Modeling a part using surfaces

Self-Paced Training

spse01560

Contents

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Course introduction

Welcome to Solid Edge self-paced training. This course is designed to educate you in the use of Solid Edge. The course is self-paced and contains instruction followed by activities.

Start with the tutorials

Self-paced training begins where tutorials end. Tutorials are the quickest way for you to become familiar with the basics of using Solid Edge. If you do not have any experience with Solid Edge, please start by working through the basic part modeling and editing tutorials before starting self-paced training.

Tutorials

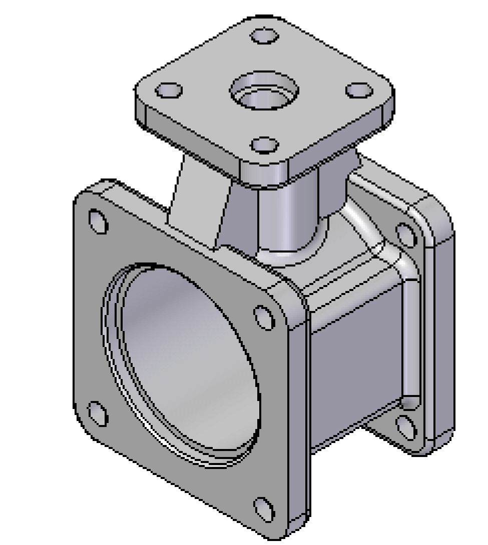
Adobe Flash Player required for videos and simulations

To watch videos and simulations, you must have the Adobe Flash Player version 10 or later installed as a plug-in to your browser. You can download the Flash Player (free) at the http://get.adobe.com/flashplayer

Lesson 1: Surface construction

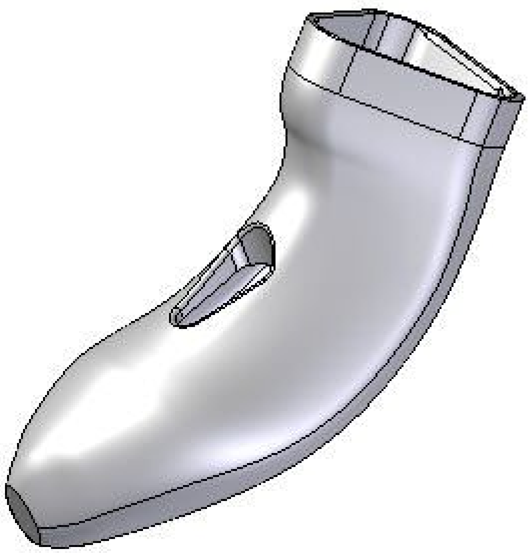
Solid Edge provides two distinct 3D modeling styles: solid modeling and surface construction.

The solid modeling method



1. A product's function is the primary concern and aesthetics are purely an afterthought.
2. Solid Edge is an industry leader of this modeling style and exhibits these additional characteristics:
   - The various modeling operations are identified as features.
   - A history tree of features is maintained.
   - All properties used in defining a feature can be edited at any time.

The surface modeling method



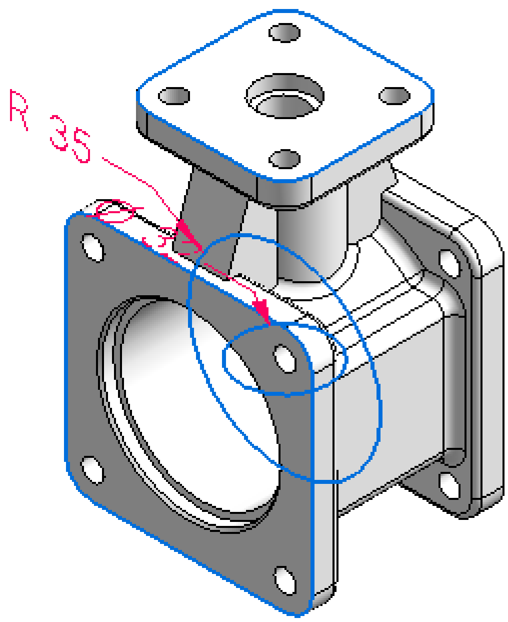
1. Many consumer products are designed using surface modeling techniques due to the market's emphasis on style and ergonomics; therefore, a model's aesthetics is the number one concern and key element in the design process. Product function is only a secondary consideration.

2. Like the solid modeling features, Solid Edge extends this style by making each point, curve, and surface an entity that knows how it was created, and can be edited at any time.

What is surfacing and why use it?

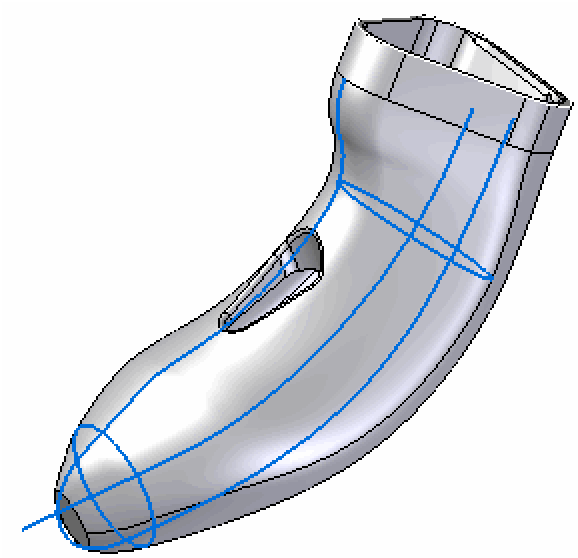
The solid modeling method is typically used when modeling with solid features. The following are key features of the solid modeling approach:

- It is characterized by 2D sketches/profiles used in creating extrusions, revolutions, and lofts to form solids, and blends on the edges of solids.



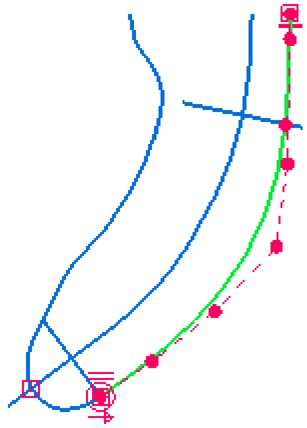
- It most often involves the addition or subtraction of material using analytic shapes.
- The model's topology is driven by faces.
- Holes are used for alignment.
- Feature faces are used for alignment as well as for mating with other features.
- Edges are rounded for safety and strength.
- Edges and faces are primarily analytic-based.

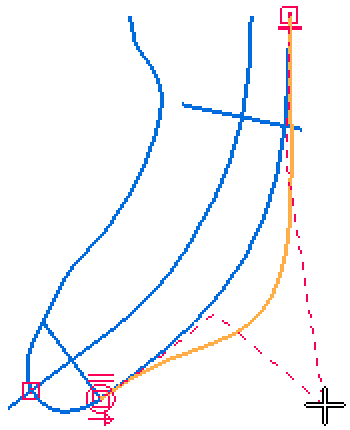
Modeling with surface-based features typically begins with a wire frame, from which surfaces are generated. Key features of surface modeling:

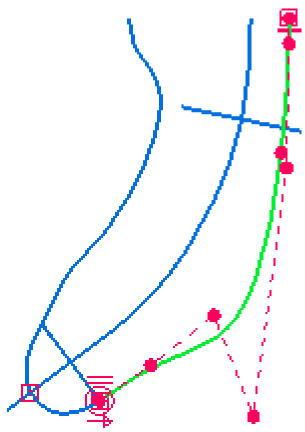


- It is characterized by control points used to define 2D and 3D curves.

- A model's topology is driven by edges and curves. Edges and faces are mainly based on splines.



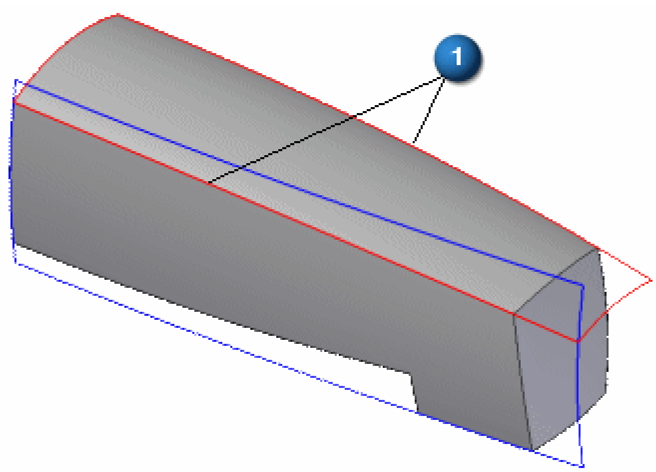




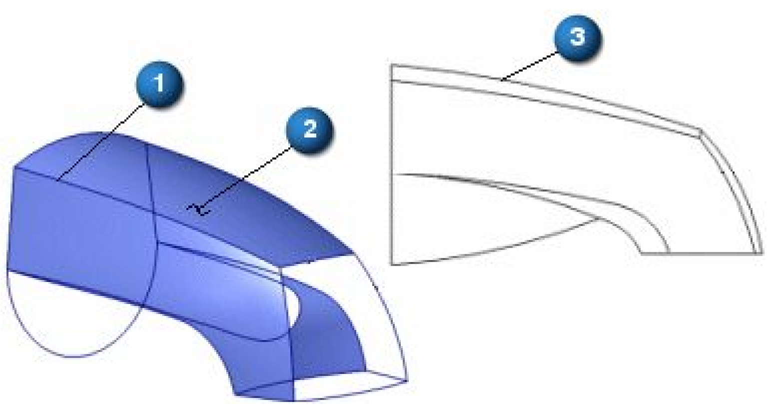
- Surface shapes are very important, therefore the ability to directly edit underlying curves and edges is crucial.
- Highlight lines, silhouette edges and flow lines of a model are important.

Surface construction advantages

For some types of parts, the surface modeling approach offers distinct advantages. For example, when modeling the faucet shown using revolved features, the shape of the edges (1) is the result of two intersecting surfaces. To change the shape of the edges, you must edit the surfaces. Often it is difficult to get the surface aesthetics you want.



With a surface modeling approach, you have much more control by using character curves. Character curves can be hard edges or soft edges. Hard edges are actual model edges (1), while soft edges are theoretical, view-dependent edges, such as when viewing a curved surface (2) from the side (3). Soft edges are also known as silhouette edges. Both types of edges are important for defining the flow, aesthetics, and overall shape of a surface.



Surface construction overview

Solid Edge provides two distinct 3D modeling styles: solid modeling and surface construction.

Many surface construction features require you to define cross section and guide curves. You can define cross section and guide curves using analytic elements or B-spline curves.

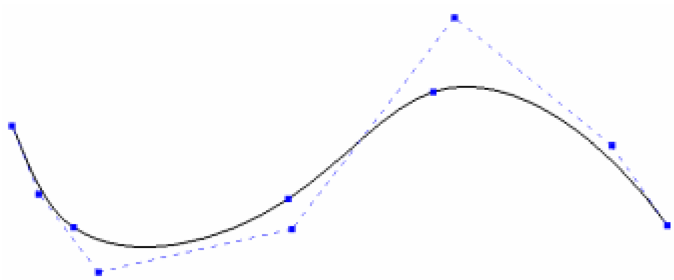
An analytic element can consist of:

- A 2D element: Line, arc, circle, ellipse, parabola, or hyperbola.
- A derived element: such as the intersection of a cone and a plane.
- A 3D element: a cube, sphere, cylinder, cone, or torus.

A b-spline element can consist of:

- A 2D element, such as a B-spline curve.
- A derived element, such as the intersection of two non-planar surfaces.
- A 3D element, such as a 3D B-spline curve or free-form surface.

