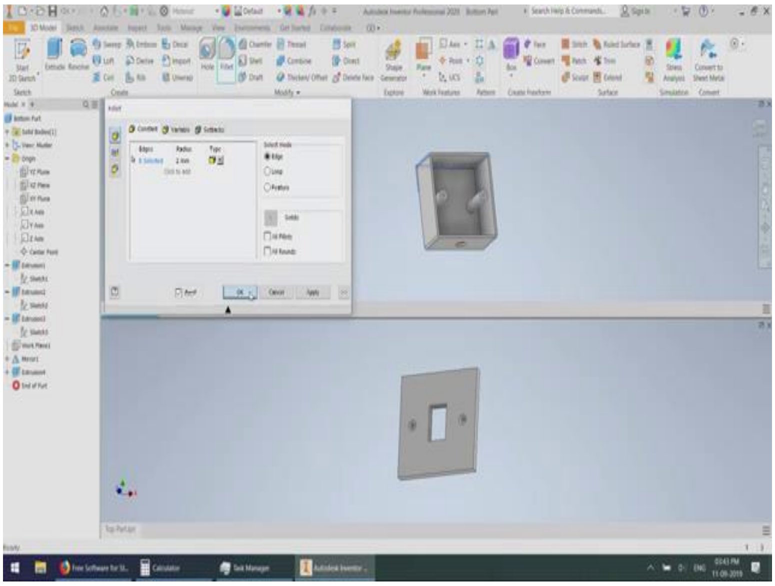

This is the assembly. And we click the bottom part. We will show here. And if we click the top part, it will always show. So, I want to make a fillet on the edges. I will select this one edge and I will click on this fillet option in the 3D model. I will click on the fillet option. I want these edges to be curvy. The default value is 2 mm. I will stick on to that I will select this edge, then I will select this one and this one. I want to make this edge, this edge, this edge and this one to be curvy at the same time. And I will click then save go to the enclosure assembly.

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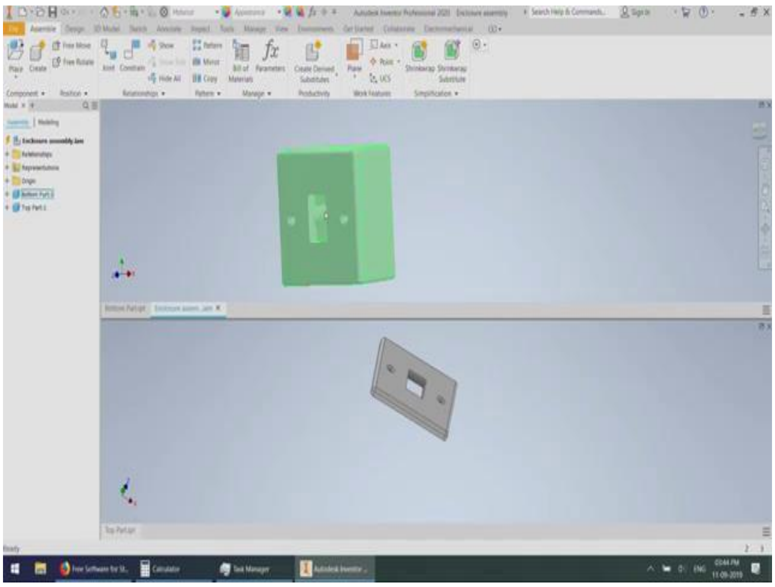

Here we can see there are is a little bit curvy. So, I want to do the same with the top part also. I will select this edge, and select this edge I will and then I will click on the fillet, I will click on this edge, this edge and this edge, this edge, this one, this one and this one. I will click on save. Now, here we can see the edges are curvy. So, it is looking a bit more aesthetic. One advantage of making assembly from parts is if we miss some dimension or we if we ended a wrong dimension value of the top part, and here in this case top part and the bottom part we can see from this assembly.

