Engineering Graphics and Design
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Week 11: Assembly
Assembly with Constraints

Welcome back to the NPTEL course on Engineering Graphics and Design. We are on week 11. In this week we will discuss how to assemble components together. So that at the end we get the machine or a device in an assembled manner.

So last few weeks we discussed about how to create the part models. We will use those part models to create what we now call it as assemblies. For creating these assemblies, we will be using constraints. And the options of how to create these constraints will be discussed with the software in this lecture.

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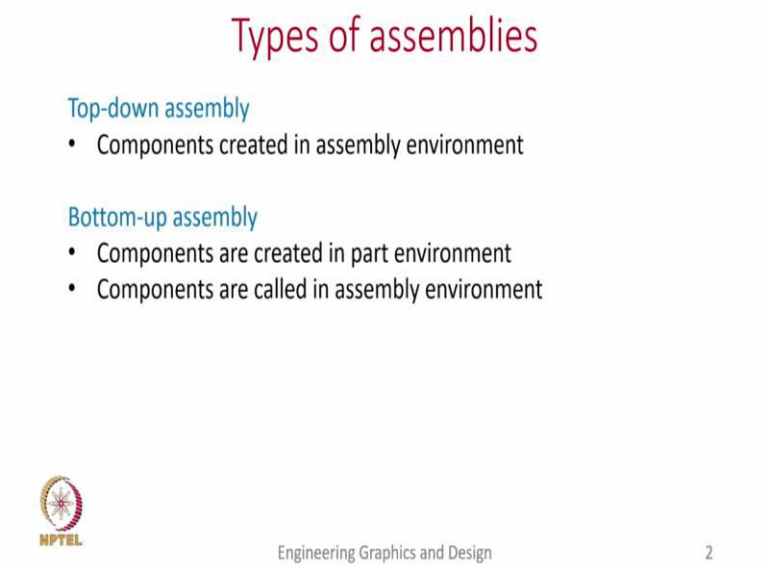

So, there are two kinds of assemblies one can create using the inventor software. One is the Top-down assembly, the other is the Bottom-up assembly. So, starting with the top-down assembly, here inventor has an assembly environment.

So, in that assembly environment you can start by creating the part models and then assemble these components to create an assembly. In the second approach which is the bottom-up assembly, you create your components in the part environment, which we have already discussed in the last few weeks. So, once you have created those components using the part environment, then we go into the assembly environment we call those parts that are already created.

So, we start with the created parts in the assembly environment and then assemble them using constraints to come up with the final device of the mission. So, there is of course a hybrid approach where you can start with a bottom-up assembly approach and later depending on the needs you can create some more components in the assembly environment itself. So, all these approaches are possible with the inventor software. So, let us see how to get started with this assembly environment.

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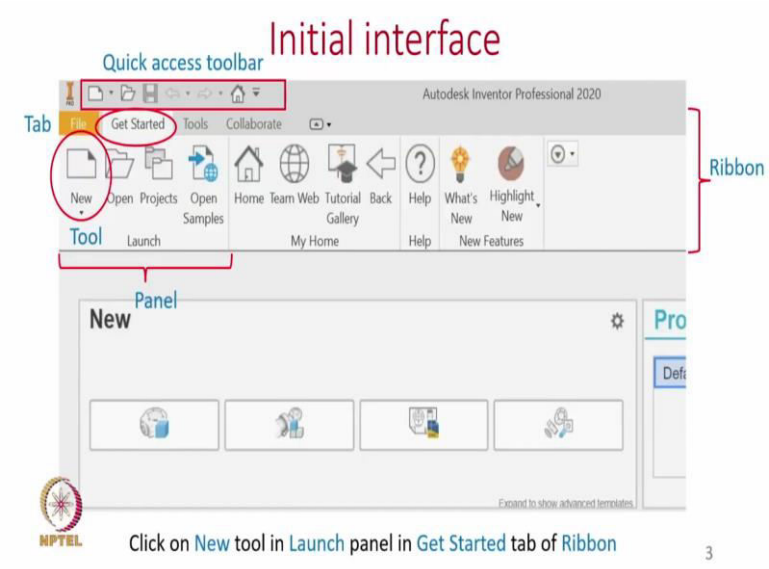

So, this is the initial interface which by now we are all familiar with. To open or to get started with the assembly environment we need to go to this ribbon then the launch panel, within the launch panel go to the new tool.

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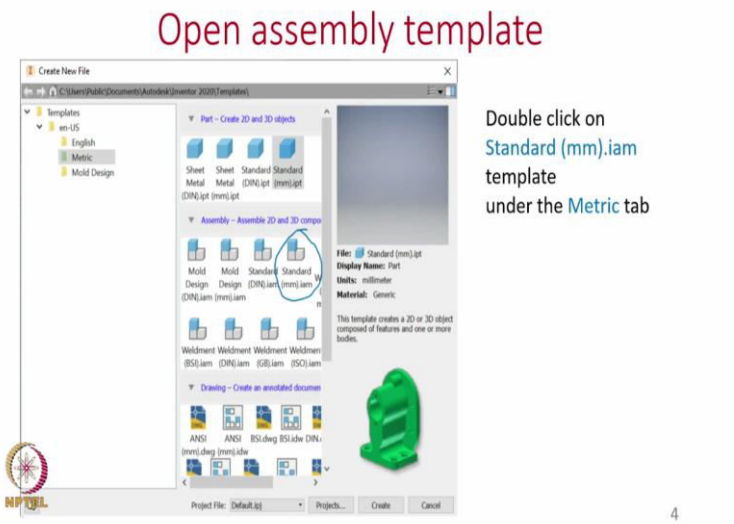

So, once you click on the new tool, we get a dialogue box, where we can select what templates, we want to start with. So, once we go to the metric templates then it suggests whether we are looking for a part model or an assembly model or drawing as well as presentation. Once you select this standard mm.iam.