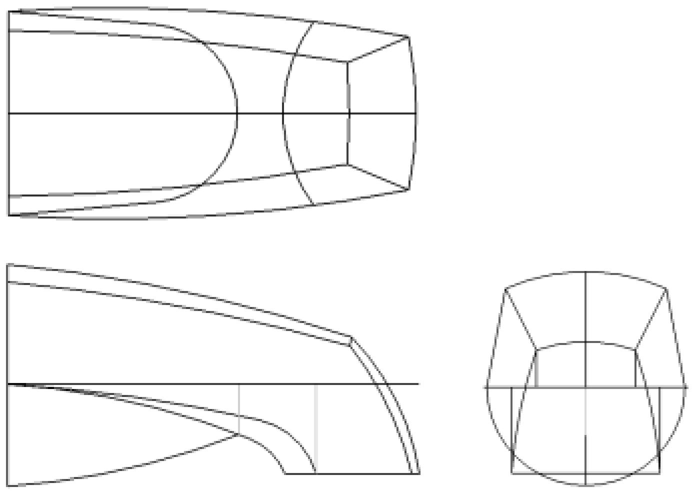
> **Note**
>
> A spline was originally a tool made from wood or thin metal which was used to draw a curve through points.

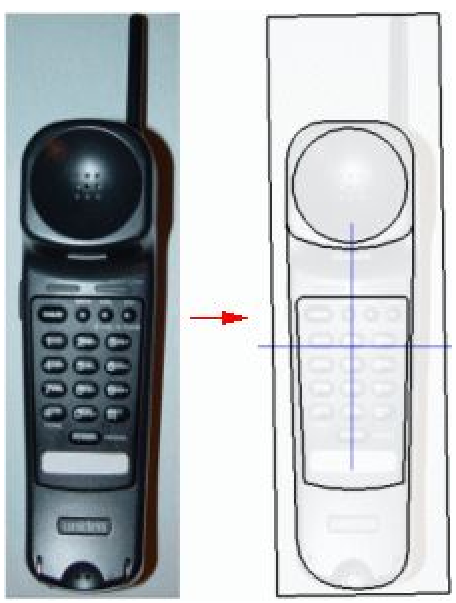


General surface modeling workflow

1. Create control drawings.

   Definition: Control Drawings are 2D drawing views defining the top, side, and end views. Typically one or two views dominate (define the majority of the shape).





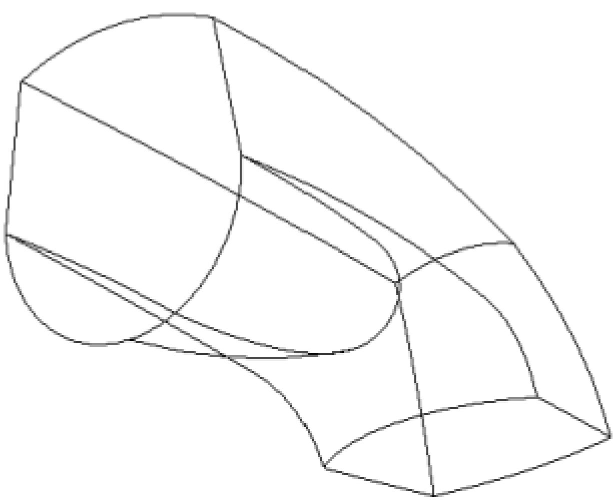
Part environment: You can create control drawings directly by drawing on reference planes. *Pierce points* aid in connecting curves.

Draft environment: You can create control drawings in 2D, then use Copy and Paste to transfer the 2D elements from Draft into Part. Also, you can use *Create 3D* or import sketches.

> **Tip**
>
> While drawing any sketch element in either the Part or Draft environment, use *Line Color* to help distinguish edges and construction edges in the control drawing.
>
> 

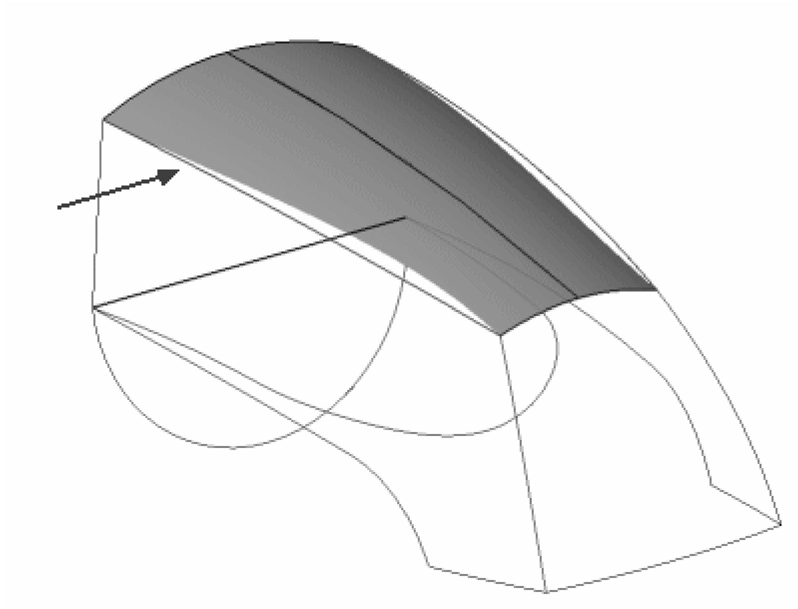
- Draw all character curves.
- Do not over draw. Do not model rounds, ribs, or features best created with solid features.
- Capture design intent. Add dimensions and constraints.
- Create simple B-splines with few Edit Points.
- Make sure view sketches register.
- Build edge continuity into your sketches.

2. Use 2D geometry to develop 3D curves.
   - Project curves from control drawings.
   - You may need some construction surfaces to generate 3D curves. This is very important in reducing modeling steps.
   - 3D curves give simplified control over edges.
   - Capture your design intent by using control drawings.

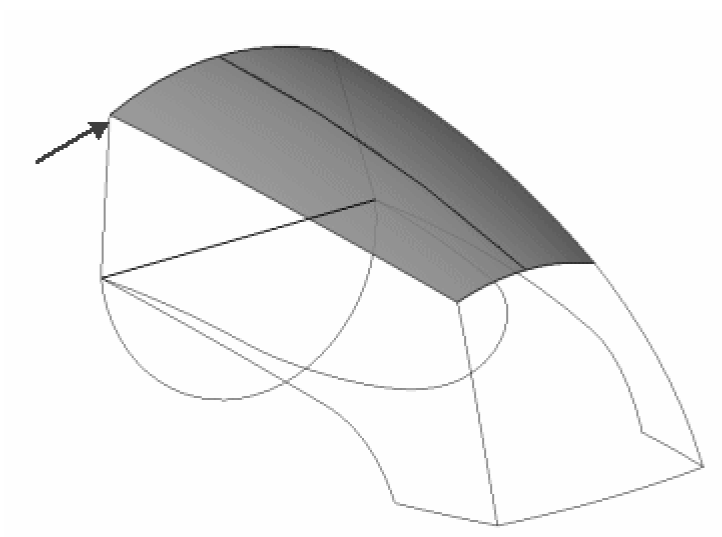
   

   - Without 3D curves, character edges may not be captured.

- Lack of 3D edges eliminates design intent and adds more modeling.

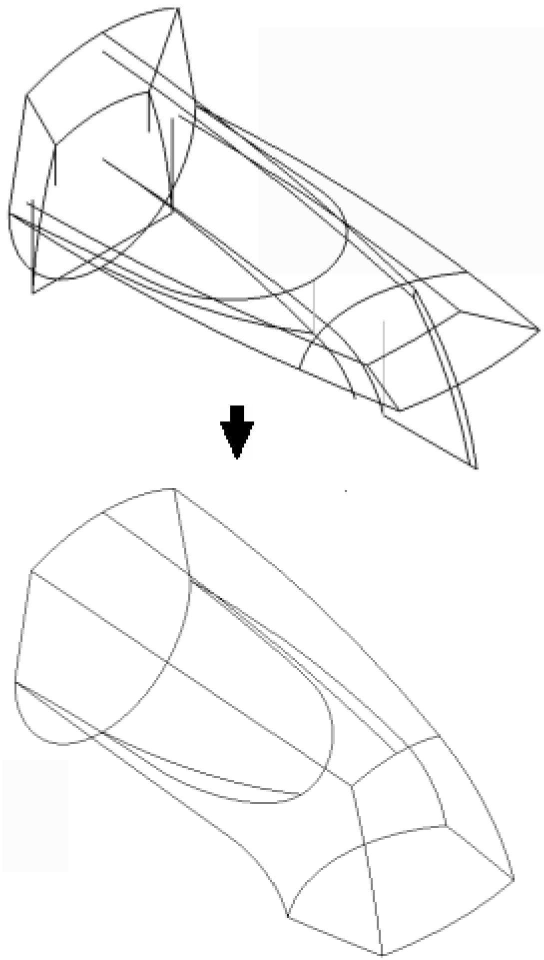


- With 3D curves, design intent is preserved and modeling is reduced.
- You can easily change the shape by altering the character curves for the respective view.
- Creating 3D edges guarantees an accurate design and reduces modeling steps.



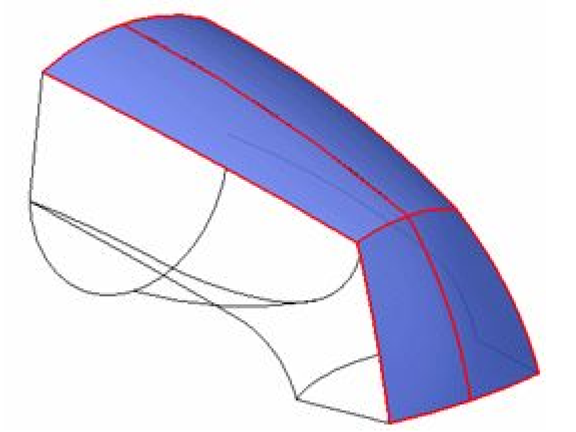
- Making changes to the 3D curve is simple. Edit the character curve in the control drawing.

- Repeat the process until all 3D curves are created
  - A wireframe representation of the model should result.
  - All 3D curves should be touching.



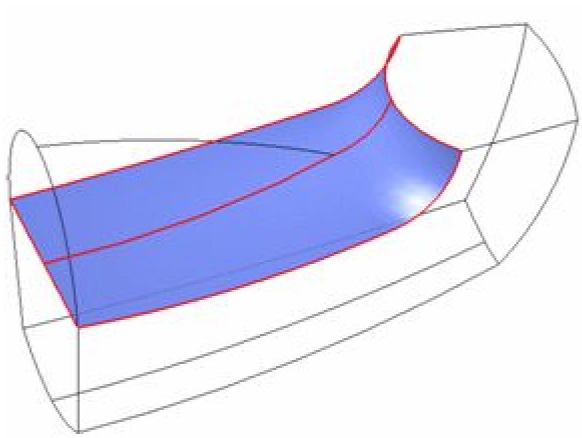
3. Use 3D curves to develop surfaces.
   - **BlueSurf** command.

     Inputs are guides and sections.



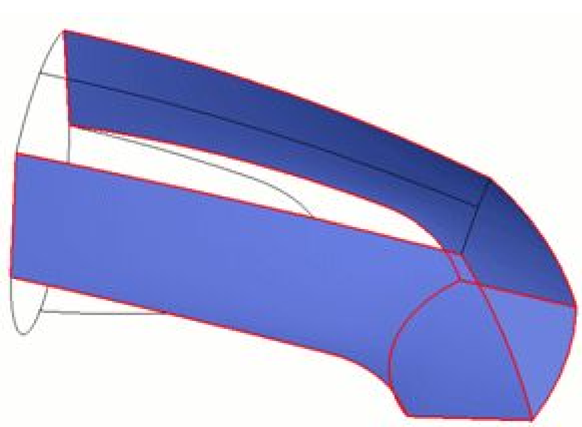
   - **Swept** command.

     Inputs are guides and sections.



