Engineering Graphics and Design
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Week 10: Part Modelling 2
Lecture 4
Drawing from Solid Model

Welcome back we are in week 10 Part Modeling 2. Until now we have been looking at how to develop solid models. But in this lecture we will look at once we have developed the solid model how to extract the 2D drawings of this solid model. In this lecture, let us look at how to get these drawings once we have the solid model.

So, for this inventor software has a drawing environment altogether just like how it has a part model environment. So, we will be looking at that environment and see what are the tools we can use to come up get those drawings.

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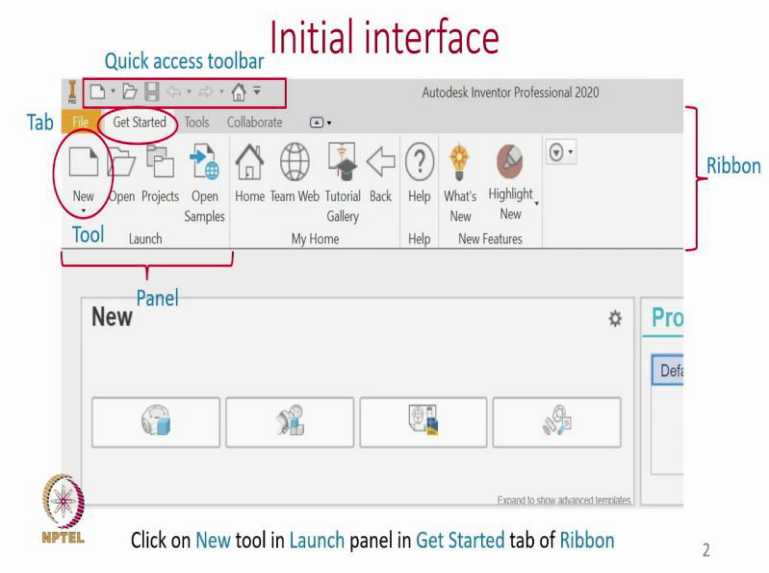

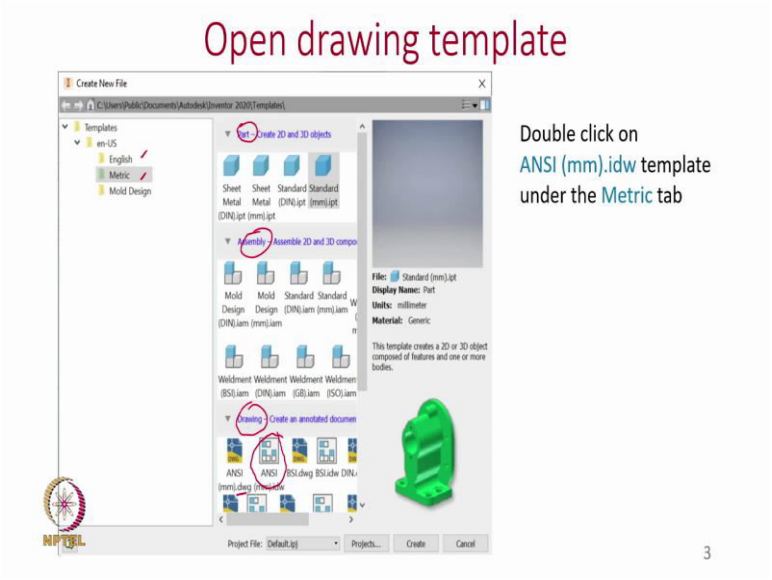

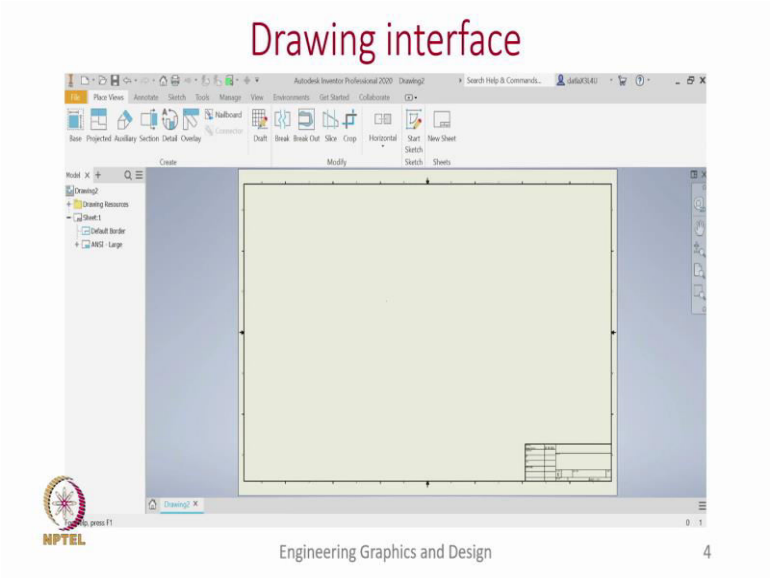

So, this is the initial interface of the inventor software as you all know by now. So, we will be looking at how to get started into the drawing interface. So, just a quick review of what we see in this initial interface, we see this quick access toolbar at the top and then we have this ribbon. So, within the ribbon, we have different tabs like file, get started, tools, and collaborate tabs. For the moment we are interested in this get started tab which is highlighted here.

So, within this get started tab again, we have different panels like launch, my home, help, and new features. We are interested in this launch panel within that if you click on this new tool that is when we can get into the choosing the templates. So, it will open up a dialog box which has a list of templates. So, as you can see to the left in the browser we see these different units the English units, metric units, and within once I choose the metric units it asks

for which kind of template I want to create. Is it a part template, an assembly template or a drawing template?

For now, we are interested in this drawing template. So, within which again we see there are several other templates. For now, let me focus on this ANSI mm idw, idw is the format it saves for the drawing files in inventor. You can also choose . dwg format that will make it compatible with the AutoCAD softwares. So, let us double click on this ANSI mm .idw template under the metric tab, that will take us to the drawing environment.

And this is how the drawing interface or the drawing environment looks like. So, at the start, it will show you a empty sheet. Of course with the border and the title block and the drawing area is empty. Let us get to the software from now on and see how we create these drawings.

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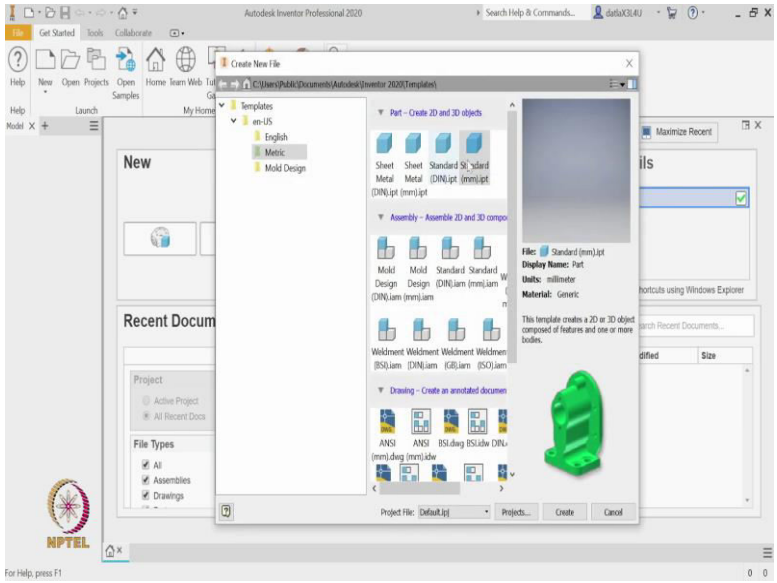

