Microsensors, Implantable Devices and Rodent Surgeries for Biomedical Applications
TA: Ajay Krishnan A, Course Instructor: Dr. Hardik J. Pandya
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore
Week - 08
Lecture - 31

Greetings everyone, welcome back to part 3 of our NPTEL lecture, *Microsensors, Implantable Devices, and Rodent Surgeries for Biomedical Applications*. In the past two lectures, we discussed different kinds of 3D printing techniques, how they differ, and how we can use them for specific applications. In this lecture, we will discuss how to make a 3D model. In previous lectures, we explored different methods of making a 3D model. One method is using STL files, like diode scanning. Another is creating a file from scratch, using 3D modelling software and converting it into an STL file for 3D printing. Another method is photogrammetry. So, there are different ways to create a 3D model. In this lecture, we will focus on how to make a 3D model using 3D modelling software. We will discuss SolidWorks, how to use it, and some of its features, providing an introduction to designing with SolidWorks software.

I hope you can see my screen. When you open the SolidWorks software, which is paid, this screen will be available to us. The specific software I am using is SolidWorks 2021, but even if you are using the 2017 or 2013 versions, there will be slight variations in the layout, but almost all features will be the same across versions. You can compare one version with another.

This is the screen that will be available once you start or open SolidWorks. First, you need to go to the "File" option here, and in the "File" option, select "New." Once you open "New," three options will be available to us. The first one is the "Part" option, where we create a single part, a single 3D model, or a single 2D sketch, whatever it may be. Then you have the "Assembly" option.

In the "Assembly" option, we combine multiple parts to create an assembly. Then you have the "Drawing" option, where once we've created a 3D model, we need to create 2D sketches of that model. We will use this option. Out of these, we will currently use the "Part" option. First, we will try to design a 3D printable keychain. So, I select the "Part" option and then click "OK" here.

This will be the window available to us. You can see a ton of options or keys here, so don't worry about that. I will guide you through some of the most important options for getting started with 3D modelling. Here, you can see the Feature Manager Tree, Property Manager, Configuration Manager, Design Performance Manager, and Display Manager.

We will mostly use the Feature Manager and Property Manager. On top, you will see options like Features, Sketch, Surface, Sheet Metal, Markup, Evaluate, and some embedded dimensions. We will mainly focus on Features and Sketches. I'll tell you one thing: if you need to add something that is not visible in this tab, you can always right-click on any of these buttons, and there will be options like Tabs. Just select the Tabs feature, and suppose you need the Weldments feature, you can select "Weldments" here.

You will be able to see the Weldments option here. If you decide you don't need Weldments and your tab is overcrowded, you can do the same: right-click on any of these features, click "Tab," and unselect "Weldments." That's how we add or delete options from our Feature Manager. The base of a 3D sketch is always a 2D sketch.

We will start by designing a 2D outline of what we need to create. In the Feature Manager, you will see Annotation, Materials, Plane, and different planes because we can use these three planes to start building our 3D model. We also have the option to add an unlimited number of planes in our sketches. That is a more complicated topic we will discuss later. First, I will select this front plane to start my sketch. I will right-click on this plane, and then there is an option for "Sketch."

Once I select the Sketch option, SolidWorks will automatically set the front plane for sketching. The front plane is the one I am using to start my sketch. First, I will try to make a rectangle, and I will build new features from that rectangle. In the Sketch option, you have different tools like Exit Sketch, Make Sketch, Smart Dimension, and other tools to draw shapes. Since I will make a rectangle, I will select the Rectangle option.

Once you select the tool you need to use, the property manager will open, and here you have an option to create different kinds of rectangles. This particular rectangle is used to make a rectangle with two points, meaning with two corners. This one is used to make a rectangle with one point at the centre and one point at the corner, and this one with three points or three corner points. You have different options. For now, I will select this option for a centre rectangle. Once I select this, this is the plane or work area where we create our model.

In this work area, I will select the origin point, which is fixed by the software itself. When I approach this point, you will see that our pen tool gets locked to the origin point. I will right-click here and then move my mouse to create a rectangle in whatever shape I want. I will try to make something like this, and then again, I will click my left button to create the rectangle. Now you can see that the rectangle is made up of blue lines.

