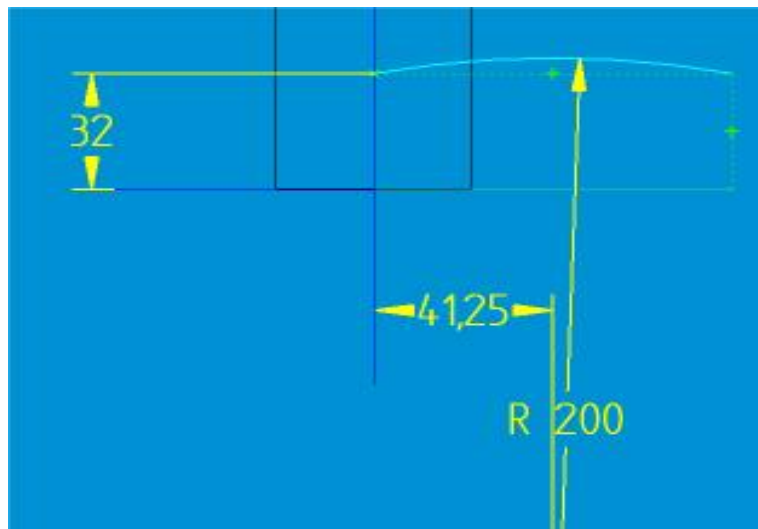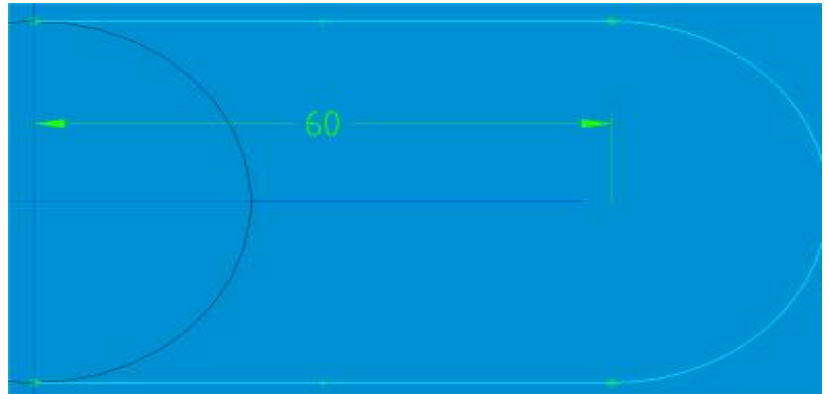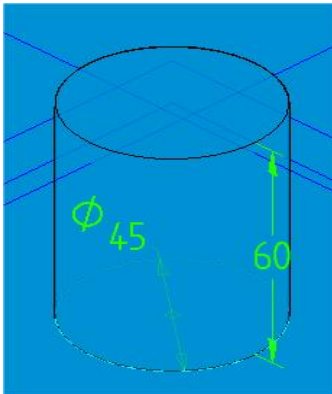# Activity 1 – Construction Surfaces

When you complete this activity, you will know how to use construction surfaces to construct a B-spline path for use in a sweep feature. You should also know how to use a construction surface to modify model topology.

1. You will learn the following skills:

- How to use construction surfaces to create a part.

- How to use intersection curves to create a swept protrusion.

- How to use the replace surface command.

2. You will now create a two construction surfaces.

❑ On the Constructions toolbar, click the Extruded Surface command. If this toolbar is not active in the Part environment, go to View > Toolbars > Constructions.

❑ Using the Extruded Surface command on the construction toolbar, construct two construction surfaces. Extruding the U-shaped bottom sketch upward 40 mm, and extrude the arc symmetrically 70 mm. If the sketches are not displayed, in EdgeBar turn on the display of Sketch1 and Sketch2.
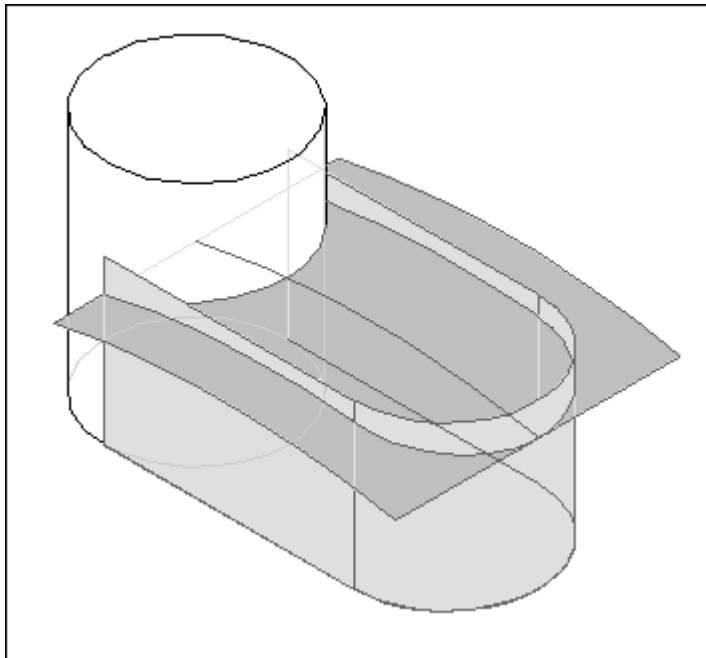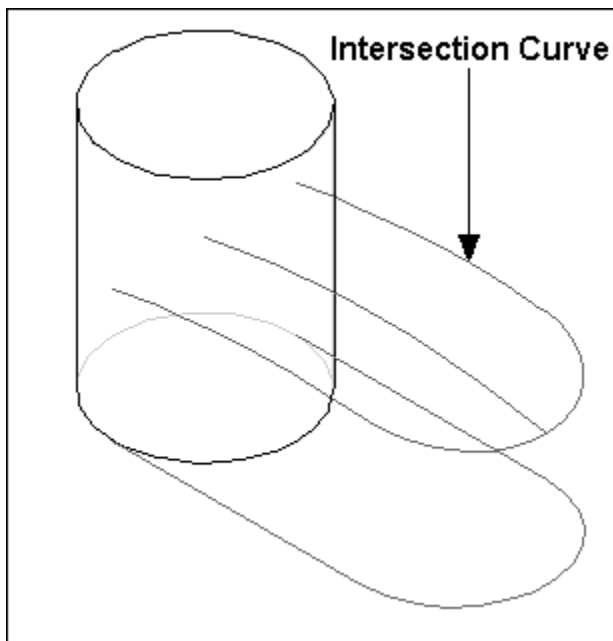

3. You will now create an intersection curve between the two construction surfaces.

❑ On the Constructions toolbar, click the Intersection Curve command, and construct an intersection curve between the two construction surfaces.

❏   Select the surface created from the arc sketch.

❏   On the ribbon bar, click the accept button.

❏   Change the selection filter on the ribbon bar from Face to Chain.

❏   Identify the surface created from the U-shaped sketch.

❏   On the ribbon bar, click the accept button, and then click Finish. When the curve is constructed, look in Feature Pathfinder for the curve entry.

❏   To hide these construction surfaces, right-click on the construction surface entry in PathFinder, and then click Hide.

❏   Fit the view to see the intersection curve.

4. You will now create a swept protrusion using the intersection curve as the path curve.

❏   In Feature PathFinder, use the shortcut menu to show Sketch 3 and Sketch 4 and to hide Sketch 1 and Sketch 2.

