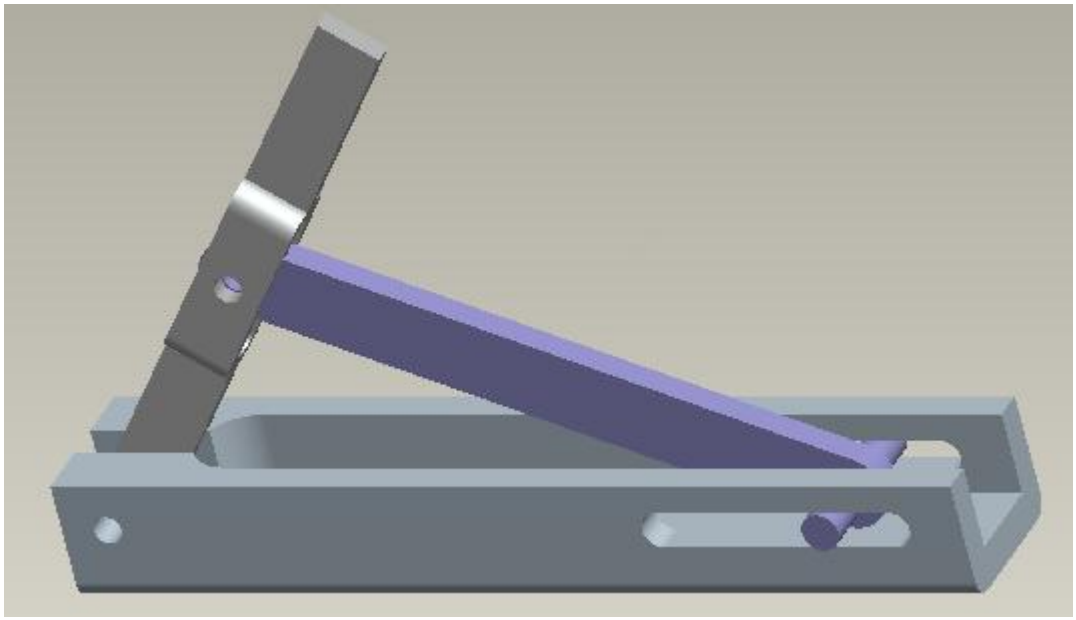
Exercise 3

In this exercise we will create mechanism connections for an assembly that is designed using the Standard Skeleton model. We will also see how to simulate a mechanism in the sketcher.

Set the working directory to CRANK_SLIDER folder and open the assembly **CRANK_SLIDER.ASM**

It will appear as shown below.

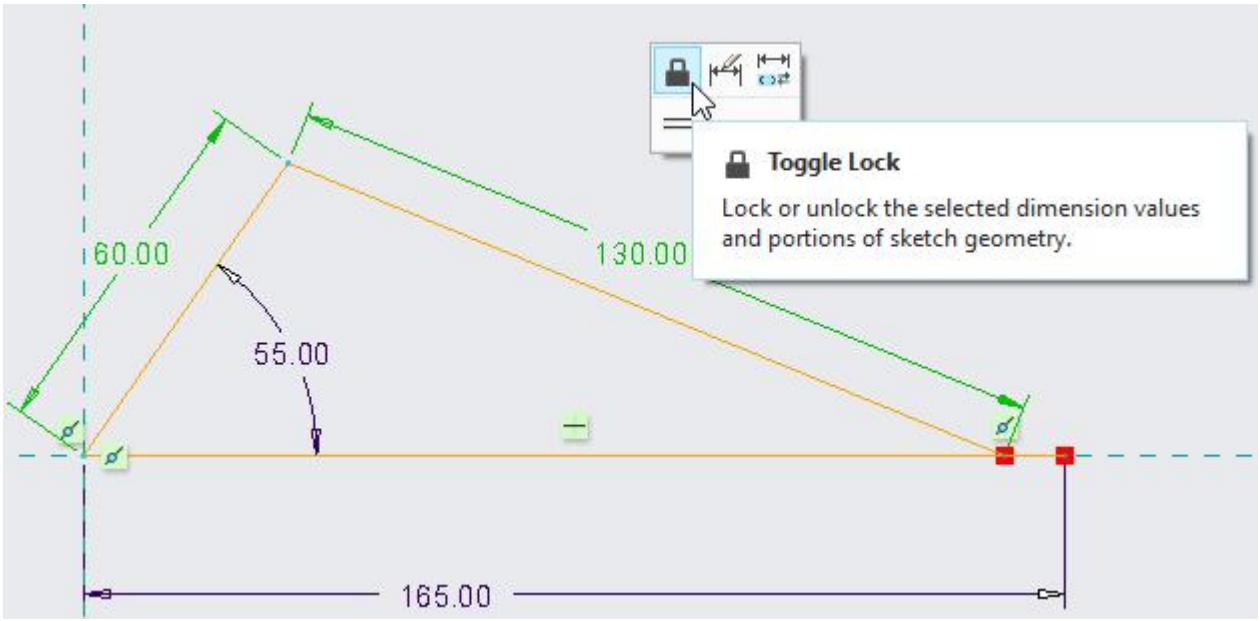


All the components have been placed in the assembly by using the Default constraint.

Open the skeleton model in a new window.

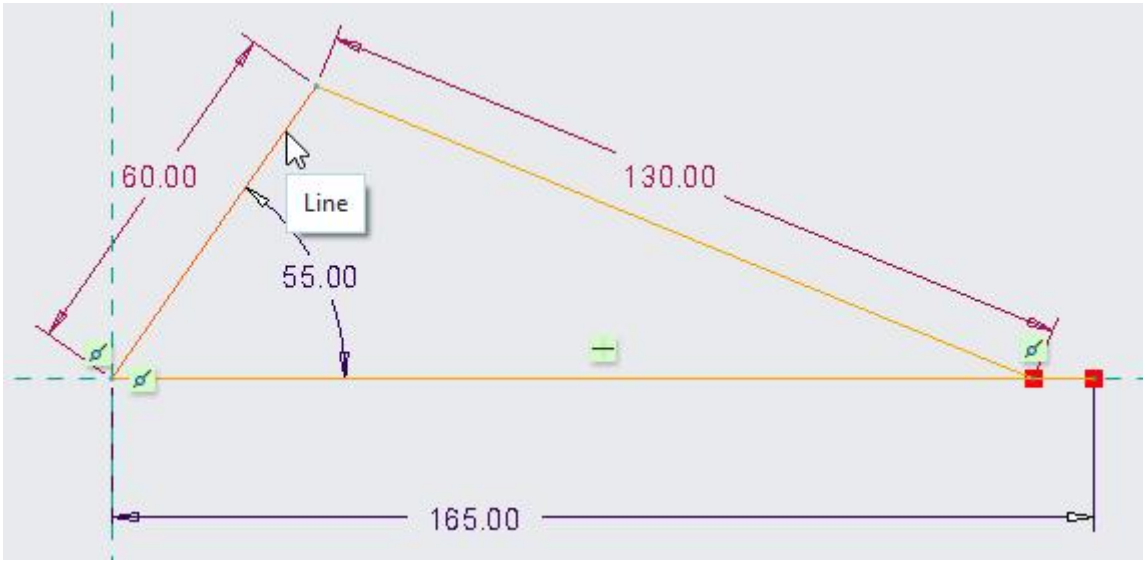
Select the “Sketch 1” and pick  and sketch will appear as shown below.

Select the “60” and “130” dimension values and pick **Lock** in the short-cut menu.

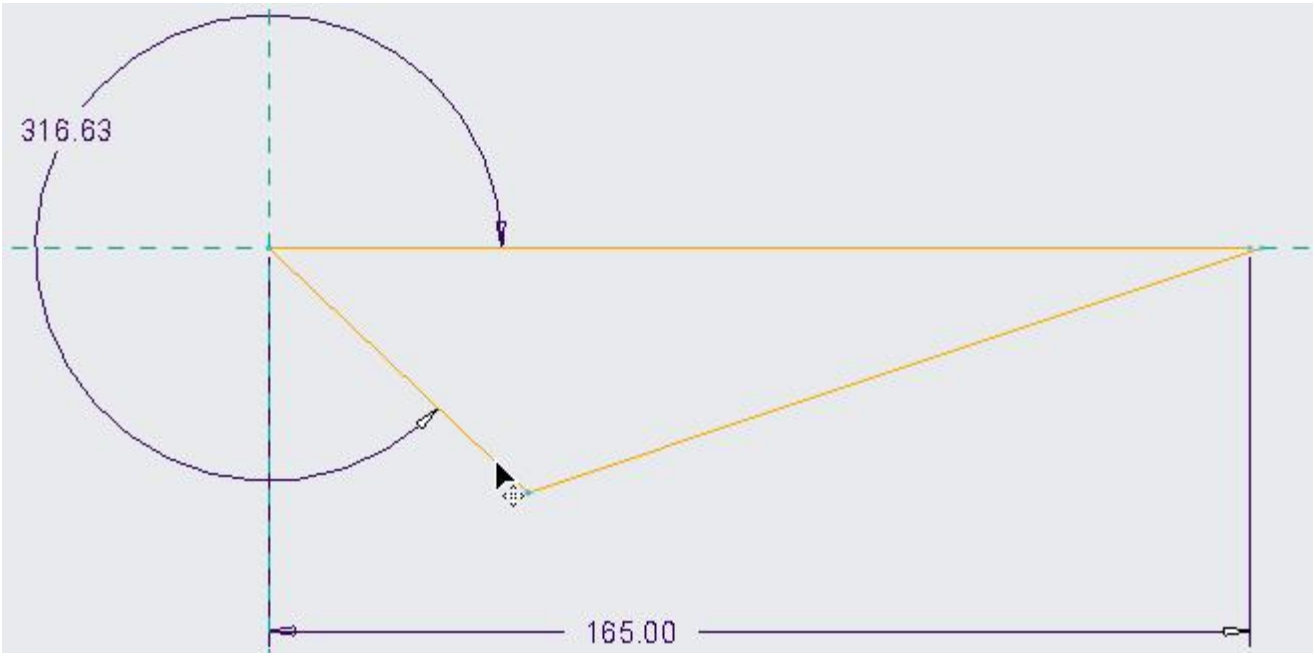


The locked dimensions will appear in different color as shown below.

Now take the mouse pointer over the left line as shown below.



Drag the line either to left or right and rotate the mouse pointer around the end point of line. Notice that sketched entities will simulate the motion of mechanism as shown below.



Pick ↶ to undo all drag section operations.

Pick ✔ to apply the changes and exit sketcher.

Locking of dimensions has no effect on the parts that are dependent on the sketch. We did so just to simulate the mechanism motion in the sketcher.

⇨ Creating Mechanism Connections

Now we will change the assembly constraints to mechanism connections. After this we will be able to simulate the mechanism.

Before proceeding hide the 01__ASM_ALL_DTM_PLN layer to unclutter the screen.

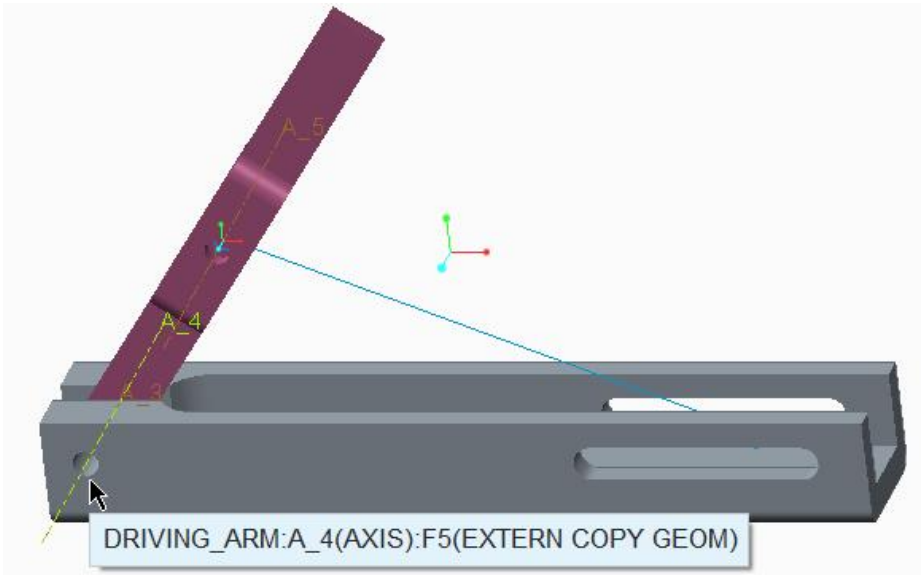
BASE.PRT is the ground body in the mechanism so we do not need to change the assembly constraints as it will not move with respect to the assembly.

Select the DRIVING_ARM.PRT and pick  .

Right-click the Default constraint in the Placement slide-up panel and pick **Delete**

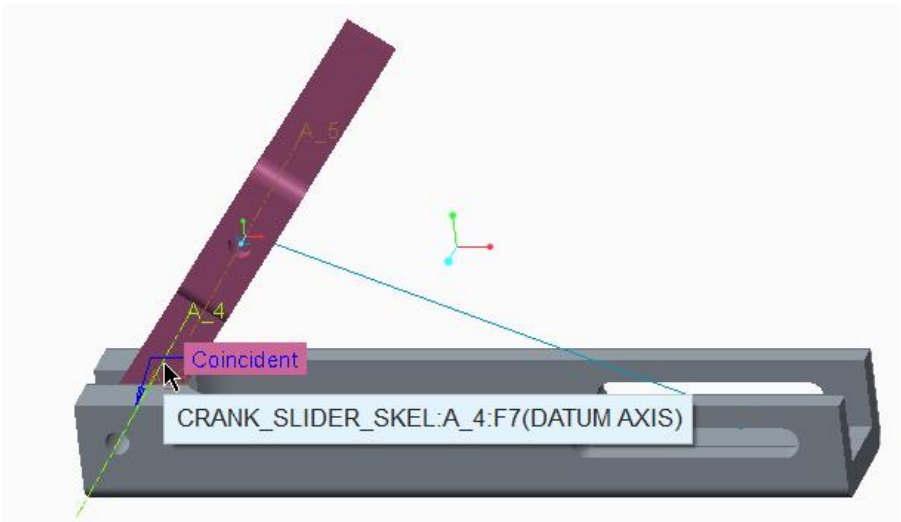
Select the **Pin** connection from the list.