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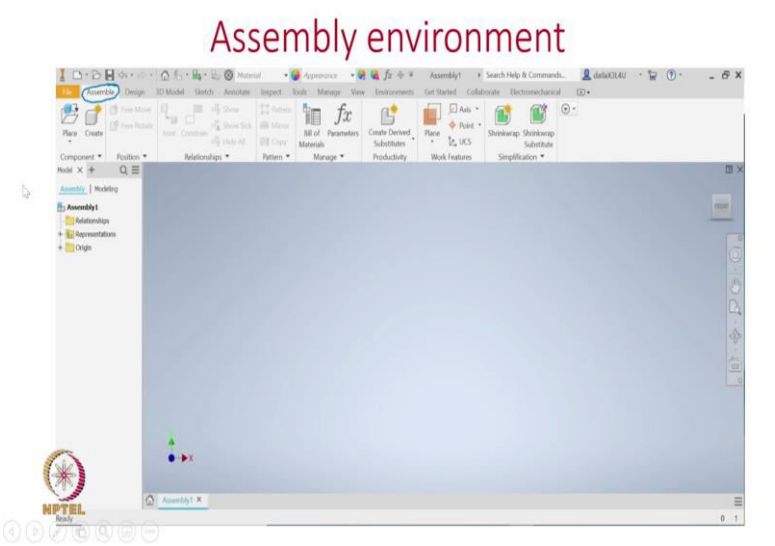

We then enter into the assembly environment. Here is the initial interface of the assembly environment where there are several tabs depending on what is our intent. By default, we start with the assembly tab where we can build an assembly from the components which we will import here from the part models that are already created. So let us get to the software and see how we can start building up assemblies.

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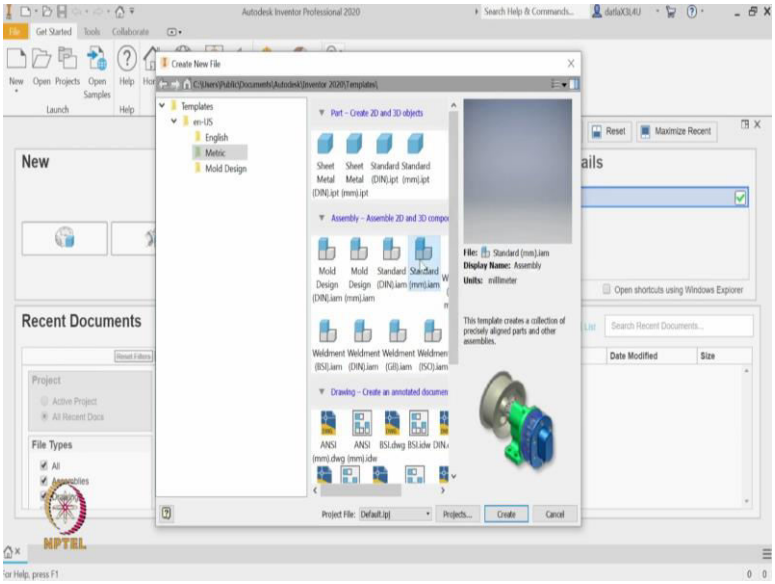

So here is the initial interface. Let us select the new template. So, we will select this metric system and we have a bunch of templates starting with the part model templates, assembly model templates, drawing and presentation templates. Since we are interested in the creating an assembly model. So let us go to these standard mm.iam, double clicking on it will take us to the assembly environment.

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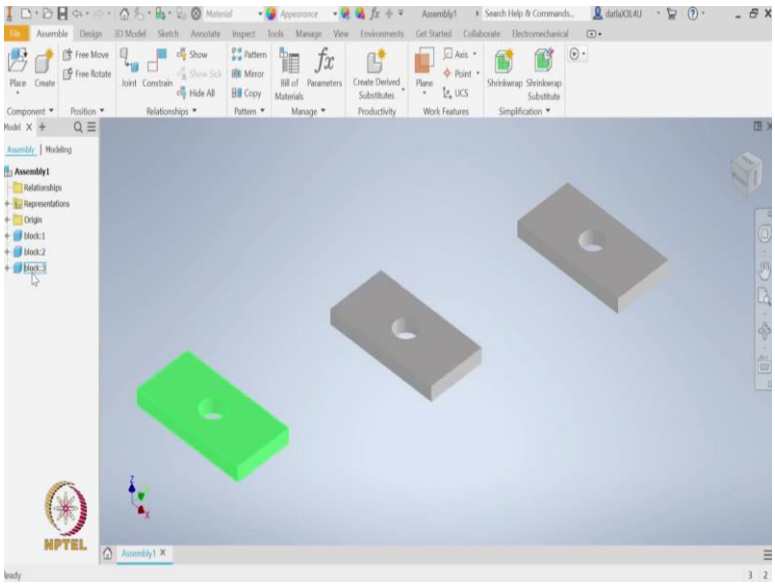

So, this is the assembly environment where there are several tabs, by default we are on the assembly tab. But there are other tabs like the design tab, 3D mode, sketch, annotate, inspect and several other tabs. Let us go back to the assemble tab. For us to start we previously said we can either go with the top-down approach where we start by creating the models.

So, for that we can click on this create and then start creating a part model in the assembly environment. But today we will be discussing more about the bottom-up approach. Because we have already created few parts in the last few weeks. So let us say if we want to start with the bottom-up approach, the first thing we need to do is bring those parts that we have already created. So, we can bring those components using this place tool. Let us left click on the place tool. Then it opens up this dialogue box where we can select a part model.

So, I will select this block. So now I can left click to place a model and I can continue to create more copies by doing more left clicks. So let us say now I will click here for the second, click one more time for the third. But let us say if I only need three copies of it, I right click and then say ok. So ideally what we are doing is we are bringing in one component and making multiple copies of it. These copies are also called as instances.