Blue lines indicate that the rectangle is under-defined, meaning we haven't given complete dimensions to our rectangle. Since it is not fully defined, it will have a blue colour, and this dark blue shaded area means it is a closed figure. If I make something like a "C", you will see that it does not have this kind of dark or shaded blue region. This means the shape is open. I will simply delete it.

To delete it, you can select the entire portion. Once you select the entire portion, the line colour will change from dark blue to light blue, and then you can press the delete option. That shape will be deleted. Now, I need to define it. I need to give dimensions like the length and width of my rectangle.

To do that, we have a smart dimension tool. The smart dimension tool is usually used to give dimensions like length, angle, and more. It will automatically detect what kind of dimension we need to give and apply the required dimensions to the figure. I will select this smart dimension tool and then select this line because this is the line I need to initially dimension. I will select this line and then keep the smart dimension tool here.

Currently, it is about 186.5 mm. I need to reduce it to around 50 mm, which is 5 cm. So, I will type 50 in the dialogue box and press enter.

Now, this line will automatically become 50 mm in length. Next, I need to specify the width. To do this, I can either select this line to give it a dimension or press escape, which will delete the selection, and select the line again. I will then give it a width of 28 mm. So, I type 28, and now I have a rectangle with a width of 28 mm and a length of 50 mm. Now you can see that the outline of our rectangle has changed to black.

Black means it is fully defined. Blue means it is under-defined, and black means our rectangle is now fully defined, meaning we have given all the necessary dimensions to our rectangle. Now, I will show you one more thing. I will make another rectangle here, and I will give this dimension 10 mm and 20 mm. But even if I give it 20 mm and 10 mm, the same dimensions as before, the figure is still not fully defined because earlier, we started our rectangle from the origin point, but here we do not have an origin point.

As you can see, when I press escape to exit the smart dimension tool, I can move this shape with my mouse, but it cannot be dragged because the selection is fixed. But here, it is not like that. I can define this figure as our master figure or the first figure. I can select the smart dimension tool again and give this line a length of around 10 mm and the other line around 15 mm. Now you can see this figure is also completely defined. This is how you fully define or dimension your figure so that it is aligned or locked in position.

If I need to delete just a part of a line, I can add a centerline. The centerline is a construction line used to create a figure. It doesn't have any direct influence on our shapes. I will make a centerline here. If I need to delete just the end of this line, I can select it and press the delete button.

This will delete the end of the line. But if I only need to delete part of the line, you have an option called "Trim Entities." You can select that trim tool and drag it over the portion you need to delete. Once I drag it over the portion, you will see that only half of the line is deleted, while the remaining half stays. Now, to get rid of the entire thing, I will select the entire area and delete it.

Now we only have this rectangle here. On the right side, you can see two buttons. One is "Exit Sketch After Saving." If I click this button, the sketch will be saved and exited.

But if I click the arrow mark, it is just an exit button, and it won't save your sketch. Here, I am exiting the sketch after saving it. Now you can see the sketch here. To make this 2D sketch into a 3D model, I need to extrude the sketch. To extrude the sketch, you need to select the sketch, go to the "Feature" option here, and then select the "Extruded Base" option. Select this option, and now I am planning to give an extruded base of 5 mm. So, just select 5 mm here in this depth option, and then click the button. Now you will have a 3D block from a 2D sketch.

This is how we will build our 3D sketch or 3D model from a 2D sketch. Now, I need to add text, such as "3D Printing," on this face. To do that, I need to extrude the "3D Printing" text here on this face. I will right-click on this face and start sketching here.

Then, I will make a centerline and press Escape to leave that line. There is an option known as the text option, which I will click. Here, you have the option to write whatever you want. Now, I will write "3D Printing." So, "3D Printing" is here. If I need to change the font and increase its size, I can uncheck the "Use document font" option and select the font option.

If you need to change the font, you can scroll through the available font options. I will select the Copper font and increase the font size a little bit, maybe to 4, and then select OK. Now you can see that the font and its size have changed. However, it is positioned outside my box. To place it inside the box, I have a reference line here.

Now, I am selecting this reference line. You can see that "3D Printing" is almost aligned with my reference line. I will now exit this sketch by saving it. Now, wherever I move my reference line, my "3D Printing" text will move as well. I will try to adjust this reference line so that it is almost the same size as the "3D Printing" text. However, I need to place this line at the centre of my extruded box.