Select the **A_4** axis from the component as shown below.

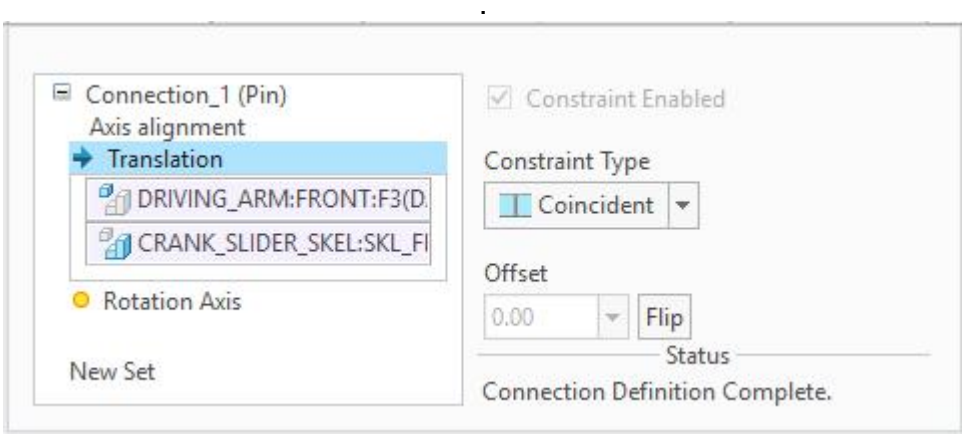


Notice that the selected axis belongs to external copy geometry feature. Although we can select an axis from the part geometry but it is not as stable reference as the axis from external copy geometry. So it is preferable to select the references copied from skeleton.

Select the **A_4** axis from the skeleton model as Assembly reference.



Now system will activate the Translation constraint so select the **FRONT** datum plane as component reference and **SKL_FRONT** as assembly reference as shown below.



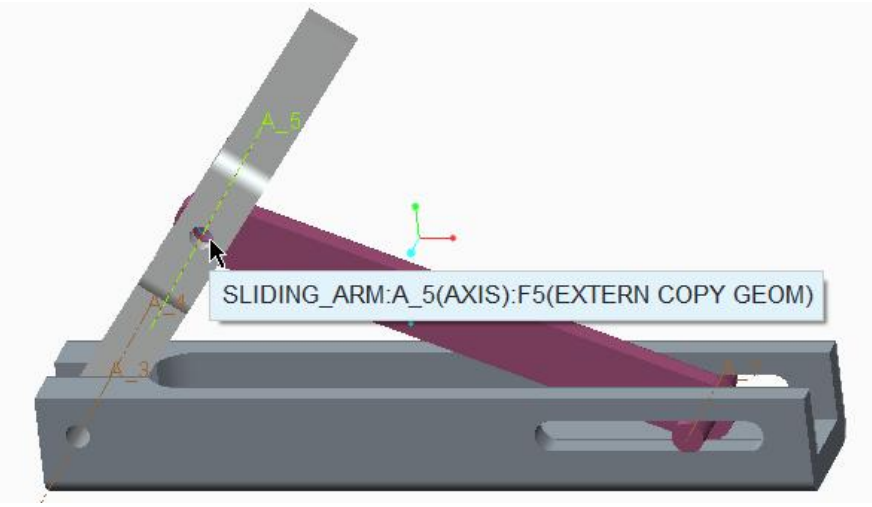
Pick to apply the changes and exit the dashboard.

Select the SLIDING_ARM.PRT and pick .

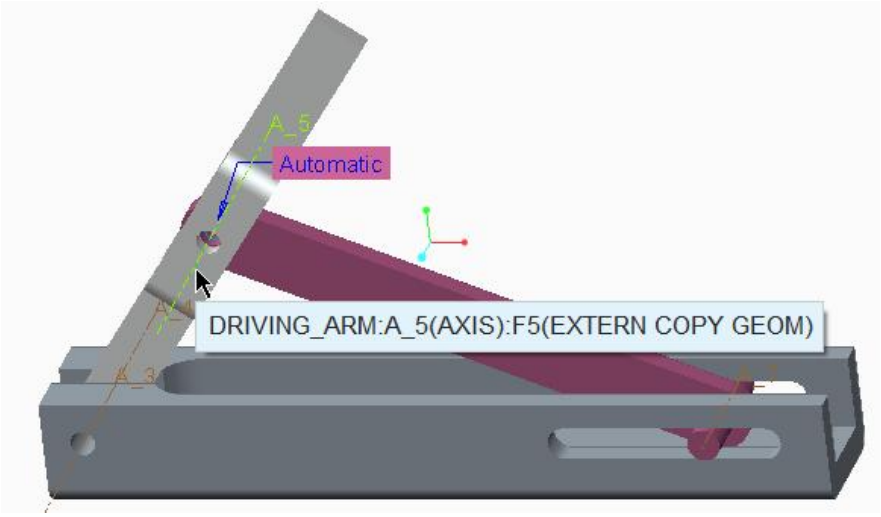
Right-click the Default constraint in the Placement slide-up panel and pick **Delete**

Select the **Cylinder** connection from the list.

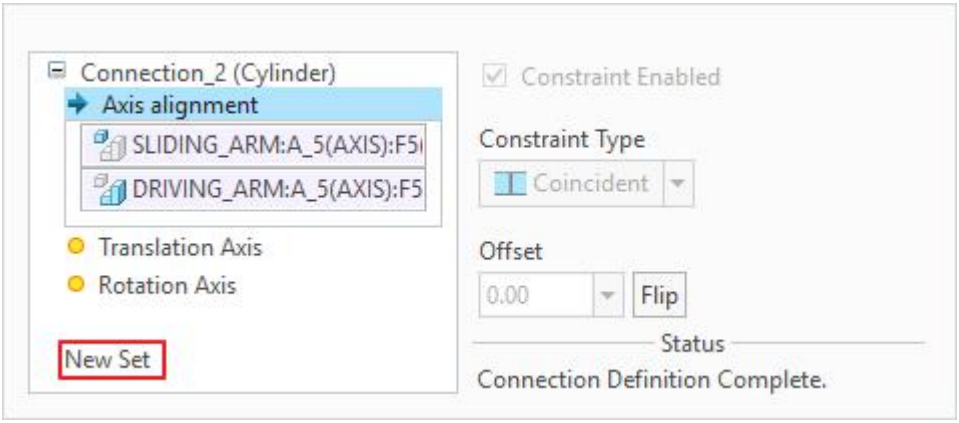
Select the **A_5** axis from the component as shown below.



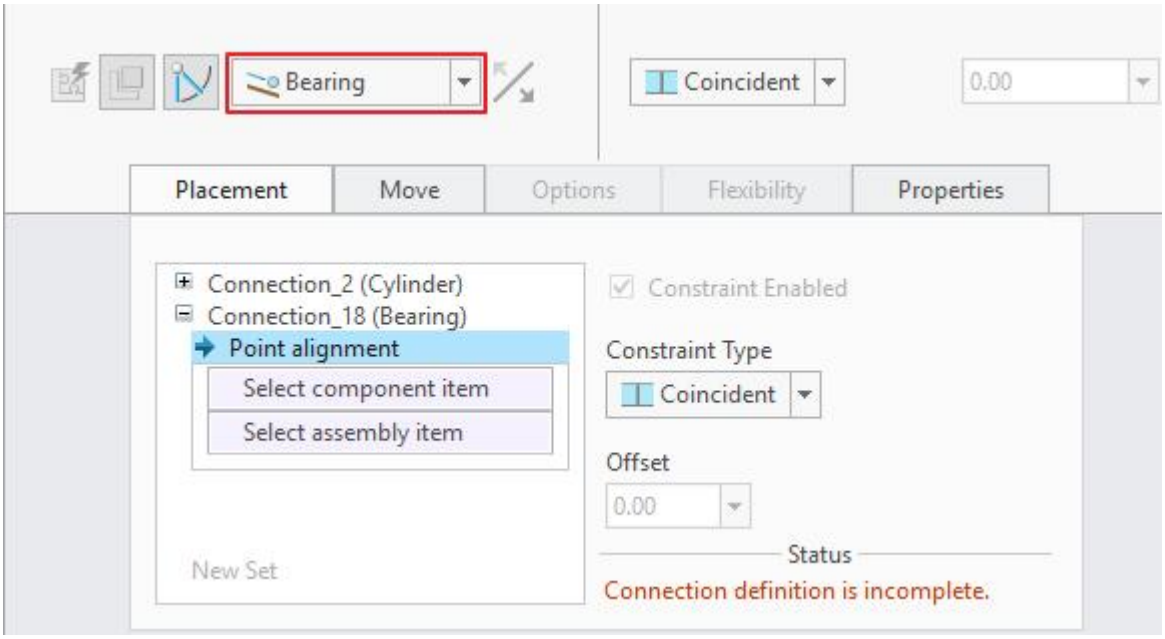
Select the **A_5** axis from the DRIVING_ARM as Assembly reference.



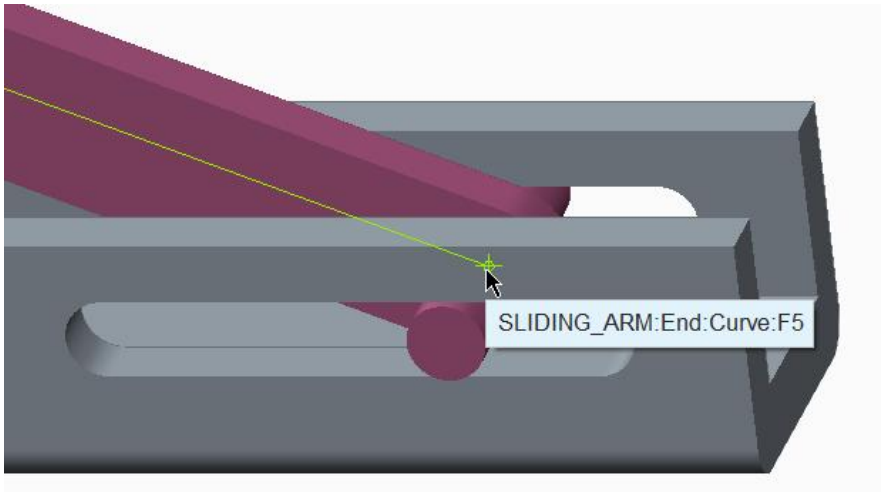
Pick on the New Set (text icon) to create a new connection.



Select the **Bearing** connection from the list.



Select the end vertex of the datum curve from the SLIDING_ARM as shown below.



Select the horizontal curve form the skeleton model as Assembly reference. The desired curve is shown in the figure below.

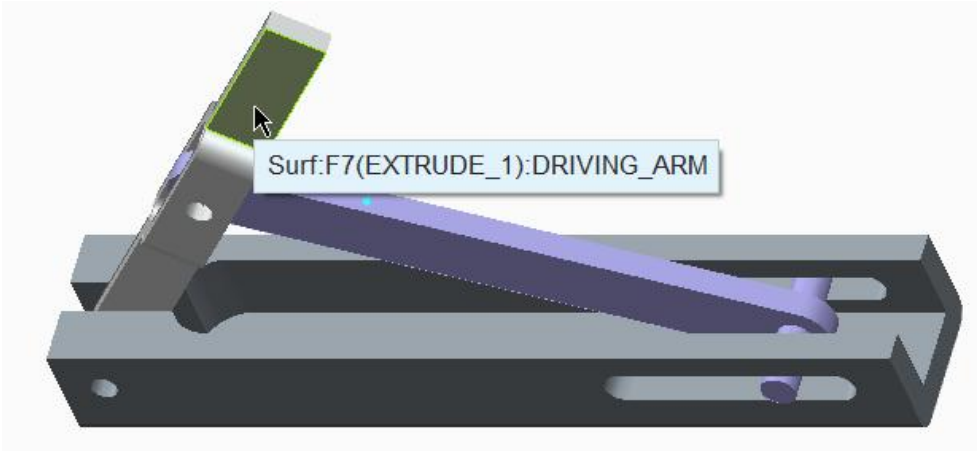
Pick  to apply the changes and exit the dashboard.

Moving the Bodies

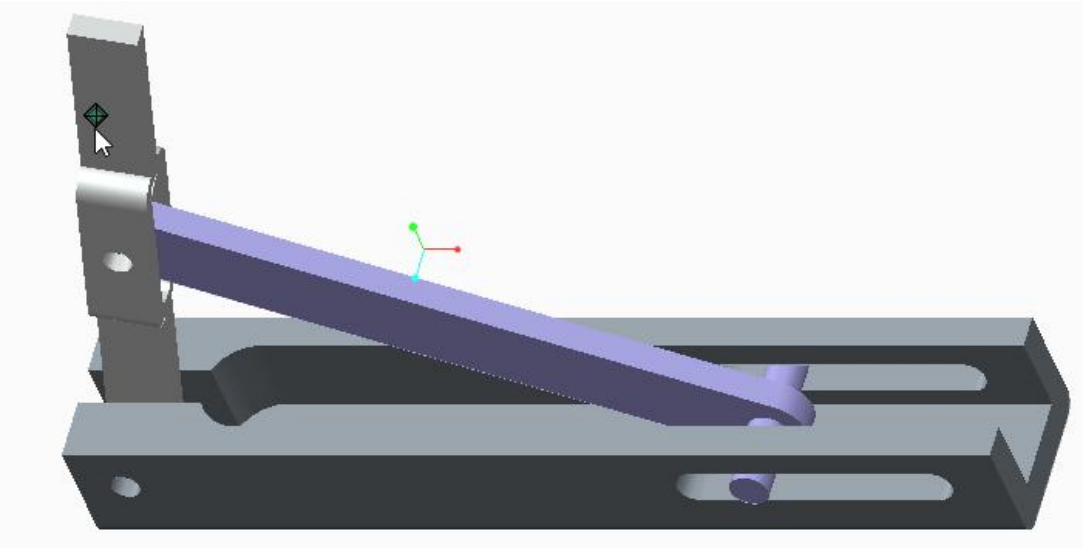
Now we will use the drag functionality to move bodies. Before that hide the 07___ASM_ALL_SKELETONS and 03___PRT_ALL_CURVES layers.

