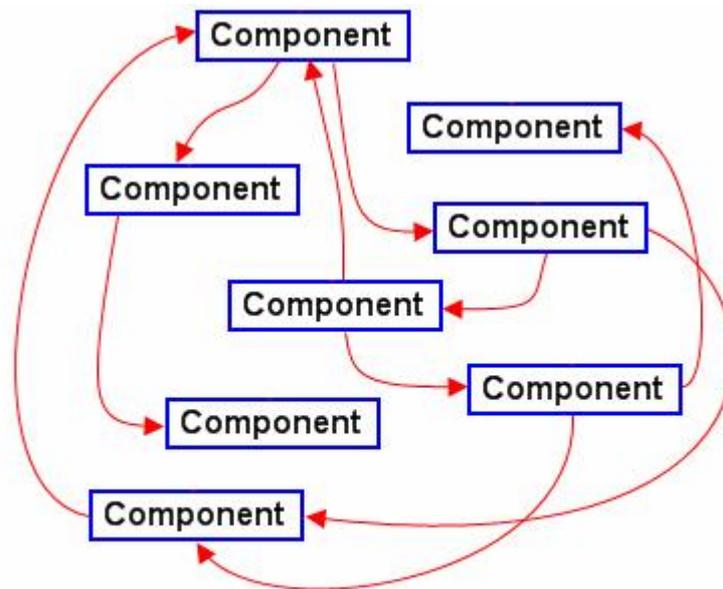


# Managing External References

External References (ER) are essential for a design project to ensure that components interface and fit together according to design intent. They make sure that models update automatically upon regeneration.

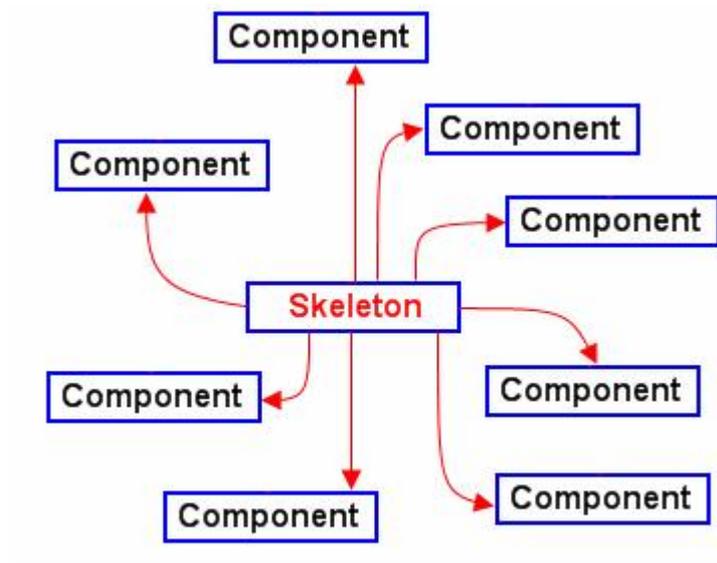
Generally, external references should be created only between a skeleton and a given model. External references between any other models are difficult to manage and should be avoided. Furthermore it is recommended that a model should copy references only from a skeleton part and skeleton should never copy references from any other model except another skeleton.

The following figure shows how difficult it becomes to manage all the relations when you create the references between parts. So this approach should never be used.



Wrong way of creating references

The following figure shows the recommended way of distributing references. In this case the project is properly structured and easy to manage. We always know where to look for controlling the interface and fit of different parts



Recommended way of creating references

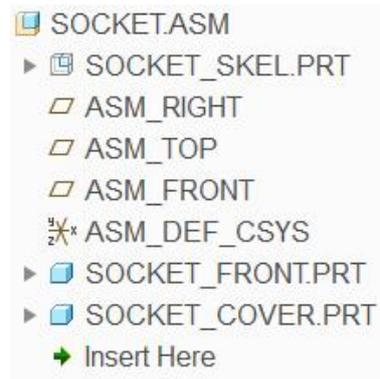
Special attention should be given while creating external references as if not managed properly external references may become bottleneck.

## Exercise 1

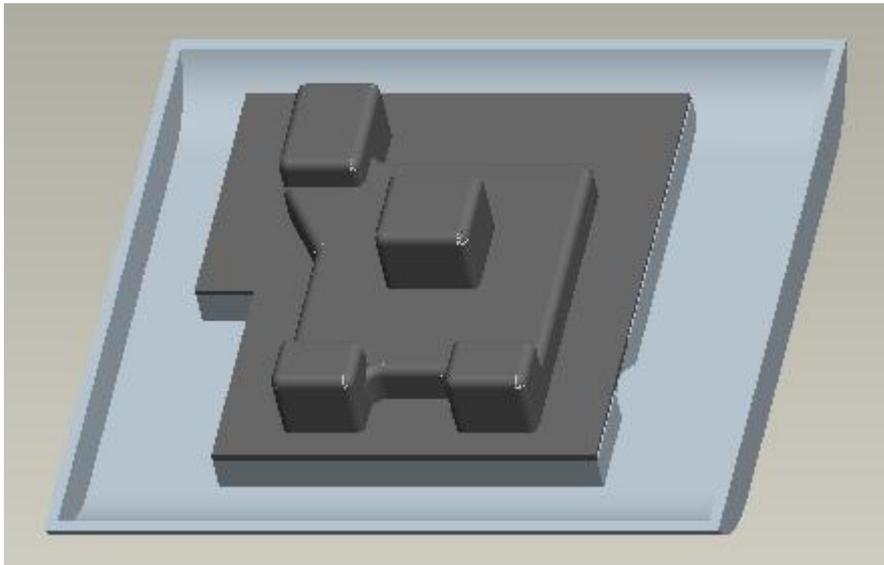
In this exercise we will see the preferred and wrong way (that is often tempting) of creating external references

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

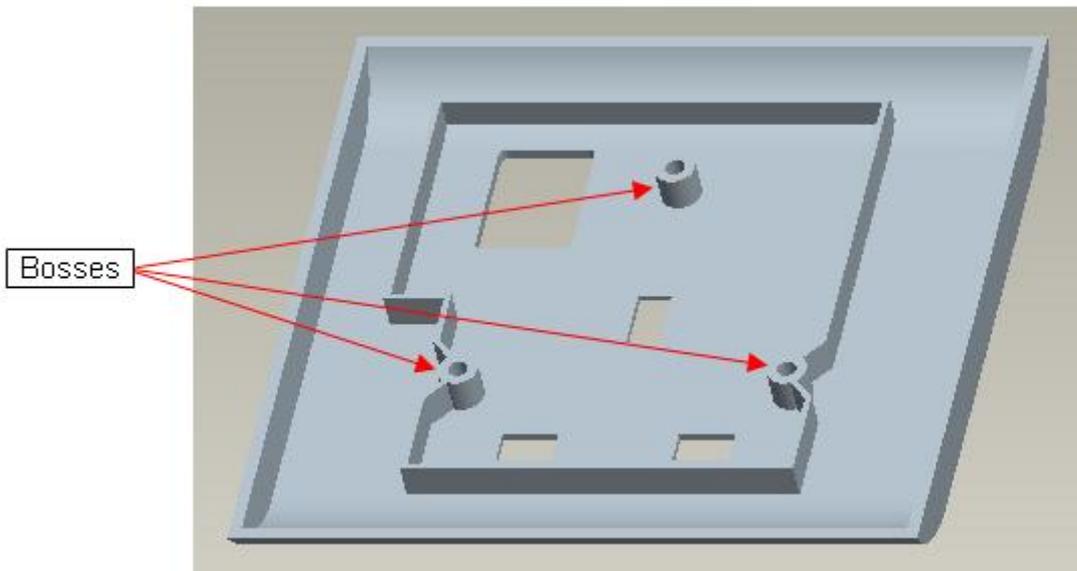
This assembly consists of two parts as shown below.



The assembly of both parts is shown below.



If you look at the SOCKET\_FRONT you will notice that it contains three bosses as shown below.