   - **Bounded** command

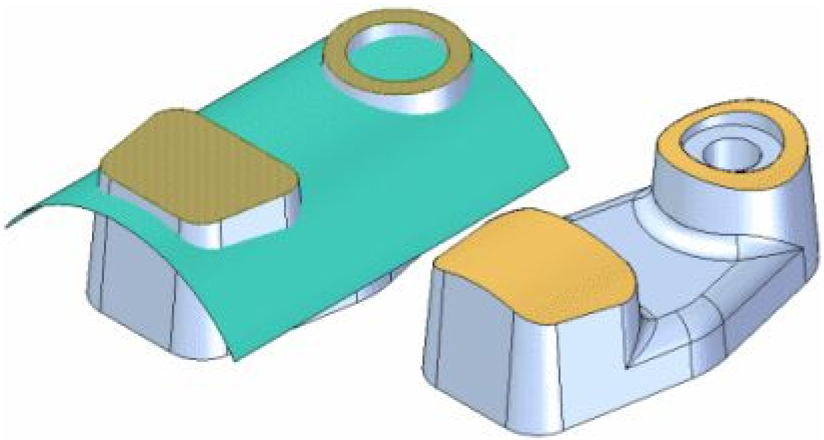
     N-sided patch.



4. Create a solid and add appropriate solid based features.
    - Stitch together surfaces
    - Add solid-based features
        - o Thinwall
        - o Stiffening Rib
        - o Hole
        - o Round
        - o Web network
        - o Lip
5. Tweak.
    - Analyze edge continuity using:
        - o Curvature Comb
        - o Zebra Stripes
    - Edit character curves
    - Edit tangent vectors
    - Edit vertex mapping

Working with points, curves, and surfaces

You can use commands in Solid Edge to create points, curves, and surfaces. These elements are typically used to construct part features, and are often referred to as construction elements. For example, you can use a single, curved surface to replace several planar faces on a model. Using points, curves, and surfaces helps you model complex design scenarios more quickly.



You can also use these commands when working with foreign data that you have imported into Solid Edge.

For some model types, you may not use the solid modeling commands until very late in the modeling process. Complex, free form parts often require that you begin the modeling process by defining points and curves that are used to define and control the surfaces that comprise the model. Surfaces are then generated, and in the final steps, the surfaces are stitched together to form a solid model. For more information on this type of workflow, see the Surface construction Help topic.

Note

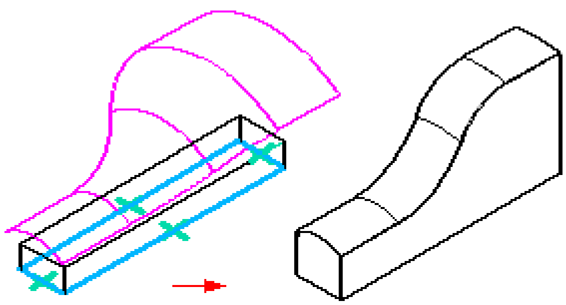
Construction elements that drive other features have a parent-child relationship with the features they drive. If you delete a construction element that is a parent to another feature, you can invalidate the other feature.

Displaying points, curves, and surfaces

The construction elements you create are listed in the **Feature PathFinder** window. You can control whether construction elements are listed in **Feature PathFinder** using the PathFinder Display commands on the Feature PathFinder shortcut menu. For example, to display construction elements in **Feature PathFinder**, click the right mouse button within **Feature PathFinder**, then point to PathFinder **Display**, and then set the **Constructions** option.

> **Note**
>
> You can change the default color for construction elements using either the **Color Manager** command or the **Colors** tab on the **Options** dialog box.

When you use construction elements to help you construct new features on a solid model, the construction elements are not consumed by the new feature. For example, if you use a construction surface to help you define the extent for a protrusion, a trimmed copy of the construction surface is used to create the protrusion. The construction surface remains, but it is hidden automatically.



You can control the display of construction elements in the graphic window using the **Construction Display** command or the **Show** and **Hide** commands on the shortcut menu. When you hide a construction element, its entry in Feature PathFinder changes to indicate that it is hidden.

When working with Solid Edge documents that contain construction surfaces and a solid design body, it can be useful to hide the design body while you are working with the construction surfaces. You can use the **Show Design Body** and **Hide Design body** commands to control the display of the design body.

Creating points, curves, and surfaces

You can create these elements using the following methods:

- Generate them using other geometry on the model. For example, you can create points and curves at the intersection of other curves and surfaces.

- Create them from scratch. For example, you can create extruded, revolved, and swept surfaces using the Solid Edge construction surface creation commands.

- Generate them using an external file. For example, you can create a helix curve using the **Helical Curve** command or coordinate data in a spreadsheet using curve by table.

- Import them from another CAD system. For example, you can import surfaces and solids from a third-party CAD system.

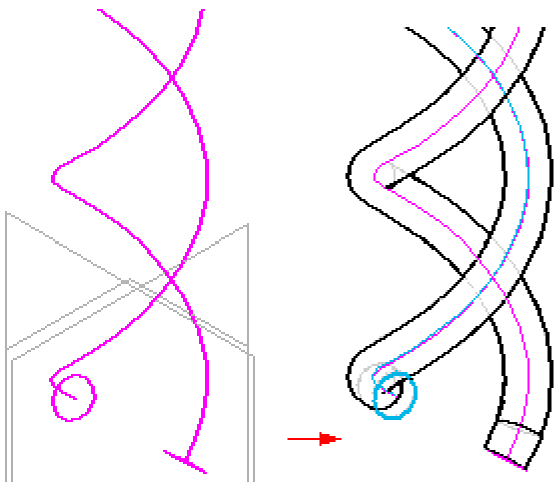
- Generate them as a part copy from another Solid Edge part. For example, you can create construction geometry using the **Part Copy** command.

Using points and curves

You can use points and curves in the following ways:

- To help you create other features, you can use a construction point or curve as a path or cross section for lofted and swept features.
- To help you create a reference plane, you can use a construction curve as input to the **Plane Normal To Curve** command.
- To help you define the extents of another feature, you can use keypoints of construction curves to define the extent for a feature.

For example, you can use 3D construction curves as paths during the creation of swept features.



You can use the **Intersection Point** command to create associative points at the intersection of edges and other curves. You can then use these points as input to define the extents of a feature. You can also use points as cross sections when creating lofted features.

You can create open or closed curves using the **Intersection Curve**, **Keypoint Curve**, **Derived Curve**, and **Curve By Table** commands. You can then use these curves to define paths and cross sections for lofted and swept features, and as profiles for profile-based features and construction surfaces.

The **Project Curve** command projects a curve onto a part face. You can then use the projected curve as a profile for either a protrusion or a cutout feature. This is a useful technique for creating embossed text on a curved surface.

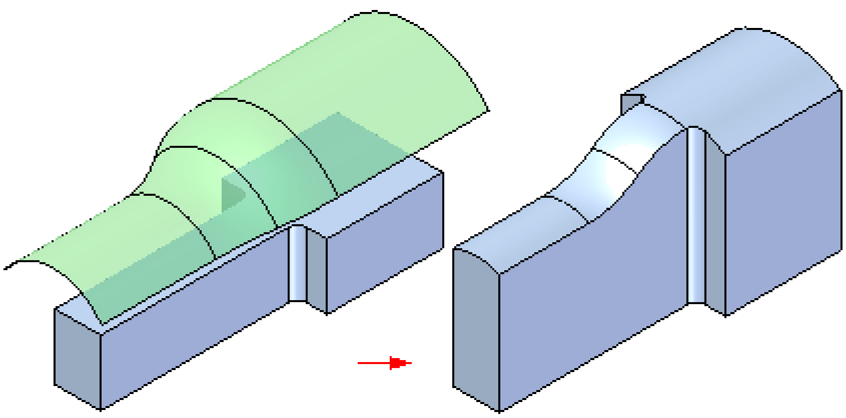
The **Split Curve** command splits a curve into multiple curves. Splitting a curve can make it easier to create other geometry, such as a surface by boundary or a normal protrusion.

Using surfaces

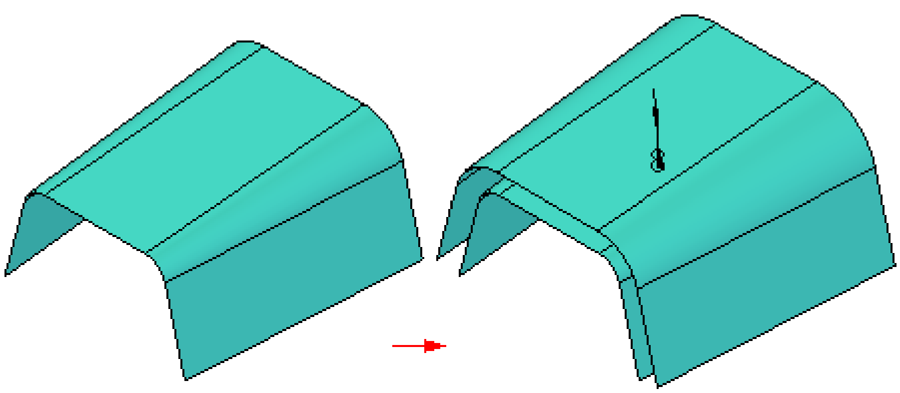
The surfacing commands help you create complex parts and surface topology more easily. You can use surfaces in the following ways:

- To define the projection extents when extruding a feature.
- To replace existing part faces.
- To divide a part into multiple parts.
- To create a new surface or solid by stitching together separate surfaces.
- To repair a model you imported from a third-party CAD system.

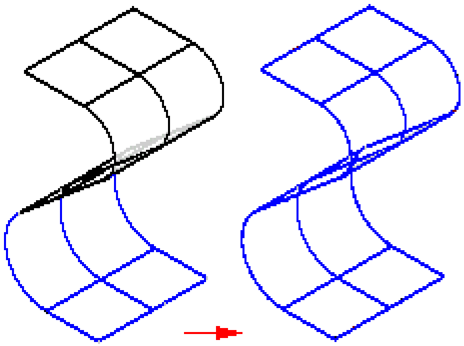
Construction surfaces are commonly used as projection extents when extruding a feature. For example, you can create a construction surface, then use the surface as input during the Extent step when constructing a protrusion.



You can use the **Offset Surfaces** command to offset a new surface. The options on the command bar allow you to specify whether you want to offset a single face, a chain of faces, or all the faces that make up a feature.



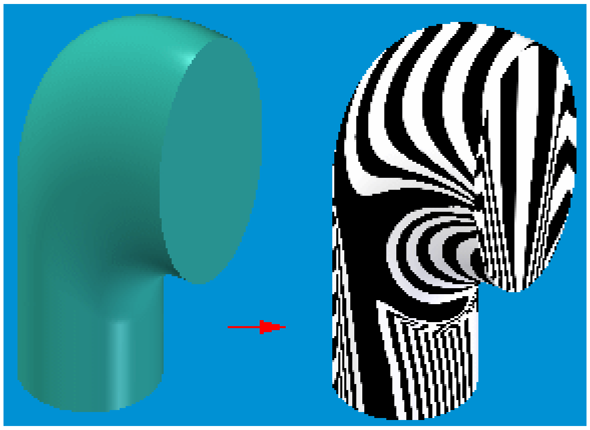
You can use the **Stitched Surface** command to stitch together Solid Edge surfaces, as well as surfaces created with another CAD system and then imported into Solid Edge.



You can also create surfaces using the **Part Copy** command. If the **Copy As Construction** option is set in the **Part Copy Parameters** dialog box, the part copy is created as a construction surface.

Evaluating surfaces

When working with surfaces, it is sometimes useful to visualize the curvature of a surface to determine if there are surface discontinuities and inflections. You can use the **Zebra Stripes** command to display zebra stripes on the model.



You must also shade the active window using the **Shaded or Shaded With Visible Edges** commands to display zebra stripes.

