Computer Integrated Manufacturing
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Lecture 42
Laboratory Demonstration
CAD using Fusion 360, an Introduction

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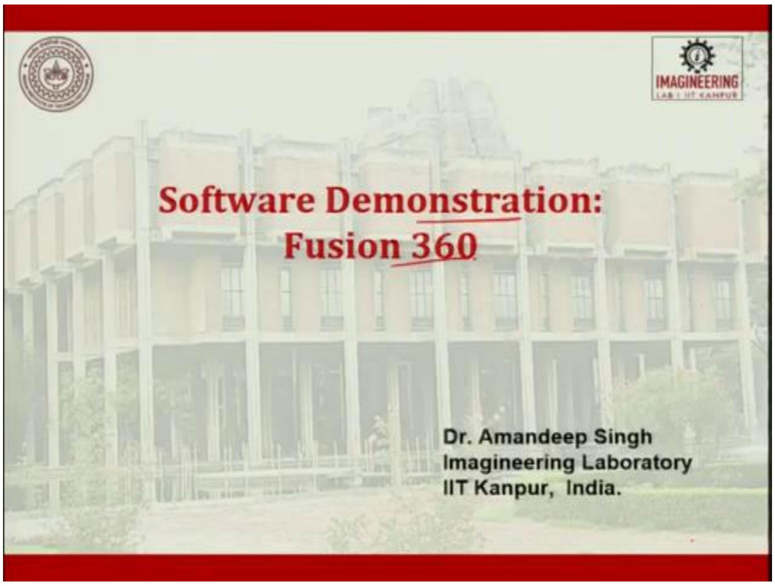

Good morning. Welcome back to the course Computer Integrated Manufacturing. We are talking about rapid manufacturing in this week, and on the request of the candidates, and the participants, and the learners, I have come up with this software demonstration on Fusion 360. We have discussed SolidWorks in the previous weeks when we discussed about the CAD modeling and CNC programming.

It was asked for some freeware software or the cloud based software I will just come up with the differences between Proprietary, and the freeware software or the Proprietary and the cloud based software here and Fusion 360 is a very common platform that is being used it was launched a few years back and after that, it had been one of the very commonly used software by the students, by the learners, hobbyist, by the small startups and so.

Because it has the cloud based structure in it, in which we do not need a great system, we do not need a high end computer to work with because only the computer with basic requirements can just get the system installed, Fusion 360, interface installed in it and all the testing, all the analysis, all the calculation, all the drawings are made on the cloud. So this is a software demonstration on Fusion 360. I will come back with a difference between SolidWorks and Fusion 360 from first.

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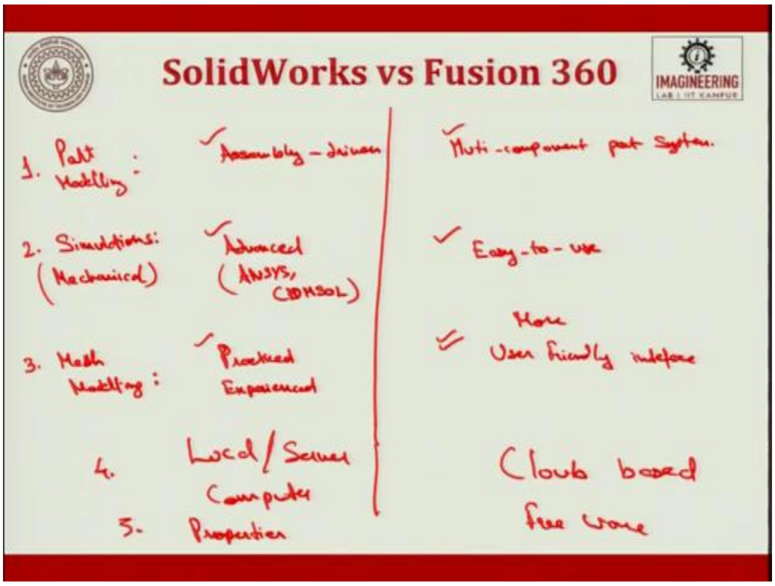

What are the differences? Okay, I will divide it into 2-3 categories. First is part modeling, category one. In part modeling, SolidWorks is actually assembly driven software, and Fusion 360 is more known as multi-component part system. So what does this mean? When I say assembly driven software that means it build a part individually before assembling them in a separate files.

So, when the parts are used in many assemblies as well as for documentation purposes SolidWorks is more used. It does not mean that SolidWorks does not have the multi-component part system it do have but provide the amount of the flexibility, amount of the features those are there in Fusion 360 are tremendous in comparison to what SolidWorks has, but in an assembly driven software, it means that if we change the dimensions of one component in the whole assembly, whole assembly would be changed.

If the dimensions of the part is changed wherever this part is used, that assembly would also reflect the change, but in multi-component part system the components can be individually separated, the change in one component might or might not affect the overall assembly this is the basic difference. Fusion 360 is a multi-component part system in which the components of an assembly are build, and assembled in the same file.

This actually makes easier for the designers to refer other components, and to build them within an assembly. So, unlike the SolidWorks, Fusion 360 does not need to refer multiple files when building an assembly, although larger assemblies in Fusion 360 can cause issues those fusion 360 is a freeware, its applications are limited, always the Proprietary software the people who are charging or selling the software they provide many functions with that.

But for the immature designers, or for the startups or for the students this is a good option to choose. So smaller assemblies with many cross reference parts are easy to do in Fusion 360. Now, comes second difference I will put regarding the simulations. When I say simulations, I will talk about the mechanical simulations. Simulations in SolidWorks though it is a Proprietary software those are advanced.

SolidWorks, similar to SolidWorks, we have ANSYS and COMSOL which are even more advanced than this right, but these are quick and easy to use simulations that can be carried in the Fusion 360. Fusion 360 offers many easy to use analysis packages to understand how a part will respond to the external characteristics or other external environments. This simulation package include the static stress analysis, thermal analysis, shape optimization or so. So, this is basic and easy to use.

In SolidWorks, the simulations are advanced one, they offer simulation packages with varying sets of features depending upon the purchase license, depending upon the full version or the limited version whatever is purchased by the user. The SolidWorks their simulation package lets you evaluate the linear and non-linear responses as well. Dynamic loading, composite materials, linear and non-linear dynamics, and so on.

And next important factor that difference between them is the mesh modeling. Mesh modeling is used by the designer with the points along the surface can be pushed or pulled to create an organic geometry or surfaces. The Fusion 360 excels in this area because the mesh modeling can be quick here, and this is cloud based. So it all happens when the cloud, not a very heavy evaluations are done on the computer.

So it can directly import a project into a small feature where user can instantly start mesh modeling with user friendly interface. I would better put here, user friendly interface, or more user friendly interface in comparison to the SolidWorks. So designers then generally tend to prefer Fusion 360 based on its ability to easily create smooth geometry, consumer products some practice, or experienced learning is required here.

These are the basic differences between them. First thing is this is cloud based, and this works on your local, or your server computer. This is completely freeware, this is Proprietary, point number 4, and point number 5. Now let me come to the Fusion 360 platform.

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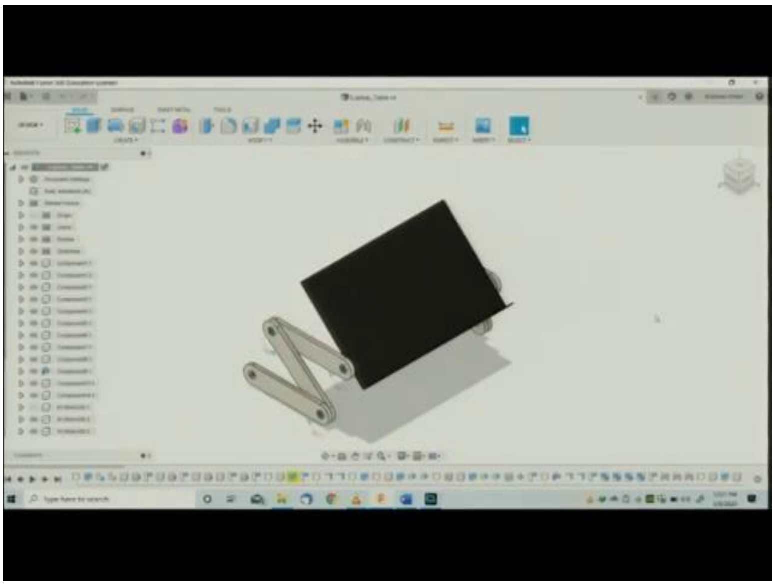

So this is Fusion 360 software provided by Autodesk. So this is a 3D modeling software and assembly software. So we have an education license of the software here, this is for part modeling this is a multidiscipline software. We can use it for 3D modeling, simulation, realistic rendering, and to make small animations.

So this software is a part driven software, not an assembly driven. It is very popular among the young startups, hobbyists, students, and education institution as well are using this because it has easy to use interface. And it is free licenses there I just mentioned before you can see the education license here.

So this is a free license here taken by a student in IIT Kanpur. This certain advantages I will just recall, it provide cloud support for various applications, here we can see the basic user interface of Fusion 360.

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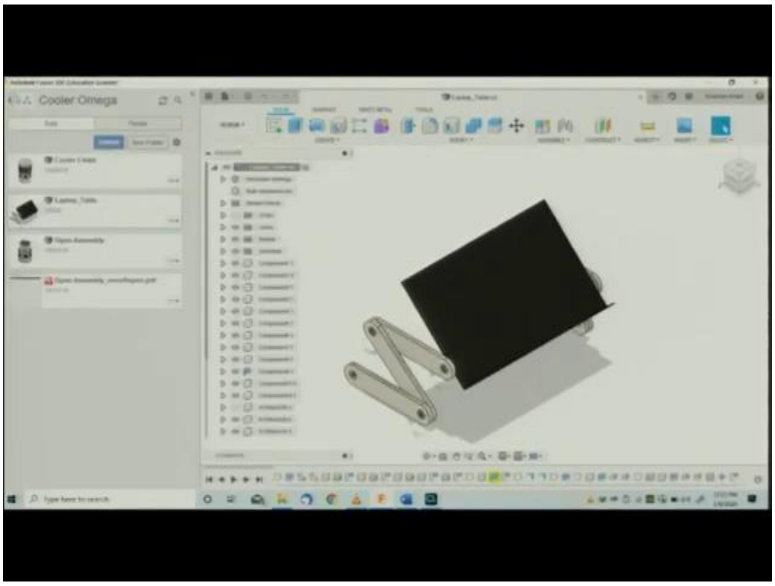

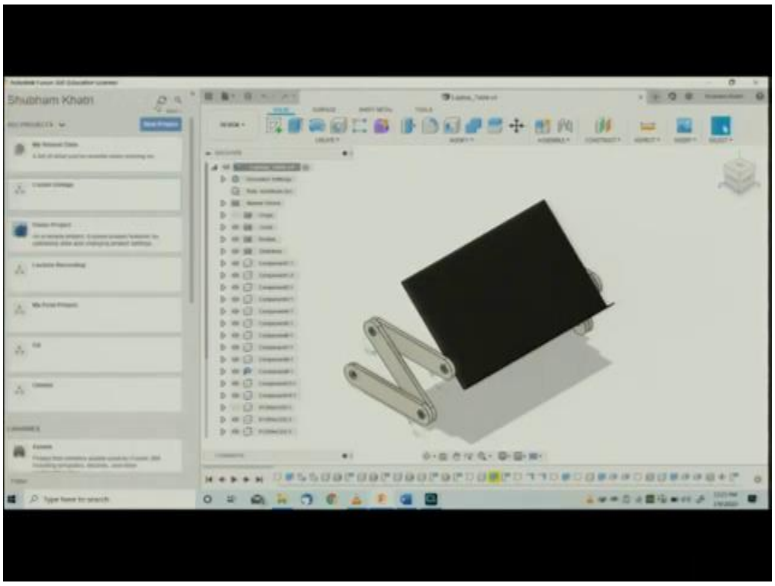

On the left of the corner, we have the data panel, we have this tab where we can access all the previous files on the different portion we can also, you can see the files by clicking on them. All the files are synchronized here with a 360 cloud, it is synchronized in real time. So, you can access the 3D model at any place.

For instance, you create a model here at IIT Kanpur you can access this model without even taking along at some other place may be in different countries in US, you can install it anywhere because everything is saved on the cloud. So you can access it anywhere.

You can edit here and design in your computer, in your laptop the same software with your login can be accessed anywhere. So, it will just be saved in your Autodesk account you just have to login with your ID, and access that because this synchronizes our design in the cloud system.

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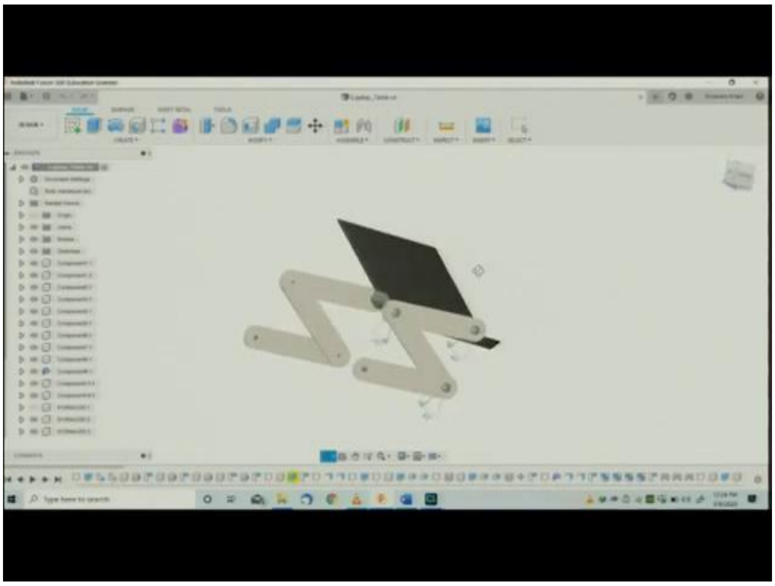

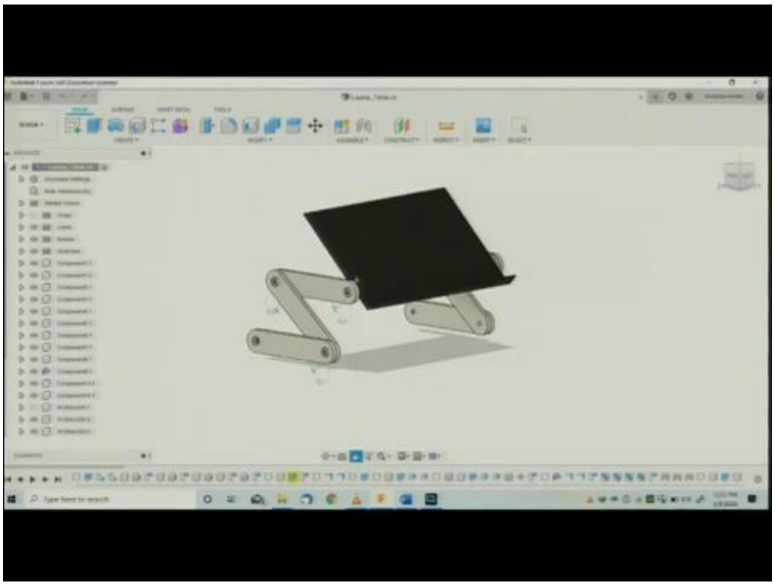

Here what you can see is, it is a small laptop stand affordable laptop stand majorly made out of the sheet metal components. So we can view, we can rotate and pan it here, so in your default browser, you can just I am panning it so rotating orient then you can also zoom in, zoom out.

So, many students and educators can take an advantage of this feature. So you can see, you can orient the design here. You can make a team where different users can use the same software or the same components and they can make a team to work on it. Only the thing that you need to do here is you can just invite people, how do we do that?

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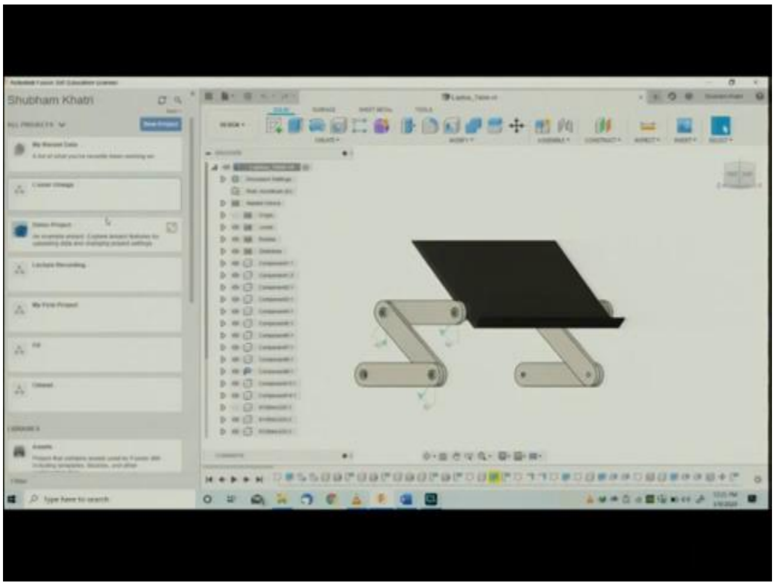

You can just come here. You can share the folders, need to invite them, we allowing people invite. We can just type the email here. So this email whomsoever you like to invite you can put that email address here and that person can be invited to access your data, you can provide the user writes to them as well.

So you can also install this in Android or IOS application your tablet or IPad or a smart phone anything can be used to use the software. This is actually the essence of using cloud based software. Cloud based software can be assessed from different platforms, from different when I say platform I am talking about different gadgets. So, thus Fusion 360 I believe is best suited for the beginners.

It is not only limited to do small things we can make professional design assembly which are actually manufacturable, and to some extent, it can also be tested. A small stress analysis, small thermal analysis can be done on this software. Again the cloud based system provides with an elite advantage, a special advantage of using the extensive Autodesk computational help that is just happening on the cloud itself.

So, you can use that, and you can just make your drawings on your computer. So even a computer with basic specifications can used to all the computer testing for the photorealistic rendering, and some basic simulations this can also be done. So when does that happen. You can use your cloud gadget to do some computation on the cloud computing, and you can use their own library.