To do this, I will right-click here and select the midpoint of this line. Then, I will find the midpoint of our reference line. Press Control and select this line. A dialogue box will appear here, where you can see the relationship options, indicating the type of relation you want to establish between these two points.

Now, I need to align them vertically. So, I will choose "Vertical Alignment" to align the centre point of the reference line with the centre point of our main 3D model or main rectangle. Now, I will slightly adjust its position. As you can see, the "3D Printing" text is almost aligned with the centre of my rectangular box.

Now, I need to extrude it. To extrude the "3D Printing" text, go to the Feature Manager, where you will find an option called "Extrude Base," the same tool we used earlier. I am planning to give it a depth of around 2.5 mm. You can see a preview of how it will look. If you are satisfied with it, press the OK button.

Now you can see the 3D text model on our base. However, it looks quite plain, so I will add a fillet to this edge to make it more aesthetic.

To add a fillet, there are multiple ways. One way is through the Feature option itself, where you will find the "Fillet" option. Select this option, and it will be in manual fillet mode. Then, select the edges where you want to add a fillet. Here, I select this edge, and it gives a 10 mm fillet.

Similarly, I will reduce the fillet size to 5 mm or around 8 mm. Then, I can select all the edges where I want to add a fillet. You can see the live preview there.

I select this edge and this edge. I am happy with this fillet. Now, I will exit the fillet after saving it. So, you will have a fillet added to your original rectangular base 3D model. Now, I will add a step here to make it look even better.

So, I need to add a step from this face. I will select this face, right-click here, and then choose the sketch option. Now, our sketch will be activated. You can see that now I can only sketch on this plane or surface.

To create a 3D model, I have two options. I can either use my sketching tools, like the line tool and arc tool, to replicate the existing shape exactly, or I can use a feature called "Convert Entities." The "Convert Entities" tool replicates the underlying features, which can be a 3D sketch, 3D model, or anything else.

I need to have a replica of this corner. So, I will select this surface and then choose the "Convert Entity" option. Now, you can see that the exact borders of our base layer are added to the sketch. Next, I need to create a step with a width of around 2 mm.

You can either draw the same shape and give it the dimensions or since it's uniform, I can offset this sketch. To offset it, I will use another tool called "Offset Entities." Select the "Offset Entities" tool, click on this shape, and since the "Select Chain" option is enabled, it will select the entire shape. If it's not enabled, it will only select the individual parts.

I will keep "Select Chain" enabled so that the entire shape is selected. Now, I need the offset to be around 2 mm, or let's say 2.5 mm, and I need it to be inside this base rectangle. So, I will select "Reverse." Now, you can see that the rectangle is offset inside our base rectangle. I will save it, so now we have a closed figure with two rectangles or a rectangle with filleted edges.

Next, we will extrude this shape. Go to Features, select "Extrude Base," and SolidWorks will automatically detect the area to extrude. You can see the preview here. But you might wonder why SolidWorks isn't extruding both shapes, even though there is one larger rectangle and one smaller rectangle.

You can do that too. To extrude both areas, use the "Select Contours" option and choose the additional area. Now, only the selected area will be extruded. If you need to extrude both areas, you can simply click on them. Now, you can see that both areas have been extruded into 3D shapes.

However, I only need this area to be extruded. So, I will cancel the other selections and proceed with extruding this area only. I will give it a height of 2.5 mm.

Press OK. Now, you can see a keychain-like shape. For a keychain, I need to add a hole here to attach a metal ring. I will do this on the same plane. You can select whichever plane you want, but I will select this one. I will right-click here, select this plane, and make a hole with a diameter of around 4 mm.

Not fully defining it, just exit the small dimension by pressing escape. I am placing this circular thing here. Then in the future, I am going to extrude cut it. Extrude base means it has built up the object. Extrude cut means it will remove the object or remove some part, like removing a shape from our base shape. So I am clicking extrude cut, and I am giving a blind depth of around 10 mm or more than 5 mm or 3 mm, which is our base depth.

