Electronics Equipment Integration and Prototype Building
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Lecture – 27
Ease of editing redesign

I am continuing the session which I had stopped earlier. One of the things I mentioned is it is about the support bush that was required for putting one jack onto the part a chassis of some vehicle, the chassis has a rectangular cross section tube which is probably yeah it says probably it is one and a quarter inch wide and depth is sufficient. Depth which is you know probably 3 inches; one and a quarter inch wide and 3 inches depth because that is a beam cross section. And, at the bottom we have a plunger which belongs to the jack; the jack plunger has to align itself to that.

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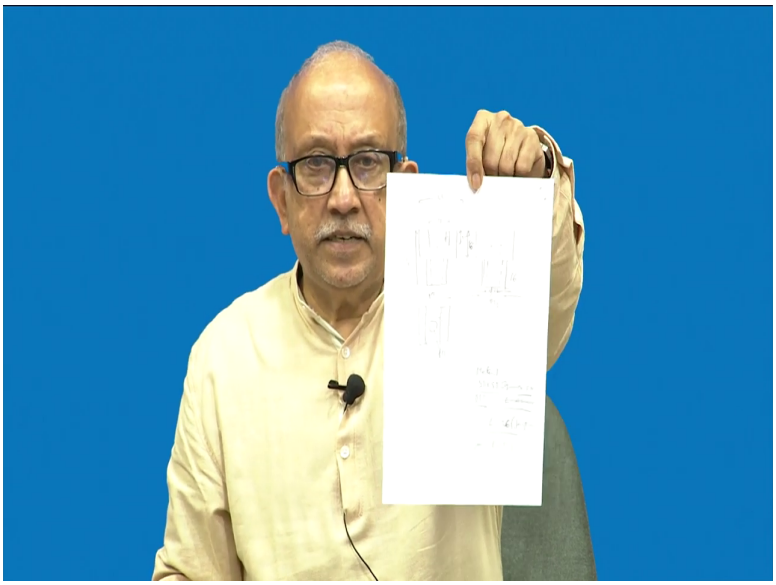

So, considering all this and after taking my concept drawing which I made earlier or fabricator or in our case there we have a small workshop on the premises, they have come out with this dimension. So, one of the things which I mentioned a little earlier is saying we start with a stock material which is a 50 millimeter square.

So, now, I will try to see if I can model it here. So, if I see here I am starting with the top view and there is several ways of doing it is there. One of them is I can in fact, talk with the directly because it is a solid modular it may be appropriate for us to start directly with a solid.

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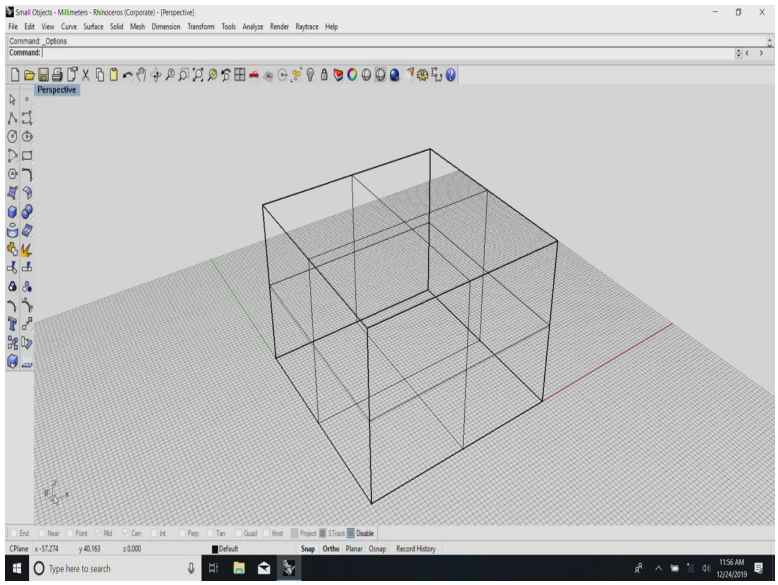

In my case, see here I am looking at it at the bottom. I have a 50 mm cross section and height being the job height is actually 34 millimeters.

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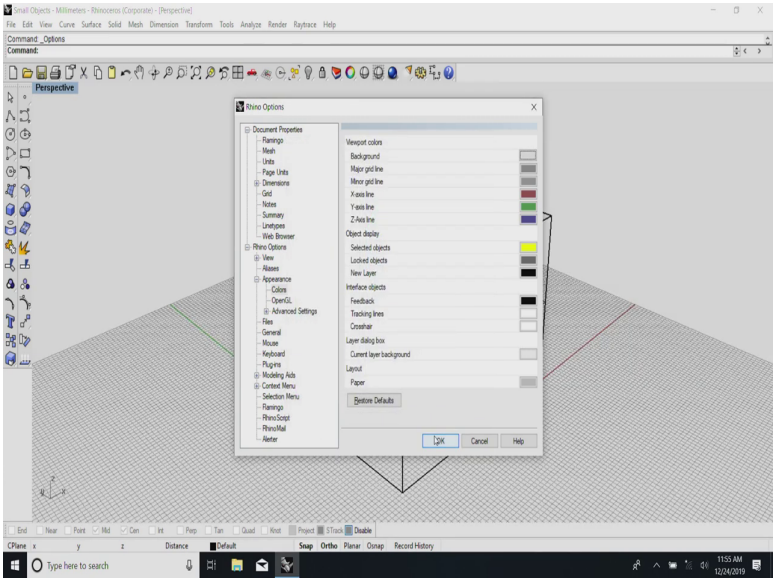

And, if I now come here you will notice that the basic what you call raw material is ready. I will just need to modify a little about the background color. Yeah, this is better. See, here this is the basic raw material within which I have started it. So, if you have to dimension it automatically whatever the dimension which I have picked will come there.

Now, in this I need to make a 31 millimeters groove front to back. So, at the corner if you see I have here at the corner this word icon axis. So, I have a x which is a conventionally left to right is x and y is front to back and then a z coordinate which is top and bottom.

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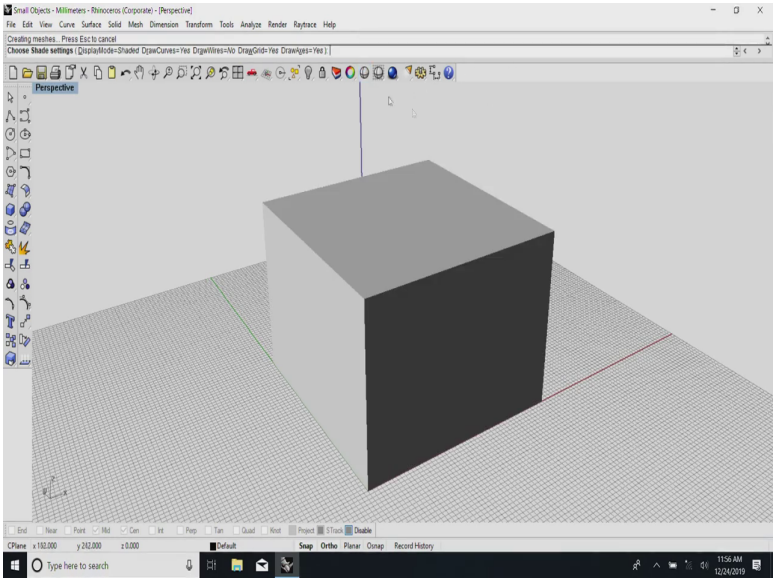

So, if I now put it in this mode I see that I have this nice thing sitting here. Now, I need to make a groove which is 31 mm wide and 11 mm deep. Now, we will see here looking at the.

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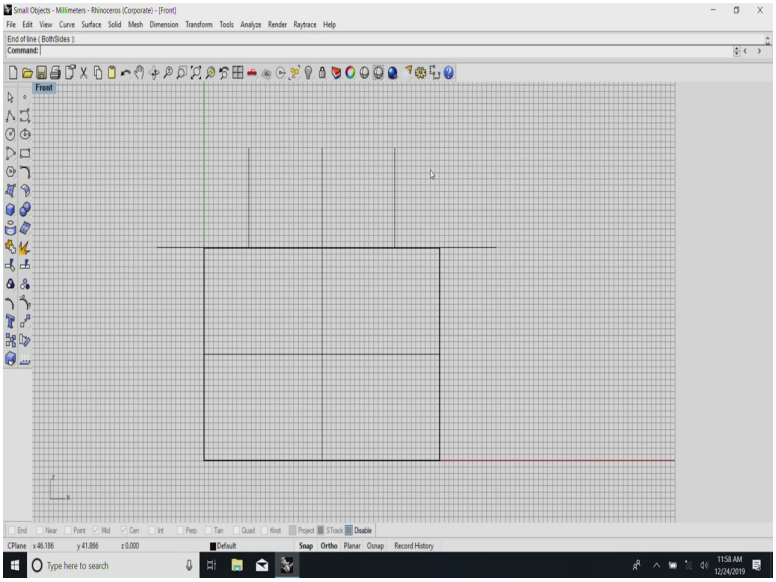

And, this point and trying to make another solid to do the thing will be slightly tricky. So, in that case what I do is I will try to create a solid by entering all the dimensions manually.

So, I have this typically the center line. I offset this by that being 31 I need to offset it by 15.5 millimeters on one side another 15.5 millimeters on the other side. Now, I need to make a depth of 11 millimeters. So, easiest thing for me will be to join these two ends this snap setting has been made such that its half millimeter snaps it will not accept those things.

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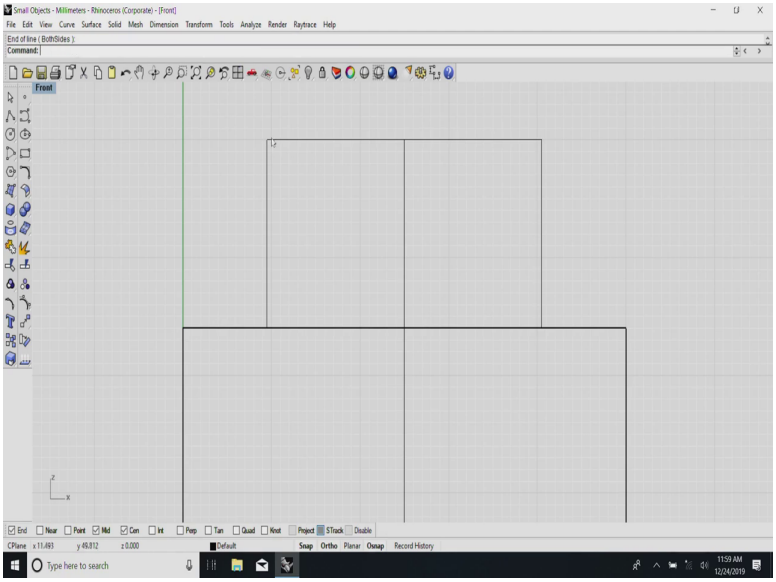

See here because of the rounded of grid and 31 and a half of it is a little less it does not come well.

Now, fortunately for me I can make a rectangle using the built-in provision. Now, I try to create you see here if you see top to bottom it is more than 10 millimeter something which is required like this is there and then after that I tried to now create this whole thing is a rectangle.

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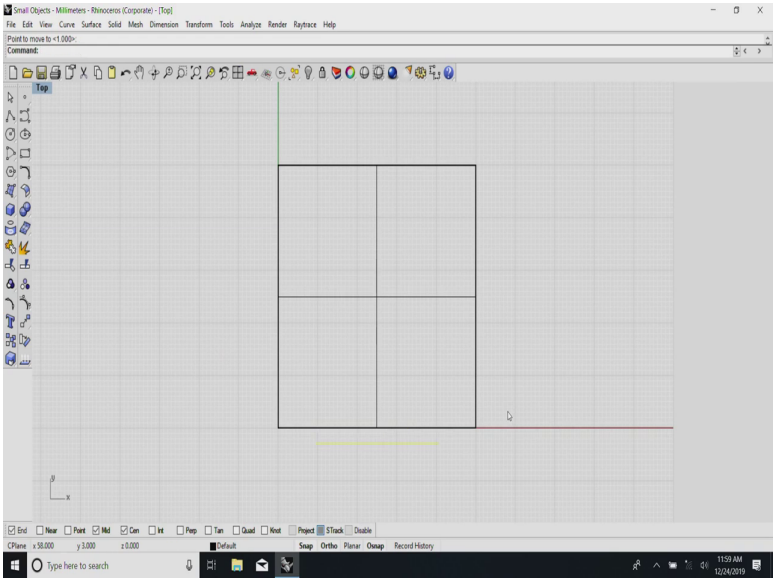

Now, I if I go to the other view I will notice that it is sitting here.

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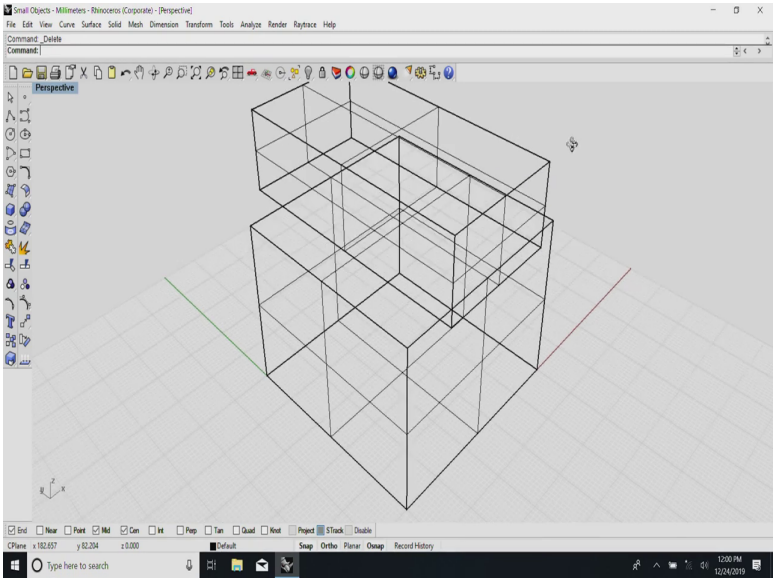

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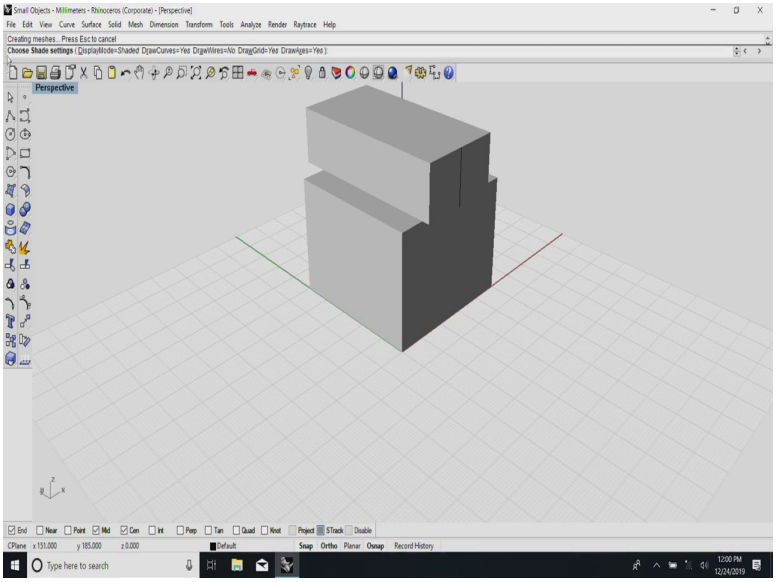

Using this I make a see here what I managed to make is one more intersecting solid with which I can remove the material. Why I am showing you this is. This is exactly the way the fabrication shop works here.

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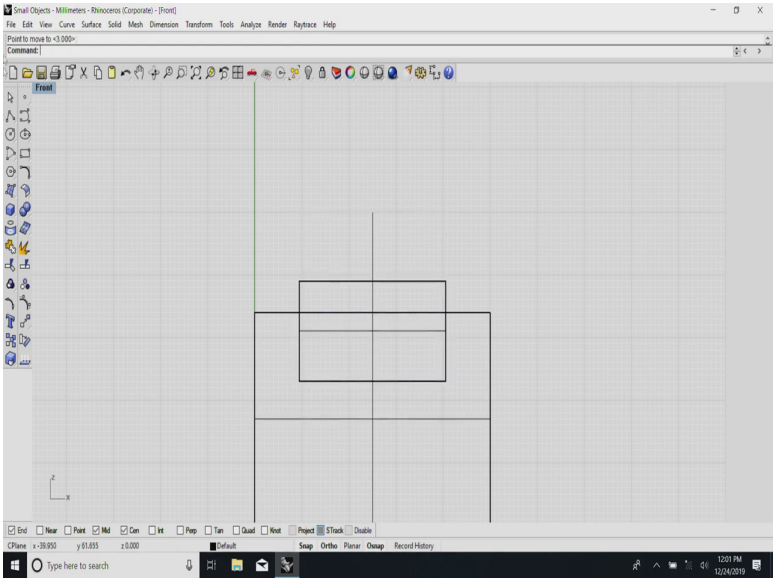

Now, the only thing is this is touching here. You have seen this? This is almost touching here. Now, I need to move it to a depth of the necessary depth. In this case this is 11 millimeters.