So, if you look at this browser it will say there are block first instance, block second instance and block third instance. Let us say you do not want this second and third, you can always right click and delete. But later let us say you want to create more components, you can select that component, control C, control V should give you the copy of these components.

So, it has created a second instance after I did control C and control V. And you can click onto any of these components, once it is highlighted you can move the mouse with the left button on and you can drag it to any place in this environment. So let me again delete the second instance. So currently let us say we do not need.

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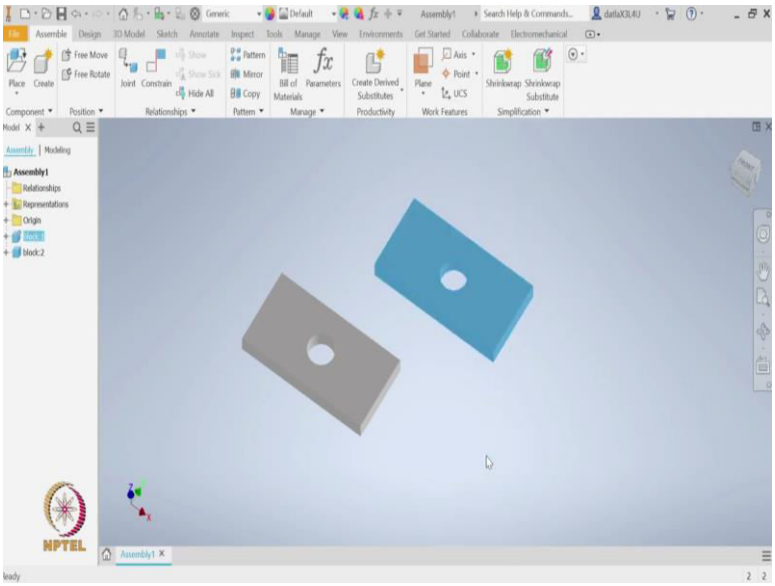

So, as we previously mentioned we use constrains to define how two components interact with each other. So here we have this first component. Now I will again do this control C and control V to create a second instance. Let me place it here. So now let us say these are two components we have. We need to assemble them together. Let us say to start with we want to place them one on top of each other. How do we do that?

So, let us select this constrain tool. This constraint tool will give us a dialogue box where it has five options on the kind of constraint you want to apply. So, first look, first let us look at what are the different kinds of constraints we can do. The first one is the mate, second is the angle, third is tangent, fourth is insert and last is symmetric or mirror. Before we get started with creating constraints it is a good practice to first ground one of these components.

So let us say we will select the first instance, right click on it and then make it grounded. What does grounded does? It constraints all the degrees of freedom for this component. So,

each component will have six degrees of freedom, three translations along the x, y, and z direction and three rotations. Rotation about x, y and z.

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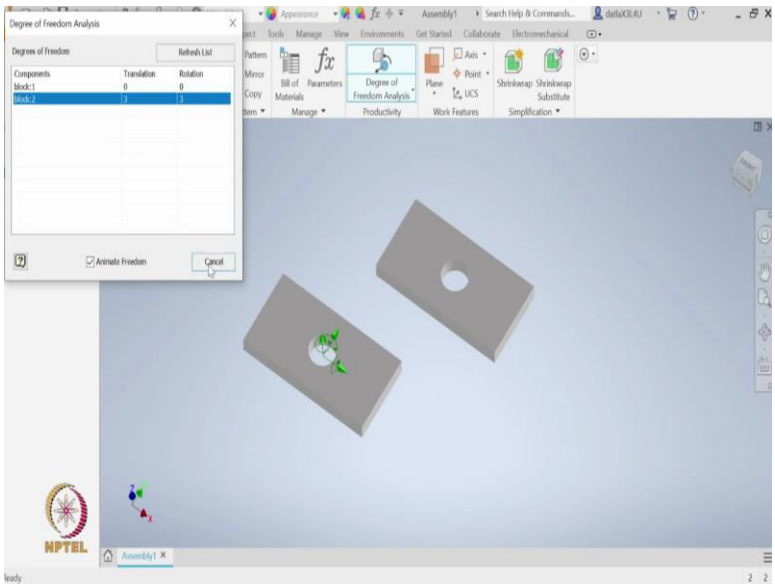

So that you can also find out using this degree of freedom analysis. So currently it is saying that you have two components, block one and block two. For block one there are zero degrees of freedom in translation and zero degrees of freedom in rotation. That is because we have just grounded it. But for block two all these degrees of freedom are open, all three in translation and all three in rotation.

We can also check what are those degrees of freedom if you go to this again analysis, select this block two and say animate degrees of freedom. So, it is now showing it can move in translate in x, y and z and then rotate in x, y and z.

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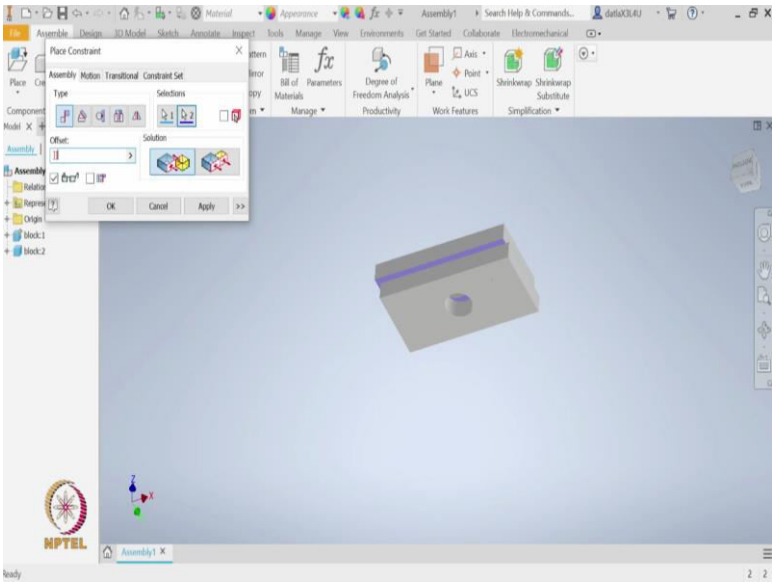

So let us get back to the constraint and now come back to the problem of how to bring these two blocks on top of each other. So, I will go use the constraint tool, I will use the first option which is mate. In mate it is asking for first selection.