Now we will create three holes in the SOCKET\_COVER, aligned to the axis of bosses in SOCKET\_FRONT

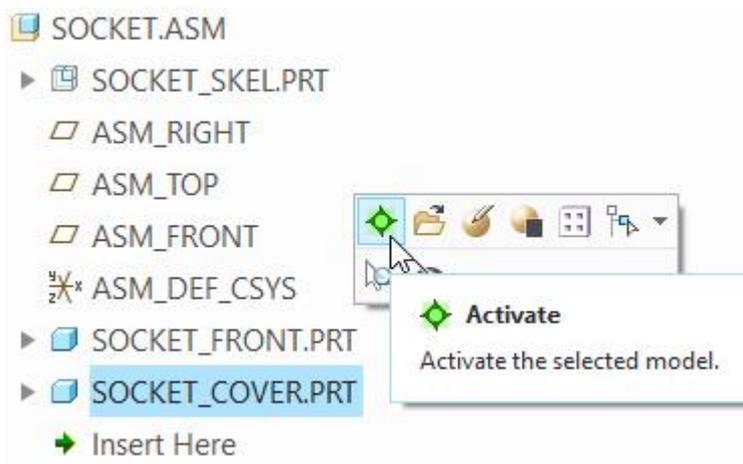
### ⇒ Wrong Way of Creating References

First of all we will show the wrong way to create the holes.

The most tempted way to create the holes is to create a cut feature in assembly mode and use the datum axis in the SOCKET\_FRONT part as reference.

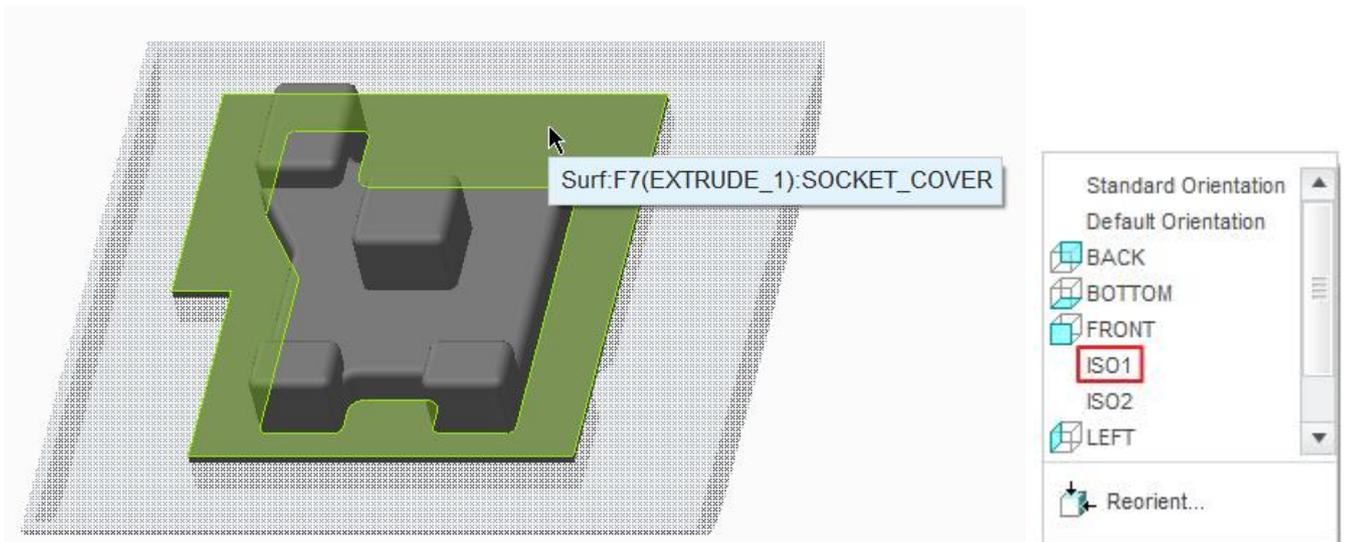
So we will create an extruded cut feature while in assembly mode.

Select the SOCKET\_COVER.PRT in the assembly window and pick **Activate**



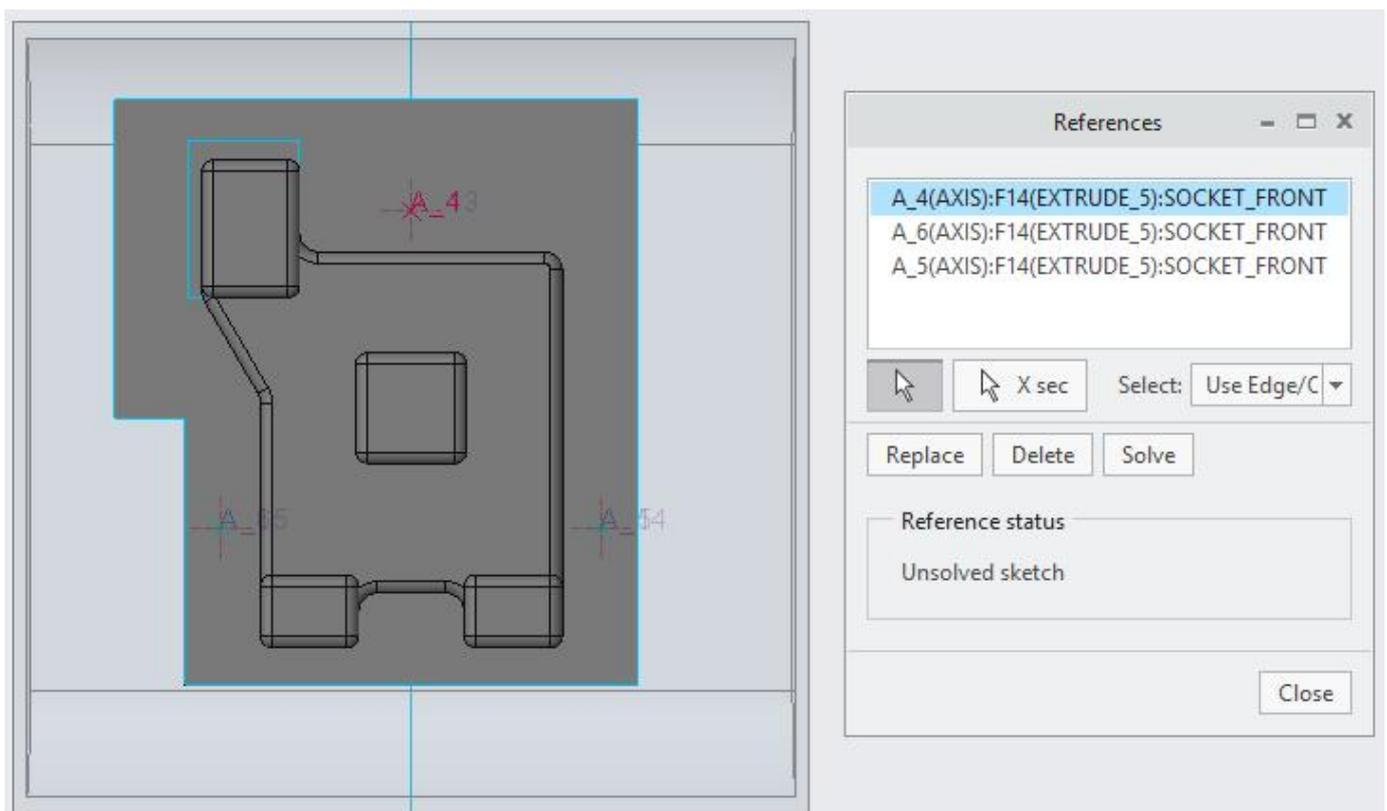
Pick  to invoke Extrude tool.

