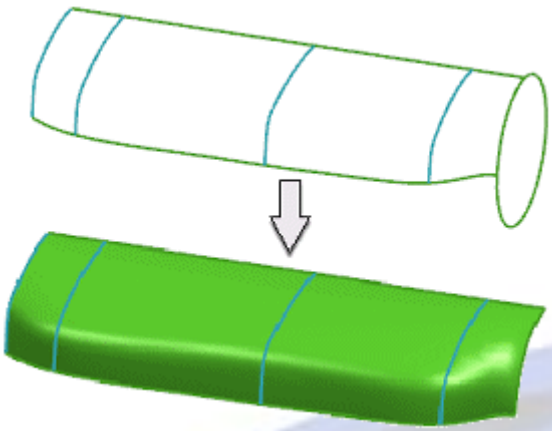
Content

Activity: Divide a curve using multiple bounding objects

Estimated time to complete: 3–6 minutes

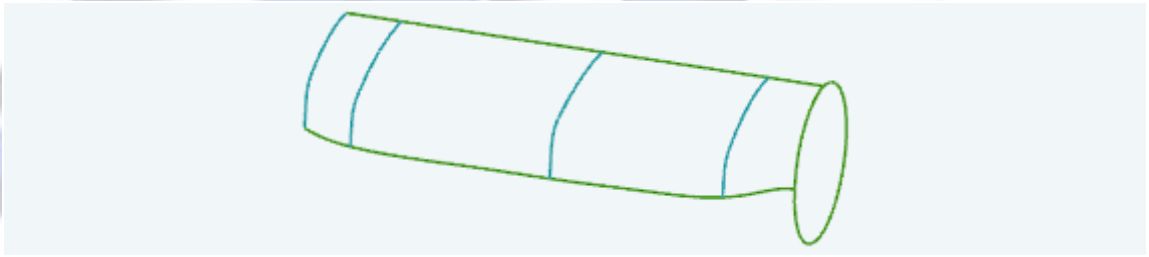
In this activity, you will divide a circle into segments so you can use a curve that starts and ends at intersecting curves for creation of a curve mesh surface.



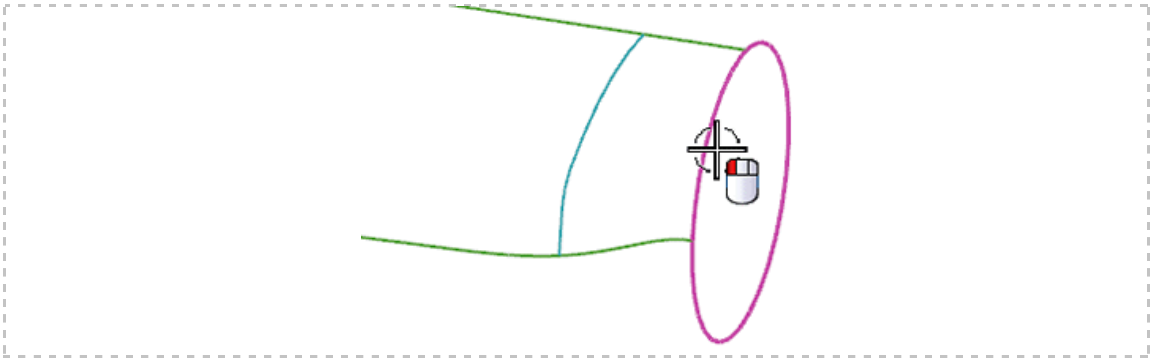
Launch the Divide a curve using multiple bounding objects activity.

Open the part and start the curve dividing

1. Open *des18_divide_curve*.

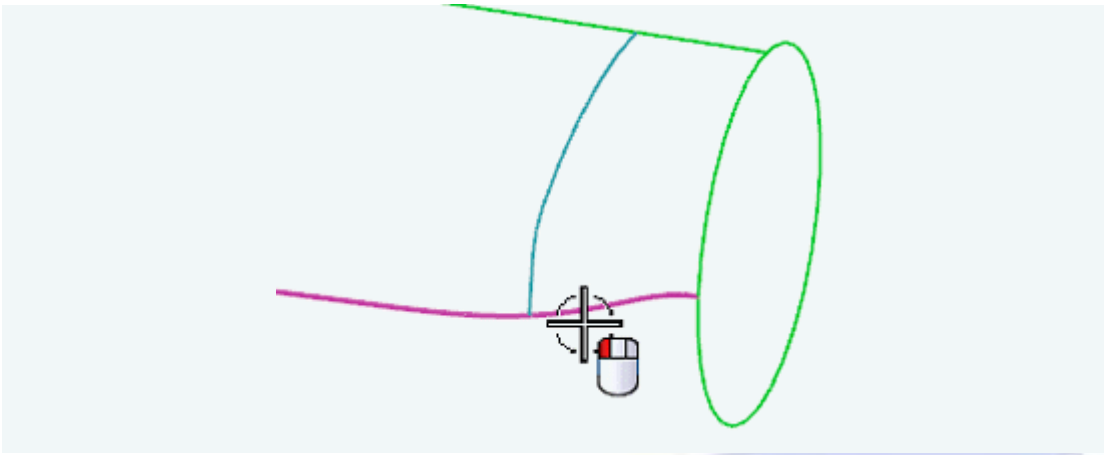


2. On the **Edit Curve** toolbar, click **Divide Curve**  .
3. In the **Divide Curve** dialog box, from the **Type** list, select **By Bounding Objects**.
4. Select the circle.

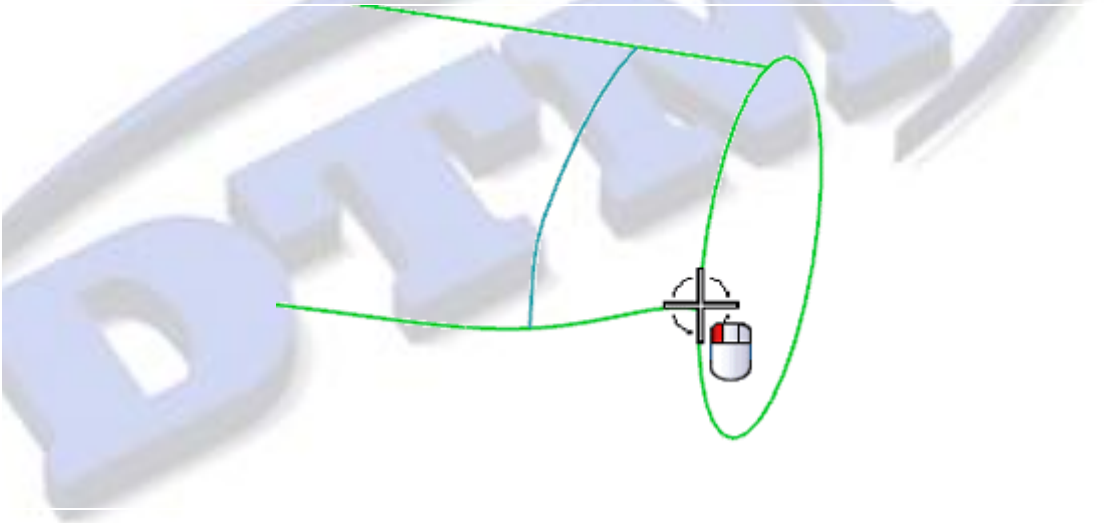


Select the bounding objects and then delete the unwanted portion

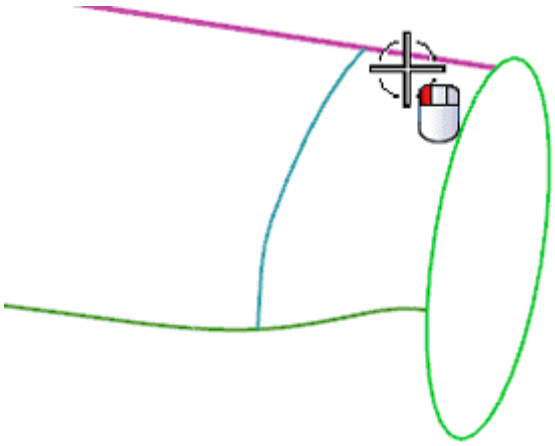
1. In the **Bounding Object** group, with **Select Object** active, select the lower spline.



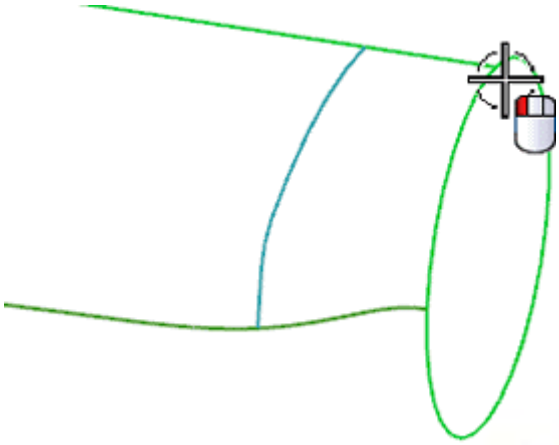
2. Select the intersection point you want.



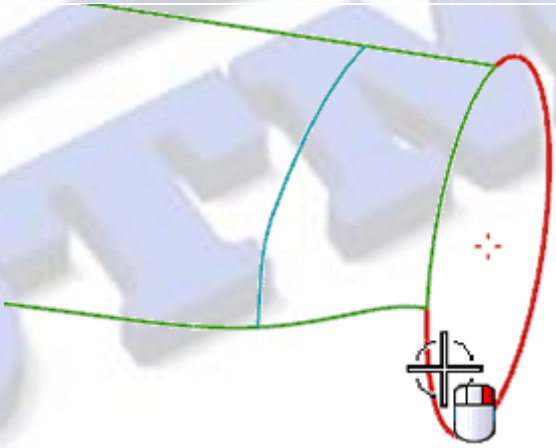
3. With **Select Object** still active, select the upper spline.



4. Select the intersection point you want.



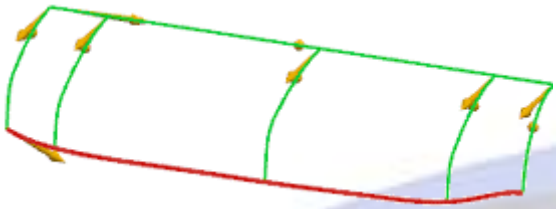
5. Click **OK**.
6. Right-click the arc outside the splines and choose **Delete**.



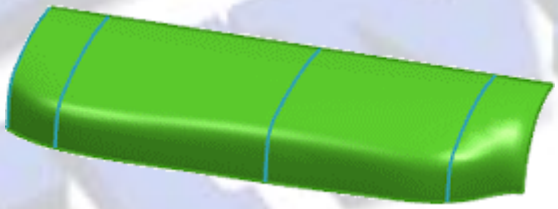
Create a curve mesh surface (optional)

1. On the **Surface** toolbar, click **Through Curve Mesh**  .
2. Select the five short strings of curves as **Primary Curves**. Make sure they are selected as 5 sets (use **Add New Set** ), and that the directions are all the same.

3. Select the two long curves as **Cross Curves**. Make sure they are selected as 2 sets (use **Add New Set** ), and that the directions are the same.



4. Click **OK**.



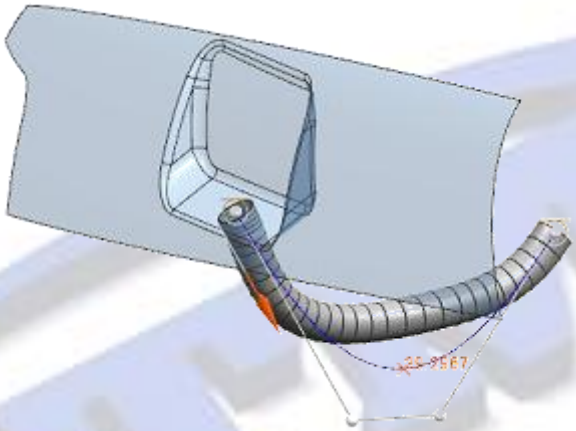
You completed the activity.

Activity: Create a studio spline to be used as a centerline of a tube

Estimated time to complete: 5–8 minutes

In this activity, you will create a studio spline to be used as a centerline for a connecting hose. You will:

- Create an associative **Through Points** studio spline normal to a face of a surface.
- Edit the surface to see how the associated studio spline behaves.
- Change the type of the studio spline from a **Through Points** to a **By Poles** spline.
- Modify the degree of the spline and extend one end of the spline.

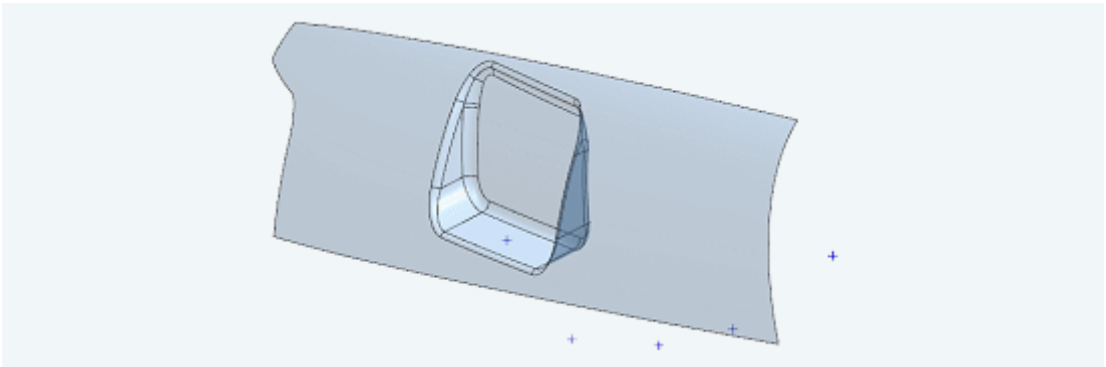


Launch the Create a studio spline to be used as a centerline of a tube activity.

Create a studio spline to be used as a centerline of a tube

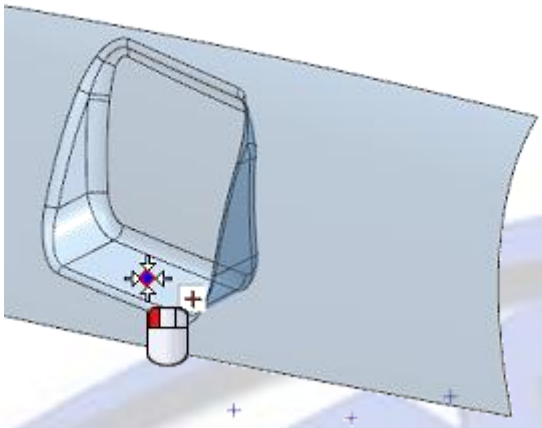
Open the part and create a Through Points studio spline

1. Open *des18_spline_normal_1*.

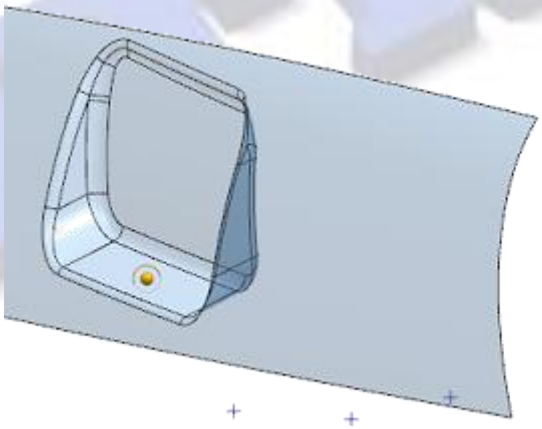


2. Choose **Preferences→Visualization**, **Visual** tab and select the **Translucency** check box.

3. On the **Curve** toolbar, click **Studio Spline**  .
4. Make sure **Type** is set to **Through Points**.
5. In the **Point Location** group, expand **Constraints**.
6. On the **Snap Point** toolbar, click **Existing Point**  .
7. Select the first point for the spline.

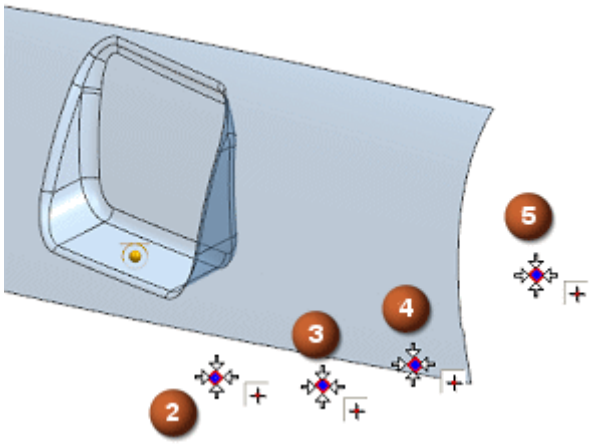


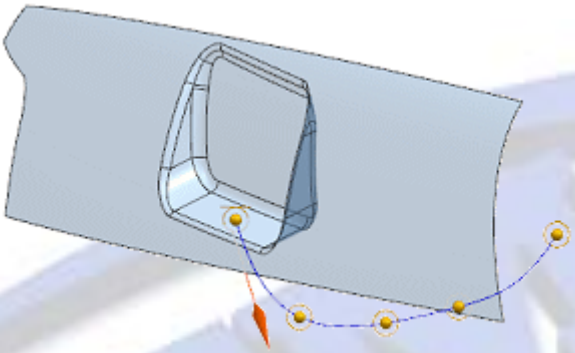
The point is associative with the center of the surface



8. Under **Constraints**, select the **Constrain to Attachment Parent** check box.
9. From the **Inferred Type** list, select **Normal**.

   This creates a constraint that maintains its tangent to the surface's natural normal.
10. Continue selecting points to create the spline.

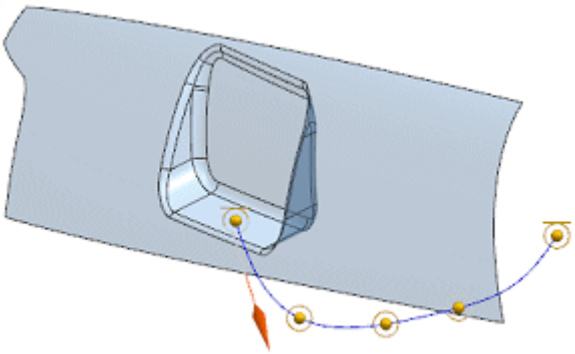




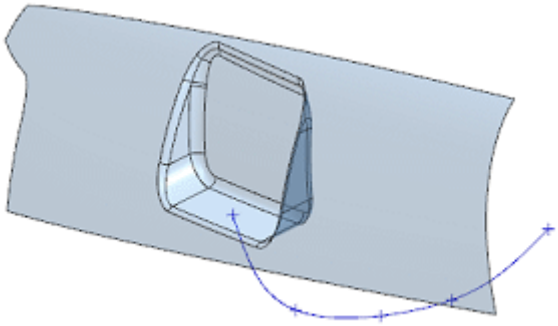
Because you selected existing points for the Through Points spline, each interior point has a position constraint (G0) and the end points have tangency constraints (G1).

11. Select the **G1 Magnitude** check box.
12. In the **G1 Magnitude** box, type **500** then press Enter.

This creates a tangency magnitude constraint on the last point.

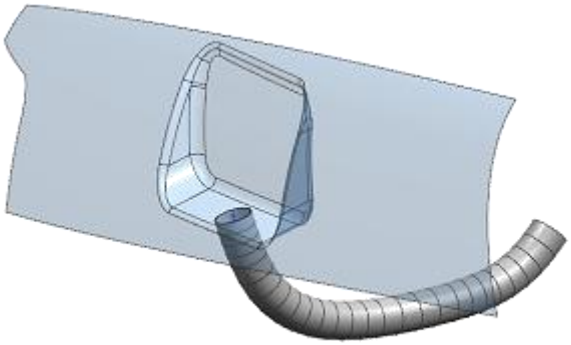


13. Click **Apply**.

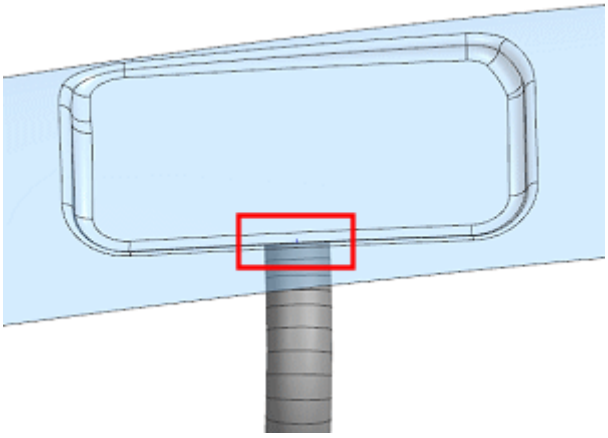


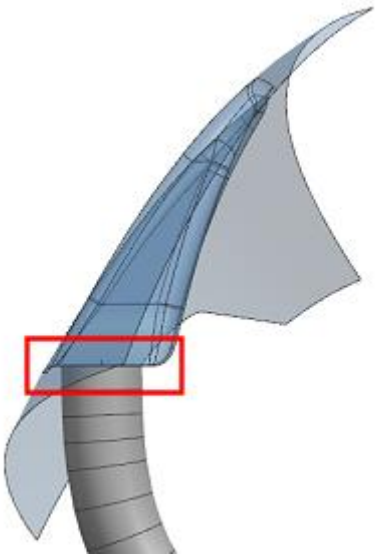
Create a tube using the Studio Spline

1. Choose **Insert** → **Sweep** → **Tube**.
2. Select the spline you just created as the path.
3. In the **Outer Diameter** box, type **30** then press Enter.
4. In the **Inner Diameter** box, type **28** then press Enter.
5. Click **OK**



6. Rotate the model around the connection to check for the normal constraint.



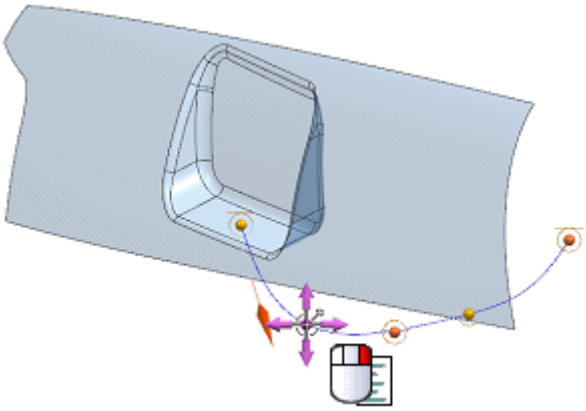


Modify the Through Curves surface and proportionately update the spline

The bottom face of the pocket was created using a projected curve to define its orientation. You must change the direction of the projection to change the angle of the pocket sides.

Because you want to proportionately update the spline, you will have to remove two position constraints to allow those spline points to move.

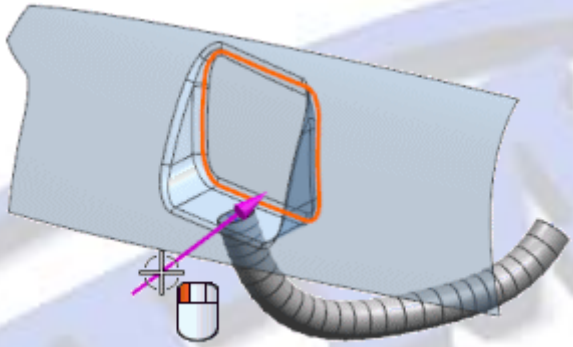
1. In the **Part Navigator**, expand the **Cameras** node and double-click the **start** camera.
2. In the **Part Navigator**, right-click **Spline (62)** node and choose **Edit Parameters** to edit it.
3. Position the cursor over the spline point as shown, right-click and select **Delete G0** from the list.



4. Repeat to delete the position constraint on the next point to the right.
5. Expand the **Settings** group and select the **Proportional Update** check box.

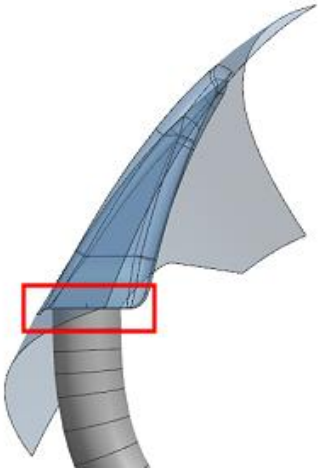
When **Proportional Update** is selected, when you modify the studio spline, all the points or poles between the updated point and the next fixed point are moved proportionally.

6. Click **OK**.
7. In the **Part Navigator**, under the **Model History** node, right-click **Fixed Datum Axis (43)** and select **Show**.
8. Still in the **Part Navigator**, under the **Model History** node, right-click **Projected Curve (44)** and select **Edit Parameters**.
9. In the **Project Curve** dialog box, in the **Projection Direction** group, from the **Specify Vector** list, select **Inferred Vector**  .
10. Select the **Datum Axis** as the new vector.

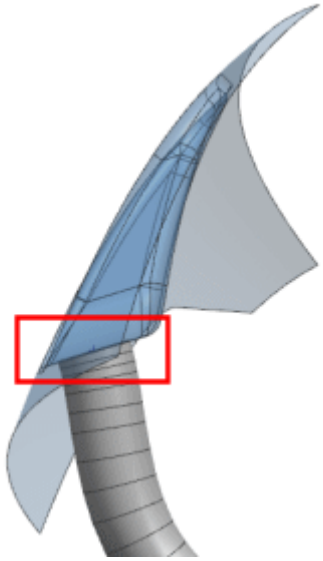


11. Click **OK**.
12. Once the model has updated, hide the **Datum Axis**.

Rotate the model and notice that the tube, which is associated with the spline, stays normal at the start point of the spline, which is associative to the face of the surface.



Before

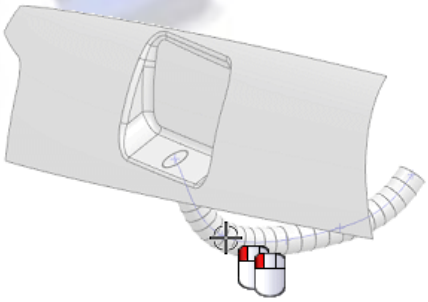


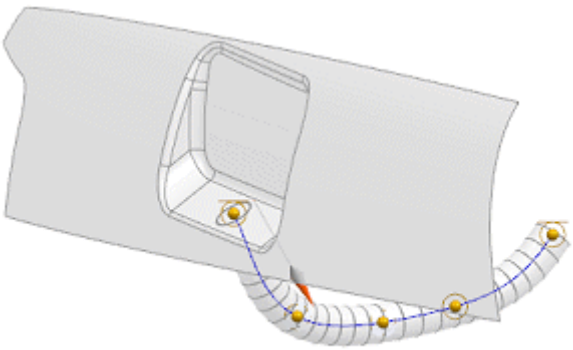
After

13. In the **Part Navigator**, expand the **Cameras** node and double-click the **start** camera.

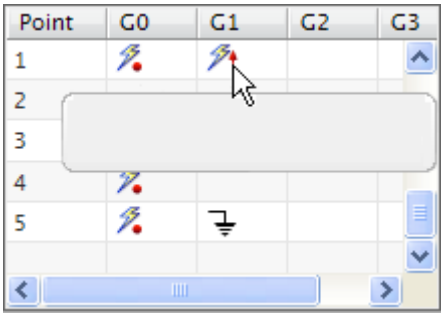
Change spline types

1. On the **View** toolbar, click **See Thru All**  .
2. Double-click the spline to edit it.





This current condition is reflected in the **List**.



Tip If you position the cursor over a constraint symbol in the list, constraint information is displayed. The balloon above states, Inferred Tangent Parent: Face of Through Curves (47)

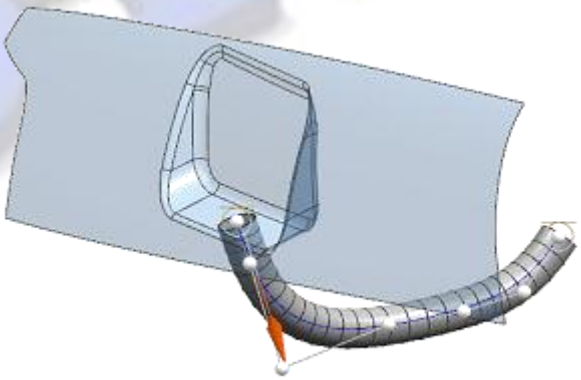
3. On the **View** toolbar, click **See Thru All** .
4. Set the **Type** list to **By Poles** .

An **Alerts** message displays

The defining data of Through Points type cannot be retained. 'Undo' can be used at this time to revert the data.

When you switch from **Through Points** to **By Poles** spline type, the through points and any internal constraints are deleted.

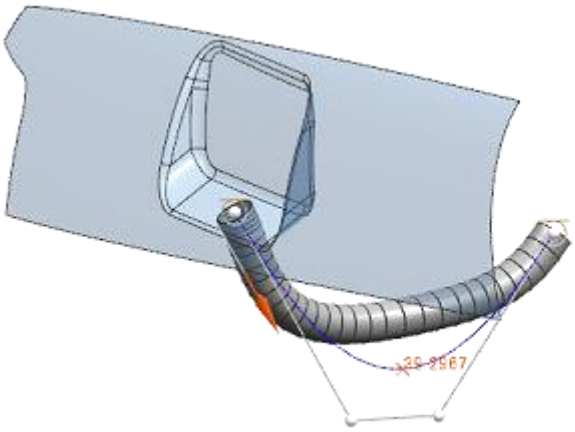
Only the start and end constraints are retained.



Convert the spline to single segment and change its degree

It has been decided that the spline is to be a single segment **By Poles** spline with a degree of 3.

1. In the **Parameterization** group, select the **Single Segment** check box.
2. In the **Degree** box, type **3** then press Enter.



When you specify degree control for a single segment studio spline, the number of poles is automatically adjusted depending on the degree you specify.

The apex point of the spline is displayed with the arclength percentage of its location on the spline.

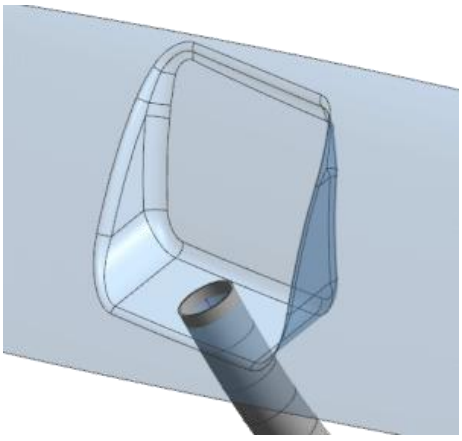
Extend the spline

There is a design requirement to extend the tube beyond its limiting face.

Since the tube is associated with the spline, you can extend the spline to meet this requirements.

1. In the **Extension** group, from the **Start** list, select **By Value**.
2. In the **Value** box, type **7** then press Enter.

When you extend a studio spline, it creates a natural extension on top of the spline.



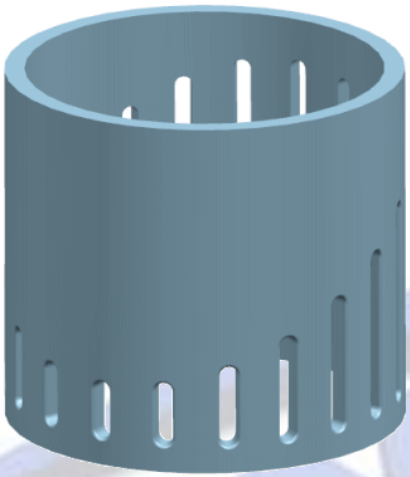
3. Close the part.

You completed the activity.

Activity: Create a feature pattern using sketch references

Estimated time to complete: 5–8 minutes

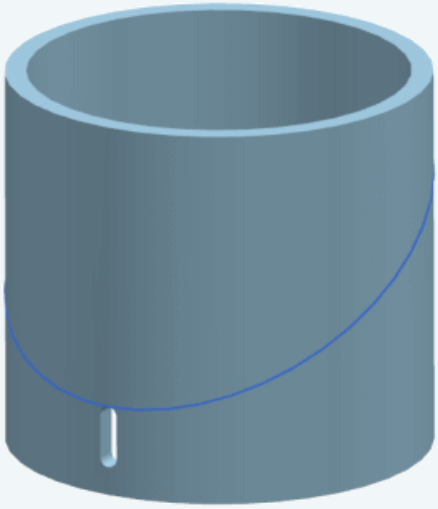
In this activity, you will create a pattern of features that reuses sketch geometry to control the length of slots as they traverse around a cylindrical tube.



Launch the Create a feature pattern using sketch references activity.

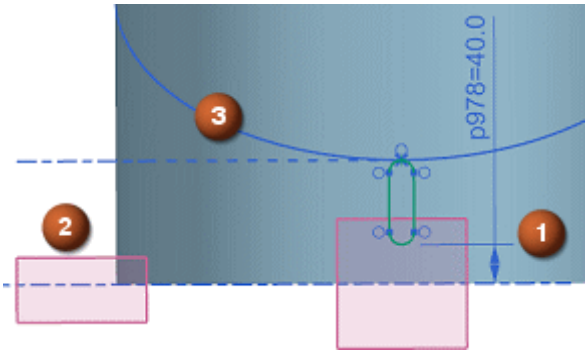
Open and examine the part

1. Open *des18_pattern_feature_reuse_reference*.



The part is made up three extrusions, two that define the main cylinder and one for the slot. There is also an edge blend which is suppressed – you will use that later in the activity.

2. In the **Part Navigator**, with the cursor over **Extrude (15)**, right-click → **Edit Sketch**.
3. Rotate the sketch, as shown, and examine the constraints of the slot sketch.



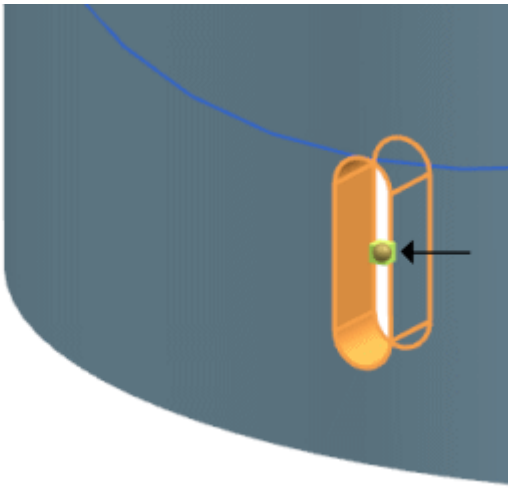
The sketch is constrained dimensionally to the base of the large cylinder using a datum plane (1) and tangentially constrained to an angled datum plane (2) which also is used to define the elliptical intersection curve (3).

4. Click **Finish Sketch**.

Select features to pattern

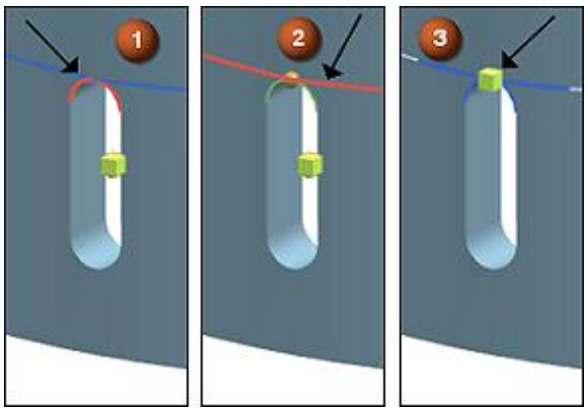
1. On the **Feature** toolbar, click **Pattern Feature**  .
2. On the rail of the **Pattern Feature** dialog box, click **Reset**  to reset the default dialog box options.
3. Select **Extrude (15)**.

The Reference Point is displayed (center of mass).



4. In the **Reference Point** group, from the **Specify Point** list, select **Intersection Point**  .

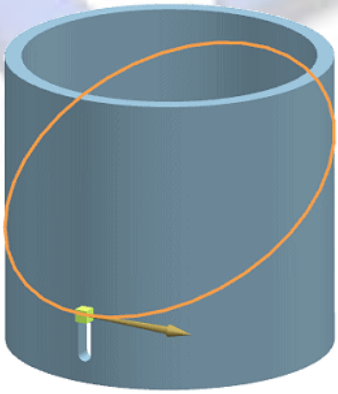
5. Locate the reference point at the intersection between the upper arc of the slot and the intersection curve on the cylinder.



You want the reference point of the initial slot feature to reference the intersection curve so as the pattern of features is created, the reference point of each instance of the pattern will follow the intersection curve.

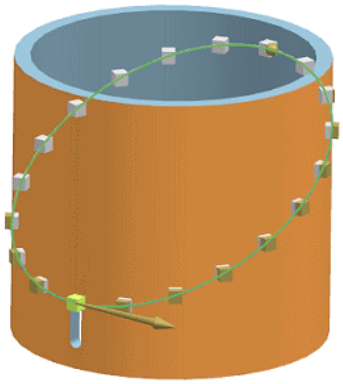
Define directional and orientation parameters

1. In the **Pattern Feature** dialog box, in the **Pattern Definition** group, set **Layout** to **Along**  .
2. Make sure **Path Method** is set to **Offset**.
3. Select the intersection curve on the large cylinder.



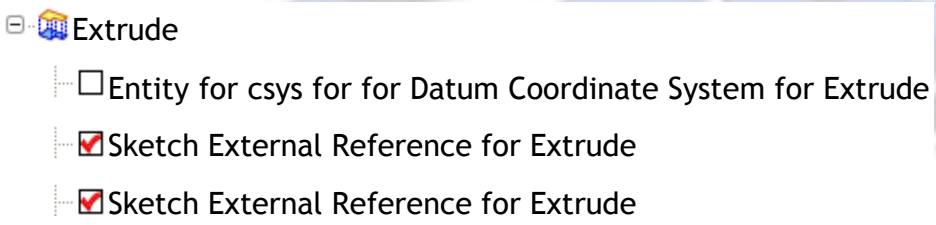
4. In the **Direction 1** section, set the following parameters:
   - **Spacing** = **Count and Span**
   - **Count** = **20**
   - **Location** = **% Arc Length**
   - **% Span By** = **100**
5. In the **Orientation** section, set the following parameters:
   - **Orientation** = **Same as Input**.
   - Select the **Follow Face** check box.

6. Select the outside face of the large cylinder.

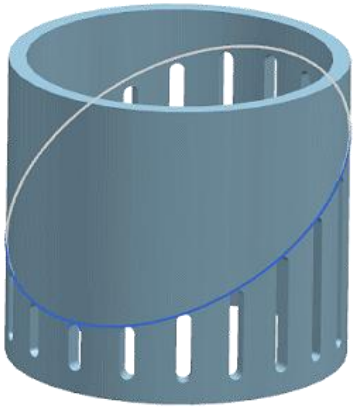


Identify reusable references

1. In the **Pattern Method** group, make sure **Method** is set to **Variational**.
2. In the **Reusable References** list, select the check boxes for the two external sketch references.



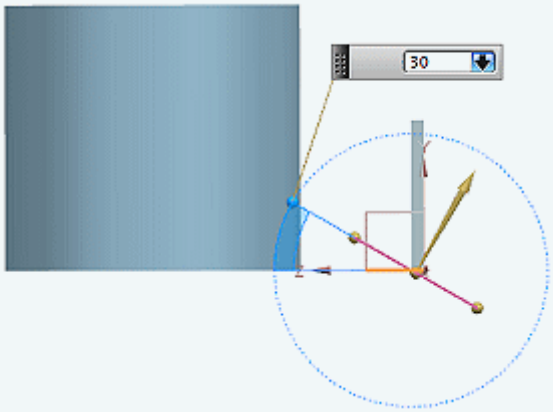
3. The first sketch reference is to the tangency constraint between the top of the slot and the intersection curve. The second sketch reference is to the dimensional constraint that controls the height from the floor plane to the bottom of the slot.
4. In the **Preview** group, click **Show Result** to check the pattern.



5. Click **Undo Result**.
6. Click **OK** to create the pattern of features.

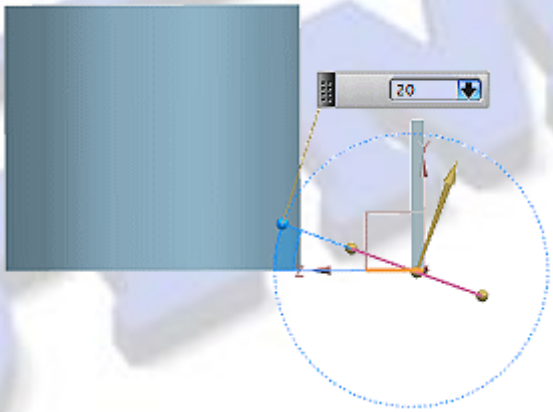
Test associativity of slot geometry

1. In the **Part Navigator**, double-click **Datum Plane (13)** to edit it.



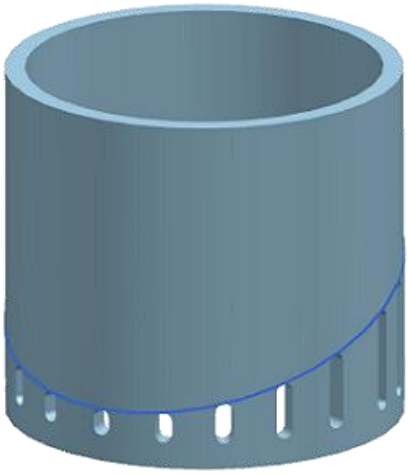
The angle of this datum plane controls the angle of the intersection curve on the cylinder. It is currently at a 30 degree angle.

2. In the **Angle** type **20**. and press Enter.



3. Click **OK** in the **Datum Plane** dialog box.

The pattern feature instances update based on the new angle of the datum plane, which controls the intersection curve to which the seed slot is associated.



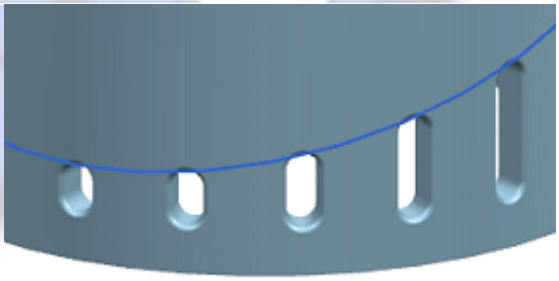
Edit the pattern to add an edge blend to the feature pattern

There is an edge blend on the initial slot which is suppressed. You want that blend on all the instances of the feature that were patterned.

1. In the **Part Navigator**, select the **Edge Blend** check box to unsuppress the blend.
2. In the **Part Navigator**, double-click the **Pattern** node.
3. In the **Pattern Feature** dialog box, in the **Feature to Pattern** group, make sure **Select Feature** is highlighted.
4. In the **Part Navigator** and using the Ctrl key, select the **Edge Blend** node so that both the extrude feature and the edge blend feature are selected.

Note In the **Pattern Feature** dialog box, the **Select Feature** option now has a **(2)** after it, signalling that both the edge blend and extrude features are selected.

5. In the **Pattern Feature** dialog box, click **OK**.



The first edge blend is propagated to the other feature instances of the pattern.

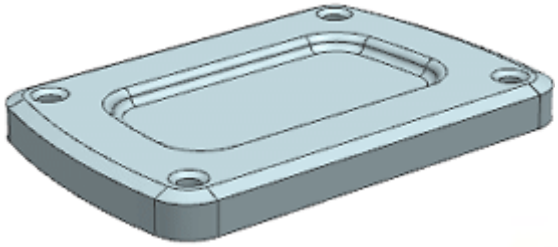
6. Close all parts.

You completed the activity.

Activity: Explore object and feature relations with the Browser

Estimated time to complete: 5–7 minutes

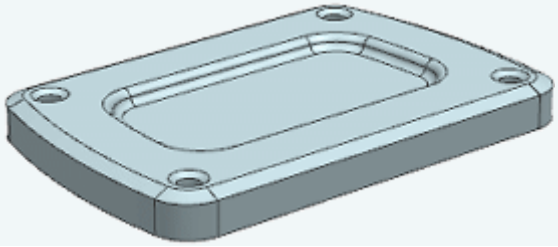
In this activity, you will use the following part to explore some of the uses of the **Browser**.



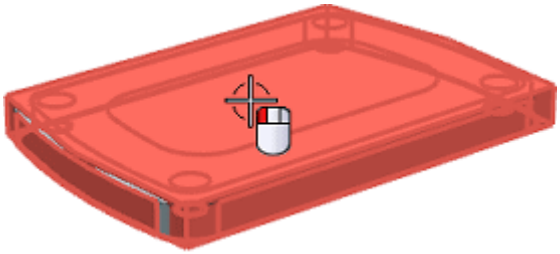
Launch the Explore object and feature relations with the Browser activity.

Open the browser graph

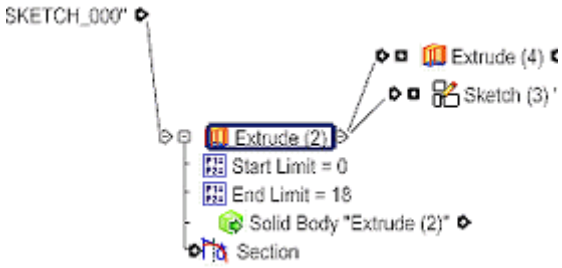
1. Open *des18_browser*.



2. Choose **Information→Browser**.
3. In the browser graph background, right-click and make sure the **Show Expressions** check box is selected.
4. In the browser graph background, right-click and clear the **All Relations View** check box.
5. In the graphics window, select **Extrude (2)**.



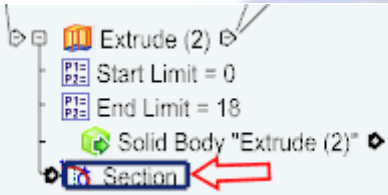
6. Notice what is focused in the browser and what is highlighted in the graphics window.



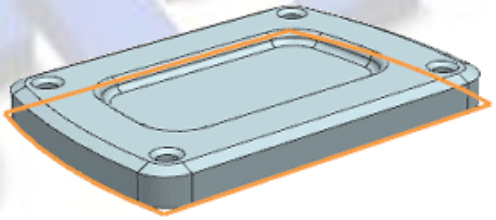
The parents are shown to the left, and the children to the right.

Explore the object relationships

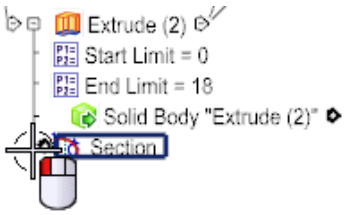
1. In the browser, select **Section**.



2. In the browser, with the mouse over **Section**, notice what highlights in the graphic window.



3. In the browser, on **Section**, click the Input parent handle.

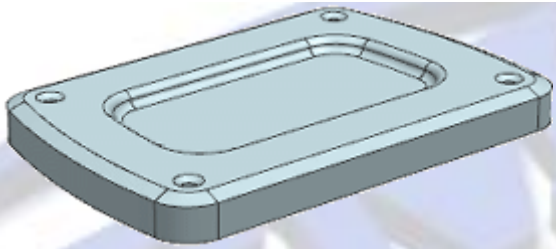


4. Notice that **Sketch (1)** is related.

5. In the browser graph, select **Extrude (2)**.

Edit some features from the browser

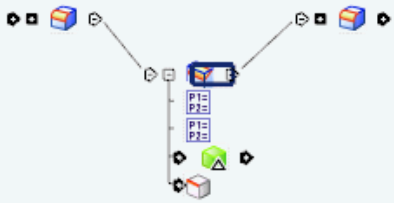
1. In the browser window, in the **Settings** group, on the **Browse** page, make sure all three check boxes are selected – **Isolate on Select**, **Center on Select**, and **Animate**.
2. In the browser graph, select **Extrude (4)**.
3. In the browser graph, select **Edge Blend (5)**, and notice the parent and child relationships.
4. In turn, select **Chamfer (6)**, **Edge Blend (7)**, **Edge Blend (8)**, and **Simple Hole (9)**. As you select those features, notice how the display in the browser moves.
5. In the browser window, with **Simple Hole (9)** highlighted, double-click the **Diameter** dimension expression.
6. In the **Diameter** text entry box, type **8** and press Enter.



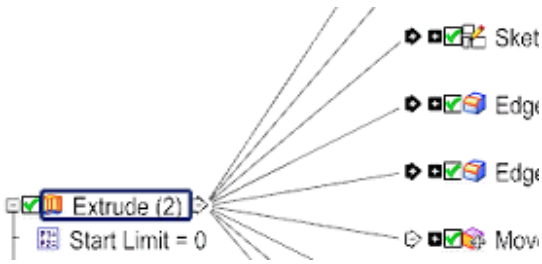
Suppress a feature with the browser

1. On the browser **Settings** group, select the **Browse** tab, and from the drop-down list, select **Icon**.

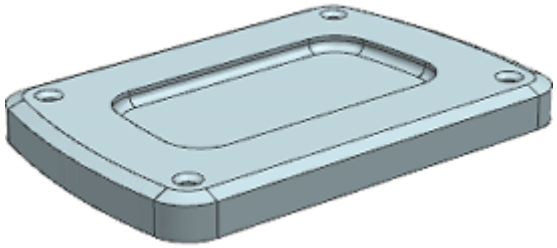
   Notice the browser display.



2. From the **Node Style** drop-down list, select **Icon + Text**.
3. In the browser graph background, right-click and choose **All Relations View**.



4. In the browser graph, double-click **Shell (11)**, and change its **Thickness** to **2**.
5. In the browser graph, on the **Edge Blend (8)** feature, clear the check box.



The blend is suppressed.

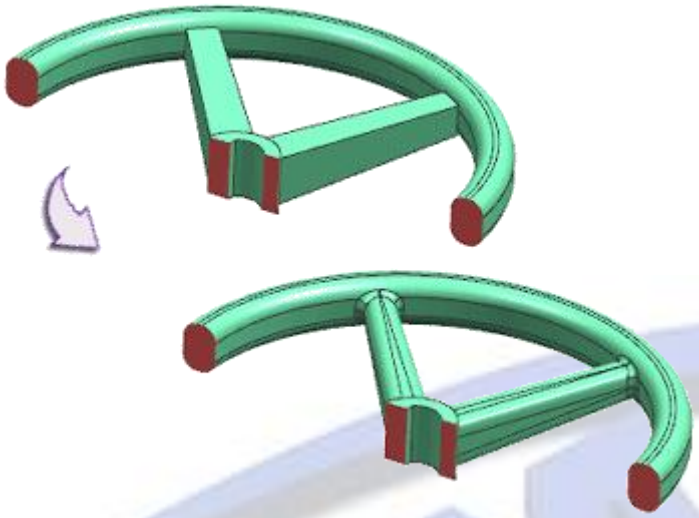
6. (Optional): Use the browser to explore the other object relationships in the part file.
7. Close the part.

You completed the activity.

Activity: Create circular and conic edge blends

Estimated time to complete: 6–9 minutes

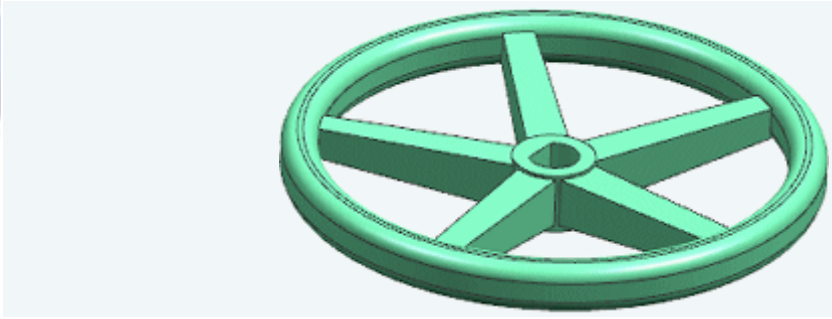
In this activity, you will create circular edge blends and conic edge blends and then compare them.



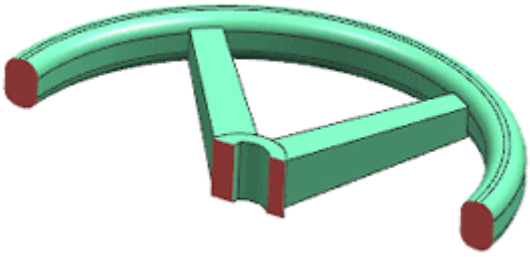
Launch the Create circular and conic edge blends activity.

Open the part and prepare the view

1. Open *des18_blend_enhancements*.



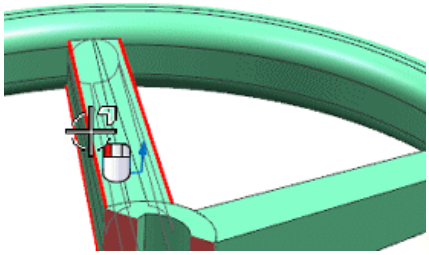
2. On the **View** toolbar, click **Clip Work Section**  .



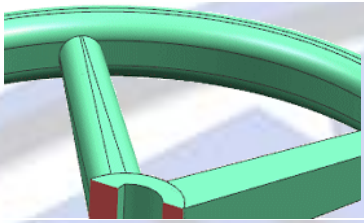
This will help you better see the blends you create.

Create the first circular edge blend

1. On the **Feature** toolbar, click **Edge Blend**  .
2. From the **Shape** list, select **Circular**.
3. In the **Radius 1** box, type **0.18** and then press Enter.
4. Select the 4 long edges of one of the spokes as shown.