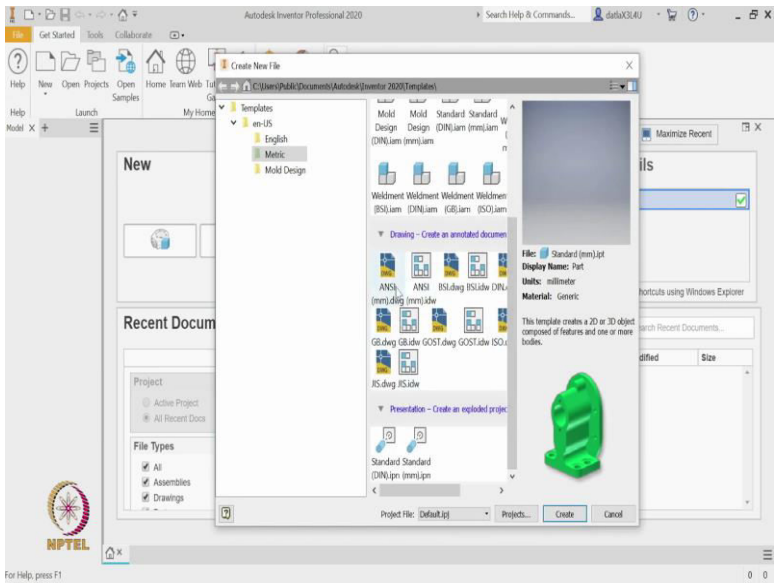

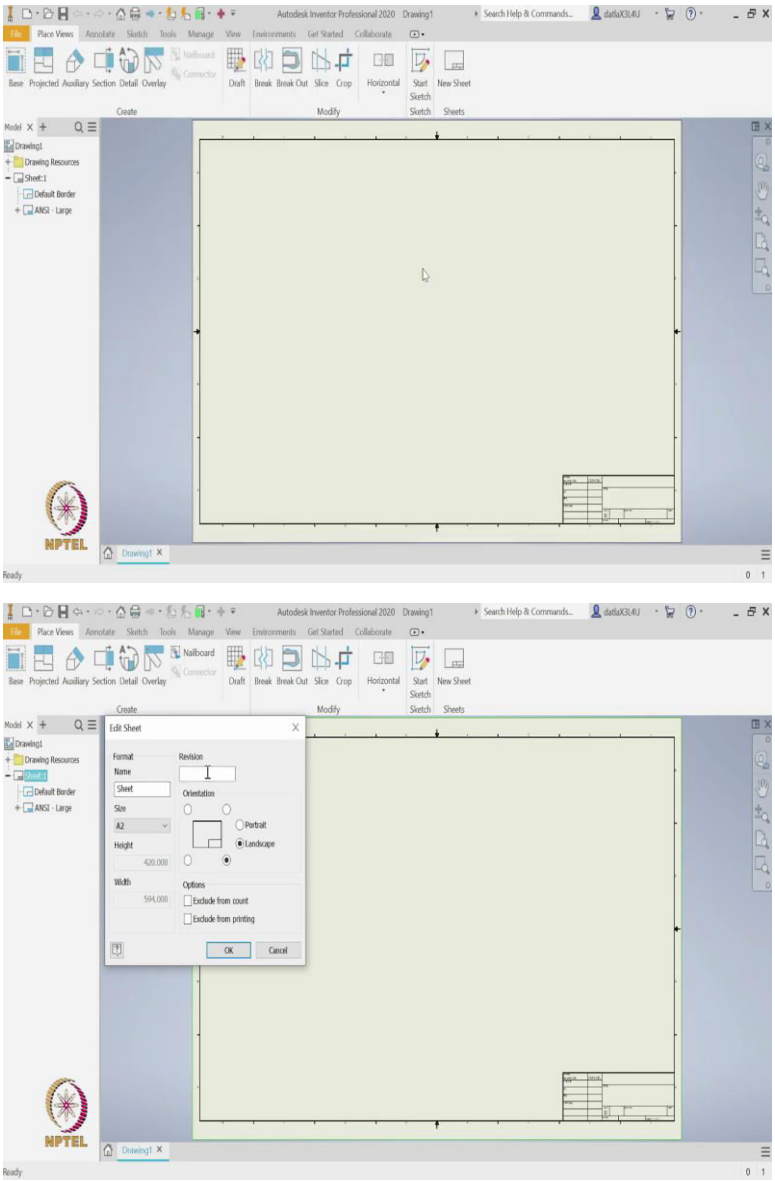

So, we are at the initial interface like we just discussed let us say we have to click at this new tool. And then once we go down into the drawings, we see that there are different templates in ANSI, BSI, DIN depending on the standards we work with we can choose any of these. But now let us go ahead with this ANSI mm .idw template. So, when we are now in this template; to the left, we see in the browser it started with sheet 1. If you right click on it, you can go into the edit sheet. In the Edit sheet we can choose what is the size of the drawing we need to work with.

So, by default it shows size D. But since we are familiar with these A4, A3, A2 and all. Let us start with this A2 drawing. So, I will choose this or there are several other options you can choose from. You can name the sheet you are working with. And of course you can also choose the revision numbers you are working in, let us say in an industry where you revisit the designs. Sometimes you need to also mention the revisions of these drawings.

And then you can also choose whether you want this sheet to be in landscape or portrait. Currently it is in landscape. We can also go with the portrait and then there are options of where to place your title block. As of now it is in the bottom right. But yeah you can choose any of the 4 corners. But let us stick to landscape and title block to the bottom right and click okay.

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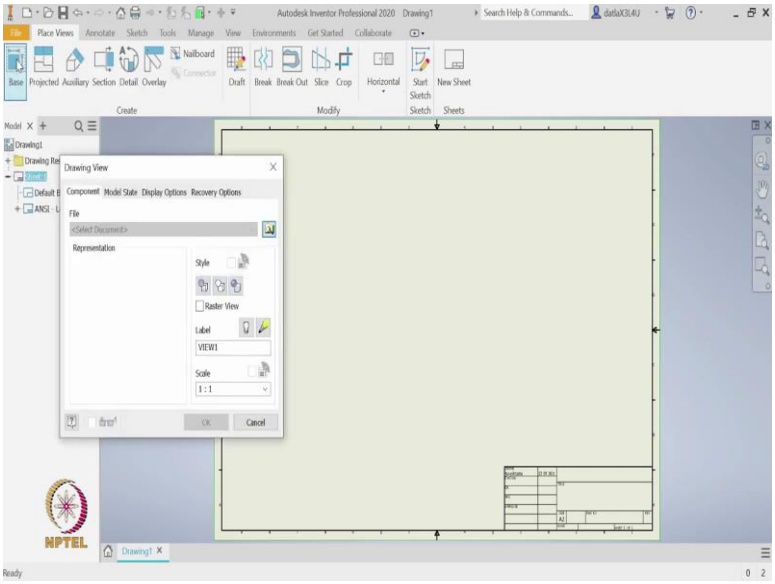

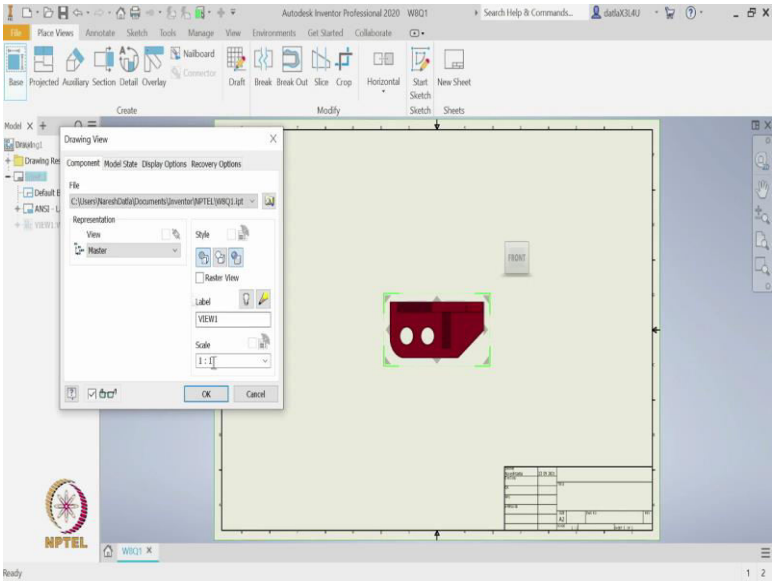

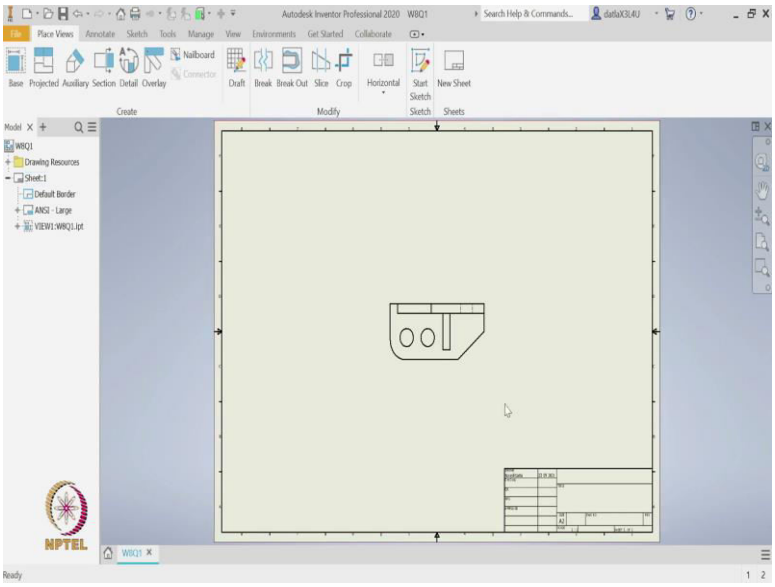
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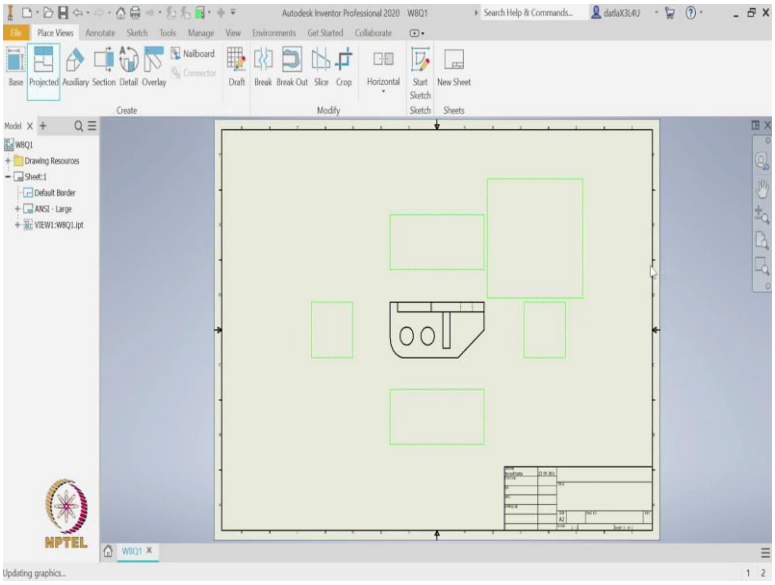
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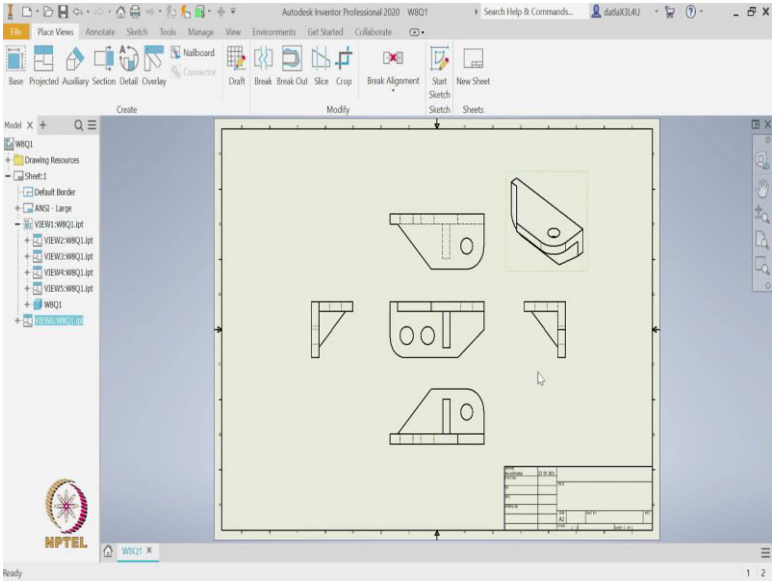
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So, now it has a readjusted the sheet says that it is an A2 sheet. Now, when we try to do the drawings there are two essential steps we do. First thing is to bring the views that is done using this tab of place views where we bring the views whatever it is so it could be normal views like front views, top view, left and right side views and other normal views. We can also bring in the auxiliary views, section views and the details.