Pick the Drag Packaged Component icon 

Pick on any entity belonging to SWIVEL_ARM as shown below.



Now move the mouse to drag the selected body. After moving towards left it should appear as shown below.

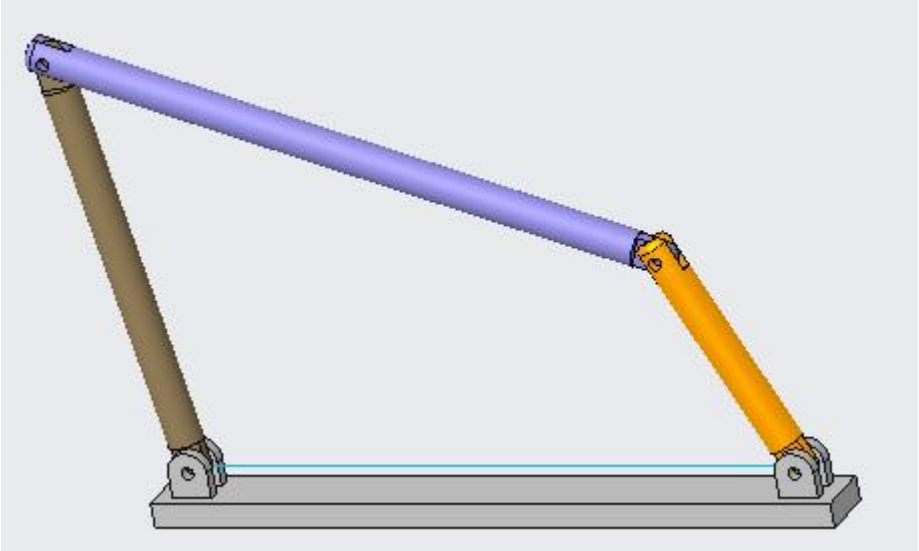


If the mechanism moves correctly it means you have successfully completed the exercise.

Exercise 4

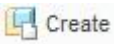
In this exercise we will learn how to use the motion skeleton to incorporate motion at the top level of a design project.

We will create a four bar mechanism driven by the skeleton as shown below.

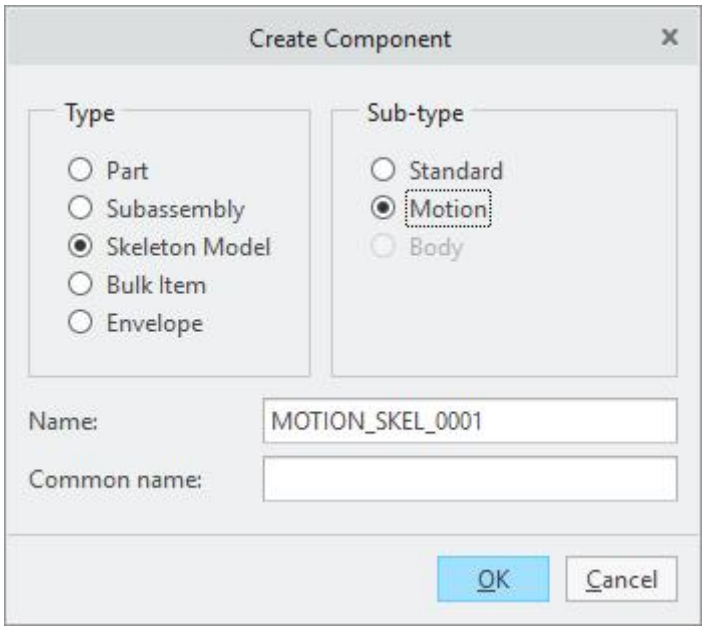


Set the working directory to FOUR_BAR folder and open the assembly FOUR_BAR.ASM

Creating Motion Skeleton

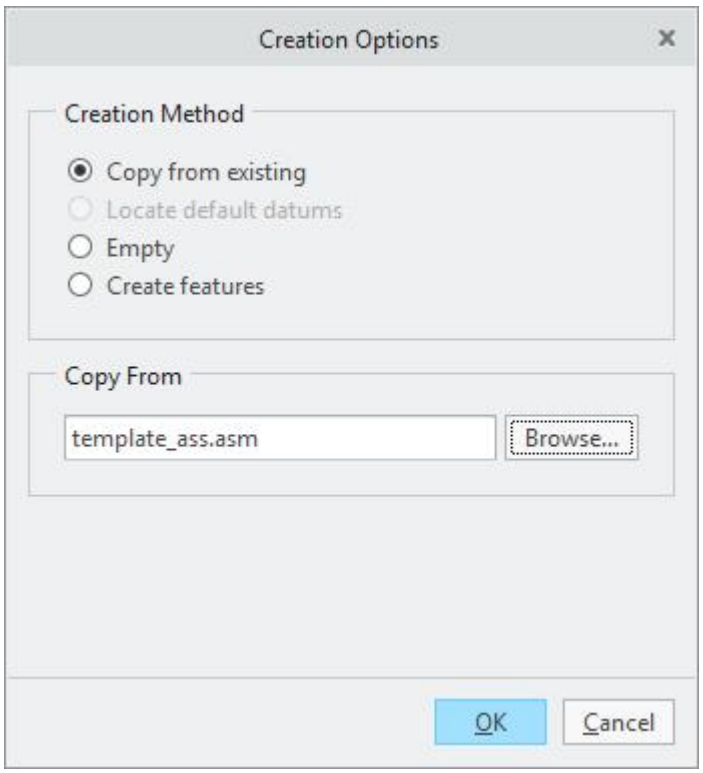
Pick the Create a component icon 

Component Create dialog box will appear. Here change the Type to **Skeleton Model** and Sub-type to **Motion** as shown below.



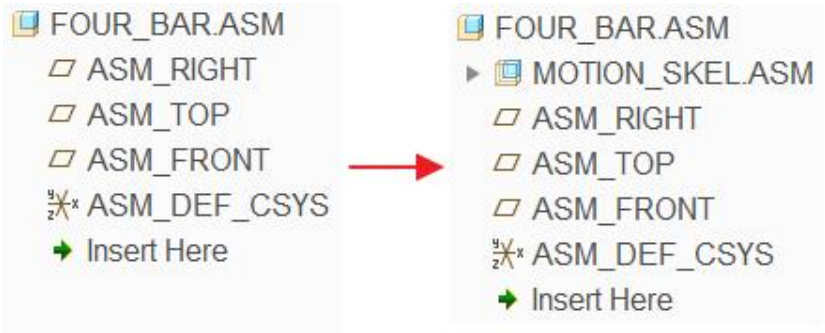
Change the Name to ***MOTION_SKEL*** and pick OK

Pick OK tab and Create Options dialog box will appear. Pick Browse... tab and select the TEMPLATE_ASS.ASM as shown below.



TEMPLATE_ASS.ASM is the template file that contains the default datum features, layers and accuracy settings.

Pick OK and newly created motion skeleton model will appear as the first feature in the model tree as shown below.



Notice that motion skeleton model is a subassembly.

Motion skeleton models are subassemblies that contain conceptual design of a mechanism and body skeletons.

You should notice that motion skeleton has a unique icon in the model tree and is different from the Standard skeleton.

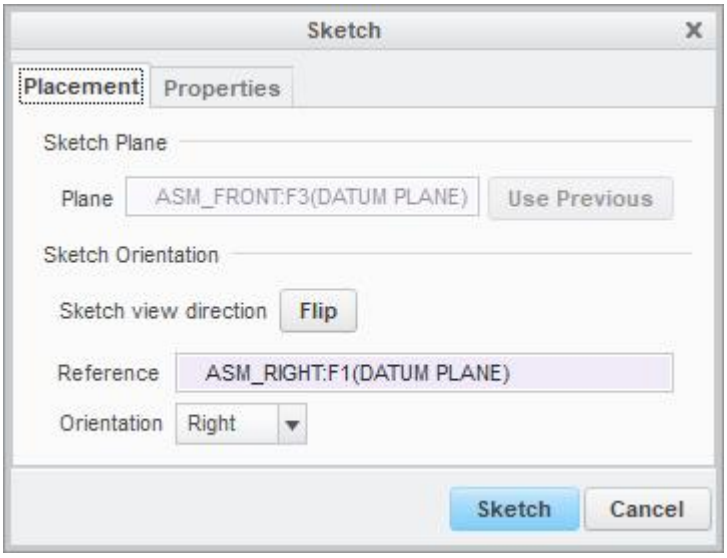
Icon for Standard Skeleton Icon for Motion Skeleton

Creating Conceptual Sketch for Mechanism

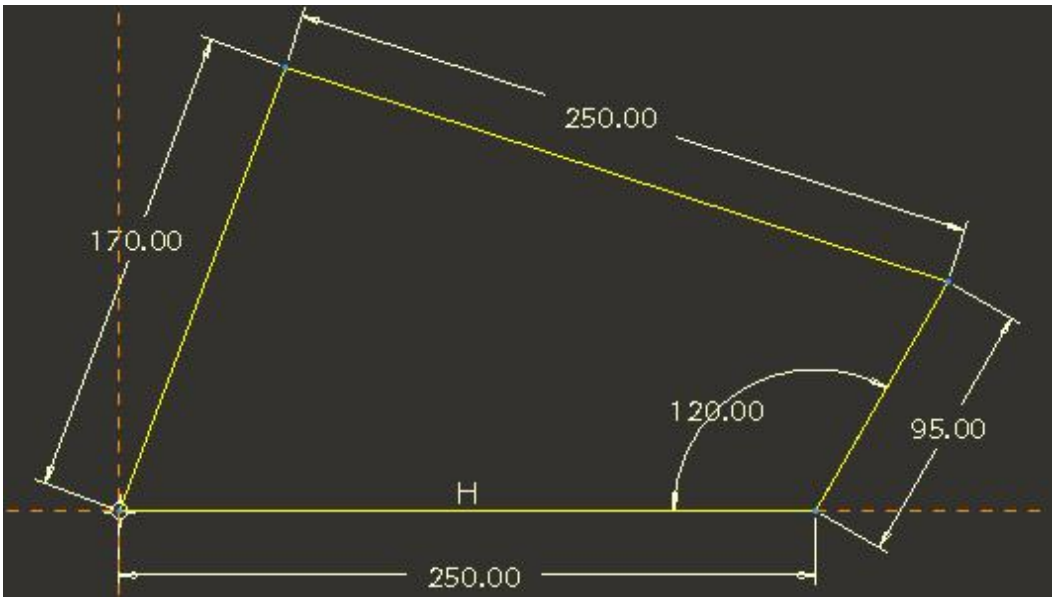
Open the skeleton model in new window and notice that it has the default datum planes and coordinate system.

Now we will create the conceptual mechanism by creating a sketch feature.

Pick and select the sketching references as shown below



Now sketch the section as shown below.

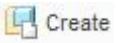


Pick to complete the section.

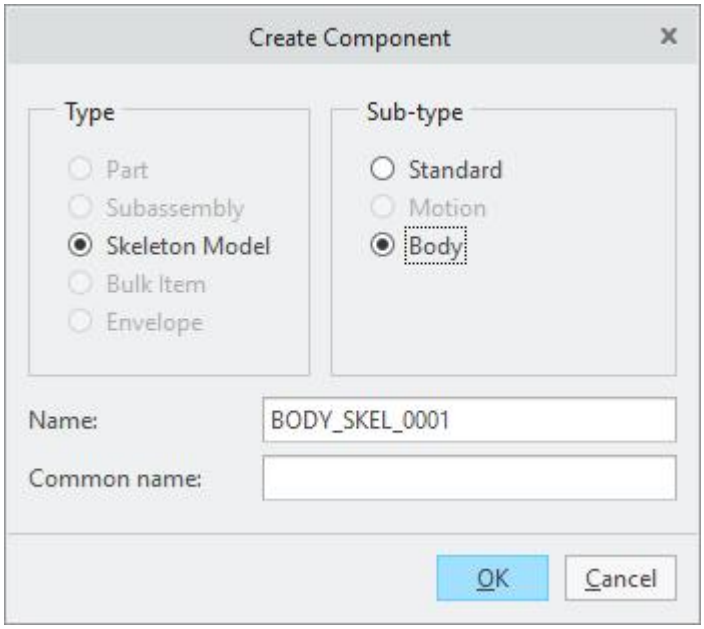
Creating Body Skeletons

In the following section, we will create body skeletons.

First of all we will create the ground body of the mechanism.