Select highlighted surface as the Sketch Plane

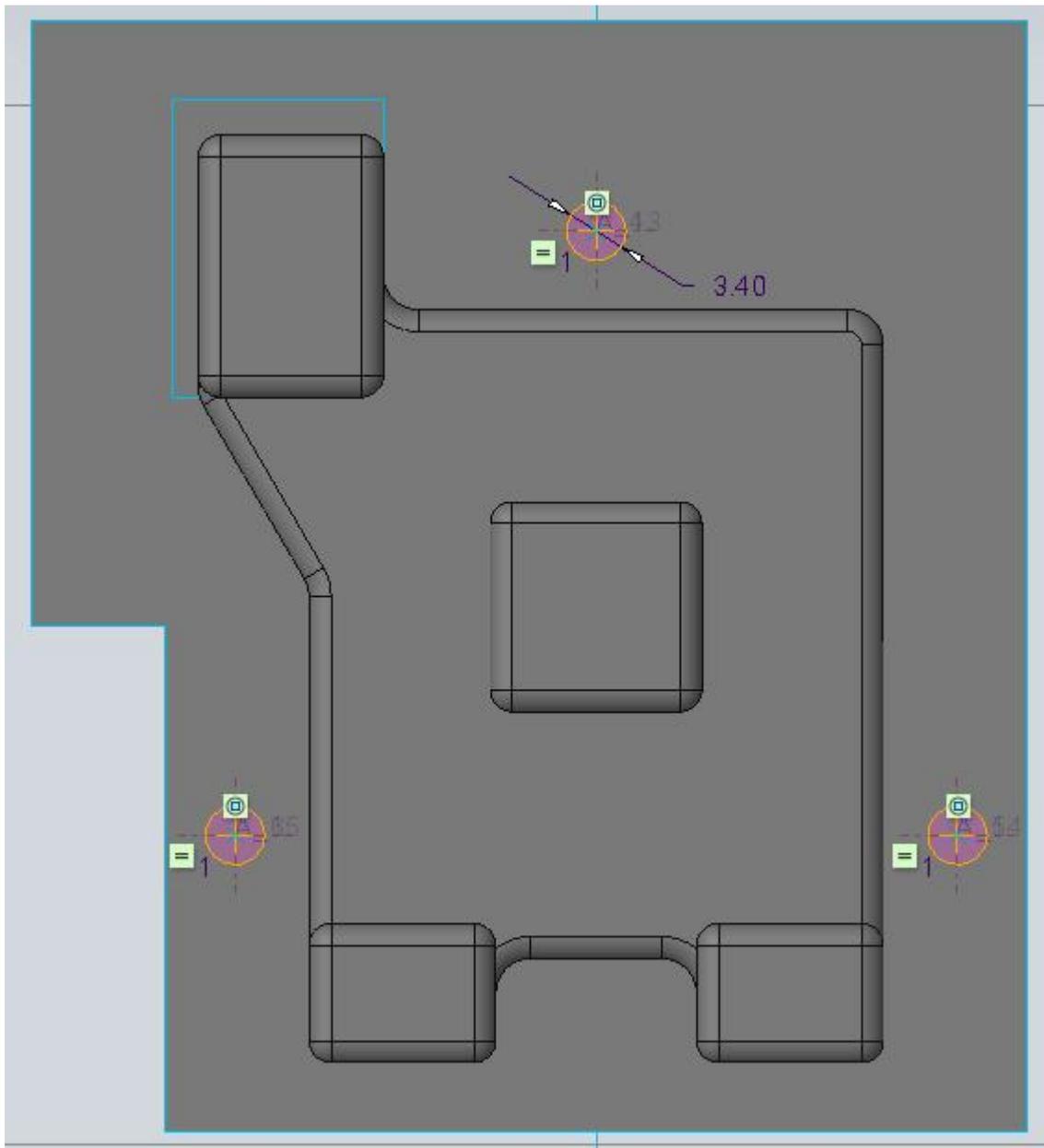


System will select the orientation reference automatically.

Select the three axis in the SOCKET\_FRONT as reference as shown below.



Now sketch three circles, their centers should be aligned to the referenced axis, as shown below.



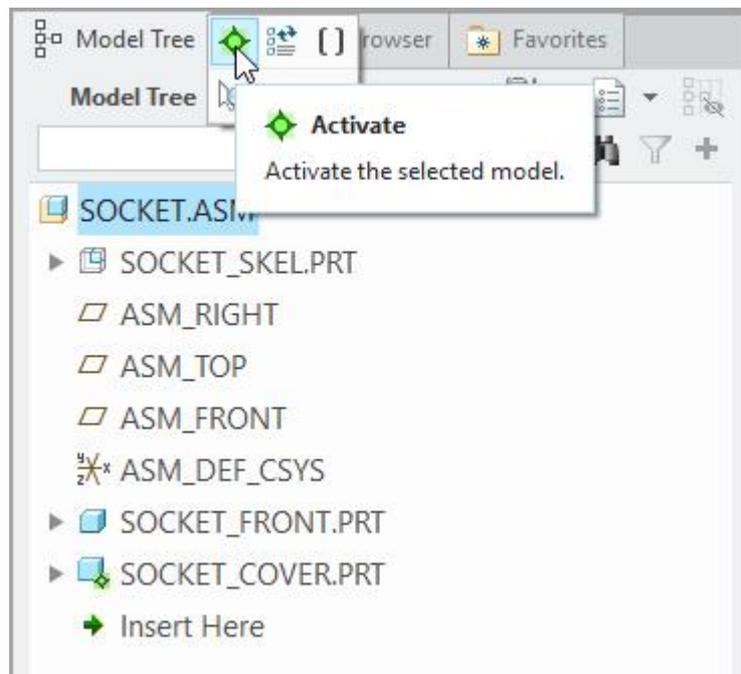
After completing the sketch pick  icon to create the feature as cut.

Specify the depth option to **Through All** icon in the Depth options list

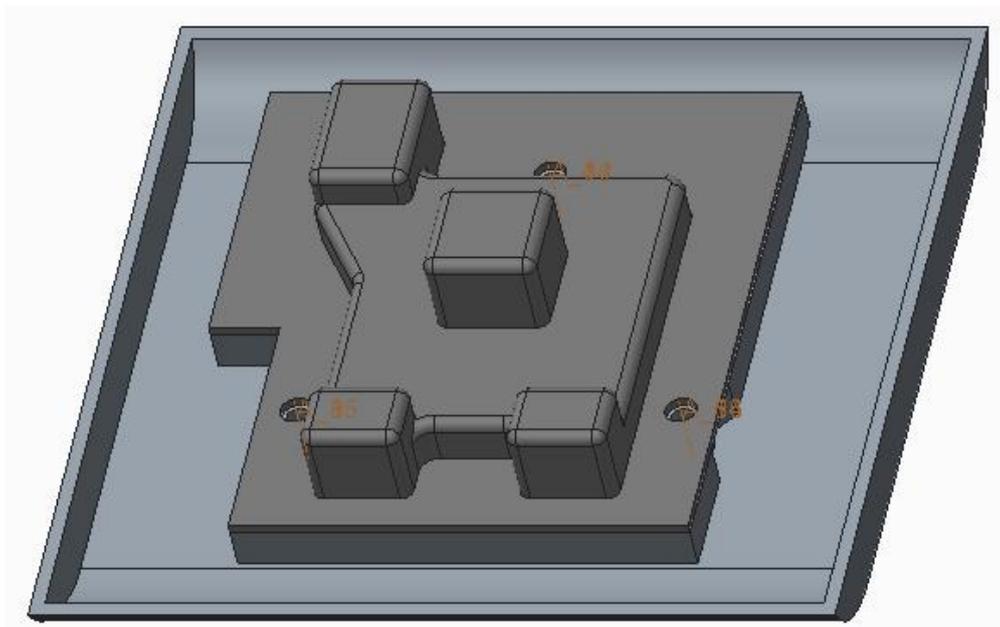
Pick the  icon to reverse the depth direction.

Pick  icon or middle-click to complete the feature.

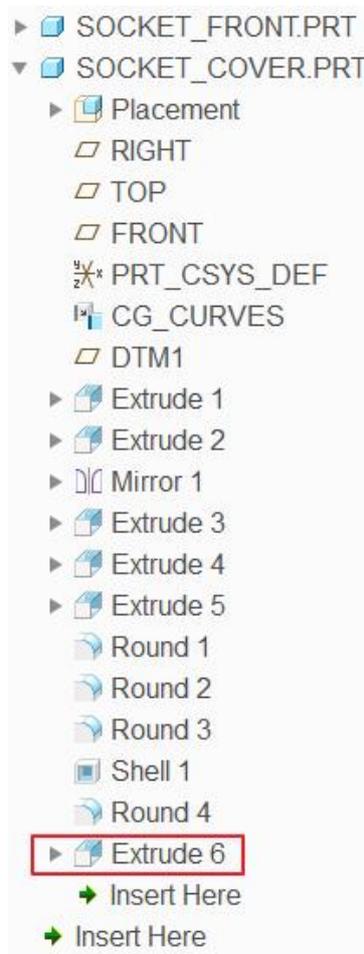
Select the SOCKET.ASM in the assembly window and pick **Activate**



The assembly will appear as shown below.

