Engineering Drawing and Computer Graphics
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Module – 07
Lecture – 54
Overview of Computer Graphics – IV

Hello everyone. Welcome to our NPTEL online certification courses on Engineering Drawing and Computer Graphics. In the earlier classes, we have covered manual drawing, and also then went ahead to try to introduce about surface modelling. And now, we are going to introduce software, mainly named Solidworks. This is quite popular software in the manufacturing sector, and we will try to have some idea about this software uses also.

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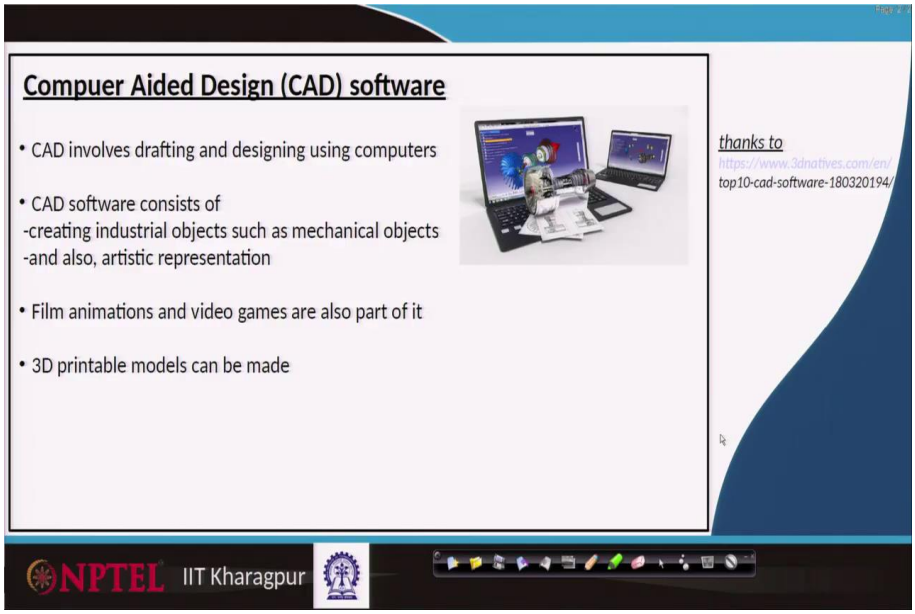

So, to begin with, any software, we use that to design and draft the things especially we use computers to use in designing objects that kind of process what we call computer-aided design. And the software whatever we use is popularly called as CAD software.

This Computer-Aided Design involves one drafting and the second one designing. The main difference between drafting and designing is, you come up with one kind of sketch part it has to be in that way, but when you are designing there might be other issues like manufacturing issues or perhaps in terms of strength of materials what we are using and so on.

These things may have to be changed. So, usually, this drafting always is clubbed with designing and most of this CAD software does both the thing. The CAD software consists of one creating industrial objects such as mechanical objects, and also they will represent artistic views, both are possible by any CAD software.

Even for film animations and video games, these days people are using CAD software. Especially, these day in manufacturing sector 3D printing is a popular mantra and CAD software helps a lot in terms of preparing these 3D materials.

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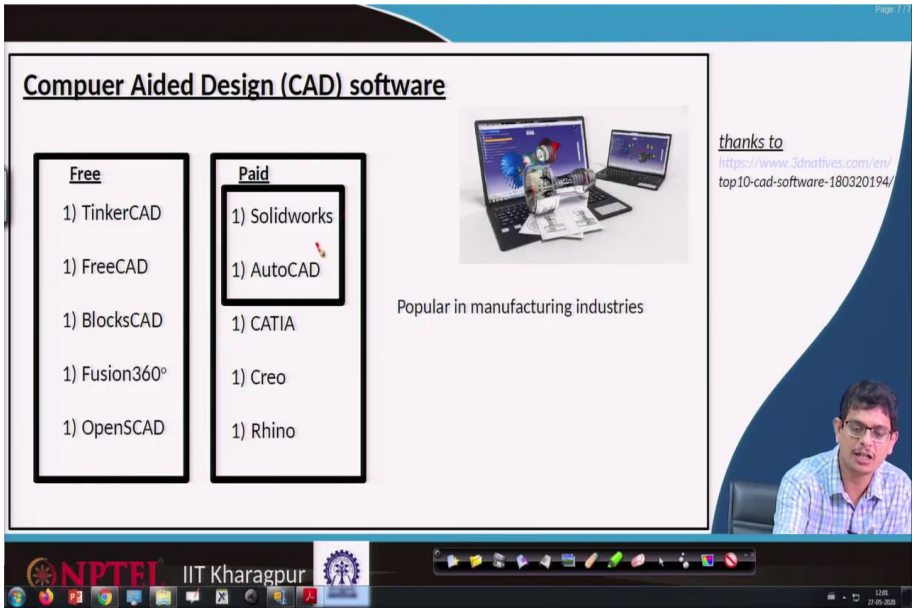

Based on my experience, we are dividing this software into two parts, one freely available or you pay a very partial amount and the other one is paid software. Each one has its advantages. Example free software you do not have to pay any money, you can just practice it, construct simple sketches, they might be very simple things, they might not have very big objects, they may not be supporting other things and many options might be missing, but still, it is a good step, to begin with, that.