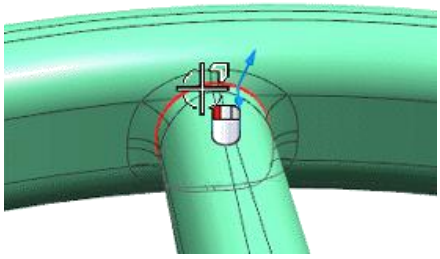


5. Click **Apply**.

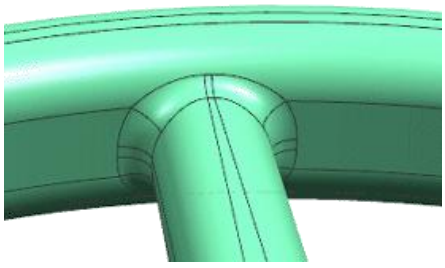


Create the second circular edge blend

1. In the **Radius 1** box, type **.09** and then press Enter.
2. On the Selection bar, from the **Curve Rule** list, select **Tangent Curves**.
3. Select any of the edges at the far end of the same spoke, as shown.

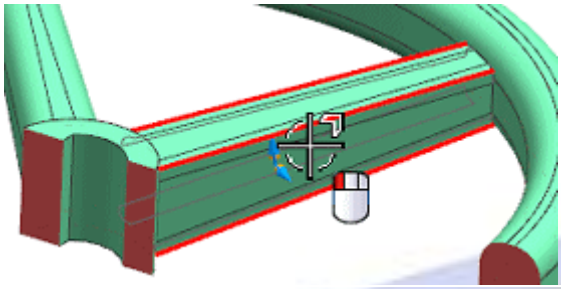


4. Click **Apply**.

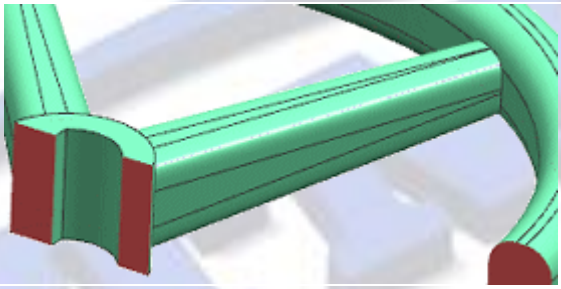


Create a conic edge blend

1. From the **Shape** list, select **Conic**.
2. From the **Conic Method** list, select **Boundary and Center**.
3. In the **Boundary Radius 1** box, type **0.18** and then press Tab.
4. In the **Center Radius 1** box, type **.06** and then press Enter.
5. Select the 4 long edges of the unblended spoke as shown.

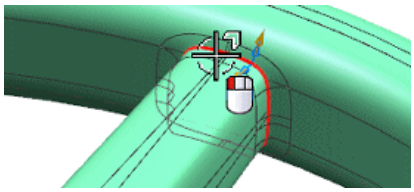


6. Click **Apply**.

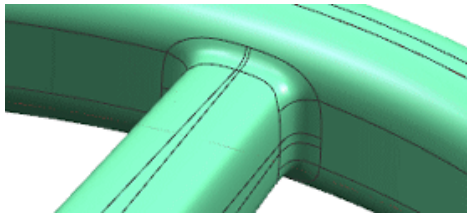


Create another conic edge blend

1. In the **Boundary Radius 1** box, type **.09** and then press Tab.
2. In the **Center Radius 1** box, type **.04** and then press Enter.
3. Select any of the edges at the small end of the spoke.

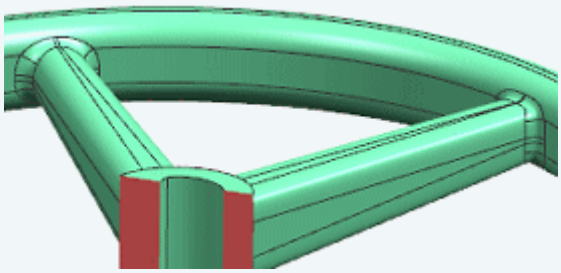


4. Click **OK**.



Compare the blends

1. Rotate the part and compare the conic edge blends with the circular edge blends.



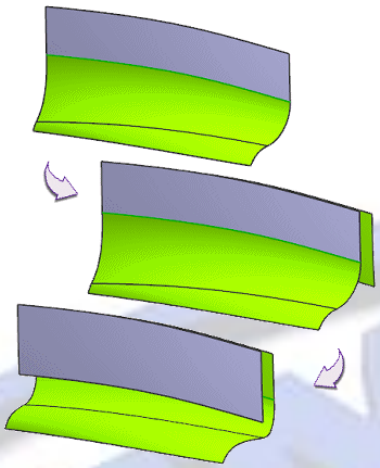
Notice the conic blends appear smoother at the edges of the blended faces.

You completed the activity.

Activity: Replace a feature with one of its dependencies

Estimated time to complete: 8–12 minutes

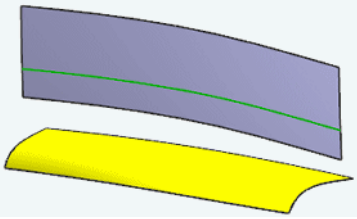
In this activity, you will create a face blend and then sew the surfaces together. Next you will create a surface offset from one of the original surfaces. Finally you will replace the original surface with the offset surface. The previous dependent features will also be moved to the replacement feature.



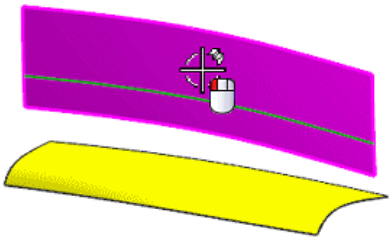
Launch the Replace Feature activity.

Start the face blend

1. Open *des18_replace_feature*.



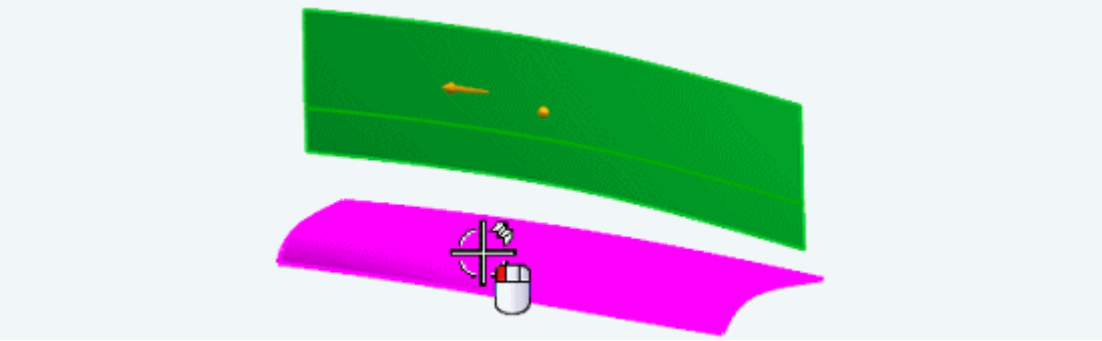
2. On the **Features** toolbar, click **Face Blend**  .
3. From the **Type** list, select **Two Defining Face Chains**.
4. With **Select Face Chain 1** active, select the upper surface.



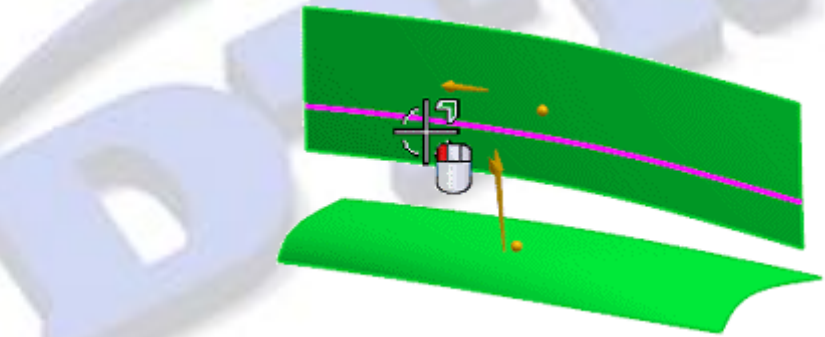
5. If the direction arrow is not pointing toward you and to the left, click **Reverse Direction** or double-click the arrow.

Complete the face blend

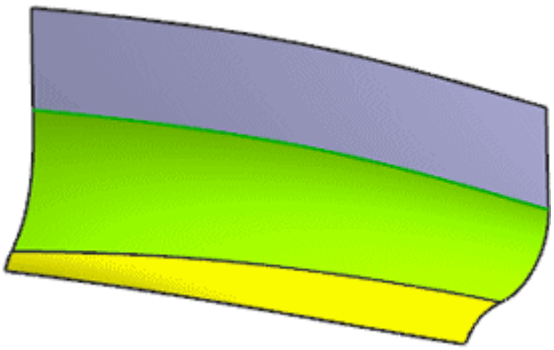
1. With **Select face Chain 2** active, select the lower surface.



2. If the direction arrow is not pointing up, double-click the arrow.
3. In the **Cross-Section** group, from the **Radius Method** list, select **Tangency Constraint**.
4. In the **Constraining and Limiting Geometry** group, with **Select Tangent Curve** active, select the curve on the upper surface.

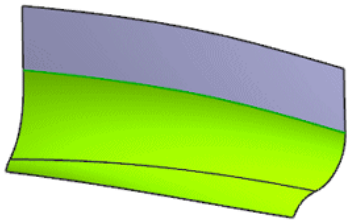


5. In the **Trim and Sew Options** group, clear the **Sew All Faces** check box.
6. Click **OK**.



Sew the surfaces together and make the first enlarge surface the current feature

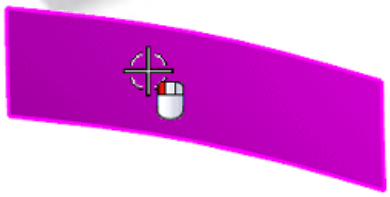
1. On the **Feature** toolbar, from the Combine drop-down list, select **Sew** 
2. From the **Type** list select **Sheet**.
3. Select any surface as the **Target**, select the other two as the **Tool**, and click **OK**.



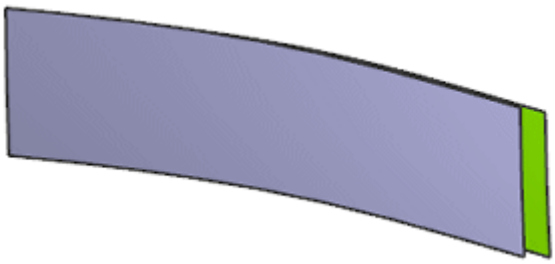
4. In the Part Navigator, right-click **Enlarge Surface (16)**, and choose **Make Current Feature**.

Create an offset surface

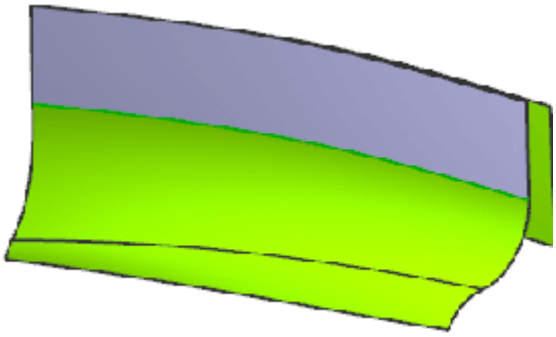
1. On the **Feature** toolbar, click **Offset Surface**  .
2. With **Select Face** active, select the face of **Enlarge Surface (16)**.



3. If the direction arrow is not pointing away from you, reverse its direction by double-clicking it.
4. In the **Offset 1** box, type **250** .
5. Click **OK**.



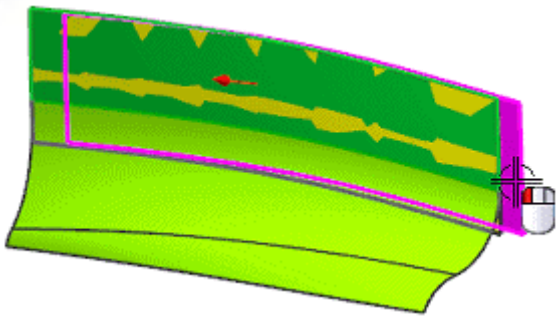
6. In the Part Navigator, right-click **Sew (10)**, and choose **Make Current Feature**.



Replace the original surface and its dependent features with the offset surface

1. In the toolbar area, right click and activate the **Edit Feature** toolbar.
2. On the **Edit Feature** toolbar, click **Replace Feature**  .
3. Select **Enlarge Surface (5)** as the feature to replace.

4. Click the middle mouse button.
5. Select the offset surface.



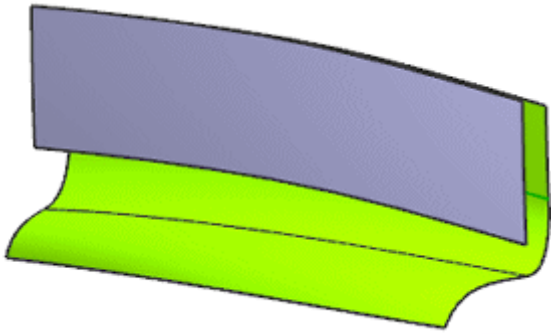
Map the new parents to the original parents

1. Click the middle mouse button.
2. In the **Original Parent** list, make sure **Face for Offset in Face (8)** is selected.
3. In the Replacement Feature window select the replacement face.
4. In the **Original Parent** list, highlight **Curve for Offset in Face (8)**.

5. in the Replacement Feature window select the bottom edge of the surface.

   Make sure the direction arrow is the same as the original.

6. Click **OK**.

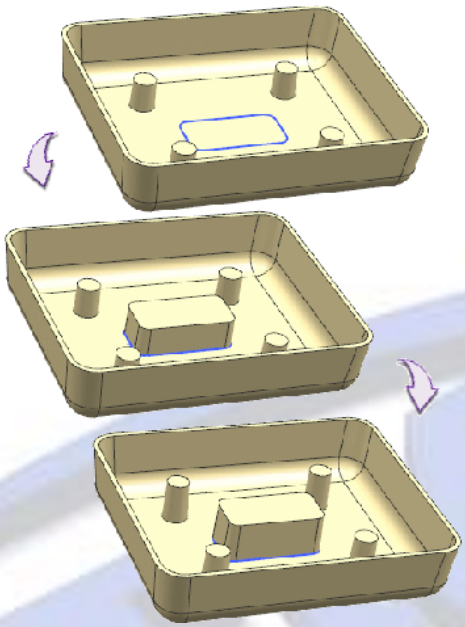


You completed the activity.

Activity: Create an extruded feature by referencing other feature dimensions

Estimated time to complete: 3-5 minutes

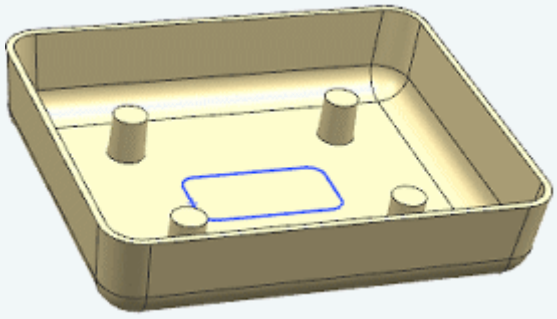
In this activity, you will create an extruded feature by selecting other feature parameters references from the graphic window. You will then test the associativity.



Launch the Extrude referencing feature dimension activity.

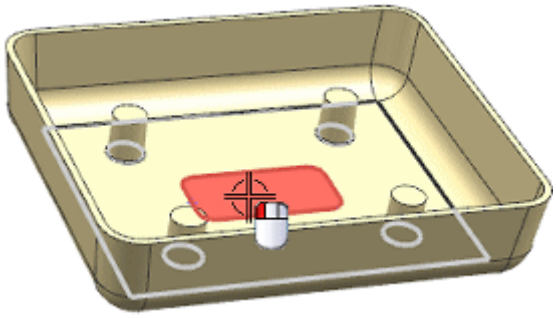
Select the area to extrude

1. Open *des18_dimension_references*.



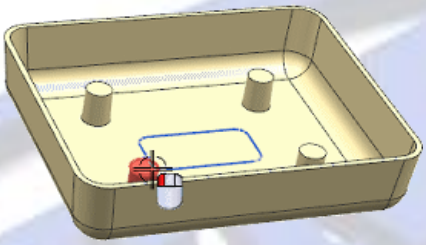
2. On the **Features** toolbar, click **Extrude**  .
3. In the **Extrude** dialog box rail, click **Reset**  .
4. On the Selection bar, from the **Curve Rule** list, select **Region Boundary Curves**.

5. In the **Section** group, with **Select Curve** active, select the portion of the face inside the curves.

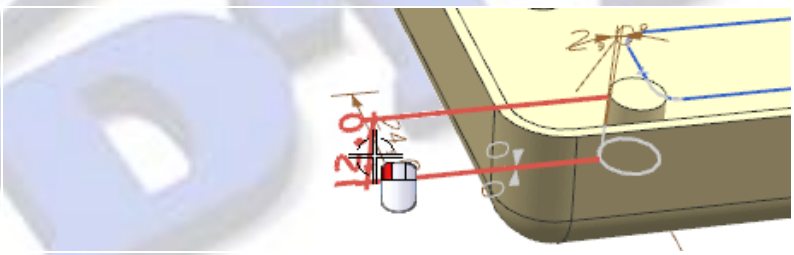


Select the dimensions from a different feature

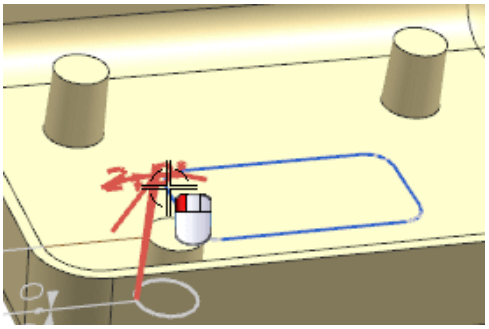
1. In the **Limits** group, from the **End Distance** parameter list, choose **Reference**.
2. In the graphics window, select the extruded boss.



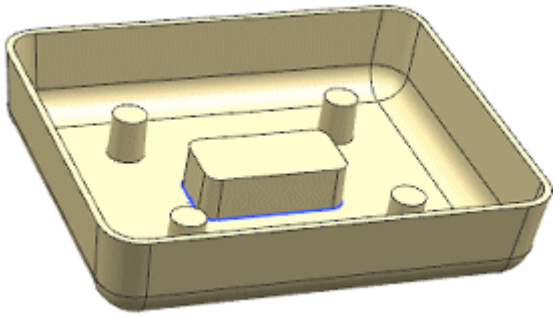
3. Select the boss height dimension.



4. Expand the **Draft** group, and from the **Draft** list, choose **From Section**.
5. In the **Draft** group, from the **Angle** parameter list, choose **Reference**.
6. In the graphics window, select the draft angle dimension.

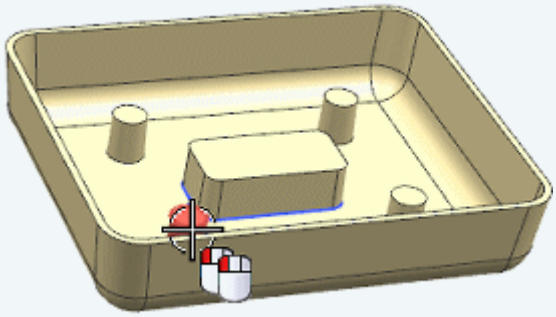


7. Click **OK**, and then press **F5** to refresh the screen.

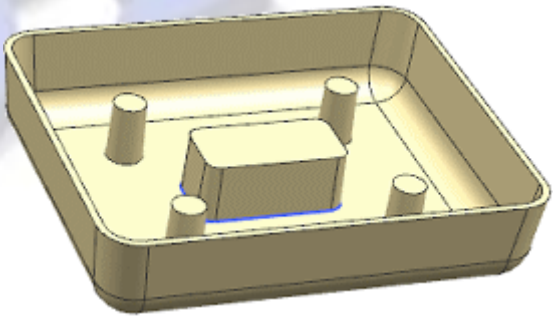


Edit the referenced feature

1. In the graphics window, double-click the extruded boss feature.



2. In the **Edit Parameters** dialog box, choose **Feature Dialog**.
3. In the **End Distance** box, type **16** and press Enter.
4. Click **OK** twice.



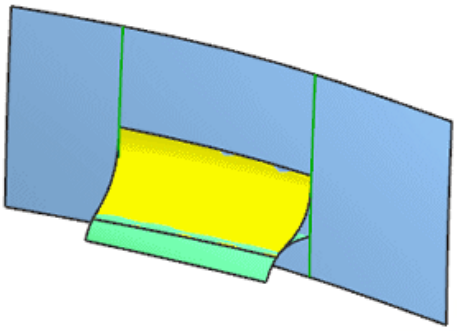
Notice both the extrusions changed.

You completed the activity.

Activity: Create a bridge surface with offsets

Estimated time to complete: 3-5 minutes

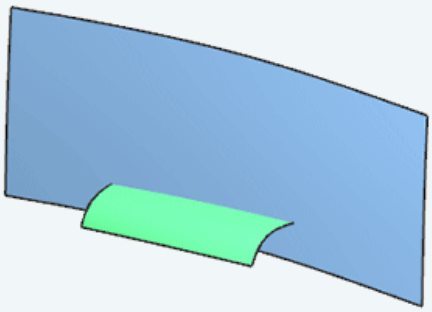
In this activity, you will create a bridge surface between offset edges and limits. You will create section curves for two of the limits.



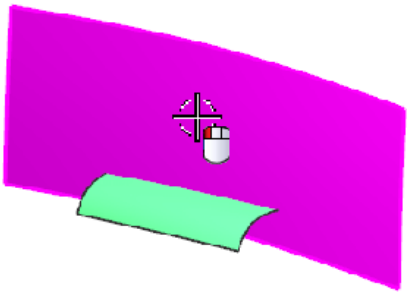
Launch the Create a bridge surface with offsets activity.

Begin creating section curves

1. Open *des18_bridge_offsets*.

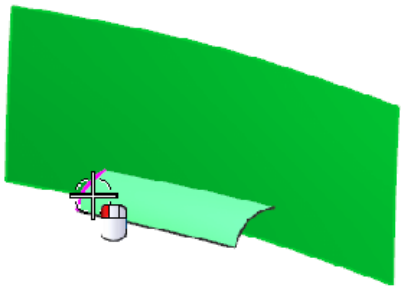


2. On the **Curve** toolbar, click **Section Curve**  .
3. In the **Section Curve** dialog box, from the **Type** list, select **Selected Planes**.
4. In the **Object to Section** group, with **Select Object** active, select the large surface.

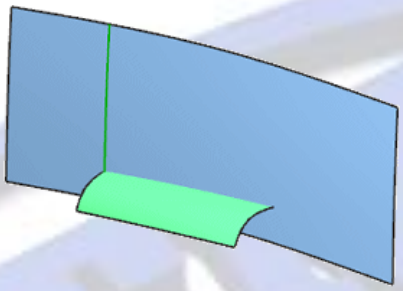


Complete the section curves

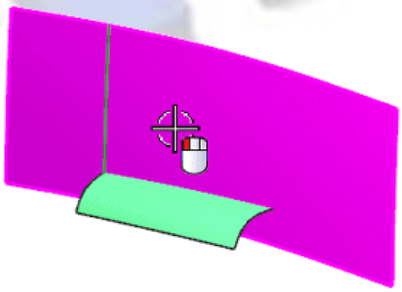
1. In the **Section Plane** group, from the **Specify Plane** options list, select **Through Object**  .
2. Select the far edge of the small surface.



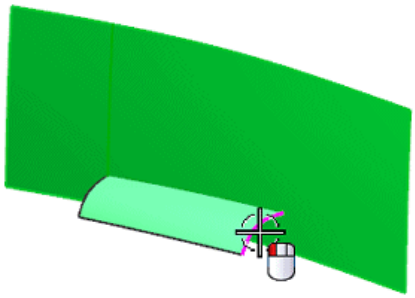
3. Click **Apply**.



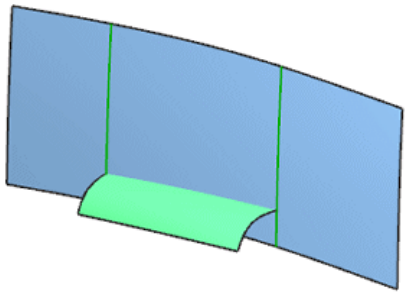
4. With **Select Object** active, select the large surface.



5. Click **Specify Plane** and select the near edge of the small surface.

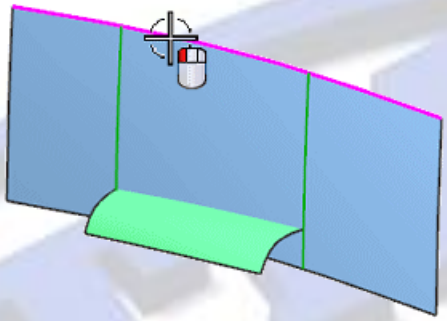


6. Click **OK**.

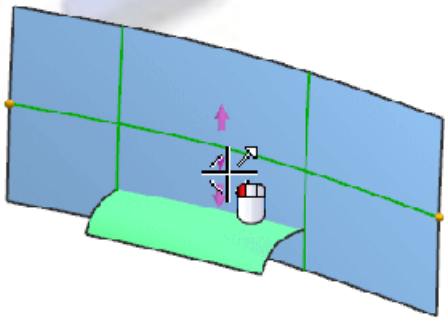


Create a bridge surface

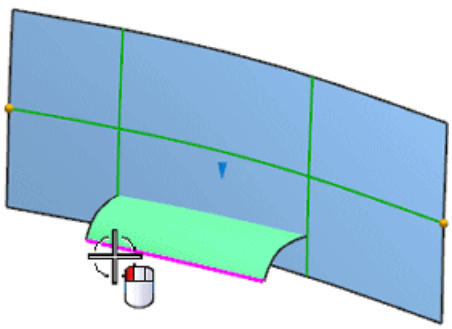
1. Choose **Insert** → **Detail Feature** → **Bridge**.
2. With **Select Edge 1** active, select the top edge of the large surface.



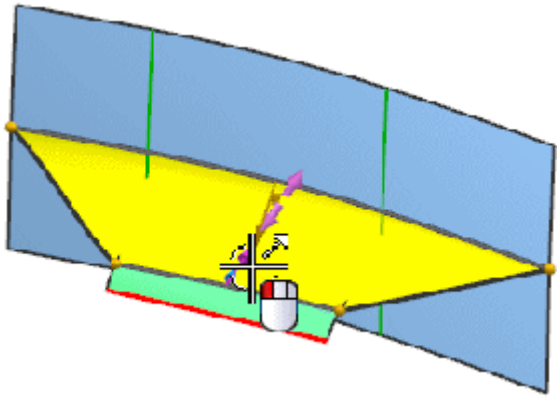
3. Drag the **% Offset** handle to **50**.



4. With **Select Edge 2** active, select the front edge of the small surface.

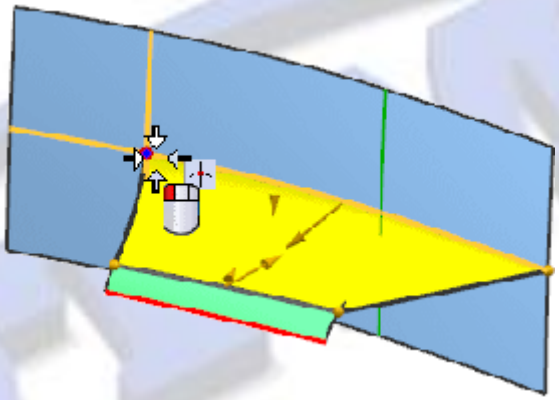


5. Drag the **% Offset** handle to **30**.

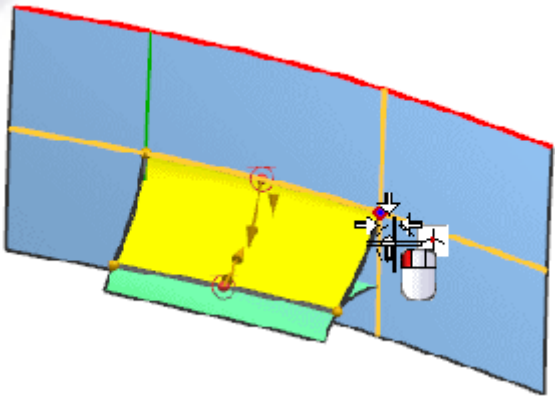


Adjust the bridge surface limits

1. In the Selection bar, make only **Intersection Point** active.
2. Drag the **% Start** handle to the right to the intersection point.

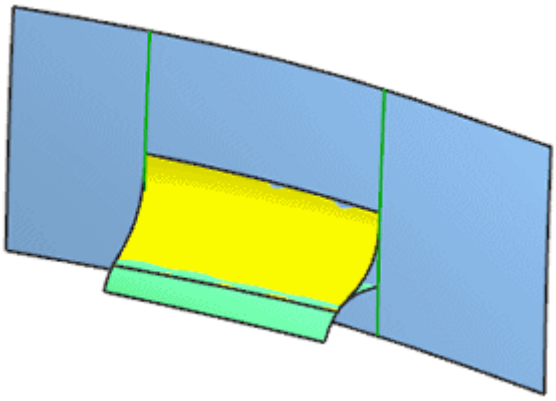


3. Drag the **% End** handle to the left to the intersection point.



4. In the **Constraints** group, under **Continuity**, set both **Edge 1** and **Edge 2** to **G2 (Curvature)**.
5. In the **Constraints** group, under **Flow Direction**, from the **Edge 1 and 2** list, select **Isoparametric**.

6. Click **OK**.

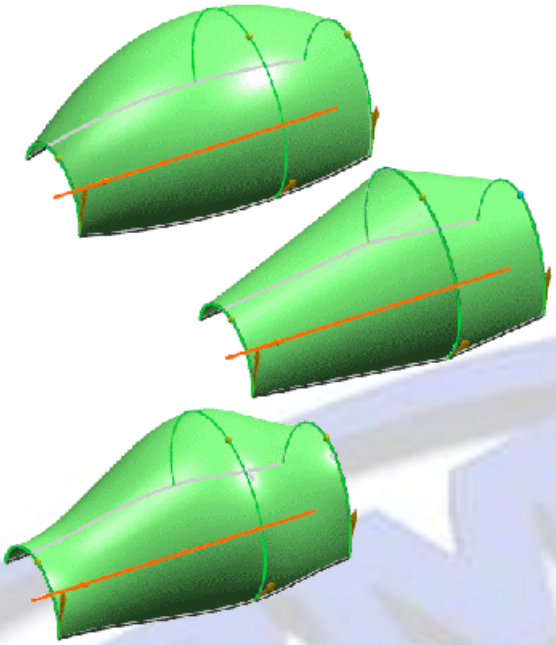


You completed the activity.

Activity: Compare swept options

Estimated time to complete: 5–8 minutes

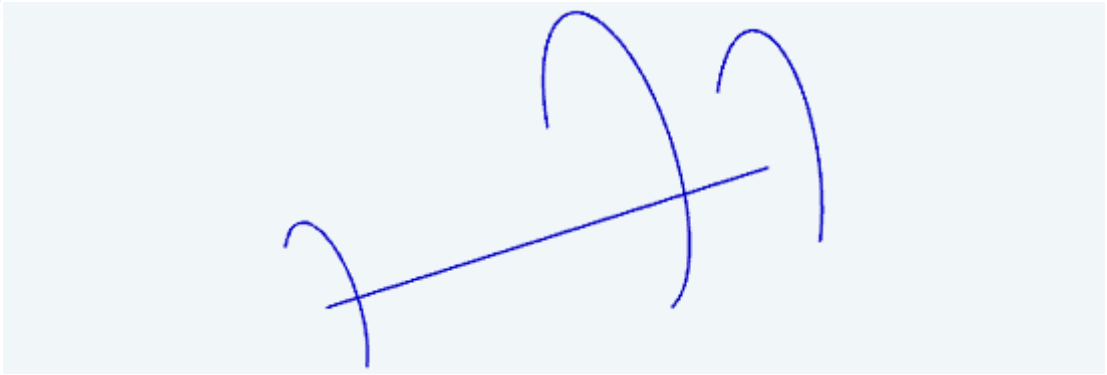
In this activity you will create swept surfaces and compare some of the options available.



Launch the Compare swept options activity.

Set the dialog box

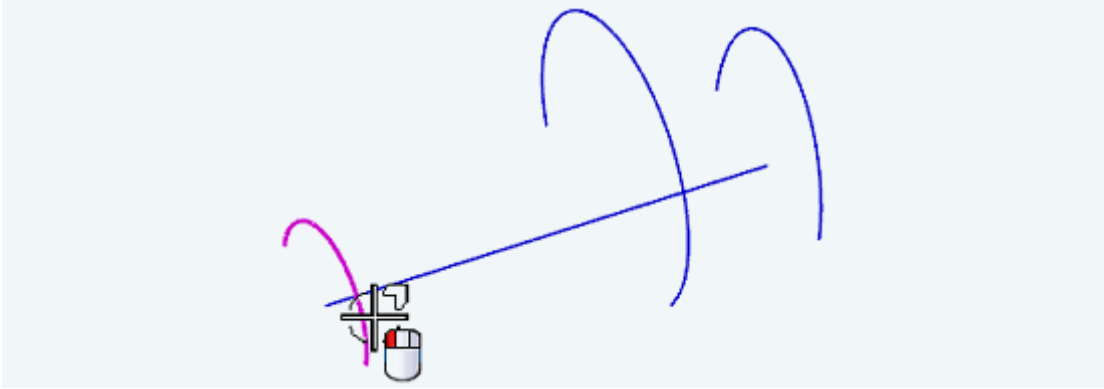
1. Open *des18_swept*.



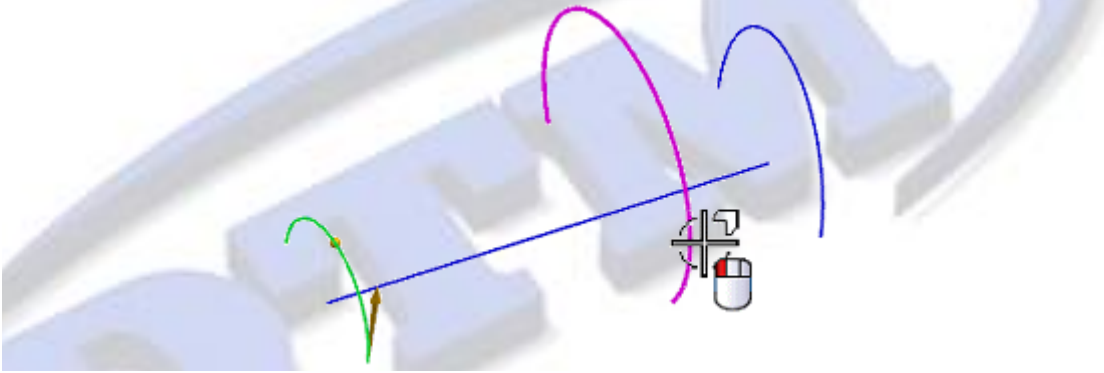
2. On the **Surface** toolbar, click **Swept**  .
3. In the **Swept** dialog box rail, click **Reset**  to reset the dialog box to the default settings.

Create the swept surface

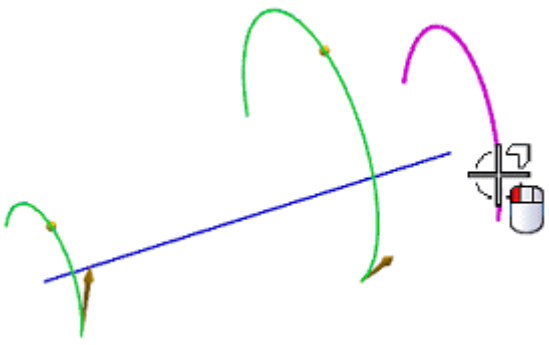
1. In the **Sections** group, with **Select Curve** active, select the arc on the left.



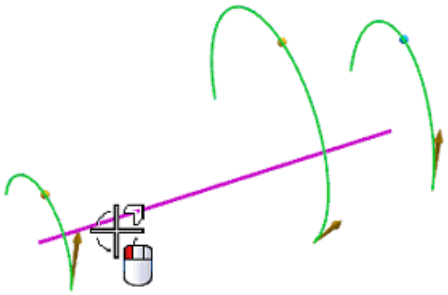
2. Click the middle mouse button or in the **Sections** group, click **Add New Set**  .
3. Select the large arc in the center.

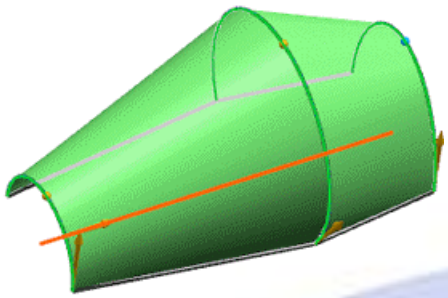


4. Click the middle mouse button or click **Add New Set**  .
5. Select the ellipse on the right.



6. Make sure all the direction arrows are pointing in the same direction.
7. Click **Select Curve**.
8. Select the line.

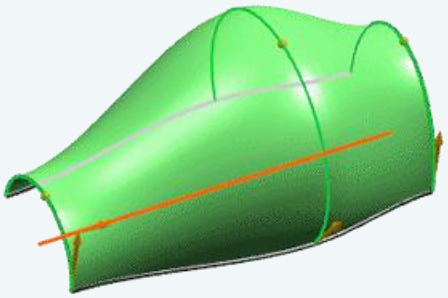




Notice the linear interpolation between the section curves.

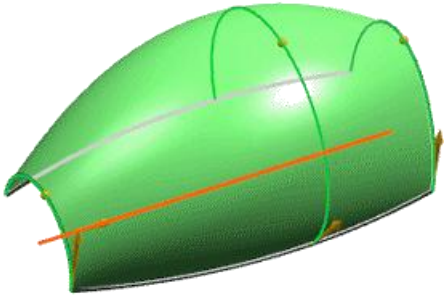
Change the interpolation

1. In the **Section Options** group, from the **Interpolation** list, select **Cubic**.



Notice the start and end of the surface are normal to the section curves.

2. From the **Interpolation** list, select **Blend**.



Notice the change in shape.

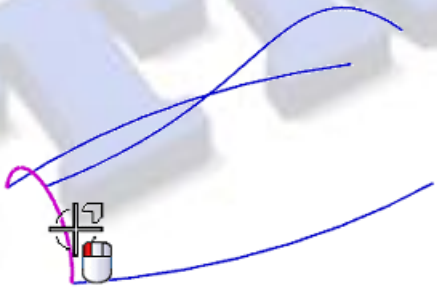
3. Click the middle mouse button twice or click **OK**.

Change the layer settings

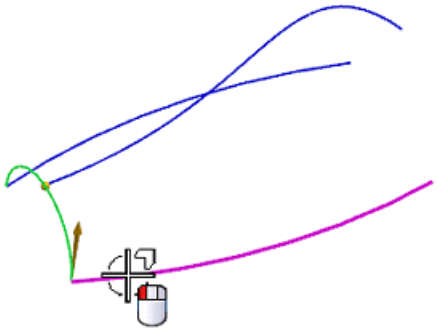
1. On the **Utility** toolbar, click **Layer Settings**  .
2. In the **Work Layer** box, type **82** and then press Enter.
3. Clear the layer **42** and **81** check boxes.
4. Select the layer **43** check box, and click **Close**.

Create another swept surface

1. In the graphics window, right click and choose **Repeat Command→Swept**, or on the **Surface** toolbar, click **Swept**  .
2. Select the arc on the left.

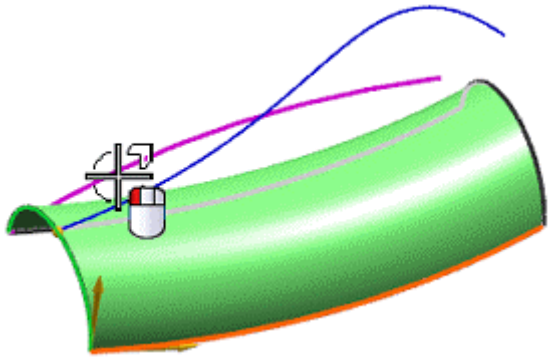


3. Click the middle mouse button twice, or in the **Guides (3 Maximum)** group, select **Select Curve**.
4. Select the curve as shown.

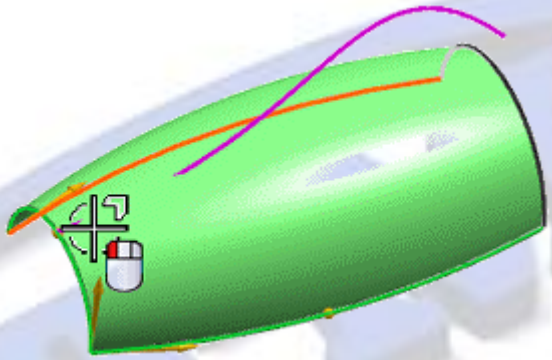


5. Click the middle mouse button or click **Add New Set**  .

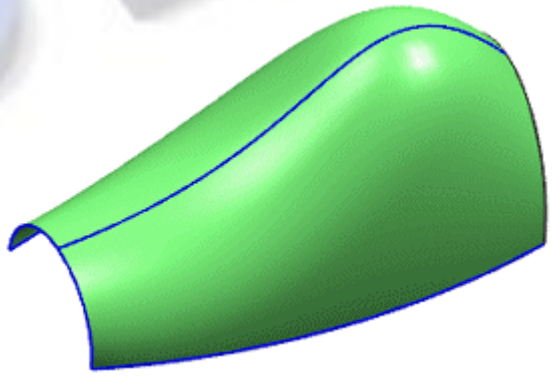
6. Select the far large arc.



7. Click the middle mouse button or, click **Add New Set**  .
8. Select the spline between the two guides.



9. Make sure all the guide direction arrows are pointing in the same direction and click the middle mouse button or click **OK**.

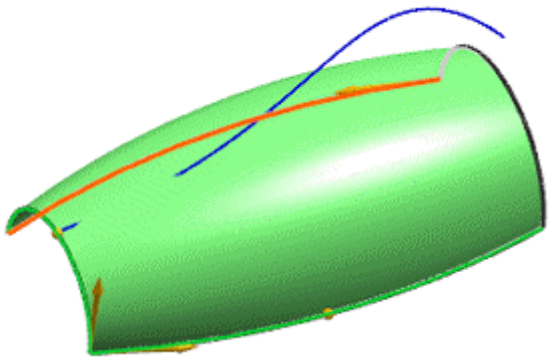


Notice the shape of the surface—it lies on all three guide curves.

Use the different scaling methods

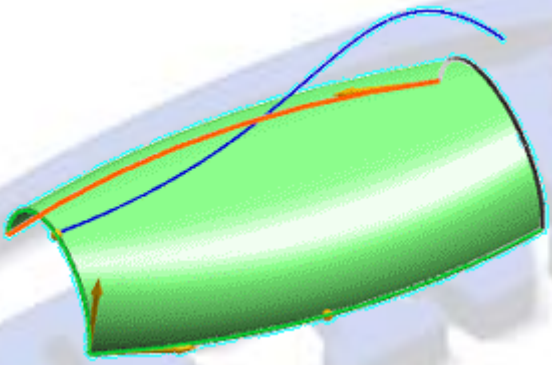
1. In the graphics window, double-click the swept surface you just created.
2. In the **Guides (3 Maximum)** group, expand the list, and select **Guide 3**.

3. Click **Remove** .



Notice the shape of the surface.

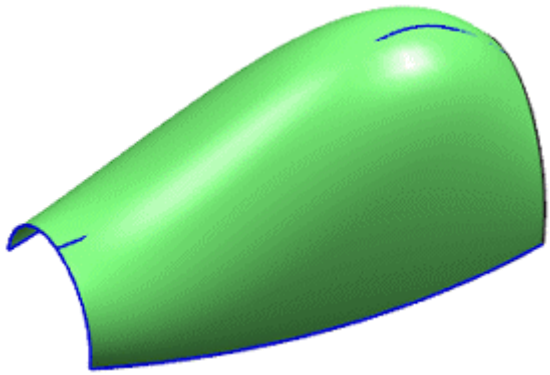
4. In the **Scaling Method** group, from the **Scaling** list, select **Lateral**.



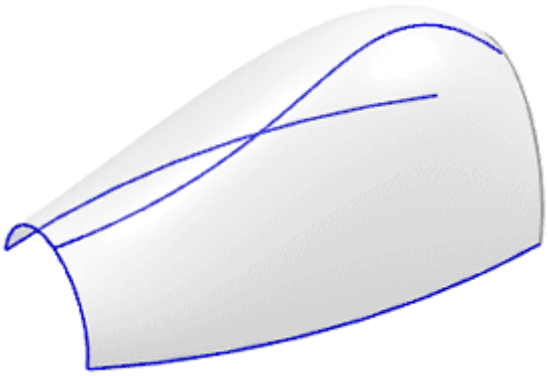
Notice the shape of the surface.

Use another curve for scaling

1. From the **Scaling** list, select **Another Curve**.
2. Select the spline, and click the middle mouse button, or click **OK**.



3. On the **View** toolbar, click **See-Thru All**  .



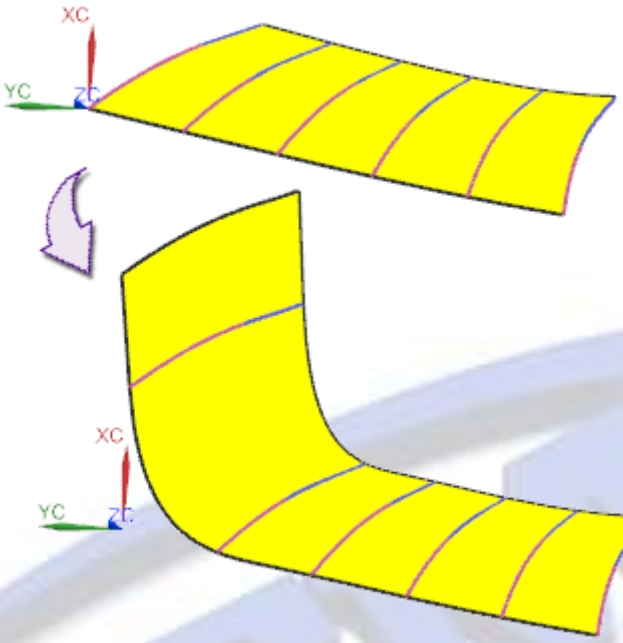
4. Compare shape of the curve selected for scaling and the shape of the surface.
5. Click **See-Thru All**  to turn it off.

You completed the activity.

Activity: Create and edit an extension surface

Estimated time to complete: 3–4 minutes

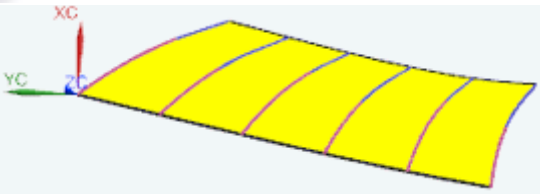
In this activity, you will create an extension surface, edit its shape, and then move one of the curves that drive the original Through Curve surface.



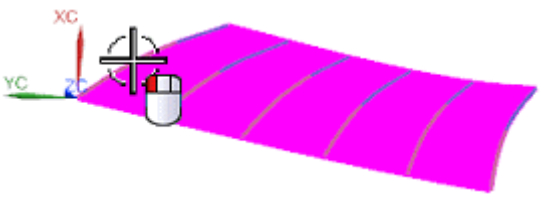
Launch the Create and edit an extension surface activity.

Create an extension surface

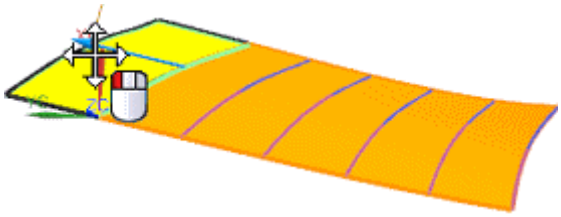
1. Open *des18_extension*.



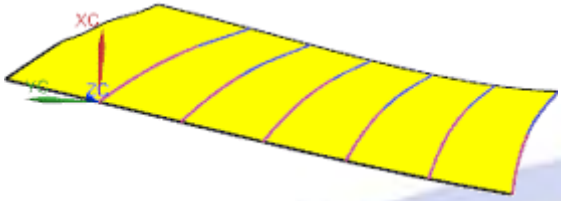
2. On the **Surface** toolbar, from the **Flange Surface** drop-down list, click **Extension Surface**  .
3. Make sure **Edge** is selected from the **Type** list, and with **Select Edge** active, select the left edge of the surface.



4. In the **Extension** group, from the **Method** list, choose **Circular**.
5. Drag the **Length** handle to **8**.

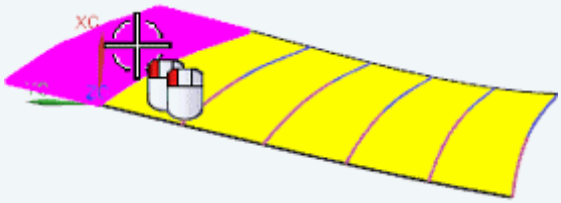


6. Click the middle mouse button, or click **OK**.

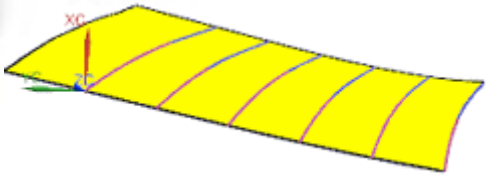


Edit the extension

1. Double-click the extension you just created.

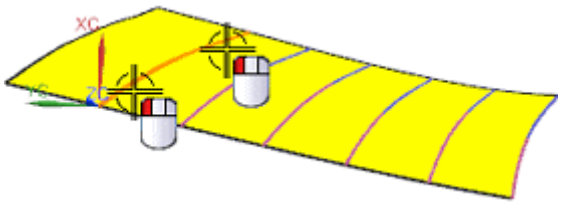


2. From the **Method** list, choose **Tangential**.
3. Drag this **Distance** handle to **7**, and click **OK**.



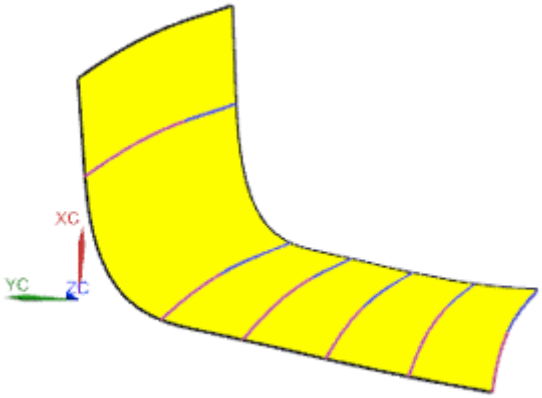
Move one of the parent curves of the through curve surface

1. On the menu bar, click **Edit→Move Object**.
2. Select the two splines at the left end of the original Through Curves surface.



3. From the **Motion** list, choose **Distance**.
4. From the **Specify Vector** drop-down list, choose **XC**.

5. Type a **Distance** value of **10**, and then press Enter.
6. Click the middle mouse button, or click **OK**.

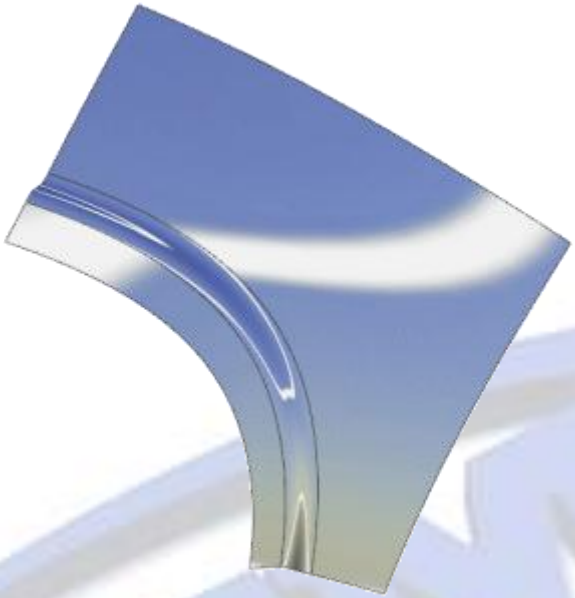


You completed the activity.

Activity: Rebuild a law extension surface

Estimated time to complete: 2–3 minutes

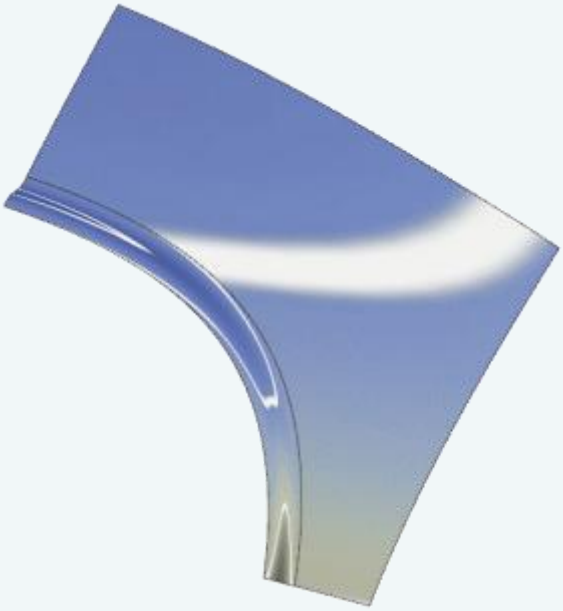
In this activity, you will use **Law Extension** to create an extension from a base sheet and examine the updated Rebuild options for the law extension surface.