Once we have the solid model we can extract any of these views. So, that will give us all the views. But once we are done with bringing all the views and then we need to give the dimensions that is when we go to the second tab, which is the annotate. That is where we have all the options to give specific dimensions and mentioned details about those drawings. Let us get started with this place views.

The first thing we do is to select this base. So, first, let us start with this base view. So, once I click on that. So, it opens a dialog box within which I can open the file solid file. Let me choose this W8Q1 file. So, once I double click on it, it is showing me a preview of the solid model that I have already created. But let us look at there are few more features in this dialog box. Here we can choose what scale you are interested in.

So, as of now it is 1 : 1 I can choose a enlarge scale of 2 : 1 or smaller scale. But looks like 1 is to 1 should be good enough for the A2 sheet we have chosen. We can again name this rename this view as by default it is saying it is view 1. There is also option about styles. There are 3 options. The first one is with the hidden lines. Usually most drawings we have the outline as well as the hidden lines.

But if you choose that there are too many details and you want to remove those hidden lines we go with the second option where it says hidden lines are removed. And last option is the shaded meaning that it will show you with like a it appears like a solid model. So, we usually go with the hidden line by default. But later you can choose if you want to go with an other option. Now it has gone with the first option where it shows with the hidden lines the base view.

So, here when we said it is the base view how do we get the remaining views for that we need to go to the projected. So, once we go to projected it asks where is your base view. So, let us select our base view. And once we select the base view depending on where you place the cursor it either shows the now my cursor is to the right of the base view. So, it is showing me the right side view. If it is to the left it will be the left side view. If the cursor is above, it is showing the top view and if it goes below it shows the bottom view.

I can also get the isometric drawings by just taking the cursor in the diagonal sense. So, I can place the isometric view here as well as here in any other direction. But let us stop with one isometric view I can click right click and then select create. So, then it will create all the projected views we asked for. So, now if you look at the browser first it has created this base view under this base view, under this base view it has created the other views.

So, right side view, left side view, top view, bottom view and for the isometric view it also created but it is different from the base view. So, essentially all these projected views are connected to the base view. If I make any changes to the base all these projected views will also change. So, let us delete few of these because maybe we do not need all of these.

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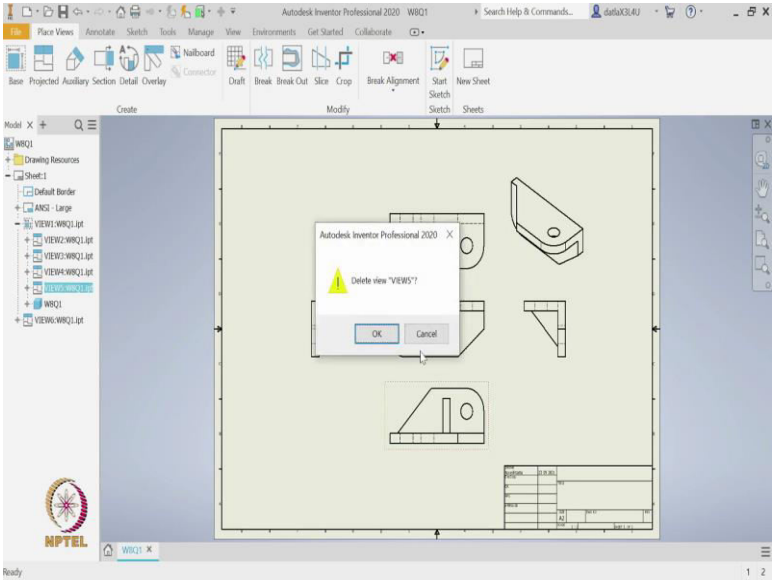

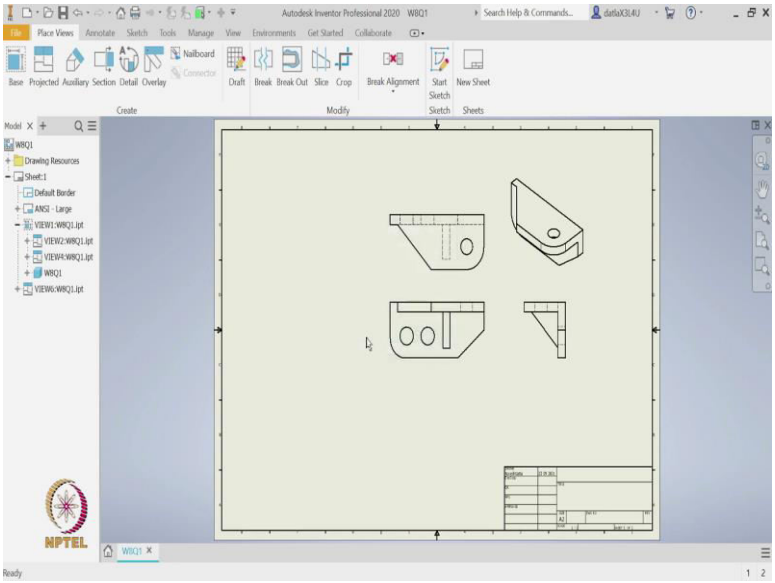

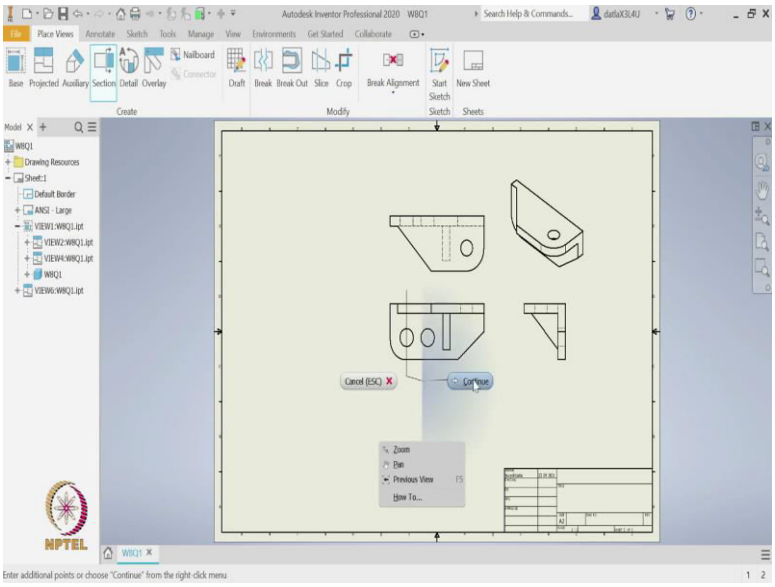

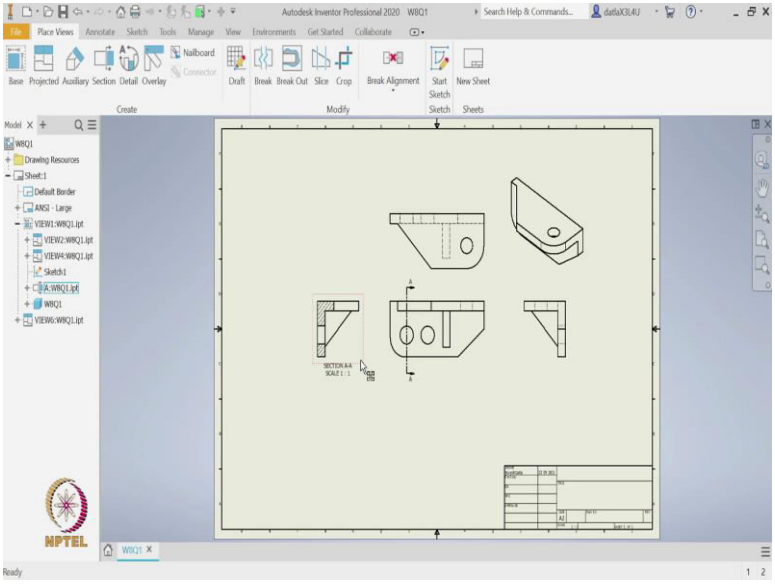

So, let me say I will select this bottom view and select delete. Similarly, let me select these and again delete. So, let us see how to now do the section views. For that, let us select this section. And then it is asking where do you want to draw the section line in which view. So, let us select this base view where we want to draw the section line. So, once I select you see that the cursor has changed to this + sign.

So, let us say now I want to draw this section line from here to here and then right say continue. So, once I say continue, when I move the mouse to the left you can see the arrows on the section line keeps changing whether my mouse is to the left or to the right of the section line. So, let us place it to the left. We can see that it has created the section view with all these sectioning's are also given. And it has named it as section AA of scale 1 : 1.