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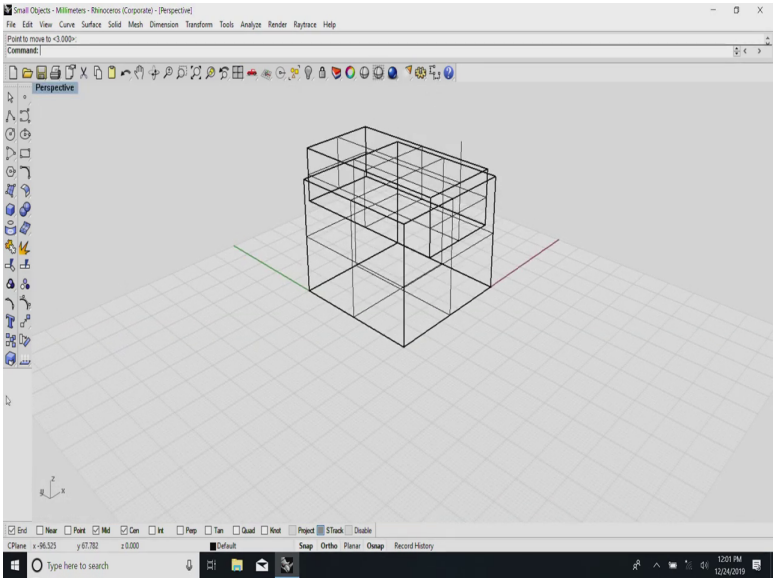

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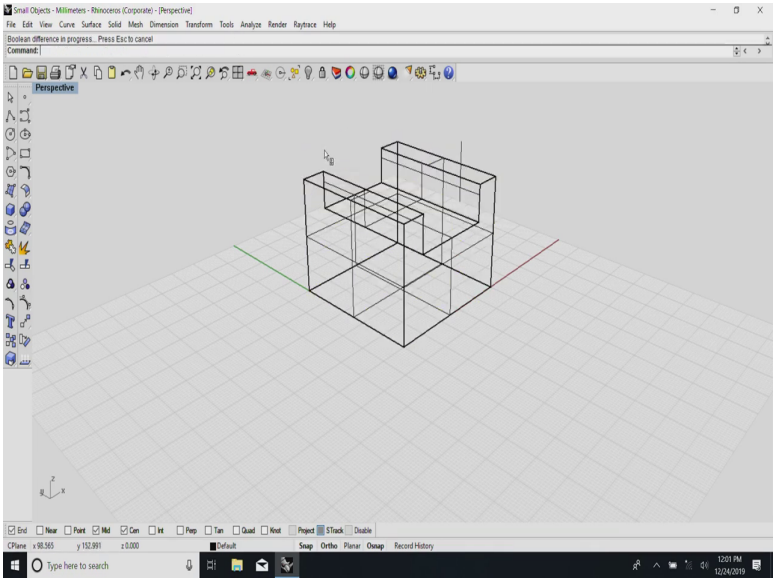

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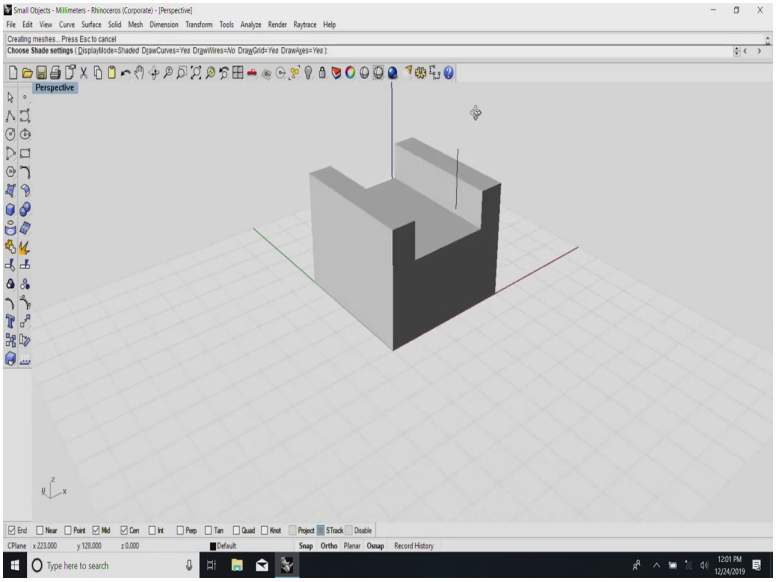

Now, I have two solids which if I make the difference, see my first cut top groove is ready.

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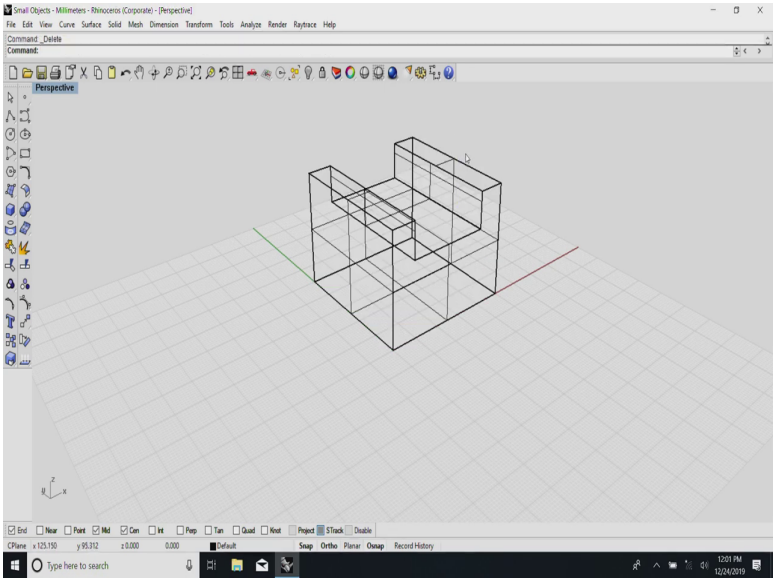

Several advantage it has – one of all one of us first term it is perfectly to scale; second thing is it follows a little bit of the operation sequence which we do in the work shop. Now, there needs to be a groove at the bottom. In this case I have measured it out saying it requires a hole of 22 millimeters.

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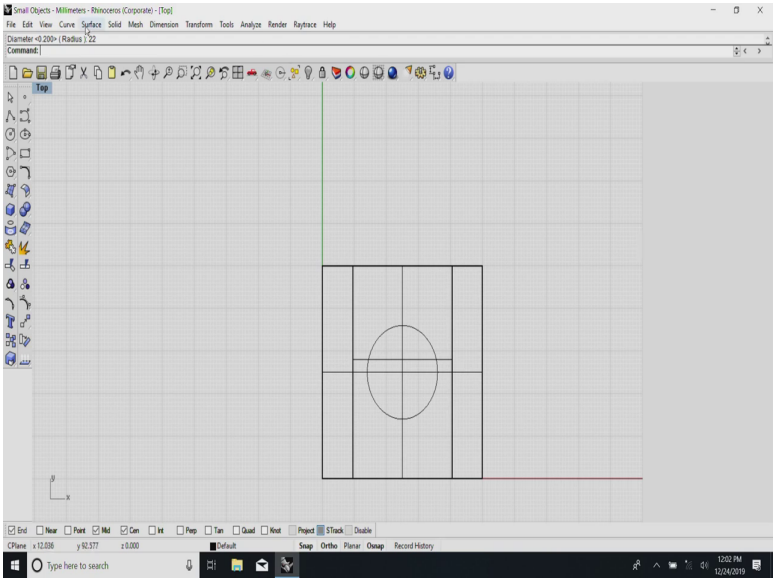

So, it is just logical for us to make an opening like this such that this is the one that goes and sits on top of the plunger at the bottom.

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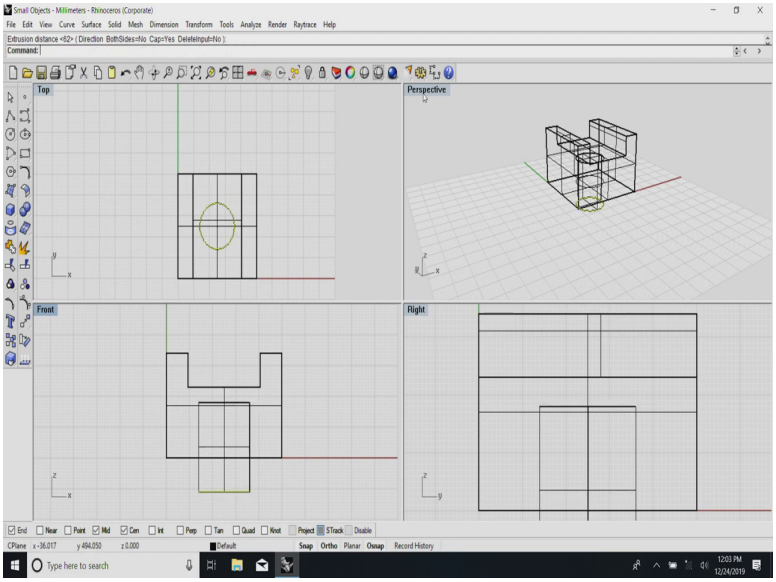

So, in principle this is sufficient for me and that needs to go to a depth of some 16 millimeters it says. I will see how deep I can go with this. Like before, I tried to make a cylinder by extruding the same planner curve.

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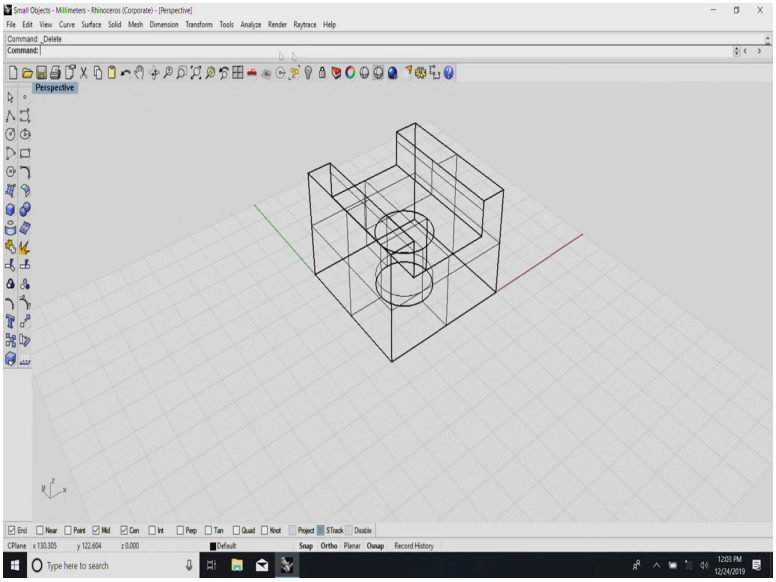

Now, once again I try to do the same difference. See here something functionally what I want has come. On this side there is an opening.

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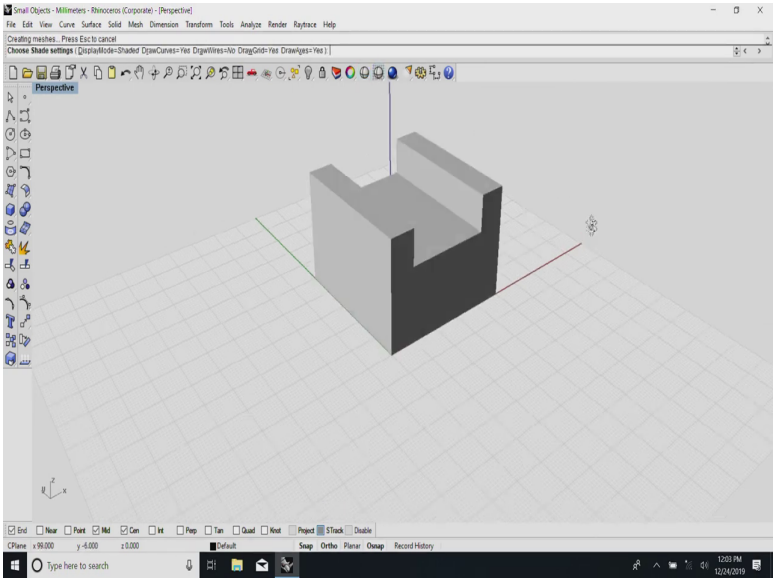

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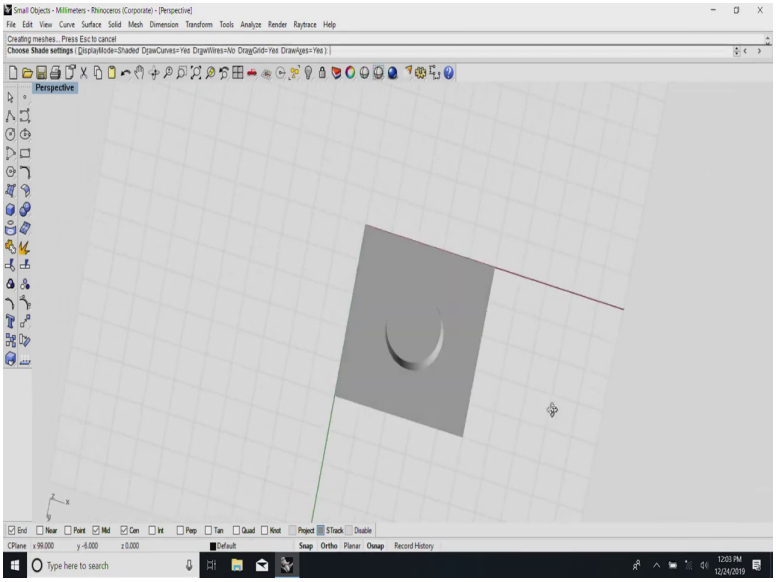

And, this can go sit and directly sit on top of the plunger.

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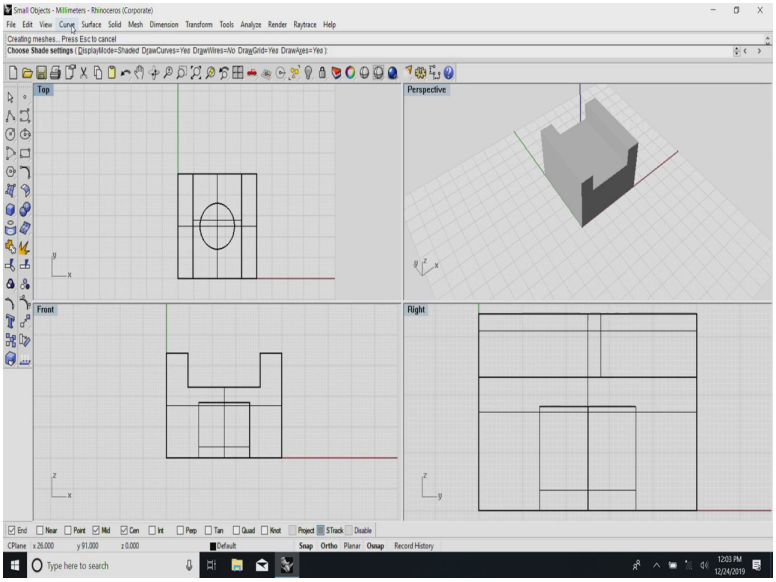

And, for purposes of drilling a hole and making this easy they have also made a radius or a diameter of 15 millimeters and they have removed this material.

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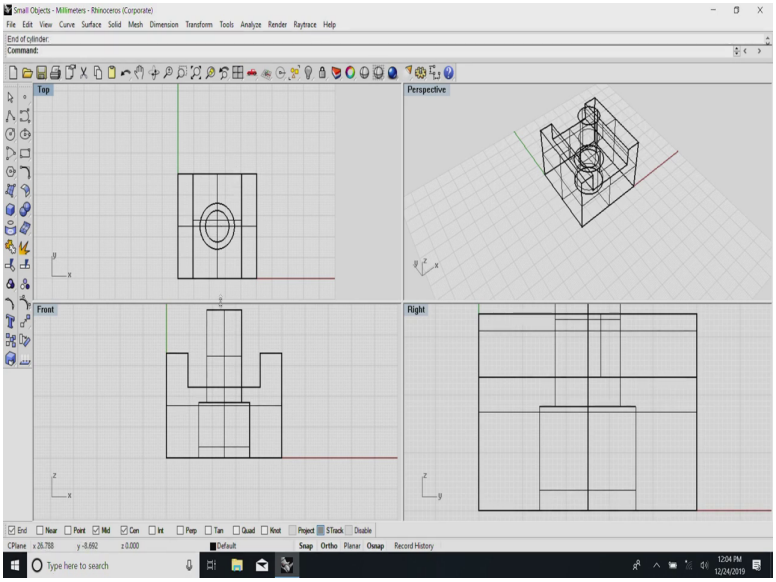

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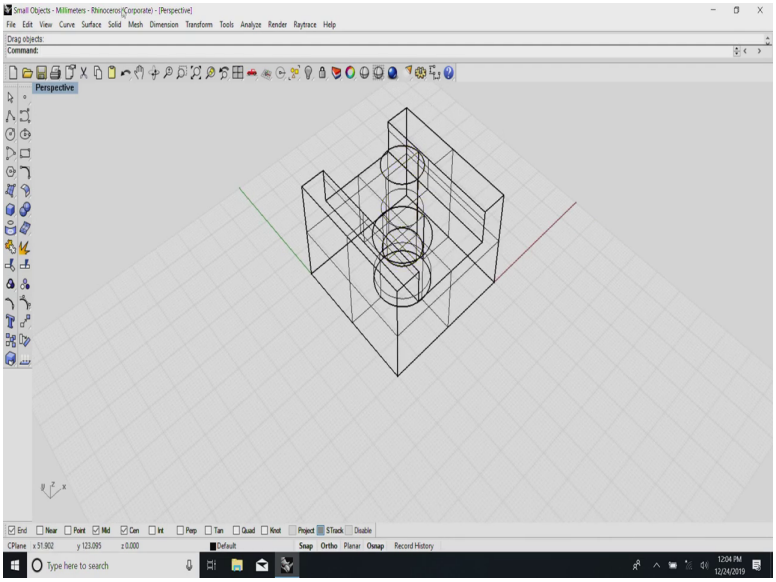

Now, I will get back to this and say why it is.

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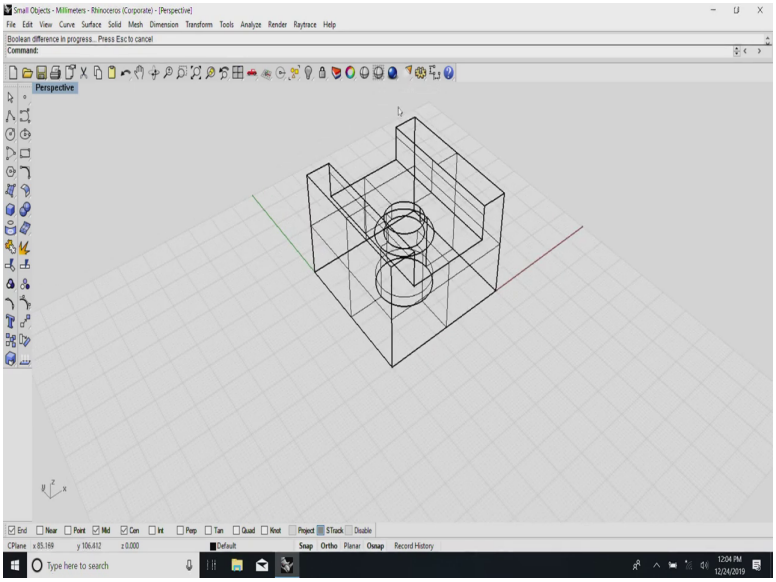

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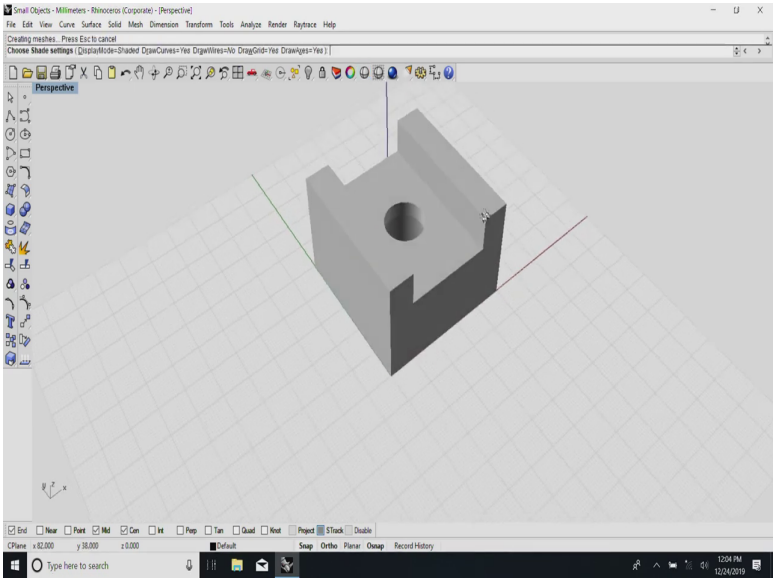

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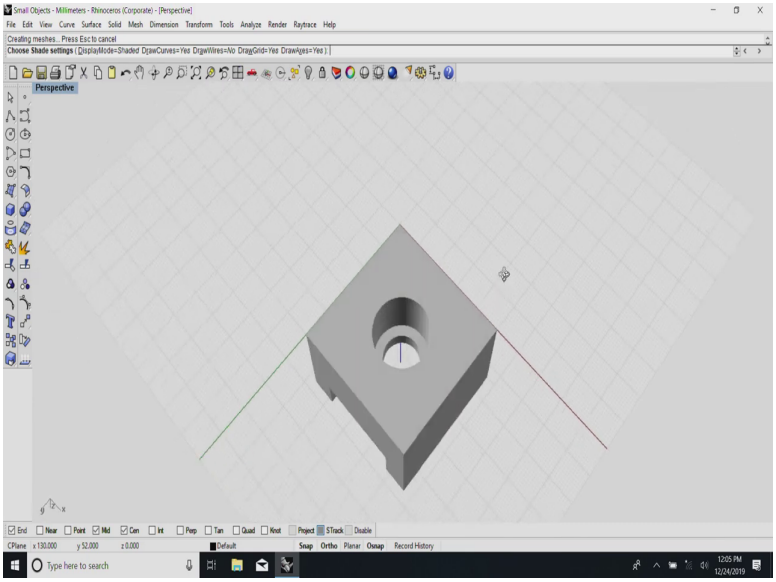

This is typically a process requirement. When they try to drill a hole here instead of using a milling cutter and try to make the end mill do the job it is a very conventional for us to generally try to make a small pilot hole using a regular drill which has a twisted cutting edge. It is much faster and much easier. You see the basically the basic functional thing is now ready for us.

