Metal Additive Manufacturing
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Lecture 34
CAD and Topology Optimization and Laboratory Demonstration 1

Hello, everyone. Welcome back to the course on Metal Additive Manufacturing. This is another laboratory demonstration session, in which I would like to demonstrate small introduction to the CAD softwares, and I would also to like to take small part building, using Solidworks, that we will use for topology optimization in the next part of this lecture series.

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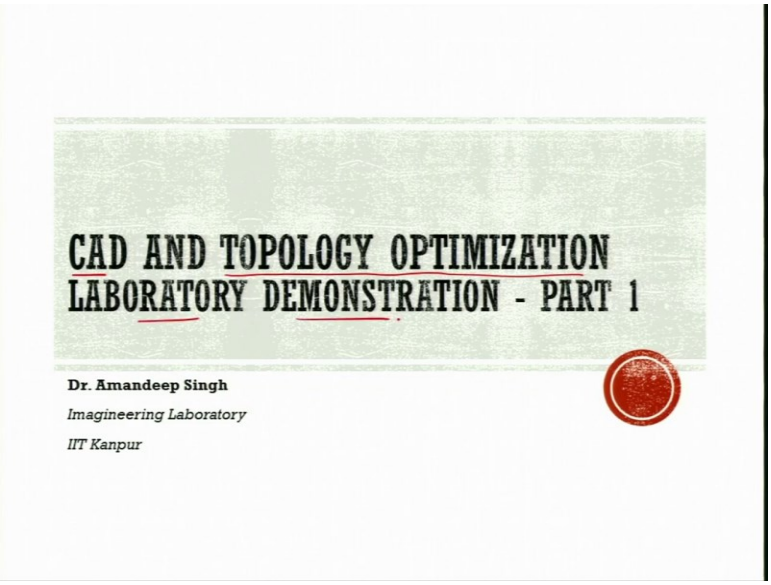

What is CAD and what is topology optimization this is all discussed in the lecture series taken by Professor Ramkumar. I will only have a demonstration on laboratory. I will try to see how various icons, various tabs are there.

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Before that, let us have a quick look on what are various general CAD softwares available. Then I will come to the demonstration.

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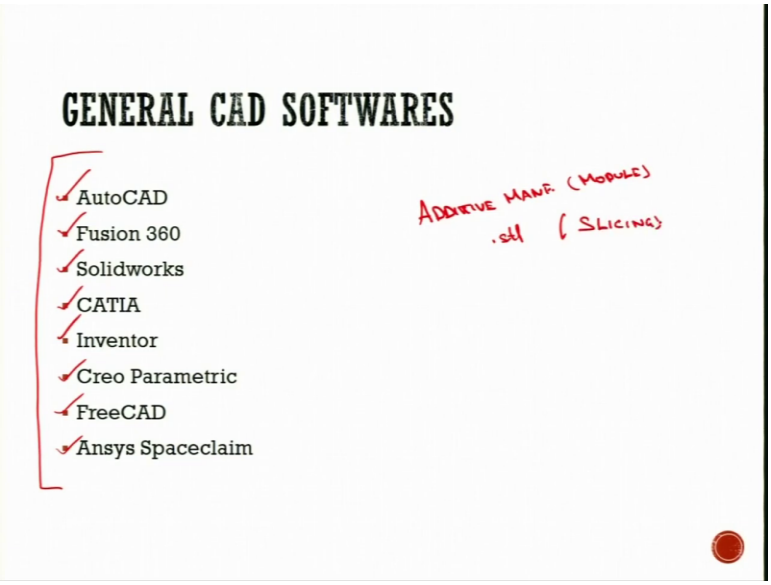

The general CAD softwares which are available in the market are AutoCAD, Fusion 360, Solidworks, CATIA, Inventor, Creo Parametric, FreeCAD, Ansys Spaceclaim. Some of these also support for the computer aided manufacturing, some of these also have the simulation inbuilt within them, simulation systems, like Solidworks. And Fusion 360 can also help us to have small simulation processes. For the detail, I will have a detailed

discussion upon the assembly based and the multi component based softwares in the next slides.

So, the purpose of CAD as it was discussed in the previous lecture, is used for architectural designing, drafting, for electrical engineer, he also uses this to design the wiring system the batting layout and everything. Mechanical engineers definitely use it for design and modeling. Interior design, industrial design, manufacturing design, any of your designs should come using these softwares. Each software has its own I would say, features or USPs.

AutoCAD is the base software that helps us to build the 2D drawings. Fusion 360 is a freeware cloud-based software, Solidworks is a full-fledged software with a simulation system itself. So, each of these softwares have its own features based upon which these are available in the market and people use them. Some of them provide the student lessons as well, like Solidworks also provide, Fusion 360 is completely free.

In the industry, sometimes it is not the single software that industry purchase, they purchase a bunch of softwares depending upon their capability. Some softwares are useful in developing these surfaces only, some are useful in having the thermal strength analysis only. So, it depends upon what applications we have. So, industry or a factory or a group of the designers, they keep multiple softwares with them to operate or to conduct different kinds of experiments that they have or the designs that they work upon.

Now, the features of a CAD software, basically if we say it could be a modern and ergonomic user interface. Now, these days, maximum of the softwares are click based only, with the click of the mouse you can just use the software. There is minimal requirement of the keyboard. However, learning keyboards are always easy and faster way to process the software. That is an easy interface for the user to have just mouse and keep on dragging, panning, and using the software.

So, these are user interface must be ergonomic, the colors have to become right, then associativeness of the software to different designs, the smart components. Nowadays, specifically, now softwares must have the additive manufacturing module in them. That

means in the stl format, the file should be able to be saved so that this it could be taken for the printing and the slicing.

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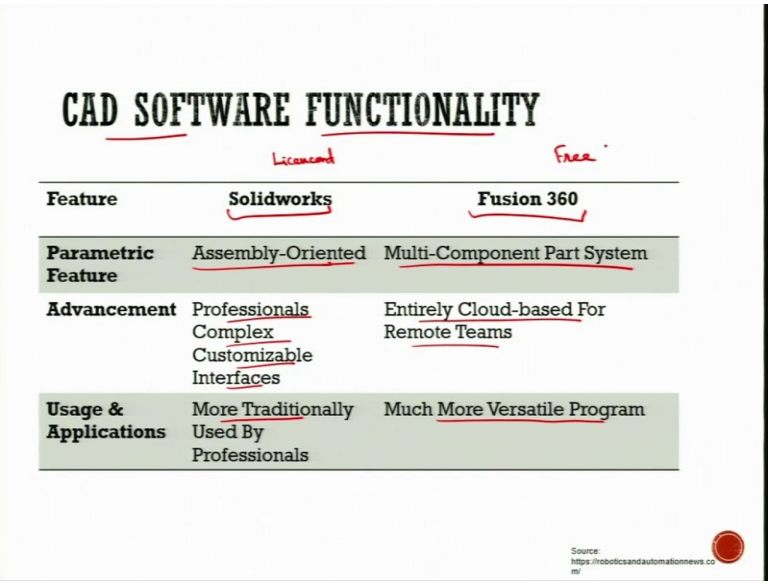

Now, I have a comparison of the functionality of the CAD software. Majorly, the softwares, it could be cloud based software, it could be computer-based software, it could be provided as a lesson that you can just take it from the internet, or it could come in the form of a dongle. Different kinds of the provisions of the softwares are there. Majorly, a classification that I see here is assembly-oriented softwares and multi-component part systems, in which Solidworks and Fusion 360 are compared.

Solidworks is an assembly-oriented software. When I say assembly oriented, that means it is a parametric program. And this provides advanced features for the professionals who like to have complex interfaces, complex designs and these are majorly for the people who are quite experienced in using or applying the softwares. So, it is for professionals for complex customizable interfaces. Fusion 360 is entirely cloud based software and it is for remote teams.

Usage and applications- Solidworks is more traditional, used by professionals, it is a much, much more versatile program. Fusion 360 is also used by the school kids. So, the Solidworks has an early subscription or something in licensing. For example, one has to pay maybe a few lakhs of rupees to keep it up, whereas student version of that has limited

capability. However, Fusion 360 is an almost free software. So, this is free, this is licensed. But we have an advanced version of Fusion 360 as well which is licensed and the cost of the license is quite lesser than what is for the Solidworks.

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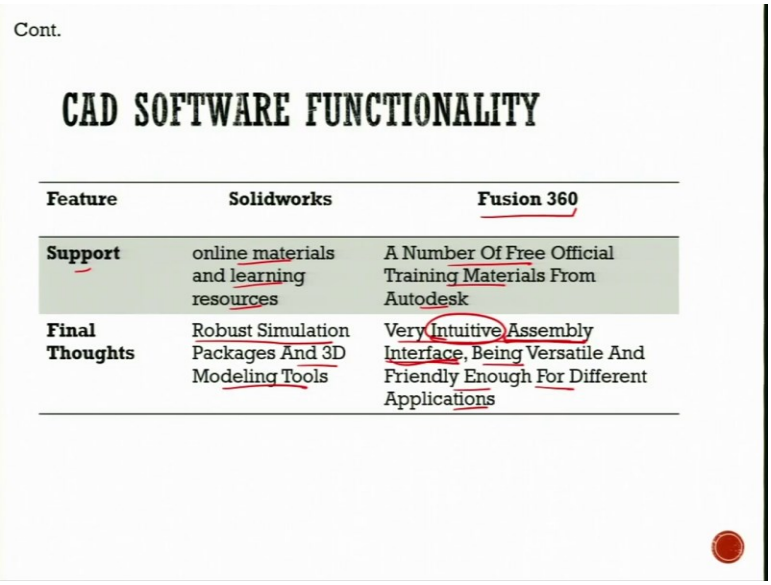

Then, support for the Solidworks in general, if we see, the system hardwares, that is required is quite bigger because this requires the resources or this requires the system that must have features, and everything must be saved on the computer itself. Fusion 360 is a software that is cloud based. So, the space requirement or the system requirement is not extremely high.

For the support, online materials and learning resources are extensively available in Solidworks, or in general assembly-based softwares, and several free official training materials are available from Autodesk, because Autodesk was the pioneer or the very leader who brought the CAD into existence. Autodesk was the first two-dimensional software.

And Fusion 360 is also brought up by Autodesk only, to support the Autodesk files to take an extension to that and to support the whole mechanical or electrical or design fraternity and to provide them the free platform for this, Autodesk has now come up with Fusion 360, which is widely used.

Now, final thoughts, Solidworks has robust simulation system packages and it is also used for 3D modeling tools very heavily. Fusion 360 is very intuitive. The assembly interface of Fusion 360 is intuitive only. So, being versatile and friendly enough for different applications. With this, let us come to the laboratory demonstration where I will start demonstrating the Solidworks software.