For that, we are categorizing TinkerCAD one can use, or FreeCAD, Blocks, Fusion 360 degrees and OpenSCAD. These are the freely available software. There are other varieties also. The industry mainly goes with paid packages because they have multiple advantages in terms of preparing and design materials.

Perhaps you want to prepare a gas turbine blade with the jet engine also. Then, its enormous amount of data one has to represent. Free software may not be in a position to handle it, but this paid software

may work, in terms of storage, multiple options, giving a different kind of colours, maybe giving you simulations, this extra information also this paid software provides.

One of the popular things is Solidworks. The others are AutoCAD, CATIA, Creo and Rhino. There are other things also there. But these are the popular software used at the industry level. Especially, in manufacturing industries, Solidworks and AutoCAD are the popular choices.

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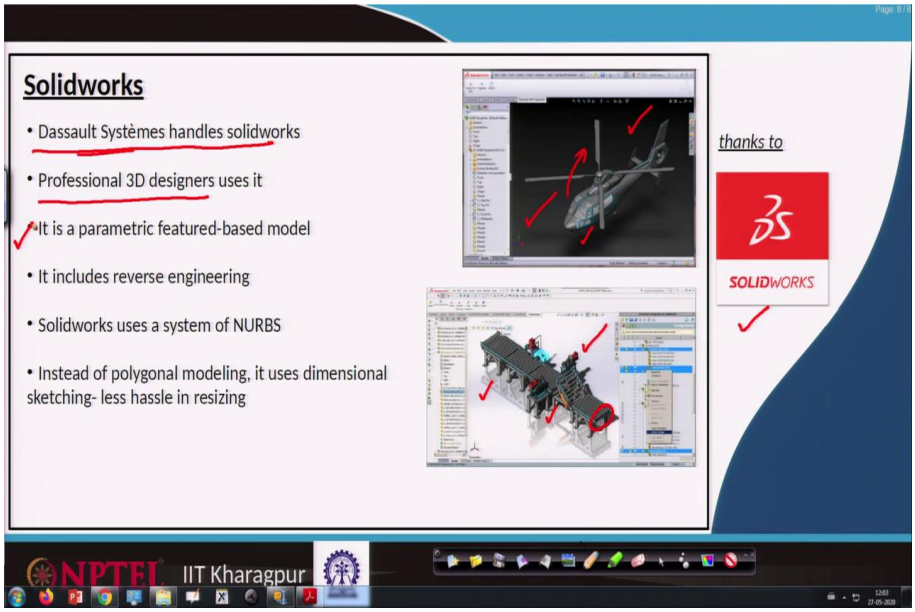

First of all, let us quickly revisit this Solidworks software. The Solidworks software mainly handled by Dassault Systemes and especially for professional 3D designing this Solidworks helps a lot. Here we are showing a picture constructed using Solidworks.

There is a helicopter blade, there are nice shade and other materials. That kind of objects we can construct it using Solidworks. You can rotate the blades and try to observe the things also. Even if someone is trying to use other specialized continuum mechanics for the grid generation and other techniques, Solidworks always be helpful.

Similarly, here I am showing you another industrial process, something like conveyor belts. It includes many parts like rollers, bearings, main structure, there is something like a shade and many things. Such kind of visualization is polished possible by Solidworks software.

And this software is based on the parametric featured based model. In the last classes, we have seen parametric variations means you want to construct something like surfaces, you have only discrete information, from there one will be in a position to construct remaining points and visualize in a better way.

That kind of parametric variation is possible by this software. And also it includes reverse engineering; that means, you already have constructed the object, then you can disassemble it, observe what kind of material, what kind of dimensions, how to analyze that also possible.

And especially Solidworks uses a system of NURBS. This is the last classes, we have seen, when we are going to construct this surface modelling we have different strategies like Bezier curves, Bezier surfaces, and coon surfaces and so on. One of the popular method what Solidworks is using NURBS, this support is available.

And it mainly uses polygonal modelling. So, for surface constructions, we have seen how these polygons are helpful in terms of better visualization. And also, it supports dimensional sketching and creates less handling in terms of constructing these sketches. When we are going to learn about Solidworks try to practice a couple of examples, we will see how easy it is to construct 2D objects and 3D objects.

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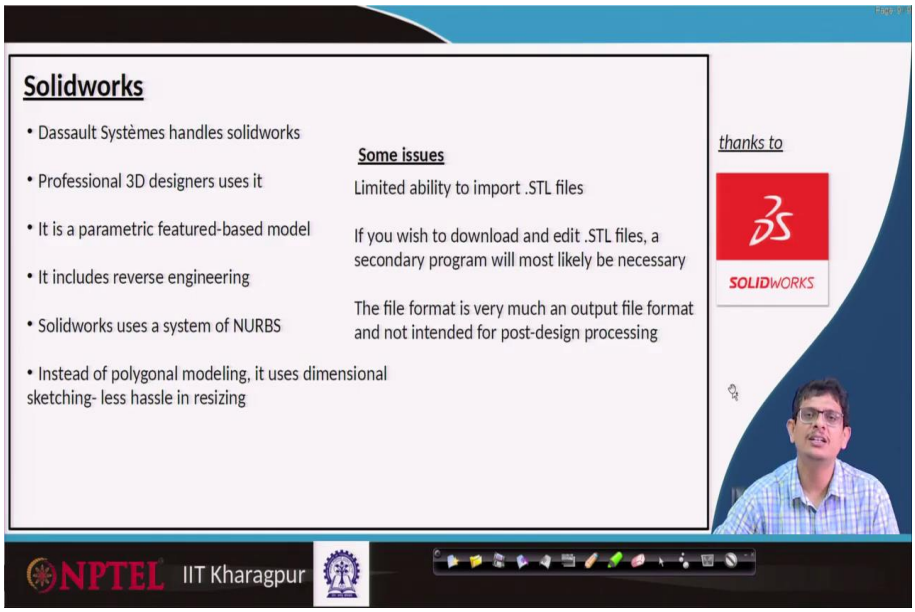