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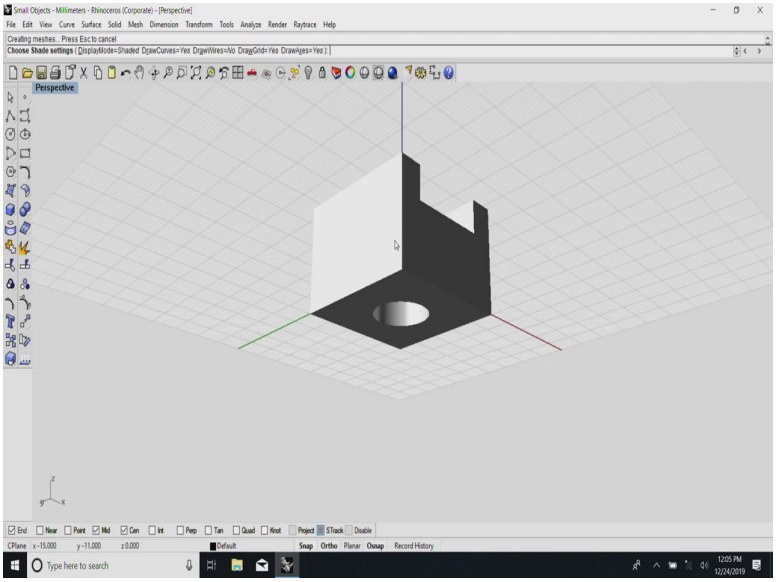

But, then that next little small thing after conceiving the part next to the small thing was you know thought of one of them saying all these extra material all around first of all it makes it heavy and possibly interferes with something else which is around.

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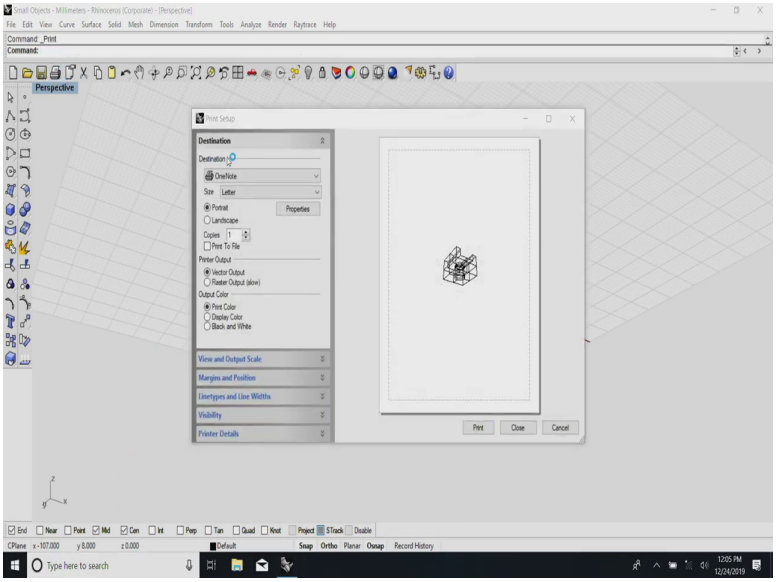

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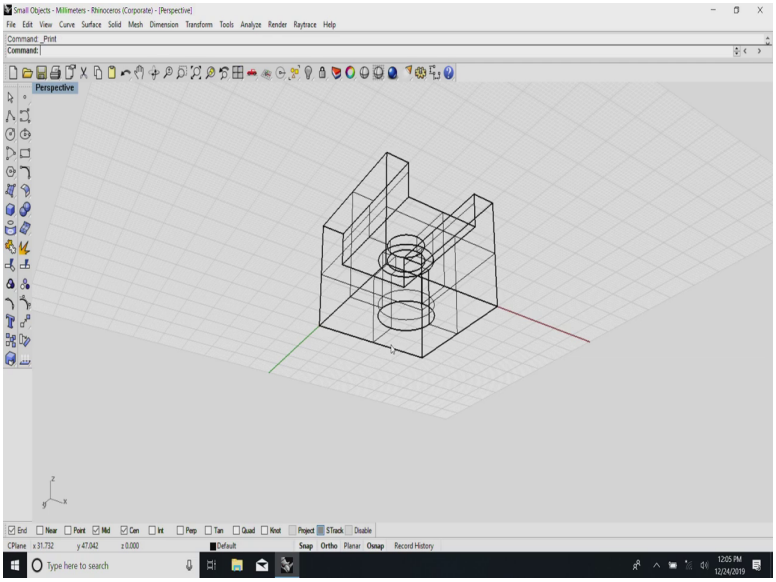

So, it was decided that it was decided that we remove all the material around just leaving only a little bit of around 5 millimeters all around.

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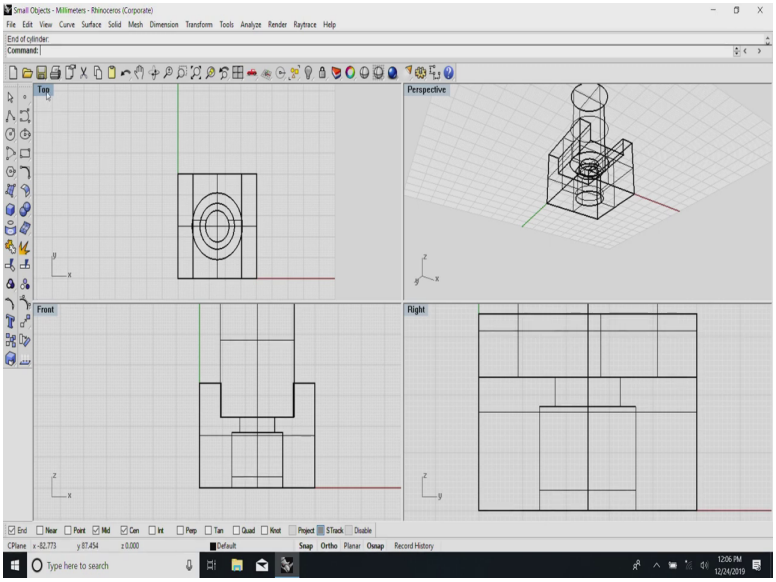

So, a outside diameter 33 cylinder was prepared. You have seen this? Now, I will come to the more interesting or.

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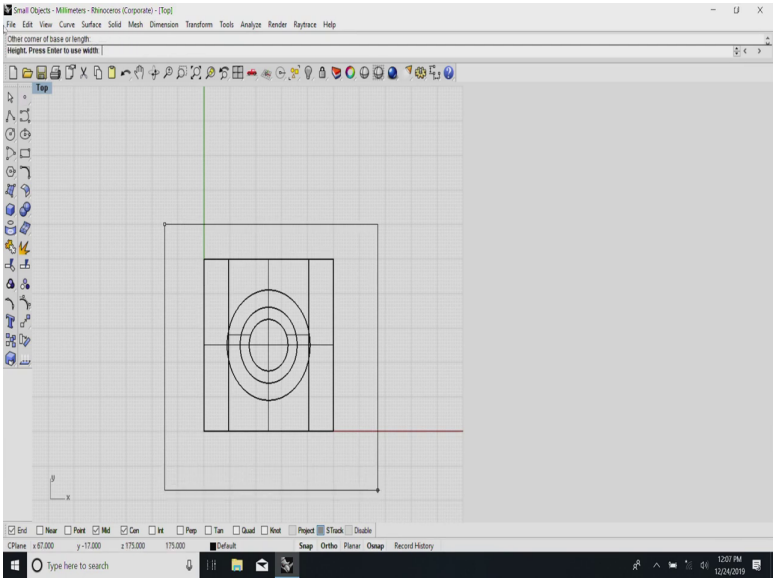

These are tricks which actually are not this is not the way to do it, but still it is very convenient for us. I will try to now make another box around it they are just used for what you call removing material all around.

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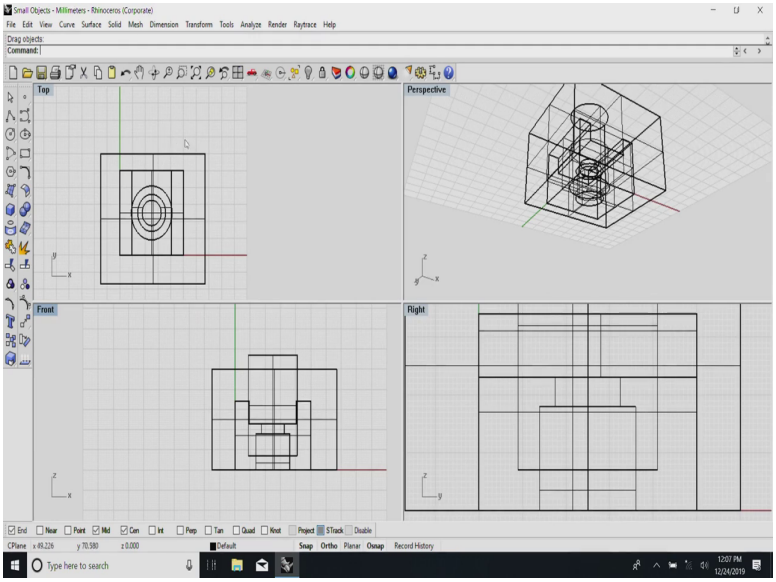

You just watch. I am just trying to make an annular piece from this solid I remove this.

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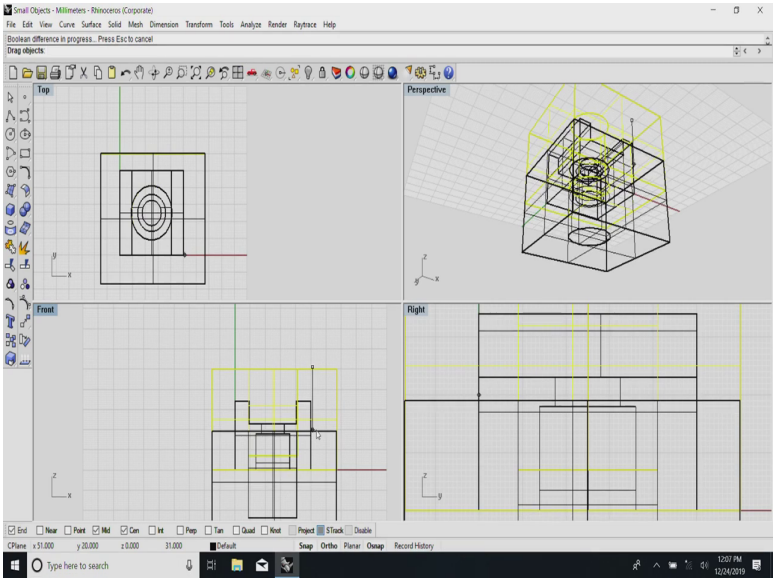

Now, I have something which I will use to remove the material all around from this.

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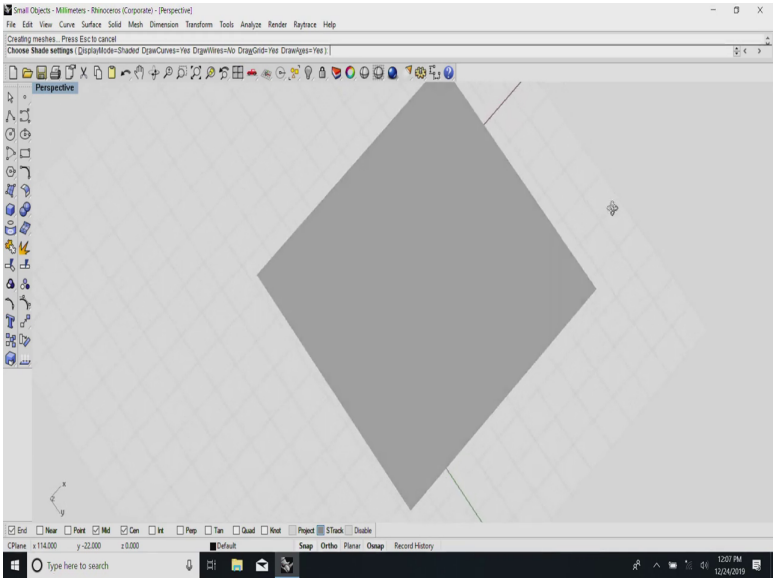

See here.

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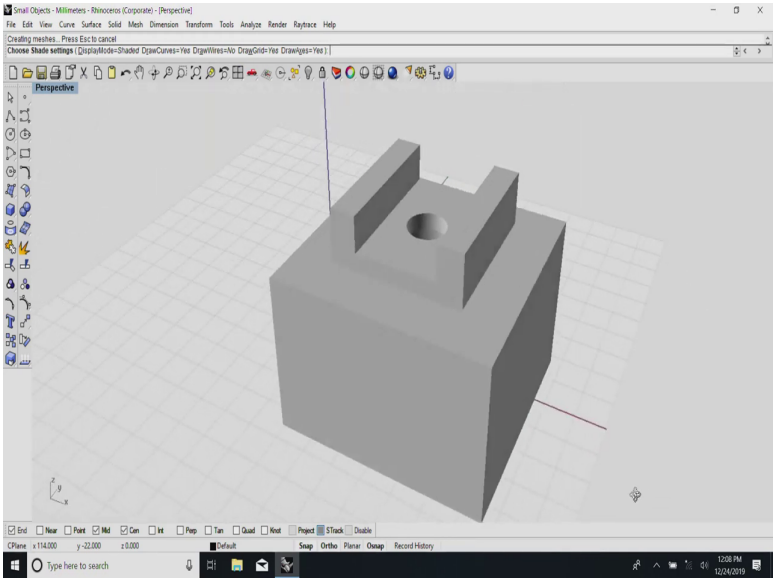

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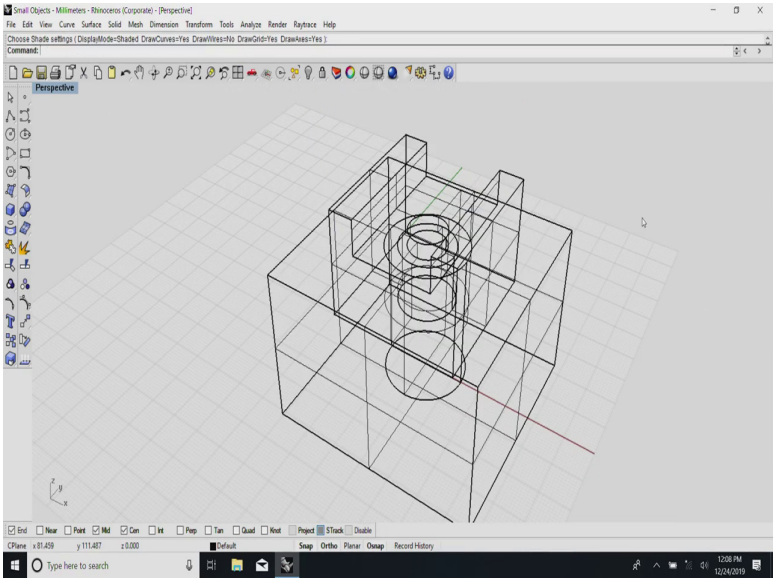

This could have been done in a different way but right now let me proceed only using the very simple Boolean difference.

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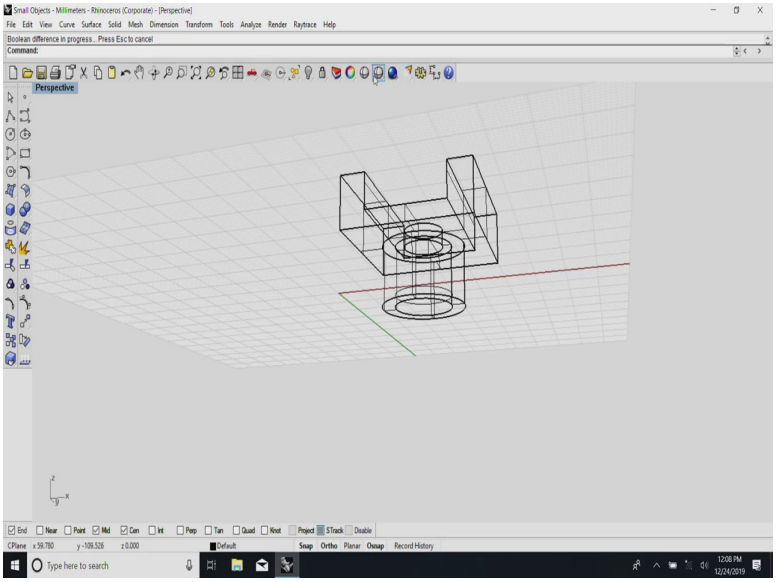

From this object I remove externally all this material. You see here what I wanted to make has been created.

