INSTITUT NATIONAL DES SCIENCES APPLIQUÉES STRASBOURG

EC Construction_3
CAD

GM4-MIQ4-PL4
2022/2023

Hands-on Training No.1

Assemblies Design and Analysis

- **Part 1 : Top-Down Design**
- **Part 2 : Double Cylinder Machine**

We want to acknowledge PTC for making available to us some training resources.

Contents

Part 1
Top-Down Design

I. Introduction

1. Top-Down Design versus Traditional Design Approach

Top-Down Design refers to the method of placing critical information in a high-level location and then communicating that information to the lower levels of the product structure. As the design develops, more specific information becomes available and is incorporated into the design. By capturing the overall design information in one centralized location, it becomes easier to make significant design changes. Because the information is contained in one place (with all subcomponents looking to that location) if you change this information, the system updates all other components.

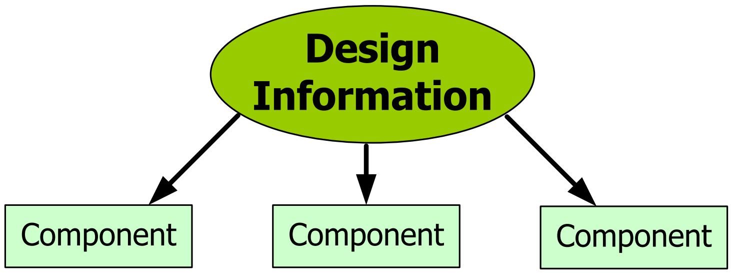


Figure 1 – Top-Down Design

Traditional Design Approach uses an approach to design individual components independently of the assembly and a manual approach to ensure that components fit properly and meet the design criteria. Components are placed in subassemblies and those subassemblies are brought together to develop the top-level assembly.

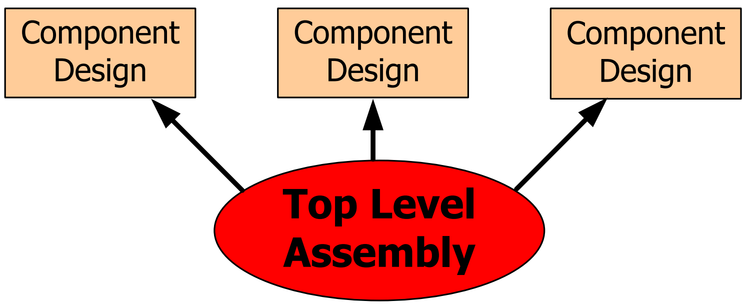


Figure 2 – Traditional Design Approach

Often, after creating the assemblies, models are discovered that do not meet the design criteria (e.g. critical interface on two models do not match). This is manually identified and modifications to each component are made to make the adjustment. As the assembly grows, detecting these inconsistencies and correcting them can consume a considerable amount of time.

2. Creo Top-Down Design Tools and Steps

The following Creo **tools** enable you to successfully capture design intent using the top-down design approach:

- Layouts;
- **Skeletons**;
- Packaging;
- **Publish Geometry**;
- **Copy Geometry**;
- Relations.

To successfully use the top-down design approach in Creo, you would generally follow these **steps**:

- Define the global product information in a **layout**;
- Generate an initial assembly structure;
- Create **skeletons** to define the product information in a 3D form;
- Document design information using **Publish Geometry** features;
- Communicate design information to the individual components using **Copy Geometry** features;
- Create model geometry using the central design information;
- Populate assemblies with solid models;
- Establish interchangeability to account for design variations.

Note - This is an iterative process in which the design becomes more detailed and specific throughout the project. You should, therefore, expect to perform the sequence of steps listed above more than once in order to complete the project.

3. Understanding Skeletons

Creating an assembly structure and using skeletons enables you to plan and design stages of assemblies.

a) Understanding Skeletons Theory

Skeletons are powerful tools. You can use them to create the 3D layout or framework of an assembly design. They can serve as a common reference source for geometry and assembling components. Any changes to the skeleton will automatically update those components that utilize shared geometry, and components which are assembled to it will also update.

Skeletons are commonly used to:

- Create space claims for components;
- Create interfaces and placement references between components;
- Provide motion between components – either by modifying skeleton dimensions, or through the use of mechanism connections.

b) Properties of a Skeleton

Skeletons are components that are given special properties. When created, a skeleton becomes the first component of the assembly automatically, so that each subsequent component can reference it. Skeletons do not affect the mass properties of the assembly, regardless of the geometry created in them. Skeleton models do not show up in the Bill Of Material (unless you specifically include them). In addition, skeletons are easily recognized in the assembly due to the blue color assigned by default to help distinguish them from other components.

c) Contents of a Skeleton

Skeletons typically contain a variety of datum features such as datum planes, axes, points, and coordinate systems. Skeletons also commonly contain sketches and surfaces to represent geometry that will be created in components. Although not as commonly used, a skeleton can contain solid geometry, however keep in mind that you can only share the surfaces of the solids.

d) Benefits of a Skeleton

An example of the advantage of using a skeleton when assembling components can be seen in the figure with the following scenario:

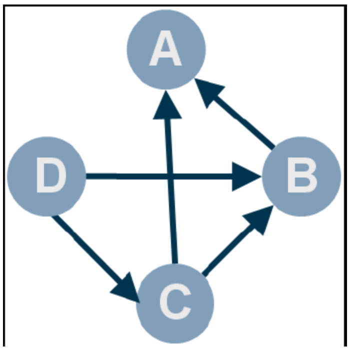


Figure 3a – No Skeleton

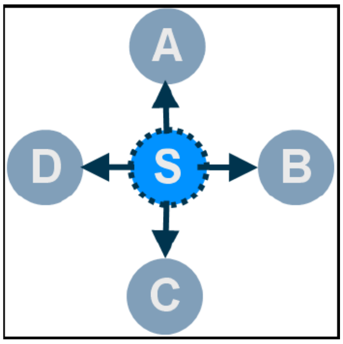


Figure 3b – With Skeleton.

In this non-skeleton design scenario:

- Component A is assembled first;
- Component B is then assembled to A;
- Component C is then assembled by referencing component A and B;
- Component D is then assembled by referencing component B and C. No Skeleton

This scenario can cause the following problems:

- Components A, B, and C cannot be deleted since all others ultimately reference it;
- If major changes are needed to multiple components (for example: overall length change), they will each have to be modified separately, possibly causing a regeneration failure of the next downstream component.

In this skeleton-based design scenario:

- Components A, B, C, and D are each assembled to the skeleton and not to each other.

- Components A, B, C, and D each contain shared geometry reference from the skeleton.

This skeleton-based design scenario has the following advantages:

- Components A, B, C, and D can each be deleted separately since they are assembled to the skeleton independently;
- If major changes are needed to multiple components (for example: overall length change), the skeleton can be modified, propagating changes to each one of the components that share skeleton geometry.

II. Partial Design of a 1-Cylinder Reciprocating Engine

In this example you learn how to create an assembly structure in the model tree to plan the assembly. You also learn how to use skeletons for space claims, interfaces, component placement and motion. Finally, you will learn how to create components from shared skeleton references.

Figure 4 – Cylinder Reciprocating Engine

1. Creating and Using Assembly Structure

Task 1 – Create an assembly structure in the empty assembly

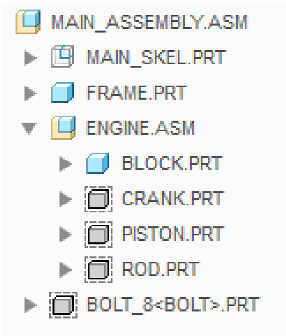
- Create a work directory on the local machine **C:\temp\Construction3\TP1** or **Z:\xxxyyy01\Construction3\TP1** and copy the necessary files to your working directory.
- Start **Creo** in **english** language, set the work directory to **~\Part1\Engine** and open the assembly file called **Main_Assembly.asm**.
- Click **Create Component** . Select **Skeleton Model** as the type, edit the name to **Main_skel** and click **OK**. Use the [en_mmNs] part solid template and click **OK** from the Creation Options dialog box.
- Click **Create Component** . Select **Part** as the type, edit the name to **Frame** and click **OK**. Ensure that the **Leave Component Unplaced** check-box is cleared and that the [en_mmNs] part solid

template is selected then click **OK**. To place the component in the graphics area, right-click and select **Default Constraint**. Click **Complete Component** .

- Click **Create Component** . Select **Subassembly** as the type, edit the name to Engine and click **OK**. Click **Browse** from the Creation Options dialog box. Select mm_kg_sec_assy.asm located in your working directory from the Choose Template dialog box and click **Open**. Click **OK** from the Creation Options dialog box. To place the component in the graphics area, right-click and select **Default Constraint**. Click **Complete Component** .
- In the model tree, **activate** Engine.asm. Click **Create Component** . Select **Part** as the type, edit the name to Block and click **OK**. Ensure that the [en_mmNs] part solid template is selected and click **OK** from the Creation Options dialog box. Place the component in the graphics area with **Default Constraint**. Click **Complete Component** .
- In the model tree, click **Settings** > **Tree Filters**, clear the **Features** and **Placement folder** check-boxes, and click **OK**. Click **Create Component** . Edit the name to Crank and click **OK**. Select the **Leave Component Unplaced** check-box and ensure that the [en_mmNs] part solid template is selected then click **OK**.

Note - Unplaced components are identified with the special gray icon .

- Click **Create Component** . Edit the name to Piston and click **OK**. Click **OK** from the Creation Options dialog box.
- Click **Create Component** . Edit the name to Rod and click **OK**. Click **OK** from the Creation Options dialog box.
- In the model tree, select Main_Assembly.asm, right-click, and select **Activate** (or Ctrl A). From the main toolbar, click **Assemble** > **Include**. Select Bolt.prt and click **Open**. Select Bolt_8 from the list and click **Open**.



SUMMARY

Creating Assembly Structure Theory

In the design process, you can create the structure of an assembly first, without defining any component geometry or specific placement constraints to locate components. Creating an assembly structure helps you to plan your project by enabling delegation of tasks to team members. You can also assign non-geometric information up-front in the form of parameters such as part number, cost, supplier, and material.

Component Types

New components for assembly structures are typically created from model templates. You can create model templates for standard (and sheetmetal) parts, as well as assemblies. You should also use these templates when creating skeletons. In summary you can create the following types of components in an assembly structure:

- **Skeletons** – You can create one or more skeletons per assembly. The first skeleton is assembled by the system using a default constraint. Subsequent skeletons are assembled like typical components – you can assemble them by default or by using constraints/connections. You can have multiple skeletons at the top level, or a top level skeleton and then skeletons at the subassembly level.
- **Parts** – You can create standard (or sheetmetal) parts. You can assemble them by default if suitable geometry does not exist, or you can assemble them using constraints or connections to the skeleton or other components.
- **Sub Assemblies** – You can create subassemblies as desired to organize the assembly.
- **Unplaced Components** – You can create a new component or subassembly and then select the Unplaced option. This leaves the new component listed in the model tree, but it is physically unplaced and not visible in the model. You can redefine the component to locate it with constraints or connections at any time. Unplaced components are shown in gray with a dashed rectangle in the model tree.
- **Included Components** – You can include existing components or subassemblies in your assembly structure. This is useful for standard components or hardware that has been previously modeled. You can redefine the component to locate it with constraints or connections at any time. Included components are shown in gray with a dashed rectangle in the model tree.
- **Packaged Components** – For new or existing parts or assemblies for which you know an approximate location, you can assemble them, drag to a location, and then complete the component with zero or partial constraints. This is particularly useful for situations when a component will be mated to a plane for example, but the exact location is not known yet. You can redefine the component to locate it with additional constraints or connections at any time. Packaged components appear with a small hollow rectangle next to the component icon in the model tree. Children of packaged components appear with a double rectangle icon.
- **Bulk Items** – You can also add Bulk Items to an assembly structure to represent components that do not require design geometry. Typical bulk items include lubricants, adhesives, paint, and so on. Bulk items are components that appear in the model tree and the bill of materials, but do not contain any modeled geometry. However, you can add parameters such as a description, type, or quantity.