❑
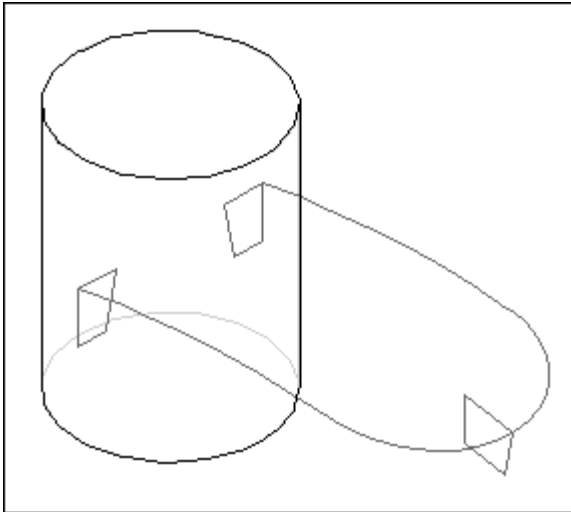
❑   To construct a swept protrusion, use the intersection curve as the path and the sketches as the cross-sections. Provide a consistent start point for the multiple cross-sections used in this command, or the swept surface may twist.
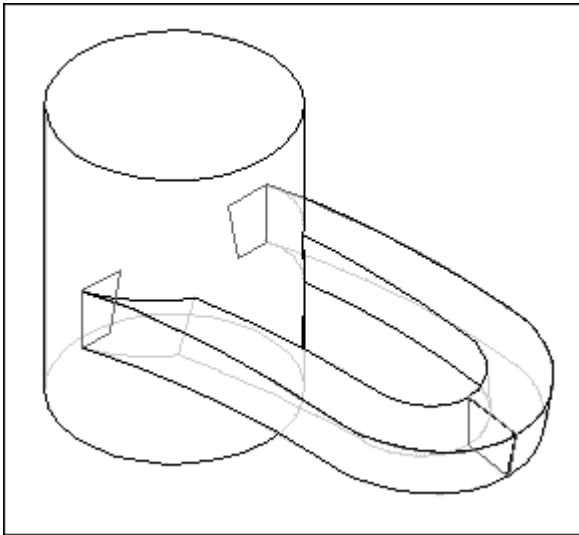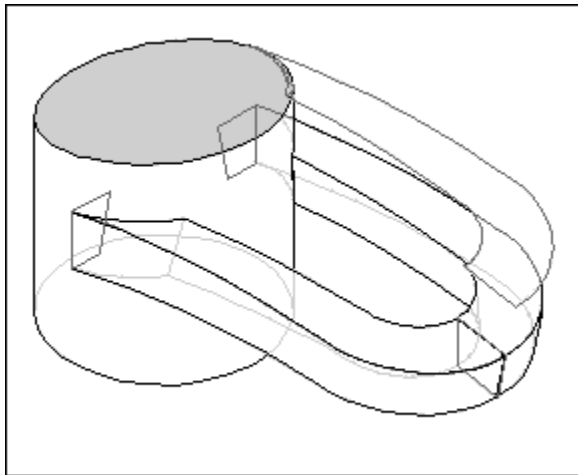

❑   Hide all sketches, intersection curves, and construction surfaces.

5. You will now modify the top face of the cylinder using the offset surface and replace face commands.

❑   Offset one side of the top handle surface 12 mm in the direction shown.

❑
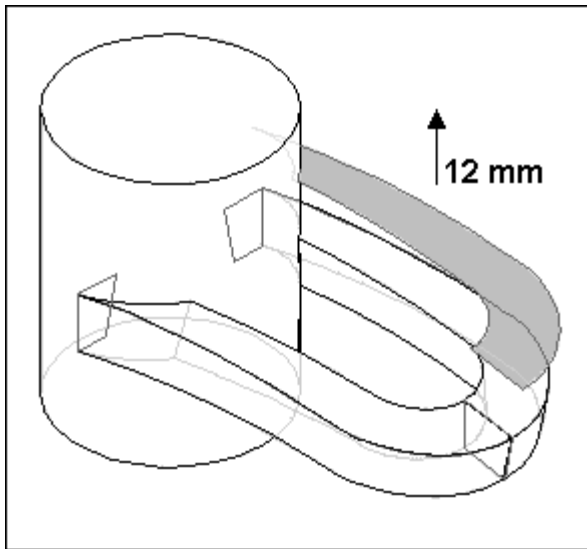
❑   On the Constructions toolbar, use the Replace Face command to replace the top of the cylinder with the surface you just offset. Follow the prompts in the lower left of the screen.
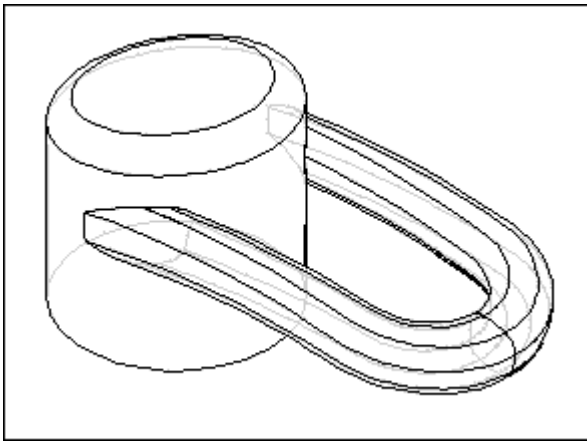

❑   Hide all construction surfaces, intersection curve and sketches.
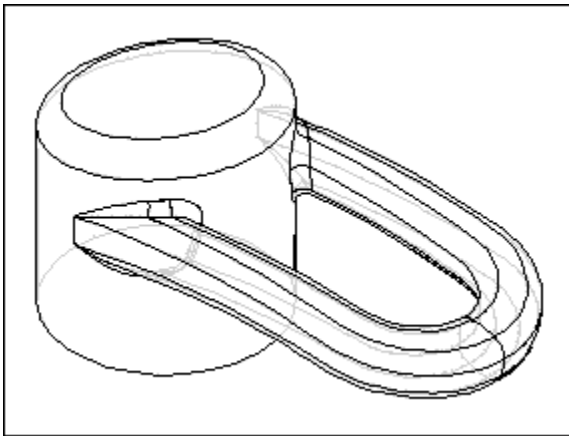
6. You will now add rounds to part edges.

❑   Round the edge of the top of the cylinder with a 5 mm round.

❑
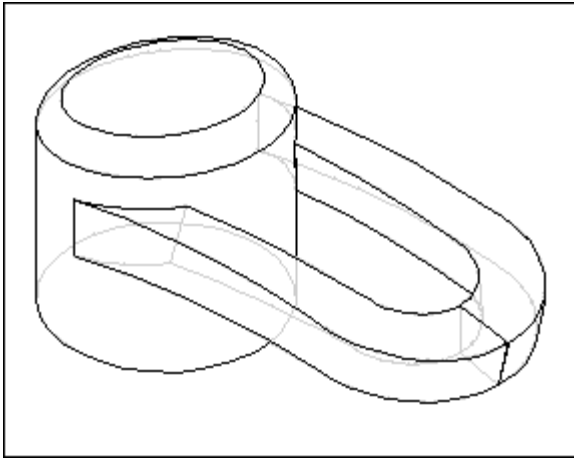
❑ Round the four handle edges (the two top and two bottom edges of the handle) with a 3 mm round.

❑ Round the connect edge between the handle and the cylinder with a 2 mm round.

