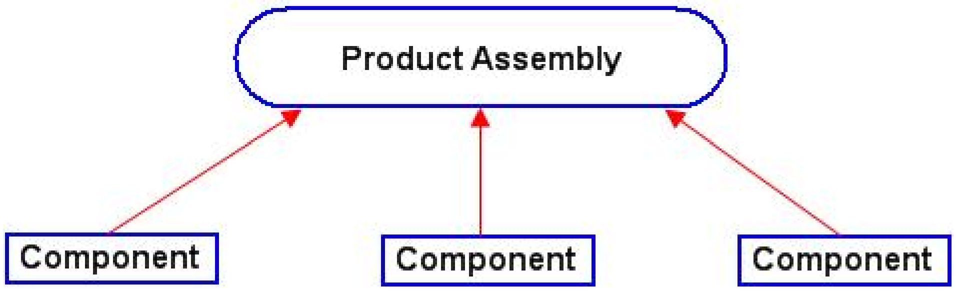
Introduction

While designing real-world products, mere the knowledge of solid and surface modeling capabilities of Creo Parametric is not sufficient, because a product consists of a lot of parts and they have to interface well with each other. So each component must perform its intended purpose within the limits of other components. It should fit well with other components and not interfere with them and meet the design intent. To achieve these goals two design methodologies are used.

1. Bottom-Up Design
2. Top-Down Design

Bottom-Up Design

It is a design methodology in which each component is designed as a separate part without any reference to other components in the assembly.



It is a manual approach to ensure that components fit properly and meet the design criteria. First the individual parts are designed in great detail. The parts are then assembled together to form assemblies of a product. Components are independent of each other. So that if one component is modified, the related components have to be changed manually.

Errors are manually identified and modifications to each component are made to make the adjustment

Advantages

1. You can jump start your product design

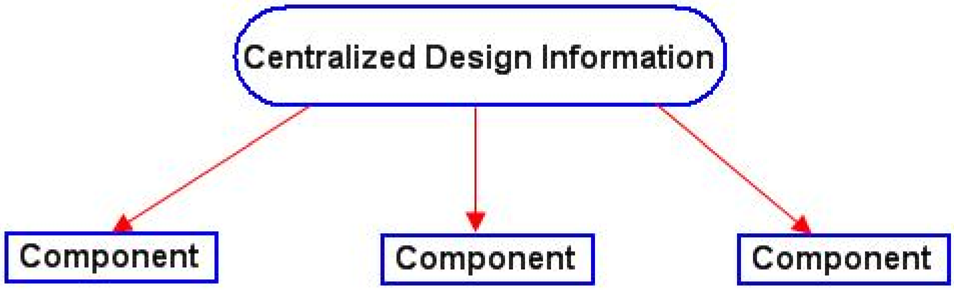
Disadvantages

1. Detecting the problems related to form and fit is a difficult process
2. Considerable amount of time is consumed for fixing individual components.
3. Components in an assembly are dependent on each other so you cannot remove, suppress, or replace a component without affecting other components.

Top-Down Design

It is a design methodology that starts from the highest level of a design concept and proceeds towards the lowest level of detail.

You start with the broad project specification in mind and put that information in a centralized location. Then you progress from this information to the individual parts.



Top-down design (TDD) approach allows to control the relations of components with each other so that if one component changes the related components change automatically. If implemented properly you can have full control of the assembly at a central location.

It provides a mechanism for breaking complex design projects into manageable units so that multiple teams can work on a project simultaneously.

Advantages

1. You can make changes from a single location that will propagate to all levels of the design
2. All of components will fit seamlessly in the final product.
3. Design information is at a central location so design teams can work on a project concurrently.
4. A component in the assembly can be removed, suppressed, or replaced without affecting other components.

Disadvantages

1. Top-down design requires more work in the beginning of a project although it saves time at the end.

Basic Elements of Top-Down Design

A top-down design system consists of following basic elements

1. Central Location for Design Data
2. Data Communication Tools

Central Location for Design Data

In top-down design methodology, the known project specification and information is placed at a centralized location. This information can be copied from this location to any component in the project assembly. This makes sure that all the components are based on same design criteria. So if some specification or information related to the design is changed in the central location, all the components referring this will also update.

Placing information in a central location is not a one-time process. You start a project by placing known information in central location and then work on individual components and assemblies. As the design develops you get more insight into the project and more information and constraints become known. Then you put this information and constraints into central location and refine your components and assemblies. So this is an iterative process.

Creo provides us two types of objects to capture the design data at a central location.

a. Notebook
b. Skeleton Model

a) Notebook

A notebook can be described as an engineering notebook where we can define parameters and relations related to a project. It maintains design intent in a central location.

Part and assembly models can both access these parameters and information in a notebook. In addition to parameters we can also create 2D sketches that represent the conceptual design of our product. Then we can annotate these sketches and show how the parameters are related to the conceptual design.

b) Skeleton Model

A skeleton model is a special type of model that acts as a central location for storing design criteria relating to a project assembly, specifically surface geometry, curves, and datum features. When the skeleton changes, the components connected to it also change.

A skeleton model, like any other assembly component, has features, layers, relations, views, and so on.

The functional characteristics of skeletons are different from other components. Therefore a skeleton is identified by a different icon in the Model Tree. Skeleton models do not show up in the Bill Of Material (unless you specifically include them) and they do not contribute to mass properties.

A skeleton model can only be created in an assembly but it can be retrieved, manipulated, and saved as ordinary parts. When you regenerate an assembly, skeleton is regenerated before all other components and assembly features.

There are two types of skeletons

1. Standard Skeleton
2. Motion Skeleton

Motion Skeleton is used when we want to incorporate motion into the design project at the beginning. So in the initial chapters we will only discuss Standard Skeleton. Keep in mind that Standard Skeleton is usually called only the Skeleton.

Data Communication Tools

Creo provides us following tools to communicate the design references or information from the skeleton to individual components

a. Copy Geometry Feature
b. Publish Geometry Feature
c. Inheritance Feature
d. Merge Feature
e. Shrinkwrap Feature

a) Copy Geometry Feature

The Copy Geometry feature is used to pass any type of geometric reference information to and from parts, skeleton models, and assemblies. You select individual references that may include curves, surfaces and datum features.

When you create Copy Geometry feature, you can reference either individual references like curve, surface and datum features or a Publish Geometry feature.

You cannot copy solid features with copy geometry features.

The part from which reference geometry is copied is called Source and to which is copied is called Target.

There are two types of copy geometry features.

1. Internal Copy Geometry
2. External Copy Geometry

1. Internal Copy Geometry

The internal Copy Geometry feature copies geometry from model to model in the context of the assembly. It follows the entire path within the assembly between the target part and the source part. We need all the assemblies, in the path of source and target parts, in session to regenerate the internal copy geometry feature.

Internal Copy Geometry Feature is usually called only the "Copy Geometry Feature".

2. External Copy Geometry

The External Copy Geometry functionality copies geometry from model to model without copying the geometry in the context of the assembly. Dependency on the assembly and all models along the path between the two components is avoided. Source and target components must be relatively positioned, but are independent of the assembly context.

We need only source and target parts in session to regenerate the external copy geometry feature.

b) Publish Geometry Feature

Publish geometry feature (PGF) allows to mark the references, in a source part, that can be referenced by a copy geometry feature altogether. So it is a consolidation of multiple local references that can be copied to other models.

This method makes it easy for the designer working on a model (usually skeleton) to identify which features are stable and appropriate to use as a reference in the downstream models.

When you create an Internal Copy Geometry or External Copy Geometry feature, you can reference a Publish Geometry feature.

It is a good practice to give unique names to Publish Geometry features. This will make it easier to determine which model or lower level skeleton, a Publish Geometry feature is intended for.

c) Merge Feature

A Merge feature allows associative propagation of complete geometry of a source part to target part.

The part from which reference geometry is copied is called Source and to which is copied is called Target.

Merge feature copies the complete geometry of source part to the target part including all surfaces and solid geometry.

Merge feature is usually used to add or subtract the material of one part to or from another part. It can also be used in place of Copy Geometry features when we need to reference the complete geometry of a source part.

d) Inheritance Feature

An inheritance feature allows associative propagation of geometry and feature data from a source part to target part. An inheritance feature begins with all of its geometry and feature data identical to the reference part from which it is derived.

The source part is the original part and the target part contains the Inheritance features.

Although an inheritance feature begins with all of its geometry and data identical to the reference part, from which it is derived, but you can identify the geometry and feature data that can change in the inherited feature without changing the source part

e) Shrinkwrap Feature

A Shrinkwrap feature allows associative propagation of exterior surfaces of a source model to target part.

You can use a part, skeleton, or top-level assembly as the source model for a Shrinkwrap feature.

A Shrinkwrap feature is a collection of surfaces and datum features of a model that represents the exterior of the source model. Shrinkwrap feature is associative and automatically updates to reflect changes in the parent models.

The Shrinkwrap feature gives an accurate representation of the exterior of a model without the need for having the entire model in memory so less memory is required.

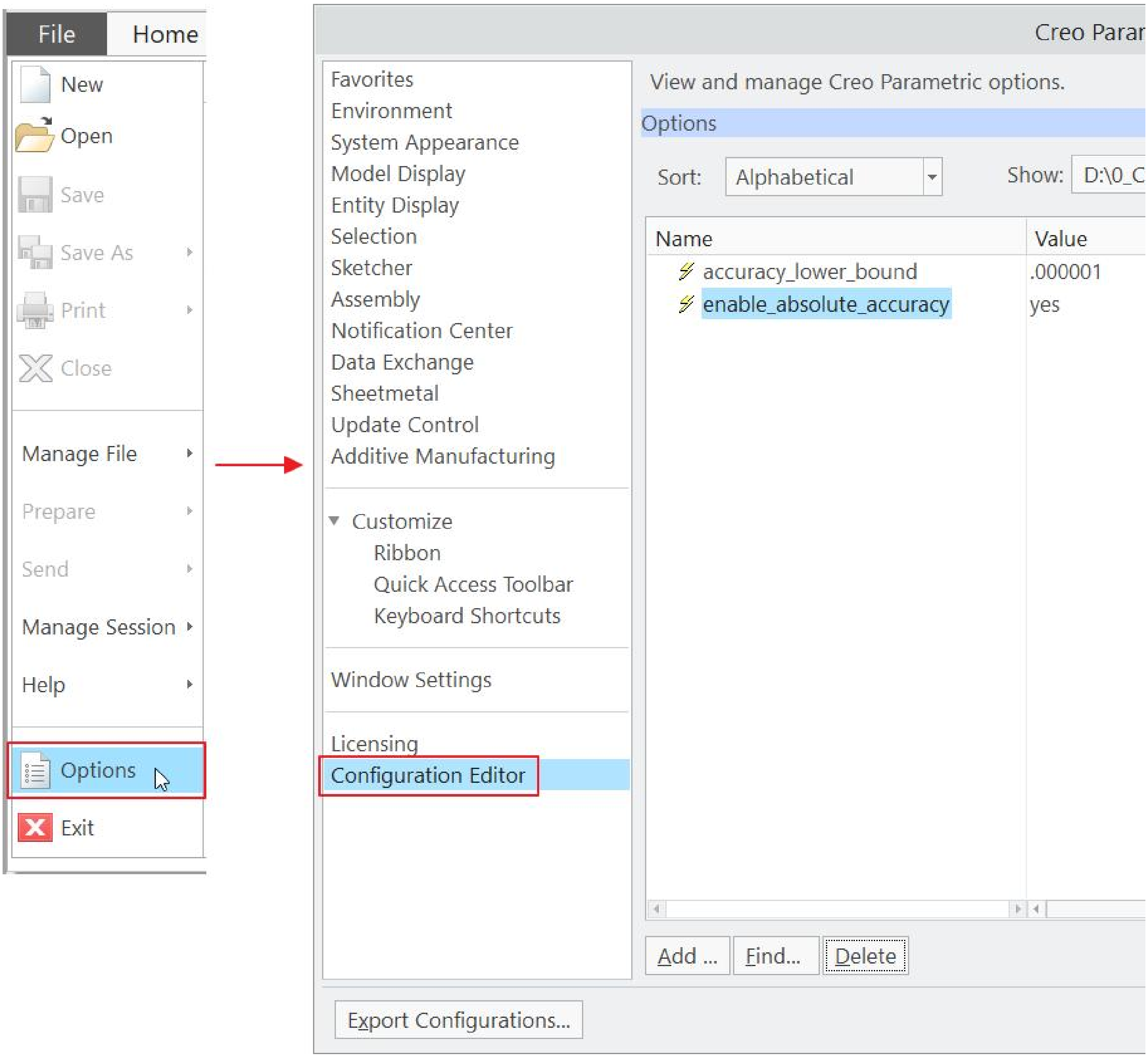
Before Starting the Exercises

1. Before starting the exercises, it is recommended to set the following Configuration file options

   enable_absolute_accuracy Yes

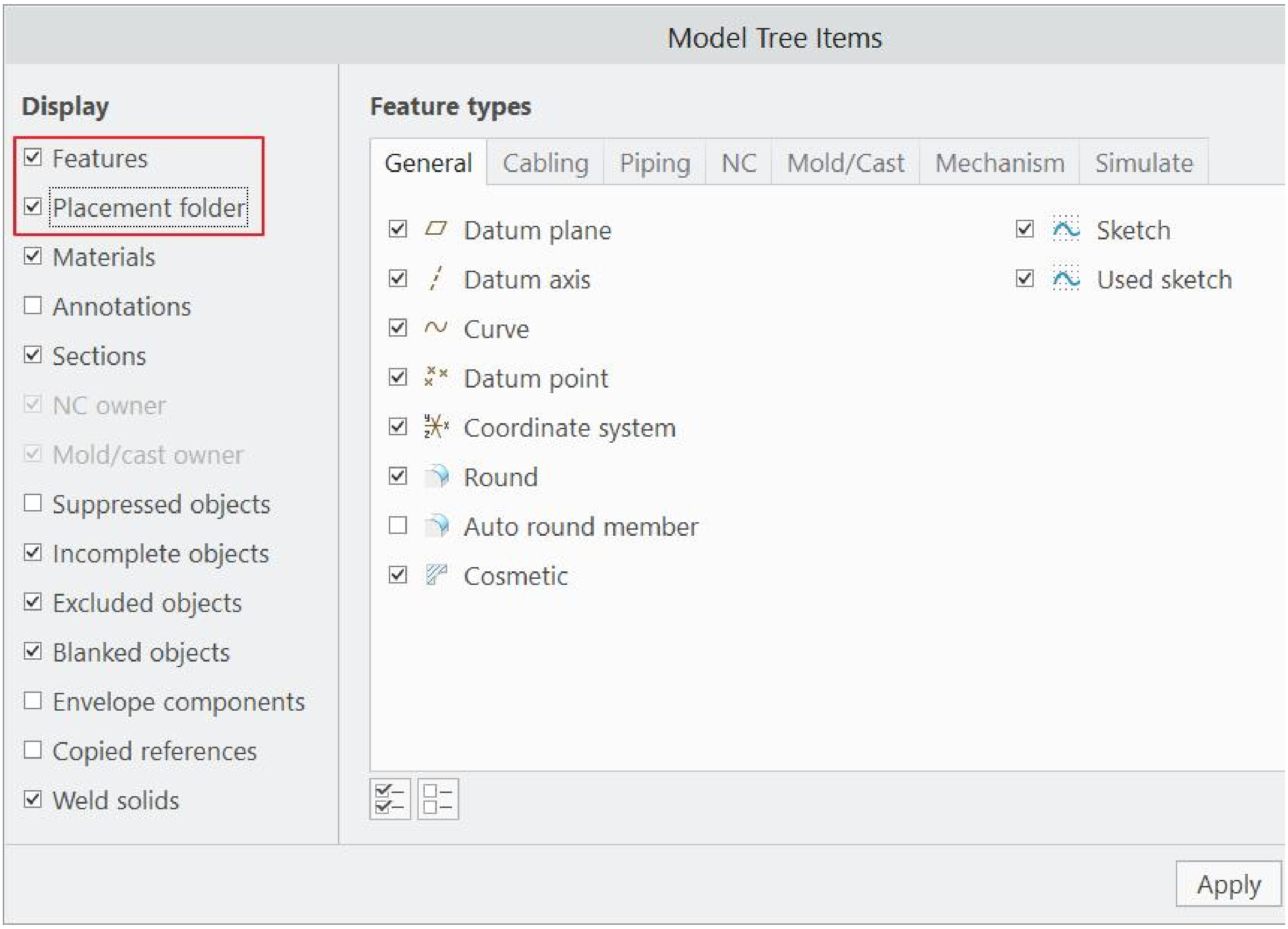
   accuracy_lower_bound 0.000001

   These option can be set by picking **File > Options > Configuration Editor** as shown below.

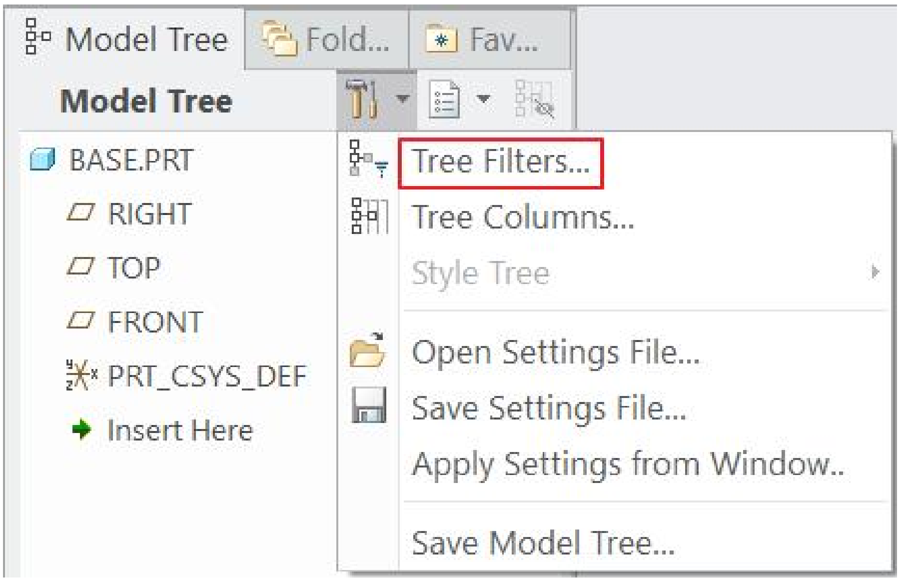


accuracy_lower_bound value is used to override the default lower accuracy limit of 0.0001

2. To display the features in the Model Tree, make sure that Features option is checked in Model Tree Items dialog box. Also make sure that Placement folder option is checked to view the placement constraints



This dialog box can be accessed at



Displaying the features in the Model Tree is necessary as most of the work will consist of creating features.

Typical Flow For Top-Down Design

There can be a number of ways to progress through the top-down design process of a product. A typical workflow is

1. Define Assembly Structure
2. Build Skeletons
3. Communicate Design Information
4. Create model geometry

1. Define the Assembly Structure

Usually the first step in the definition of a top-down project is to create the main assembly, sub-assemblies and related components (without any geometry). This creates the basic structure of a product and individual tasks or assemblies can be assigned to individual designers or teams.

By creating the assembly structure you can easily manage the project and identify the areas of the assembly that still need to be worked on. This also lets the other members of the team to know what the assembly structure of the product is. You can create the virtual BOMs at the start of the project that will help to monitor the overall progress of the project.

2. Build Skeletons

After creating the assembly structure, the next step is to create the skeleton model (or models). A skeleton model can only be created in an assembly. The system always inserts the newly created skeleton before all other components and assembly features. So no matter, how many components exist in an assembly, skeleton model will appear at the top of the model tree.

Tip: There may be situations when you want to create the skeleton geometry before defining the assembly structure. In this scenario you can create the desired geometry and features in a standard Creo component. Later, while creating the skeleton model, you can copy all the features of that component in the skeleton.

Keep the following points in your mind while creating the geometry in the skeleton.

1. The geometry that affects or crosses more than one component (or has to be referenced by more than one component) should be placed in the skeleton.
2. When some geometry or feature is only related to a single component in the assembly then there is no need to place a reference for it in the skeleton.
3. All the geometry that can cause the failure of a shell feature in a component should be created in the component after shell and not in the skeleton model.

3. Communicate Design Information

After creating the geometry in the skeleton, the next step is to communicate that design information to individual components (or skeletons in the sub-assemblies) in the assembly. This makes sure that all the components confirm to same design criteria and update as we make changes in the skeleton. Creo provides different tools to communicate this information to individual components. The most widely used are the Publish Geometry and Copy Geometry features. These features give you the flexibility to select the geometry from the skeleton that you need for a specific component. Other features i.e. Merge and Inheritance copy the complete geometry of source component so the target component becomes very heavy. Therefore Merge and Inheritance features are usually used to create the derivative models.

4. Create Model Geometry

Once the information is captured from the skeleton model into the components, you can use the Copy Geometry feature as a reference to build parts. Thus parts are dependent on the geometry of the skeleton and hence will update to the changes in the skeleton. Thus a designer can easily control a design project from a central location.

Exercise 1

In this exercise we will learn how to create a simple assembly of parts that are driven by a skeleton model.

Here we will use the following simplified approach for our design project.

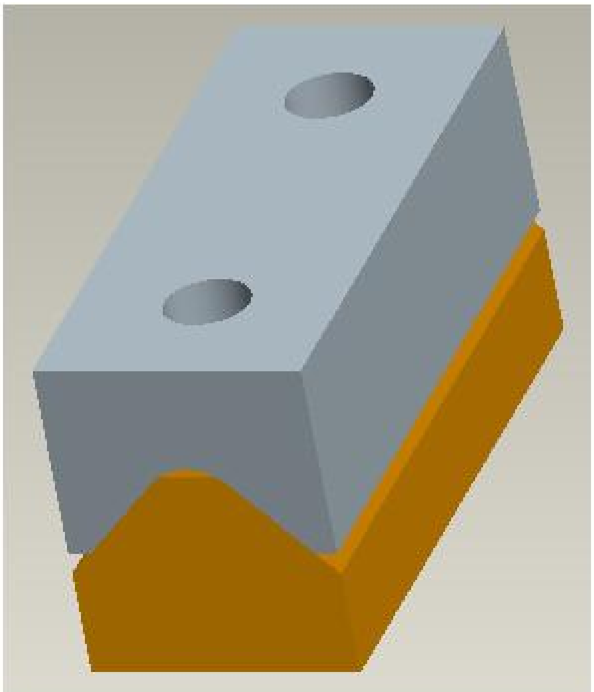
1. Creating a Skeleton
2. Assembling the components
3. Using Copy Geometry feature to communicate references
4. Creating model geometry

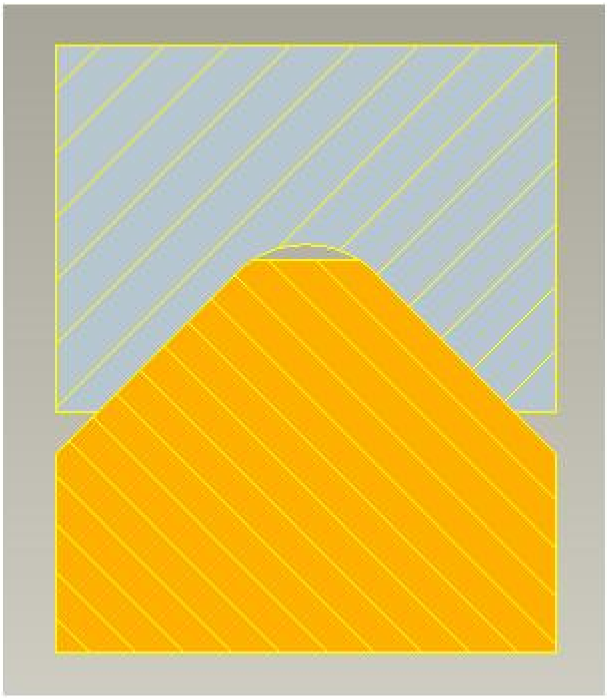
The purpose of this exercise is to introduce you to the Top-Down design process with the help of a very simple example.

Set working directory to the BLOCK_SET folder and open the assembly GUIDE_BLOCK.ASM

Notice that the assembly consists of only default datum features as shown below.

We will assemble two empty components and then create the geometry that will be driven by a skeleton. After completing the components, the assembly will appear as shown below.





Before starting a design project, you should determine what the relationship is or common attributes between the components. For the components shown in above figure, following relationship is desired:

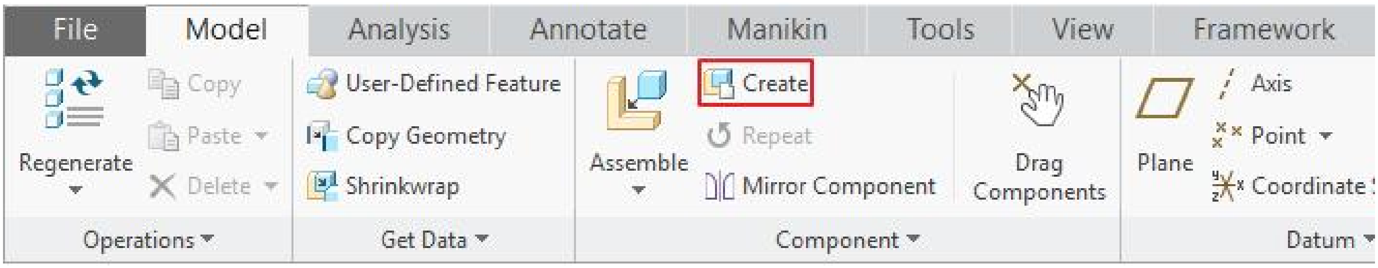
1. Both components should be equal in length and width.
2. The shape of the mating surfaces should be same.
3. There should be two holes in each component. The diameter of holes should be same and their axis aligned.

Such information that affects more than one component should be placed in the skeleton model.

Creating the Skeleton

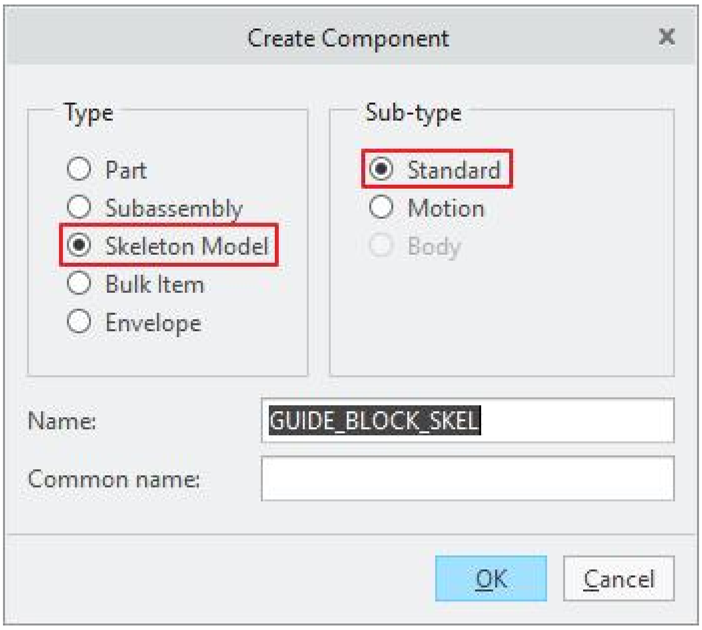
Now we will create the skeleton model in the assembly.

Pick the Create a component icon ()

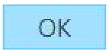


Skeletons models can only be created in an assembly

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

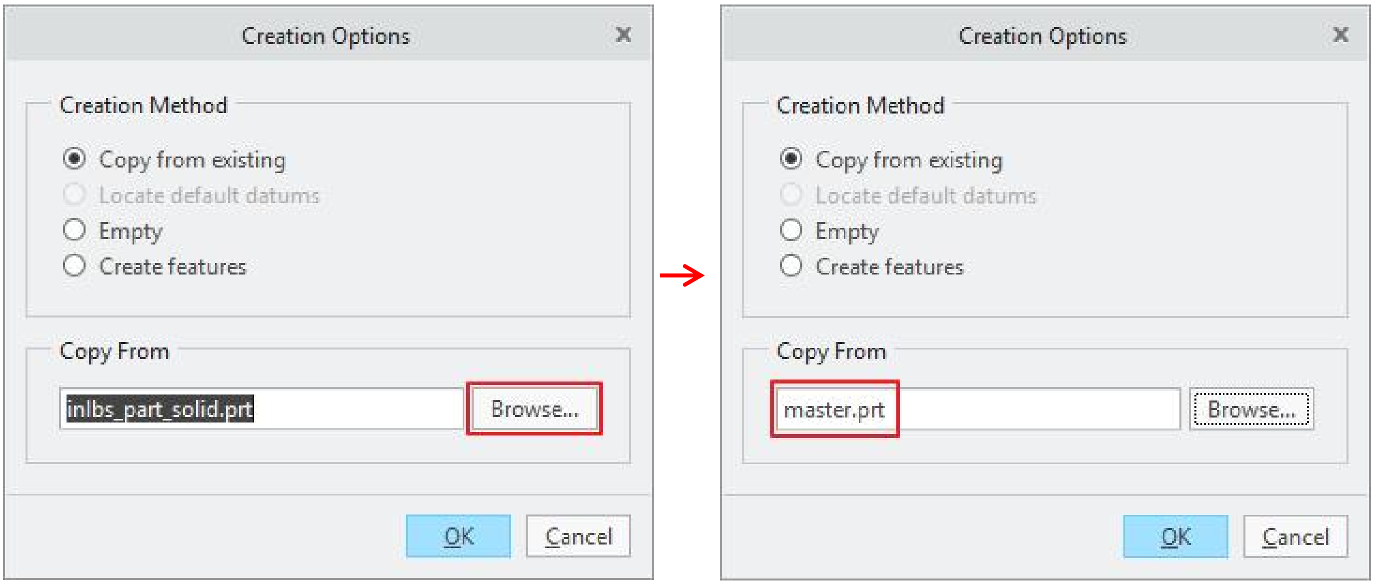


Notice that system automatically gives a new name to the skeleton model.

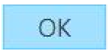
Pick 

and Create Options dialog box will appear. Pick 

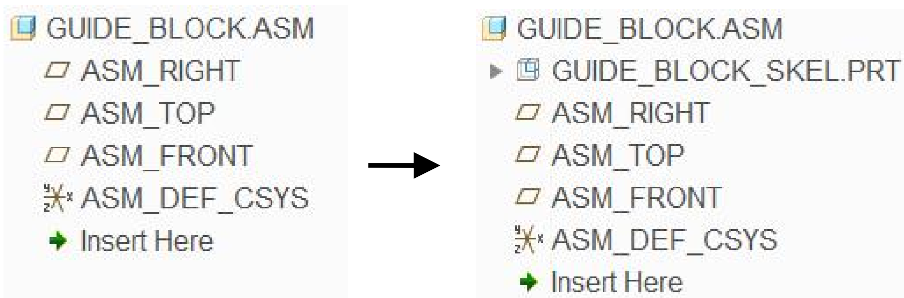
and select the MASTER.PRT as shown below.



