Engineering Graphics and Design
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Week 8
Autodesk Inventor Environment

Welcome back. In this lecture we will be discussing about Autodesk Inventor environment. In the previous lecture we discussed a brief introduction to the CAD softwares. In this lecture we will be focusing on one particular software called as the Autodesk Inventor. So we will be using this software throughout the remaining part of this course. I hope you can follow the instructions available on the website of the Autodesk Inventor to download this software and be ready so that you can follow this lecture when you have installed the software.

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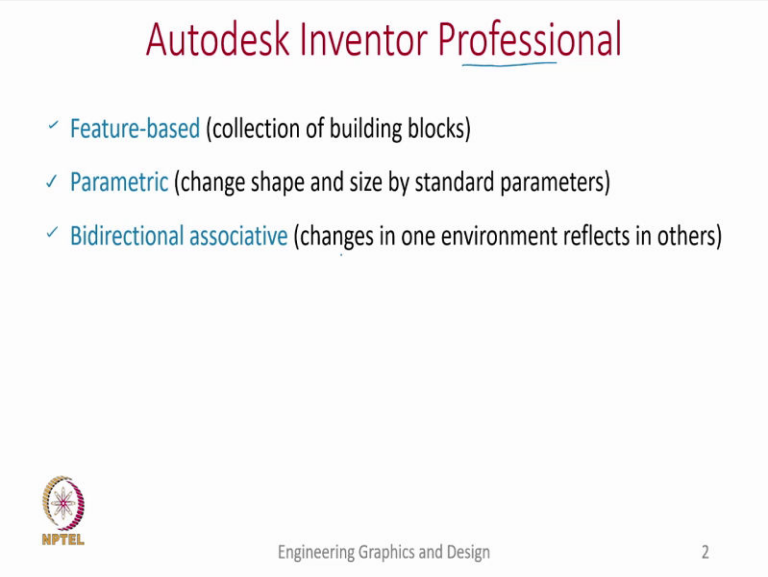

So we will be focusing on the Autodesk Inventor Professional. So this is the software you need to proceed with this course. So few features of this Autodesk Inventor are, first it is a featured-based modeling software which means it collect the building blocks and does the Boolean operations to develop complex objects. Second, it has a parametric modeling feature which means you can change the dimensions and shape by defining standard parameters. And lastly bidirectional associativity, here changes in one environment reflects in others.

So usually this CAD software has many design environments. So we have Part module, Assembly module, Drawing module and many other kinds of modules. These are the different

design environments we have in the CAD software. So let us say you make a small change in the Drawing module. So that reflects in the Assembly as well as in the Part module. So more of it we will be discussing in the next slide.

But the idea we are trying to say is you can work in any of these modules and any changes made in one particular module will be reflected in the other modules. So this is the bidirectional associativity we are talking about. You can either go and change at the part level or you can change it at an assembly level or you can change it at a drawing level. It doesn't matter. It can take changes at any environment.

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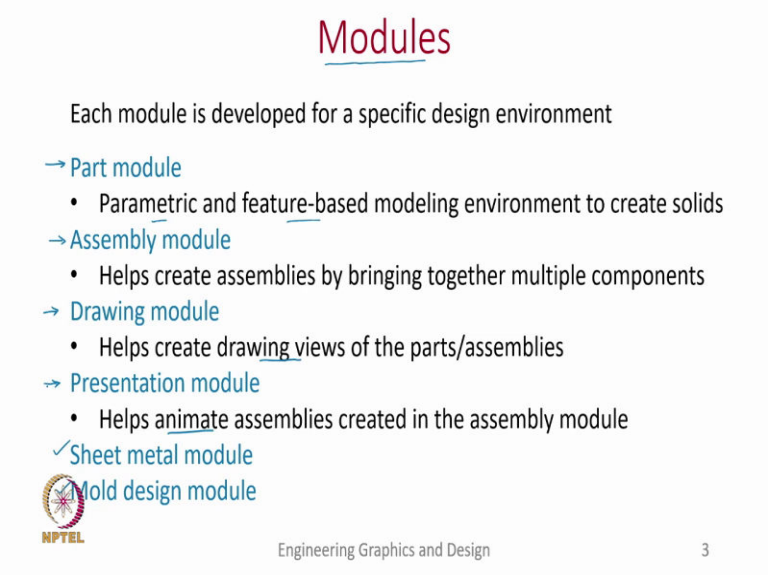

So these environments is what we are discussing as the modules. This Autodesk Inventor has many design environments starting with Part module. Essentially Part module is where you create your objects. So it has the parametric modeling features and the feature-based modeling features to help you develop those solid models of a component.

But let us say once we have developed these components then we can go into the Assembly module which will help you to assemble these individual components to make a device. And when you are assembling it you can apply constraints by which you can say what is the degree of freedom each of these component have, because that will help in the later part when you want to animate and see how these assemblies are positioned.

So next is a Drawing module which will help you create the drawing views of both parts as well as assemblies. So each of these is a different module, all within the same software. So you just need to switch gears between Part module to Assembly module and again from, after you are done with the design and you are in a position of getting the design detailing that is when you go to the Drawing module to create those views. So in addition to this you also have the Presentation module where you can animate the assemblies created in the Assembly module.

So these are usually used for presentation purposes or for marketing purposes where you want to show the working of this device or the machine you have created or designed. So there are other modules like Sheet Metal module or the Mold design module which are more specific for specific purposes. You also have Analysis module for very specific purposes.

In this course we will not be going into all of it. But we will be touching upon these first four; the Part module where we will help you to create objects and then we will also help you to use this software to develop Assembly modules, which means assemble those components and then show the complete device. And later we will also show you how to extract these drawings from the components and assemblies you have created. And lastly we will also show how to use this Presentation module which will help you to animate the assemblies you have created.

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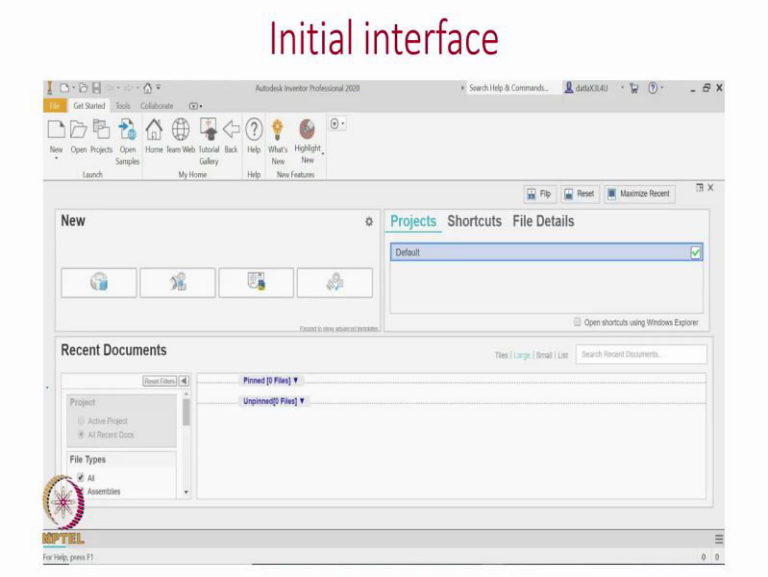

So let me get started with the software, and here I am assuming that you have already downloaded this software. The Student version is free from the Autodesk website. So once you download this software and click the Start button in the software the first thing you see is this interface which we call as the Initial interface of the software. So let us look at what are the features we see in this first Initial interface because that will help you to understand what options you have to choose when you get started.