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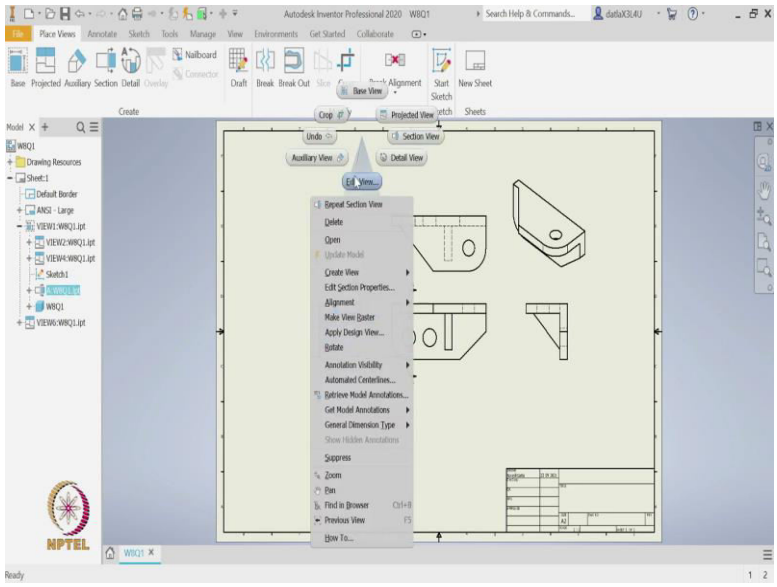

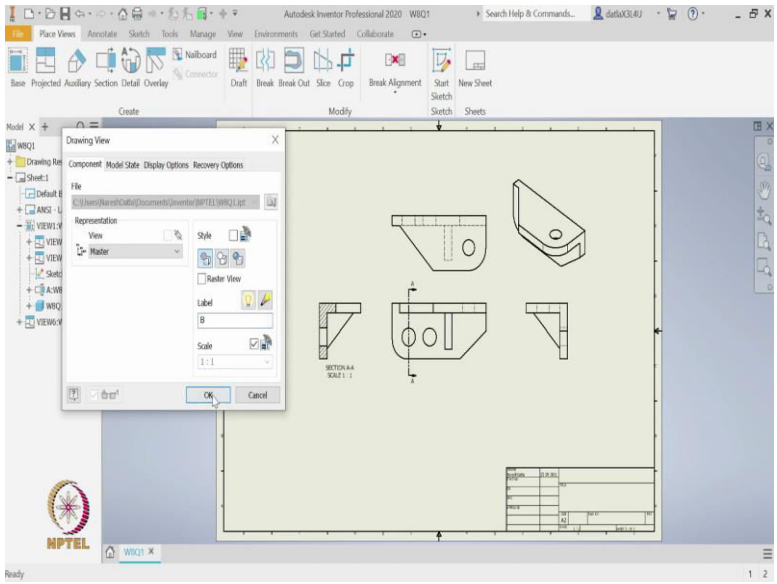

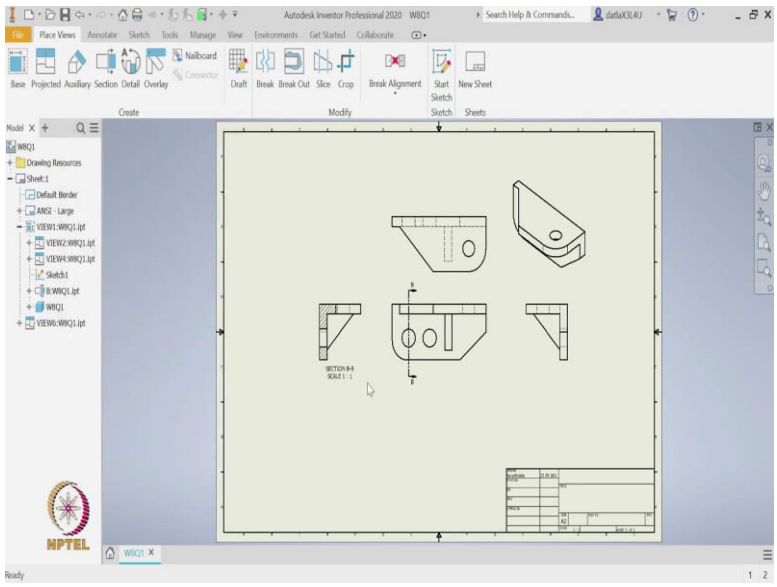

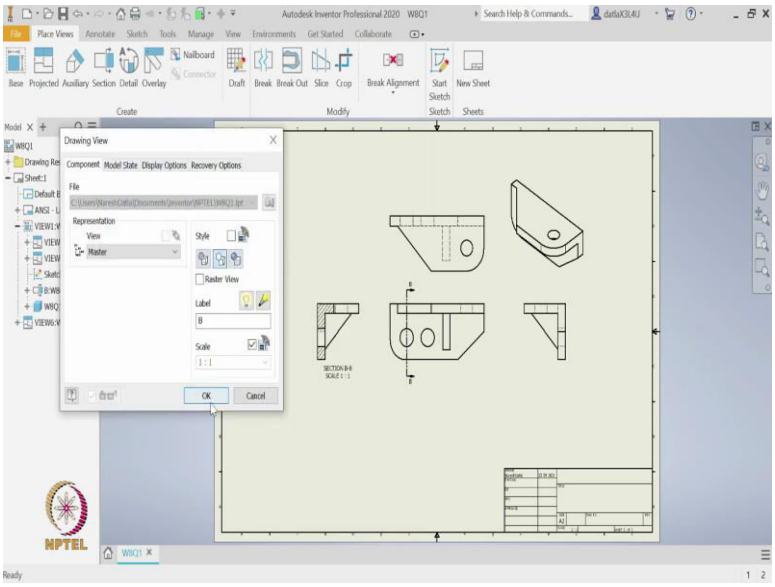

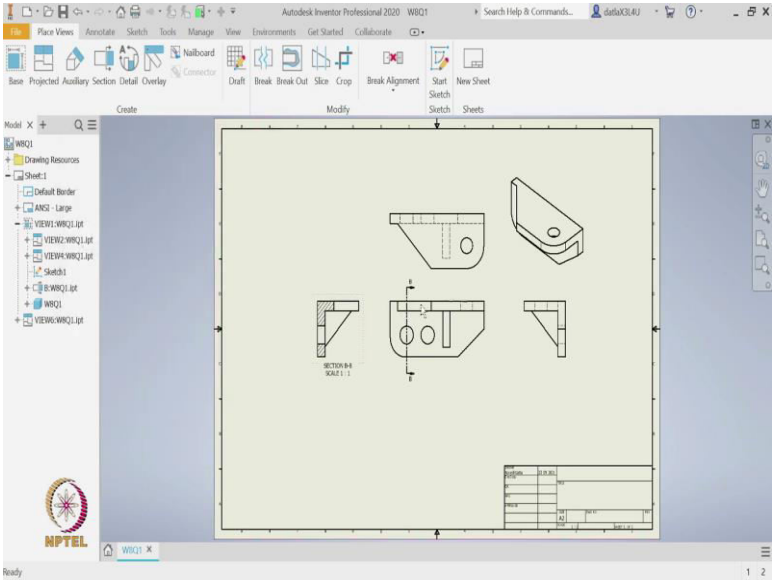

Let us say you want to edit this section view, we just need to go there right click and say edit view. So, by default it is section scale is 1 : 1 because even the base is at 1 :1 but this level of AA let us say I want to change it to B that you can change it here. And here by default first section it is choosing this style where it removes all the hidden lines. Usually that is what we do. But let us say you insist on showing the hidden lines you can always do that. And click okay. But let me remove those because that is not what we do usually. So, let us go with the second option of without hidden lines.

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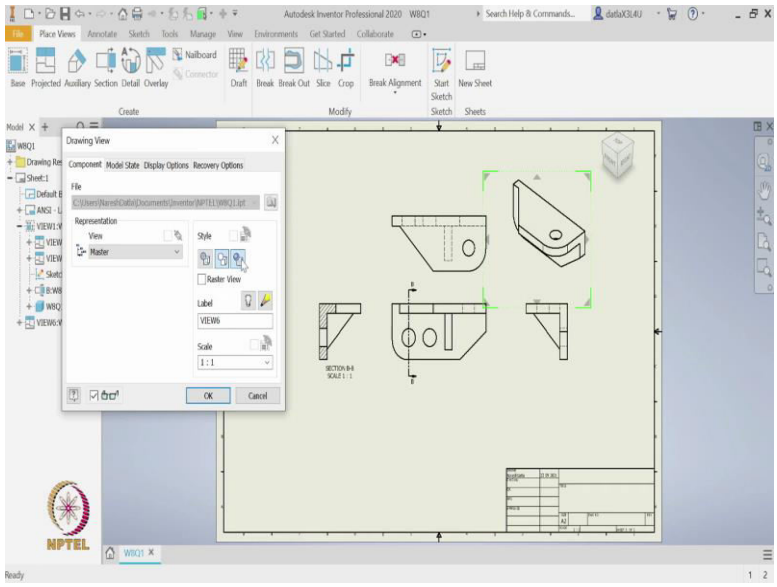