2. Creating Skeletons for Space Claims

Note - In the examples below, a skeleton model contains surfaces used to claim space for the interior engine components, such as the piston, connecting rod, and crankshaft. An offset datum plane is used to help claim space for the overall engine block height.

Task 2 – Add surface geometry to the skeleton model

- In the model tree, select **Main_skel.prt**, right-click, and select **Open**. Start the datum **Plane** tool. Select datum plane **TOP** and drag the drag handle to an offset of **54**. Select the **Properties** tab, edit the datum's name to **Height**, and click **OK**.

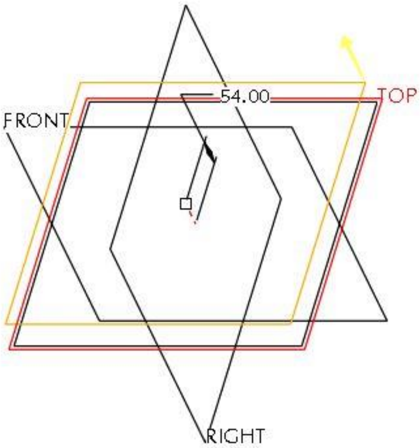
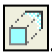
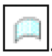


Figure 5 – "Height" datum

- With the datum plane **Height** still selected, start the **Extrude** tool . Sketch a **21** diameter circle centered on the sketch references. **Close** the Sketch menu. In the dashboard, click **Surface** . In the dashboard, edit the depth to **To Selected** and select datum plane **TOP**. Click **Complete Feature** . Click **CTRL+D** to reorient.

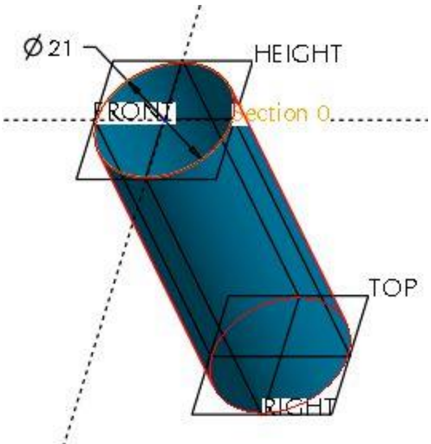
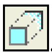
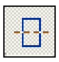
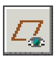


Figure 6 – The First Extruded Surface

Note - Notice the white breaks color applied to skeleton geometry.

- In the model tree, select datum plane **FRONT** and start the **Extrude** tool . Sketch a **28** diameter circle centered on the sketch references. **Close** the Sketch menu. In the dashboard, click **Surface**  Edit the depth to Both Sides  and type **32**. Click **Complete Feature** . Click **CTRL+D** to reorient. Click **Plane Display**  to disable display of datum planes.

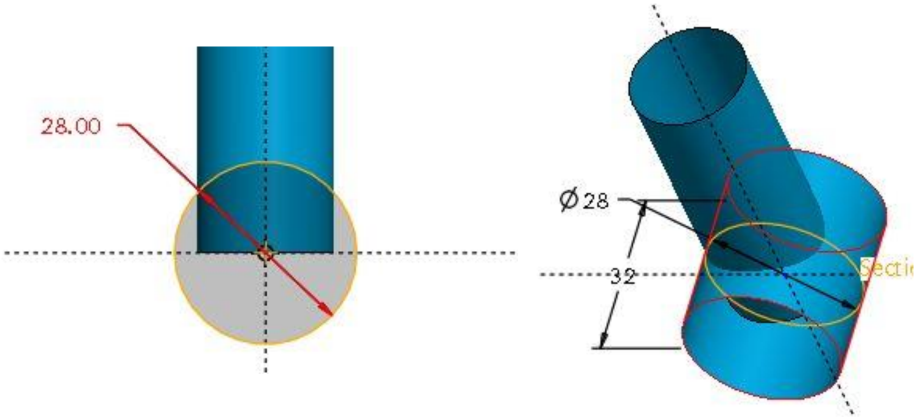


Figure 7 – Extruded Surfaces

Task 3 – Merge the surface geometry

- Click in the graphics area to deselect all geometry and features. In the model tree, press **CTRL** and select **Extrude 1** and **Extrude 2**. Click **Editing** > **Merge**. In the graphics area, click the yellow direction arrows so that the merge preview appears, as shown. Click **Complete Feature** .
- **Save** the skeleton part.

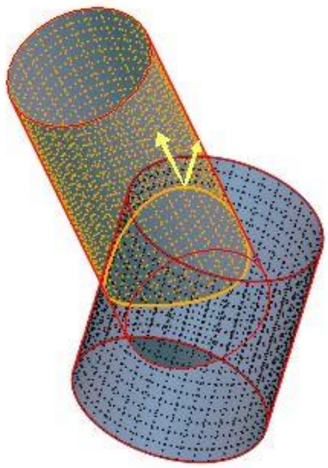


Figure 8 – Merged Surfaces

SUMMARY

Creating Skeletons for Space Claims Theory

You can use skeletons to allocate or 'claim' 3-D space before modeling or assembling the components. This can provide a common reference for geometry to be shared to multiple components that will reside at a particular location. You can then modify the space claim features in a skeleton, propagating change to one or multiple components.

Contents of a Space Claim Skeleton

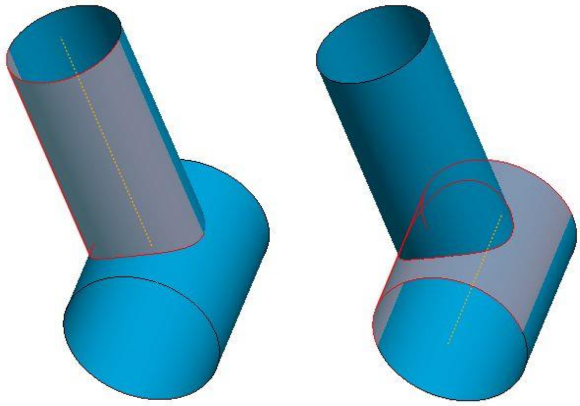
- **Surfaces** – Typically, surfaces and datum planes are used for space claim references in a skeleton. You can create open or enclosed surfaces to define the occupied volume for components or subassemblies. The surface can be as simple or as complex as required to convey the design intent. You can also use the external shrinkwrap functionality to quickly create an engulfing surface over an existing part or subassembly to represent a component that is to be placed in the assembly.

- **Datum Features** – A series of offset datum planes can also be useful to claim space for components. Offset planes are easily adjustable in size, and you can use them alone or in conjunction with surfaces in the skeleton.

3. Creating Skeletons for Placement References

Task 4 – Create axes for use as assembly placement references

- Ensure that the Main_skel.prt skeleton is still opened.
- Start the datum **Axis** tool . Select a surface of **Extrude 1**, as shown. In the Datum Axis dialog box, select the **Properties** tab. Edit the name to **Cyl** and click **OK**.
- Start the datum **Axis** tool . Select a surface of **Extrude 2**, as shown. In the Datum Axis dialog box, select the **Properties** tab. Edit the name to **Crank** and click **OK**.



Cyl axis　　Crank axis

Figure 9 – Datum Axis

Task 5 – Create datum planes for use as assembly placement references

- Click **Plane Display** to enable display of datum planes. Select datum plane **RIGHT** and then start the datum **Plane** tool . Drag the drag handle to an offset of **20**, as shown. Select the **Display** tab, click the **Adjust Outline**, and select **Reference** from the drop-down list. Select the upper cylindrical surface of **Extrude 2** as the reference. Select the **Properties** tab. Edit the name to **Offset** and click **OK**.

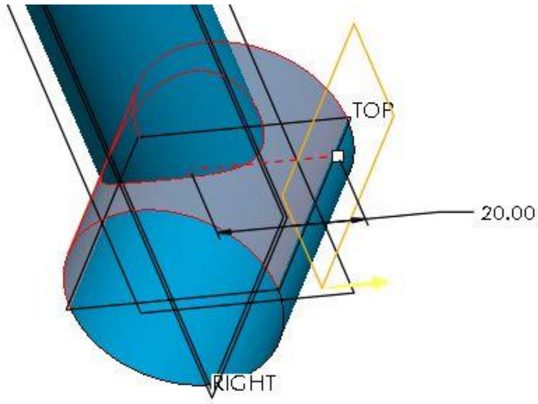


Figure 10 – Datum plane "Offset"

Task 6 – Create axes, referencing the datum plane OFFSET

- Select datum plane **Offset** and then start the datum **Axis** tool . Move the Datum Axis dialog box as necessary and start the datum **Plane** tool . Select datum plane **FRONT** and drag the drag handle to a forward offset of **10**, as shown Figure 11. Click **OK** to close the Datum Plane dialog box. Select the **Display** tab, click the **Adjust Outline** button, and select **Reference** from the drop-down list. Select the upper cylindrical surface of **Extrude 2** as the reference. Select the **Properties** tab. Edit the name to **Hole1** and click **OK**.

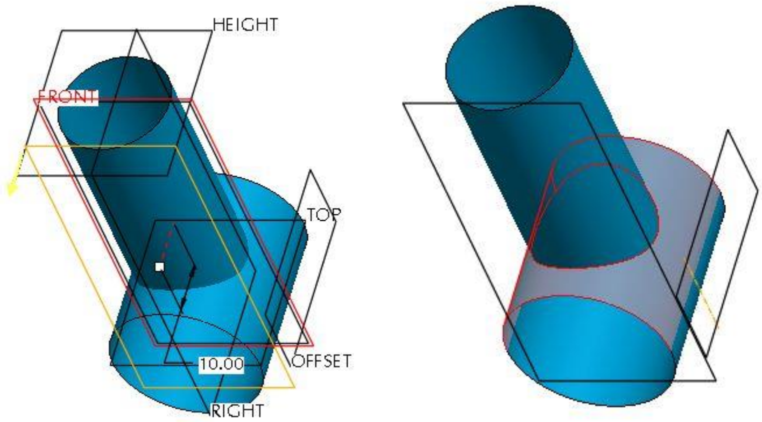


Figure 11 – Datum Axis "Hole1"

- Create a new datum axis named **Hole2** on the other side of the datum plane FRONT as shown Figure 12.

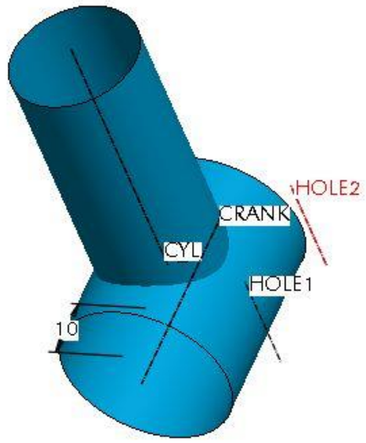


Figure 12 – Datum Axis "Hole2"

Task 7 – Create sketch, referencing the datum plane OFFSET

- Start the Sketch Tool and create a sketch on the datum plane **TOP** as shown Figure 13.