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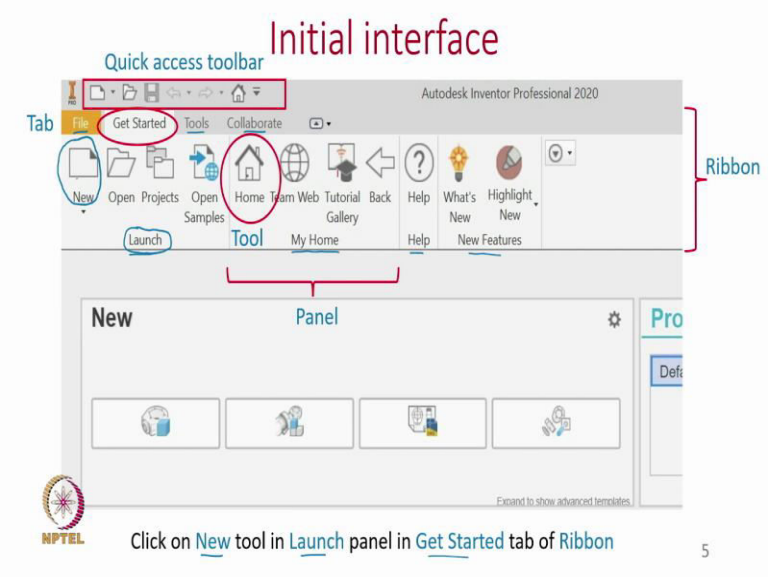

So let me zoom this Initial interface to the top left corner. So this is the zoomed version of the same Initial interface. I want to show you what are the features we see here and what are the terminologies involved because that makes all of us on the same page when we discuss in the next lectures.

So to start with we have a Quick access toolbar at the top. So what do this Quick access toolbar have? It has something called the New. So if you want to start a new part or an assembly that is where you can use. You can open a file, save a file, undo, redo, going to the home page. So these are all the shortcut buttons, let us say, you have in this Quick access toolbar.

Below this Quick access toolbar you have a long horizontal thing called the Ribbon. So in this Ribbon you have many features which keeps changing depending on where you are in the software. So in this Ribbon it is divided into tabs. So for example as of now, we are in this Get Started tab. But what are the other tabs available?

We have this File tab, Tools tab and the Collaborate tabs. And within this Get Started tab we have different panels. So what we are showing now is My Home panel. But what are the other panels we have here? We have this Launch panel, Help panel and New Features panel. And within the panel we have tools. So for example, in this My Home panel we have this Home tool, Team Web tool, Tutorial Gallery tool and the Back tool.

So what I was trying to do is to introduce the terminologies we will be using as we go for forward, saying the long Ribbon, and then it has several tabs in it, and each tab has multiple panels. Within the panel you have multiple tools to pick. So let us see what is the first thing you do when you come to open the software. So let us say you are here to start a new drawing or start a new part or start a new assembly.

There are multiple ways of doing it. But a preferred way is to click on the New tool in Launch panel in the Get Started tab of the Ribbon. So where is the Get Started tab? Get Started tab is already open. We need to go into the Launch panel. So this is the Launch panel. In the Launch panel we are asking you to select the New tool. So where is the New tool? It is here.

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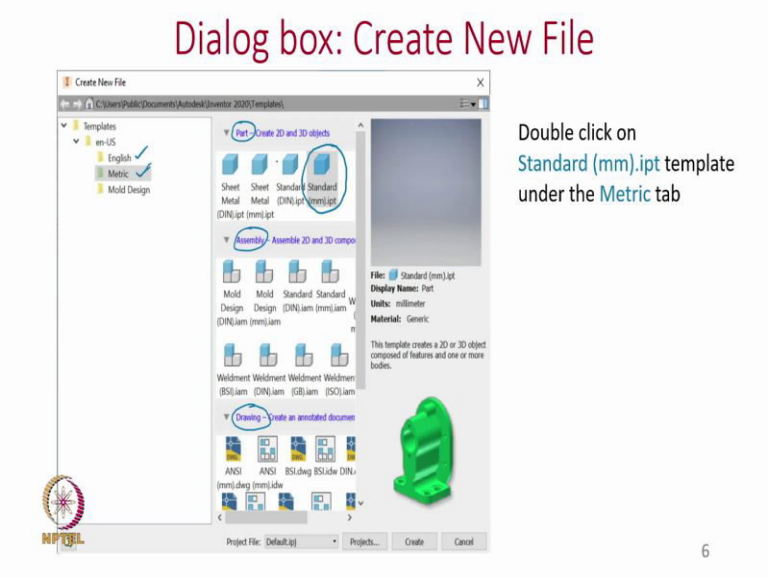

So if you select that we get a dialog box which looks like this. So once you open a dialog box this is called the 'Create New File' dialog box. This will give you a list of templates you can choose to start on. So depending on whether you are doing part modeling, assembly modeling or drawing modeling you can choose different templates. And even in these templates here we have different options. Like you can choose English units, Metric units and there is a Mold design.

So essentially I prefer to using the Metric. But depending on the need you can choose whether it is English or Metric. So if you are planning to use the Metric or the millimeter we prefer you to choose this option. So you just need to double click on the standard mm ipt template. So this is the template. I suggest you to click, double click on it to open the template. So this template we have chosen by going into this Metric tab.