Launch the Rebuild a law extension surface activity.

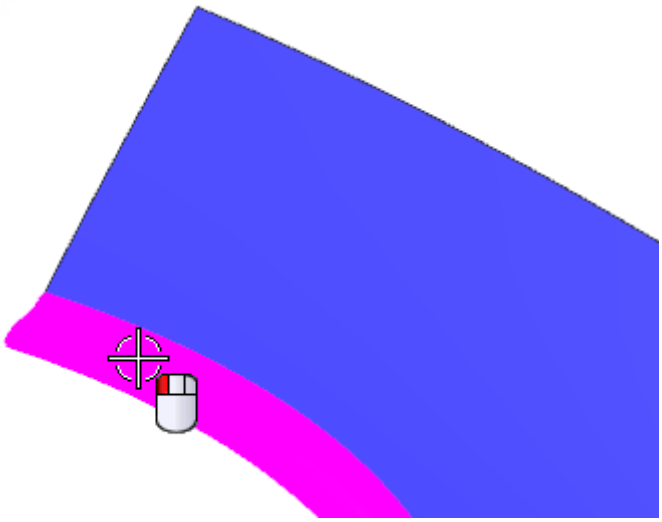
Open the part and show the poles of the surface

1. Open *des18_keep_parameterization_3*.



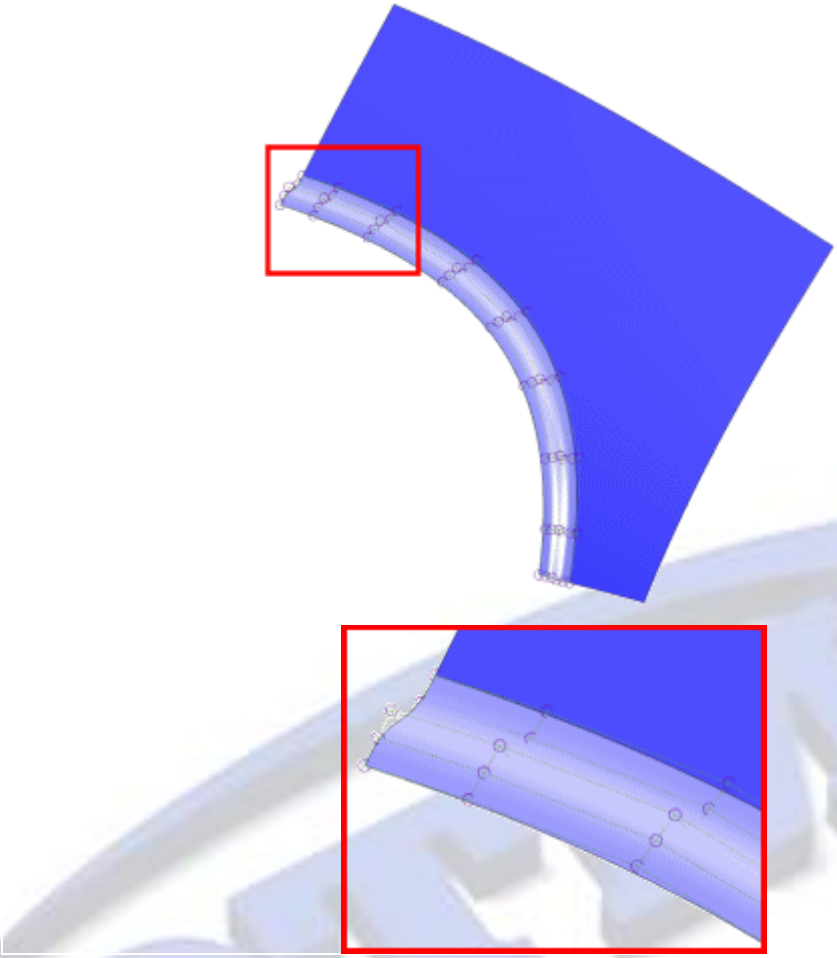
2. With the cursor in a blank area of the graphics window, right-click and hold until the radial menu appears, then hover over **Shaded with Edges** and release.

3. If the **Analyze Shape** toolbar is not available, position the cursor in an empty area of a toolbar and right-click and select **Analyze Shape** to display the toolbar.
4. Select the curved surface as shown.



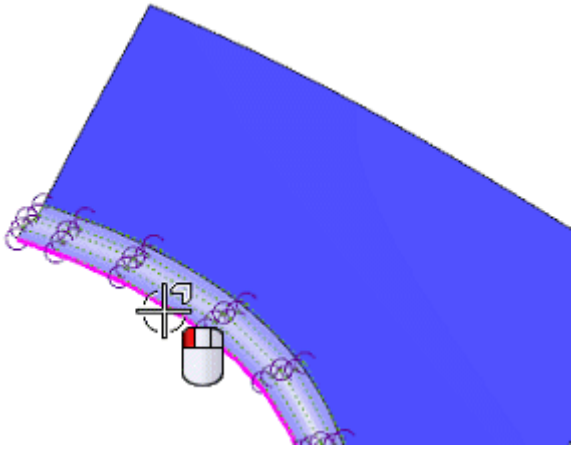
5. On the **Analyze Shape** toolbar, click **Show Poles** .

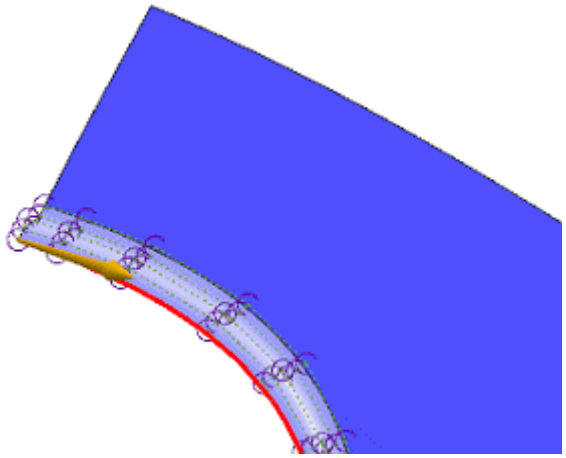
6. Shift-select the surface to deselect it.



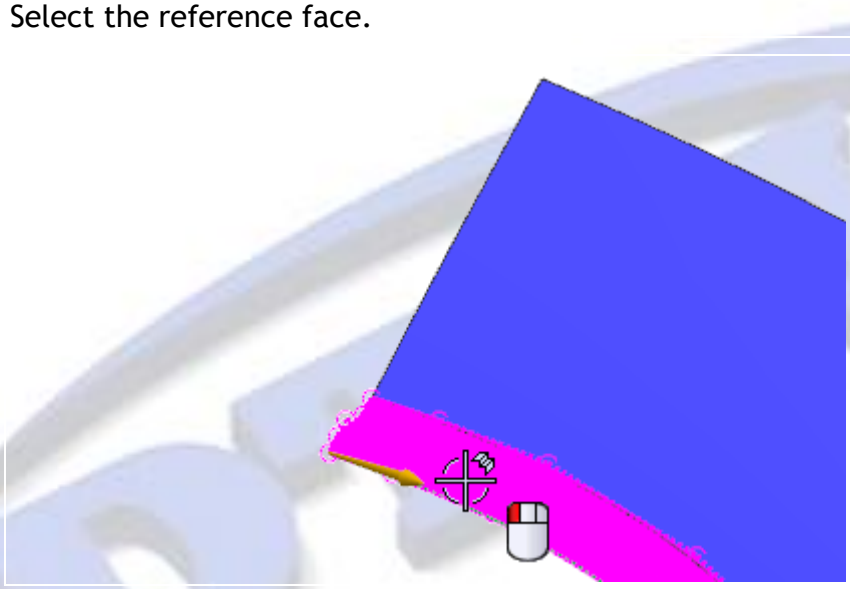
Create an law extension surface

1. On the **Surface** toolbar, choose **Law Extension**  .
2. Select the base profile curve.

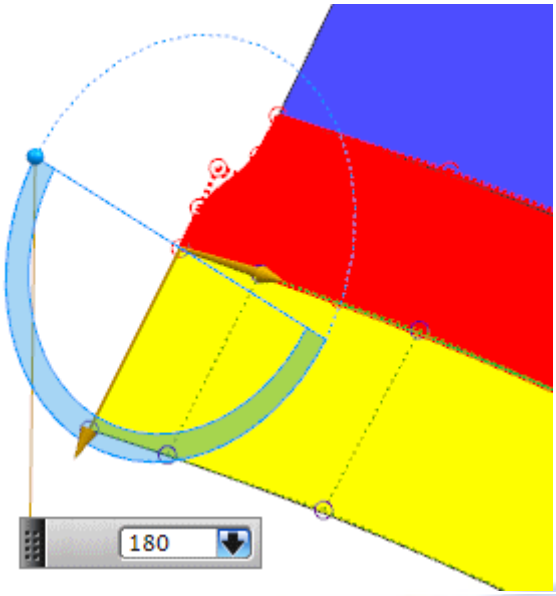




3. Click the middle mouse button.
4. Select the reference face.

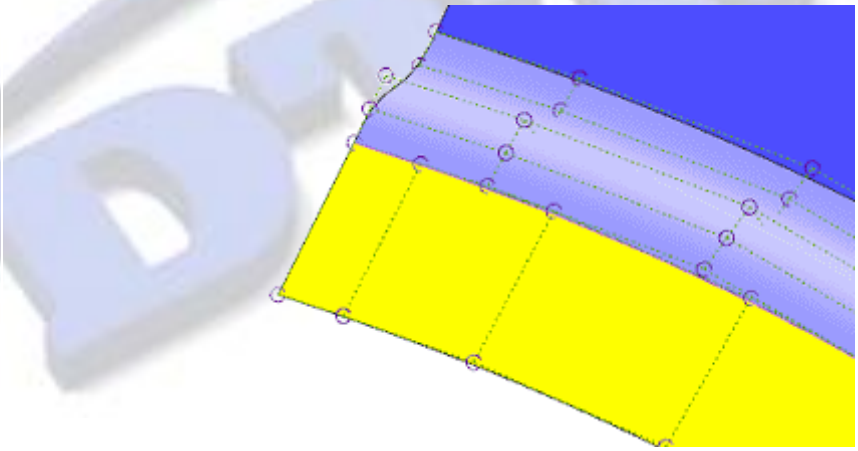


5. In the **Length Law** group:
    - Make sure **Law Type** is set to **Constant**.
    - In the **Value** box, key in **50** then press Enter.
6. In the **Angle Law** group:
    - Make sure **Law Type** is set to **Constant**.
    - In the **Value** box, key in **180** then press Enter.



7. On the **Analyze Shape** toolbar, click **Show Poles**  .
8. In the **Law Extension** dialog box, expand the **Preview** group and click **Show Result**  .

   Notice that the poles of extension surface do not match with the poles of the parent surface. This is because there are no rebuild settings in effect.



9. Click **Undo Result**  .

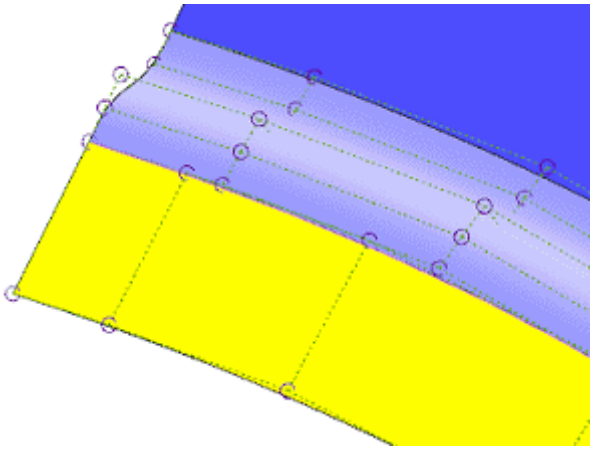
Check Rebuild settings

1. Expand the **Settings** group.

   Notice that the **Merge Faces if Possible** check box is selected and that **Rebuild** is set to **None**.

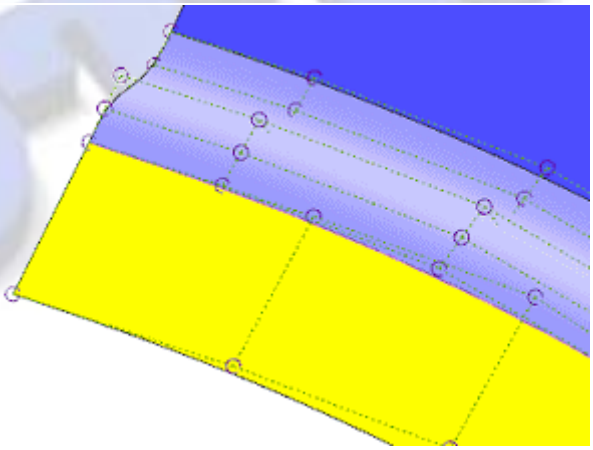
2. From the **Rebuild** list, select **Degree and Tolerance**.

3. Click **Show Result** .



Degree and Tolerance lets you specify the maximum degree and the tolerance to control the parameterization of the output surfaces.

4. Click **Undo Result** .
5. From the **Rebuild** list, select **Auto Fit**.
6. Notice the **Alerts** message.
7. Click **Show Result** .



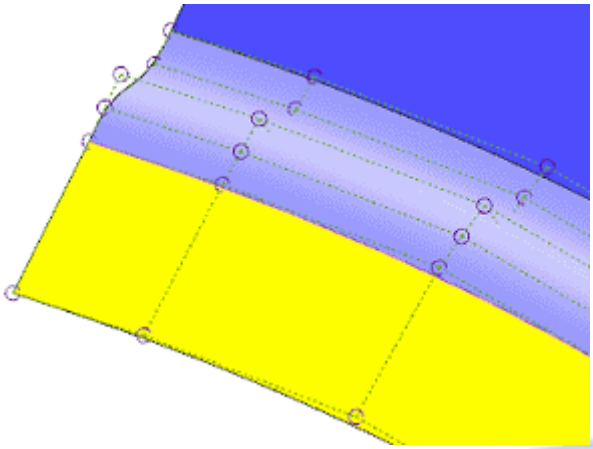
Auto Fit creates the smoothest possible surface within the specified tolerances.

You can specify **Maximum Degree** and **Maximum Segments** to rebuild the surface without adding segments up to the maximum degree.

8. Click **Undo Result** .
9. Clear the **Merge Faces if Possible** check box.

The Rebuild option of Keep Parameterization is not available unless the Merge Faces if Possible check box is selected.

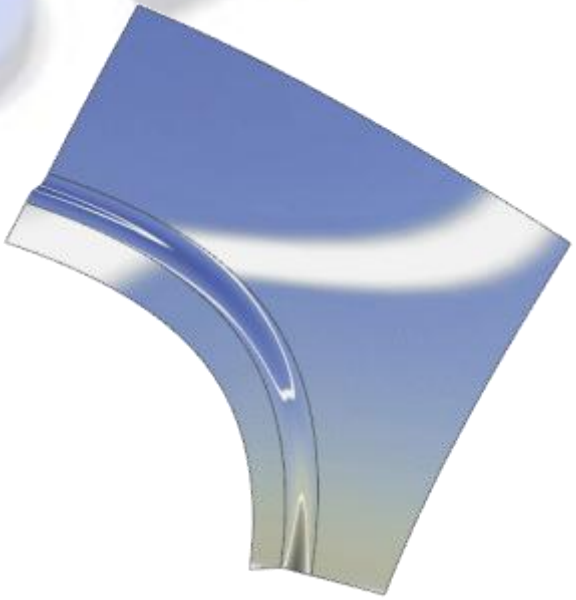
10. From the **Rebuild** list, select **Keep Parameterization**.
11. Click **Show Result**  .



Keep Parameterization inherits the degree, segments, pole structure and knot structure from the input surfaces and applies it to the extension surface.

Keeping the parameterization between input curves and derived surface allows working in both mesh and sweep based workflows and a pole editing based workflow.

12. Click **OK** to create the law extension.
13. (Optional) On the **Analyze Shape** toolbar, use **Show Poles**  to turn off the pole display and **Face Analysis - Reflection**  to add the law extension surface to the reflection display.



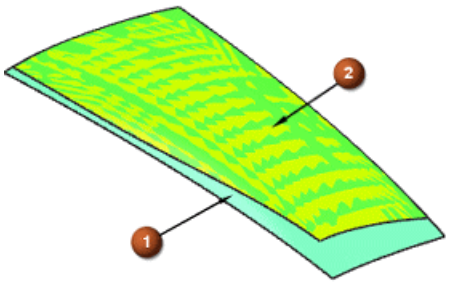
14. Close the part.

You completed the activity.

Activity: Compare Delete Edge with Untrim

Estimated time to complete: 2–3 minutes

In this activity, you will use both the **Untrim** and the **Delete Edge** commands to remove edges of a surface. You wilt then compare the results.



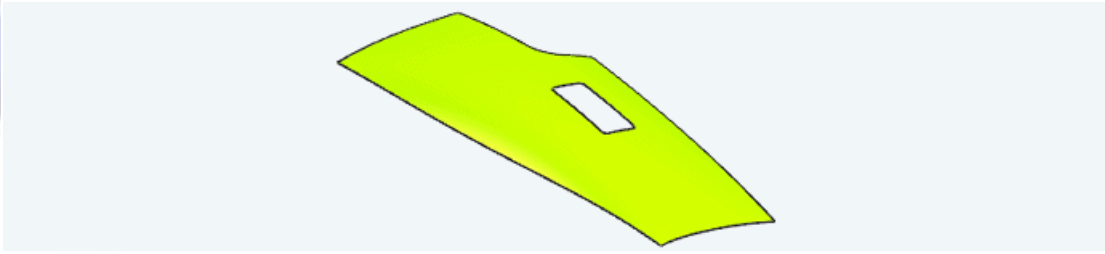
- 1 Untrimmed
- 2 Edges deleted

Launch the Compare Delete Edge with Untrim activity.

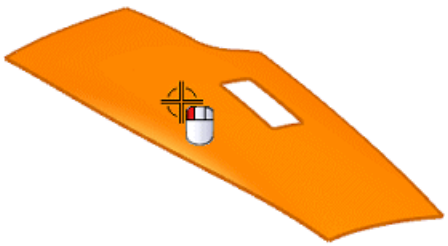
Untrim the surface

The objective is to remove some of the edges of a surface. You will use both **Untrim** and **Delete Edge** and compare the results.

1. Open *des18_delete_edge*.

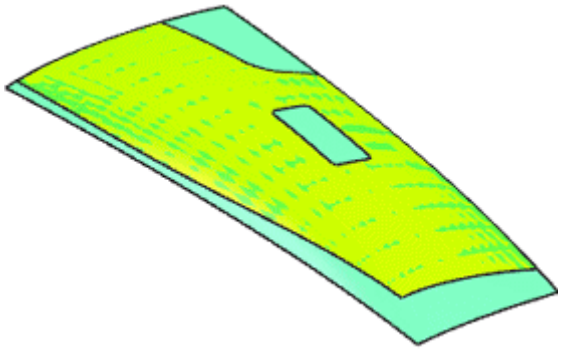


2. Choose **Insert→Trim→Untrim**.
3. Select the face.



4. In the **Settings** group, clear the **Hide Original** check box.

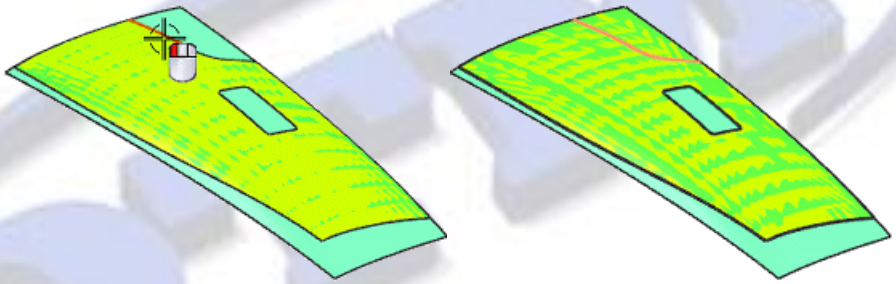
5. Click the middle mouse button, or click **OK**



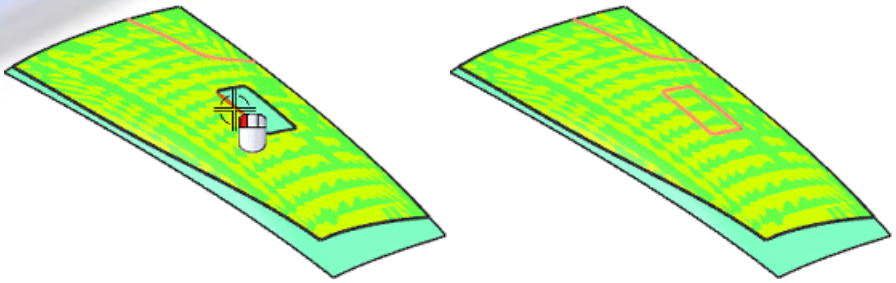
Notice most of the edges changed.

Delete some edges of the original surface

1. Choose **Insert→Trim→Delete Edge**.
2. From the **Curve Rule** list, choose **Tangent Curves**.
3. Select any curve in the notch area.

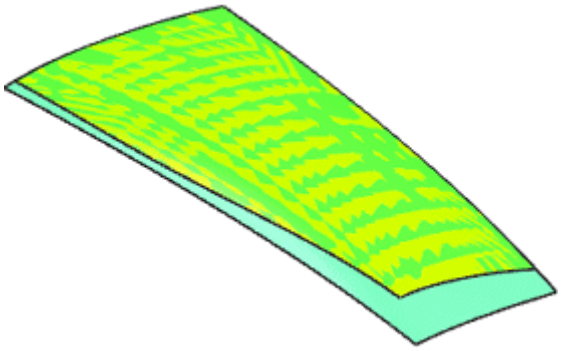


4. Select any curve at the edge of the hole.



5. Click the middle mouse button, or click **OK**.

6. Compare the untrimmed surface with surface with the removed edges.



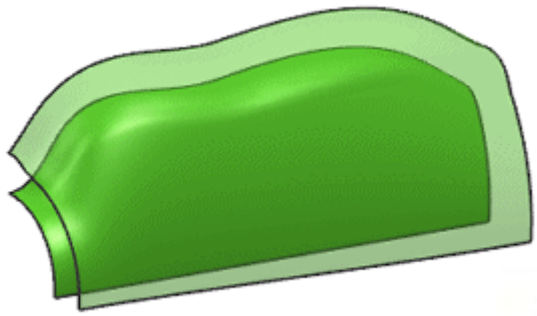
Notice that only the selected edges were untrimmed.

You completed the activity.

Activity: Create a variable offset surface

Estimated time to complete: 4–8 minutes

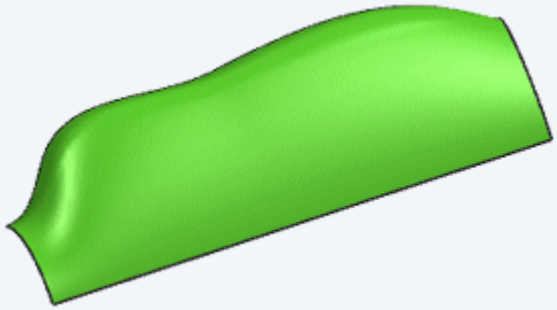
In this activity, you will create a variable offset surface with different offset locations and values.



Launch the Create a variable offset surface activity.

Open and set the object preferences

1. Open *des18_variable_offset*.



This is a curve mesh surface. All the generating sketches, curves, planes and axes are hidden.

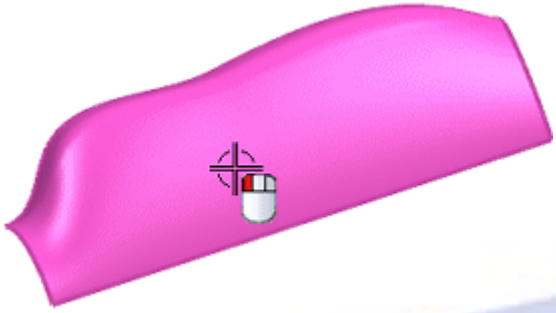
2. Choose **Preferences→Object**.
3. In the **Object Preferences** dialog box, on the **General** page, set the **Translucency** slider to approximately **50** percent.

   This will make it easier to compare the offset surface that you create with the original.
4. Click the middle mouse button, or click **OK**.

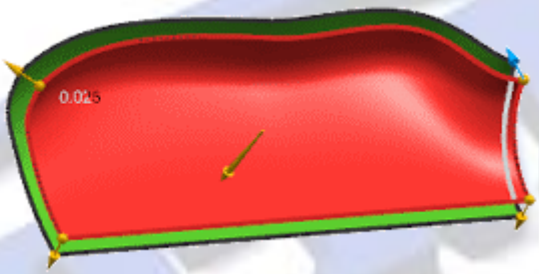
Select the base surface and set some the offset parameters

1. Choose **Insert→Offset/Scale→Variable Offset**  .

2. In the **Variable Offset** dialog box rail, click **Reset** to reset the dialog box to its default settings..
3. In the **Offset** group, select the **Apply to All** check box.
4. In the **Offset at A** entry box, type **4**, and then press Enter.
5. Select the face of the surface.

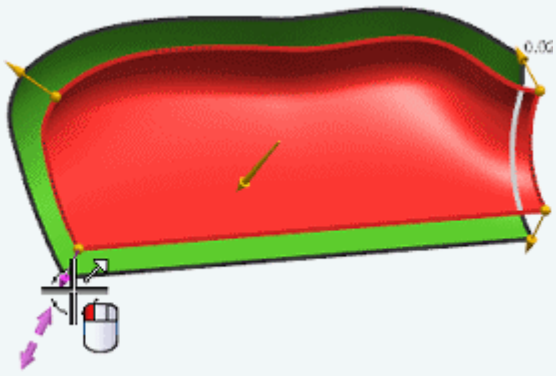


6. Rotate the view so you can better see the surfaces.



Specify the offset locations and values

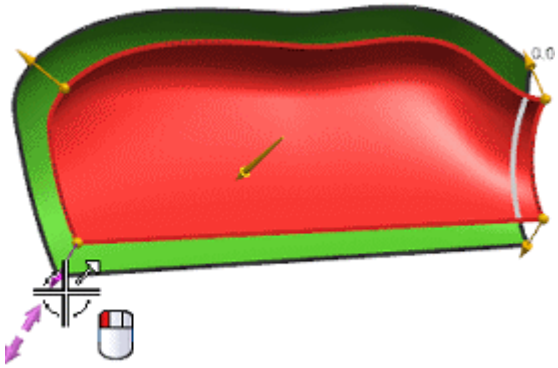
1. Drag the offset handle shown below to **7**.



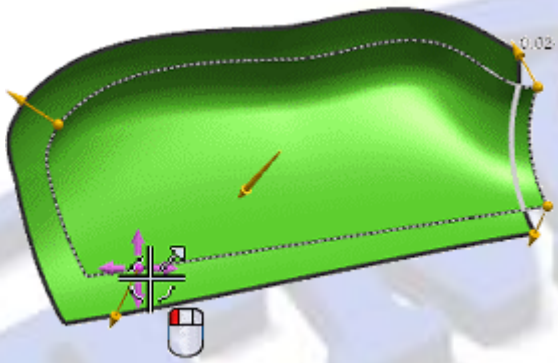
Notice the **0.024** value in black at one corner. This is the location and value of the maximum deviation of the fitted surface from the true offset surface. If the value exceeds the Modeling Distance tolerance, it is displayed in red.

2. Clear the **Apply to All** check box.

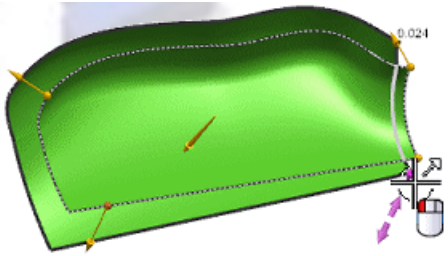
3. Drag the offset handle shown below to **12**.



4. On the Selection bar, make sure only **Point on Curve**  is active.
5. Drag the origin handle along the edge, approximately as shown below.

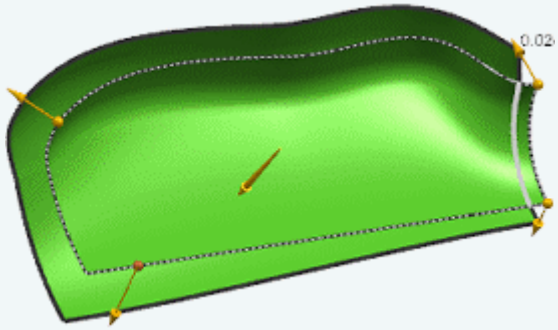


6. Drag the offset handle shown below to **5**.



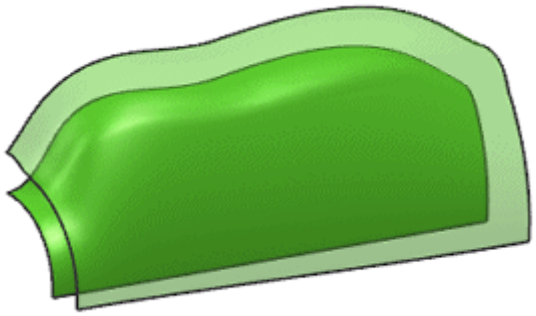
Change a parameter and complete the offset

1. In the **Settings** group, from the **Method** list, select **Cubic**.



Notice the shape of the edges of the previewed surface.

2. Click the middle mouse button, or click **OK**.
3. Rotate and examine the two surfaces.

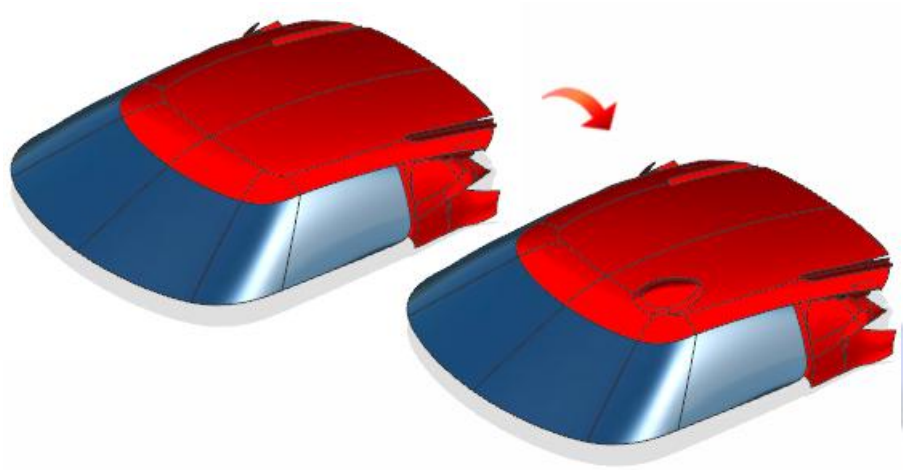


You completed the activity.

Activity: Delete faces

Estimated time to complete: 3–5 minutes

In this activity, you will delete a face without trimming the surrounding faces so that a new face can be sewn in its place.

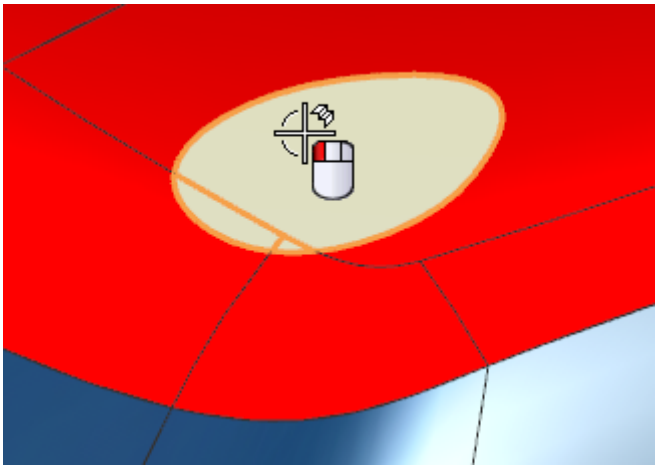


Launch the Delete faces activity.

Delete faces

You need to create a bump in the roof of this race car.

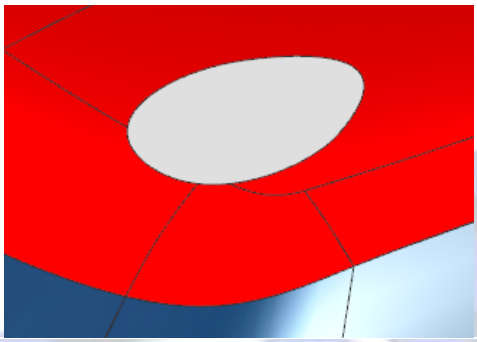
1. Open *syn2_delete_faces*.
2. On the **Synchronous Modeling** toolbar, click **Delete Face**  .
3. In the **Delete Face** dialog box, in the **Settings** group, select the **Heal** check box.
4. Select the three faces as shown, and click **OK**.



With **Heal** selected NX attempts to close the hole by extending the surrounding faces. In this example NX is unable to extend these faces. For this activity you need to delete the faces without healing the surrounding faces because you want to use a new face in its place.

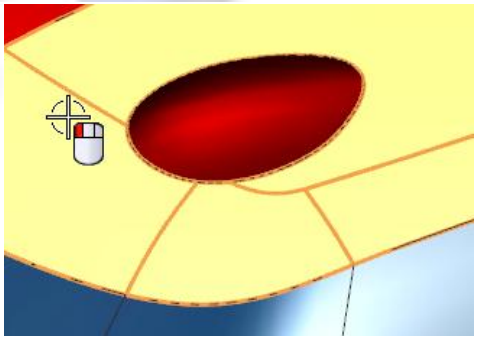
5. In the **Delete Face** message box, click **OK**.
6. In the **Delete Face** dialog box, in the **Settings** group, clear the **Heal** check box, and click **OK**.

   The three faces have been deleted and the surrounding faces have not changed. A new face can be created and sewn into the existing body.

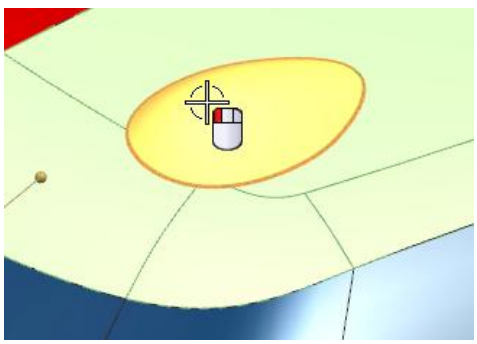


Sew in new faces

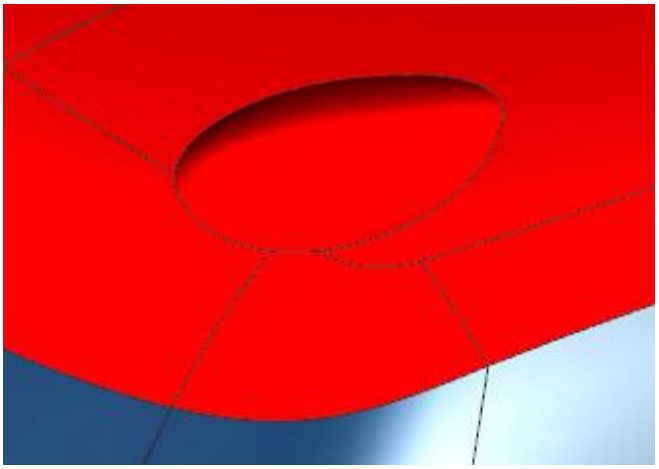
1. In the **Part Navigator**, right-click **Through Curves (63)** and select **Show**.
2. On the **Feature** toolbar, from the **Combine Drop-down** list, select **Sew**  .
3. Select the face as shown.



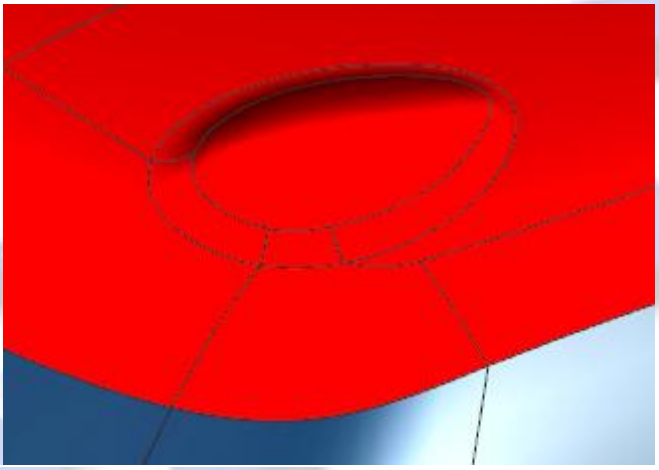
4. Select the face as shown, and click **OK**.



The bump is now part of the race car roof.



Blends and other changes can be made to the new body.



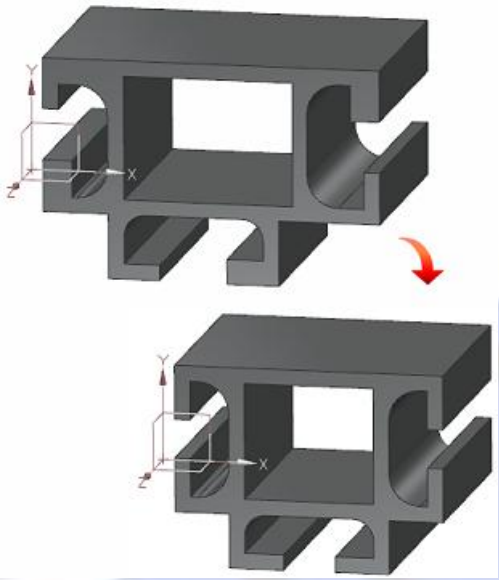
5. Close the part.

You completed the activity.

Activity: Edit Cross Section

Estimated time to complete: 8–12 minutes

In this activity, you will modify an extruded aluminum part that has no feature history. You will create a cross section through the part and modify the cross section in a sketch.

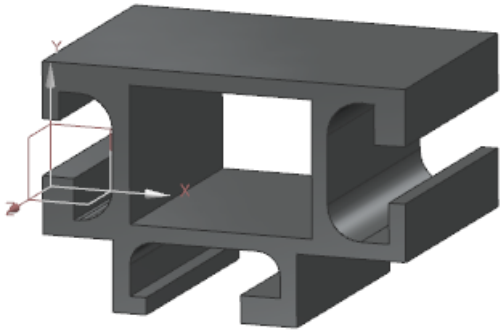


Launch the Edit Cross Section activity.

Define a section

First you will edit the part's cross section in History-Free Mode. Later you will edit the part's cross section in History Mode.

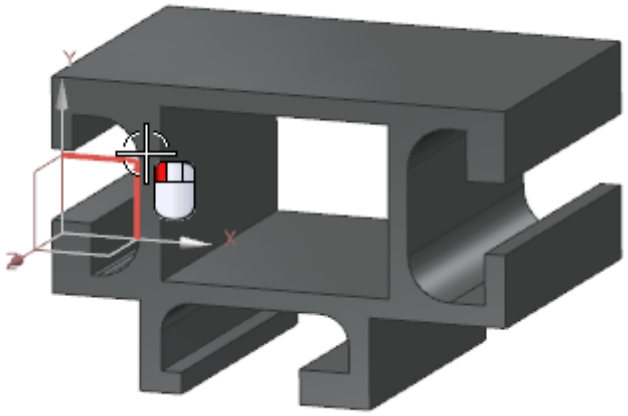
1. Open *syn2_edit_cross_section*.



2. On the **Synchronous Modeling** toolbar, make sure **History-Free Mode** is selected.

3. On the **Synchronous Modeling** toolbar, click **Edit Section** .

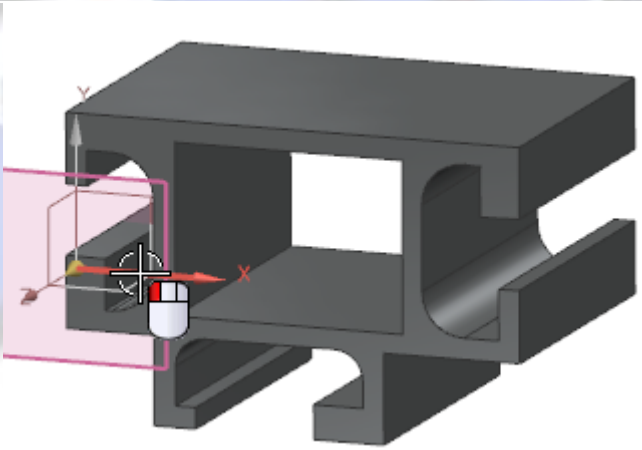
4. In the **Cross Section Edit** dialog box, from the **Type** list, select **On Plane**.
5. In the **Sketch Plane** group, from the **Plane Method** list, select **Create Plane**.
6. Select the X-Y **Datum Plane** as shown.



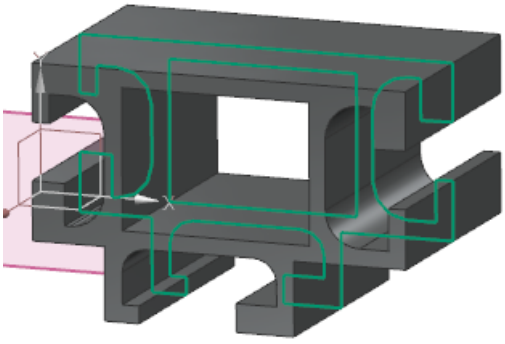
7. To advance to defining the horizontal reference of the sketch, click the middle mouse button.

   Notice that **Select Reference** is now highlighted in the dialog box.

8. Select the X **Datum Axis** as shown.

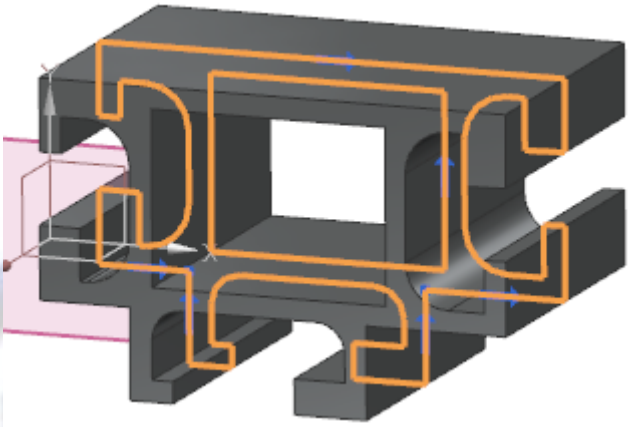


9. To accept the sketch plane and enter the Sketch task environment, click the middle mouse button.



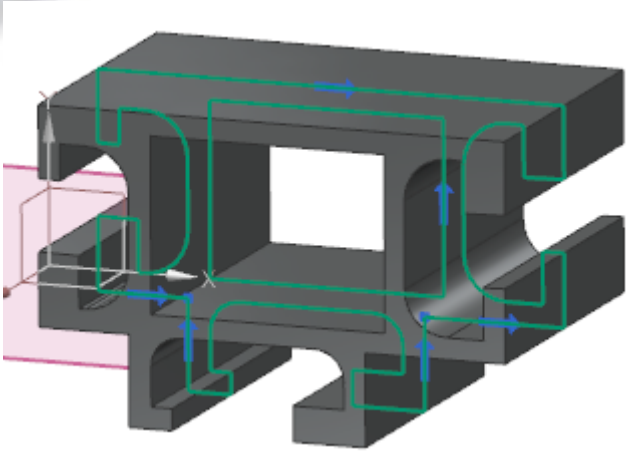
Edit the sketch

1. On the **Sketch Tools** toolbar, click **Auto Constrain**  .
2. In the **Auto Constrain** dialog box, in the **Constraints to Apply** group, select the following check boxes:
   - **Horizontal**
   - **Vertical**
   - **Collinear**
   - **Coincident**
3. To select all the curves in the sketch, press Ctrl+A.



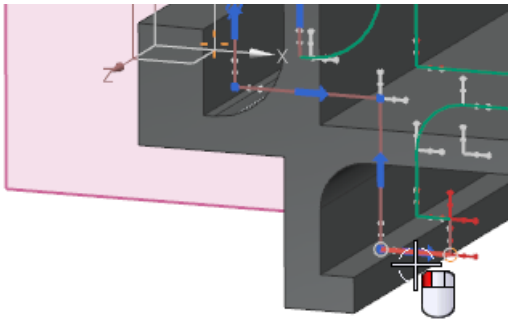
4. To create the constraints, click the middle mouse button.

On the **Sketch Tools** toolbar, if **Show All Constraints**  is turned on, you will see the geometric constraints that have been created.

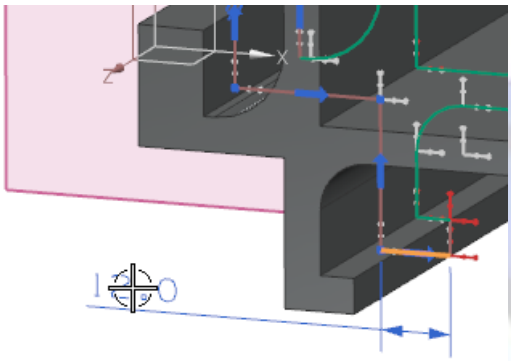


5. On the **Sketch Tools** toolbar, click **Inferred Dimensions**  .

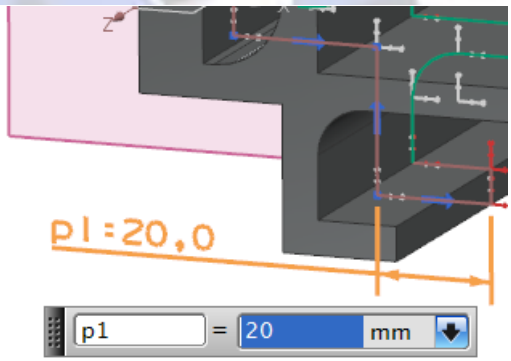
6. Select the horizontal line as shown.



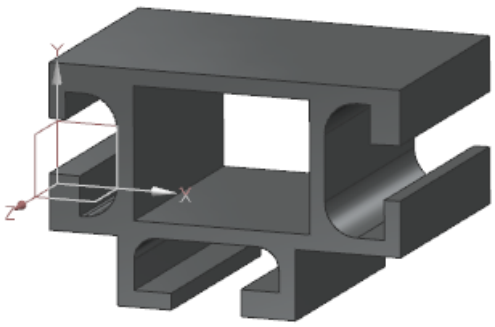
7. Click a screen position to locate the dimension origin.



8. In the on-screen input box, type **20** and press Enter.



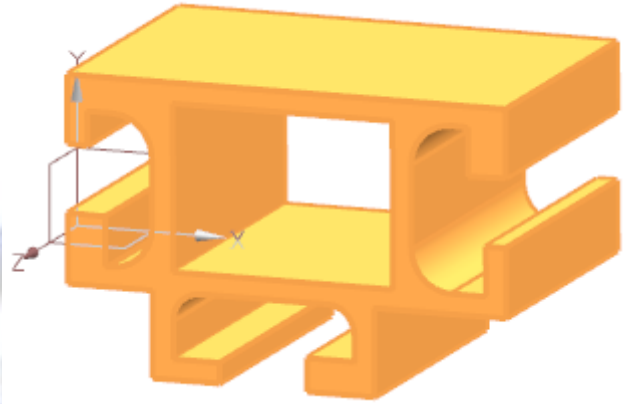
9. Right-click in the background of the graphics window and choose **Finish Sketch**  .



Because the solid body has been modified in **History-Free Mode**, the sketch used to cross section the part is deleted. If you want NX to maintain the sketch and the constraints, use the **Edit Cross Section** command in **History Mode**.

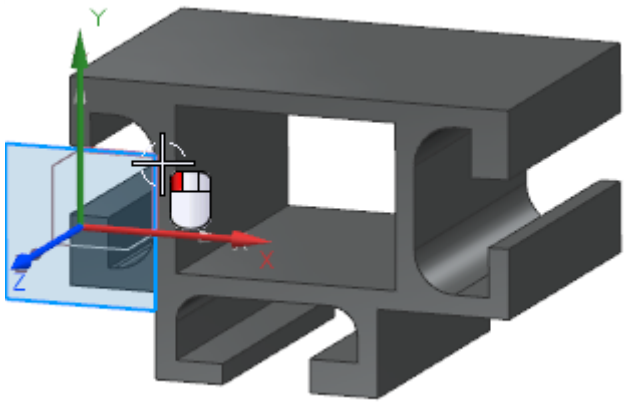
Edit the part in History Mode

1. On the **Synchronous Modeling** toolbar, click **History Mode**  .
2. In the **Modeling Mode** message box, click **Yes**.
3. On the **Synchronous Modeling** toolbar, click **Edit Cross Section**  .
4. To select all the faces, press Ctrl+A .



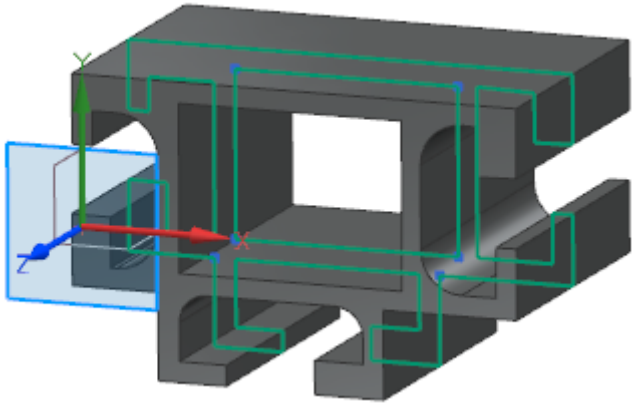
5. To advance to defining the cross section plane, click the middle mouse button.

   Notice that **Select Plane** is now highlighted in the dialog box.
6. Select the X-Y **Datum Plane** as shown.



7. In the **Edit Cross Section** dialog box, in the **Preview** group, clear the **Preview** check box.

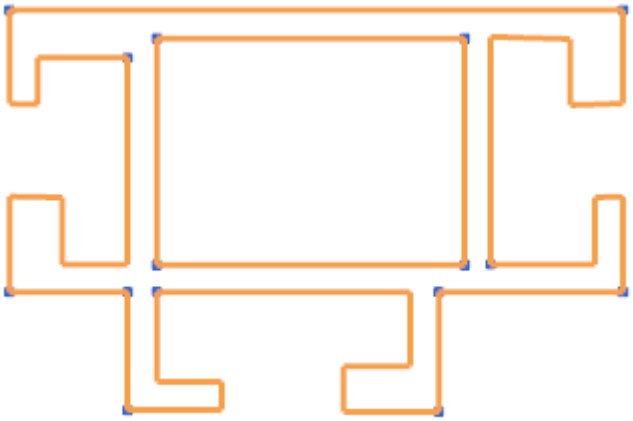
8. In the **Section Curve** group, click **Section**  to enter the Sketch task environment.



9. To make the sketch easier to visualize, in the **Part Navigator**, right-click **Body (1)**and choose **Hide**.

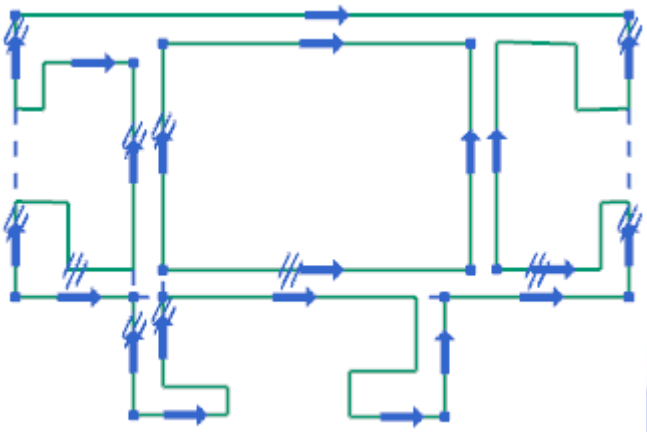
Edit the sketch

1. On the **Sketch Tools** toolbar, click **Auto Constrain**  .
2. In the **Auto Constrain** dialog box, in the **Constraints to Apply** group, select the following check boxes:
   - **Horizontal**
   - **Vertical**
   - **Collinear**
   - **Coincident**
3. To select all the lines us the sketch, press Ctrl+A.