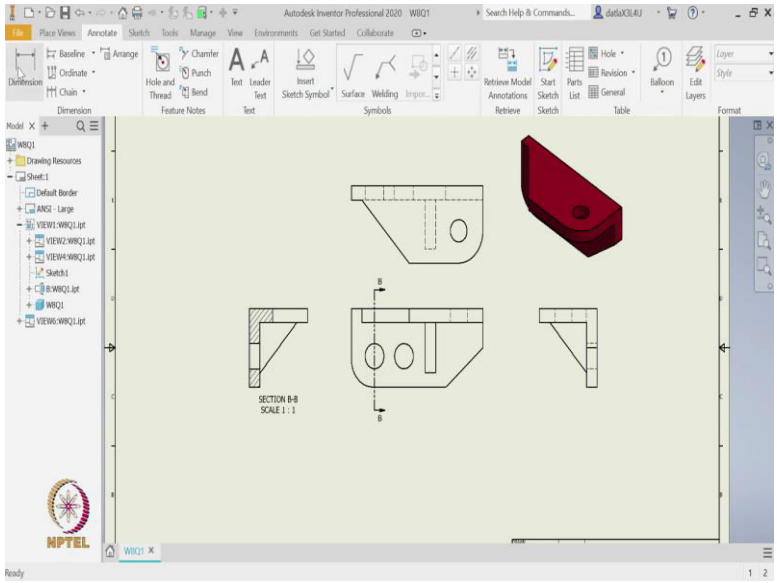

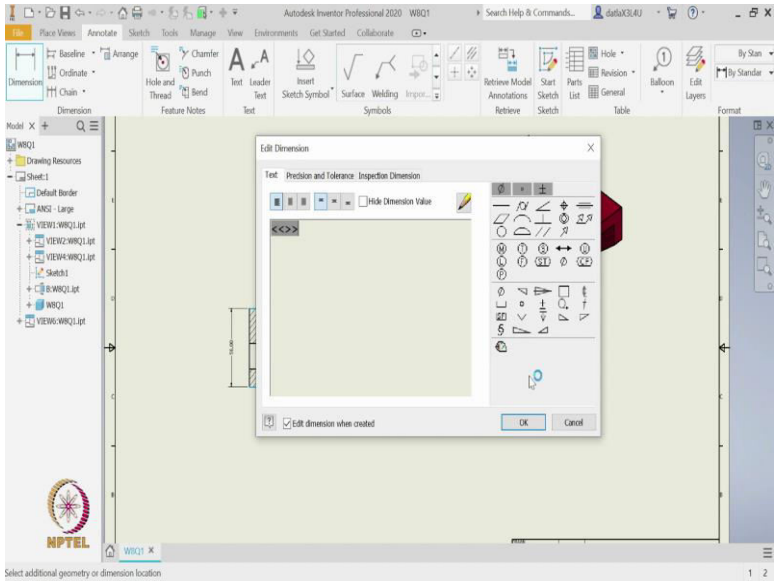

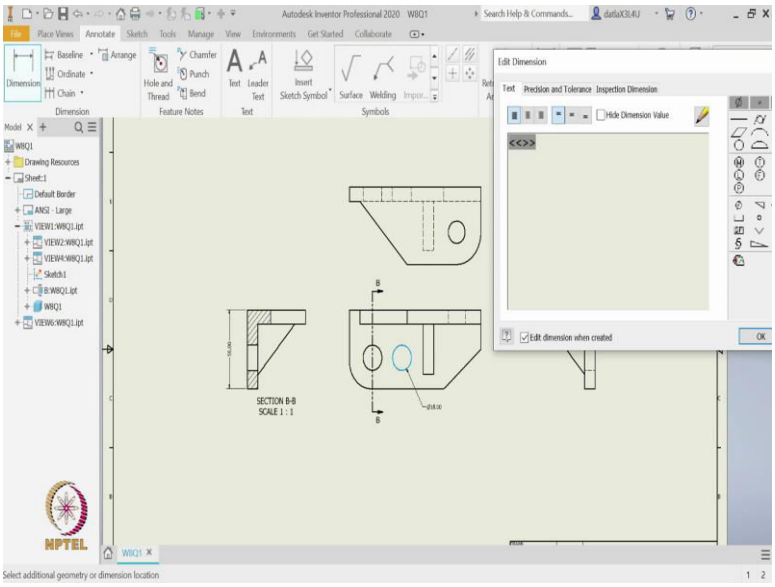

And for the isometric view let us edit view. And then say instead of showing it in normal outlines, we will show it as a shaded solid. So, now we have changed this isometric into a shaded solid we just need to right click on those views and go to edit view. So, now that let us say we have extracted these views. Now we want to give those dimensions for that we go into this annotate tab within which again the tools are all modified as per the requirements.

So, most of the dimensions can be done using the single dimension tool. Let us say you want to dimension this overall dimensions for that you choose 2 parallel lines and then just move the mouse. So, it will suggest where to place it. But you can place it anywhere you want. Let us leave it here and click okay. So, here it brings up a dialog box where it lets you add few more information if you want but as of now let us say okay.

Because we will go with the default information or just the dimension of this line. So, let us keep doing these dimensions. So, let us say we have done the linear dimension but how about the radius. So, let us choose this circle. So, it has understood that this is a circle so, it is giving a diameter of 18. Now this time, we noticed that there are 2 circles of the same diameter. So, I can add that information here. So, let us say at the start I will say it is 2x. And let us click okay.

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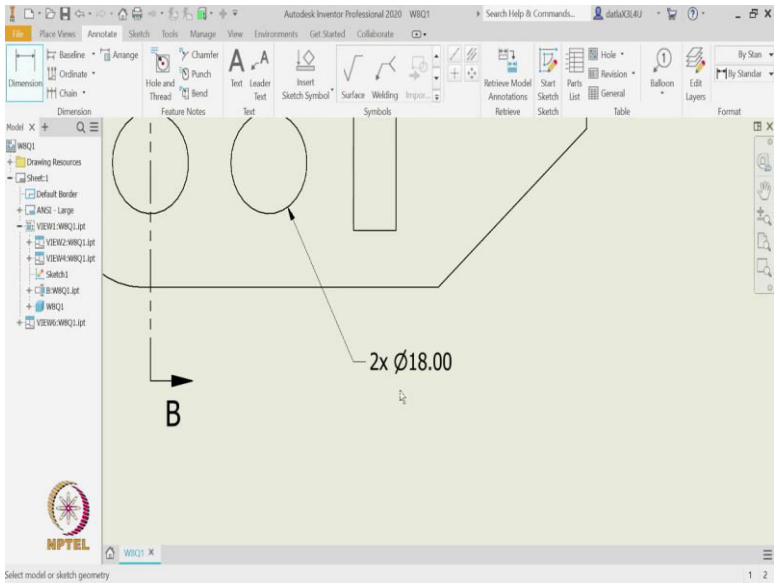