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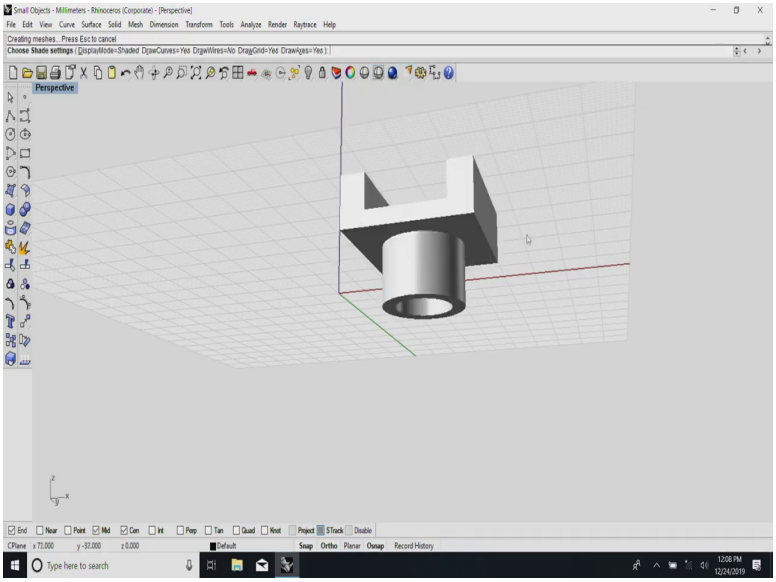

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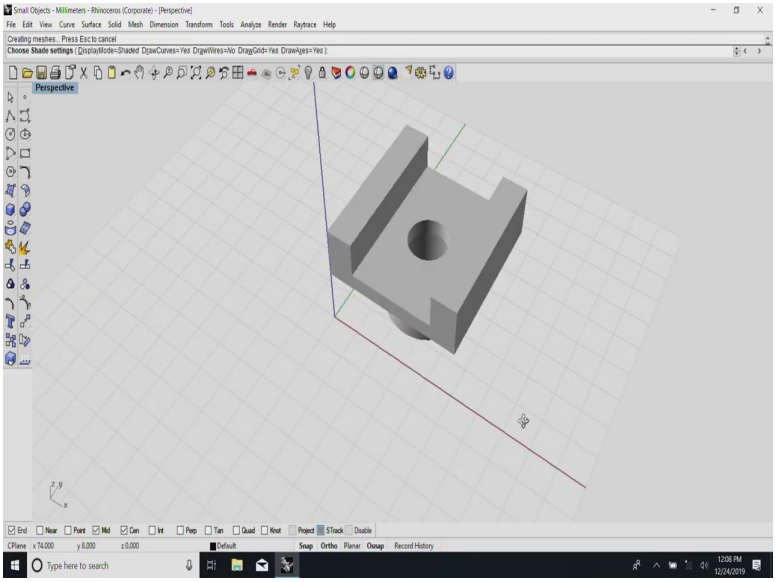

I am sure it would have been easily possible for us to do several other ways. One of the ways is trying to try to make a cross section of this extrude it; that extrusion and that systems will come if actually in reality we are having an extrusion.

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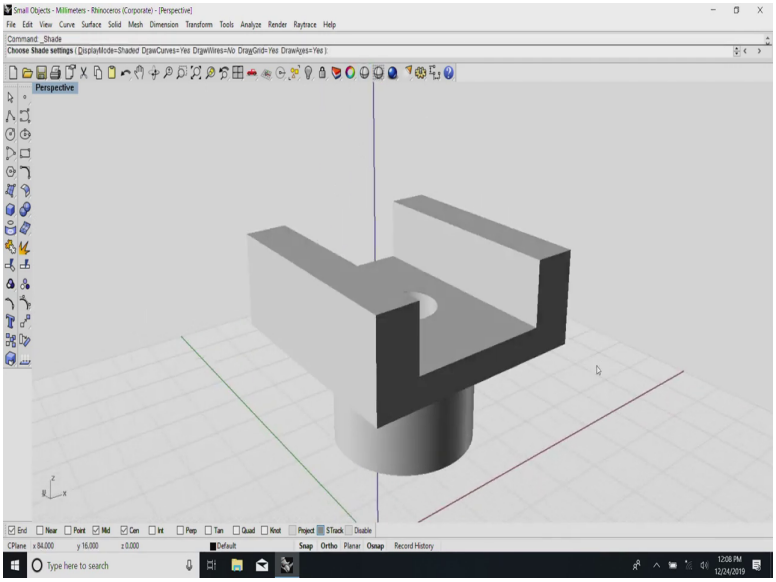

And, afterwards you drill a hole in the extrusion on the other side we make a pipe and fit it to that.

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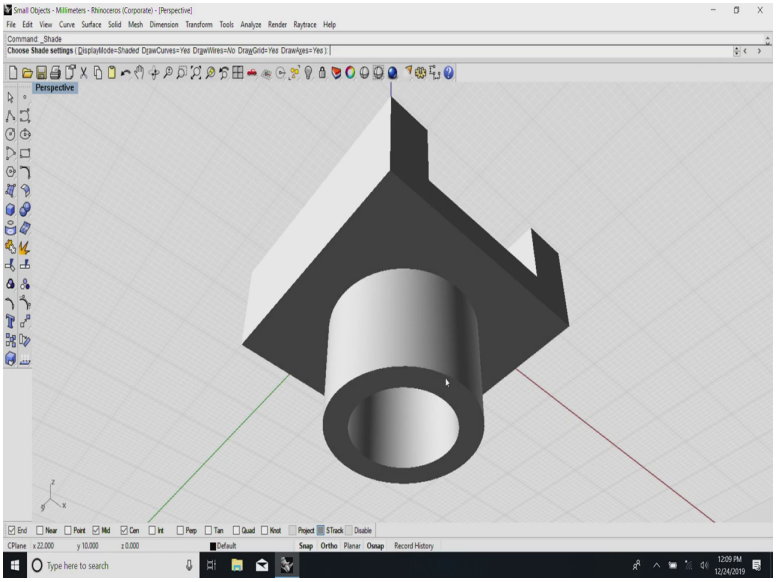

And, then one short equivalent thing would have been produced very easily and now to make the extrusion I just need all this four things and then take it long here. And, then to make this pipe I need to have the internal and external dimensions and eventually when we join them together we will have the same job which has come here. Yeah, I hope you are with me at this point this has been created in one way. There are any number of ways in which we can be created.

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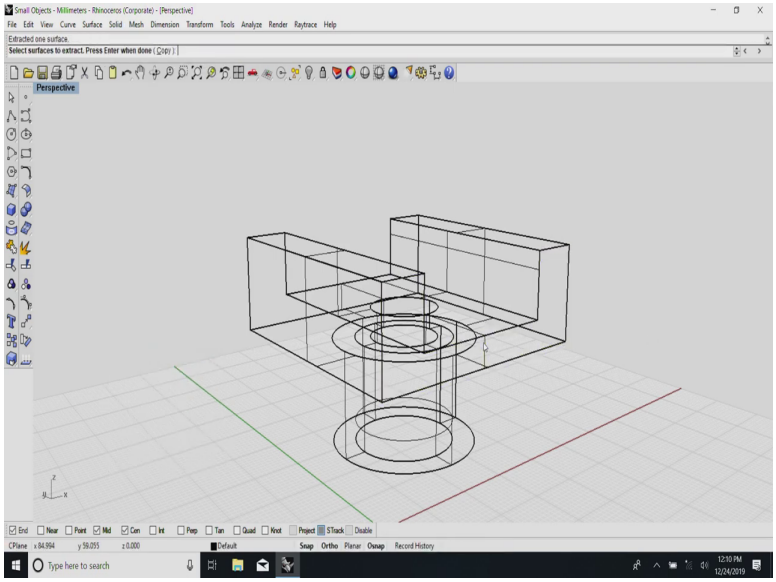

Now, what we generally choose is depending on the thing which you are familiar with. If it is a extrusion, imagine I have now I go to this surface and try to extract this surface. See, I have a surface here, I have extracted.

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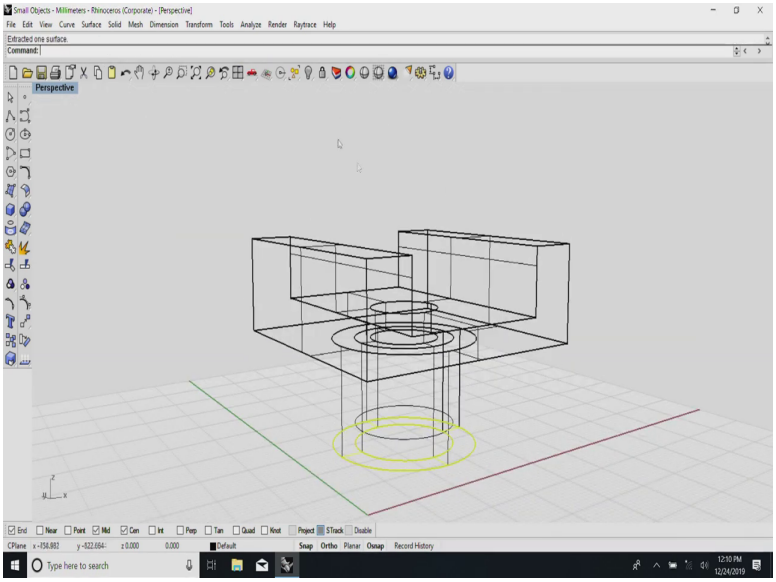
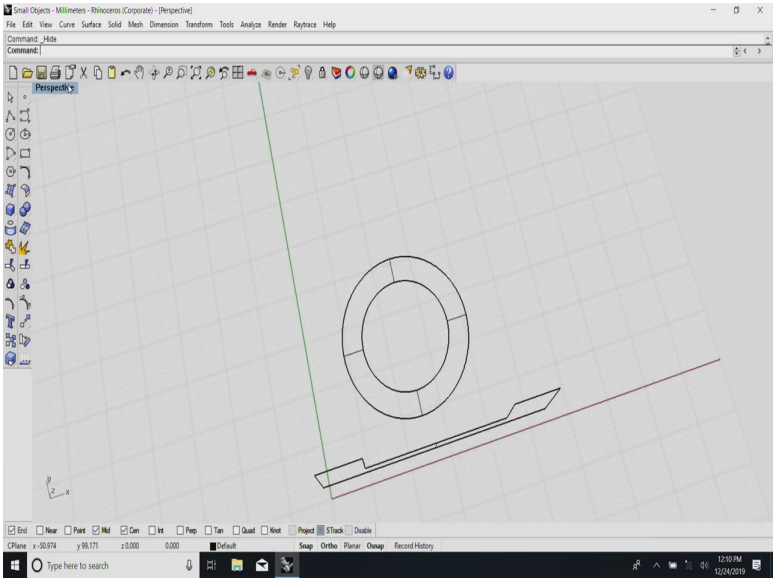


Similarly, I have another surface here.

(Refer Slide Time: 15:04)



Now, if you see here the same object can be prepared by using these things if it were a what you say a steel angle or a steel c-section or some other thing.

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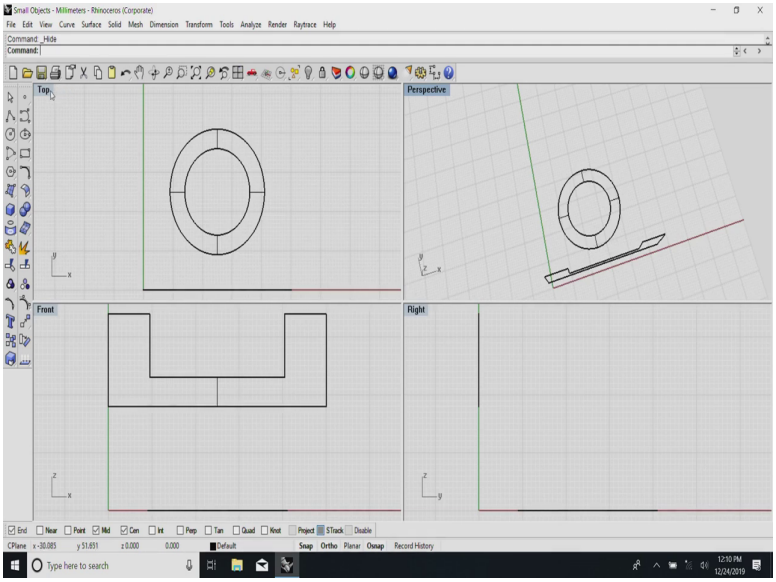
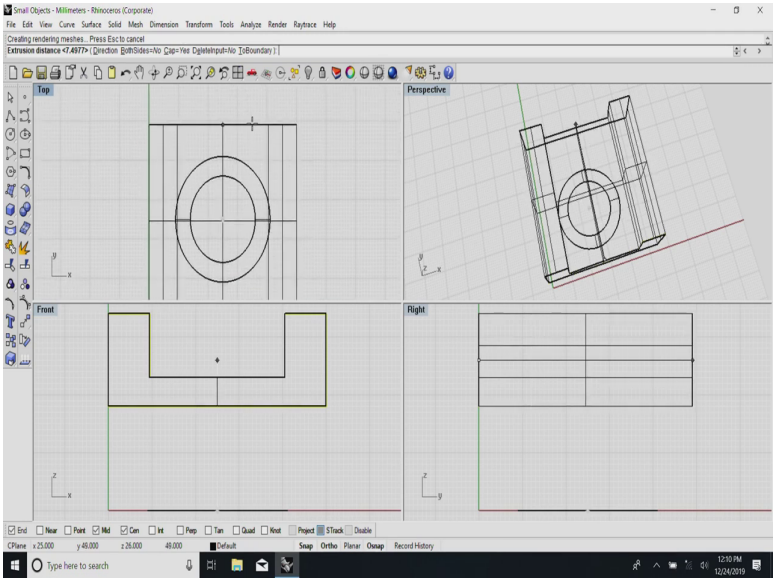


I have all the necessary information which is already built into it. Instead of going the other way I take this try to make, see same thing has been achieved here.

(Refer Slide Time: 15:32)



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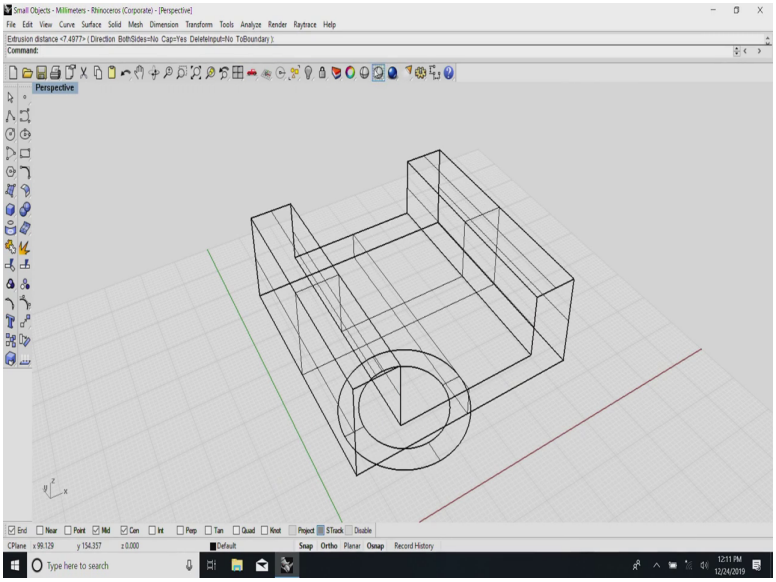
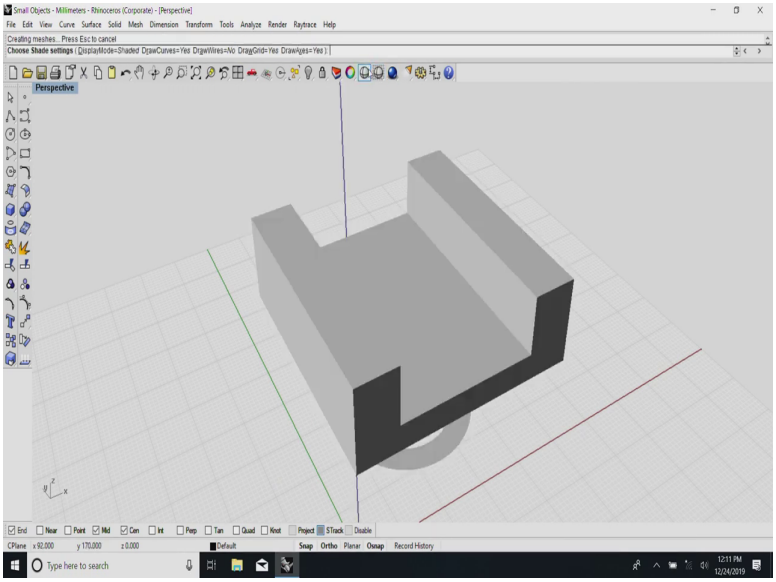
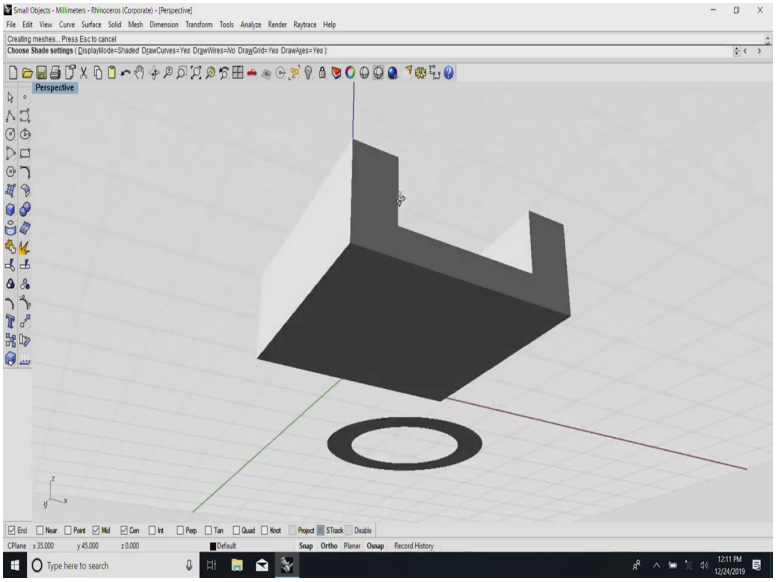


(Refer Slide Time: 15:43)



And, why one is preferred over the other is that one was based on a 40 mm square or length 36 mm. In this case let us say I already have some usually an extrusion of the nearest size imagine I have a c-section which approximately has the same this thing here.