So since our base depth is 5 mm, I am giving here an extrude cut depth of 5 mm. So you can see a hole here. Now, to make it a little bit better looking, I am using a fillet option. Then I am just giving a fillet of around 1 mm here and then a fillet of around this corner, this corner, this corner. I am giving a fillet of around 0.5 mm and then just press ok. So now we created a 3D printable keychain. This is a 3D file that you can save in different formats, like the base format or the normal format in SolidWorks, which is a solid part or part file. So you can save the part file by file, save it as I will go to my desktop. Now I am creating a new folder in ptl.io. Now here, I will rename it as a keychain, and solid part is the usual file format that we will be using for saving SolidWorks files.

So I am clicking save. So now this format is saved. So if you want to 3D print it, now or then, you need to save it in some kind of mesh file. You have mesh files like 3MF or you have mesh files like STL. Usually, we will be saving this file in STL format. So for saving this in STL format, you can again go to file, save as then here you can select which type you need to save our file as.

So just click on solid parts, you will be having STL here. So just select STL. So earlier, it was not 3MF, it was 3MF. So STL then click save. So now you will be having a preview like this.

This is called a mesh file. So then just check yes. So now this file is saved as an STL file as well as a SolidWorks solid part file. So this is how you will be creating a basic shape, a basic part in SolidWorks. Now we will be seeing how to create an assembly model in SolidWorks. For example, now I need to give a cap to this keychain. What will I do? I need to make an extra cap so I can put this 3D-printed file into that cap, and I can close it like this.

So we will now be making a cap for this thing. So for that, I will head to file, new, then I will create an assembly, a new assembly here, and then just click ok. So you will have a new window here. You will have a new window exactly like the part file, but here you will have some options like assembly, layout, sketch, and all. So first you need to head to assembly.

Then you need to insert the earlier-made keychain into this thing. To do that, just click Insert. Here, you will have the keychain option because we have just opened it. It's already opened in SolidWorks. Otherwise, you can use the browse option, and then you can head to whichever folder you have saved this thing in, and then just import the solid part file.

So just click on this keychain, and then just click open. Place it here. Place it here. So now you will be having that, sorry, assembly. So place it here. Now you will have that keychain in your assembly window.

So I will be making a small cover for this keychain. So to make that cover, I need to have a new part. So to make a new part, you can go to this insert component option, press this arrow, and then just press the new part. So here, I am going to rename it as a keychain cover.

Right-click, just press here. Right-click here. Then rename the part. Rename part, I am going to rename it as a keychain cover. After renaming it, press enter. Then you need to edit; you need to make a new part on this keychain cover part.

So just right-click here again. Then you will have tons of options here. Just go to the edit part option. So press this edit part button. Now whatever you are making, that will automatically go to this keychain cover part.

So I need to make a cover here. To make a cover here, I will start with this face. So I need to make a cover that will come till this part. So I will select this part, and press sketch with this plane. So you can always press the space key to change your orientation. So once you press the space key, you will have an orientation view like this, and just head to whatever place you need to go, whatever view you need to go, and then just press this kind of window.

So you will be having an orientation of that plane exclusively. So you will be having something kind of this, and you have already selected which plane you need to start your sketch from. So before starting the sketch, I need to check the thickness from this plane to this plane. So how will we do that? The smart dimension is actually used to give some dimensions, but if I need to measure the dimension from this plane to this plane, then I can go to the evaluate window, and then there is an option called a measure. So select this measure option, then select this plane, and then just select this plane. Once you select both of these planes, you can see the distance here, which is 7.5 mm, or else you can go to, like here is the measure dialogue box, then you can see the distance here too, which is also 7.5 mm. So I know the maximum height is 7.5 mm. So now I will start sketching. As earlier, I will give an offset from this set. So it will be like selecting this face, going to sketch, offset entities, and then I will give an offset to this place.

I need to reverse it, but I need to only give an offset of around 0.25 or 0.2 mm. So you might be thinking why I should give an offset instead I can just convert entities, which means converting the same face, because since we are 3D printing it, we need to give a small offset, then only our two parts will mate with each other. So the offset depends on which kind of printer you are using.

Generally, you can give an offset in the range of 0.1 mm to 0.3 mm depending on which printer you have seen. Currently, I am giving an offset of 0.2 mm so it will easily go in and it can easily come out. So I will select this face, then I will select this edge, then this edge. Likewise, I will be selecting all of my edges. Now it has become a closed shape, but it is just like once I select other edges, it just reverses direction, so I just unclick this reverse dialogue box so you can see that it goes just out of my shape.