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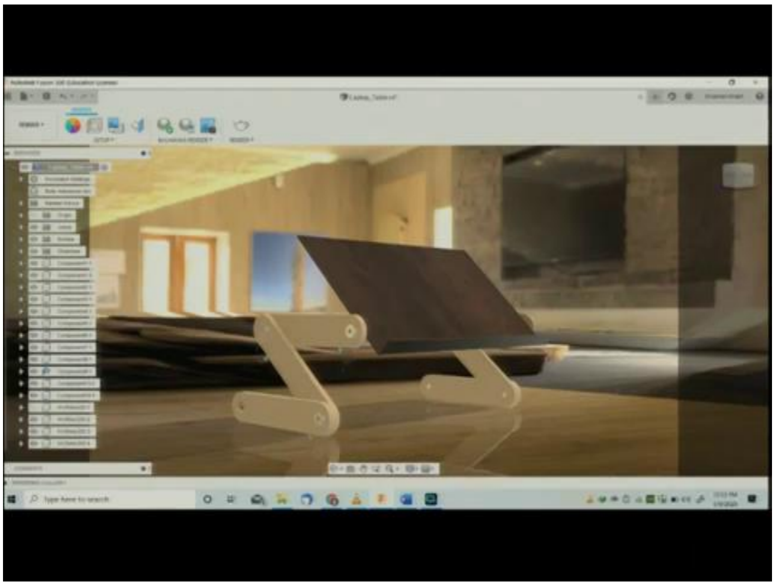

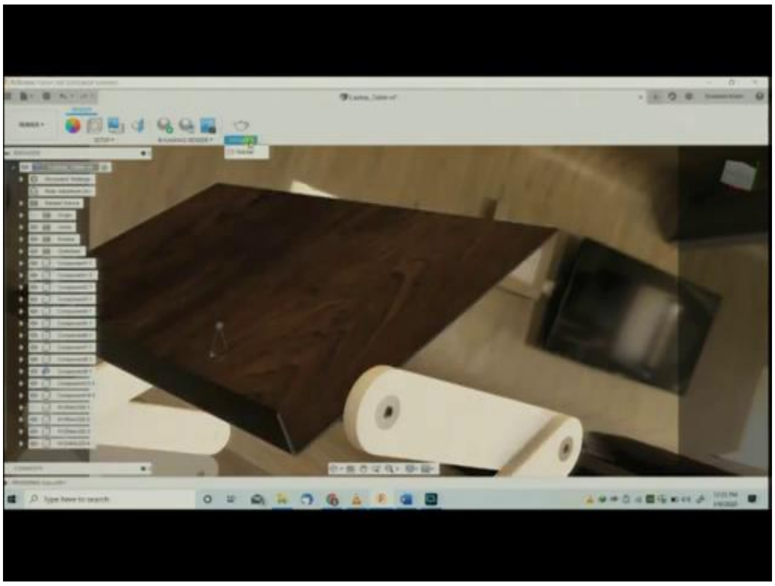

You can create your own library, and their library look like here you can see render it is a foldable laptop stand we can try to render this as well, again render. Yeah, this is the library, this is a foldable laptop stand here. This is again HDRI, photo realistic render, what is HDRI? High dynamic range image. You can have a photo realistic render here so this entire computational process is happening on the cloud.

So there will be no load on your machine or on your computer. See different orientations, you can use the software in a very smooth manner even when you are working with multiple models you can design solid geometries, sheet metal parts, and also create PCB Printed Circuit Boards. This software helps to create 3D PCB that is available here.

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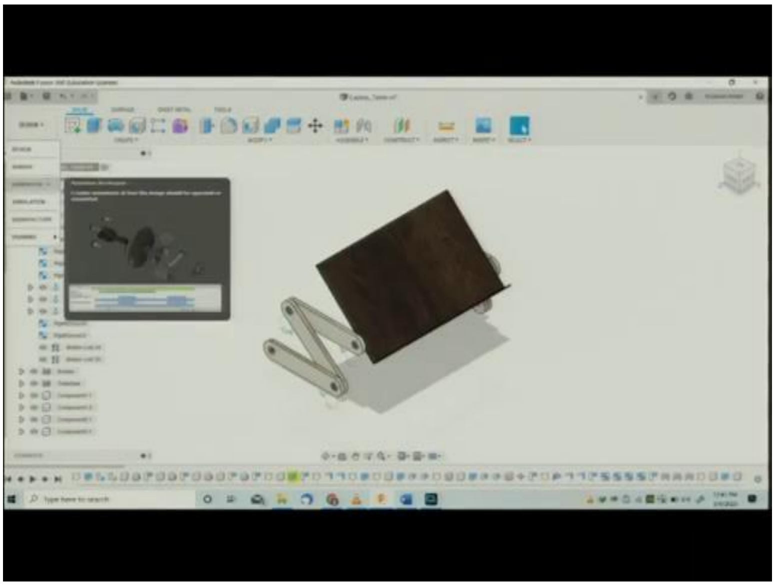

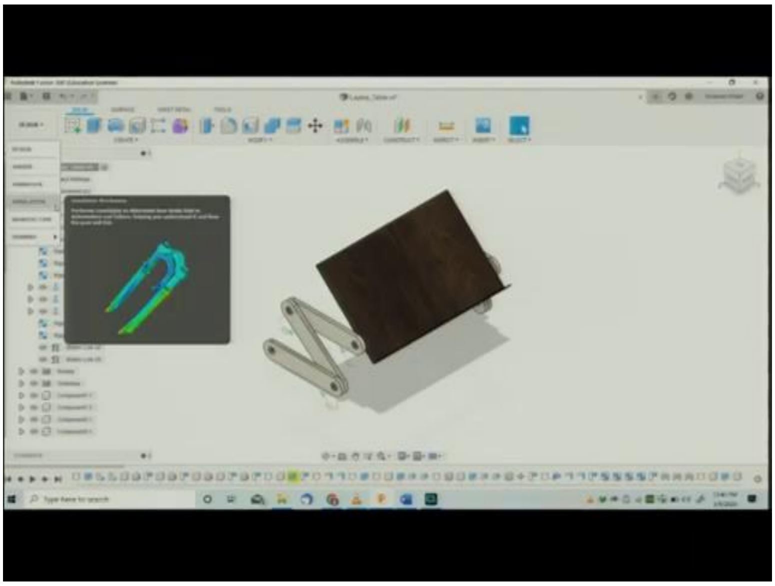

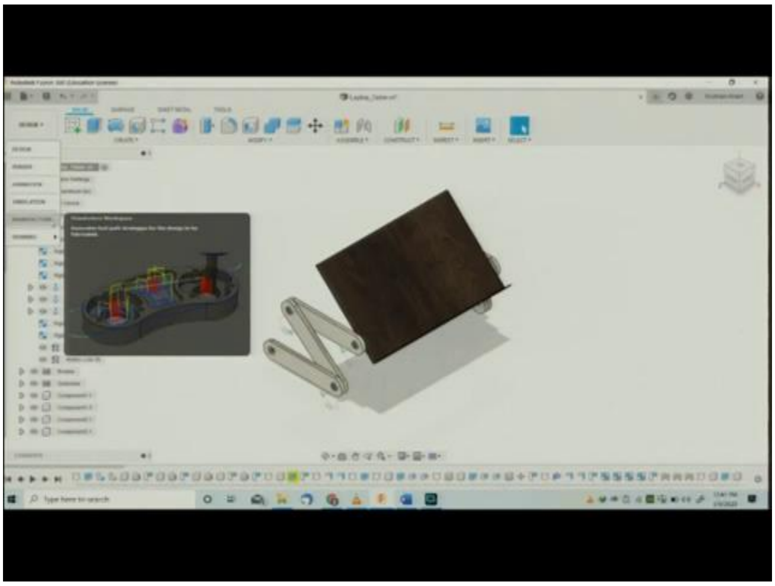

Here we have environment options. We can animate, either go for designing, rendering, animates, small simulations, basic simulations like stress and thermal, and for advanced simulation definitely, we would need some high end software like ANSYS and COMSOL and also SolidWorks can also help in that. In manufacturing environment you can see this, these are the features like to generate tool path, like we discussed in the computer manufacturing demonstration.

Like milling, turning and CNC operations those can also be done here itself in the software, but since I have introduced Fusion 360, and I have also mentioned differences between SolidWorks and Fusion 360. Let me go through the UI of Fusion 360, I will go through the various commands and features one by one. In that tab at the top, you can see we have solid surface sheet metals and tools. The tools tab is now open.

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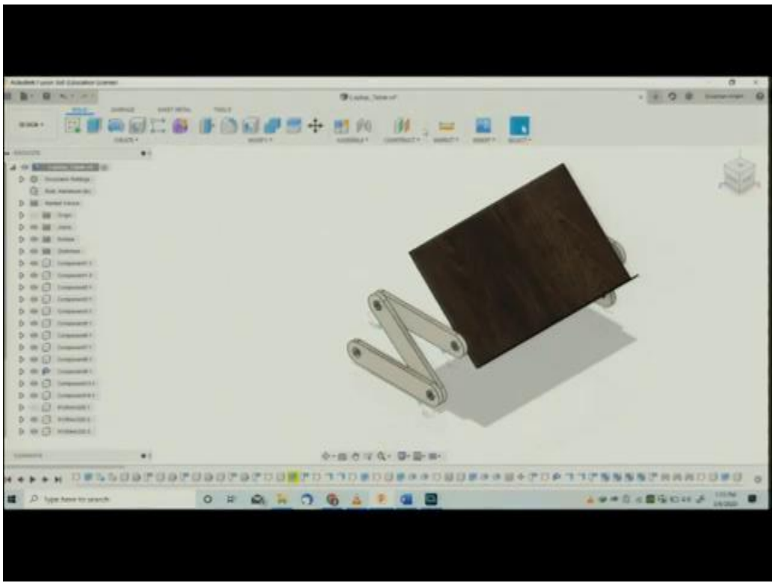

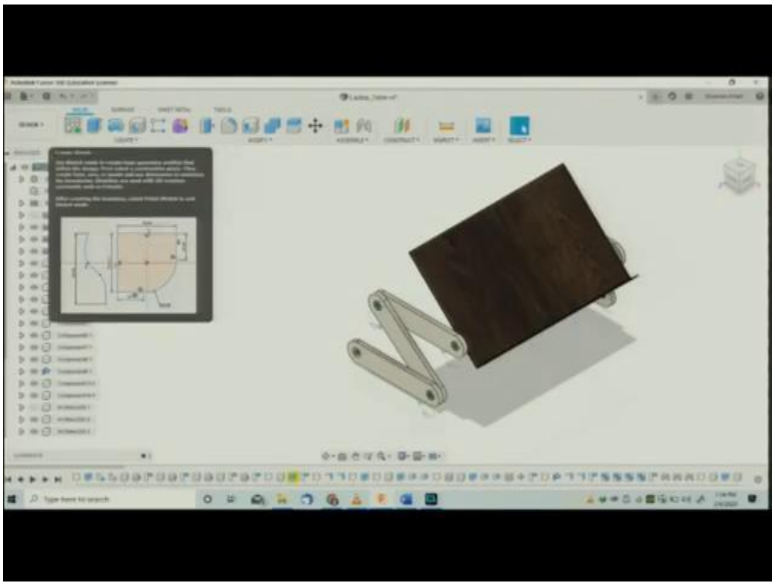

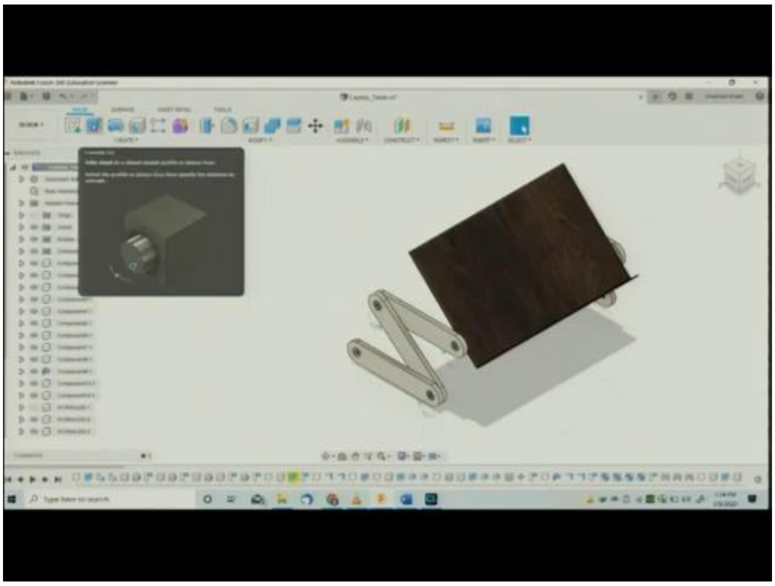

Here tool options here before that like these are the solid options. We can create sketches from there, we can create sketches like in SolidWorks we can extrude, we can create 3D models from this sketches or this is extrusion command here, this is revolve pattern, rectangular, and sheet metal command are there, therefore, more options we have certain tools here.

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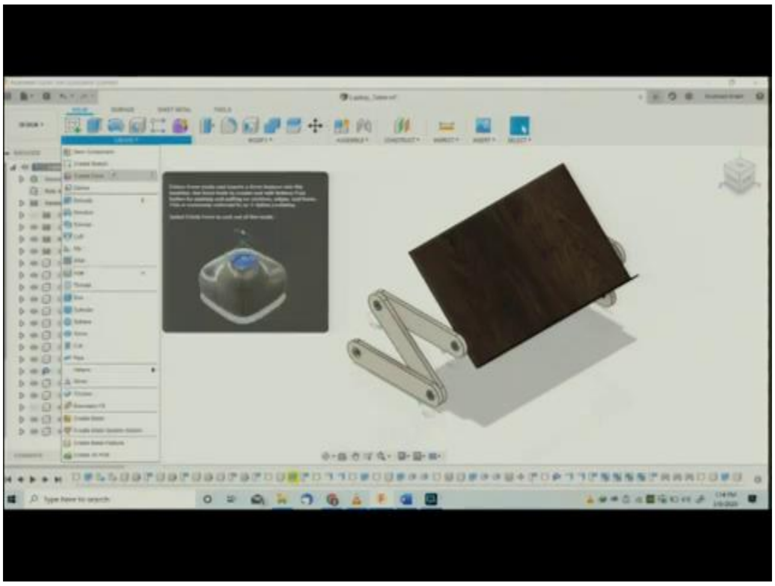

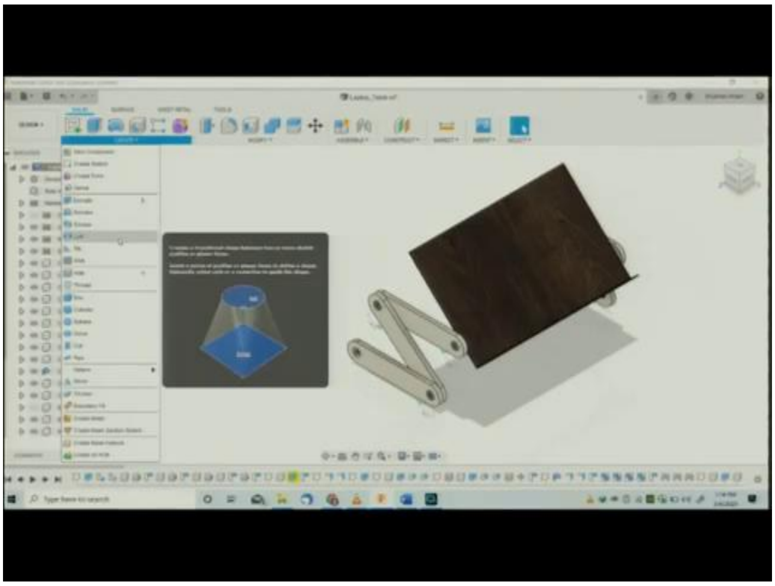

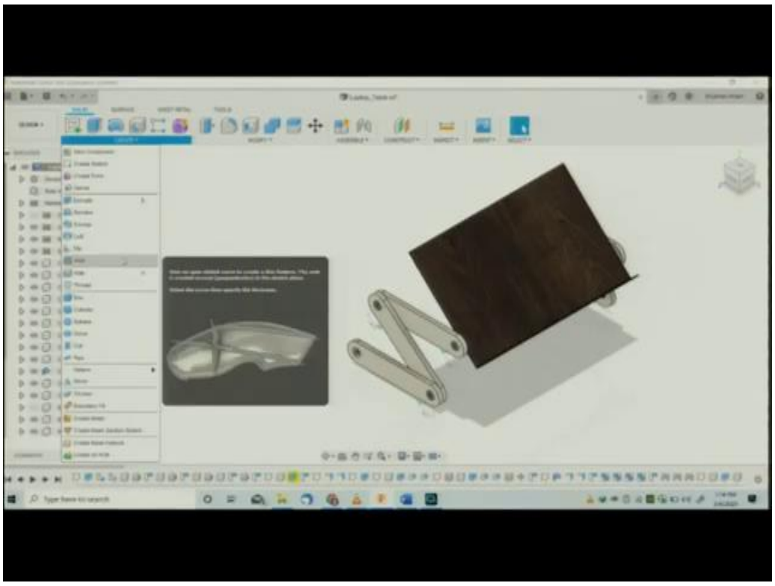

We have for more options we have modify command, and also in the top down menu here we will create we have certain operations like box, cylinders sphere, revolve. We can make any 3D modeling parts it has all the options, and the other 5 they are on the tab. So we have assembly here in assembly options here and in construction also we can offset plain we can decide to join the axis we can decide to join the point at axis.

So these options, construction options are very crucial to properly design the product like point at certain plane, and so this is inspect we can measure, and we can put dimensions, we can measure the dimensions. We can put draft and so in insert we can insert mesh like 3D or 2D file as well.