MASTER.PRT is a regular Creo part that contains the geometry that we want to use in our skeleton model. Although you can create that geometry directly in the skeleton part but for convenience it has already been created in this part. Skeleton part is just copying the geometry from this existing part.

Pick 

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



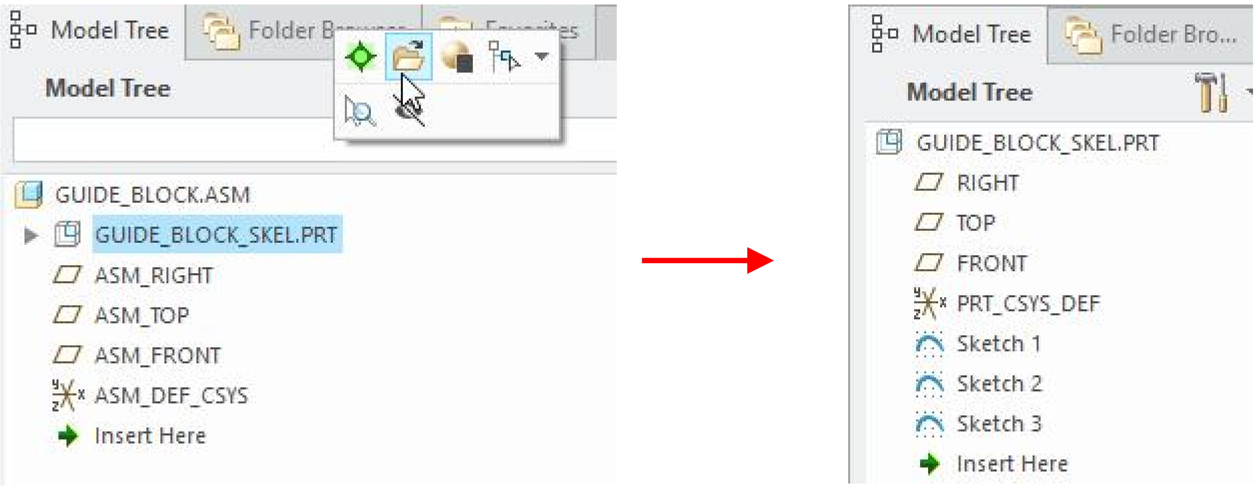
The system always inserts the newly created skeleton before all other components and assembly features.

You should notice that skeleton model has a unique icon in the model tree.

 Icon for Skeleton Model	 Icon for a Regular Component

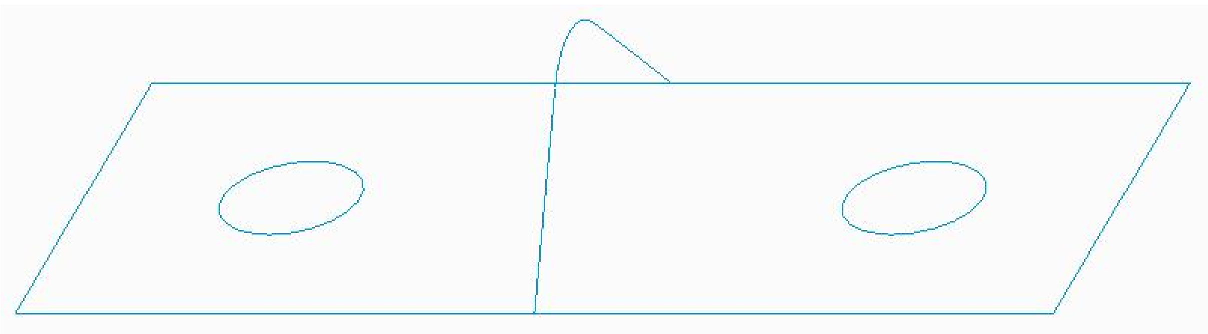
Because the functional characteristics of skeletons are different from other components, they are identified by a different icon in the Model Tree

Now open the skeleton model in the new window and notice that it has the features like any other component as shown below.



A skeleton model, like any other component, has features, layers, relations, views, and so on.

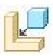
The geometry of the skeleton model will appear as shown below.

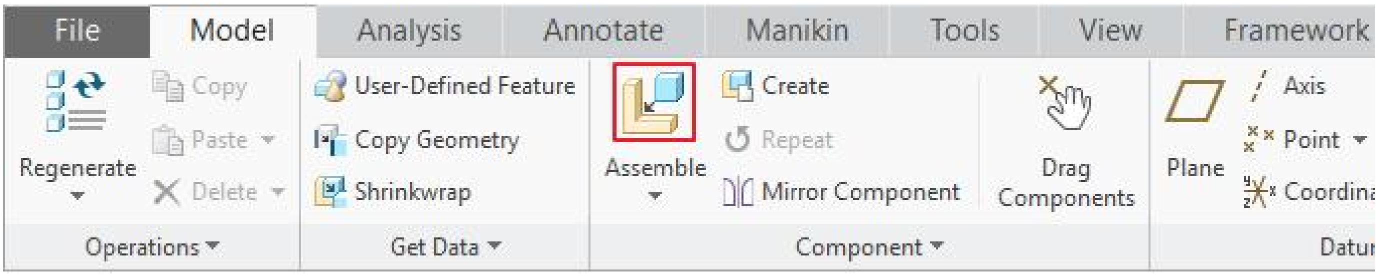


These datum curves will be used to drive the geometry in individual components according to design intent. Both components will reference these curves therefore the relationship or common attributes between the components will always be maintained.

Building the Assembly Structure

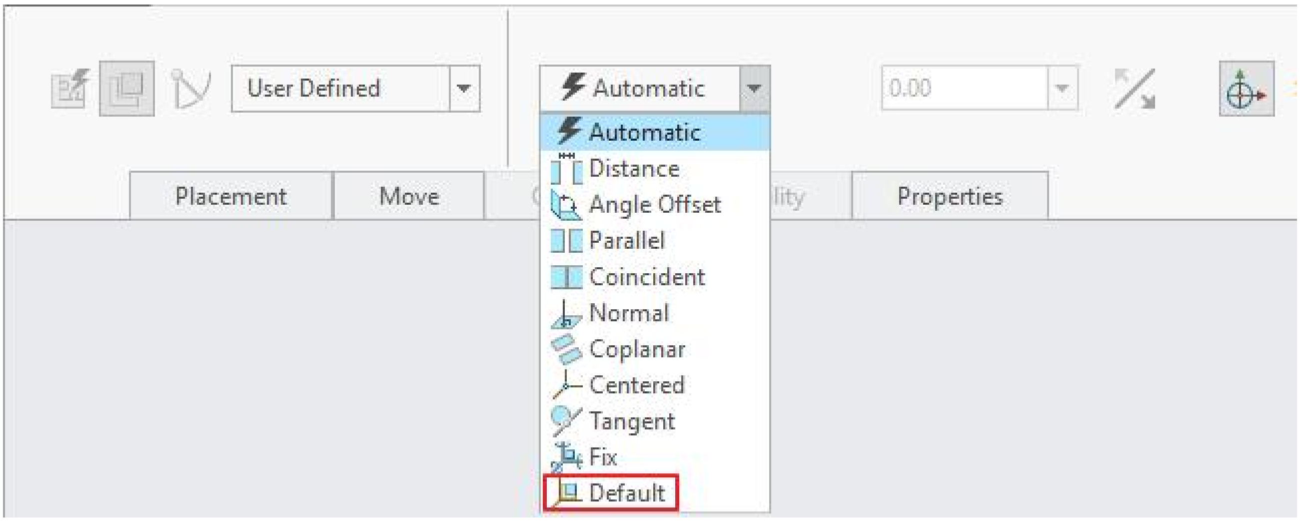
Now we will assemble two empty components that are regular Creo components. These components can also be created in the assembly environment but we have created them just for convenience.

Pick the Add a Component icon ().



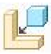
Select the BLOCK_ML.PRT in the open dialog box then pick Open .

In the Component Placement dashboard select the **Default** constraint as shown below.



The Default constraint aligns the default system-created coordinate system of the component to the default system-created coordinate system of the assembly.

Notice that after picking the Default constraint, STATUS changes to “Fully Constrained”. So pick  to apply the changes and exit the dashboard.

Again pick  and select the BLOCK_FM.PRT in the open dialog box then pick Open .

In the Component Placement dashboard select the **Default** constraint.

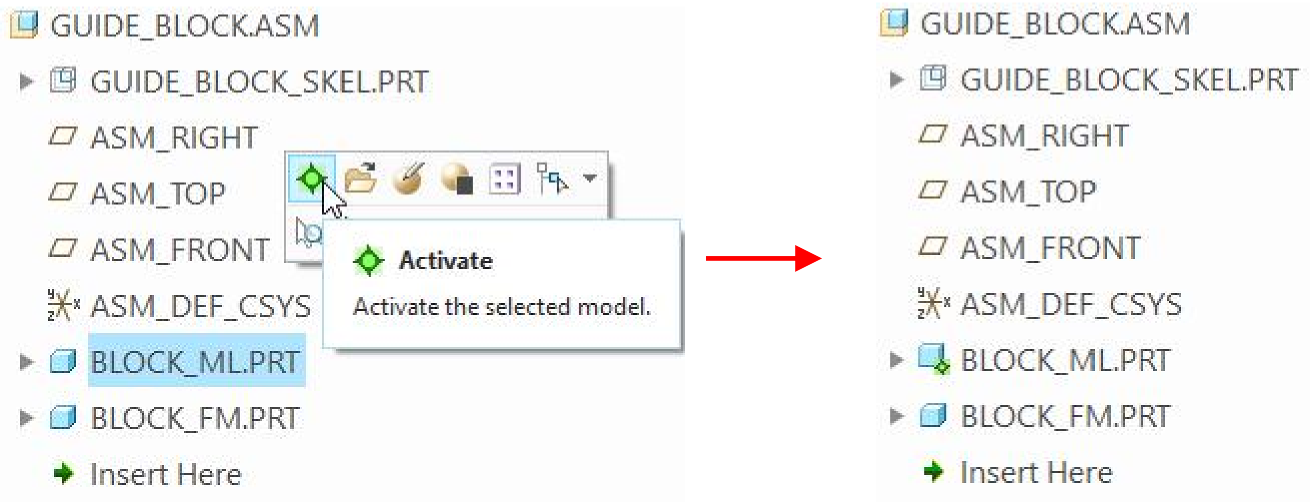
Pick  to apply the changes and exit the dashboard.

Communicating the Design Information

Now we will use Copy Geometry feature to communicate design references from the skeleton to the assembled parts.

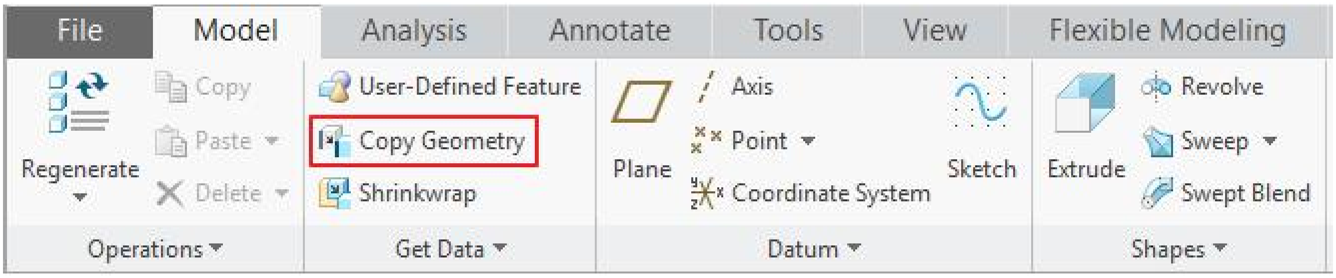
First we will create a copy geometry feature in the BLOCK_ML.PRT

Pickthe BLOCK_ML.PRT in the model tree and select **Activate**.

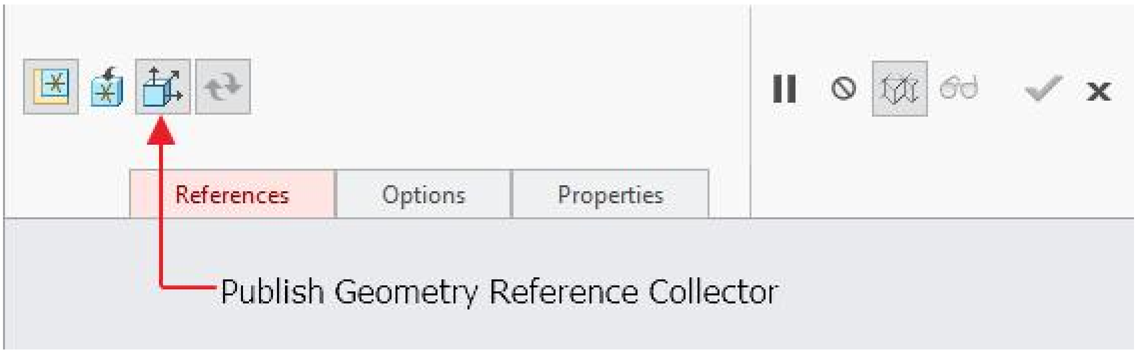


To create a copy geometry feature, pick 

on the Model tab.



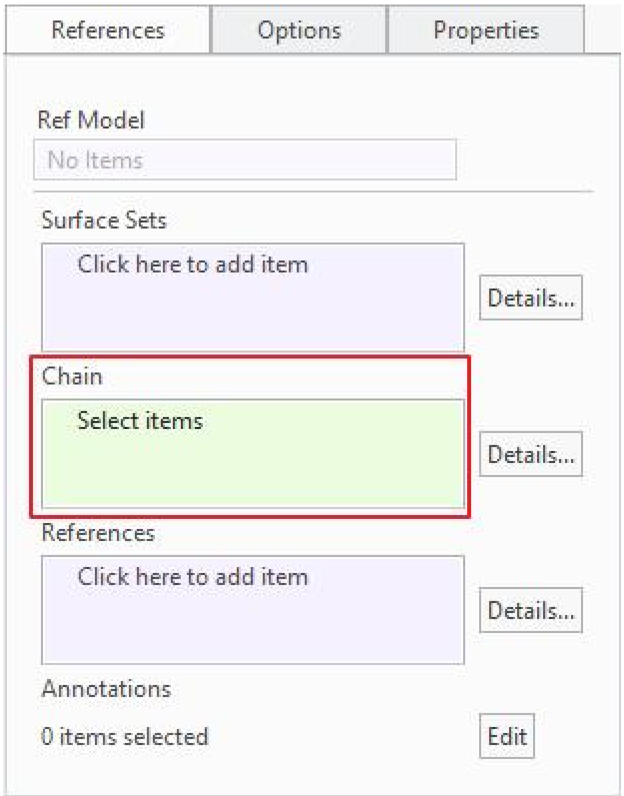
The Copy Geometry dashboard will appear as shown below.



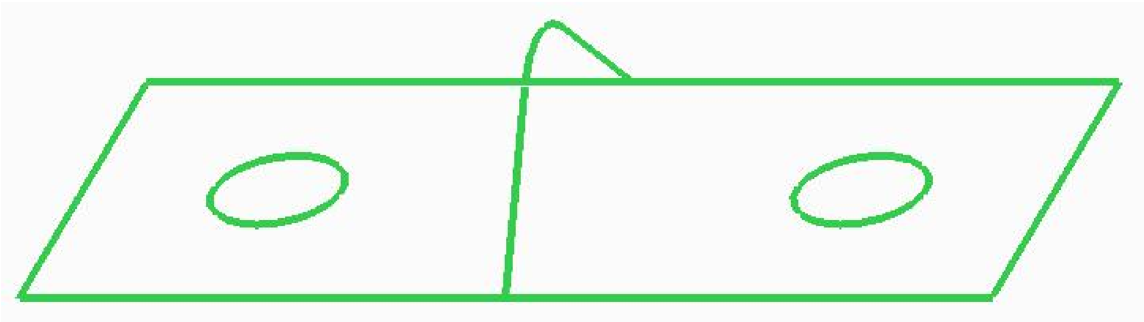
Notice that Publish Geometry reference collector is active by default. For this exercise we do not want to use the publish geometry feature, so pick  in the dashboard to deactivate the publish geometry reference collector.

When you create a Copy Geometry feature, you can reference either individual references like curve, surface and datum features or a Publish Geometry feature.

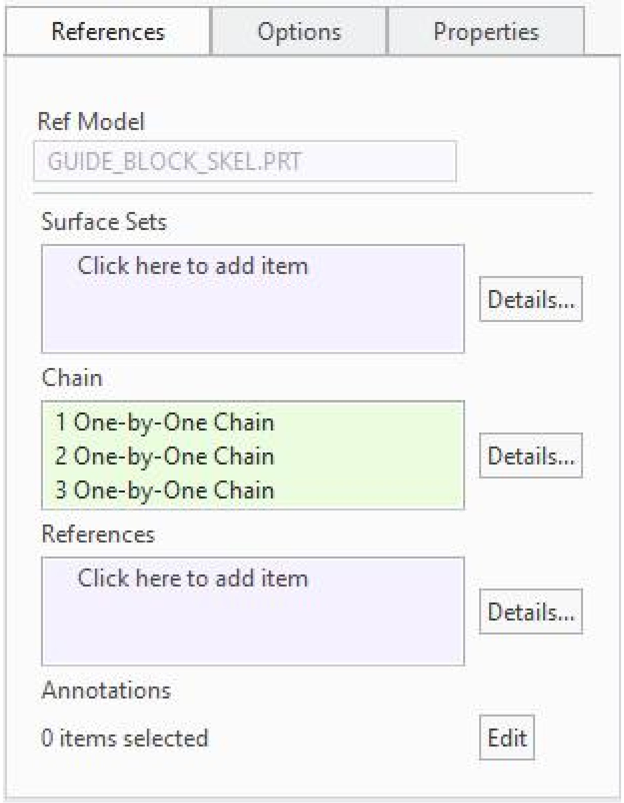
Now pick **References** tab to access References panel and pick in the Chain collector to activate it.



Hold down the Ctrl key and pick all the datum curves in the skeleton model as shown below

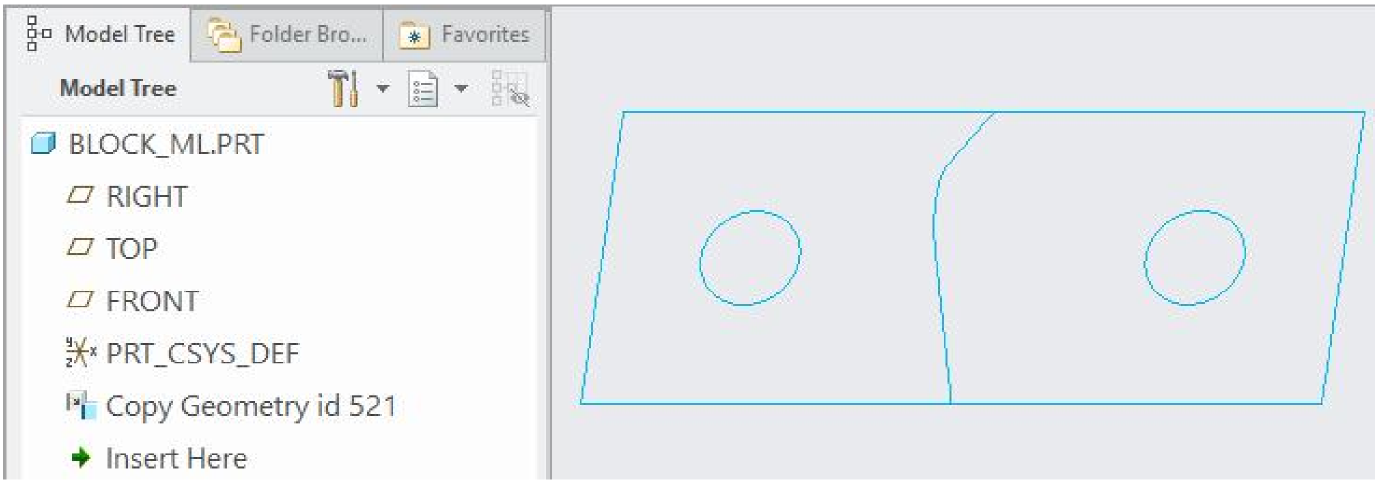


All the selected references (chains) will appear in the Chain Collector as shown below.



Pick  to apply the changes and exit the dashboard.

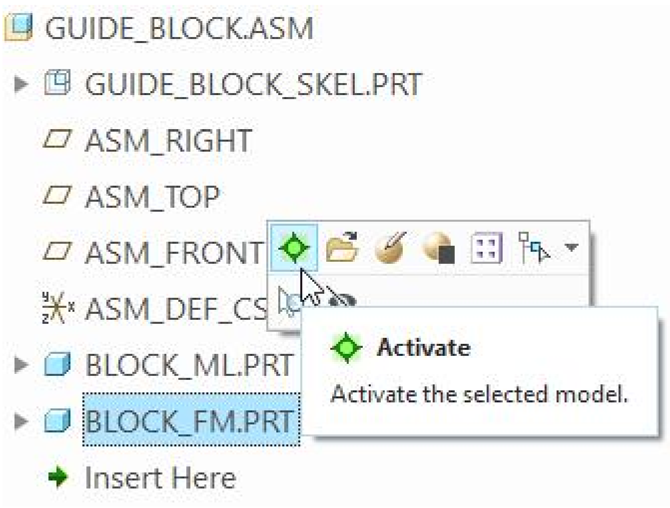
Now open the BLOCK_ML.PRT in the new window and notice that it has a new feature (Copy Geometry) in the model tree as shown below.



You can see the copied curves in the graphics window.

The Copy Geometry feature can be used to pass datum features, curves and surfaces, but not the solid geometry.

Now switch to the assembly window and activate the BLOCK_FM.PRT as shown below.



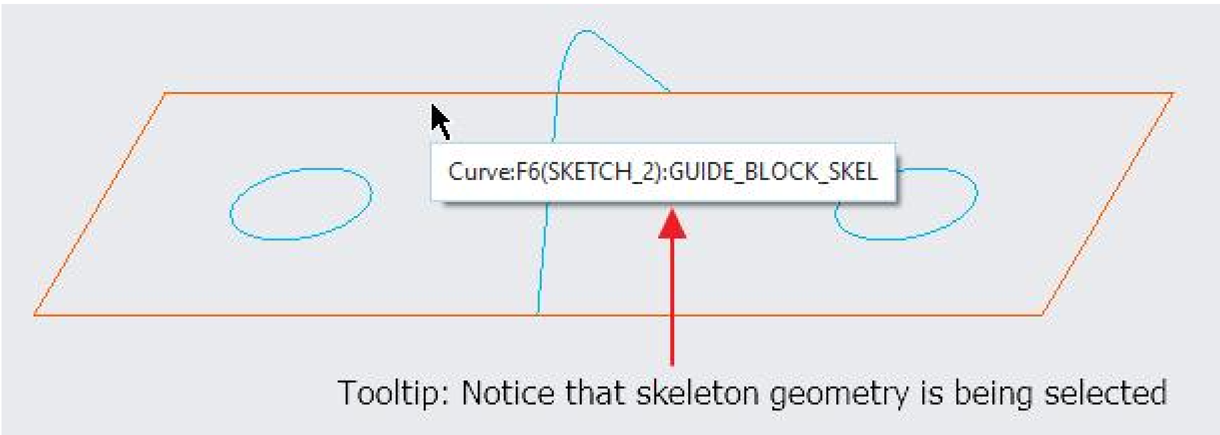
To create a copy geometry feature, pick Copy Geometry on the Model tab.

In the Copy Geometry dashboard pick to deactivate the Publish Geometry reference collector.

Pick **References** tab to access References slide-up panel.

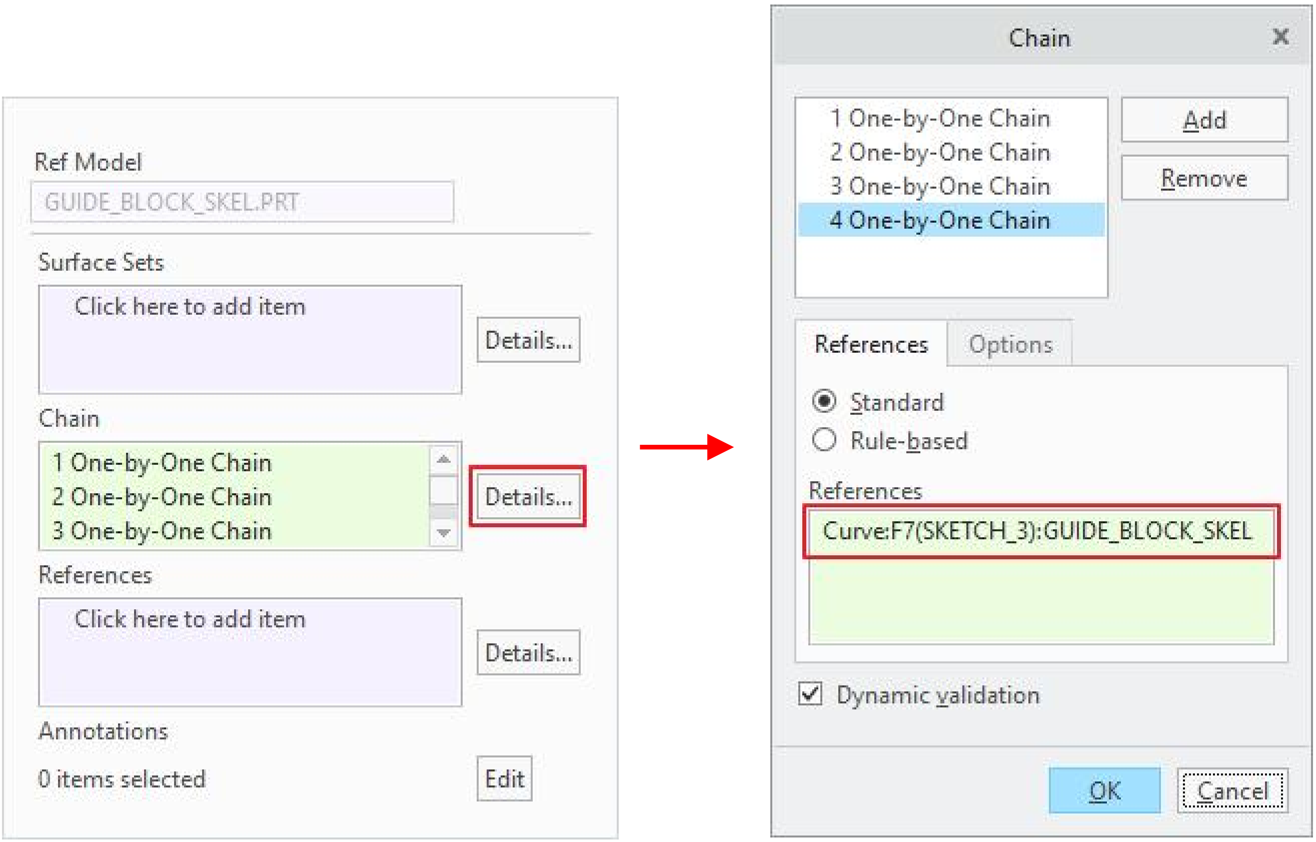
Pick in the Chain collector to activate it.

Hold down the Ctrl key and pick all the datum curves in the skeleton model. You should make sure that you pick the curves of the skeleton model and not from the BLOCK_ML.PRT. For this purpose you should keep an eye on the tool-tip that appears at the bottom of the mouse pointer.



Another way to confirm that the selected references belong to the skeleton model is to pick Details... icon and looking at the References collector.

Skeleton model name should appear in the References field as shown below.



Pick  to apply the changes and exit the copy geometry dashboard.

Creating Model Geometry

Now the information is captured from the skeleton model into the components. We will use the Copy Geometry feature as a reference to build parts.

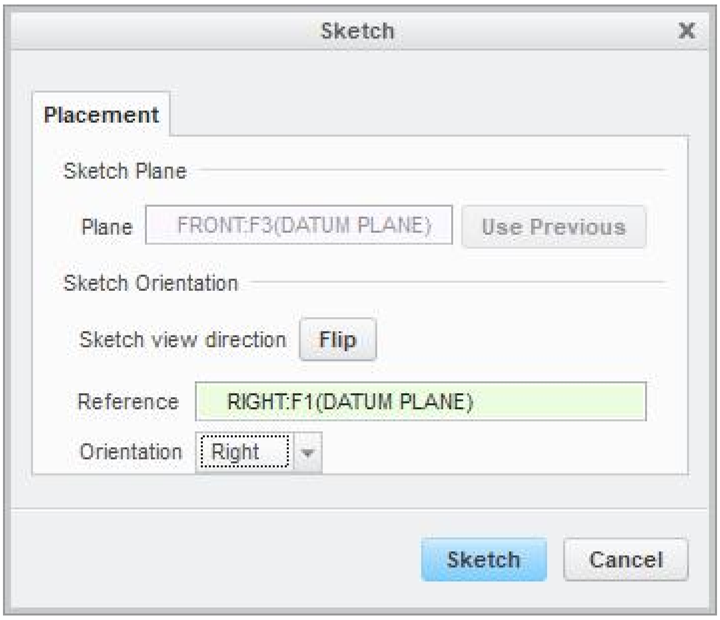
First Part

Open the BLOCK_ML.PRT in new window

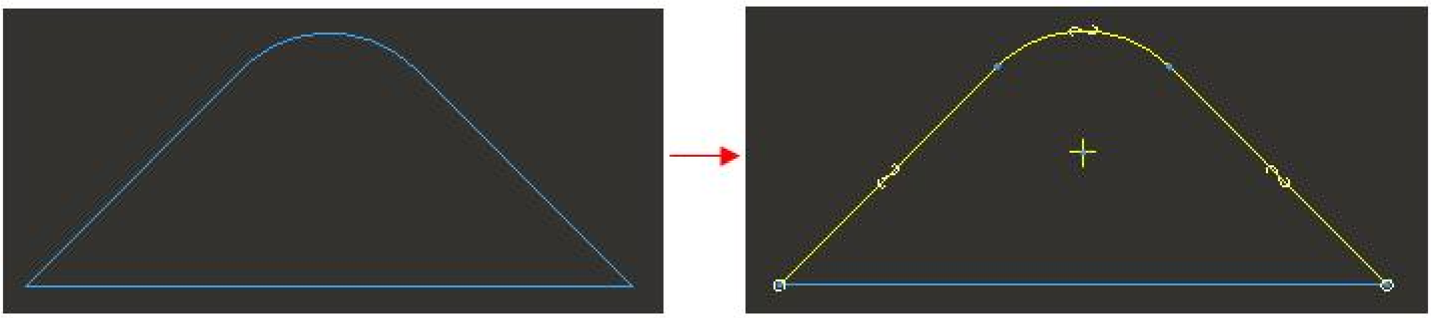
We will create two extrude features to build the geometry for this part.

Pick the Extrude Tool icon ().