Pick the Create a component icon 

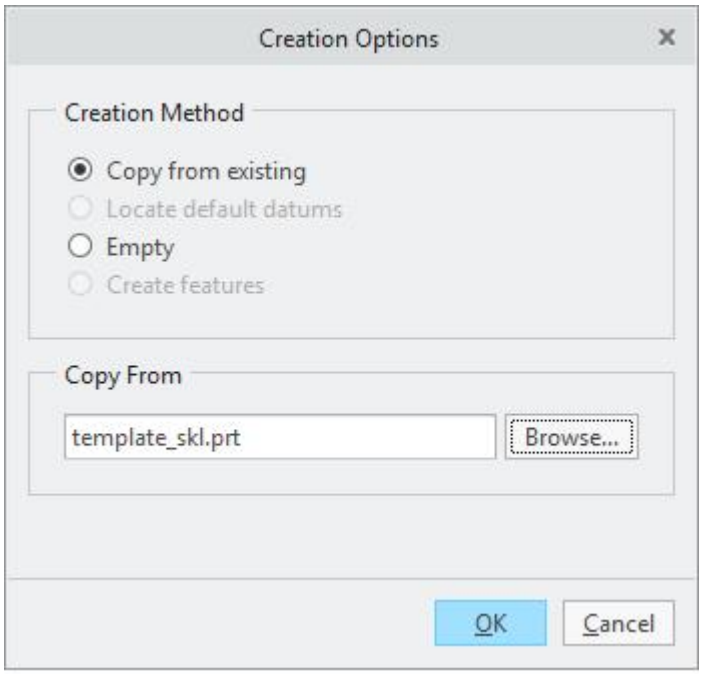
In Component Create dialog box, change the Type to **Skeleton Model** and Sub-type to **Body.**



Change the Name to ***BODY_SKEL_1*** and pick 

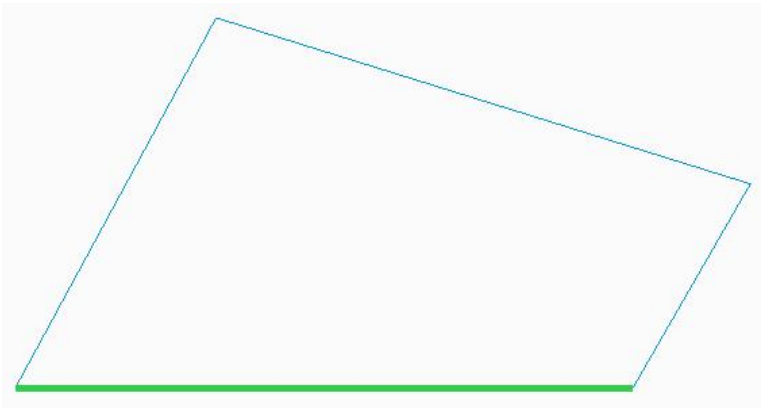
In the Create Options dialog box, pick 

tab and select the TEMPLATE_SKL.PRT.



Pick OK and BODY DEFINITION dialog box will appear.

Select the highlighted entity of the sketch shown in the figure below.



Pick OK to complete the definition of the body.

The first body is always categorized as Ground Body. There are no connection definitions for the ground body.

Now we will create the second body skeleton.

Pick the Create a component icon Create

In Component Create dialog box, change the Type to **Skeleton Model** and Sub-type to **Body**

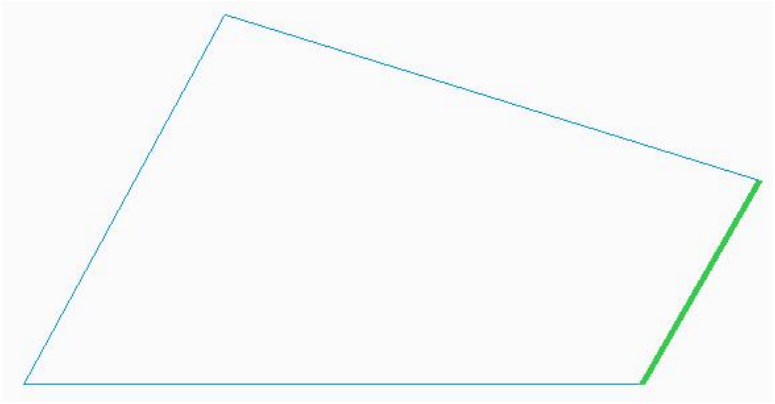
Change the Name to ***BODY_SKEL_2*** and pick OK

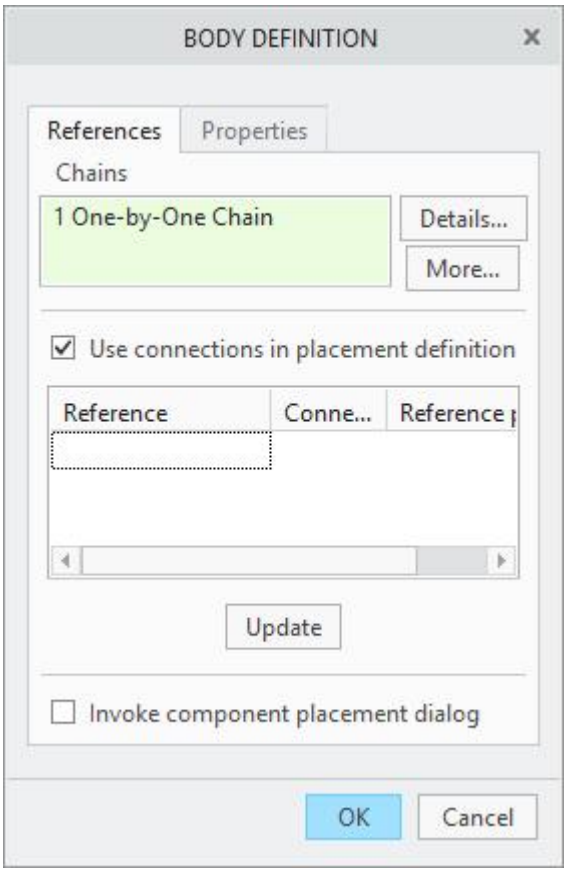
In the Create Options dialog box, pick Browse... tab and select the TEMPLATE_SKL.PRT as shown below.

Pick OK and BODY DEFINITION dialog box will appear .

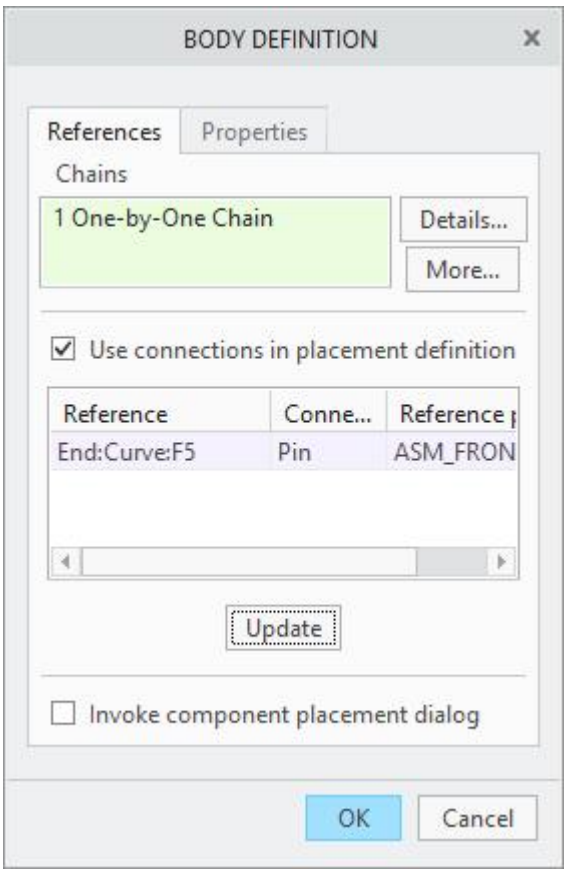
Notice that now the BODY DEFINITION dialog box is different than the dialog box for ground skeleton.

Select the highlighted entity of the sketch shown in the figure below.





Pick Update and system will apply the appropriate connection for this body skeleton as shown below.



System automatically creates the required axes for the connections in model tree.

Pick OK to complete the definition of the body.

Now we will create the third body skeleton.

Pick the Create a component icon Create

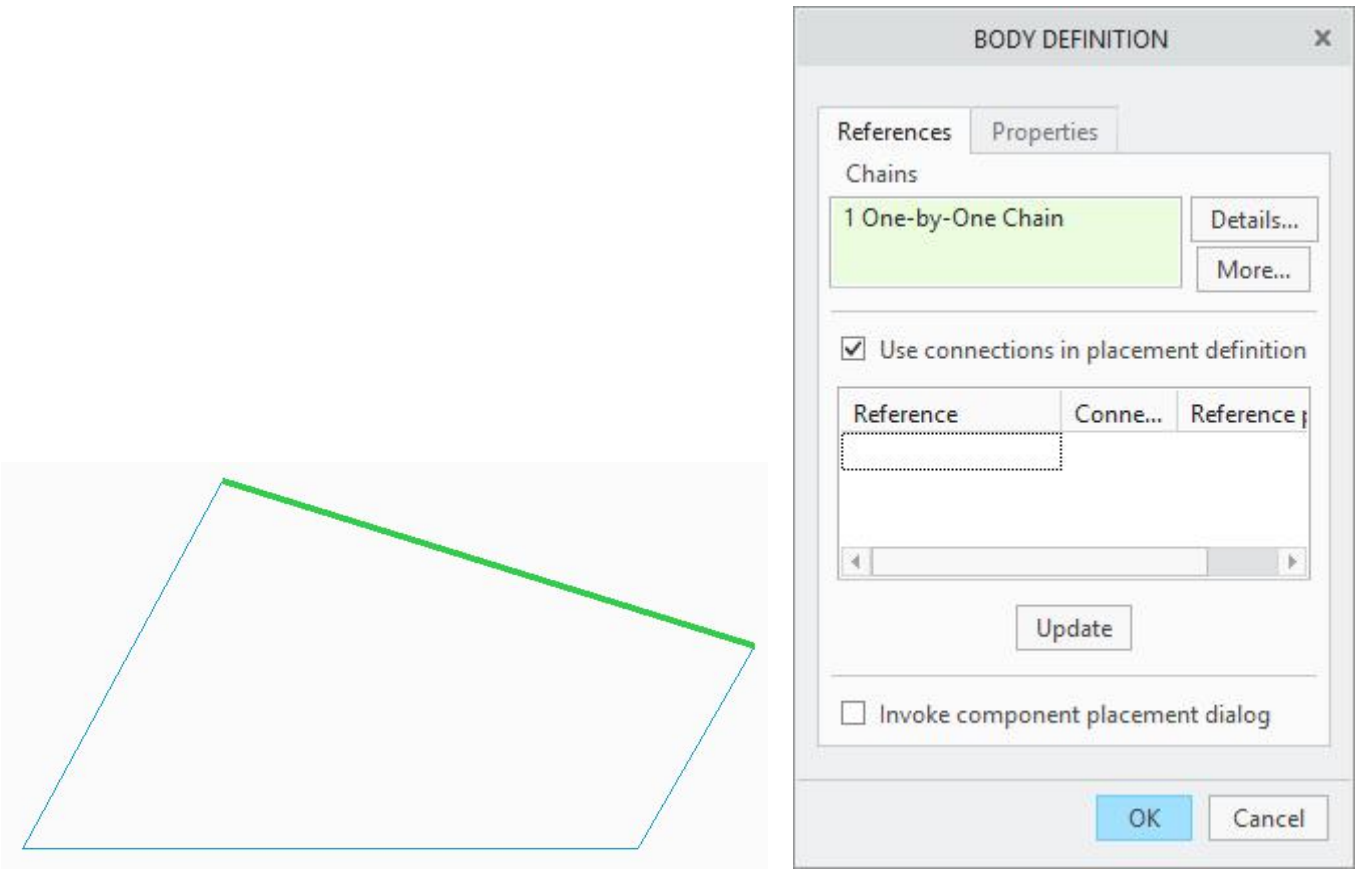
In Component Create dialog box, change the Type to **Skeleton Model** and Sub-type to **Body**

Change the Name to ***BODY_SKEL_3*** and pick OK

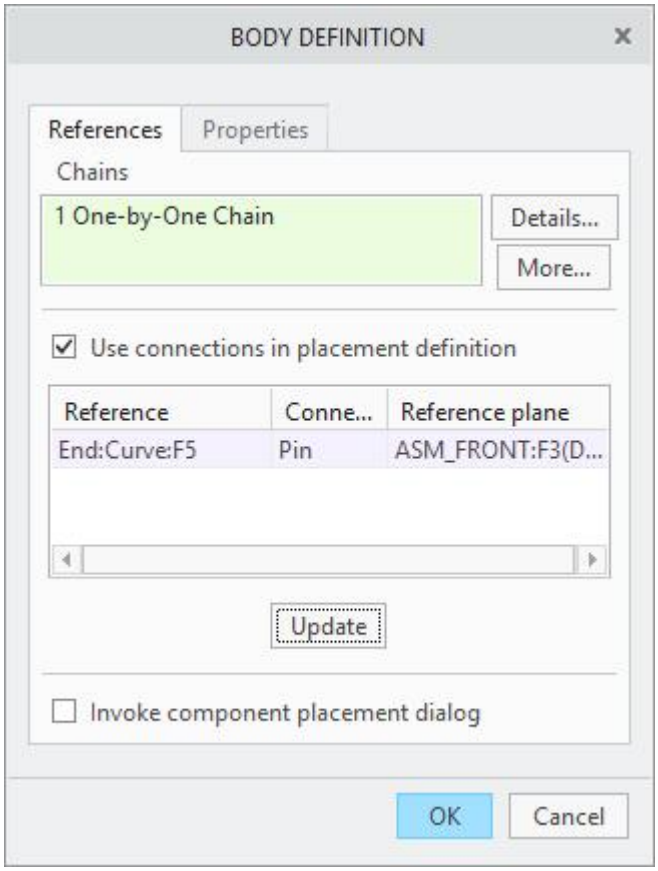
In the Create Options dialog box, pick Browse... tab and select the TEMPLATE_SKL.PRT

Pick OK and BODY DEFINITION dialog box will appear.

Select the highlighted entity of the sketch shown in the figure below.



Pick Update and system will apply appropriate constraints as shown below.



Pick OK to complete the definition of the body.

Now we will create the fourth body skeleton.