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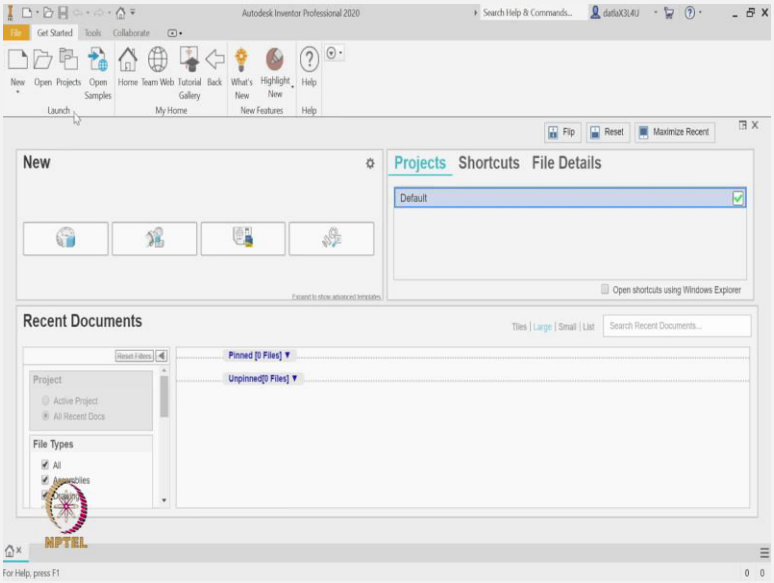

As of now, now I will show you the software. So here is the Initial interface I was talking. So once you are in the Initial interface these are the Quick tool, Quick access tool bar where we have the New, Open, Save, Undo, Redo, Home. And then we had these different tabs. This is the File tab, Get Started tab, Tools tab, Collaborate tab. So let us get back to this Get Started. I prefer you to do this. Go to the New in the Launch panel.

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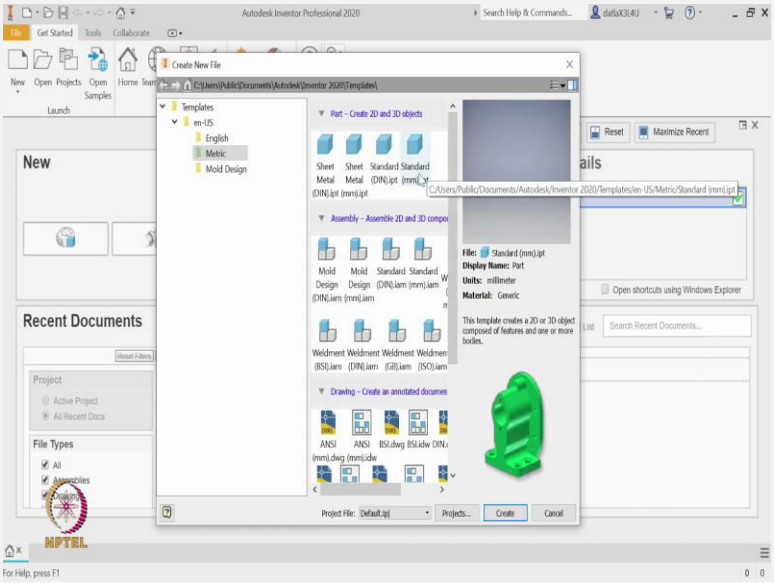

So if I double click the New we get this dialog box. In this dialog box you can go to the English. Then the there are different set of templates. And If you go to Metric there is a different set of

templates. So here is the part templates, assembly templates, drawing templates. If I go further down there are few presentation templates as well.

So let us double click on this standard millimeter ipt. ipt will be for part drawings. So if you go to assembly it will be iam. And once you go to drawings it will dot idw. Similarly for presentations it is ipm. But now let us do this ipt for standard millimeter.

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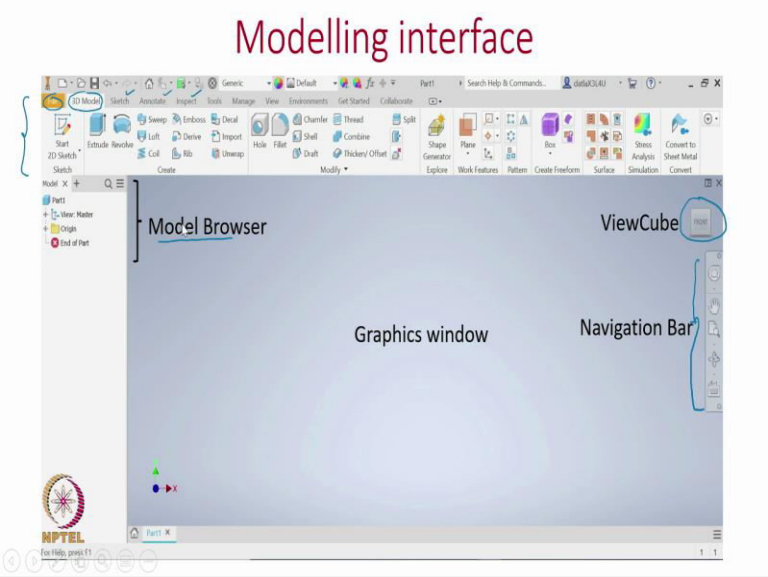

So if I double click on it then it opens up the modeling environment. So as you can see in this modeling environment the Ribbon is completely changed. This is what we call it as a dynamic because the software dynamically changes the options we work with depending on the design environment we are in. So previously we were showing this is the design environment and we showed the Graphic window, ViewCube, Navigation Bar and the Model Browser. Let us see how to use this.

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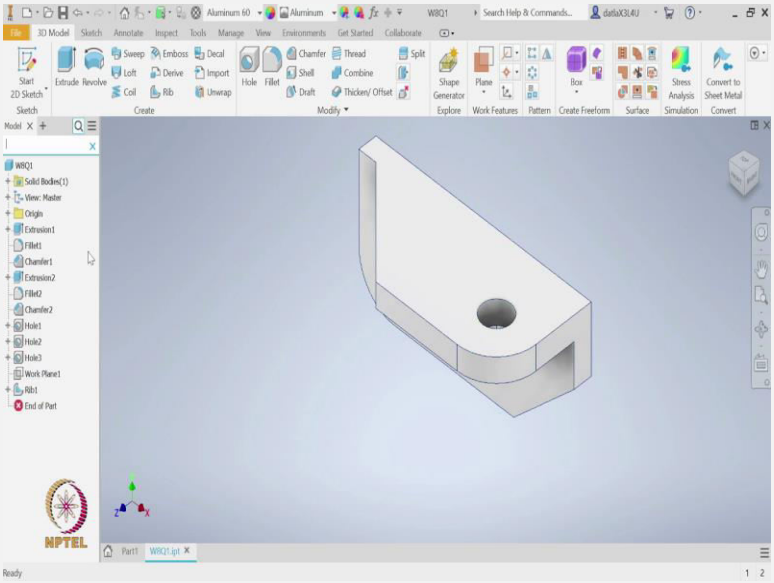

So in the software since we started a new component everything is empty. So by default it starts with part1. But let me open a model, solid model that I have already created. And so this is a component I have already modelled. So let us see what are the changes you notice. So first thing is if you look into the left thing, previously we called this as a Model Browser.

So the Model Browser has several activities in it. So previously we only had this View Master and Origin. So in addition we have this Extrusion, Fillet, Chamfer, Extrusion, Hole, Work Plane, Rib and lastly the End of Part. So these are all the activities which I have previously done to arrive at this final object. So this you can also use the search command to go into any of it.