There are some issues also because every software has its own merits and also demerits. Some of the demerits which are very minor things are a limited ability to import STL files. But these days, Solidworks is coming up with new features even to handle these specialized formats for these files.

For example, it supports its kind of files like SLDPRT format, basically Solidworks Part modelling, solid work assembly kind of things and so on. But if someone is accustomed to old kind of format like STL kind of forms, then importing those things slight issue and one may require additional

supports and modules which is a very minor thing. For a beginner that might be an issue, so that can be easily handled.

And the second thing, if you wish to download and edit STL files a secondary program must be required as we can easily get it on online resources with the help from groups.

And the third one, the file format is very much an output file format and not intended for post design processing. That means once you construct that you can use only Solidworks to edit that parts and assemblies, you cannot use some other software to edit that file. That support is very minimal. These are minor issues.

As long as we are sticking with Solidworks trying to learn these issues are not any hassle thing.

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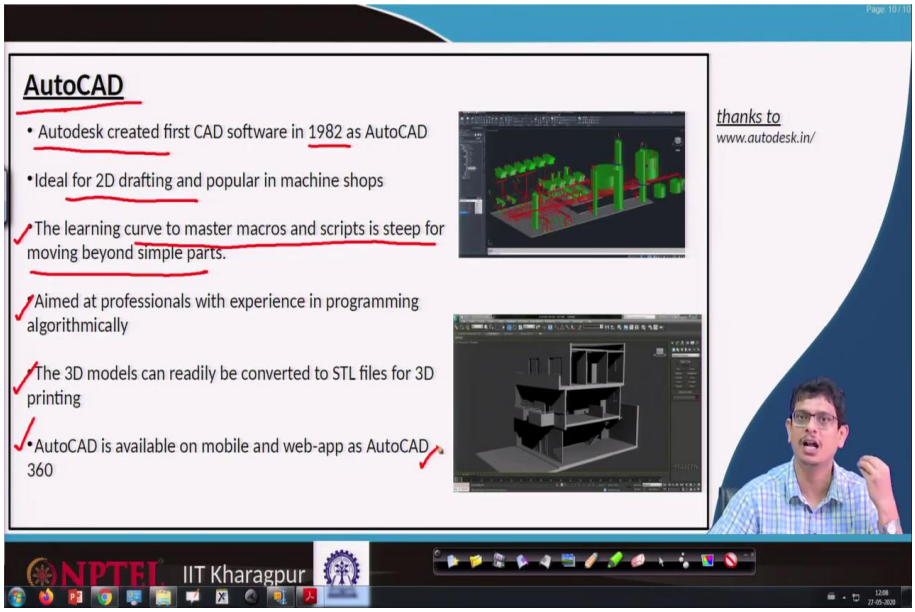

The other popular software in designing is in manufacturing AutoCAD. And this AutoCAD is mainly first created by Autodesk, as early as 1982. These are the first pioneers in terms of constructing these drafting using computers.

If you are mainly going for 2D printing or 2D drafting, blueprints and other things this is the best tool what one can have. This is the easiest way of constructing things. And especially if you are going to any workshops machine shops still AutoCAD is quite popular, because you want to send your information in a very effective way AutoCAD gives you that kind of advantage.

The learning curve to master macro scripts is steep and moving beyond simple parts. If you are reasonably good with programming and you can easily construct macros then you can build even

complicated structures. And this is mainly aimed at professionals, and who love to go for the algorithmic way of looking at the things.

And the third one, the 3D models can readily convert into STL files for 3D printing. So, when you are going for old-style formats like STL things, AutoCAD enormous help it gives us an advantage. Nowadays, AutoCAD is available even on mobile and web apps as AutoCAD 360. It has limited versions, so one has to be careful with that in terms of paying money and other things.

So, it is not free software you have to pay but the advantage is you will have very beautiful designs one can construct it. If you are a professional this gives you enormous help in terms of designing the objects.