I will, the first selection will be top of this block and for the second selection I will select the bottom of the second block, bottom face of the second block. Now you see that they are, both the surfaces are in a single plane. But then you have two options, whether both of them are mate or flush.

Mate means, these two surfaces are facing each other. Whereas in flush both these blocks are in one side of the common frame. So let us select the first option which is mate, which is what we require.

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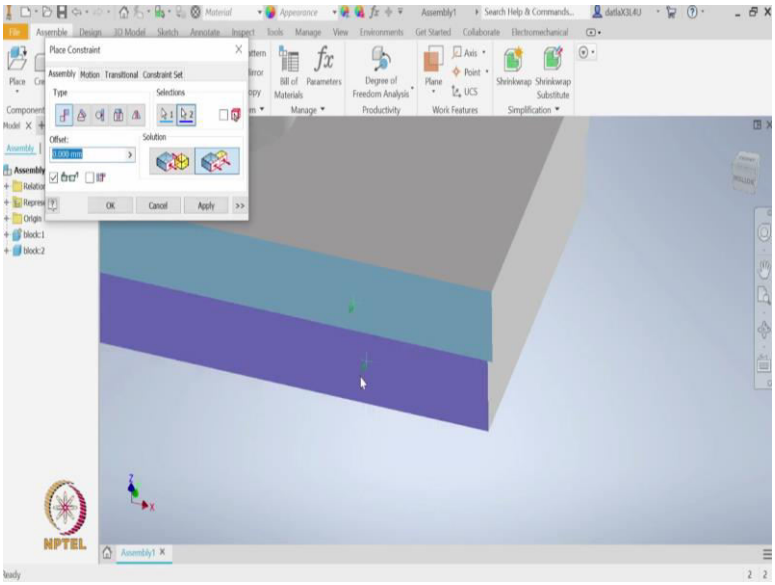

After we apply this mate constraint, we see that both these components are now sitting on top of each other. But are they properly aligned? No.So, let us zoom in and see that this face is not properly aligned with the other face. For that how do we do that? We again go to the constraint tool and this time I will take flush between the face of this top component and the face of the bottom component. Now both of them are flush, which means they both are on the one side of the common plane. So let us apply this flush.

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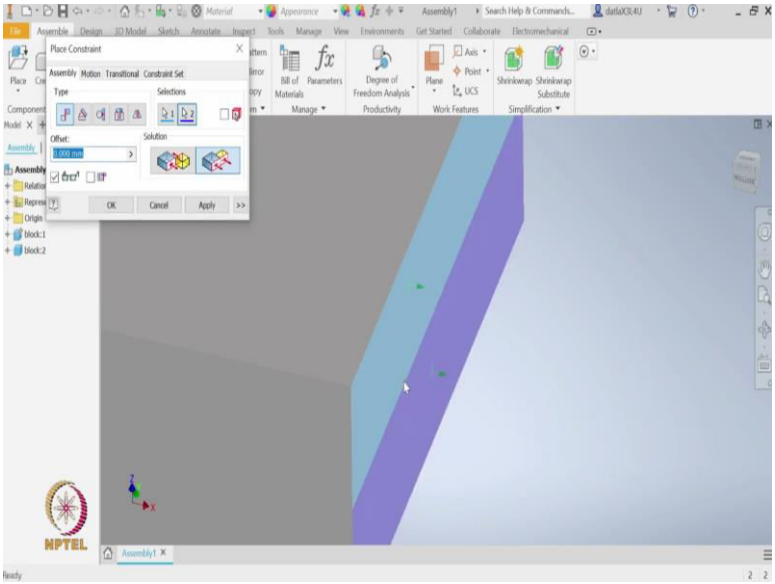

Now, we see that both of them are not exactly matching. So, this face and this face are in different planes. How do we bring them together? Again, we can use this flush option. I got

to the flush option and select this phase from the top component and then selection two is the, this face from the bottom component.

And now both of them are flush. Which means both of them are in the same plane. Now we again need to apply to finish the operation. So now we see this, both these blocks are perfectly aligned and sitting on top of each other.

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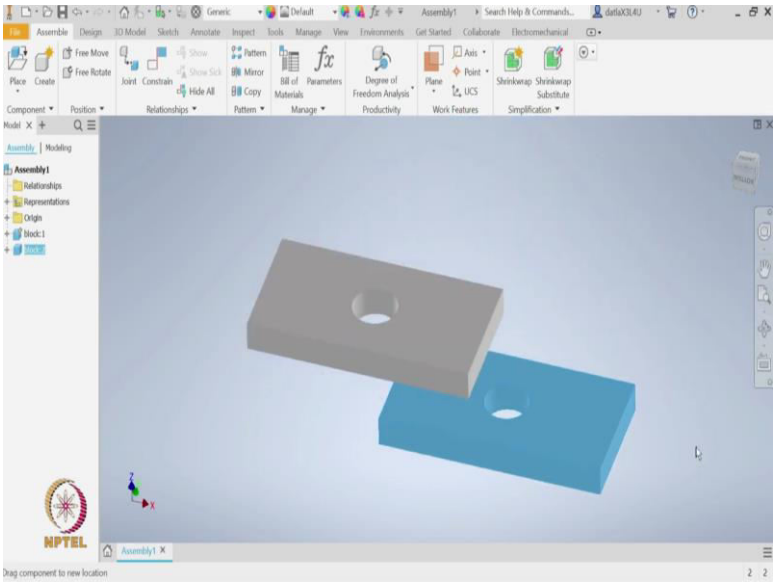

So, after this now let us see the second option. Within constraint the second option is this, angle. Before I apply the angle, I can go back and delete the previous constraints we have defined.