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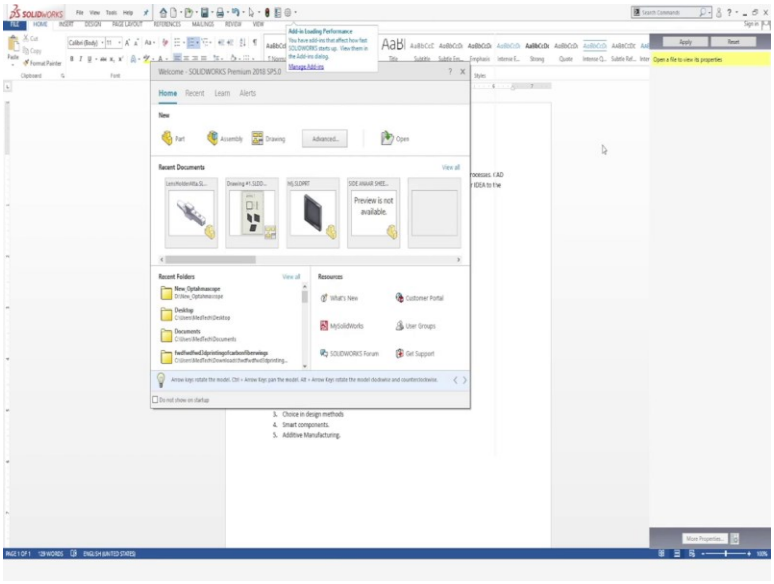

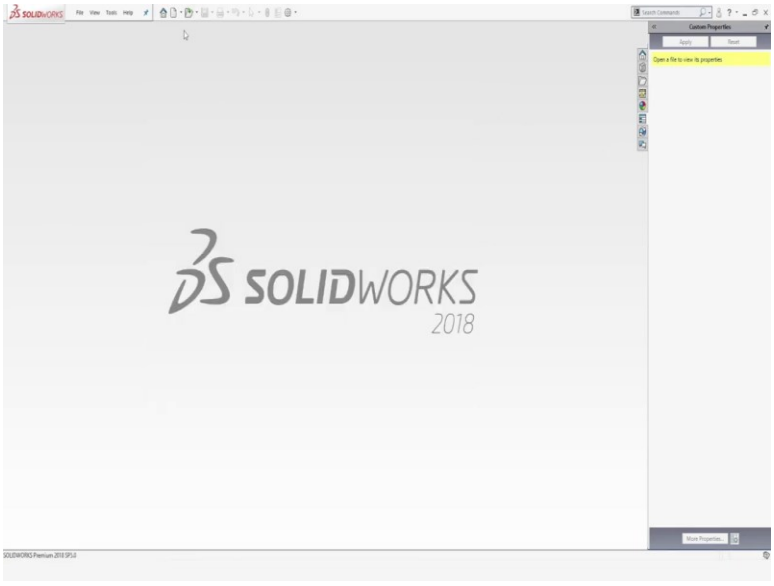

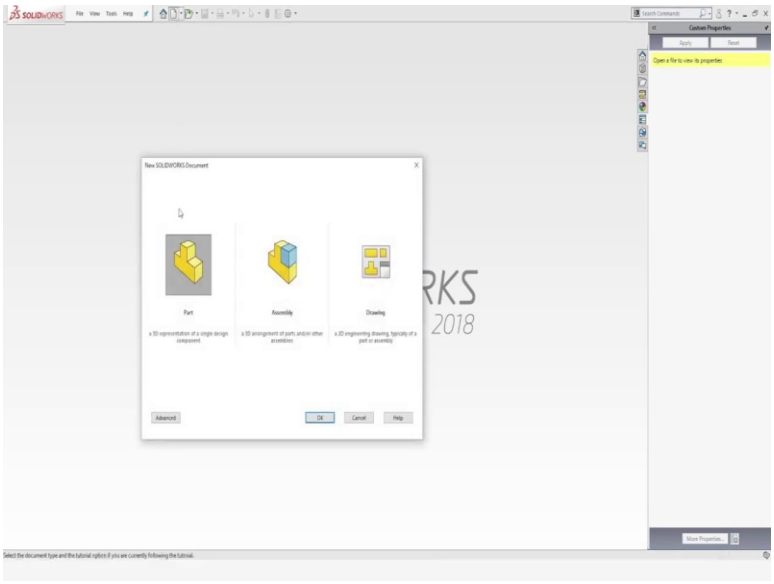

This is Solidworks 2018. So, it contains number of modules. Designing, simulation for different purposes. So, this is the main page of Solidworks when we open it. If we select, there are options which come. We can open a new part, a drawing or assembly. So, these can be opened with 2D representation, that is, the drawing, with 3D representation could be taken in part and assembly.

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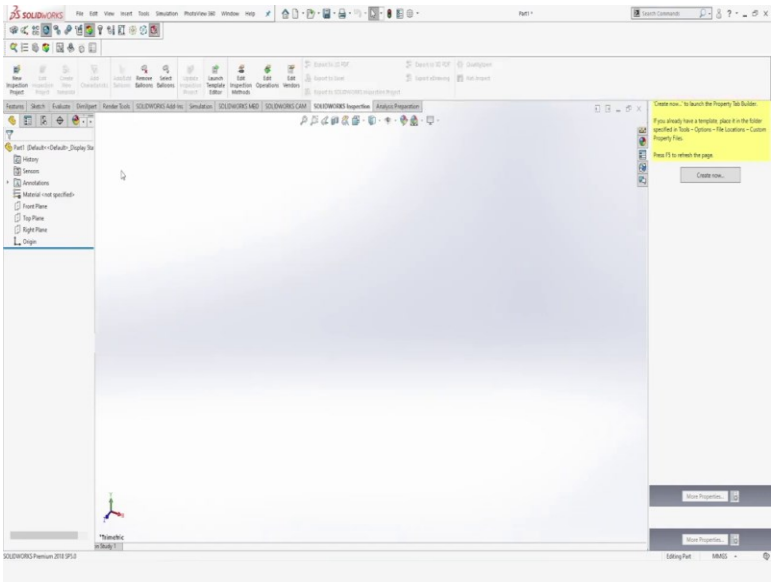

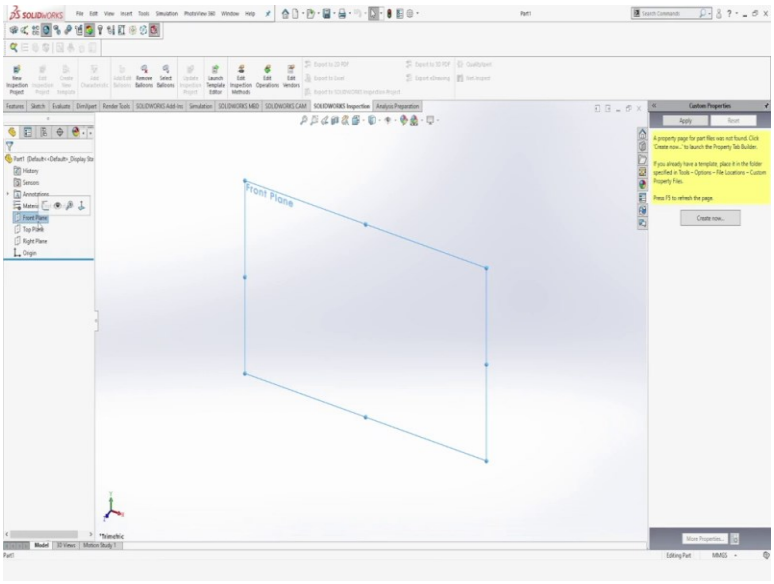

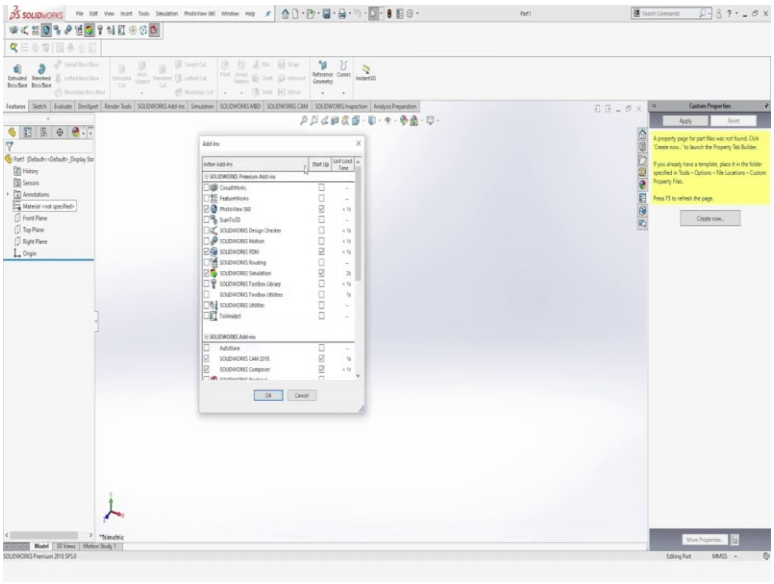

So, if I select part here, so, it will try to open the window, the work window and the work place here. So, this is how it shows. So, for each drawing, for anything to start with, we need a plane and a sketch. So, you can see in the left menu here. So, there are planes kept here, front plane, top plane, right plane. So, I will pick a plane here.

So, picking a plane, before starting any drawing, I need to add the add-ins here. So, there are certain add-ins here for example feature works, photo 3D, scan 3D, solid design checker, Solidworks simulation, if it is checked in. Checked means, this will be, this is already selected. So, in the Solidworks add-ins, for these, we have the cam.

So, if we need to take it to the manufacturing, we can also have these add-ins Solidworks cam 2018. Also, from the library we can pick certain different already made components. So, flow simulation is also possible. For electrical designs, there is a specific module, it has given. So, for additive manufacturing also the Solidworks cam also helps to slice the component or the part that we have manufactured.