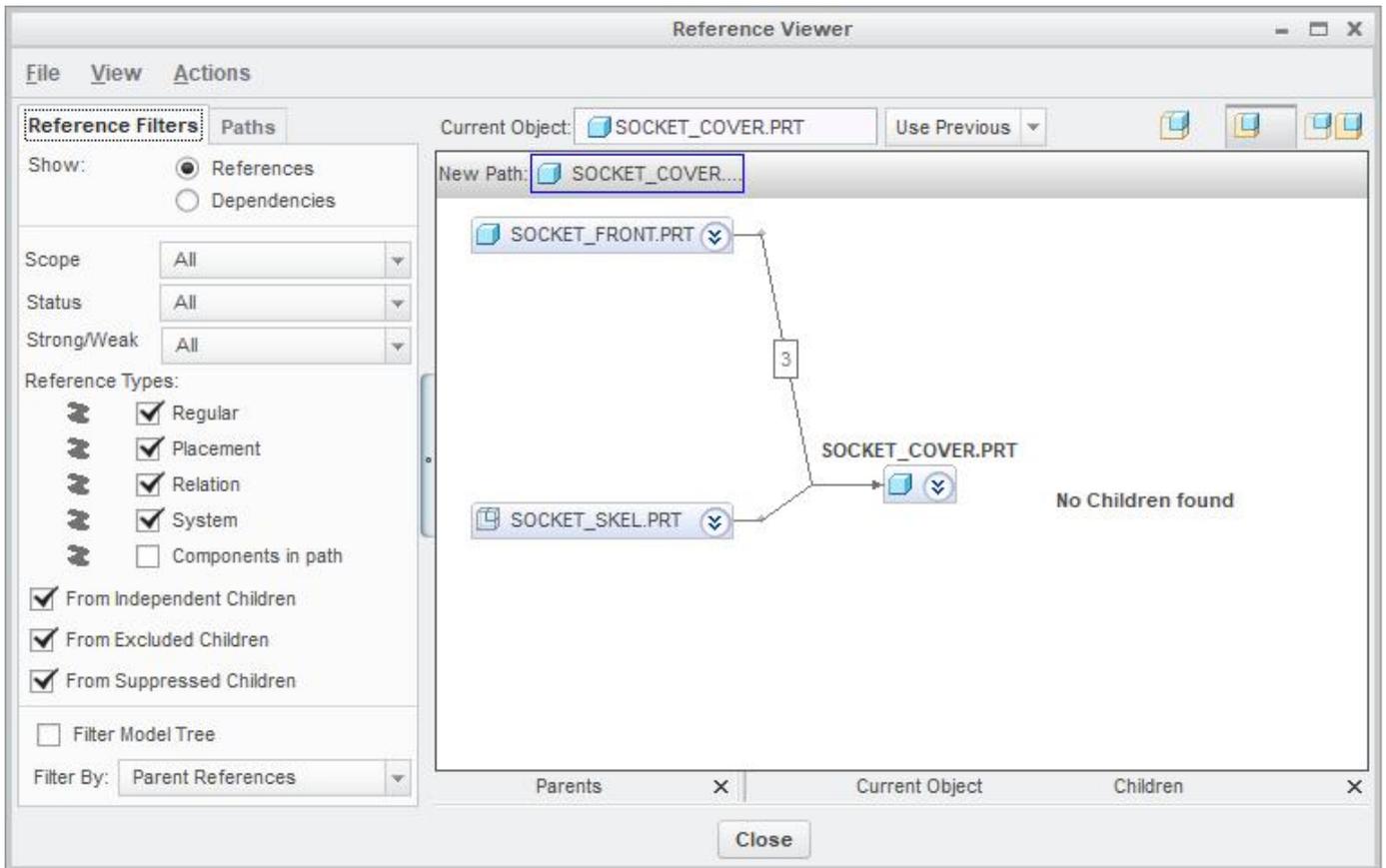


Look in the model tree and notice that a new feature appears in the SOCKET\_FRONT.PRT as shown below.



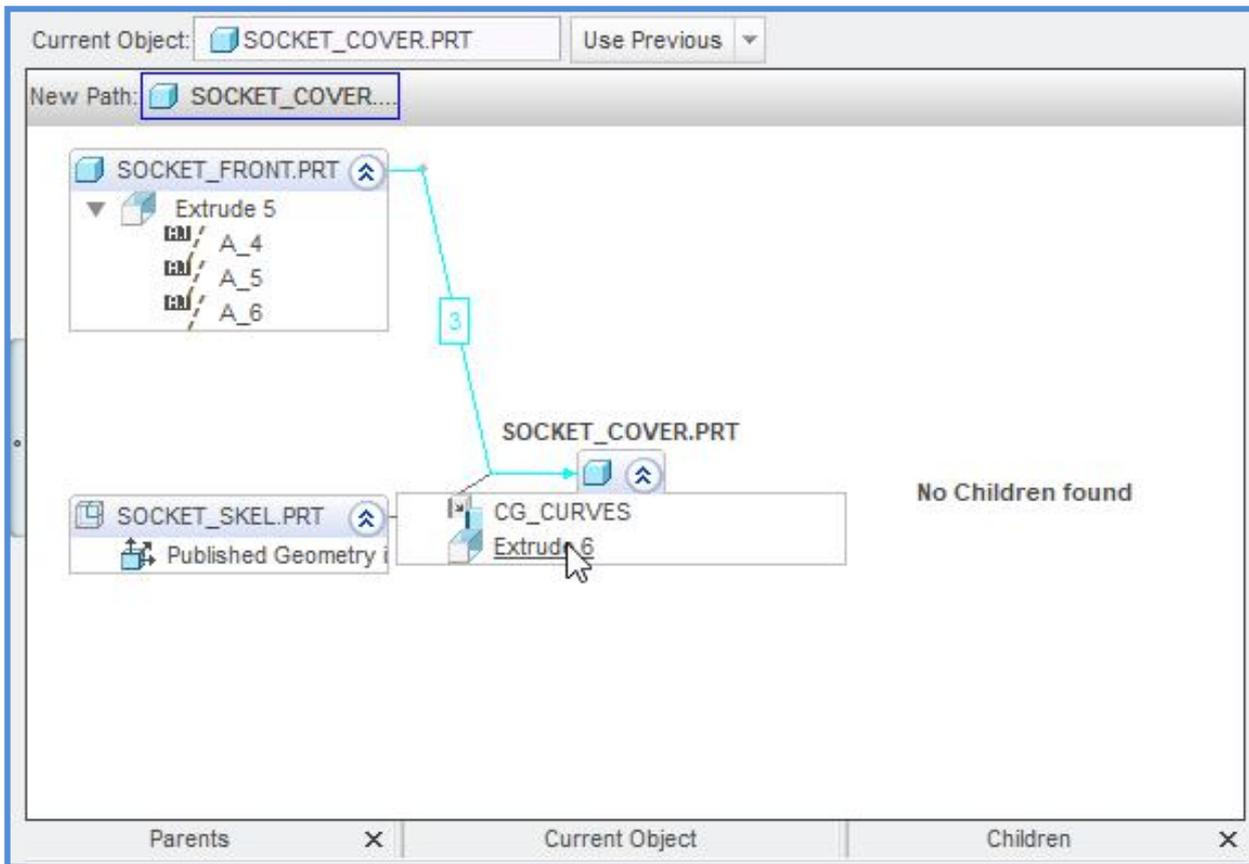
Right-click the SOCKET\_COVER.PRT in the model tree and pick **Information > Reference Viewer** to open the Reference Viewer.

Pick  to set the model as current object and system will show the dialog box as shown below.



It can be seen that SOCKET\_COVER has two parts as parents i.e. SOCKET\_FRONT and SOCKET\_SKEL. Furthermore current object is referencing three features in SOCKET\_FRONT

Now pick on  in front of SOCKET\_COVER.PRT to see the details. It can be seen that two features are referencing from other parts. If you place mouse pointer over “CG\_CURVES”, system will highlight the path toward SOCKET\_SKEL.PRT. If you place mouse pointer over the “Extrude 6” feature, system will highlight the path to SOCKET\_FRONT as shown in figure below.



The highlighting of path towards SOCKET\_FRONT implies that it contains the parents of this feature. Keep in mind that “Extrude 6” is the feature that we just created.

So we can say that one component of the assembly is referencing another component. This practice is called “Part to Part” referencing. Although it is a quick way to maintain fit between parts but it can become very difficult to manage especially when number of parts increases. You cannot keep track of all the references and relations between components. So it must be avoided.

External references should only be created to the skeleton model, and not to any other model.

Pick  to close the Reference Viewer.