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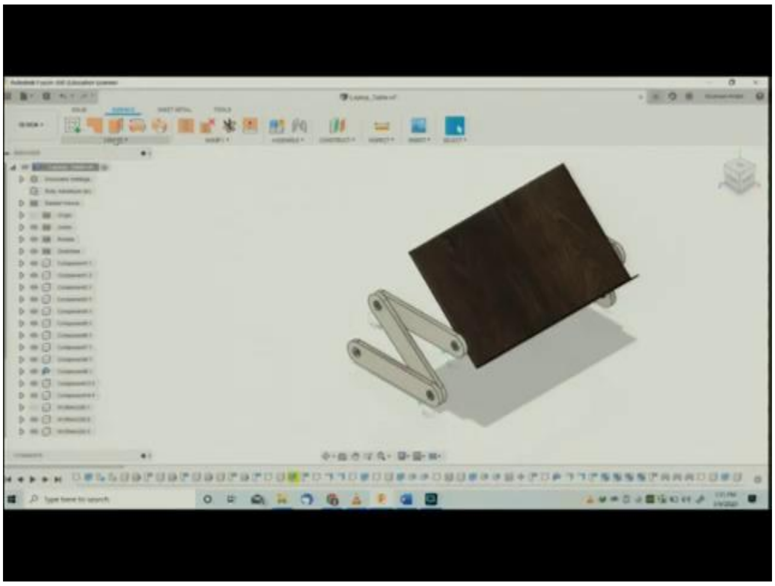

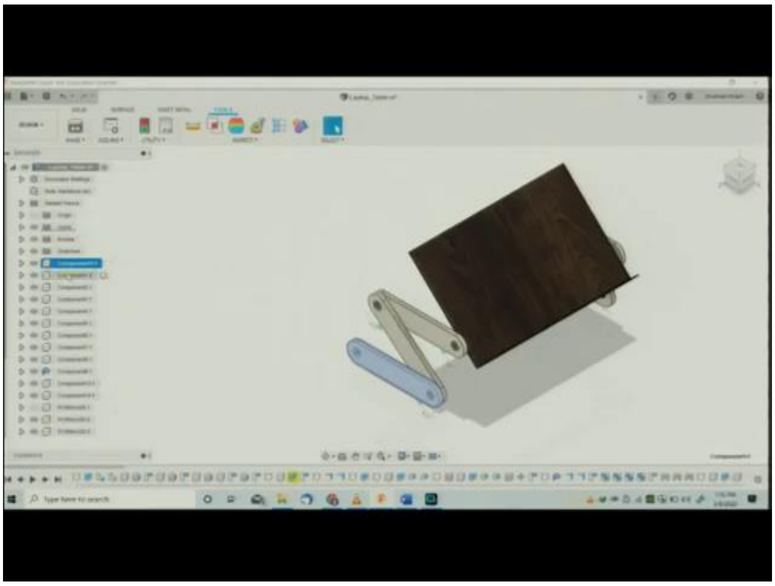

In the surface options, we can also create different kinds of surfaces, we have tools here or these are tools for creating structures. And we can modify the parts to make it solid geometry finally. The Fusion 360 also allows to perform sheet metal operations, we have dedicated library for sheet metals as well. In the sheet metal as well we can select any material and thickness, and its properties, there are couple of tools in sheet metal, we have certain tools.

So we have further tools here to make or to render so we can in add-on we can write our own scrip if you wish or we can directly script from the store provided by Autodesk. Here on the left edge in the left window, we have this navigation window, we have separate commands all the components doors we have already made those are kept here. First, we need to understand the difference between the component, and the bodies, so what is the difference?

Every component contains a single body or a multiple bodies. Whenever we change any dimension of a body of a component it would eventually reflect in our whole assembly. So all the components are divided separately into different bodies. So bodies are actually the parts of the components. So we can get solid components as well as E12 components into an assembly here.

Since Autodesk Fusion 360 is a cloud based system as I have been telling before it has several advantages over the other 3D modeling software. For example, here you can see it is a foldable laptop stand that we will make here. We can view and edit via our software on your computer and the computation is happening on the cloud.

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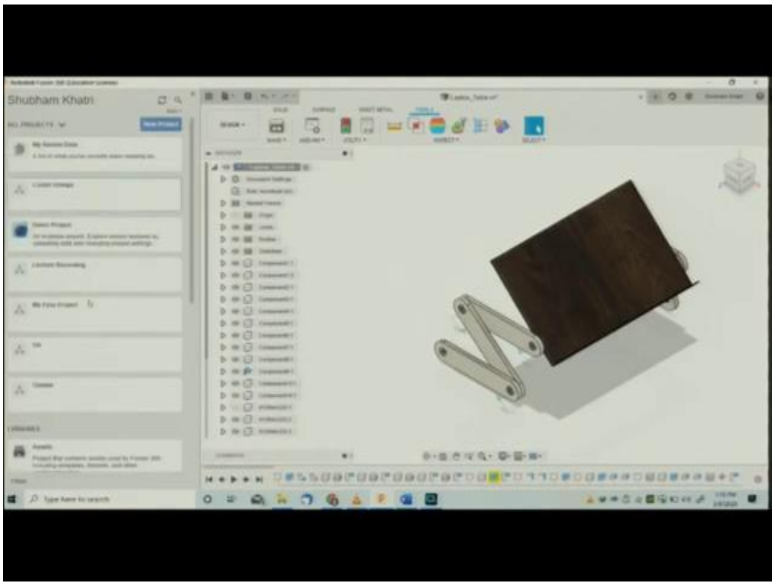

On the left top corner you can see we have access to all the libraries and products that Autodesk Fusion 360 has. Every change that we make here would be automatically uploaded on the cloud. Also it has an online portal, you need to install the Fusion 360 on your computer the online portal itself would help you to see the design that you have saved.

The similar laptop stand can be seen, can be panned, can be seen on the online portal. So there are so many interesting aspects that can be discovered using Fusion 360 a few of them we will discuss, and we will discover in the coming sessions here.

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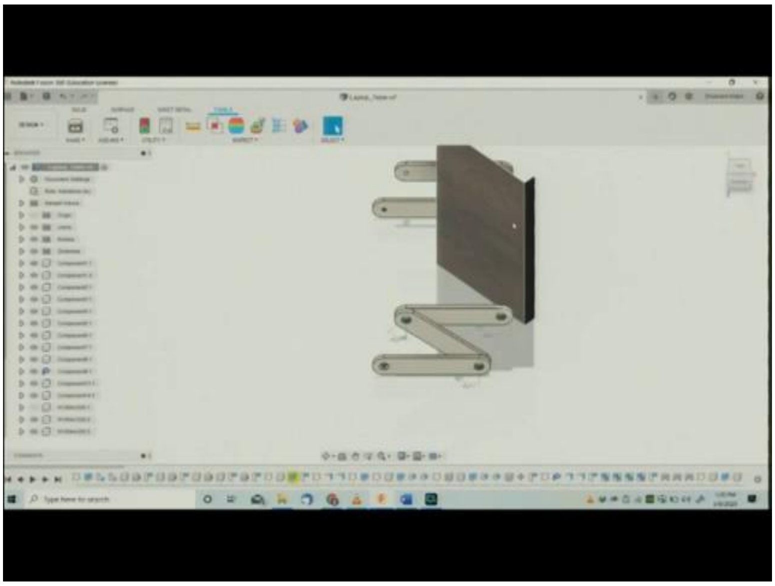

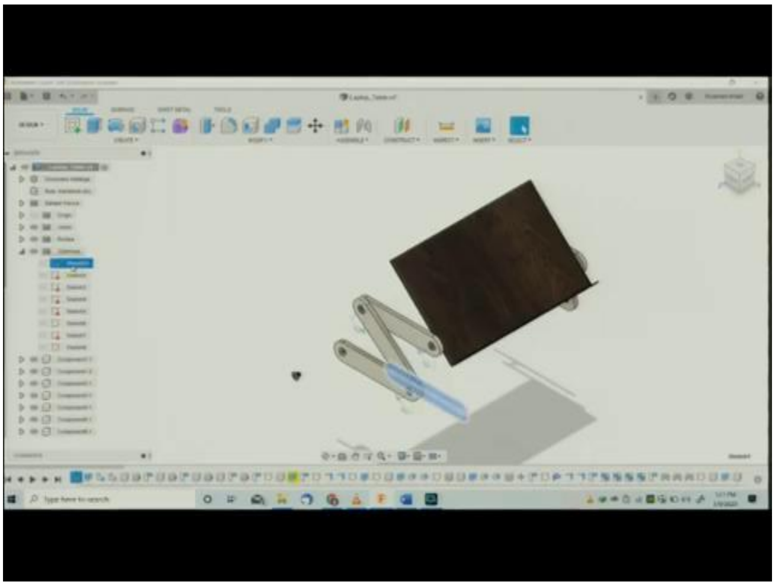

So I will just start constructing this component and before that I would like to tell you that something about the Fusion 360 environment. We can select which component we need to edit. This is the drawing already made it has an history the components are there.

We can take the advantage of the parametric modeling, and edit any of the components that we would like to change. If we make a 3D geometry of a 2D sketch and after that you would like to change the dimensions if we only change the section of the sketch. If we change the dimension of the sketch the 3D geometry would itself be changed.

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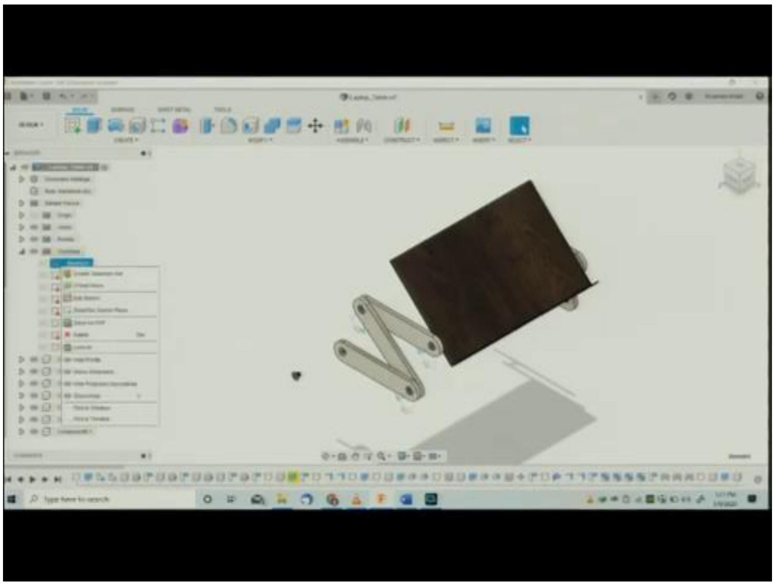

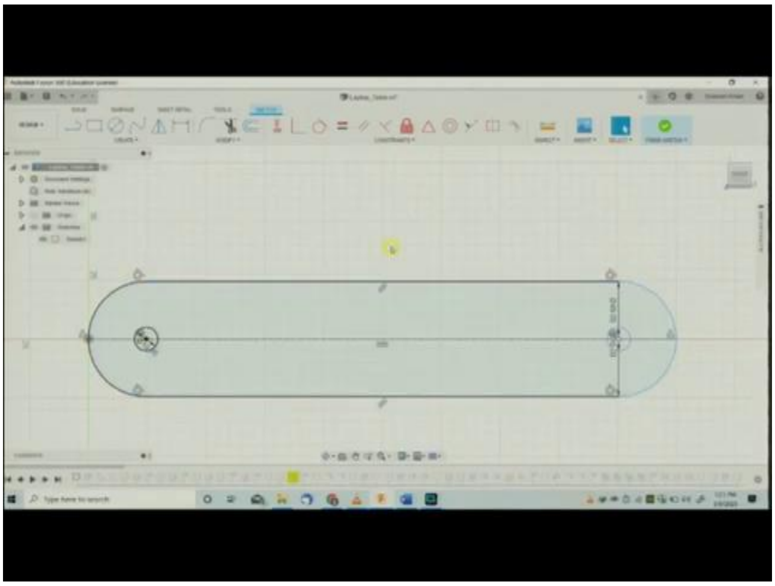

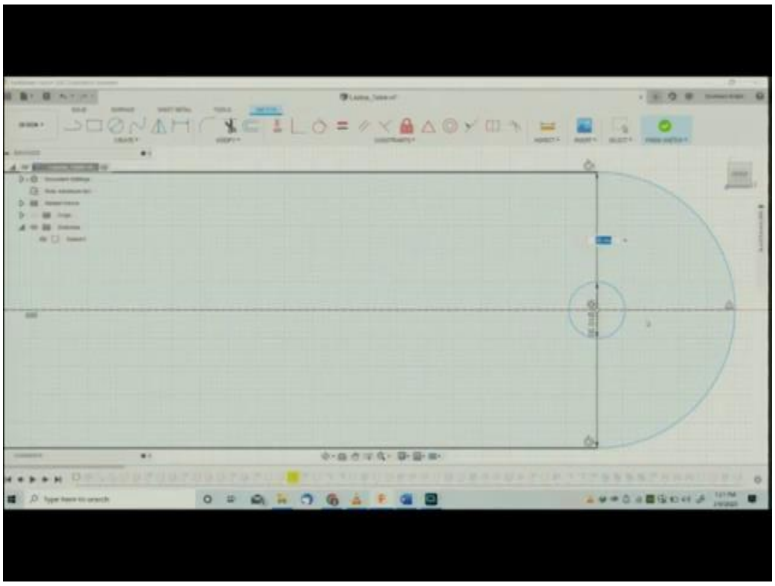

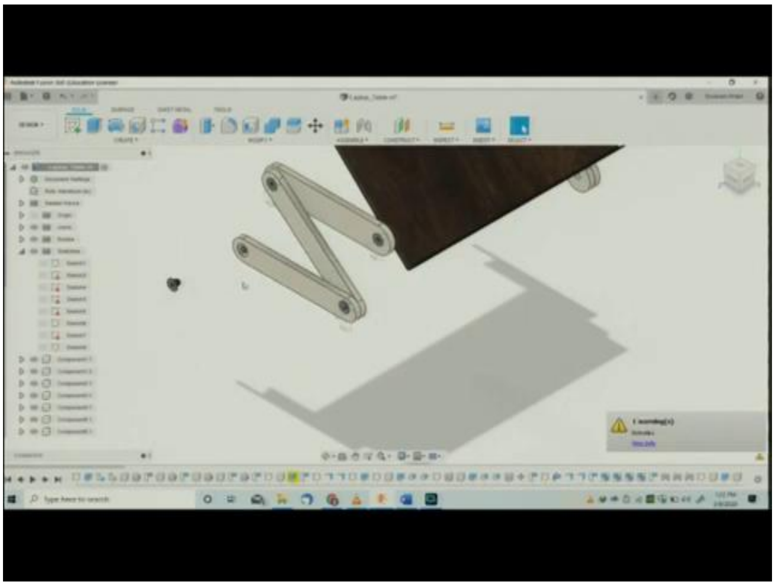

So let me try this is sketch right click on the sketch, edit sketch. So this sketch we have the re-dimensions here we are allowed to access the 2D sketch environment here directly, here already a dimensions were put as 49 mm dia, and 10 mm dia. For this component, if we make any change here for instance if I make change from 49 to 39.

See, this dimension is changed, I am saving it. It would automatically reflect in our 3D geometry, you can see.

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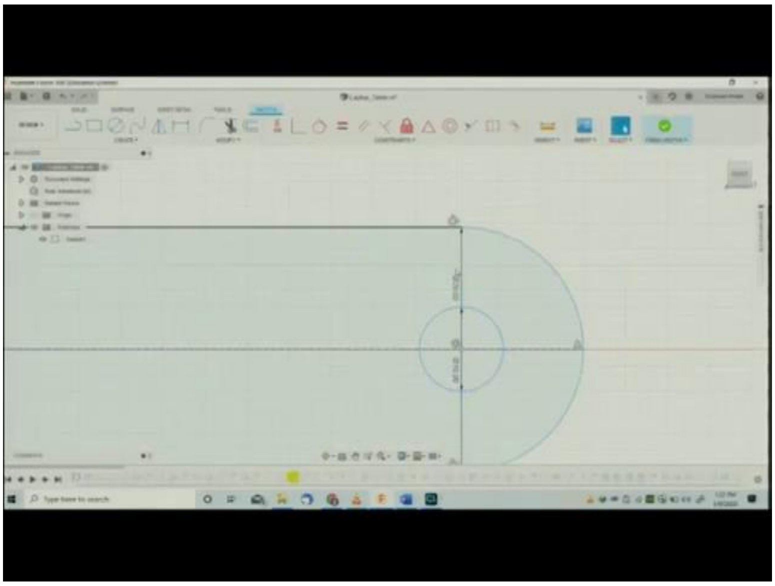

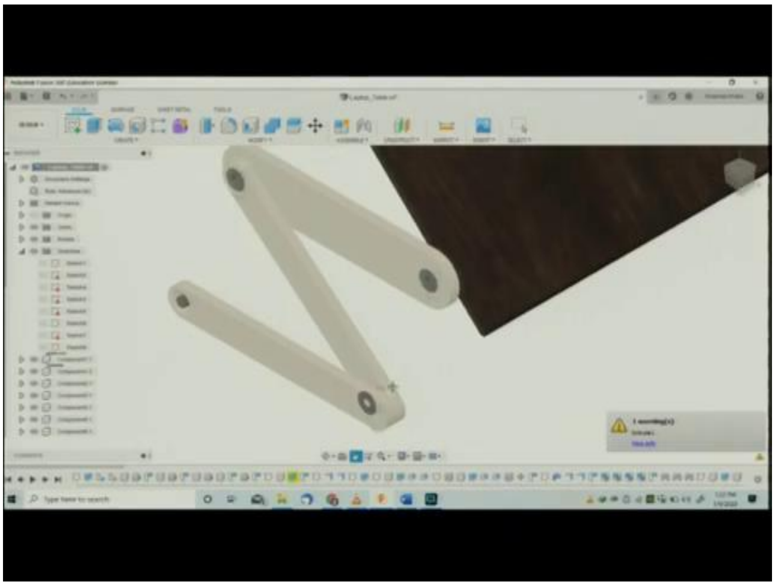

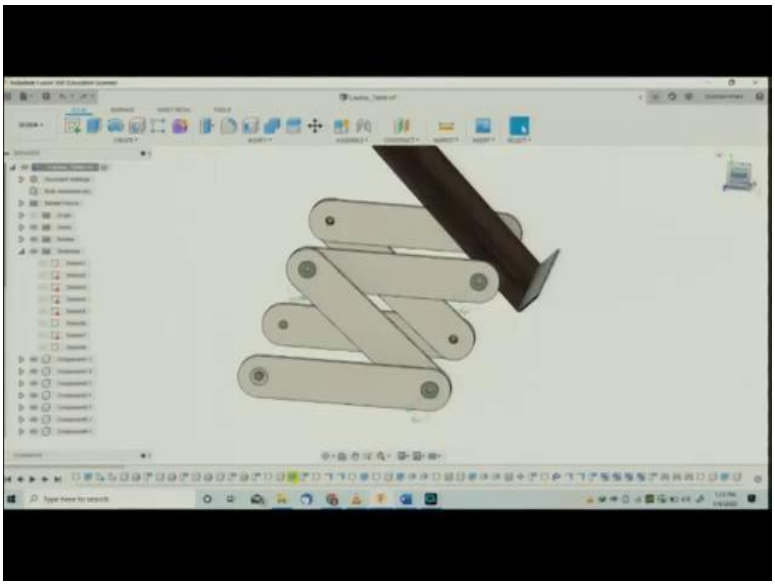

We can use standard options for planning other dimensions are 39 now. It is 29 now just to see the difference somewhere it is a little more smaller. See this is now 29 this link, and the angle is 29 mm wide. Okay, the lower one is 39, the center one is 29, the top one is 49. Now I will come back to our original component.