So, for this I will go to these relationships. There I see the constraints we have applied, first is the mate and then flush one and flush two. So now what I will do is I will select each of those. I will select these constraints and delete them. So, since I have already deleted those constraints, now I can move these parts, they are free to move. So, the second component again got back its six degrees of freedom.

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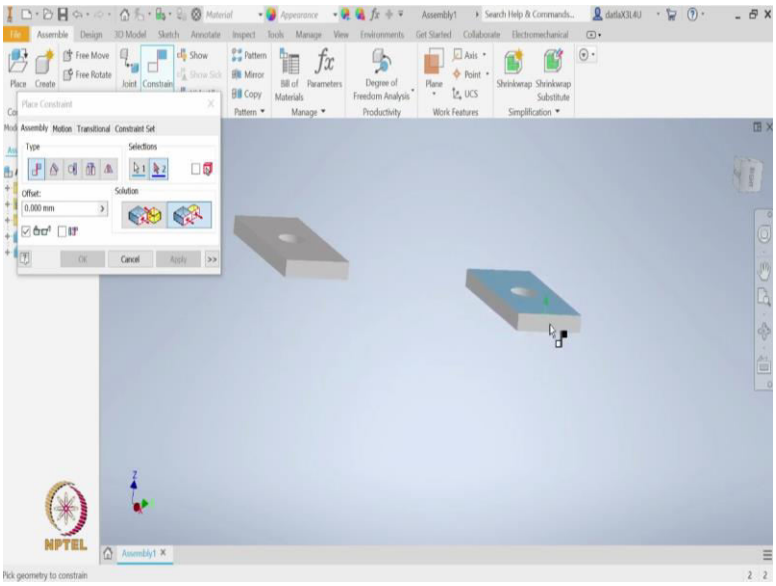

Now we will look into the second type of the constraint, which is the angle constraint. Before I apply the angle constraint let me make sure both these top faces are lying in the same plane. For that I can do with the flush option.

I will start with the mate type and then select top face of object one and top face of object two. And then apply this flush. So that both of them are on one side of the common plane and apply.

So, let me move the second component away. But now probably if I rotate you can see that even if I move, they are still in the same plane. So, probably if I look at the right side view it will be more clear. I can only move it in this plane but not out of the plane.

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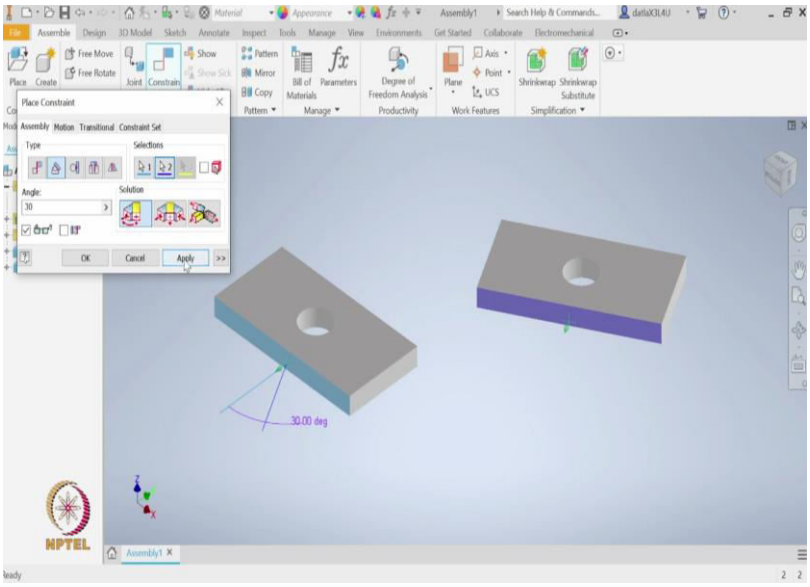

Now let us apply the angle constraint. For this we need to select the second type which is the angle. And I will select this first option which is the directed angle. For this we need to make two selections, the first selection is this face here and the second selection is this face here. By default, the angle is zero degrees. But let us say now we change it to 30. So, now it is showing me a preview of what happens if I apply this angle of 30. If this is what we want we can simply apply and then we have applied this angle constraint.

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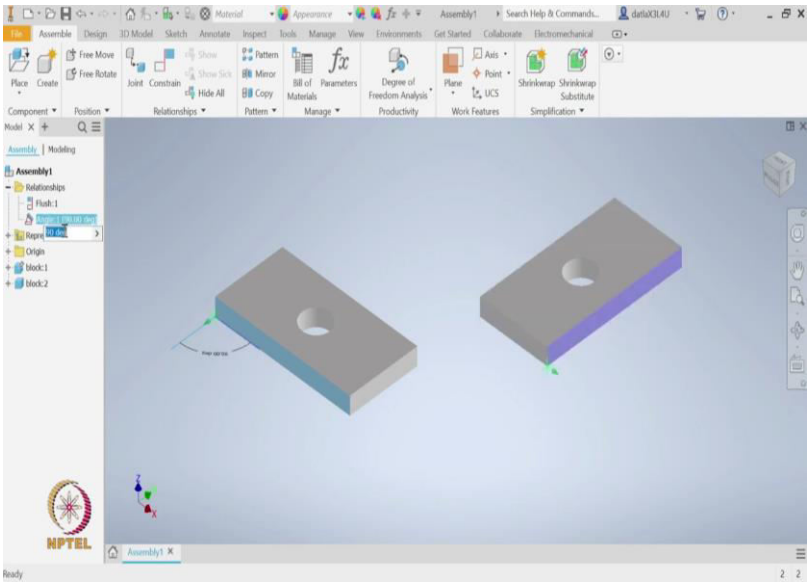

But now let say we want to change this angle. So, instead of 30 you want 90 degrees. So, you can always go back to relation, click on it and then it gives you an option to change. So, instead of 30 let me change it to 90 degrees.