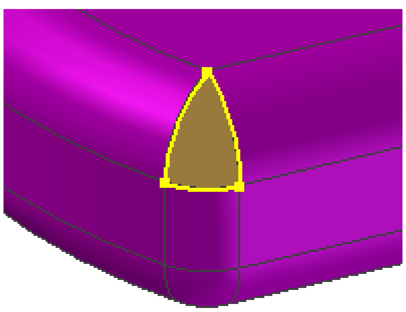
Evaluating and repairing foreign data

When you import surfaces that do not form a closed volume, they are imported as construction geometry. If the imported surfaces form a closed volume, you have the option to create a solid body.

If the imported surfaces do not form a solid body in Solid Edge, but were created as a solid body in the other CAD system, the accuracy of the data prevented it from being converted into a solid body in Solid Edge. Typically, the surface-to-surface matching tolerances used in the source system were larger than the Parasolid modeling kernel requires for successful stitching of the surfaces into a solid body. Some CAD systems allow surface-to-surface matching tolerances that are quite large, in some cases larger than the manufacturing tolerances of the manufactured part. The surface-to-surface matching tolerance requirements of the Parasolid modeling kernel are more exacting.

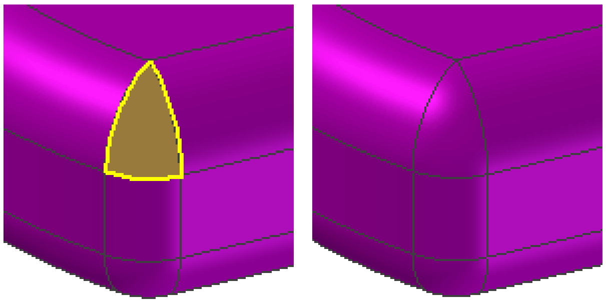
You can use the **Geometry Inspector** command to determine what problems the model has and then you can use the construction commands to modify the model to repair the problem areas. For example, there may be surfaces that did not import correctly, or there may be gaps or overlaps between individual surfaces in the model. Geometry Inspector evaluates the model and builds a list of the problem areas and provides suggestions as to how you can repair the problems.

If there are areas that did not stitch properly, you can use the **Show Non-Stitched Edges** command to display the non-stitched areas. You can then use the other commands on the **Surfacing** tab to repair the existing surfaces, or create new surfaces and stitch them into the model. You can also delete surfaces that would be easier to recreate from scratch than to repair.



Both curve and surface manipulation commands are available for creation and modification of construction elements. You can use the **Derived Curve**, **Split Curve**, **Project Curve**, and **Intersection Curve** commands to create new curves or modify existing curves. You can use the **Trim Surface**, **Extend Surface**, and **Delete Face** commands to modify or delete construction surfaces. You can use the **Extruded Surface**, **Revolved Surface**, **Swept Surface**, **Lofted Surface**, and **Bounded Surface** commands to create new construction surfaces. For example, if an imported surface overlaps another surface, you can use the **Derived Curve** command to extract a curve from the edge of the surface it overlaps, then use the new derived curve as input with the **Trim Surface** command to trim the existing surface.

If the non-stitched edges are the result of a missing surface, you can use the construction commands to create a new surface and stitch it into the model. For example, you can create an extruded, revolved, swept, and lofted construction surface to close a gap in a model.



When repairing imported data, you may need to try several approaches before finding one that succeeds. For example, if you are unsuccessful creating a revolved surface, try creating an lofted surface. The tolerance issues inherent with imported data can make model repair difficult.

After you have repaired a surface, or created a new surface, you can then use the **Stitched Surface** command to add the new surface to the model. If the stitched surfaces form a closed volume, you have the option to create a solid body. You can then use the solid body to complete the modeling process.

Lesson 2: Creating and editing curves

Objectives

After completing this lesson, you will be able to:

- Create curves.
- Edit curves.
- Analyze curves.
- Create BlueDots.
- Edit BlueDots.

Lesson 3: Surface modeling approach

The backbone of surface modeling is made up of cross sections and guides. Cross sections and guides can be of entity type analytic or spline.

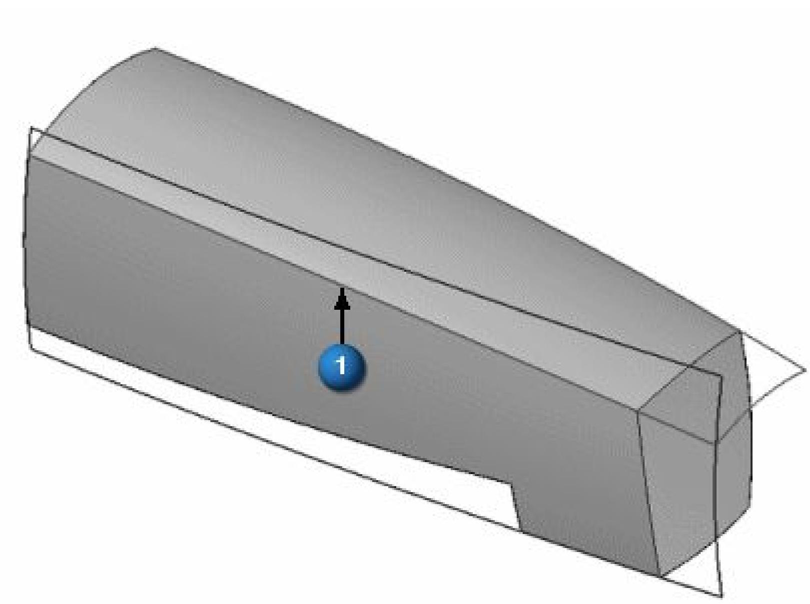
An *analytic* entity type consists of:

- 2D: Lines, arcs, circles, ellipse, parabola, hyperbola.
- The intersection of a plane and a cone.
- 3D: Cubes, spheres, cylinders, cones, tori.

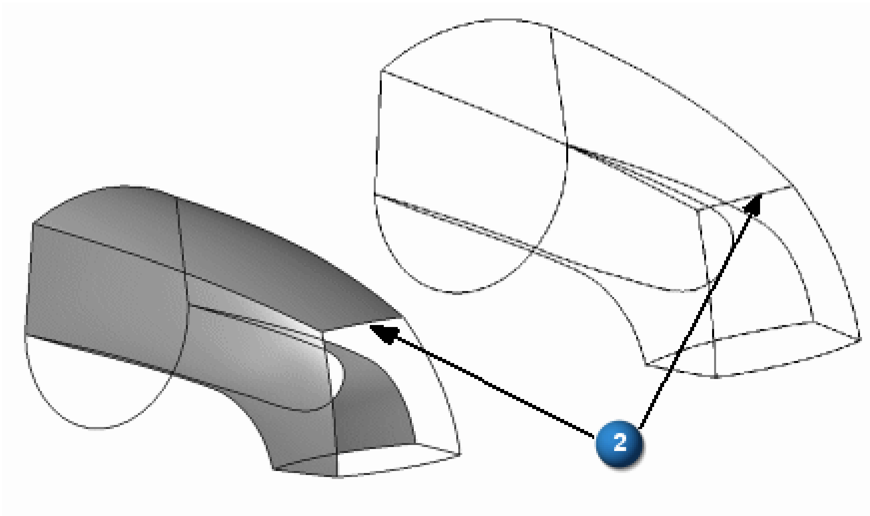
A *spline* entity type consists of:

- 2D: constructed spline curves, derived curves.
- 3D: derived spline curves.

A *solid* modeling method using revolved features results in no edge control and difficult edits. Edge (1) is a result of the intersection of two revolved surfaces. You do not have direct control over the result.



A *surface* modeling method results in exact edge control and edges are based on character curves. You have direct control over edges such as (2).



Lesson 4: Overview of splines

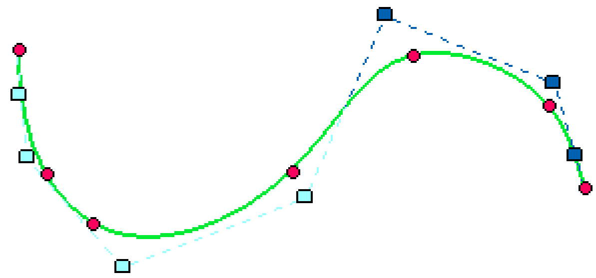
A *spline* is a standard curve in most CAD systems. Unlike lines and conic curves, generally categorized as *analytics*, the spline can be adjusted to virtually any shape in two or three dimensions. Their flexible nature makes splines the foundation for surface modeling.

A spline entity type consists of:

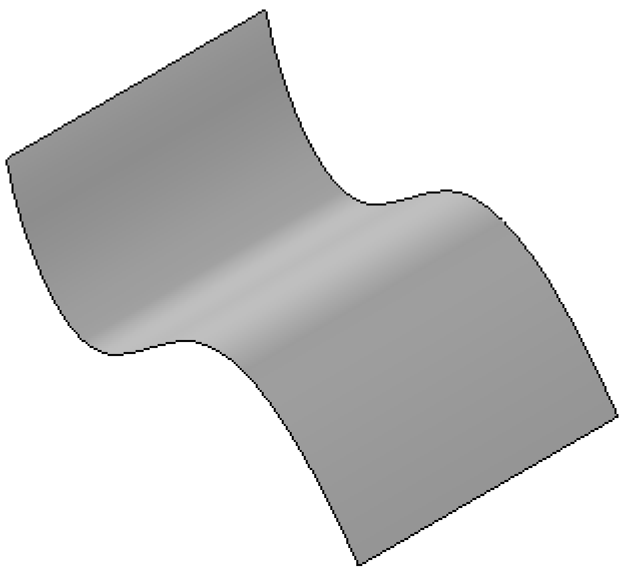
- 2D: constructed spline curves, derived curves.
- 3D: derived spline curves.

> **Note**
>
> Originally, a spline was a stylist's tool made from wood or thin metal and used to draw a curve through points.

Shown below is a 2D spline.



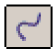
Shown below is a 3D surface based on a spline.

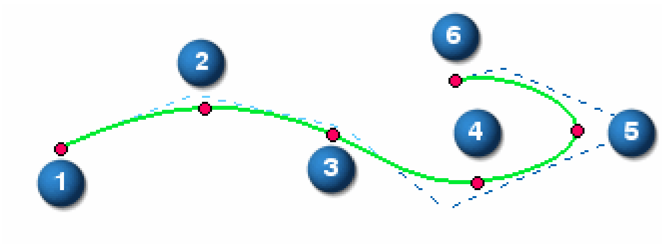


For the remainder of this course, the term *curve* is used instead of splines. Just remember that curves are splines. Two types of curves are discussed:

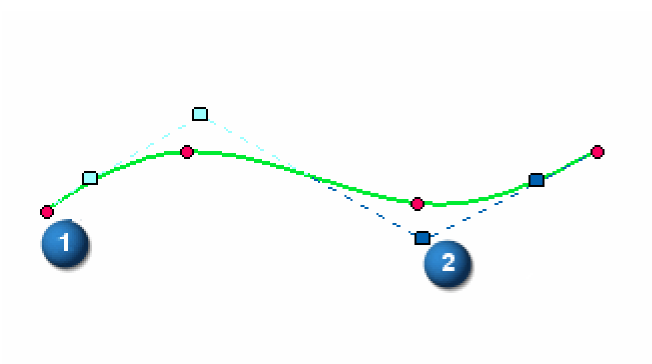
- *Constructed*–You have direct control of constructed curves.
- *Derived*–Derived curves are controlled by the method used to create them. Derived curves cannot be edited directly.

Lesson 5: Curve command

Use the **Curve** command  to draw a smooth, B-spline curve by points. You can click and drag to define a freehand curve, or you can click to create edit points to define the curve. If you click edit points, you must define at least three points to create the curve.