With the help of assembly options, we can provide joints here that I will also show you when I will make this drawing. We can also set the movements of the joints according to our requirements.

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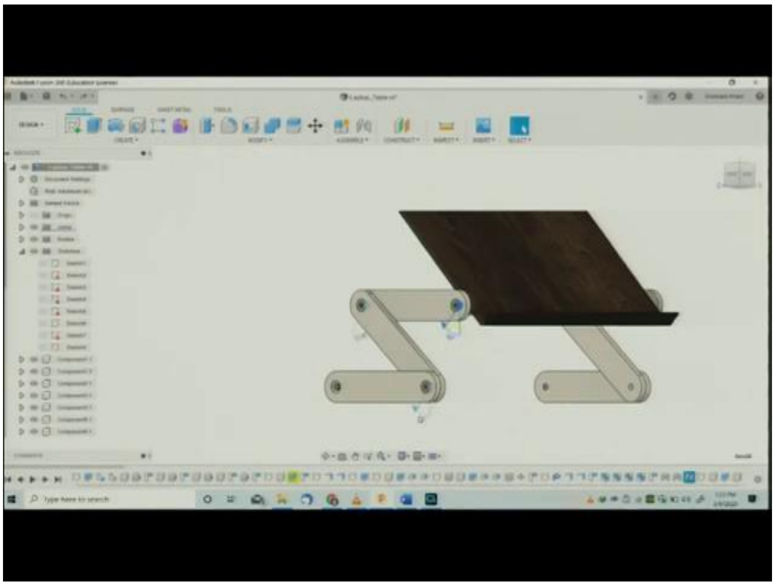

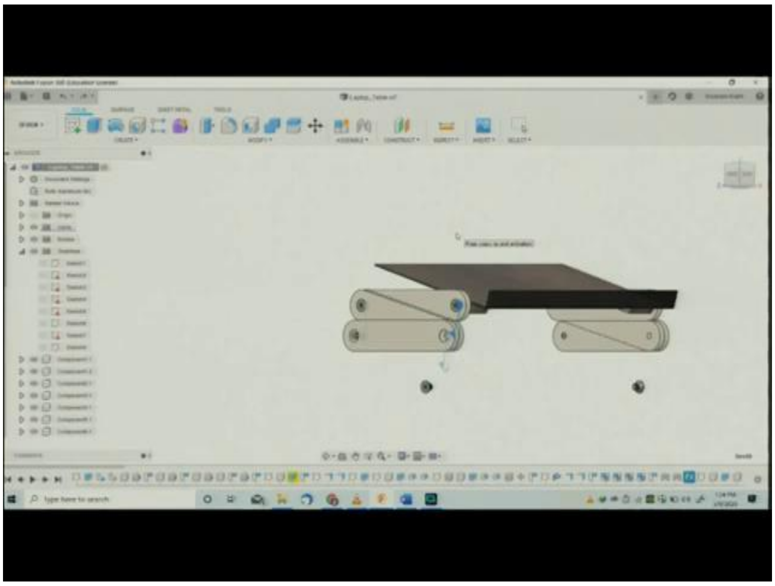

Here we have multiple relative joints over three different positions, but this is the model that we will make. We can animate your model with the help of the joints that you have made here.

We just need to write click on animated model, see the animation is being shown here. The angles are between 0 to 45 degrees. We can press escape button to escape this animation pressing escape button will stop the motion like in SolidWorks, I would again suggest to keep your left hand one of the fingers of your left hand close to the escape button bed because that is most of the times requires to keep escaping from the points.

Now, in the assembly chain in the bottom, you can see the entire history of the operations, entire history of the steps that we went through while making this. We can just click at any step and make changes we can move to that step directly.

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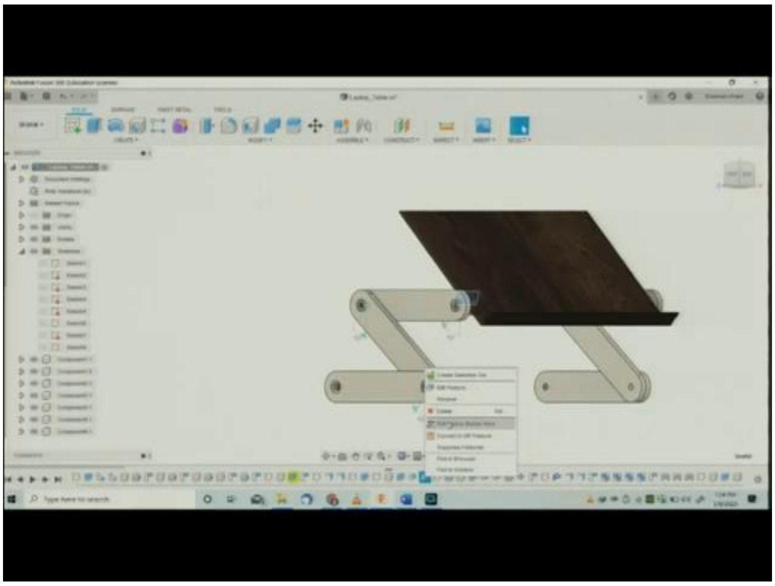

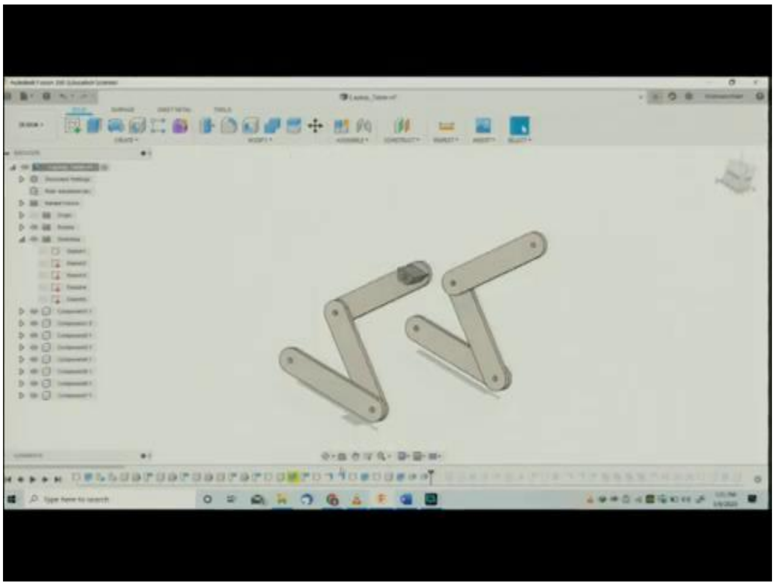

Like for instance if I click here, and that this specific step right click here and I come to role history marker. It is showing history marker at this stage. This was a position this was a status of the drawing that I was making, so we can make changes here as well. These changes whatever I make here would again happen in the final assembly.

If we wish to see the in-time process we can go through the first step in the history marker and, we can play the history button, is play, either we can play to open the whole history or we can just keep clicking on the next button.

You can see, that the system is guiding us step by step through the whole process how the drawing was made. In this way how can you benefit? For suppose if one of the friends in your group is a little expert or little comfortable while using this software, that person can make the drawings, and the beginners can just watch the history that how the drawing should be at any point you can go comment, see the dimensions.

They can see the steps those were followed while making this drawing, home. So this is the original position now. So this is the component that we will make, and I will just go through the various steps in this I will try to make links first, the links, then I will try to make the hinges or the hinge support for our laptop top then I will make the sheet. The sheet actually this is our top face of the laptop that I will make in the later parts.

This component I will just make, and also the screws are also required. The screws and I will also try to use the screw as a tool to cut the hole in my link. So let us go through the steps for making this.

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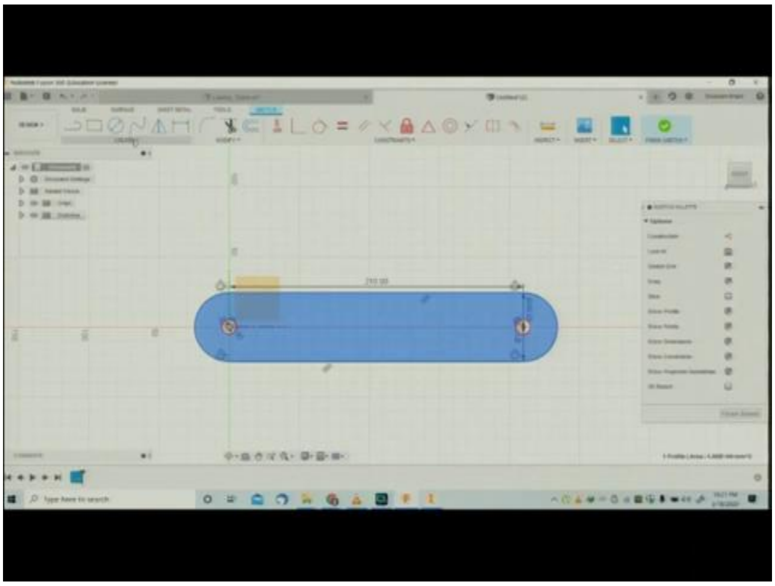

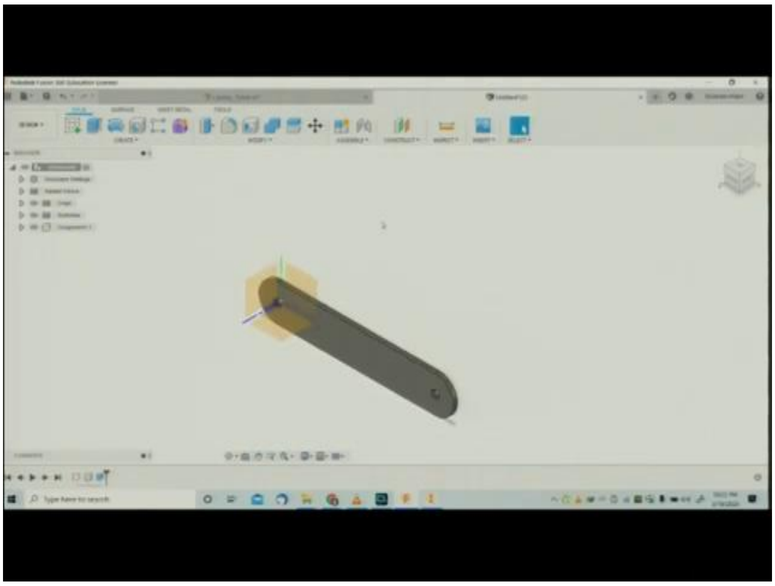

So I have come to sketch here, sketch environment. So in this, this is a center here, how do we start it? First I will make the part that is link I will come to create tab in this, in create I will select line, select a line, click on center, and we can put dimension 210.

Now I will create new sketch I will like to create here a slot, center to center, this center to this center it will create a slot here. Now, I need to provide the width of my slot, I will make it to 49, 49 is the dia that becomes the width of the slot. Now I will create a circle here, this is the hole for my screw or the pins dia is 10 mm. So this is another circle again 10 mm dia.

Now, we can take this constraint by clicking and then pressing the delete button. So we select the profile here and finish the sketch while clicking finish. Now we are in the 3D environment here now in solid environment. I will select the profile and extrude that, extrude it as a new component, new body options are always there.

As a new component we can make changes independently for the new body it will become as if the part of the any changes in the previous body would reflect in the further body as well. So I will it can just extrude it as a new component, 5 mm thickness. Now this is the link that is ready.

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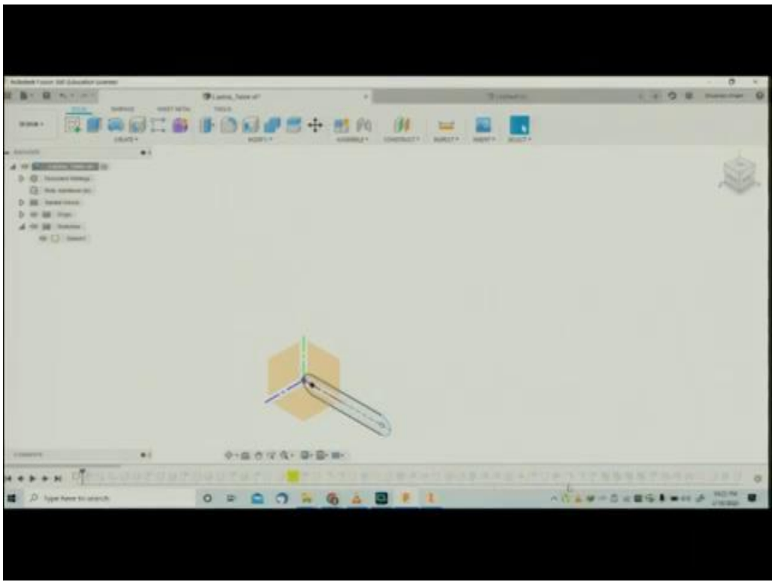

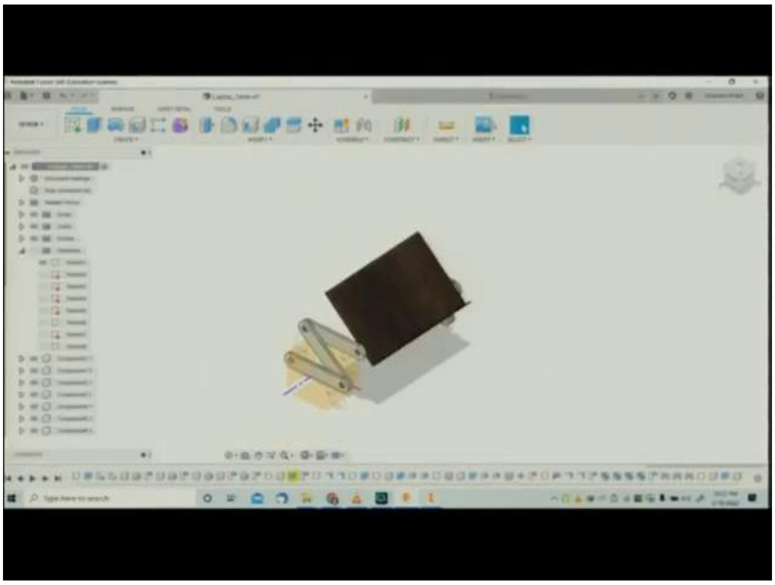

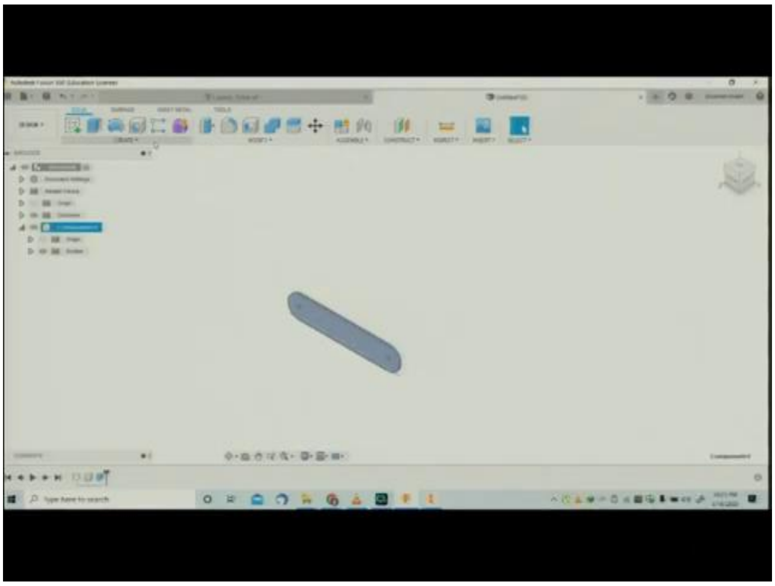

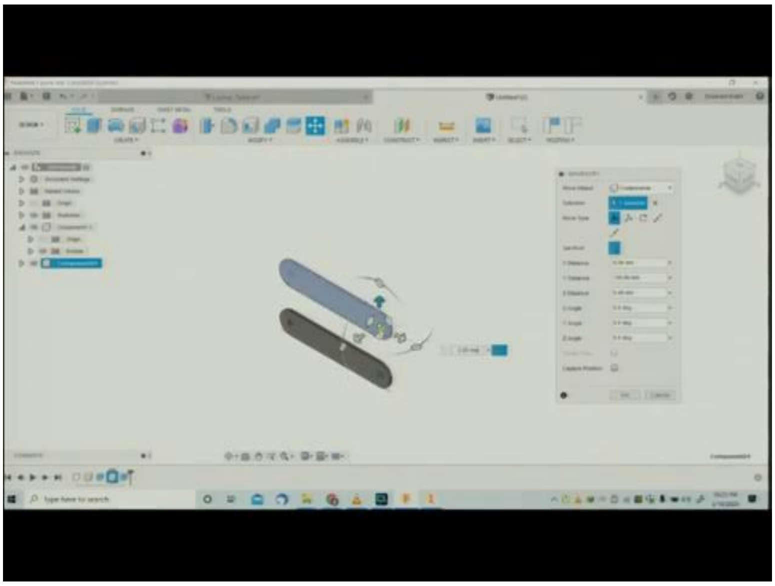

Now, our objective was to create links like these, like in this design you can see. There are 3 links on the left, and 3 on the right. So total 6 links of same dimensions. So, let us try to continue this link we need to create copies of this link. We can show or hide the constraint, geometries, or sketches here. So this is sketch.

Now this component is to be copied, what do I do? I double clicked on my geometry, select the part that is right click on it, then I copy it from here, and we can paste it here, paste and paste new. Again paste, and paste new two options are there when I say paste it would just paste as a copy of this, and any change in the primary component would reflect in the any further components those are made from that.

Paste new would be an independent component in which the changes can be very independently. So paste new is the one that I would like to do here. So, a new link is created at that place itself so we can move it in different directions, we can arrange it or actually, this has to be, center has to be aligned. So I will select a pivot point here, a pivot point pivot, this from point to point or center to center is better.