So, once we have updated with 90 both these selected faces, we see are at an angle of 90 degrees to each other. Now I can always again use the flush to bring them close to each other. So, let us say this face and that this face I can flush, apply.

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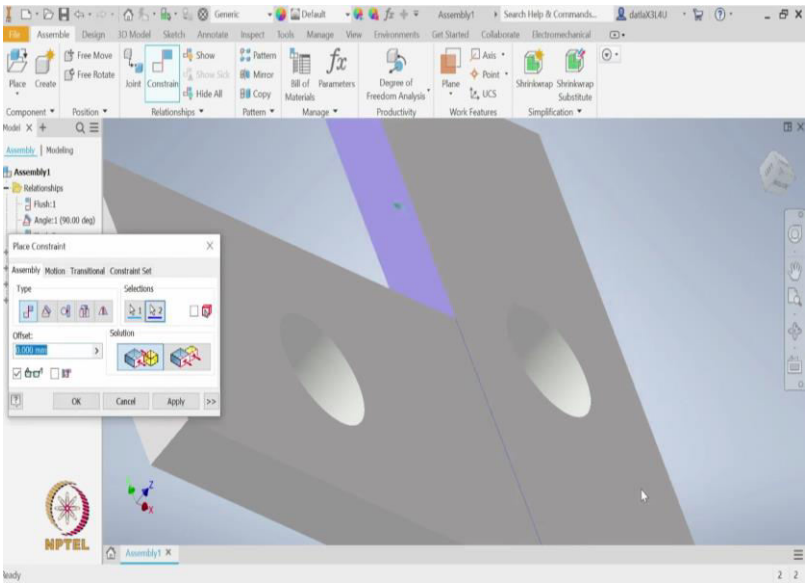

And then let me see is if there is any interference between them or not. So, we are not sure whether they are touching each other or not. So, for that let us apply this constraint and option mate. So, we will pick this face and the corresponding face on the other side. Now they both are joining each other without any gap. So now I can apply this mate constraint. These are options one and two where we have showed you how to use options of mate as well as apply the angle constraint.

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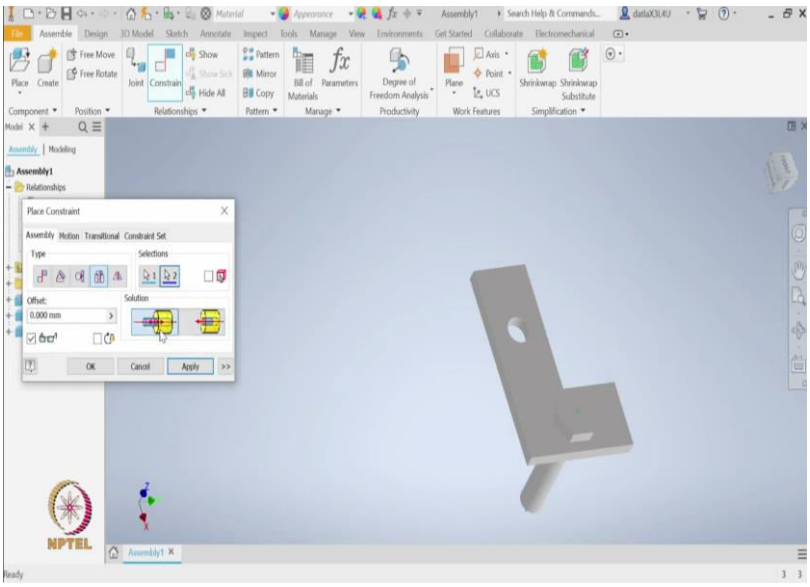

Now let us look at other options. So, for example we can start with this insert. Before I apply this insert let me import another component that we have created. For this I will go to the place tool and select this bolt. Again, let me place it here, right click and okay.

So, now let me go back to the constraint tool. We said we will go with this insert type. For this we need to apply these two selections. First selection let us take the bolt and the second selection let us take this hole here. So, the moment we do that then it understands that these are the two objects which needs to be inserted one inside the other.

So, here we have two options, either we can say opposed or aligned. So, currently it is in the opposed option. Now let us see what happens in the aligned option. Now we see the orientation of the bolt is different. So, if this is what you want you can leave it here. But for the moment let us select the opposed.

So, essentially it is looking at the axis of the bolt and the axis of the hole and now looking whether these two axes should be aligned in the same direction or in the opposite directions. So, let us apply this constraint and then we are done.

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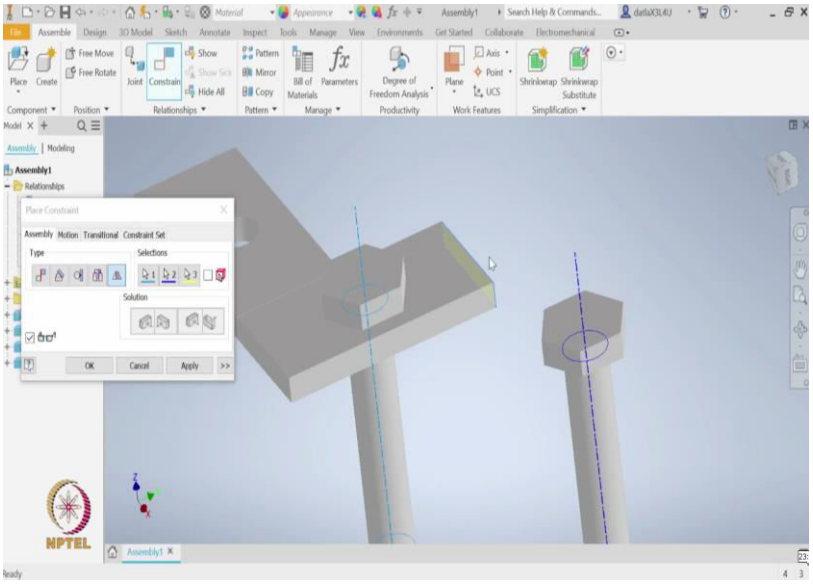

So, similarly there are two other options we have, like. So, let us now look at the symmetry constraint. So, let me see how we can apply this symmetry constraint here. Before I place the symmetry constraint let me create one more copy of the bolt. So, let us say I will take control C and control V and the second bolt I can, let me place it here. So, now let us use this symmetry. Now what I will do is first I will select the first bolt and then the second bolt.