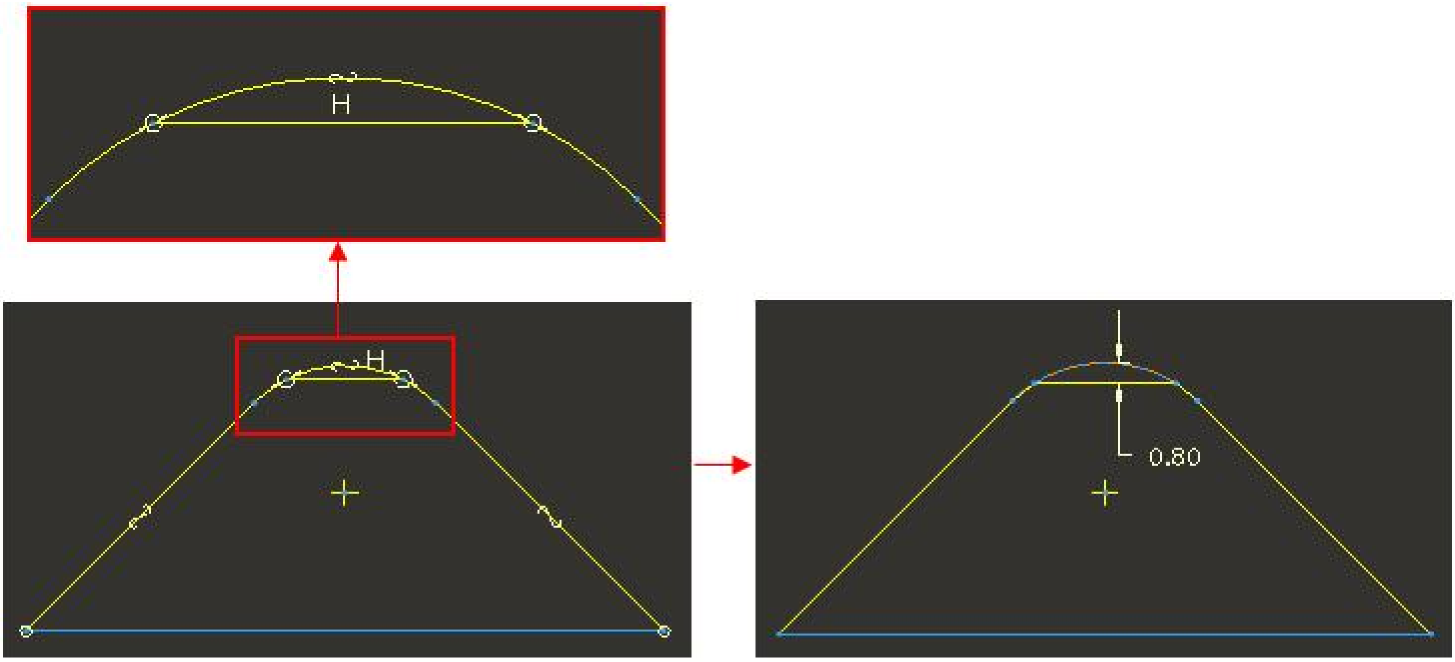
Select the **FRONT** datum plane as the Sketch Plane and **RIGHT** datum as the Right Orientation reference.



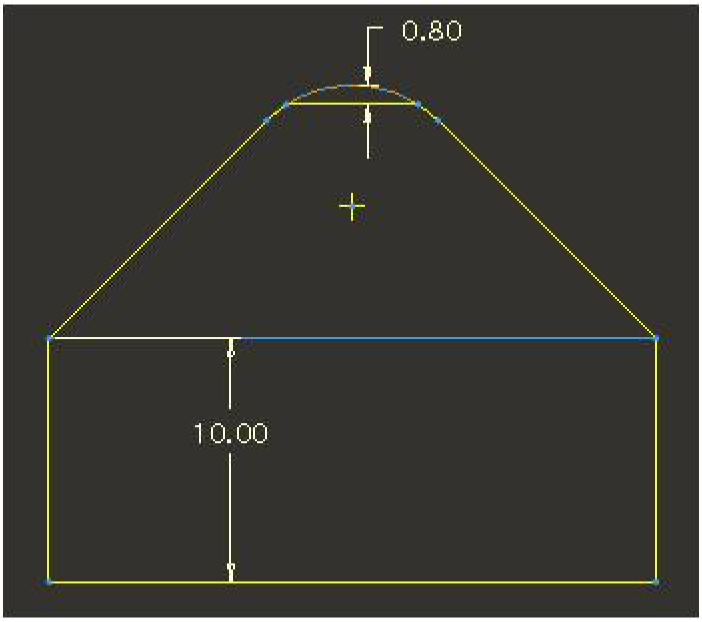
Pick Project icon and select the following loop. (Note: Sketch entities are shown in such colors so that they are easy to distinguish. You will see different colors depending upon the configuration of your system.)



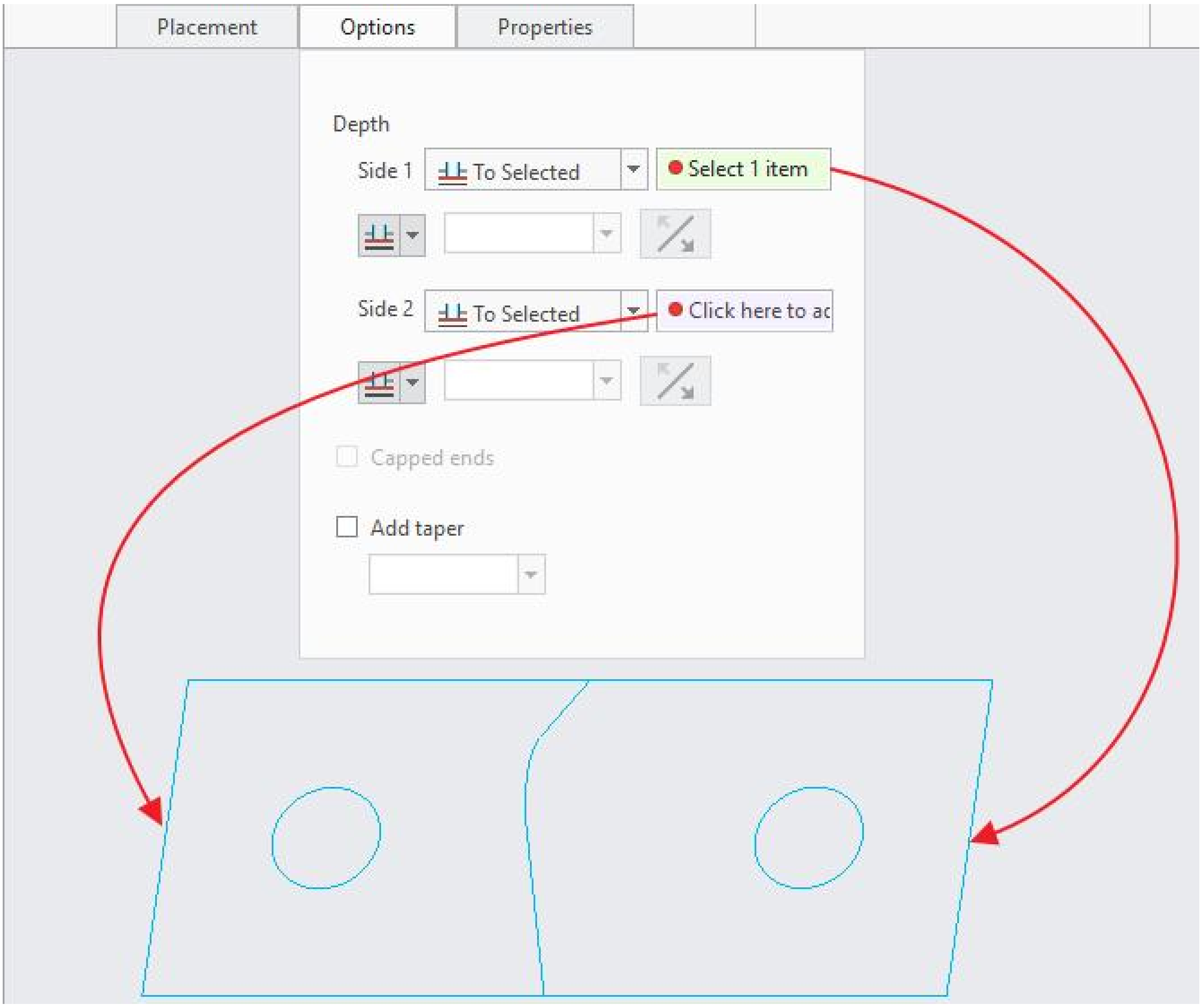
Now sketch a line then trim and dimension the section as shown below.



Complete the section as shown below.

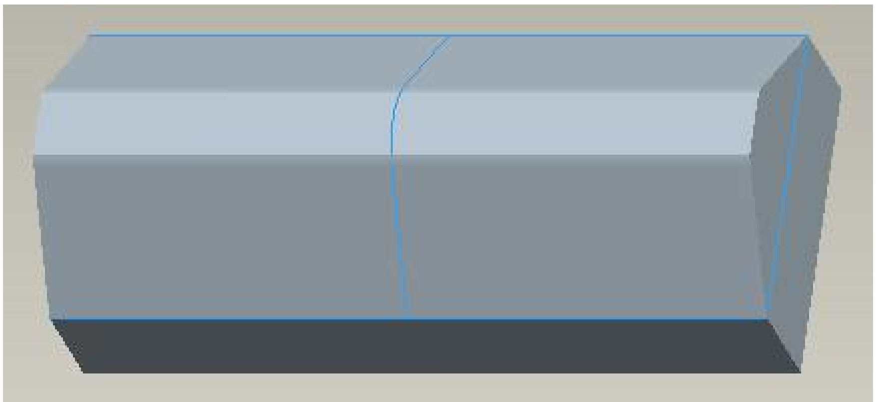


Pick ✔ to complete the sketch, select the depth references as shown below.



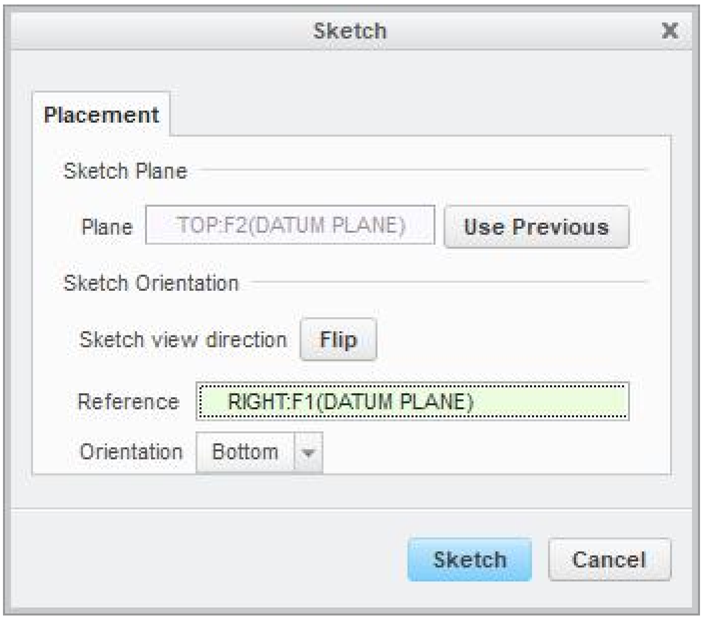
Pick  icon or middle-click to complete the feature.

The part will appear as shown below.



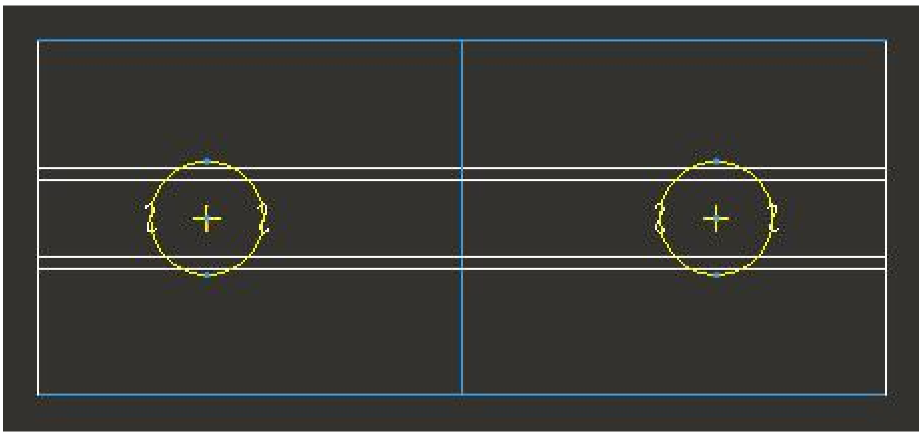
Again pick  icon to create the second extrude feature.

Select **TOP** datum as the Sketch Plane and **RIGHT** datum as the Bottom Orientation reference.

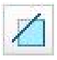


Pick 

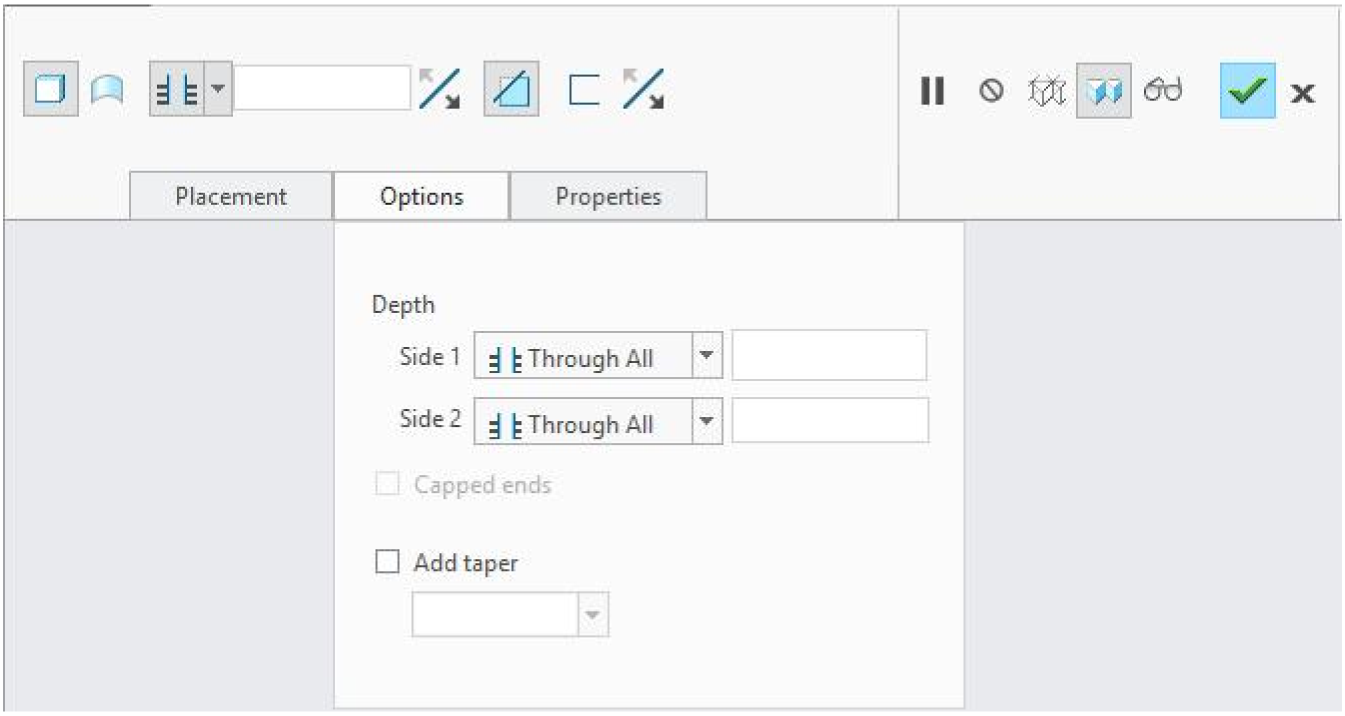
icon and select the following loops.



Pick  to exit the sketcher.

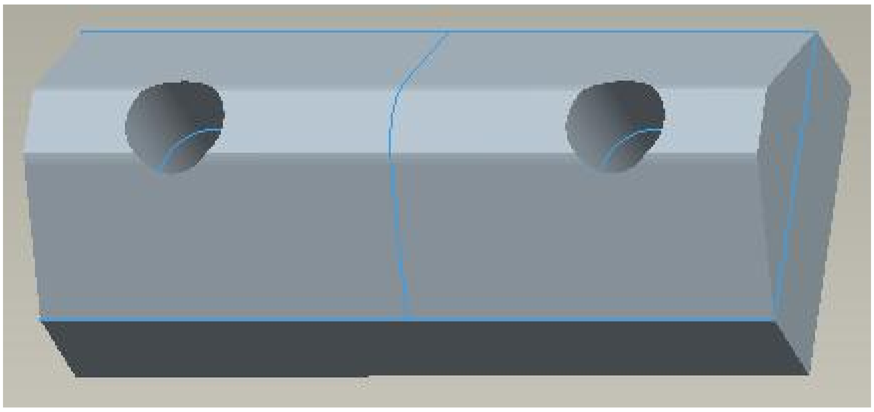
Pick  icon to create the feature as cut.

Select the depth option to **Through All** for both directions as shown below.



Pick  icon or middle-click to complete the feature.

The part will appear as shown below.



Second Part

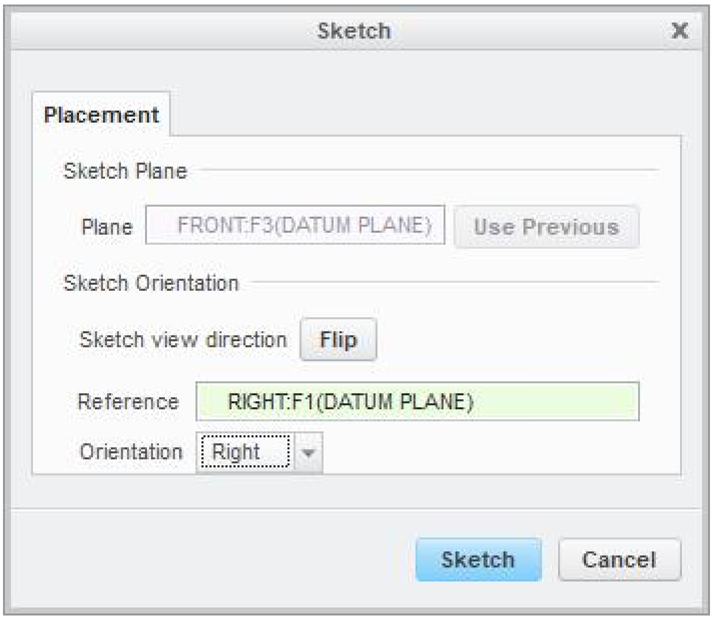
Now we will create the geometry for the BLOCK_FM.PRT

So open the BLOCK_FM.PRT in a new window

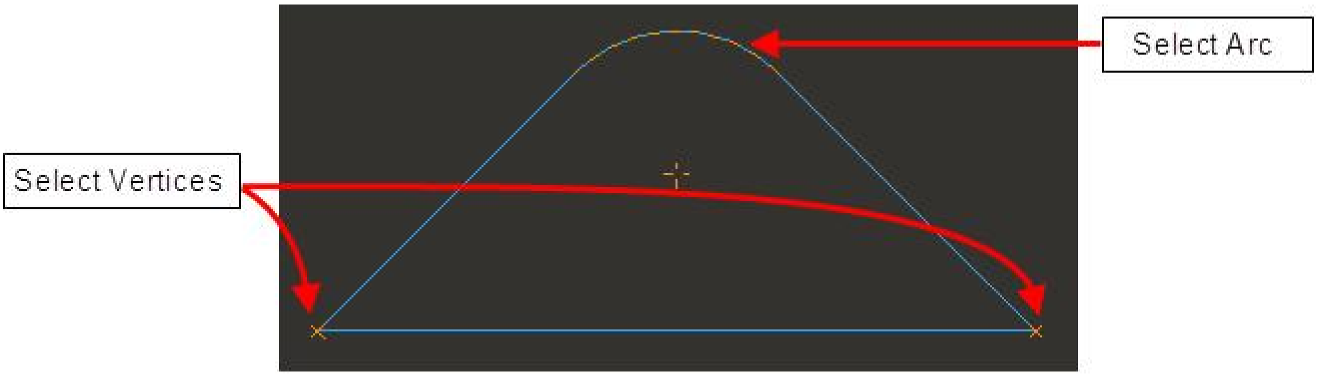
We will create three extrude features to build the geometry for this part.

Pick the Extrude Tool icon 

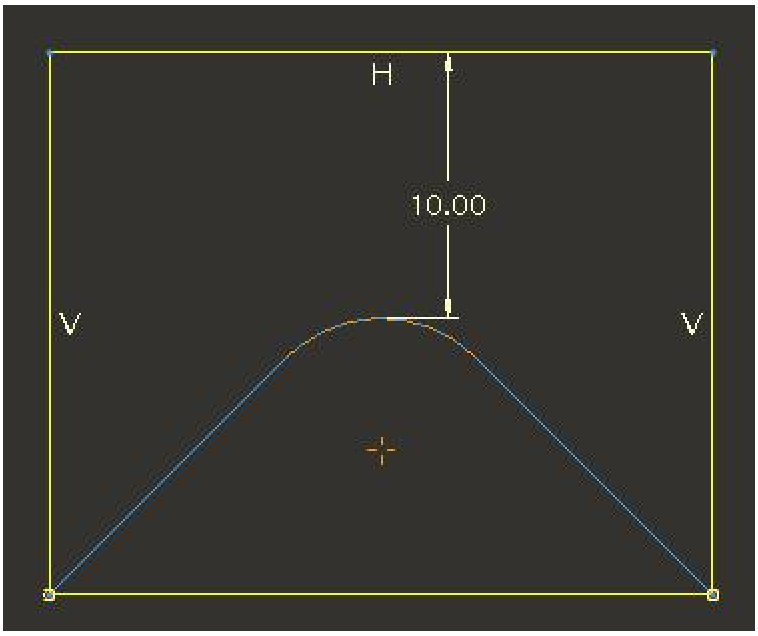
Select **FRONT** datum as the Sketch Plane and **RIGHT** datum as the Right Orientation reference.



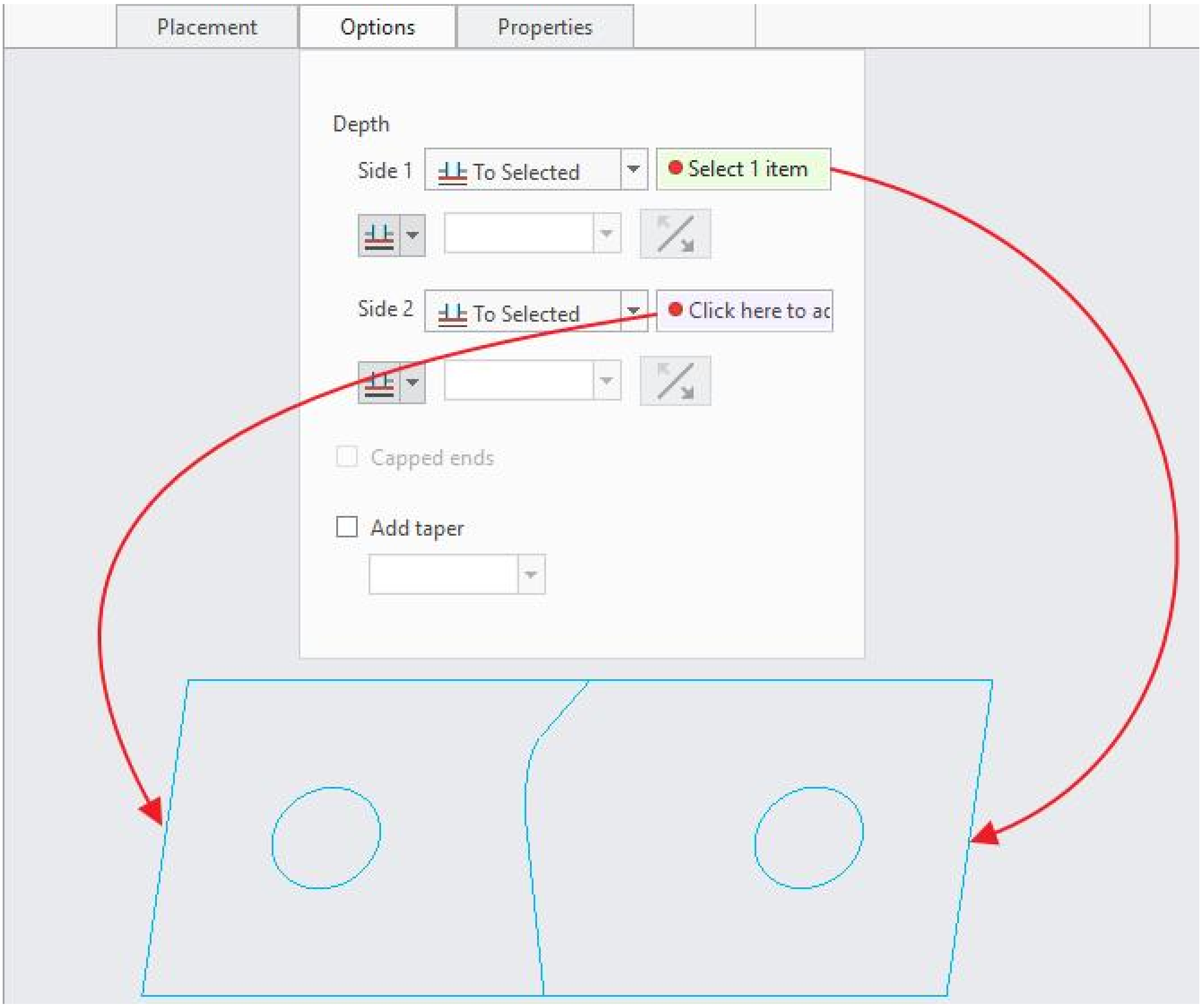
Select the following references for dimensioning and constraining the section geometry.



Sketch the section as shown below.

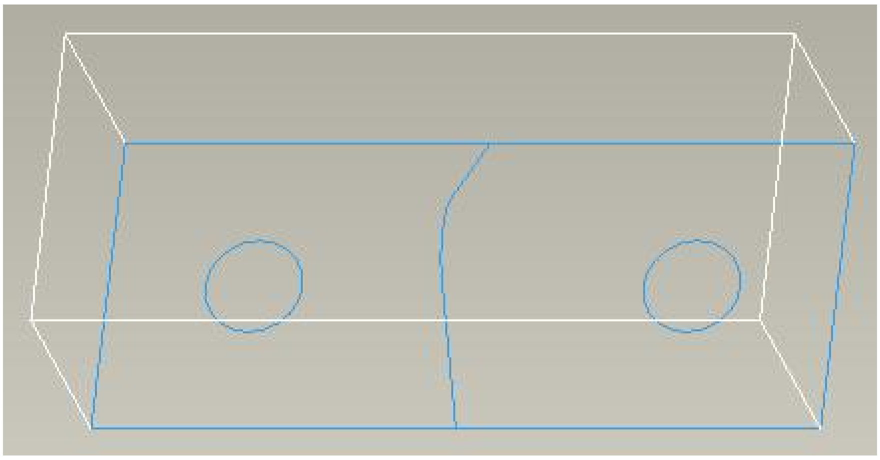


After completing the sketch, select the depth references as shown below.



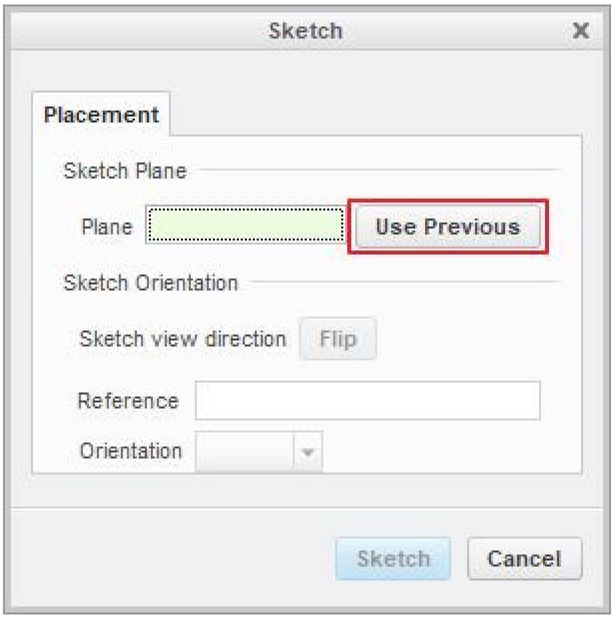
Pick ✓ icon or middle-click to complete the feature.

The part will appear as shown below.



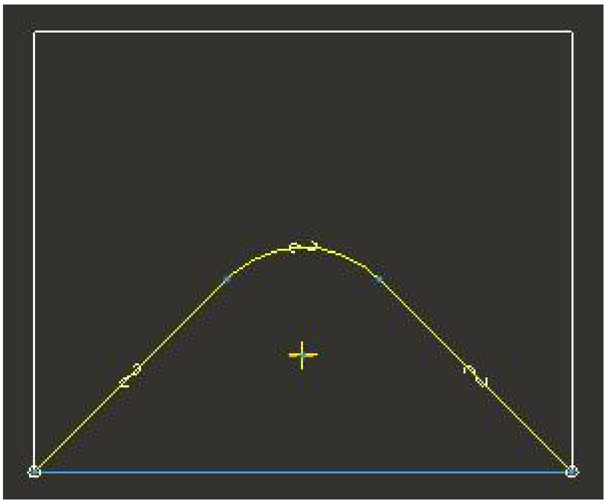
Again pick  icon to create the second extrude feature.

Select the same sketching references that were used for previous section by using the **Use Previous** option in the Sketch dialog box.

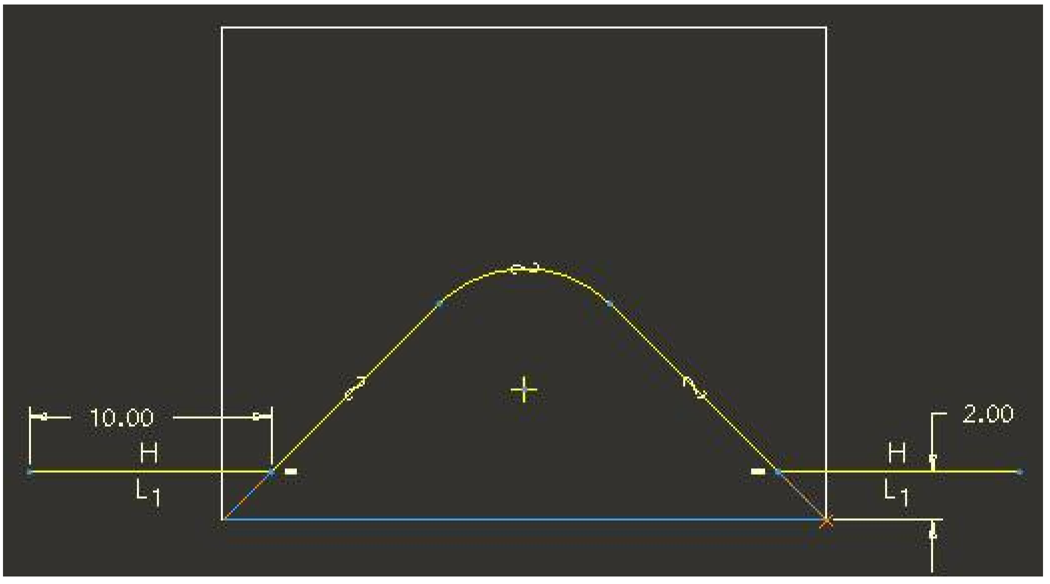


Pick 

icon and select the following loop.