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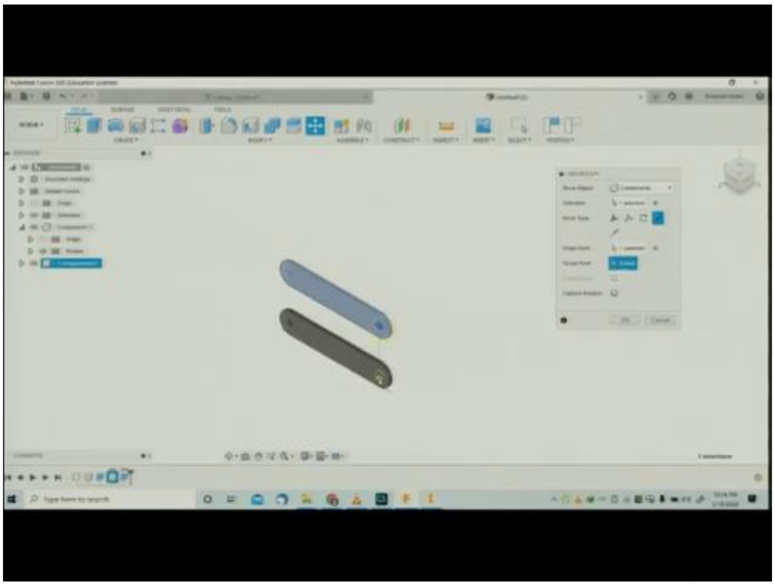

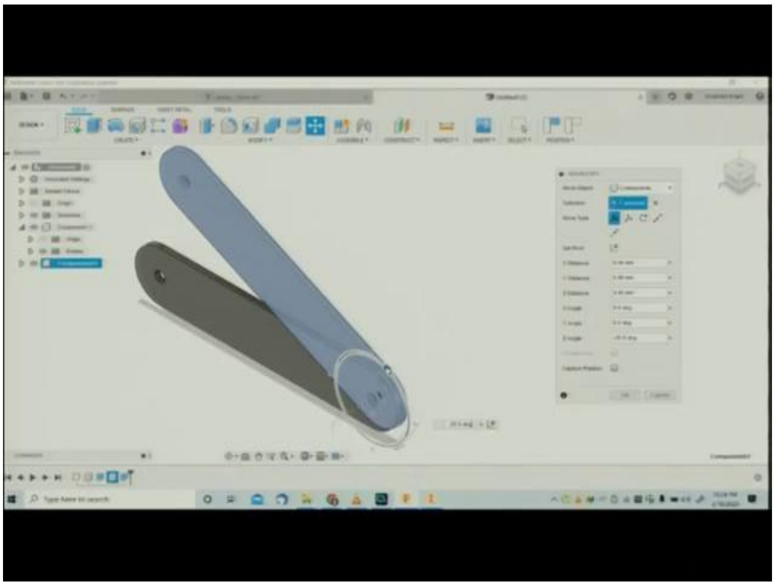

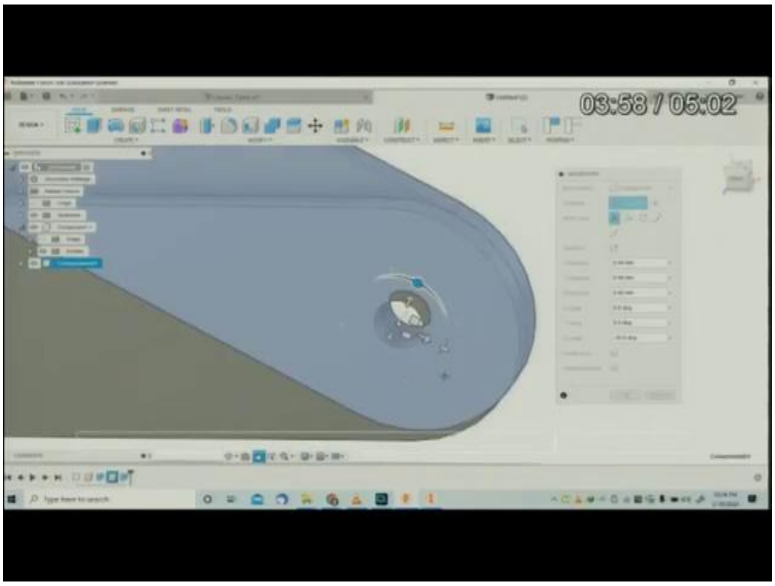

We move from center to center this point and this point is aligned as a center, then I can move this. Now these two points are aligned, and also we can see this component is minus 5, now the components are completely attached to each other because the thickness was 5, so minus 5 has taken it the 5 mm distance.

Now they are completely rubbing with each other, so we can rotate it around the central line, they are both link because it is pivoted here. So let me put some value rotated up to some angle let me say minus 40, minus 40 is okay.

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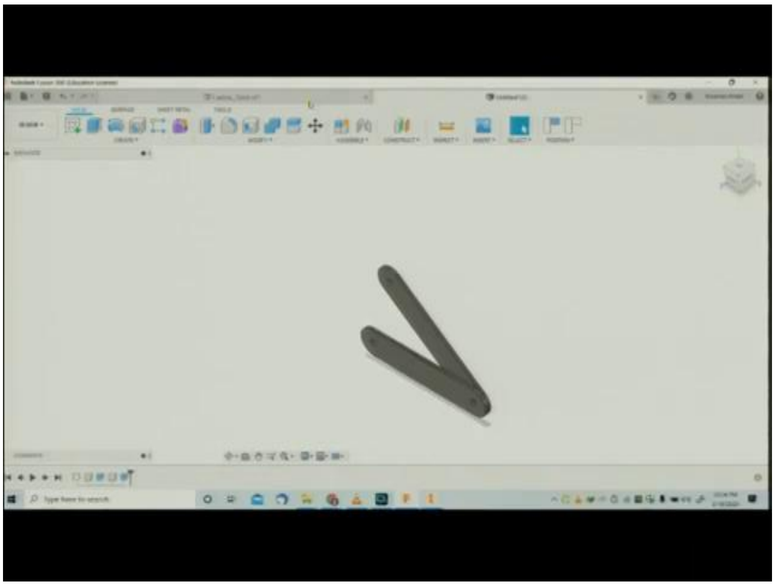

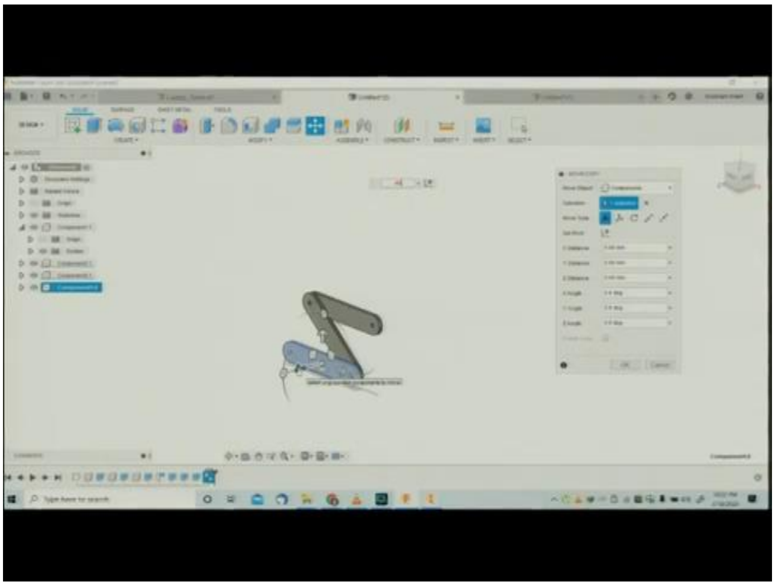

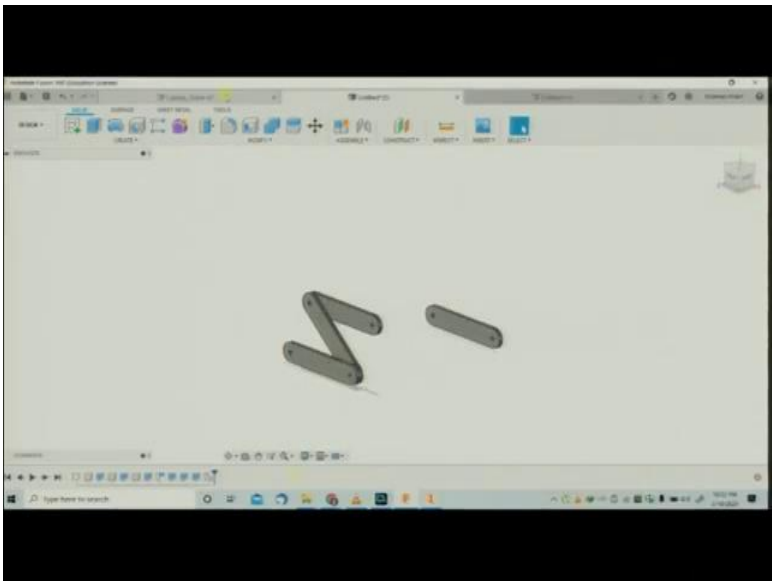

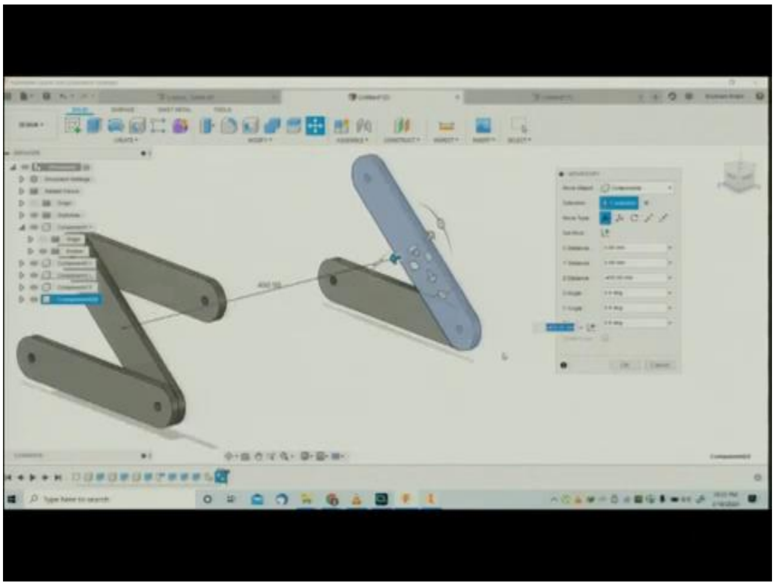

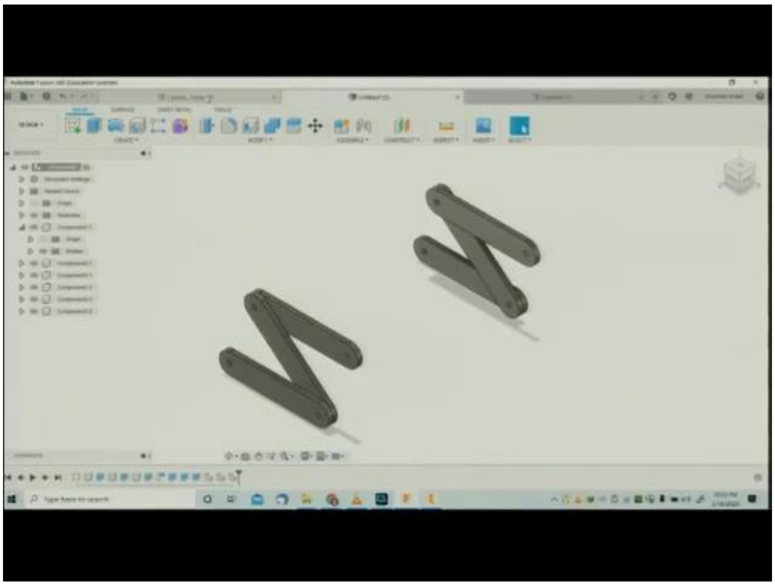

Now these two links are created. So third link can also be created in the similar fashion by right clicking on the component, select copy, and then I go to the open space here, work place here, and click there and paste it here.

It is pasted this component on the component it is complete overall overlapping. So I select a pivot point here the components are rotating. Okay, the component has to be offset by 5 mm (it can offset for 5 mm) minus 5 mm here to make it look like this.

Now, this links are ready another 3 links and I can select the components again right click and copy. Okay, control C and control V it can also be used as like in MS word, control C for copy, now control V is pasted here.

So, I can provide the distance between them the distance would be equivalent to my sheet, my top sheet, it is minus 460, minus 460 value that is put here. This distance let me check it here, yes it is 460 mm. Now, similarly, we can complete the second link, copy C and copy V, again the distance between them, minus 440, again control C and control V, it was 440 10 mm on this side so it will be minus 420. So, I have similar links here.

Next thing that I will design here is my sheet metal component, the top phase has to be designed.

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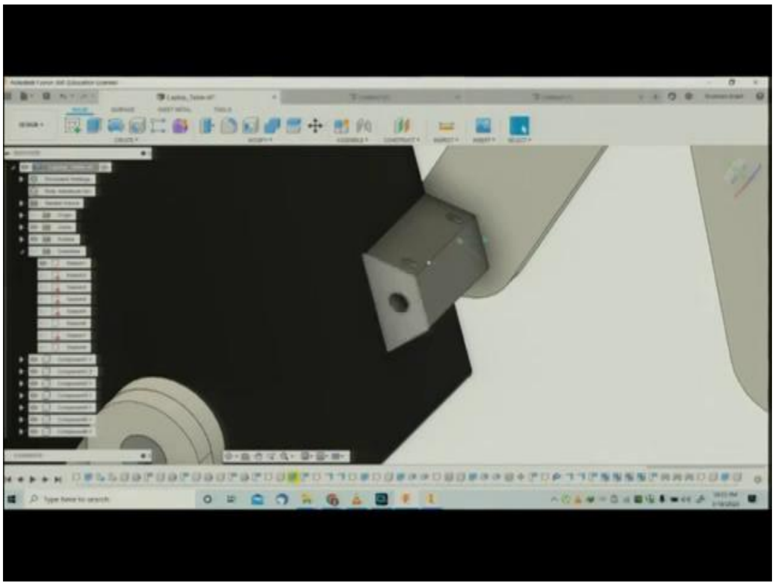

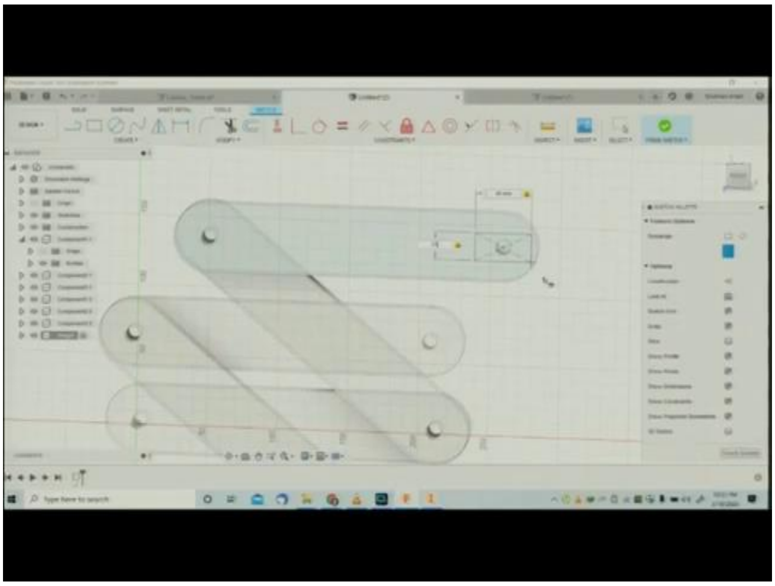

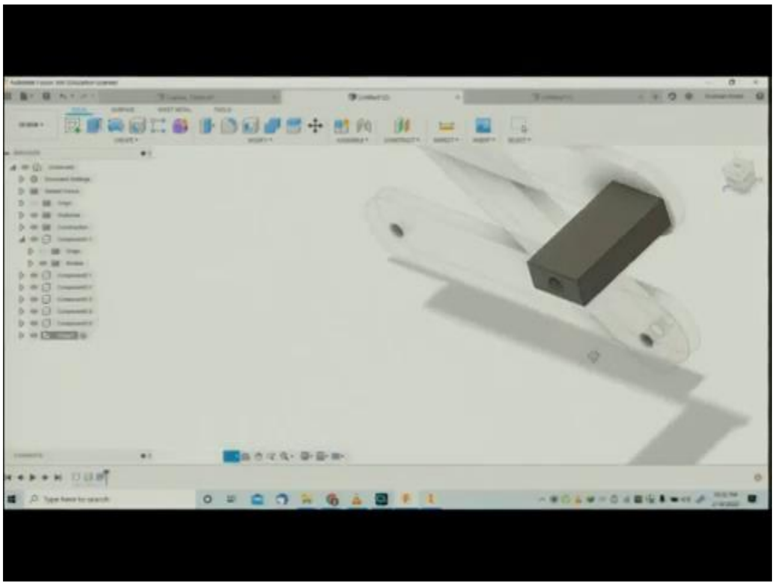

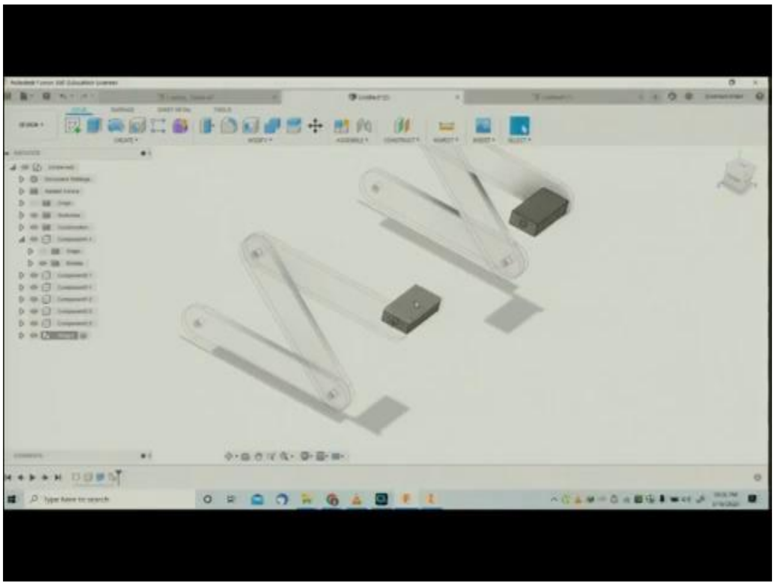

Before that, I have to design a hinge here, this hinge. It will fixed below the sheet metal component. This hinge support I would say for sheet metal this is to be created. I will just make a simple rectangular hinge here.

So, first, we need to make a sketch here, for that I will pick rectangle a 2 point rectangle or a center rectangle, therefore, we have a center there we select a center here and make rectangle. The dimension for rectangle is dimensions of 40 cross 20. Oh, I got it the other way it is 20 cross 40, I will make this dimension as 40, and this side it is 20, 40 by 20 rectangle is there.