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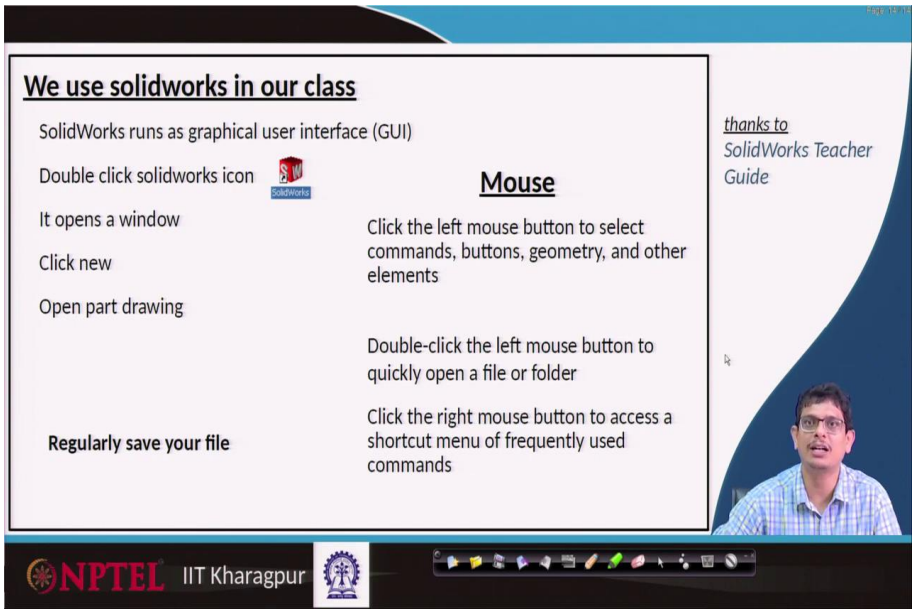

Now, for our course, we prefer Solidworks uh because this is meant for very beginners where they do not even require any little bit about programming skills, ok. You can find these SolidWorks, you can go through SolidWorks, most of the colleges these days might be having solid work, so, based on that we are going to teach you basic preliminaries on SolidWorks.

Once you go through this class you will be more or less comfortable in constructing 3D objects. We will practice a couple of 3D objects also in the later classes. So, Solidworks is a graphical user interface based application. You click the things you get the things.

The first step is you always have to double click the Solidworks icon, so it opens it. It opens a window. Then, you have to click something new to construct an object. Then, you go with part drawing construct it step by step and finish the thing. When we are practising we will see. The first step is

always double-clicking, then it opens a window, you pick part drawing, then go draw the path drawing, and then regularly save the file by drawing the things. ah

Because uh some of the students are not pretty familiar with this mouse left-click, right-click kind of thing, so we are going to recap that information. You always use the left mouse button, something like if you have a mouse, in that mouse the right one, left one will be there. So, the left mouse what we are going to use? To select any commands, buttons, geometry or other elements. You double click that left mouse to quickly open a file or folder.

These are the kind of shortcuts what we use. And the right side of that mouse button any shortcut menu which we will like to quickly update it we use that. So, the mouse is an enormous help for us in terms of picking the points or updating the points or curves. And the most important thing throughout this Solidworks practice what we are going to do you have to regularly save the drawing.

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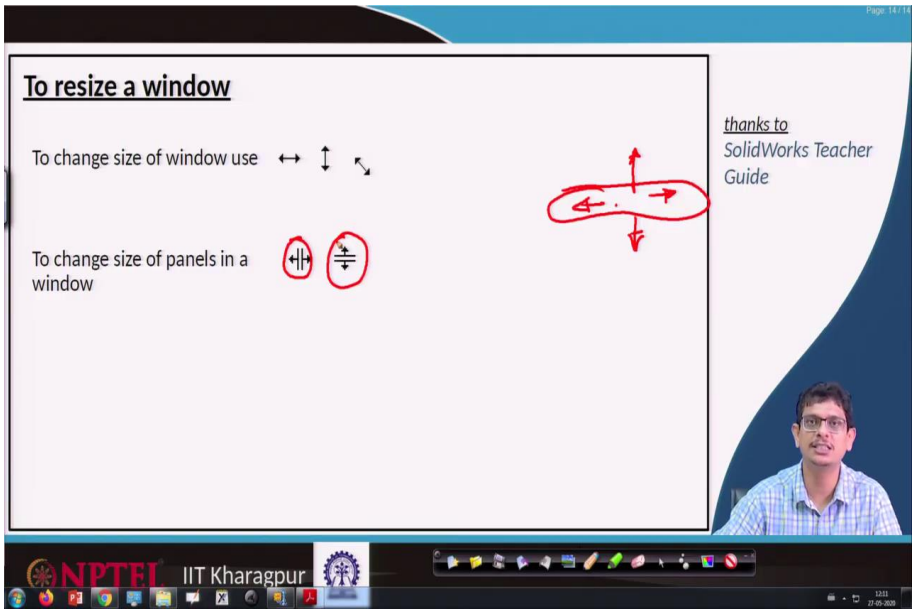

You want to change the size of this window which we will see it later. You use on your keypad there are buttons like up arrow, side arrow, another side arrow, and down arrow. You use these combinations to change the size of the window, maybe moving up, and down increasing, enlarging and other things. You want to increase the size use these side buttons together. Similarly, you want to increase that size use these two buttons together, you press them together so that it enlarges or change the size. So, these are the combinations what we will use.