So now I press ok. Now I need to have a vault. So to have a vault again you need to offset it for 2 mm now I am giving a vault thickness of 2 mm so again click this offset button click on this button and then just select the chain button but I need to give an offset of about 2 mm so instead of 0.2 mm to give here 2 mm.

Then just press ok. So now you will be having a vault kind of structure here. So now I need to extrude it to 7.5 mm. But it extrudes in the opposite direction. To change the direction you can have a change direction arrow here or a reverse direction arrow here. Just select this one so it will reverse the direction.

So this is how it is now just a vault. Now I need to have a top cap kind of thing. The top cap needs to be on this face. So I will just select this plane, and then select the sketch option. Then, exactly like I need to have a proper copy or proper replica of the outer edge. So I select this face completely, click on or else click on "convert entities," then I can select edge by edge. You can select either of them. You can just select these faces, so it will convert the outermost entity. So you can select whatever edges if you need this edge. If I need the inner edge to be converted, you can just select this inner edge, then I can just click OK. So you can see how this edge, like the outer edge, and you can see a line here.

Now I don't need it, so I am just going to trim it. Or you can even delete that thing too. So now you have a shape here. I am going to extrude it again. I am going to extrude it for around 2 mm. So now you have a cap for your earlier-built keychain. I just did this to show how the assembly part can be made. This is a simple one, but SolidWorks is so powerful that you can create a highly complicated assembly feature inside this assembly part, assembly window, or assembly option, and then you can check whether this thing works exactly as we planned.

So then only you need to go for manufacturing. That’s a major advantage of this assembly window. You can make different parts; you can bring those different parts to our assembly window. You can mate each other using the mate tool, which means you can give the relationship towards each surface, and then you can check whether this thing works as we planned or not. This means if it’s having some kind of interference in some phases, then you can modify it and make it work perfectly as we planned. This is a simple example, and now, to make it a little bit more beautiful, I am giving a small fillet here.

Just one more fillet here. So now you have a complete keychain and a cover for that keychain. This just looks like a box, but yeah. So if you need to save this shape or if you need to open this part as an only part file, now it’s in an assembly file. So if you need to save this as a part file only, just open this as a part file, and if you need to edit it, just right-click here, and then you have an open part option. So just click on the open part. So now you will only have that part file.

Now here, you can also edit it, whatever you want, and that will reflect on your assembly file too. So I will show that too. So if I need to have an array of holes here, I will just select this face, and then I will put a small hole here.

I will give a dimension of around 2 mm. So let this hole be here. Now I need to extrude it. Sorry, I need to extrude and cut it. So this is how it will be. Suppose, imagine if I need to give this hole through this entire line, then fix a distance of 2 mm. What should I do? I just need to make each circle as it is, but it's a time-consuming process. So SolidWorks has an option called a linear pattern. So you can select this linear pattern option, and then first, you need to select which direction you need to have this hole pattern.

So I will be selecting this line as the direction. So you can see this arrow. Then, if I go to features and select this option, you can see now I have two holes, but I need to have at least two holes 10 mm apart, and I need to have holes 5 mm apart. So I will put 5 here, and then I can increase the number of holes. So I will be having around 10 holes here, and suppose I need to have the same thing above also.

So I will just select this direction 2 option here, and then in this direction, I want. So I will just select this key here, and then I will have one more, around 5 mm apart. So I will have one more. Then you can see the preview. If you are satisfied or okay with the preview, then press okay.

So now you will have a hole that is patterned. We only made one hole, and then we patterned this many holes. So that's why the pattern is useful. Then just click Ctrl+S because it's already saved in the assembly. Then you can head back towards the assembly option here.

You can see the changes that we have done there actually reflected here also. So now we are going to save this assembly. To save this assembly, go to file, save as, and here we already created an option keychain cover.

Now it will be saved as a solid assembly format. Now click save. Just click this option. Now this assembly is entirely saved. But the assembly is saved as a solid assembly file. If we need to 3D print it, we need to save it in an STL format. Either you can save the assembly as an STL format, which will create two separate files of these two separate parts and save them, or else you can save this part individually.

You can just open this part, then head to file, save as. Here we already saved the keychain that we already created. Now we need to save this as an STL file. So go to save as type, click here, click STL, and then click save.