(Refer Slide Time: 16:04)



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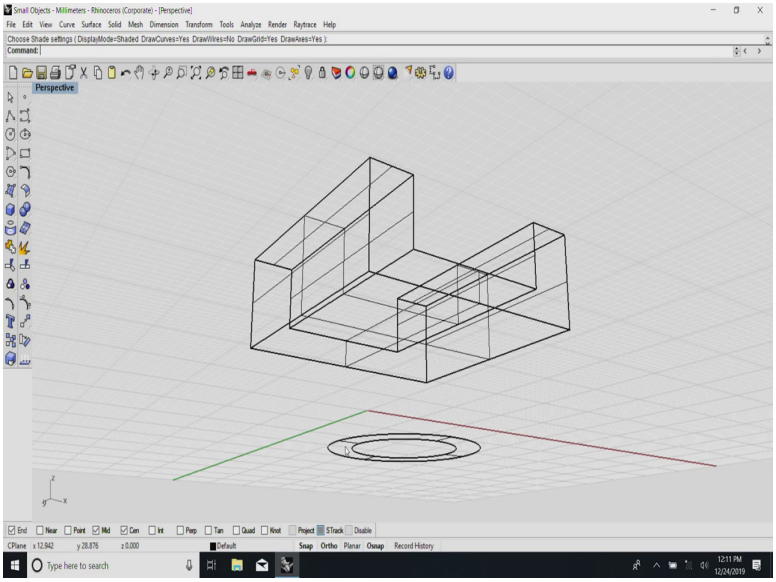
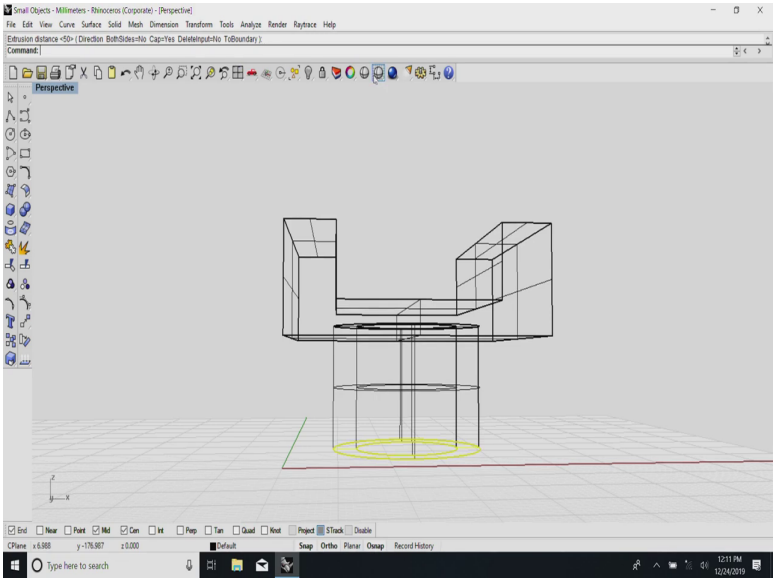


I can use this c-section cut it to length and at the other side I have a small pipe. And, pipes of various diameters are already always available.

(Refer Slide Time: 16:13)



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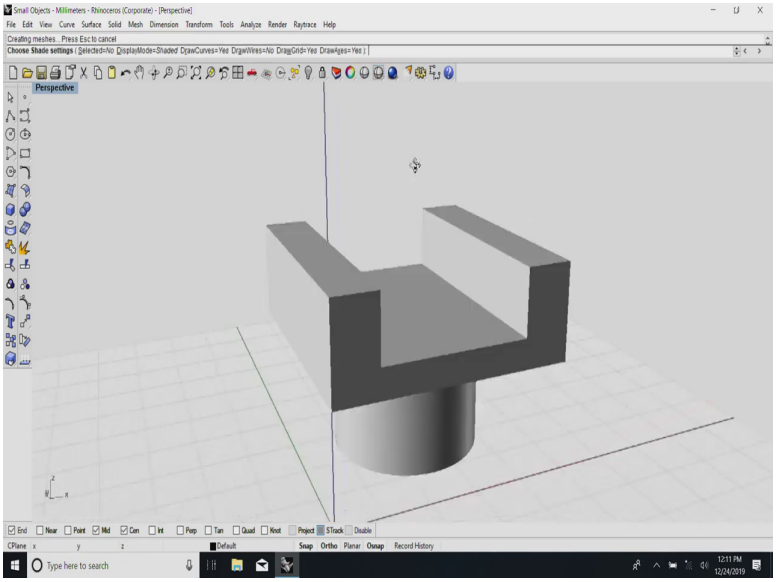

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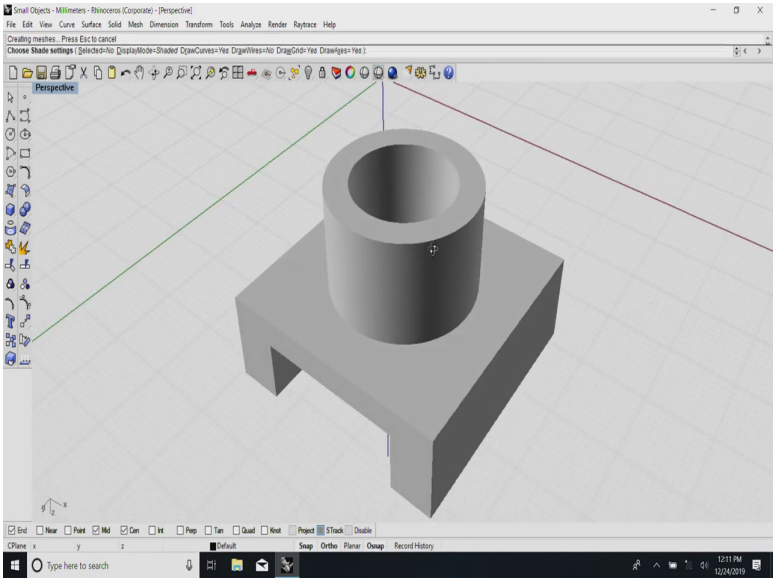
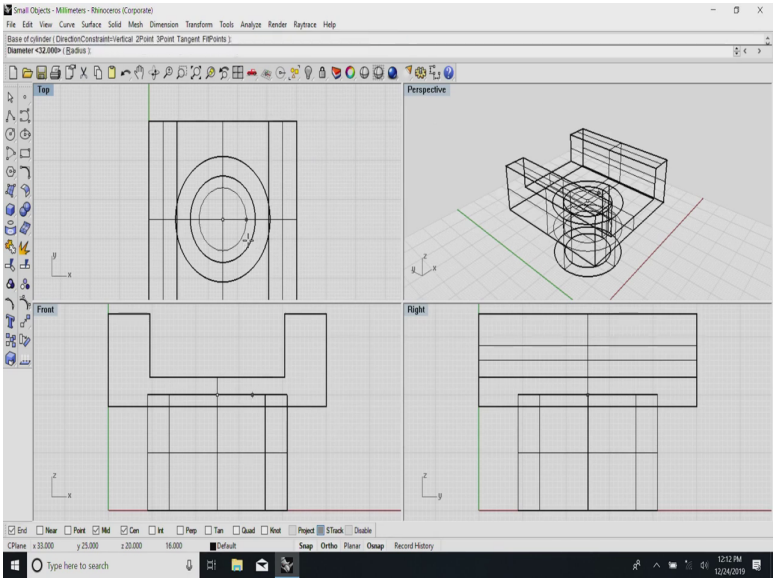


See, in principle it is the same object which have created I have hidden the object I will try to show there in of the middle there is a 15 mm hole. Now, I will try to make the hole here.

(Refer Slide Time: 16:48)



So, as before I tried to make a cylinder and see what best I can. So, diameter of the cylinder is 15 millimeters, yeah.

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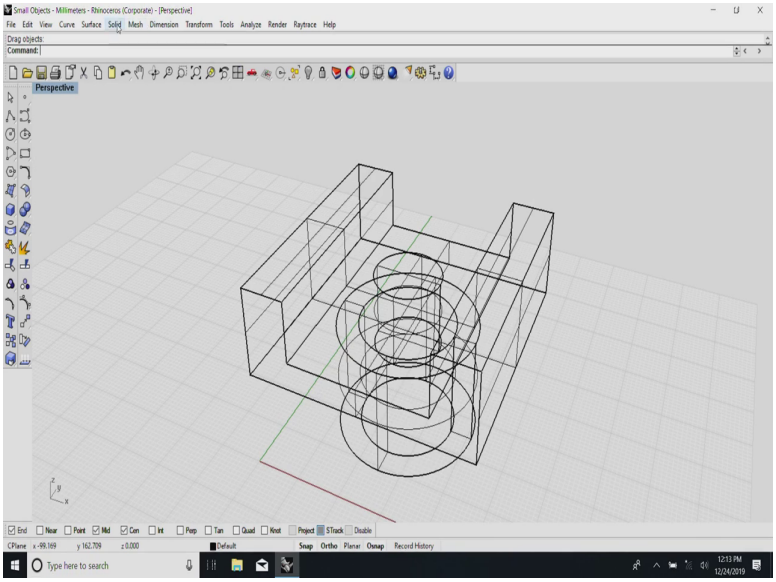
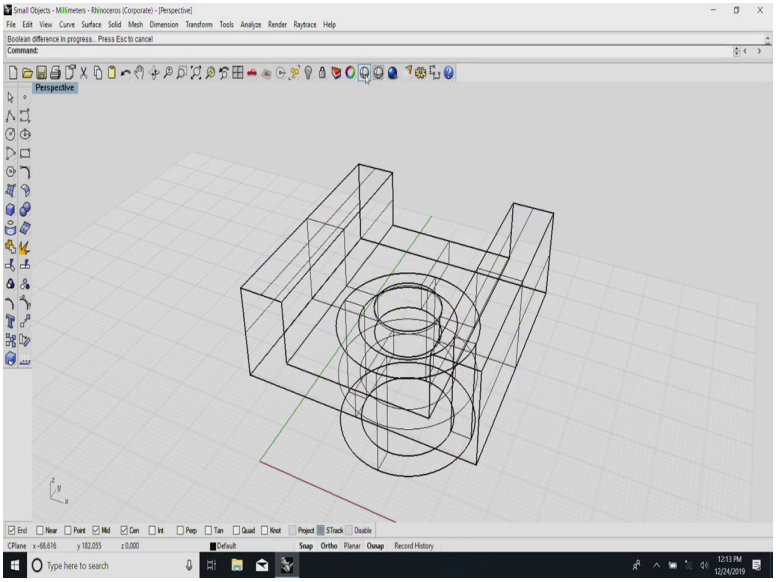


This is the opening I am trying to make in this. Now, as before I tried to make the union I am sorry the subtraction from this you see what happens. I have the same object created by a different method.

(Refer Slide Time: 18:37)



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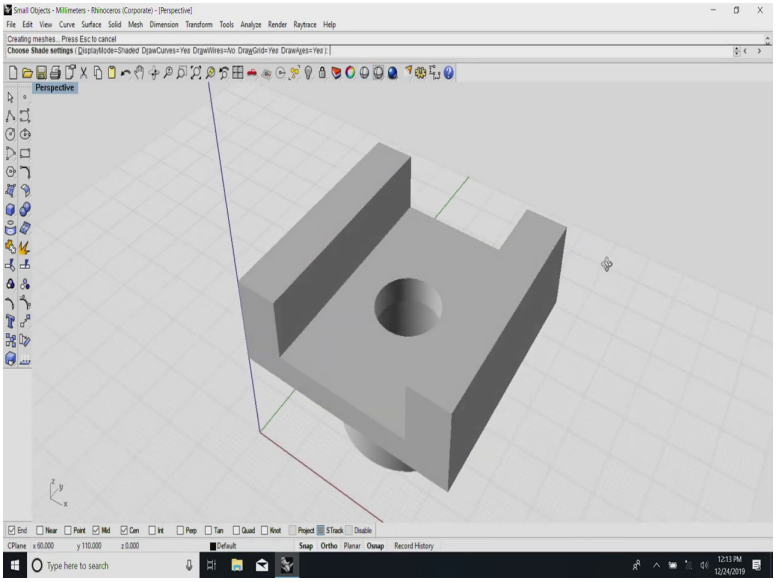

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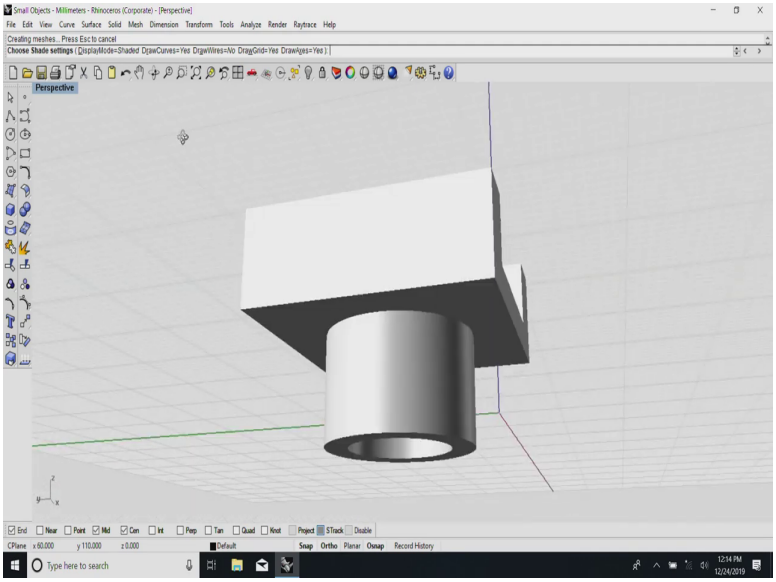
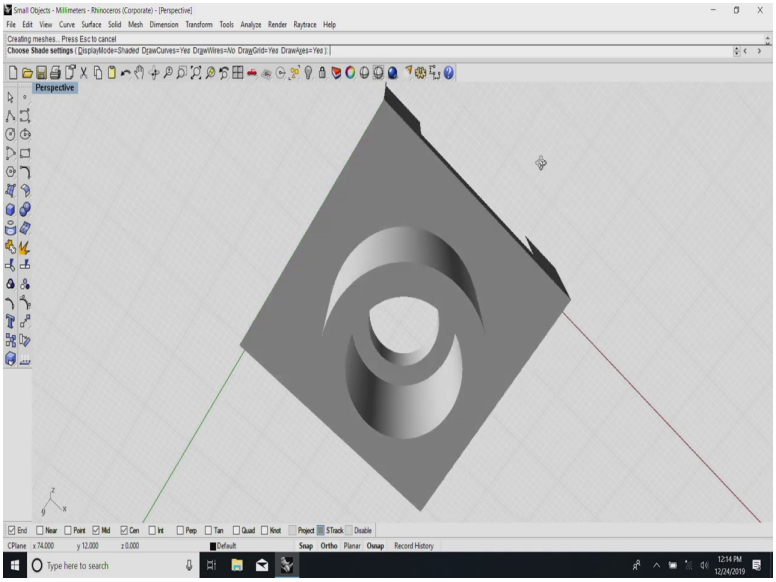


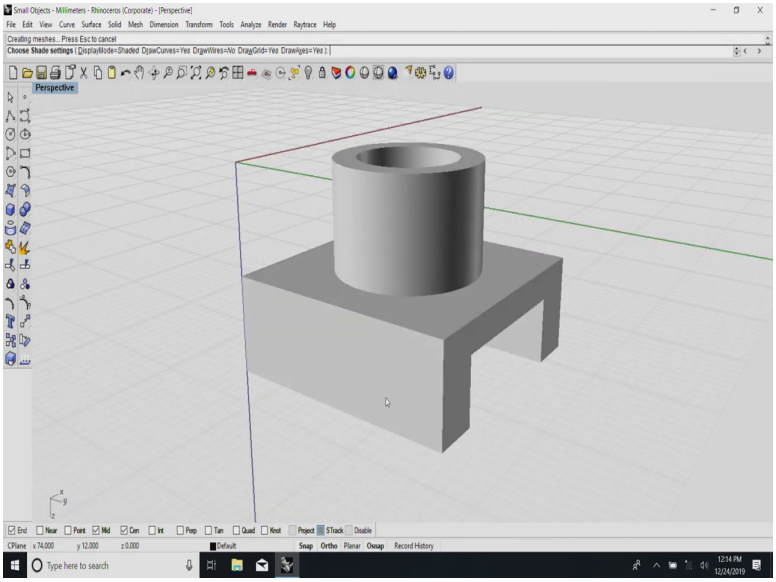
So, what is the difference one of the first things in the difference is I have a probably an extrusion. In this case a C channel of internal width of 31 mm which is typically that one and a quarter inch I am talking about is the all that is required. And, the thickness really does not matter I can I get things which are 3 millimeters and 4 millimeters wide. So, a C channel I have used the profile of the C channel to take it forward and show you that whole thing is possible.

(Refer Slide Time: 19:14)



And, at the bottom I have attached a tube to this, you have seen this and in this case comes the quite important things saying this is where our CAD packages are able to really make a difference where I can do a filleting which is equivalent to.

(Refer Slide Time: 19:21)



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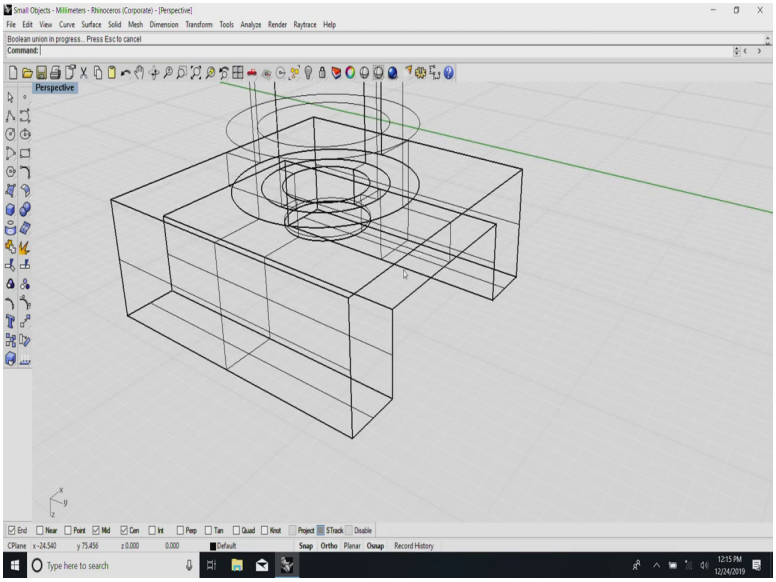
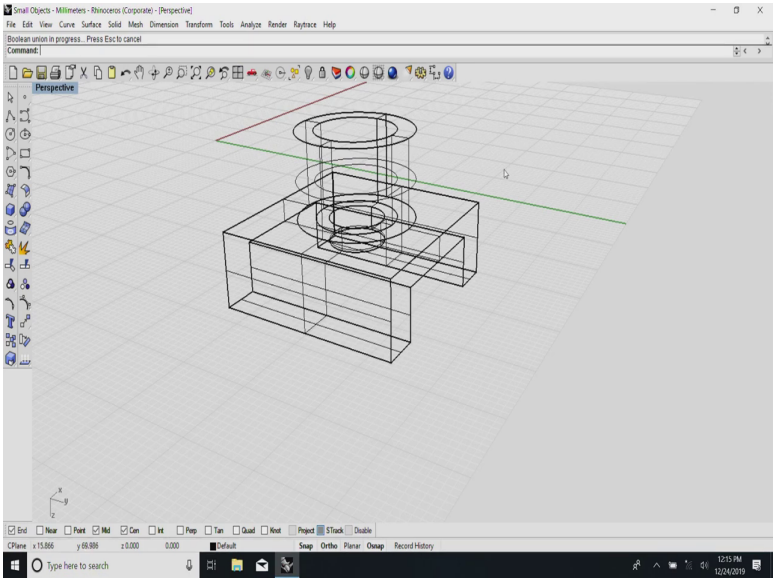


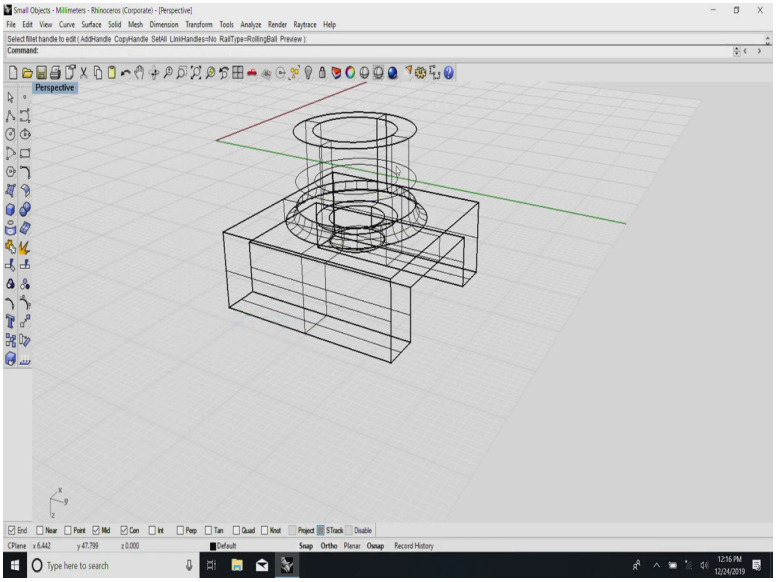
The welding operation that we carry out actually complete the Boolean, now it has become a genuine full-fledged solid.