We select a profile and I will extrude it to make my hinge. On your keyboard, we can press E for extrude commands. So in extrude, I have put 80 mm, 80 mm distance is the extruded length. So, this is created as a new component. Now we have this hinge support here.

So again we can control C and control V a new component is created. For alignment yes it is different options is translate-rotate we can select point to point. We can just put this point, this point here, this is first point and this is the second point, this center point are not aligned. Now, we have created two hinge supports.

Now, as the geometries are not independent, the parent geometry, any change in the parent geometry would bring the change in the other geometries created from it.

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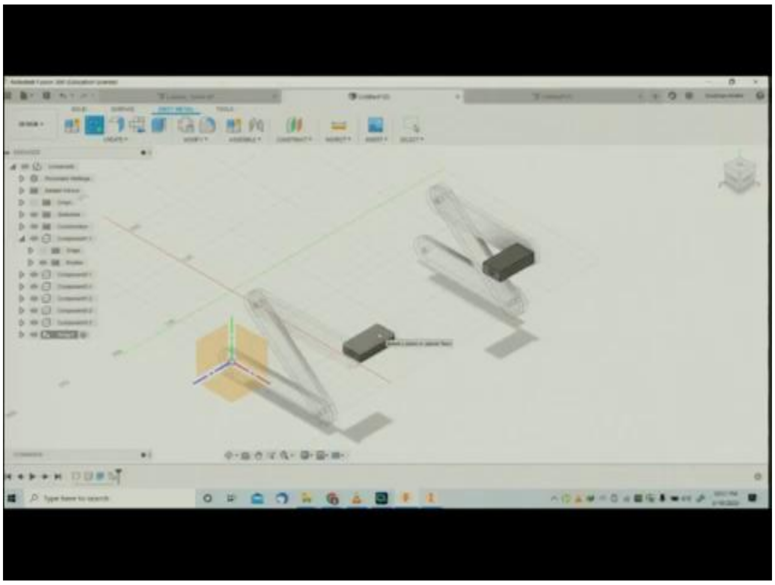

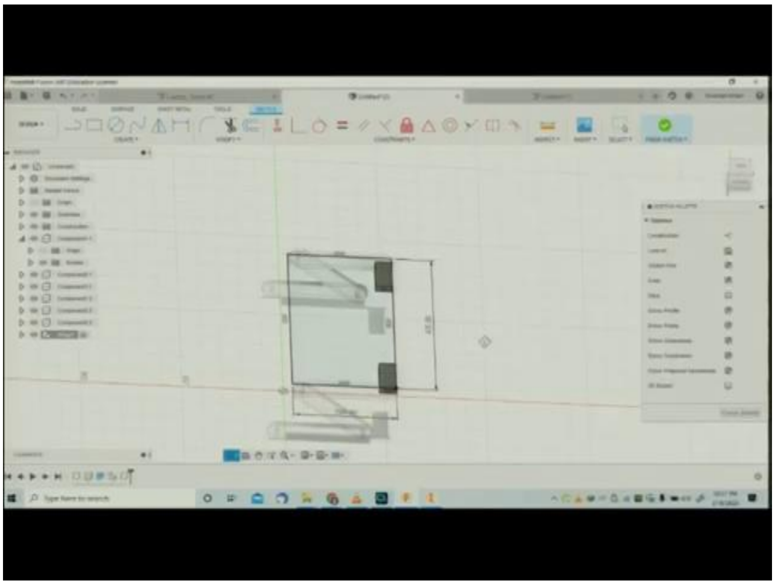

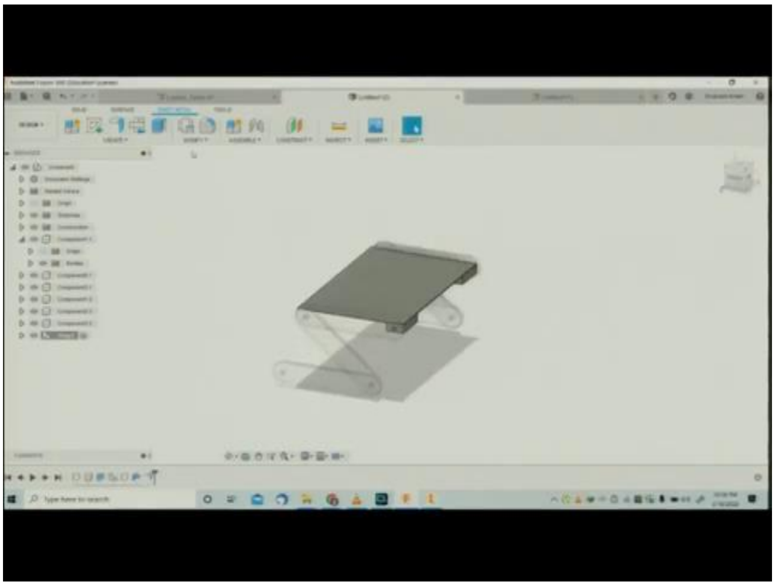

Now, we will make this top sheet, this is a sheet metal, sheet metal component. For this we can select sheet metal option here, sheet metal, okay now this is the sheet metal environment. First, we need to make a sketch, select a sketch here, then define my plane, now I have selected the plane that is the top surface of my hinge support it is selected this plane. We can make our geometry here, now the size of the geometry is to be put here. And this side is, in our case, it is 420 cross 256, and 420 is my length and 256 is the width of the top face or the top plate.

Now 420 has got overlap it, I will make it 410. Now yes it is 410 by 256 actually before making the drawing you have the dimensions ready with you. I am just trying to make a model just to portray the different features or to show a different feature. So I am selecting the dimensions accordingly.

So we can make a flange here. So we need to set geometry to make the flange. So in this you can see the different materials options are there steel, aluminum, SS and so we can select it as a new geometry or new body, and always it is important to select it as a new component. If we make a new body, we would not be able to make an independent change in that we have to select as a new component here only.

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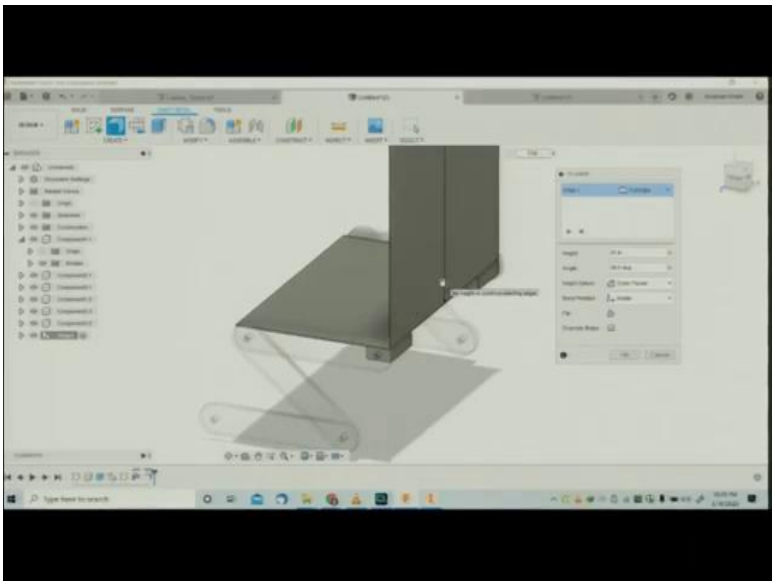

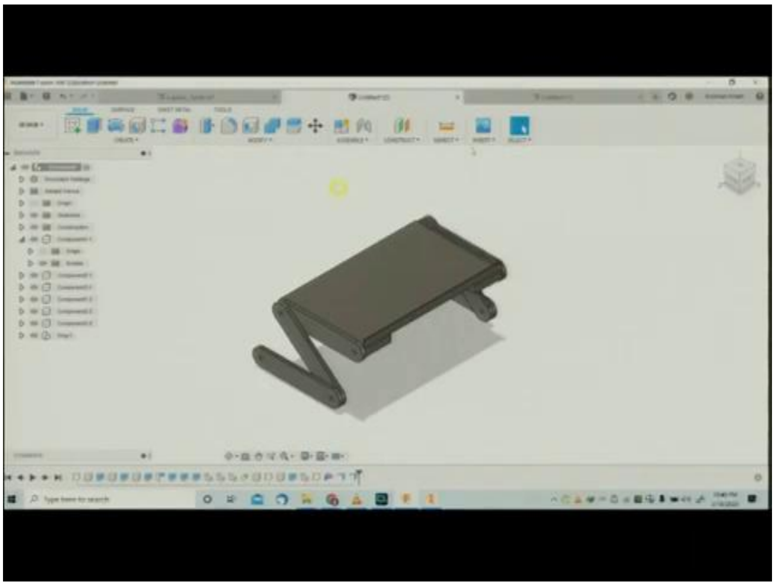

I select it okay, I have come to the base design here. I need to create flange here. You can make a bend or a flange, again I will make a flange select the flange option. From the parent sheet metal, I need to select the geometry, geometry where I like to make the flange, flange in this section okay. Now it is making the flange. All I need to do is just to pull it in the distance, any distance I can really put a distance here. Let me take it 25 mm here.

Okay, the basic dimensions here 25 mm. If I need to changes the dimensions into inches, only I need to do is I will need to put 25 in or it has taken it as 25 inches, but I need to put that in mm, so it is 25 mm, select okay, and this is done. So come back to solid geometry. Now I will start to make joints before that I need to insert pivots.

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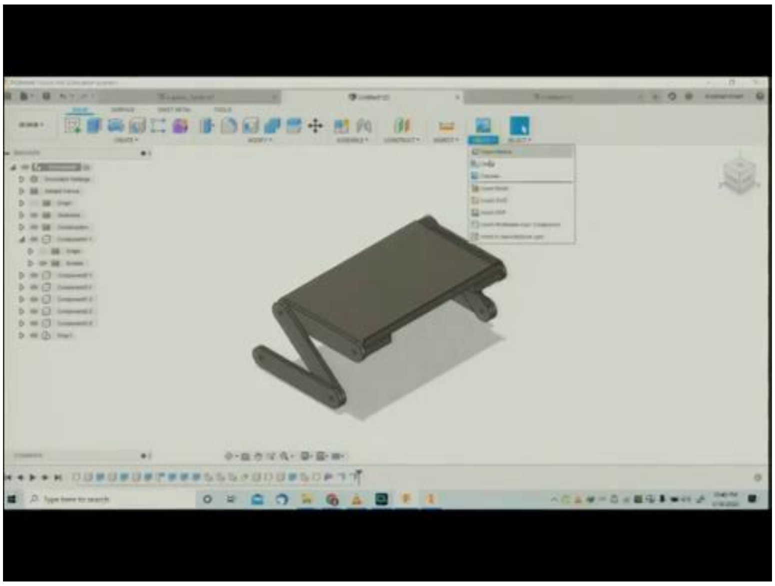

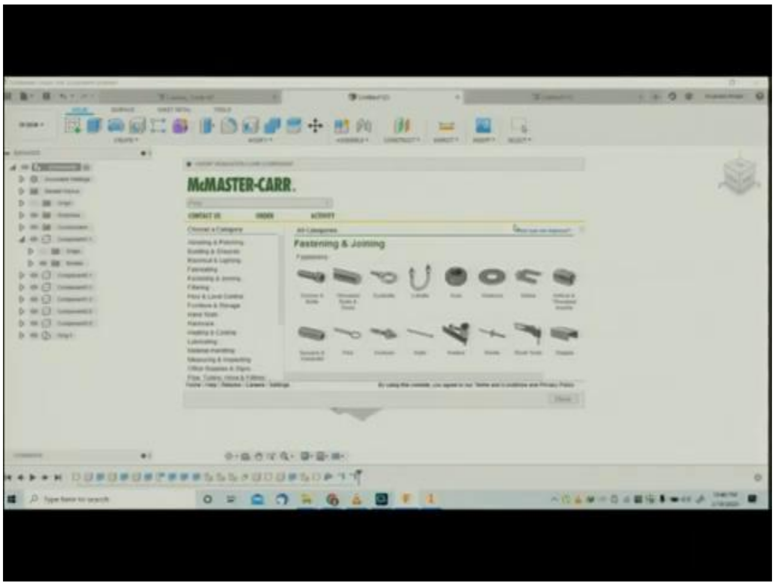

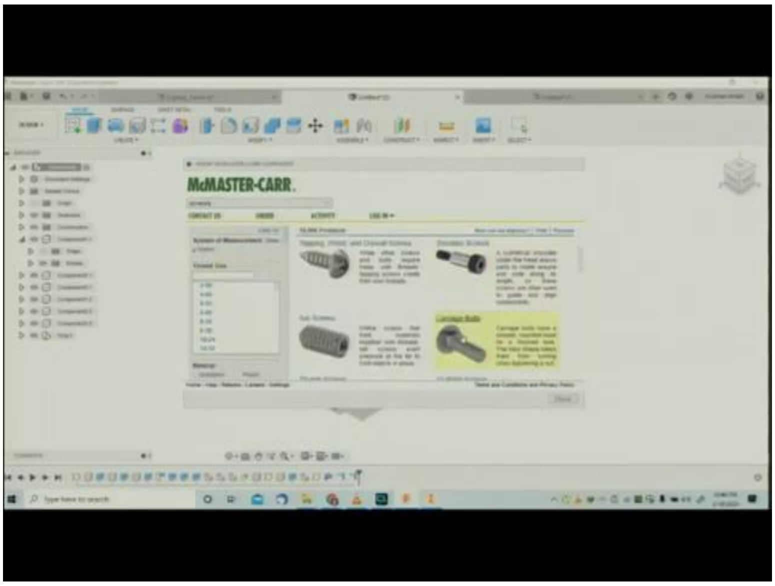

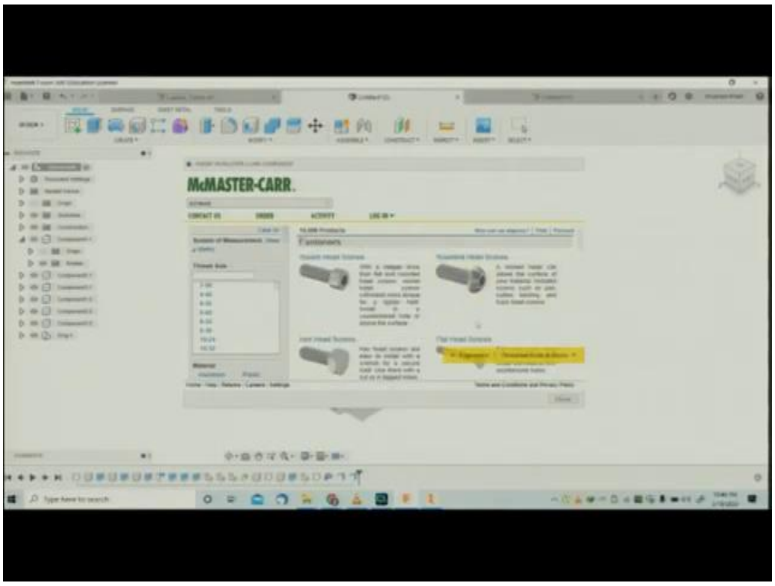

Now in Fusion 360, we can import different components here. We can feed many computer components with using several libraries provided like this in the McMaster-Carr in which we can select these different kinds of components it is just showing a different kind of fastening and joining. So, you can see the different options are available electric and lighting components, fabricating, fastening, joining, filtering, flow or level control, furniture, hand tools, and so on like material handling maturing and inspection.

So they are fastening and joining different kinds of joining components, there we have screw and bolts, studied rods and studs, I-bolts, U-bolts, nuts, washers, shims, spacers, pins and so on nails are also there, rivet etcetera are there, staples are also there. So, we will select the screws and bolts here.

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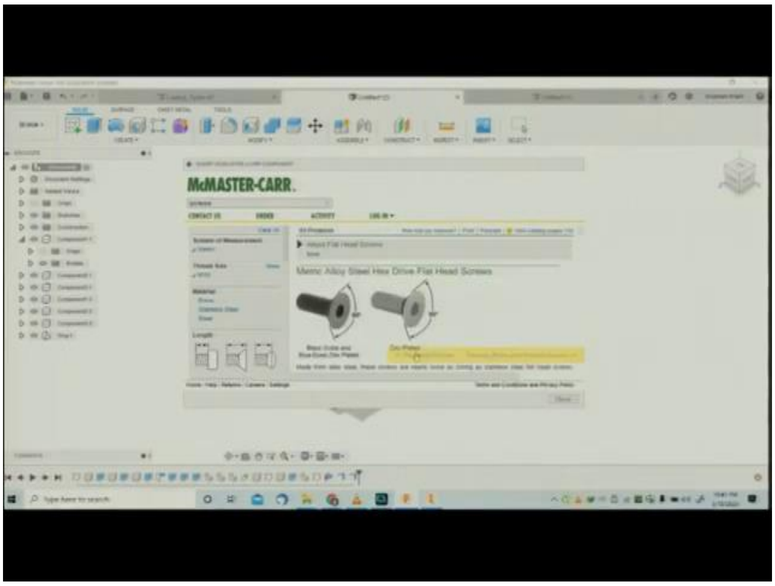

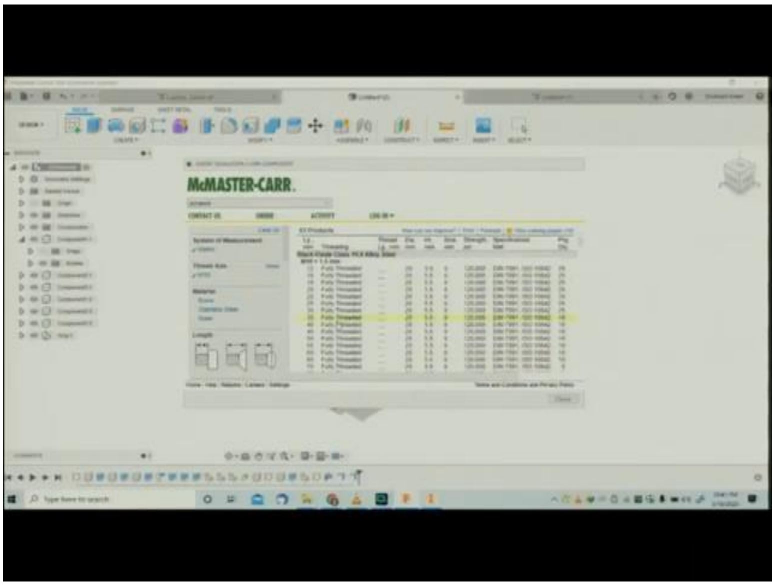

In this screw I will select metric threads screws with contrasting geometry screws, I need to select with metric threads, and that dimensions were 10 mm. Flathead screw I will just select once and I need to select M10 okay this is selected, it is now trying to search for the screw and provide us with the one.