## ➡ Breaking Part to Part References

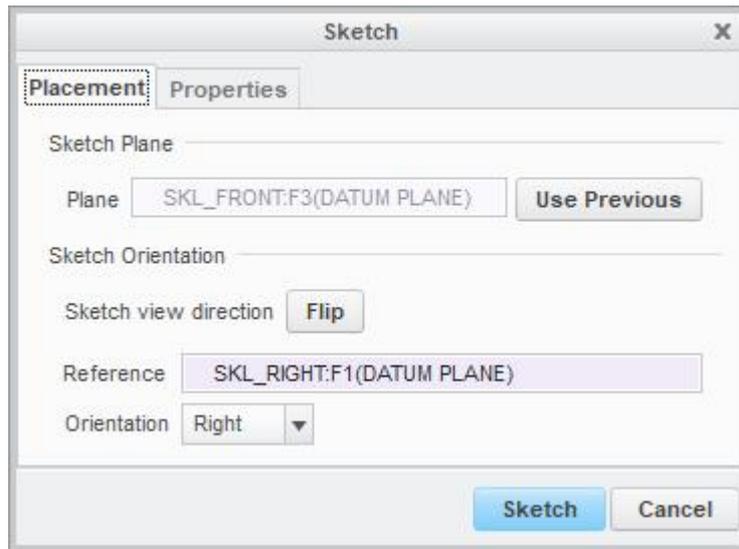
Now we will show you how to break the external references between parts and instead control their geometry from skeleton.

First we will create the datum points in the skeleton model and then use them as references to control the location of bosses and holes in individual parts.

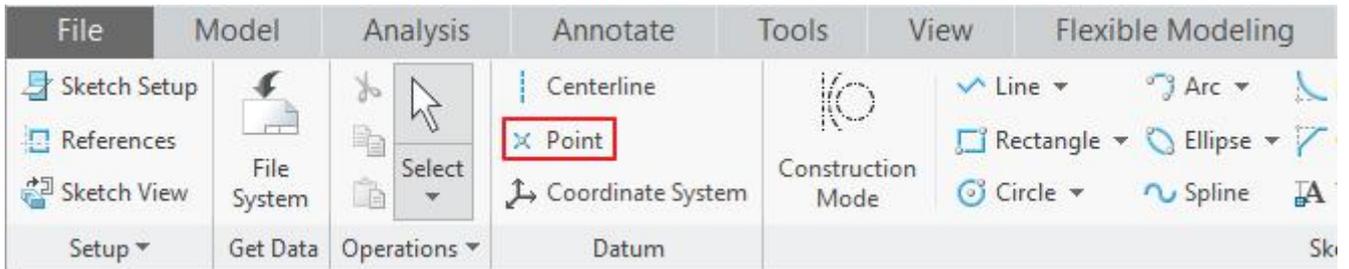
Open the skeleton SOCKET\_SKEL.PRT in a separate window.

We will create a sketch containing geometry points. So pick  on the Model tab.

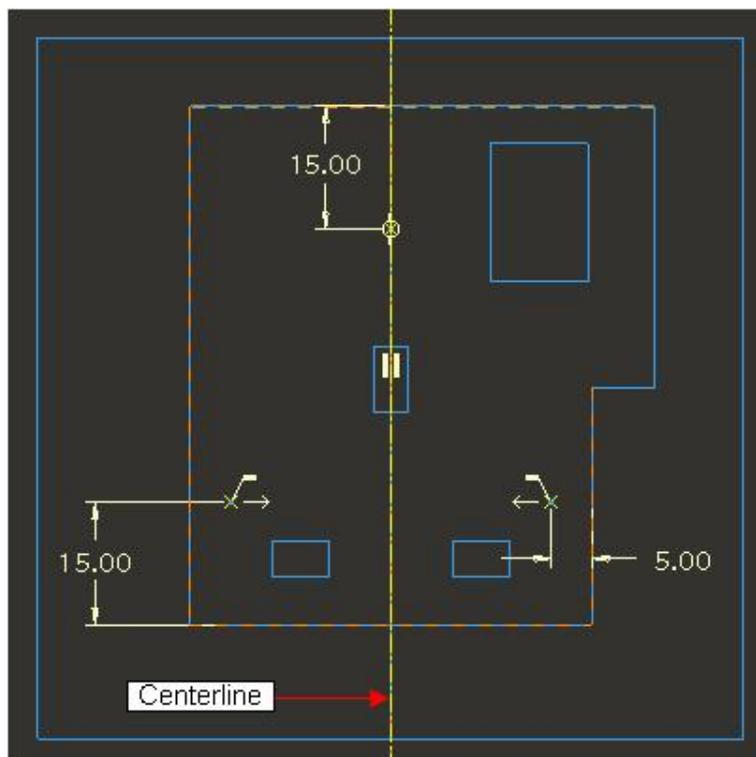
Select the sketching references as shown below



Pick Geometry Point icon 



Place three datum points and dimension them as shown below.

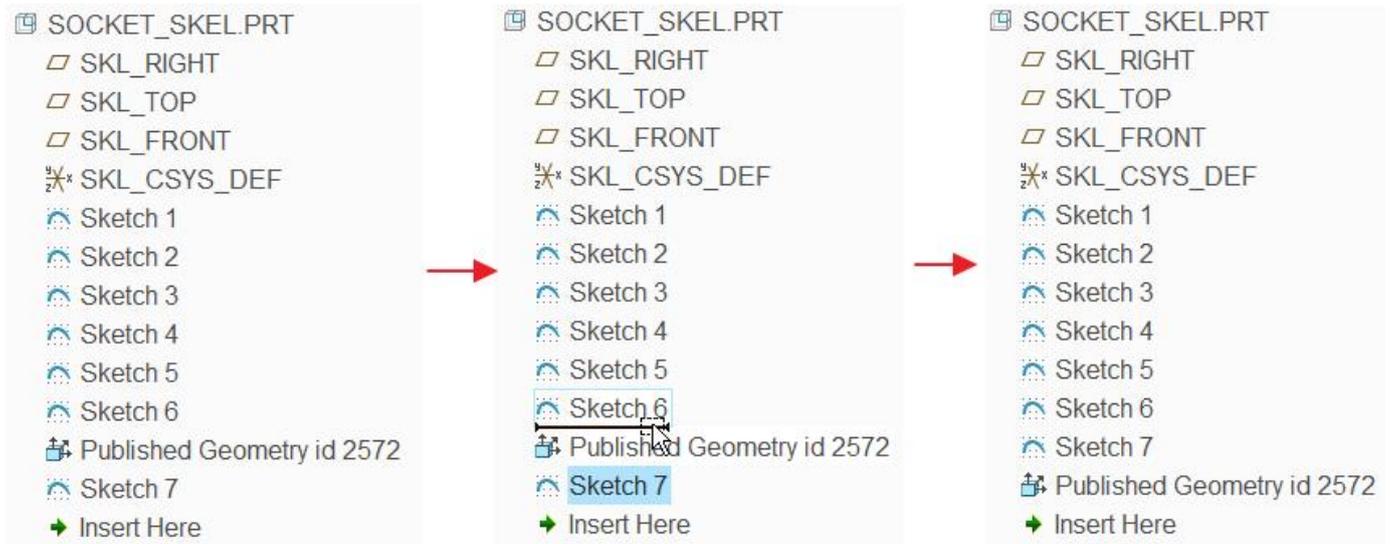


At the moment, don't worry about the exact dimensions. We will redefine this feature in the assembly mode and set the dimensions.

Pick  to complete the section.

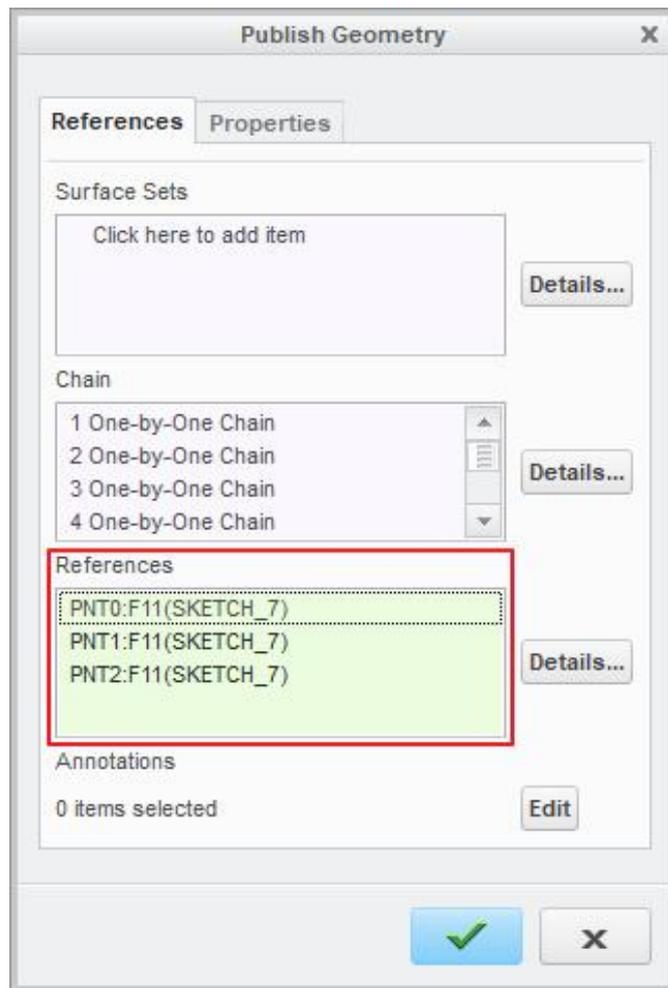
Now we will redefine the publish geometry feature and add these points. But first we should reorder this datum point feature to appear before the publish geometry feature.