(Refer Slide Time: 19:55)



Now, when I do the filleting operation see what will happen.

(Refer Slide Time: 20:02)



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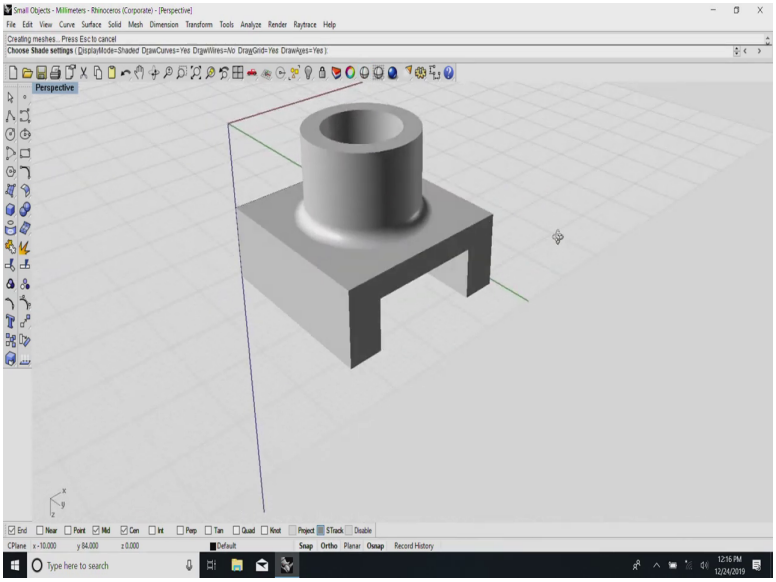
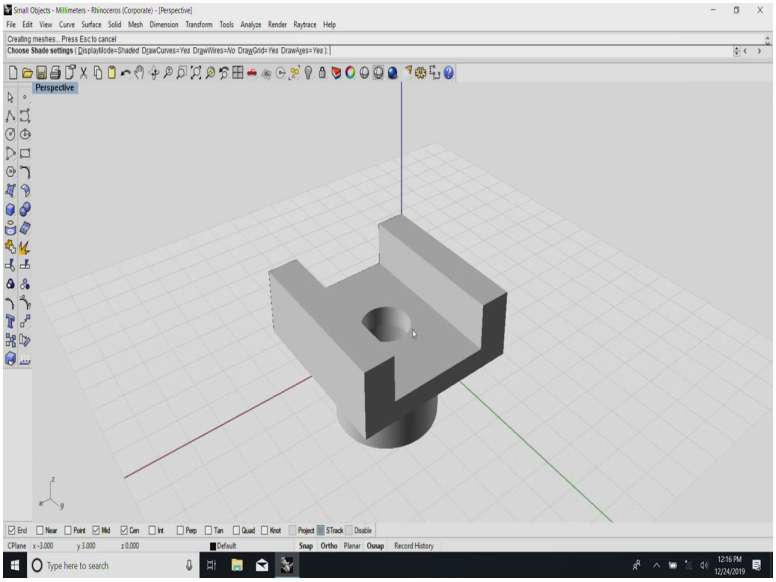


Now, my job with taking a cross section and then trying to what do you call weld it to this and then eventually drilling a hole is ready.

(Refer Slide Time: 20:19)



Except that is in this case probably this drilling hole is redundant I do not need this hole. You have seen that just to make it compatible with the earlier thing I have gone through these operations. Strictly speaking I do not need this hole in this case because I just have a tube here tube or a pipe and then I attach it to this.

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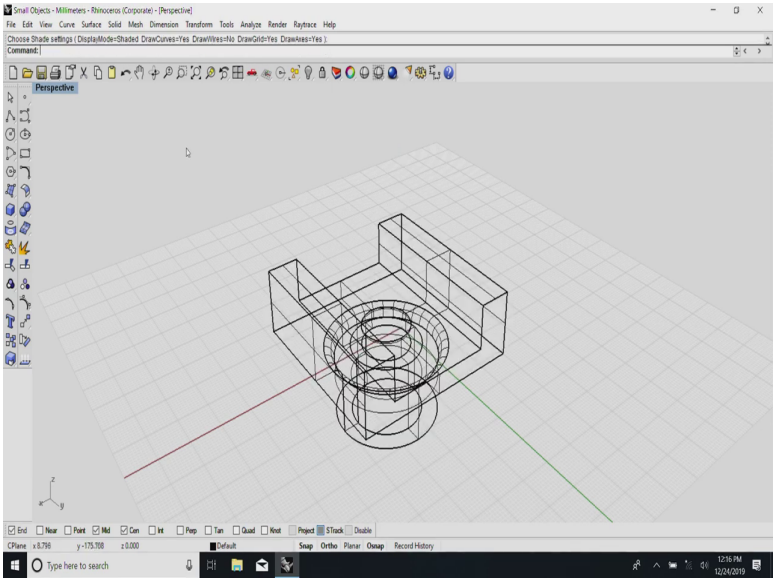

This hole was actually a process dependent hole which in reality I probably do not need this.

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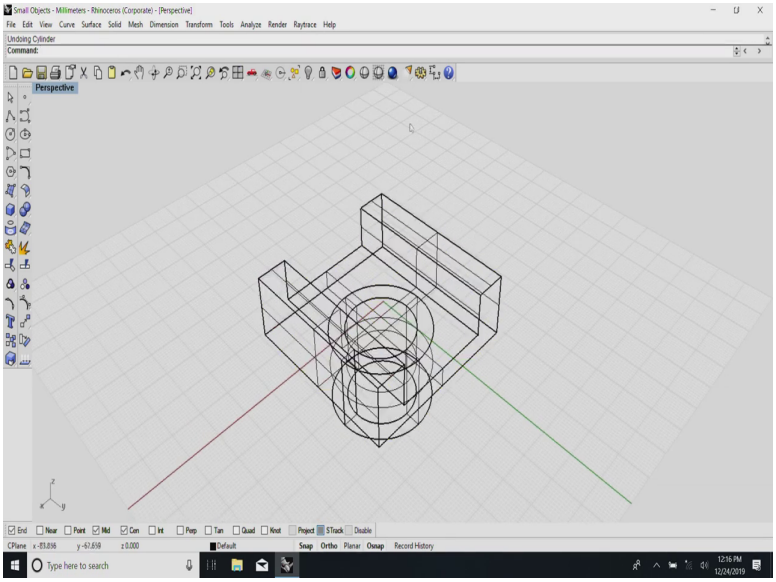

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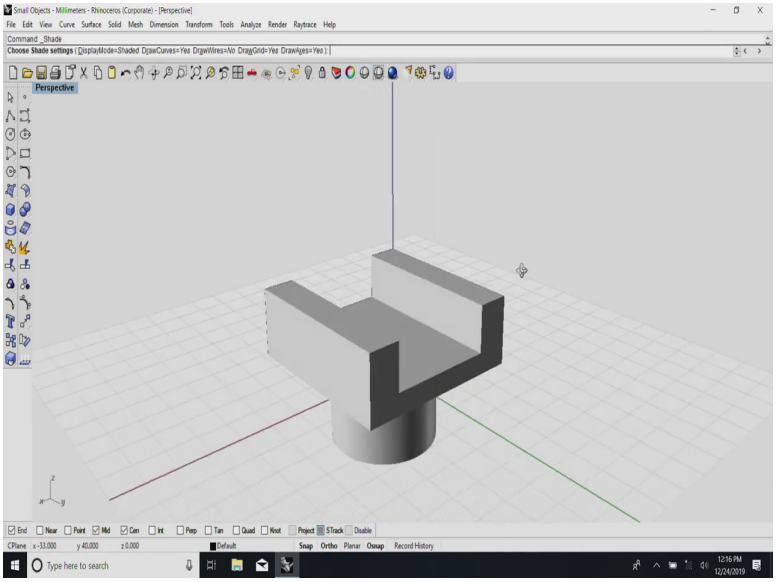
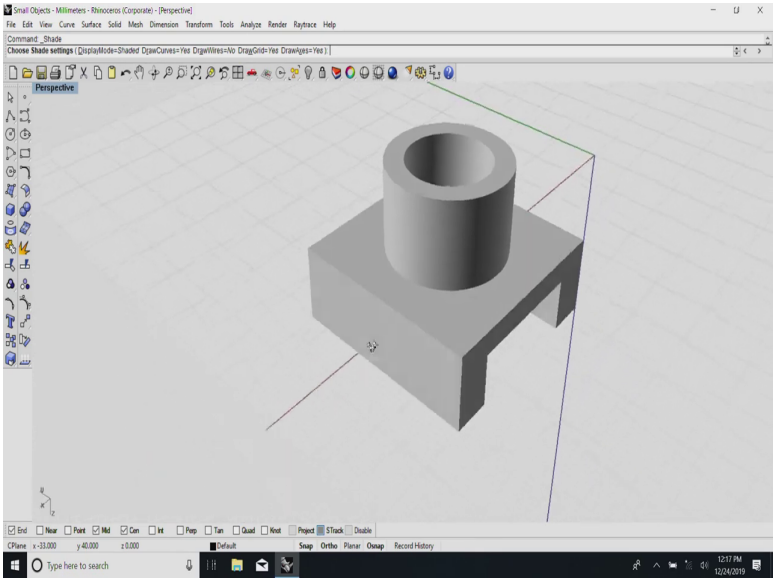


That is a where advantage is we can always this is all that is required a small section profile, then I have a tube welded to it and after welding the tube as before I try to see functionally.

(Refer Slide Time: 21:05)



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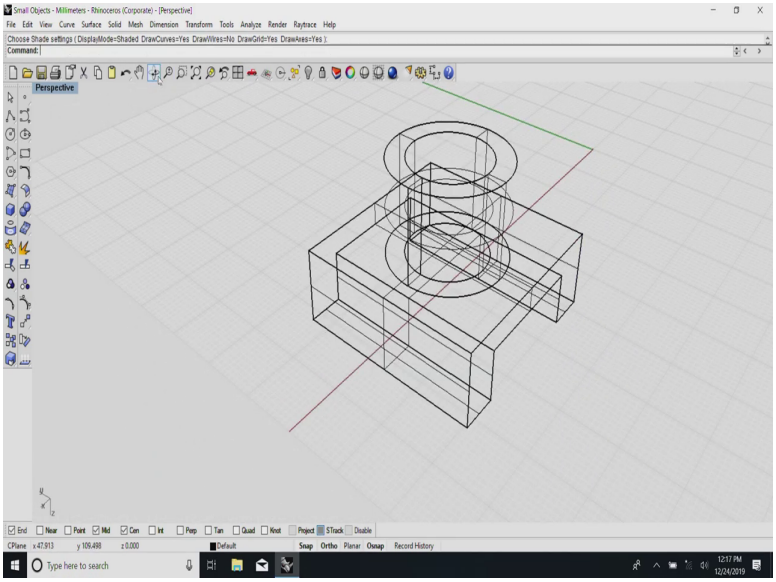

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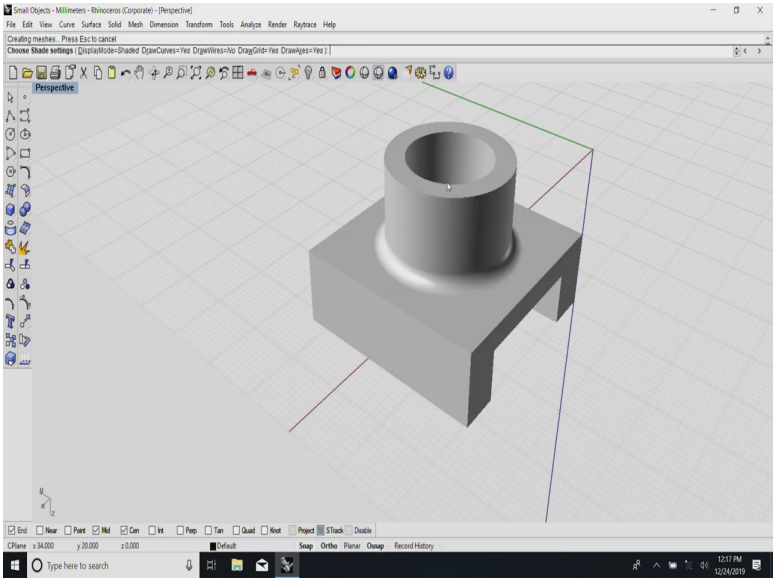

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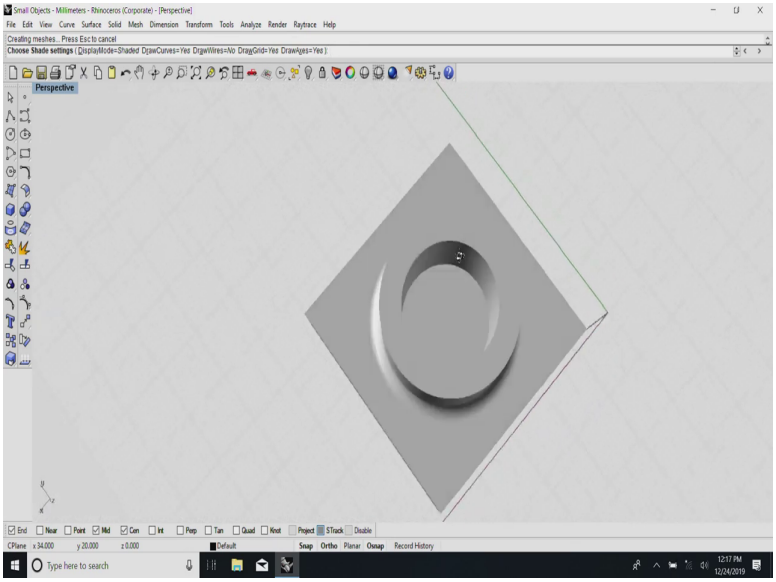

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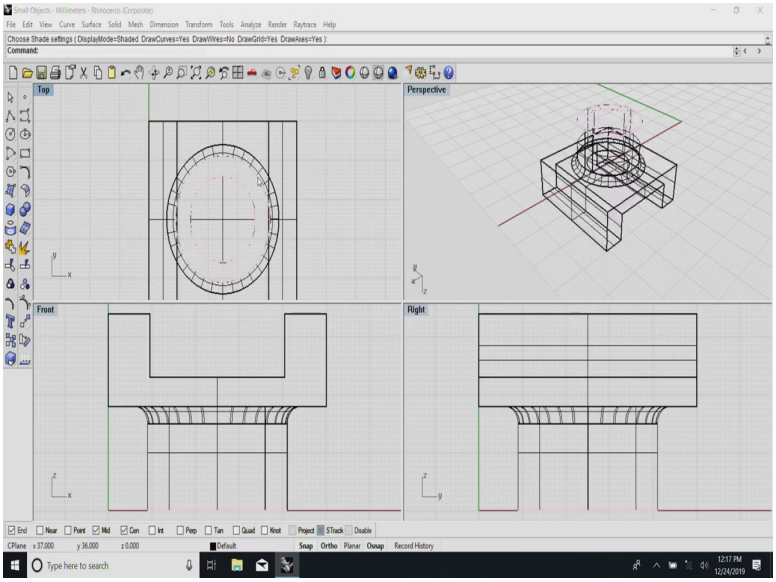

This is very good absolutely no problem which is exactly similar to the earlier job.

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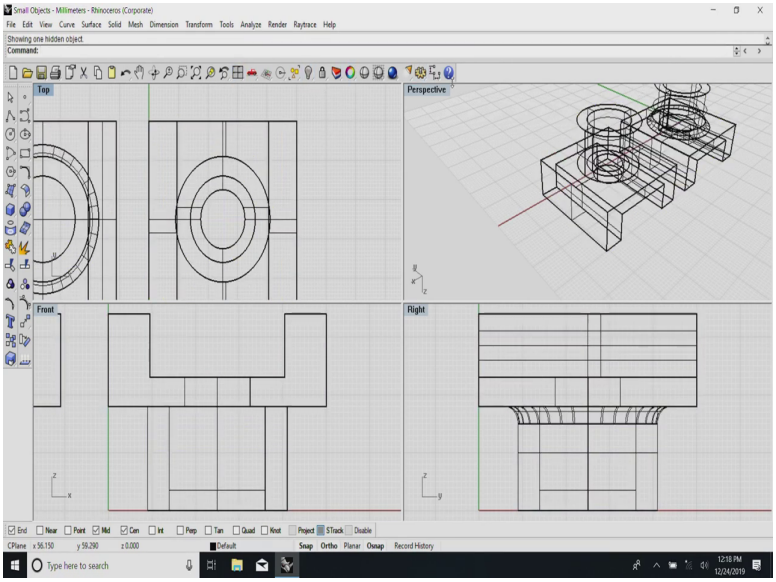
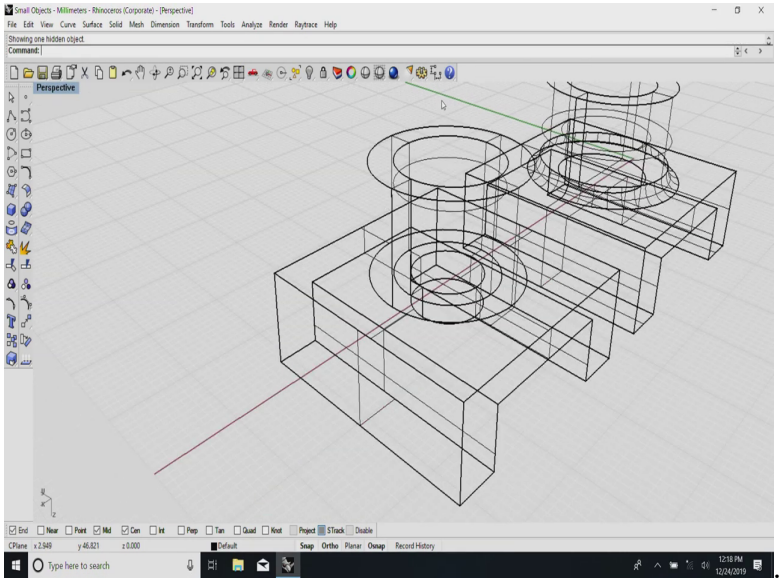


Except that, in this case, the fabrication has been based on at different type of processes.