As of now it is just an bunch of 10 activities. But probably later when you are working with a more complex model you will have a big list of activities and then this search feature will help you go into the right activity.

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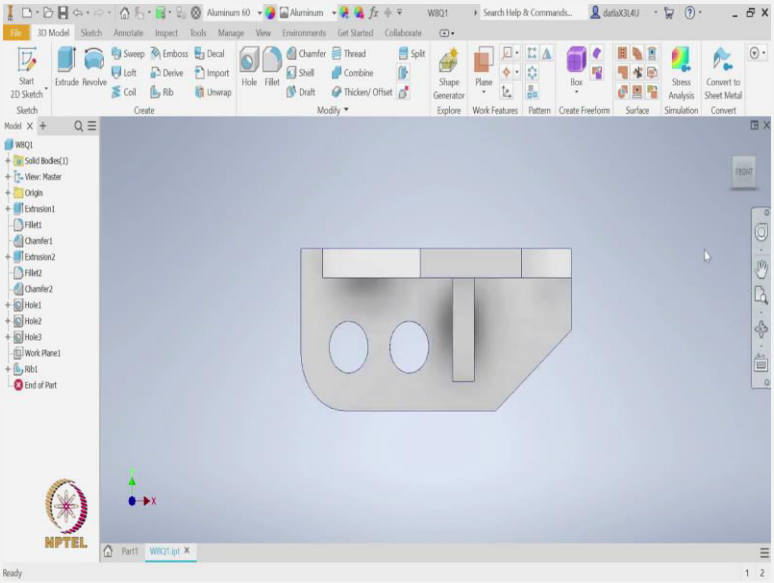

But let us look at this ViewCube. So we said at the top right we have this ViewCube. How do we use? So if you look carefully it has front, right, top. So if I move my cursor to the front it highlights. You can see the change in color to blue. Say if I click on it is showing the front view.

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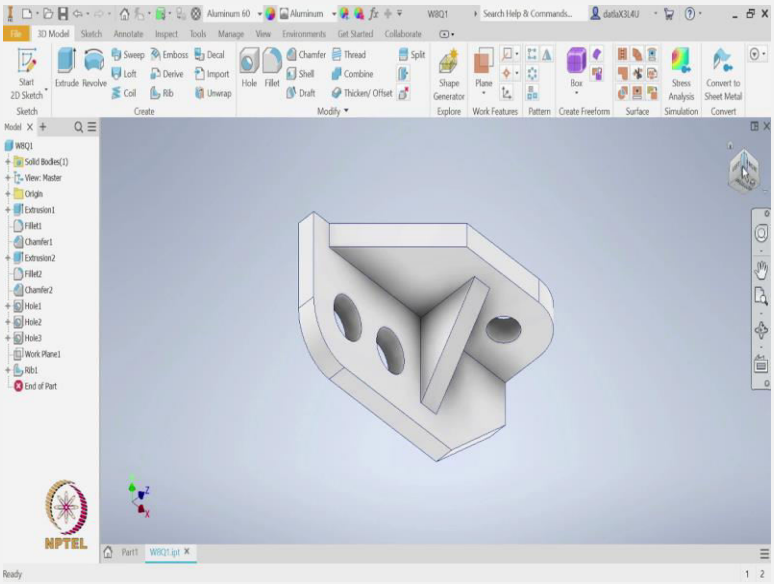

So in the front view, again if go to corner of this ViewCube it highlights a corner. So if I highlight this corner and then click on it, left click, then it is showing me a view which has the left, front and bottom.

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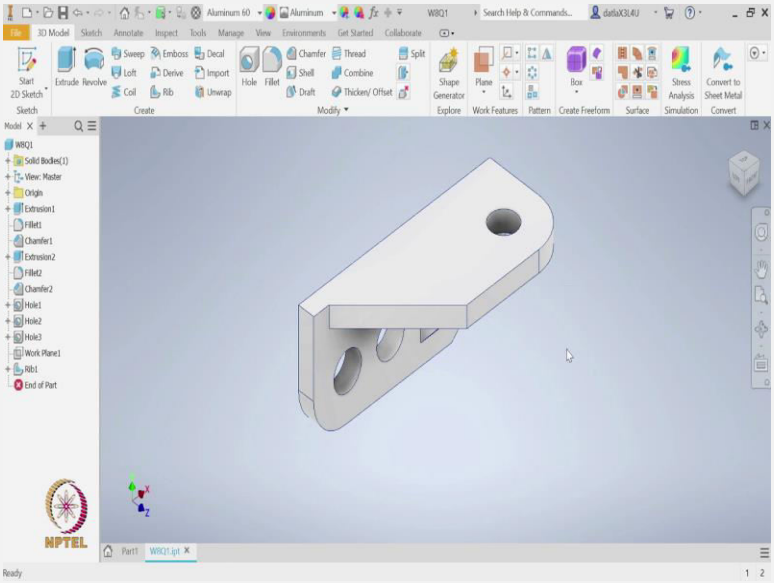

Similarly I can go to other corners as well. So I have chosen a corner which is showing me top, left and front. Like this I can look at the object from different view angles. I can either have this normal front view, side view and top views or the isometric views or from the corners. There are other ways to manipulate this object. Just for viewing purpose. I am not talking, discussing about modifying the shape of this object. We are only discussing about how to visualize this object.

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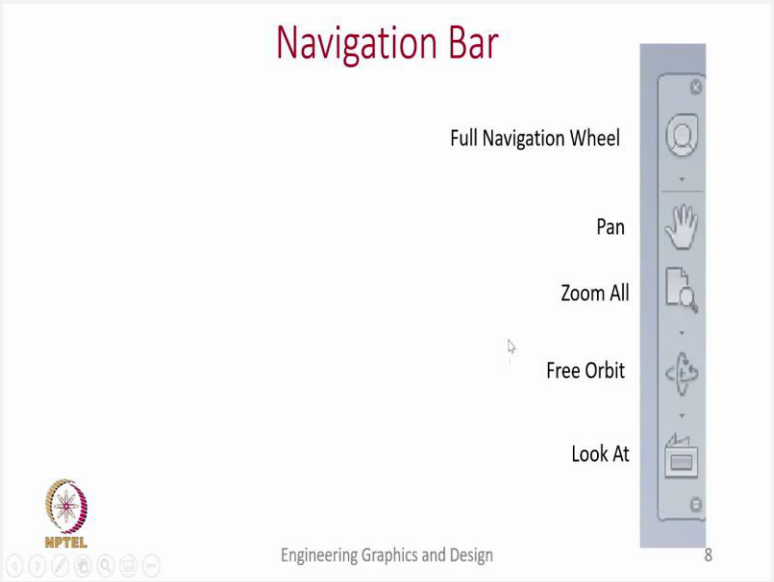

So here is the Navigation Bar. And these are the different options we have. The first one is the Full Navigation Wheel. Second is Pan which will help you to move the object to the left, right,

top or bottom. And Zoom In and Zoom Out also is there. So then there is a Free Orbit which is a dynamic tool to rotate the object. And lastly Look At. Let us look at all these features in the software itself.