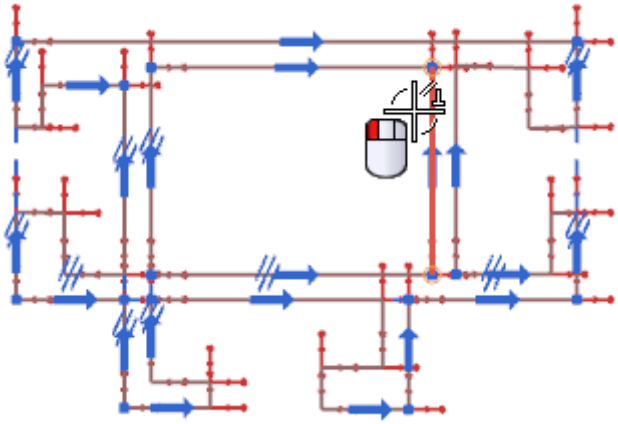


4. To create the constraints, click the middle mouse button.

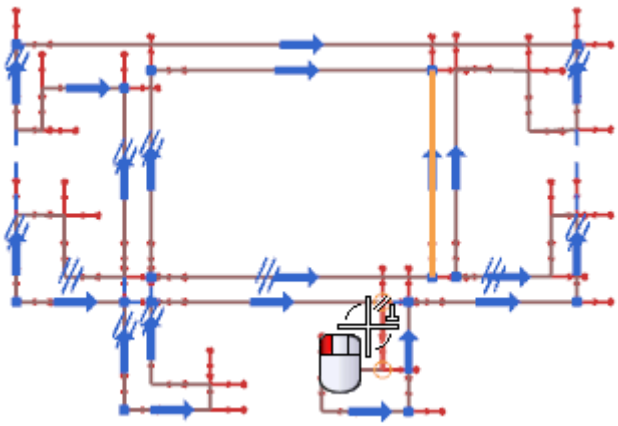
   On the **Sketch Tools** toolbar, If **Show All Constrains**  is turned on, you will see the geometric constraints that have been created.



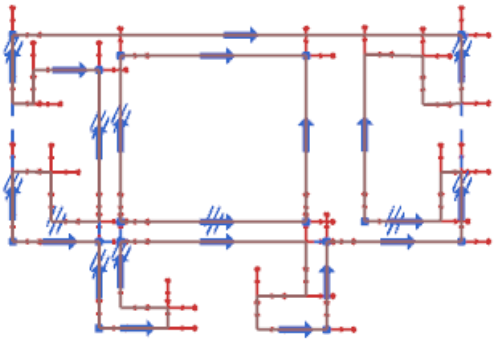
5. On the **Sketch Tools** toolbar, click **Constraints**  .
6. Select the vertical line as shown.



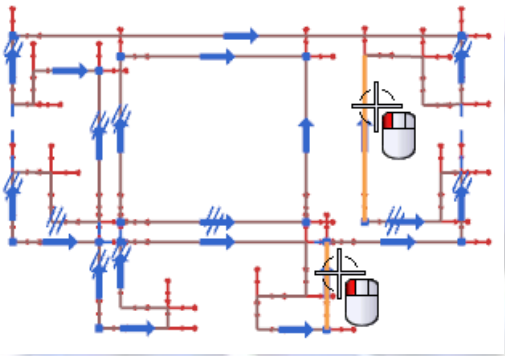
7. Select the vertical line as shown.

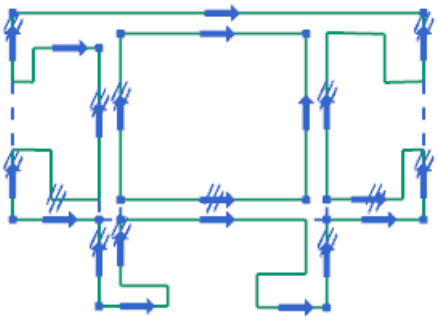


8. Right-click one of the selected lines and choose **Collinear**  .

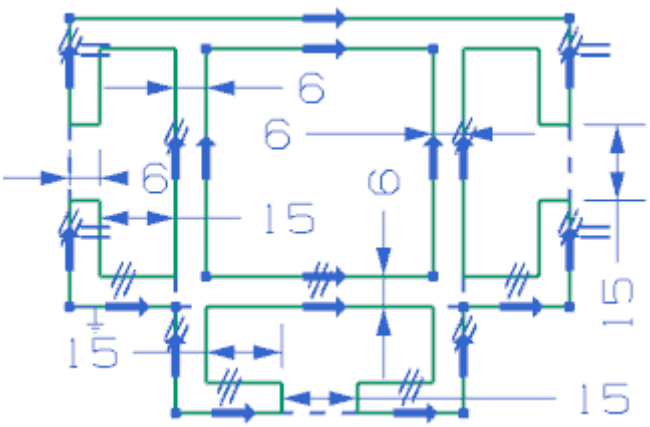


9. Make the two vertical lines as shown collinear.

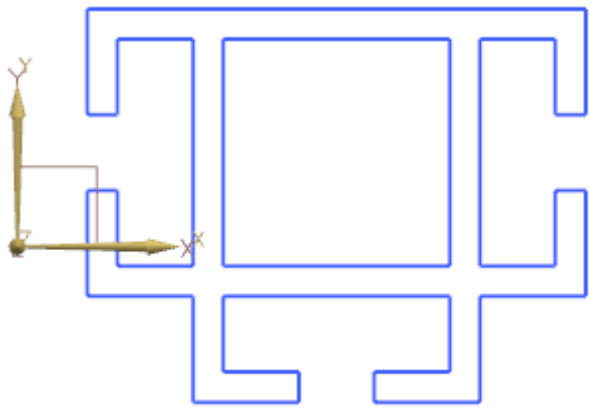




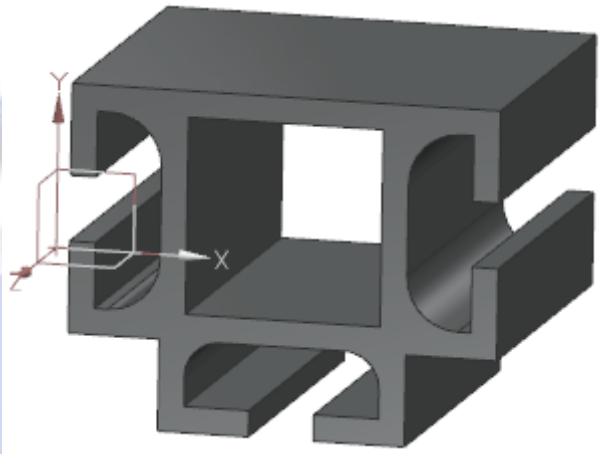
10. You can continue to constrain the sketch or go to the next step.



11. When you finish adding constraints, right-click in the background of the graphics window and choose **Finish Sketch**  .

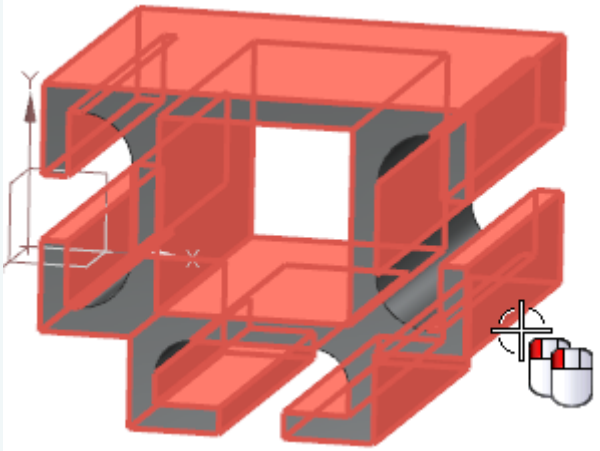


12. In the **Edit Cross Section** dialog box, click **OK** to complete the edits.
13. In the **Part Navigator**, right-click **Body (1)** and choose **Show**.

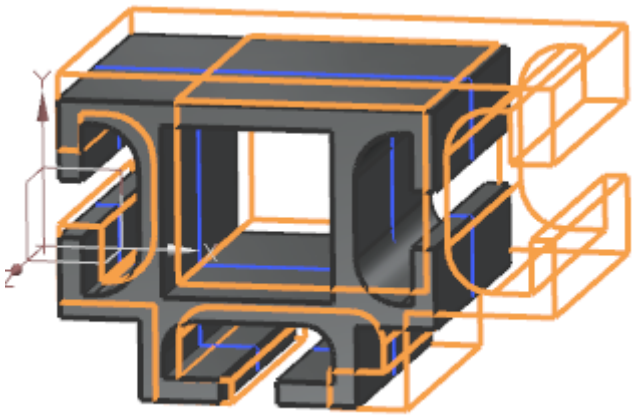


Edit the existing cross section sketch

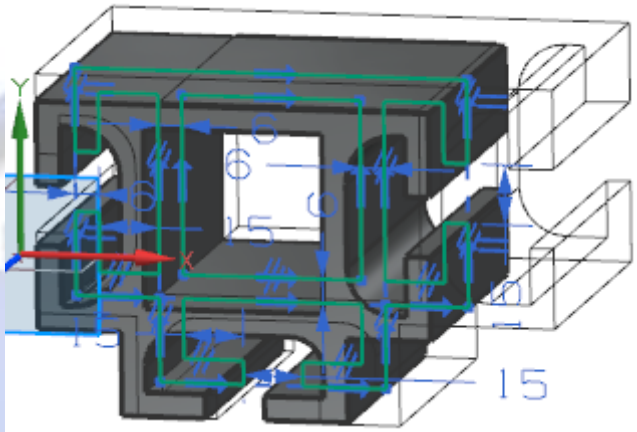
1. To edit the cross section, double-click the face as shown.



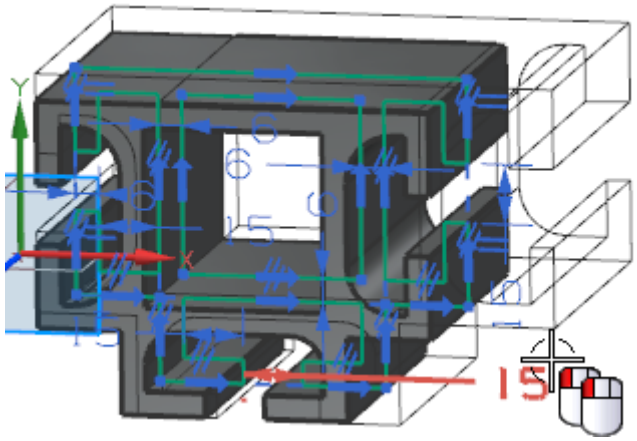
2. In the **Edit Cross Section** dialog box, in the **Preview** group, select the **Preview** check box.

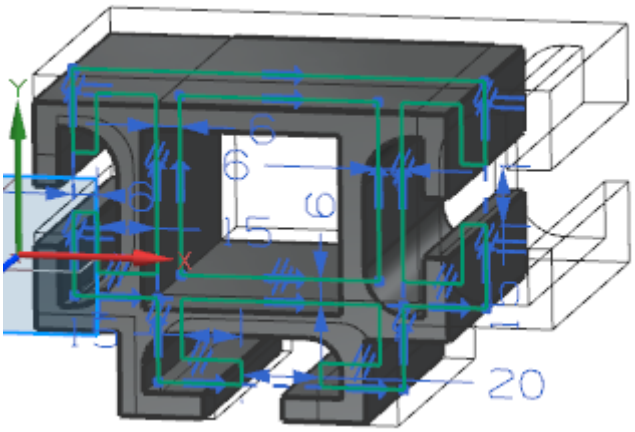


3. To enter the Sketch task environment, in the **Section Curve** group, click **Section**  .

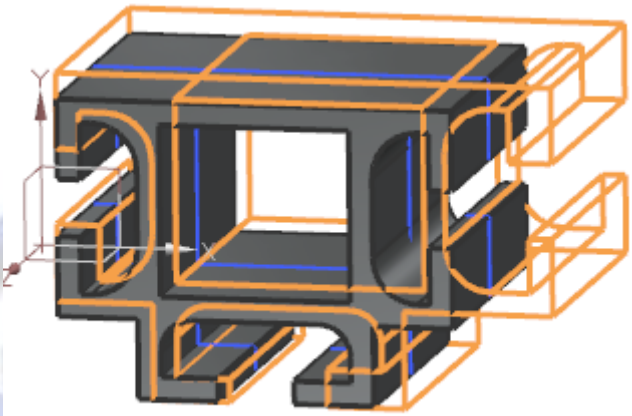


4. Edit one of the sketch dimensions or constraints you created, or create a dimension or constraint.

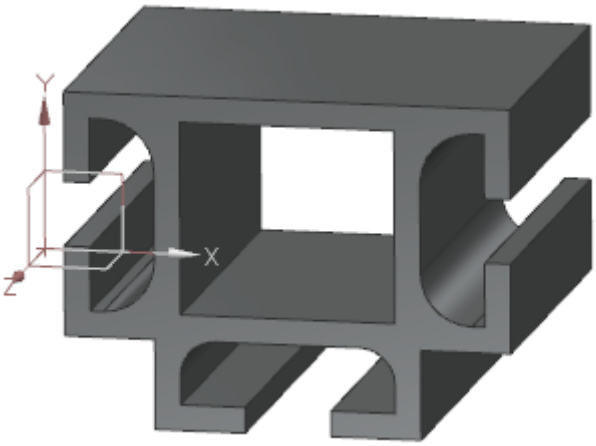




5. When you finish editing the sketch, right-click in the background of the graphics window and choose **Finish Sketch**  .



6. To complete the edits, in the **Edit Cross Section** dialog box, click **OK**.



You can use the **Edit Cross Section** command in **History Mode** to add design intent to models with or without feature history.

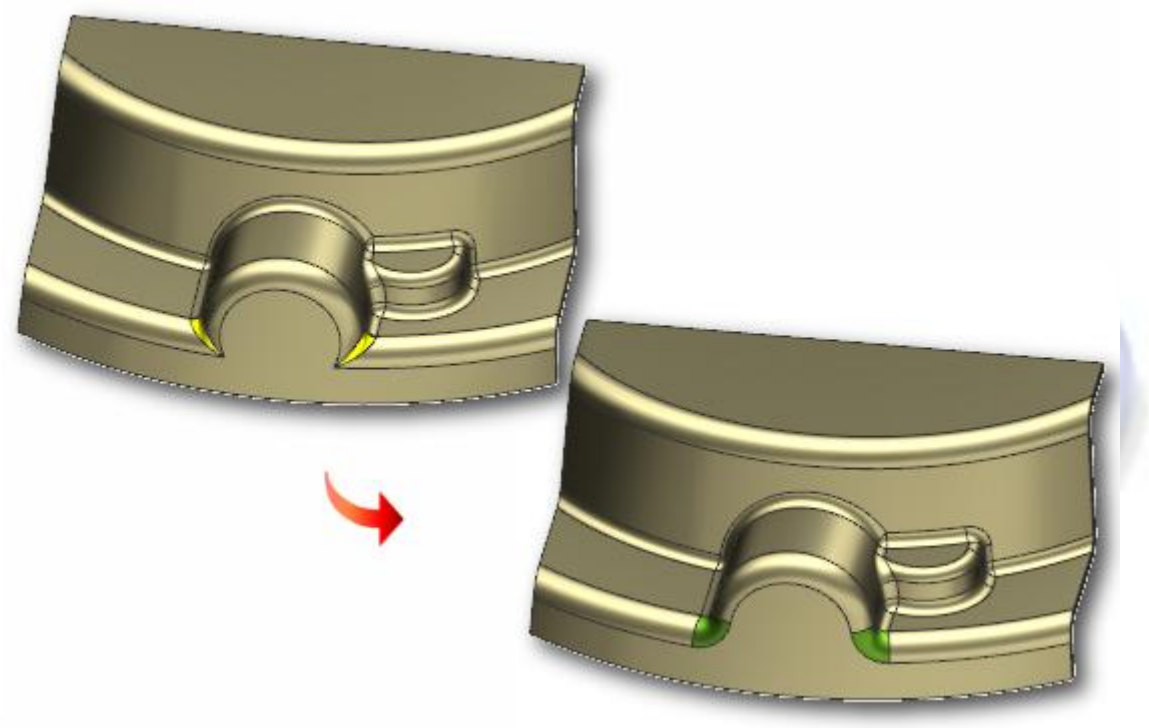
7. Close the part now or close all parts when you exit the session.

You completed the activity.

Activity: Reorder Blends

Estimated time to complete: 3–5 minutes

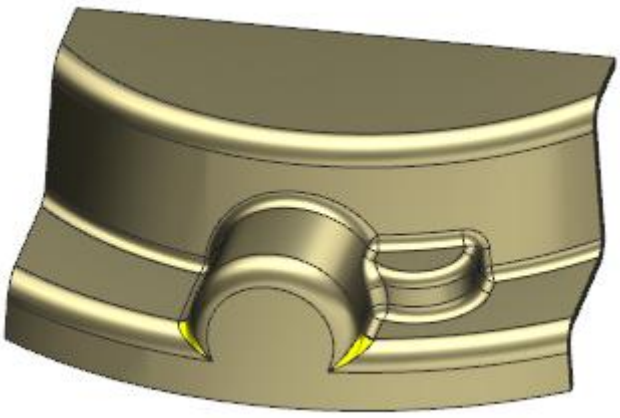
In this activity, you will change the order of blends in a molded part without feature history. Using the **Reorder Blends** command lets you change the order of two intersecting blends of opposite convexity without deleting and recreating blends.



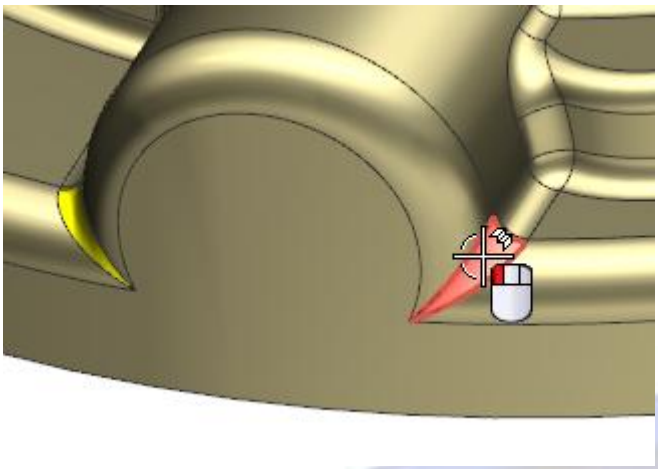
Launch the Reorder Blends activity.

Reorder Blends

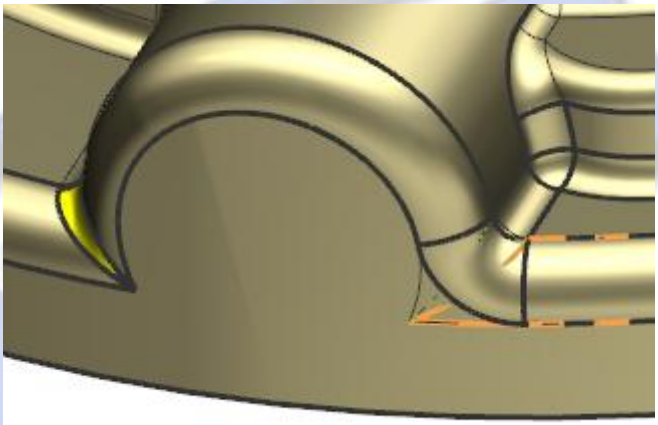
1. Open *syn2_reorder_blends*.



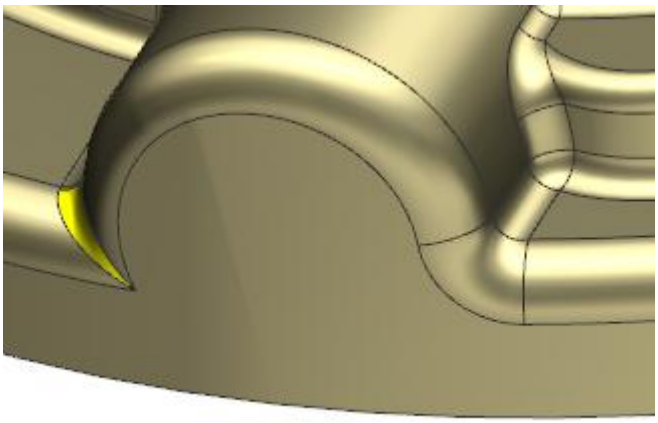
2. On the **Synchronous Modeling** toolbar, from the **Detail Feature Drop-down** list, click **Reorder Blends**  .
3. Select the blend face as shown.



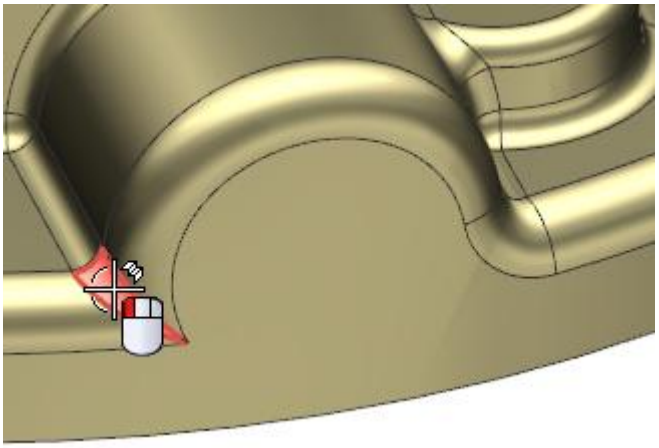
NX selects the **Blend Face 2** and displays the preview of the change.



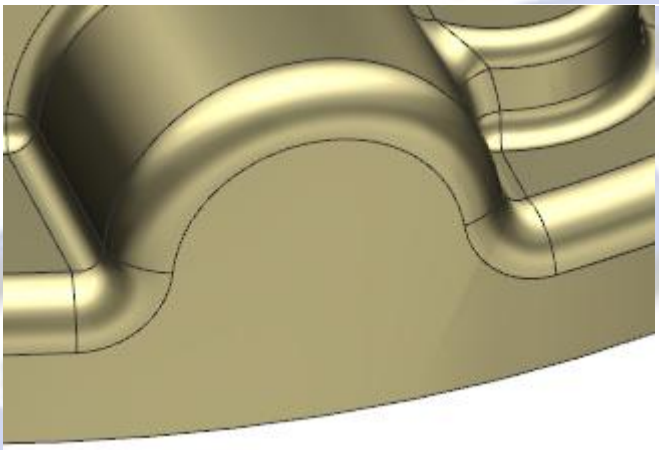
4. In the **Reorder Blends** dialog box, click **Apply**.



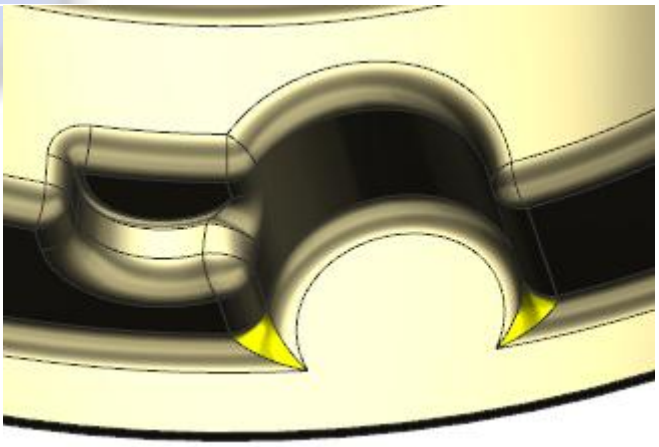
5. Select the blend face as shown.

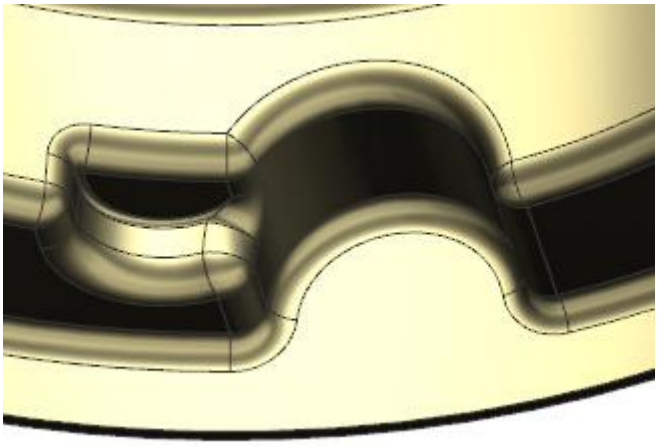


6. In the graphics window, click the middle mouse button to click **Apply**.



7. Reorder the two blends in the underside for the part.





8. Close the part now or close all parts when you exit the session.

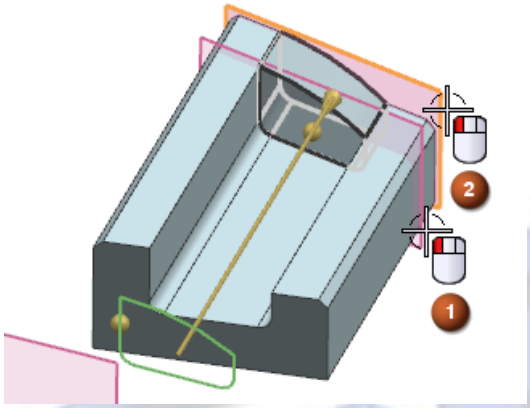
You completed the activity.

Activity: Project curves to a sketch

Estimated time to complete: 3-5 minutes

In this activity, you will:

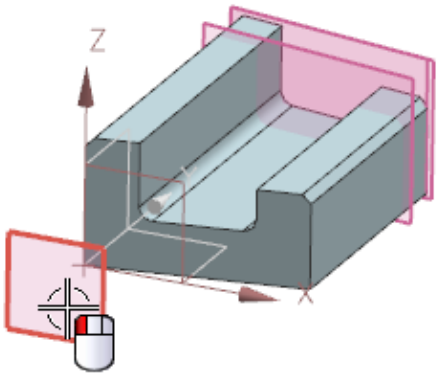
- Add a sketch feature to an unparameterized part.
- Associatively project curves from the unparameterized part into the sketch and use them to create a feature.
- Modify the part in such a way that the sketch-based feature updates.



Launch the Project curves to a sketch activity.

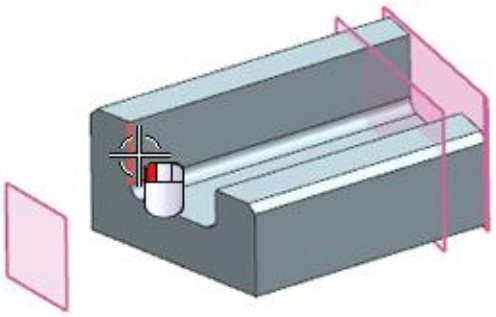
Project the groove edges to a sketch

1. Open *skt13_project_curves*.
2. On the **Direct Sketch** toolbar, click **Sketch**.
3. In the **Create Sketch** dialog box, from the **Type** list, select **On Plane**.
4. Select the datum plane shown and click **OK**.

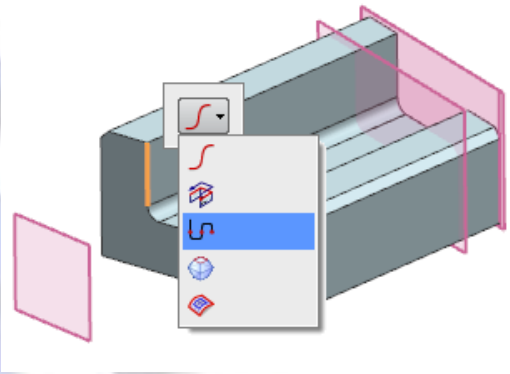


5. On the **Direct Sketch** toolbar, from the **Curve from Curves Drop-down** list, select **Project Curve**.

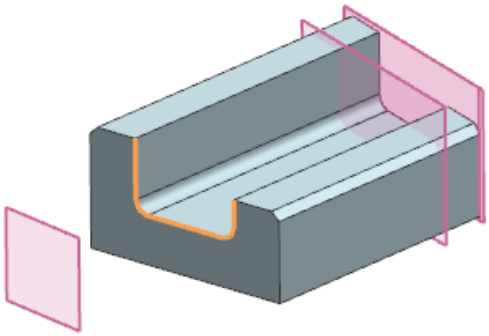
6. In the **Project Curve** dialog box title bar, click **Reset** to return the dialog box and the **Selection Bar** to the default settings.
7. In the **Settings** group, make sure the **Associative** check box is selected.
8. Select any curve on the edge of the inner groove.



9. From the on-screen **Curve Rule Drop-down** list, select **Tangent Curves**.

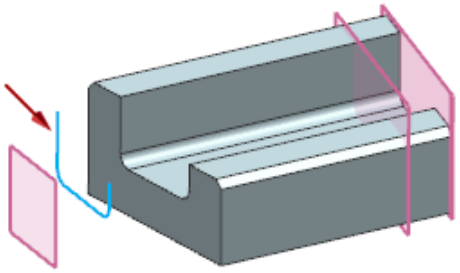


The tangent edges are selected. You get this same result had you selected **Tangent Curves** from the **Curve Rule** list on the Selection bar before you selected the edge.



10. Click **OK**

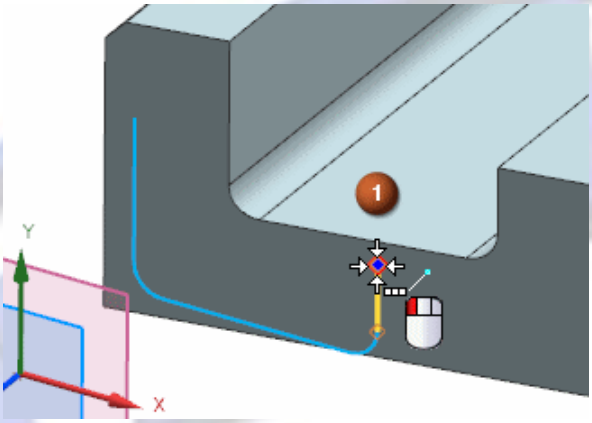
NX projects the curves to the sketch plane.

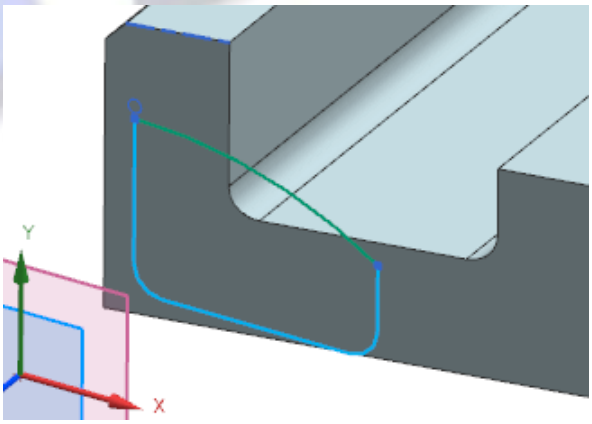


Extrude the sketch and modify the part

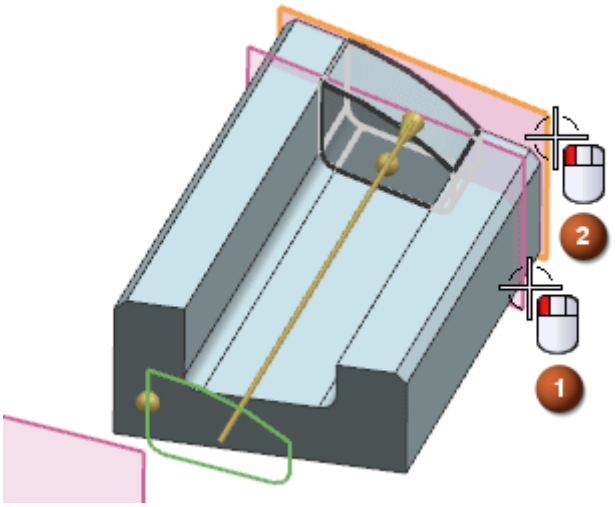
1. On the **Direct Sketch** toolbar, click **Arc**  .
2. Create an arc that closes the projected profile.

   As you create the arc, be sure to infer a tangent constraint with the top edge of the solid body as shown.

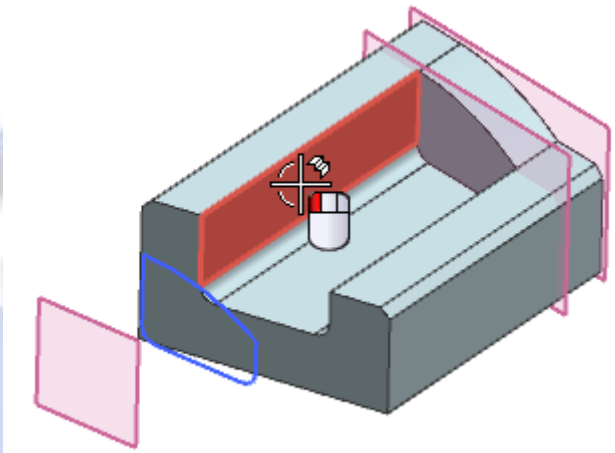




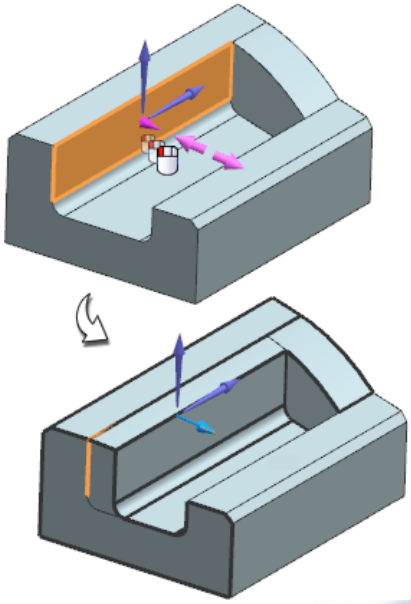
3. On the **Feature** toolbar, click **Extrude**  .
4. In the **Limits** group, for both **Start** and **End**, select **Until Selected** and select the datum planes as shown.



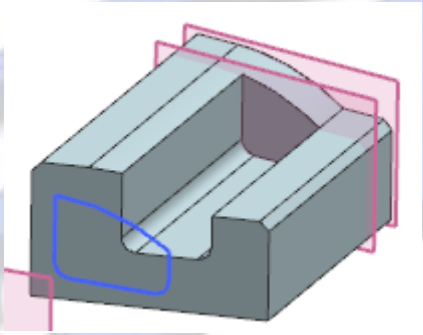
5. On the **Synchronous Modeling** toolbar, click **Move Face**  .
6. Select the face shown.



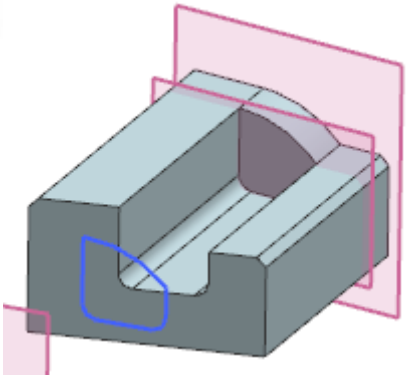
7. In the **Transformation** group, from the **Motion** list, select **Distance**.
8. Drag the handle into the part as shown. Click **OK** to complete the move face command.



Because the sketch appears before the **Move** feature in **Timestamp** order, the sketch and extrude do not update.



9. In the **Part Navigator**, drag **Move Face (8)** before **Sketch (6)**.



NX updates the sketch curves and the extrude feature.

10. Close the part now or close all parts when you exit the session.

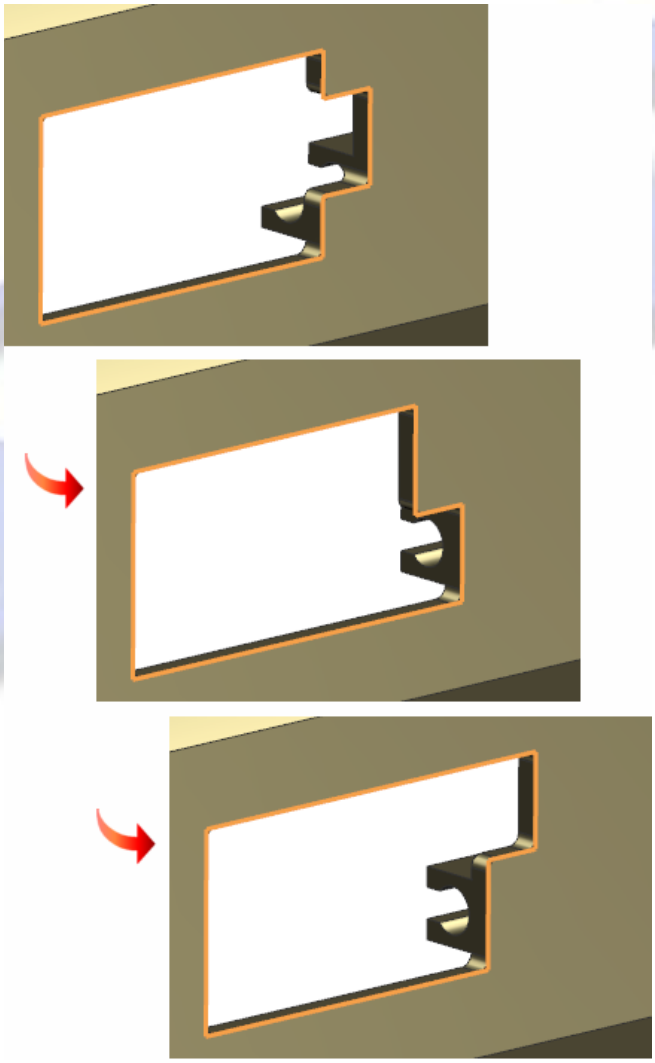
You completed the activity.

Activity: Zero length lines in a sketch

Estimated time to complete: 3-5 minutes

This extruded aluminum heat sink needs to be configured for three different assemblies. The three different cutout configurations shown below can be made with the same sketch. In this activity, you will:

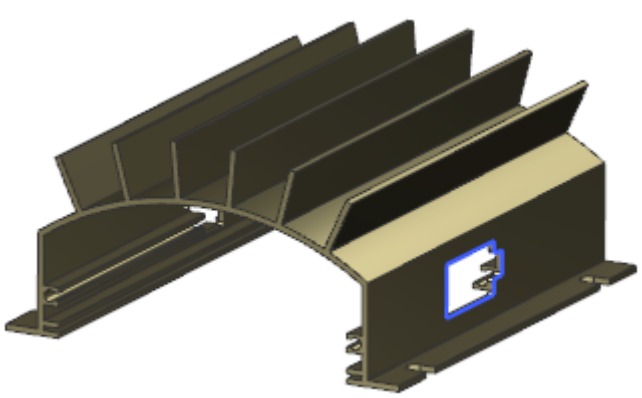
- Add a dimension to a sketch.
- Edit the dimension value to equal zero.
- See that a **Perpendicular** dimension maintains its direction when the expression value is set to zero.
- See that you can have zero length lines in a sketch.



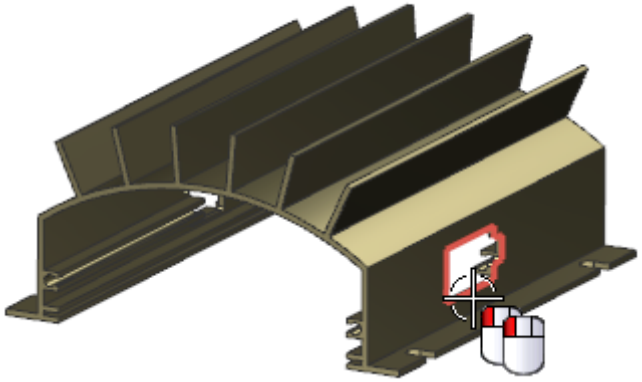
Launch the Zero length lines in a sketch activity.

Add sketch dimension

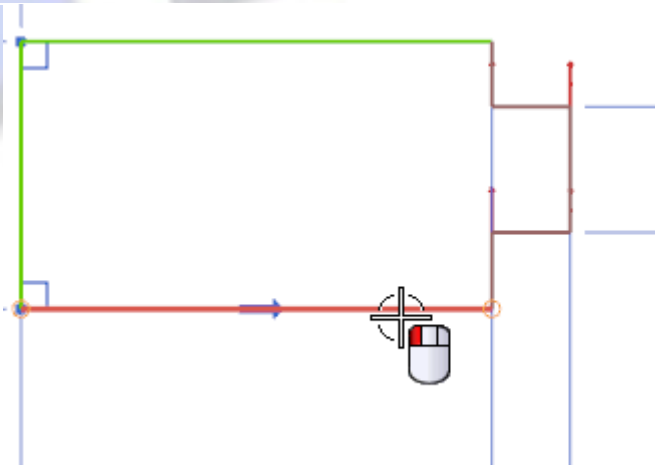
1. Open *skt10_sketch_dimensions*.



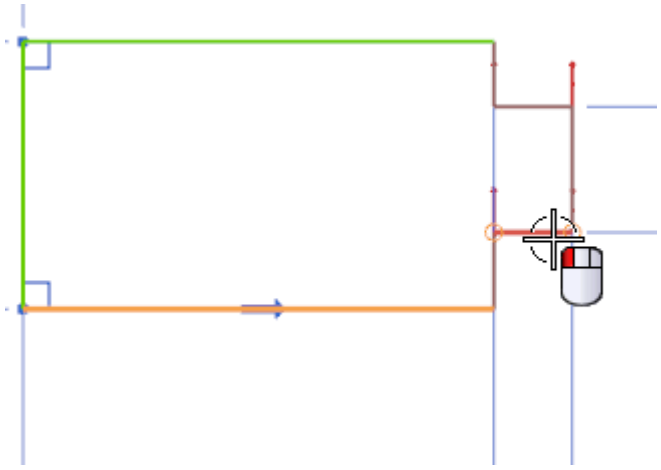
2. Double-click the sketch as shown to open it for edit.



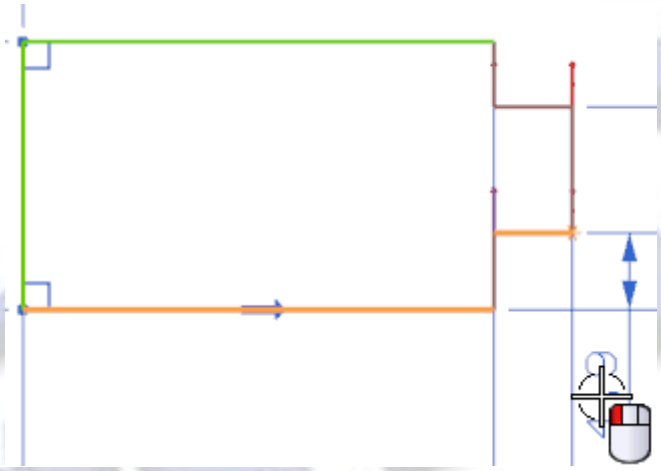
3. On the **Direct Sketch** toolbar, click **Inferred Dimensions**  .
4. Select the horizontal line as shown.



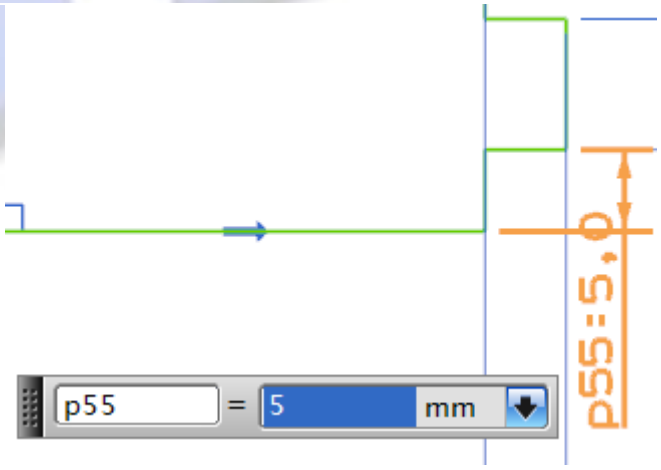
5. Select the short horizontal line as shown.



6. Click to place the dimension.



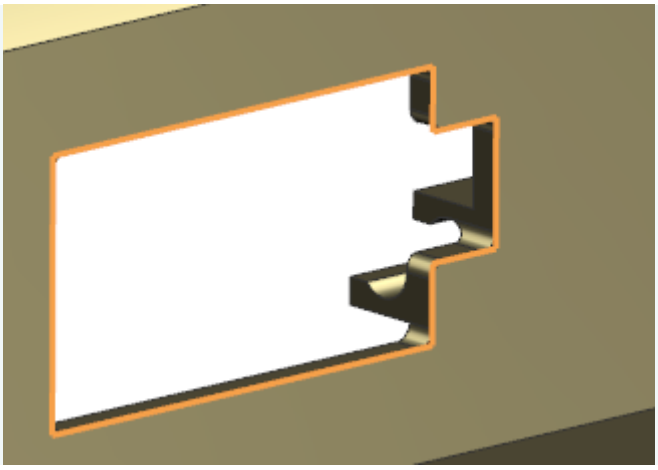
7. In the on-screen input box, type **5** and press Enter.



8. Right-click in the background of the graphics window and choose **Finish Sketch**  .

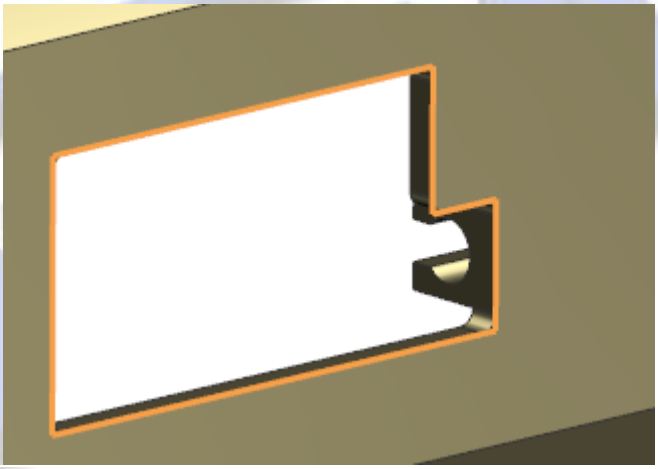
Edit sketch dimension

1. In the **Part Navigator**, click **Sketch (10)**.



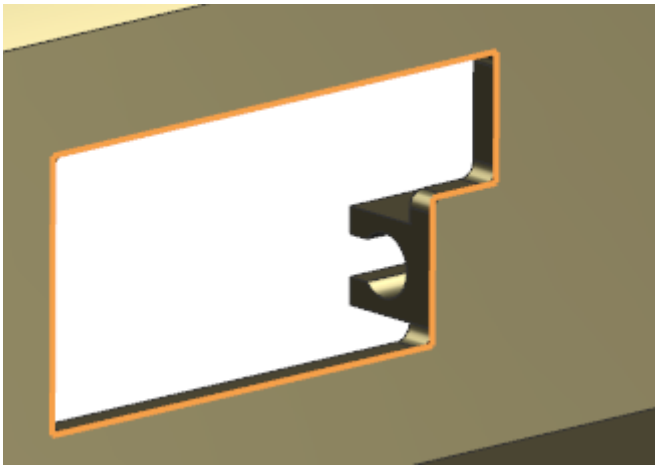
2. In the **Part Navigator**, expand **Details**.
3. In the **Expression** column double-click **p55=5**, type **0** and press Enter.

   The notch has moved to the bottom of the sketch and the lower short vertical line is removed.



4. In the **Expression** column double-click **p55=0**, type **9** and press Enter.

   The notch has moved to the top of the sketch and the lower short vertical line appears.



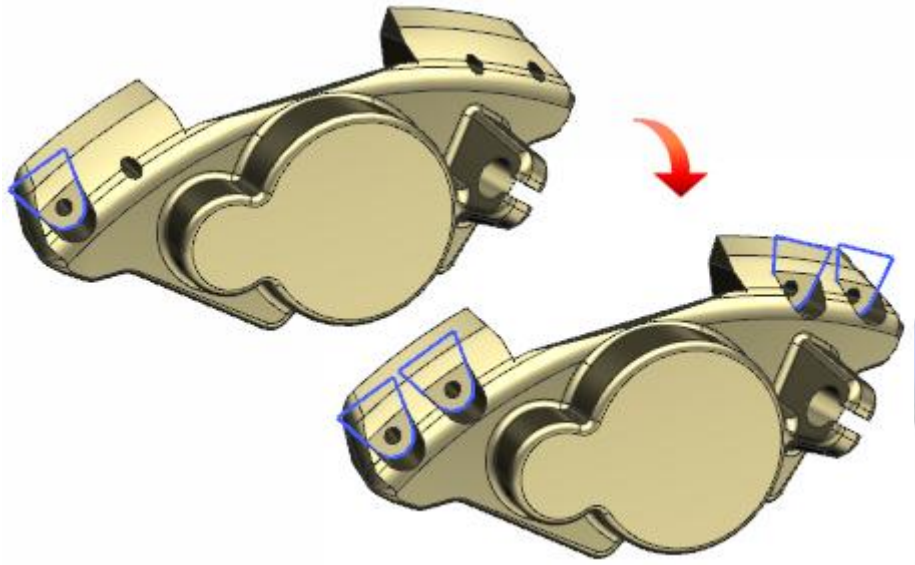
5. Close the part now or close all parts when you exit the session.

You completed the activity.

Activity: Create a general pattern in a sketch

Estimated time to complete: 5–8 minutes

In this activity, you will pattern the existing sketch to add bolt clearance pockets on a brake caliper.

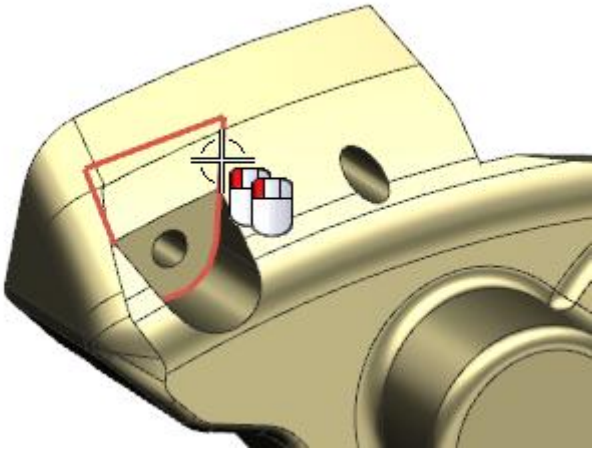


Launch the Create a general pattern in a sketch activity.

Pattern sketch curves

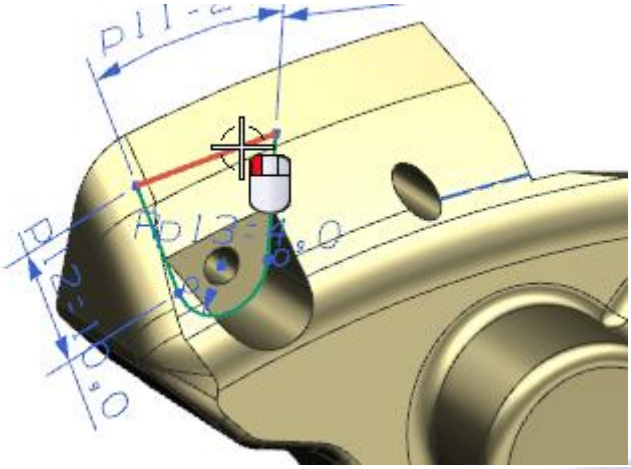
Edit the existing sketch.

1. Open *skt13_general_pattern*.
2. Double-click any curve in the sketch to open it for edit.

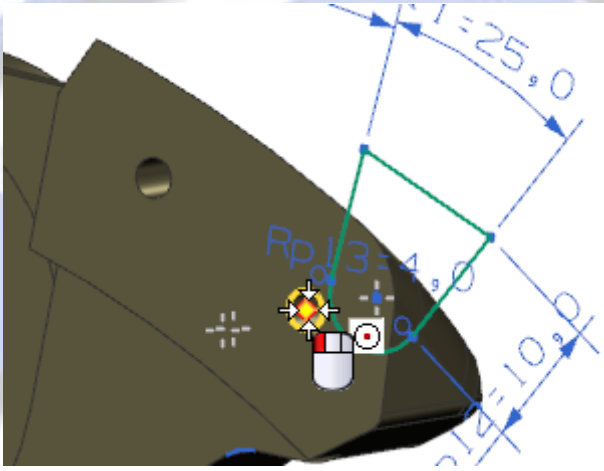


3. On the **Direct Sketch** toolbar, from the **Curve from Curves Drop-down** list, select **Pattern Curve**  .

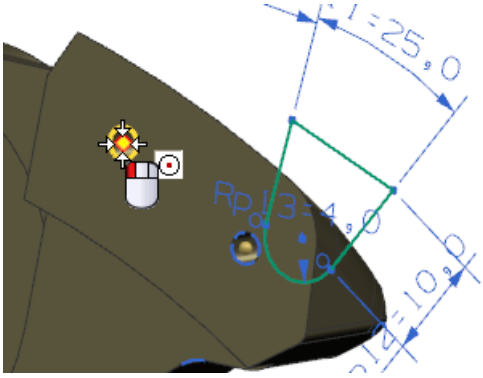
4. From the **Curve Rule** list, select **Connected Curves**.
5. Select the sketch curves to pattern as shown.