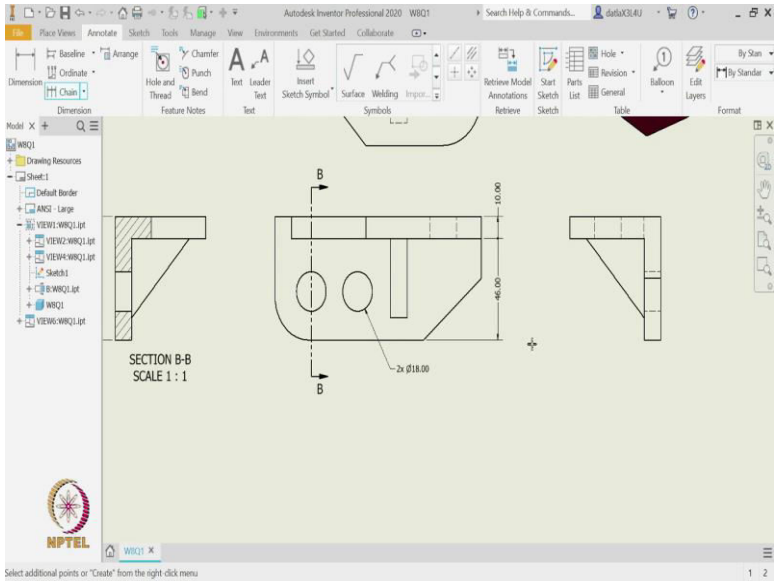

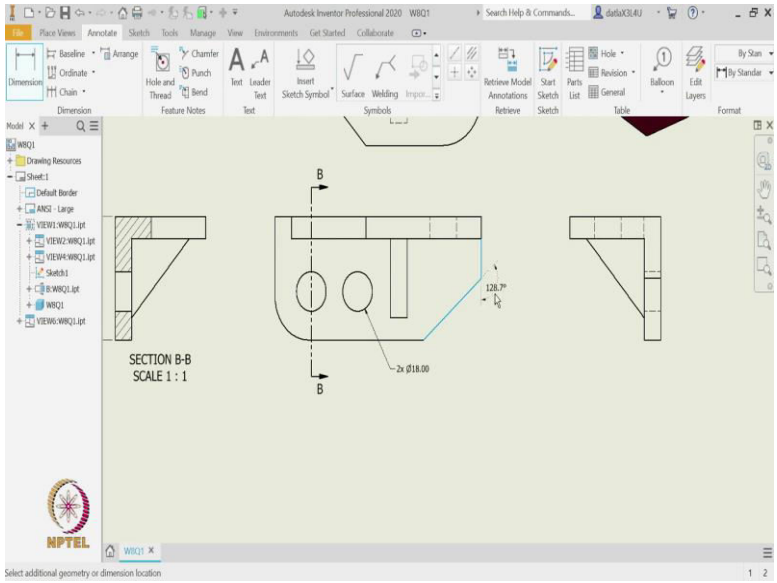

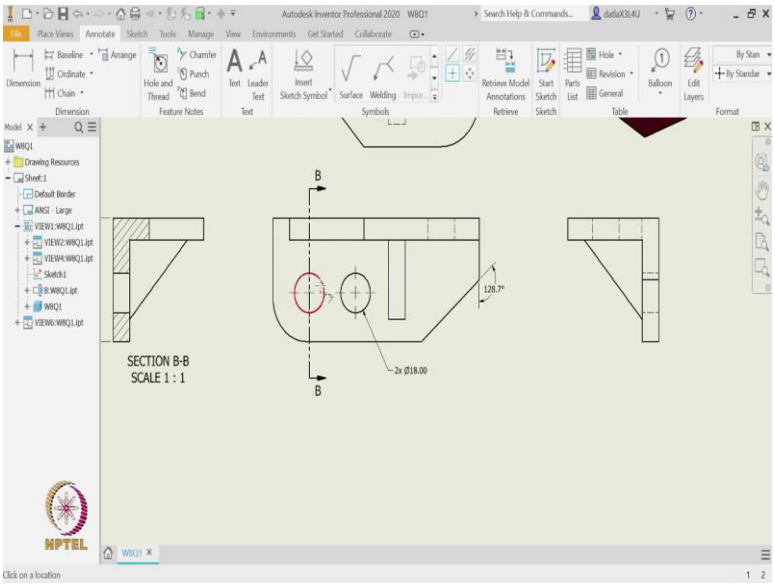

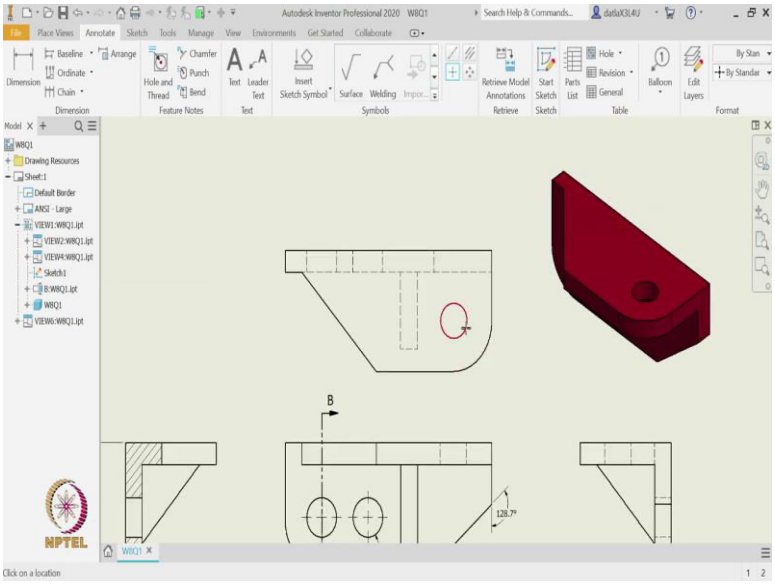

So, as you can see now it has mentioned that there are 2 circles with diameters of 18. So, within this dimension, we have other options like we can choose a baseline and dimension multiple objects. So, let us do this. So, let us say using the leftmost one as the baseline I will dimension these 2 vertical lines. Right click on it say continue. So, you can place wherever you want. Let us say I will place it here.

Now you can see taking the leftmost line as the base it has dimensioned both these dimensions both 70 and 125. Similarly, you can also choose the chain dimensioning, for chain dimensioning let us choose some horizontal lines. Let us say this, this, and this right like say continue. And let us place it here. So, now we can see it is using the chain fashion of dimensioning. So, we can also define angles for that we need to use dimension and specify, let us say 2 sides.

So, now it specifies what is the internal angle between these 2 lines you can either show it inward or outwards, let us show it outwards. There are other features you can specify here. For example, when we have a hole we usually add these center lines, right? So, that we can do using this option where you add a center mark you just need to set the circle and it will add a center mark. So, let us add center mark here as well. Here too we can add.

There are other features as well like, you can add either the center line or center line as a bisector. So, let us add the center line as a bisector. So, these 2 vertical lines we know are representing this hole here. So, if I choose those 2 lines, line 1 and line 2, it added a center line.

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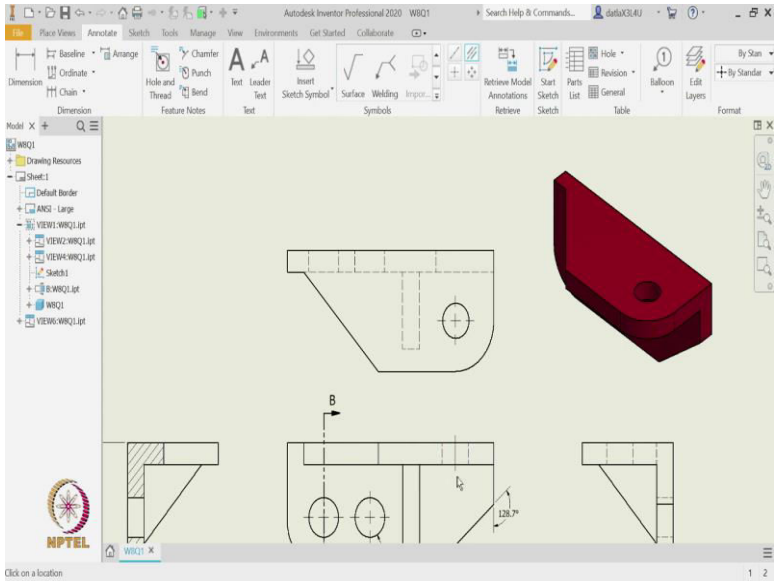

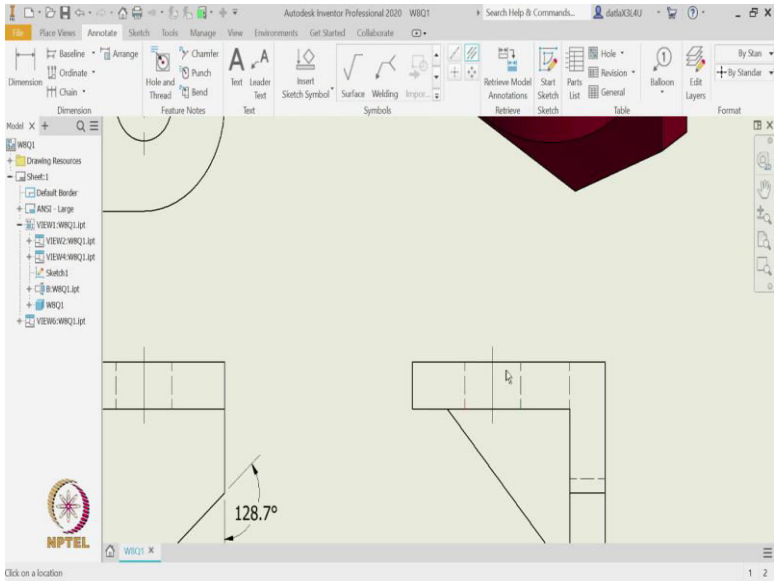

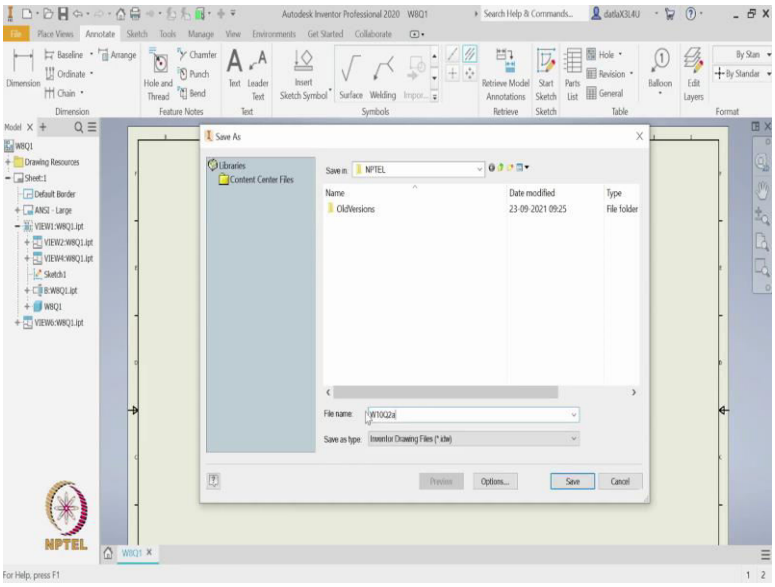

Similarly, let us see where else can we add it. So, here as well. Let us zoom in. So, there you go we added that center line. So, these are most of the features we can add on. So, I am sure this dimensioning is not complete but we have looked into most of the common features we use to dimension your object using which you can go ahead and dimension most of the features.

One interesting feature which we already discussed is saying that what happens if we change the solid model will that be reflected in the drawing or not. So, let us look at it whether it does that or not. We already mentioned that it gets reflected because you remember the first step we have done while creating these view is we have created a base view where we have selected a solid model.

So, in future if you go back to that solid model and change any dimensions or the shape that gets reflected in the drawing as well because these both are connected to each other. So, first let us save this, we will save it in the name of W10 Question 2a.

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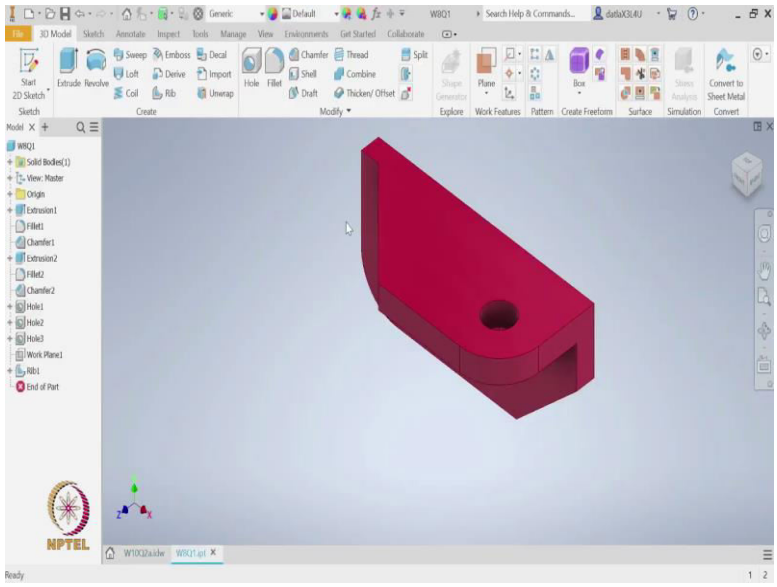