7. Save and close this file. This completes the activity.

# Activity 2 – Plastic Injection Molded Part

The following activity is designed to reinforce concepts examined throughout this module. This activity describes only one of many possible ways to create the part shown below.
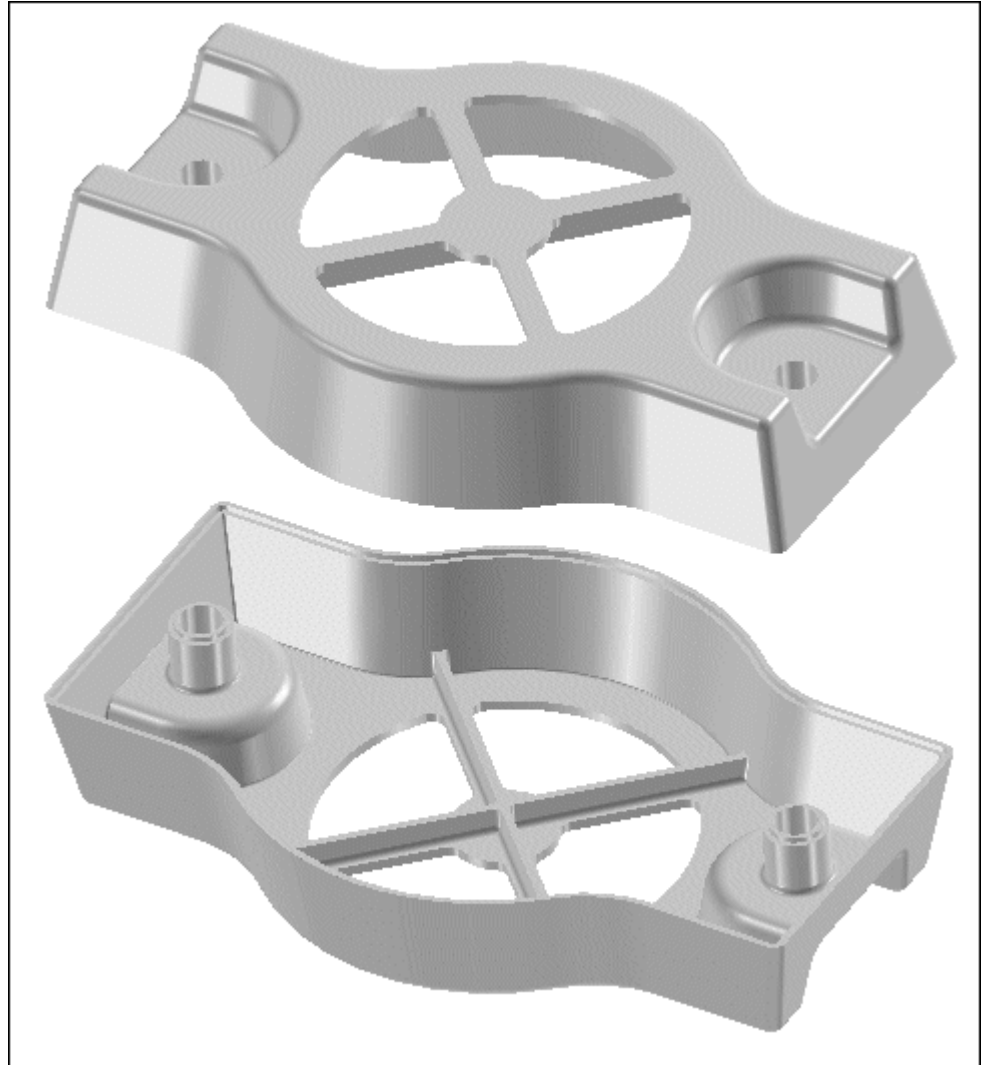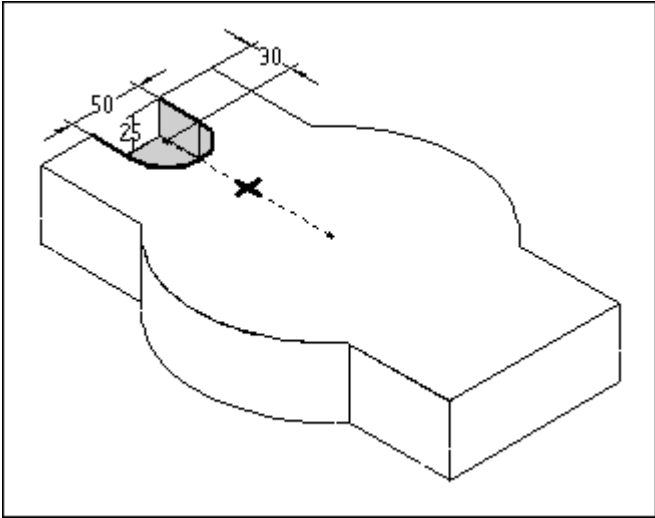
1. Create a new part file using the metric template.

2. You will now construct the base feature.

   ❑ Construct a protrusion on the reference plane shown. Draw the profile shown and anchor and center the profile to the center of the reference plane. The protrusion height is 50.8 mm.
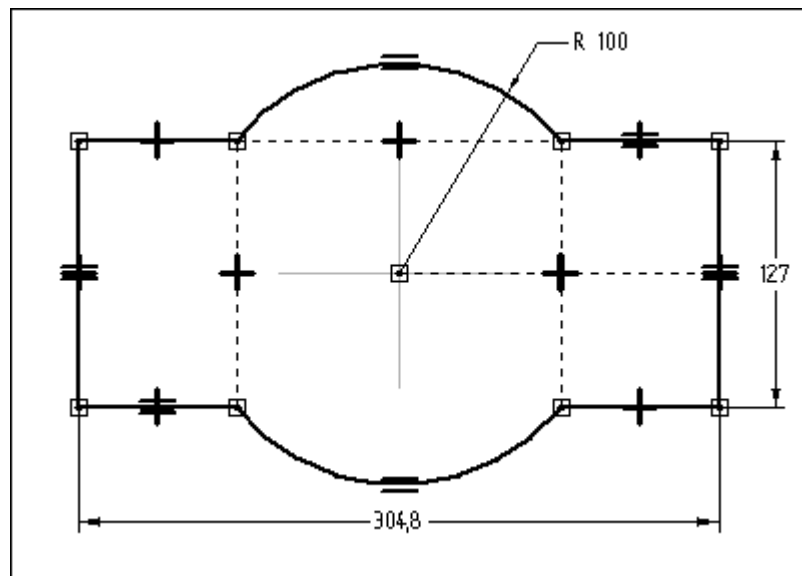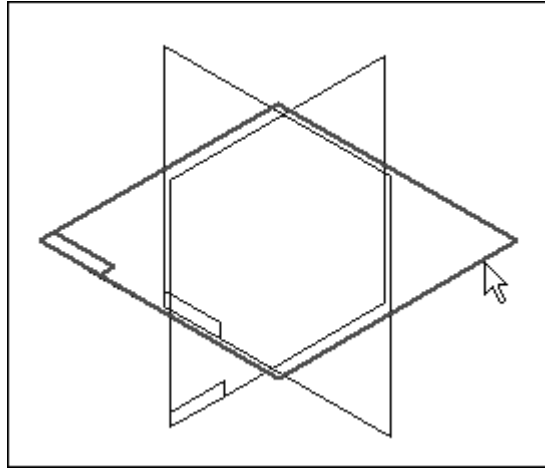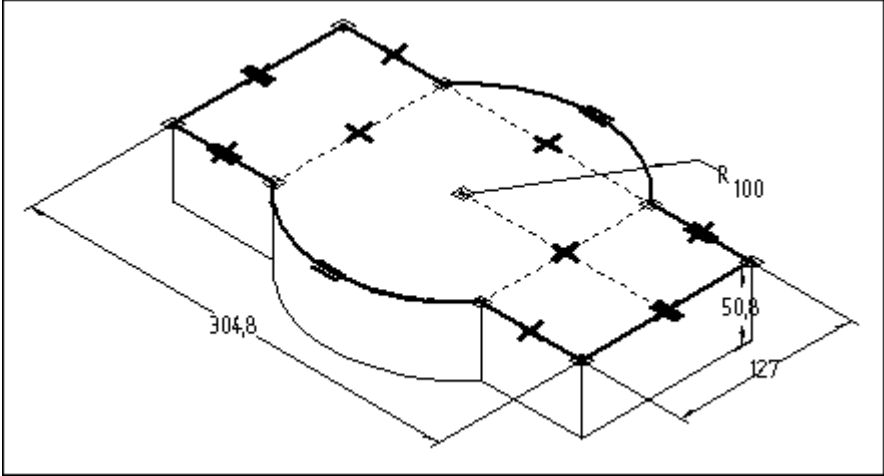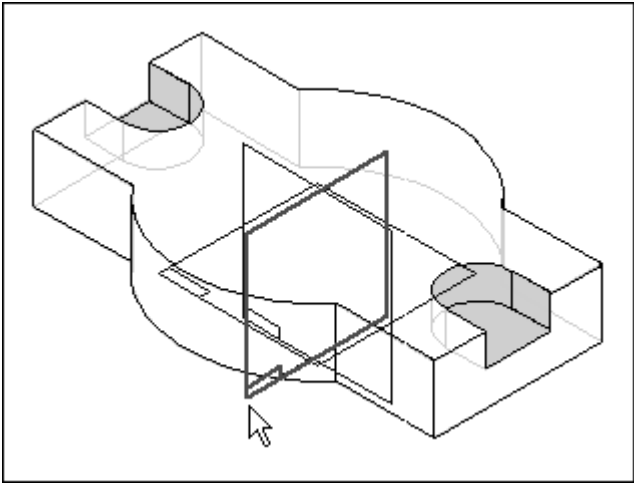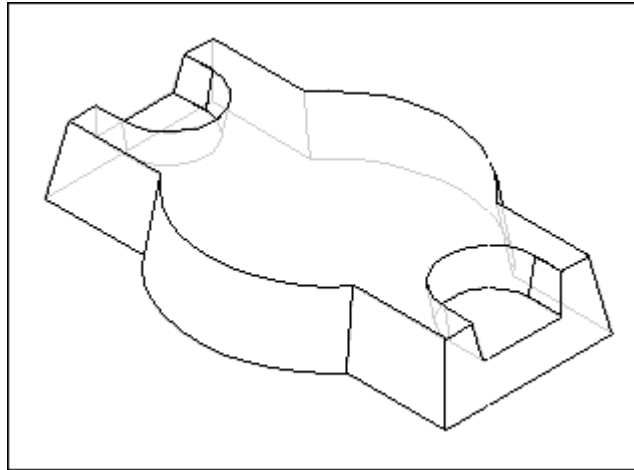
3. You will now add a cutout 25 mm deep as shown.