Pick the Create a component icon Create

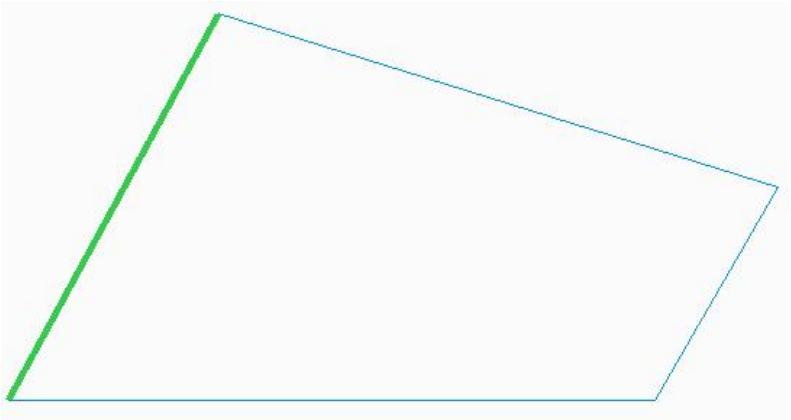
In Component Create dialog box, change the Type to **Skeleton Model** and Sub-type to **Body**

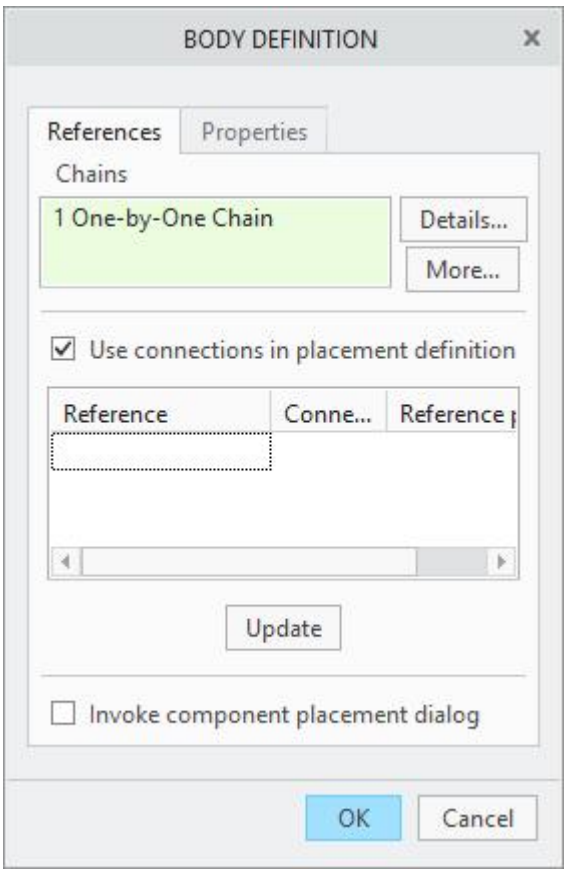
Change the Name to ***BODY_SKEL_4*** and pick OK

In the Create Options dialog box, pick Browse... tab and select the TEMPLATE_SKL.PRT

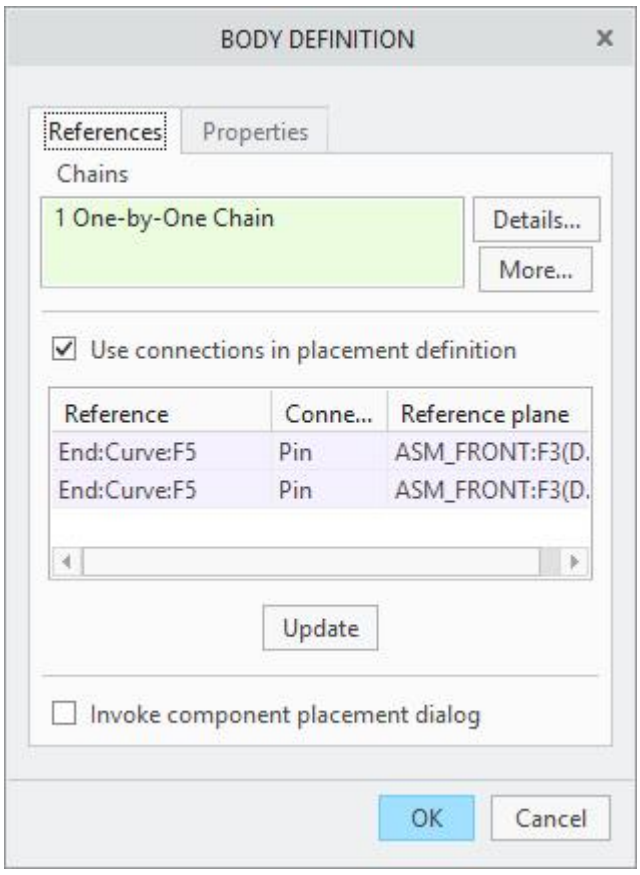
Pick OK and BODY DEFINITION dialog box will appear.

Select the highlighted entity of the sketch shown in the figure below.



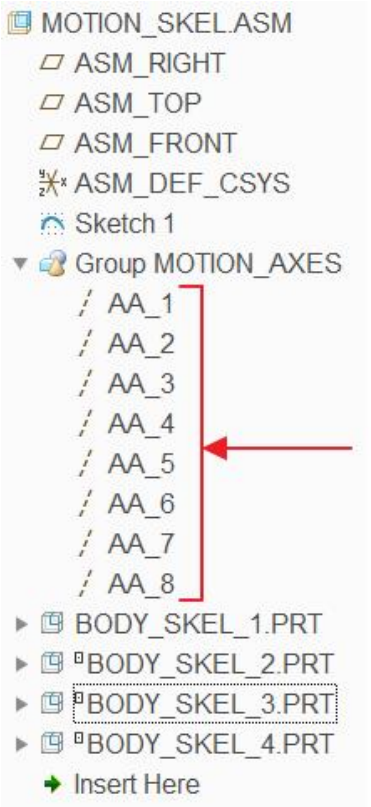


Pick Update and system will apply the appropriate constraints for this body skeleton as shown below.



Pick OK to complete the definition of the body.

Notice that system automatically creates the required axes for the connections in a separate group as shown below.

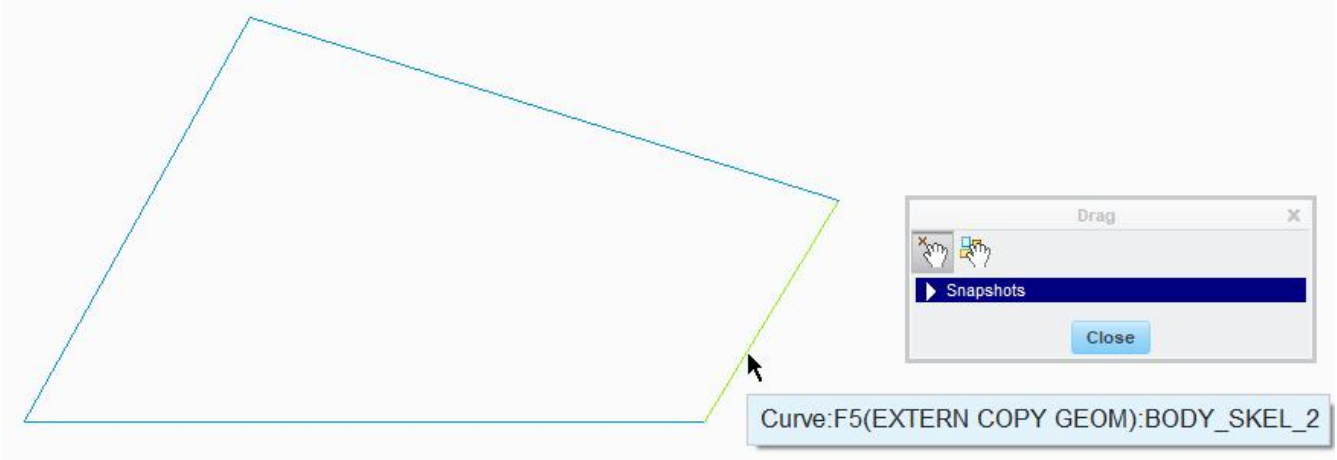


Moving Body Skeletons

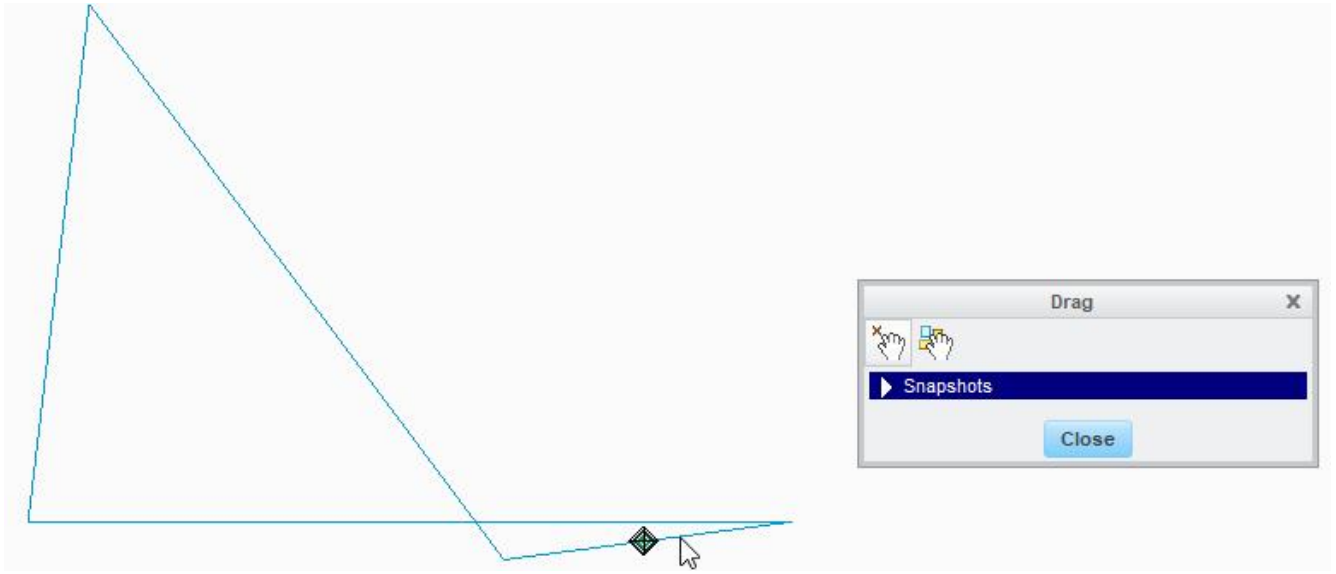
Now we will use the drag functionality to move assembly entities (body skeletons) to see if the mechanism works properly or not.

Pick the Drag Packaged Component icon ( )

Pick on any entity to select a body as shown below.



Now move the mouse to drag the selected body. You will notice that the other bodies also move according to the mechanism connections as shown below.



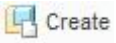
Left-click when you want to finish the dragging.

Look at the message area to see how the mouse buttons will act in the drag mode.

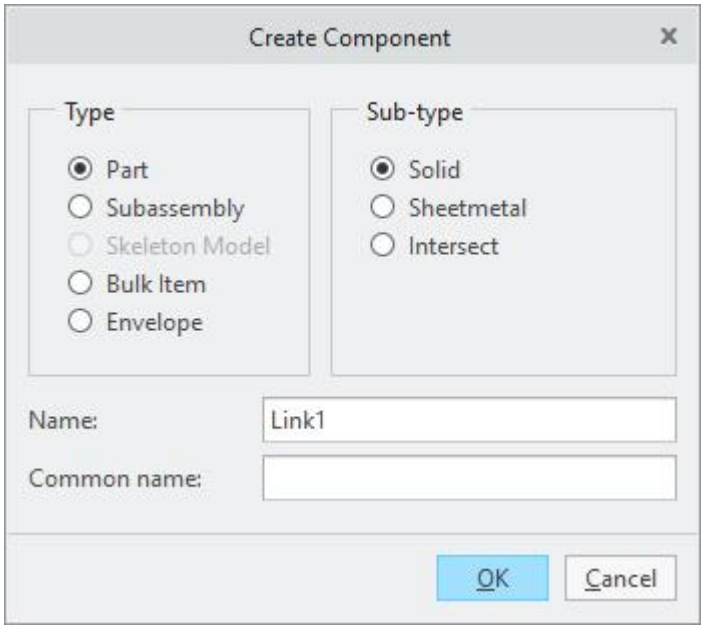
Creating Body Parts

Now we will create body parts by referencing the body skeletons.

Switch to the FOUR_BAR.ASM window.

Pick the Create a component icon 

Component Create dialog box will appear. Enter ***LINK1*** as the Name.

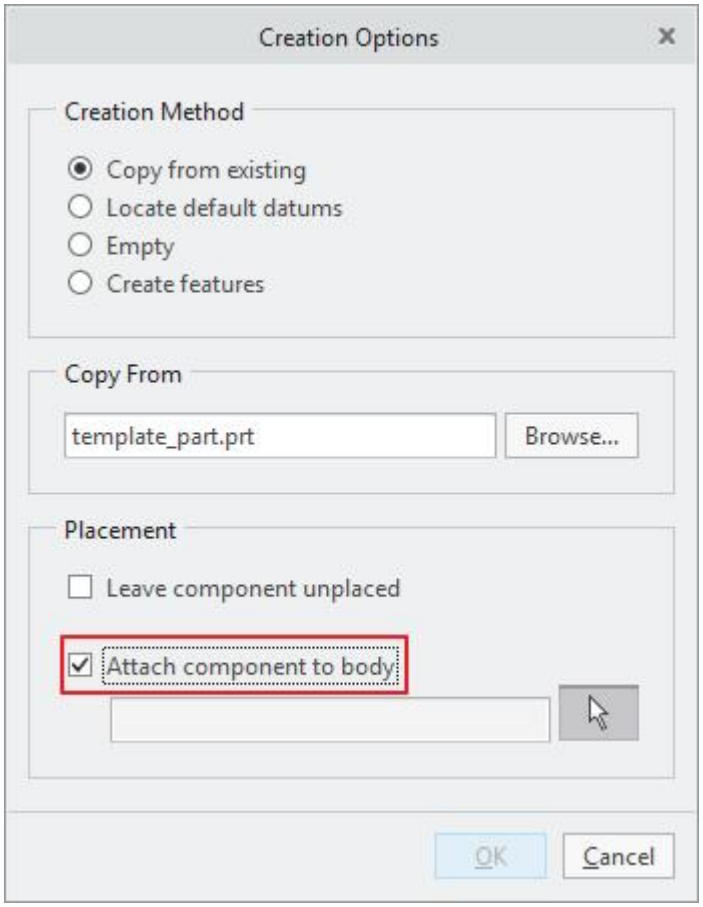


Pick 

and Create Options dialog box will appear. Pick 

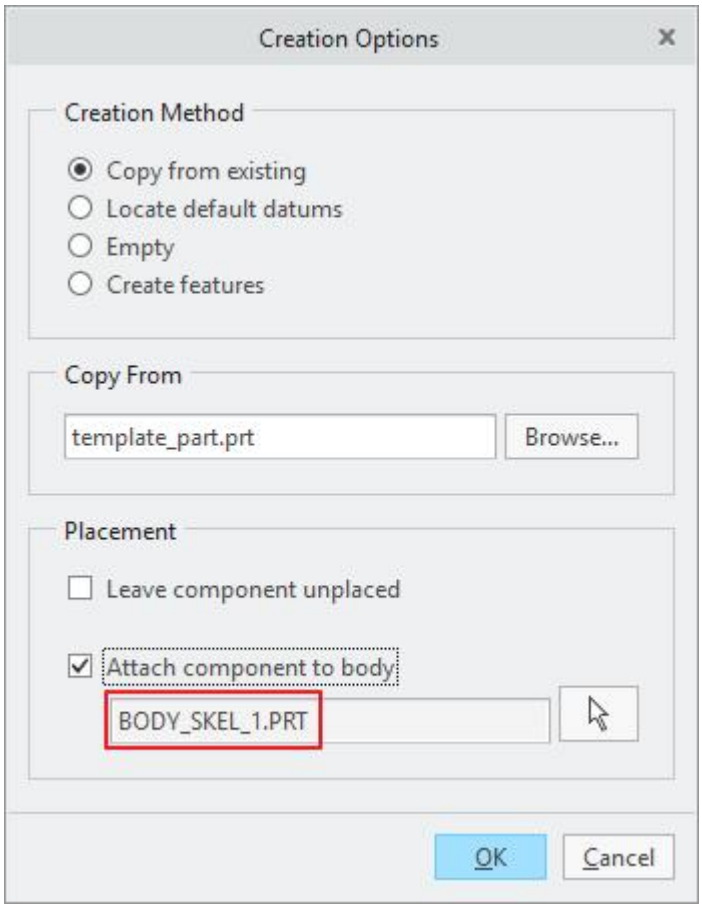
tab and select the TEMPLATE_PART.PRT as template.