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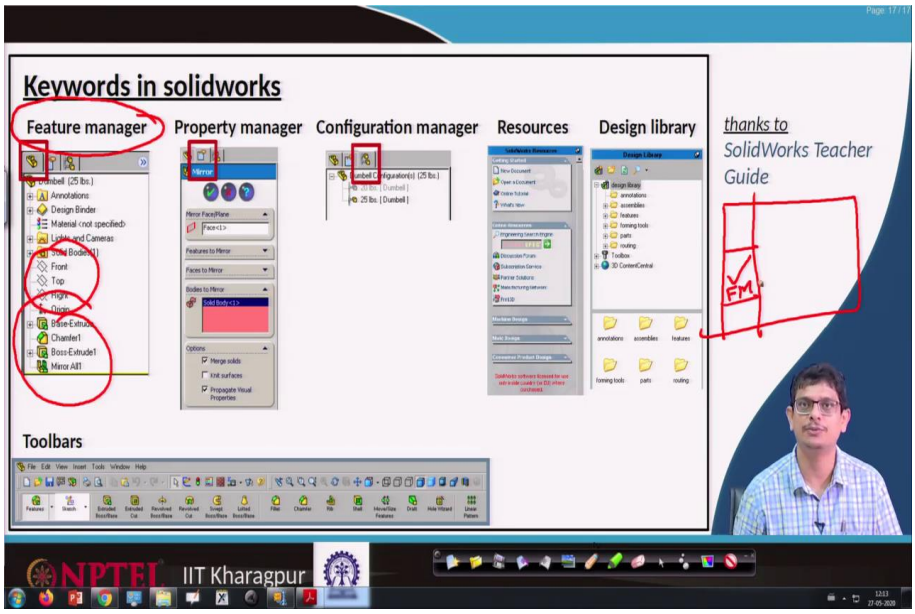

Now, first of all, we have to learn a few keywords in SolidWorks. After double-clicking these Solidworks we will have something like a panel, where some of the things mention like these are the data, there will be some other places some other data mentions, some other locations some other data will be mentioned. There will be something like toolbars menu and many things will be there.

Most of our drawing we work in this X location. Remaining shaded portions we pick appropriately based on that we use those tools. Now, let us look at how they look like. For example, in that, we might come across a keyword name feature manager. In that feature manager, we might be having something like front, top, right planes.

When we are constructing these objects we may have to occasionally click whether I would like to construct front view, right view, side view or on which side we would like to construct. For that purpose, we have some of the options.

Similarly, whatever the operations like whether I am extruding the object, making a cut, fill filling some surface, filleting, chamfering this kind of things are also some of the objects it will mention. Whatever these operations we are doing features we call and that manager or that portion of your Solidworks we call feature manager. So, usually for Solidworks, we see it on the left-hand side, somewhere here we have that feature manager.

There is something like the property manager also. So, if you are looking at that feature manager there are 3 buttons, 1, 2, and 3, each one represents each of these managers. So, the first button is the feature manager.

Now, let us move on to that second button that is what we call property manager In that you will be in a position to know which face I am looking at, whether that face is added by any edge or not, what kind of solids are have been meshed and other kinds of details. So, those properties we will get it at this second button where you will have an icon like a blank page.

And there is a third one option, that is what we call configuration manager. In that, it shows what might be the material properties, size, weight, dimensions and other things we will have it. Over time we will acclimatize with these words and will be in your position to sense what is happening.

And most of the resources you require any technical help regarding learning Solidworks one on online you will learn, another way you will have something like how to create a new document, how to open a new document or perhaps how to get online tutorials or perhaps some other subscription services you would like to have these kinds of details we will get at this resources.

So, there always be a title like solid work resources. Anything you can post it there and you will be in a position to get the information about that. And there is one more keyword name design library. That tool also comes either on the left side or right side based on how we are dragging and dropping at this suitable location, where you will see some of the things like whether we are going with assemblies, whether we have touched any features, whether we have given any annotation, all these details we will be having it.

And that information again we will have something like background information. When you are looking at floor shopping and other things, we will extensively use that. For our course, we may not require all that information. The basic things like how to sketch, how to make 3D objects, how to assemble made them is the most important thing for us.

And there is another important object which we call toolbar. This is more like a workbench where you get most of the tools. For example, you want to sketch the object, that object sketch portion will be there, you want to fill the material, something like extrude will be there.

You want to revolve the things in the last classes we have seen how to revolve these surfaces based on different kind of strategies, how to extrude the objects that kind of strategies also we have seen, how to what it means chamfering, what it means filleting of surfaces also we have seen. So, that specialized algorithms it takes and by just clicking it automatically takes and chamfers the thing. We may have to provide certain additional features for that, ok.

And there will be additional things like I would like to see the isometric view. Maybe I would like to see the only top view, I would like to see the only bottom view, that kind of details also we will get

it or we want to give some specialized focus, we want to give something like a tapering that is also available. So, almost all these steels tools available at this level. So, based on your expertise how much you are going to prepare you will be in a position to effectively use these things.