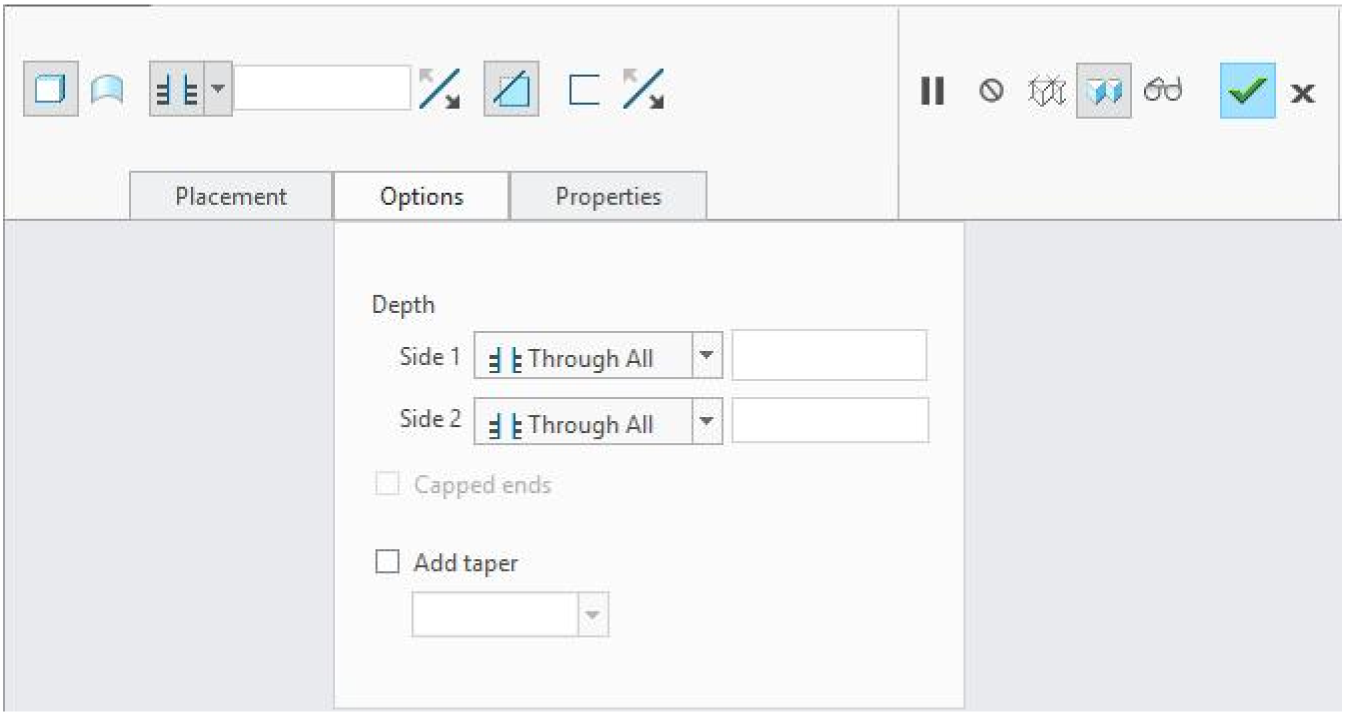


Complete the section as shown below.



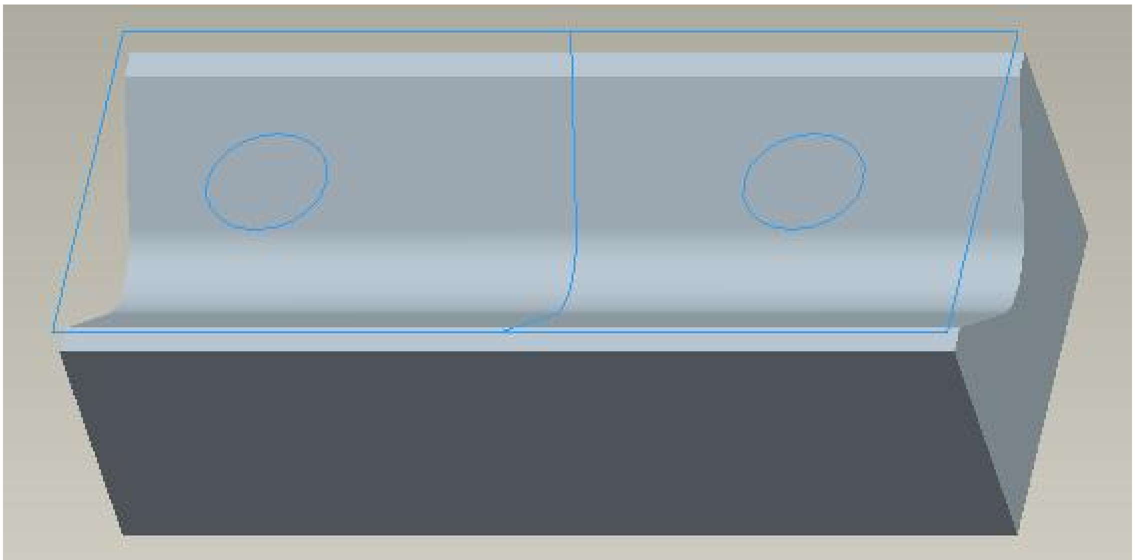
After completing the sketch, pick  icon to create the feature as cut.

Select the **Through All** as depth option for both directions as shown below.



Pick icon or middle-click to complete the feature.

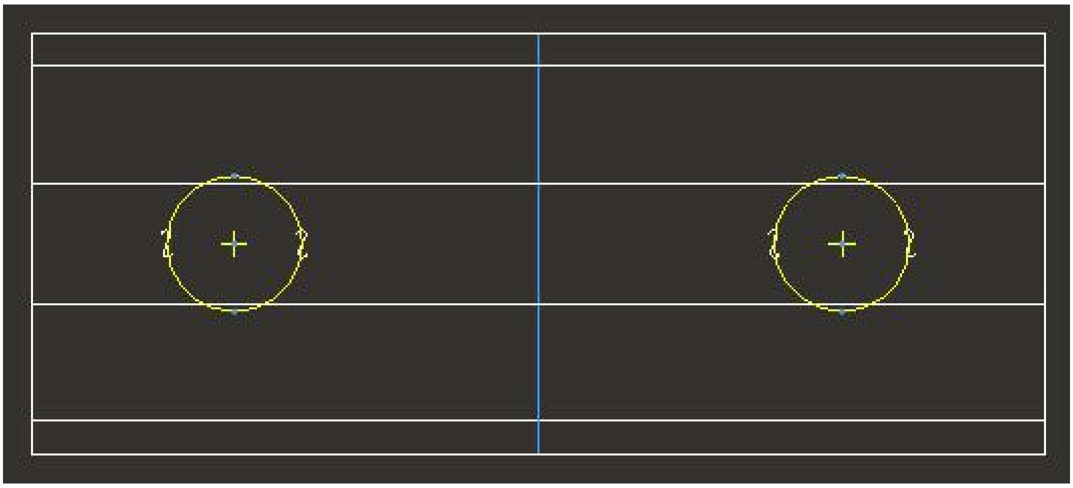
The part should appear as shown below.

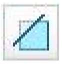


Again pick icon to create the next extrude feature.

Select the **TOP** datum as the Sketch Plane and **RIGHT** datum as the Bottom Orientation reference.

Pick Project icon and select the following loops.

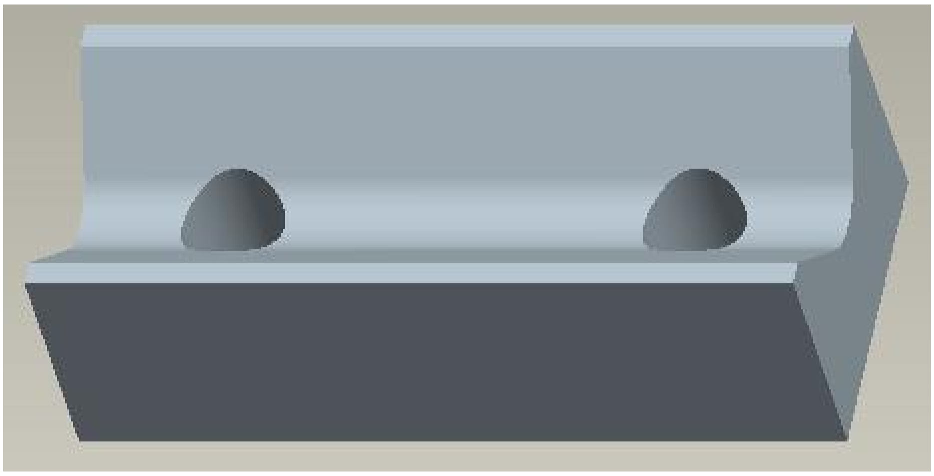


After completing the sketch pick  icon to create the feature as cut.

Change the depth option to **Through All** in the Depth options list for both directions.

Pick  icon or middle-click to complete the feature.

The part will appear as shown below.



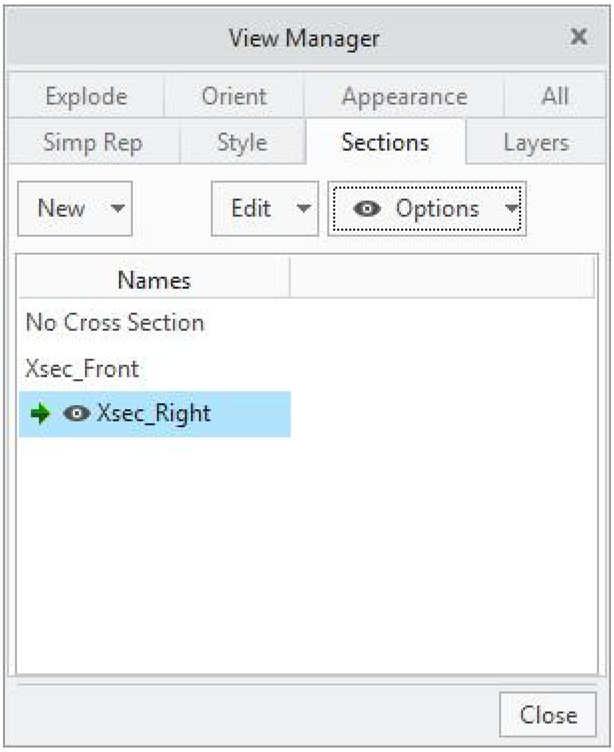
Verifying Your Design

Switch to assembly and look carefully at the parts.

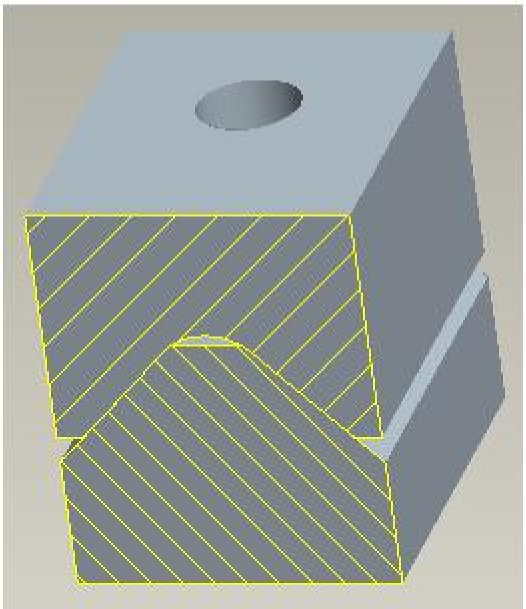
To analyze the assembly clearly we need to view the cross-sections at different planes.

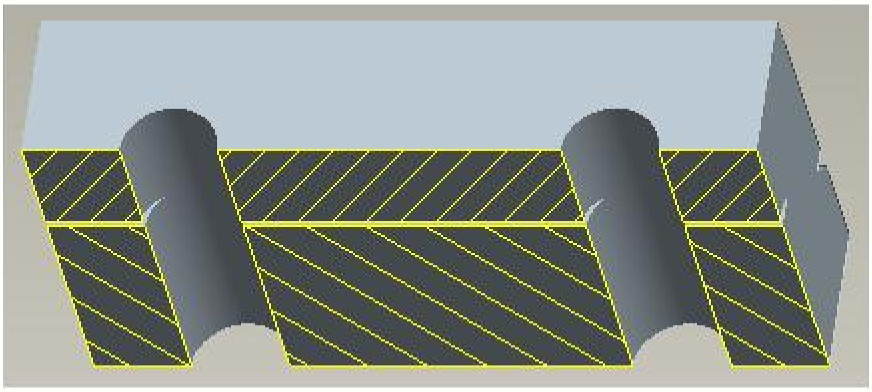
So pick  to launch the View Manager Dialog box

Select **Sections** tab and double click the predefined cross-sections one by one.



Following figure shows the Xsec_Front and Xsec_Right cross-sections of the assembly.





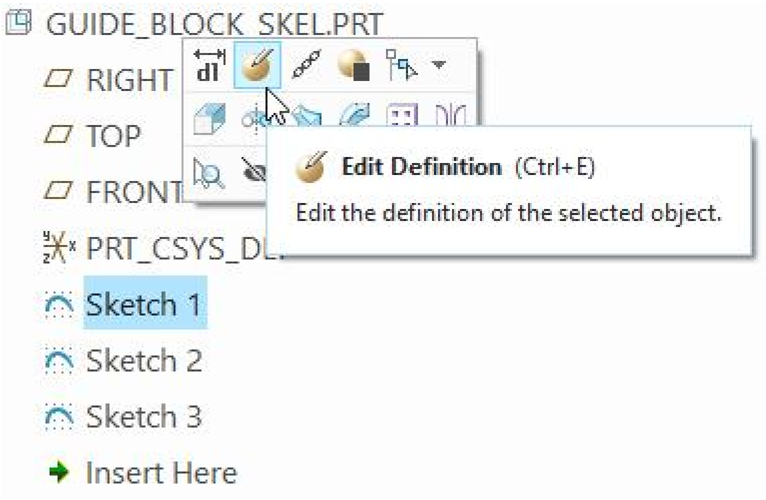
Cross-Sections of the assembly

You can notice that both parts confirm to the design intent in terms of size, form and location of holes.

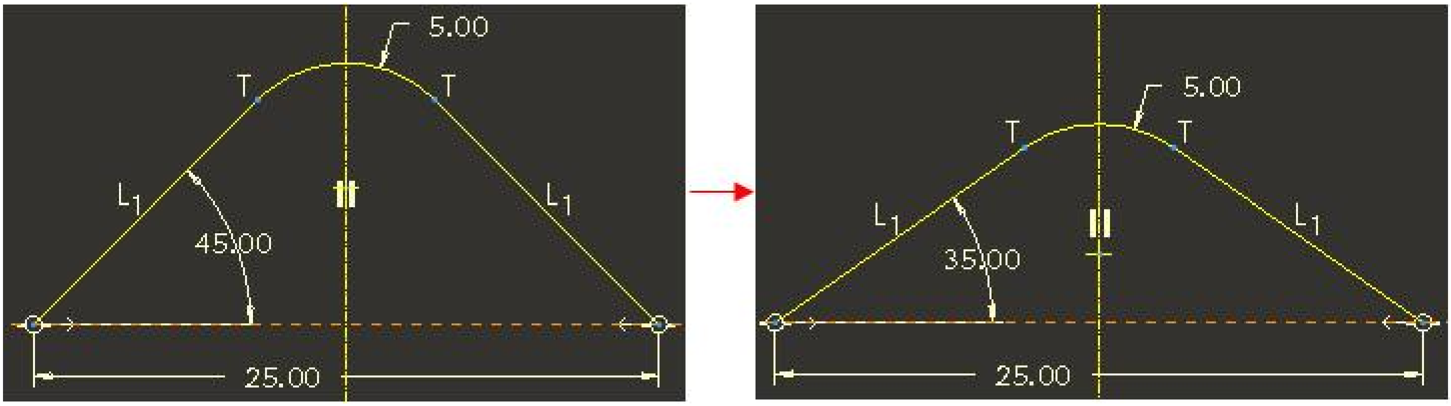
Now we will modify the skeleton and regenerate the assembly. You will see that the parts confirm to the design intent after regeneration.

Open the skeleton in a separate window.

Select the "Sketch 1" feature and pick in the pop up menu as shown below.



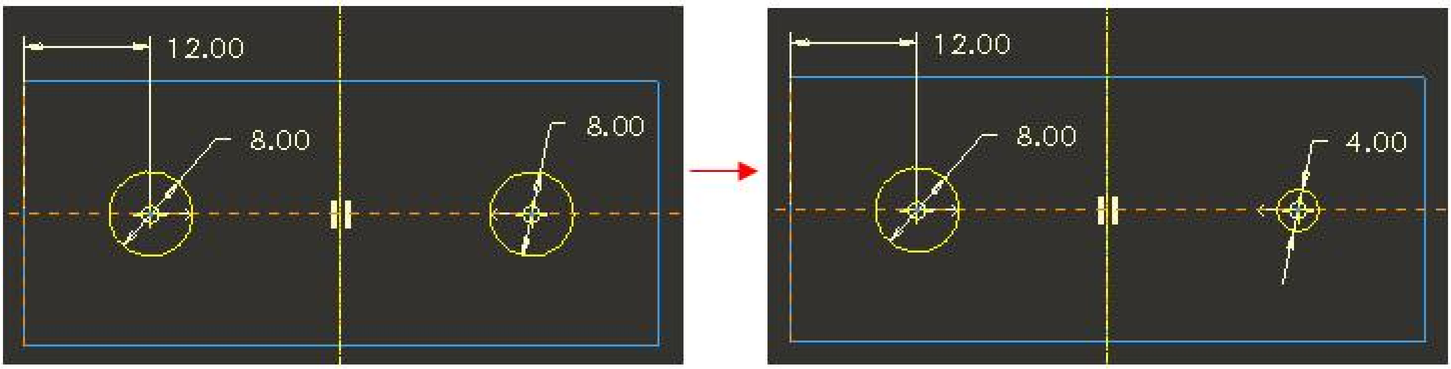
Now modify the angular dimension as shown below.



Pick after modifying the dimension.

Select the "Sketch 3" feature and pick in the pop up menu

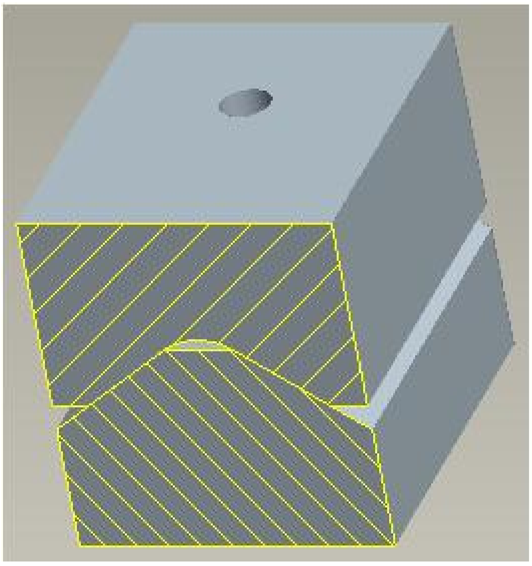
Now modify the diameter of circle as shown below.

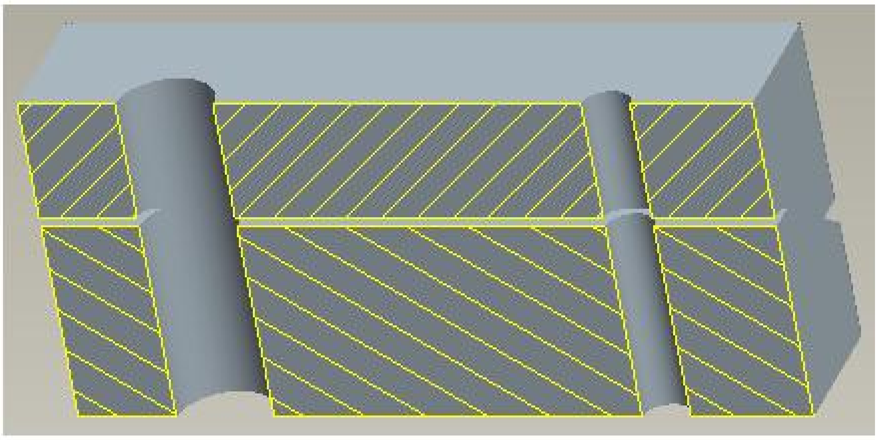


Pick after modifying the dimension.

Now switch to the assembly window.

Regenerate the assembly by picking icon. The following cross-sections show the parts after regeneration.





You should notice that geometry in both parts update according to the changes made in the skeleton.

Exercise 2

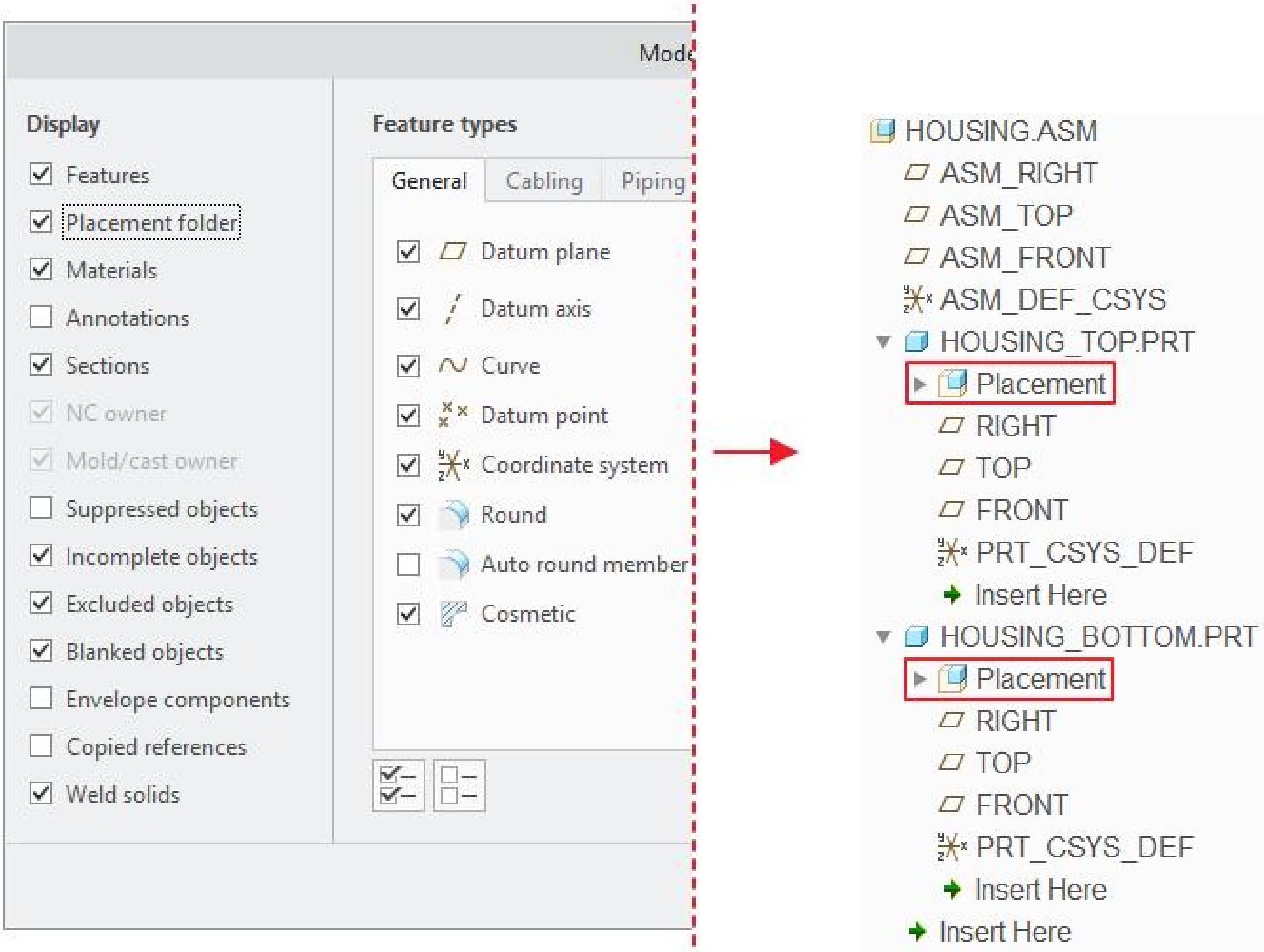
In this exercise we will create a simple housing consisting of two parts.

Here we will use the following simplified top-down design approach.

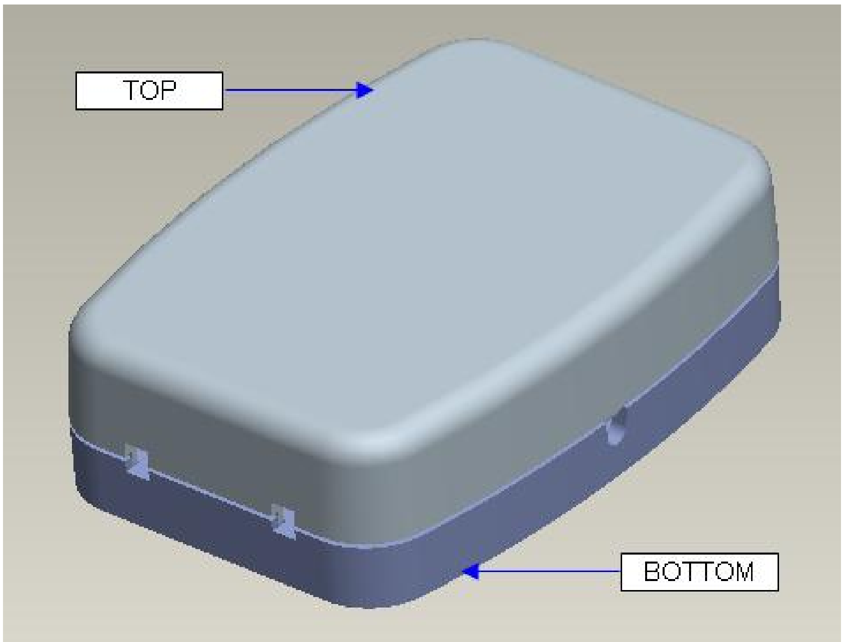
1. Creating the geometry in Skeleton model
2. Using Publish Geometry and Copy Geometry features to communicate references
3. Creating model geometry

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

Notice that there are two components assembled with Default constraint. Also notice that both components have only default datum features as shown below. (Note: Placement constraint of a component is only visible if Placement folder option is checked in Model Tree Items dialog box.)



The housing assembly in the completed form is shown below.



The common attributes between both components are as follows

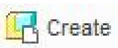
1. Profile of both components should be same.
2. Two rectangular holes in side walls of the components should be aligned and of same width.

So we will create the geometry in the skeleton model that controls the above common attributes.

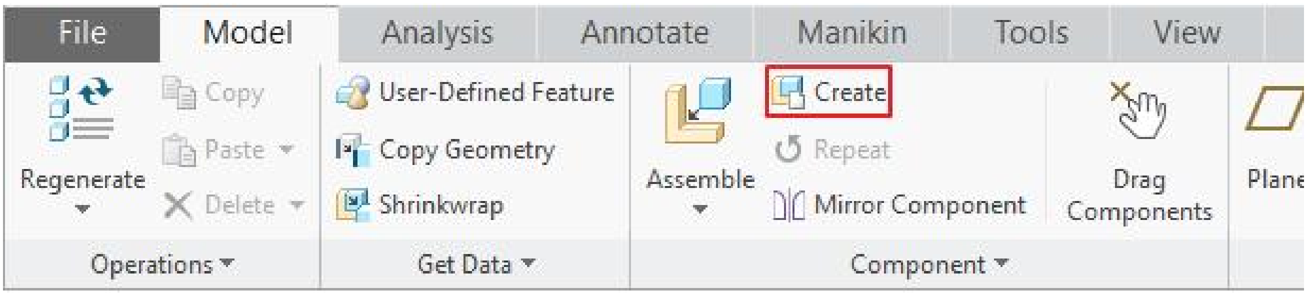
Only, the geometry that affects or crosses more than one component (or has to be referenced by more than one component) should be placed in the skeleton.

⇨ Creating the Skeleton

Now we will create the skeleton model in the assembly mode.

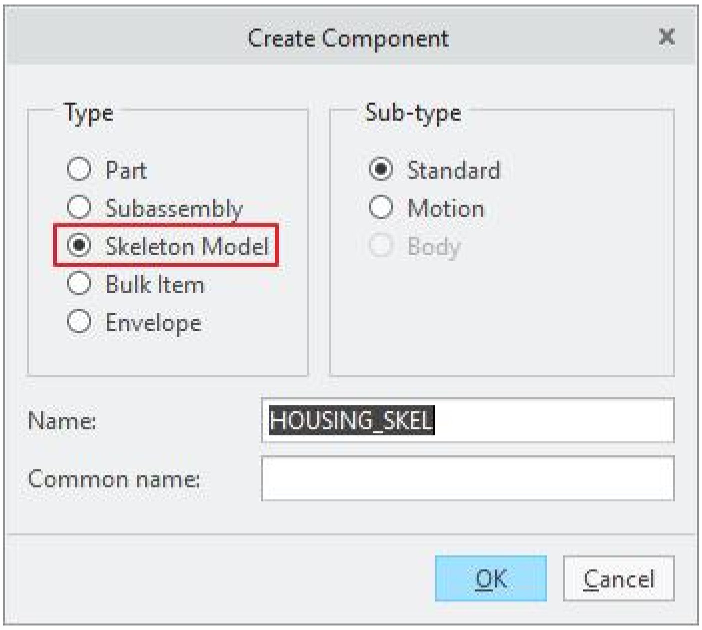
Pick 

to create a new component.



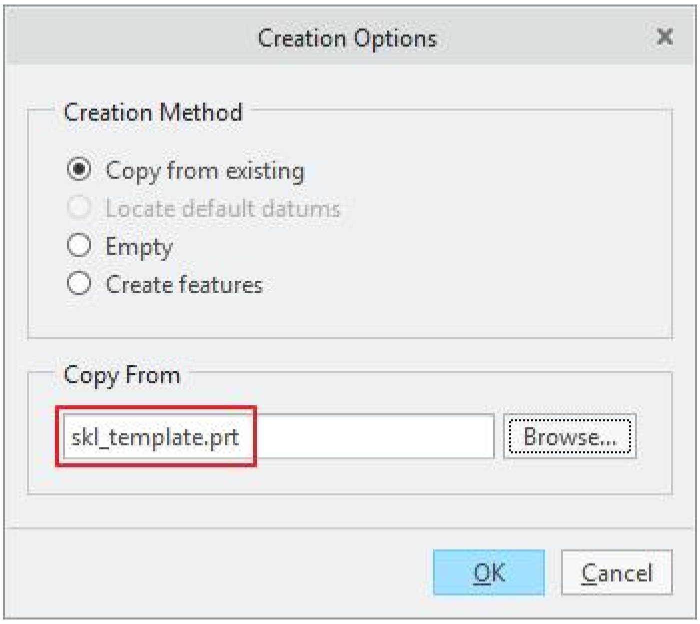
Skeletons models can only be created in an assembly.