Now both these bolts I want to place them in such a way that they are symmetric to a plane. For that I need to select a plane. Let me select this plane. So, the moment I select it, it has understood that. The distance between the second bolt from the plane is exactly same as the distance between the first bolt from the plane.

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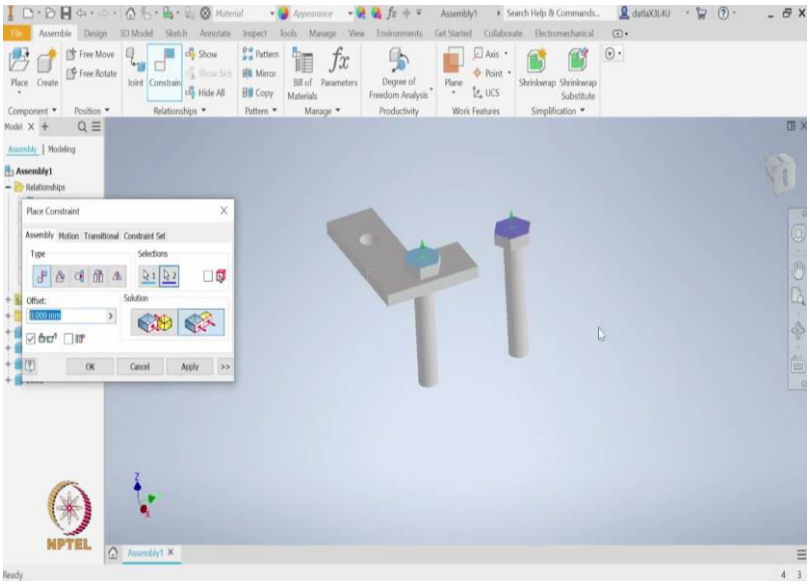

Now we notice that these both bolts are not in the same vertical position. So, for that we can simply do that by going to constraint and apply flush between the top face of first bolt and top face of the second bolt.

Now both of them are in the single plane. And that symmetry had made sure that both these bolts are symmetric about this common plane that we have selected previously. Apply, cancel.

Let me go back to bolt two and see if there are any constraints. So, let me delete these and flush as well. Now I can always inspect the degrees of freedom. Now I see for bolt two both the translation and rotation, all the degrees of freedom are available.

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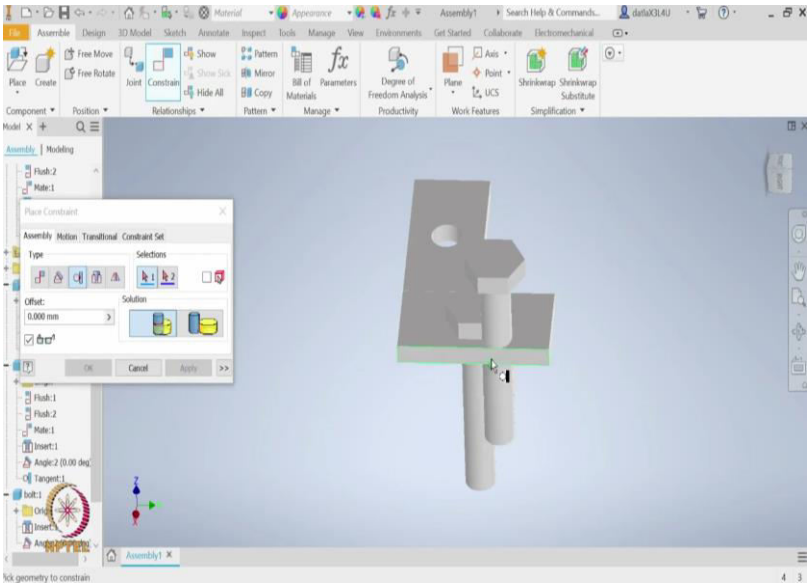

So let me go to the constraint and see if we can apply this tangent. What this tangent does it ensures that the two surfaces we have selected are tangential to each other. So first let me select this circular surface here. And maybe for the second selection I will select this surface. Let me zoom in and select this surface. Now we see that both these surfaces are tangential to each other.

Here too there are two options within solution. The first one is the inside and the other is outside. Depending on which one you need you can choose either of these tangential constraints. So, let me say inside and apply. So, now if I zoom in, I can see at this line for example, both these surfaces, the circular surface of the bolt as well the planar surface of this block, both of them are tangential to each other.

So, let me move this slightly away. So, these are the several constraints one can apply. But if you look closely there are other options like motion, translation and constraint set. It will be helpful to animate the device. But for the current lecture we will focus on the assembly tool.

So, now once we are done with the assembly, let me delete this bolt. So that let us say that we have only three components here, two blocks and one bolt. And let us say we want to get the drawings of this assembly.