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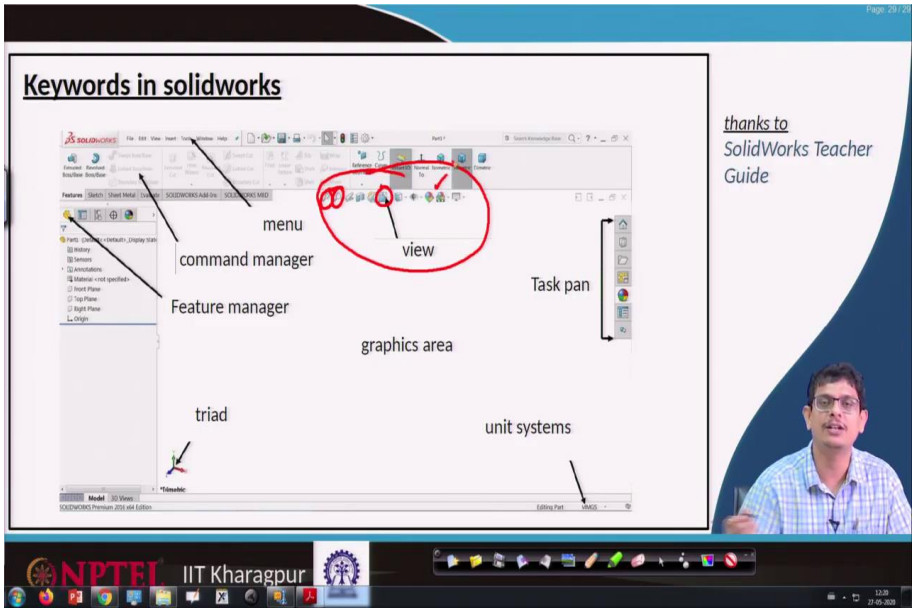

Now, if I would like to summarize this entire keywords information, it is not only this way we can have it, sometimes because of your software issues you might be dragging from one location to other location also you might be leaving. But in general, by looking at those words we will be in a new position to sense which part it belongs to. Let us look at that.

The first one triad. So, whatever you are drawing happens that entire drawing you will be constructed within this red portion of that object. This is the terminal where you pick the point, draw connected by line draw anything you will do it. On the terminal, it always shows you the x-axis direction, y-axis direction, z-direction also. There is an option to rotate when you rotate it this triad also rotates. So, most of the drawings what you work on this area graphics area what we call will consist of a triad.

And the second part is the dimension, the units whether you are working on centimetres, millimetres, inches that kind of units will be mentioned there. You click that option it shows you different things. So, you have to adjust it.

For practice, we will go with one kind of system of units. Once we are acclimatized we will learn how to change those dimensions by clicking it and change that. But throughout your geometry remember that you have to work on one system of units. You create something object, save it like

part drawing, then you want to make it assemble unless these units meet you will not be in a position to get effective drawing. So, be consistent with all the time with one kind of units.

And there will be views available here. So, the isometric views, top view, front view these things, you will have it by clicking that. There will be zoom in, zoom out buttons also will be there. Colour combinations also available there, which one can use that. And most of the menu we will be having here. You want to save a file, you want to edit it, you want to insert tools, other objects, you will have it.

And features managers like I said, those things will be available at the first button here. So, which contains your front, top, right and whatever the case study we have done. And additional task what we would like to do, that will be given at this task pan. When we are opening clicking the things we will better understand these things.

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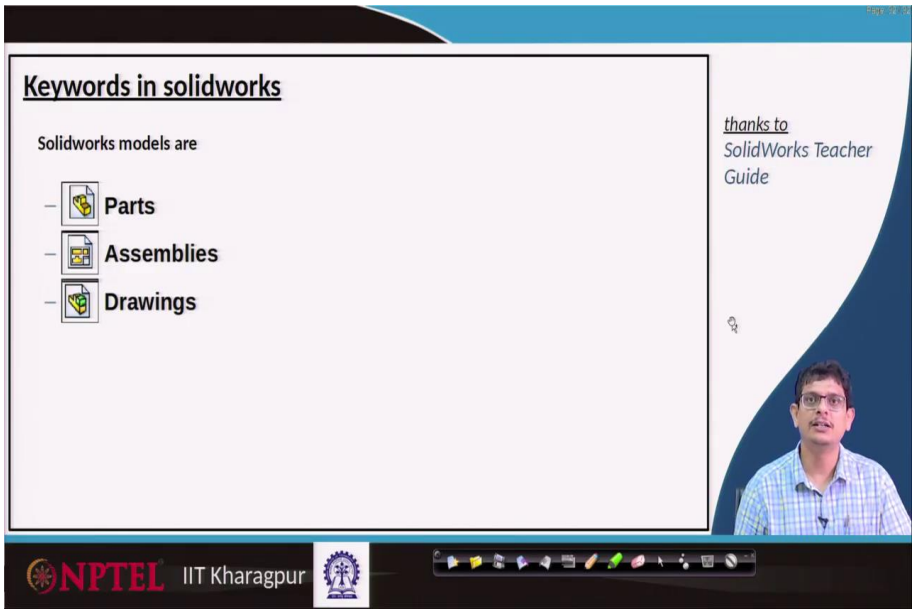

Especially, the Solidworks, if we are going to divide it consists of mainly 3 parts. One you prepare individual parts, save the individual files, then import those individual parts to make assemble. That means, for example, I want to construct a by skill, maybe I want to construct something like a pen, what I will do is I will prepare a cap, I will prepare a refill, I will prepare something like bottom portion, save individual parts, then carefully load it, tighten them, so that a combined part like a pen can be made.

A similar thing happens even for a cycle, you prepare something like a seat, chain, sprocket, wheel, tire, individual parts you carefully prepare it, save them, then load them, assemble them and visualize

the objects. And you want to transfer that information to someone like local shop guy, then the way is you make something like a drawing sheet.