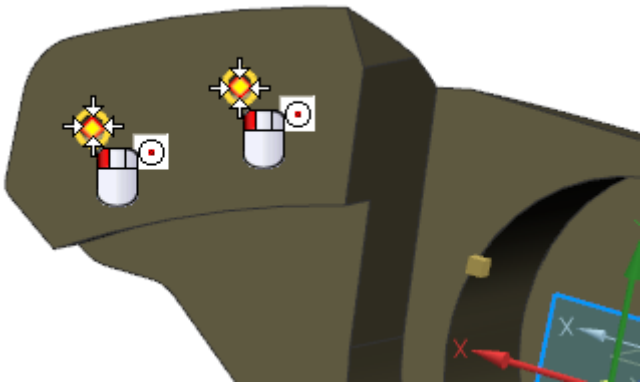
6. In the **Pattern Curve** dialog box, in the **Pattern Definition** group, from the **Layout** list, choose **General**.
7. In the **From** section, click **Specify Point**.
8. On the opposite side of the part, select the arc center of the edge shown.



9. Select the arc center of the edge as shown for **To** location.



10. Select the remaining two arc centers of the edges as shown for additional **To** locations.



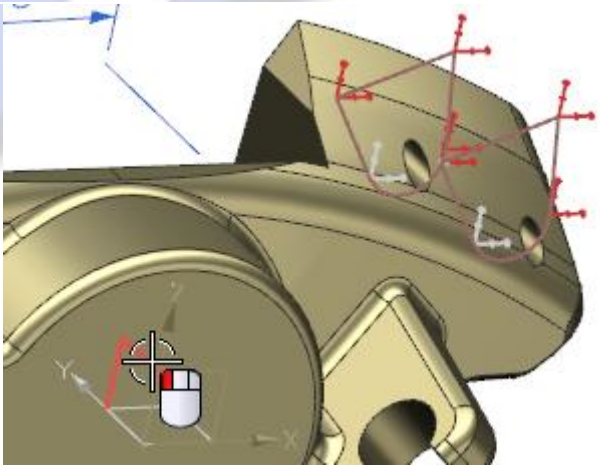
11. In the **Pattern Curve** dialog box, in the **Pattern Definition** group, clear the **Lock Orientation** check box.

    Because the patterns were created with the lock orientation check box cleared, you can set the rotation angle for each pattern.

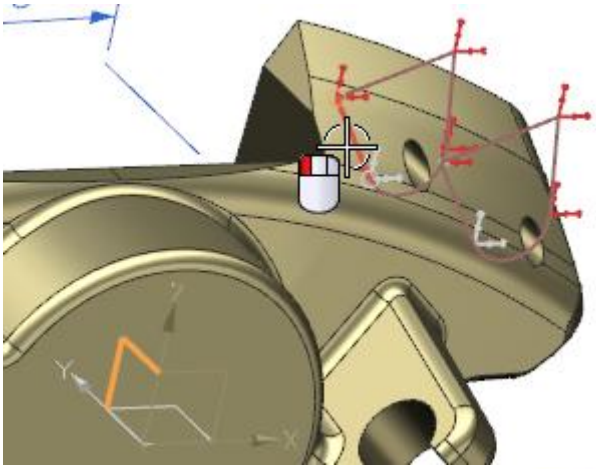
Constrain patterns

Constrain the rotation angle for each pattern.

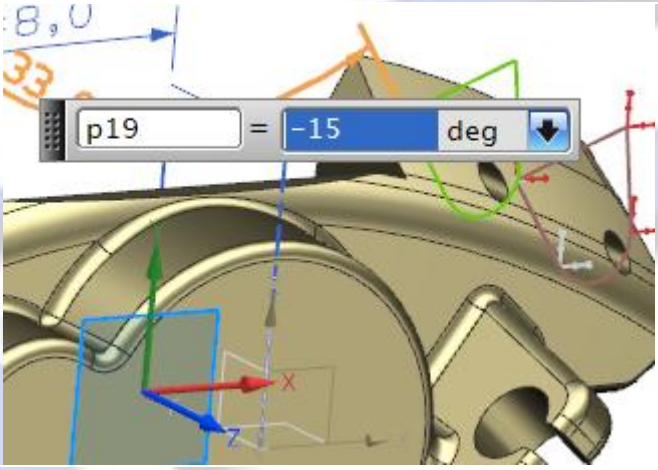
1. On the **Direct Sketch** toolbar, click **Inferred Dimensions**  .
2. Select the Y-Z datum plane as shown.



3. Select the sketch curve as shown between the end points, and click to indicate the dimension origin.

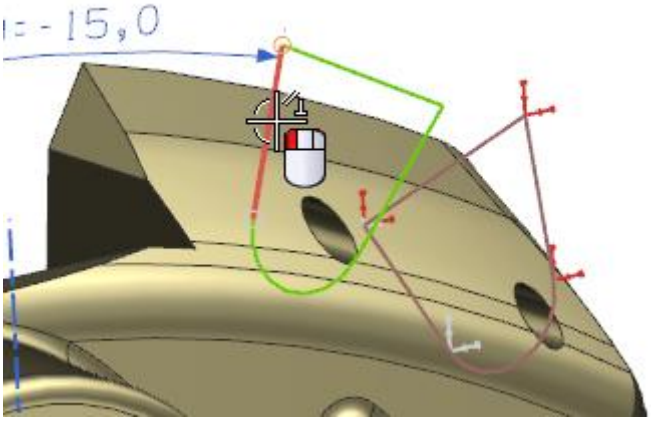


4. In the on-screen input box, type **-15**, and press Enter.

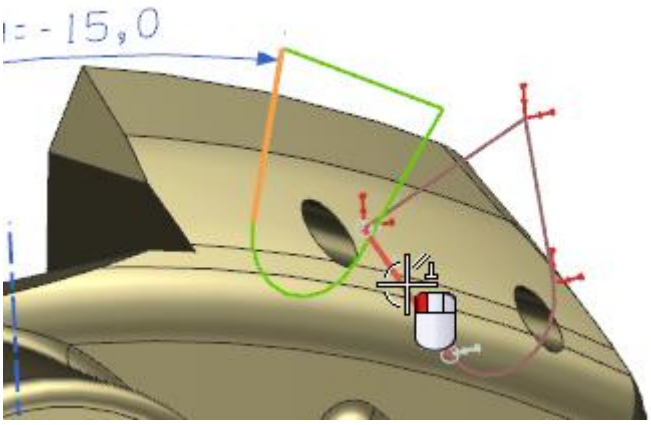


Note You can enter a negative value for angle dimensions to get the alternate angle.

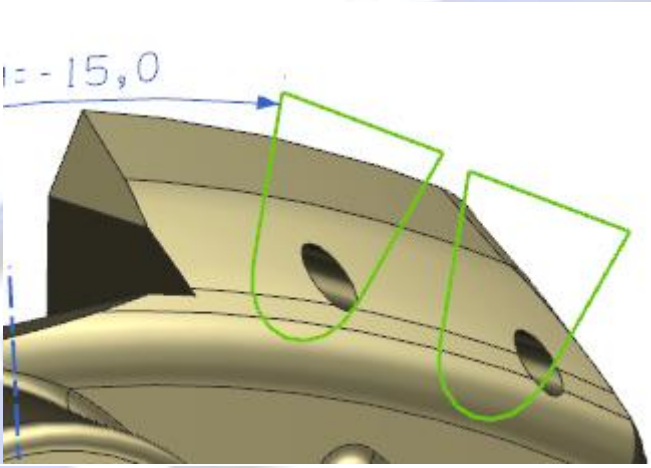
5. On the **Direct Sketch** toolbar, click **Constraints**  .
6. Select the sketch curve as shown.



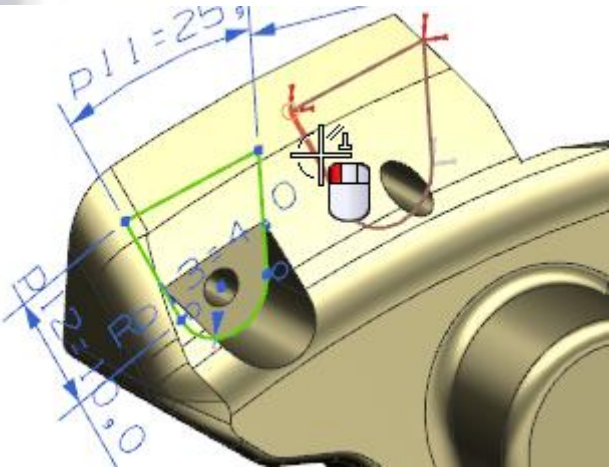
7. Select the second sketch curve as shown.



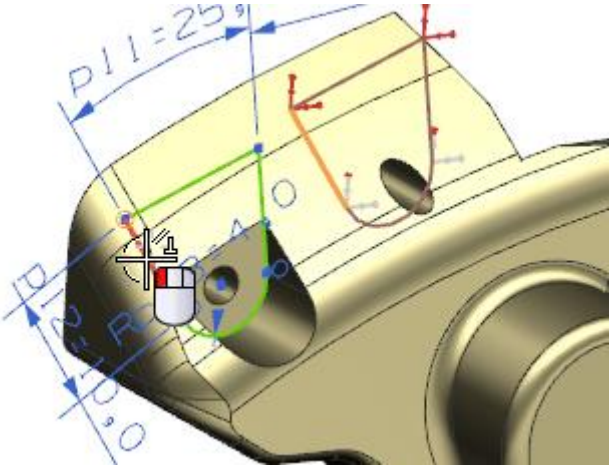
8. Right-click one of the selected curves and choose **Parallel**  .



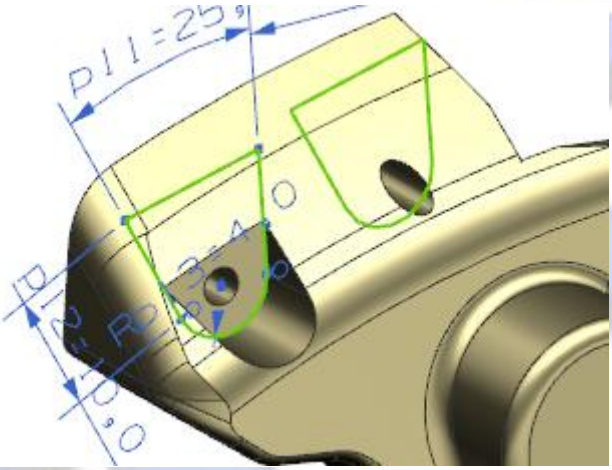
9. Select the sketch curve as shown.



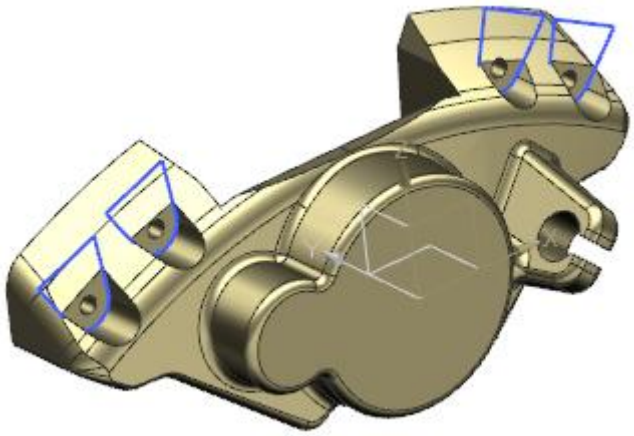
10. Select the second sketch curve as shown.



11. Right-click one of the selected curves and choose **Parallel**  .



12. In the background of the graphics window, right-click and choose **Finish Sketch**  .



In the **Part Navigator**, the **Extrude (6)** node has an information alert icon  .

13. Move your cursor over the **Extrude (6)** feature to display the **Information** window.

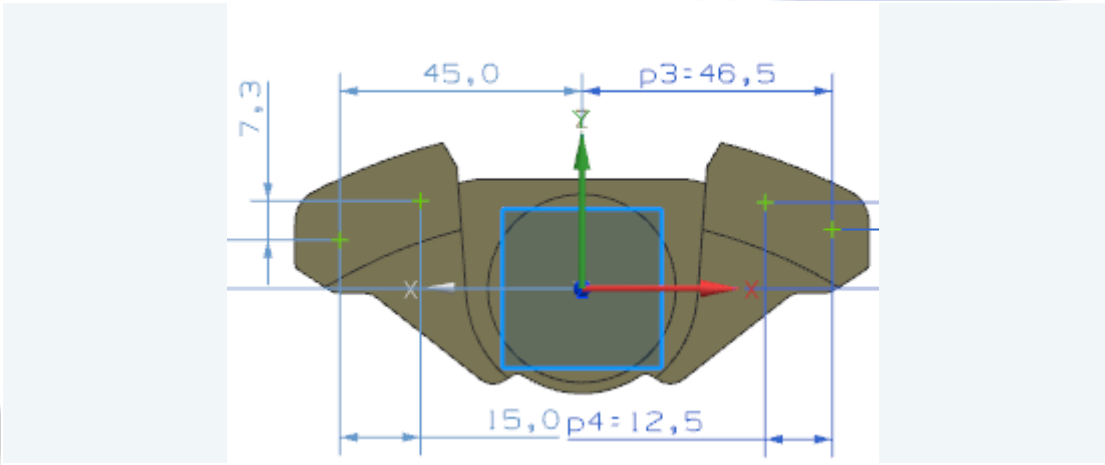
When the original extrude feature was created, the selection curve rule was set to **Feature Curves**. Curves that were added to the sketch were also be added to the extrude feature.

14. To close the **Information** window, right-click the **Extrude (6)** node and choose **Clear Feature Info Alerts**.

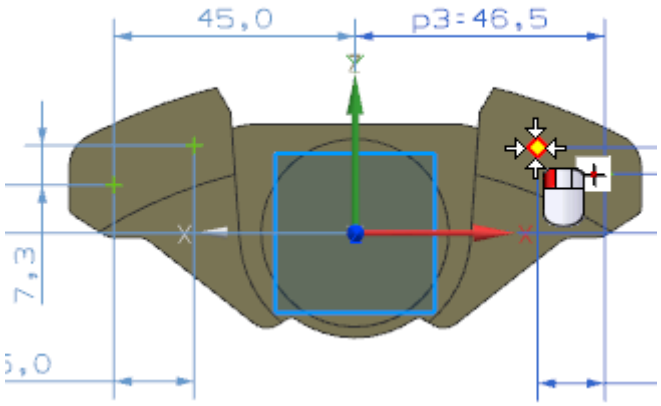
Edit hole locations

The through holes, and therefore the sketch patterns, are not located correctly. You need to change the hole locations. The patterns are associative to the hole locations and will be updated when the holes move.

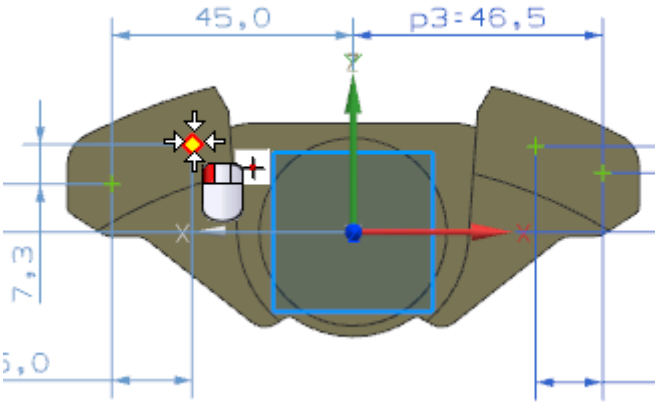
1. In the **Part Navigator**, right-click the **Simple Hole** and choose **Edit Sketch**.



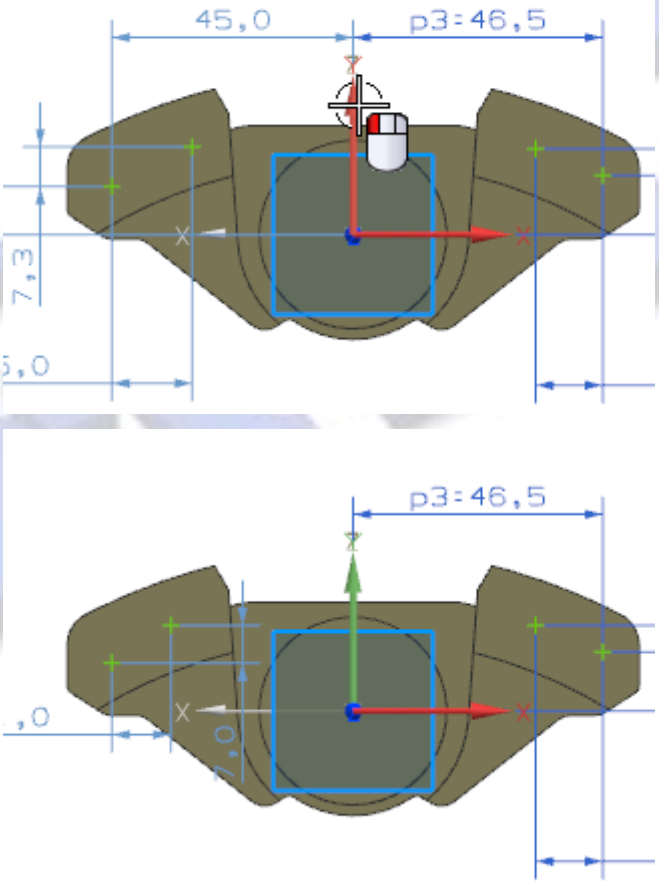
2. On the **Sketch Tools** toolbar, click **Make Symmetric** to create a symmetric constraint between the inner and outer two holes.
3. Select the point as shown for **Primary Object**.



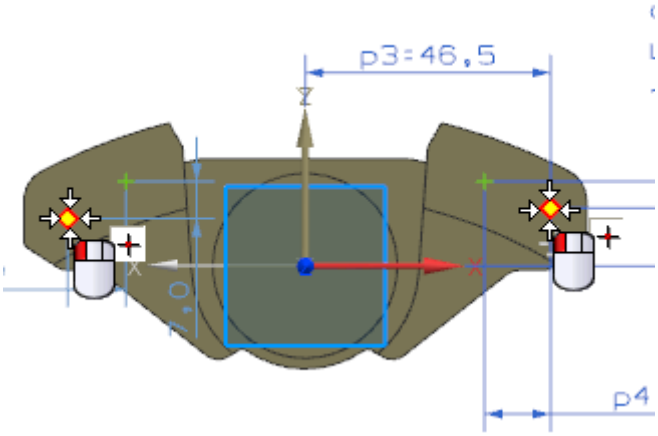
4. Select the point as shown for **Secondary Object**.

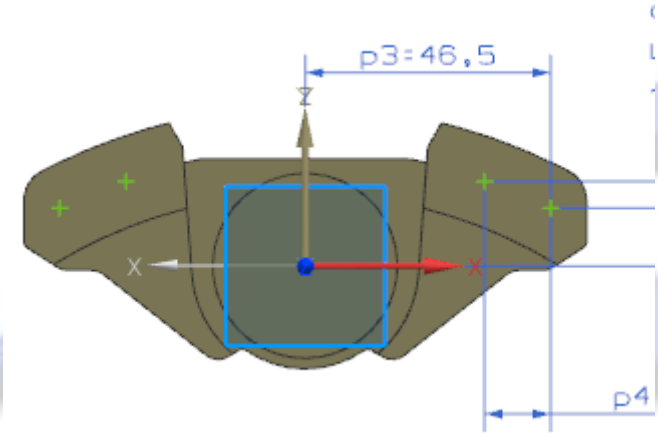


5. Select the datum axis as shown for **Symmetry Centerline**.



6. Create another symmetric constraint between the outer two points.

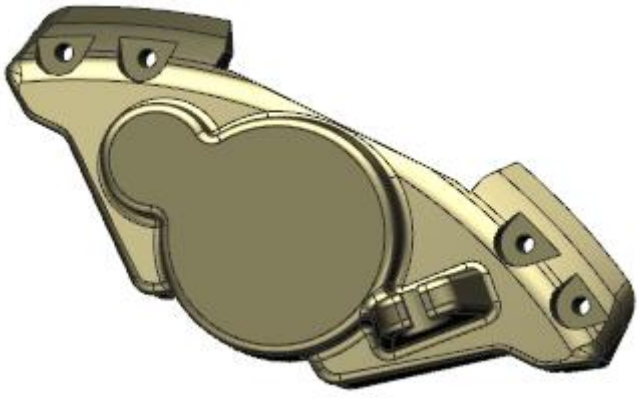




Note The automatic dimensions on the two left points are removed because the symmetric constraint now applies.

7. Right-click in the background of the graphics window and choose **Finish Sketch**  .

   The sketch patterns are associative to the hole centers and are now located correctly.

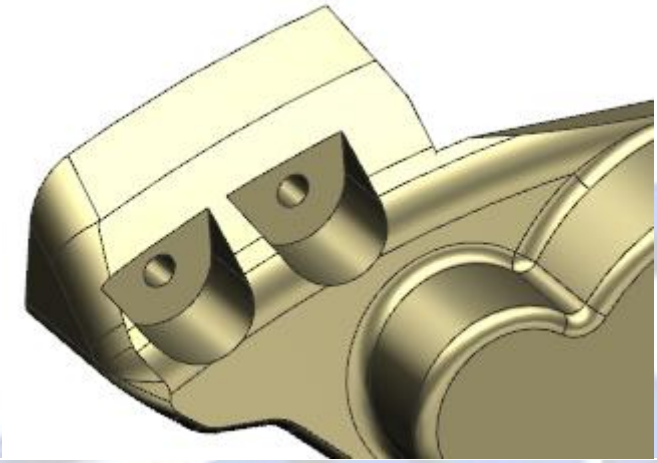


8. Close the part now or close all parts when you exit the session.

Edit patterns

The sketch patterns are associative to the original curves. You need to edit one of the sketch dimensions to allow for a larger bolt head clearance.

1. In the **Part Navigator**, select the **Sketch** node.
2. In the **Part Navigator**, expand the **Details** panel.
3. In the **Details** panel, slowly double-click the **p13=4** expression, type **5** and press Enter.



All four patterns and extrusions now have larger radii to accommodate a larger bolt head clearance.

You completed the activity.

Assembly

Constraint Navigator

1. Open the part

   Open *asm3_ws_assembly_1* loading all components with **Use Partial Loading**.

   1.1 On the **Standard** toolbar, click **Open**  .

   1.2 In the **Open** dialog box, click **Options**. The **Assembly Load Options** dialog box appears.

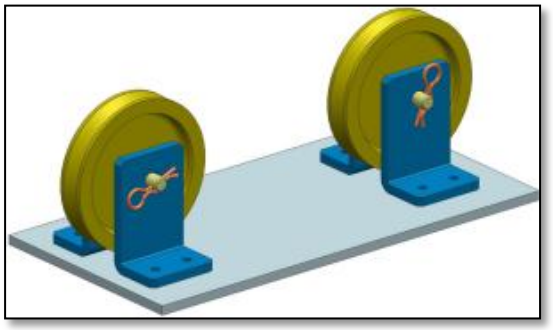
   1.3 In the **Part Versions** group, expand the **Load** list and select **From Folder**.

   1.4 In the **Scope** group, expand the **Load** list and select **All Components**.

   1.5 Ensure that the **Use Partial Loading** check box is selected.

   1.6 Click **OK**.

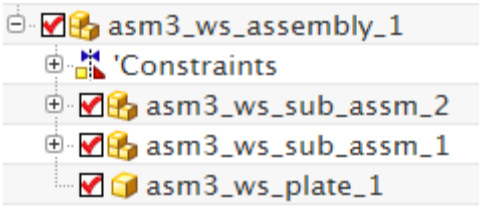
   1.7 Open *asm3_ws_assembly_1*.



2. Review the assembly

2.1 Open the **Assembly Navigator**.

2.2 Notice there are 2 subassemblies and a component.



2.3 In the **Assembly Navigator**, expand the sub-assemblies.

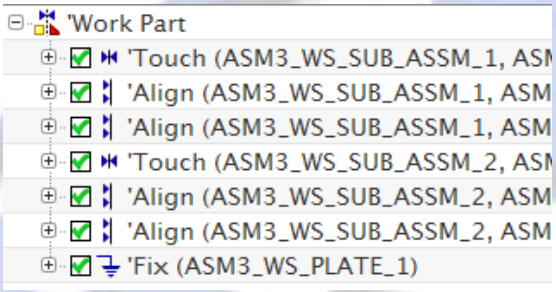
2.4 Observe that the top level assembly and both sub-assemblies have Constraints nodes.

3. Review top level assembly constraints

3.1 In the **Resource Bar**, click the **Constraint Navigator** and pin it open.

3.2 Below the last constraint listed, right-click and choose **Group by Constraints**.

This setting lists the constraints by pairs of components. Also notice that the constraints are only listed for the Work Part.



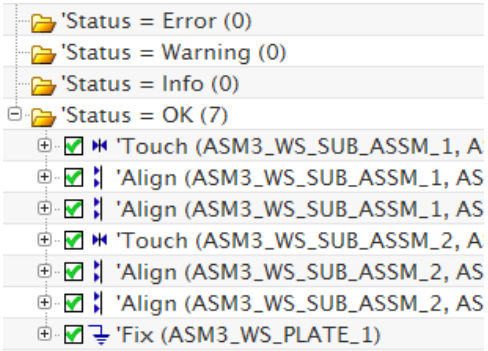
4. Group constraint status

4.1 In the **Constraint Navigator** below the last constraint listed, right-click and select **Group by Constraint Status**.

4.2 Observe that the navigator lists by status.

4.3 Expand **Status = OK (7)**.

Now you see the same constraints as before.

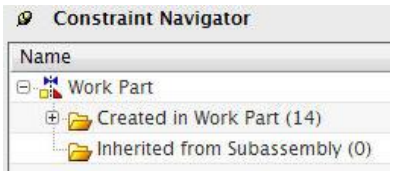


5. Group by inherited

5.1 In the **Assembly Navigator**, double-click *asm3_ws_sub_assm_2* to make it the work part.

5.2 Return to the **Constraint Navigator**.

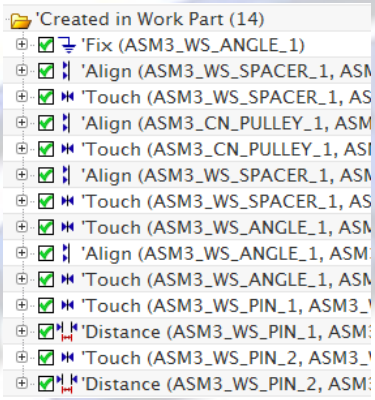
5.3 In the **Constraint Navigator** below the last constraint listed, right-click and select **Group by Inherited**.



5.4 Observe that there are no inherited constraints.

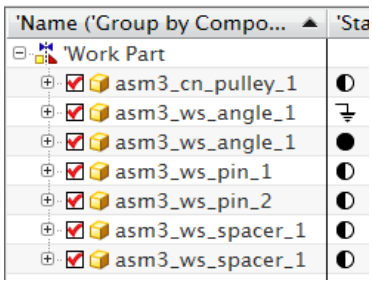
5.5 Expand **Created in Work Part (14)**.

Listed are all of the constraints of the sub-assembly.



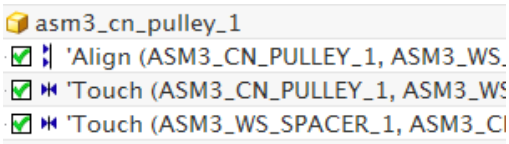
6. Group by components

6.1 In the **Constraint Navigator** below the last constraint listed, right-click and select **Group by Components**.



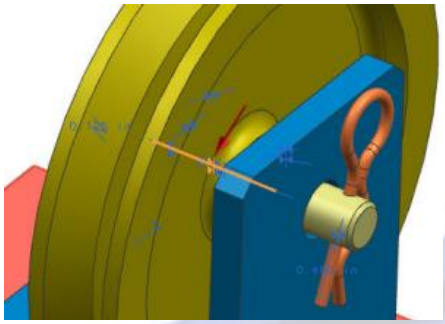
6.2 Observe the **Status** column. Notice that one of the angle components is fixed and the other is fully constrained. The rest of the components are partially constrained.

6.3 Expand *asm3_cn_pulley_1*.



6.4 Click **Align (ASM3_CN_PULLEY_1, ASM3_WS_ANGLE_1)**.

6.5 Observe in the graphic window that the constraint is highlighted.

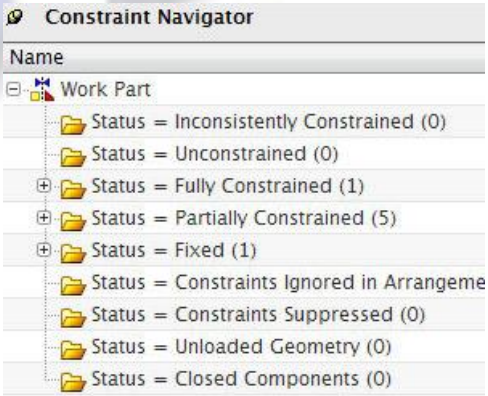


6.6 Review the other 2 constraints of the pulley.

With a little interrogation you can see that the pulley rotation is not constrained, thus the pulley is partially constrained.

7. Group by component status

7.1 In the **Constraint Navigator** below the last constraint listed, right-click and select **Group by Component Status**.



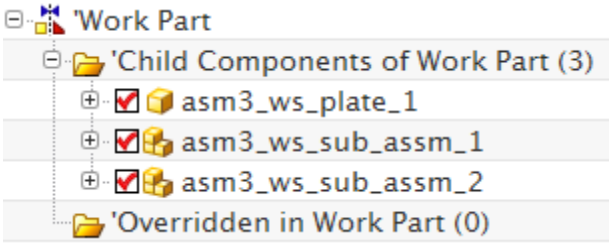
8. Group by component level

8.1 In the **Assembly Navigator**, double-click *asm3_ws_assembly_1* to make it the work part.

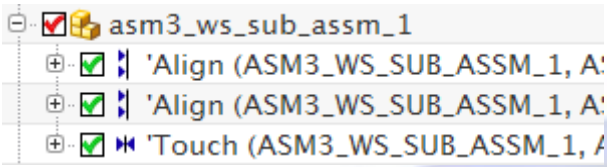
8.2 In the **Constraint Navigator** below the last constraint listed, right-click and select **Group by Component Level**.

8.3 Expand **Child Components of Work Part (3)**.

This grouping allows the investigation of constraints at the component level.



8.4 Expand *asm3_ws_sub_assm_1*.



8.5 Click **Touch (ASM3_WS_SUB_ASSM_1, ASM3_WS_PLATE_1)**.

8.6 Observe in the graphic window that the top of the plate becomes highlighted.

Since this is a Touch constraint, you could assume that it is constrained to the bottom of one of the angle components.

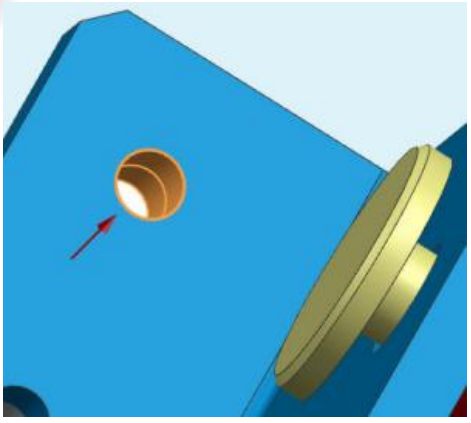
8.7 Click of the **Align (ASM3_WS_SUB_ASSM_1, ASM3_WS_PLATE_1)** constraints.

8.8 Observe in the graphic window that 2 holes are highlighted, 1 in the plate and 1 in the angle.

8.9 Click the other **Align (ASM3_WS_SUB_ASSM_1, ASM3_WS_PLATE_1)** constraint.

8.10 Observe in the graphic window that 2 holes are highlighted, 1 in the plate and 1 in the angle.

Now it's easy to see how the **ASM3_WS_SUB_ASSM_1** is fully constrained.



8.11 Close all parts; do not save.

Unique part files from existing components

1. **Open the part**

   Open *asm3_ws_assembly_1* loading all components with **Use Partial Loading**.

   1.1 On the **Standard** toolbar, click **Open**  .

   1.2 In the **Open** dialog box, click **Options**.

   The **Assembly Load Options** box is displayed.

   1.3 In the **Part Versions** group, from the **Load** list, select **From Folder**.

   1.4 In the **Scope** group, from the **Load** list, select **All Components**.

   1.5 Ensure that the **Use Partial Loading** check box is selected.

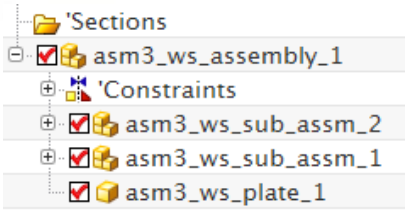
   1.6 Click **OK**.

   1.7 Open *asm3_ws_assembly_1*.

2. **Review the assembly**

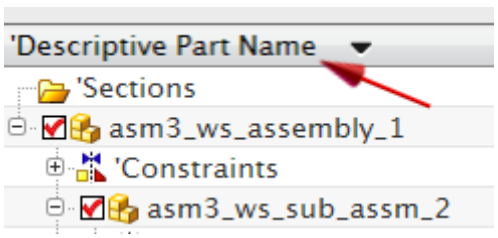
   2.1 Verify that the **Assemblies** application is running.

   2.2 In the Assembly Navigator, notice that there are 2 subassemblies and a component.



   2.3 Expand the node for the subassemblies.

   2.4 Click the **Descriptive Part Name** column heading.

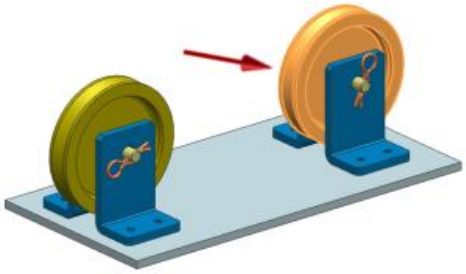


   2.5 Observe that the components of the two subassemblies are identical.

3. **Make a pulley unique**

The design requirement is that the pulley on the right side needs to be increased 4” in diameter. You will use the Make Unique command to achieve this.

3.1 Right-click the right side pulley and choose **Make Unique**.



3.2 In the **Make Unique** dialog box, click **Name Unique Parts**.

3.3 If necessary, from the **Project Assignment** message box, click **OK** until the message is dismissed.

3.4 In the **Name and Location** group, in the **Name** box, type **xxx_pulley_1**, where xxx represent your initials, and press Enter.

3.5 In the **Folder** box, enter and address to a directory to which you have write access.

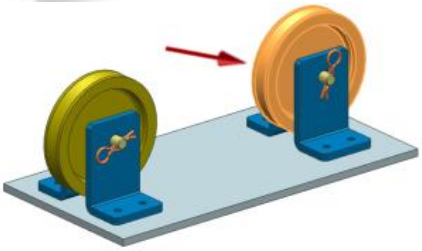
3.6 In the **Name Unique Parts** dialog box, click **OK**.

3.7 In the **Make Unique** dialog box, click **OK**.

3.8 Click **OK** if you get a **Read-only Part** message.

4. **Edit the pulley**

4.1 Right-click the right side pulley and choose **Make Work Part**.

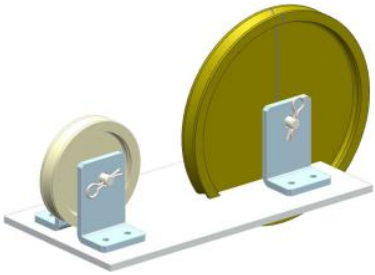


4.2 Choose **Tools⟶Expression**.

4.3 In the **Expressions** dialog box, from the **Listed Expressions** list, select **All**.

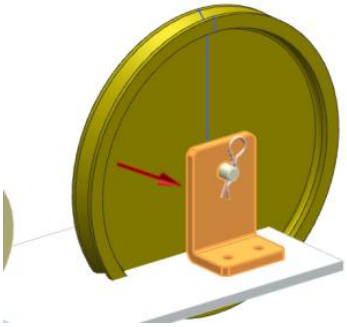
4.4 Select the **p8** expression and change the **Formula** value to **4**.

4.5 Click **OK**.



5. Make the angle unique

5.1 Right-click the right angle, as shown, and choose **Make Unique**.



5.2 In the **Make Unique** dialog box, click **Name Unique Parts**.

5.3 If necessary, from the **Project Assignment** message box, click **OK** until the message is dismissed.

5.4 In the **Name and Location** group, in the **Name** box, type **xxx_angle_1**, where xxx represent your initials, and press Enter.

5.5 In the **Folder** box, enter an address to a directory to which you have write access.

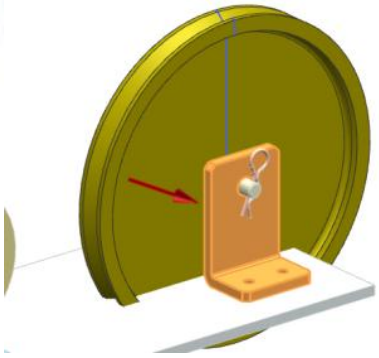
5.6 In the **Name Unique Parts** dialog box, click **OK**.

5.7 In the **Make Unique** dialog box, click **OK**.

5.8 In the **Make Unique** warning box, click **No - Discard Changes**.

6. Edit the angle

6.1 Right-click the angle and choose **Make Work Part**.



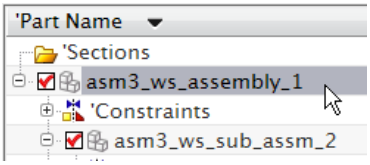
6.2 Choose **Tools⟶Expression**.

6.3 Select the **p0** expression and change the **Formula** value to **5**.

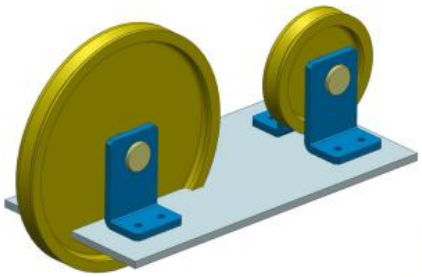
6.4 Click **OK**.

7. Replace the angle

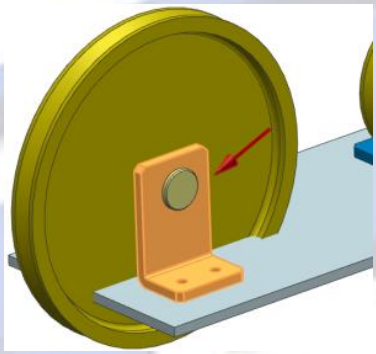
7.1 In the **Assembly Navigator**, double-click *asm3_ws_assembly_1* to make it the work part.



7.2 Rotate the view to the orientation shown below.



7.3 Right-click the angle, as shown, and click **Replace Component**.



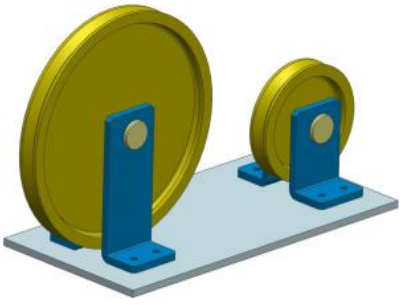
7.4 In the **Replace Component** dialog box in the **Replacement Part** group, select **xxx_angle_1**.

7.5 In the **Settings** group, verify that the **Maintain Relationships** check box is selected.

7.6 Click **OK**.

7.7 In the **Replace Component** warning box, click **Yes**.

7.8 Choose **View⟶Operation⟶Regenerate Work**.



7.9 Close all parts without saving them.

Load Interpart data enhancements

1. **Verify current load options**

   1.1 Choose **File**⟶**Options**⟶**Assembly Load Options**.

   1.2 In the **Part Versions** group, from the **Load** list, select **From Folder**.

   1.3 In the **Scope** group, from the **Load** list, select **All Components**.

   1.4 Ensure that the **Use Partial Loading** check box is selected.

   1.5 Ensure that the **Load Interpart Data** check box is not selected.

   1.6 Click **OK**.

2. **Save the plate component**

   2.1 Open *asm3_interpart_assembly_1*.

   2.2 In the **Assembly Navigator**, double-click **asm3_ipu_plate_1** to make it the work part.

   2.3 In the **Read-only Part** message box, click **OK**.

   2.4 Choose **File**⟶**Save As**.

   2.5 From the **Save in** list, select a directory in which you have write access.

   2.6 In the **File name** box, type ***_plate_1** where *** represents your initials.

   2.7 Click **OK**.

   2.8 In the **File name** box, type ***_interpart_assembly_1** where *** represents your initials.

   2.9 Click **OK**.

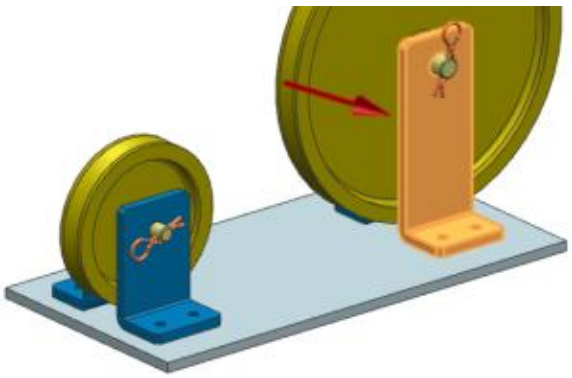
   2.10 In the **Save As** message box, click **Yes**.

   2.11 In the **Save Warning** message box, click **OK**.

   2.12 Close the **Information** window.

3. **Save the angle component**

3.1 Right-click *asm3_ipu_angle_1* component as shown, and choose **Make Work Part**.



3.2 Choose **File⟶Save As**.

3.3 From the **Save in** list, select a directory in which you have write access.

3.4 In the **File name** box, type ****_angle_1* where *** represents your initials.

3.5 Click **OK**.

3.6 In the **Save as** dialog box, click **Cancel**.

3.7 In the **Save As** message box, click **Yes**.

3.8 In the **Save As Report** message box, click **OK**.

3.9 Close the **Information** window.

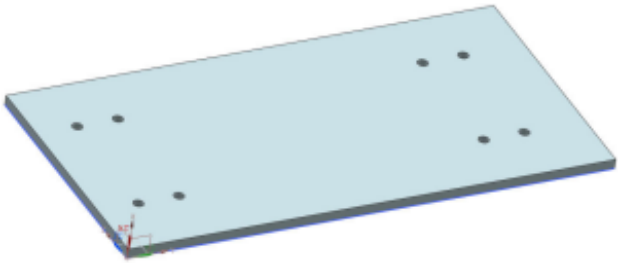
3.10 In the **Assembly Navigator**, double-click ****_interpart_assembly_1* to make it the work part.

3.11 Choose **File⟶Save**.

3.12 Choose **File⟶Close⟶All Parts**.

4. **Change a hole size in the plate component**

4.1 Open ****_plate_1*.

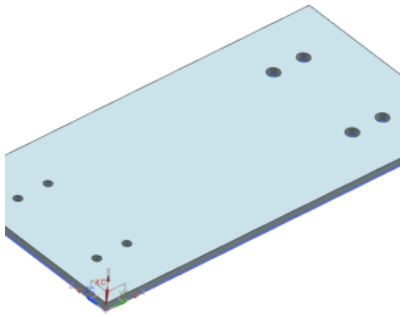


4.2 Choose **Tools⟶Expression**.

4.3 Select the **p11** expression and change the **Formula** value to **0.375**.

4.4 Click **OK**.

4.5 Observe that the holes on the right side of the plate are now larger.



4.6 Choose **File⟶Close⟶Save and Close**.

5. **Open the assembly**

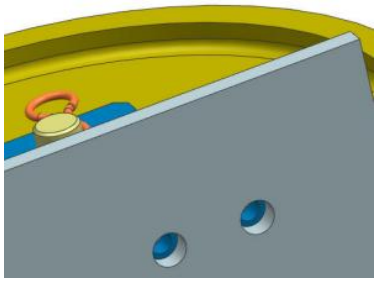
5.1 Choose **File⟶Options⟶Assembly Load Options**.

5.2 In the **Part Versions** group, in the **Load** list, select **As Saved**.

5.3 Click **OK**.

5.4 Open ****_interpart_assembly_1*.

5.5 Observe the size of the holes in the angle and plate components under the large pulley. Notice that the holes are of two different sizes. An interpart expression links the holes from the plate to the angle so that they will be the same size. However, because of the **Partial Loading** and **Load Interpart Data** settings used from the **Assembly Load Options** dialog box, the expressions are not updated.



5.6 Close all parts without saving.

6. Open the assembly

6.1 Choose File⟶Options⟶Assembly Load Options.

6.2 In the Part Versions group, in the Load list, select As Saved.

6.3 In the Scope group, ensure that the Load Interpart Data checkbox is selected.

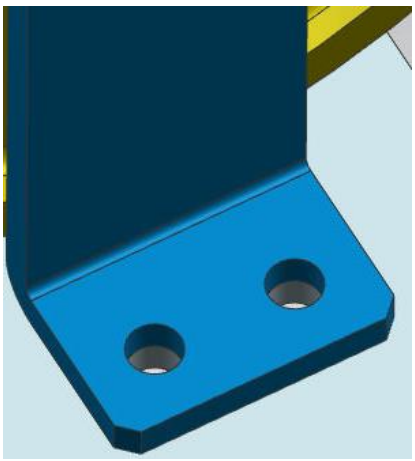
6.4 From the Load Parents list, select All Levels.

6.5 Click OK.

6.6 Open ****_interpart_assembly_1*.

6.7 Observe the size of the holes in the angle and plate components under the large pulley.

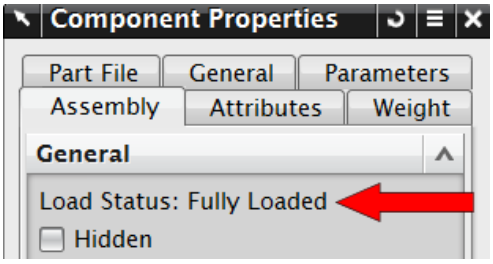
The holes in the plate and angle are now the same size.



6.8 From the Assembly Navigator, right-click ***_angle_1 and choose Properties.

6.9 From the Assembly page, review the Load Status.

Even though you used the Partial Loading option, the Load Interpart Data option automatically fully loaded the ***_angle_1 component part.



6.10 Click OK.

6.11 Close all parts without saving.

7. Open the assembly by group

7.1 Choose File⟶Options⟶Assembly Load Options.

7.2 In the Scope group, from the Load list, select Specify Component Group.

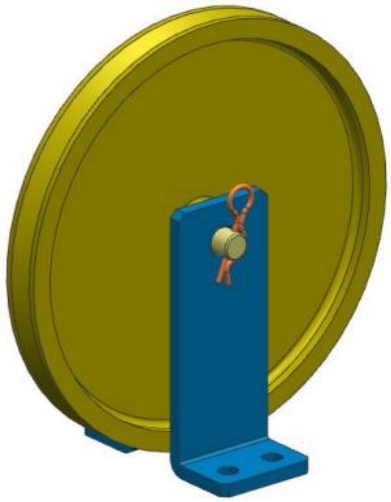
7.3 Ensure that the **Load Interpart Data** check box is selected.

7.4 Click **OK**.

7.5 Open ****_interpart_assembly_1*.

7.6 In the **Open Components** dialog box, select **By Name Pulley_2** and click **OK**.

The assembly opens with only the selected component group and this time the holes in the angle components have been updated as defined in the plate component.



7.7 Observe the size of the holes in the angle components under the large pulley.

The holes have been updated to reflect the size in the plate component.

7.8 Close all parts without saving.

7.9 Open the Assembly Load Options and clear the Load Interpart Data check box.

Reopening modified parts

4.1 Open the part

Open *asm3_ra_assembly_1* loading all components with **Use Partial Loading**.

1.1 On the **Standard** toolbar, click **Open**  .

1.2 In the **Open** dialog box, click **Options**.

The **Assembly Load Options** dialog box appears.

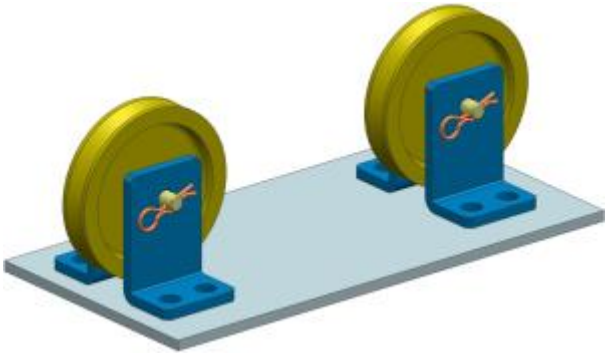
1.3 In the **Part Versions** group, expand the **Load** list and select **From Folder**.

1.4 In the **Scope** group, expand the **Load** list and select **All Components**.

1.5 Ensure that the **Use Partial Loading** check box is selected.

1.6 Click **OK**.

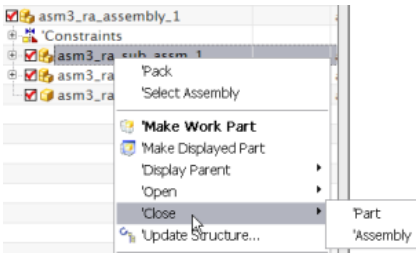
1.7 Open *asm3_ra_assembly_1*.



2. **Review close options**

2.1 Open the **Assembly Navigator**.

2.2 Right-click *asm3_ra_sub_assm_1* and select **Close**.



2.3 Observe that the only selections for **Close** are **Part** and **Assembly**.

2.4 Right-click *asm3_ra_plate_1* and select **Close**.

Again, the only selections for **Close** are **Part** and **Assembly**.

3. Edit the pulley

3.1 In the **Assembly Navigator** expand the first *asm3_ra_sub_assm_1*.

3.2 Double-click *asm3_ra_pulley_1* to make it the work part.

3.3 Click **OK** if you get a **Read-only Part** message.

3.4 Choose **Tools⟶Expression**.

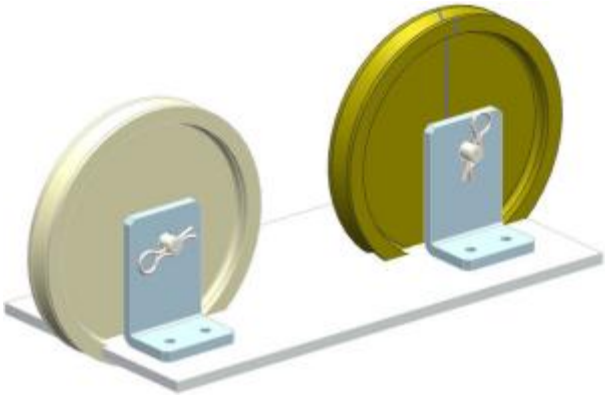
3.5 From the **Listed Expressions** list, select **All**.

3.6 Select the **p8** expression.

3.7 In the **Formula** box, type **3**.

3.8 Click **OK**.

The pulleys are now larger and intersect the plate.



3.9 Click **OK** if you get a **Read-only Part** message.

4. Edit the angle

4.1 In the **Assembly Navigator** double-click one of the *asm3_ra_angle_3* components to make it the work part.

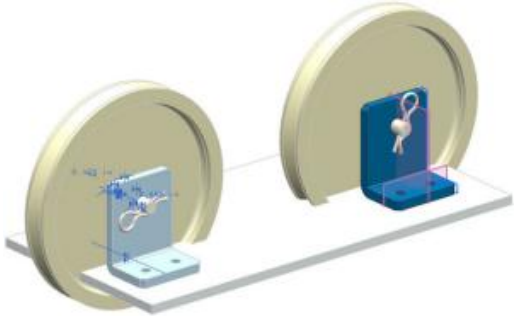
4.2 Choose **Tools⟶Expression**.

4.3 Select the **p0** expression.

4.4 In the **Formula** box, type **2.5**.

4.5 Click **OK**.

4.6 Click **OK** if you get a **Read-only Part** message.



5. **Edit the plate**

5.1 In the **Assembly Navigator** double-click *asm3_ra_plate_1* to make it the work part.

5.2 Choose **Tools⟶Expression**.

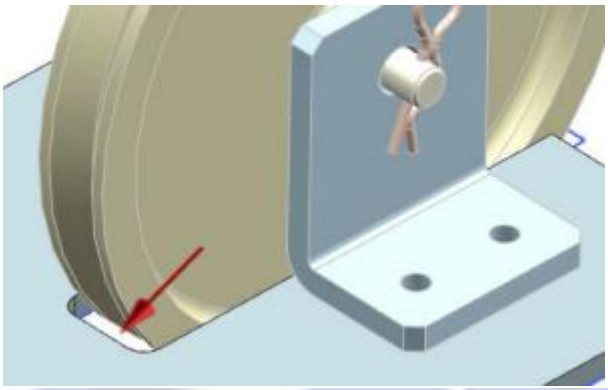
5.3 Select the **p23** expression.

5.4 In the **Formula** box, type **1**.

5.5 Click **OK**.

5.6 Click **OK** if you get a **Read-only Part** message.

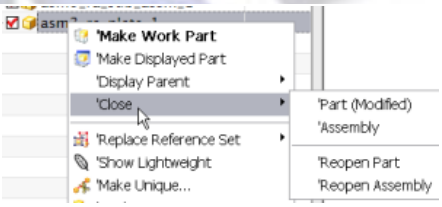
5.7 Observe that there are now slots in the plate.