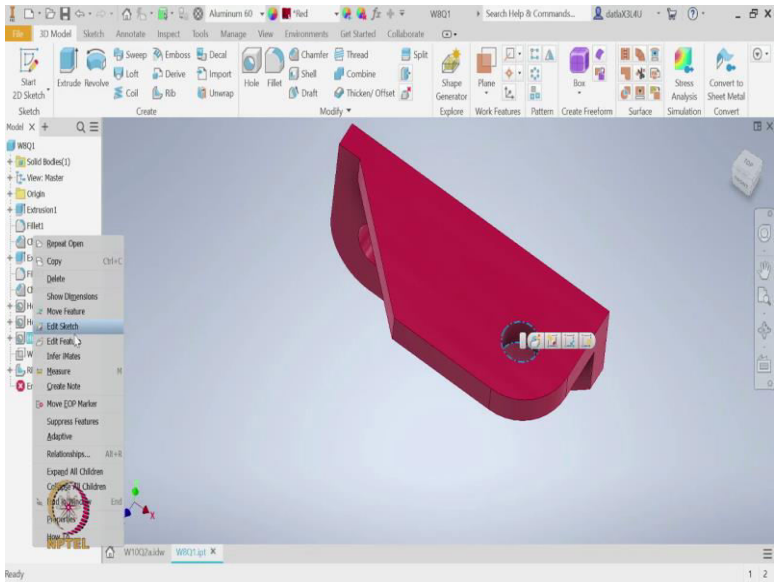

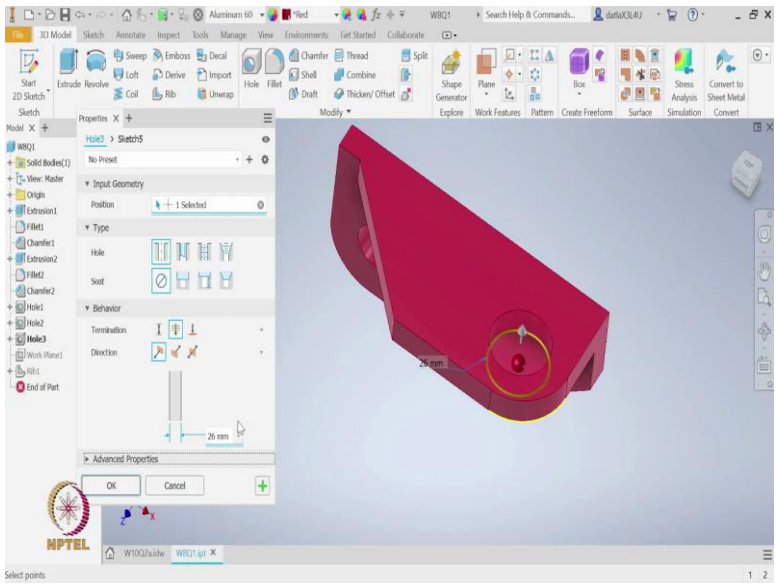

So, let me open that solid model I have W8Q1. So, here is the solid model that we have created in week 8. Now let us say we modify something like let us say this hole we will enlarge the diameter. So, let us see which feature is this right click on it say edit feature. Currently the diameter is 16. Let us make it for example 26. So, we see now that this diameter is the hole is enlarged. Let us save it and now let us go back to the drawing and see how are things changed.

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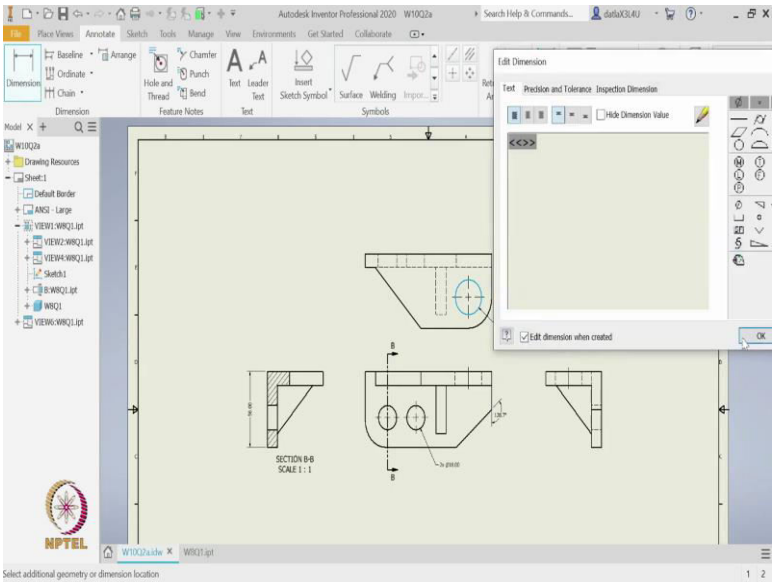

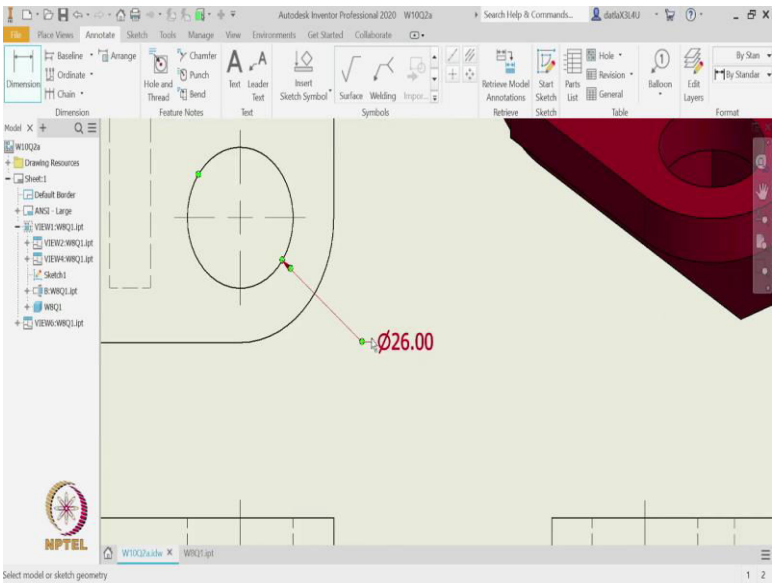

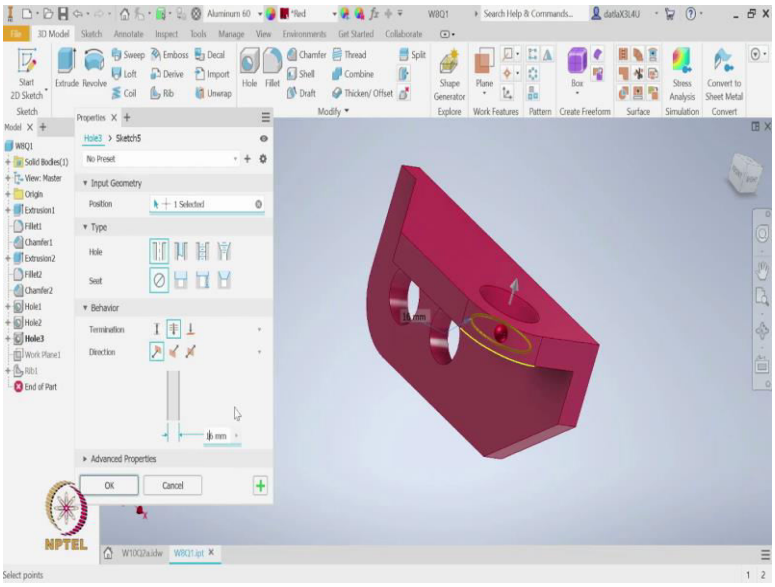

So, now you see that it has automatically updated the feature. So, that let us now dimension this diameter and see, what is the value? We see that the diameter is 26. Just for the sake of clarity we will again go back there and get it back to the initial value by going to edit feature and then specify this as 16. So, let us save it here and then go back to the drawing.

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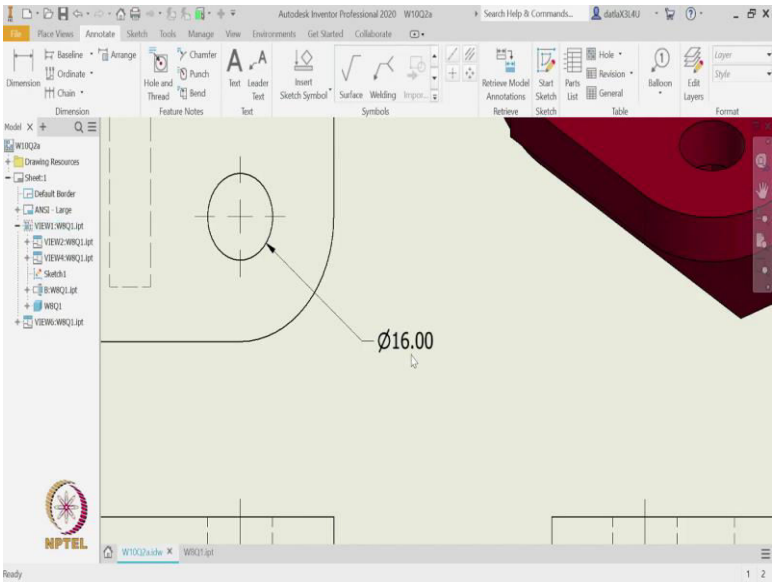