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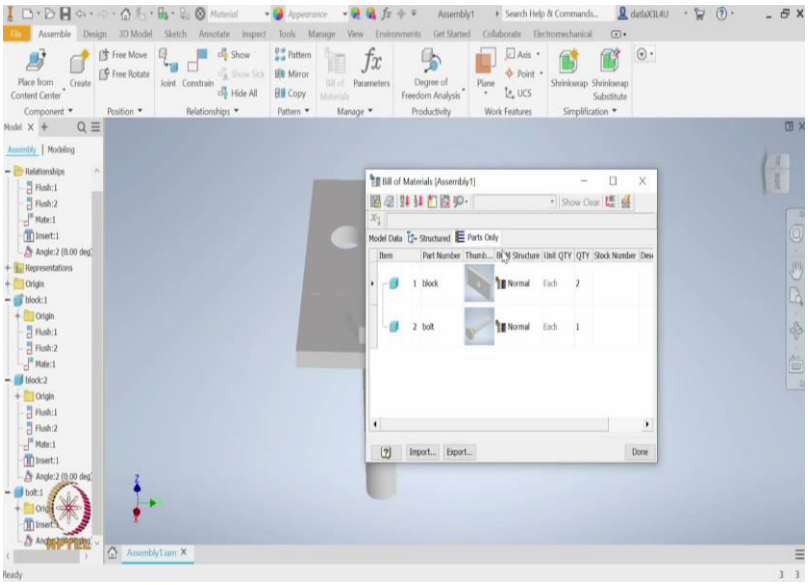

Before we get there let us once look at the bill of materials. We need to select this tool here. And in this dialogue box if you right click here, you can either enable or disable the bill of materials. Let us keep it in the enabled, both for the structured as well as the parts. Right click and enable and let us say done.

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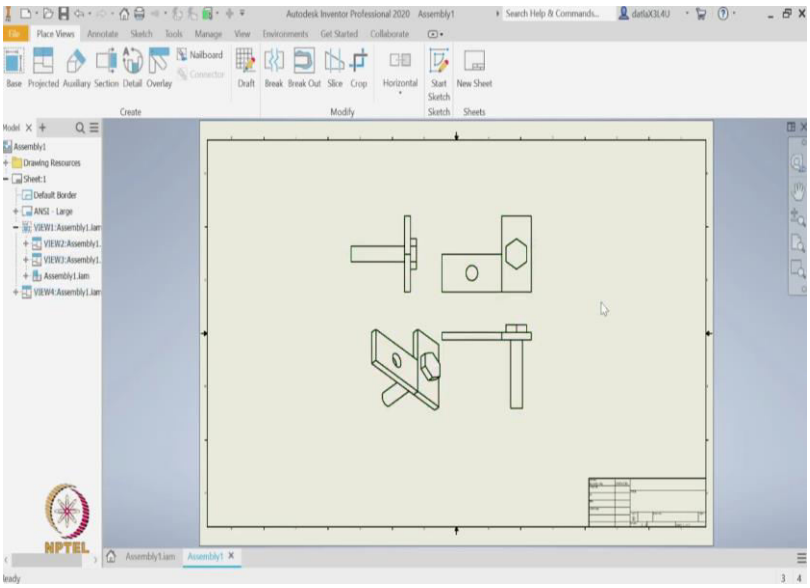

Now let us save it and now open a template for the assembly drawing. So, it is the same as the previous drawings we have looked at. So, we go to these ANSI (mm).idw. So, once we are in this drawing environment, to get the drawings it is the same procedure we used with the part models.

So, first we need to go to the base. By default, it is giving me the assemblies since it is already opened. Otherwise, I can browse it and choose the assembly. So, once I choose the assembly let us say I want to place it here. Choose the scale lightly bigger. Let us say 2 : 1 and select okay.

See it has created an assembly view of it. We can as well create more views of this assembly by first selecting the base and let us say two more views. Right click and create. So, let us select all of these and move them closer. Maybe let us place it at the middle. now for us to place the bill of materials we need to go to annotate and then go to this part list. So, it asks us to select the assembly. We will go and select the base view and then click ok.

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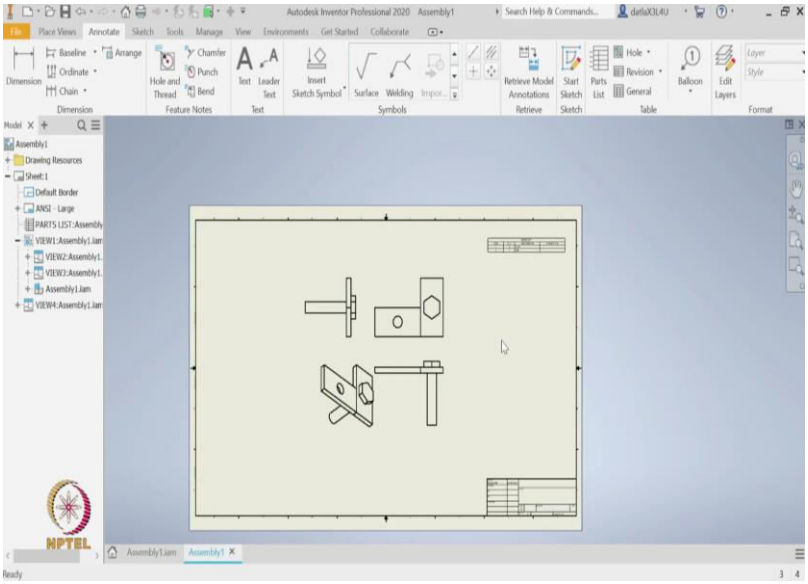

Now it is asking where to place the bill of materials. Let us say you want to place it in the top right corner. So, let me zoom in and see. So, it has two rows, one is the item one and two. For the item one which is the part number we named it as block, it has two quantities. So, this is how you need to place the bill of materials in the assembly drawing.

So, in summary what we have looked in today's lecture is, how to get started with the assembly environment and then in the assembly environment we said there are two approaches, either we can go with the top-down approach or the bottom-up approach. In this lecture mostly we followed the bottom-up approach where the components are already created in the part environments.

So, what we do is start with an assembly environment and bring those part models as components into the assembly. So, once we have this part models, we apply these constraints

such that we can assemble these components together in a specified manner. And once we are done with the assembly, we can open up a drawing template where we can capture the views as well as the dimensions and also the bill of materials which is required.

With this let us conclude the lecture and we will meet again in the next lecture where we will take a problem and then assemble it using constraints as well as get to the drawings. Thank you for your attention.