So drag the "Datum Point" before the publish geometry feature as shown below.



Select the publish geometry feature and pick 

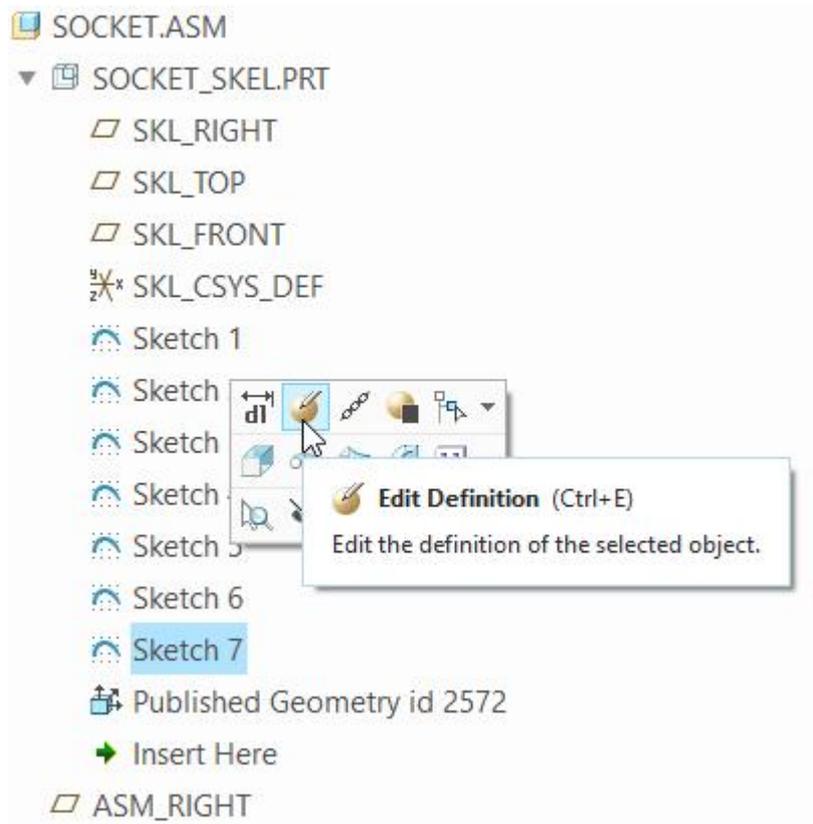
Activate the References collector and pick the three datum points, that we created above.



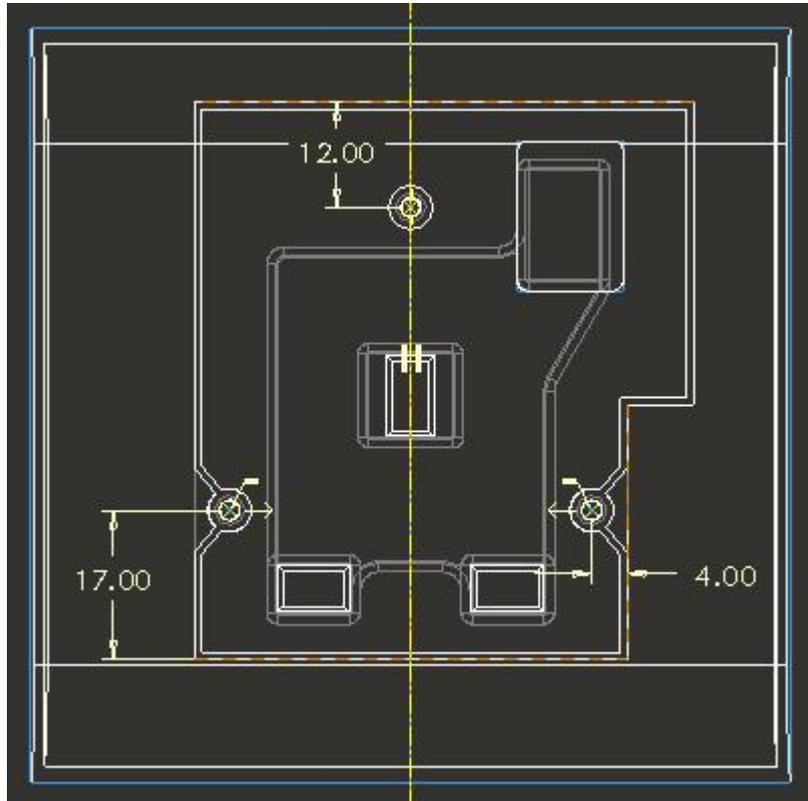
Pick  to apply the changes and exit the dialog box.

We assume that locations specified for the bosses in SOCKET\_FRONT represent design intent. So we will modify dimensions of the points in the skeleton so that they confirm to the design intent.

Switch to the assembly model then select newly created sketch feature and pick  as shown below.



Now change the dimensions as shown below.



Notice that now the points lie exactly over the centers of the bosses. Also observe that we are not using any reference from any other part. We are just visually aligning the points to the existing axis. We can achieve this intent by other methods e.g. measuring the distances of the

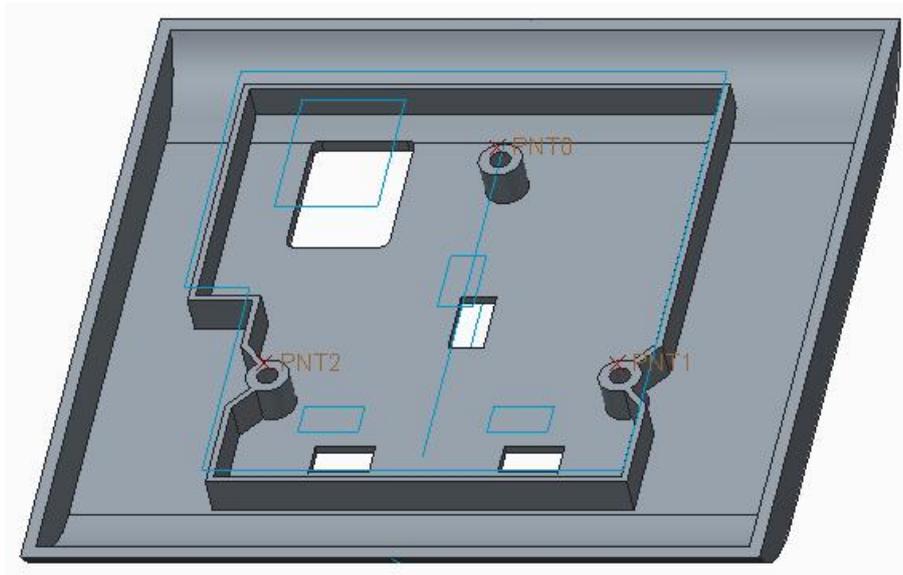
axis from some references and then modifying the dimensions. But this technique seems more robust for this case.

Pick  to complete the section.

Unhide the 07\_\_\_PRT\_COPY\_GEOM layer in the assembly model.

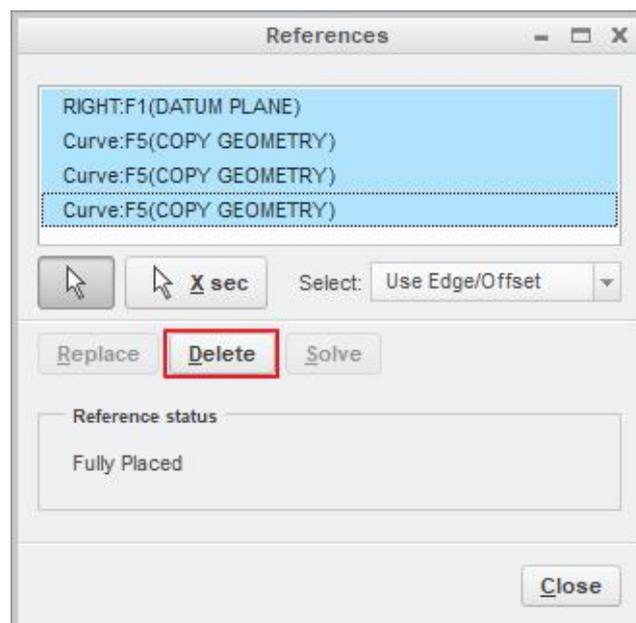
Now open SOCKET\_FRONT.PRT in separate window and regenerate the part by picking .