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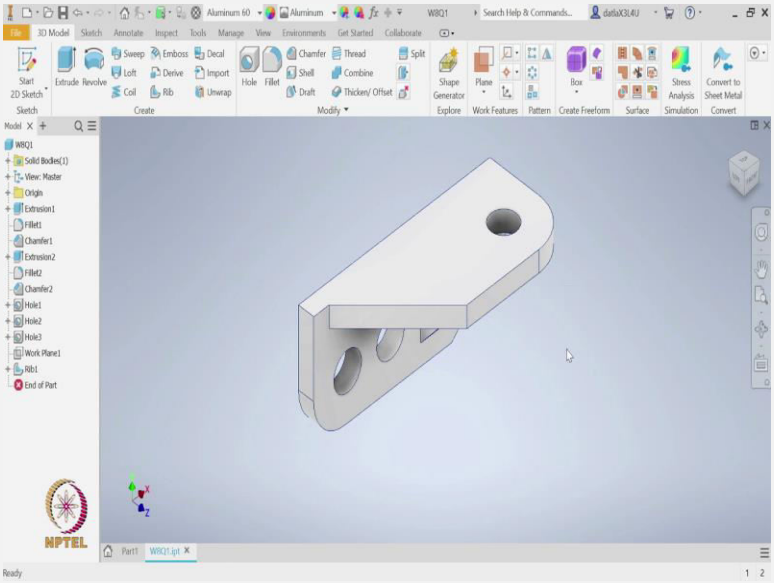

So let us start with Pan. So Pan, and after I click this Pan button I need to come to the Drawing area and just click and drag. So click and hold. So once you release the hold that will be your final location. So let me repeat. First click then hold and move it and then leave the click button.

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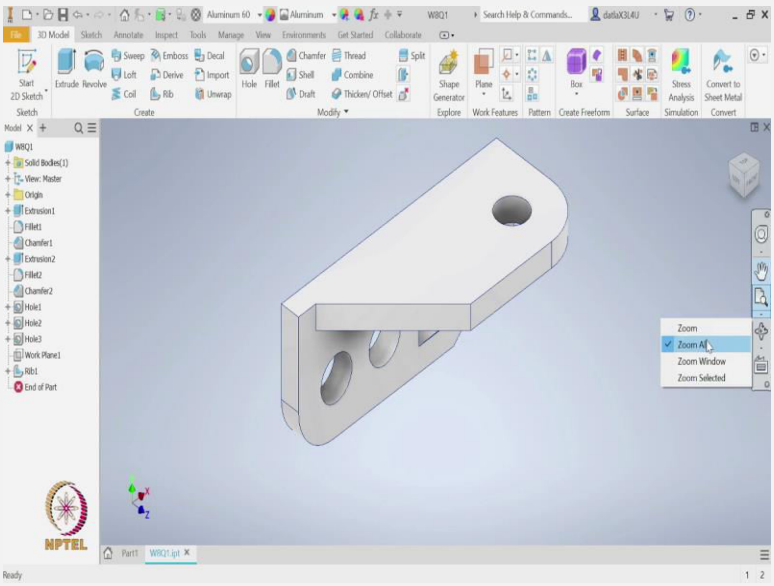

So the next one is the Zoom button. So what it does is it has again multiple options saying that Zoom All, Zoom. So let us look at one at a time.

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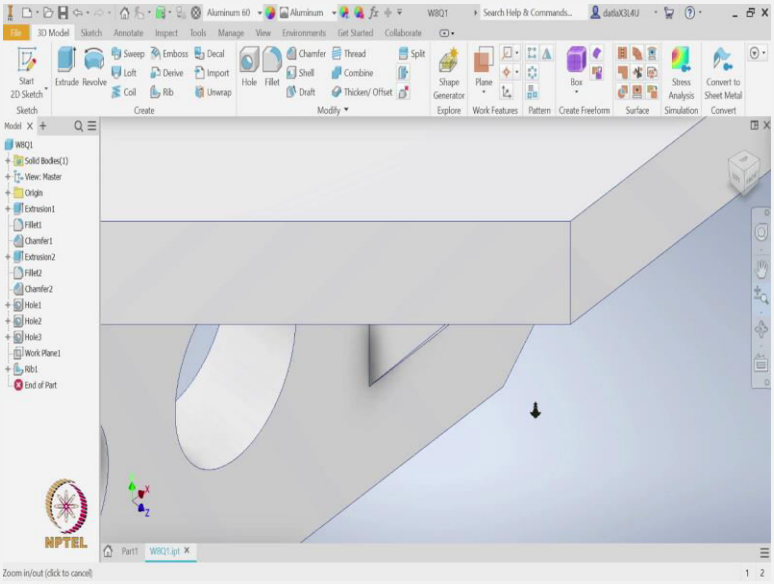

Zoom All; let us first start with Zoom. So once you click the Zoom button you can go Zoom In or Zoom Out. You just need to click and then move, hold it on and then move your mouse either up or down. If you move up it is decreasing the size and if you move it down it is increasing the size. So essentially it is Zoom In and Zoom Out.

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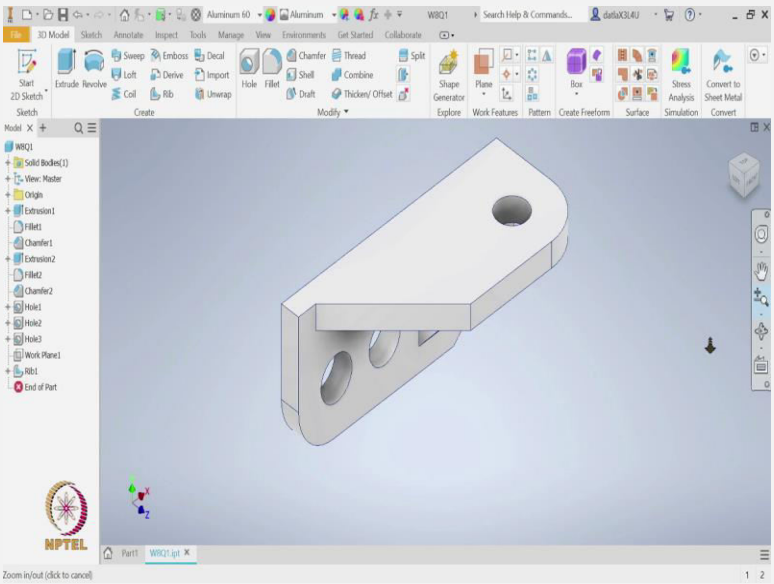

So now when I say Zoom All it fits the whole object into the screen.