Component Create dialog box will appear. Here change Type to **Skeleton Model** as shown below.



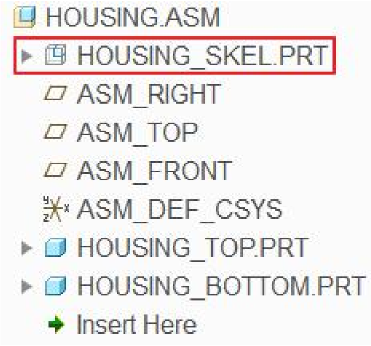
Notice that system automatically gives a new name to the skeleton.

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



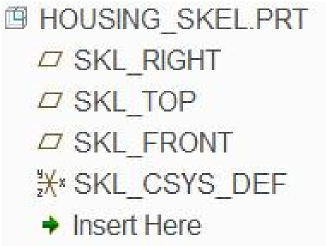
SKL_TEMPLATE.PRT is the template file that contains the default datum features, layers and accuracy settings.

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



A skeleton model can be created at any time in the design process. The system always inserts the newly created skeleton model before all other components and assembly features.

Open the skeleton model in a separate window and notice that it only consists of default datum features as shown below.

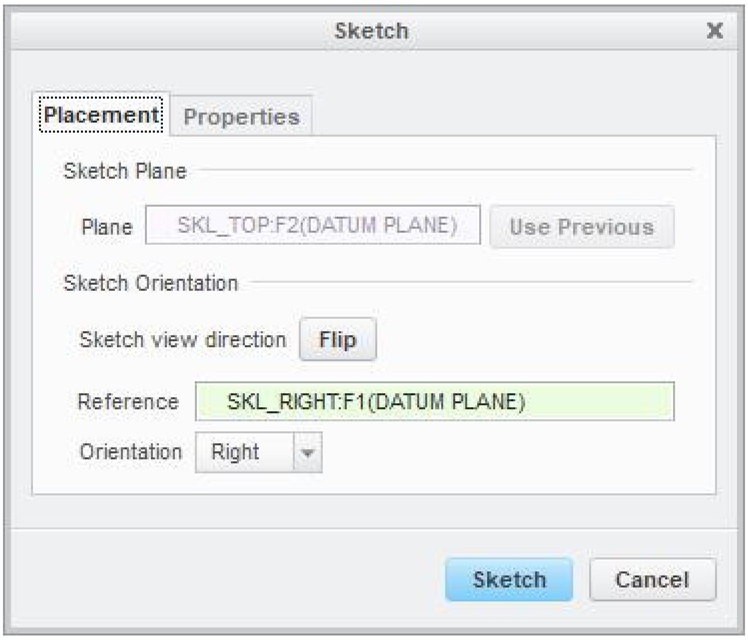


Notice that all the datum features of skeleton have “SKL_” as the prefix. It is to distinguish them from the datum features of other components.

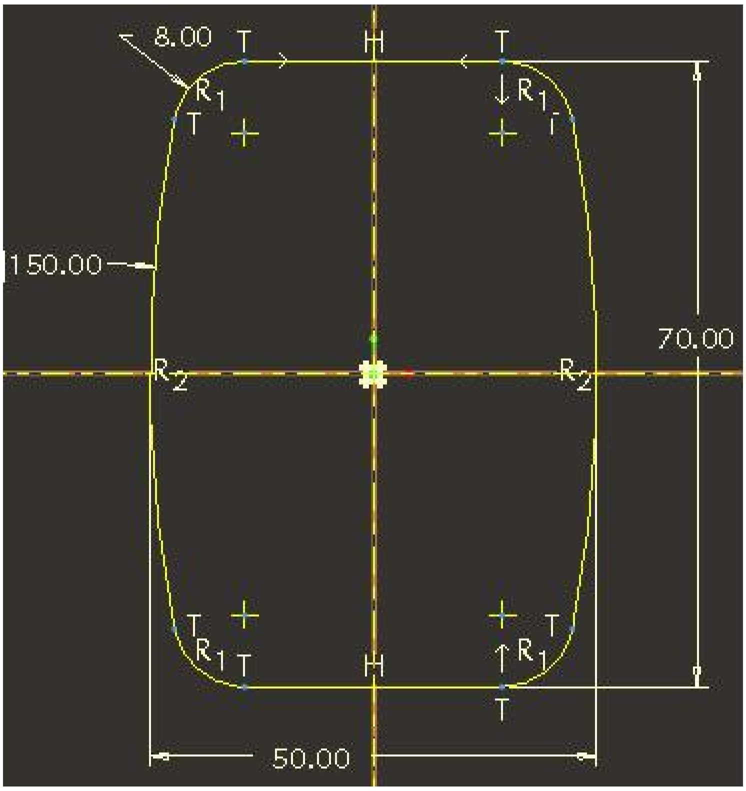
Now we will create the geometry in the skeleton model that will drive the housing assembly.

Although skeletons models can only be created in an assembly, they can be retrieved, manipulated, and saved as ordinary parts.

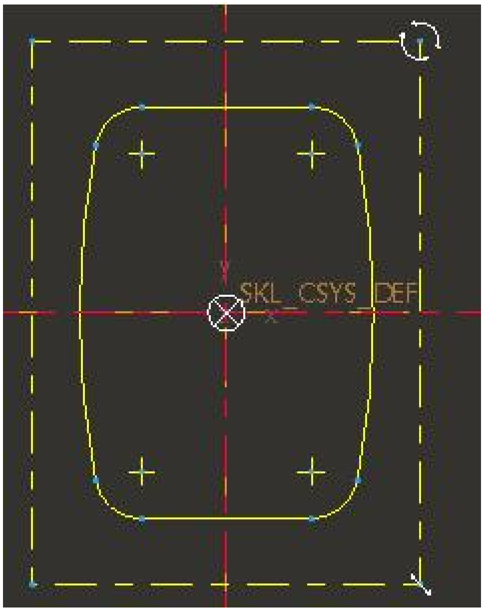
Pick and select the sketching references as shown below

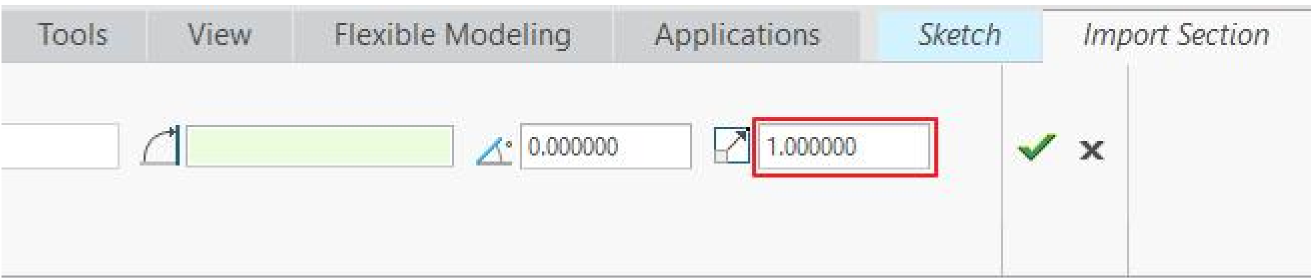


Now sketch the section as shown below. (For your convenience the section has been predefined. You can place this predefined section by picking  and opening the profile.sec)



Make sure that centerlines are aligned to the datum planes and Scale should be set to 1 as shown below.

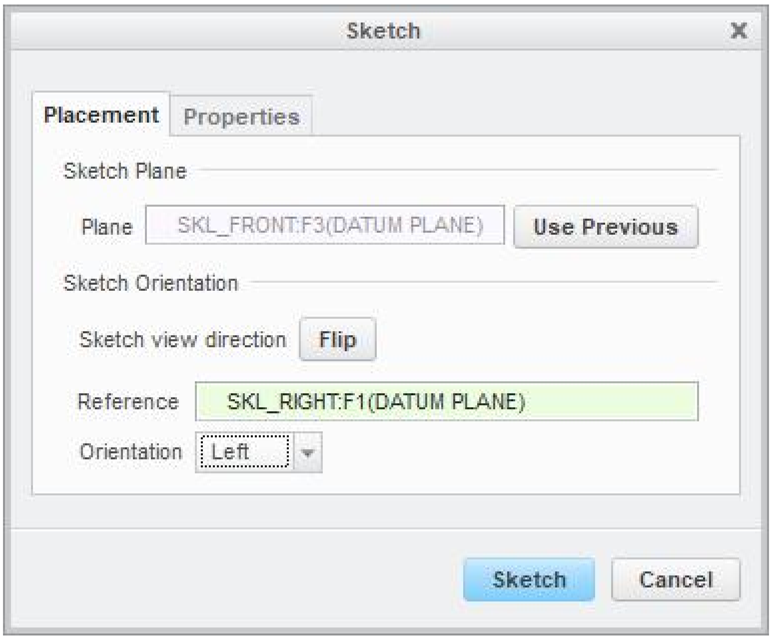




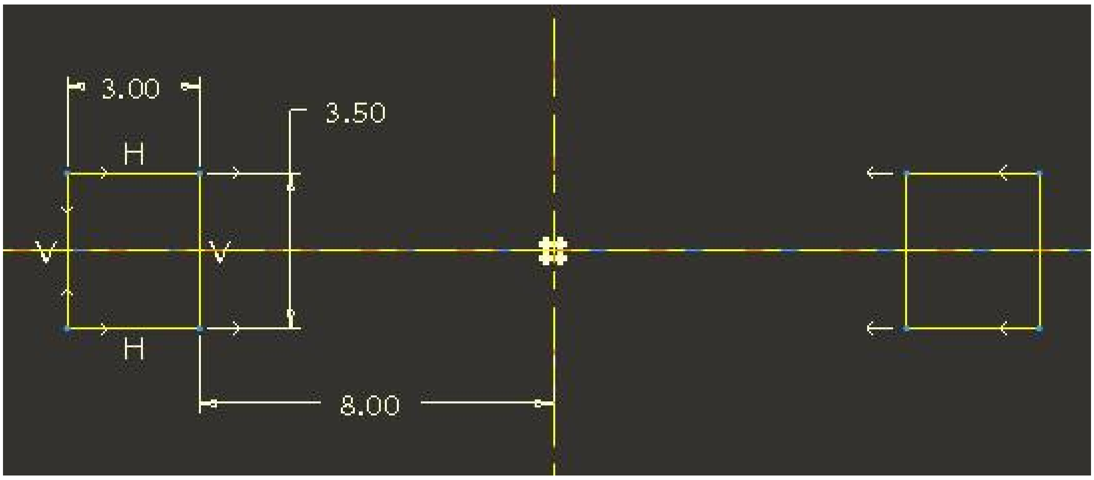
Pick to apply and exit the Import Section dashboard.

Again pick to complete the section. This datum curve will be used to drive the profile of both components.

Again pick and select the sketching references as shown below



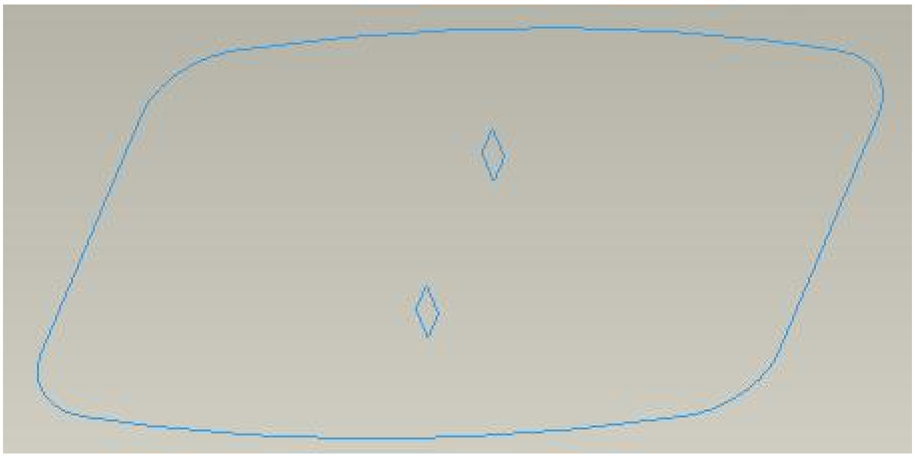
Now sketch the section as shown below.



Make sure that centerlines are aligned to the datum planes.

Pick to complete the section.

The Skeleton model will appear as shown below.

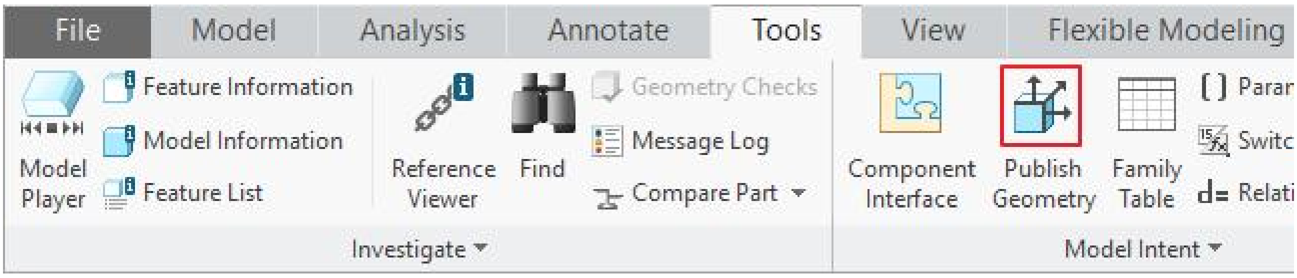


Communicating the Design Information

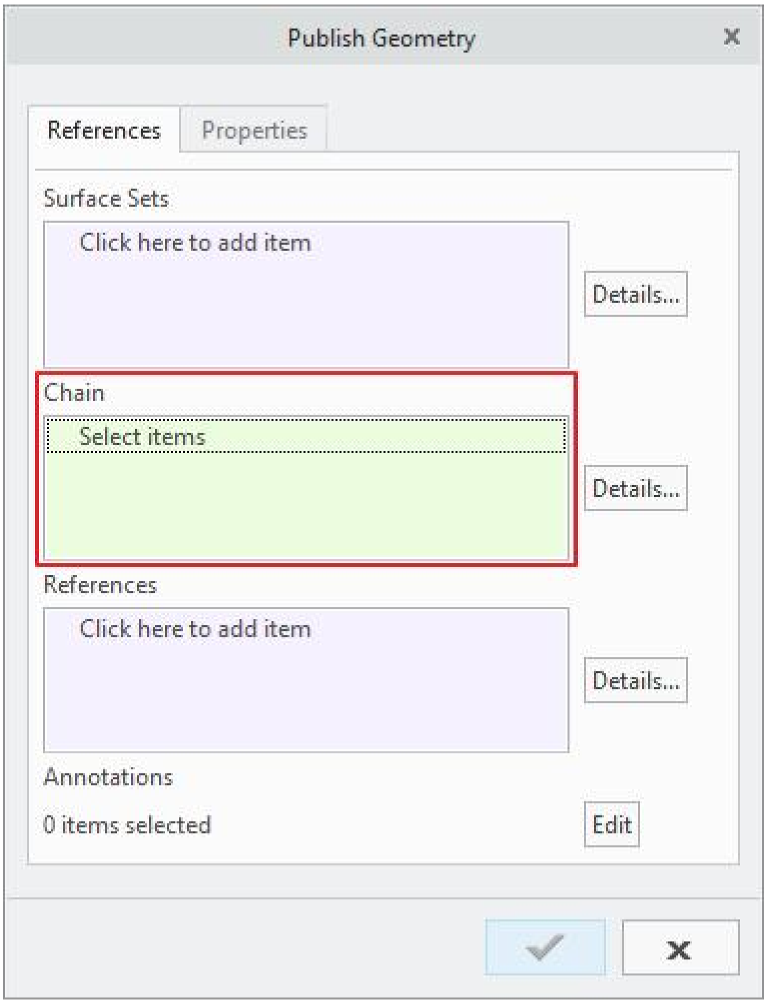
For this exercise we will use the Publish Geometry and Copy Geometry features to communicate design references from the skeleton to the assembled parts.

First we will create the Publish Geometry in the skeleton model. So make sure that skeleton model window is active.

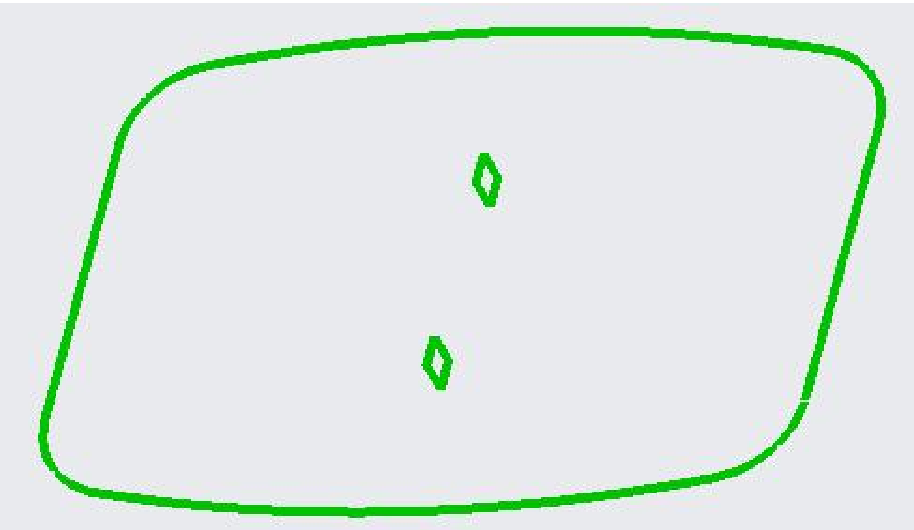
Pick  on the Tools tab as shown below.



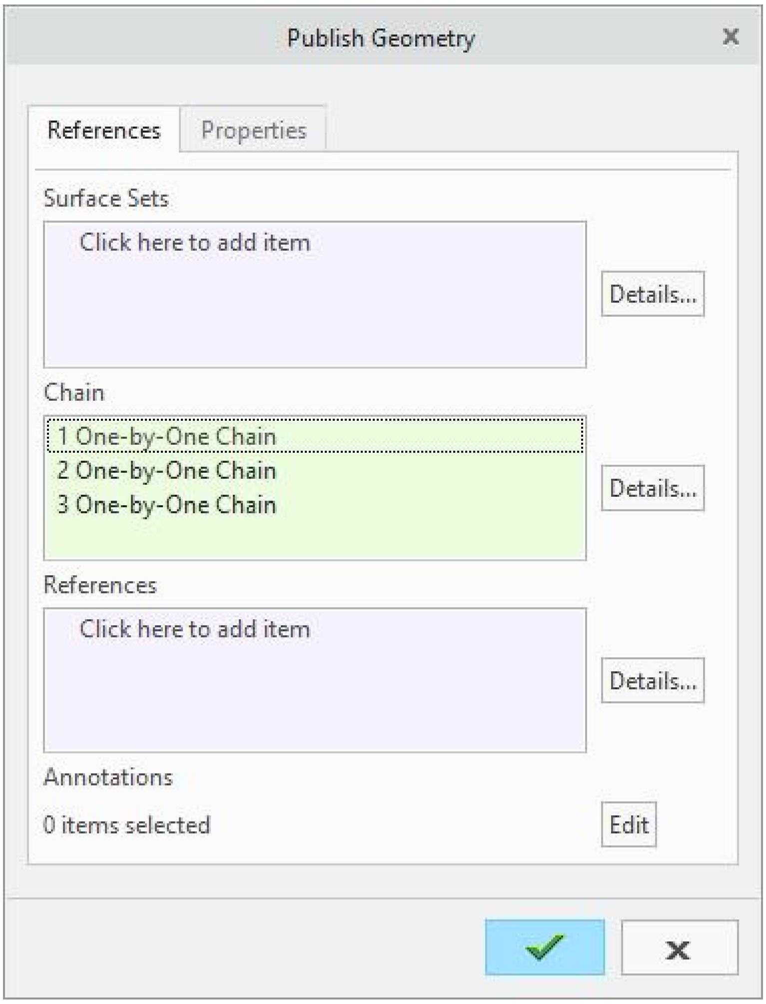
The Publish Geometry dialog box will appear. Pick in the Chain collector to make it active.



Hold down the Ctrl key and pick all the datum curves in the skeleton model as shown below



All the selected references (chains) will appear in the PUBLISH GEOMETRY dialog box as shown below.



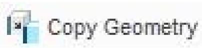
Publish geometry feature allows to mark the references, in a source part, that can be referenced by a copy geometry feature altogether. So it is a consolidation of multiple local references that can be copied to other models.

Pick  to apply the changes and exit the dialog box.

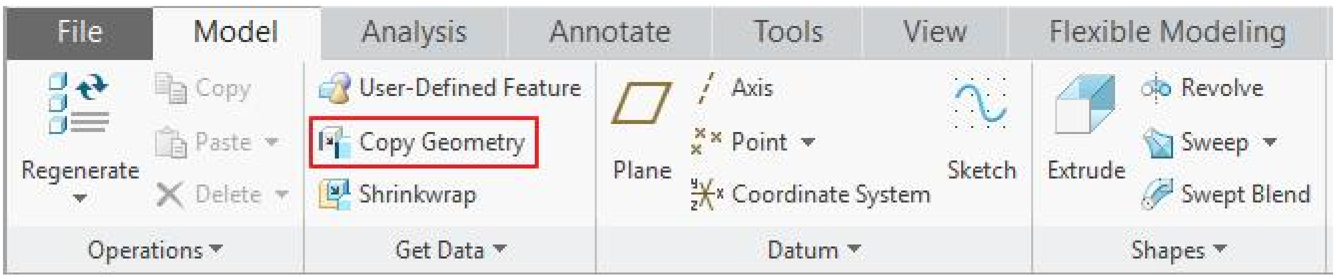
Now we will create Copy Geometry features in individual parts. This Copy Geometry feature will reference the Publish Geometry feature created in the skeleton part. We will create the Copy Geometry feature in the assembly environment so make the assembly window active.

First we will create a copy geometry feature in the HOUSING_TOP.PRT

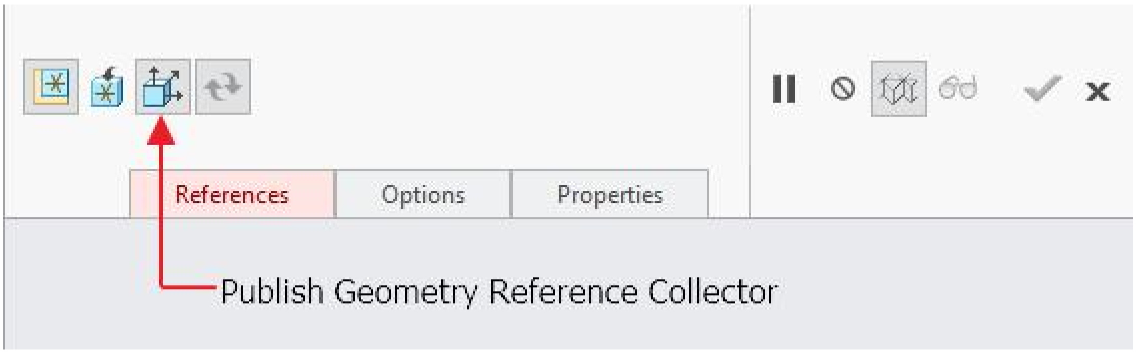
Pick the HOUSING_TOP.PRT in the model tree and select **Activate**.

To create a copy geometry feature, pick 

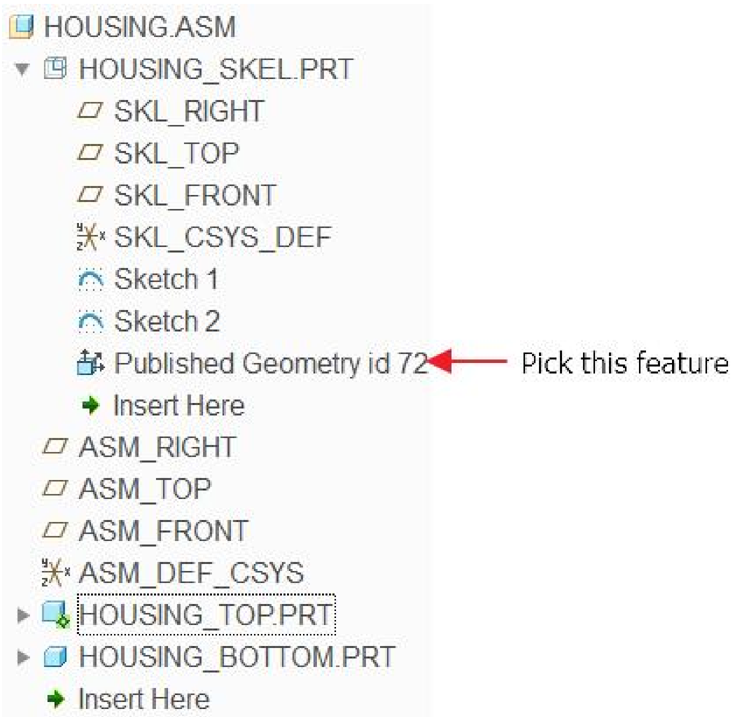
on the Model tab.



The Copy Geometry dashboard will appear as shown below.



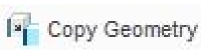
Notice that Publish Geometry reference collector is active by default. So select the publish geometry feature in the skeleton part by picking it in the model tree as shown below.



When you create a Copy Geometry feature, you can reference either individual references like curve, surface and datum features or a Publish Geometry feature.

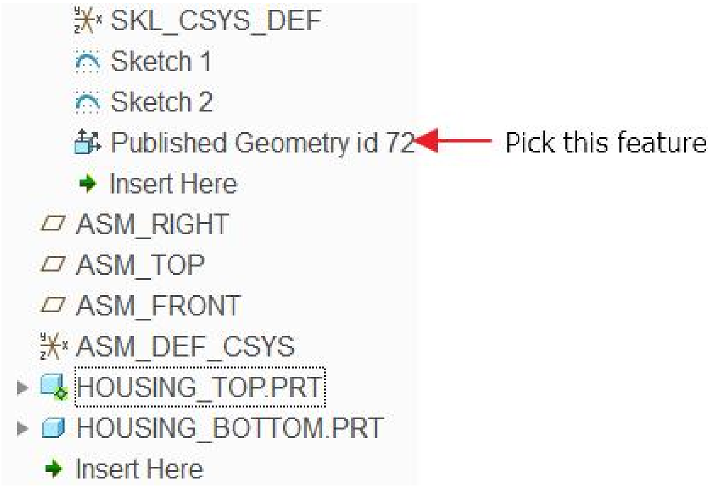
Pick  to apply the changes and exit the dashboard.

Now pick the HOUSING_BOTTOM.PRT in the model tree and select **Activate**.

To create a copy geometry feature, pick 

on the Model tab.

The Copy Geometry dashboard will appear. As Publish Geometry reference collector is active by default so select the publish geometry feature in the skeleton part by picking it in the model tree.



Pick to apply the changes and exit the dashboard.

Creating Model Geometry

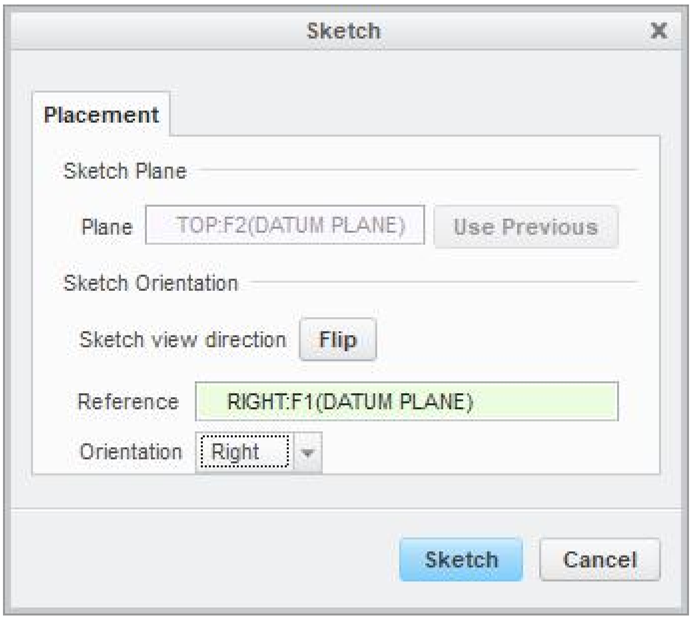
Now the information is captured from the skeleton model into the individual components. We will use the Copy Geometry features as a reference to build geometry in the parts.

First Part

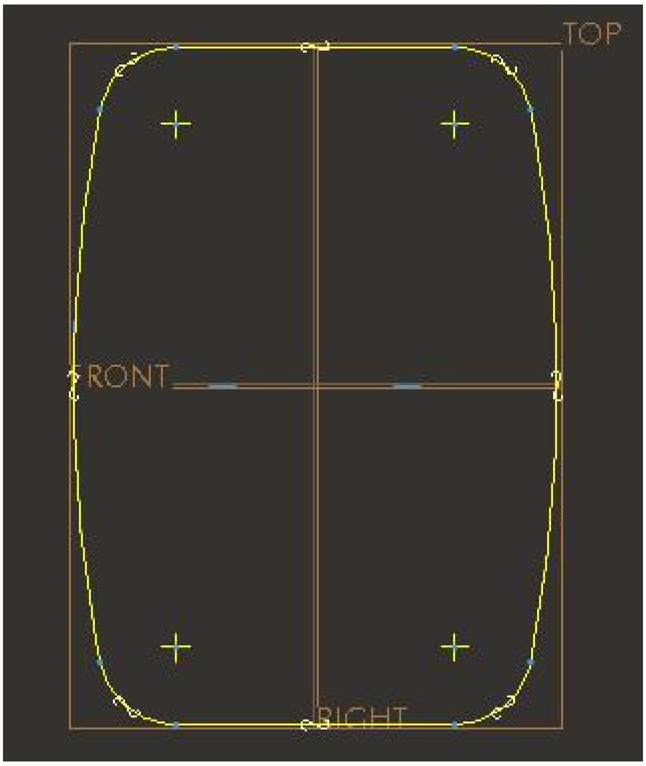
Open the HOUSING_TOP.PRT in the new window

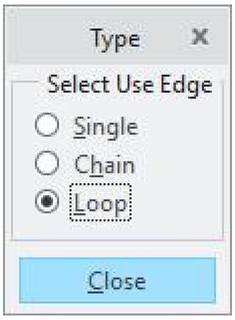
Pick the Extrude Tool icon .

Select the sketching references as shown below.