Notice that the datum points that we just added in the publish geometry appears in this part as shown below. (Make sure that layer 07\_\_\_PRT\_COPY\_GEOM is not hidden)

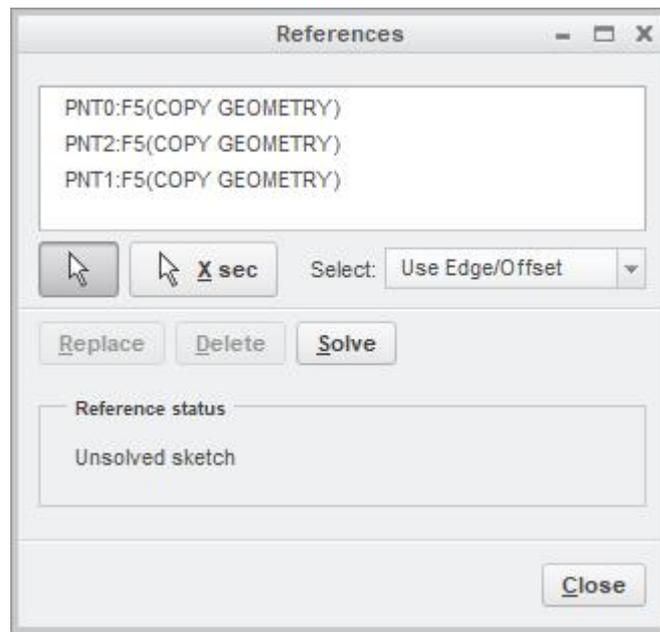


Select the “Extrude 5” feature in the model tree and pick .

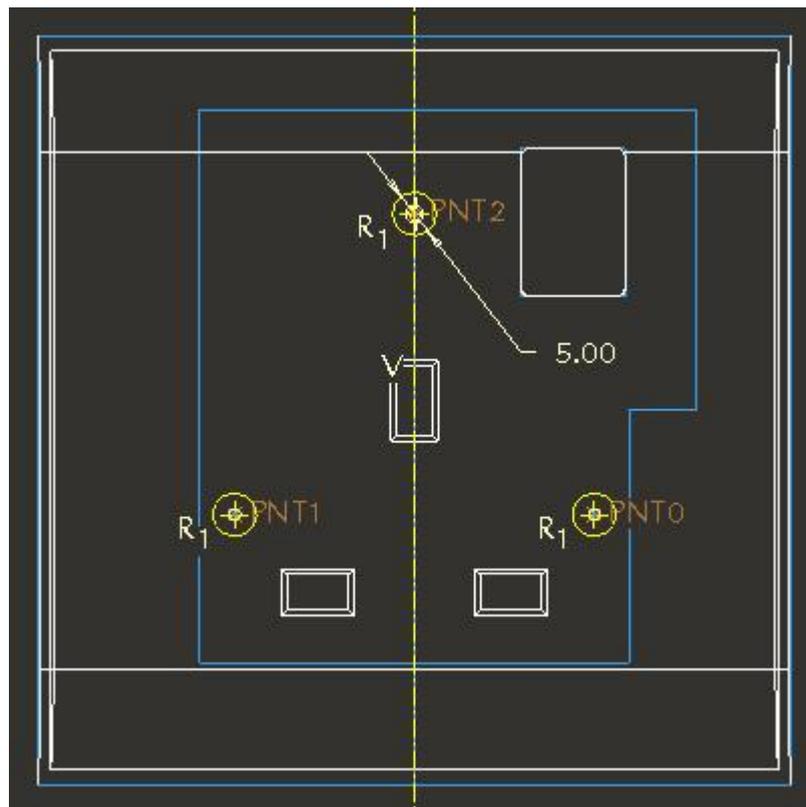
In the sketcher mode, delete all the existing references as shown below.



Pick the datum points as new references as shown below.



Now pick **Coincident** and align the centre of circles to these new references. You should delete any other constraints if they conflict with new constraints. Now the sketch will appear as shown below.

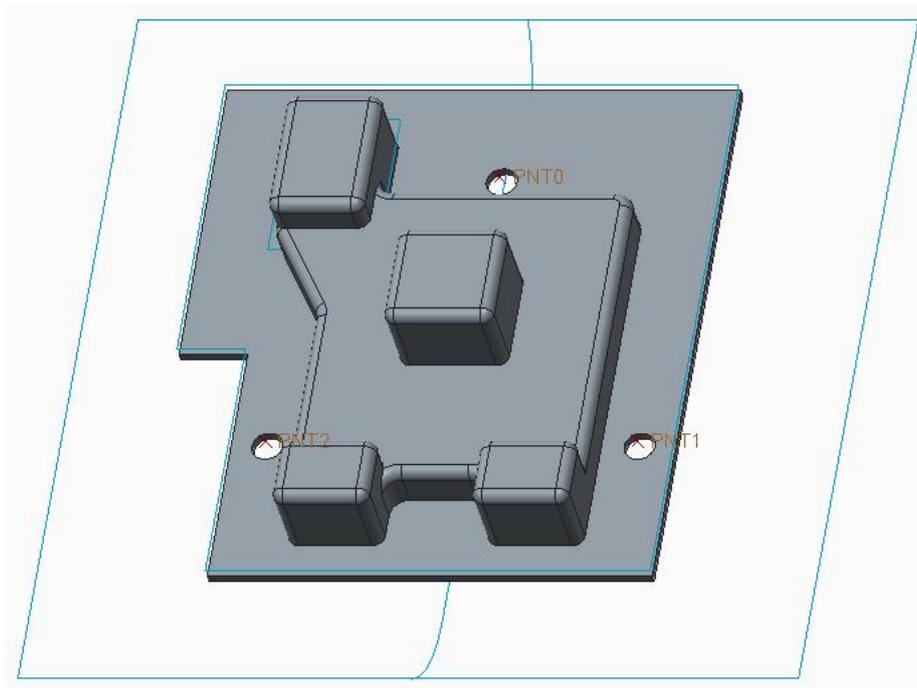


Pick  to complete the section.

Pick  to apply the changes and exit the dialog box.

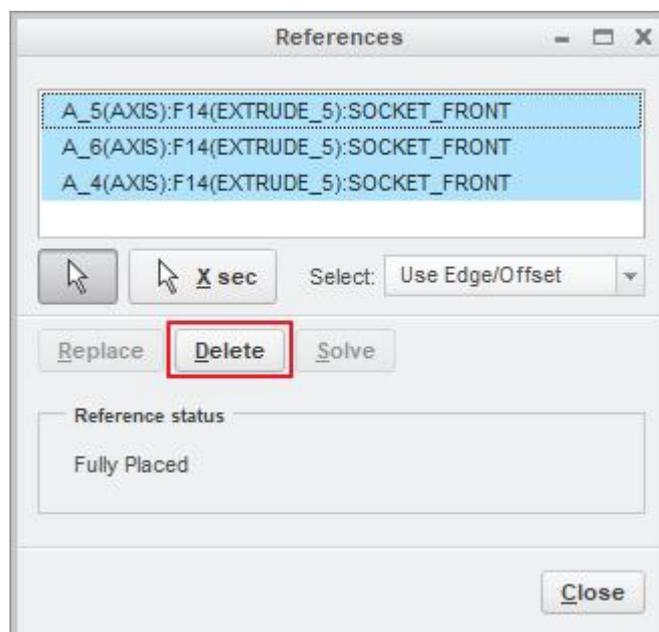
Now open the SOCKET\_COVER.PRT in separate window and regenerate the part by picking .