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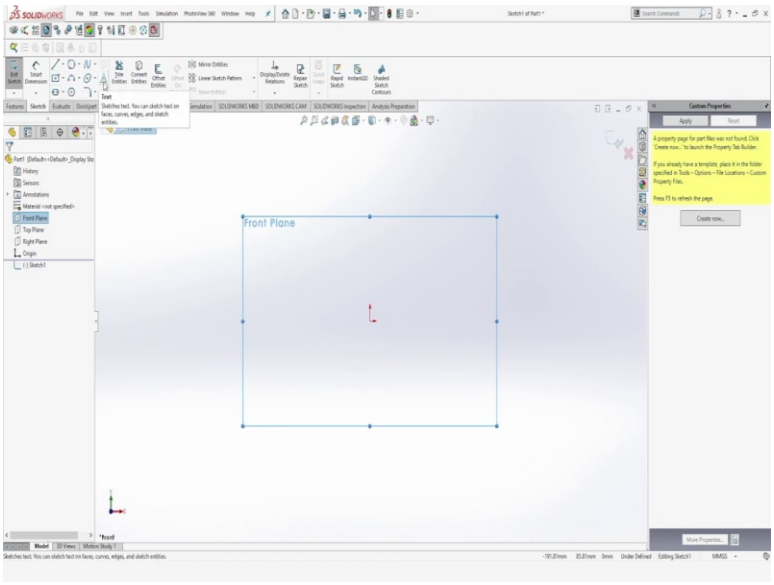

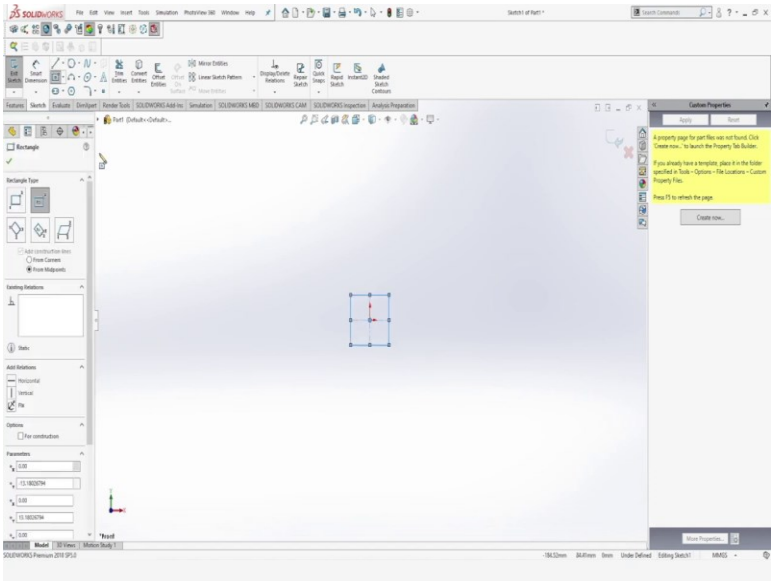

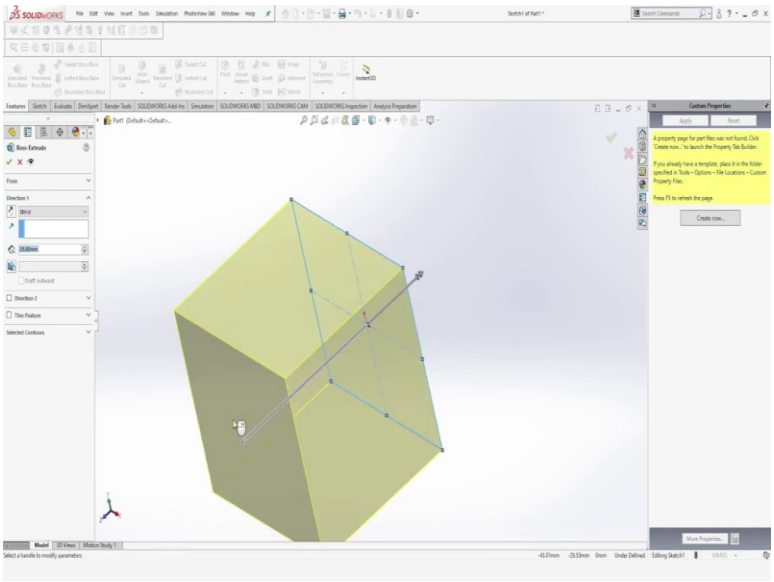

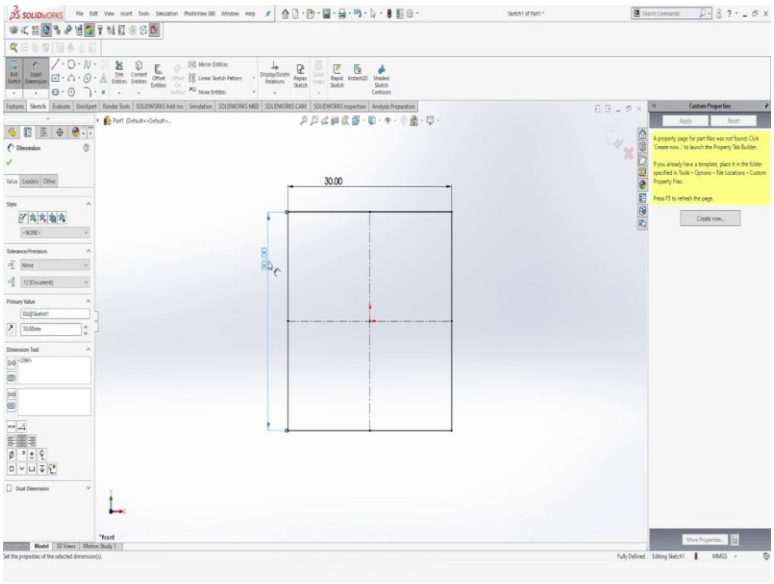
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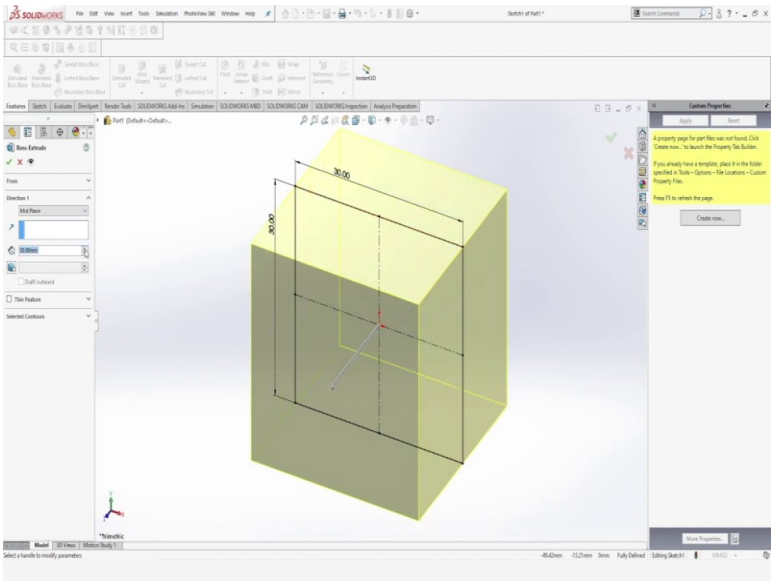
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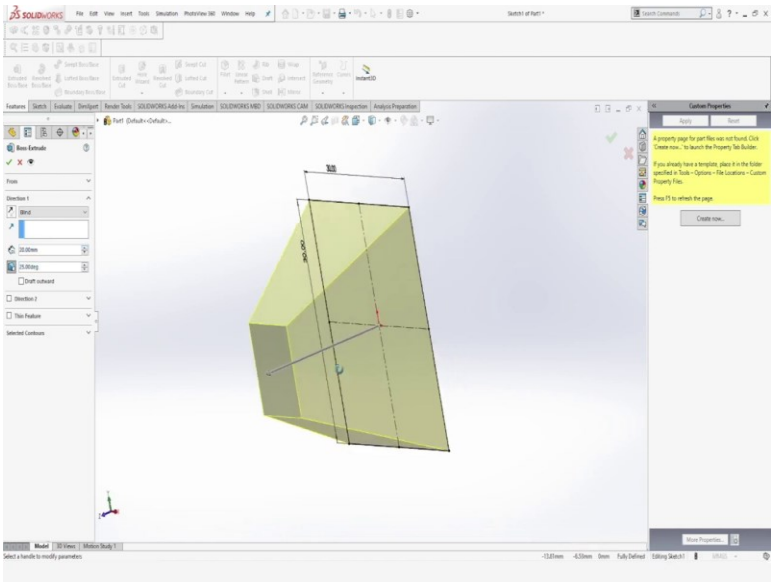
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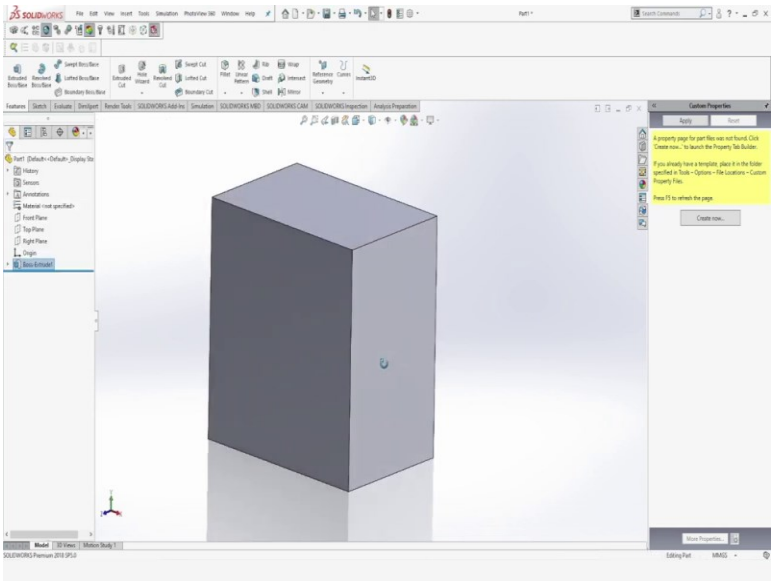

Now, starting the use of the software. So, it is, the first step, that is, selecting a plane. Selecting a plane is, the analogy is like when we need to draw something, we need to pick a scale, a paper compass, etc. Similarly, we have picked a plane here. We will try to draw something. So, if you see this sketch tab here.

So, we have various tabs in the sketch tab. We have this ribbon, in which different sketches are kept, line, circle, fillet, ellipse, parallel ellipse, parabola. Then we have circle, in the circle, we have perimeter circle and circle. We have splines. Splines, we can keep on clicking at certain points, we can keep on snapping and extract the point. Then three point arc, different polygons are also there.

So, we can specify the polygon, the number of edges and angle. So, let me select a simple square here. So, I can just extrude it while coming to the features, and try to say, box extrude. This is a square that is selected here. So, if I take a square, so, I can choose smart dimension and horizontal smart vertical ordinate dimensions, different ways are given here to dimension or to give the size to the square or the edges that we have put here.

So, if I select this, I come to smart dimension and gave here as 30 mm dimension. This is, vertical and horizontal, both are kept 30 mm now. So, this has become a square. So, this is a two-dimensional drawing. So, to make it 3D, we can come to features and try to extrude it. So, this extrudes depth or length, we can give it here. So, we can give up to vertex, up to surface.

In an assembly, if suppose there is vertex, so, for some other part or so. We can extract up to that. So, up to a body, up to a mid plane. So, let us say mid plane. So, now the square, the 2D body has come at mid. Whatever dimension you will give, it will go to both of the sides. So, I can put blind here. Blind means directly whatever dimension I give here, it will go to that.

Also, I can give some, some angle here. So, you can see I am just changing the angle. Let me increase the angle to maybe 25^0. So, it has made this kind of a pyramid here, a truncated pyramid. So, this angle is again turned back to 0. Select. Always, when we put some command, there is a tick, green tick, and also, there is an enter button at the right top. With that, we select, we approve that, this drawing, what we have made, we enter this.