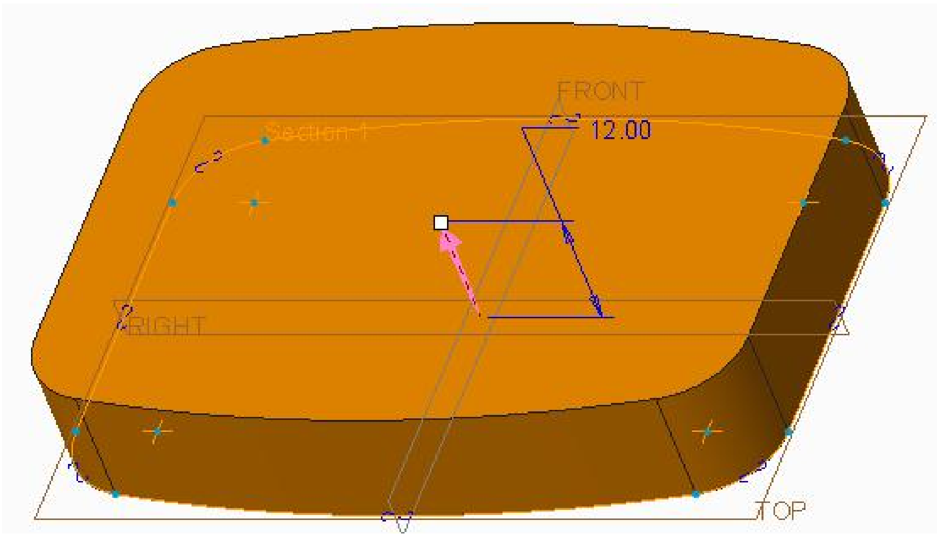


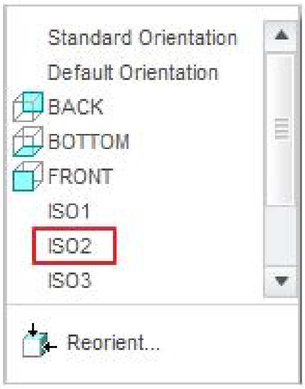
Pick Project icon and select the following loop.





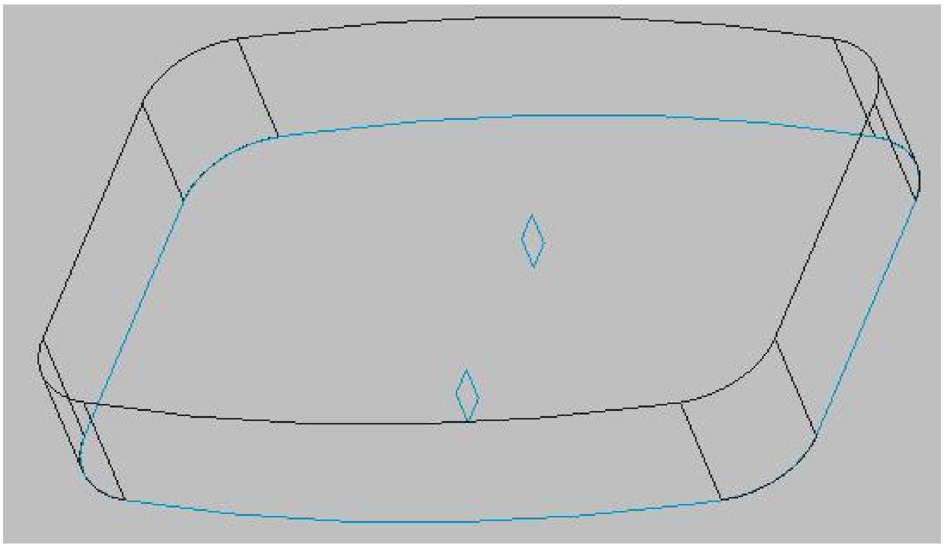
After completing the sketch specify the blind depth of **12** in the direction as shown below. (Note that extrusion direction is towards brown side of the datum plane/positive direction of y-axis. Set the view to **ISO2** to avoid any ambiguity.)





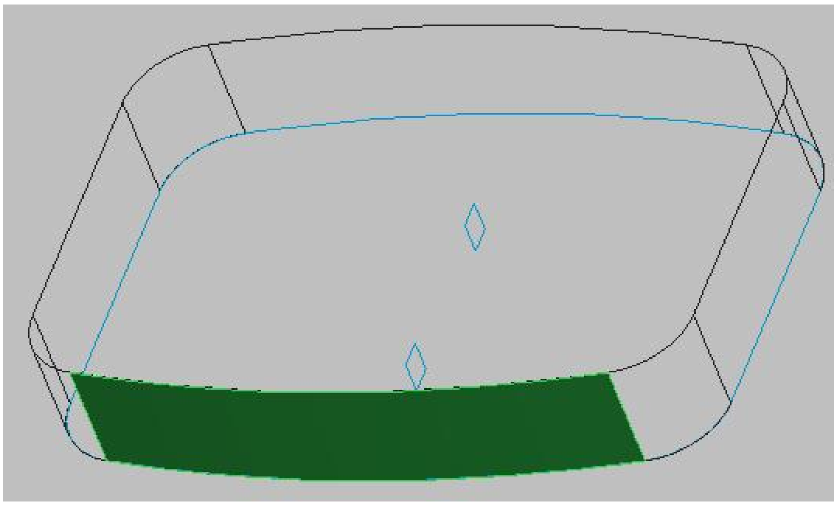
Pick  icon or middle-click to complete the feature.

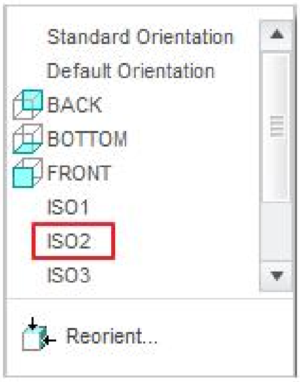
The part will appear as shown below.



Now we will apply draft and rounds to this part then shell it.

Select the following surface. Set the view to **ISO2** to avoid any ambiguity.

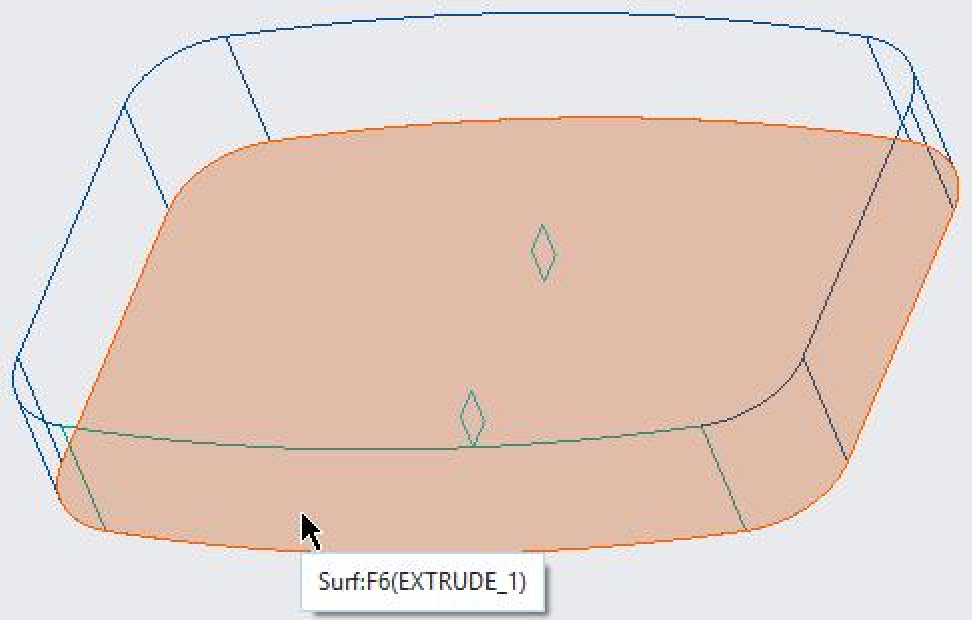


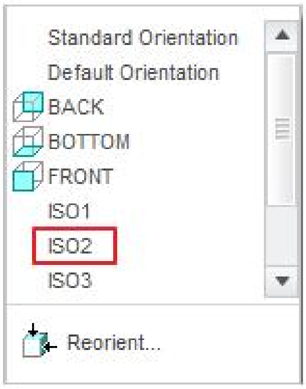


Pick 

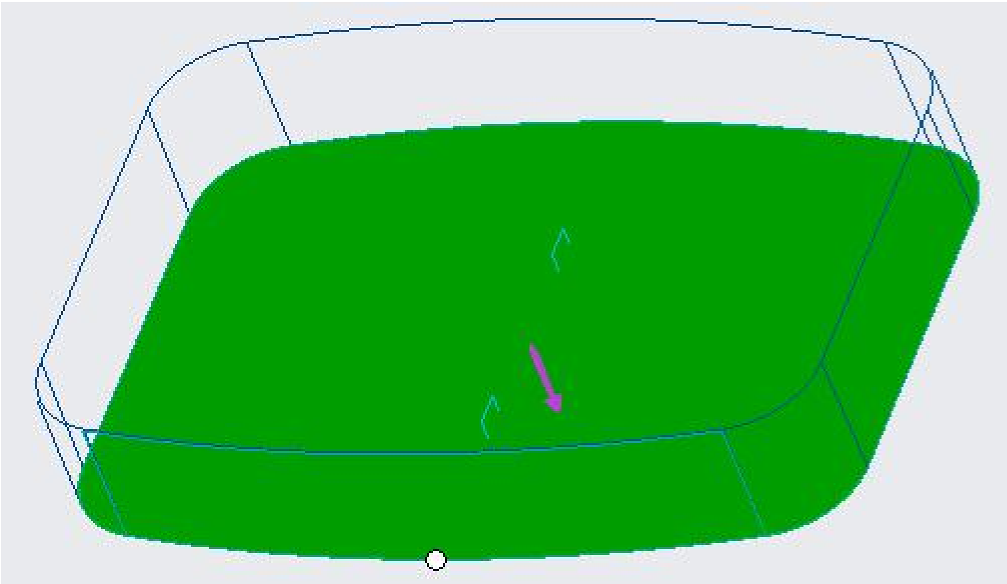
to access the Draft tool.

Notice that Draft hinges collector is active so pick highlighted surface as the draft hinge.





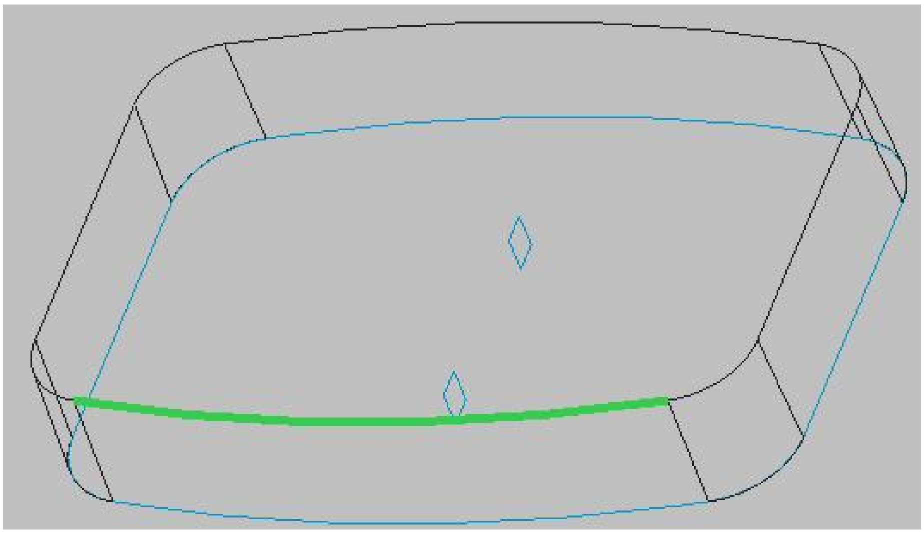
Enter ***2*** in the Draft angle field on the dashboard. The preview of the drafted surface will appear as shown below.

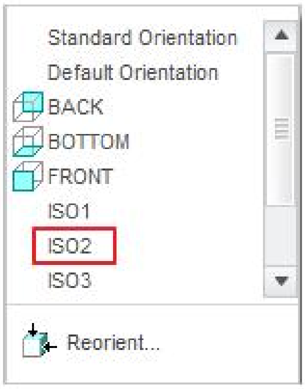


Make sure that Pull Direction arrow points in the direction shown.

Pick  icon to complete the feature.

Select the following edge for applying round.





Pick 

to access the Round tool.

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

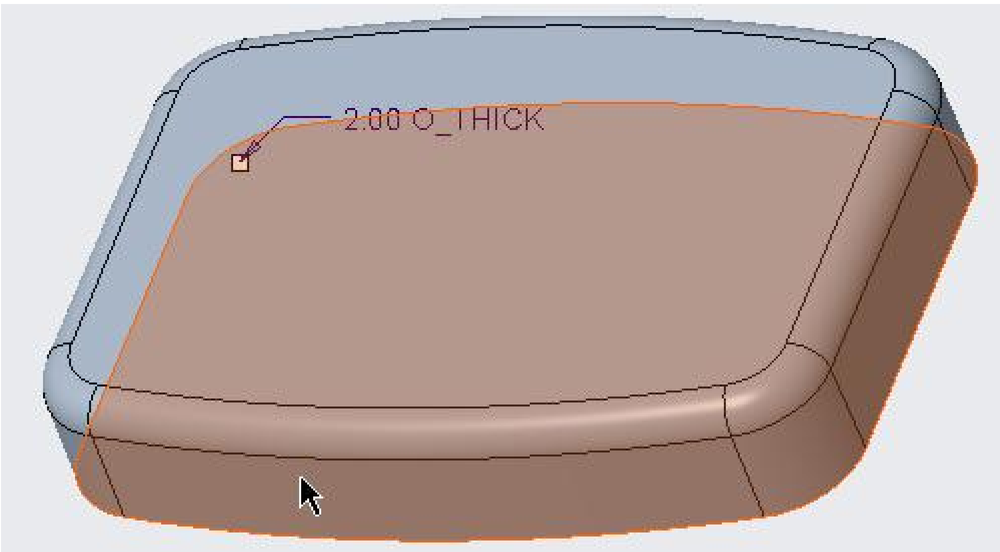
Pick  icon to complete the feature.

Pick 

to access the Shell tool.

Enter ***2*** as the thickness value

Select the following highlighted surface to add it to Removed surfaces collector.

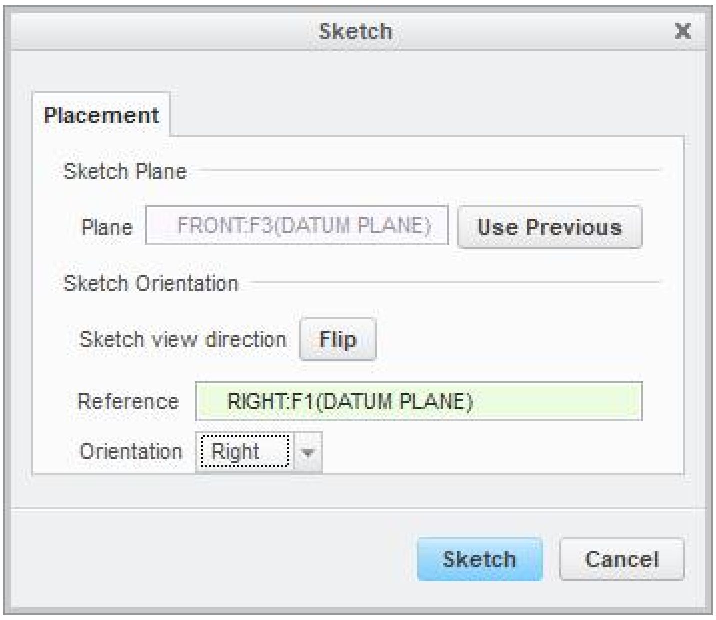


Pick  icon to complete the feature.

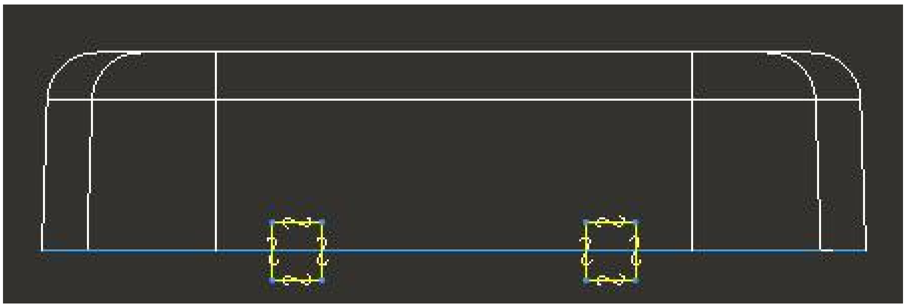
To create the cuts in the side wall we will create an extrude feature.

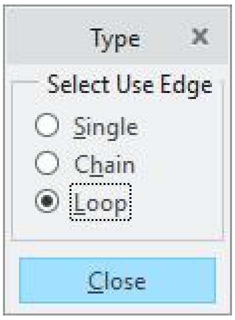
So pick  to invoke Extrude tool.

Select the sketching references as shown below



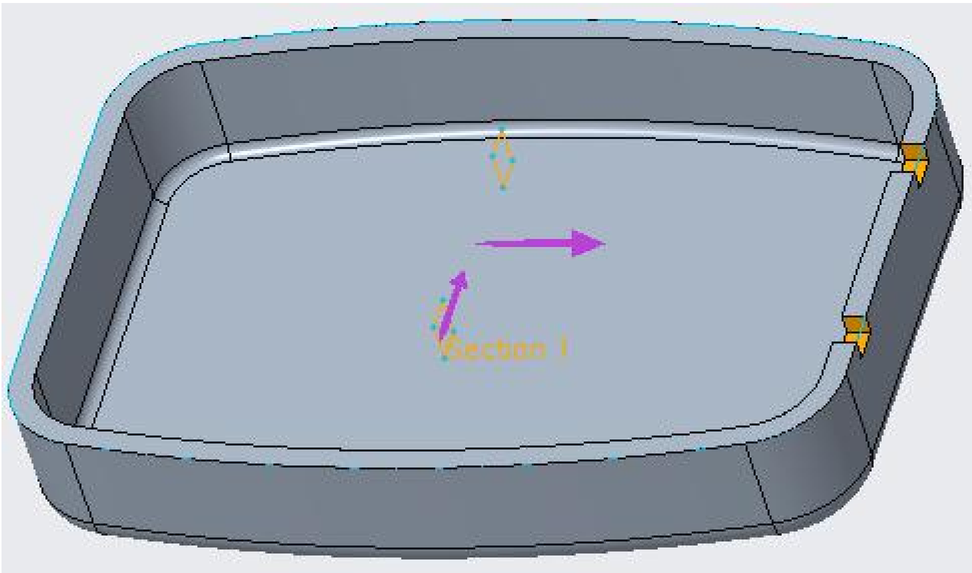
Pick  icon and select the following loop.

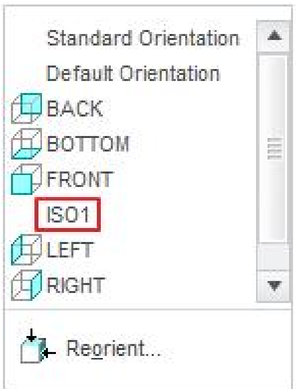




After completing the sketch pick  icon to create the feature as cut.

Specify the depth option to **Through All.**





Pick icon or middle-click to complete the feature.

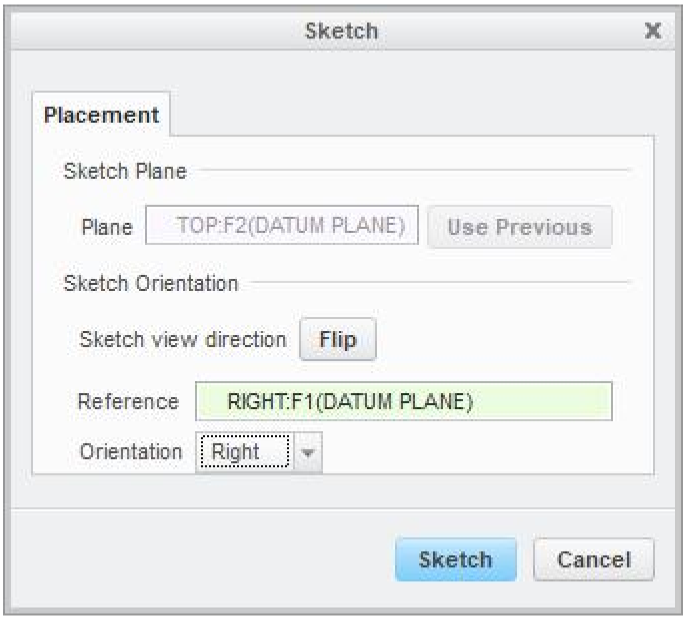
This completes HOUSING_TOP except the interlock detail which we will create later.

Second Part

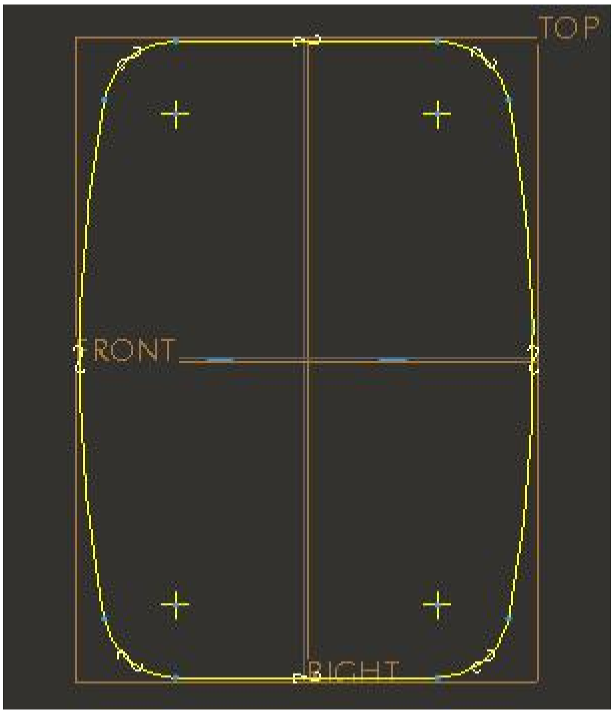
Now we will create the geometry for second part. So open the HOUSING_BOTTOM.PRT in the new window.

Pick the Extrude Tool icon

Select the sketching references as shown below

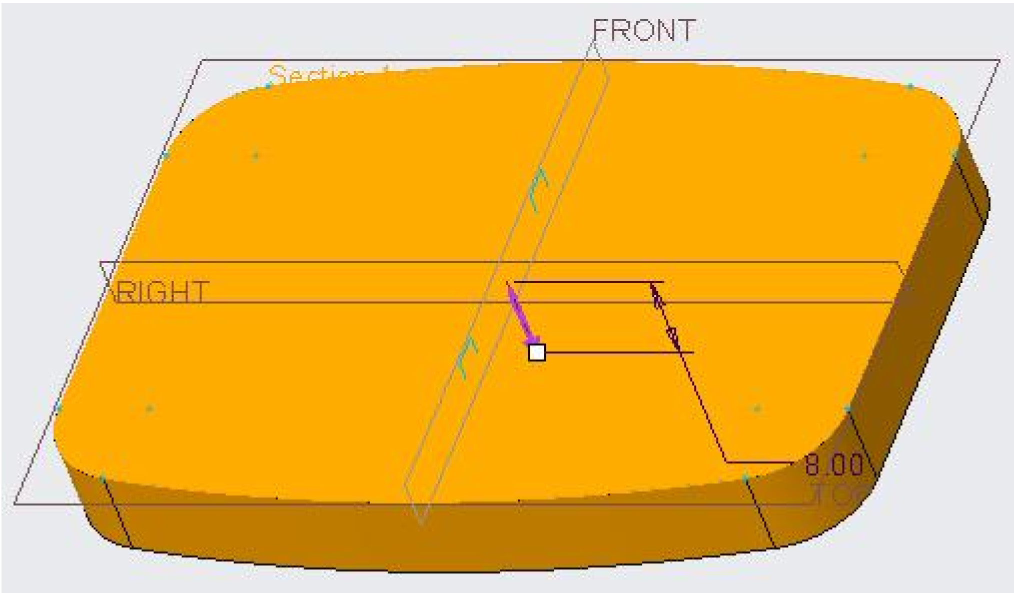


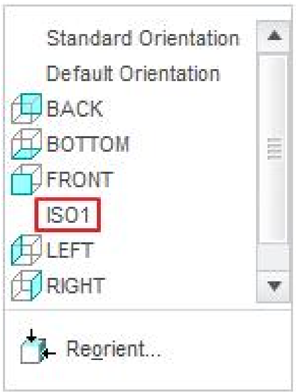
Pick Project icon and select the following loop.



After completing the sketch specify the blind depth of **8**

Pick to flip the direction of feature creation.



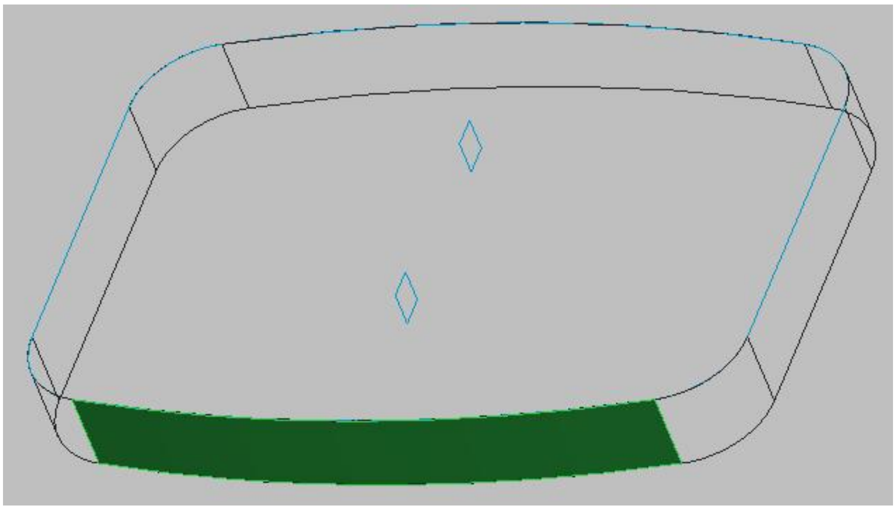


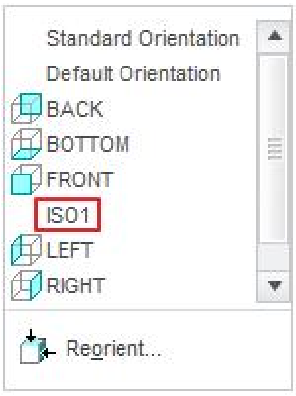
Make sure that extrusion directions is as shown above.

Pick icon or middle-click to complete the feature.

Now we will apply draft and then shell the part.

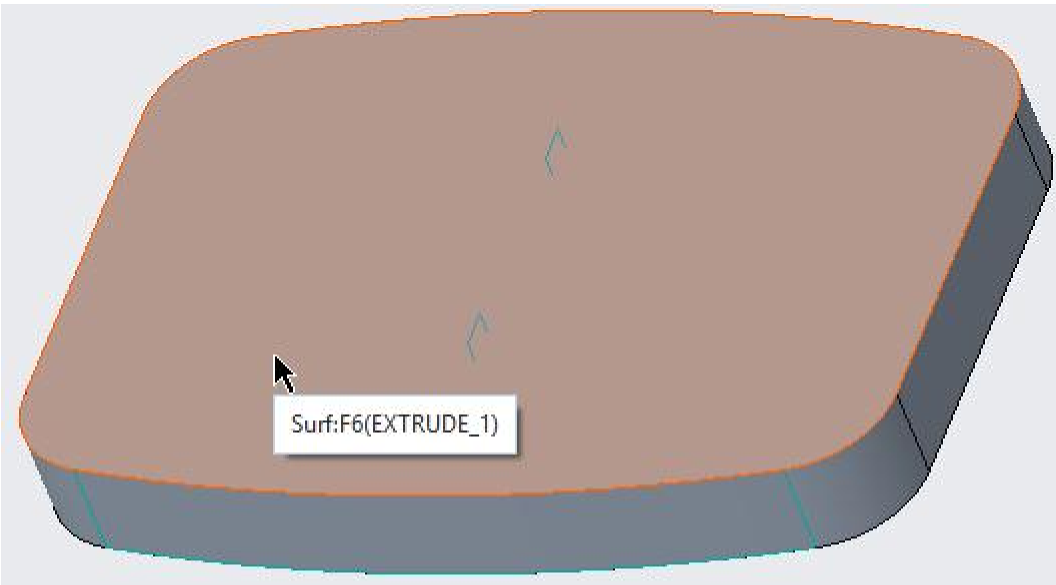
Select the following surface.





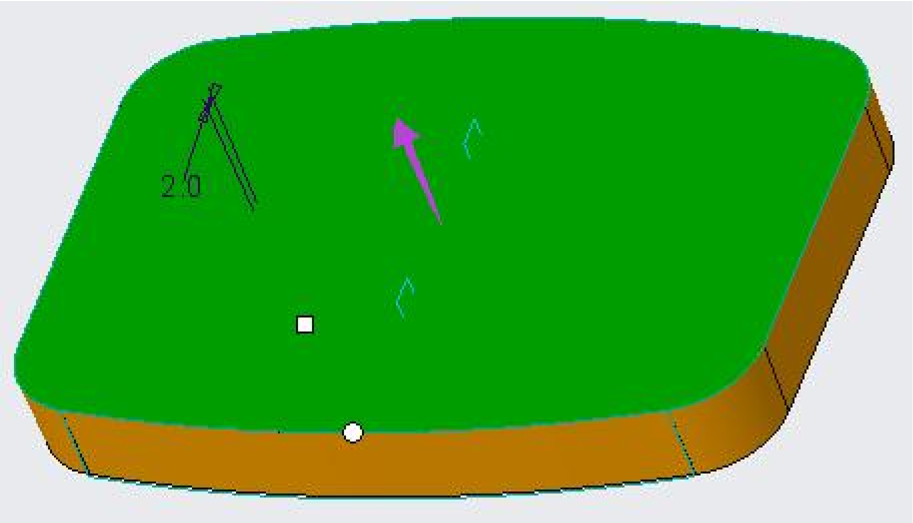
Pick Draft to access the Draft tool.

Notice that Draft hinges collector is active so pick the following surface as the draft hinge.



Enter ***2*** in the Draft angle field on the dashboard.

The preview of the drafted surface will appear as shown below.