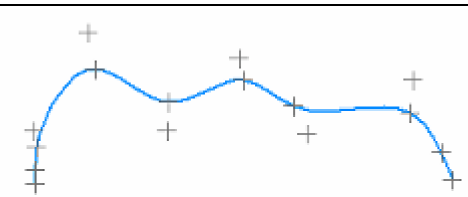


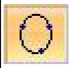
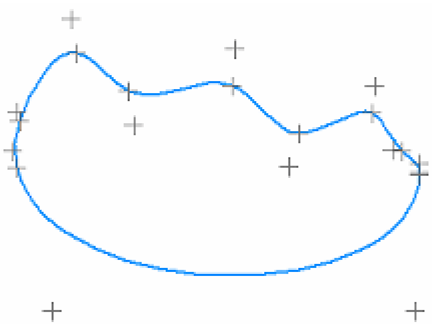
When you create a curve, edit points (1) and curve control vertex points (2) are created to help you edit and control the shape of the curve.



Closing curves

You can use the **Closed** option on the Curve command bar to create a continuous line that forms a closed curve connected tangentially at the first and last point you click.

Closed option	Result
Off 	

Closed option	Result
On 	

When editing a curve created from edit points, you also can use the **Closed** option to:

- Close an open curve, without adding points.
- Open a closed curve, without deleting points.

You cannot use this option to modify a freehand curve.

Displaying curves

You can use the options on the Curve command bar to control the display of a curve.

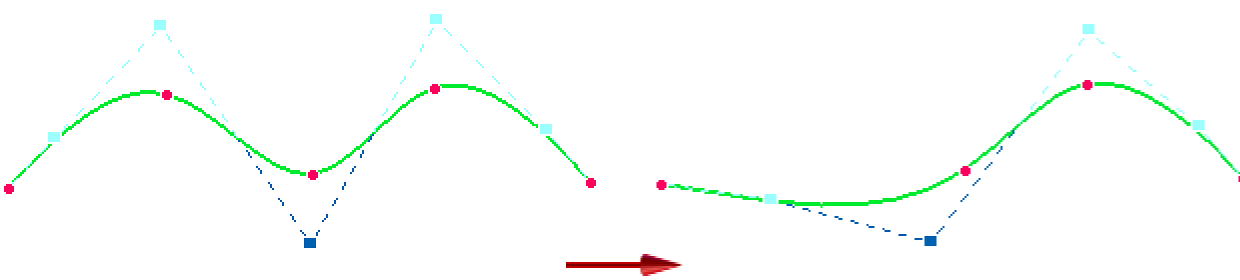
The **Add/Remove Points** button adds or removes edit points along the curve. When you add an edit point, the shape of the curve does not change. If the number of edit points on the curve is the same as the number of control vertex points, adding an edit point adds a corresponding control vertex point. The control vertex point moves to maintain the shape of the curve.

When you remove edit points, the control vertex points move, and the shape of the curve changes.

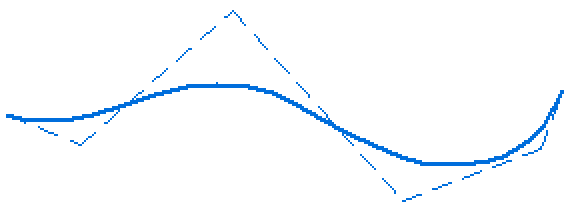
Note

If there are only two edit points on the curve, you cannot remove an edit point from the curve.

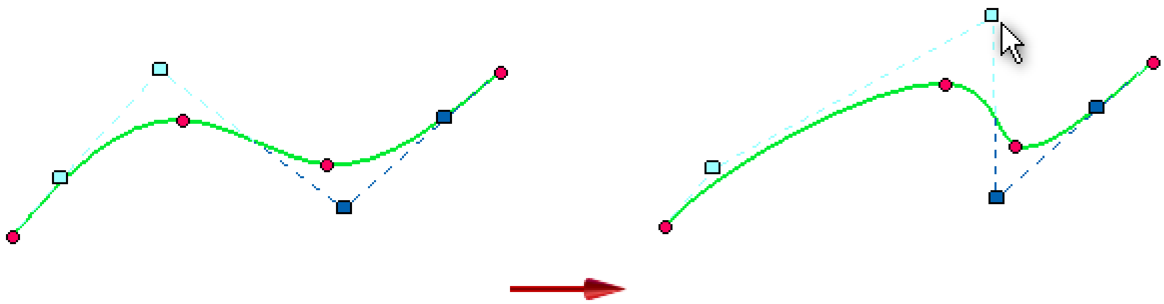
See Insert or remove points on a curve.



The **Show Polygon** button turns on the control polygon of the curve. This polygon displays when the curve is not selected.



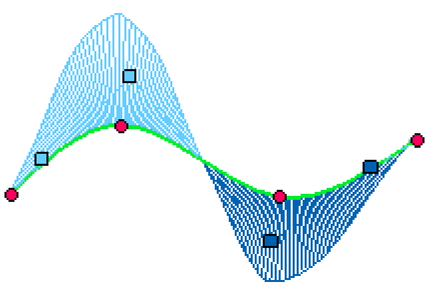
The edit points and control vertex points are handles that you can drag to change the shape of the curve.



Note

You can also use these points as keypoints for relationships and dimensions.

The **Show Curvature Comb** button displays the curvature comb for the curve. This helps you determine how quickly or gradually curves change and where they change direction.



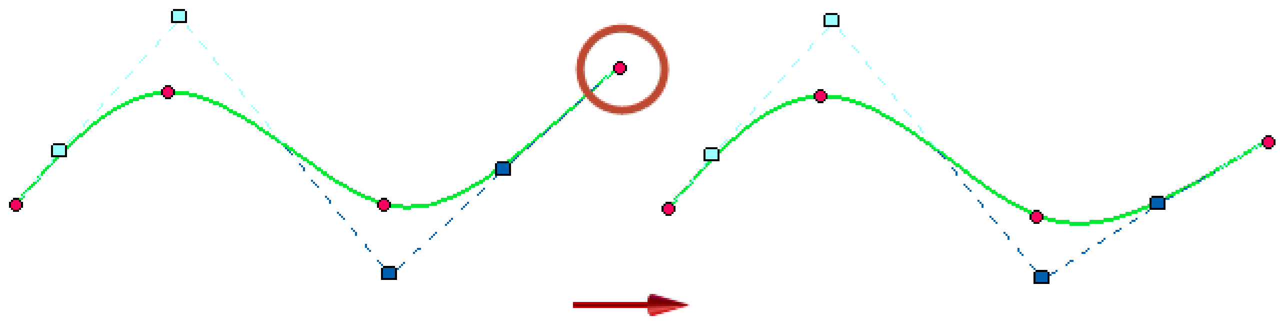
You can use the **Curvature Comb Settings** command to control the density and magnitude of the curve.

Editing curves

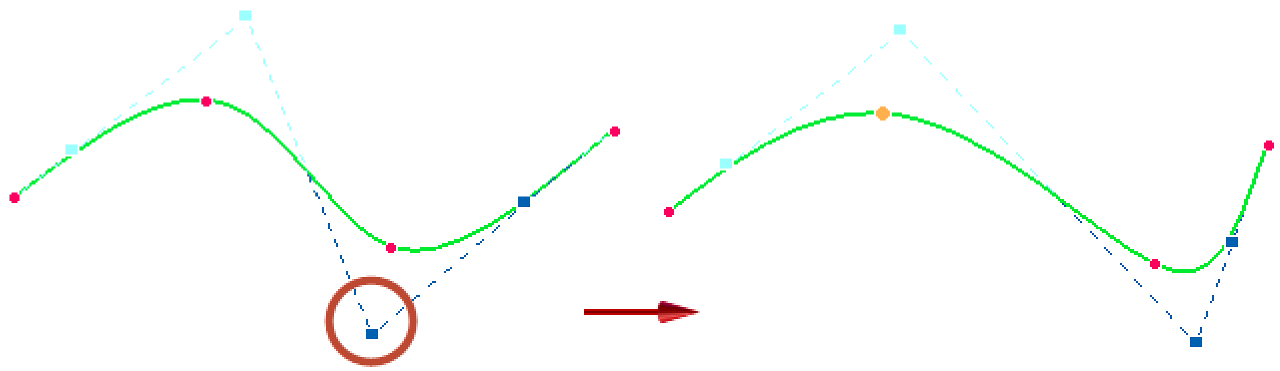
The Curve command bar controls how the shape of the curve changes when you make changes to the edit points and control vertex points.

The **Shape Edit** and **Local Edit** buttons control the shape of the curve when you move a point on the curve.

When you select the **Shape Edit** button, you affect the shape of the entire curve when you move a point on the curve.



When you select the **Local Edit** button, you affect the shape of the curve around the edit point.



With Local Edit, if you drag a vertex point on an unconstrained curve, no other vertex points will move. However, if you drag a vertex point on a curve that has some relationships, then other vertex points may move as well. This allows the curve to adapt to the new location of the vertex point you moved while still maintaining the relationships.

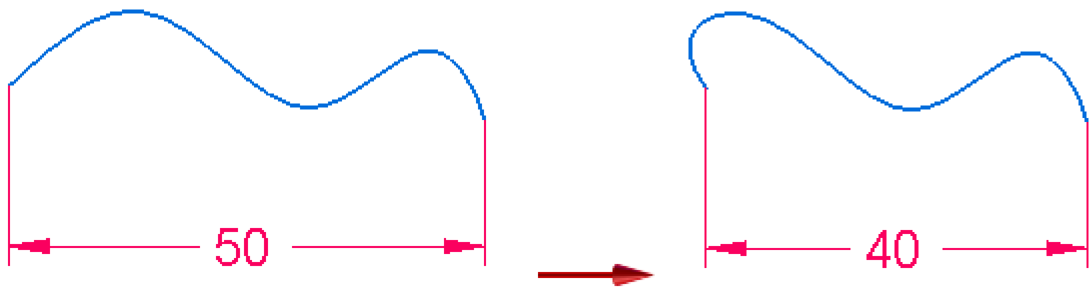
Note

You cannot drag an edit point that is fully constrained.

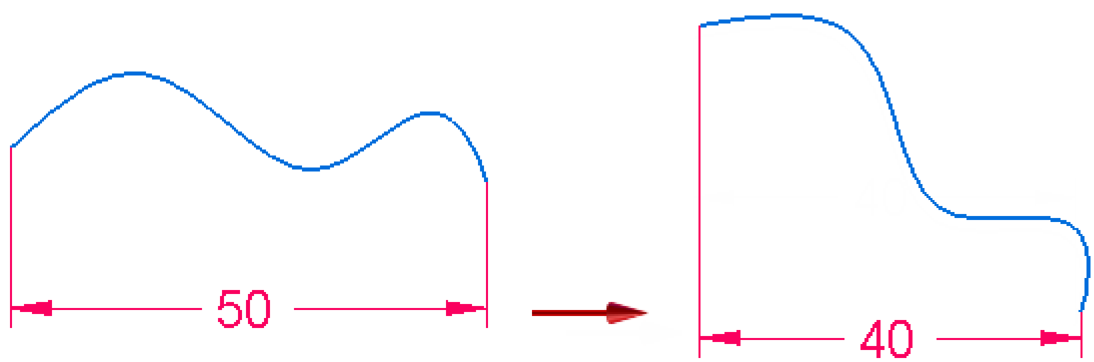
You can select the **Curve Options** button to display the Curve Options dialog box. You can use this dialog box to change the number of degrees for the curve and to specify the relationship mode for the curve. You can set the relationship mode to:

- Flexible
- Rigid

In **Flexible** mode you can use external relationships to control the shape of the curve. For example, you can apply a dimension relationship on the curve and as you make changes to the dimensions, the shape of the curve automatically updates.