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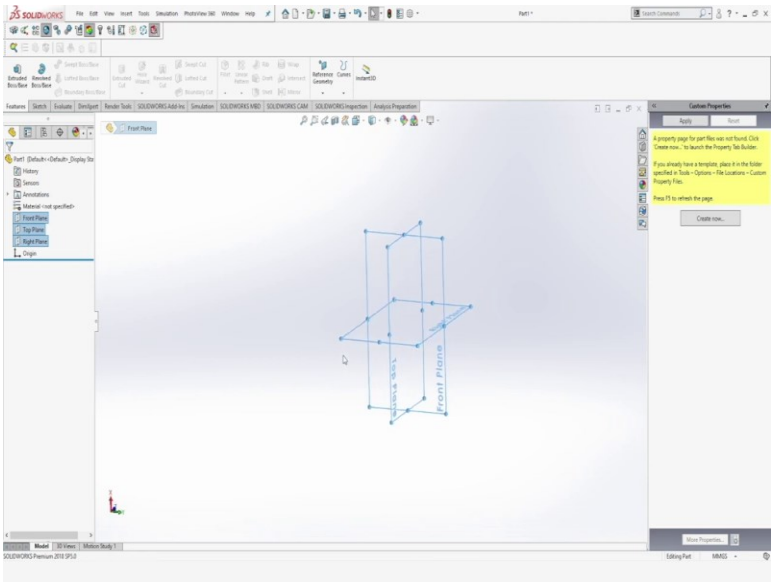

Again, if I come to sketch, from the right, I say okay, this is my extruded part. So, this was one of the examples. For another example, so, let me try to see what kinds of the different planes are there if you put them in a single origin, a single location. So, these three planes are selected, and I have zoomed them to fit in the screen. In the center, they are all, all the zoom options and feature and the view options. So, front, top and right plane, we can see all together.

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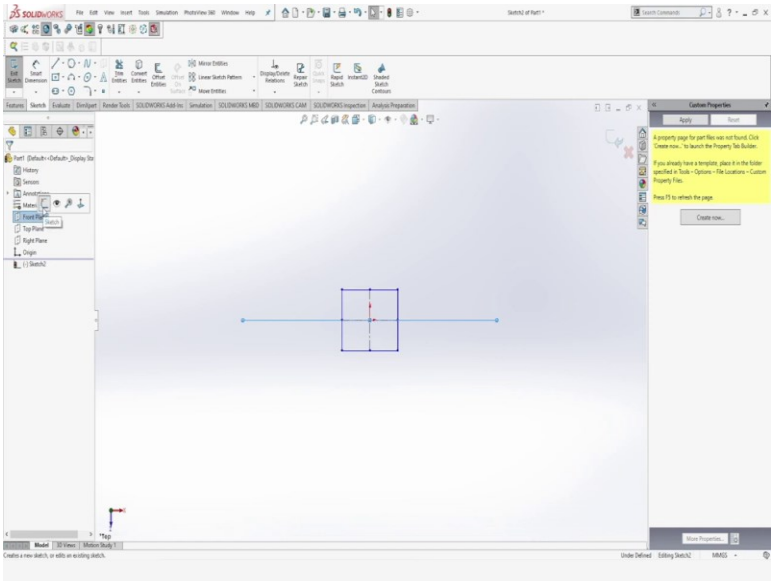

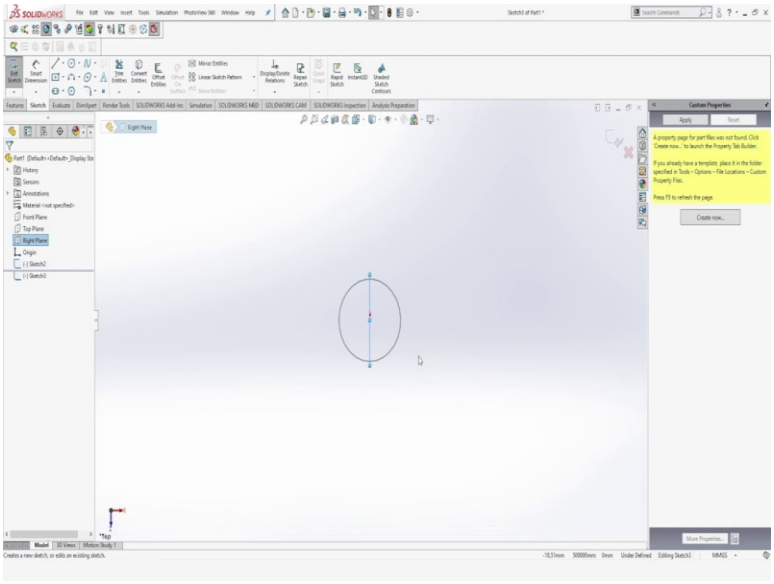

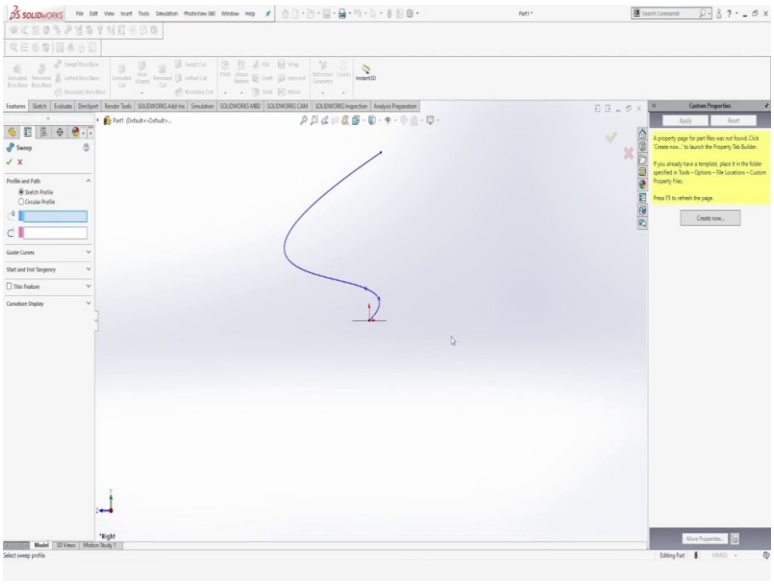

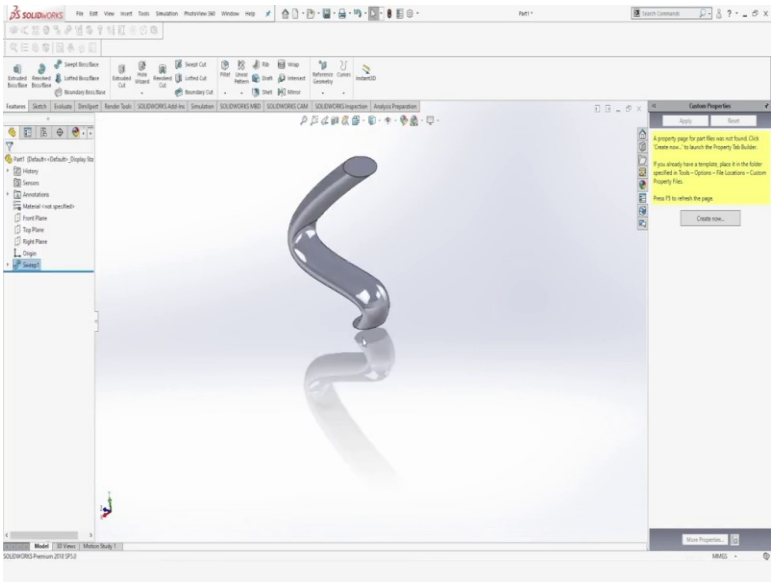

We can now draw a sketch on the top plane. So, let me again select a rectangle. And I say okay. yes, blue lines mean it is now ready, it is selected. One small feature I would just like to put here, but this is just a trivial demonstration of Solidworks. There is a, we can design, from the simple to the complex components using this.

Since it is told already that Solidworks has a capacity to build complex components like a carburetor like a full machine tool, different components or different ways of designing are there. We can pick components from the library; we can design something. Right now, I have picked a circle here, I have just drawn a circle. On this circle, I will draw a spline perpendicular to the plane of the circle.

So, while just clicking or snapping the points here, a spline is being drawn. So, we can select further, extruded boss, swept command or so, to sweep this. So, I can select the profile that is to be swept out. So, it will now ask along which dimension do you wish to sweep. So, it, I select that line. So, this swept one has helped me to bring up this model.

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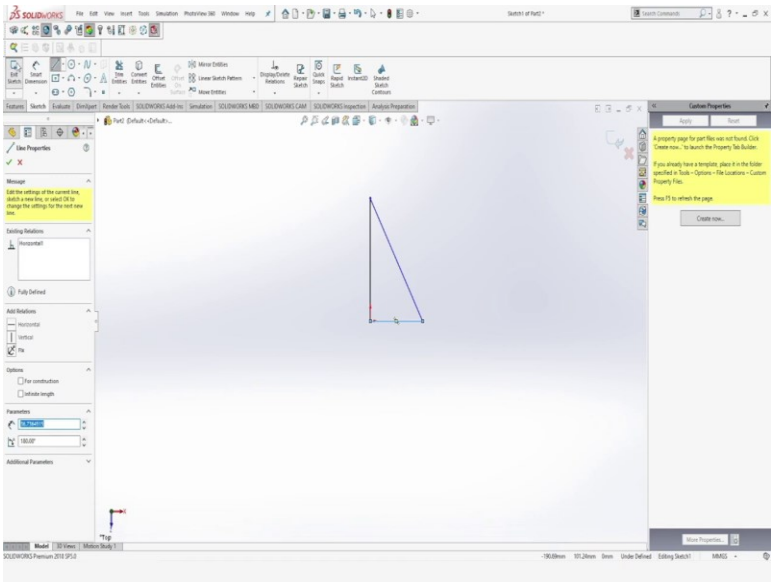

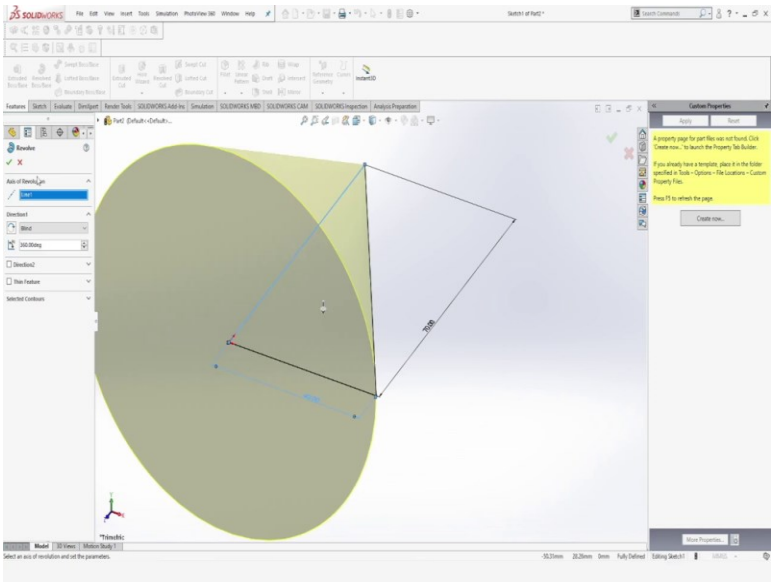

So, also, revolve command is also there, another example. So, in a 30 minute lecture, we cannot explain the complete software features. So, just to give the basic information, I am just trying to make, a triangle is made. This triangle could be rotated along its

perpendicular axis, along any of the sides of the triangle, it could be rotated. So, let me rotate it.