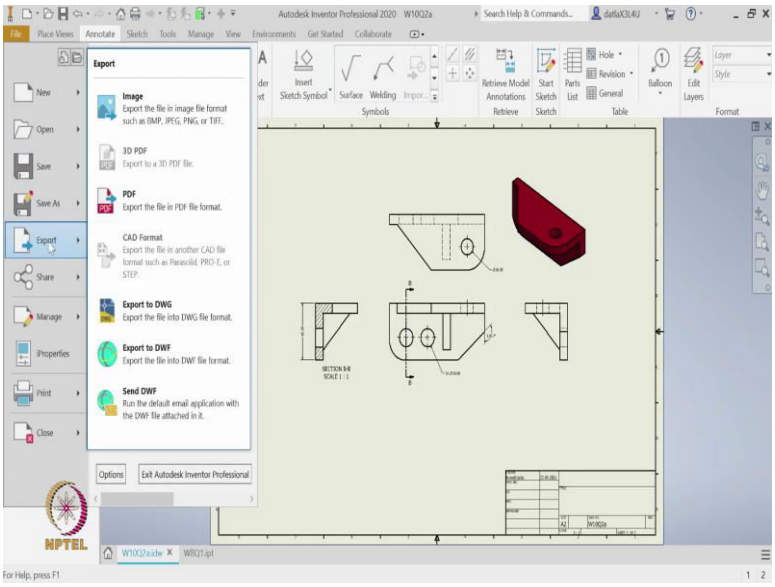
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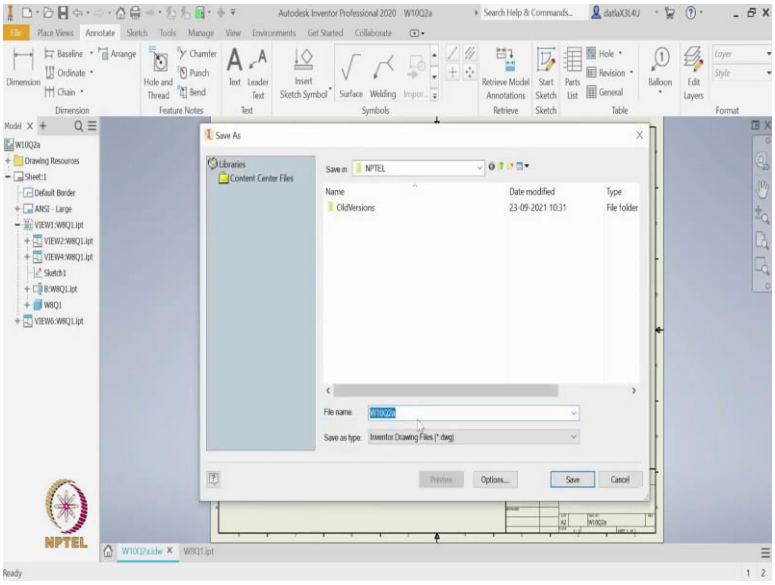
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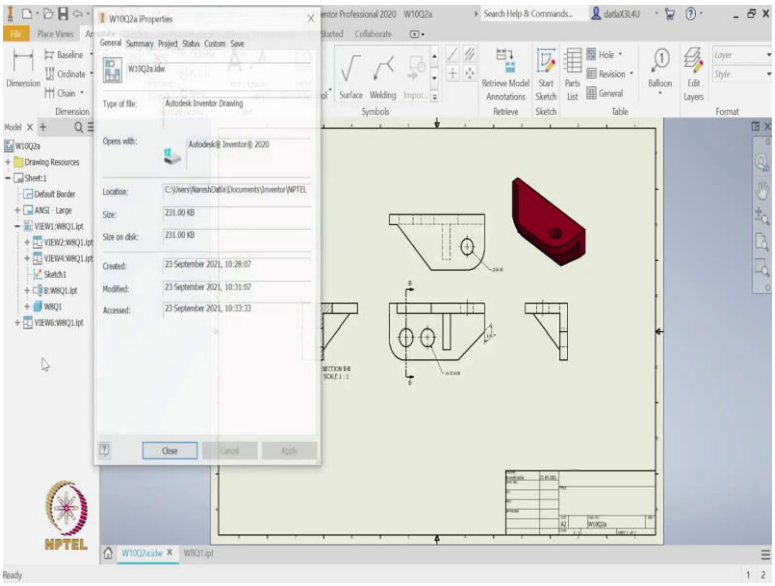
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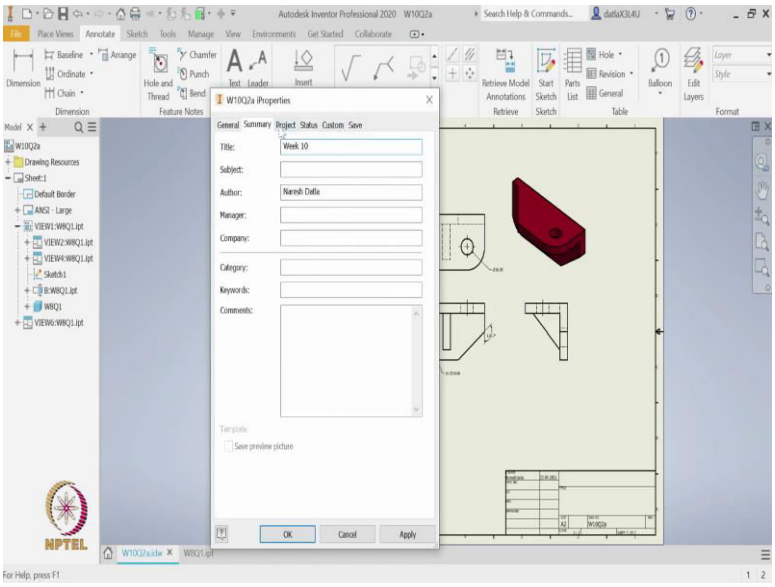

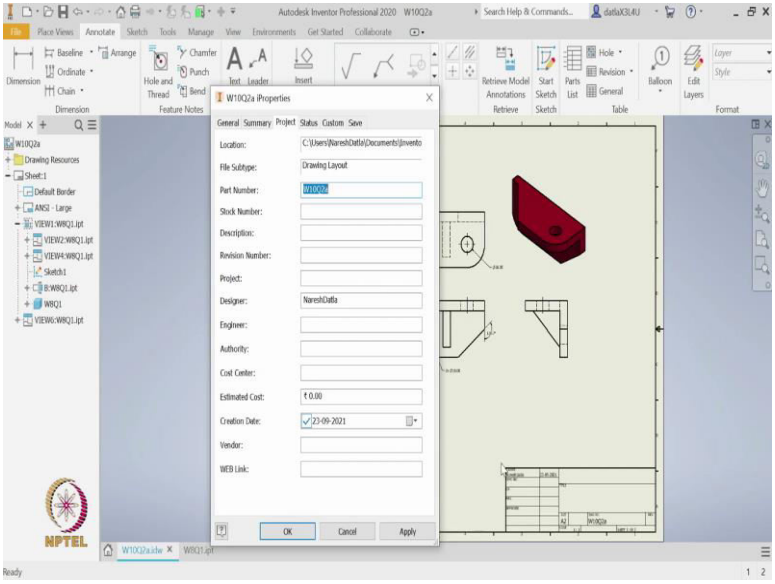

We see it has automatically updated from diameter of 26 to diameter of 16. So, once we are done with extracting the views as well as annotating the drawings. Then we can save it by default it will save with the template we started with. But we can also export this. When we choose to export it gives you options like you can save it as an image, a PDF or export it to DWG. So, let us say if it is exported DWG we can just save it.

So, in addition to this if you go to the file tab you see there is something called as I properties. So, once you open the I properties it will give you the properties of these file we are working with. So, if we go to the summary I know you can change the author here and you can add details about what are there are some of the details you have in the title block like what is the title of sheet let us say it is week 10 and subject.

So, we can also go into the project where it will ask you what is the part number so, which gets reflected in the title block. And it is of course, it is asking who is the designer engineer authority and all these details, one can, see based on the template you only have few features in the title block, but one can always go and edit those templates to include other features based on the requirements of your industry or your work environment. So, we can, let us say click okay.

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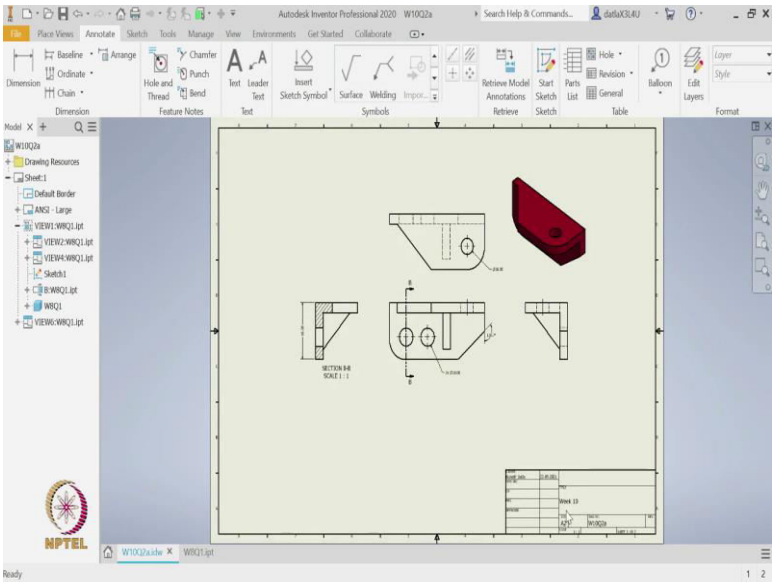

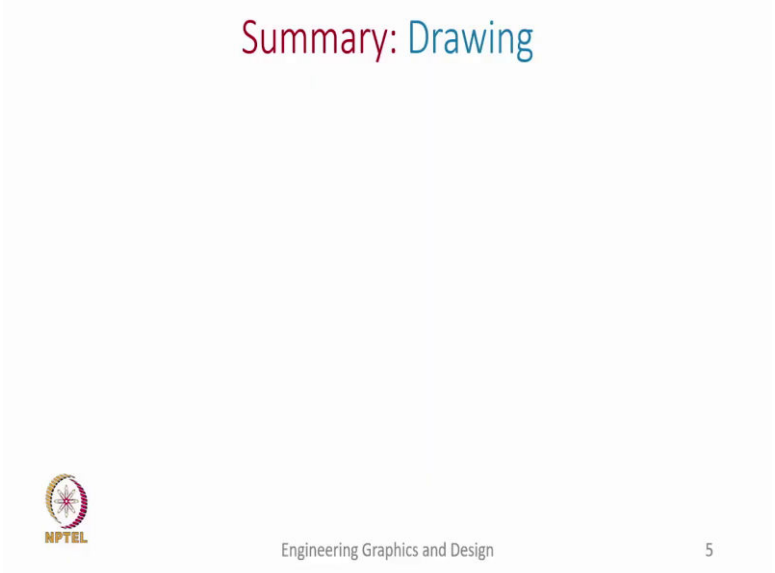

So, we have entered the title which is reflected here. So, this should give us an idea about how to also edit the title block based on the work environment we are at. So, in summary, what we have looked in this lecture is we have started from a solid model and extracted the drawings 2D drawings from those solid models. We have done essentially 2 important

activities. One is first to extract the views, by first extracting the base view and the projected view, we can also get into the other views like the sectional view, auxiliary or the detail views.

So, once we get all those views, then we have gone into the annotate tab where we can dimension all features of that drawing. So, with that let us conclude today's lecture. We will meet again in the next week. Thank you for your attention.