Then click yes. So now this thing, the cover, is also actually saved as an STL part. So now we need to 3D print it. We already discussed that to 3D print something, first, we will make a 3D model. That can be either by making a model from scratch using some 3D modeling software like SolidWorks, AutoCAD, or even Blender, or anything. Then we need to convert it into a mesh file.

Currently, we are using the STL file format. Then we will use it, and then we will slice it. Only then can we send that file to our 3D printer. It can either be online, or it can either be a pen drive or any means depending on which printer we are using. So currently, we are using the Form 3BL 3D printer. So we will be using the slicing software of Formlabs.

So for that, we can just open software known as PreForm. PreForm is the slicing software that we use for slicing STL files for Formlabs printers. So we will be using PreForm files and PreForm software. It's the official Formlabs slicing software. So just wait for it to operate. So this is the user UI for Formlabs slicing software.

Once you go to this window, you can select which printer, and which type of printer you are using for Formlabs. There are different printers like Form 2, Form 3, Form 3L, 3BL, and Fuse 1. We are using the Form 3BL printer, and since Form 3BL is an SLA printer, we have a different variety of material choices.

So here we will be using a grey air asymptote printer. So just choose the material that is loaded onto your printer. Now we are using grey material. Each material has its different properties. So we select grey material and then just click apply. So you will have your printer and the material that is loaded into that printer on your slicing software itself.

Now you need to import the model. To import the model, go to file, click open, and I have saved my STL files in this folder. So you can see only STL files have been detected by our printer, and here are the options you can select STL OBJ or even Form file. The form file is a file that is saved from the PreForm software itself. Since we have saved it in the STL file, we are going to select these two things.

So just select both, then click open. So you will have both of these files here. But you can't print this as it is. You need to orient this. You need to arrange the orientation and give support. Since it is an SLA printer, we already discussed that SLA printers need support. So since this is an SLA printer, we need to orient this thing in the best format or best way possible, and then we need to generate support. So to orient it, you have different options here. So this is, if you need to scale this model, if you need to scale this model like now it is in a particular dimension if I need to double the size, then I can put two here.

So it will increase its size in the X, Y, and Z direction by a factor of two. But now I am going to revert this change. I just need it to be in the usual size. Then you have an orientation option. So orientation means you need to orient this thing in a particular format. So you will get the best print quality out of that printer.

So now I am going to make this like this and we know that support will be built from this surface. If I make our part like this, if I orient our part like this, then supports will be attached to this surface. So if the supports are here, then we know that once we remove the support, our particular face will not look that good. So we will try to arrange the model in such a way that support will come to the least visible faces.

So I will make it like this. So I am happy with this orientation for the cover and the keychain, I will make an orientation like this. So this is also okay. So the support will be coming in base structure. Now I will generate the support. For that, you need to go to this menu and then auto-generate the surface.

So I am waiting for the support to be generated. So now you can see how our part will be printed. So this is how our supports will be generated or this is how exactly our part will be printed using our software. And in each software, each licensed software, the workflow will be different in a little bit manner. But this is the generic form of how you need to do an SLA 3D printing. So I am happy with this.

Suppose I need to make two parts of this thing, then what will I do? Do I need to import the same model and give an orientation and everything? No. I can select this model, head towards this layout option and see how many duplicates I need to create. There is an option for me can create how many duplicates of this shape I want. Now I need to make one more duplicate.

So I just select this thing and then just click the create option. So you can see one more duplicate is made with the exact supports and everything. If I need to delete it, just click here. You can hide it or else just click here and press the delete option. So it will be gone.

So now we created everything and then this is an online printer. Once you generate, and slice everything, you can just pass this file to our printer online itself. You do not need to copy this thing to a pen drive and plug that pen drive into our printer. We can just pass this or we can just send this thing automatically to our printer.

For that, you can go to this option. Press start a print and already our printer is selected. I am going to change it to a keychain with a cover. Then once you upload the job, it will slice the file and it will send this file to our printer. So this is how we will be generally doing 3D printing on an SLA printer.

So once it is uploaded, we need to go to our printer. We need to prime it. This means we need to check all the functions like everything is okay or everything is ready for printing. Then we can start the print. So those are things we will be seeing in our next lecture. We will be directly going to our printing room and then we will be having a hands-on experience of how a 3D print can be done with a real 3D printer. So those things we will be checking on our next lecture. So that's for now. Thank you.