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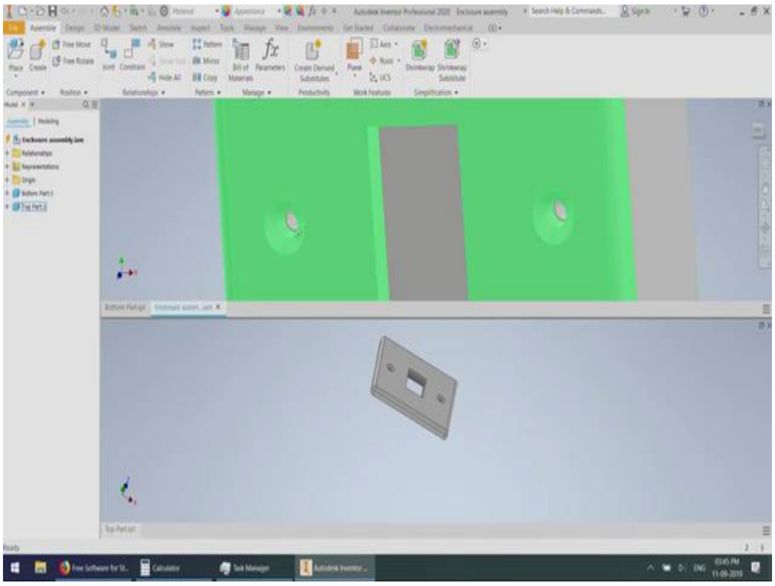

Here we can see these two holes are aligning. Here there is a little bit different because this circle diameter is 3 mm, and here this is only 2.5 mm. Similarly, on this circle also this hole also.

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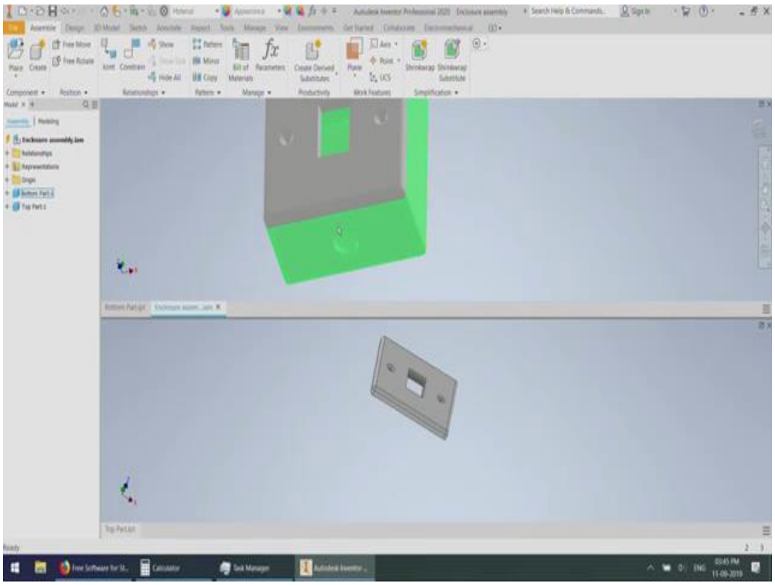

And there is a hole for taking the (Refer Time: 38:21) out.

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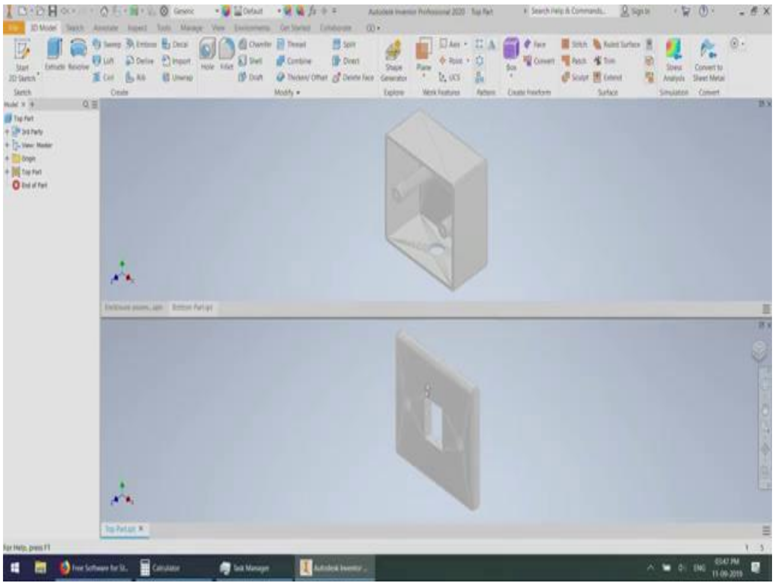