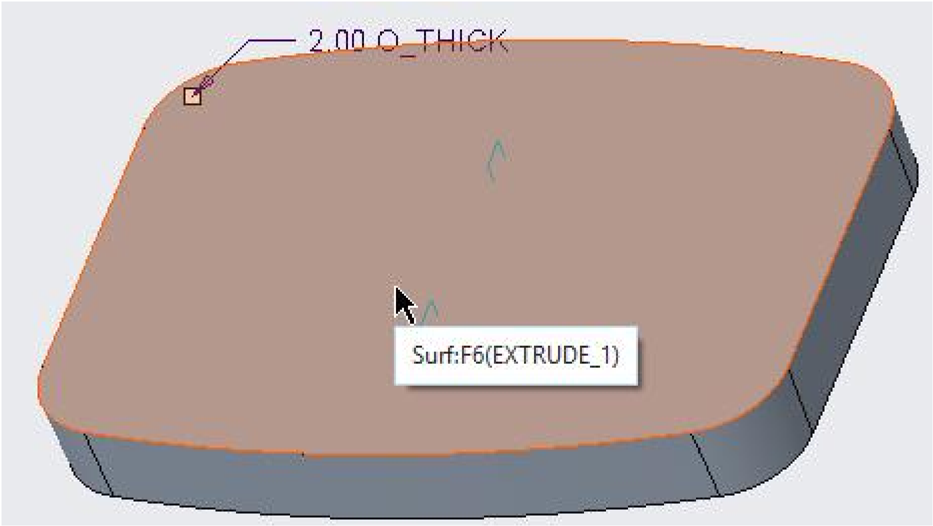
Make sure that Pull Direction arrow points in the direction shown.

Pick icon to complete the feature.

Pick Shell to access the Shell tool.

Enter ***2*** as the thickness value

Select the following surface to add it to Removed surfaces collector.

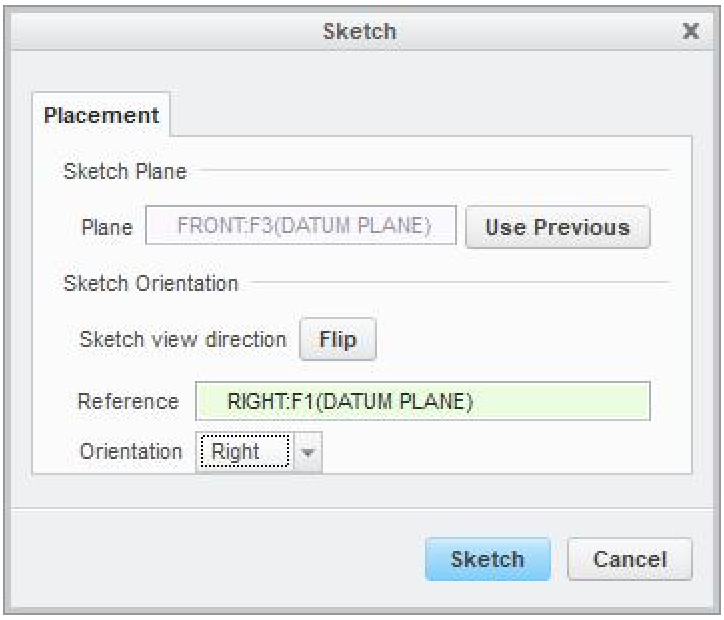


Pick icon to complete the feature.

To create the cuts in the side wall, we will create an extrude feature.

So pick to invoke Extrude tool.

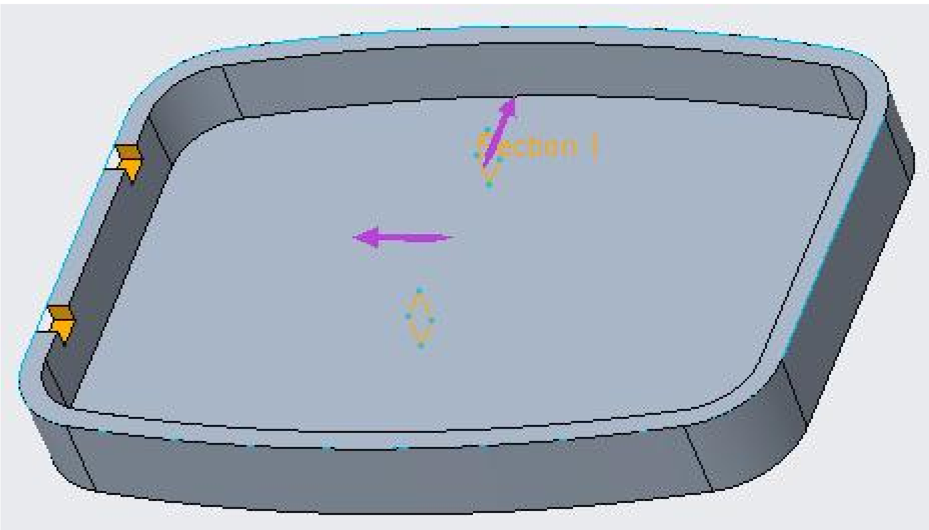
Select the sketching references as shown below

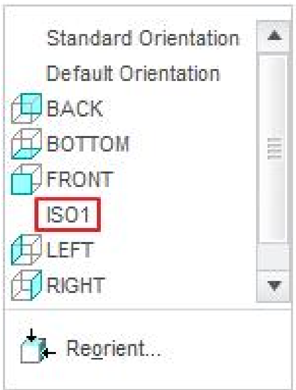


Pick  icon and select the following loop.

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

Specify the depth option to **Through All.**



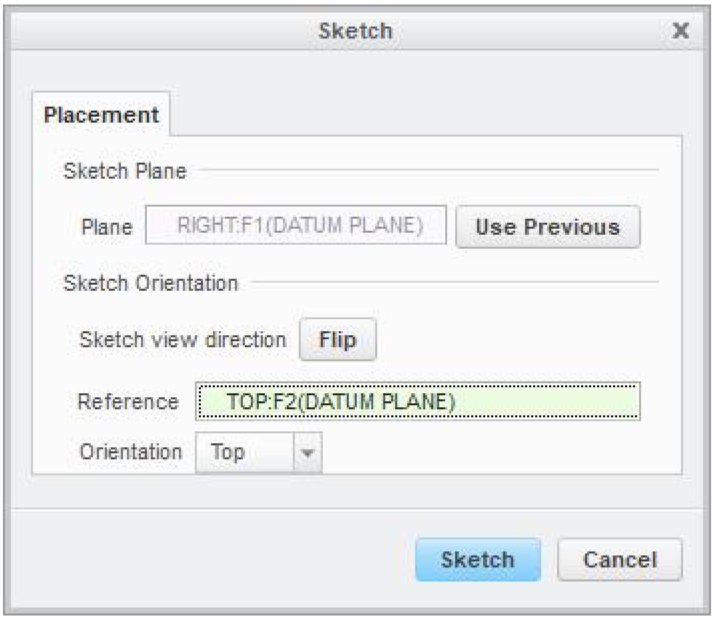


Pick  icon or middle-click to complete the feature.

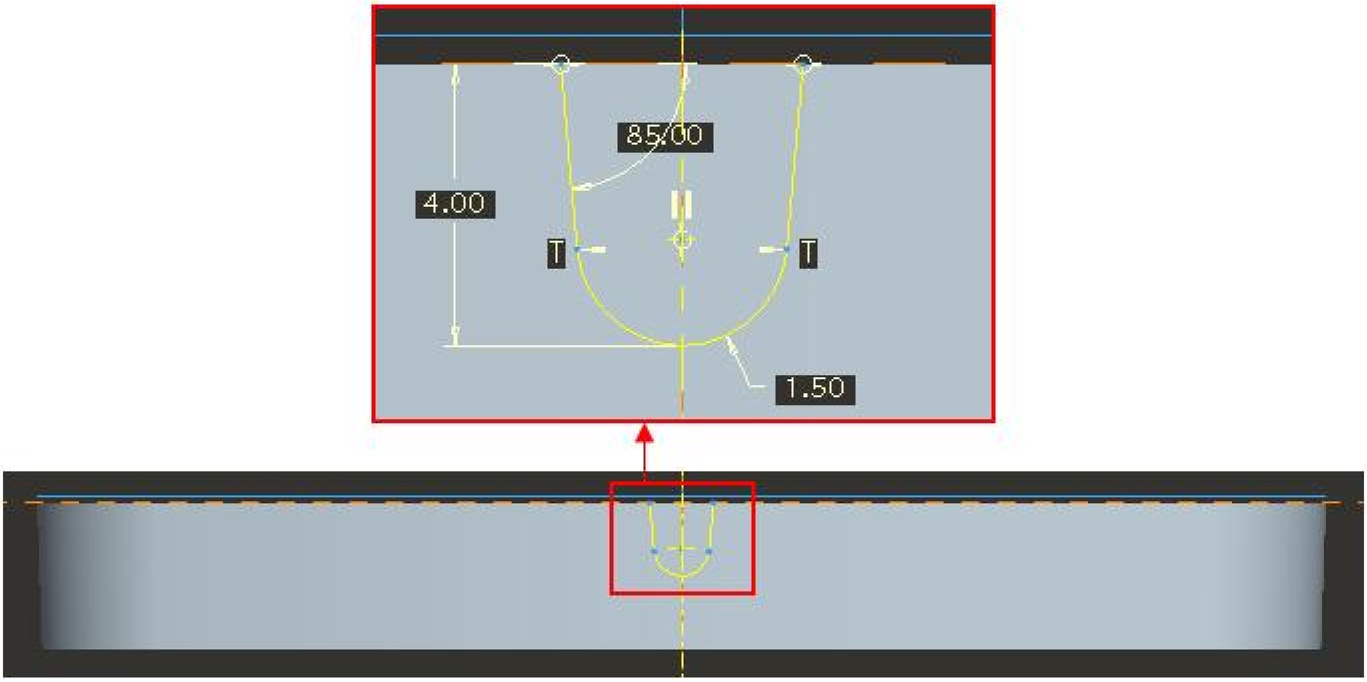
Now we will create a cut in the front wall.

Pick to invoke Extrude tool.

Select the sketching references as shown below.

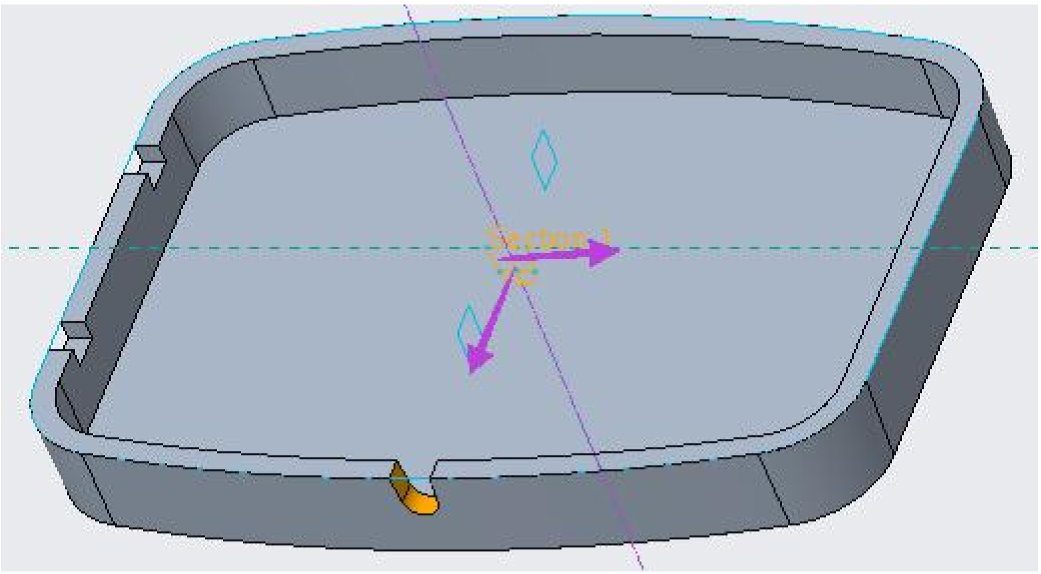


Sketch the section as shown below.



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

Specify the depth option to **Through All.**

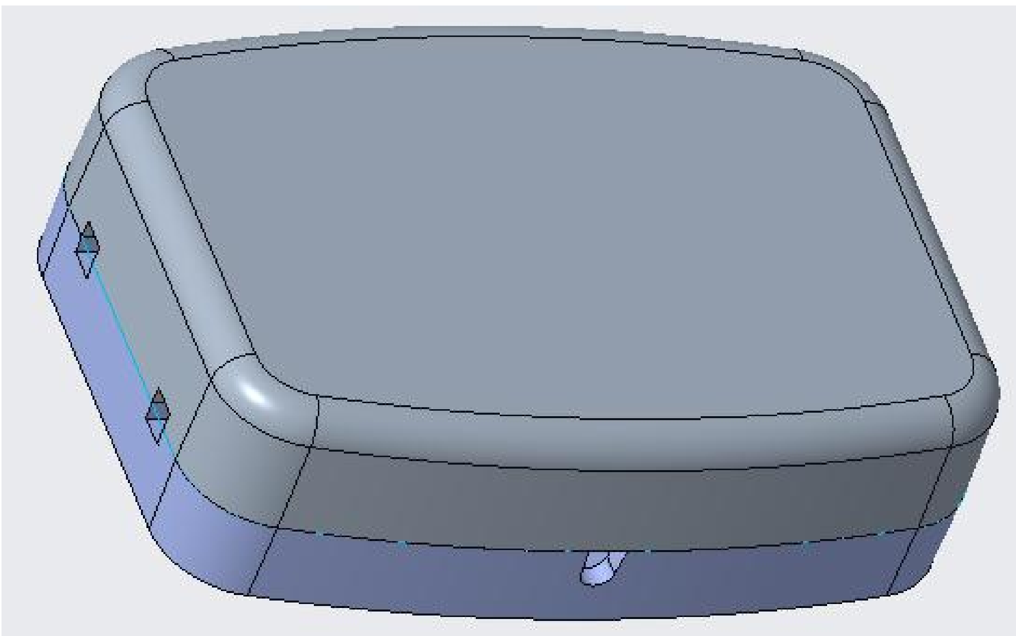


Pick icon or middle-click to complete the feature.

Notice that we did not create any geometry in the skeleton model for this cut feature. It is so because the location and size of this cut feature does not relate to any geometry in any other parts of the assembly.

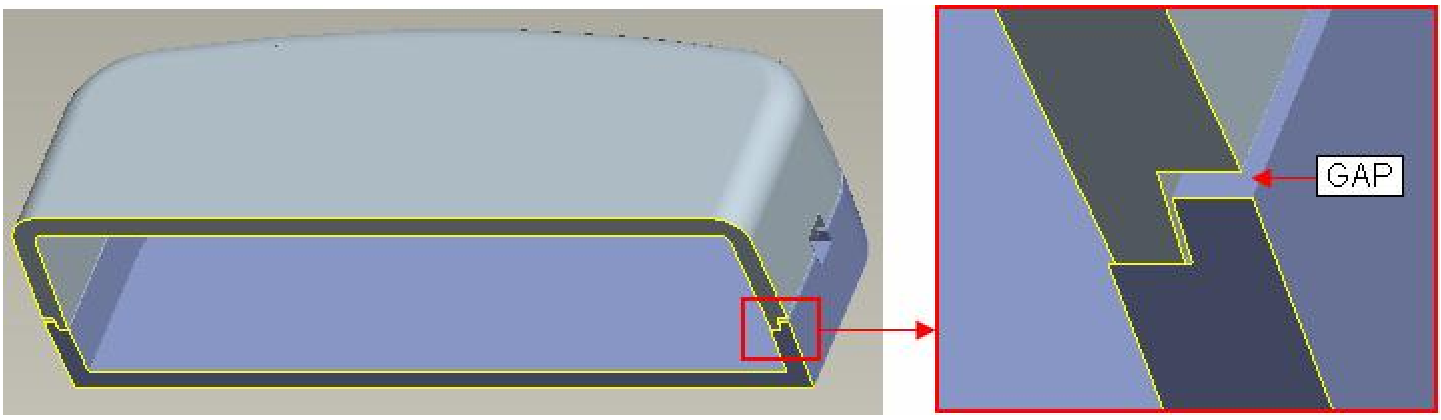
When some geometry or feature is only related to a single component in the assembly then there is no need to place a reference for it in the skeleton.

The assembly of parts will appear as shown below.



Interlock Details

Now we will create the interlock detail so that both parts are aligned and positioned relative to each other during the assembly. The cross-section of assembled parts, after creating the interlock geometry, will appear as shown below.



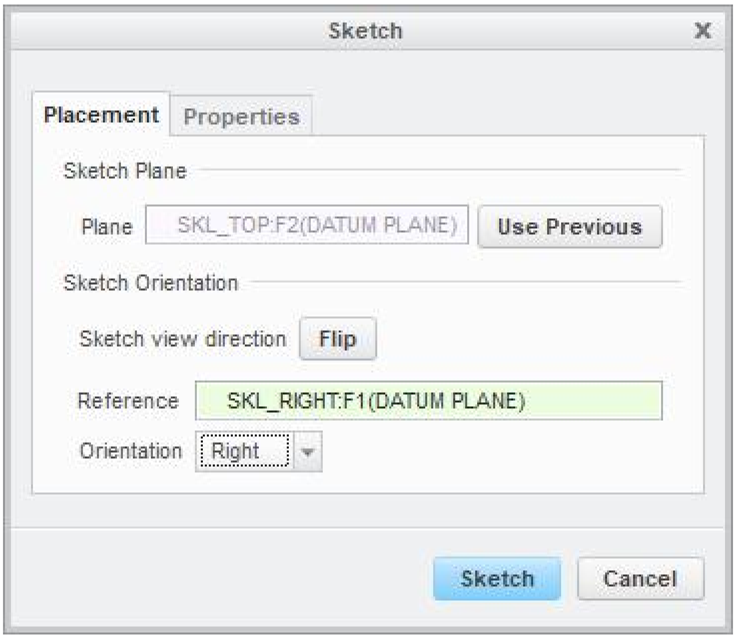
You should notice the gap between Top and Bottom halves. The purpose of this gap is to disguise any mismatch between these parts.

First we will create a datum curve in the skeleton and add it to the publish geometry feature. This datum curve will be used to make sure that interlock geometry in both parts does not interfere.

The interlock lip is used to aid the alignment and positioning of two mating components during assembly process.

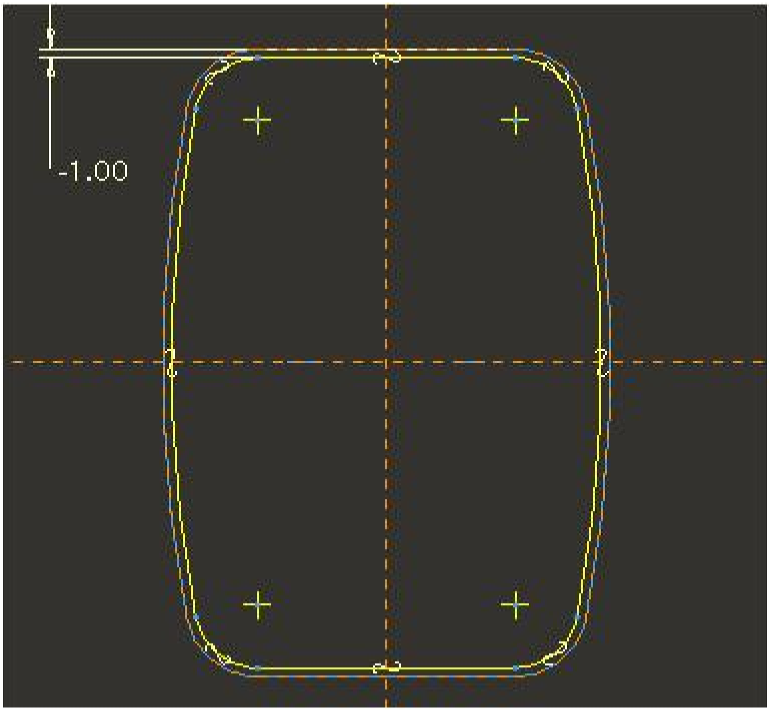
Open the skeleton model in a separate window.

Pick  and select the sketching references as shown below



Pick 

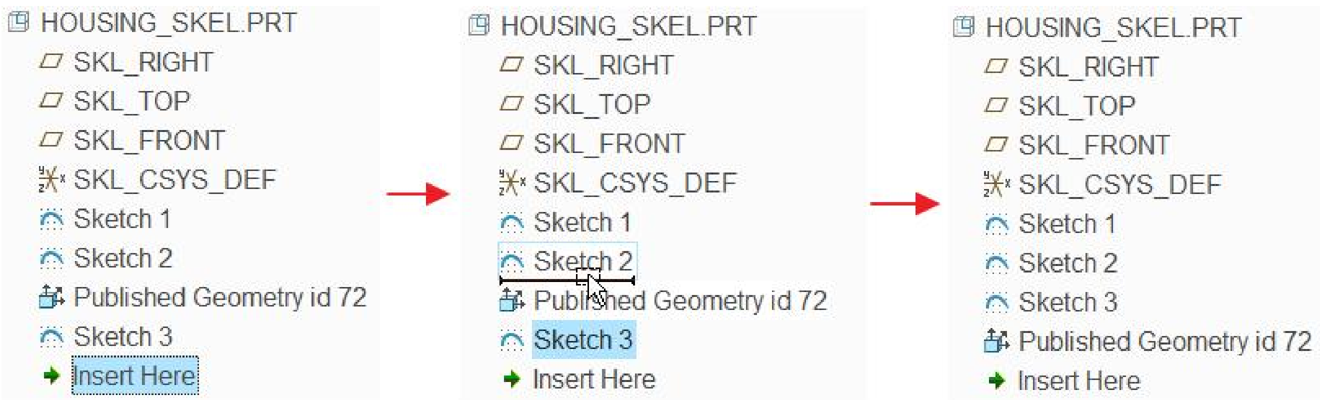
icon and create a loop that is offset by ***-1*** from the profile curve as shown below.



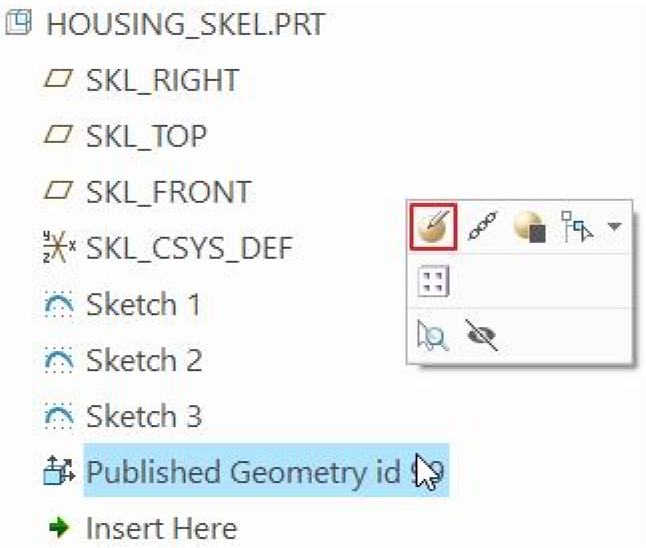
Pick  to complete the section.

Now we will redefine the publish geometry feature and add this new curve. But first we should reorder this new sketched curve to appear before the publish geometry feature.

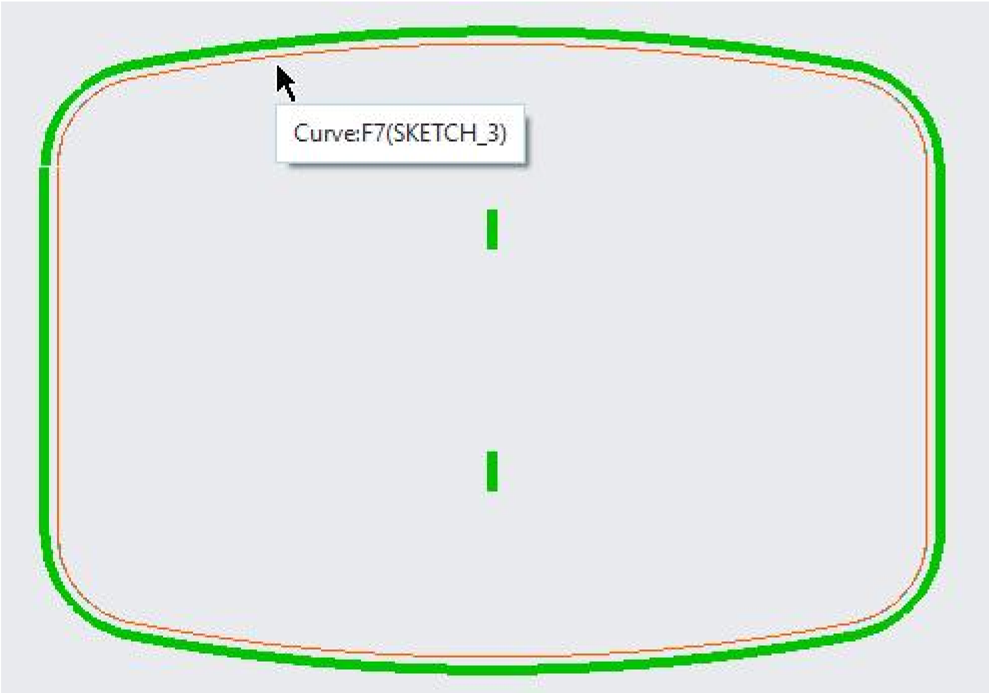
So drag and drop the "Sketch 3" before the publish geometry feature as shown below.



Select the publish geometry feature and pick 

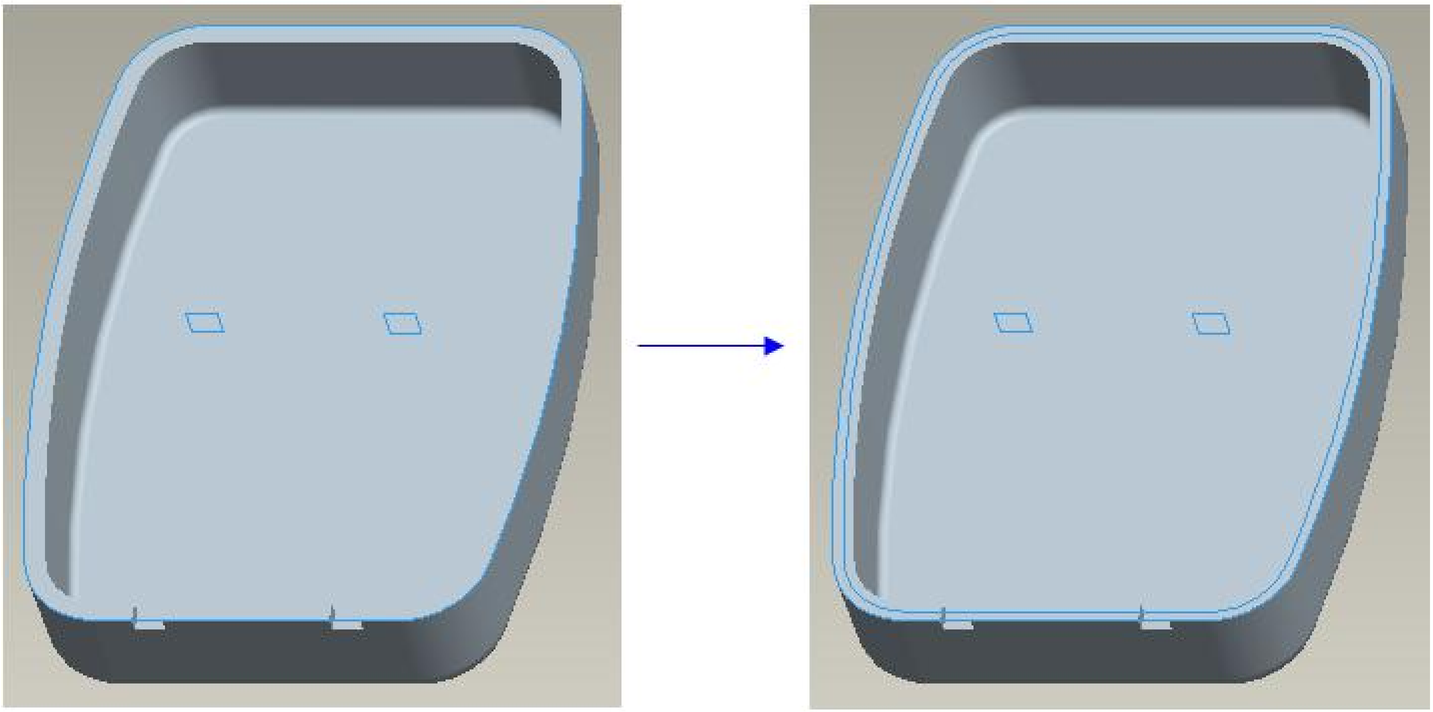


Activate the Chains collector and pick the curve formed by SKETCH_3 feature while holding down the Ctrl key.



Pick ✓ to apply the changes and exit the dialog box.

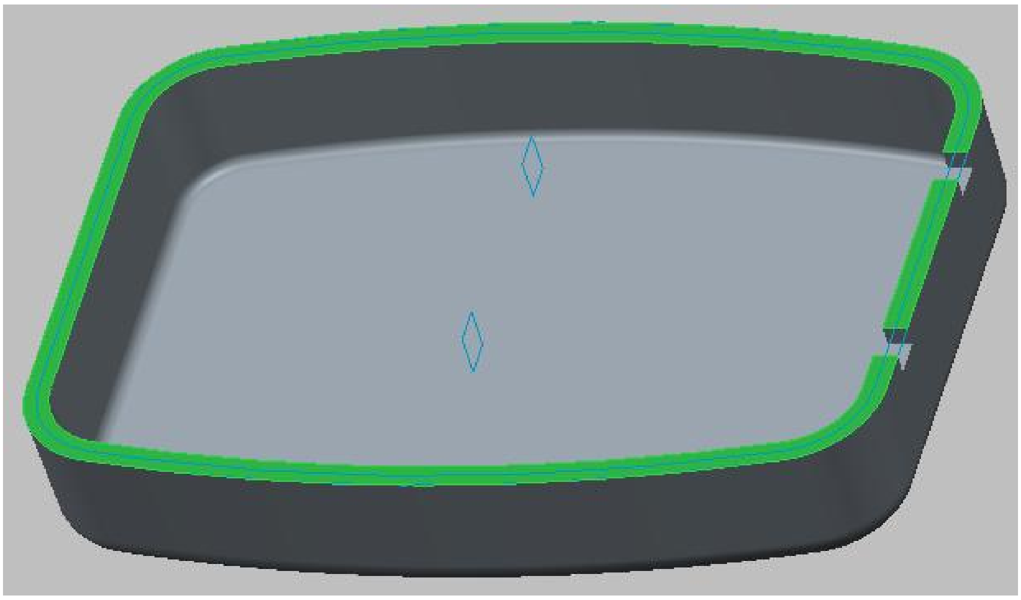
Now open the HOUSING_TOP.PRT in separate window and regenerate the part by picking icon. Notice that the curve that we just added in the publish geometry appears in this part as shown below.



It is because the copy geometry feature in this part is referring the publish geometry feature in the skeleton model. Therefore the geometry added or removed in publish geometry feature will be reflected here.

Now we will define an Offset feature to create the interlock lip.