In the features, I will say revolve command, removal boss, along this axis. So, it has converted a triangle into a cone while revolving it. So, similarly we can keep playing, we can keep adding our dimensions, dividing the overall assembly or the part into different components or different areas, we can design, and we can keep on building up the features. We can build holes, threads.

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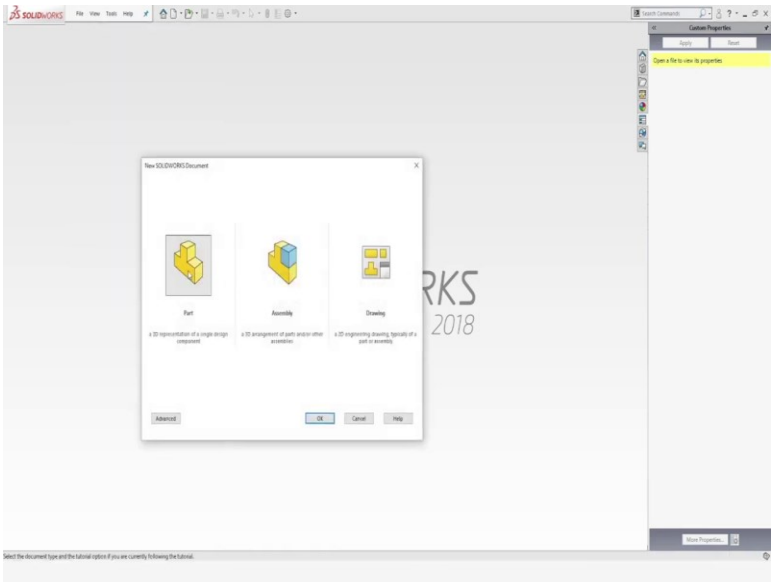

Now, to make a CAD model that I will use for additive manufacturing or for topology optimization, to put the mesh over it. So, I will just select new part and try to make a small body that could be developed or that could have different arms at different angles, for example 45^0, $< 45^0$, $>45^0$, so that I show you that how supports are required and what topology optimization helps us to reduce the material.

So, we can select different features here, certain features like this circuit works, photo view, within Fusion 360, scan through 3D, Solidworks motion, so, we can give motions to gears, motion to the cam and the follower, these motions could be given when we try to build an assembly here.

So, Solidworks simulation for static structure, for the movement, all the simulation could be carried out here. Then, Solid works toolbox is here so, nuts and bolts, preview for nut and bolts are there in the library, that is, the library is called as toolbox. So, those could be picked from here. So, small gear etc. could also be taken from here. Spur, bevel, or different kinds of gears. There is a Solidworks flow simulation. So, for the fluid dynamics, the computational fluidics is the flow simulation used.

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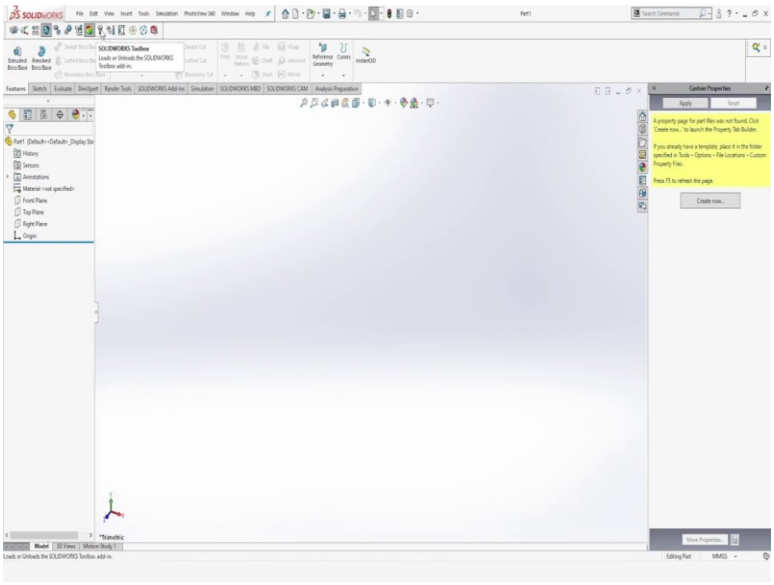

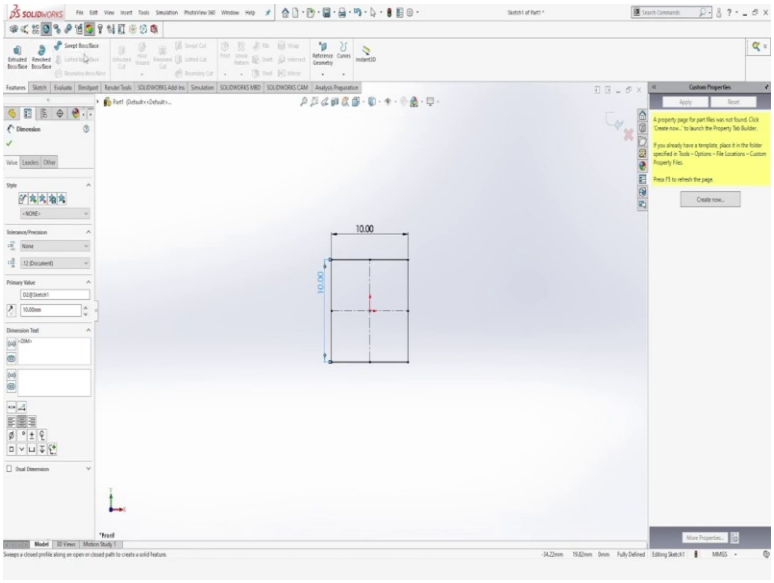

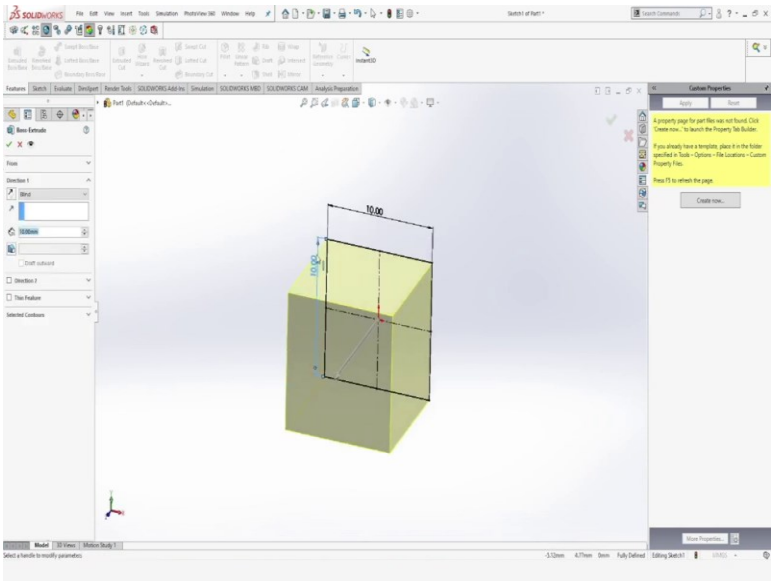

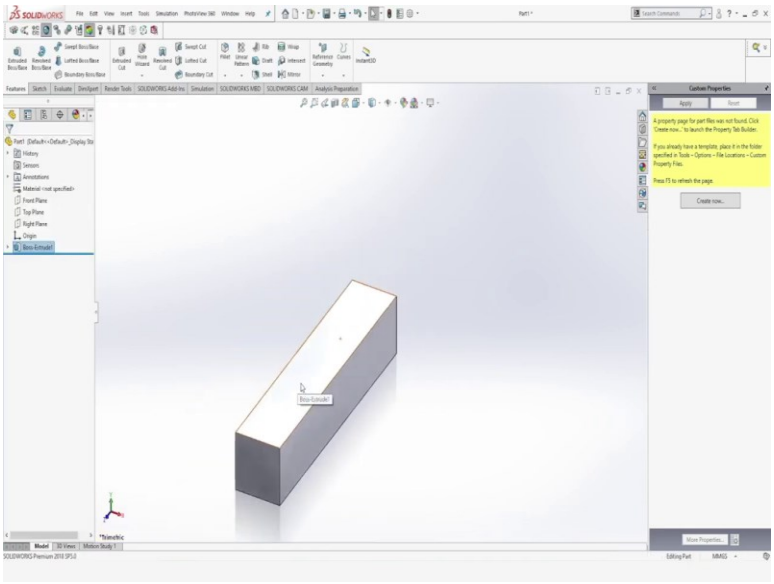

Now, let me select a plane, that is, the front plane. So, sometimes, we start from the material as well, but here, I will just take a design. Material means for example some simulation is to be taken, then the material could be taken that would take the material as SS or MS, whatever we select. Based upon that, the strength of the material will be then fixed. And then, done based upon the parameters of the specific material.

I have now made a square. So, again extruded boss, revolve boss can help me, swept boss, this can help me to create or sweep or revolve the object. Then extrude cut, then whole wizard, revolve cut, sweep cut, these all, negative substances, negative volumes could also be produced.

So, to increase the strength of the base or the small edges where the edges are sharp, fillet could be applied, then rib could be given there. So, then mirror could also be there to have the copies of the component. So, this is all for taking the material off, that is producing a negative volume there if that material is not required. The intersect quantity is also there, if the material is intersecting there, so, we can also take off the material from there.

So, I will just extrude this and give the length of extrusion as 50 millimeter. So, now we have got a rectangular bar here. So, different arms at different angles only I am going to add here so that we try to see the CAD design in additive manufacturing. So, with using and without using the support.