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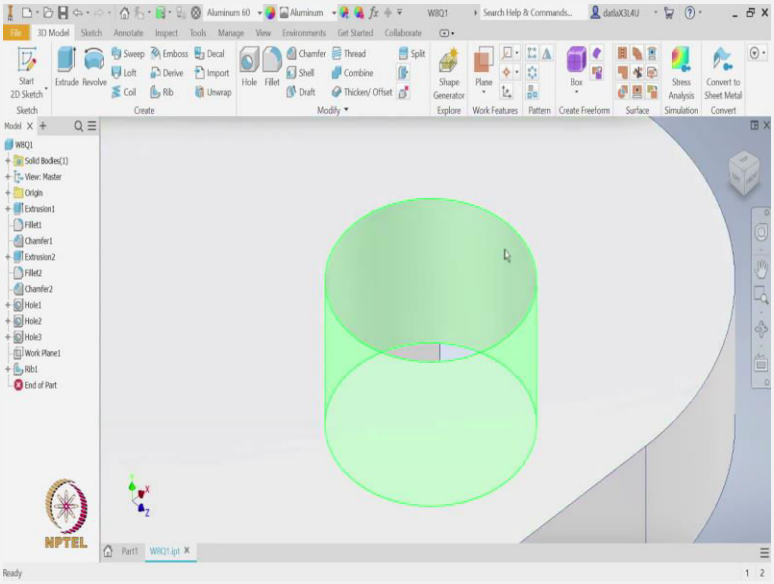

Zoom Window, so you can also create a window. Let us say this is the hole on the object and I just want to zoom in at the location. So I create a window and then I go, zoom in into that feature.

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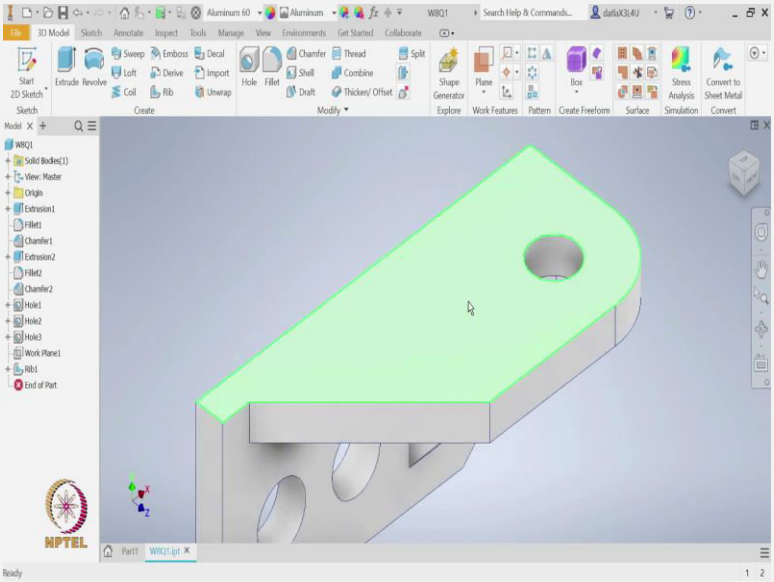

So I can point out to any feature. For example the mouse is pointing on to the top of this object. If I click there I am going closer. So only that top face is centered here.

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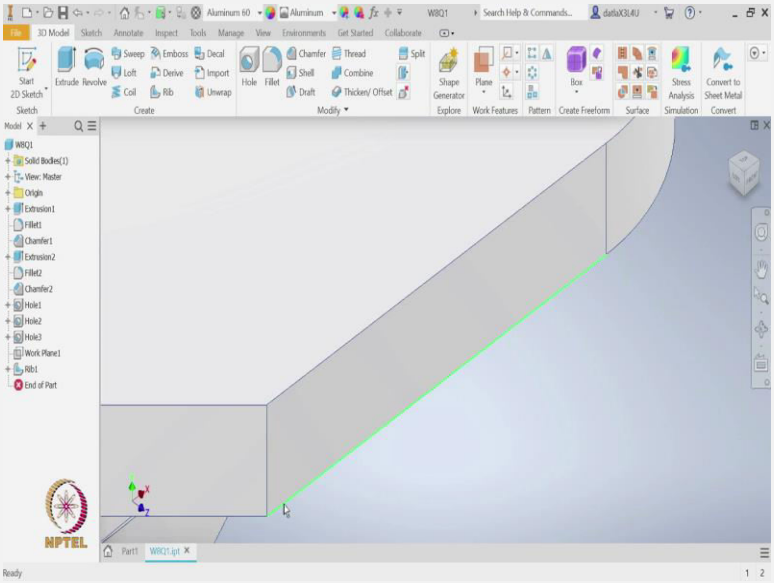

So but again let us repeat it for this front surface. So this front surface is fitting the complete view.

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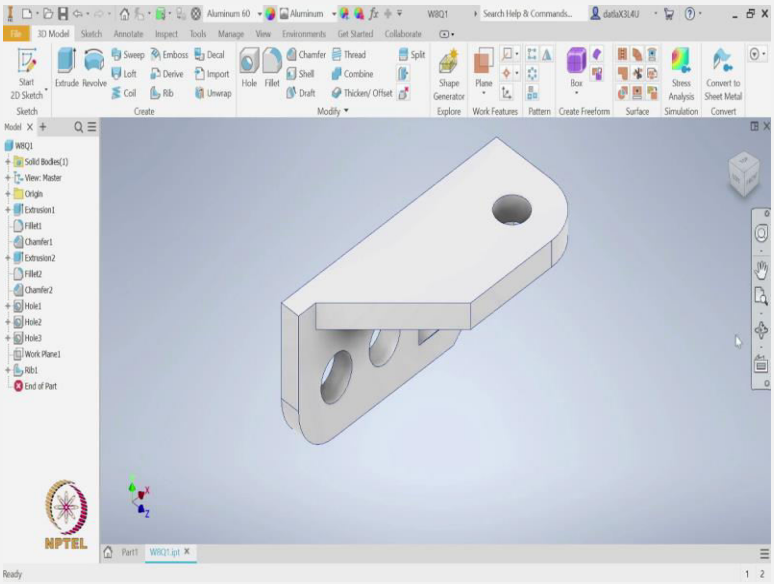

So mostly with use this Zoom All so that you can see the complete object and then decide on what operations to be done.