4. Mirror the cutout you created in the previous step using the reference plane shown.
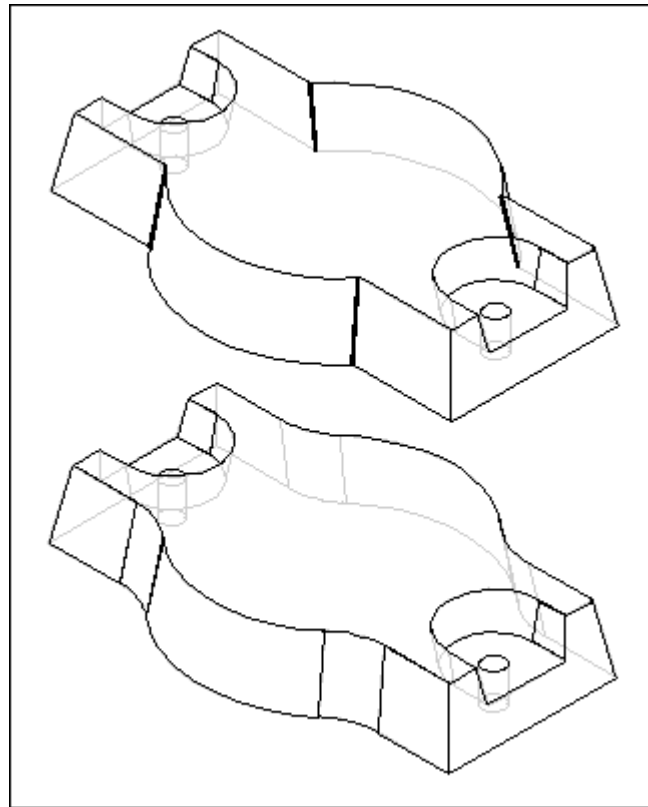
5. Add a 10-degree draft to the model. Use the bottom face of the model as the drafting plane, and draft inward from all sides perpendicular to the bottom of the model, including the sides created from the cutouts.
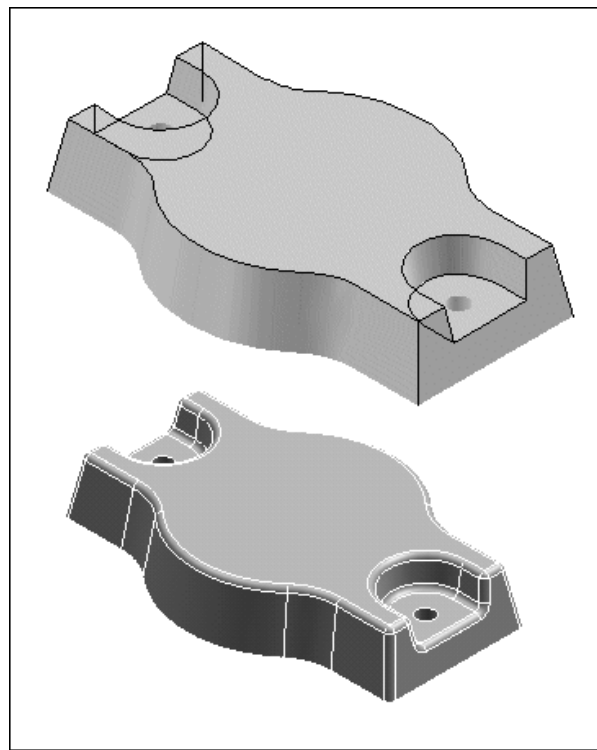

6. Add two 16 mm holes centered on the arcs of the cutouts.
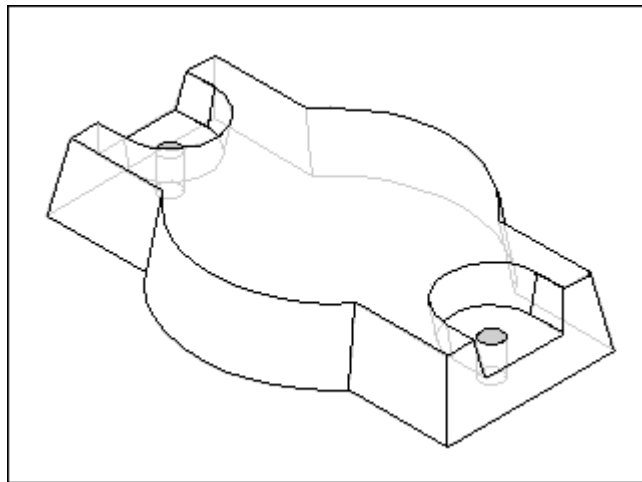
7. Add a 50 mm radius round to the four edges shown in the illustration.

8. Add a 5 mm round to all edges shown in the illustration. Do not select hole edges nor edges around the perimeter of the bottom face.