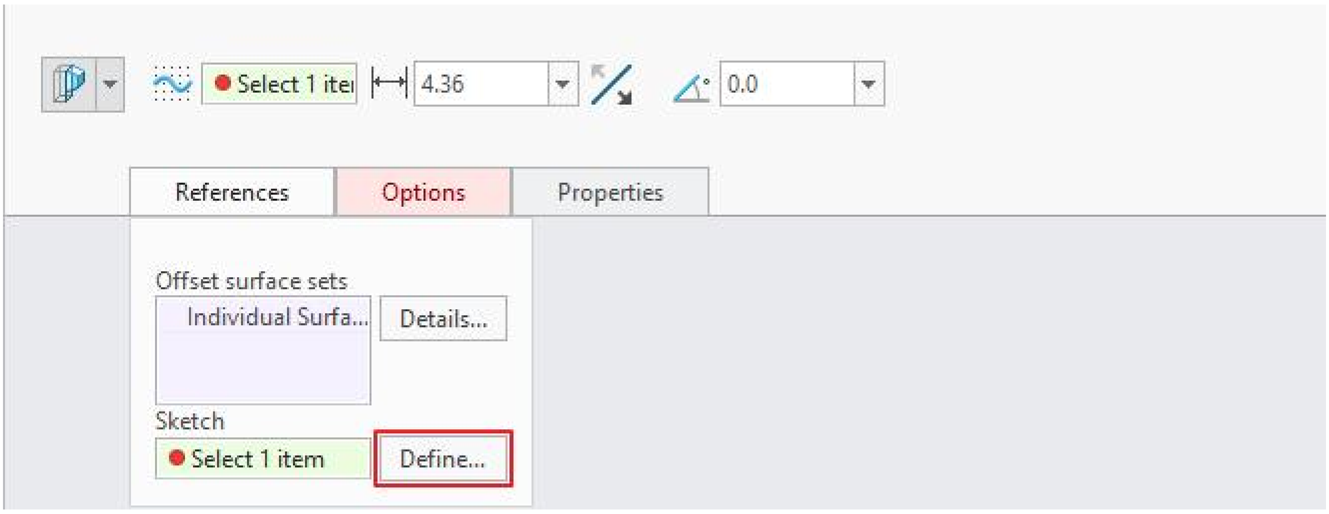
So first select the following surface of the part.



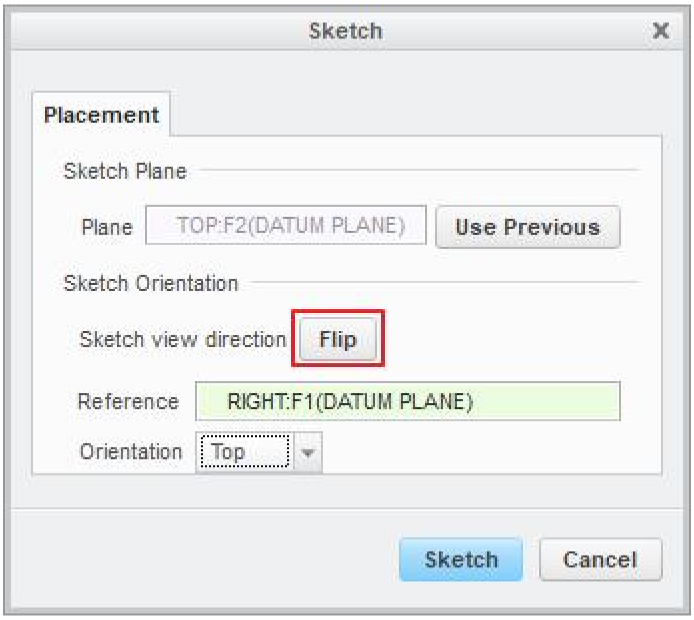
Pick Offset icon on the Model tab to access the Offset tool.

Change the Offset type to **With Draft** by picking the icon in the Offset Type list.

Pick Define... tab in the References slide-up panel.

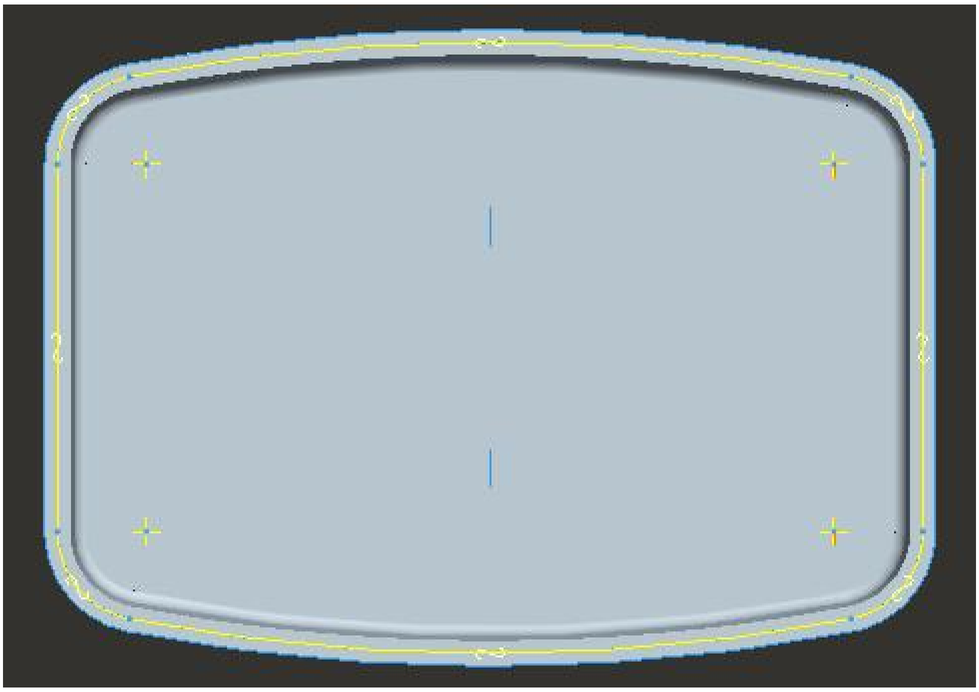


Select the sketching references as shown below. Also flip the direction of sketch plane for clear view.



Pick 

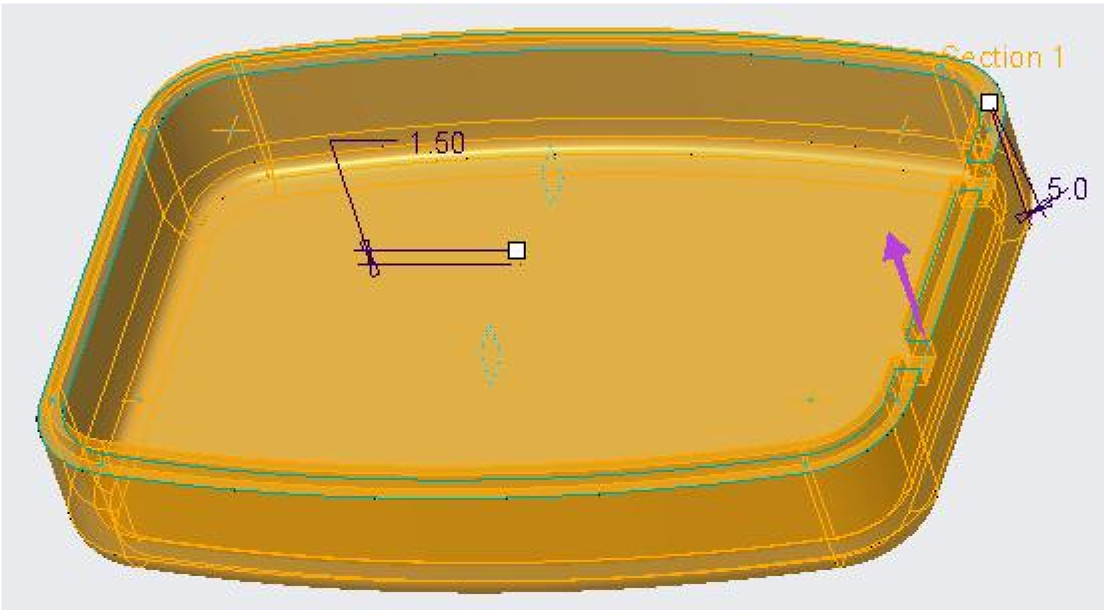
icon and select the newly added loop as shown below.



After completing the sketch enter ***1.5*** as the offset value and hit Enter key.

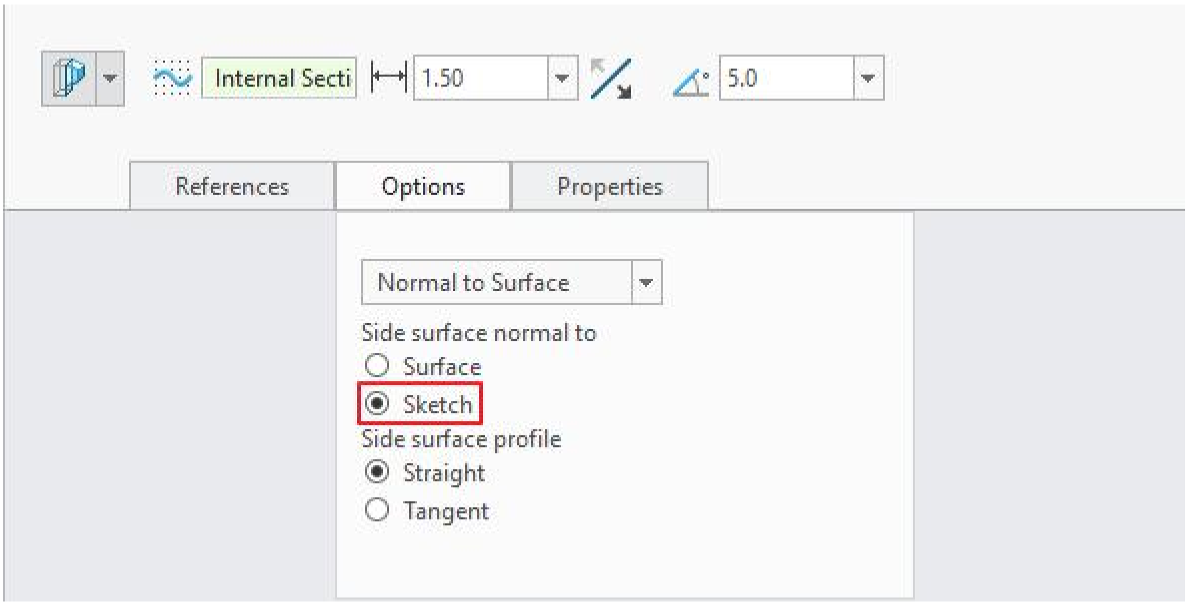
Enter ***5*** as the draft angle and hit Enter key.

The preview will appear as shown below.

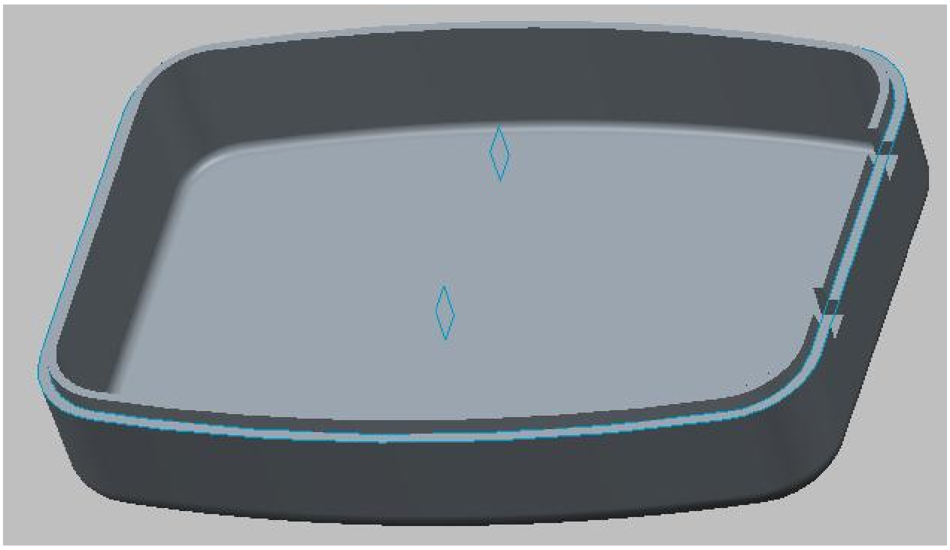


Make sure that Offset Direction arrow points in the direction shown. It will make sure that material is added to the part.

Pick Options tab and change the “Side surface normal to” option to **Sketch**.

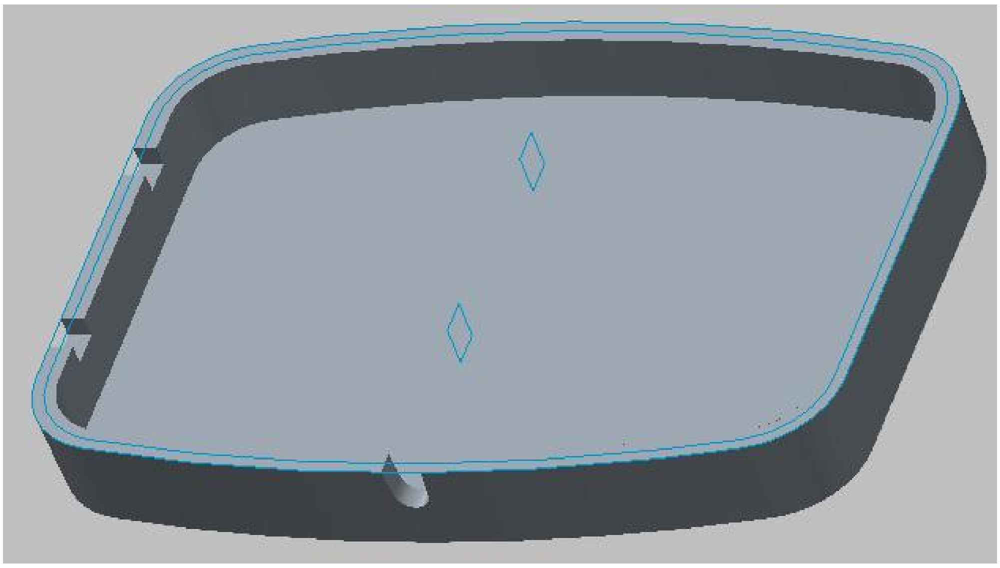


Pick icon to complete the feature. The part will appear as shown below.



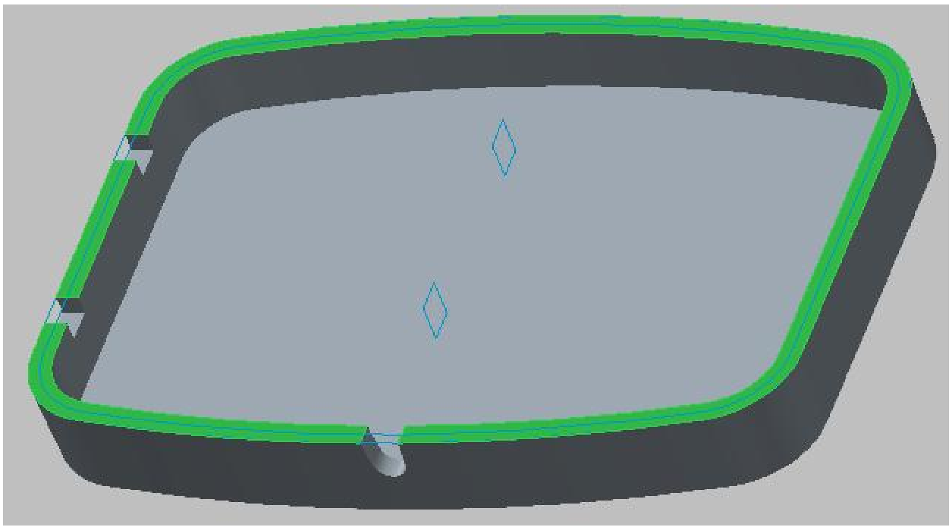
If you look at the model carefully, you will notice that outer side surfaces of the lip are drafted and inner side surfaces follow the topology of existing geometry of the part.

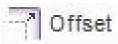
Now open the HOUSING_BOTTOM.PRT in separate window and regenerate the part by picking icon. Notice that the curve that we just added in the publish geometry appears in this part as shown below.



Now we will define an Offset feature to create the interlock lip.

So first select the following surface of the part.



Pick 

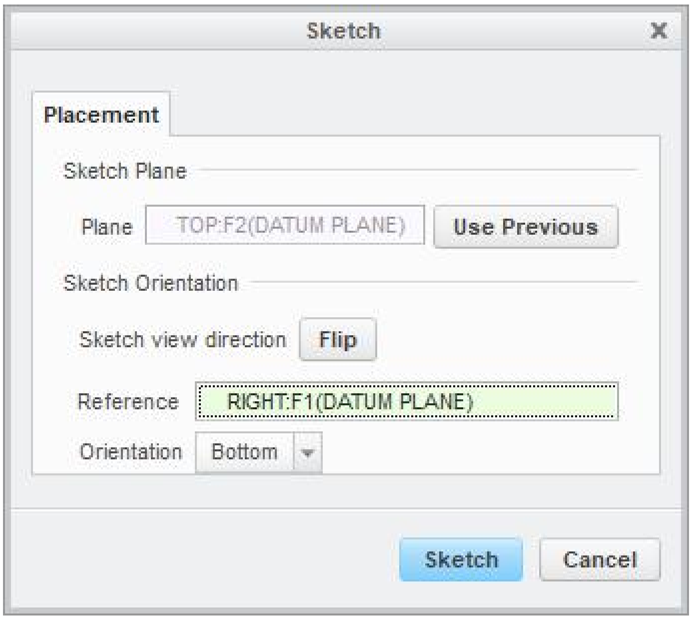
icon on the toolbar to access the Offset tool.

Change the Offset type to **With Draft** by picking the  icon in the Offset Type list.

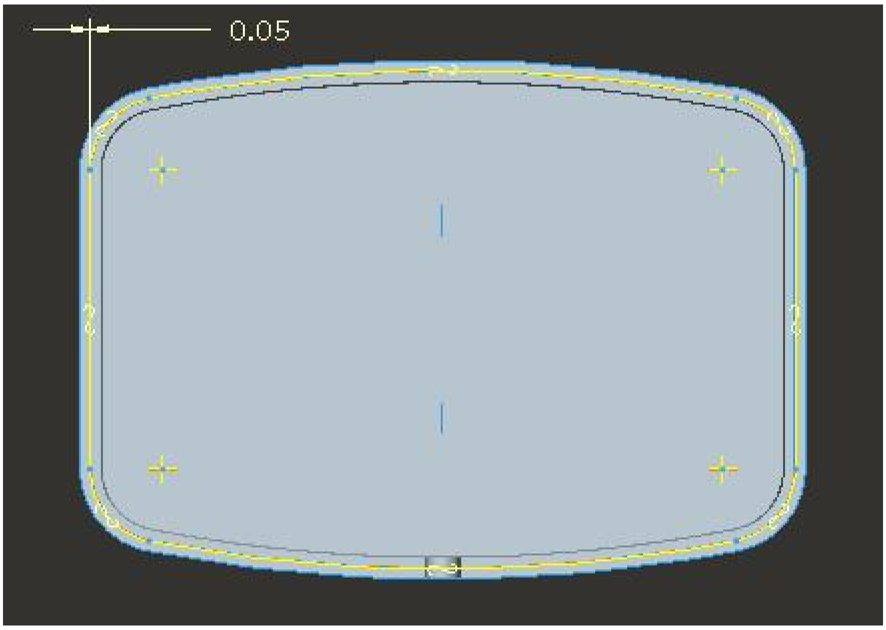
Pick 

tab in the References slide-up panel.

Select the sketching references as shown below.



Pick Offset icon and create a loop that is offset by ***0.05*** outward from the newly added curve as shown below.



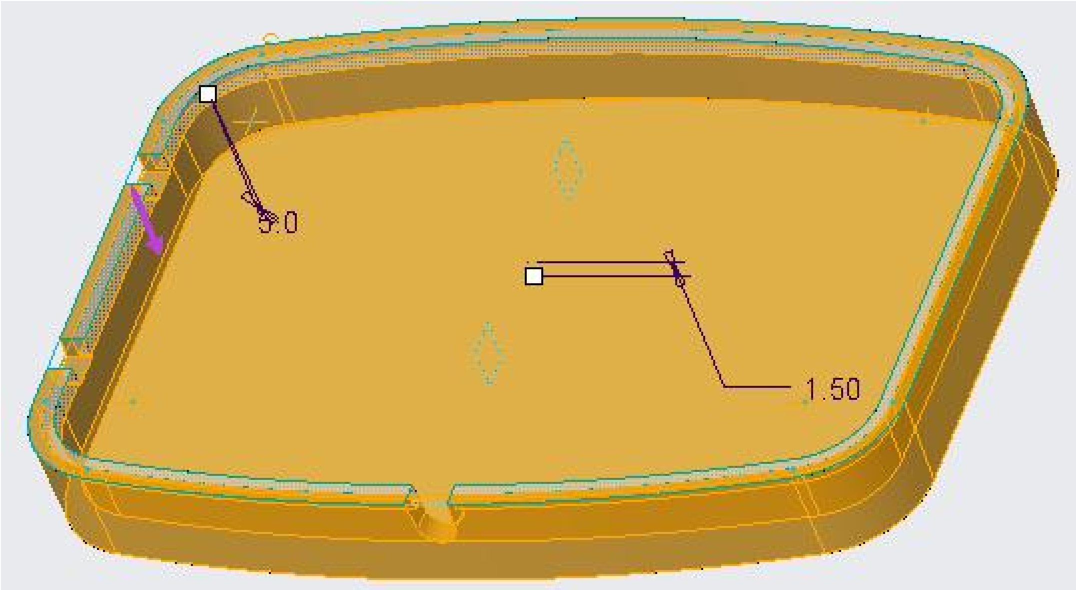
The purpose to this offset is to have some clearance between lip geometry of both parts.

After completing the sketch enter ***1.5*** as the offset value and hit Enter key.

Enter ***5*** as the draft angle and hit Enter key.

Pick to flip the direction of feature creation.

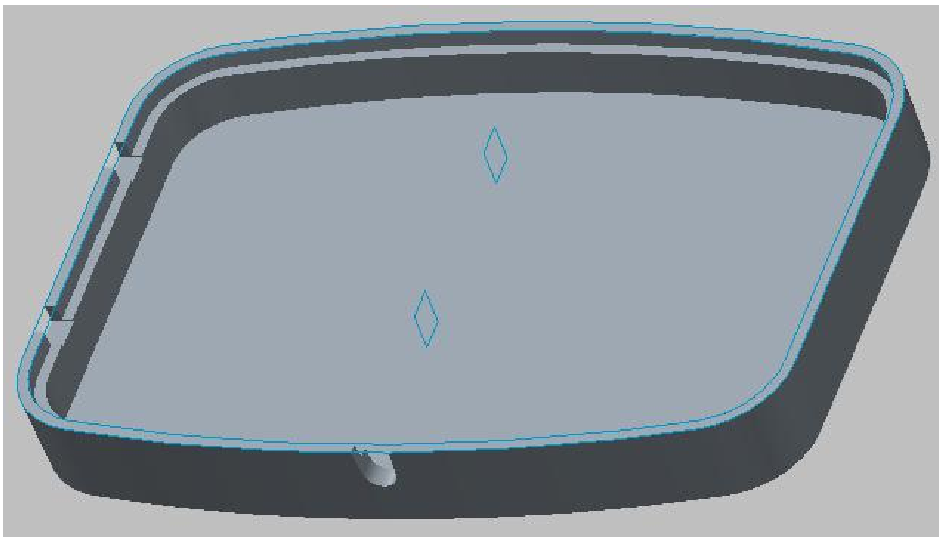
The preview will appear as shown below.



Make sure that Offset Direction arrow points in the direction shown. It will make sure that material is removed from the part.

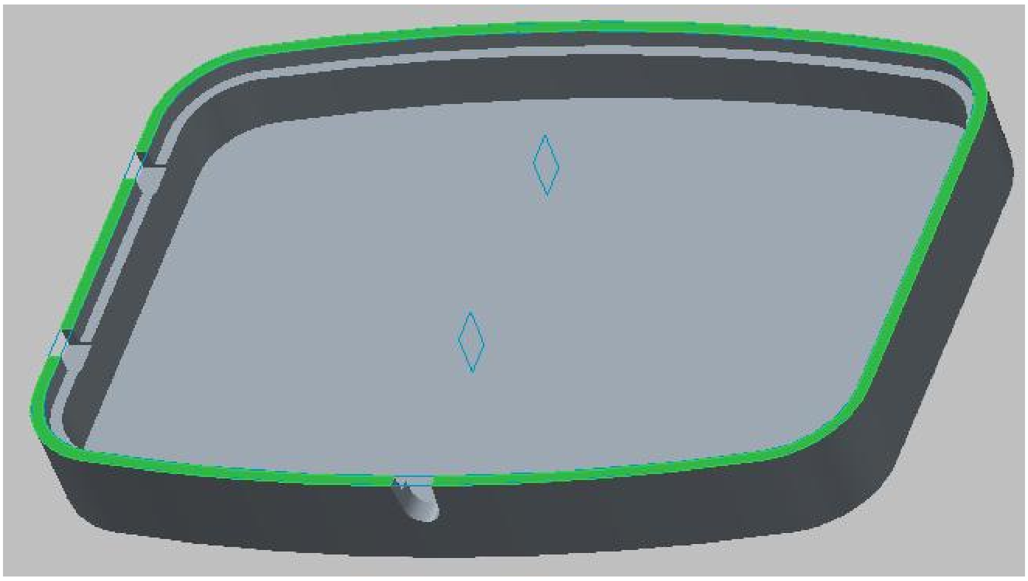
Pick Options tab and change the “Side surface normal to” option to **Sketch**.

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



Now we will create an offset feature that will create a gap between top and bottom halves to disguise any mismatch between both parts.

So select the following surface of the part.

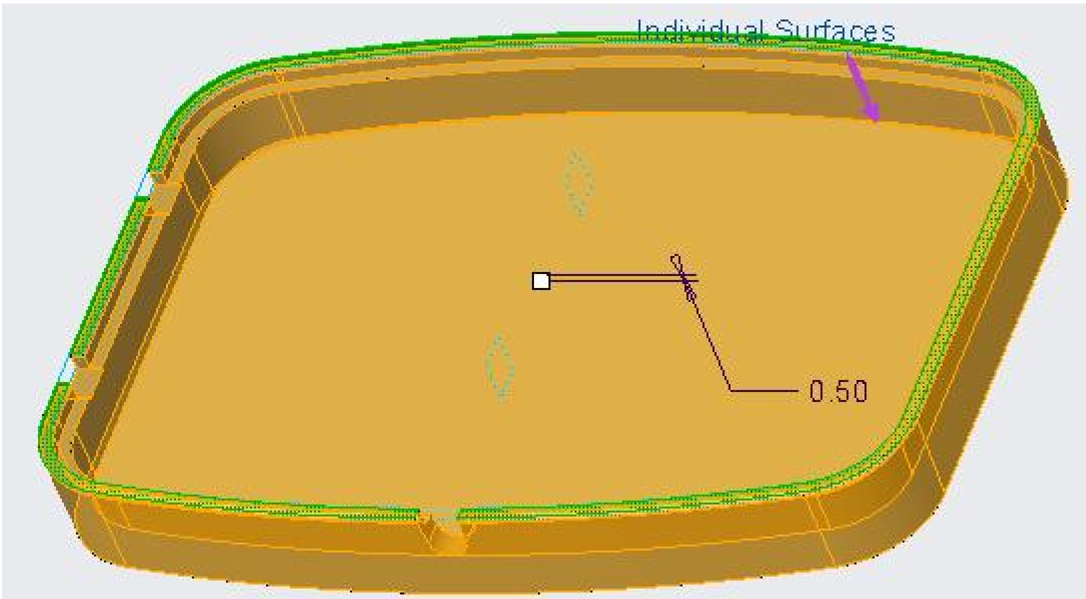


Pick Offset to access the Offset tool.

Change the Offset type to **Expand** by picking the icon in the Offset Type list.

Enter ***0.5*** as the offset value

Pick the icon to reverse the direction of offset and the preview will appear as shown below.

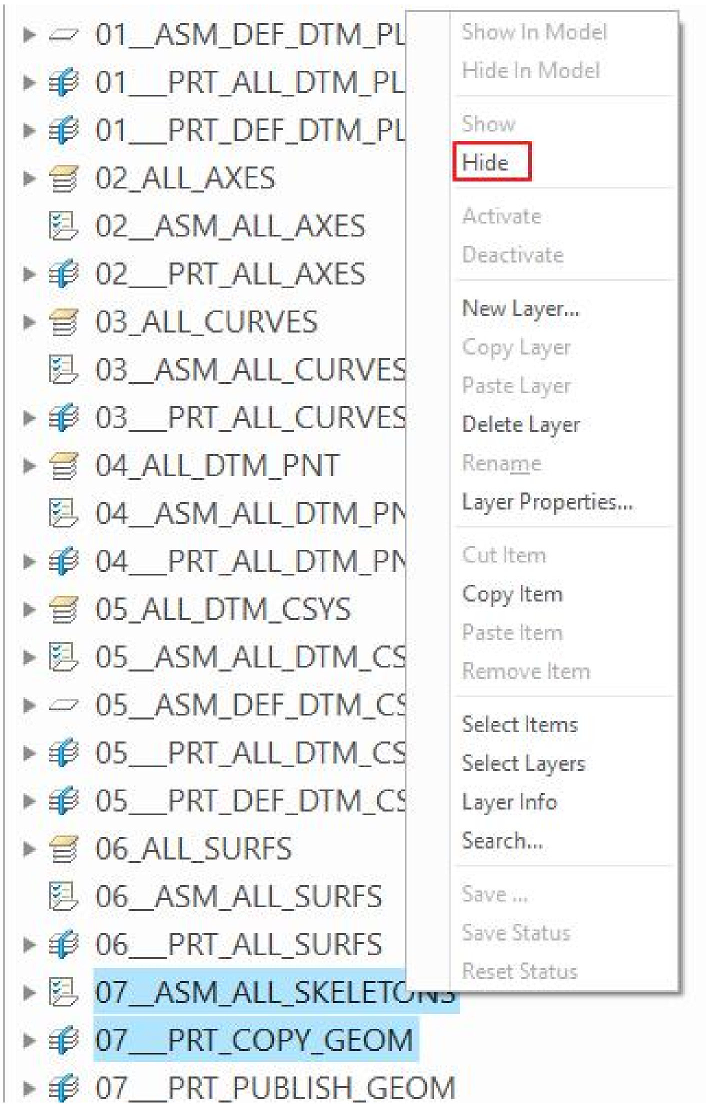


Make sure that Offset Direction arrow points in the direction shown. It will make sure that material is removed from the part.

Pick icon to complete the feature.

Switch to assembly window and pick icon on the View tab.

Hide the 07___PRT_COPY_GEOM and 07___ASM_ALL_SKELETONS layers.



If you analyze the assembly using cross-sections, you will notice that both parts confirm to the design intent. Following figure highlights this fact.