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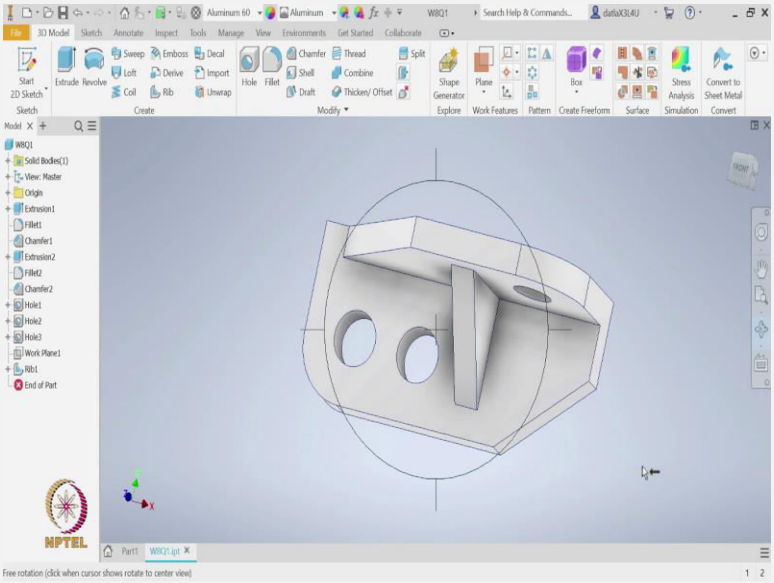

Next we move to this Free Orbit. So you just need to click at some place and hold on and move. So depending on how we move you can look at different angles of this object. And once you release the left button that will be your final orientation.

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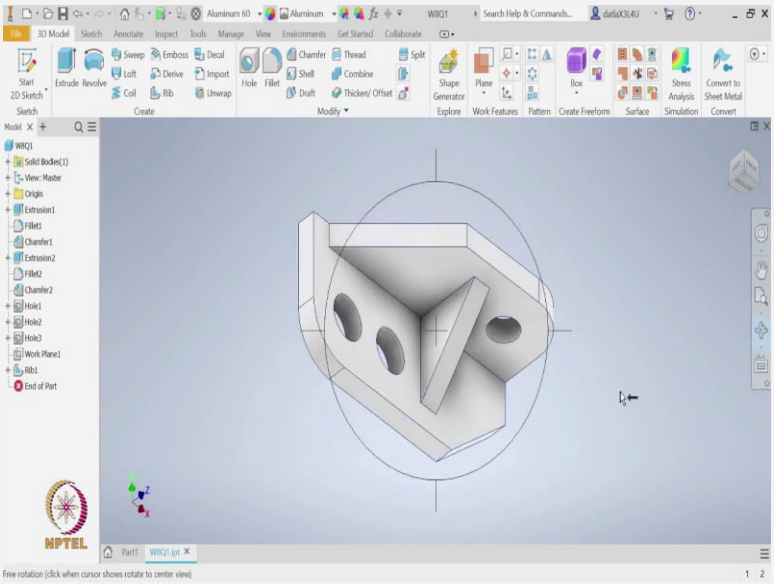

But let us say if you want to go back to the regular orientations we can always use this ViewCube; front, corners.

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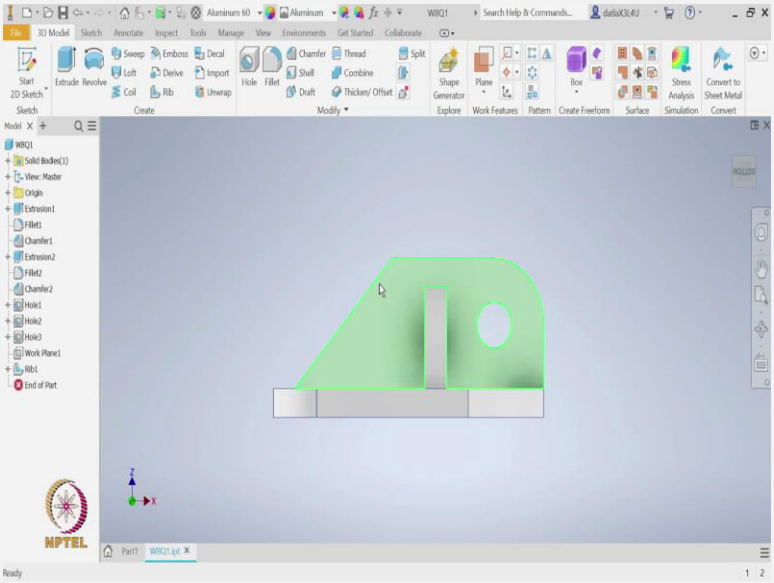

And lastly is this Look At. So if you want to look at, let us say this inclined surface that is oriented normal to your view. And if I repeat that, let us say, to this vertical surface now it is reoriented such that the surface we selected is normal to our view.

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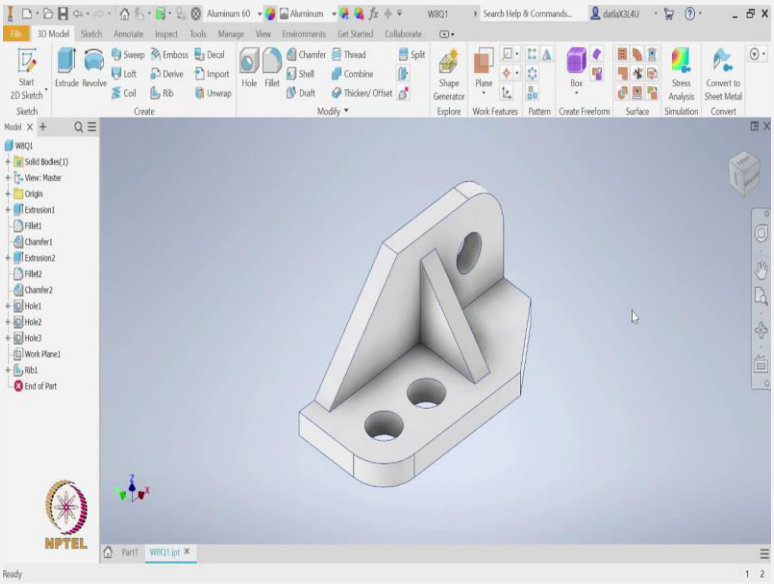

So let me do this Zoom All and maybe we will have isometric view. So with this what we have shown is if you already have a component we have showed you how to open that component, and then if you want to make changes you can always go to these activities. And once you click

on that activity you can go into that specific activity and make changes there. And once you say Ok it will rechange the shape or the size of this object again.

And similarly during this process if you are looking at different angles or different locations of this object you can either use this ViewCube or the Navigation Bar for viewing the object. And let us say after you have done with the modifications you want to save the file.