9. Thin wall the model, specifying a wall thickness of 4 mm. Leave the bottom face of the model open. To view the thin wall result more clearly, the model shown in the illustration has been rotated 180-degrees.
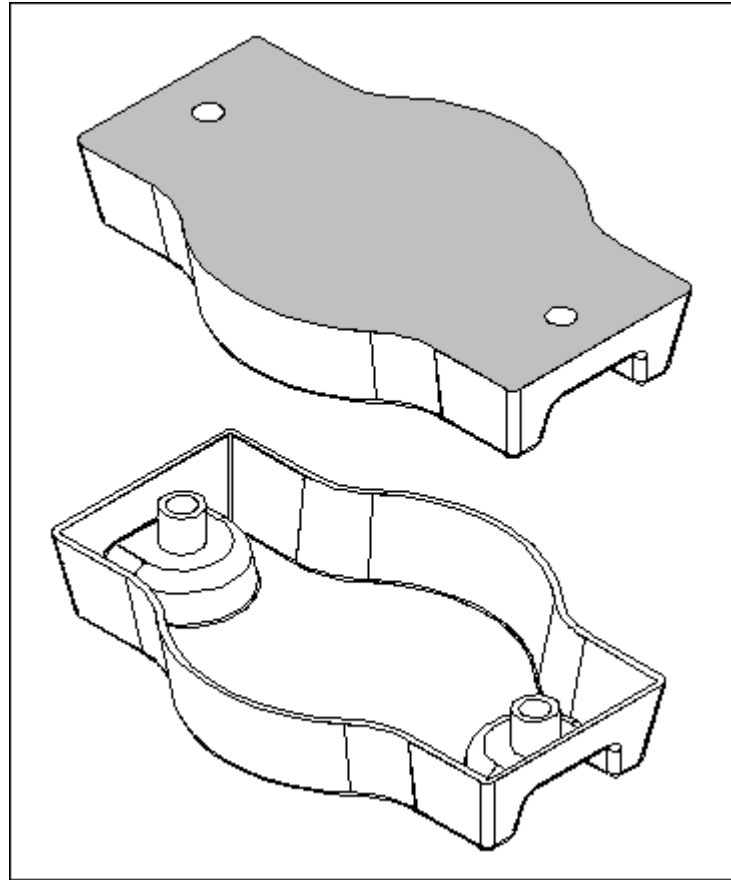
10. Use construction geometry to construct a cutout as shown.

11. Add a 5 mm radius round to the four thickness edges created by the cutout you constructed in the previous step.
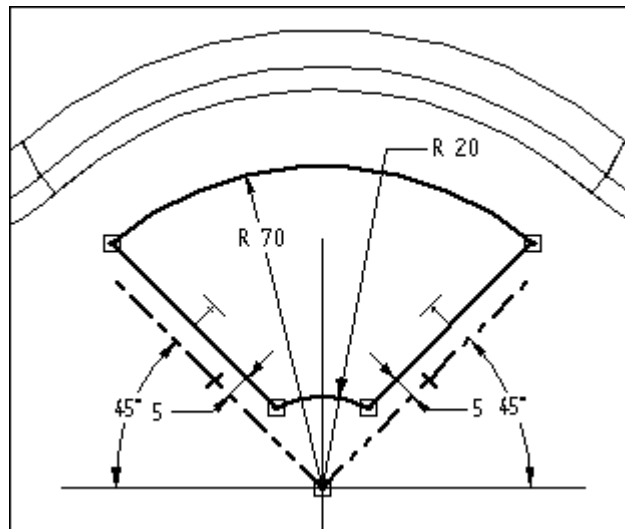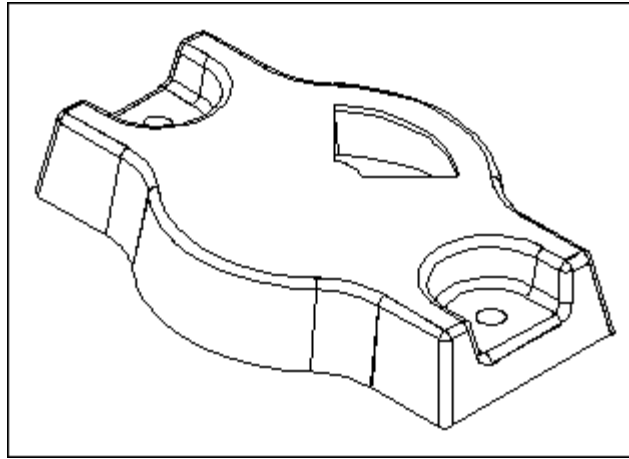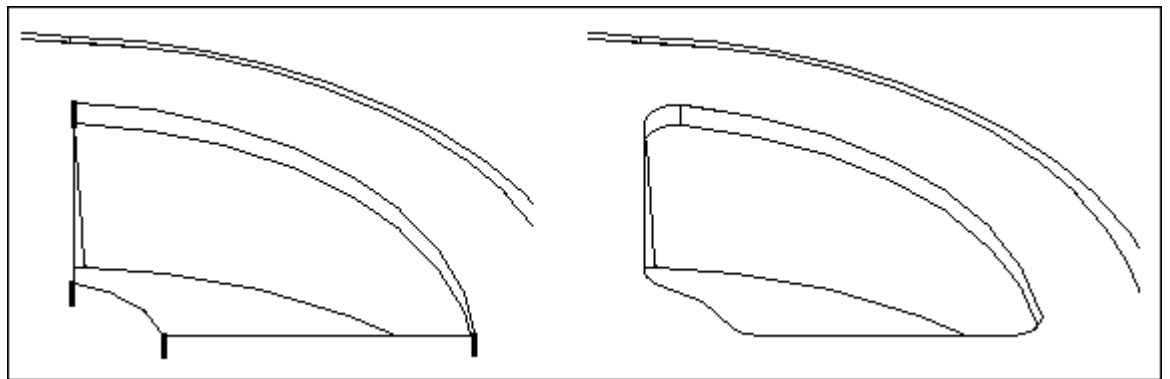