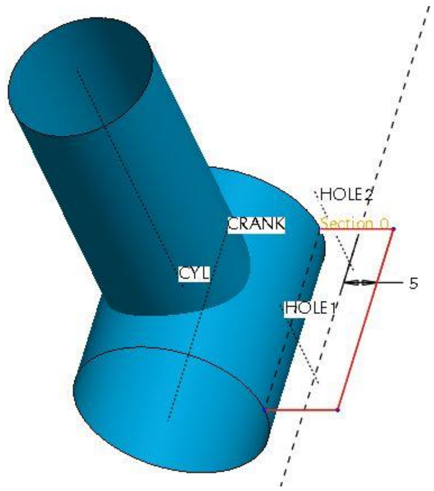


Figure 13 – Datum Sketch

- **Save** your skeleton part and **Close** the window.

SUMMARY

Creating Skeletons for Interfaces and Placement Theory

You can use skeletons to define the interfaces between adjacent or mating components, and also to provide common references for assembling components.

Skeleton features created to represent interfaces can provide the shape and/or location of the interfaces between components. You can copy the interfaces to multiple components to provide a common reference for creating solid geometry. You can modify the interface features in the skeleton, propagating changes to one or multiple components.

You can also use skeletons to define component placement. You can therefore modify the placement of single or multiple components with changes to the skeleton. Component placement examples include:

- An axis in the skeleton to provide a partial reference for assembling a component;
- A coordinate system to provide a complete reference for assembling a component;
- A series of sketched lines representing an adjustable mechanical linkage. Once the components are assembled to the skeleton, you can change lengths and/or angles of the skeleton sketches to update the model locations and/or size.

Typically, sketches, surfaces, and various datum features are used for interface and placement references in a skeleton. These features can be as simple or as complex as required to convey the design intent.

4. Creating Multiple Skeletons

Note - The example below illustrates multiple independent skeletons in use. You use the **Main** skeleton to share references to the **Frame** and also the **Block** in the **Engine.asm**. The **Crank**, **Piston**, and **Rod** skeletons are additional skeletons assembled to the **Main** skeleton using connections to provide mechanism movement.

Task 8 – Create and assemble the crank skeleton

- Press **CTRL** and in the model tree, select **Frame.prt** and **Engine.asm**. Right-click, and select **Hide**. Select **Main_skel.prt** and click **View** > Manage **Views** > **View Manager** > **Style** > **Properties** > **Transparent**.

- Click **Create Component** . Select **Skeleton Model** as the type, edit the name to **Crank_skel** and click **OK**. Click **Browse**, select **Crank_geom.prt** from the Choose Template dialog box, and click **Open**. Ensure that the **Leave Component Unplaced** check box is cleared. Click **OK** from the Creation Options dialog box. Click to place the component in the graphics area. In the dashboard, select **Placement** and **Pin** from the User Defined drop-down list. Select the **CRANK** axes in both models to define axis alignment. Select the **FRONT** datum plane in both models to define translation. Click **Complete Component** .

Task 9 – Create and assemble the piston skeleton

- Click **Create Component** . Edit the name to **Piston_skel** and click **OK**. Click **Browse**, select **Piston_geom.prt** from the Choose Template dialog box, and click **Open**. Click **OK** from the Creation Options dialog box. Click to place the component in the graphics area. In the dashboard, select **Placement** and **Slider** from the User Defined drop-down list. Select the **CYL** axes in both models to define axis alignment. Select the **FRONT** datum plane in both models to define rotation. Press CTRL+ALT and right-click to drag the skeleton into the approximate position shown. Click **Complete Component** .

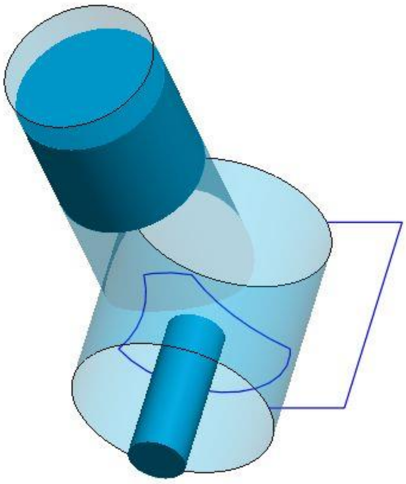


Figure 14 – Crank and Piston Skeletons

Task 10 – Create and assemble the connecting rod skeleton

- Click **Create Component** . Edit the name to **Rod_skel** and click **OK**. Click **Browse**, select **Rod_geom.prt** from the Choose Template dialog box, and click **Open**. Click **OK** from the Creation Options dialog box. Click to place the component in the graphics area. In the dashboard, select **Pin** from the User Defined drop-down list. Select the **PIN_1** axes in both models to define axis alignment. Select the **FRONT** datum plane in both models to define translation. Right-click and select **New Set**.

Note - The assembly references above should reference the **Crank_skel.prt**.

- In the dashboard, edit the connection to **Cylinder** . Select the **PIN_2** axes in both **Rod_skel.prt** and **Piston_skel.prt** to define axis alignment. Click **Complete Component** .
- Click **Axis Display** and **Plane Display** . Click **Drag Components** , select the cylindrical shaft surface of the **Crank_skel** and drag it through its motion.

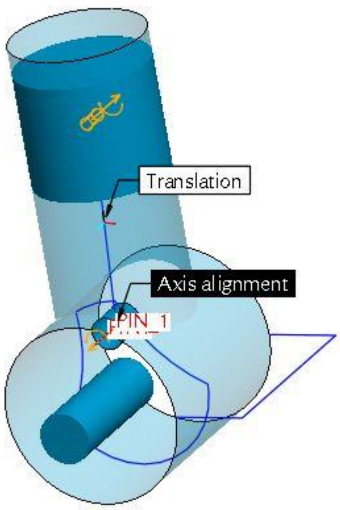


Figure 15 – Rod Skeleton

- **Save** your assembly.

SUMMARY

Creating Multiple Skeletons with Connections Theory

In the design process, you can create the structure of an assembly first, without defining any component geometry or specific placement constraints to locate components. Creating an assembly structure helps you plan your project by enabling delegation of tasks to team members. You can also assign non-geometric information up-front in the form of parameters such as part number, cost, supplier, and material.

Assemble Skeletons

The first skeleton is assembled by the system using a default constraint. You can assemble subsequent skeletons like typical components – you can assemble them by default or by using constraints or connections. You can create multiple skeletons at the top level, and also at the subassembly levels.

Techniques

There are several different techniques that you can use when working with multiple skeletons, including:

- Creating the multiple skeletons independent of each other, meaning you can modify each skeleton independently without affecting the other skeletons.
- Using multiple skeletons in a hierarchy. In this technique, you create a main skeleton, and then share references to several other skeletons. These additional skeletons can be at the top level also, or in a subassembly. Modifying the "main" skeleton would then propagate change to multiple other skeletons, which would then in turn update several components.
- When creating multiple skeletons, consider whether each skeleton will represent one or many components. A skeleton can also represent an entire subassembly.

5. Sharing Skeleton Geometry

Note - The example below illustrates a selection of surfaces and datum features being copied from the skeleton using a Copy Geometry feature. The resulting copy geometry feature is also shown along with the model tree.

Task 11 – Copy references from Main_skel.prt to the Block.prt

- **Unhide** the Engine sub-assembly.
- In the model tree, select **Block.prt**, right-click, and select **Open**.
- Click **Get Data** > **Copy Geometry**. In the dashboard, deselect **Publish Geometry Only** and then click **Open** . Select **Main_skel.prt** and click **Open**.
- In the Placement dialog box, click **Coord Sys** and select **PRT_CSYS** in both models. Click **OK** to close the Placement dialog box.
- In the dashboard, click **References** and activate the **Surface Sets** field in the References tab. Query select the entire surface quilt of **Main_skel.prt**. Activate the **Chains** field and select the curve feature shown Figure 16. Activate the **References** field. Press **CTRL** and select datum plane **OFFSET** and the axes **HOLE1** and **HOLE2**. Click **Complete Feature** .

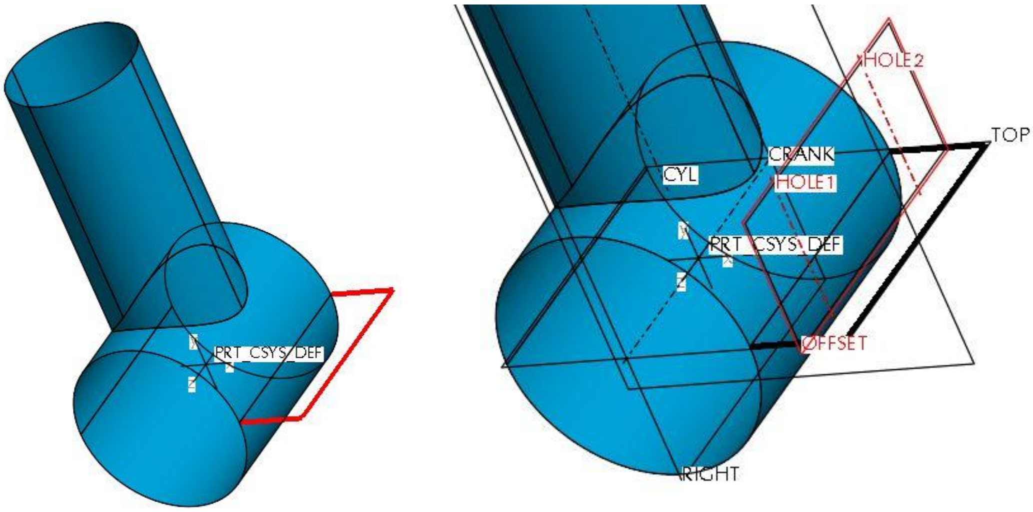


Figure 16 – Chains and references to copy

- **Save** your part and **close** the window.

Task 12 – Copy references from Crank_skel.prt to the Crank.prt

- Open the **Crank.prt** component. Click **Get Data** > **Copy Geometry**. In the dashboard, deselect **Publish Geometry Only** and then click **Open**. Select **Crank_skel.prt** and click **Open**.
- In the Placement dialog box, click **Coord Sys** and select **PRT_CSYS** in both models. Click **OK** to close the dialog box.
- In the dashboard, click **References** and activate the **Surface Sets** field in the References tab. Query select the entire surface quilt of **Crank_skel.prt**. Activate the **Chains** field and select the outer curve feature shown. Activate the **References** field. Press **CTRL** and select axes **CRANK** and **PIN_1**. Click **Complete Feature** .

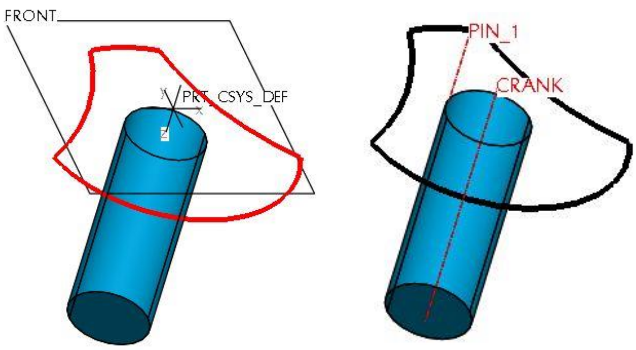


Figure 17 – Chains and references to copy

- **Save** the part and **close** the window.