So, once you prepare that there is an option to make a drawing sheet, you click that it gives you a two-dimensional picture which you can easily share it with anyone. So, whoever so going to manufacture they will prepare that.

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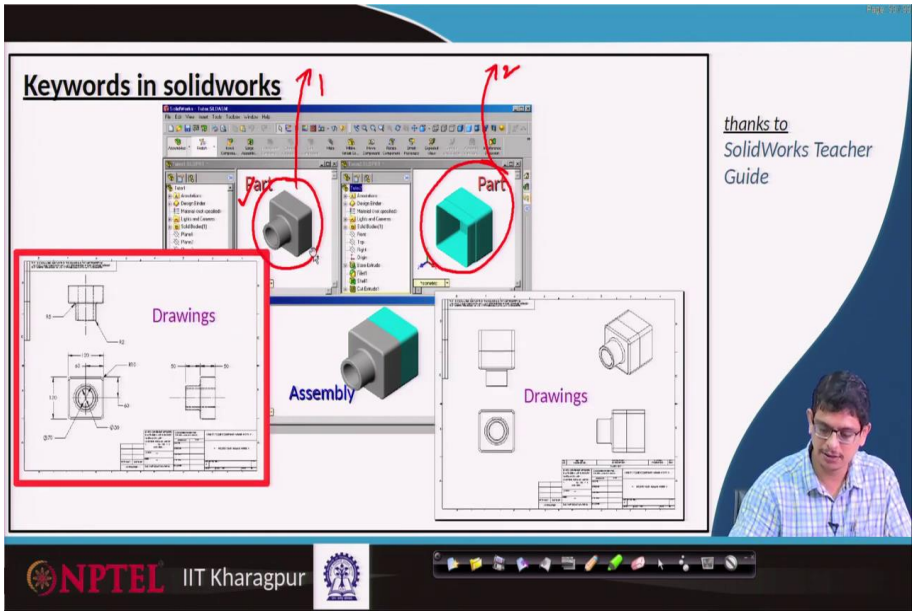

So, if we are going to summarize this thing, you prepare the objects.

If it is one single object, what you are going to prepare? Then, you call part drawing. Here also we are going to prepare only one single object part. This I will save it as part 1, this I will save it as part 2. Then, I want to assemble it. Then, I load each part, I will prepare each part, load it and make a mating so that a single object as an assembly we will get it. This is the way most of these things happen.

And over that, we want to prepare a drawing sheet, there will be buttons options once you create this assembly thing from individual parts, there will be buttons which you click it. So, it makes something like what is a top view, what is the front view, what is the side view and what are the materials have been used, whose name is there, and when they have prepared and so on so things and gives you those dimensions also at manual level.

We are the ones deciding what should be that sheet, draw it carefully, divide the task like in the earlier classes we have seen something like division a, b, c, d, 1, 2, 3, 4 and this system of units, and perhaps something like titles labels, all these things we are manually preparing it. But here at the software

level you straight away use dimensions create the part, you assemble it, then you want to separate the parts, you can straight away construct those individual drawings, either for assembly or for part drawings you can make.

Here we have that part drawing information. If it is assembly you will have individual parts and also the isometric view you will straight away get it. So, all that manual laborious task will be very easily taken care of by this software.

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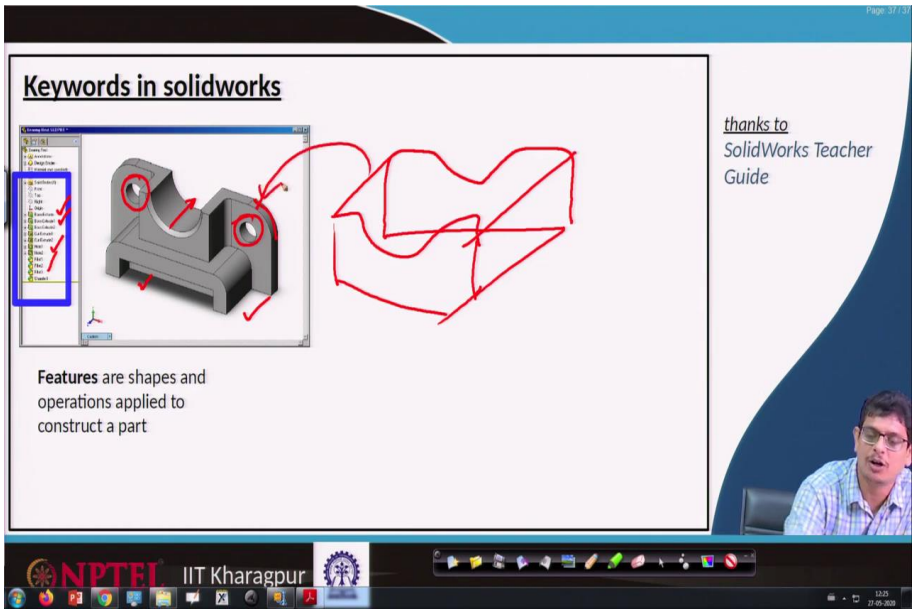

There are other keywords also involved in Solidworks. For example, once you prepare this Solidworks drawing, there will be something like boss, extrude, cut, hole kind of thing. So, whatever the operations we are making this is the cut. This might be something like a fillet kind of surfaces.