12. Create a circular pattern using the cutout and the rounds you created in the previous two steps. The pattern should have four evenly spaced instances as shown in the illustration.
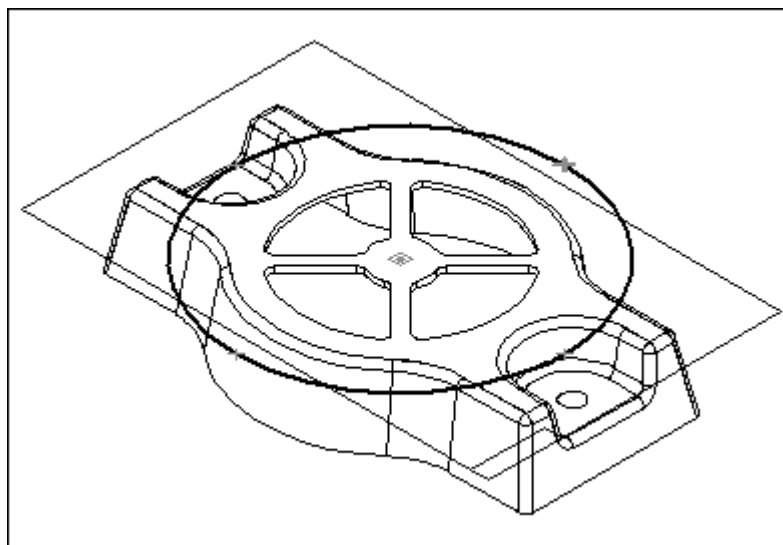
13. Construct reinforcement ribs as shown. Use the Web Network command with a parallel reference plane 10 mm off the face shown in gray. Make the ribs 3 mm thick.
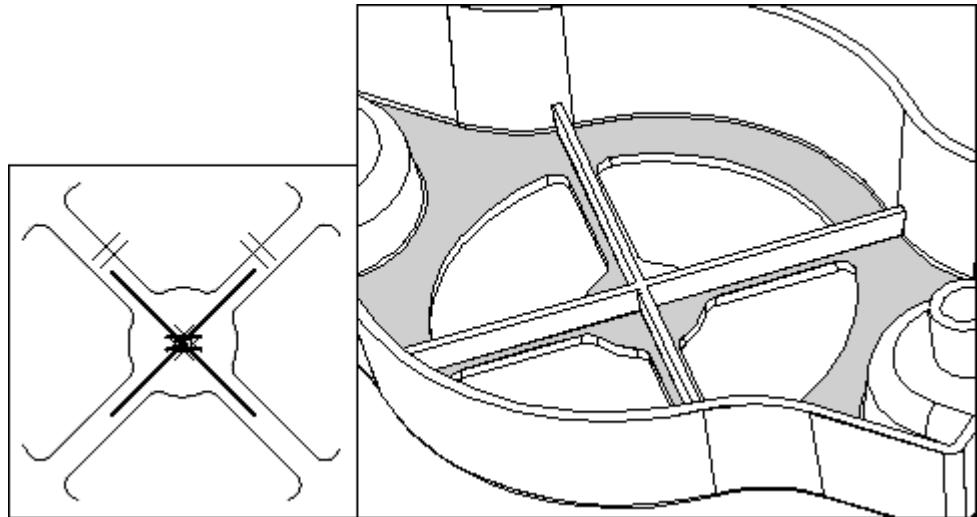
14. Add a 2.5 mm radius round to the edges of the ribs and the bosses.
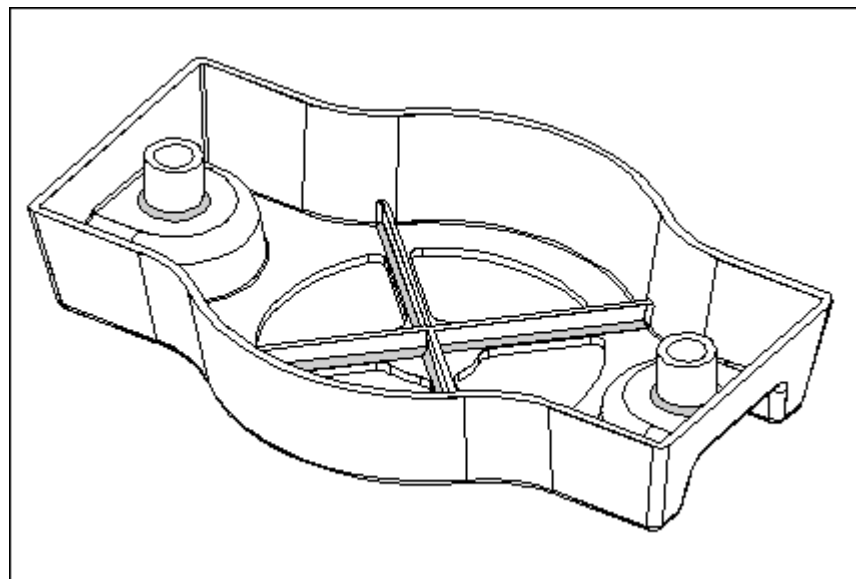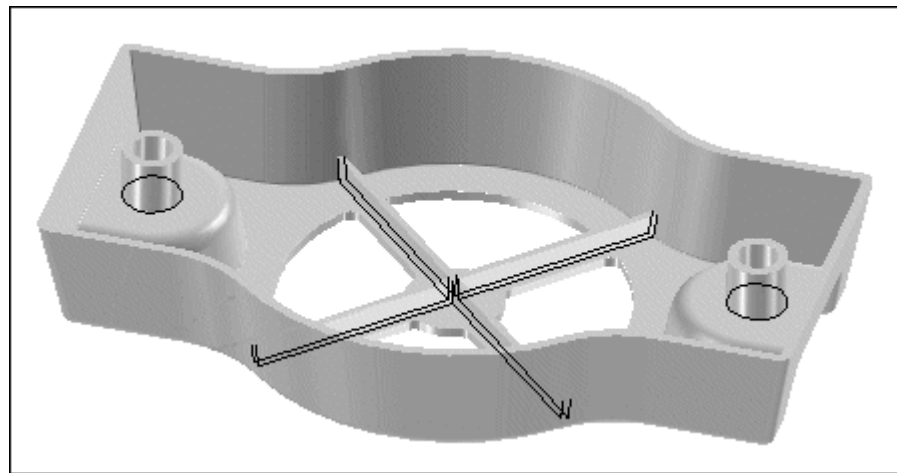
15. Add a 2 mm square groove to the perimeter of the model's bottom and a 2 mm lip to the bosses. Use the Lip command for this.
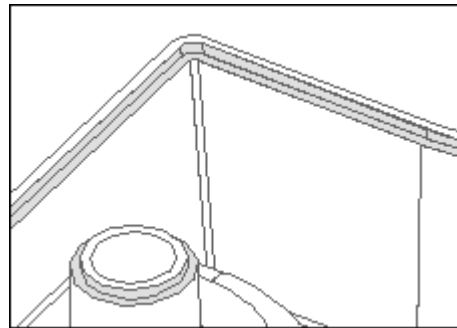
16. Save and close the file. This completes the activity.

Construction Surfaces

After you have broken down the part, established the feature order, and ascertained what profiles are needed, you must decide if construction surfaces will play a role in helping to define part feature shapes. If construction surfaces are needed, you must determine the best way to work the surface into the model file. Solid Edge can develop construction surfaces from the following methods:
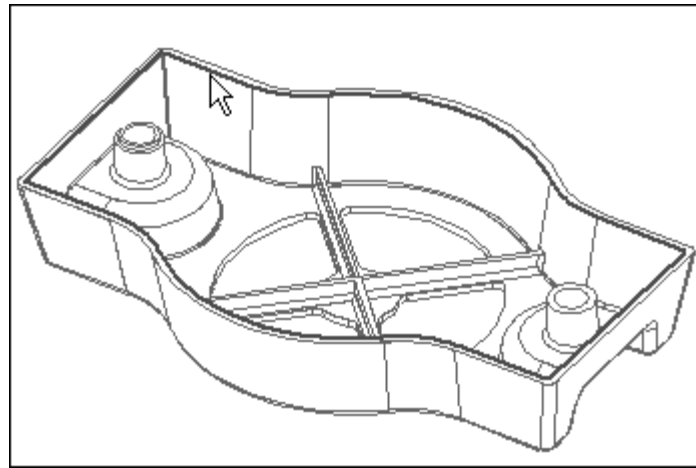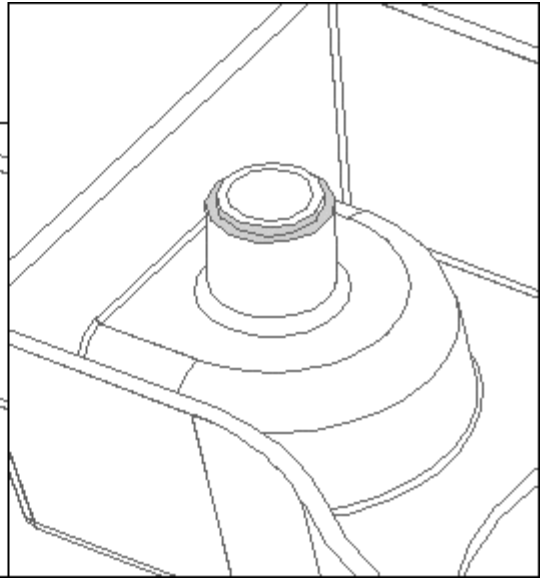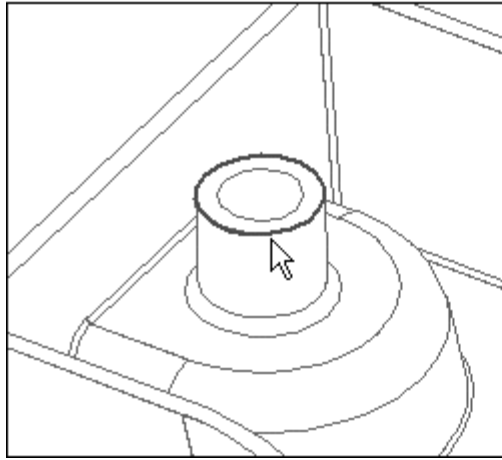
- Generate them directly from the object model.