6. **Change the work part**

6.1 In the **Assembly Navigator** double-click *asm3_ra_assembly_1*.

6.2 Right-click *asm3_ra_plate_1* and select **Close**.



6.3 Observe that there are now two new selections for **Close**, **Reopen Part** and **Reopen Assembly**.

6.4 In the **Assembly Navigator** right-click *asm3_ra_plate_1* and select **Close⟶Reopen Part**.

An information window pops up informing you that the *asm3_ra_plate_1* has been successfully reopened.

6.5 Close the information window.

6.6 In the **Assembly Navigator** right-click *asm3_ra_sub_assm_1* and select **Close⟶Reopen Assembly**.

An information window pops up informing you that the *asm3_ra_angle_3*, *asm3_ra_pulley_1*, and *asm3_ra_sub_assm_1* have been successfully reopened.

6.7 Close the information window. And Close all parts; do not save.

Edit Suppression State enhancement

1. **Open the part**

   Open *asm3_suppress_assembly_1* loading all components with **Use Partial Loading**.

   1.1 On the **Standard** toolbar, click **Open**  .

   1.2 In the **Open** dialog box, click **Options**.

   The **Assembly Load Options** dialog box appears.

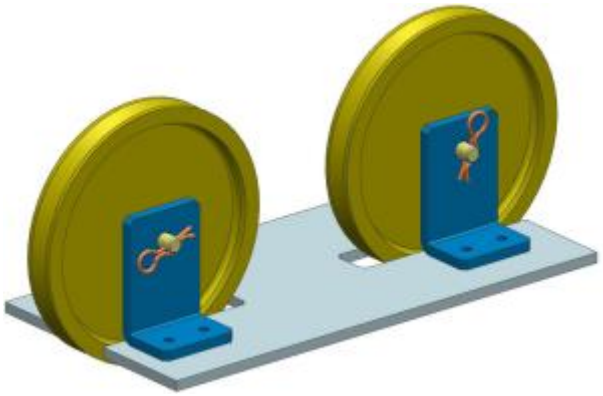
   1.3 In the **Part Versions** group, expand the **Load** list and select **From Folder**.

   1.4 In the **Scope** group, expand the **Load** list and select **All Components**.

   1.5 Ensure that the **Use Partial Loading** check box is selected.

   1.6 Click **OK**.

   1.7 Open *asm3_suppress_assembly_1*.

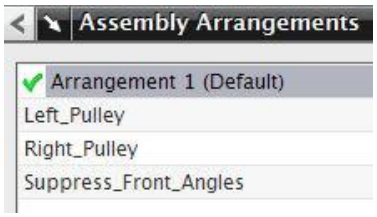


2. **View the arrangements**

   2.1 Select **Assemblies**⟶**Arrangements**.

   2.2 Observe that there are 3 additional arrangements.

   However, at this time they are only copies of Arrangement 1 that have been renamed.



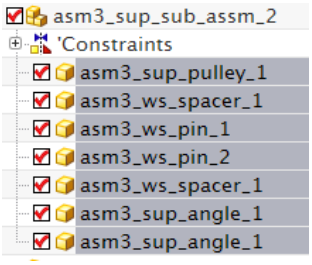
   2.3 Click **Close**.

3. Edit the left pulley arrangement

3.1 Open the **Assembly Navigator** and pin it open.

3.2 Expand *asm3_sup_sub_assm_2*.

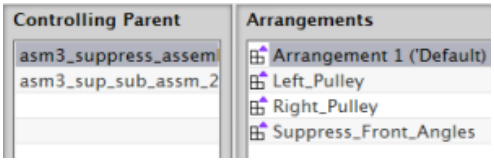
3.3 Select all of the components of *asm3_sup_sub_assm_2*.



3.4 On one of the selected components, right-click and select **Suppression**.

3.5 In the **Suppression** dialog in the **Controlling Parent** group, select *asm3_suppress_assembly_1*.

3.6 Observe that the **Arrangements** group is now populated with arrangements that you had reviewed earlier.



3.7 In the **Arrangements** group, select **Left_Pulley**.

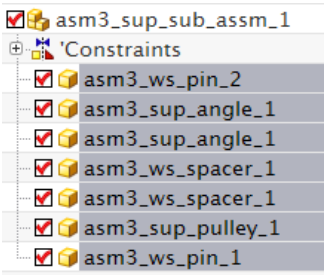
3.8 In the **State** group, click **Always Suppressed**.

3.9 Click **OK**.

4. Edit the right pulley arrangement

4.1 In the **Assembly Navigator** expand *asm3_sup_sub_assm_1*.

4.2 Select all of the components of *asm3_sup_sub_assm_1*.



4.3 On one of the selected components, right-click and select **Suppression**.

4.4 In the **Suppression** dialog in the **Controlling Parent** group, select *asm3_suppress_assembly_1*.

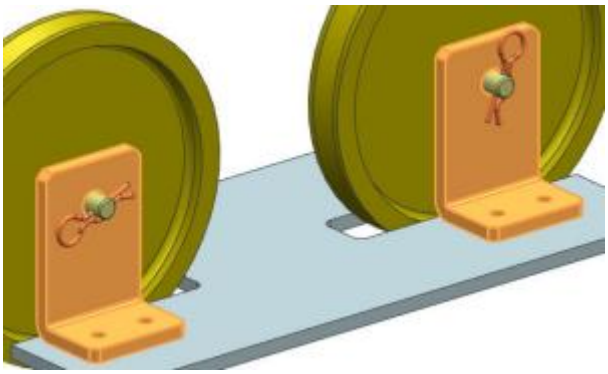
4.5 In the **Arrangements** group, select **Right_Pulley**.

4.6 In the **State** group, click **Always Suppressed**.

4.7 Click **OK**.

5. Edit the Suppress Front Angles arrangement

5.1 In the graphics window, select the 2 angle components as indicated below.



5.2 In the graphics window, right-click one of the highlighted components and select **Suppression**.

5.3 In the **Suppression** dialog box in the **Arrangements** group, select **Suppress_Front_Angles**.

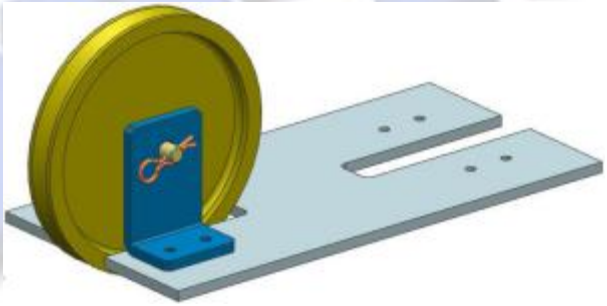
5.4 In the **State** group, click **Always Suppressed**.

5.5 Click **OK**.

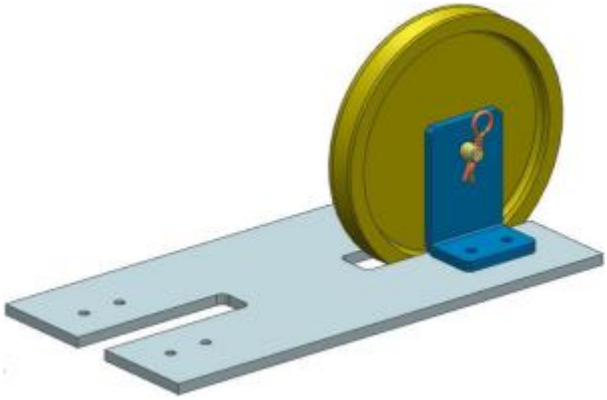
6. Test the arrangements

6.1 Select **Assemblies**⟶**Arrangements**.

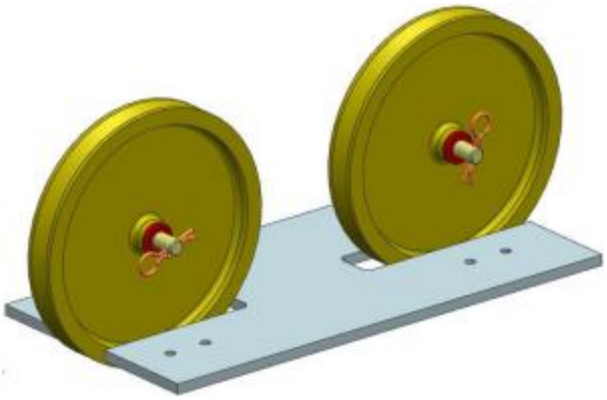
6.2 Double-click **Left_Pulley**.



6.3 Double-click **Right_Pulley**.



6.4 Double-click **Suppress_Front_Angles**.



6.5 Click **Close**.

6.6 Close all parts; do not save.

Synchronize Links

1. Open the part

   1.1 Choose **Tools⟶Update⟶Interpart Update**.

   1.2 Verify that **Delay Geometry, Expressions, and PMI** is selected.

   1.3 Choose **File⟶Options⟶Assembly Load Options**.

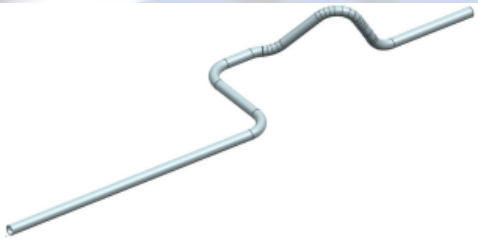
   1.4 In the **Part Versions** group, expand the **Load** list and select **From Folder**.

   1.5 In the **Scope** group, expand the **Load** list and select **All Components**.

   1.6 Ensure that the **Use Partial Loading** check box is selected.

   1.7 Click **OK**.

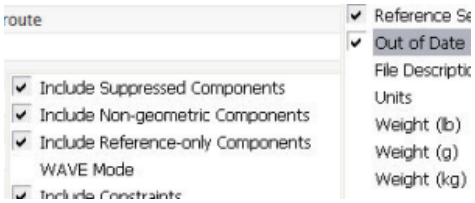
   1.8 Open *asm2_epr_assm*.



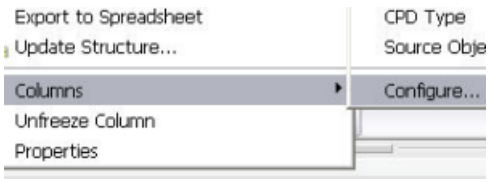
2. Add a column to the Assembly Navigator

   2.1 Open the **Assembly Navigator**.

   2.2 Below the last component node, right-click and choose **Columns⟶Out of Date**.

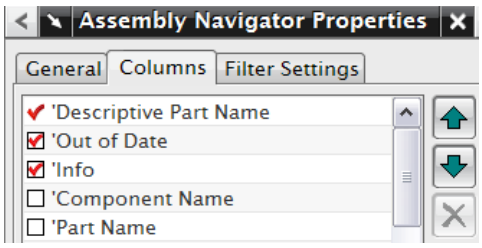


   2.3 Below the last component node, right-click and choose **Columns⟶Configure**.



2.4 In the **Assembly Navigator Properties** dialog box on the **Columns** tab, select **Out of Date**.

2.5 Click the **Move Up** button until **Out of Date** is second from the top.



2.6 Click **OK**.

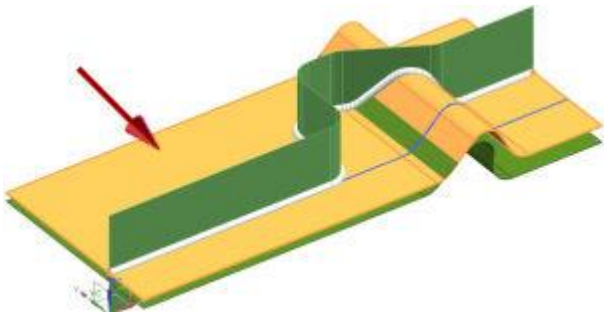
3. **Edit the guide curve**

3.1 In the **Assembly Navigator**, double-click *asm2_pipe_route* to make it the work part.

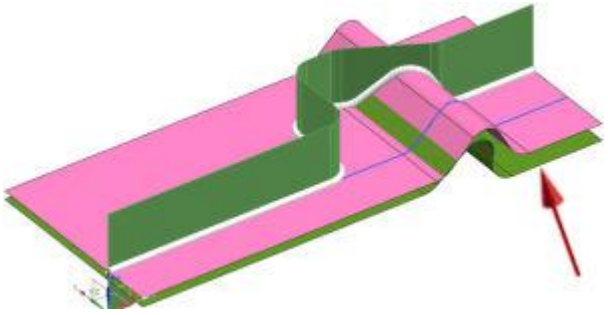
3.2 In the **Part Navigator**, double-click **Intersection Curves (7)**.

3.3 In the Selection bar, insure that the **Face Rule** is set to **Tangent Faces**.

3.4 Hold the Shift key and select one of the highlighted faces as indicated below.



3.5 Select the lower sheet solid as indicated below.



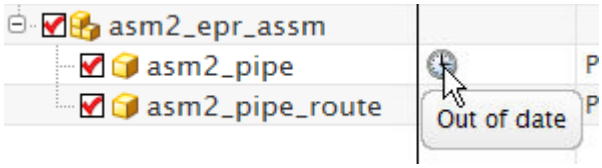
3.6 Click **OK**.

4. **Return to the assembly**

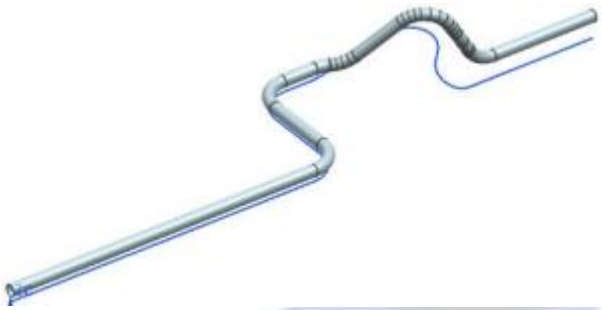
4.1 In the **Assembly Navigator**, double-click *asm2_epr_assm*.

4.2 In the **Assembly Navigator**, right click *assm2_pipe* and select **Open⟶Component Fully**.

4.3 Observe in the **Assembly Navigator** that there is an **Out of Date** icon present for the *asm2_pipe* component.



4.4 Observe in the graphic window that the *asm2_pipe_route* and *asm2_pipe* components are no longer coaxial.



5. **Update the pipe component**

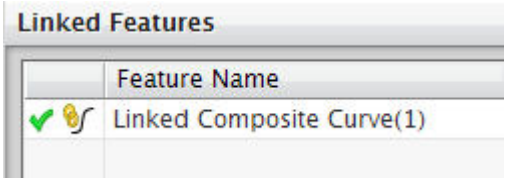
5.1 In the **Assembly Navigator**, double-click *asm2_pipe* to make it the work part.

5.2 Choose **Assemblies⟶WAVE⟶Synchronize Links**.

5.3 In the **Synchronize Links** dialog box, select **Linked Composite Curve(1)**.

5.4 Click **Synchronize Linked Feature**  .

5.5 In the **WAVE Geometry Linker** dialog box, click **OK**; the feature is updated.

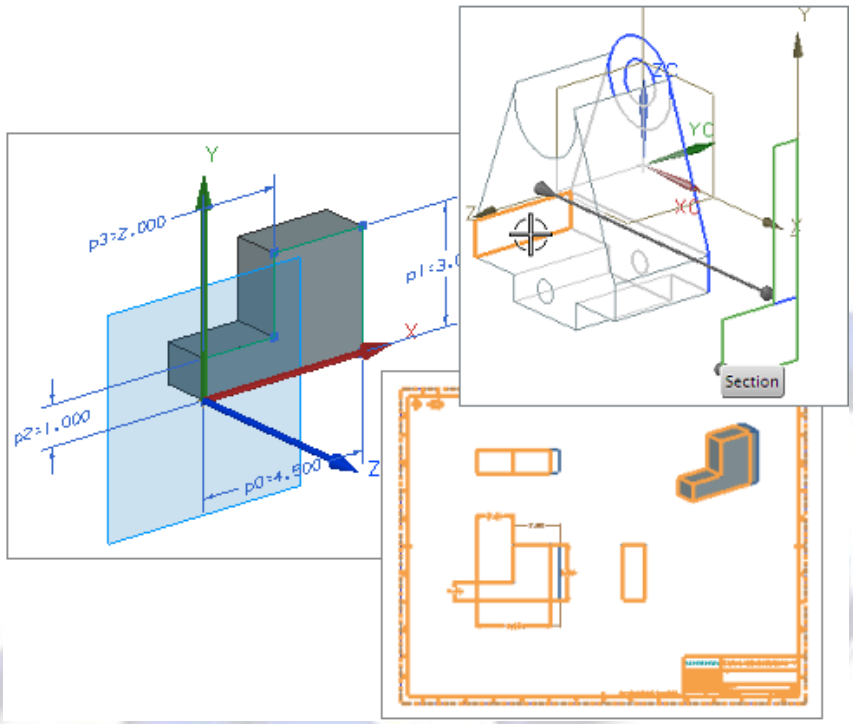


5.6 In the **Synchronize Links** dialog box, click **Close**.

5.7 In the **Part Navigator**, right-click **Tube (2)** and select **Make Current Feature**.

5.8 Close all parts; do not save.

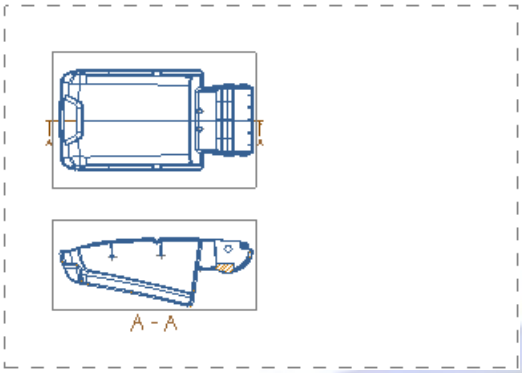
NX 8 Drafting Transition



Borders & Zones

1. **Open the part and set the work layer**

1.1 Open *drf7_borders_and_zones_dwg*.



1.2 On the **Utility** toolbar, click **Layer Settings**  .

1.3 In the **Work Layer** box, type **256** and then press Enter.

1.4 Click **Close**.

2. **Apply standard zones to the drawing sheet**

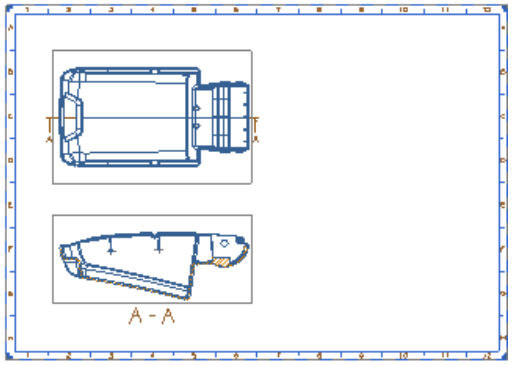
2.1 On the **Drawing Format** toolbar, click **Borders and Zones**  .

2.2 In the **Borders** group, in the **Width** box, type **15**.

2.3 In the **Zones** group, make sure the **Method** is set to **Standard**.

2.4 In the **Zone Labels and Markings** group, in the **Label Height** box, type **10**.

2.5 Click **OK**.



Sheet “Sheet 1” Work

The **Standard** method places equally spaced zones along the horizontal and vertical borders of the drawing sheet.

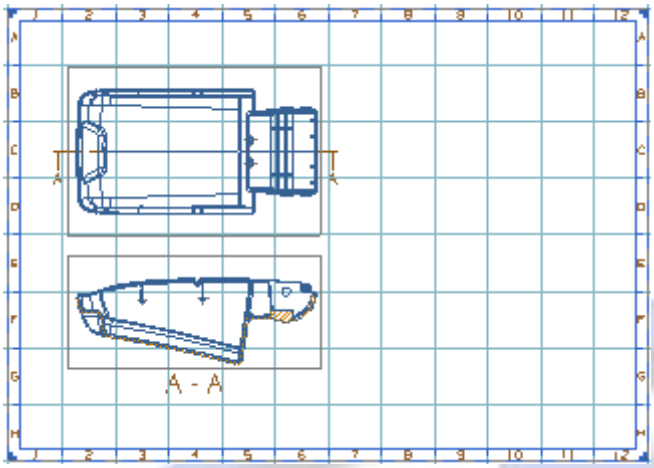
3. **Turn the sheet zone grid on**

3.1 Choose **Preferences**⟶**Drafting**.

3.2 At the bottom of the **General** tab, from the **Grid Settings** group, click **Use Sheet Zone Grid**.

3.3 Click **OK**.

3.4 In the **Part Navigator**  , right-click the **Drawing** node and select **Grid**.

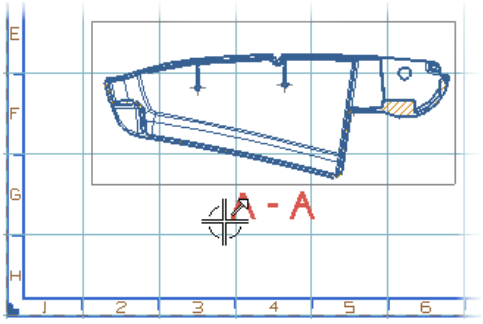


Sheet "Sheet 1" Work

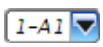
Associative grid lines appear on the drawing sheet. The major grid spacing coincides with the zone markings on the border.

4. **Convert the section view label to a sheet and zone callout**

4.1 Double-click the **View and Section Label**, or right-click it and choose **Edit View Label**.

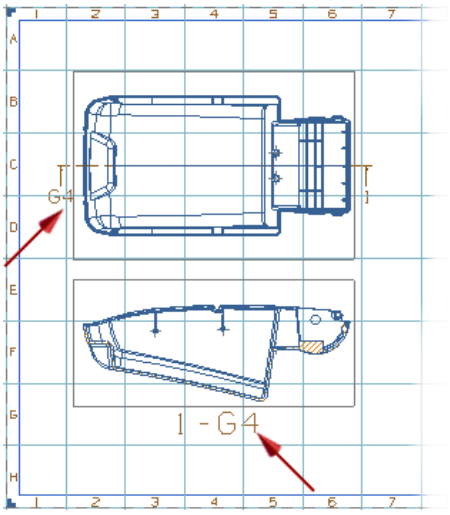


4.2 In the **View Label** group, from the **Letter Format** list, select **1–A1**.



4.3 From the **Reference to Show** list, select **Sheet and Zone**.

4.4 Click **OK**.



The zone information appears in the label of the section view, and on the section line symbol in the parent view. The label's alignment position determines its zone location.

5. **Edit the border and zone layout**

5.1 Click **Borders and Zones** .

5.2 In the **Zones** group, from the **Method** list, select **Custom**.

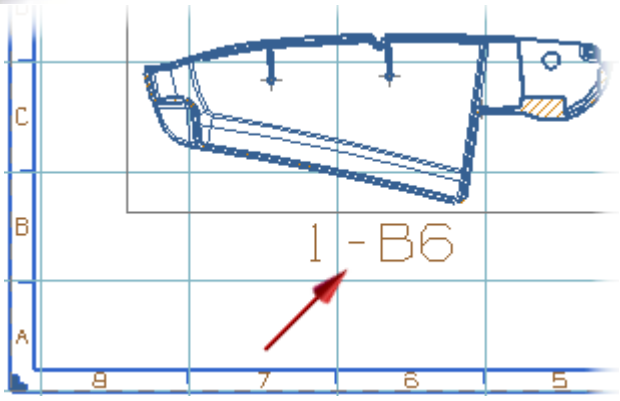
The **Custom** method lets you configure your own border and zone layout.

5.3 In the **Horizontal Size** box, type **100**.

5.4 From the **Origin** list, select **Bottom Right**.

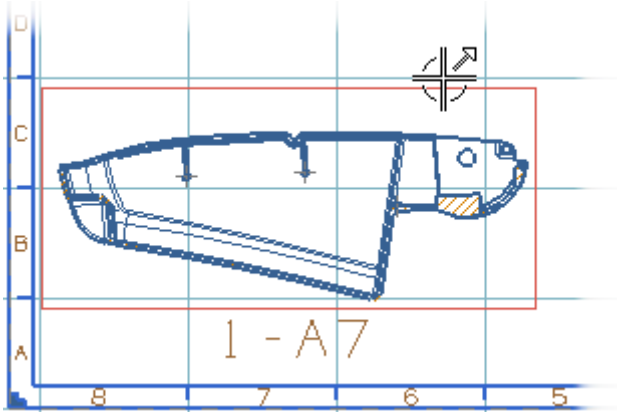
5.5 Click **OK**.

Zone 'A1' is placed at the bottom right of the drawing border.



The section view label updates to reflect the new zone configuration on the drawing sheet.

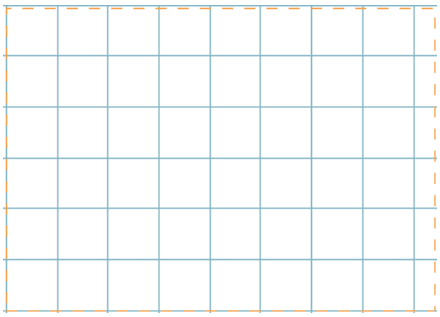
5.6 Drag the section view to a new location on the drawing sheet.



The label updates as you move the view from one zone to another.

6. **Open Sheet 2 and add borders and zones to it**

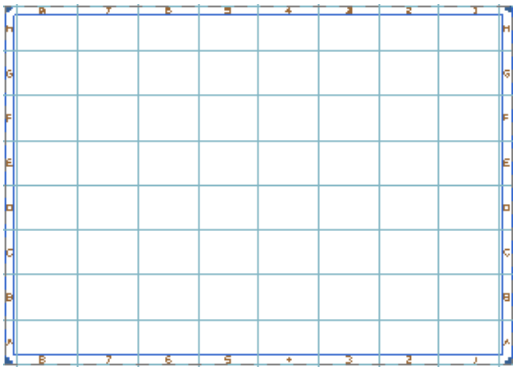
6.1 In the **Part Navigator**  , double-click the **Sheet "Sheet 2"** node to open it.



Sheet "Sheet 2" Work

6.2 Click **Borders and Zones** 

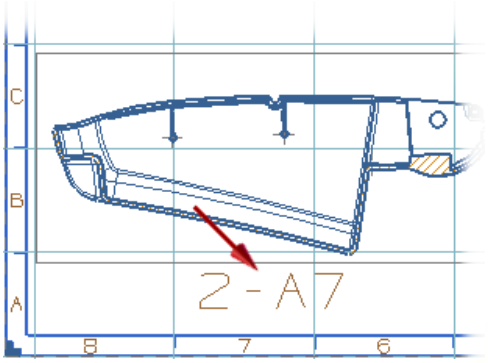
6.3 Accept the current border and zone settings by clicking **OK**.



Sheet "Sheet 2" Work

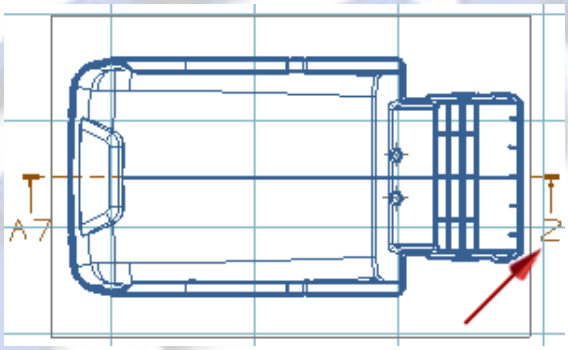
7. **Move the section view to Sheet 2**

In the **Part Navigator**  , use your mouse to drag the '**Section "SX@2" A** node down to the '**Sheet "Sheet 2"(Work-Active)** node.



Sheet "Sheet 2" Work

The section view moves to the same relative location on Sheet 2, and the section view label updates. If you look at Sheet 1, the section line symbol on the parent view also reflects the corresponding update.



8. **Add a note with a zone callout in it**

8.1 On the **Annotation** toolbar, click **Note**  .

8.2 In the **Text Input** box, type:

Zone

8.3 In the **Symbols** group, from the **Category** list, select **Relationships**.

8.4 Click **Insert Sheet Zone**  .

8.5 In the **Sheet Zone Reference** dialog box, from the **Attributes** list, select both:

Sheet Number
Zone Reference

8.6 From the **View Reference** group, select the **Show All Views in Drawing** check box.

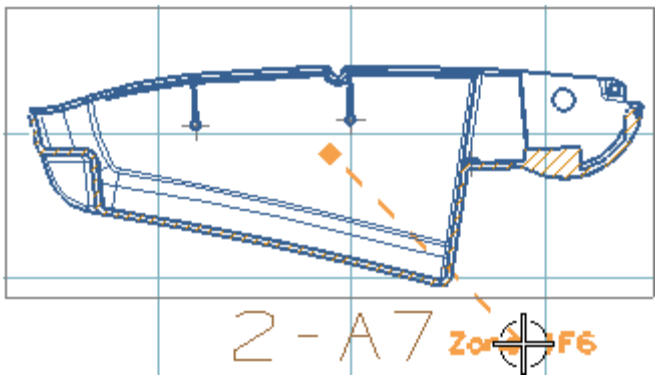
8.7 From the **Views List**, select:

Top@1

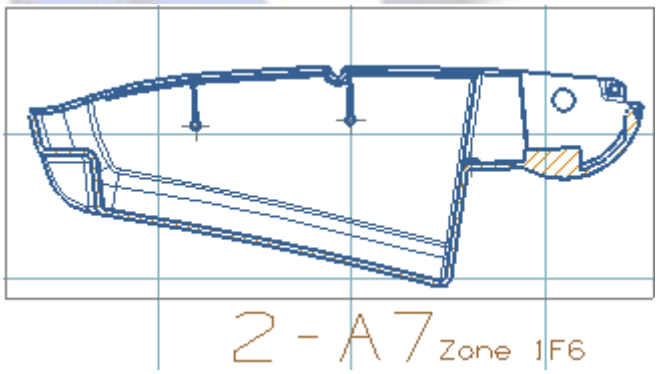
8.8 Click **OK**.

The character sequence for the sheet and zone callouts appears in the **Text Input** box.

8.9 Drag the note over the view so that an associative helper line extends from it to the center of the view.



8.10 Click to place the note next to the view label.

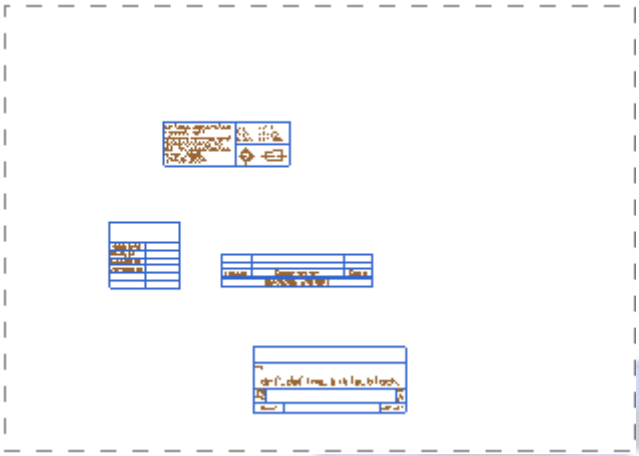


8.11 Close all parts.

Create and edit a title block

1. Open the drawing and add borders and zones to it

1.1 Open **drf7_define_title_block**.



Sheet “Sheet 1” Work

1.2 On the **Utility** toolbar, click **Layer Settings**  .

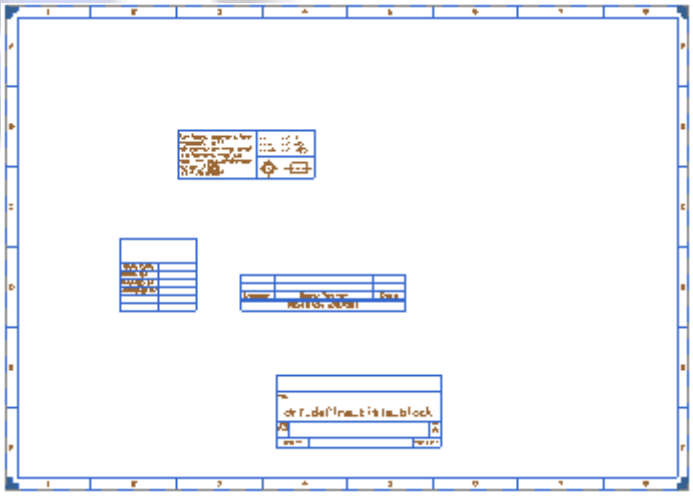
1.3 In the **Work Layer** box, type **256**, and then press Enter.

1.4 Click **Close**.

1.5 On the **Drawing Format** toolbar, click **Borders and Zones**  .

1.6 On the dialog rail, click **Reset**  .

1.7 Click **OK**.



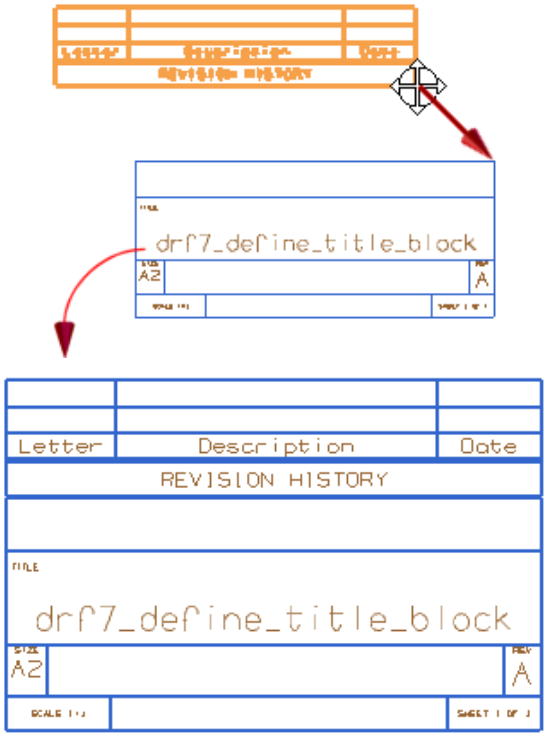
Sheet “Sheet 1” Work

A set of associative borders and zones is placed on Layer 256.

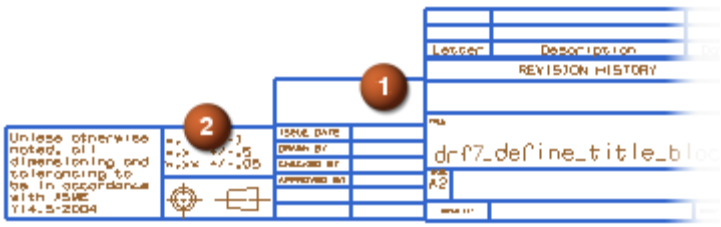
2. Move the tabular notes

2.1 Move your cursor over the origin of the revision block.

2.2 Drag the revision block to the upper right corner of the main title block note.



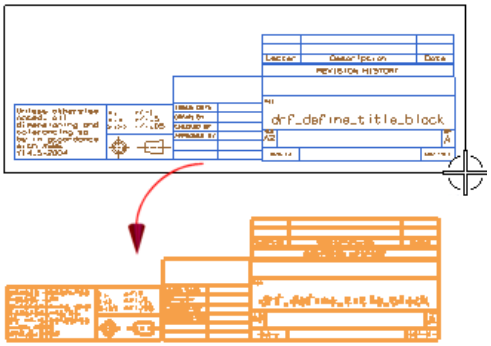
2.3 Drag the signature (1) and tolerance block (2) to the positions shown.



3. Create the title block

3.1 On the **Drawing Format** toolbar, click **Title Block**  .

3.2 Rectangle select all of the tabular notes.

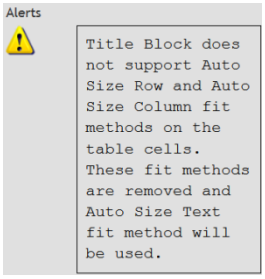


Their contents appear in the **Cell Properties** list.

List

Lock Status	Content Type	Value	Label
	Text		Label1
🔒	Mixed	TITLE	Label2
🔒	Mixed	SIZE A2	Label3

Observe that some of the cells of title block are automatically locked. For instance, **LABEL2** and **LABEL3** are automatic text attributes and cannot be edited, thus they are locked. An alert also appears in the graphics window:



It informs you that only Auto Size Text can be applied to the cells of the custom title block. All other fit methods are ignored.

3.3 Scroll down to the bottom of the list.

List

Lock Status	Content Type	Value	Label
	Text	Unless othe	Label33
	Text	x +/- 1x.x	Label34
🔒	Symbol		Label35

3.4 The text in **Label33** is currently unlocked and can be edited.

3.5 Click **OK** to create the custom title block.

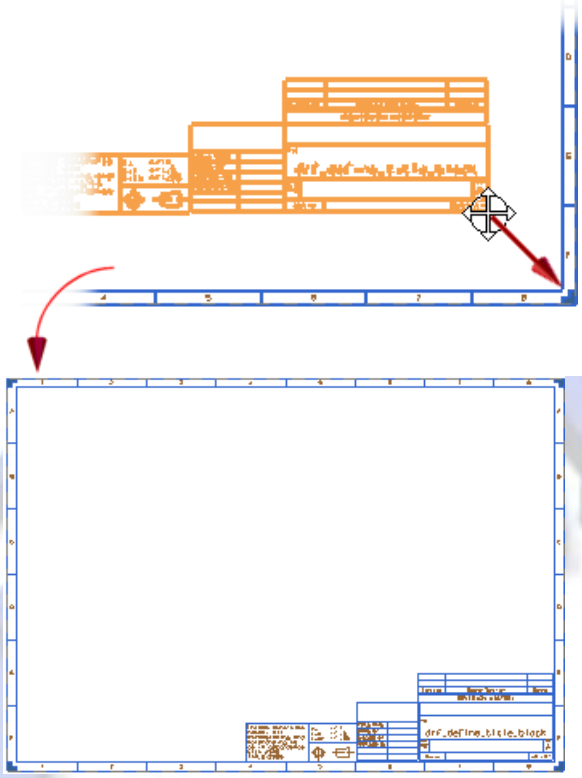
4. **Drag the title block to its position on the drawing sheet**

4.1 Right-click the title block and choose **Style**.

4.2 From the **Alignment Position** list, make sure **Bottom Right** is selected.

4.3 Click **OK**.

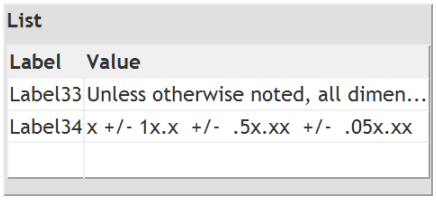
4.4 Drag the title block to the lower right corner of the drawing sheet.



5. **Edit the title block**

5.1 Right-click the title block and choose **Populate**.

5.2 Scroll down to the bottom of the list.

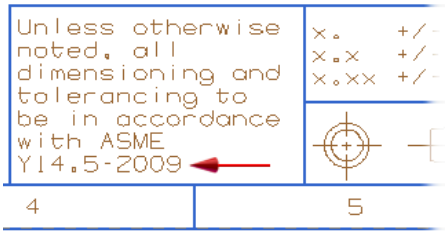


5.3 Select the **Label33** row.

5.4 Click  .

5.5 In the **Text** editor, scroll to the end of the character string and replace **–2004** with **–2009**.

5.6 Click **OK**.



The note updates.

5.7 In the **Populate Title Block** dialog box, click **Close**.

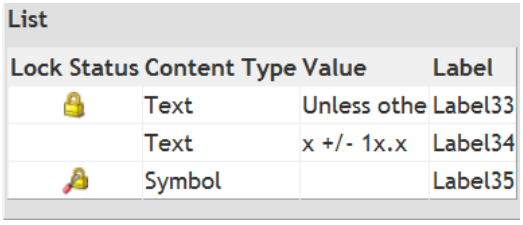
6. Lock the contents of the edited cell

6.1 Right-click the title block and choose **Edit Definition**.

6.2 Select the **Label33** row.

6.3 Select the **Lock** check box.

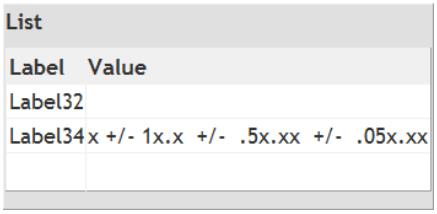
The lock symbol appears in the cell row.



6.4 Click **OK**.

6.5 Right-click the title block and choose **Populate**.

Label33 no longer appears in the list and cannot be edited.



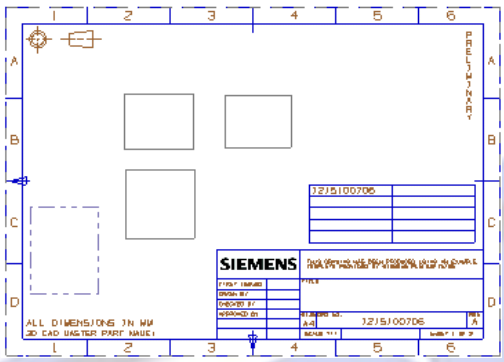
6.6 Click **Close**.

6.7 Close all parts.

Mark as Template

The **Mark as Template** command creates a sheet or drawing template based on the configuration of the currently open drawing sheet. The template can include associative borders and zones, views, notes, template regions, symbols, and custom title blocks.

Marking a drawing as a template gives you access to the template regions and rules functions as well as allows you to choose to add the template to your collection of sheet and drawing templates.



If you elect to save the template in your template directory, you must specify a presentation name, description, template type, and the .pax file location.

Three different template types are available:

Sheet

Creates a sheet template from the current drafting part. A sheet template is used to add a new sheet to an existing drawing.

Reference Existing Part

Creates a master model drawing template from the current drafting part. A master model drawing template is used to create a separate drafting part that includes the current model as a component.

Standalone

Creates a 2D drawing template. A 2D template is used to create a standalone drawing that does not reference a master model part.

When you save the template part, the specified .pax file is created or updated and the template is displayed by its presentation name on the Resource bar palette.

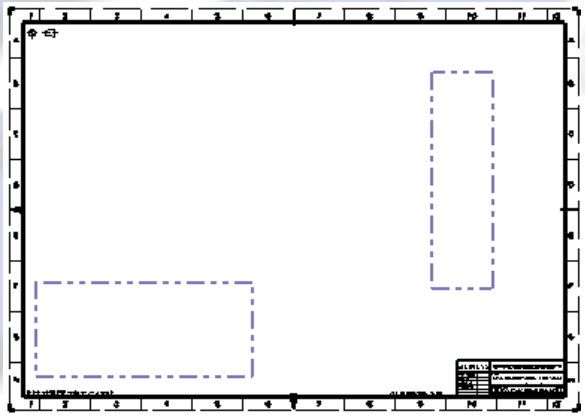
Note : You must have write permission to the .pax file and file directory to add a new template or update an existing template.

Template Region

Template region is a rectangular area on a drawing template designated to accept only certain types of drafting objects. The **Template Region** command defines the origin and the rectangular dimensions of the area on the drawing template and the specific types of drafting objects that can be contained therein.

You can designate a template region as a note, symbol, table, or view type region. This designation determines the type of drafting object that can be added to the template region. You can apply Knowledge Fusion rules to template regions to govern the behavior of objects in them when the drawing template is incorporated into another part.

Note : Although this command is only available for drawing templates, it is not necessary to create template regions in order to add drafting objects to the drawing template. You can add views, custom symbols, title blocks, and notes to your drawing template without template regions.



Phantom lines denoting the boundary of a template region

When the contents of a template region grow beyond its borders, the region behaves in one of two ways:

As a growing region

Contents that extend beyond the template region's boundary continue into other template regions of the same object type. Depending on how you set the Continuation for the template region, the overflow extends to other template regions of the same type on the same sheet, or to template regions of the same type on another sheet.

As a fixed region

Contents that extend beyond the template region are displayed outside the boundary of the region and do not continue to any other region.

Note : View regions are always fixed regions.

After you create a template region, you can re-size it, move it, or delete it in the drawing template.

However, after you import a region as part of a drawing template, you cannot edit it, move it, or delete it.

Priority and Content Control

If content from two growing regions grow into each other, the region with the larger priority number will move to the continuation area. You may also control whether the entire contents of the lower priority region move to another region, or if only the overlapping portion moves.

Customer Defaults

Customer defaults govern the display of template regions. The defaults are contained in the **Customer Defaults** dialog box, on the **Drafting⟶Drawing Automation ⟶General** tab:

- **Display Region in Non-Template Part** — Displays the template region's border when a drawing template is imported into another part.
- **Display Region Label** — Displays the template region's name, type, and the rule file name at the center of each region.

You can also set color, font, and width settings for region borders here.

These options also control the default setting for the template region options on the **General** tab of the **Drafting Automation Preferences** dialog box.

Support for Standard Fonts

Standard font types available in the FreeType Font library are supported in the PMI and Drafting environments.

Note : Although the FreeType Font library supports non scalable and other font types, only scalable TrueType, OpenType, and PostScript fonts are supported.

When setting your character font, it is important to remember the following points:

- Existing NX font types are still available and are located in a directory set by the **UGII_CHARACTER_FONT_DIR** environment variable.
- Standard Fonts files are found in your normal font directories, usually in the *C:\Windows\Fonts* on Windows or in a configuration file on Unix.
- You can use custom fonts by placing the font files in a directory referenced by the **UGII_STANDARD_FONT_DIR** environment variable.
- NX will search for the character font in the following order:
  1. Standard font location (that is, *C:/Window/Fonts*).
  2. The directory specified by **UGII_STANDARD_FONT_DIR**.
  3. The directory specified by **UGII_CHARACTER_FONT_DIR**.
- A default font is used when a font cannot be found. The default font can be changed, but is initially set to Arial Unicode MS. If Arial Unicode MS is not found, the default font is set to Tahoma. If Tahoma is not found, then the default font is set to Arial.

Why should I use it?

Use Standard Fonts to replace, enhance, or supplement your current set of drafting and PMI character fonts.

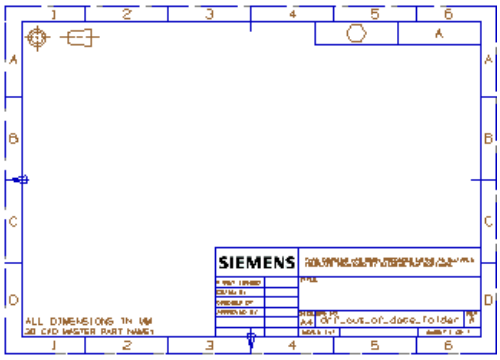
Where do I find it?

All fonts found in standard directories and in the custom directory are automatically available in customer defaults, preference dialog boxes, style dialog boxes, and any other dialog box that lets you specify a character font.

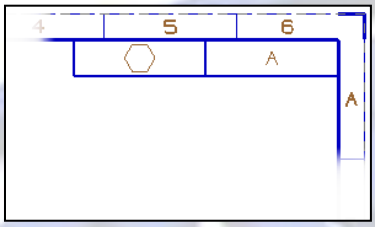
Out of Date Folder

1. Open the part and review the drawing

   1.1 Open *drf7_out_of_date_folder*.



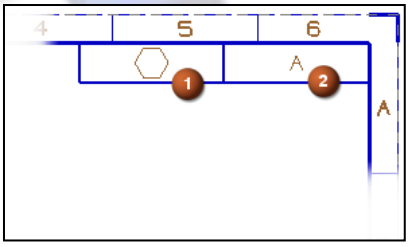
   1.2 Zoom in on the tabular note at the upper right corner of the part.



   The note contains:

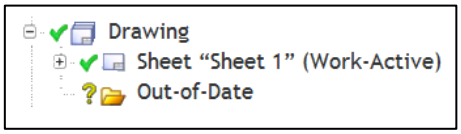
   1.2.1 A custom symbol.

   1.2.2 The part attribute DB_PART_REV.

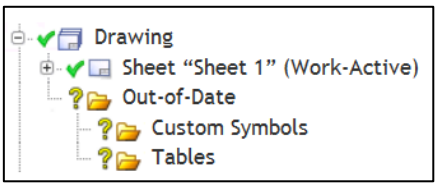


2. Review the Out-of-Date folder

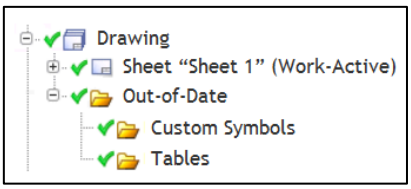
   2.1 On the Resource bar, click the Part Navigator tab.



   2.2 Right-click the Out-of-Date folder and choose Refresh.



2.3 Folders for the custom symbol and tabular note are added under the **Out-of-Date** folder.



2.4 Right-click the **Out-of-Date** folder and choose **Refresh All**.

2.5 NX performs an out of date check on the contents of the selected folders and verifies that the objects in them are up to date. It then marks each folder with the green check mark symbol.

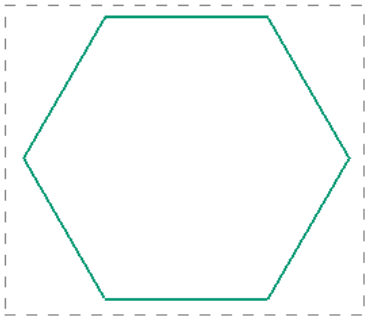
3. Edit the custom symbol definition

3.1 On the Resource bar, click the **Reuse Library** tab.

3.2 Expand the **Custom Symbol Library** node and select the **Part Symbols** folder underneath it.

3.3 On the **Member Select** panel, right-click **Hex** and choose **Edit**.

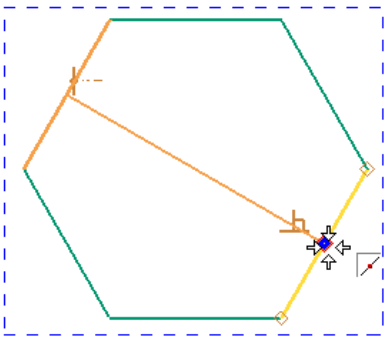
You are placed in the symbol editor mode.



Sheet "Symbol Editor" Work

3.4 On the **Sketch Tools** toolbar, click **Line** .

3.5 Use **Mid Point** to construct a line diagonally from one side of the hexagon to the other.



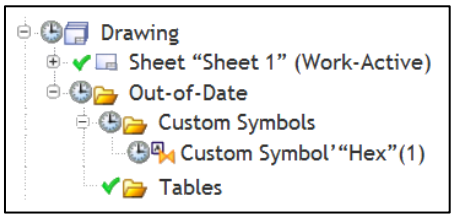
Sheet "Symbol Editor" Work

3.6 On the **Symbol** toolbar, click **Finish Custom Symbol Edit** .

3.7 In the **Finish Custom Symbol Edit** dialog box, click **OK**.

4. Update the symbol from the Out-of-Date folder

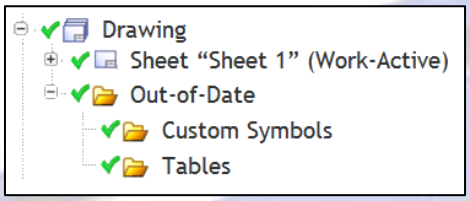
4.1 In the **Part Navigator**, right-click the **Out-of-Date** folder and choose **Refresh All**.



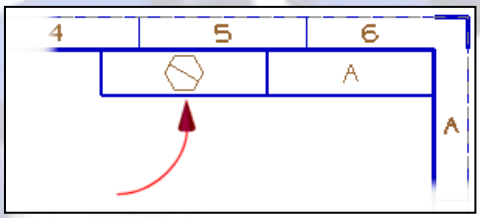
The custom symbol is marked out of date.

4.2 Right-click the **Out-of-Date** folder and choose **Update All**.

The **Out-of-Date** and **Custom Symbols** folder underneath it are marked up to date, and the object is removed from the **Custom Symbols** folder.

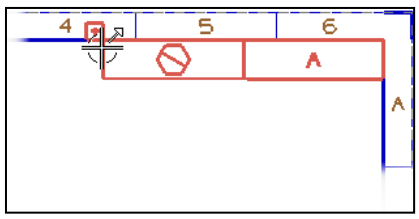


The custom symbol is likewise updated in the tabular note.



5. Edit the part attribute

5.1 Right-click the tabular note and choose **Style**.



5.2 Click the **Tabular Note** tab.

5.3 Clear the **Automatic Update** check box.

This prevents the tabular note from automatically updating as you edit the part attribute.

5.4 Click **OK**.

5.5 Choose **File⟶Properties**.

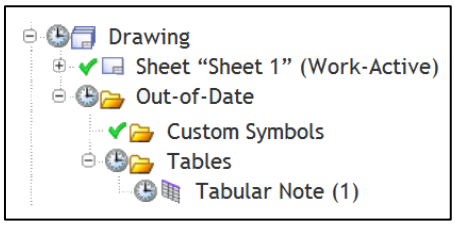
5.6 On the **Attributes** tab, select the *DB_PART_REV* row.

5.7 In the **Value** box, overstrike **A**, and then type **B**.

5.8 Click **OK**

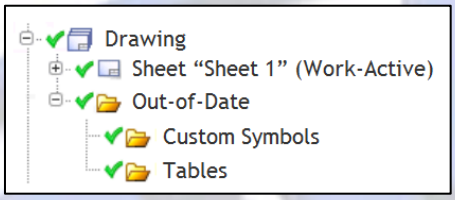
6. **Update the attribute value from the Out-of-Date folder**

6.1 Right-click the **Out-of-Date** folder and choose **Refresh All**.

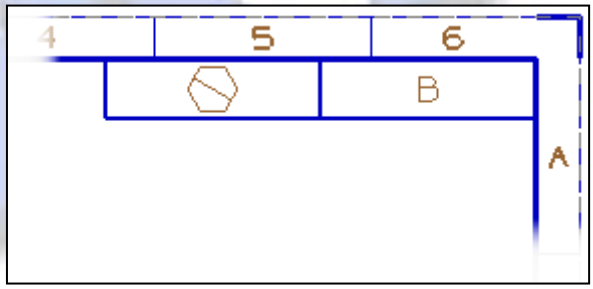


The tabular note is marked out of date.