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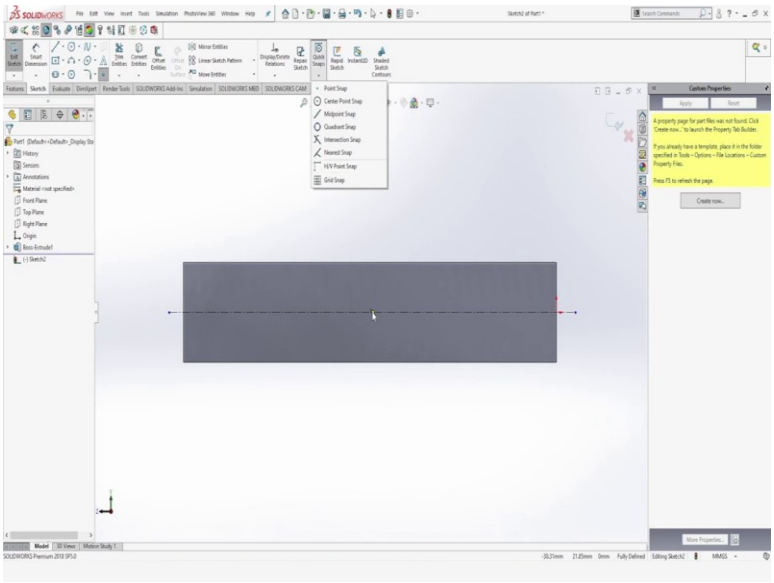

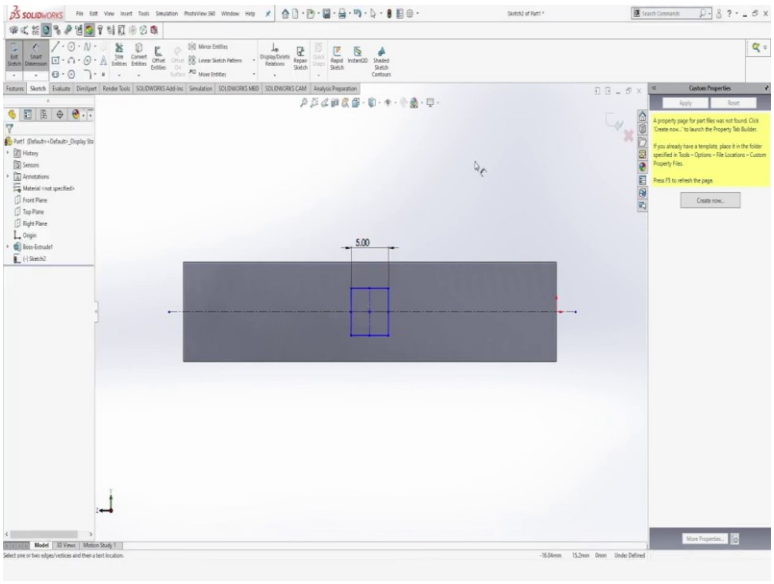
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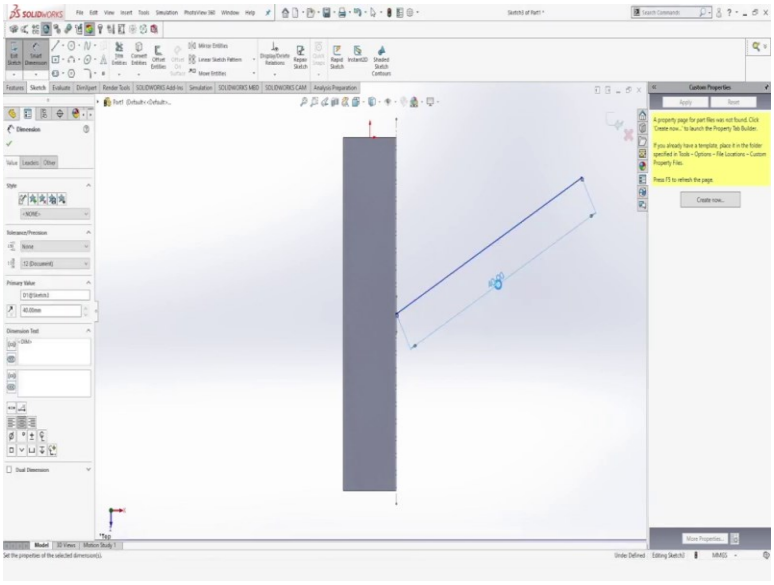
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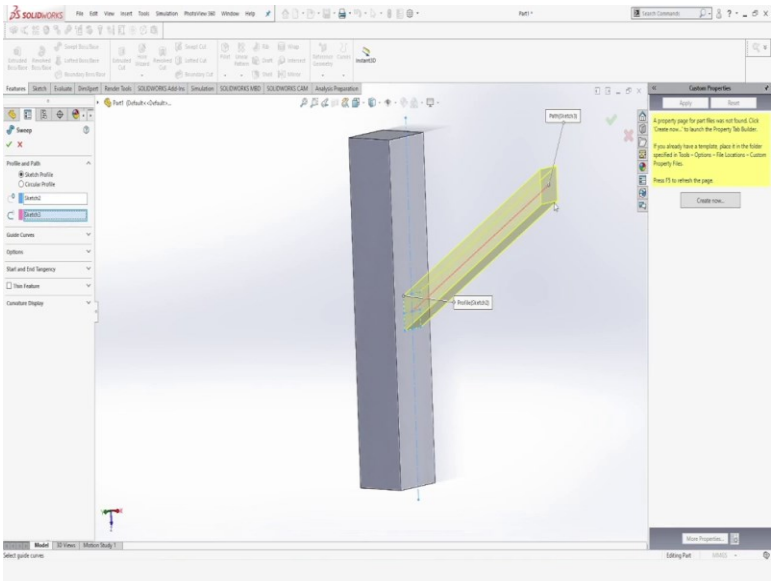

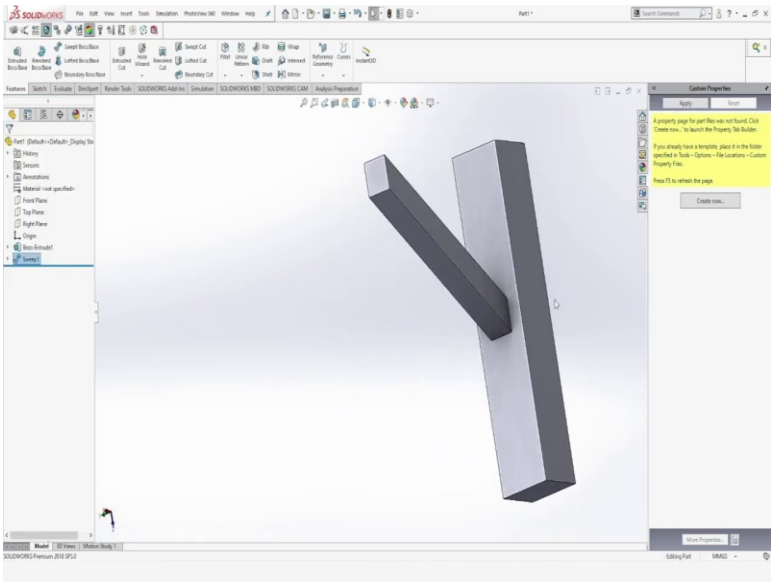

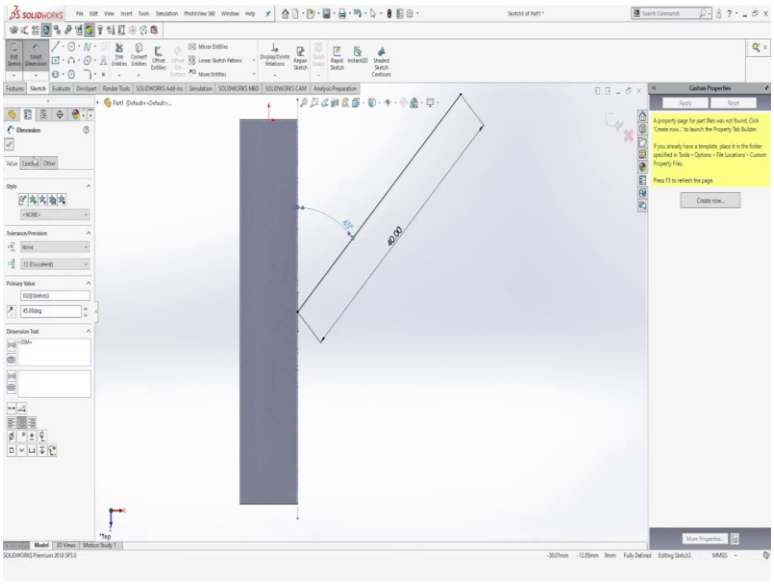

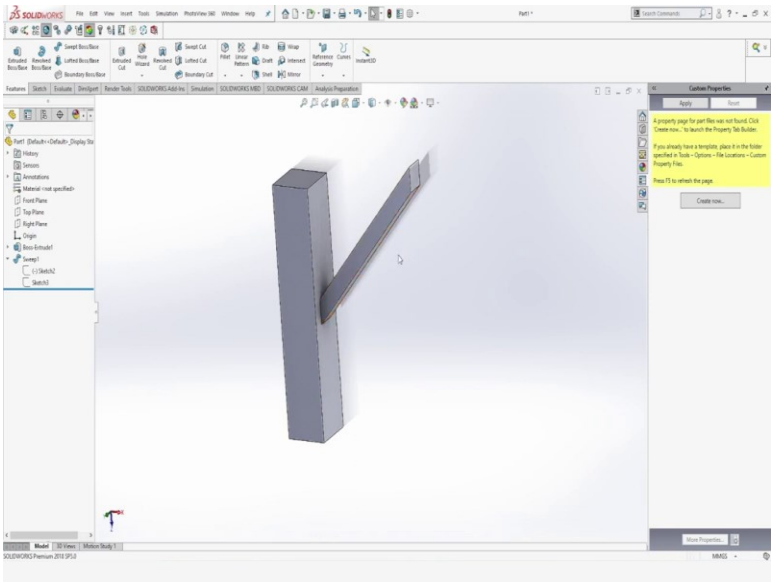

So, I can just select a surface or sketch. I just selected the surface, then I selected a sketch. I just drew an axis line here. So, the different corners are set so that this axis comes to the midpoint. So, then we have snaps here. There are multiple ways to put snap here, that what the points are to be snapped, point snap, center snap, midpoint snap, quadrant snap, intersection snap, near a snap, grid snap.

So, for our convenience, we can use this snap points to quickly go to the point or the dimension or the edge or corner. So, now we have taken a point. We select the point here and try to collect the mid here. Now, I will take a rectangle at this point, it is the mid

point of the surface from the both the sides, and give the dimensions here, 5 mm. Yeah. I enter it and get out of it.

So, in a different plane, I will again put another sketch here. So, in the top plane, this is to be put now. So, I am panning it a little, so that I have a clear view of the other side as well. So, normal to this line. Normal to this point, I have made a line. So, I just give some dimension here, maybe 40 mm, a fixed length. I select okay.

Now, this is just a line from the center of the square that was drawn. To make an arm, so I will just make a profile here using the sweep command once again. So, this is one arm now. So, to the main body, this becomes one arm, that is a cantilever arm. So, similarly, we will have cantilever arms at 45^0, $<45^0$, $>45^0$.