(Refer Slide Time: 21:36)



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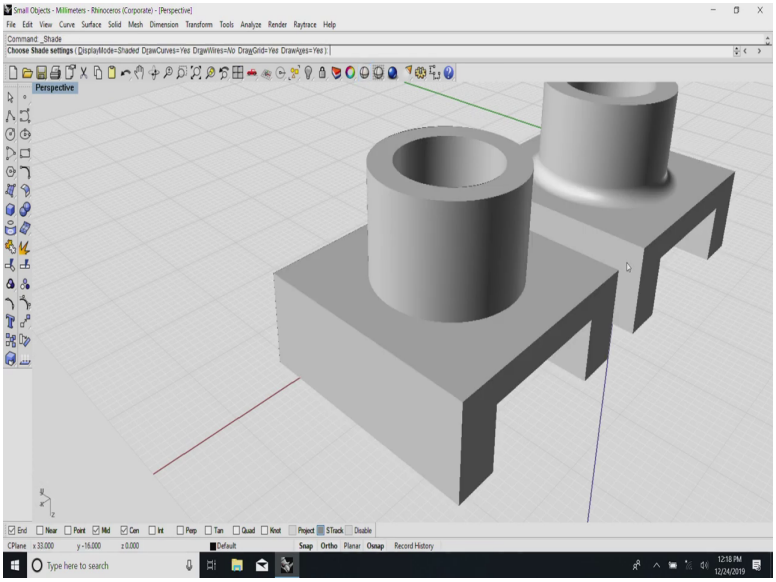
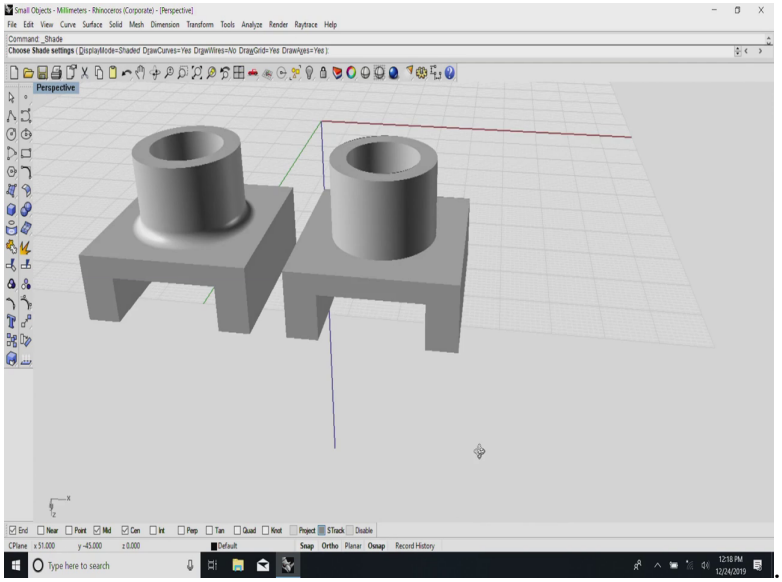


So, you see here both are about the same. This one has been done by milling. In the case of milling there still will be a small what you call fillet which I will try to show you because it is not easy to get into this. The milling part of it is on the groove on top side, bottom one is by turning it. What they have done is in fact, they have done it in the reverse they have first completed this turning operation by holding this flat in the in a forger check and completed that then held it inside and they have got it. You have seen this there both of them are a little exactly like what we are looking for.

(Refer Slide Time: 22:23)



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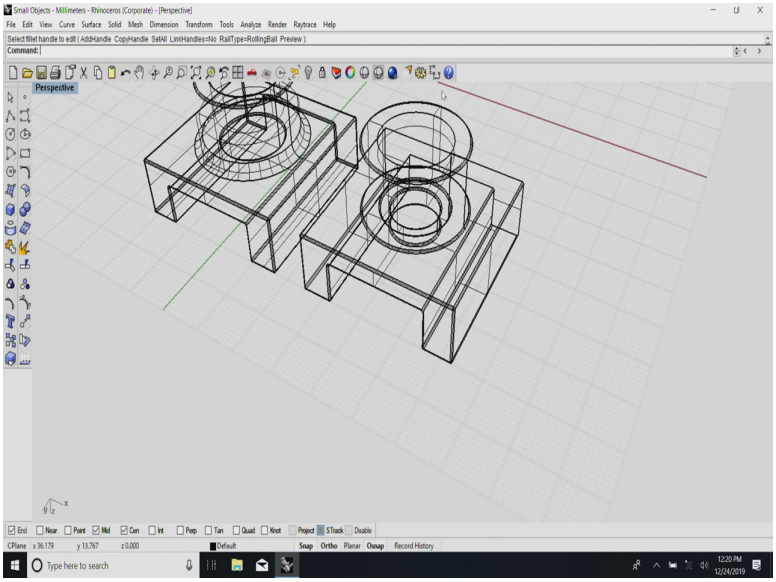
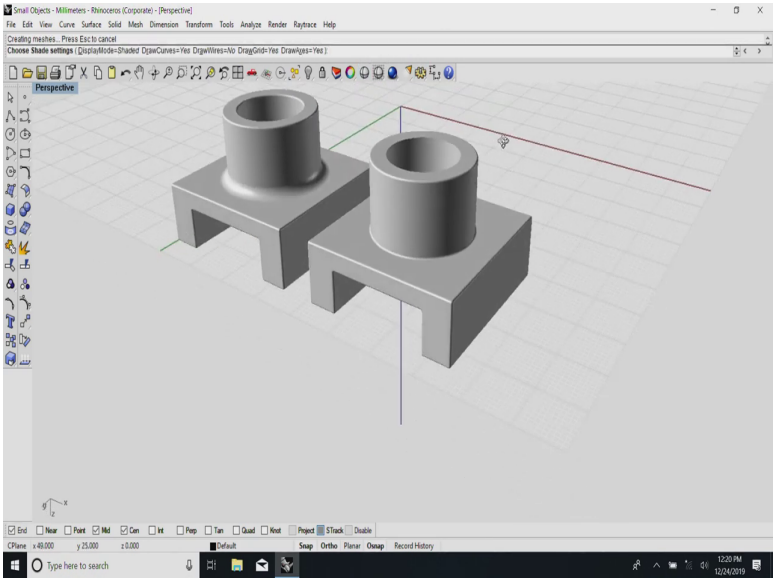


Now, to make the thing a little more presentable, I will fillet all the edges to make it such that as part of the rendering. The bomb is mine.

(Refer Slide Time: 23:12)



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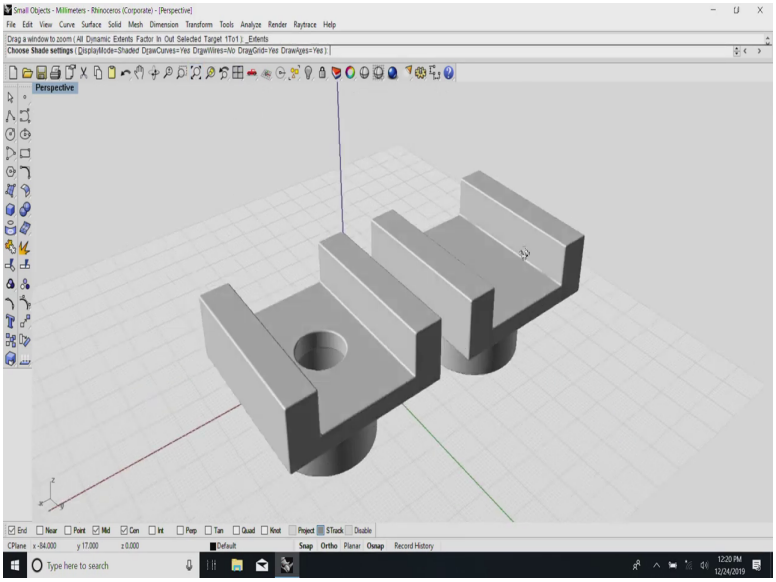
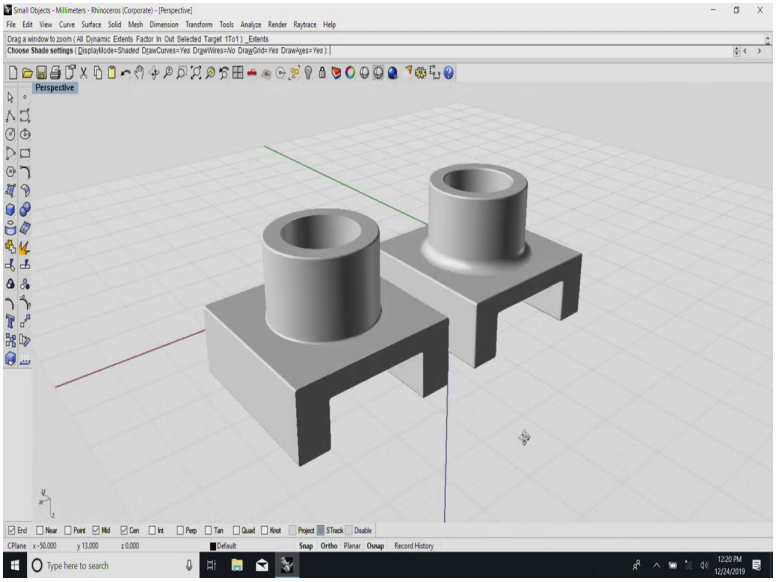


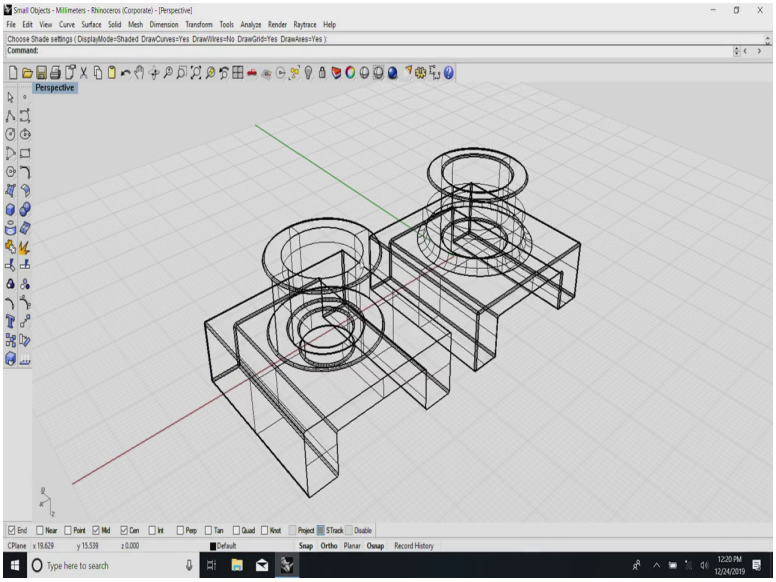
See here we have a generally very presentable object which is most likely this is how it is going to appear for us. It looks easy, is it not? Both are functionally equivalent. This process requires a small opening because we can drill and do this.

(Refer Slide Time: 24:14)



This process instead directly uses a profile which is extruded and then once it is extruded we take this pipe and weld it to that and then we have reasonably good stuff here.

(Refer Slide Time: 24:20)



You have seen this? We have two objects which are created from two different materials then what is the change which will come back to once again our drawing here. One of them the material I have mentioned saying a 50 mm by 50 mm square of length of 36 mm. If you directly take this it is valid only for one of the type of operations that has mentioned there.

The other operation was having there are two parts of it – the one part of it is the C profile. So, several c sections are available and generally C section some of them depending on they are cold rolled or hot rolled the dimensions and all are slightly different; cold rolled ones generally it will be very true. So, if this is the section there would not be any most of the time the these edges will be very square. The edges will be square in the case of a cold rolled sheet. In the case of a hot rolled there will be very peculiar profile.

Depending on the amount of precision required and depending on the condition and the tolerances we can directly use one of those cold rolled ones cut it to length, put it on type of another pipe just weld it all around. So, in the long run this much better than the original thing which I tried to conceive saying I take a sheet metal bend it into a U and then I take another what do you call something else pipe and attach it to that. This is where when you create a solid model if you create it properly the first time it can be used for all other operations.

So, if we pass it on down the line they will never be any surprises and you will not make many enemies when you look at it.

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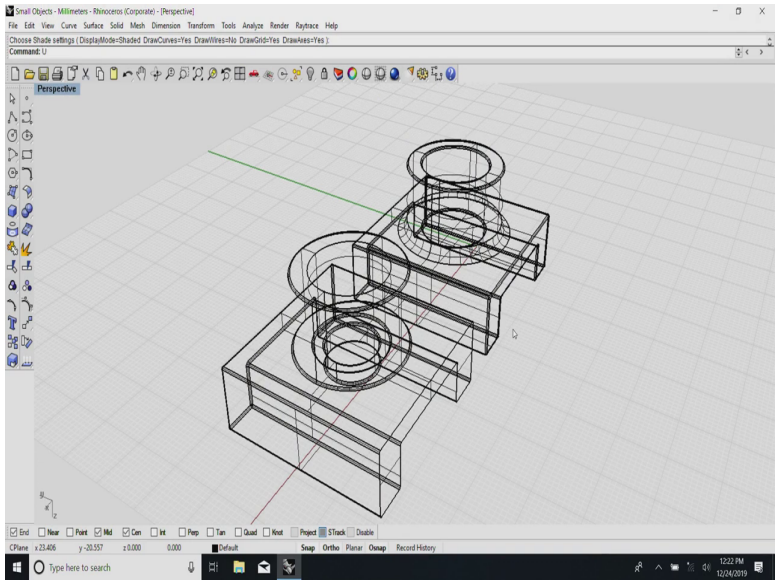

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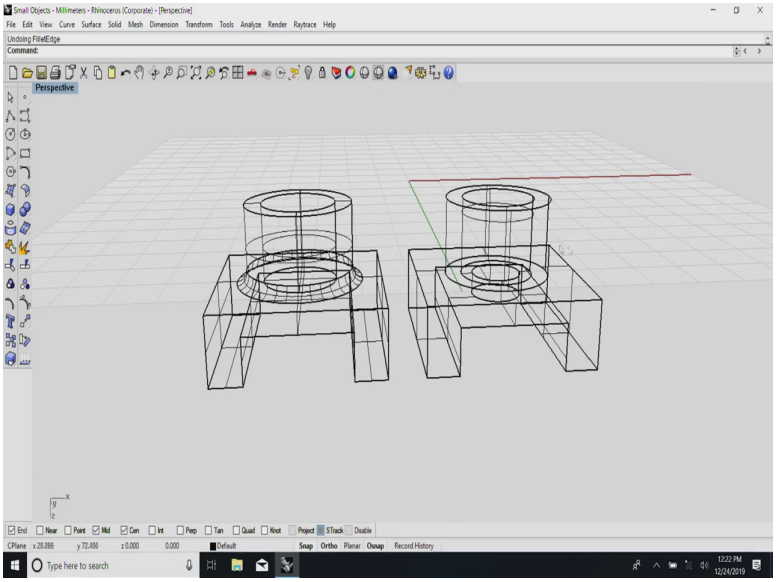

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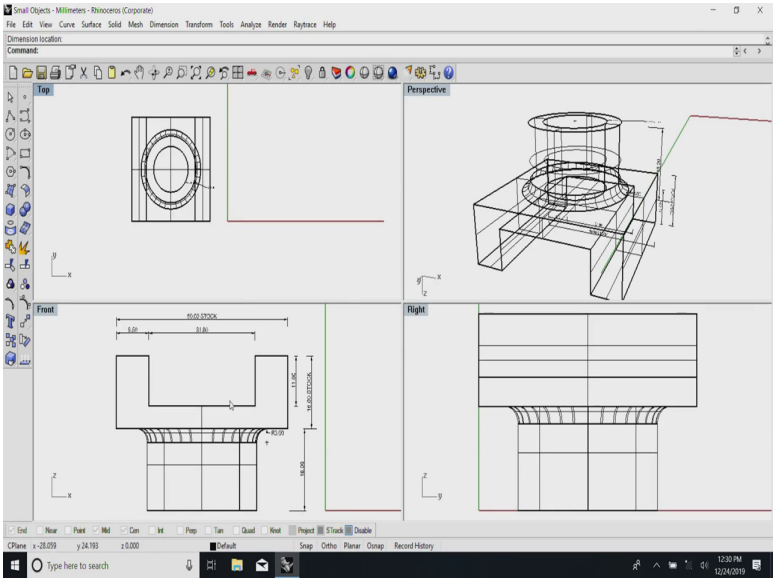

Now, let me do a little bit of if you see here we end up with having two dimensions parts.

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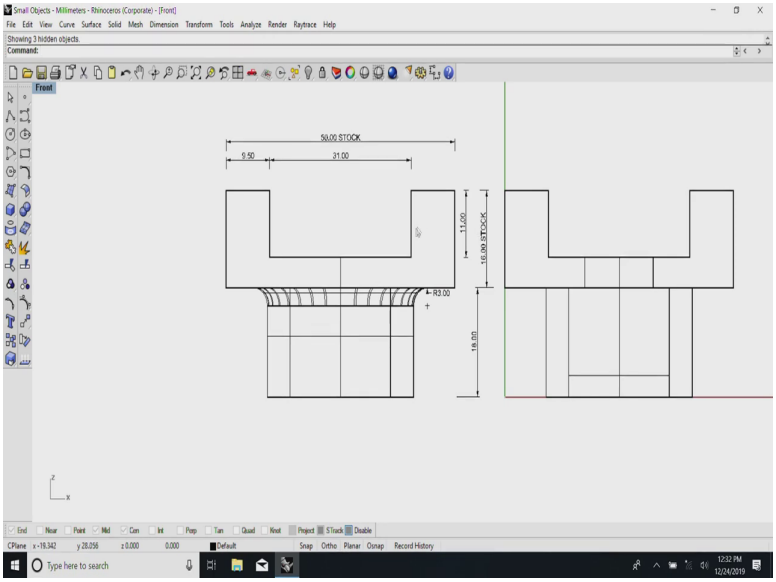

Now, we come to the all important aspect of how to dimension this parts. Fortunately, for us this is where these what you call CAD packages are useful in one way. This is where the CAD packages are also a little bit of a problem in some way. One of the obvious advantages is we can directly pick certain features here and then try to have them dimensioned automatically, but they still those dimensioning conventions which you have learnt earlier are still valid at this point.