Note - Geometry copied from a skeleton, into the design model is associative. If the skeleton changes, the referencing design model will update accordingly, as well as all geometry referencing the copied geometry.

SUMMARY

Sharing Skeleton Geometry Theory
Once Skeletons have been created, you can share geometry with the solid components. You can then utilize the shared geometry as references to build solid features. It is through the use of shared geometry that the skeleton can propagate changes to the features in the components. Note that this step is not required for those components that are only being assembled to the skeleton.

6. Creating and Placing Models using Skeleton References

Note - In the example above, the space claim surfaces from the skeleton are being used to create solid tube shapes for the beginning of the engine block model. Holes are also created using the copied axes. Finally, the unplaced bolts from the original assembly structure are now placed, referencing both solid geometry and the copied axes.

Task 13 – Reference copied features to create features in the block part

Note - The appearance manager enables you to manage your appearances. You access the appearances manager by clicking **View** > **Appearance Gallery** from the main toolbar and selecting **Appearance Manager**. The Appearance Manager dialog box contains both the contents of the appearance gallery and the appearance editor.

- Open the **Block.prt** part.

Note - In this procedure, like in a real design, use your best judgment regarding the display of datum features. Enable or disable their display as required.

- Select the External Copy feature, then click **Editing** > **Thicken**. Edit the thickness to **2** and press **ENTER**. Click **Complete Feature**.

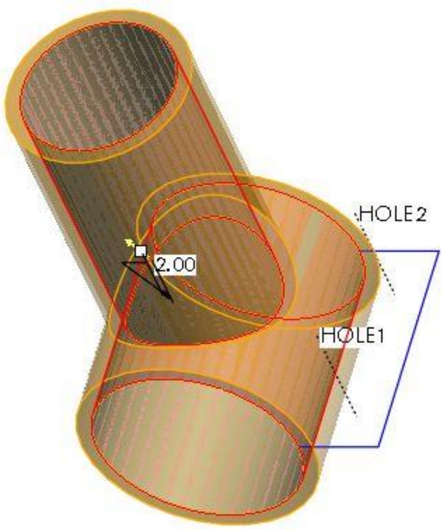


Figure 18 – First feature in the block part

- Select datum plane **TOP** and then click **Extrude tool** . In the **Sketcher** toolbar, click **Project** and select the datum curve at the right side of the model. Close the loop with a single straight line. Click **Done Section** . Press **CTRL+D**, edit the depth to **4** and press **ENTER**. Click **Complete Feature** .

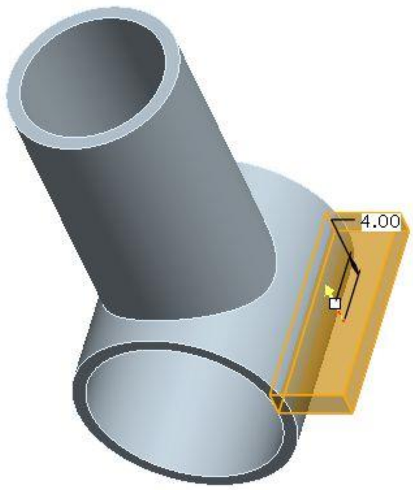


Figure 19 – Second feature in the block part

- Click the **Hole Tool** and select axis **HOLE1**. Press **CTRL** and select the hole's placement surface, as shown. Edit the diameter to **3** and depth to **Through All** . Click **Complete Feature** .
- With the hole still selected, press **CTRL+C** and then **CTRL+V**. Select axis **HOLE2**, press **CTRL** and select the hole's placement surface, as shown Figure 20. Click **Complete Feature** .

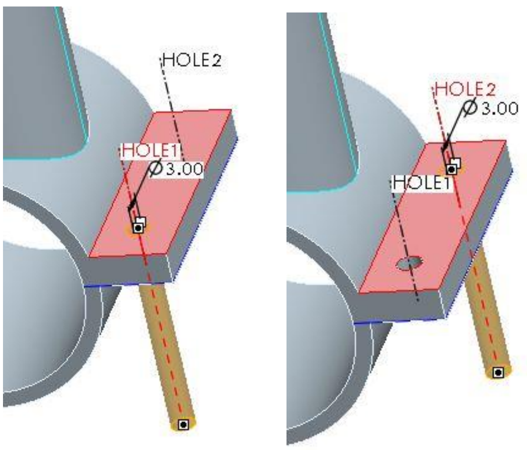


Figure 20 – Holes in the block part

- In the model tree, select the External Copy feature, right-click, and select **Hide**.
- **Save** the file and **close** the window.

Task 14 – Reference copied geometry to create features in the crank part

- Select Crank.prt, right-click, and select **Open**. Select the External Copy feature, then click **Editing** > **Solidify**. Click **Complete Feature** .

- Start the **Extrude** tool. Right-click and select **Define Internal Sketch**. Select the far end of the cylinder as the sketch plane and click **Sketch**. Remove the datum plane references, click **Close** and then **Yes** from the Missing References dialog box. In the Sketcher toolbar, click **Project**, and select the **Loop** check box in the Type dialog box. Select the datum curve and click **Done Section**. Press **CTRL+D**, drag the depth handle towards you, to a depth of **6**. Click **Complete Feature**.
- Start **Extrude** tool. Right-click and select **Define Internal Sketch**. Select **Use Previous**. Select axis **PIN_1** as a reference and click **Close**. Sketch a **5** mm diameter circle, centered on **PIN_1**, then click **Done Section**.
- Press **CTRL+D**, drag the depth handle away from you, to a depth of **6** and press **ENTER**. Click **Complete Feature**. In the model tree, select the External Copy feature, right-click, and select Hide.

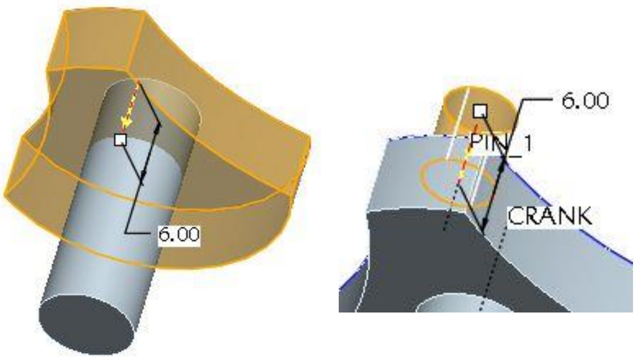


Figure 21 – Features in the crank part

- **Save** the file and **close** the window.

Task 15 – Cleanup the display and assemble bolts

- Press **CTRL** and in the model tree, select **Main_skel.prt** and **Crank_skel.prt**, right-click, and select **Hide**. Select **Engine.asm**, right-click, and select **Unhide** if necessary. Select **Block.prt** and set the model display to **transparent**.
- In the model tree, select **Bolt_8.prt**, right-click, and select **Edit Definition**. Constrain the bolt as shown Figure 22 and then click **Complete Component**.
- Disable the **Axis Display** and the **Plane Display.** Select **Bolt_8.prt**, press **CTRL+C** and **CTRL+V**. Constrain the bolt into the last hole as shown and then click **Complete Component**.
- In the model tree, expand **Engine.asm**, select **Crank.prt**, right-click, and select **Edit Definition**. In the Placement dialog box, select **PRT_CSYS** in both models (**Crank.prt** and **Crank_skel.prt**). Click **Complete Component**.
- Click **Drag Components**, select the crankshaft and drag it through its motion.
- **Save** the file.

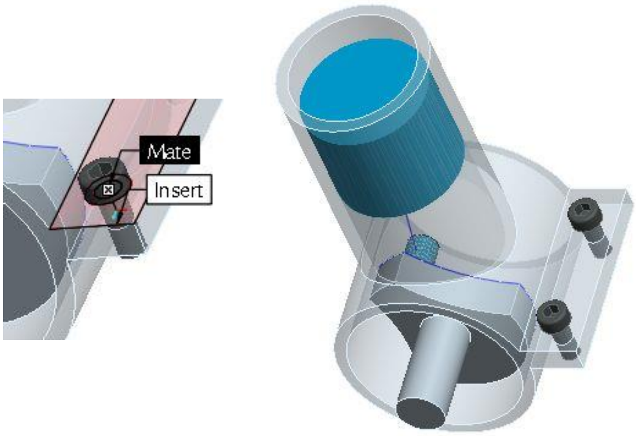


Figure 22 – Bolts

SUMMARY

Creating and Placing Models using Skeleton References Theory
Once you have used a copy geometry feature to share references, you can begin to create solid geometry utilizing the copied references. You can either open the component in its own window, or activate the component and model in the context of the assembly. You can utilize the copied surfaces, planes, axes, and so on by referencing them when creating sketches, solid features, and other datum features in the model.

Task 16 – Reference copied geometry to create features in the piston part

- Communicate the design information from the **Piston_skel.prt** component to the **Piston.prt** one and place it to the assembly.
- **Save** and **Close** all the windows.

III. Design of a Housing

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

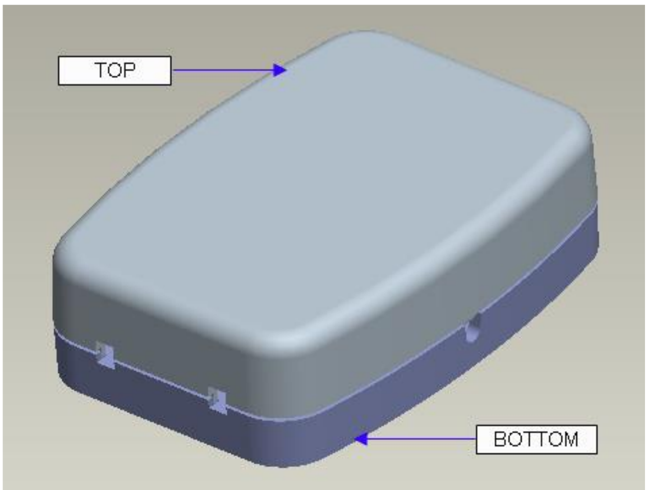


Figure 23 – Housing

The common attributes between both components are as follows:

- Profile of both components should be same;
- Two rectangular holes in the 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.

Note - 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.

- Set the working directory to **~\Part1\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.

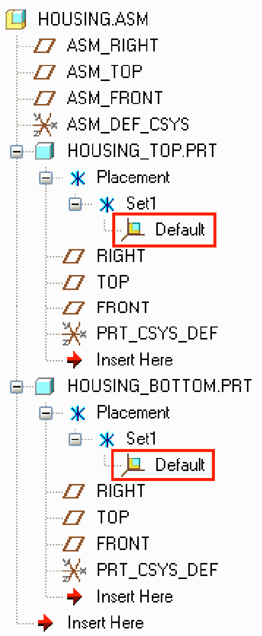


Figure 24 – Model Tree

1. Creating the Skeleton

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

Task 1 – Creating Skeleton Model

- Click **Create Component** . Select **Skeleton Model** as the type, edit the name to **Housing_skel** and click **OK**. Click **Browse** from the Creation Options dialog box. Select **Skl_template.prt** from the Choose Template dialog box and click **Open**. Click **OK** from the Creation Options dialog box.

Note - **Skl_template.prt** is the template file that contains the default datum features, layers and accuracy settings.

- **Open** the skeleton model in a separate window and notice that it only consists of default datum features.

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.

Task 2 – Creating the geometry in the skeleton model

- Pick and select the **SKL_TOP** plane as the sketch plane and click **Sketch**. Now sketch the section as shown below. For your convenience the section has been predefined. You can place this predefined section by picking the profile.sec from **Get Data** > **File System** (choose a scale of one).