So, I will keep it normal to the plane again. I will then try to specify the angle for this so, I will select this sketch, this sketch 3 of 40-millimeter length, the angle has to be specified, this angle. Right now, it was 60^0, 61^0 or so, I made it 45^0 exactly. Now, this is an arm at 45^0 to this plane, top, front plane.

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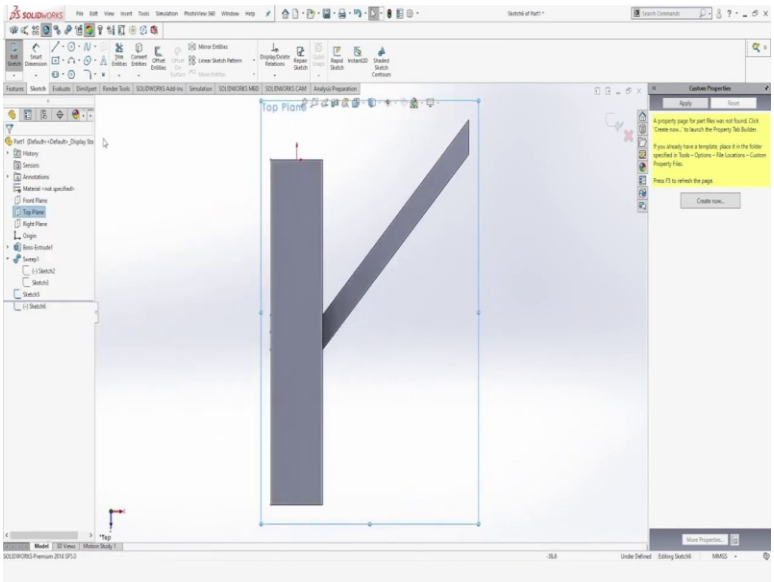
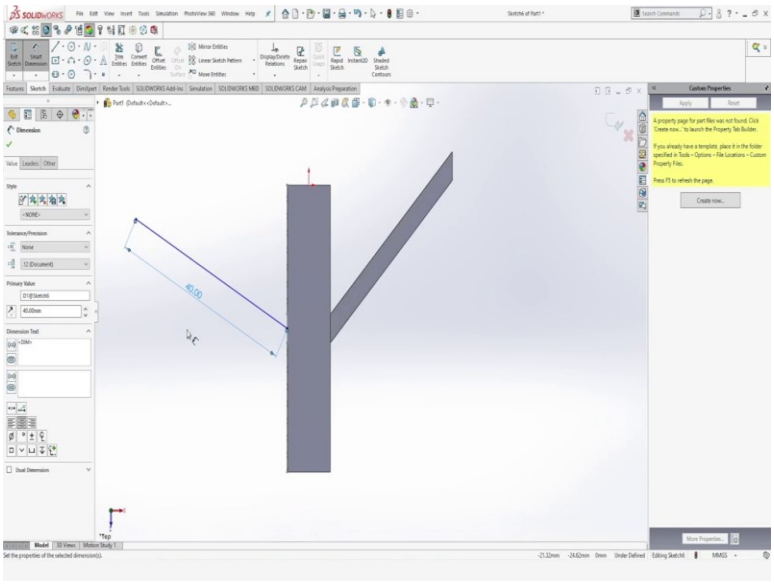
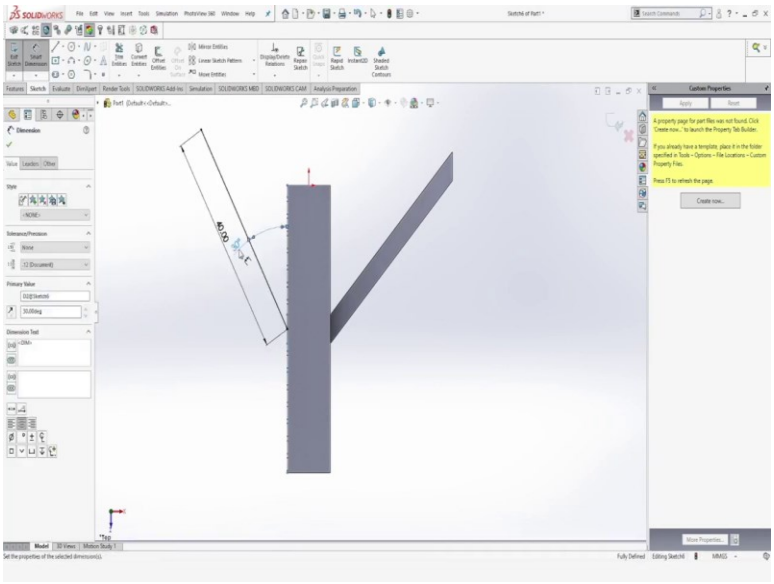
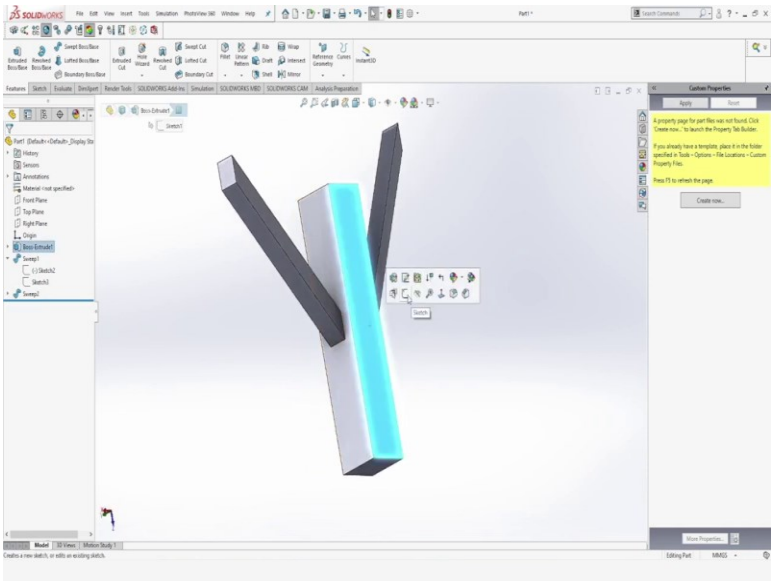



40.00
30°


40.00
30°



So, now to another plane, exactly opposite to it, I will again make another sketch. I will just again draw a center line. Then, I will put a rectangle over it, give the dimensions to it using smart dimension, again 5 mm. And on the other side, again 5 mm. So, it becomes a square for this, like the previous commands.

I will use another plane. Again, I will select a line and specify the length of this, 40 mm. The angle must be specified, angle of this line to the line adjacent to it. So, I will put it 30^0. That means angles to the horizontal base is 60^0, and angles of this left side arm to the vertical edge is 30^0, that is defined. So, for different angles, we can develop different arms just to test only.

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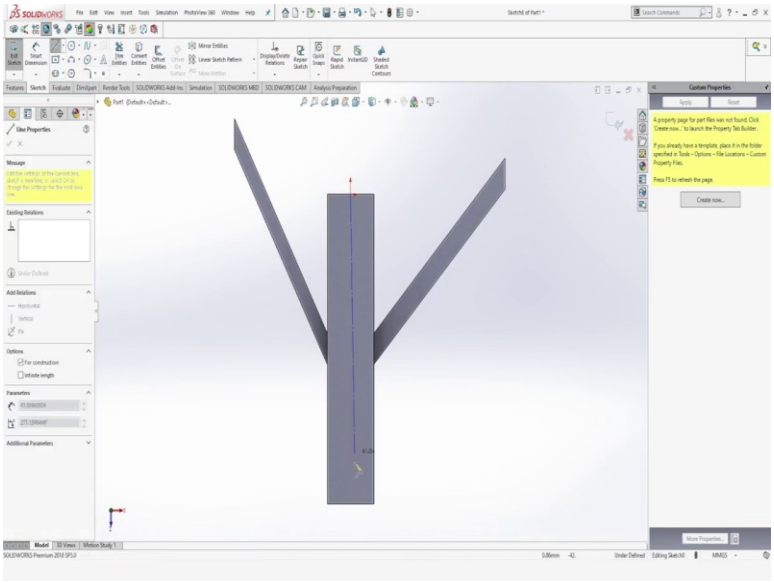