I will try to give it a what you call a small attempt for the matter of convention I switched off the whole grid and allow me to see whether I can it is possible for me to dimension is. So, as explained to you earlier we need to give the smallest first here. You see here it shows it is a 9.5 millimeter limp here, and again keeping with the same convention I will show a groove which is 31 millimeters which I have shown you.

I will rise it a little above so that it is easier for us to understand both ways are equally good one you can keep both of them in line. So, we know what it is and finally, comes to the I have tried to keep whatever I have showed you saying I take a stock material of 50, then I add this what you call one limb here and one limb here. Here there is a provision here, see here something has got added in that saying this is the basic stock material meaning we do not need to inspect it and carry out any other operations on that.

Same thing here now, we come to the depth of the groove and the height of it. Though it is convenient to directly show it here by another what you call convention rarely we show anything inside the object. I will come back to it later because other features it may interfere with the other features about it. Now, this clearly says take the stock material of 50 mm by 16 mm which is instantly also the original stock material same here.

We just need to cut it to length and afterwards make a 31 millimeter by 11 millimeter groove inside. Depending on the method of operation it is it may be needed that I show this left dimension or it may not be needed that I show the right dimension. The necessary dimensions are all already have been extracted from the drawing here. Now, I will come back to the other thing you see again at the bottom we have something else here. At the bottom the total length of the object also needs to be given.

See here it typically shows the length of the object and there is no opening here and then we again have this radius here. I will see if it is possible to do the radius here. Yeah. Anyway, I am not able to push on it properly. It is if I play around I will get these things which now brings us back to these two openings here you have seen this here I have an opening here and I have a opening here.

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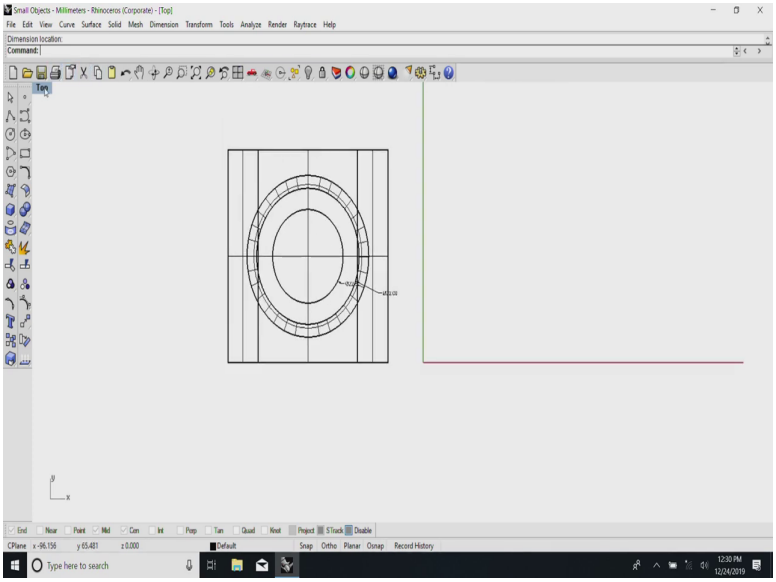

Now, depending on the type of diminishing features the things that are available it is very much possible for us to indicate the other dimension directly here. So, by carefully drawing to place it the way we wanted it is possible to indicate all these things.

Now, one of the things you will notice is all the features that are required here are mentioned here all the features that are required here are mentioned here depending on as I have said earlier my job is very much ready. Got my point so far?

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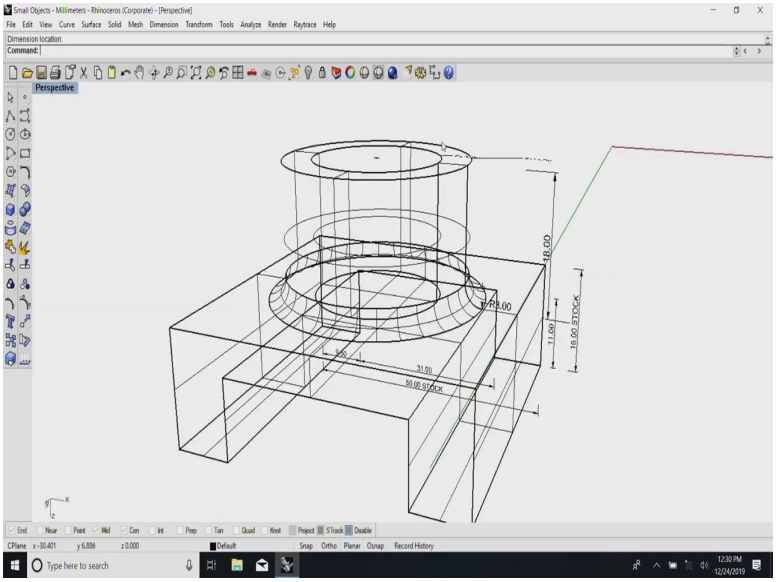

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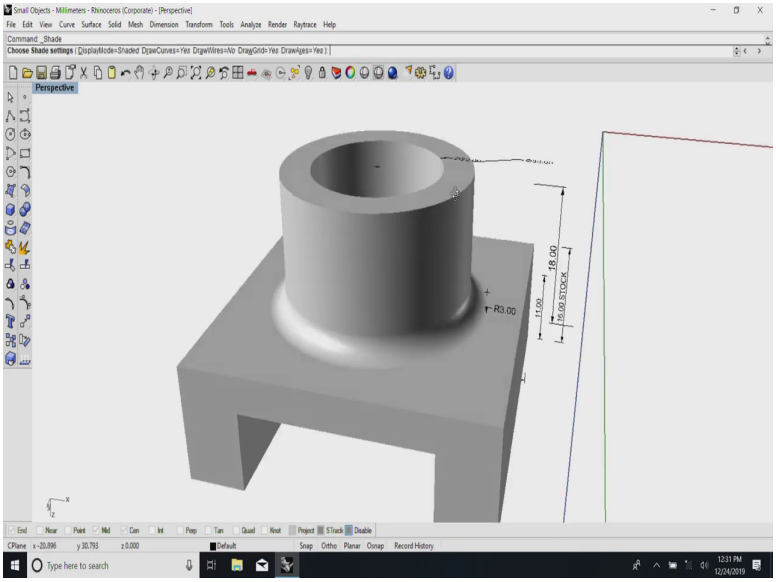

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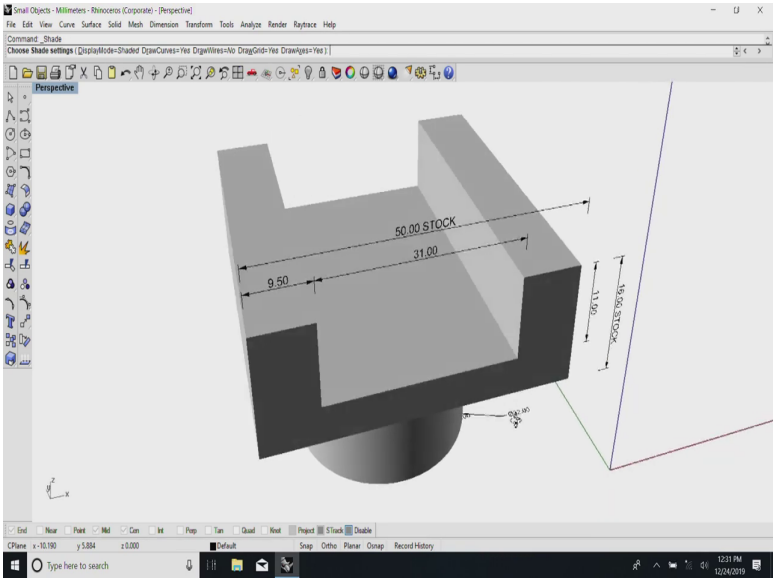

This particular what you call a package which I have managed to just to download the trial version does not do a very good job of it.

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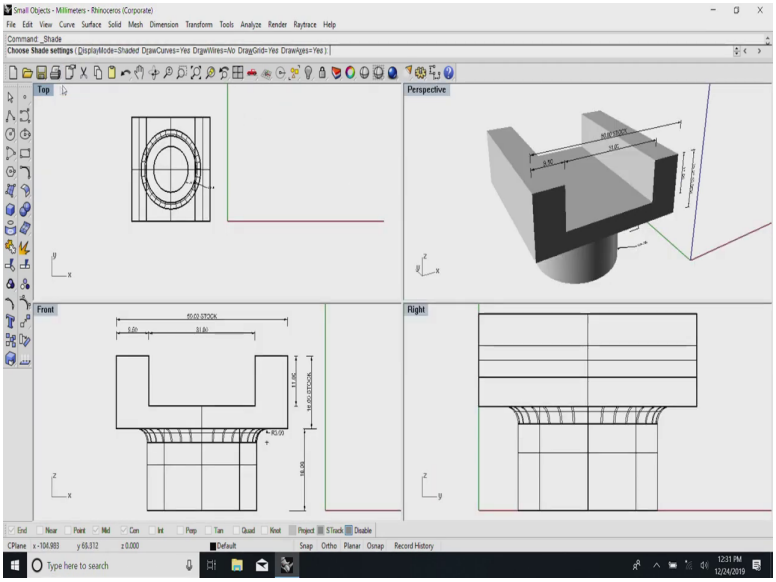

However, when you take a print out it is possible for you to make a 4 view drawing and get all these features all into one drawing as you like. Why I have started with this is now let me go back to let us say I need to modify something here.

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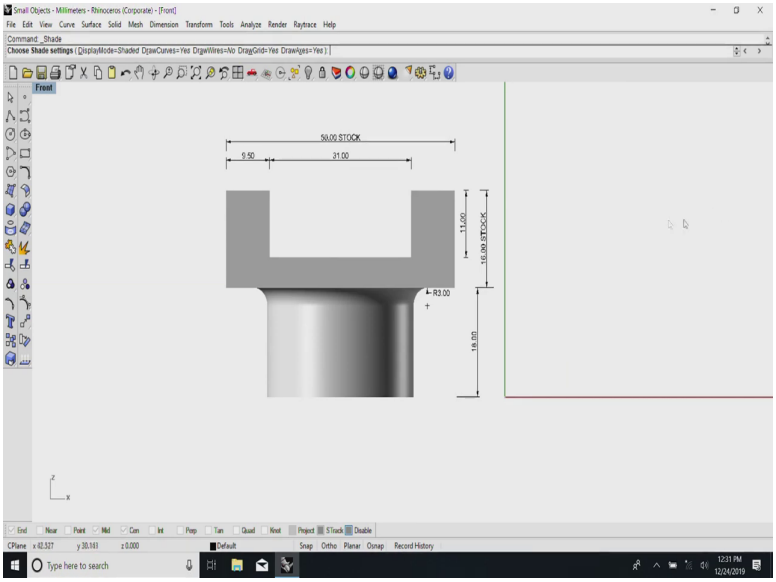

If I need to modify something here, this has been a 3D solid as I have told you earlier. It is not easy for me to actually get the other modifications which I am trying to do because the that is a shortcoming of this particular package.

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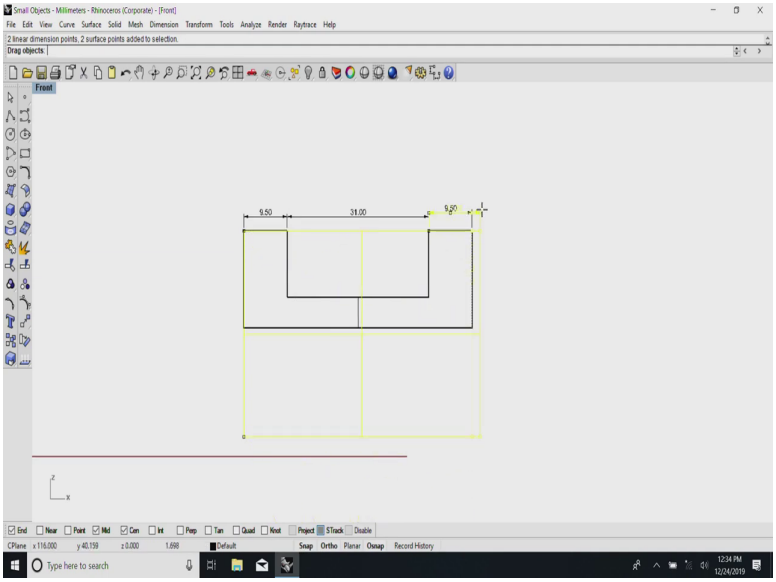

Now, instead, I will go here and see whether I can alter these things here. If you remember I have dimension and it they showed me 9.5, they showed me 31, they showed me 9.5. Once if you have made a drawing of one of these profiles advantage of using these any of these CAD packages is I can easily. See what I have done. I have modified this such that it is stretched itself across and in this case I have not taken the taken care of the dimension.

So, now, I will take care of this dimensional also and now, all the defining points of the dimensions are also there. See, what has happened total length has changed and proportionately the width of this has been changed.

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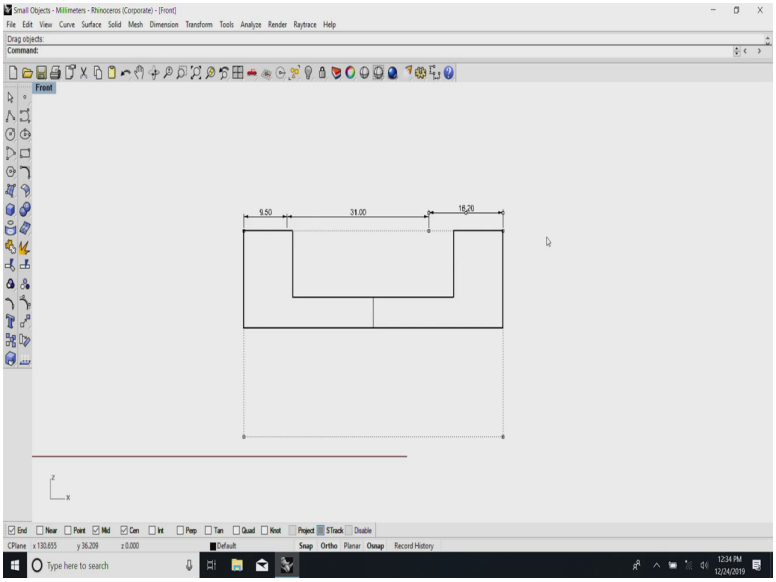

I can avoid this by one more time by appropriately selecting these control points which are there. If I want to only the total length to change I need to select the various other things like this.

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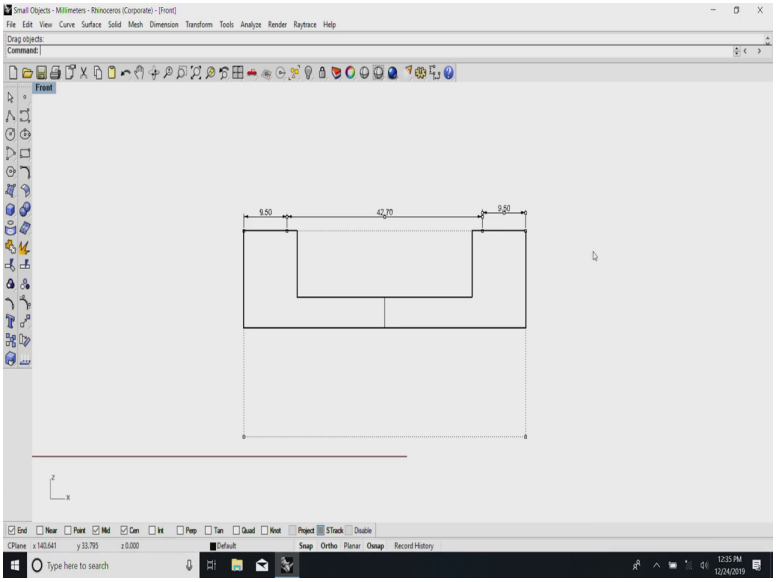

And, you see here if you see that 9.5 has not changed, but the other features have changed which is related to this. This has happened because it is a surface and surface you cannot really pick all the points. I will continue with this what you call exercise in making we will take a break here.

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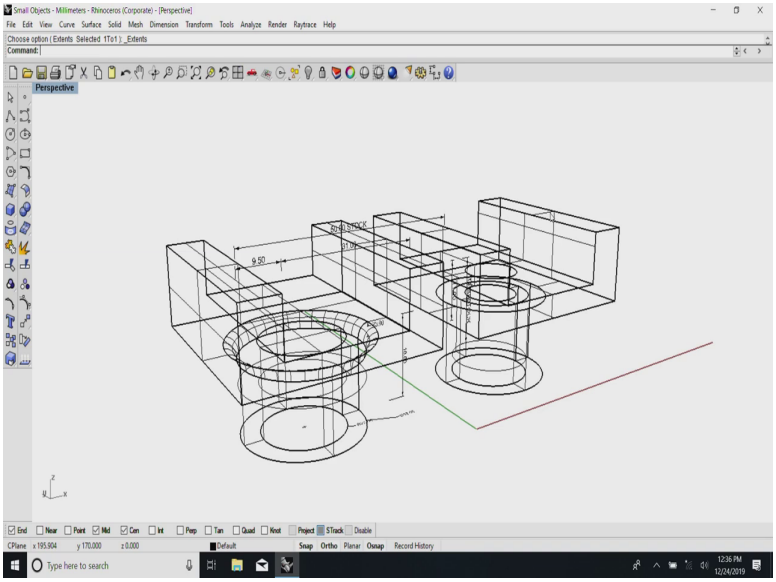

I just wanted to demonstrate you the main advantage of trying to use solid modeling for us to conceive of a actual object which is useful. What looked like a simple bush which can go on top of another thing obviously, there is so many ways of making it. One of them in this case we have taken a stock square fifty mm square it would have been possible for example, I can take a stocks stock round which is a maybe around a 37 mm to and try to just cut a groove through that that will be equally useful product for us. In that case various other things including trying to make a this various other settings and all are probably saved a lot.

So, I will take a break at this point we will meet again. So, thank you what I have covered so far is how to have a concept of what an object is and then different ways of making it. There in this case I have shown you actually two ways of showing it here compared to my original thing. And, finally, even at this point the moment it comes here as we develop, we will notice

that we can take a extrusion and then on that we can take another pipe probably weld them together, drill holes and get the whole thing going.

So, thank you. I will continue with this in the next session. So, thank you.