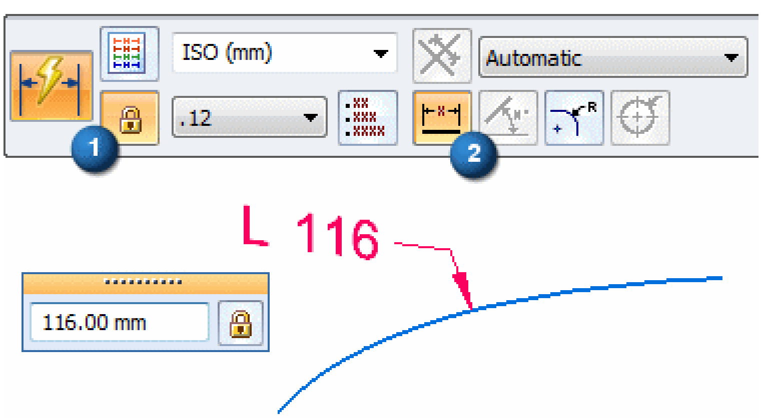


In **Rigid** mode you cannot use external relationships to control the shape of the curve. Instead, the curve shape remains unchanged and the curve simply rotates.



Dimensioning curves

When placing a smart dimension on a curve, the dimension result is measured as curve length (2) and is locked (1).



Simplifying curves

You can use the Simplify Curve command to simplify a polygon-based curve by reducing the number of edit points and control vertex points on a curve. The Simplify Curve dialog box increases or decreases a fit tolerance for the curve.

> **Note**
>
> Simplifying a curve can cause the relationships placed on a curve to be deleted.

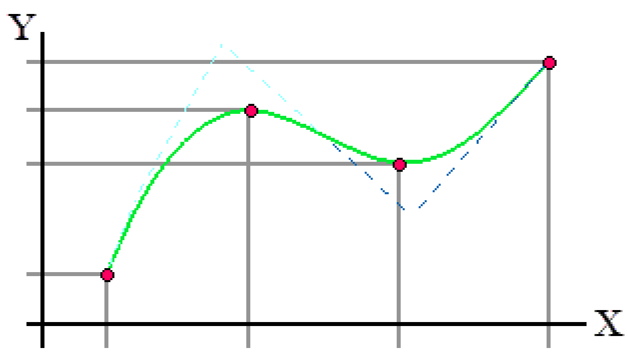
Lesson 6: Curve definition

The shape of a curve is dependent on the number of control and edit points. These elements are determined by standard polynomial expressions.

Curve order

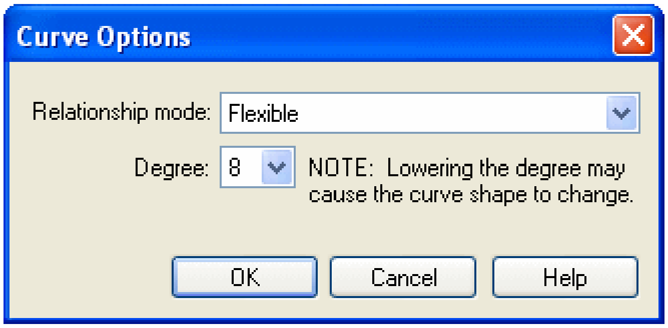
The order of a curve is equal to the degree of the curve, plus 1 (Order = Degree +1).

A polynomial curve is defined as:
x(t) = x0 +x1(t1) + x2(t2) + x3(t3)
y(t) = y0 +y1(t1) + y2(t2) + y3(t3)



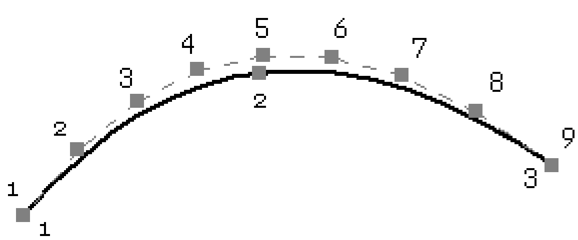
Determining control vertices

If the number of edit points is two or three, then the number of Control Vertices = Order.

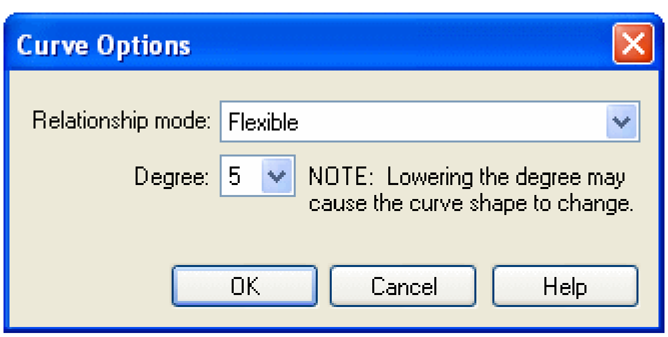


Example:

Edit points = 3
Degree = 8
Order = 9 (Degree + 1)
Control Vertices = 9

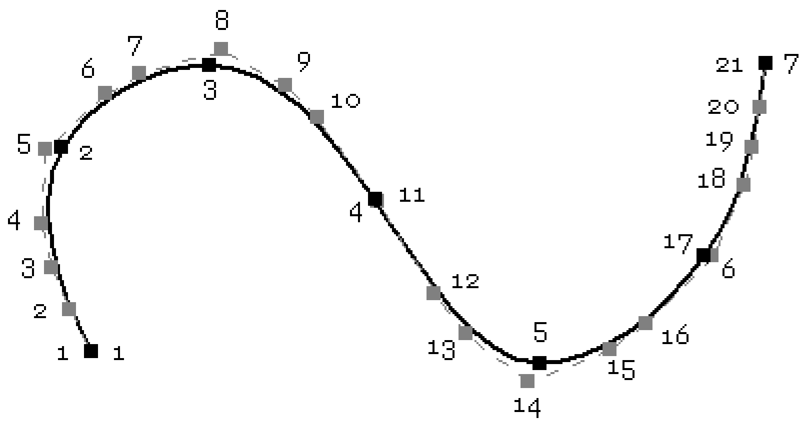


If the number of edit points is >= 4, the number of control vertices is (n+2) + {(n-1) x (k-4)}. Where n = Edit Points, and k = Order.



Example:

Edit points = 7
Degree = 5
Order = 6 (degree + 1)
Control Vertices = 21



For more information on curve options, refer to Curve Options dialog box.

Lesson 7: Curve display and edit

Displaying curves

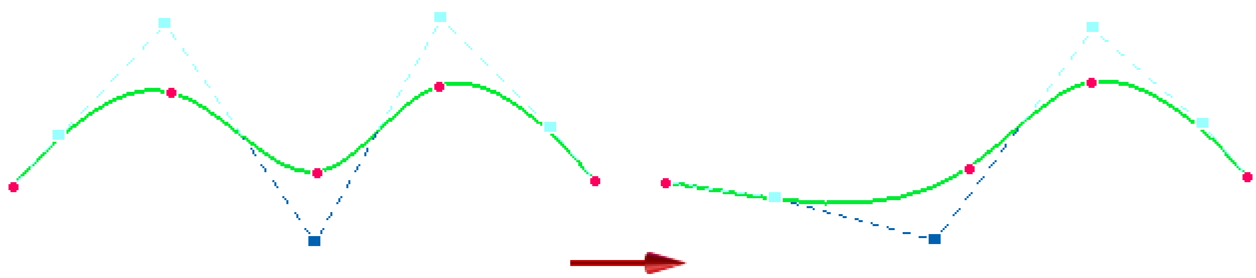
You can use the options on the Curve command bar to control the display of a curve.

The **Add/Remove Points** button adds or removes edit points along the curve. When you add an edit point, the shape of the curve does not change. If the number of edit points on the curve is the same as the number of control vertex points, adding an edit point adds a corresponding control vertex point. The control vertex point moves to maintain the shape of the curve.

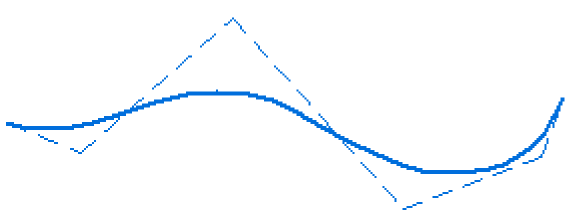
When you remove edit points, the control vertex points move, and the shape of the curve changes.

> **Note**
>
> If there are only two edit points on the curve, you cannot remove an edit point from the curve.

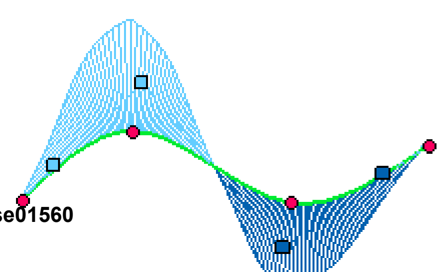
See Insert or remove points on a curve.



The **Show Polygon** button displays the control polygon of the curve, which you can use to edit the curve.



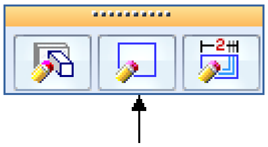
The **Show Curvature Comb** button displays the curvature comb for the curve. This helps you determine how quickly or gradually curves change and where they change direction.



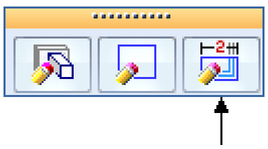
Editing curves

You can edit curves at any time via one of two methods for editing curves.

1. Edit Profile mode: Just like editing a sketch.

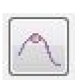


2. Dynamic Edit mode: Shows all of the control and edit points.



When you move a control point or edit point, the curve updates automatically; any surface that has the curve as one of its defining entities will update dynamically.

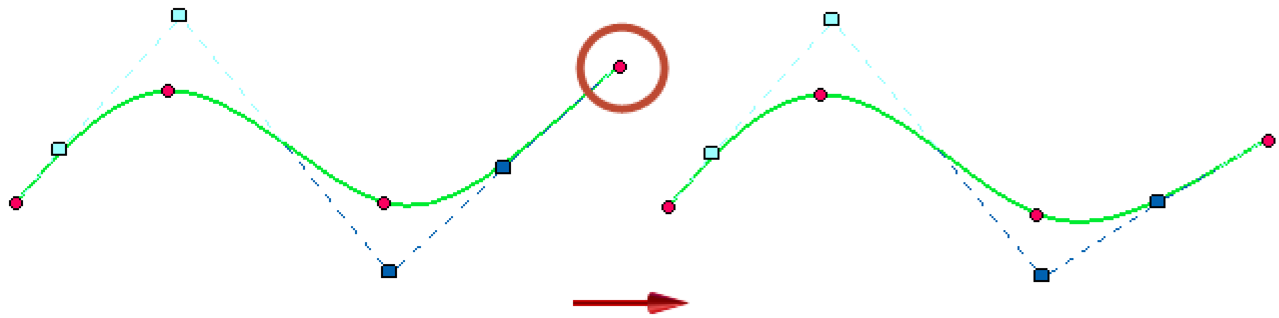
The Curve command bar controls how the shape of the curve changes when you make changes to the edit points and control vertex points. These options are only available in Edit Profile Mode

	Add/Remove Points	Adds and removes points on a curve.
	Show Polygon	Display of the control polygon on a curve
	Show Curvature Comb	Displays the curvature combs on a curve.
	Show Edit Points	Displays the edit points on a curve.
	Show Control Vertex Points	Displays the control vertex points on a curve.
	Shape Edit	Affects the shape of the entire curve when points are moved
	Local Edit	Affects the curve shape around the edited point.
	Close Curve	Specifies a open curve to be closed.
	Simplify Curve	Reduces the edit points on a curve.
	Curve Options	Displays the curve options dialog.