Check **Attach Component to Body** option to attach the skeleton body with this part.



When “Attach Component to Body” is checked, system copies the geometry of the selected Body Skeleton to this part.

Select the **BODY_SKEL_1** component in the model tree. The dialog box will appear as shown below.



Pick OK and new component will appear in the model tree.

System automatically creates the placement constraints and copies the geometry from the body skeleton to this part.

Now we will create the part for second body of mechanism.

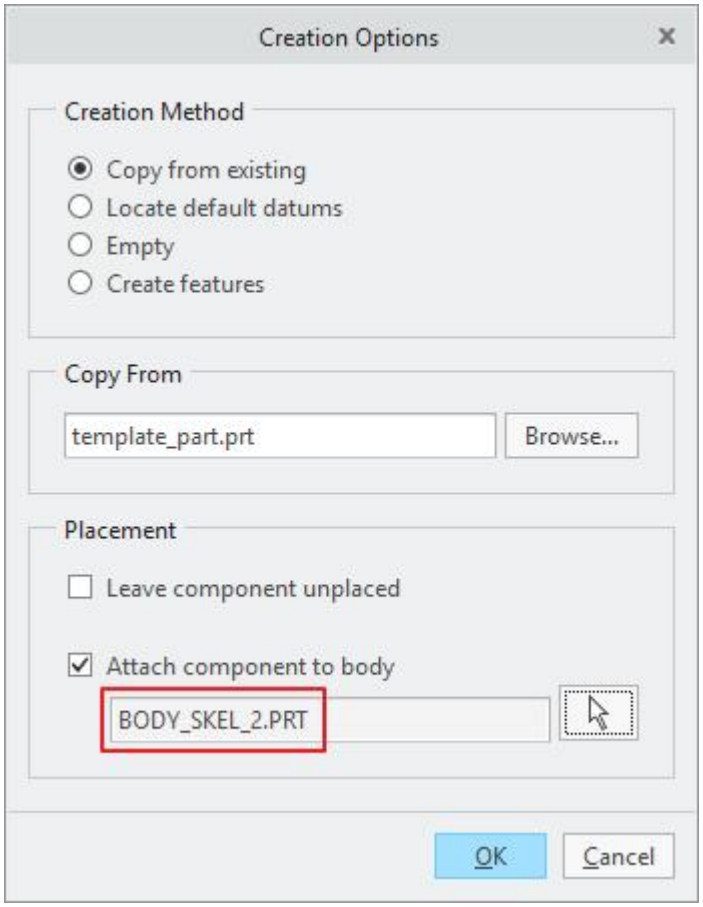
Pick the Create a component icon Create

Component Create dialog box will appear. Enter ***LINK2*** as the Name.

Pick OK and Create Options dialog box will appear. Pick Browse... tab and select the TEMPLATE_PART.PRT as template.

Check **Attach Component to Body** option to attach the skeleton body with this part.

Select the **BODY_SKEL_2** component in the model tree.



Pick OK and new component will appear in the model tree.

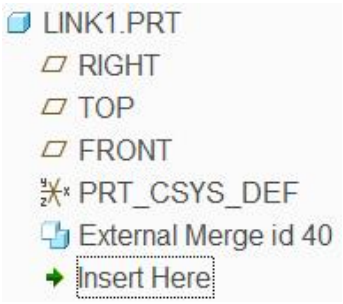
You can create LINK3.PRT and LINK4.PRT by attaching them to BODY_SKEL_3 and BODY_SKEL_4 respectively on your own. But it is not necessary for the purpose of this exercise.

Creating Geometry for Parts

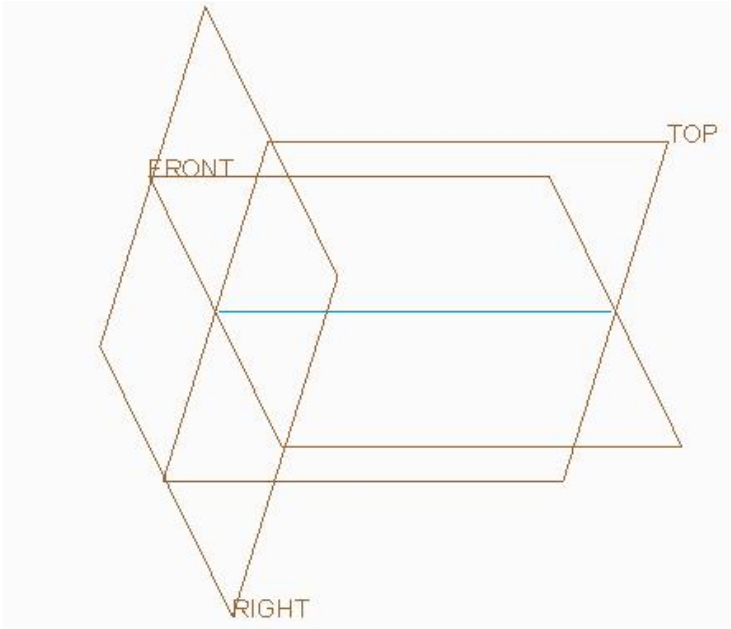
Now we will create the solid geometry in individual parts.

First Part

Open the LINK1.PRT in a new window and notice that it contains an External Merge feature as shown below.



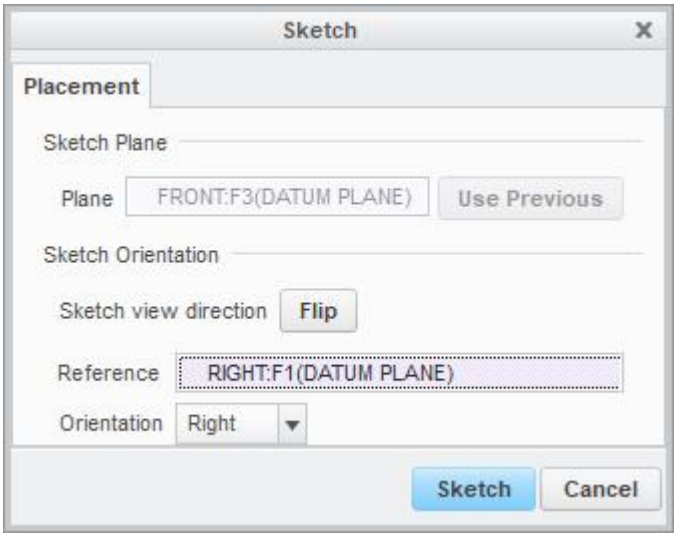
The merge feature references the geometry in the BODY_SKEL_1.PRT as shown below.



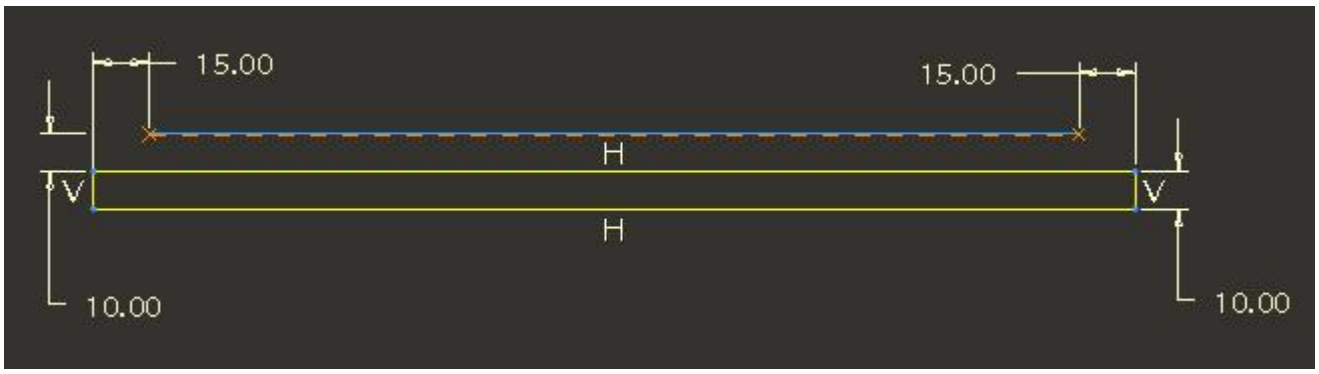
Merge feature copies the complete geometry of a source part to target part associatively.

Pick  to invoke Extrude tool.

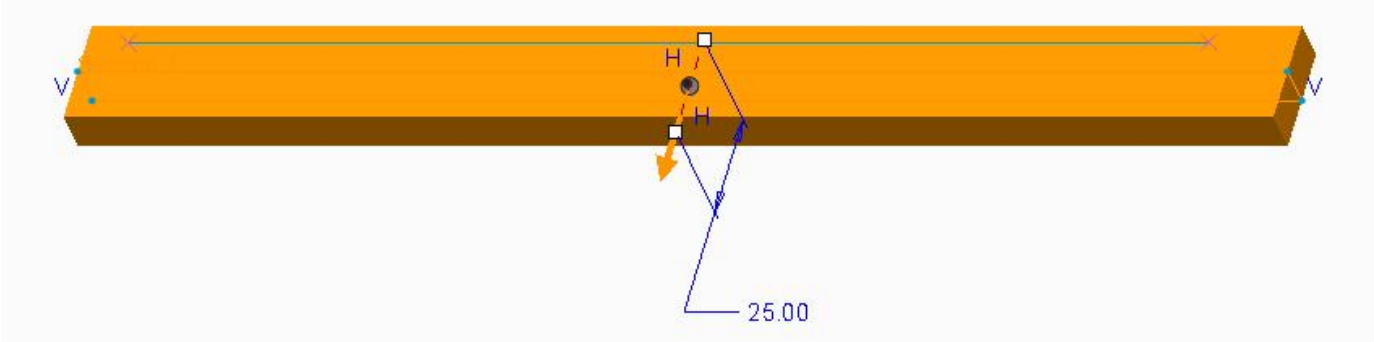
Select the sketching references as sown below.



Sketch as shown below.



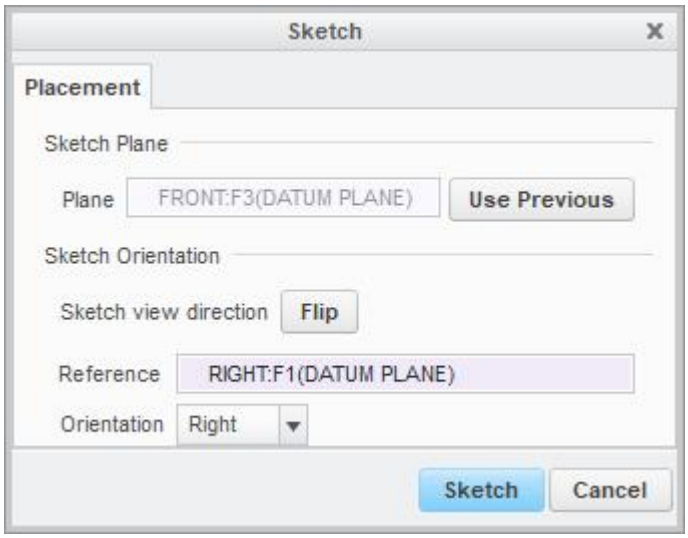
After completing the sketch, change the depth option to Symmetric ( ) and enter **25** as depth value.



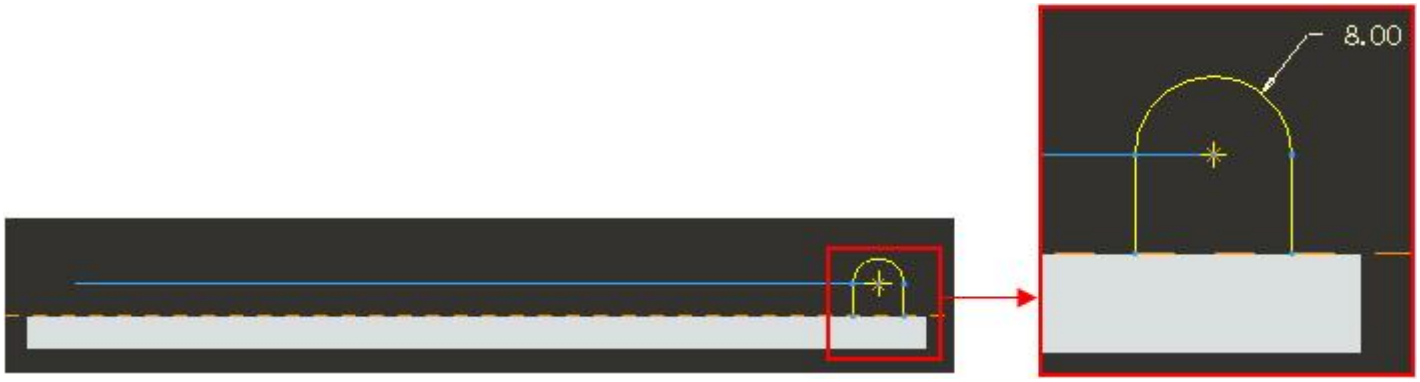
Pick icon to complete the feature.