Now, we can select the length of the screw here. I can pick maybe from 2018, 2016, or 20 would also work.

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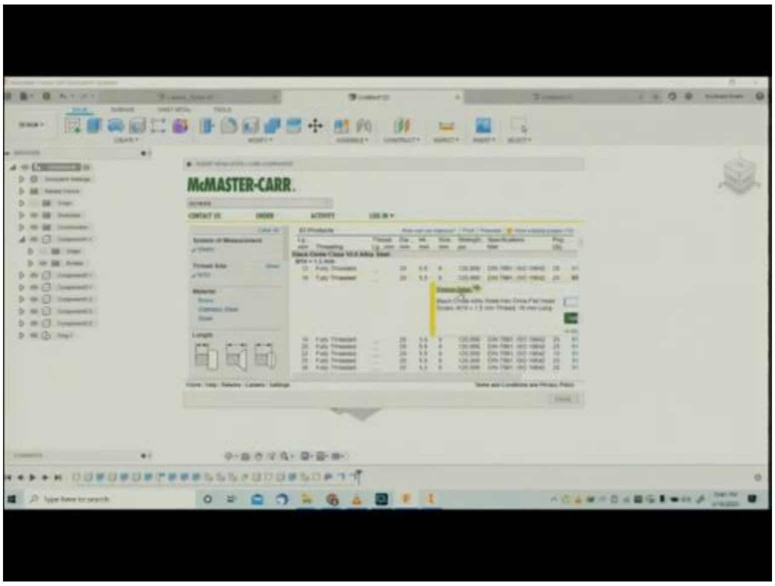

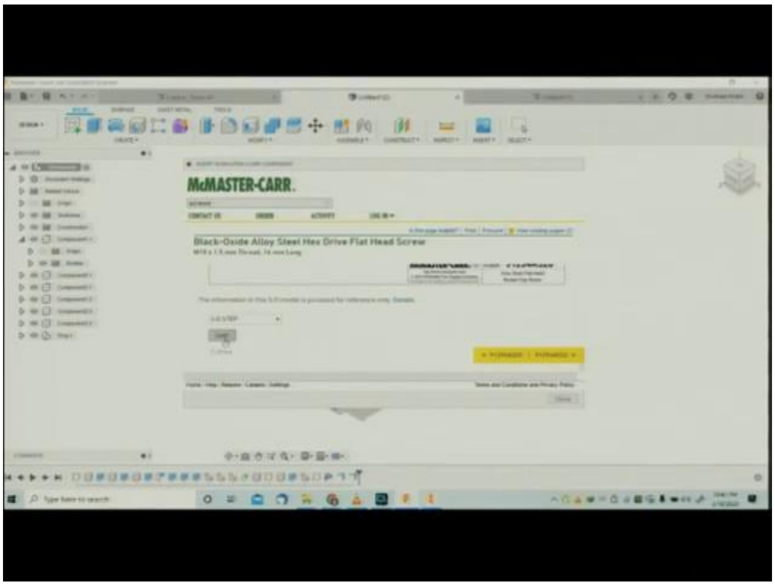

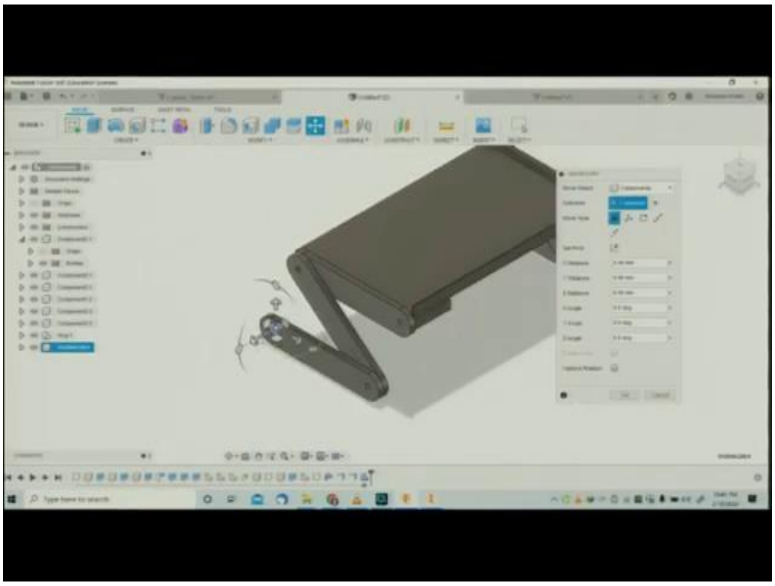

Okay let me take 16 fully threaded screw, product detail we can see it here or this is a product detail. We can donate the file in this format PDF, SAT, SolidWorks. It is generally suggested to import that in STEP format while save this screw here. Now this screw has come here.

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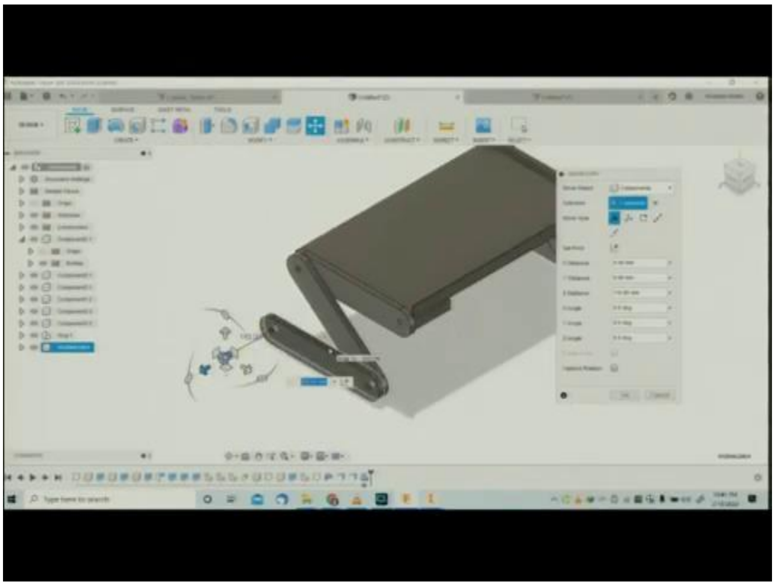

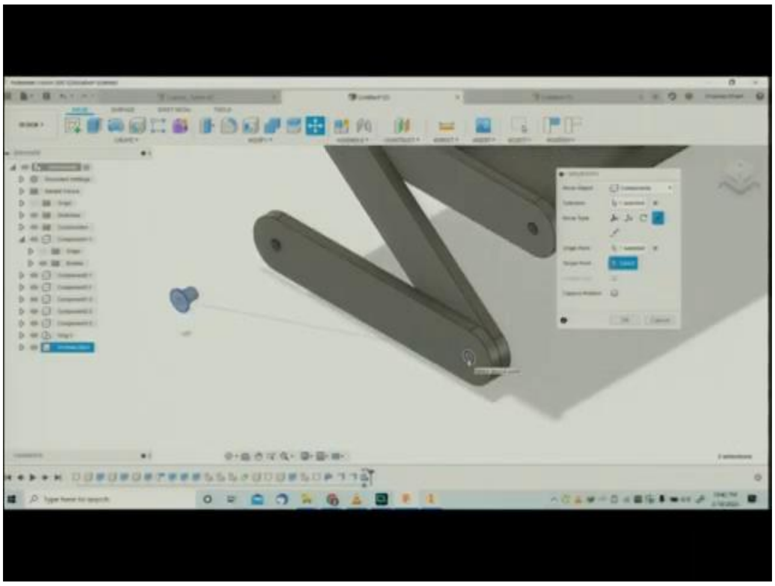

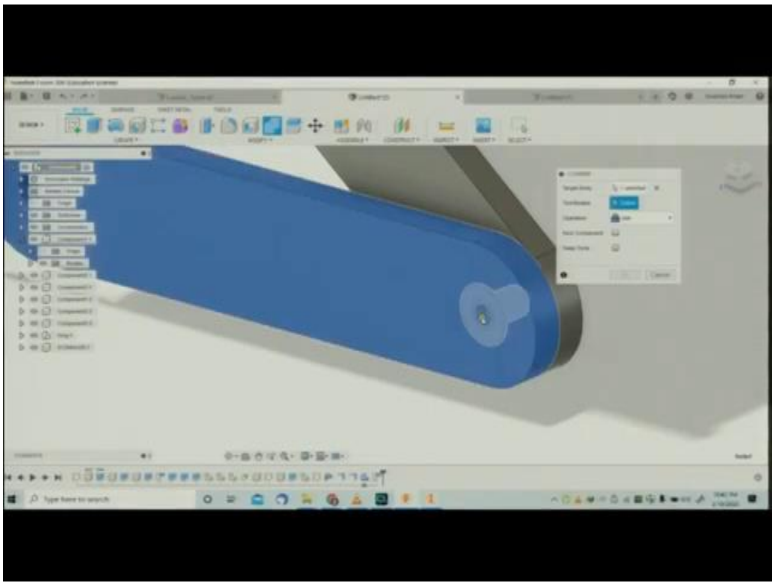

Now here is the component that we have. Now we need to place it at the right spot for which I will just move it point to point. Now, this point and this destination point, just put the screw at the right place. Now one thing is missing here the countersunk hole in our link. So you can see we need the specific geometry here. As you can see there is no countersunk hole there this countersunk profile has to be created here.

What I can do here is I can create this screw as a tool, I will select the combine option and select machine, select the target body as a tool body and cut keep the tools and okay. I will repeat this, what I am trying to do here is, I am going to use this screw as a tool and my link as the material that has to be cut.

So how do I do this again? So, if I will come to combine capture position, I will select the target body, target body is my shield, tool body is my screw and cut, cut the portion and also we need to keep the tools okay.

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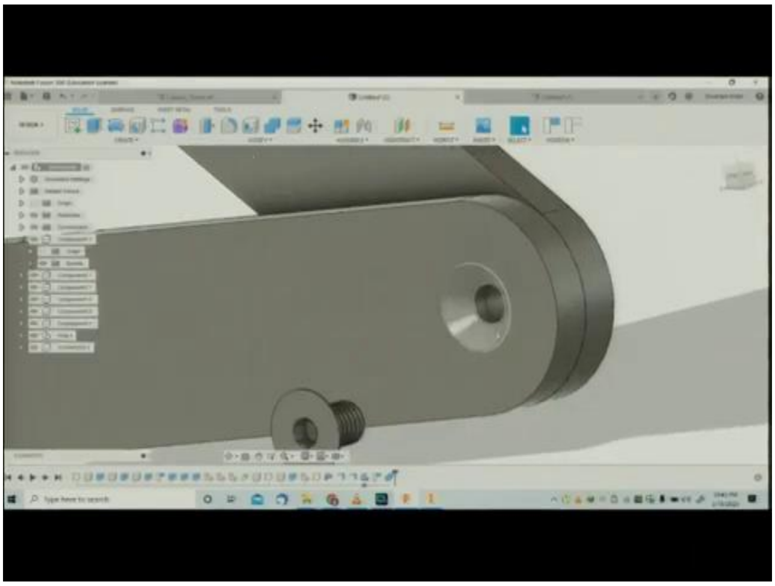

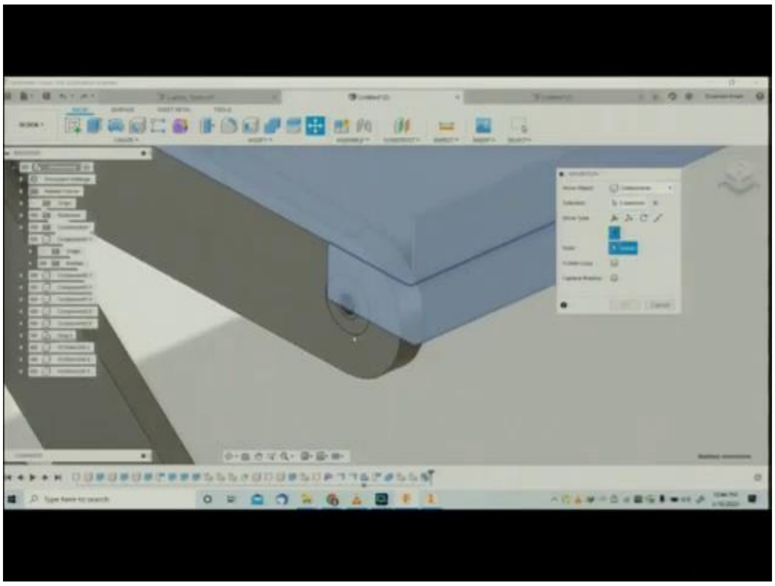

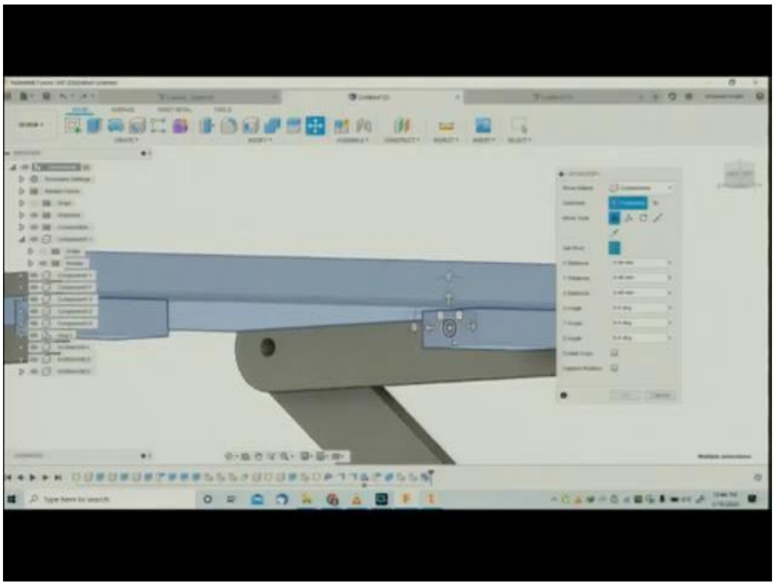

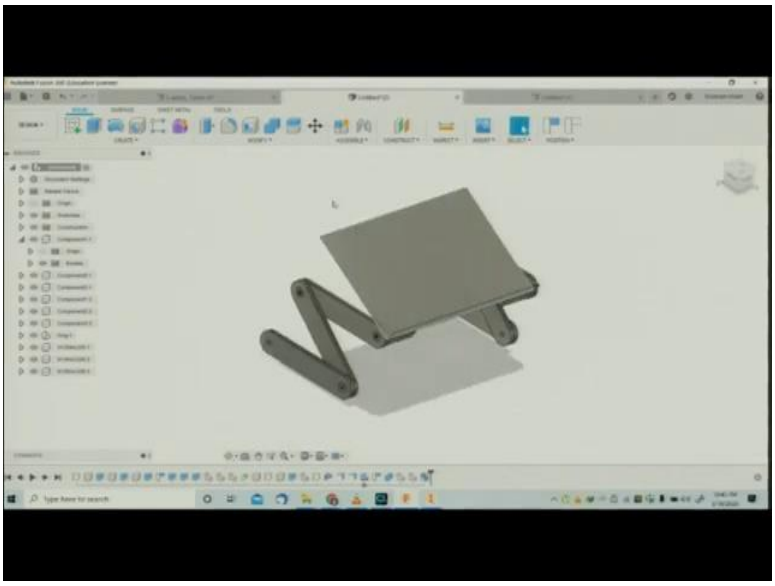

Now you can see here we have a countersunk geometry created in my 3D link here. All I need to do is the repeat the process for all the joints we have 1, 2, 3, 4, 5 other joints left. So, copy and paste, and take it to this geometry point to point, I hope you are doing it copy and paste and point to point from this point to this point.

So, I have created in this side, these two similar screws can be created on the other side. Now I will start to put joints here in the assembly. Let us make us some rigid joints, I need to make sheet metal, and the hinge support as a rigid body now this has become a rigid body.

Whenever I move this, is the kind of a this has become a kind of a permanent joint here wherever I move this body the both bodies are kind of now one body. They were created separately, but they have become a rigid joint here. Now I need to select my pivot point here, for rotation, select M for move select my pivot point, here this is my pivot point and select pivot point and we can rotate it here, yes, now let me put some value 30 degrees, okay minus 30 degrees.

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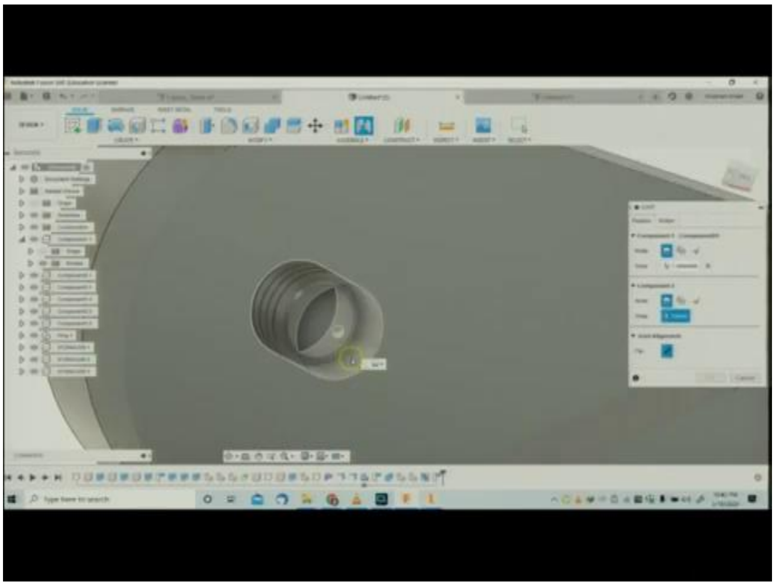

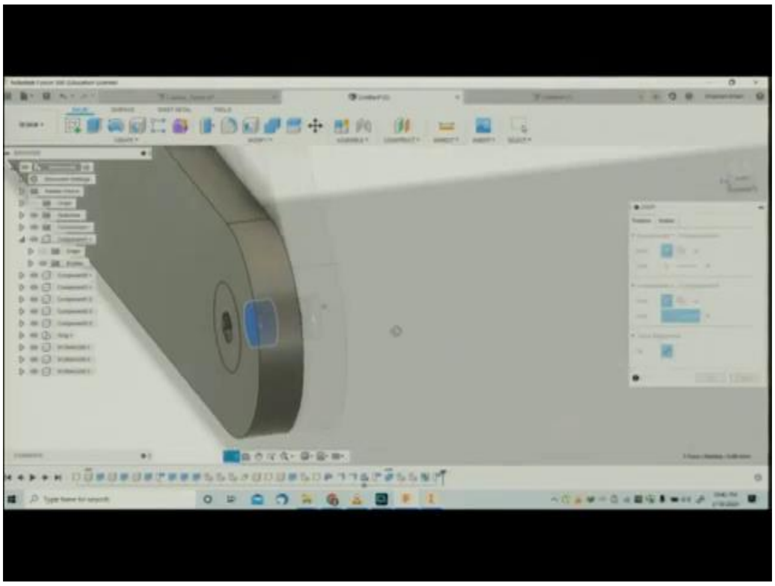

Now, we can provide different joints between the links, we have separate links here this is the first link, this is my second link, this phase. It is better to isolate the components. Since so many components I can isolate these two components from the other components, and select these two components this plus the component.