Notice that the datum points that we just added in the publish geometry appears in this part as shown below. (Make sure that layer 07\_\_PRT\_COPY\_GEOM is not hidden)

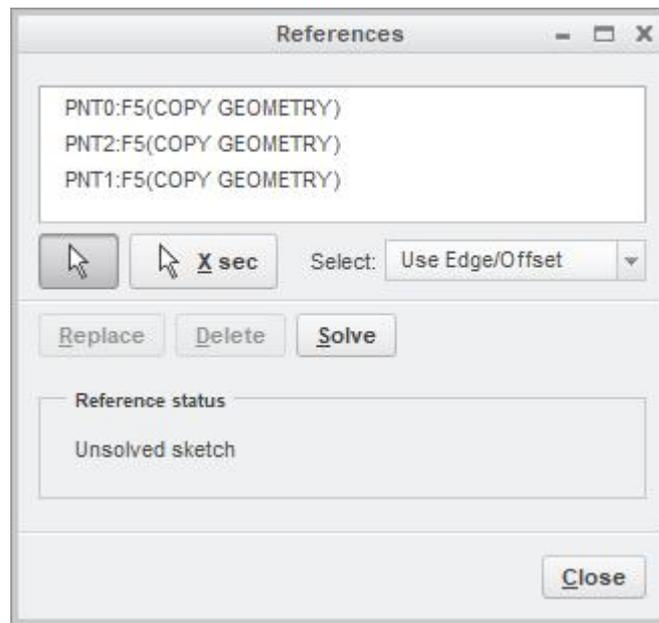


Select the “Extrude 6” feature in the model tree and pick 

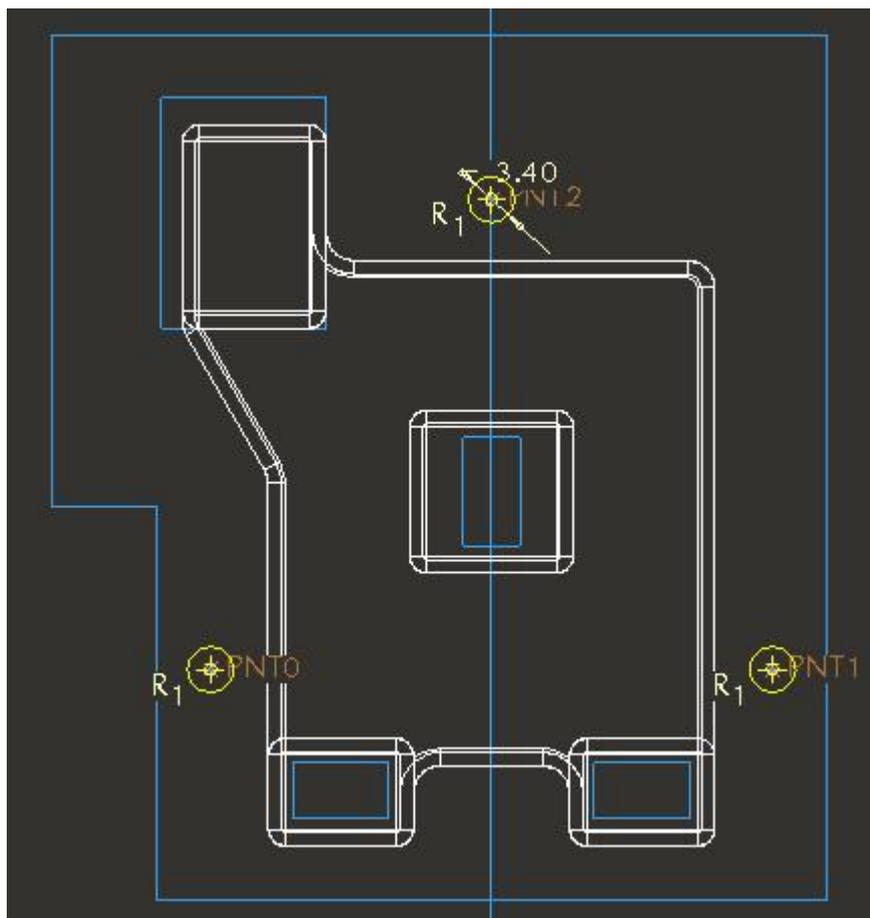
In the sketcher mode, delete all the existing references as shown below.



Pick the datum points as new references as shown below.



Now align the centre of circles to these new references. You should delete any other constraints if they conflict with new constraints. Now the sketch will appear as shown below.



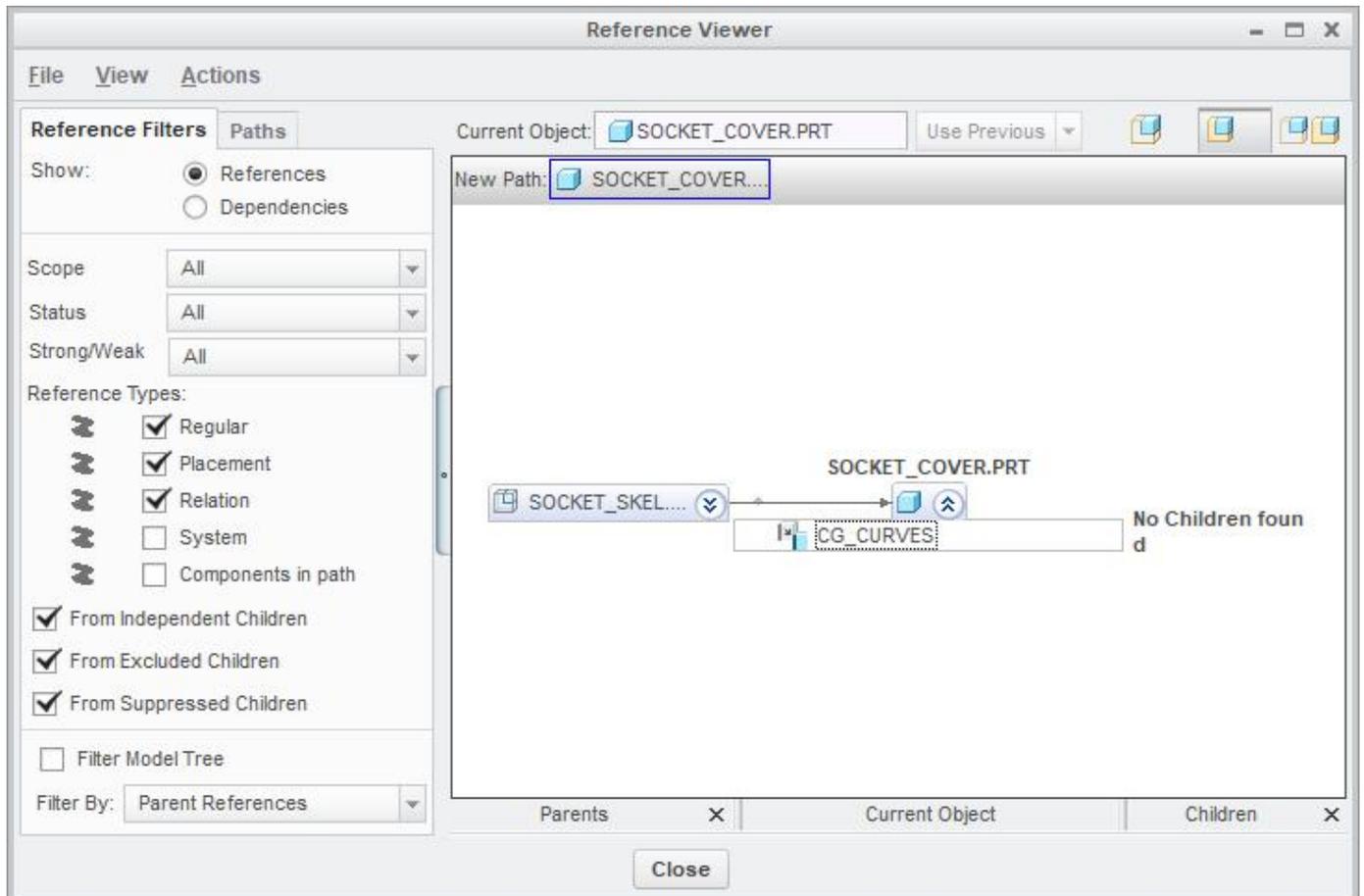
Pick  to complete the section.

Pick  to apply the changes and exit the dialog box.

Switch to the assembly model

Right-click SOCKET\_COVER.PRT in the model tree and pick **Information > Reference Viewer** to open the Reference Viewer.

Pick  to set the model as current object and system will show the dialog box as shown below.



It can be seen that SOCKET\_COVER has only skeleton part as parent. Furthermore only the Copy Geometry is the feature with external references.

[External references should only be created to the skeleton model, and not to any other model.](#)

Pick  to close the Reference Viewer.