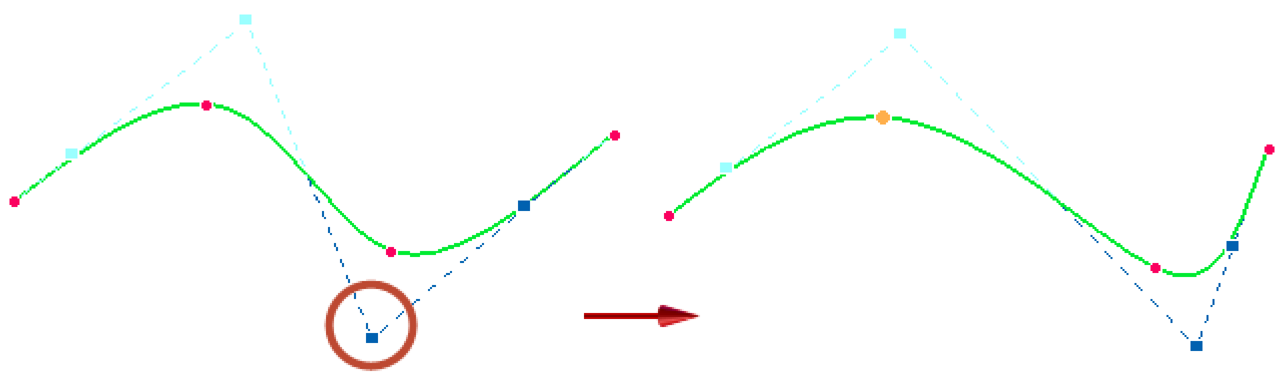
Below are some examples of curve editing

The **Shape Edit** and **Local Edit** buttons control the shape of the curve when you move a point on the curve.

When you select the **Shape Edit** button, you affect the shape of the entire curve when you move a point on the curve.



When you select the **Local Edit** button, you affect the shape of the curve around the edit point.



With **Local Edit**, if you drag a vertex point on an unconstrained curve, no other vertex points will move. However, if you drag a vertex point on a curve that has some relationships, then other vertex points may move as well. This allows the curve to adapt to the new location of the vertex point you moved while still maintaining the relationships.

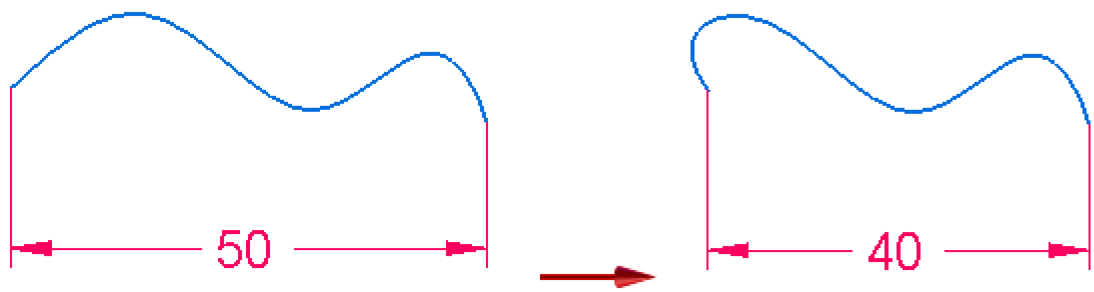
Note

You cannot drag an edit point that is fully constrained.

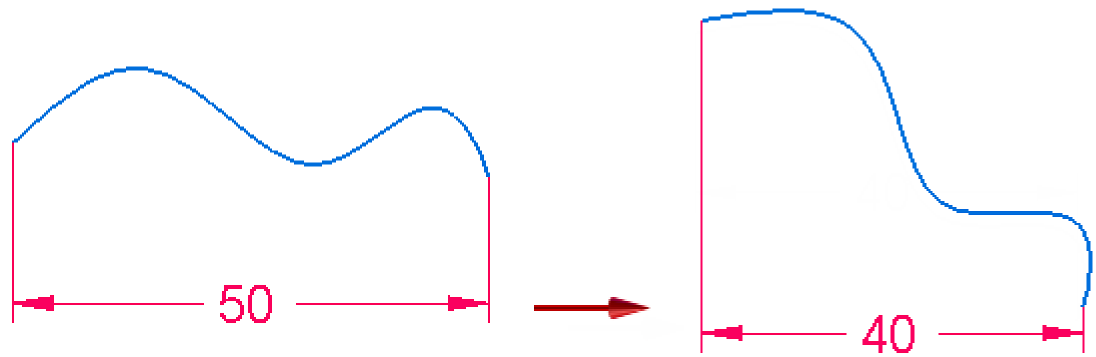
You can select the **Curve Options** button to display the Curve Options dialog box. You can use this dialog box to change the number of degrees for the curve and to specify the relationship mode for the curve. You can set the relationship mode to:

- Flexible
- Rigid

In **Flexible** mode you can use external relationships to control the shape of the curve. For example, you can apply a dimension relationship on the curve and as you make changes to the dimensions, the shape of the curve automatically updates.



In **Rigid** mode you cannot use external relationships to control the shape of the curve. Instead, the curve shape remains unchanged and the curve simply rotates.



Lesson 8: Simplify Curve command

Reduces the number of edit points for a curve.

- Curve data can be manually created or can be read in from foreign data.
- Manually created curve data usually contains a limited number of control points.
- Foreign data may come from a digitized set of control points, which could contain a large amount of points.
- Simplify curve is a tool that allows you to define a tolerance to reduce the number of edit points and control vertices.

The command works differently depending on the type of curve. For edit point based curves, once you reduce the number of edit points to two, the command reduces the control poles. Control polygon based curves only have two edit points, so the command reduces the control poles.

Select the **Simplify Curve** command from the command bar or right-click on a curve and select **Simplify** to access its dialog box; for more information, refer to the **Simplify Curve** dialog box topic:

Simplify Curve dialog box

Lesson 9: Convert To Curve command

The **Convert To Curve** command converts analytic geometry into a B-spline curve. B-spline curves are typically easier to use during surface modeling creation than analytic elements. For example, suppose you create a model with surfaces defined with analytic elements. You can use this command to convert the analytic elements to B-spline curve to obtain the control provided by B-spline curves.

Why convert?

- Analytical elements are often utilized as cross-sections and guide paths during surface creation. The resulting surface has inherent limitations on how it can be edited, as lines remain linear and arcs retain their circular definition.

- Curves provide more control, therefore are easier to use.

- Increased control facilitates edits.
  - Allows modification of a curve's properties.
  - Defaults to a degree of 2. You can increase the degree and add edit points for more control.

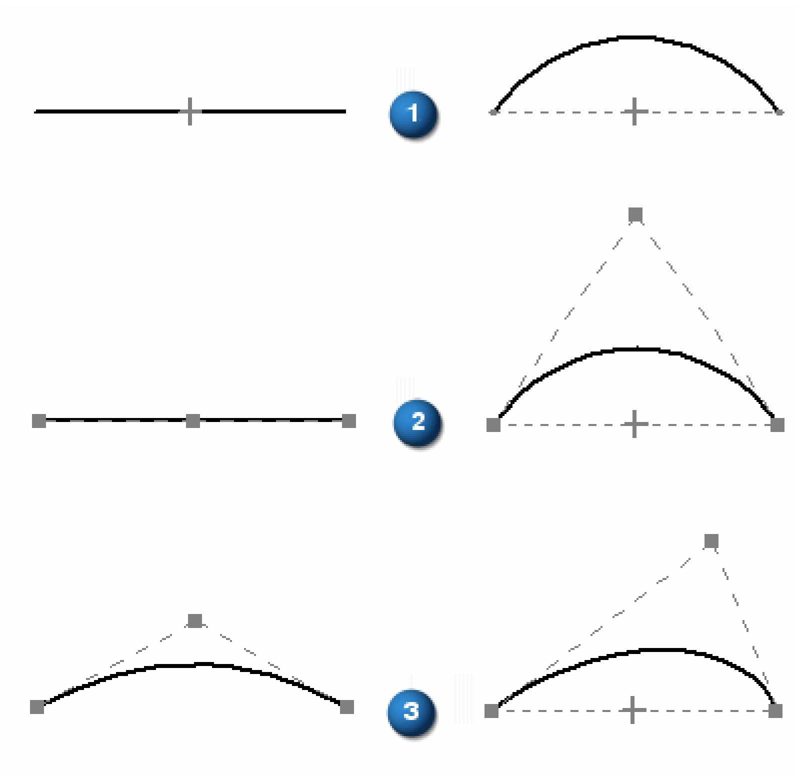
- Once converted, curve shapes will have greater control over associated complex surfaces.
  - Simplifies the manipulation of a model from initial concept through final production.

- Can be used on the following analytics:
  - Single non-connected analytic element: conversion results in a single non-connected B-spline curve.
  - Multiple connected analytic elements.
    - Non-tangent elements: conversion results in multiple connected B-spline curves with no cusps.
    - Tangent elements: conversion results in multiple connected and tangent B-spline curves.

> **Note**
>
> You cannot convert B-spline curves to analytic geometry.

Note

You can only convert analytics to curves while in the Edit Profile mode.

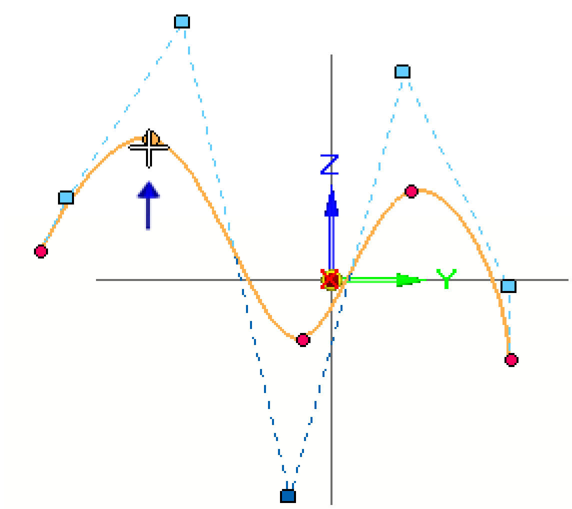


(1) Analytic line and arc element

(2) Analytic elements converted to curves

(3) Curve edits

Lesson 10: Activity: Drawing and editing a curve



Overview

In this activity, you learn to use the curve creation tools. Curves are the backbone for creating and controlling surface shape.

Objectives

After completing this activity you will be able to:

- Create curves.
- Edit curves.
- Analyze curves.

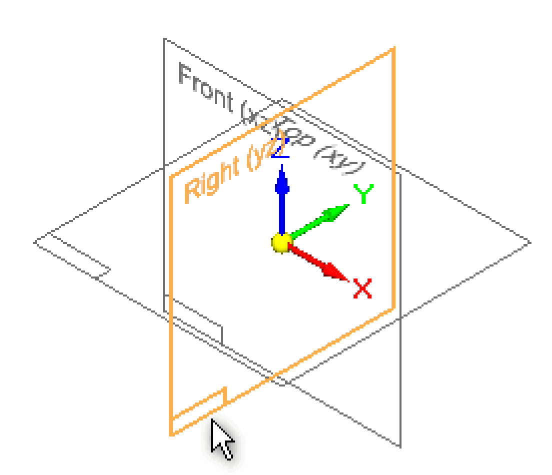
Lesson 11: Open part file

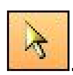
- Open **surface lab 2–01.par**.

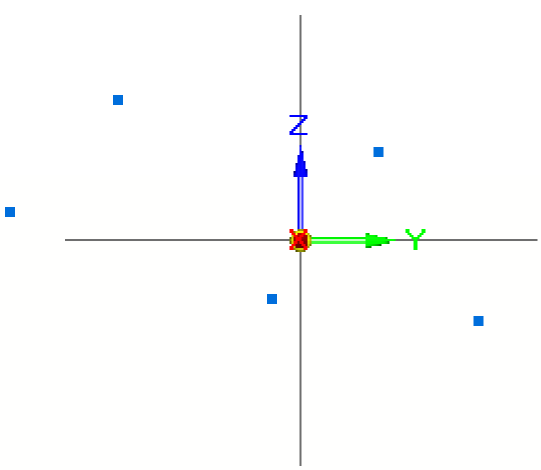
Lesson 12: Draw a curve

Begin the activity by drawing a curve with edit points in space.