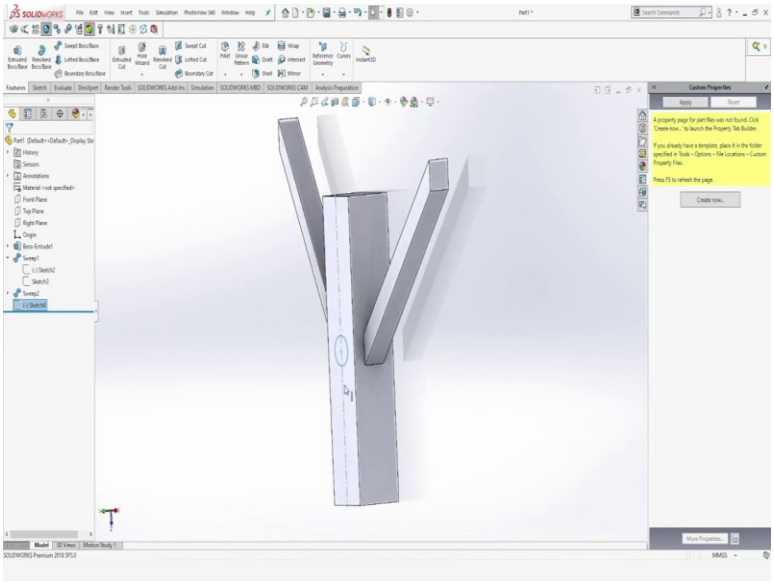

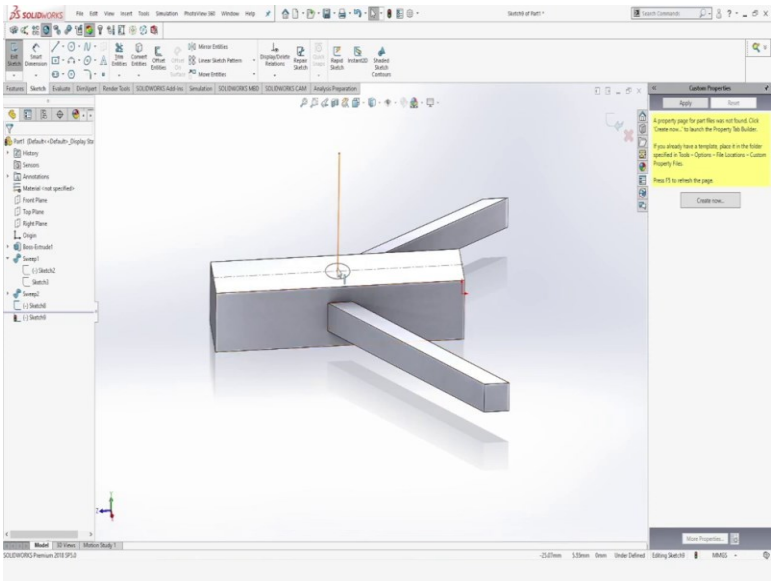

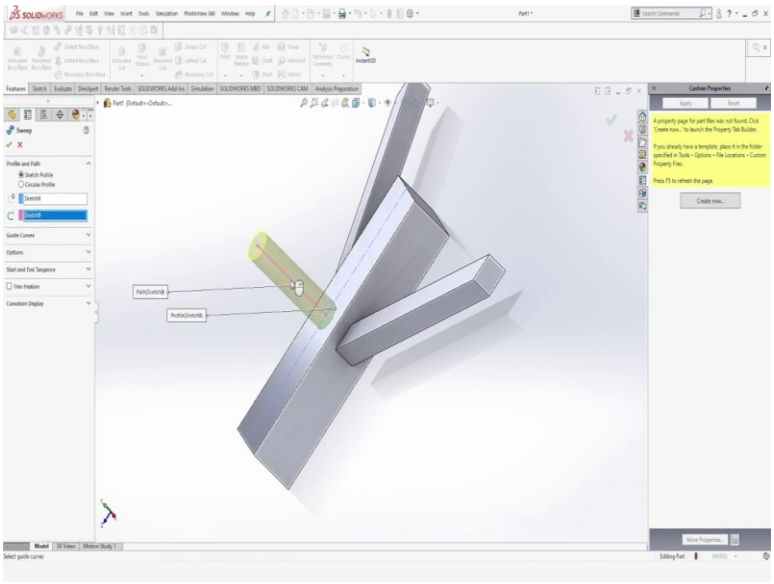

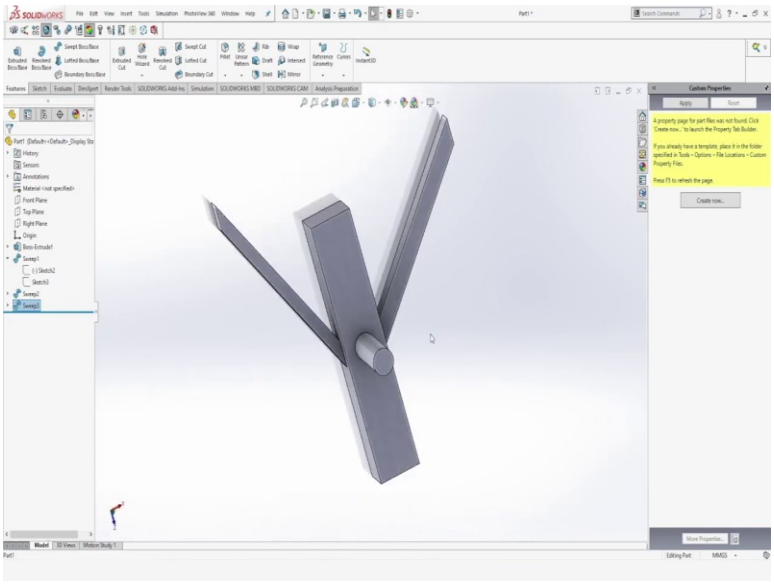

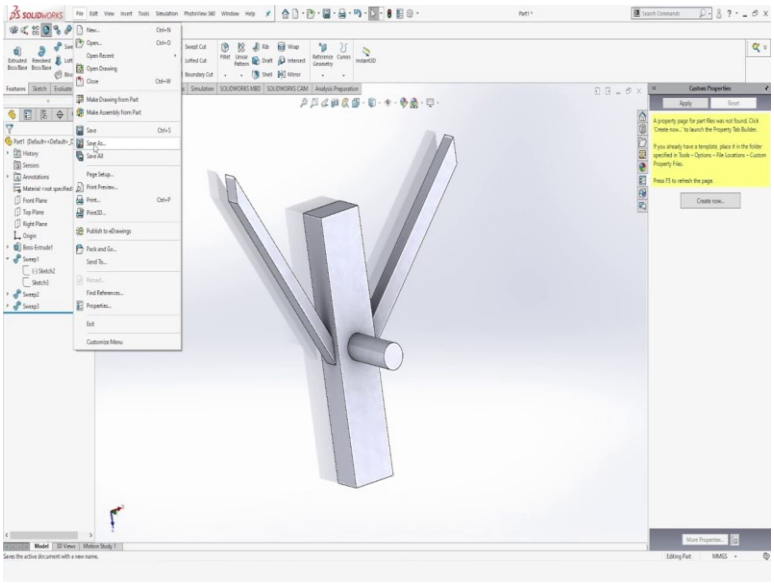

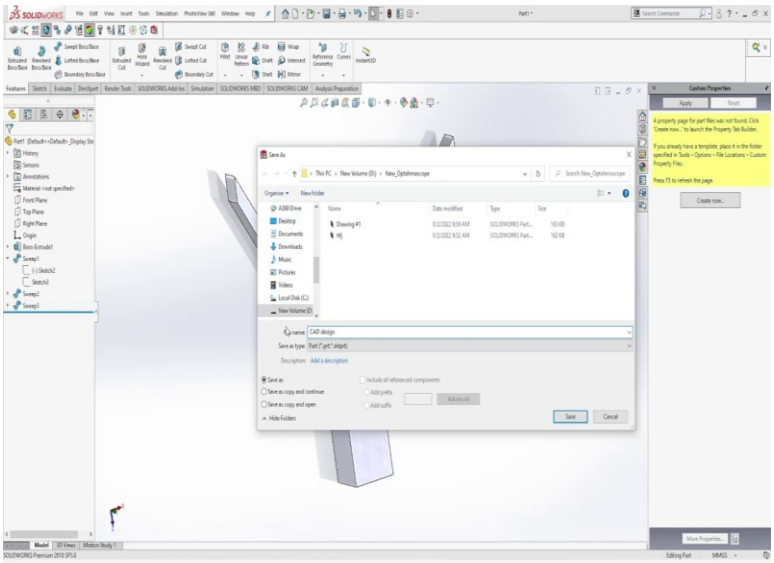

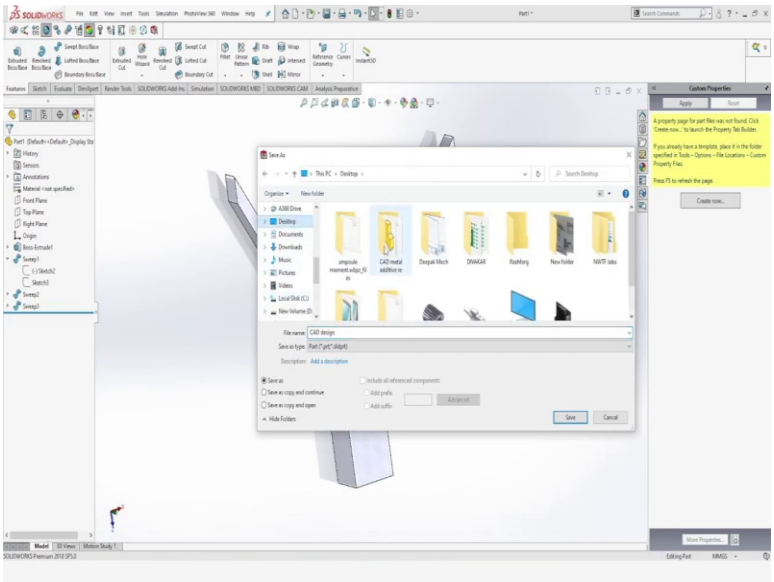

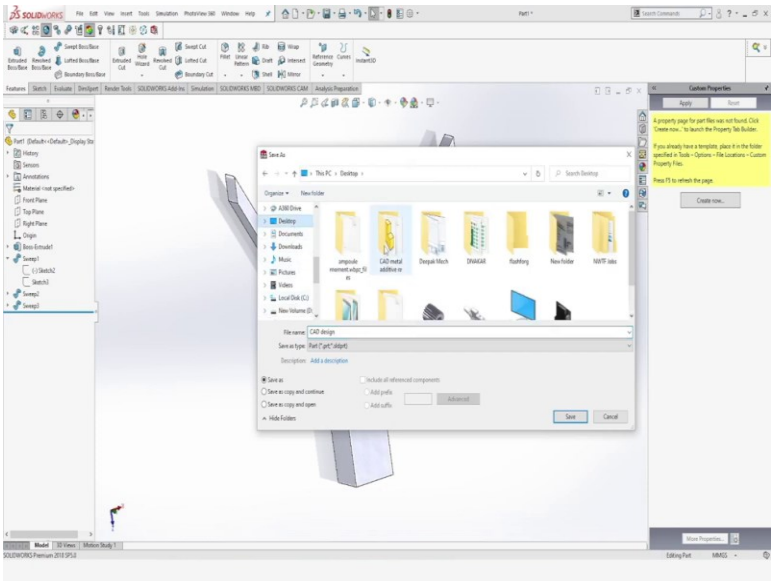

Now, if we check this, make it normal, and on this perpendicular surface, or an adjacent surface to them, another arm would be drawn, first while drawing the center line, then we can make a circular rod or rectangular load based on our choice. So, for a circular arm, we can specify the radius of the arm. That means, the radius is 2.5. So, it will make the diameter as 5 mm.

So, again, I come to the plate perpendicular to it and select this point, since the snap is already on. And I choose a line. So, this is perpendicular to the surface. So, this snap point is caught in, and this line is drawn perpendicular to the surface. So, then we sweep it. So, profile and the path are selected, and the feature is swept out. So, different planes, for different designs, we can keep on adding or updating the features that we have.

So, now we can put the component into our data directory of the Ansys. So, for that, we must save the component. So, it could be saved in multiple ways. Solidworks provides an option to save the components in different formats. I put this name as CAD design and select the folder. Then you can say sld prt or prt file. So, they are different ways.

So, IGES is the majorly used, STEP is very used, STL is for the additive manufacturing. So, then for Creo, for pro-E, prt format could be used. The e drawing is also used for having sketches over it, for additive manufacturing .amf files are also sometimes required. 3mf is also one of the additive formats that could be used.

So, right now it is saved in sld prt, but we will better save it in the step, that is, the STEP format, that could be used directly in the Ansys software. So, also, it is saved in IGES format wherein no face sets could be increased. So, it can also be used in different softwares. So, in CATIA, in Inventor, and in other softwares.

So, these different softwares could be used according to our requirements. So, I will just amend this component a little and then take it to the Ansys simulation software, where we will try to have the topology optimization. How to reduce the surface, what is the stress concentration, wherever the stress concentration is minimum or zero, no material is required, and the material is removed from there.

Then, when the material is removed, it completely becomes an abstract object, we must clean it. We must make it little rectilinear so that the 3D printing could happen with as far as possible smoothness. So, with this, I will meet you in the second part of the CAD and topology optimization laboratory demonstration, where we will see how we conduct the topology optimization. Thank you.