So, we will be using this bottom part file and the top part file for 3D printing. So, I will select this bottom part, and then I have to save this file for 3D printing. So, this Autodesk inventor software I am using is saving the parts in a dot it formats I want some other format that can be readable in the 3D printer software.

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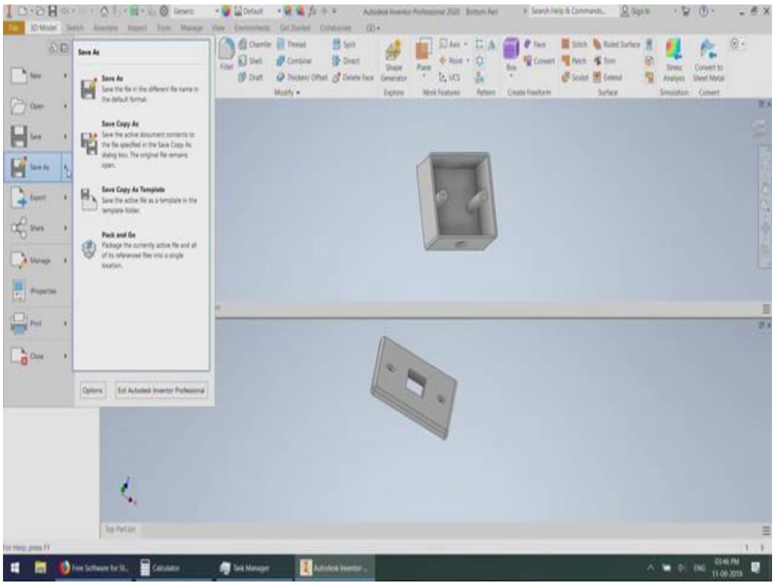
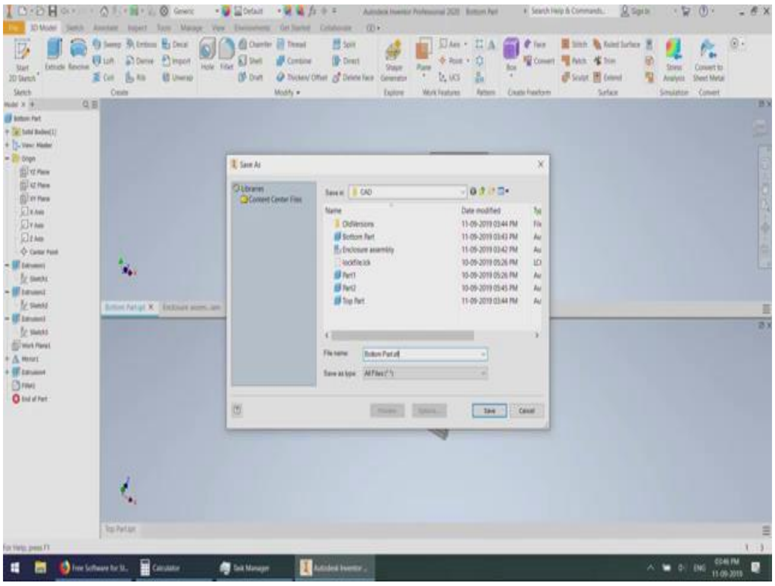


(Refer Slide Time: 39:10)



So, I will go to the file and I will go to the save as option. I will click save as, here see the default file extension is it I will select here and I will I want the file in STL format that is the default file extension for 3D printer software. Now, this is converted to a vector file in dot stl format I want this top part in the same format that is stl file. I will click here in the save as here dot stl. Now, this one also converted to a vector file.