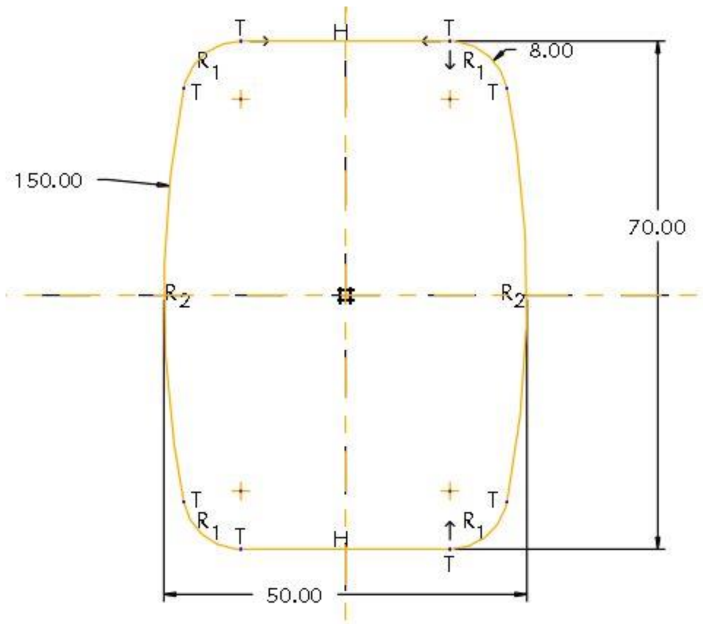


Figure 25 – Predefined section

- Make sure that centerlines are aligned to the datum planes. Pick .to complete the section. This datum curve will be used to drive the profile of both components.
- Again pick and select the **SKL_FRONT** plane as the sketch plane and click **Sketch**. Now sketch the section as shown below. Pick to complete the section.

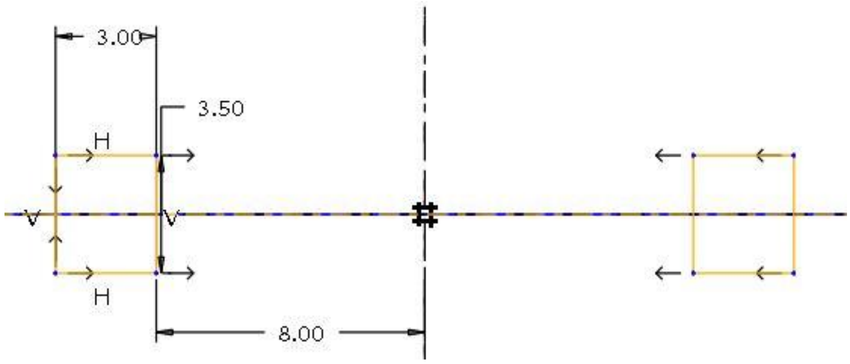


Figure 26 – New section

2. 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 the skeleton model window is active.

Task 3 – Creating the Publish Geometry in the skeleton model

- Click **Model Intent > Publish Geometry**. The Publish Geometry dialog box will appear. 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.

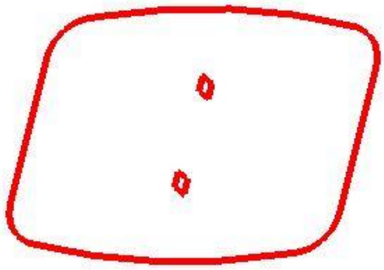


Figure 27 – Selected Chains

Note - Publish geometry feature allows to mark the references, in a source part, that can be referenced by a copy geometry feature altogether. Therefore, 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 feature 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. First, we will create a copy geometry feature in the **Housing_Top.prt**.

Task 4 – Creating the Copy Geometry in the two parts

- Make the assembly window active, right-click the **Housing_Top.prt** in the model tree and select **Activate**.
- Click **Get Data > Copy Geometry**. The Copy Geometry dashboard will appear. 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.

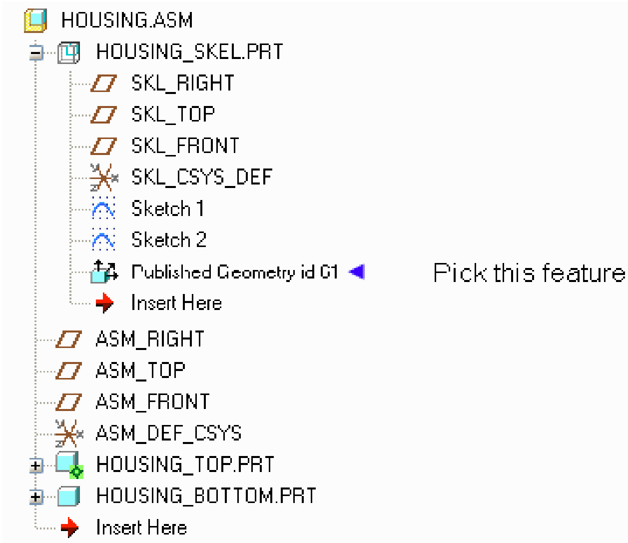


Figure 28 – Published Geometry Feature

Note - 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, by the same way, copy the publish geometry into **Housing_Bottom.prt** part.

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.

Task 5 – Creating Housing_Top Geometry.

- Open the **Housing_Top.prt** in the new window. Invoke **Extrude** tool. Select the **TOP** plane as the sketch plane and click **Sketch**. Pick **Project** icon and select the following loop.

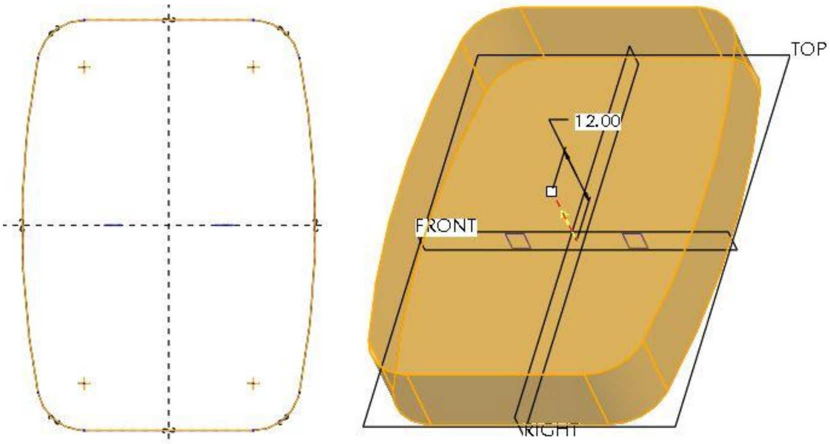


Figure 28 – Extrusion

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

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

- Select the following surface:

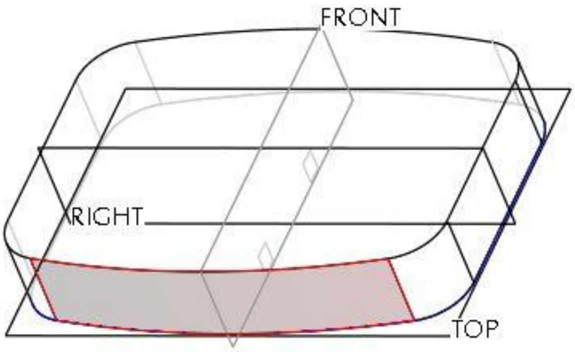


Figure 29 – Drafted Surface

- Pick  to access the **Draft** tool. Notice that Draft hinges collector is active so pick the following surface as the draft hinge.

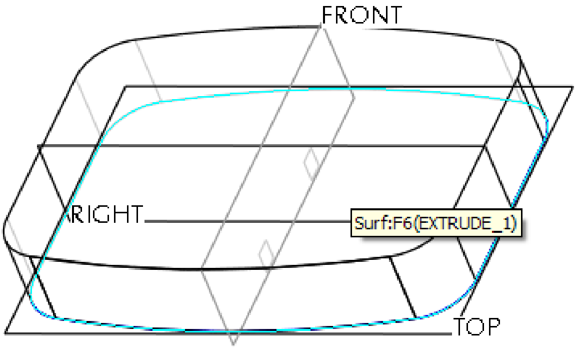


Figure 30 – Draft Hinge

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

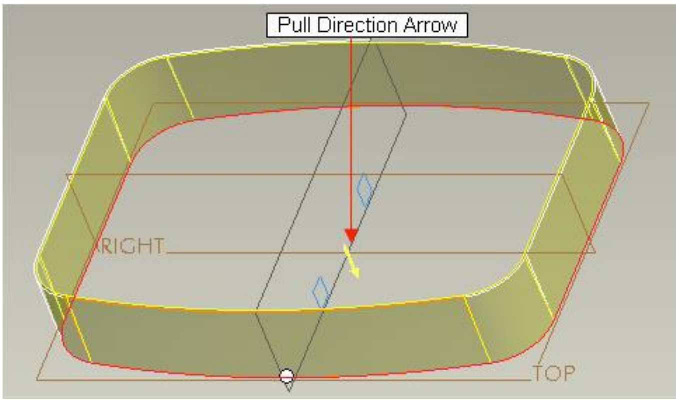


Figure 31 – Pull Direction

- Make sure that Pull Direction arrow points in the direction shown. **Complete** the feature.
- Select the following edge for applying round:

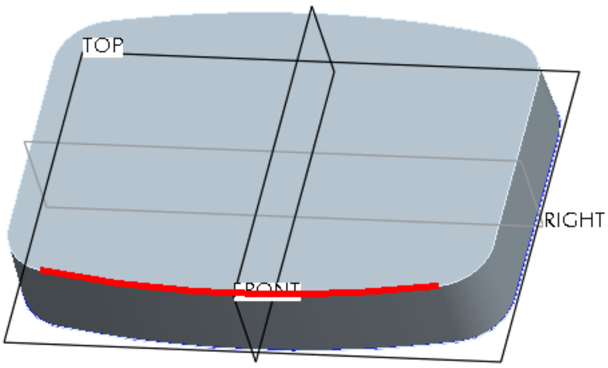
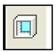


Figure 32 – Round Feature

- Pick  to access the **Round** tool. Enter **3** as the radius value. **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. **Complete** the feature.

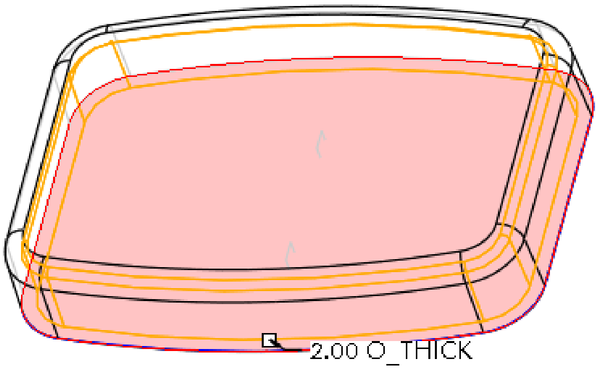


Figure 33 – Shell Feature

- To create the cuts in the side wall we will create an extrude feature. So invoke the **Extrude** feature. Select the **FRONT** plane as the sketch plane and click **Sketch**. Select the following loop:

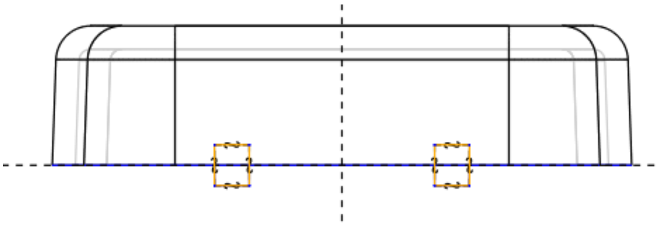
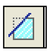
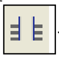


Figure 34 – Sketch

- After completing the sketch pick  icon to create the feature as cut. Specify the depth option to **Thru All** (). Notice that extrusion direction is towards the brown side of the sketching plane. **Complete** the feature and part will appear as shown below.