6.2 Right-click the **Out-of-Date** folder and choose **Update All**.



NX updates the attribute value in the tabular note, and then marks all folders up to date.



6.3 Close all parts.

View Creation Wizard

1. **Review your drafting preferences**

   1.1 Choose **Preferences**⟶**Drafting**.

   1.2 On the **General** tab, from the **Model-based Drawing Workflow** group, make sure the following options are selected:

   ☑ Automatically Start View Creation

   ◉ View Creation Wizard

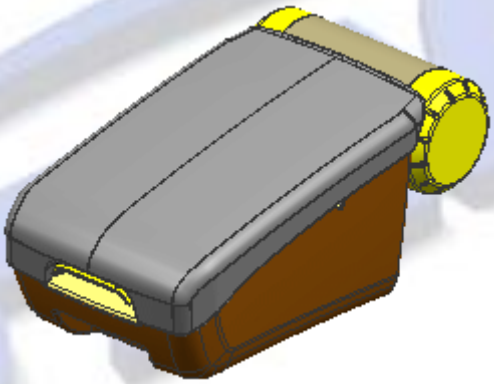
   ○ Base View Command

   Do not be concerned about selecting any other options as you do not need them for this activity.

   1.3 Click **OK**.

2. **Open an assembly and create a drawing part**

   2.1 Open *drf7_armrest_assm*.



   2.2 On the **Standard** toolbar, click **New**  .

   2.3 Click the **Drawing** tab.

   2.4 In the **Filters** group, make sure

   Relationship = **Reference Existing Part**

   Units = **Millimeters**

   2.5 From the **Templates** list, select **A0 – Drawing**.

   2.6 Under **New File Name**, make sure:

   Name = xxx_armrest_assm_dwg

   2.7 **Under** Part to create a drawing of, **make sure**:

   Name = drf7_armrest_assm

   2.8 **Click** OK.

   2.9 **Click** Close **in the** Populate Title Block **dialog box**.

3. ect the model and view options

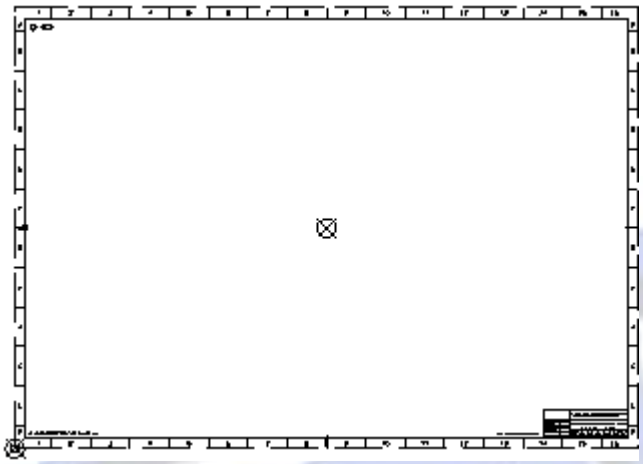
3.1 On the **Part** page, from the **Loaded Parts** list, make sure *drf7_armrest_assm.prt* is selected.

3.2 Click **Next** to advance to the **Options** page.

3.3 From the **View Boundary** list, make sure **Automatic** is selected.

3.4 Clear the **Show Centerlines** check box.

3.5 From the **Preview Style** list, select **Border**.



Sheet “Sheet 1” Work

3.6 Click **Next** to open the **Arrangement** page.

4. Select the arrangement and orientation for the parent view

The assembly was saved with the **closed** arrangement displayed. You can accept this arrangement for your assembly views by leaving it selected in the **Arrangement** list.

4.1 Click **Next** to advance to the **Orientation** page.

4.2 From the **Model Views** list, select **Front**.

The parent view serves as the base view for the other orthographic views in the layout.

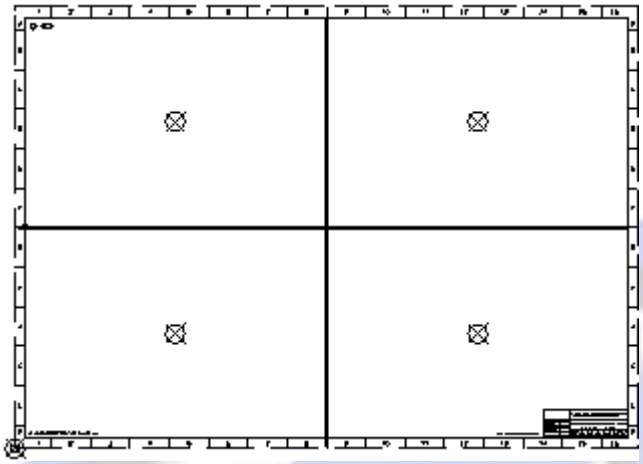
4.3 Click **Next** to advance to the **Layout** page.

5. Construct the view layout

On the **Layout** page, the middle button denotes the parent view you selected from the **Orientation** page.

Select the view layout indicated below.

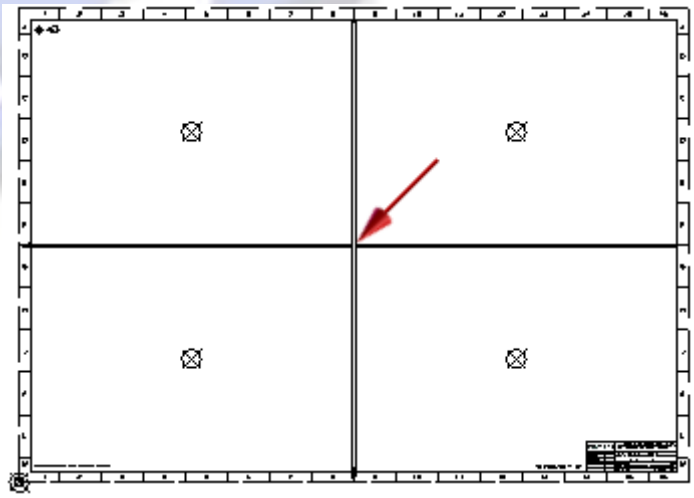
The preview on the drawing sheet updates as you select view buttons.



TIP : If you need to turn a view button off, click it a second time. You can turn off all but the parent view.

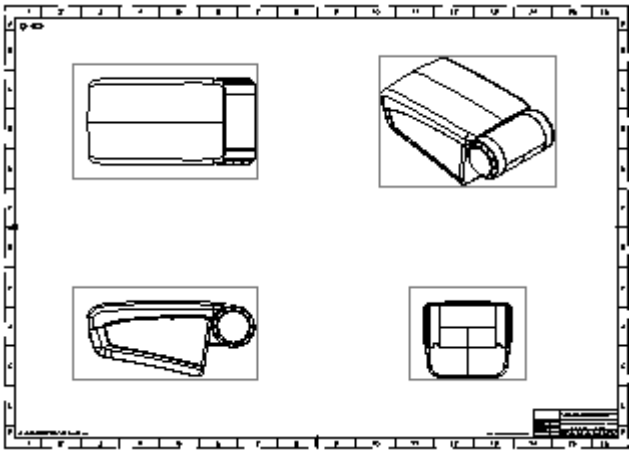
6. Place the view layout on the drawing sheet

Notice that the **Automatic** placement option centers the view layout on the drawing sheet.



If you accept the current view placement now, your views would be placed on the drawing sheet at 1:1 scale.

Click **Finish**.



eet “Sheet 1” Work

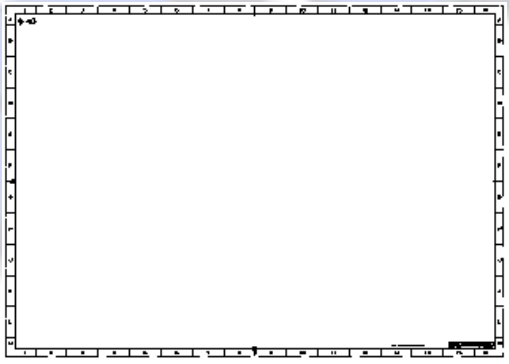
7. **Create a view layout showing an assembly arrangement**

7.1 On the **Drawing** toolbar, click **New Sheet**  .

7.2 In the **Sheet** dialog box, in the **Size** group, make sure **Use Template** is selected and **A0 – Size** is highlighted in the sheet list.

7.3 Click **OK** to create a new sheet.

7.4 Click **Close** to close the **Populate Title Block** dialog box.



Sheet “Sheet 2” Work

7.5 On the **Drawing** toolbar, from the **Add View Drop-down** list, select **View Creation Wizard**  .On the dialog rail, click **Reset**  .

7.6 On the **Part** page, from the **Loaded Parts** list, make sure *drf7_armrest_assm.prt* is selected.

7.7 Click **Next** to advance to the **Options** page.

7.8 From the **View Boundary** list, select **Manual**.

7.9 Make sure the **Auto-Scale to Fit** check box is selected.

The **Auto-Scale to Fit** applies the maximum scale to the views that still shows the geometry in its entirety within the specified area of the drawing sheet.

7.10 Click **Next** to open the **Arrangement** page.

8. **Select an assembly arrangement**

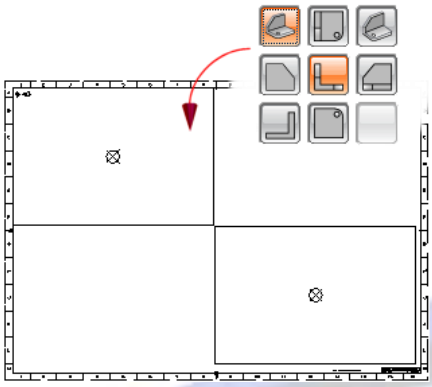
8.1 From the **Arrangement** list, select **fully_opened**.

8.2 Click **Next** to advance to the **Orientation** page.

8.3 From the **Model Views** list, select **Front**.

8.4 Click **Next** to advance to the **Layout** page.

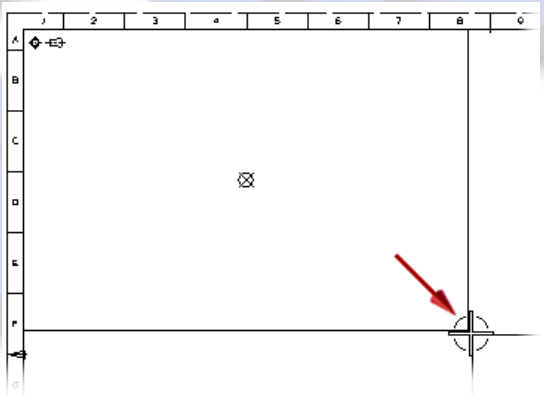
8.5 Select the view layout depicted below.



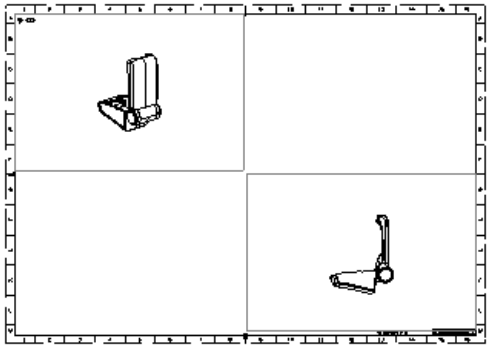
The preview on the drawing sheet updates as you select a different array of views.

8.6 In the **Placement** group, from the **Option** list, select **Manual**.

8.7 Move your cursor into zone F8.



8.8 Select the position.



Sheet “Sheet 2” Work

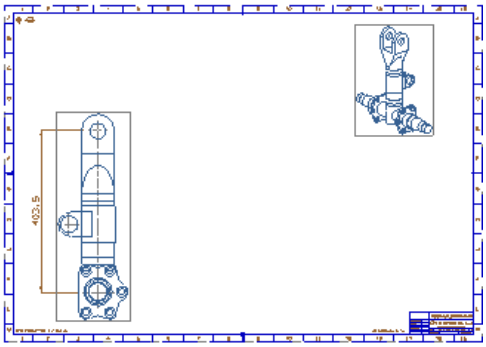
The views are placed on the drawing sheet showing the *fully_opened* arrangement and auto scaled to 1:2.

8.9 Close all parts.

View Break

1. **Open the part and review the current view break preferences**

   1.1 Open *drf7_view_break_dwg*.



Sheet “Sheet 1” Work

   1.2 Choose **Preferences**⟶**Drafting**.

   1.3 Click the **View Break** tab.

   1.4 In the **Workflow** group, make sure the **Propagate View Break** check box is selected.

   This ensures that any projected or section view contains the same view break as the parent view.

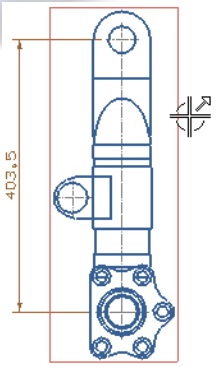
   1.5 Make sure the **Show Break Lines** check box is selected.

   This ensures that break lines are visible in all break views.

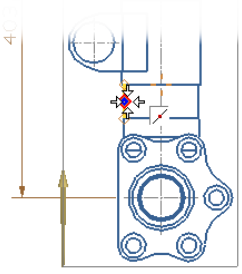
   1.6 Click **OK**.

2. **Start the view break**

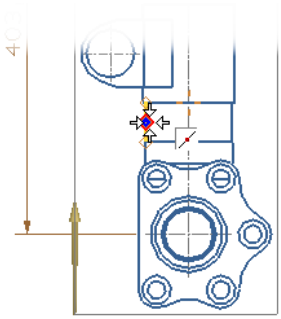
   2.1 Right-click the border of the left view and choose **Add View Break**.



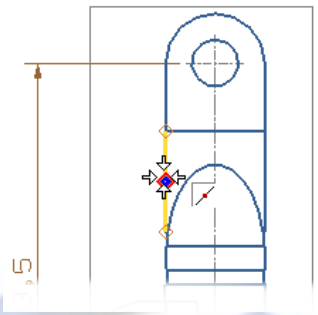
   2.2 In the **Type** list, make sure **Regular** is selected. The regular view break produces two break lines that foreshortens the view area in between them. The default direction vector points upward.



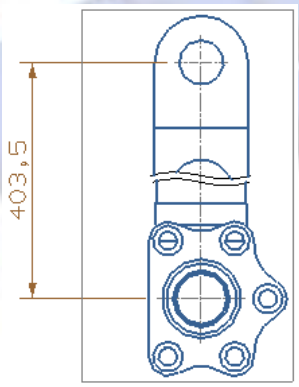
2.3 For **Break Line 1**, select the anchor point shown.



2.4 For **Break Line 2**, select the anchor point shown.

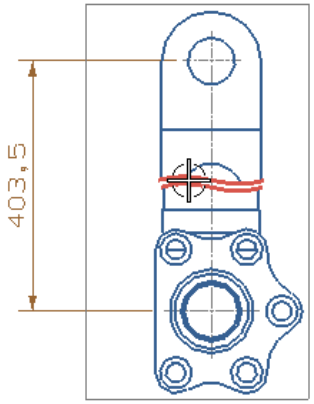


2.5 Click **OK**.



3. **Edit the style of the break**

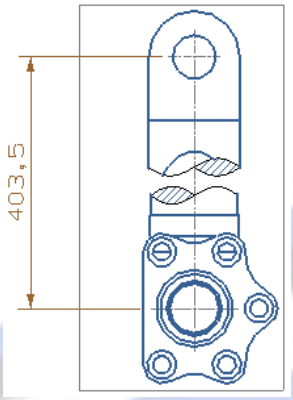
3.1 Right-click the break line in the view and choose **Edit**.



3.2 In the **Settings** group, set the following parameters:

Gap	=	26
Style	=	
Amplitude	=	16

3.3 Click **OK**.



Import AutoCAD Block

Use **Import AutoCAD Block** to create custom symbols by importing block definitions from AutoCAD DXF and DWG files. There are two methods for selecting AutoCAD files with blocks:

- Select an AutoCAD file and its list of blocks will be extracted.
- Select a folder and any AutoCAD files open and any list of blocks is extracted.

Users can mix any selection of files and folders, the blocks tree list will accumulate all selections. By default the block name will be used to define the symbol name, and the symbol part file name will be the symbol name with a .sym.prt extension (not applicable to part symbols).

The **Import Summary** will show the number of blocks processed, number of symbols created and their status.

To remove a new symbol, you can delete it after import like any other custom symbol.
You can recreate symbols for simple blocks as you migrate AutoCAD data to NX Drafting. However, in the case of more complex blocks or large block libraries, it is easier to use this functionality to create the NX symbol from the existing AutoCAD blocks. In the event of symbol name clashes, a number will be appended to the name.

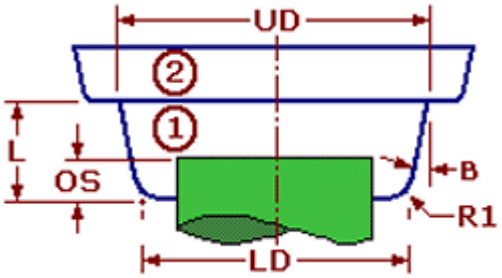
Where do I find it?

Application	Drafting
Prerequisite	The DXF / DWG Translator (dxfdwg kit) must be installed.
Menu	File⟶Import⟶AutoCAD Block
Resource bar	**Reuse Library**  tab ⟶ **Custom Symbol Library** ⟶ right-click a folder ⟶ **Import AutoCAD Block**

Activity: Create cutting tools

Estimated time to complete: 3-5 minutes

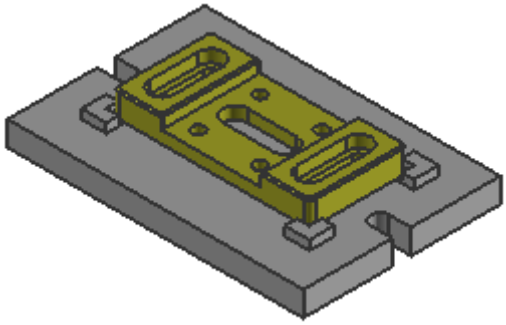
In this activity, you will create several milling tools and their tool holders.



Launch the Create cutting tools activity.

Create the Manufacturing setup

1. Open *mnf3_MFG_asmb_slotted_adjuster*.



2. On the **Standard** toolbar, click **New** or choose **File→New**.
3. Click the **Manufacturing** tab.
4. From the **Units** list, select **Inches**.
5. From the Templates list, select **General Setup**.
6. Click **OK**.

   The manufacturing assembly is created.
7. In the **Operation Navigator**, right-click in the background and choose **Machine Tool View**.
8. Right - click in the background again and choose **Expand All**.

   The machine tool view contains a carrier and pockets.
9. In the graphics window, right - click in the background and choose **Orient View→Front**.

Create a 1.0 diameter end mill with a tool holder

Since a mill is a good general purpose tool, it will be placed as tool number one. The diameter of the mill will be 1.000.

1. In the **Create** toolbar, click **Create Tool** or choose **Insert→Tool**.

   The **Type** is set to **mill_planar**.

2. In the **Create Tool** dialog box, in the **Tool Subtype** group, click **MILL**.
3. In the **Location** group, from the **Tool** list, select **POCKET_01**.
4. In the **Name** box, type **EM-1.00**.
5. Click **OK**.

   You will accept the tool parameter defaults and set some additional settings.

6. In the **Milling Tool-5 Parameters** dialog box, in the **Description** group, click **Edit**.
7. In the **Search Result** dialog box, in the **Matching Items** group, select **TMC0_00002** from the list.
8. Click **OK**.
9. In the **Milling Tool-5 Parameters** dialog box, click the **Holder** tab.

   Holders are built sequentially from the tool up to the gauge line. You will start at the connection point to the face mill.

10. In the **Holder Steps** group, set the parameters as shown.

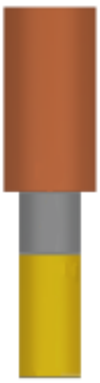
    a. **(LD) Lower Diameter** = **1.50**.

    b. **(L) Length** = **3.00**.

    c. **(UD) Upper Diameter** = **1.50**.

11. Click **OK**.

You have created a 1.00” diameter mill, defined a tool holder and placed it into pocket 1 of your machine tool. Next you will create end mills of various sizes.



Create a 1.25 diameter end mill with a two step tool holder

1. On the **Insert** toolbar, click **Create Tool**  .
2. In the **Create Tool** dialog box, in the **Tool Subtype** group, click **MILL**  .
3. In the **Location** group, from the **Tool** list, select **POCKET_02**.
4. In the **Name** box, type **EM-1.25-0**.
5. Click **OK**.
6. In the **Milling Tool-5 Parameters** dialog box, in the **(D) Diameter** box, type **1.25**.
7. In the **(L) Length** box, type **5.5**.
8. In the **Flutes** box, type **3**.

Caution Do not set the tool number. It is inherited from the POCKET_ID.

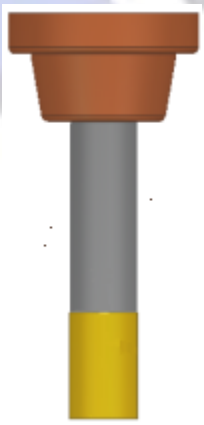
9. In the **Description** group, click **Edit**  .
10. In the **Search Result** dialog box, in the **Matching Items** group, select **TMC0_00006** from the list.
11. Click **OK**.

Define step 1 of the tool holder

1. In the **Milling Tool 5-Parameters** dialog box, click the **Holder** tab.
2. In the **Holders Steps** group, set the parameters as shown.
    a. **(LD) Lower Diameter** = **2.25**.
    b. **(L) Length** = **1.25**.
    c. **(B) Taper Angle** = **10**.
    d. **(R1) Corner Radius** = **.125**.

Define step 2 of the tool holder

1. Click **Add New Set**  .
2. In the **Holders Steps** group, set the parameters as shown.
    a. **(LD) Lower Diameter** = **3.50**.
    b. **(L) Length** = **.75**.
    c. **(UD) Upper Diameter** = **3.50**.
    d. **(B) Taper Angle** = **0**.
    e. **(R1) Corner Radius** = **.125**.
3. Click **OK**.



You have created a 1.250 end mill, defined a tool holder and placed it into pocket 2 of your machine tool. Next you will create a 1.000 diameter, 3 - fluted carbide end mill, with a .125 corner radius.

Create a 1.0 diameter end mill with a tool holder

1. Click **Create Tool**  .
2. In the **Create Tool** dialog box, in the **Tool Subtype** group, ensure **MILL**  is active.
3. In the **Location** group, from the **Tool** list, select **POCKET_03**.
4. In the **Name** box, type **EM-1.00-.125C**.
5. Click **OK**.
6. In the **Mill Tool-5 Parameters** dialog box, on the **Tool** tab, in the **Dimensions** group, set the parameters as shown.
    a. **(D) Diameter** = **1.00**
    b. **(R1) Lower Radius** = **.125**.
    c. **(B) Taper Angle** = **10**.

Caution Do not set the tool number. It is inherited from the POCKET_ID.

7. In the **Description** group, click **Edit**  .
8. In the **Search Result** dialog box, in the **Matching Items** group, select **TMC0_00002** from the list.
9. Click **OK**.

Define the tool holder

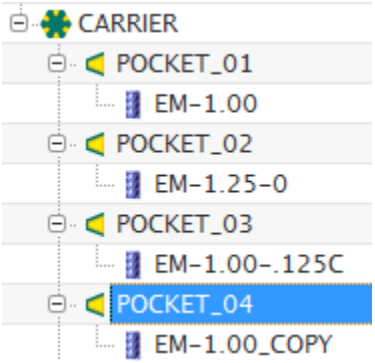
1. In the **Milling Tool-5 Parameters** dialog box, click the **Holder** tab.
2. In the **Holder Steps** group, set the parameters as shown.
    a. **(LD) Lower Diameter** = **3.5**.
    b. **(L) Length** = **.75**.
    c. **(UD)) Upper Diameter** = **3.5**.
    d. **(B) Taper Angle** = **0**.
    e. **(R1) Corner Radius** = **.125**.
3. Click **OK**.

You have created a 1.000 diameter end mill with a .125 corner radius and a 10 degree taper, defined a tool holder, and assigned it to **POCKET_03**.

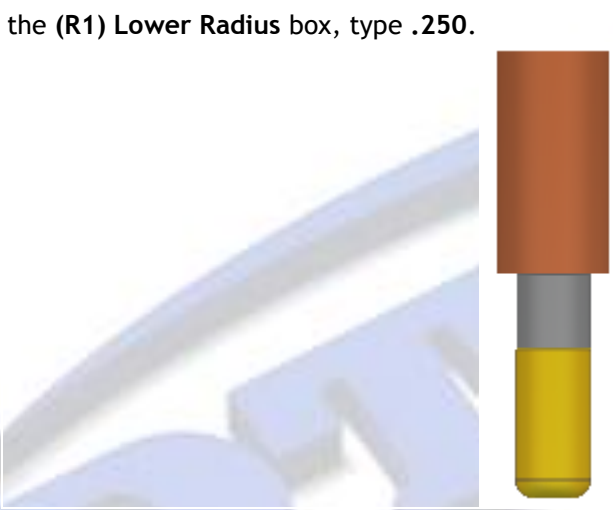
Copy and edit an existing tool

1. In the **Machine Tool** view of the **Operation Navigator**, under the **POCKET_01** node, right-click **EM-1.00** and choose **Copy**.

2. Right-click the **POCKET_04** node and choose **Paste Inside**.



3. Rename the new tool to **EM-1.0-.25C**.
4. Double-click the new tool to edit its parameters.
5. In the **Milling Tool 5-Parameters** dialog box, on the **Tool** tab, in the **Dimensions** group, in the **(R1) Lower Radius** box, type **.250**.

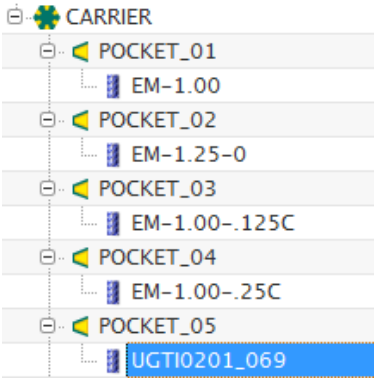


6. Click **OK**.

Retrieve a tool from the tool library

1. Click **Create Tool**  .
2. In the **Create Tool** dialog box, in the **Library** group, click **Retrieve Tool from Library**  .
3. In the **Library Class Selection** dialog box, expand **Milling** and select **End Mill (non indexable)**.
4. Click **OK**.
5. In the **Search Criteria** dialog box, in the **Search Parameters** group, in the **(D) Diameter** box, type **1.25** and click **OK**.
6. In the **Search Result** dialog box, in the **Matching Items** group, select **End Mill 1 1/4”** from the list, and click **OK**.

The tool is retrieved and placed in the next available pocket.

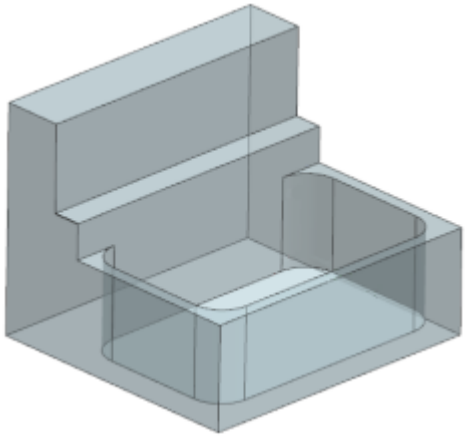


7. Click **Cancel** to close the **Create Tool** dialog box.
8. Close the part.

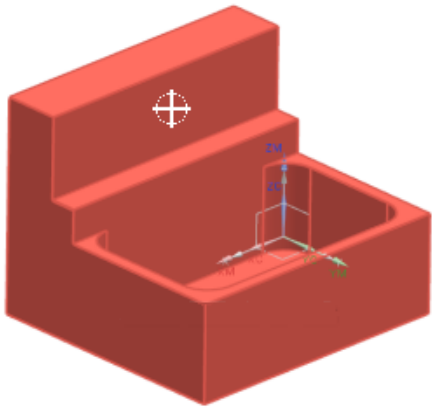
Define the shank, tool holder, and clearances

You normally define the tool, the shank, and the tool holder. In this activity, the tool was previously defined.

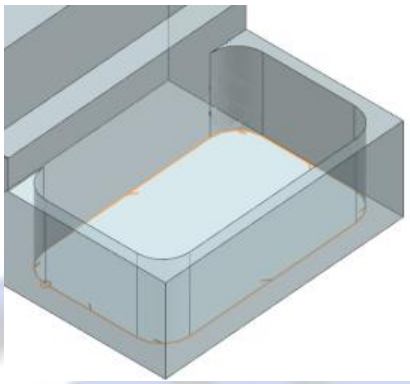
1. Open *mnf17_tool_clearance*.



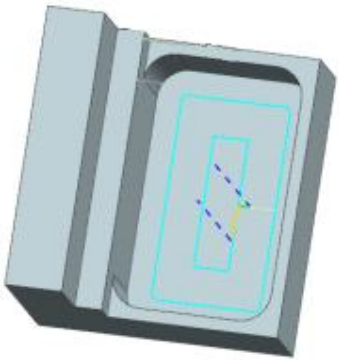
2. In the **Operation Navigator**, right-click in the background and choose **Geometry View**.
3. Right-click **WORKPIECE** and choose **Edit**.
4. In the **Mill Geom** dialog box, click **Specify Part** and select the part.



The cut region for this operation was previously defined.



5. Click **OK** twice.
6. In the **Operation Navigator**, right-click **FACE_MILLING** and choose **Generate**.



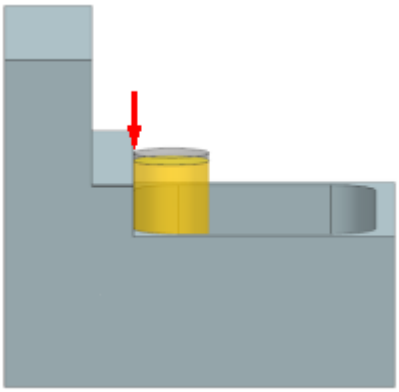
The tool path is generated. Next you will verify the tool path to visually inspect for collisions.

7. Right - click **FACE_MILLING** and choose **Tool Path→Verify**.
8. In the **Tool Path Visualization** dialog box, on the **Replay** tab, move the **Animation Speed**  slider to **1**.

   This value slows the animation speed so that you can view the tool path more easily.

9. Click **Play**  .

   The tool cuts and rubs against the highest wall of the pocket.



10. Click **OK** to accept the tool path.

    You can specify additional parameters to more accurately define your cutting tool.

11. In the **Operation Navigator**, right - click in the background and choose **Machine Tool View**.

12. Right - click **MILL** and choose **Edit**.

13. In the **Milling Tool-5 Parameters** dialog box, click the **Shank** tab.

    The tool diameter, length, and flute length were previously defined. You will now define the shape of tool shank.

14. Define the tool shank and preview the tool.

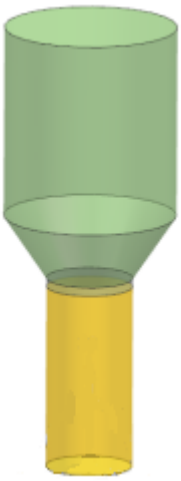
    a. In the **Shank** group, select the **Define Shank** check box. In the **Dimensions** group, set the parameters as shown.

    b. **(SD) Shank Diameter** = **2**.

    c. **(SL) Shank Length** = **3**.

    d. **(STL) Shank Taper Length**=**.750**.

    e. In the **Preview** group, click **Display**.



    f. Click **OK**.

You must update the tool path for it to inherit the new tooling parameters.

15. Verify the tool path.

    a. In the **Operation Navigator**, right-click **FACE_MILLING** and choose **Generate**.

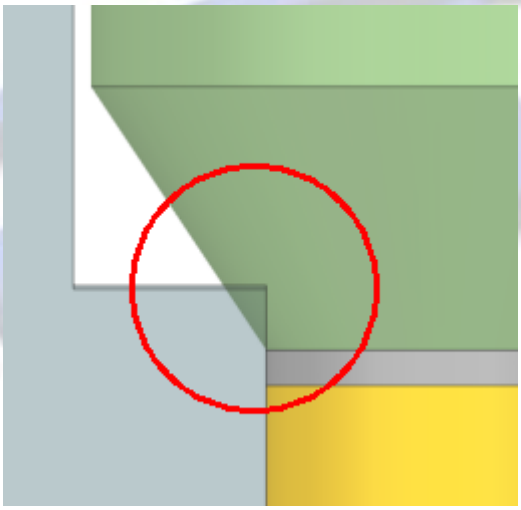
    b. In the graphics window, right-click in the background and choose **Orient View→RIGHT**.

    c. In the **Operation Navigator**, right-click **FACE_MILLING** and choose **Tool Path→Verify**.

    d. In the **Tool Path Visualization** dialog box, on the **Replay** tab, move the **Animation Speed** slider to **1**.

    e. Click **Play**.

    The shank of the tool interferes with the part. You need to check for tool and holder collisions.



    f. Click **OK**.

16. In the **Operation Navigator**, right-click **FACE_MILLING** and choose **Edit**.

17. In the **Face Milling** dialog box, in the **Path Settings** group, click **Cutting Parameters**.

18. Edit the tool path and check for tool and holder collisions.

    In the **Cutting Parameters** dialog box, click the **Containment** tab.

    a. Select the **Check Tool and Holder** check box.

    b. Click the **More** tab.

    You can specify individual clearance distances for each segment of the tool. You are going to specify clearance distances for the tool holder and the shank.

    c. In the **Clearance** group, set the parameters as shown.

        A. **Tool Holder** = **.100**

B. **Tool Shank** = **.100**.

d. Click **OK**.

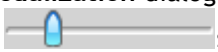
19. In the **Face Milling** dialog box, in the **Actions** group, click **Generate**  .

20. If you see the **Tool Path generate** message, click **OK**.

21. Click  to close the **Information** window.

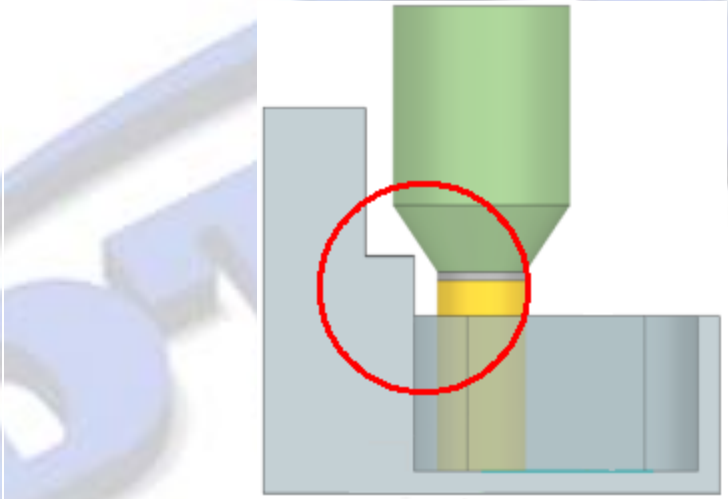
22. Verify the tool path.

In the **Face Milling** dialog box, in the **Actions** group, click **Verify**  .

a. In the **Tool Path Visualization** dialog box, on the **Replay** tab, move the **Animation Speed**  slider to **1**.

b. Click **Play**  .

The tool and shank now clear the wall of the part.



c. Click **OK** twice to close the open dialog boxes.

23. In the **Operation Navigator**, if necessary switch to the **Machine Tool View**.

24. Right - click **MILL** and choose **Edit**.

The tool diameter and length, the flute length, and the shape of the tool shank are defined. You will now specify the shape of the tool holder.

25. In the **Milling Tool-5 Parameters** dialog box, click the **Holder** tab and, enter the parameters as shown.

Holder Steps group:

a. **(LD) Lower Diameter** = **3.5**.

b. **(L) Length** = **2.0**.

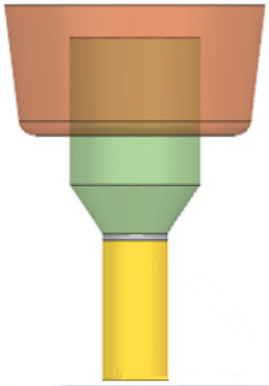
c. **(UD) Upper Diameter** = **4.0**.

The Taper Angle will be automatically calculated based on the **(LD) Lower Diameter** and **(UD) Upper Diameter** values.

d. **(R1) Corner Radius** = **.25**.

e. **Tool Insertion** group:

(OS) Offset = **1.5**.

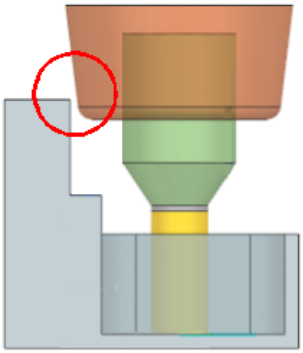


f. In the **Preview** group, click **Display** .

g. Click **OK**.

26. In the **Operation Navigator**, right-click **FACE_MILLING** and choose **Generate** to update the tool path.
27. If you see the **Tool Path generate** message, click **Yes**.
28. Click to close the **Information** window.
29. In the **Operation Navigator**, right-click **FACE_MILLING** and choose **Tool Path→Verify**.
30. Play the animation.

The tool holder stays away from the wall because NX adjusts the tool path depending on the **Tool Holder** clearance value.



31. Click **OK**.
32. Close the part.

You completed the activity.

Activity: Mill with a chamfer tool

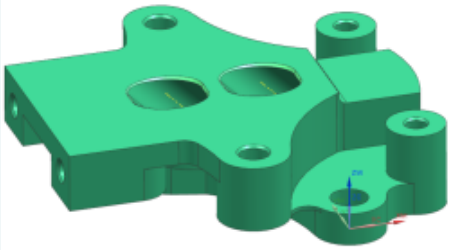
Estimated time to complete: 5-8 minutes

In this activity, you will create a chamfered end mill used to machine an edge break on two ports.

Launch the Milling with a chamfer tool activity.

Create an end mill with a chamfer

1. Open *mnf3_mount7*.



2. On the **Insert** tool bar, click **Create Tool**  .
3. In the **Create Tool** dialog box, in the **Type** group, from the list, select **mill_planar**.
4. In the **Tool Subtype** group, click **CHAMFER_MILL**  .
5. Click **OK**.
6. In the **Chamfer Mill** dialog box, set the parameters as shown.
   a. **(D) Diameter** = **10.00**
   b. **(B) Chamfer Angle** = **45**
   c. **(C) Chamfer Length** = **1.5**
   d. **(FL) Flute Length** = **1.5**
7. Click **OK**.

Create an operation to machine the chamfer

1. On the **Insert** tool bar, click **Create Operation**  .
2. In the **Create Operation** dialog box, in the **Type** group, from the list, select **mill_contour**.

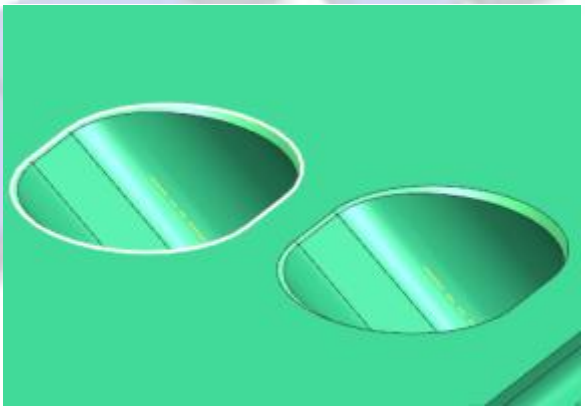
3. In the **Operation Subtype** group, click **CAVITY_MILL**  .
4. In the **Location** group, set the options as shown.

Program	**NC_PROGRAM**
Tool	**CHAMFER_MILL**
Geometry	**WORKPIECE**
Method	**METHOD**

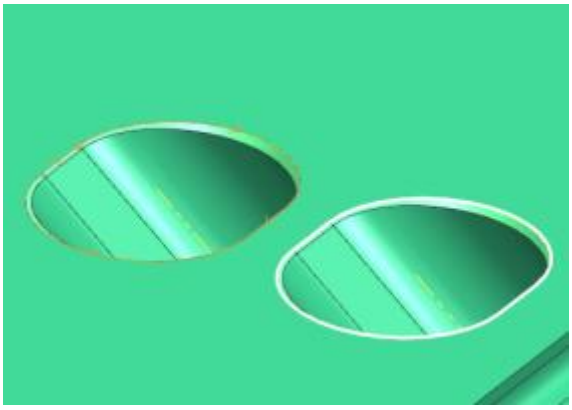
5. Click **OK**.

Select the curves surrounding the two ports to contain the tool path

1. In the **Cavity Mill** dialog box, in the **Geometry** group, click **Specify Trim Boundaries**  .
2. In the **Trim Boundaries** dialog box, under **Filter Type**, click **Curve Boundary**  .
3. On the **Selection** bar, from the **Curve Rule** list, select **Tangent Curves**.
4. Under **Trim Side**, select **Outside**.
5. Select the first edge as shown.



6. In the **Trim Boundaries** dialog box, click **Create Next Boundary**.
7. Select the second edge as shown.



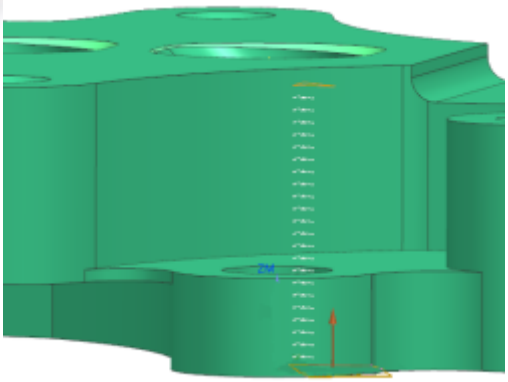
8. Click **OK**.

Set the depth of cut and specify the cut range

1. In the **Cavity Mill** dialog box, in the **Path Settings** group, from the **Cut Pattern** list, select **Profile**.
2. In the **Maximum Distance** box, type **1**.

   The **Maximum Distance** value determines how far below the specified trim boundary plane the tool will cut.
3. Click **Cut Levels**  .
4. In the **Cut Levels** dialog box, in the **Ranges** group, from the **Range Type** list, select **Single**.

   There is one range, with a 1 mm depth of cut increment through the entire thickness of the part.

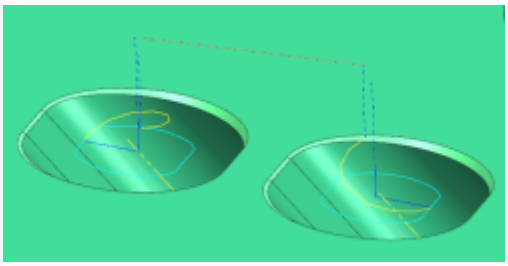


5. From the **Cut Levels** list, select **Only at Range Bottom**.
6. In the **Range Definition** group, in the **Range Depth** box, type **1**.
7. Click **OK**.

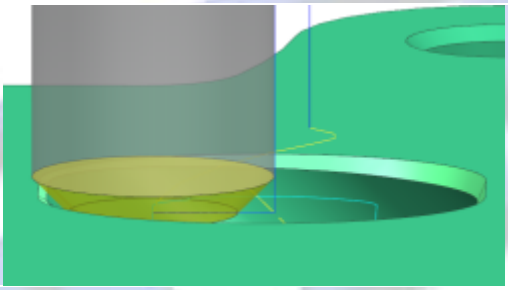
   The **Cavity Mill** dialog box is displayed.

View the tool path

1. In the **Actions** group, click **Generate**  .



2. Click **Verify**  .
3. In the **Tool Path Visualization** dialog box, move the **Animation Speed**  slider to 1.
4. Click **Play**  .



5. Click **OK** twice.
6. Close the part.

You completed the activity.

Activity: Output shop documentation

Estimated time to complete: 3–5 minutes

In this activity, you will create shop documentation, an operation list, and a tool list.

Launch the Output shop documentation activity.

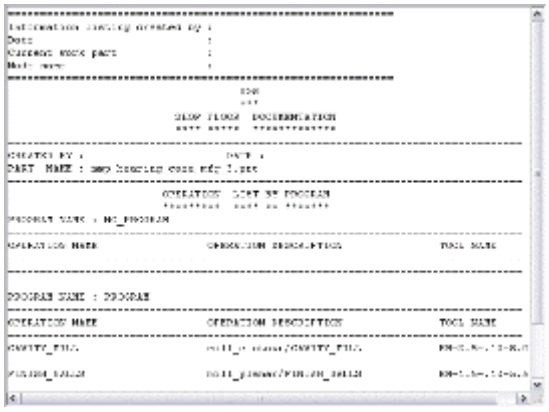
Output shop documentation for the operations and tools in a text format

Continue to use *mnf3_bearing_case_mfg_3*.

1. In the **Operation Navigator**, select the **PROGRAM** folder.
2. On the **Operations** toolbar, click **Shop Documentation**  .

   You must specify a name and location for the documentation data file that you are about to create. The default location is the current directory, which should be your home directory.
   Note that you will accept the default name in the **Output File Name** field or change it to your home directory path if your current directory is not your home directory.
3. In the **Shop Documentation** dialog box, in the **Report Format** group, from the list, select **Operation List Select (TEXT)**.
4. Click **OK**.

   The Operation List is displayed in the **Information** window.

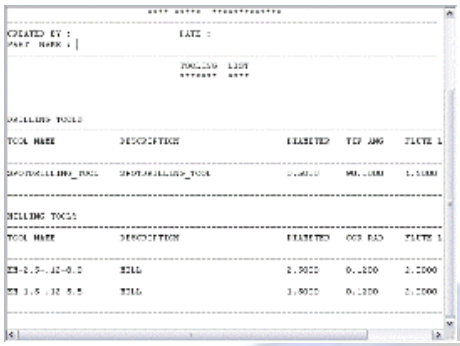


5. Examine the information in the operation description and tool name columns.
6. Close the **Information** window.

Output shop documentation for the tools in a text format

1. On the **Operations** toolbar, click **Shop Documentation** .
2. In the **Report Format** group, from the list, select **Tool List Select (TEXT)**.
3. Click **OK**.

   The Tool List is displayed in the **Information** window.



4. Examine the milling and drilling tool information.
5. Close the **Information** window.

Output shop documentation for the tools in an HTML or Excel format

1. Click **Shop Documentation** .
2. In the **Shop Documentation** dialog box, in the **Report Format** group, from the list, select **Tool List Select (HTML/Excel)**.
3. Click **OK**.

   The HTML and Excel both open.
4. Examine the tool information output and file name extensions.

5. Return to NX.

Output shop documentation for the operations in an HTML or Excel format

1. Click **Shop Documentation** .
2. In the **Shop Documentation** dialog box, in the **Report Format** group, from the list, select **Operation List Select (HTML/Excel)**.
3. Click **OK**.

4. In the **Output File** message box, click **OK**.
5. Examine the operation information output and file name extensions.

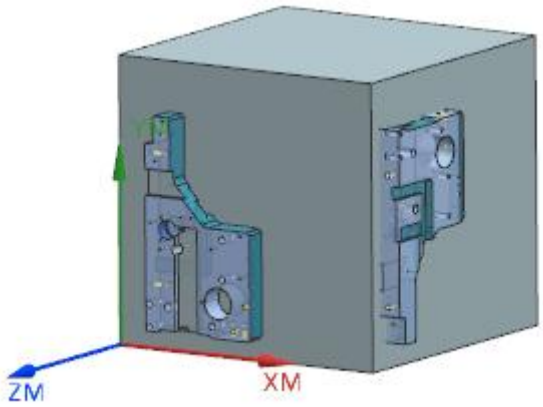
6. Close the part without saving it.

You completed the activity.

Activity: Transfer an IPW between two workpieces

Estimated time to complete: 4–8 minutes

In this activity, you will transfer an IPW between two workpieces in the same setup assembly.

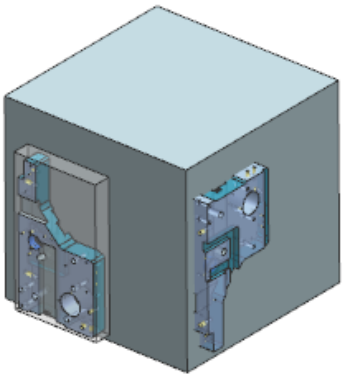


Launch the Transfer an IPW between two workpieces activity.

Open the part

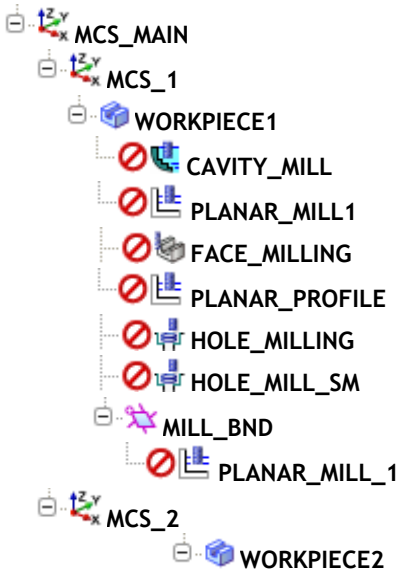
This setup assembly for this activity has a main MCS with two workpieces. There is an MCS for each workpiece.

1. Choose **File→Options→Assembly Load Options**.
2. In the **Assembly Load Options** dialog box, in the **Scope** group, clear the **Use Partial Loading** check box.
3. Click **OK**.
4. Open *mnf15_IPW_Workpart_flow_mfg_1*.

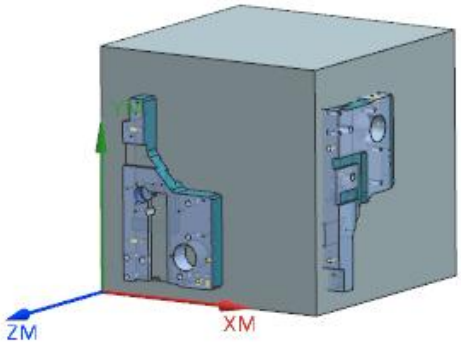


5. Review the **Geometry** view of the **Operation Navigator**.

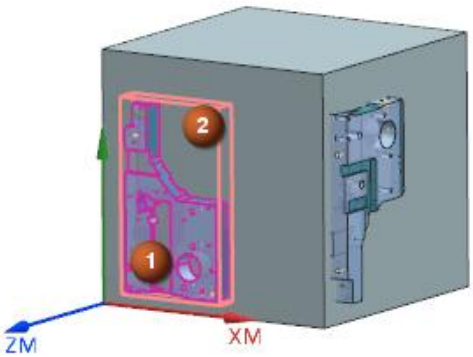
   The **WORKPIECE1** geometry parent for the first setup will become the IPW source for the **WORKPIECE2** geometry parent in the next setup.



6. Click **MCS_1** to display the MCS for the first setup.

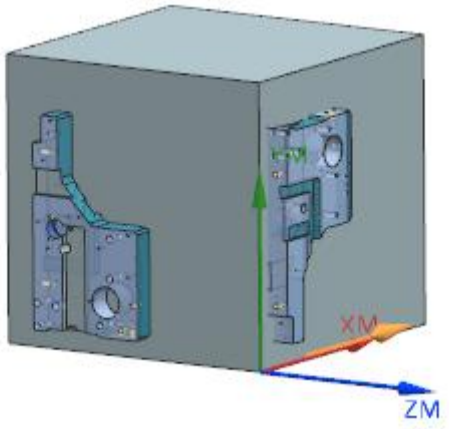


7. Click **WORKPIECE1** to display the part and blank geometry for the first setup.

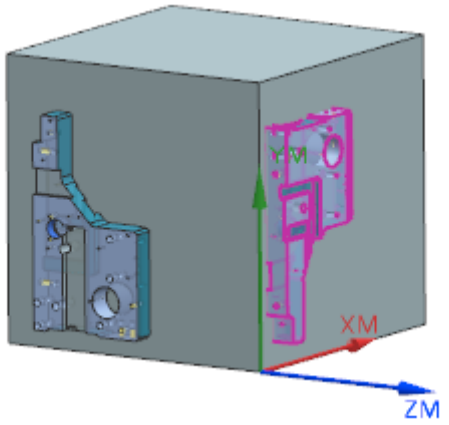


1. Part in first setup
2 .Blank in first setup

8. Click **MCS_2** to display the MCS for the second setup.



9. Click **WORKPIECE2** to display the part geometry for the second setup.



Generate and verify the operations

1. Right-click **WORKPIECE1** and choose **Generate**.

   NX generates the first tool path.

2. In the **Path Generation** dialog box, click **OK** as necessary to continue generating the tool paths for **WORKPIECE1**.

3. Click **Yes** in the **Tool Path generate** message box.

   The **CAVITY_MILL** roughing operation uses a larger diameter tool that does not fit in all areas. You can ignore this message.