This is also cut. There might be extrusion happen because first we have constructed some object like that, and then projected that into 3 dimensions and fill that material. That object we got. So, all these operations you will have it at these feature managers. So, whatever the operations we are doing it will be pretty clear. Even if we are making a mistake at one level we can delete that, and continue with the remaining things. So, this feature is a powerful concept in SolidWorks.

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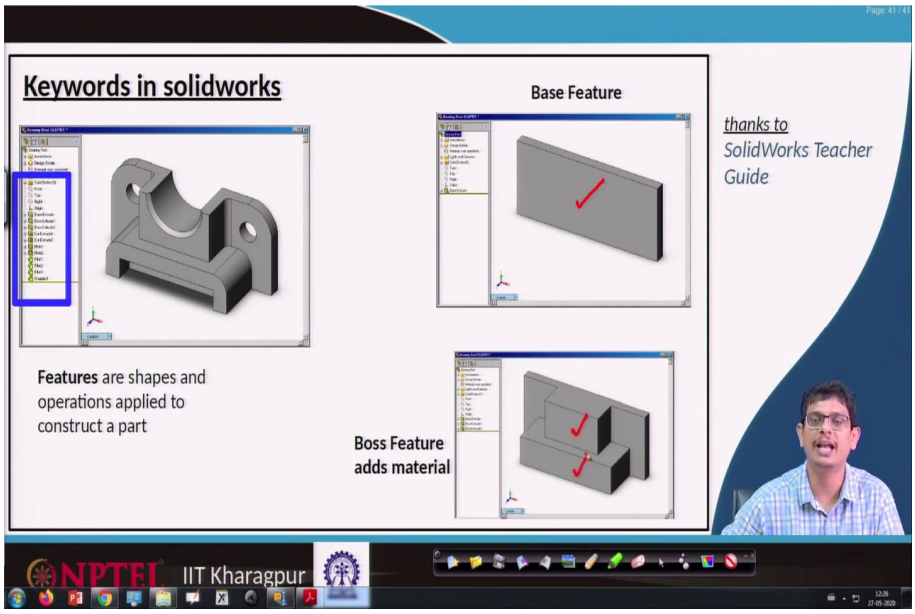

Anything that we call base feature is the preliminary version or the starting version of the drawing. For example, I want to construct this object, I may not be in a portion of straight away first of all cut the things, make holes and other things. First I will make something like a rectangular block. On that, I will make an operation like remove that portion, remove that portion. So, this entire portion we will be removing. So, the first one what we are going to construct carefully that is what we call base feature.

Boss feature is the one where you are going to extend the things add material and other things. So, we have prepared base on that we are going to add boss information, extra things.

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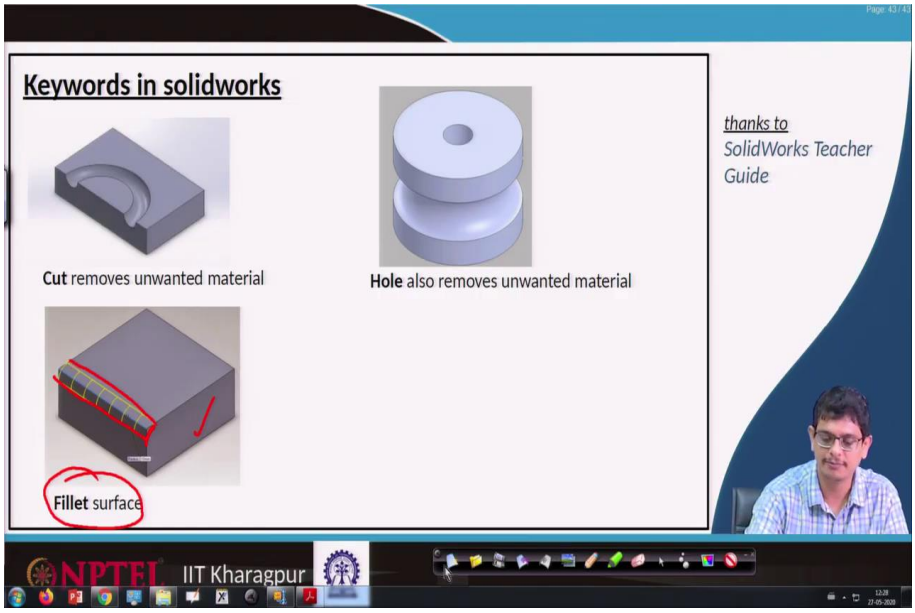

And there are other keywords also involved at Solidworks level, something like we have this object somehow we have prepared from a rectangular thing, and somehow we have created a circle and remove that surface. That kind of thing what we call cut.

For other objects also we have cut, but this is more like a uniform cut what we are having. So, one can use an option like a hole. You just want to make a drill, hole. So, that hole options straight away available. A specialized feature like cut requires additional components like what is the depth of cut, what should be the arc and other options we may have to provide by clicking the things.

There is one more object named fillet surface. So, we have already object we want to make it rounded a smooth corner, so for that purpose, we create something like a fillet surface by clicking the options. We will see that when we are learning.

And in the next class, we will begin with Solidworks as a practice thing. So, step by step we will construct rectangle, ellipse, circle, curves, and try to extrude it and so on other things we will see, ok.

Thank you.