Again pick to invoke Extrude tool.

Select the sketching references as sown below.



Sketch as shown below.



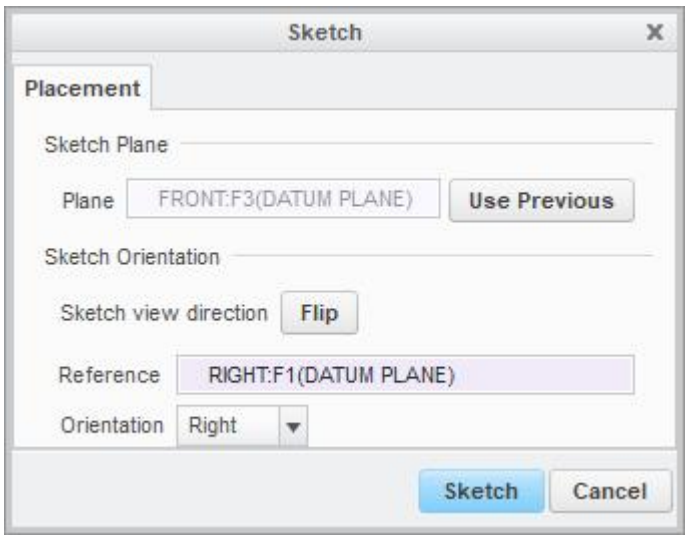
After completing the sketch, change the depth option to Symmetric ( ) and enter **15** as depth value.

Pick icon to complete the feature. It will appear as shown below.

Copy this extrude feature using Paste Special option so that part appear as shown below.

Again pick to invoke Extrude tool.

Select the sketching references as sown below.

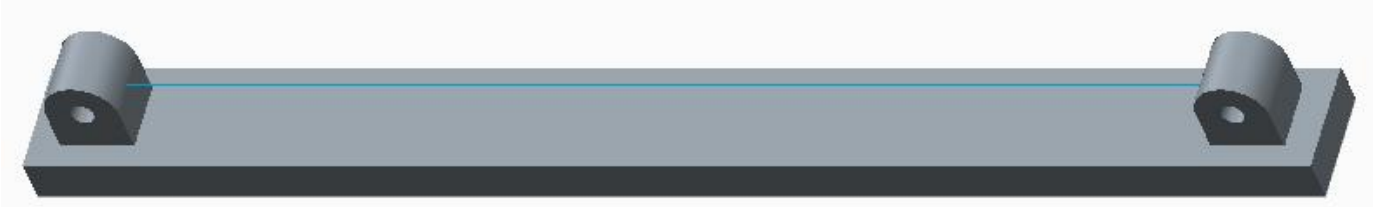


Sketch two circles aligned to the end points of the datum curve as shown below.

After completing the sketch pick  icon to create the feature as cut.

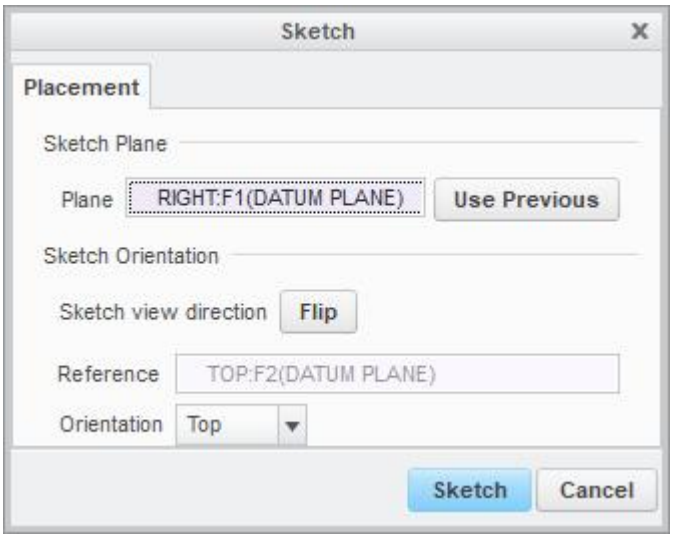
Specify the **Through All** () depth option for both directions.

Pick  icon to complete the feature. It will appear as shown below.

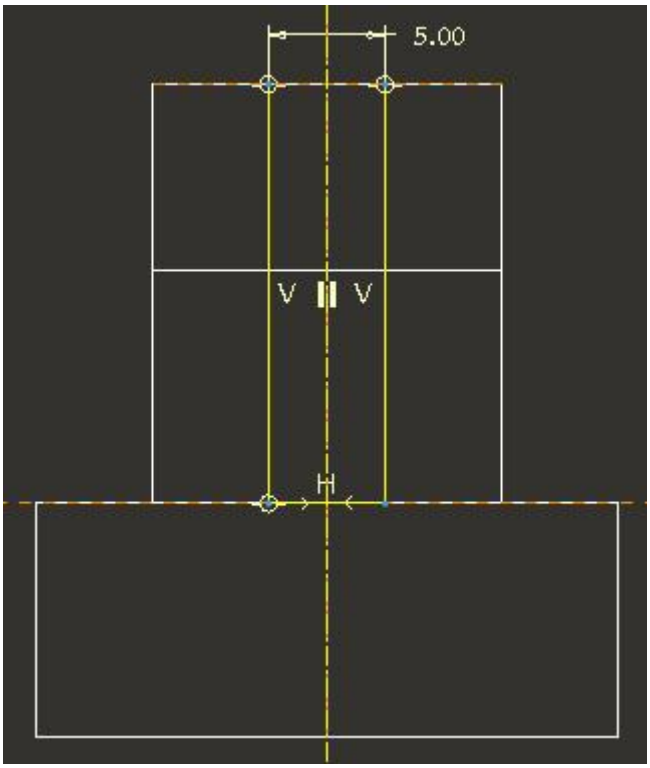


Again pick  to invoke Extrude tool.

Select the sketching references as sown below.



Sketch as shown below.



After completing the sketch pick  icon to create the feature as cut.

Specify the **Through All** () depth option for both directions.

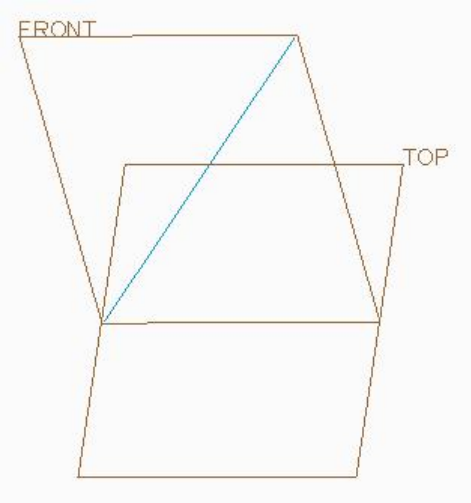
Pick  icon to complete the feature. It will appear as shown below.

Second Part

Now we will create the geometry in second body.

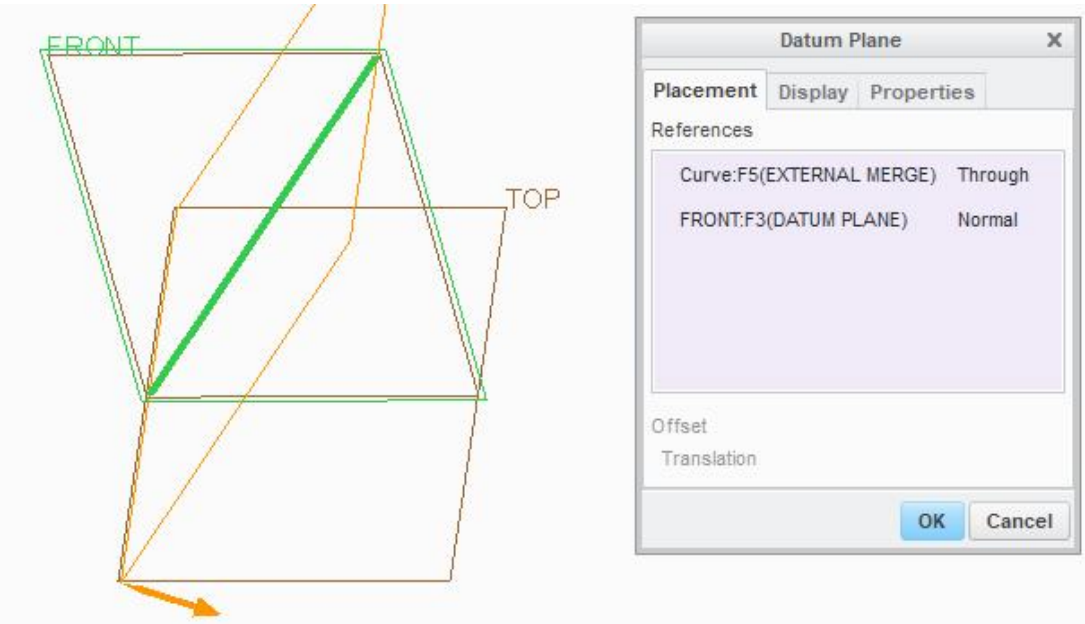
Open the LINK2.PRT in a new window and notice that it contains an External Merge feature.

The merge feature references the geometry in the BODY_SKEL_2.PRT as shown below.



First we will create a datum plane that passes through the shown datum curve.

Pick on the Model tab and select the references as shown below.

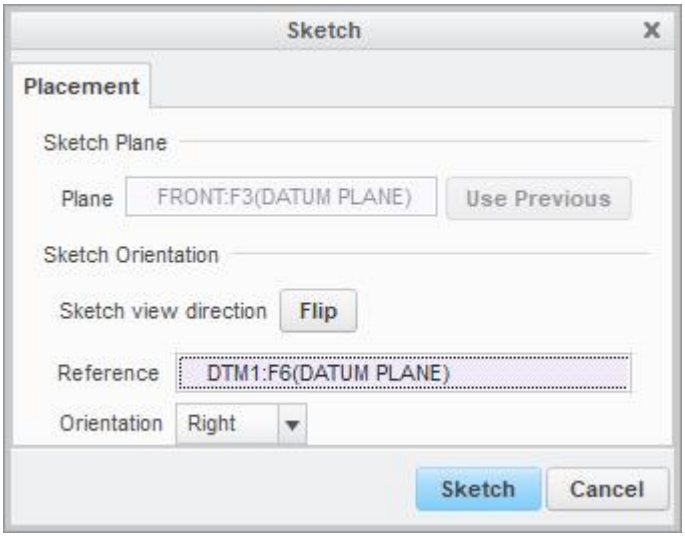


Notice that datum plane is normal to the FRONT datum plane.

Pick OK to apply the changes and exit dialog box.

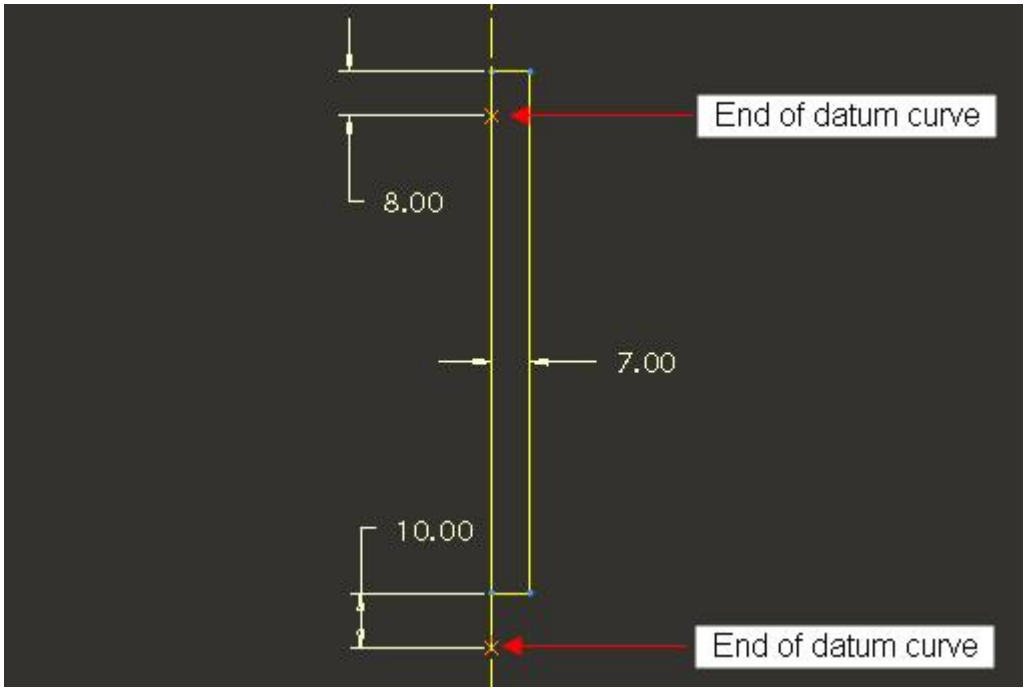
Pick Revolve to invoke Revolve tool.

Select the sketching references as sown below.



Here DTM1 is the newly created datum plane.

Select the datum curve and its end points as reference then sketch as shown below.



Notice that centerline is aligned to the datum curve.

After completing the sketch make sure that rotation angle is set to **360**.

Pick icon to complete the feature.

Select the following edge for applying round.