- Generate them using the construction projection commands.

- Generate them from a table of points.

- Import them from existing files.

Construction surfaces are completely disjoint from the solid model. They exist in the model file but are not used or computed in other environments. Construction surfaces in a model file will not display in assembly mode. If the same model file is used with the Insert Part Copy command, the model displays without any of the original construction surfaces.

The most common use for construction surfaces is for non-planar extent criteria. You can define a construction surface as an extent with the From/To extent option. Another common use for construction surface commands is developing 3-D curves for use with the Swept command. The software offers no method for interactively sketching a 3-D curve, but you can use construction geometry to develop such a curve.

Construction geometry can be created with commands on the construction toolbar shown in an illustration at the beginning of this topic. The commands on this menu work the same way as command on the Features toolbar. Selecting a construction surface feature command invokes a ribbon bar with SmartStep and filter options. Imported surfaces automatically become construction geometry since they are not solid data.

Extruded Surface

The Extruded Surface command is similar to the Protrusion command. It creates a surface by projecting an open profile across a distance. Options are available to control the extents of the surface.
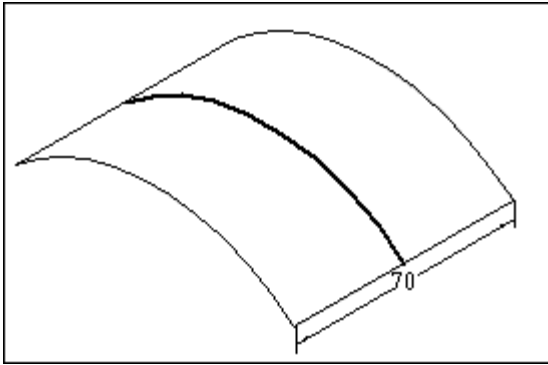
Revolved Surface

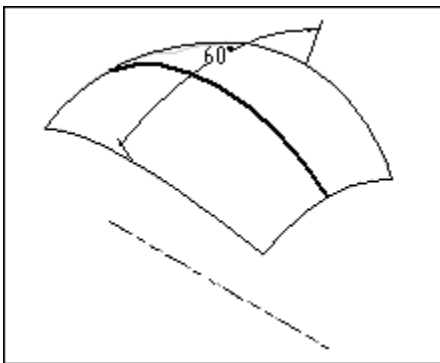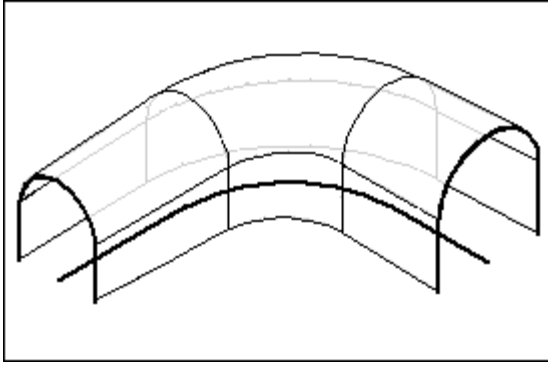   The Revolved Surface command is similar to the Revolved Protrusion command. It creates a surface by revolving a profile about an axis. Options are available to control the extents of the surface.

Swept Surface

    The Swept Surface command is similar to the Swept Protrusion command. It creates a surface based on paths and cross-sections.

Lofted Surface

The Lofted Surface command is similar to the Lofted Protrusion command. It creates a surface based on multiple cross-sections.
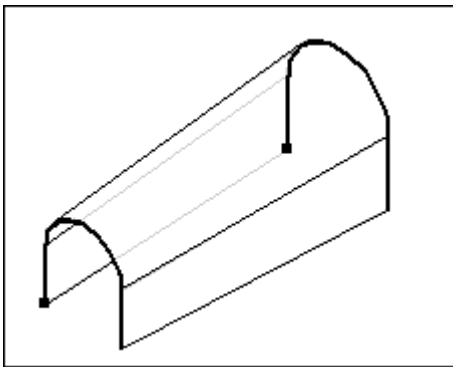
Offset Surface

The Offset Surface command constructs a surface copy of a model face, a reference plane, or a construction surface. The copy possesses identical attributes to the original surface and is offset a specified distance from the original surface. This offset surface is associative to the original surface.
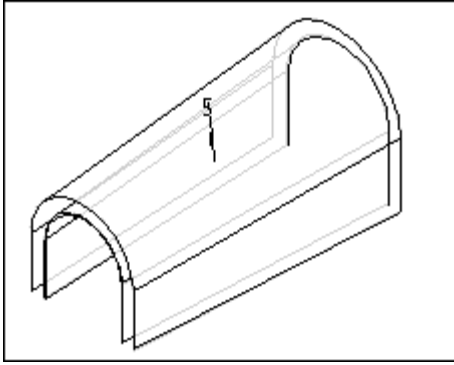
Stitched Surface

The Stitched Surface command stitches together multiple adjacent construction surfaces to form a single construction surface feature. This command is very useful for joining imported surfaces.

Intersection Curve

The Intersection Curve command creates an associative curve from the intersection of two surfaces. The surfaces can be any combination of reference planes, model faces, or construction surfaces.
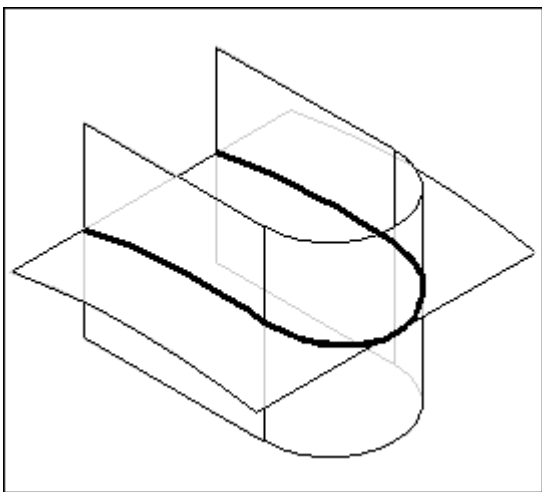
Curve By Table

The Curve By Table command uses an Excel spreadsheet to define a construction curve in the Part and Sheet Metal environments. The spreadsheet is embedded in the Solid Edge document. This allows you to better import and manage engineered curves. You can create a curve by creating a new spreadsheet or by identifying an existing spreadsheet.

Intersection Point

The Intersection Point command creates an associative point at the intersection of a curve with any curve surface, model face, or reference plane.
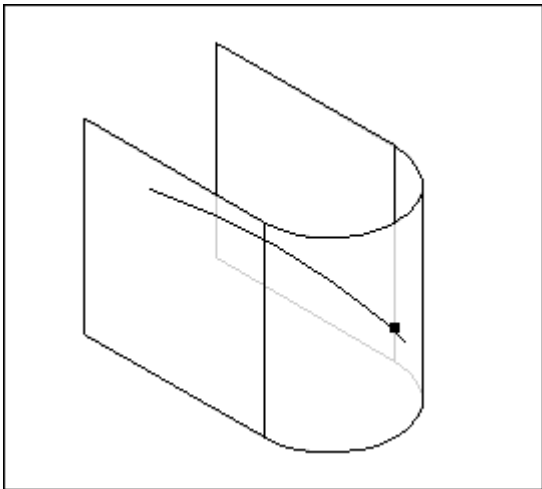
Project Curve

The Project Curve command projects a 2-D sketch onto the 3-D object. You can specify a vector projection or else project along the normals of the surface. If the curve is projected normal to the surface, the shape of the curve will change.