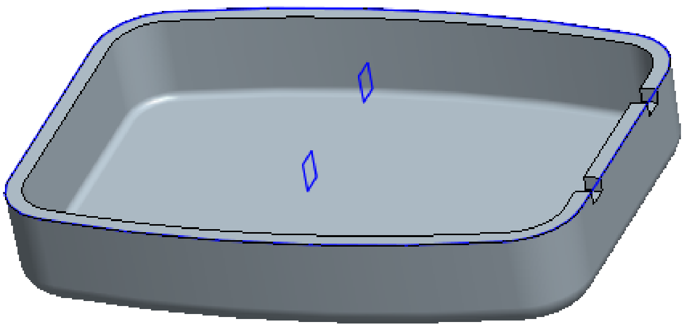


Figure 35 – Housing_Top

This completes **Housing_Top.prt** except the interlock detail which we will create later.

Task 6 – Creating Housing_Bottom Geometry

- Now we will create the geometry for the second part. So open the **Housing_Bottom.prt** in a new window.
- Pick the **Extrude** tool icon. Select the **TOP** plane as the sketch plane and click **Sketch**. Select the following loop.

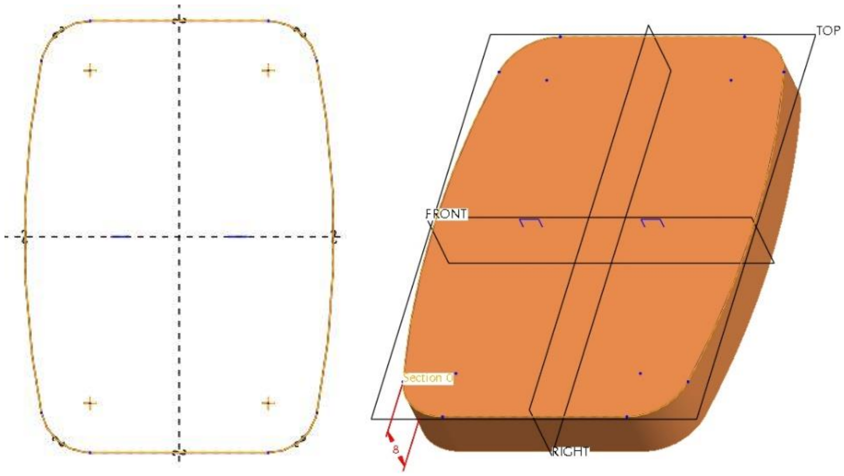


Figure 36 – Extrusion

- Make sure that extrusion directions is towards the gray side of "TOP" datum plane 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:

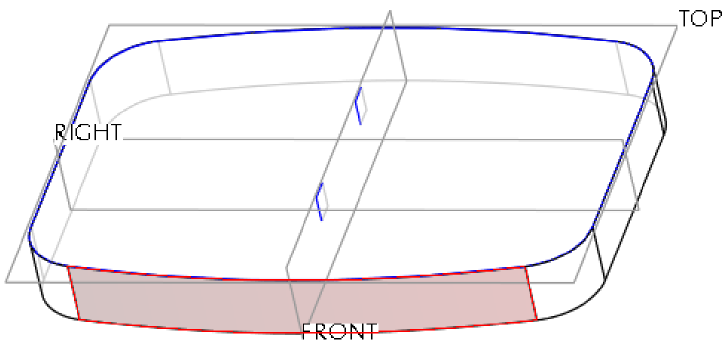


Figure 37 – Drafted Surface

- Invoke the **Draft** tool. Then pick the following surface as the draft hinge.

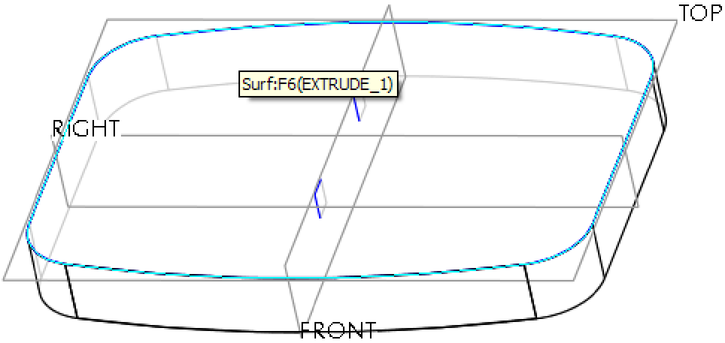


Figure 38 – Draft Hinge

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

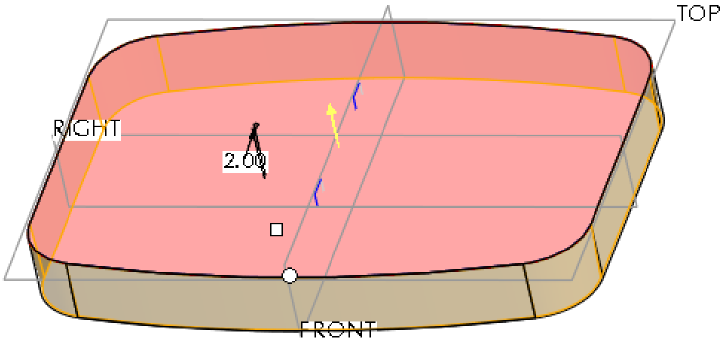


Figure 39 – Pull Direction

- Make sure that Pull Direction arrow points in the direction shown. **Complete** the feature.
- Invoke the **Shell** tool. Enter **2** as the thickness value. Select the following surface to add it to Removed surfaces collector and complete the feature.

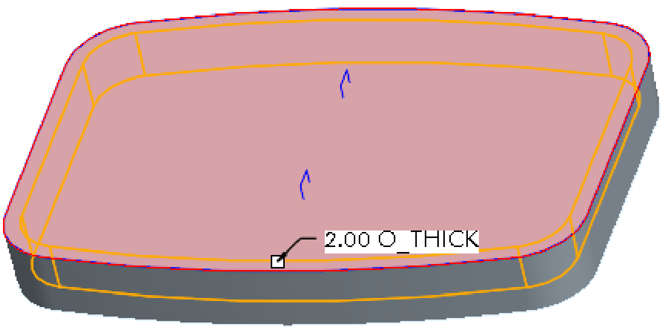


Figure 40 – Shell

- Create the cuts in the side wall, like for the **Housing_Top** part.

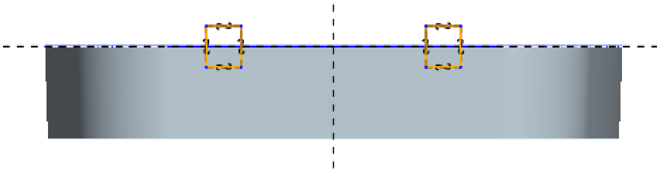


Figure 41 – Sketch

Now we will create a cut in the front wall. Be careful that the two cuts of each part are on the same side.

- So invoke the **Extrude** tool. Select the **RIGHT** plane as the sketch plane. Sketch the section as shown below.

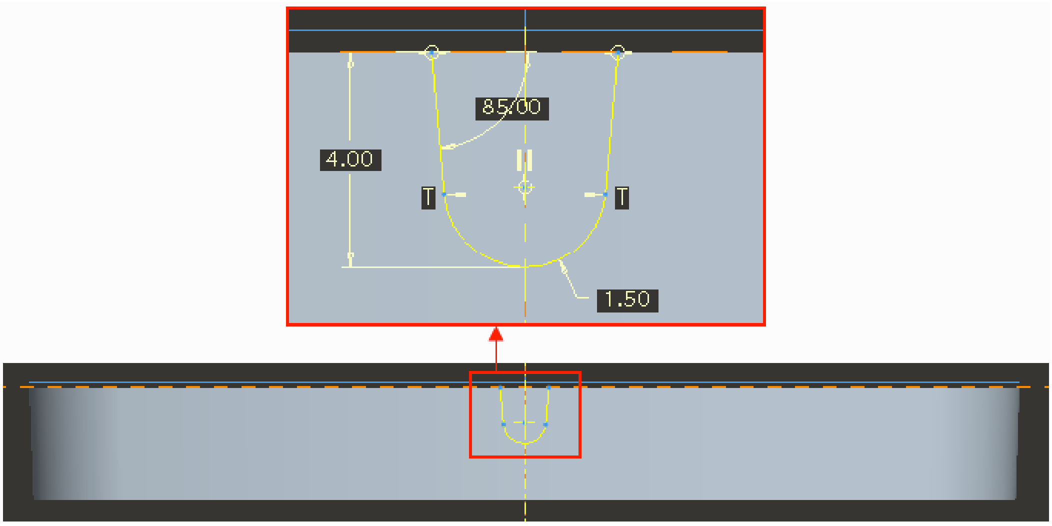
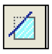
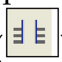


Figure 42 – New Sketch

- After completing the sketch pick  icon to create the feature as cut. Specify the depth option to **Thru All** (). 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.

This completes **Housing_Bottom.prt** except the interlock detail.

Task 7 – Interlock Details

Now we will create this 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.

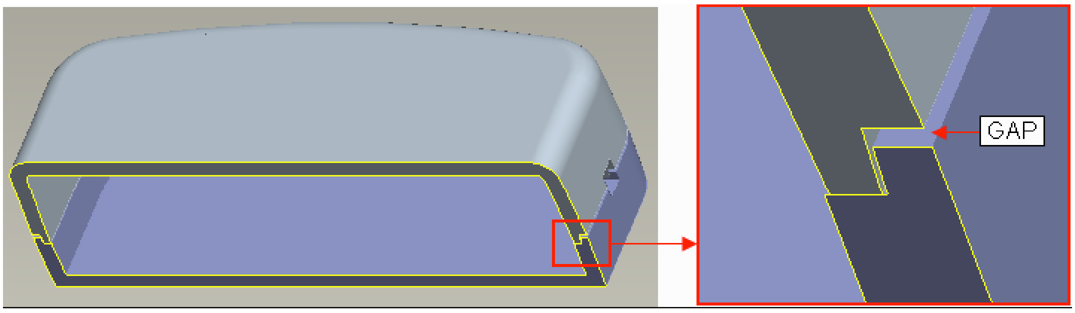


Figure 43 – Gap

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. Invoke the **Sketch** tool and select the **SKL_TOP** plane as the sketch plane. With the **Offset** tool create a loop that is offset by **-1** from the profile curve as shown below. **Complete** the feature.

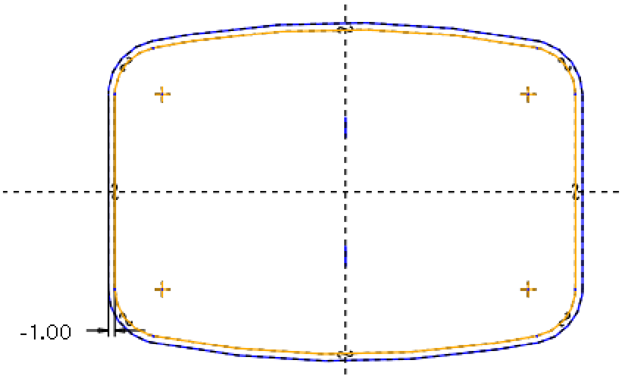


Figure 44 – Offset

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. Right click the publish geometry feature and pick **Edit Definition**. Activate the **Chains** collector and pick the following chain (pre-highlighted in green color) while holding down the **CTRL** key.