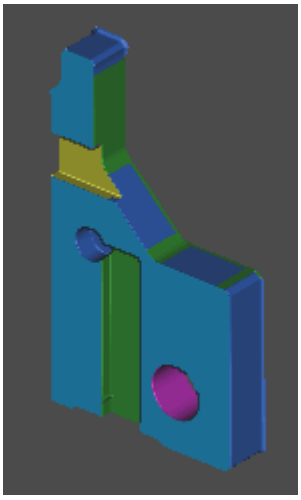
4. Close the **Information** window.

5. Right-click **WORKPIECE1** and choose **Tool Path→Verify**.

   The **Tool Path Visualization** dialog box is displayed.

6. Click the **2D Dynamic** tab.

7. Click **Play**.



8. Click **OK**.

Define the blank for the second setup

The part geometry in **WORKPIECE2** is already defined for you, and uses the WAVE linked body.

A WAVE linked body is not a prerequisite, but the part geometry in both setups must be from the same source. You can choose:

- The same WAVE linked body for both setups.
- The same assembly occurrence of the part geometry for both setups.

1. In the **Operation Navigator**, double-click **WORKPIECE2**.

   The **Mill Geom** dialog box is displayed.

2. In the **Geometry** group, click **Specify Blank**  .

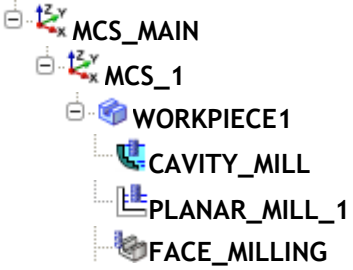
   The **Blank Geometry** dialog box is displayed.

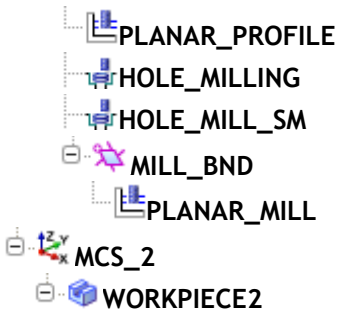
3. In the **Type** group, from the list, select **IPW — In Process Workpiece**.

4. In the **IPW Source** group, click **Select Source for IPW**  .

   The **IPW Source** dialog box is displayed. The **Copy IPW from** option is **Work Part**, which is the default.

5. In the hierarchy list, expand the collapsed nodes.

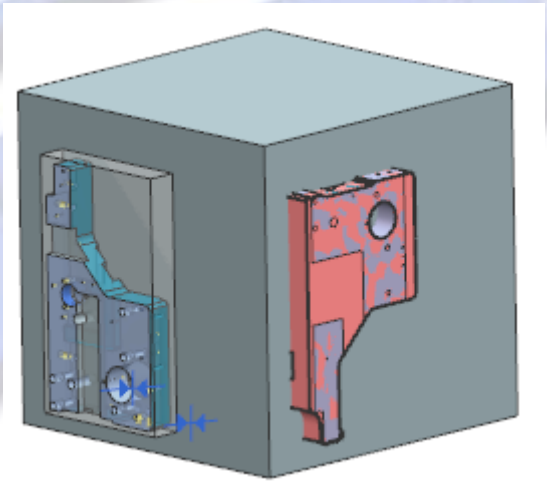




6. The operations are included in the hierarchy for reference to help you select the correct workpiece. You can only select a *workpiece* for the IPW source.
7. Select **WORKPIECE1** and click **OK**.

   The **Blank Geometry** dialog box is displayed.
8. In the **Local IPW** group, click **Update Local IPW from Source** .

   The output IPW of the last operation in the **Program Order View** for **WORKPIECE1** becomes the input IPW for your second setup. NX transforms the position and orientation of the IPW to match the position and orientation of the part geometry in the current setup.



9. Click **OK**.

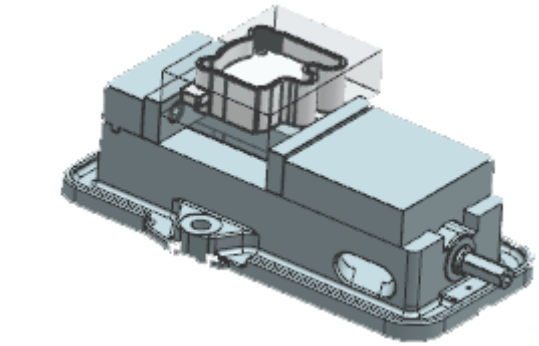
   The **Mill Geom** dialog box is displayed.
10. Click **OK**.
11. Close the part without saving it.
12. Choose **File→Options→Assembly Load Options**.
13. In the **Assembly Load Options** dialog box, in the **Scope** group, select the **Use Partial Loading** check box.
14. Click **OK**.

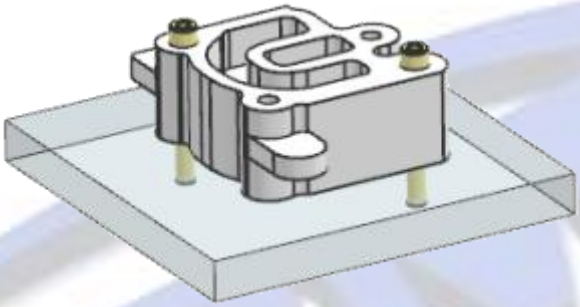
You completed the activity.

Activity: Transfer an IPW between two setup assemblies

Estimated time to complete: 6–10 minutes

In this activity, you will transfer an IPW between two setup assemblies.

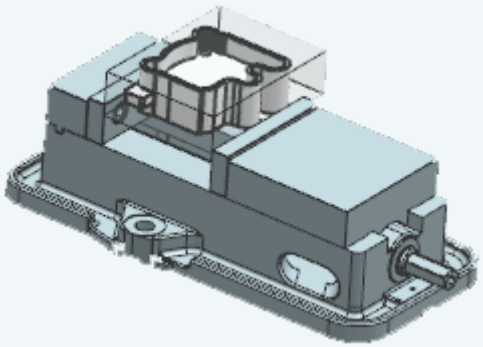




Launch the Transfer an IPW between two setup assemblies activity.

Open the first setup assembly

1. Open *mnf15_ipw_setup_1*.



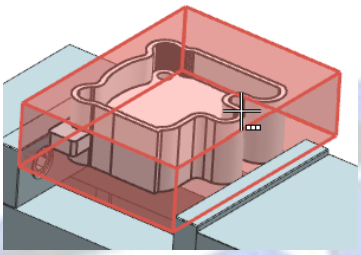
2. Notice the orientation of the part in the fixture.

Specify the blank for the first workpiece

1. Right-click in the background of the **Operation Navigator** and choose **Geometry View**.
2. Double-click **WORKPIECE**.

   The **Mill Geom** dialog box is displayed.
3. In the **Geometry** group, click **Specify Part**  .

   The **Part Geometry** dialog box is displayed.
4. Pause the cursor over the part geometry until the QuickPick cursor  is displayed, and then click.

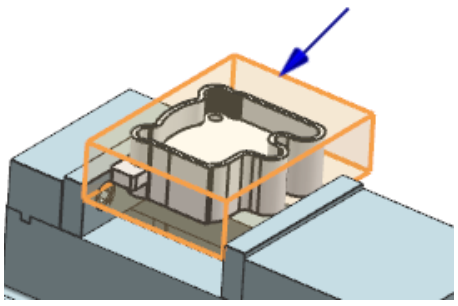
   

   The **QuickPick** dialog box is displayed.

   This step helps you to identify if there is more than one occurrence of the part geometry in the assembly.
5. Select **Solid Body in MNF15_IPW_TRANSFER**.
6. In the **Part Geometry** dialog box, click **OK**.

   The **Mill Geom** dialog box is displayed.
7. In the **Geometry** group, click **Specify Blank**  .

   The **Blank Geometry** dialog box is displayed.
8. Select the blank as shown.

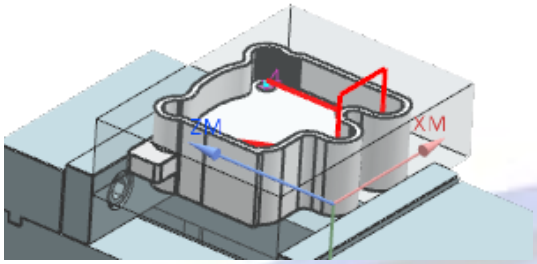
   
9. Click **OK**.

   The **Mill Geom** dialog box is displayed.
10. Click **OK**.

Generate the operations for the first setup assembly

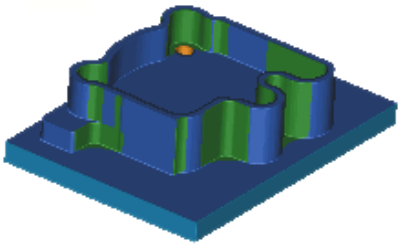
1. Right-click in the background of the **Operation Navigator** and choose **Program Order View**.
2. Right-click **PROGRAM** and choose **Generate**.

   NX generates the first tool path.
3. In the **Path Generation** dialog box, click **OK** as necessary to continue generating the tool paths for the program.



   The output IPW for the last operation in the **PROGRAM** hierarchy defines the input IPW for the workpiece in the second setup assembly.

4. Click **Yes** in the **Tool Path generate** message box.
   The **CAVITY_MILL** roughing operation uses a larger diameter tool that does not fit in all areas. You can ignore this message.
5. Close the **Information** window.
6. Right-click **PROGRAM** and choose **Tool Path→Verify**.
   The **Tool Path Visualization** dialog box is displayed.
7. Click the **2D Dynamic** tab.
8. Click **Play**.

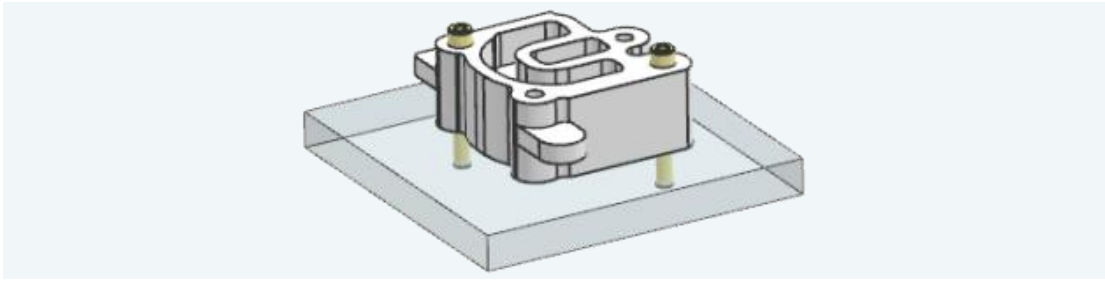


9. Click **OK**.
10. Save the part as **xxx_ipw_setup_1**.

    Replace **xxx** with your initials.
11. Close the part.

Define the workpiece for the second setup assembly

1. Open *mnf15_ipw_setup_2*.



2. Notice the orientation of the part in the fixture.
3. Right-click in the background of the **Operation Navigator** and choose **Geometry View**.
4. Double-click **WORKPIECE**.

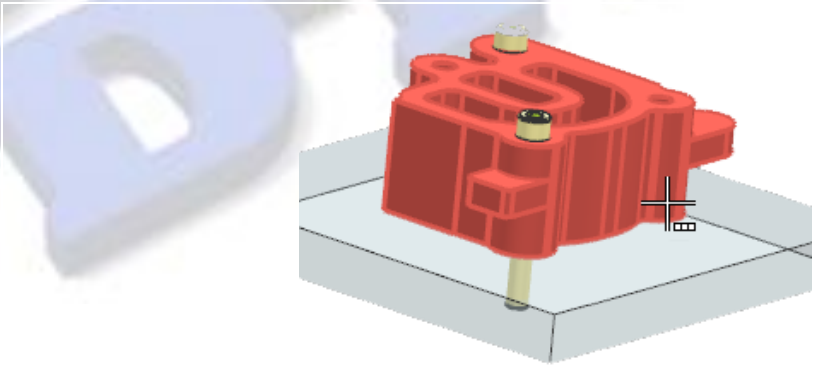
   The **Workpiece** dialog box is displayed.

5. In the **Geometry** group, click **Specify Part**  .

   The **Part Geometry** dialog box is displayed.

   You must select the same part geometry for the second setup that you selected for the first setup.

6. Pause the cursor over the part geometry until the QuickPick cursor  is displayed, and then click.



   The **QuickPick** dialog box is displayed

7. Select **Solid Body in MNF15_IPW_TRANSFER**.

   This step helps to ensure you select the same part geometry as you did for the first setup assembly.

8. In the **Part Geometry** dialog box, click **OK**.

   The **Workpiece** dialog box is displayed.

9. In the **Geometry** group, click **Specify Blank**  .

The **Blank Geometry** dialog box is displayed.

10. In the **Type** group, from the list, select **IPW — In Process Workpiece**.

    The **WORKPIECE** geometry parent from the first setup assembly is the IPW source for your blank geometry.

11. In the **IPW Source** group, click **Select Source for IPW**  .

    When you select an IPW source part, NX maintains the path to locate the part. If you copy the work part or source part to a different folder, you must modify the blank to specify the source part in its new location.

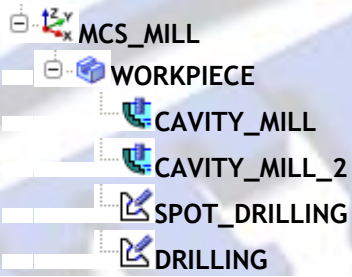
    You can also modify the blank to specify a different IPW source.

    The **IPW Source** dialog box is displayed.

12. From the **Copy IPW from** list, select **xxx_ipw_setup_1**.

    If necessary, select **Browse** and navigate to your part file.

13. In the **IPW Source** dialog box, expand the **WORKPIECE** geometry object.



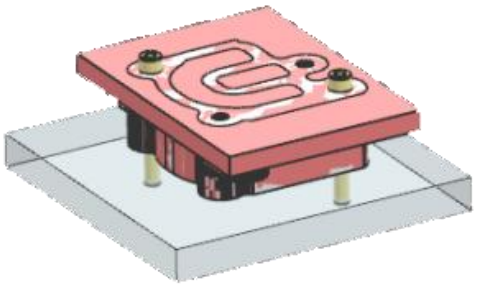
14. The operations are included in the hierarchy for reference to help you select the correct workpiece. You can select only a workpiece for the IPW source.

15. Select **WORKPIECE** and click **OK**.

    The **Blank Geometry** dialog box is displayed.

16. Review the status information in the **Blank Geometry** dialog box. Notice that the **IPW Source** group displays the name of the source part and geometry group you are using for the input IPW.

17. In the **Local IPW** group, click **Update Local IPW from Source**  .



    Notice that in the **Local IPW** group, the **Status** line displays **Up to Date**.

18. Click **OK**.

    The **Workpiece** dialog box is displayed.

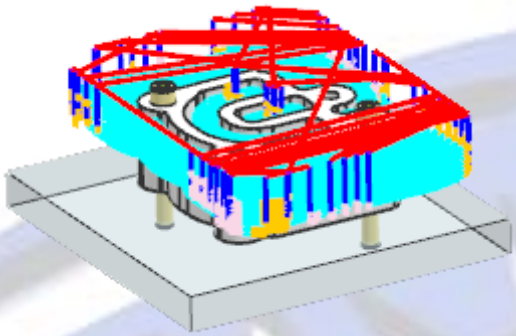
19. Click **OK**.

Generate the operations in the second setup assembly

1. In the **Operation Navigator**, right-click **WORKPIECE** and choose **Generate**.

   NX generates the first tool path.

2. In the **Path Generation** dialog box, click **OK** as necessary to continue generating the tool paths for **WORKPIECE**.



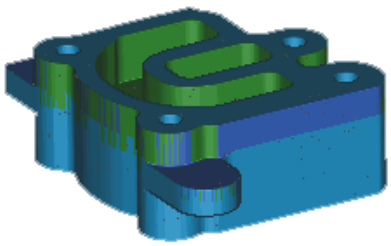
3. Click **Yes** in the **Tool Path generate** message box.

   The **CAVITY_MILL** roughing operation uses a larger diameter tool that does not fit in all areas. You can ignore this message.

4. Close the **Information** window.
5. Right-click **WORKPIECE** and choose **Tool Path→Verify**.

   The **Tool Path Visualization** dialog box is displayed.

6. Click the **2D Dynamic** tab.
7. Click **Play**.



8. Click **OK**.
9. Close the part.

You completed the activity.

Activity: Customize a Probing operation

Estimated time to complete: 15–25 minutes

In this activity, you will do the following to customize a probing operation:

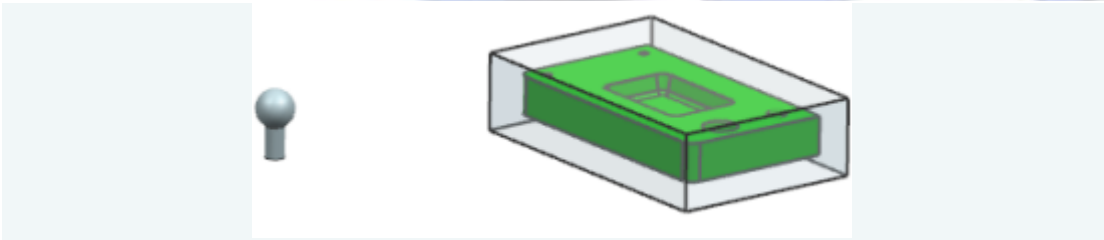
- Add a new custom sub-operation type.
- Add new parameters to an existing sub-operation type.

You must customize probing operations *before* you add suboperations. You cannot modify a suboperation once it is used in the current **Probing** or **Generic Motion** operation. However, you can modify a suboperation that is used in another operation because each operation is independent.

Launch the Customize a Probing operation activity.

Create the probing operation

1. Open *mnf16_mill_probe_mfg*.



2. On the **Create** toolbar, click **Create Operation**  or choose **Insert→Operation**.
3. In the **Type** group, select **probing**.
4. In the **Operation Subtype** group, click  **PROBING**.
5. In the **Location** group, set the options as shown.

Program	**T2345A**
Tool	**UGTI_0402_001**
Geometry	**WORKPIECE**
Method	**MILL_FINISH**

6. In the **Name** group, type **CUSTOM_PROBING**.
7. Click **OK** to create the operation.

Customize a standard suboperation type

1. In the **Probing** dialog box, in the **Options** group, click **Customize Dialog**  .
2. In the **Customize Dialog** dialog box, in the **Items Used** group, under **Sub-Operation Manager**, select **Linear Move to Point**.
3. Click **Customize Sub-Dialog**  .
4. In the **Customize Sub Dialog** dialog box, from the **Dialog Item Type** list, select **Double**.

   The **Double** item adds an input box for a double-precision number parameter.
5. In the **Label** box, enter **my_double**.
6. In the **Default Value** box, enter **5.3**.
7. Click **Add to Dialog**  .
8. Click **OK** to complete the parameters.

Add a custom suboperation type

1. In the **Customize Dialog** dialog box, in the **Items Used** group, select **Sub-Operation Manager**.
2. In the **Items to Add** group, from the **Dialog Item Type** list, choose  **User Defined Move Type** .
3. In the **Label** box, enter **my_subop**.
4. Click **Add to Dialog**  .

   NX adds **my_subop** to the **Sub-Operation Manager** list and selects it.
5. Click **Customize Sub-Dialog**  .

Add parameters to the custom suboperation type

1. In the **Customize Sub Dialog** from the **Dialog Item Type** list, select **Group**.
2. In the **Label** box, enter **my_parameters**.
3. Click **Add to Dialog**  .

   NX adds **my_parameters** to the **Items Used** list and selects it.
4. From the **Dialog Item Type** list, select **Double**.
5. In the **Label** box, enter **my_double**.

6. In the **Default Value** box, enter 5.3.
7. Click **Add to Dialog**  .

   NX adds **my_double** to the **my_parameters** group.
8. From the **Dialog Item Type** list, select **Enumeration**.
9. In the **Label** box, enter **my_list**.
10. In the **Enumeration Items** box, enter your list options.
    - Option_1
    - Option_2
    - Option_3
11. Click **Add to Dialog**  .

    NX adds **my_list** to the **my_parameters** group.
12. Click **OK** to complete the parameters.

Modify the list of available suboperation types

1. In the **Customize Dialog** dialog box, in the **Sub-Operation Manager** list, clear the check boxes for all suboperations except **my_subop** and **Linear Move to Point**.

   You clear the check box to remove a suboperation from the **Type** list of the **Create Move Subop** dialog box. You can select the check box later to add the suboperation type if your requirements change.
2. Select **my_subop** and click  until **my_subop** is below **Linear Move To Point**.
3. Click **OK** to complete your customized suboperation list.

View your customizations

1. In the **Probing** dialog box, in the **suboperations** group, click **Add New suboperation**.
2. In the **Create Move Subop** dialog box, in the **Type** group, from the list, review the available suboperation types.

   Notice there are only two suboperation types: **Linear Move To Point** and **my_subop**.
3. From the **Type** list, select **Linear Move to Point**.
4. Notice the **my_double** parameter appended to the **Linear Move To Point** suboperation.
5. From the **Type** list, select **my_subop**.
6. Review the **my_subop** parameters.

7. Click **Cancel** to close the dialog box without creating a suboperation.
8. In the **Probing** dialog box, click **OK** to save the customized **Probing** operation.

   You can use the **custom_probing** operation as a template for your probing operations.
9. Close the part.

You completed the activity.

Activity: Use the IPW to reduce non cutting motions

Estimated time to complete: 4-6 minutes

In this activity, you will modify a **Contour Area** operation to use the in-process workpiece (IPW) to remove tool path segments that do not remove material

Launch the Use the IPW to reduce non cutting motions activity.

Open the part and generate the operation

1. Open *mnf16_surface_contour_ipw*.
2. Right-click the background of the **Operation Navigator** and choose **Configure**.

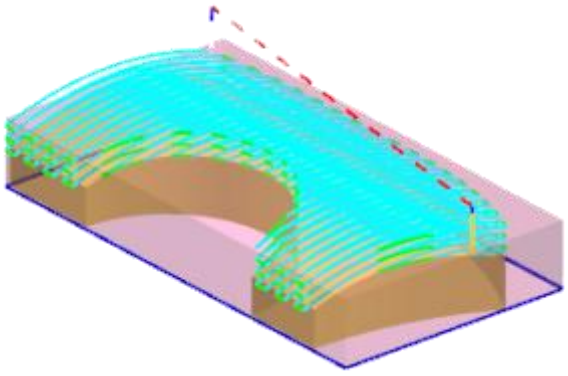
   You will configure the **Program Order View** of the **Operation Navigator** to ensure the **Time** and **Length** columns are visible.

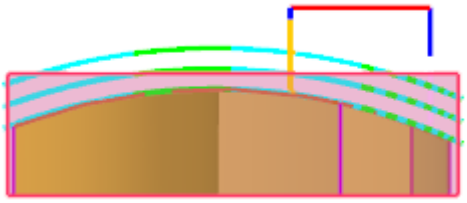
3. In the **Operation Navigator Properties** dialog box, under **View**, click **Program Order**
4. Set the options as shown.

   Name
   - ☑ **Toolchange**
   - ☑ **Path**
   - ☑ **Time**
   - ☑ **Length**
5. Click **OK**.
6. Review the **Time** and **Length** values for the **CONTOUR_AREA** operation.

   Time = 01:37:36

   Length = 1019.7
7. Double-click **CONTOUR_AREA**.
8. In the **Contour Area** dialog box, in the **Actions** group, click **Replay**.
9. Review the tool path.



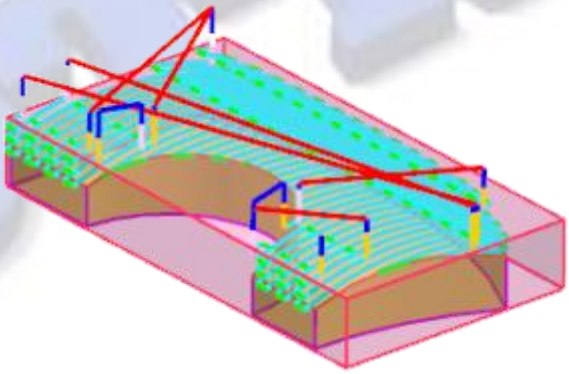


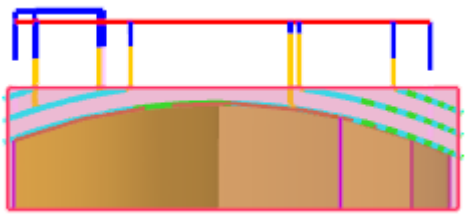
Notice the cutting motions that do not remove material. Also notice the engage and retract motions.

Change the Cutting Parameters options and generate the operation

1. In the **Path Settings** group, click **Cutting Parameters** .
2. In the **Cutting Parameters** dialog box, click the **Containment** tab.
3. In the **Blank** group, from the **In Process Workpiece** list, select **Use 3D**.

   The **Minimize Non Cutting Moves** check box is selected by default.
4. Click **OK**.
5. In the **Contour Area** dialog box, in the **Actions** group, click **Generate** .
6. Review the tool path.





Notice how non cutting portions of the tool path were removed to create a more efficient tool path. The tool is no longer cutting air.

7. In the **Operation Navigator**, review the **Time** and **Length** values for the **CONTOUR_AREA** operation.

Time and **Length** values using the IPW:

- **Time** = 01:02:55
- **Length** = 717.7

Time and **Length** values without the IPW:

- **Time** = 01:37:36
- **Length** = 1019.7

8. Click **OK**.
9. Close the part.

You completed the activity.

Activity: Align the tool to wall UV rulings

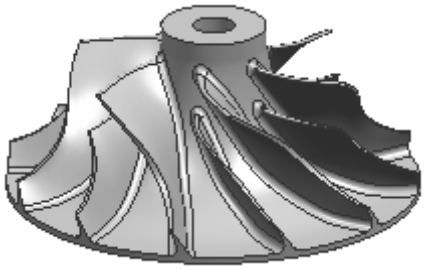
Estimated time to complete: 1–4 minutes

In this activity, you will modify a **Contour Profile** operation to align the tool with the UV rulings of the part wall.

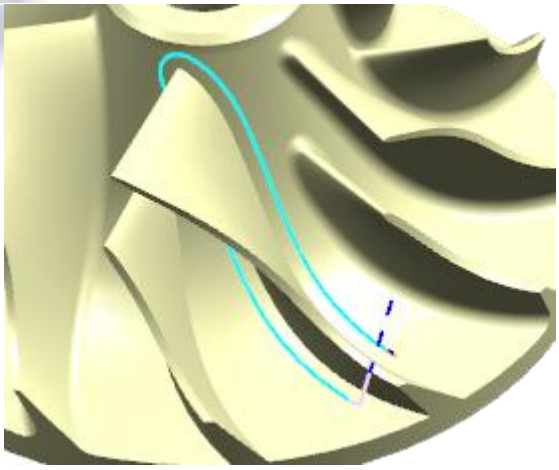
Launch the Align the tool to wall UV rulings activity.

Open the part and generate the operation

1. Open *mnf16_contour_profile_swarf*.



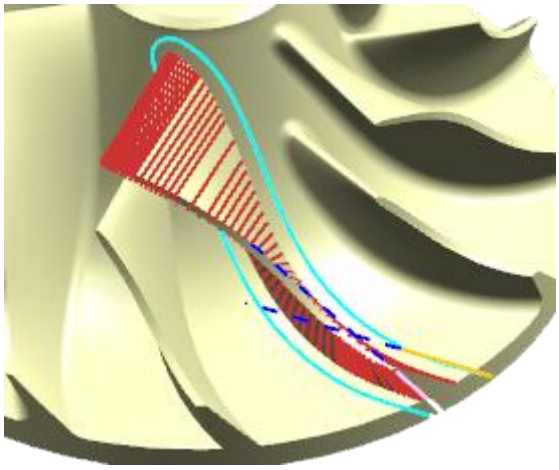
2. Right-click the background of the **Operation Navigator** and choose **Geometry View**.
3. Double-click **VARIABLE_CONTOUR**.
4. In the **Variable Contour** dialog box, in the **Actions** group, click **Generate**.
5. In the warning dialog box, click **OK**.
6. Review the information message and close the information window.
7. Review the tool path.



Change the tool axis option and generate the operation

1. In the **Tool Axis** group, from the **Axis** list, select **Swarf Base UV**.
2. In the **Actions** group, click **Generate**.

3. Review the tool path.



4. Click **Verify**  .
5. In the **Tool Path Visualization** dialog box, on the **Replay** tab, move the **Animation** slider to 1.
6. Click **Play**  .

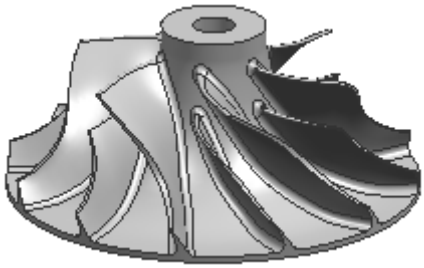
   Notice how the tool aligns with the blade contour.
7. Click **OK**.
8. Close the part without saving it.

You completed the activity.

Activity: Create a multi blade blend finish operation

Estimated time to complete: 5–7 minutes

In this activity, you will create a mill multi blade blend finish operation.

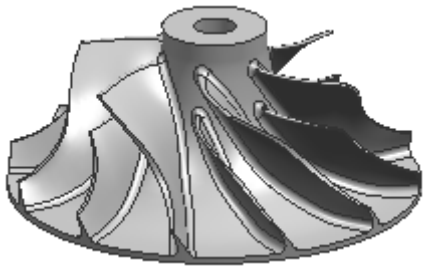


Launch the Create a multi blade blend finish operation activity.

Open the part and create the operation

The blend finish operation removes material that remains from the blade finish and hub finish operations.

1. Open *mnf16_multi_blade_blend_finish*.

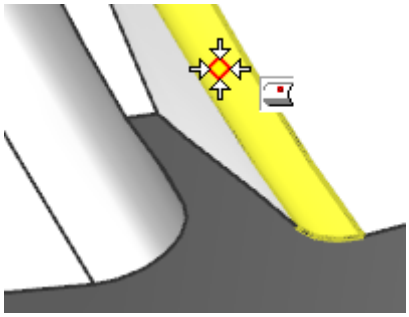


2. Choose **Analysis→Geometric Properties**

   The **Geometric Properties** dialog box is displayed.

   Analyzing the fillet geometry helps you to determine the correct tool size.

3. In the graphics window, pause the cursor over the blade blend fillet.



The **Minimum Radius** value in the **Results** group of the **Geometric Properties** dialog box is -2.250.

You will use a 4 mm ball mill to finish the blade blend fillet.

4. Click **Close**.
5. On the **Create** toolbar, click **Create Operation**  or choose **Insert→Operation**.
6. In the **Type** group, from the list, select **mill_multi_blade**.
7. In the **Operation Subtype** group, click **BLEND_FINISH** 
8. In the **Location** group, set the options as shown.

Program	**1234**
Tool	**BALL_4**
Geometry	**MULTI_BLADE_GEOM**
Method	**MILL_FINISH**

9. The geometry for the operation is inherited from the **MULTI_BLADE_GEOM** geometry parent.
10. Click **OK**.

    The **Blend Finish** dialog box is displayed.

Specify the drive settings

1. In the **Drive Method** group, click **Blend Finish**  .

   The **Blend Finish Drive Method** dialog box is displayed.
2. In the **Cut Periphery** group, from the **Geometry to Finish** list, select **Blade Blend**.
3. From the **Sides to Cut** list, select **Left, Right, Leading Edge**.
4. In the **Drive Settings** group, from the **Drive Mode** list, select **Reference Tool**.

   The **Reference Tool** option provides more control of the cut area and ensures full coverage of the area that was not finished in the previous blade and hub operations. The smaller tool in the blend finishing operation removes material in the regions where the larger reference tool could not fit due to the ball tool diameter.
5. In the **Reference Tool Diameter** box, type **5**.

   The blade and hub finish operations both use a 5 mm diameter ball mill.
6. Use the default values for the remaining options.

   Overlap on Hub = 0.0000

   Overlap on Blade = 0.0000

   Cut Band = **Stepovers**.

Number on Hub = 3

Number on Blade = 3

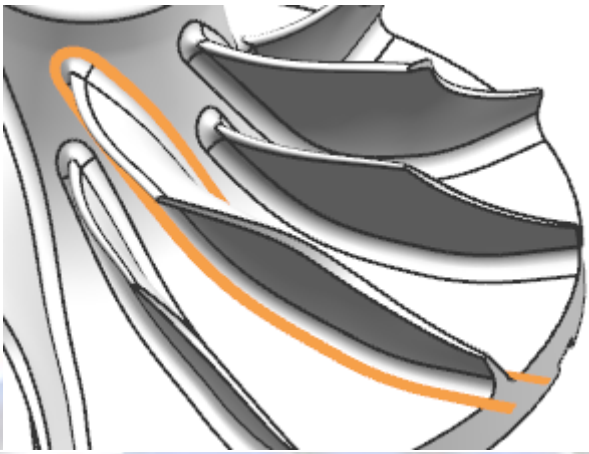
Cut Pattern = **Zig**

Sequencing = **Steep First**

Cut Direction = **Climb**

Start Point = **Trailing Edge**

7. In the **Preview** group, click **Display**  .



8. Evaluate the preview display.

   The tool path is not trimmed, so the tool fits into all regions.

9. Click **OK**.

   The **Blend Finish** dialog box is displayed.

Adjust the cutting parameters

1. In the **Path Settings** group, click **Cutting Parameters**  .

   The **Cutting Parameters** dialog box is displayed.

2. On the **Tool Axis Control** tab, move the **Axis Smoothing %** slider to **100**.

   With greater smoothing, the software may not find a gouge-free tool orientation, and will trim the cutting motions.

   You will check the tool path for any trimmed cutting motions.

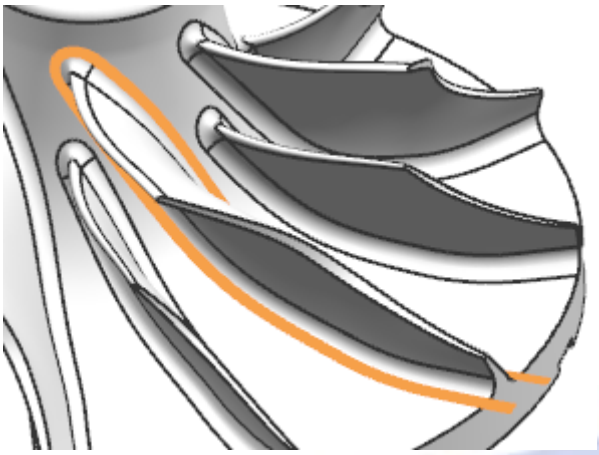
3. Click **OK**.

   The **Blend Finish** dialog box is displayed.

4. In the **Drive Method** group, click **Blend Finish**  .

The **Blend Finish Drive Method** dialog box is displayed.

5. In the **Preview** group, click **Display**  .

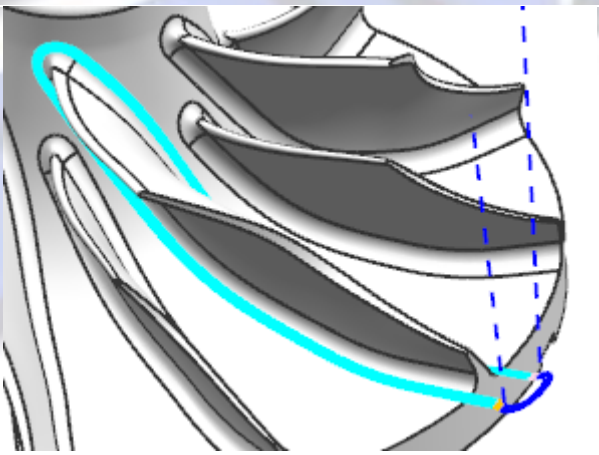


The tool path preview does not show any trimmed cutting motions.

6. Click **OK**.

The **Blend Finish** dialog box is displayed.

7. In the **Actions** group, click **Generate**  .



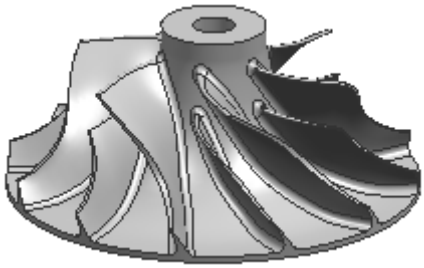
8. Click **OK**.
9. Close the part.

You completed the activity.

Activity: Create an instance of the multi blade blend finish operation

Estimated time to complete: 3–5 minutes

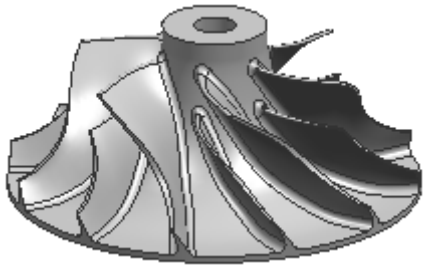
In this activity, you will create a multiple instances of the multi blade hub finish operation.



Launch the Create an instance of the multi blade blend finish operation activity.

Open the part and open the Transformations dialog box

1. Open *mnf16_multi_blade_blend_finish_instance*.

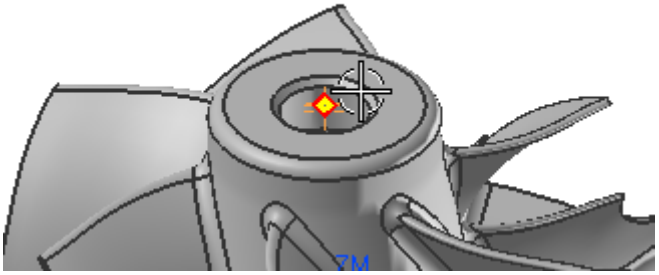


2. In the **Operation Navigator** right-click **BLEND_FINISH** and choose **Object→Transform**.

The **Transformations** dialog box opens.

Select the transformation parameters

1. From the **Type** list, select **Rotate About a Point**.
2. On the **Selection** bar, from the **Selection scope** list, select **Entire Assembly**.
3. In the **Transformations Parameters** group, click **Specify Pivot Point**.
4. In the graphics window, select the hub center arc.



5. In the **Angle** box, type **360**.
6. In the **Result** group, select **Instance**.
7. In the **Distance/Angle Divisions** box, type **6**.
8. In the **Number of Instances** box, type **5**.
9. Click **OK**

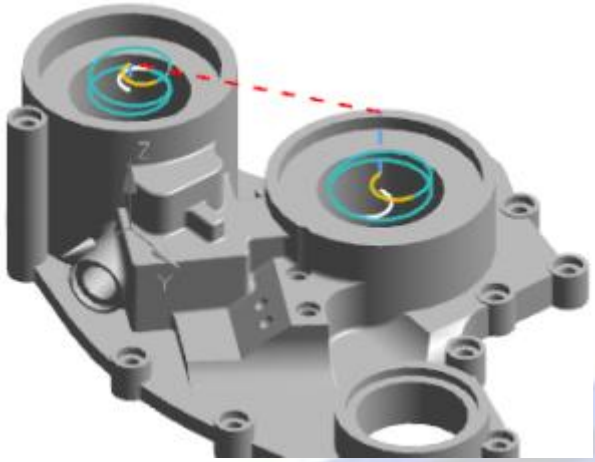
   The instances are displayed in the graphics window and added to the operation navigator.

You completed the activity.

Activity: Create a Hole Milling operation

Estimated time to complete: 15–25 minutes

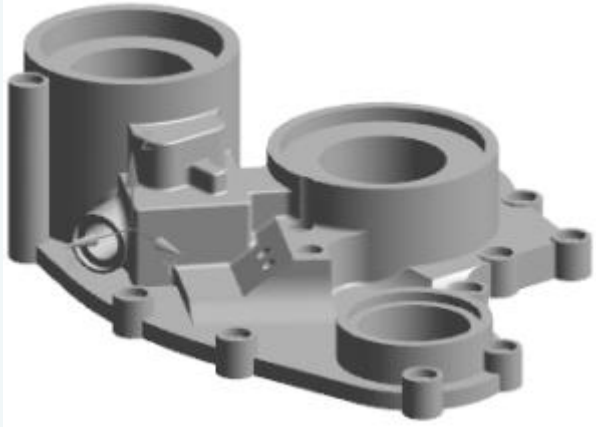
In this activity, you will create a **Hole Milling** operation with a helical cut pattern.



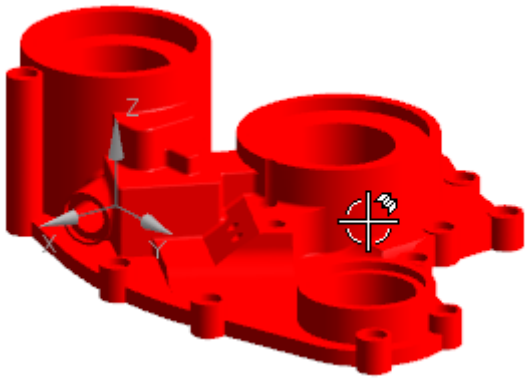
Launch the Create a Hole Milling operation activity.

Specify the Workpiece geometry

1. Open *mnf15_hole_milling*.



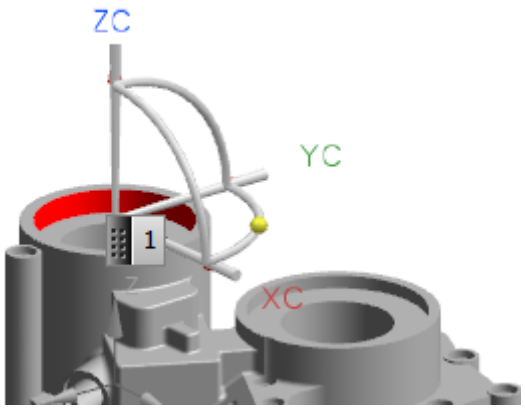
2. Right-click in the background of the **Operation Navigator** and choose **Geometry View**.
3. Double-click **WORKPIECE**.
4. In the **Workpiece** dialog box, in the **Geometry** group, click **Specify Part**  .
5. Select the part geometry as shown.



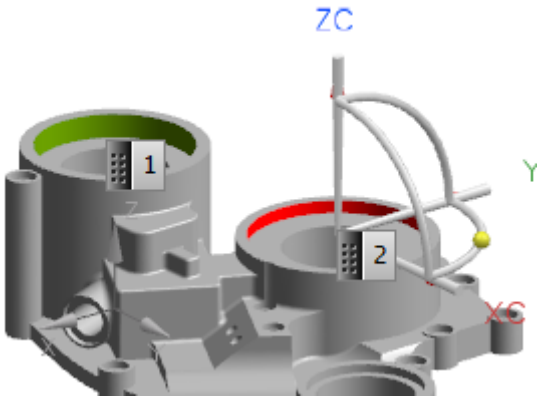
6. In the **Part Geometry** dialog box, click **OK**.
7. In the **Workpiece** dialog box, click **OK**.

Define the hole geometry

1. On the **Insert** toolbar, click **Create Geometry**  .
2. In the **Create Geometry** dialog box, in the **Type** group, from the list, select **mill_planar**.
3. In the **Geometry Subtype** group, click **HOLE_BOSS_GEOM**  .
4. In the **Location** group, from the **Geometry** list, select **WORKPIECE**.
5. Click **OK**.
6. In the **Hole or Boss Geometry** dialog box, in the **Type** group, from the list, select **Hole**.
7. Select the first hole diameter as shown.



8. From the **Depth Limit** list, select **Blind**.
9. Select the second hole diameter as shown.



10. Click **OK**.

Create the Hole Milling operation

1. On the **Create** toolbar, click **Create Operation** or choose **Insert→Operation**.
2. In the **Type** group, from the list, select **mill_planar**.
3. In the **Operation Subtype** group, click **HOLE_MILLING**.
4. In the **Location** group, set the options as shown.

Program	**PROGRAM**
Tool	**END_MILL_2**
Geometry	**HOLE_BOSS_GEOM**
Method	**METHOD**

5. Click **OK** to create the operation.

Specify the operation parameters

1. In the **Hole Milling** dialog box, in the **Path Settings** group, from the **Cut Pattern** list, select **Helical**.
2. From the **Blank Diameter** list, select **Diameter**.
3. In the **Start Diameter** input box, type **0.500**.
4. From the **Start Diameter** units list, select **in**.
5. Use the default values for the remaining options.

   Under **Axial**:

   - **Depth per Revolution** = **Distance**.
   - **Pitch** = 25.0 **%Tool**

   Under **Radial**:

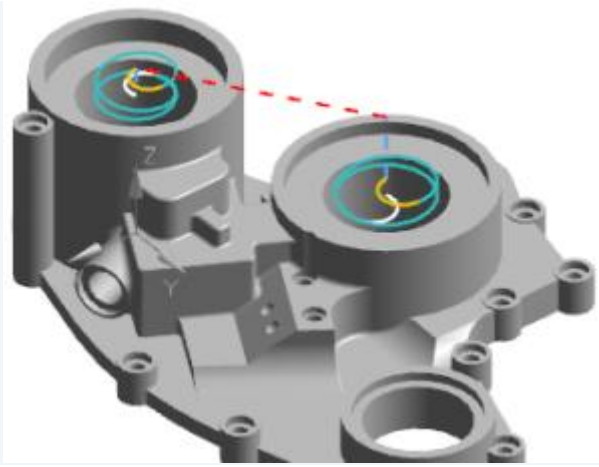
- **Radial Stepover** = **Constant**
- **Maximum Distance** = 50.0 **%Tool**

Turn on the cleanup passes

1. Click **Cutting Parameters**  .
2. In the **Cutting Parameters** dialog box, click the **Strategy** tab.
3. In the **Cleanup Passes** group, select the **Add Cleanup Passes** check box.
4. Click **OK**.

Generate the operation

1. In the **Hole Milling** dialog box, in the **Actions** group, click **Generate**  .



2. Click **OK** to save the operation.
3. Close the part.

You completed the activity.

Activity: Tilt Tool Axis

Estimated time to complete: 10–12 minutes

In this activity, you will use the **Tilt Tool Axis** option to avoid tool holder collisions with the part and reach greater depths using the shortest tools possible . NX replaces the colliding 3-axis motions with 5-axis motions to avoid the collision.

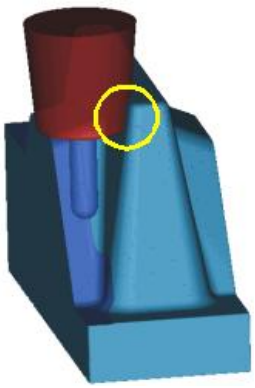
Launch the Tilt Tool Axis activity.

Verify the existing tool path

1. Open *mnf15_electrode*.
2. In the **Operation Navigator**, right-click in the background and choose→**Geometry View**.
3. Right-click **FIXED_CONTOUR** and choose **Tool Path→Verify**.
4. In the **Tool Path Visualization** dialog box, click the **2D Dynamic** tab.

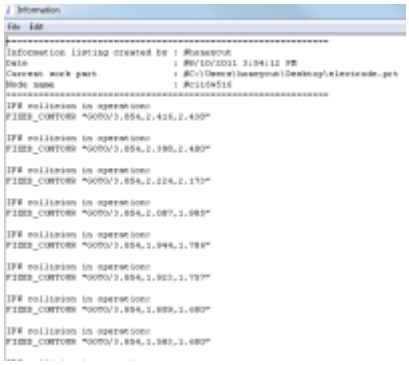
   Make sure that the **Check for IPW Collisions** and **Check Tool and Holder** check boxes are selected.
5. Move the **Animation Speed** slider to **3**.
6. Click **Play**.
7. In the **No blank** message box, click **OK**.
8. In the **Blank Geometry** dialog box, in the **Type** group, from the list, select **Offset from Part**.
9. In the **Offset** group, in the **Offset** input box, type **.030**.
10. Click **OK**.

    Notice the tool holder and part collision along the high ridge of the part. You made need to orient the view similar to the illustration shown to see the interference.



11. In the **Tool Path Visualization** dialog box, click **List**.

    In the **Information** window, NX lists the location of every collision in the tool path. Review some of the coordinates.



12. Close the **Information** window.
13. In the **Tool Path Visualization** dialog box, click **OK**.

Tilt the tool axis to avoid collisions

1. In the **Operation Navigator**, right click **FIXED_CONTOUR** and choose **Tool Path→Tilt Tool Axis**.

   NX replaces the colliding 3-axis motions with 5-axis motions to avoid the collision.
2. In the **Tool Path Tilt** dialog box, review some of the parameters settings and accept the defaults.
3. Click **OK**.

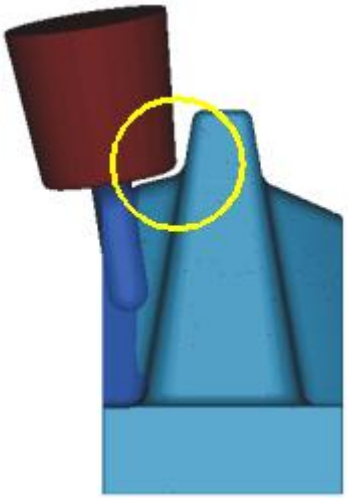
   The tool axis tilt calculation can take a few minutes to complete.

Visually inspect the tool path

1. In the **Operation Navigator**, right-click **FIXED_CONTOUR** and choose **Tool Path→Verify**.
2. In the **Tool Path Visualization** dialog box, click the **2D Dynamic** tab.

   Make sure the **Check for IPW Collisions** and **Check Tool and Holder** check boxes are selected.
3. Move the **Animation Speed** slider to **3**.
4. Click **Play** .
5. In the **No blank** message, click **OK**.
6. In the **Blank Geometry** dialog box, from the **Type** list, select **Offset from Part**.
7. In the **Offset** group, in the **Offset** box, type **.030**.
8. Click **OK**.

   Notice how the tool axis tilts away from the part to avoid collisions.



After the verification is complete, notice that the **List** option is no longer available in the **Tool Path Visualization** dialog box. The option is not available because no collisions were detected.

9. Click **OK**.

   If you regenerate the operation in which NX tilted the tool path, the tilted tool path is not maintained, and the tool path reverts to being a 3-axis tool path.

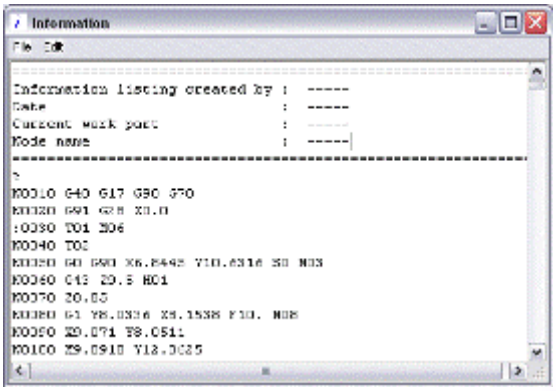
10. Close the part.

You completed the activity.

Activity: Post process tool paths

Estimated time to complete: 3–5 minutes

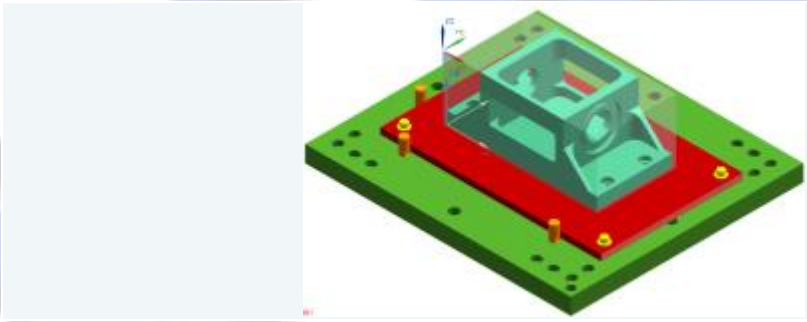
In this activity, you will post process tool paths using the NX POST post processor.



Launch the Post process tool paths activity.

Post process the tool path

1. Open *mnf3_bearing_case_mfg_3*.

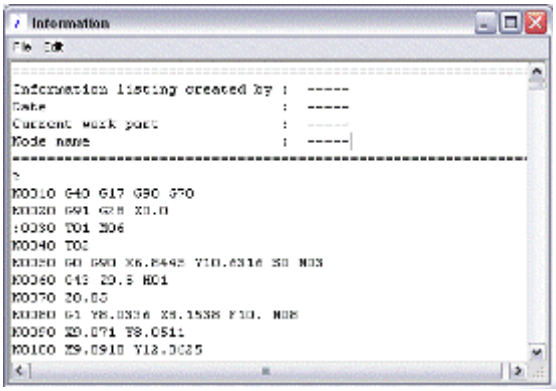


2. In the **Operation Navigator**, right-click in the background and choose **Program Order View**.
3. Right-click in the background again and choose **Expand All**.
4. Click the **PROGRAM** folder.
5. On the **Operations** toolbar, click **Post Process**  .
6. In the **Postprocess** dialog box, in the **Postprocessor** group, from the list, select **MILL_3_AXIS**.
7. In the **Output File** group, accept the default name in the **File Name** input box.

   You need to specify a name and location for the posted output file that you are about to create. The default location is your current directory.

8. Click **OK**.

   The **Information** window is displayed containing the post processed output for the three tool paths CAVITY_MILL, FINISH_WALLS, and SPOT_DRILLING operations. View the output of the program.



9. Close the **Information** window.
10. Do not save or close the part. You will use the same part in the next activity.

You completed the activity.