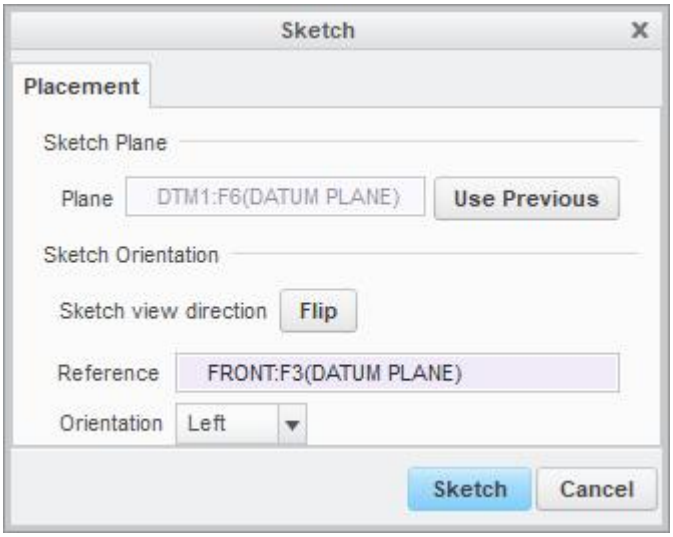
Pick Round to access the Round tool.

Enter ***3*** as the radius value

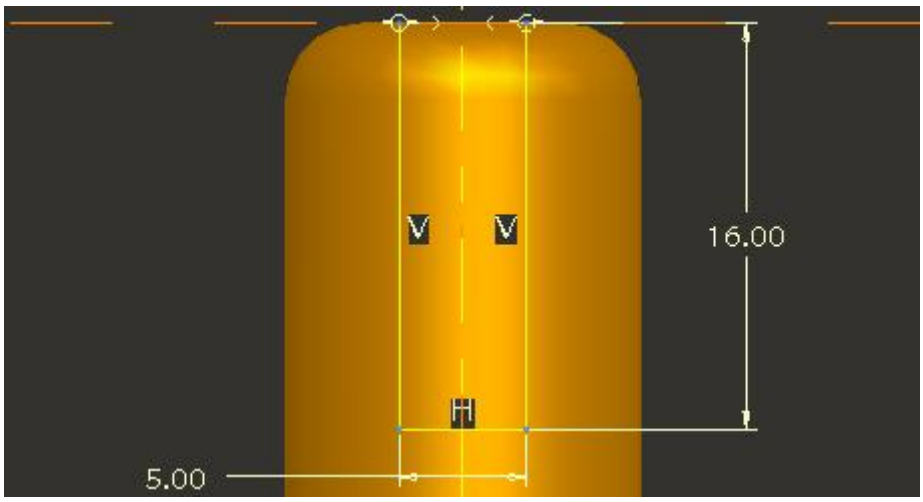
Pick icon to complete the feature.

Now pick to invoke Extrude tool.

Select the sketching references as sown below.



Sketch as shown below.



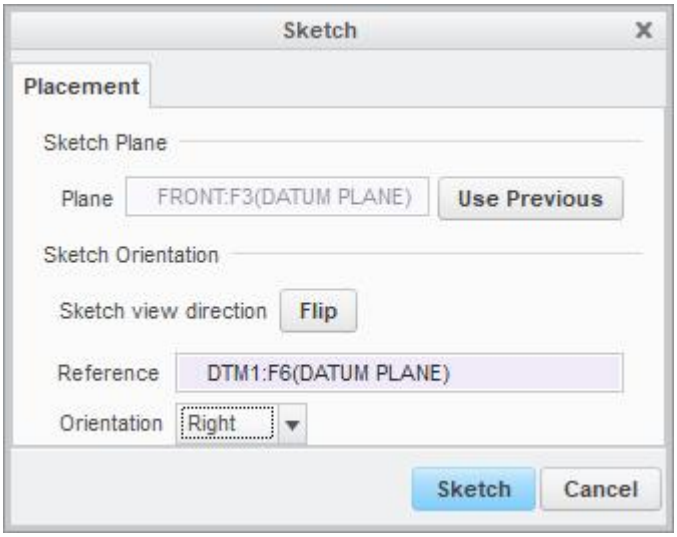
After completing the sketch pick  icon to create the feature as cut.

Specify the **Through All** () depth option for both directions.

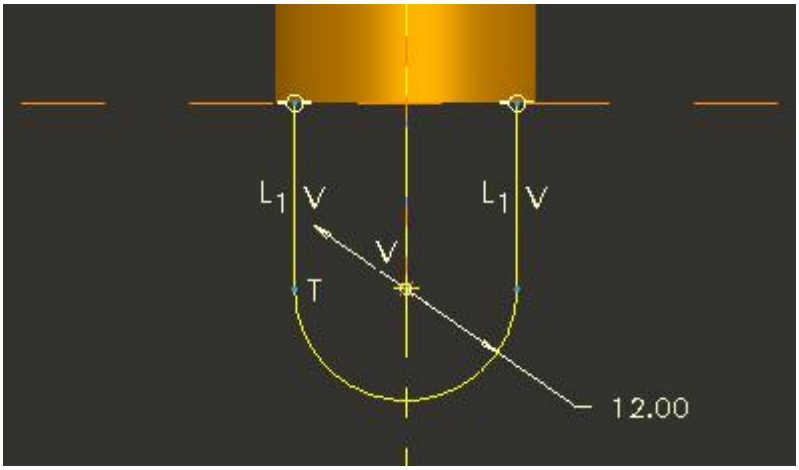
Pick  icon to complete the feature. It will appear as shown below.

Again pick  to invoke Extrude tool.

Select the sketching references as sown below.



Select the end of datum curve as reference and sketch as shown below.



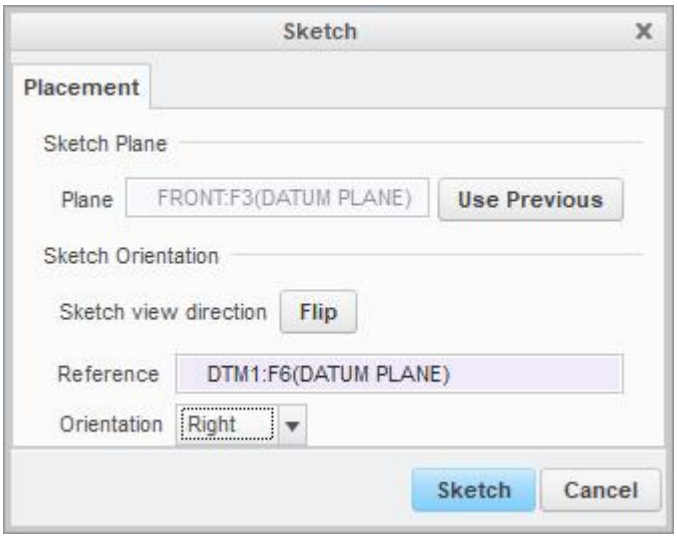
Notice center of arc is aligned to the end point of datum curve.

After completing the sketch, change the depth option to Symmetric ( ) and enter **4.8** as depth value.

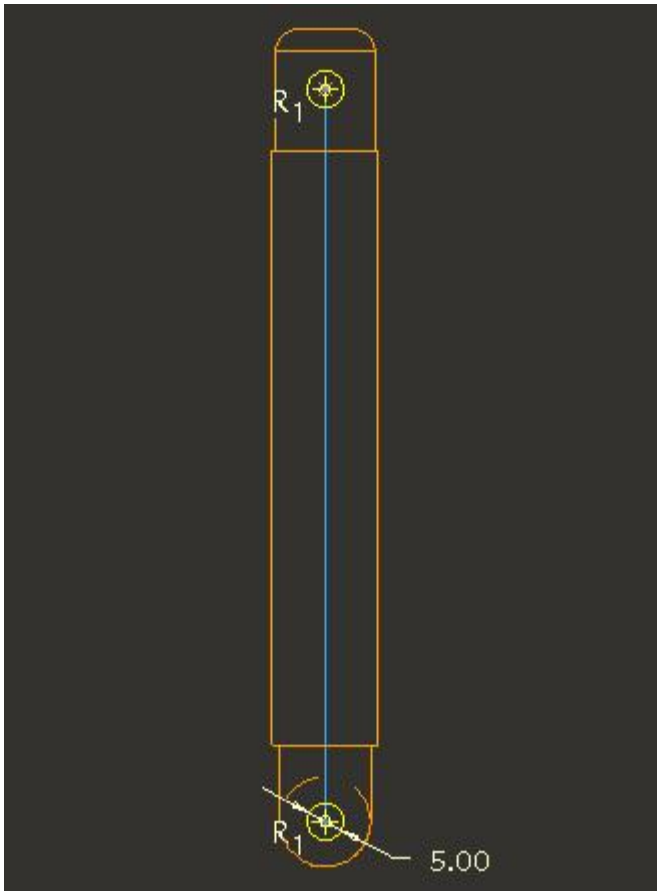
Pick icon to complete the feature. It will appear as shown below.

Again pick to invoke Extrude tool.

Select the sketching references as sown below.



Sketch two circles aligned to the end points of the datum curve as shown below.



After completing the sketch pick  icon to create the feature as cut.

Specify the **Through All** () depth option for both directions.

Pick  icon to complete the feature. It will appear as shown below.

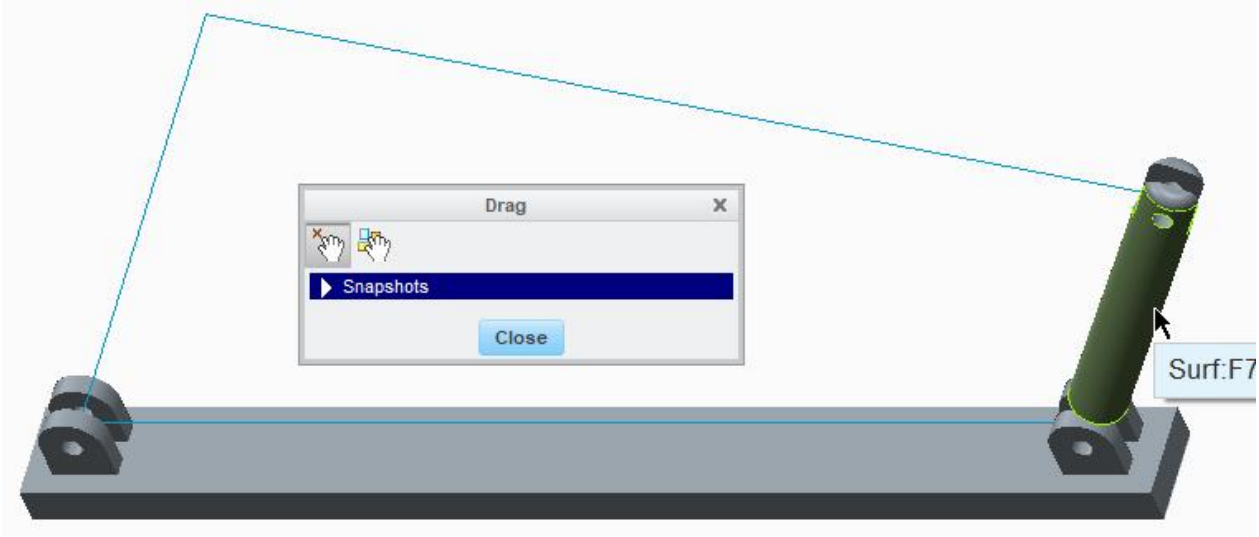
In the same way you can create geometry for other components.

Moving the Bodies

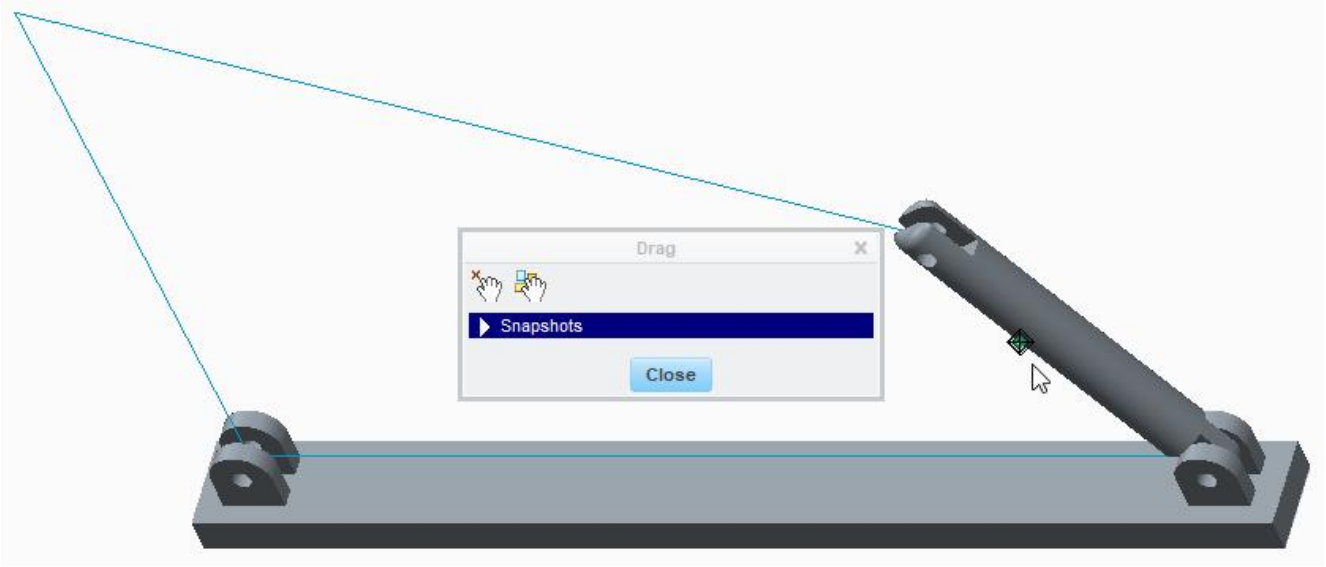
Now we will use the drag functionality to move bodies.

Pick the Drag Packaged Component icon 

Pick on any entity belonging to LINK1 as shown below.



Now move the mouse to drag the selected body. After moving towards left it should appear as shown below.



If the mechanism moves correctly it means you have successfully completed the exercise. You can complete other parts yourself.