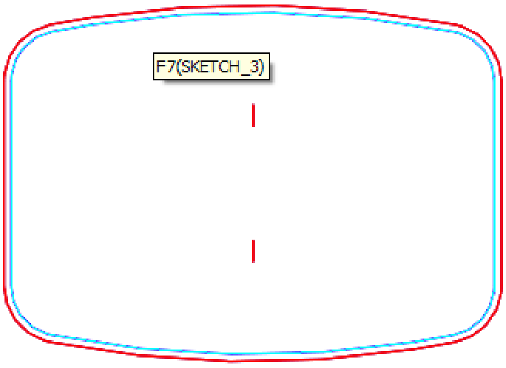


Figure 45 – Chain

- Pick  to apply the changes and exit the dialog box.
- Now open the **Housing_Top.prt** in a separate window and **regenerate** the part. Notice that the curve that we just added in the publish geometry appears in this part. 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.

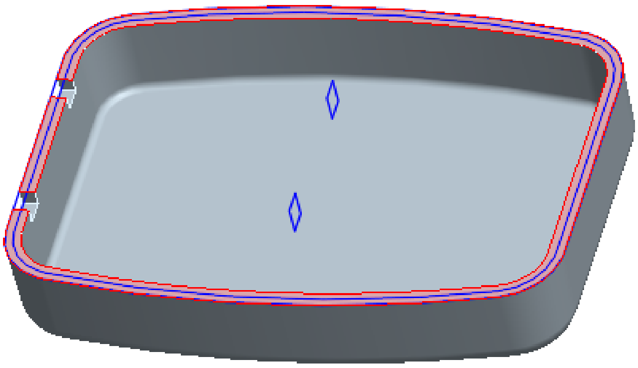
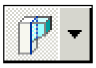


Figure 46 – Offset Surface

- Pick **Editing** > **Offset** or  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 **Define** tab in the References slide-up panel. Select the **TOP** plane as the sketch plane and click **Sketch**. Also if necessary flip the direction of the sketch plane. Select the newly added loop as shown below.

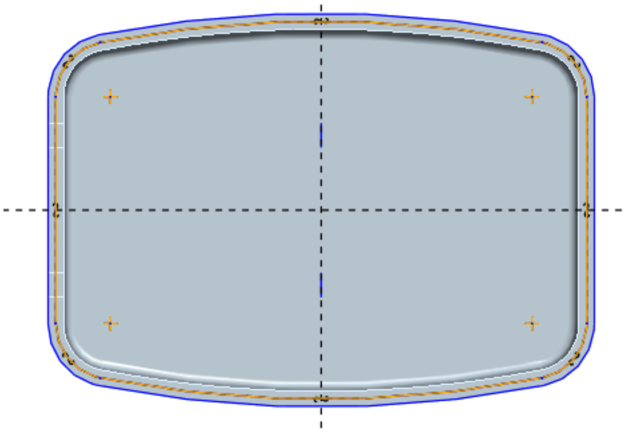


Figure 47 – Sketch

- 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.

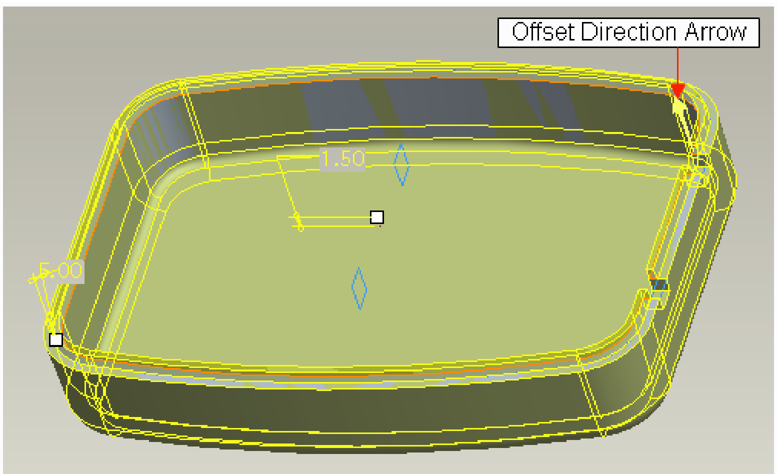


Figure 48 – Offset Direction

- 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**. **Complete** the feature. The part will appear as shown below.

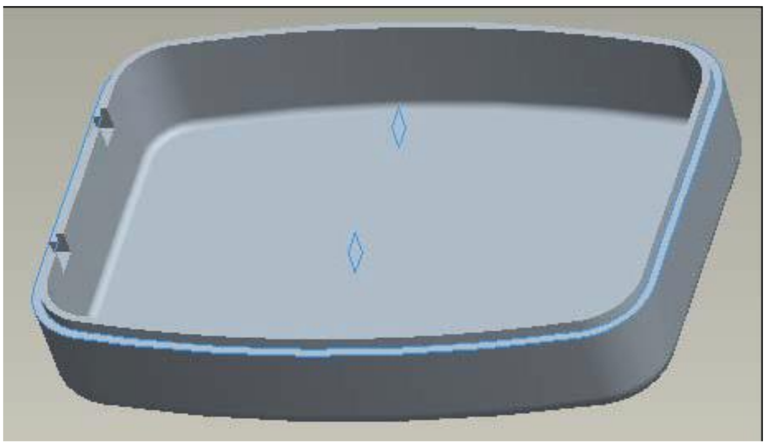


Figure 49 – Housing_Top

Note - 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. Notice that the curve that we just added in the publish geometry appears in this part.

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

- So first select the following surface of the part.

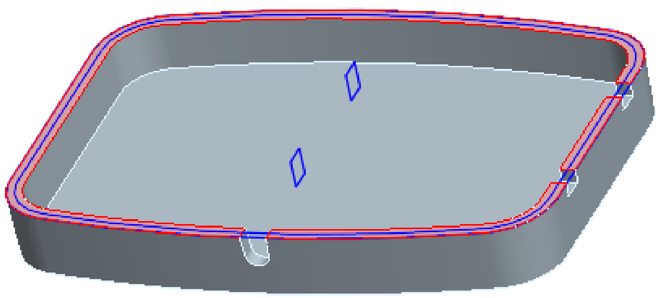


Figure 50 – Offset Surface

- Invoke 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 **TOP** plane as the sketch plane and click Sketch. Also if necessary flip the direction of sketch plane for clear view. Create a loop that is offset by **0.05** from the newly added curve. 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. The preview will appear as shown below.

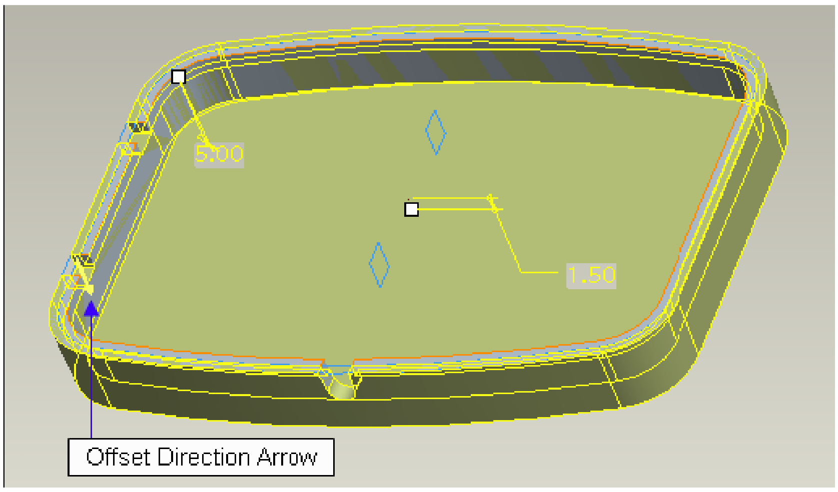


Figure 51 – Offset Direction

- 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.

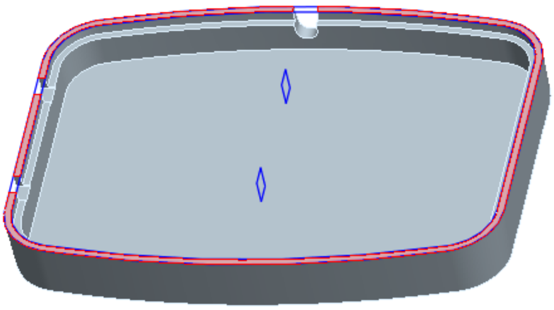


Figure 52 – New Offset Surface

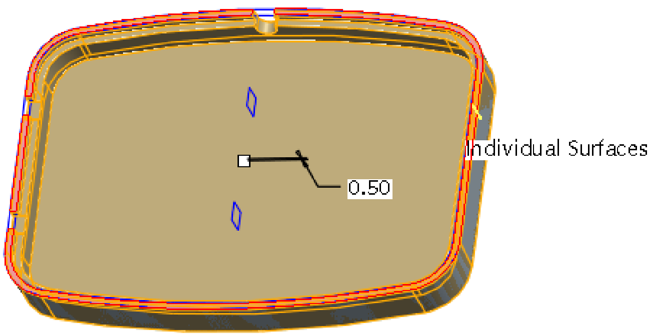


Figure 53 – Housing_Bottom

- Pick **Editing > 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 above. 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 select the Layer Tree. Hide the 07___PRT_PUBLISH_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 highlight this fact.

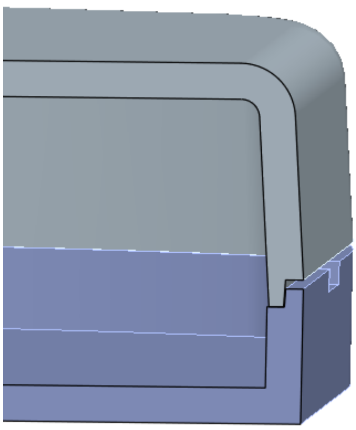


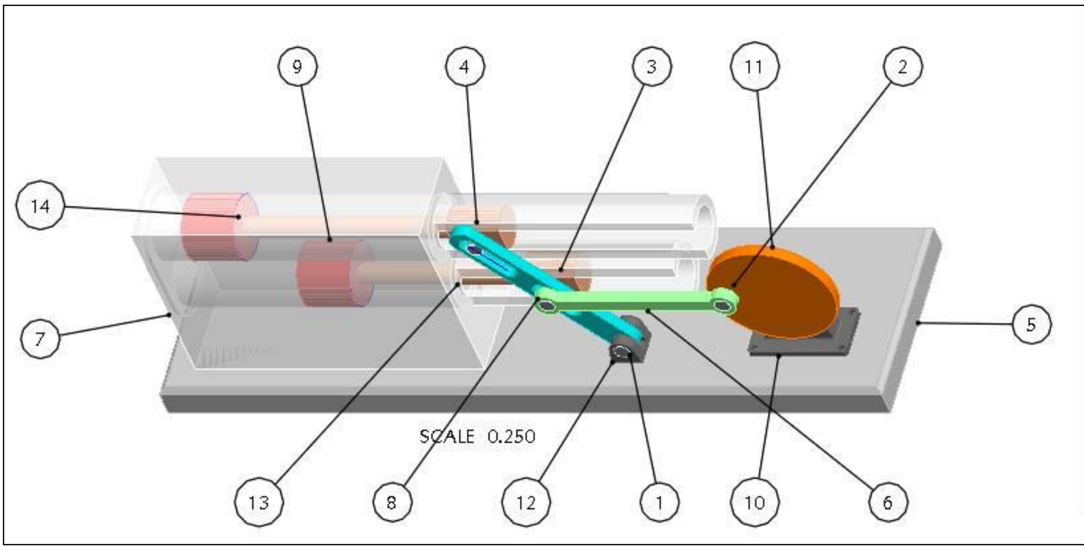
Figure 54 – Gap

Part 2
Double Cylinder Machine

Le travail demandé porte sur la mise en place de la maquette numérique d'une machine à double cylindre en vue de réaliser des tests comportementaux (simulations dynamiques et calcul de structure).