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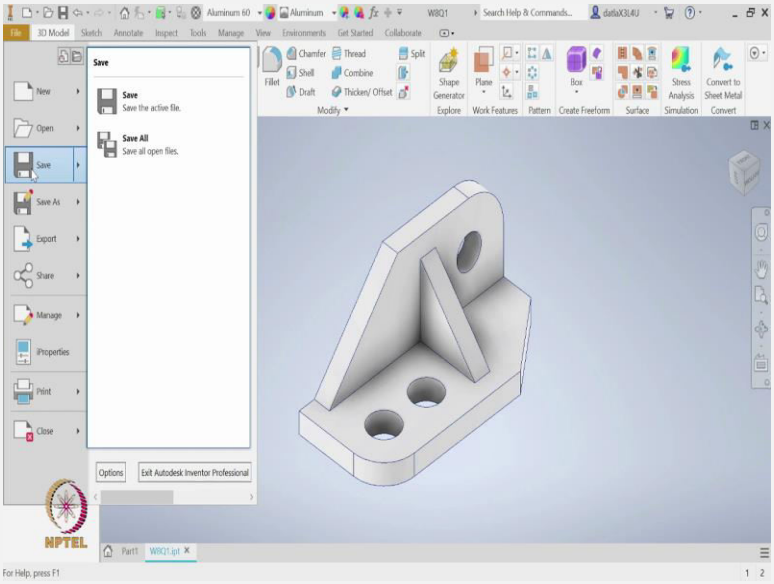

So then we go to this File tab and in this File we have either Save. It simply saves the current features saying that if you make any changes you need to keep saving so that let us say, if the software crashes or the power goes off you don't lose the object.

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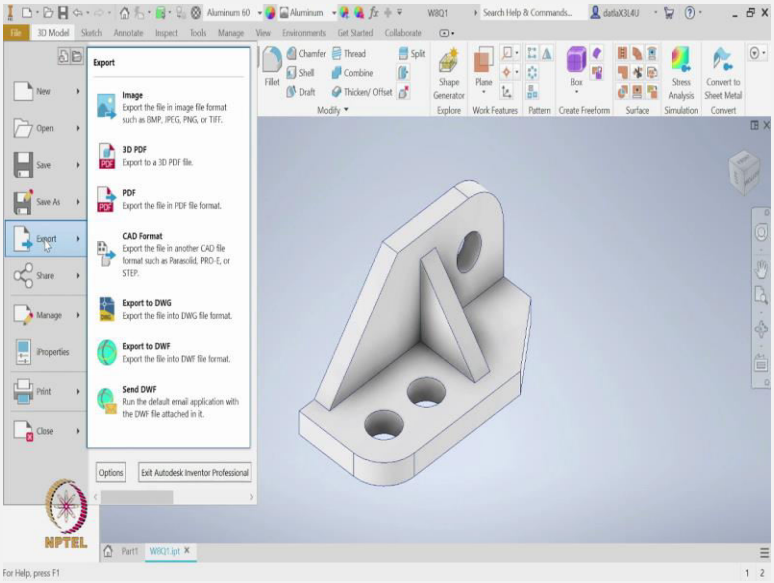

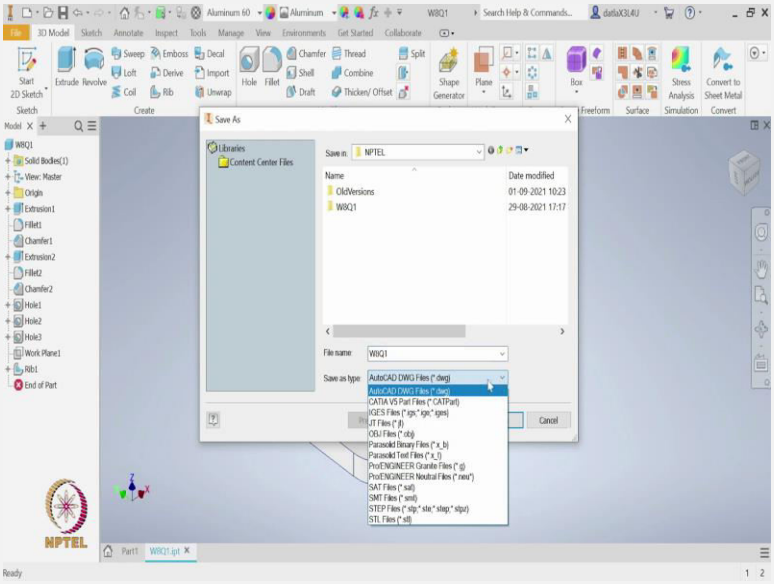

So you can also do the Export which means it will give you several options of how do you want to save this. You can save it as an image. You can save it as a PDF document, or you can save it as a CAD format. Let us say, we normally use the CAD format when you are transferring your CAD model from one software to the other software.

So once you go to the CAD format you can, if you click on this Save as Type, it will give you the different kinds or the different drawing formats you can save here. You can save it as an AutoCAD drawing, CATIA drawing, IGES files, and many other commonly used CAD formats like the SAT file, SMT file, STEP files or the STL files.

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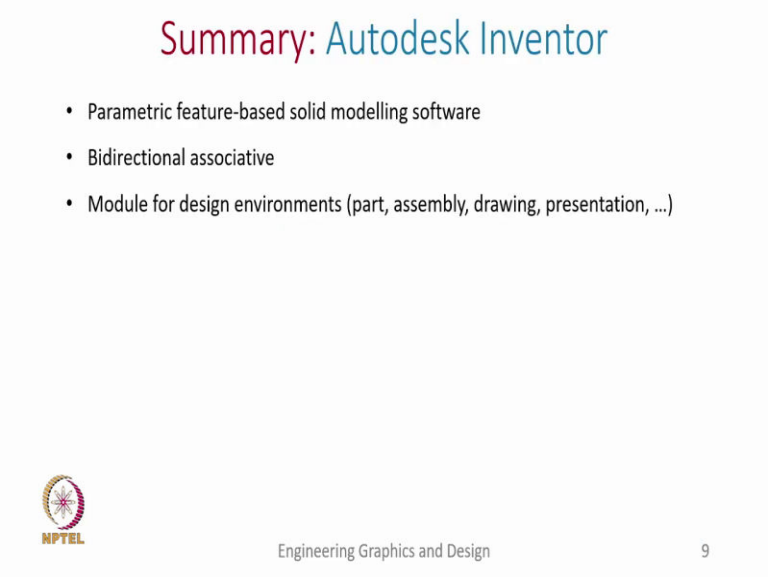

So let us conclude what we have discussed in this lecture. We introduced to you what is the AutoCAD Inventor, saying that it is a parametric, feature-based solid modeling software which has this interesting feature of bidirectional associative. This software has many design environments which we call it as modules. So you can make changes in any of these design environments and those changes reflect in all the environments.

So we have looked in different environments like part, assembly, drawing and presentation. We will start with this Part module and in the coming lectures we will help you show how to create a simple part and later we can use, go into the Assembly module to combine these parts to make a device and then come to the desired drawings as well as the presentations. With that I would conclude this lecture.