Project along Vector   Project along Normals
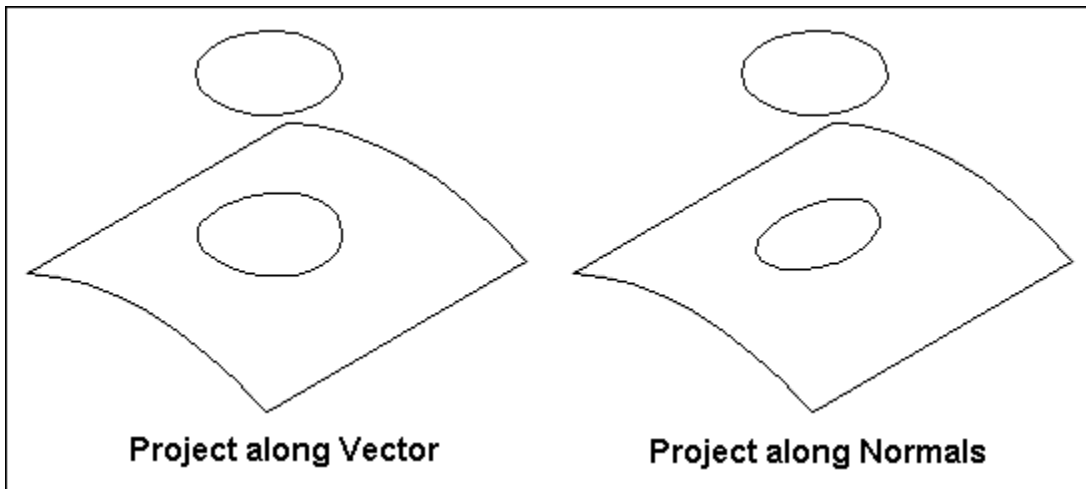
Keypoint Curve

The Keypoint Curve command creates a 3-D curve by forcing a curve through a series of points. The points can be accessed from curves, surfaces, or the solid model.


Top View   Isometric View
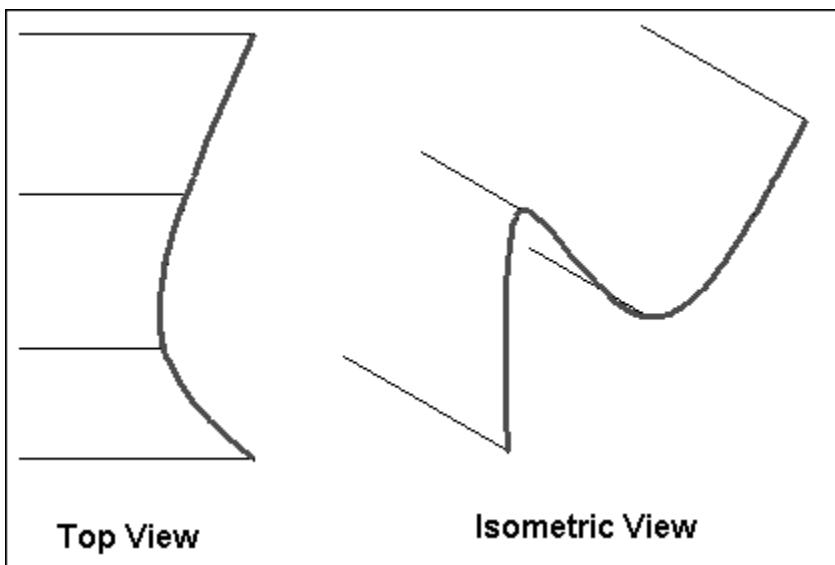
Replace Face

The Replace Face command replaces a surface on the model with another surface. The replacement surface is typically a construction surface such as an extruded or revolved surface. The adjoining faces of the solid are extended or trimmed to the replacement face. In the illustration, the top face of the solid is replaced with the lofted construction surface.
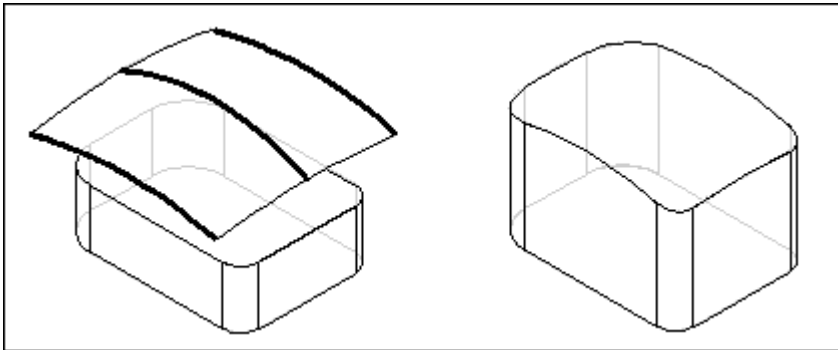
Divide Part

The Divide Part command separates a part into multiple parts and uses the Insert Part Copy command to save the parts as separate files. The new parts are associative to the original whole part. This command is typically run on molded parts. This command is very useful when you design a part and realize it should be divided into separate parts. Reference or construction planes are used as the dividing entities for the part.
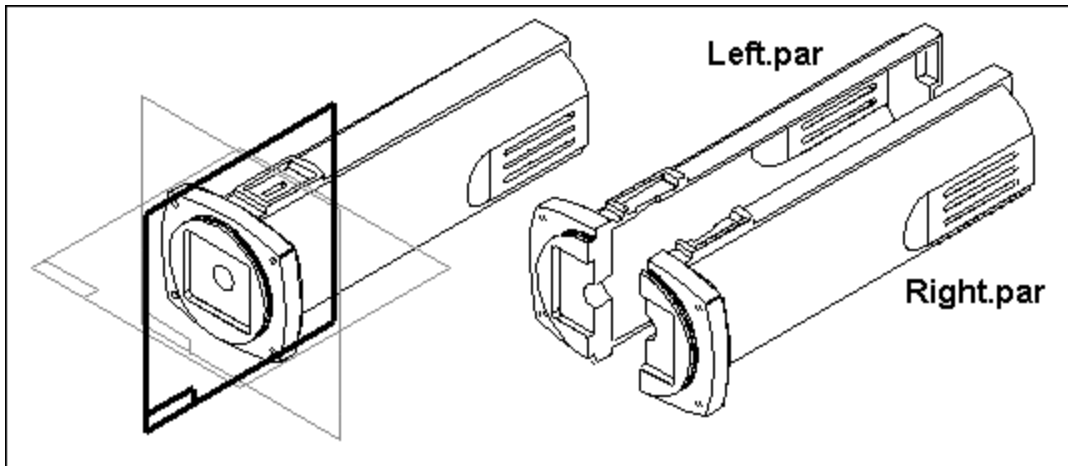
Boolean Feature

The Boolean Feature command unites, subtracts, or calculates the intersection of surfaces or solids. A typical workflow for the Boolean Feature is creating a cavity for a die. For example, if an extruded die is required, you can model the finished part and use the Insert Part Copy command to insert the part into a new file. A solid can then be created, and the Boolean feature is used to subtract the extruded part from the new solid. Use the ribbon bar to control whether the object is added, subtracted, or intersected.