Partie 1 : Création de l'assemblage

Les noms de l'ensemble des pièces constituants la machine figurent sur le dessin d'ensemble ci-dessous.



NOMENCLATURE

Rep	NB	NOM DU COMPOSANT
1	1	AXE_MANIVELLE
2	1	AXE_ROUE
3	1	AXE_TIGE_PISTON_BAS
4	1	AXE_TIGE_PISTON_HAUT
5	1	BASE
6	1	BIELLE
7	1	BLOC_CYLINDRE
8	1	MANIVELLE
9	2	PISTON
10	1	PORTE_ROUE
11	1	ROUE
12	1	SUPPORT_AXE_MANIVELLE
13	1	TIGE_PISTON_BAS
14	1	TIGE_PISTON_HAUT

Figure 1 – Nomenclature de la machine

Le mécanisme étudié est modélisé selon le graphe de liaisons donné sur la Figure 2. Les liaisons de nature mécanique sont mentionnées en caractères gras tandis que les contraintes géométriques d'assemblage sont notées en italique.

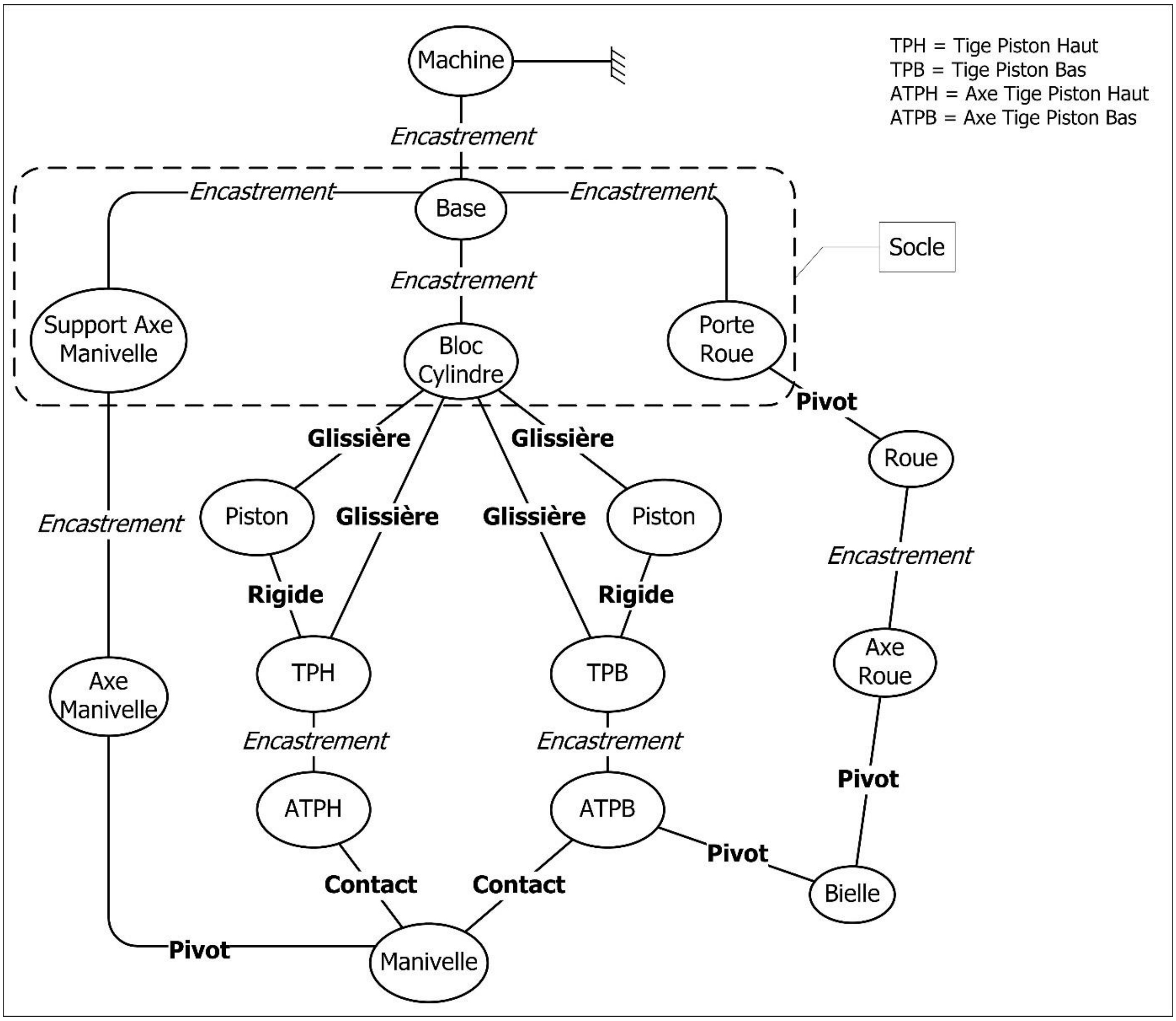


Figure 2 – Graphe des liaisons

NB : Le graphe des liaisons ne tient pas compte de la réalisation technologique du montage et notamment de l'hyperstaticité du mécanisme.

NB : Les axes et plans des liaisons ou contraintes géométriques ne sont pas spécifiés dans la mesure où aucune ambiguïté ne réside quant au fonctionnement de la machine.

1. Créer un assemblage **Machine**. Assembler le sous-assemblage **Socle** puis l'ensemble des autres pièces figurant sur le graphe des liaisons à l'exception de **Piston** (qui ne figure pas parmi les fichiers donnés).

NB: Pour établir les liaisons cinématiques requises vous pouvez, si besoin, installer toutes les entités de référence nécessaires (point, axe, plan, etc) sur les différents composants en vous appuyant sur les entités géométriques particulières des pièces telles que alésages, portées sphériques, surfaces, etc. Faire des hypothèses en cas d'information manquante.

2. Vérifier le bon fonctionnement cinématique du mécanisme à l'aide de l'outil « Drag Components ».

   Afin de finir d'assembler la machine, le composant **Piston** doit être conçu. Pour cela on souhaite pouvoir hériter, par le biais d'un squelette, du diamètre des chambres ainsi que celui des tiges au sein du composant.

3. Créer un squelette **Machine_Ske** au sein de l'assemblage **Machine**. Créer dans ce fichier deux esquisses circulaires de diamètre respectif 60 et 20 mm. Partager ces données afin de pouvoir en hériter dans tout composant.
4. Créer une pièce **Piston** sans placement. Copier dans ce composant la géométrie partagée puis dessiner le piston représenté Figure 3.

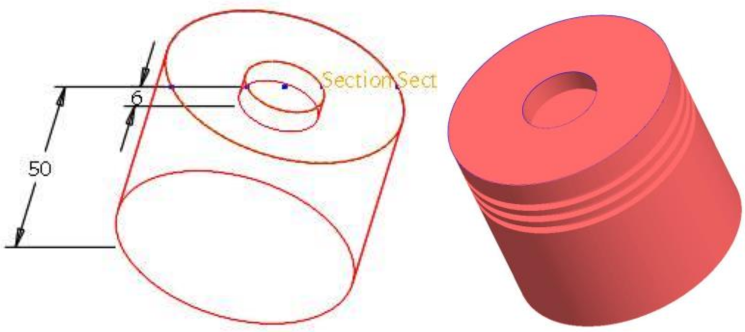


Figure 3 – Piston

5. Placer le composant **Piston** selon le graphe des liaisons.

   NB: Afin que la maquette soit totalement paramétrée, pilotée, par les esquisses figurant dans le squelette pour pouvoir totalement bénéficier d'une conception descendante, il conviendrait de redéfinir les pièces **Bloc_Cylindre** et **Tige_Piston** à s'appuyant sur le squelette.

6. Affecter le matériau acier (steel) à toutes les pièces. Donner la masse totale de l'assemblage :

   $M_{assemblage}$ =………………kg.

Partie 2 : Simulations dynamiques

On se propose maintenant de vérifier le comportement du mécanisme sous l'effet d'un moteur.

7. Créer les clichés suivants pour le mécanisme :

   - Snapshot 1 : le volume disponible dans les chambres est minimum. Nommer ce cliché **Mini**.

- Snapshot 2 : le volume disponible dans les chambres est maximum. Nommer ce cliché **Maxi**.

8. Installer un moteur sur l'axe de l'arbre d'entrée du mécanisme afin de simuler le comportement de ce dernier. Appliquer une vitesse de rotation de 120 deg/s. Nommer le moteur **Moteur1**.
9. Définir une analyse permettant de simuler pendant 6s le fonctionnement du mécanisme (prendre un intervalle de temps minimum égale à 0,02s). Définir le cliché **Mini** comme configuration initiale. Nommer cette analyse **Analyse1_Moteur1**.
10. Créer une mesure de la position puis de la vitesse de chacun des pistons. Nommer ces mesures **Position_piston_haut**, **Vitesse_piston_haut**, **Position_piston_bas** et **Vitesse_piston_bas**.
11. Quelles sont les courses des pistons haut et bas ?

    Course piston haut :………………mm

    Course piston bas : ………………mm

12. Quelles sont les vitesses et accélération max des pistons haut et bas ?

    Vitesse max piston haut :………………mm/s

    Vitesse max piston bas :………………mm/s

    Accélération max piston haut :………………mm/s²

    Accélération max piston bas :………………mm/s²

13. Tracer l'allure de la vitesse et de l'accélération sur l'intervalle de temps [0;3s]. Expliquer les différentes zones de la courbe.
14. Mettre en place un modèle de simulation (frottements et gravité désactivés) permettant de déterminer les efforts au niveau des liaisons, pendant la rotation du moteur (2 tours à partir de la position **Mini**). Nommer cette analyse **Analyse2_Moteur1**.

    - Effort radial maximum exercé par la manivelle sur le socle =………………N

- Effort radial maximum exercé par la bielle sur la tige du piston bas =………………N

- Position correspondante = ………………

15. Installer un moteur force sur l'axe de l'arbre d'entrée du mécanisme. Appliquer un couple variable de manière à obtenir un mouvement proche de la simulation précédente (2 tours effectuée par la roue en 6s). Nommer le moteur **Moteur2**. Nommer l'analyse **Analyse_Moteur2**.

A l'inverse du mécanisme étudié, les moteurs munis de pistons fonctionnent grâce à la compression des gaz dans les chambres de combustion. Ces derniers entraînent le déplacement des pistons, qui à leur tour entraîneront manivelle, bielle puis arbre en rotation.

16. Expliquer dans quelles mesures il est possible de modéliser de tel système en fonctionnement proche de la réalité.