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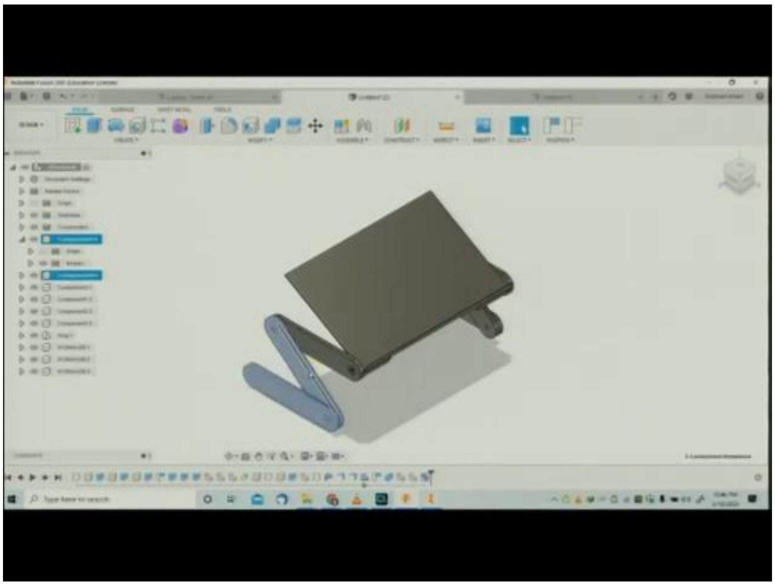

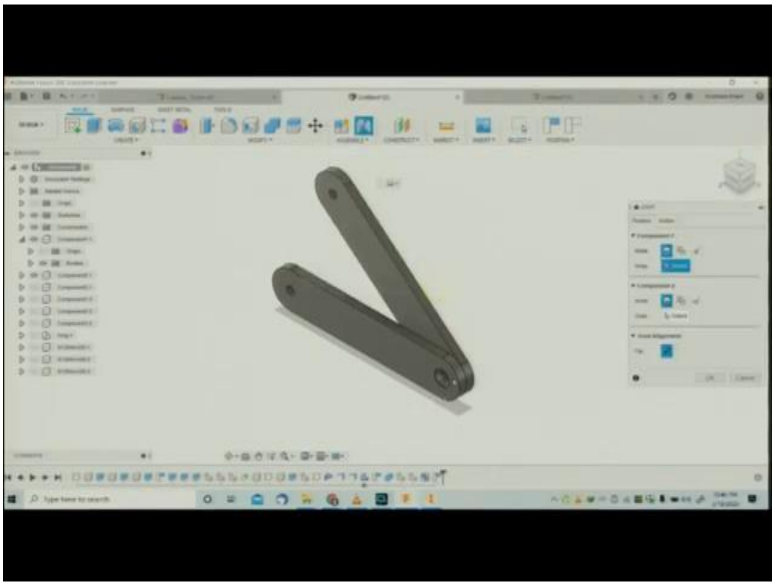

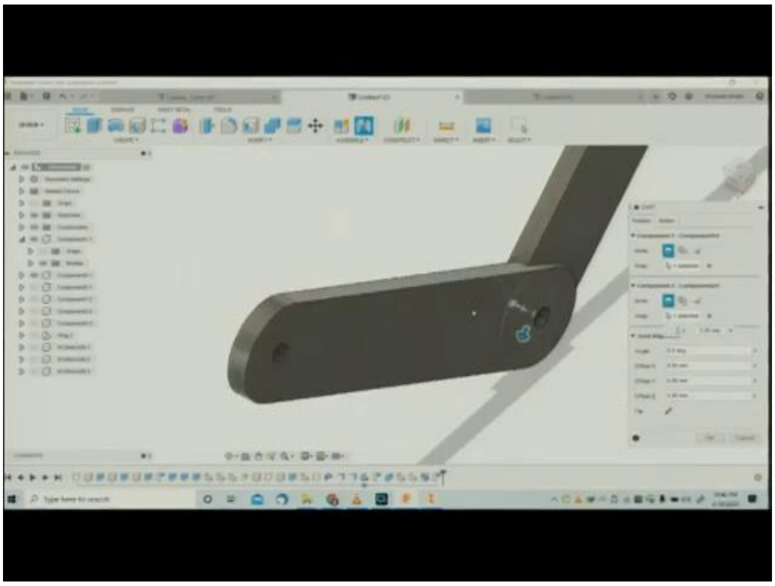

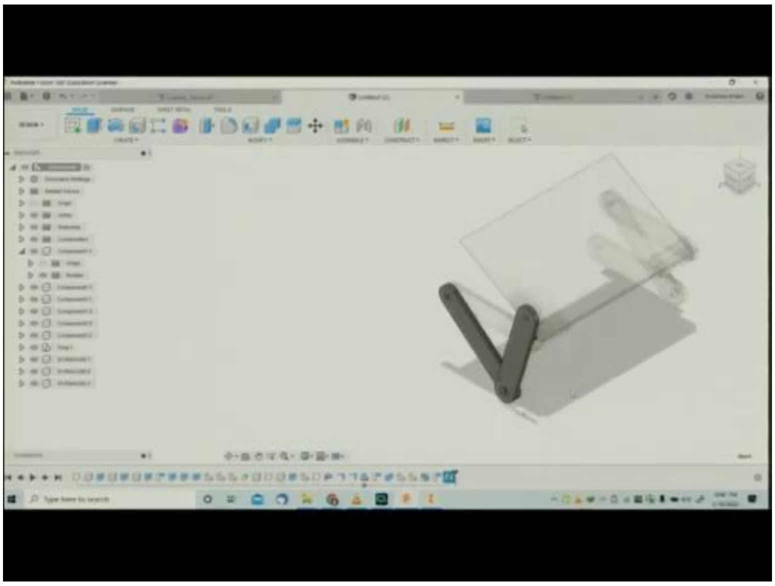

So I have selected both components while pressing control, right click to them, and isolated them. Now we can freely work on these two joints. These two components in which is one joint I can select come joints capture position, select first component, and second component.

The screw that was there before that was bringing its profile in the screw test was being shown. So, I have isolated these two components to work on only them, and I can create the joints here. The joint type here will be revolute, it is now showing a complete 360 degree revolution.

Now if we joint anything both these two components are completely in contact with each other. If I try to start rubbing them there will be lot of friction in them. So they need to some clearance between them. It should have some, it would be kept like this it should have some clearance so that the joints are free. So that clearance I will provide here. So I have provided this clearance here.

Now, I am offsetting it to provide the clearance, so it is now some clearance is here you can see here. So, let me put angle here, 40 degrees, we can also constraint the motion how to do that we need to put the minimum and the maximum angles, minimum would be 0, maximum would be 40 degrees or 45 degrees.

So, I can isolate them here to see the whole assembly, to animate the joints we can also see the animation here. So it is still showing 360 degrees for that to constraint it from 0 to some angle. I need to do this.

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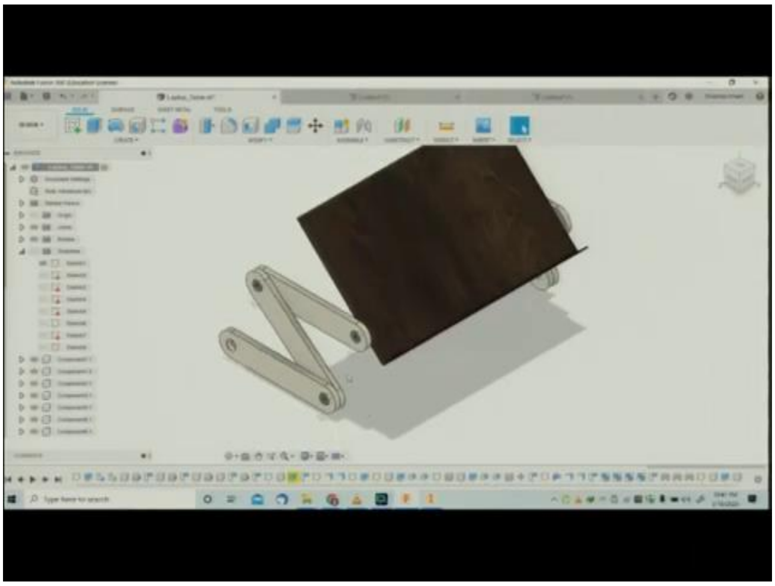

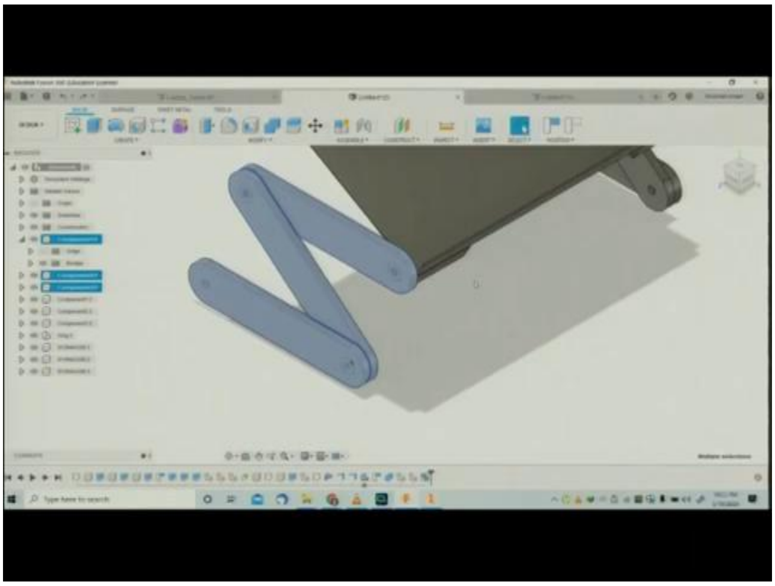

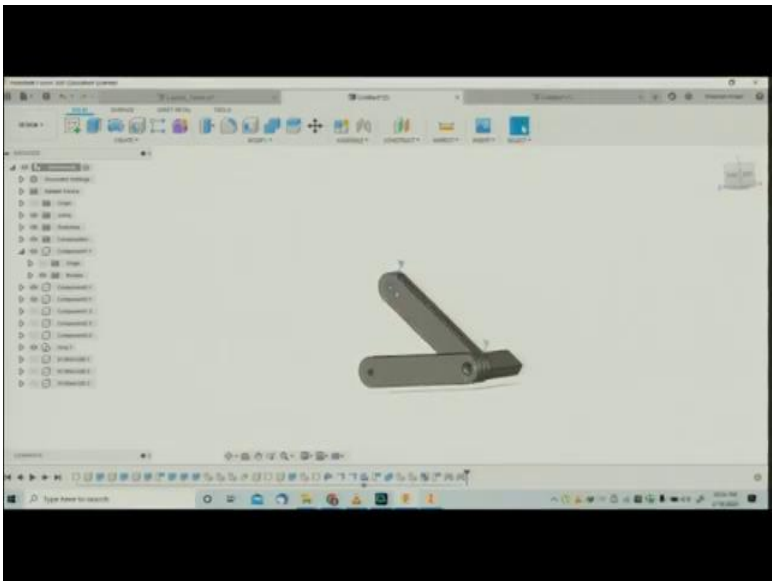

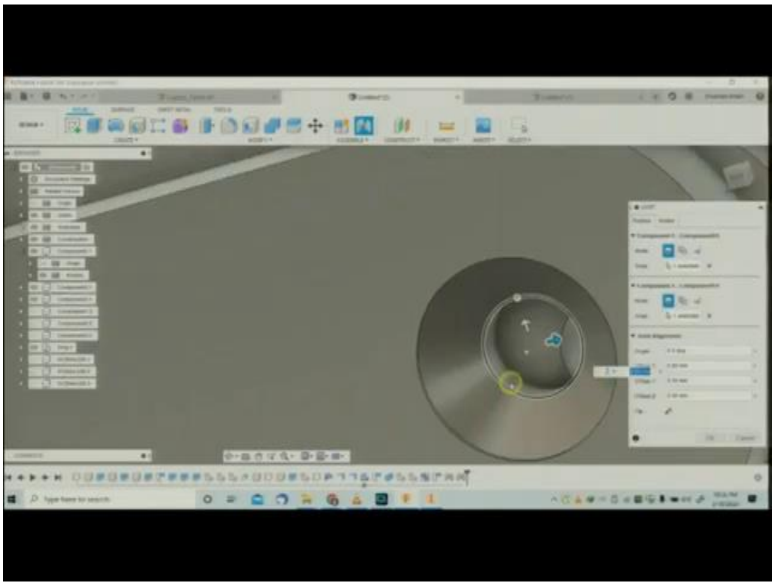

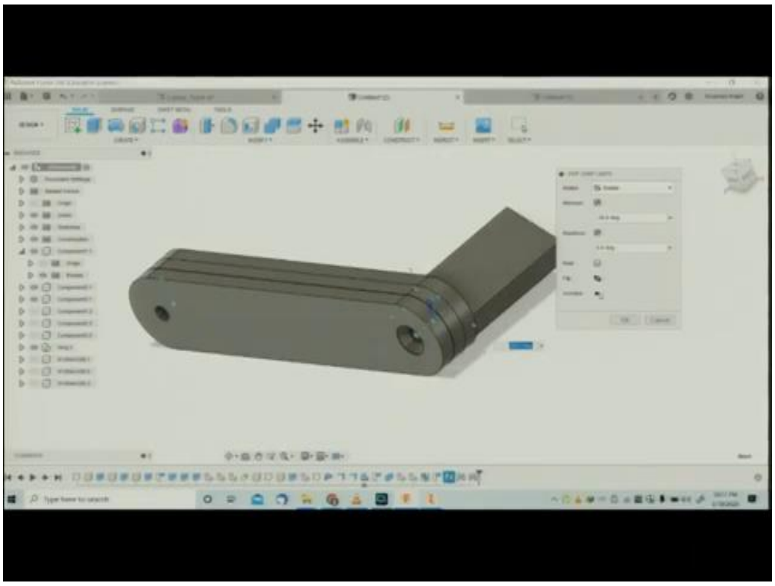

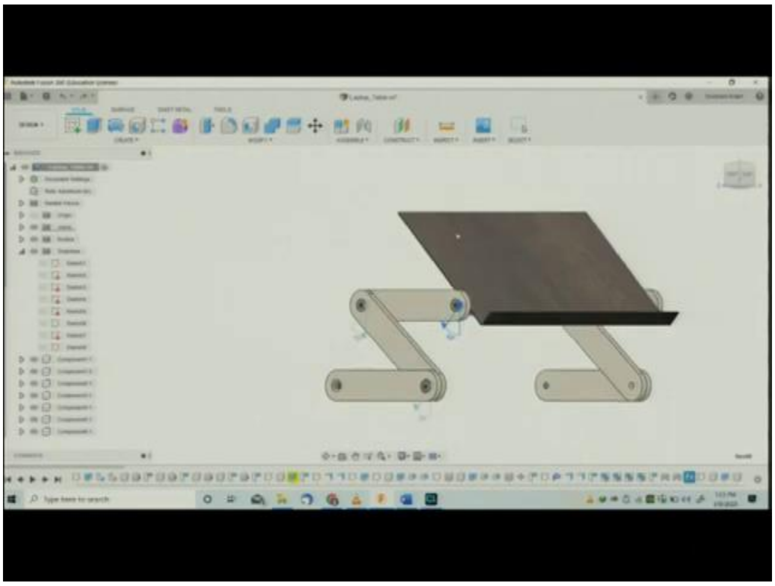

So we have already created this model we can see the properties of the geometries, and the joints here. So how did that happen we can see here. This is the joint, right click, constraint, modeling, right click, edit joint limits you can see edit joint limits you can see in the minimum and the maximum, minimum is 0 degrees and maximum is 45 degrees.

So similar things we can use here. Again here edit joint limits 0 to 45 degrees in negative direction. The direction one is direction minus 45, and another one is direction plus 45 degrees. We can also animate the models here see this animation is the one that we would wish to create in our model so how do we do that?

I will select these 2 links or these 3 links together. We can select these 3 links together, and the hinge wherever we need to join. Now these 4 components are taken and isolated we consider the joint origin capture position, and these two joints are to be rotated first phase, second phase, and we select.

Now this is 360 degree rotation once, now I need to constraint the rotation to 45 degrees. We can provide a constraint later, for this also I need to join this select join capture position, first phase, second phase, and select first profile, second profile, and revolution. This is second animation, okay, similarly this and this again, capture first profile, second profile again revolution is possible here. I will offset it by 10 mm to provide clearance.

So, I need to step the joints again here and provide a complete revolution now the whole revolution is almost ready. Again providing the offset here, and provide offset maybe minus 4.8, okay. Now, the whole revolution is almost ready.

Now, I need to provide constraints to this revolution so that the revolution is limited to some angle 45 degrees or 30 degree for the hinge. How do I do this? I will write click and edit joint limits, minimum and the maximum values could be put here the minimum is 0 degree and the maximum for hinge support is minus 30 degrees, and for this first link edit joint limits minimum is 0, maximum is I will minus 45 degrees.

This would go in minus or plus 45 degrees, minimum is 0 plus, and this is minimum 0, and maximum is minus 45 degrees. This is the product that is ready and let me try to animate this I have just clicked right click on the body on the whole product and clicked animate. So this is the component that is ready. Here I would like to stop and finish this lecture.

We will meet in the next lecture where I will discuss about the rapid prototyping concerns and also I like to discuss something about the rendering. About the appearance, about showing the components, about downloading, about saving the product that we have made in JPG format. We can choose different environments, different backgrounds for them.

We can choose environment like in a machine shop or in a room for furniture this kind of laptop stand we can consider room how does it look like in our room. So this we will discuss in the next lecture.

Thank you so much.