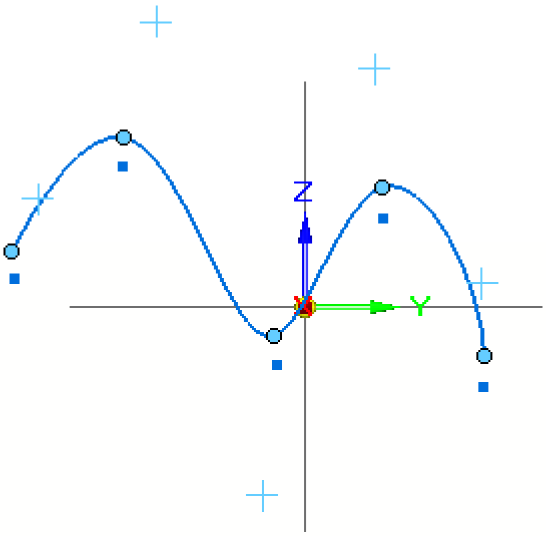
- Select **Home** tab→**Sketch** group→**Sketch** .
- Select the reference plane shown.



- Click **Home** tab→**Select** group→**Select** .
- In PathFinder, click the check box to display Sketch A. Use the sketch elements in Sketch A as a guide to where to place the edit points.

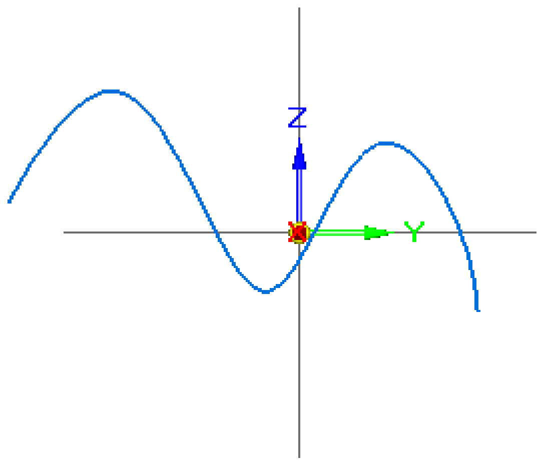


- Select **Home** tab→**Draw** group→**Curve** .
- Click just above each of the construction points from left to right as shown. After clicking above the last point, right-click the end the command. Select **Close Sketch**, then **Finish** on the command bar to create the curve.



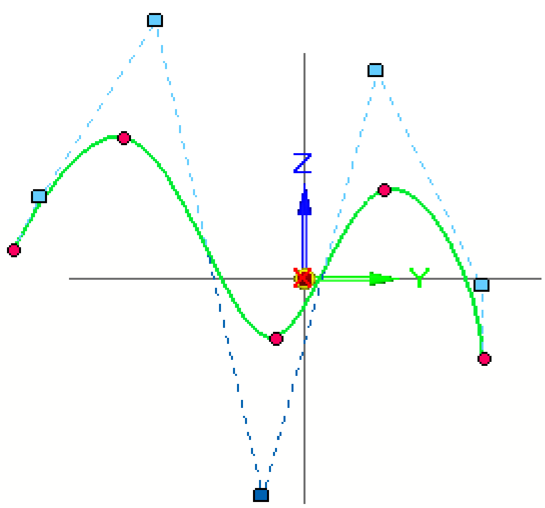
Lesson 13: Hide the sketch containing edit points

- Click **Home** tab→**Select** group→**Select**.
- In PathFinder, turn off the display of Sketch A.

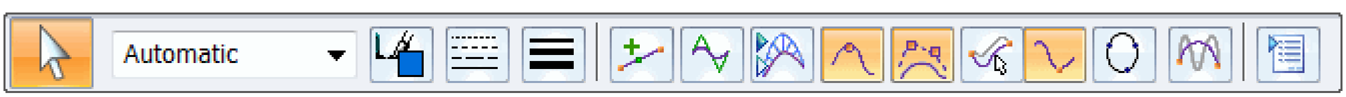


Lesson 14: Edit the curve shape

- Select the curve, and on the command bar, choose **Edit Profile**. Select the curve again, and notice the display of the edit points and control polygon.

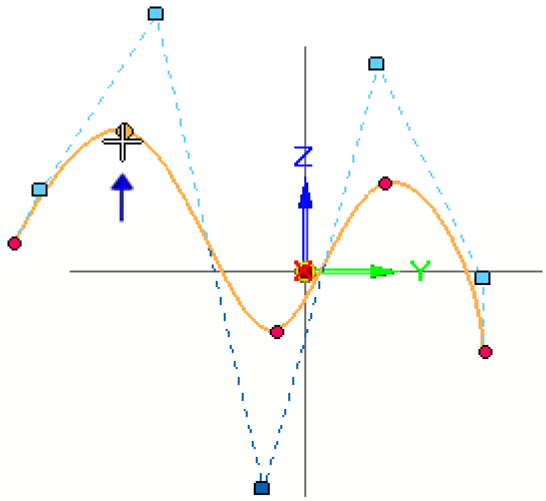


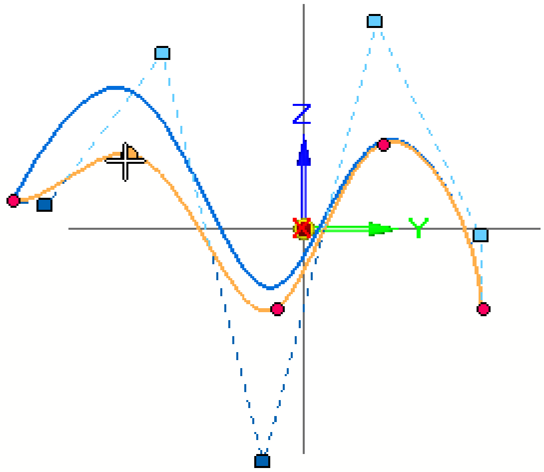
The **Edit Curve** command bar displays. On the command bar, the **Local Edit** option is on.

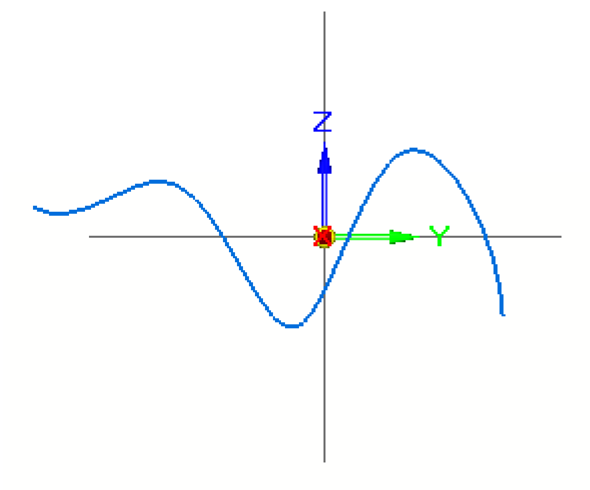


> **Note**
>
> With the **Local Edit** option, when you drag an edit point or control point, the shape of the curve changes near the point you drag. With **Shape Edit**, the entire curve changes shape slightly, preserving the overall shape of the curve.

- With the **Local Edit** option selected, drag the edit point shown to observe how the curve shape changes.


Z
Y

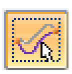

Z
Y



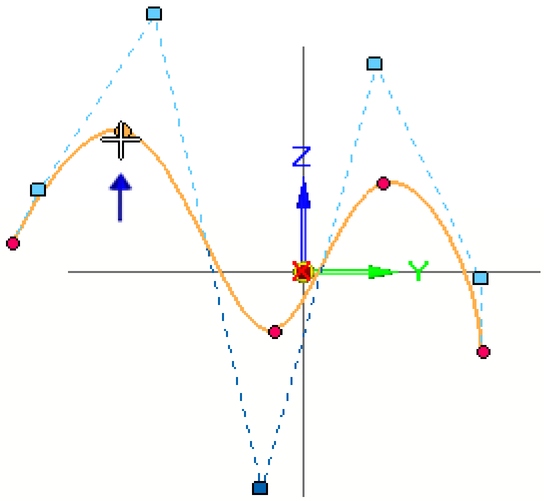
- After editing the curve, on the **Quick Access** toolbar, click the **Undo** command.

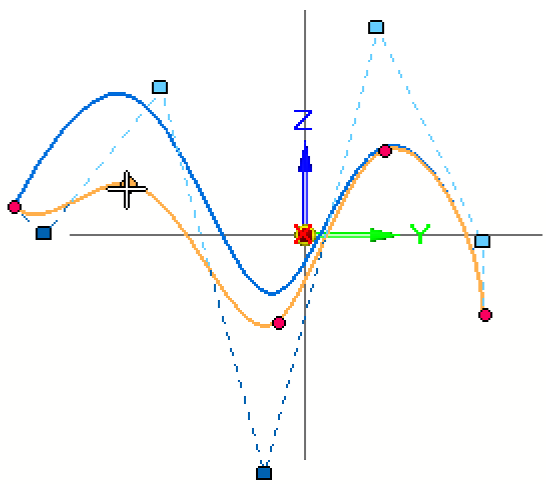
  This returns the curve to its original shape.

- Select the curve. On the **Edit Curve** command bar, select the **Shape Edit** option.

- With the **Shape Edit** option selected, drag the edit point shown and observe how the curve shape changes.

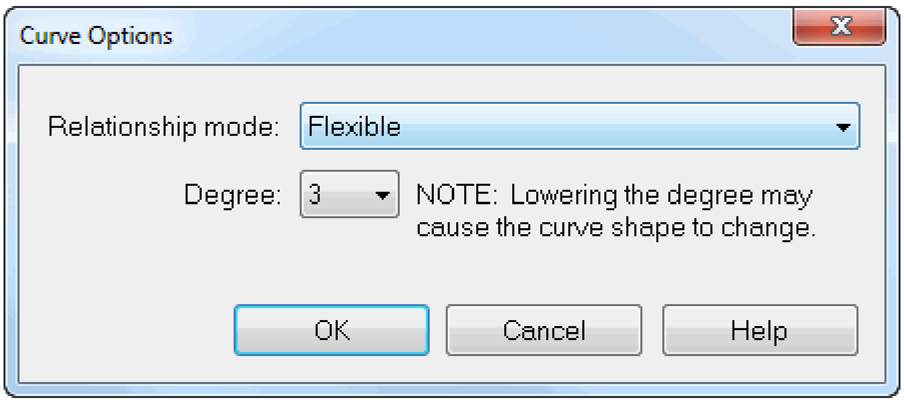




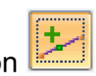
- After editing the curve, click the **Undo** command.

Lesson 15: Add more control to the curve

- Select the curve. On the **Edit Curve** command bar, click the **Curve Options** button .

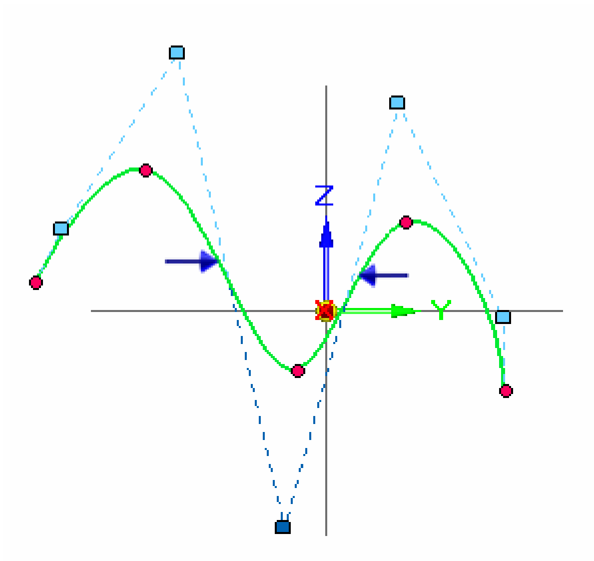


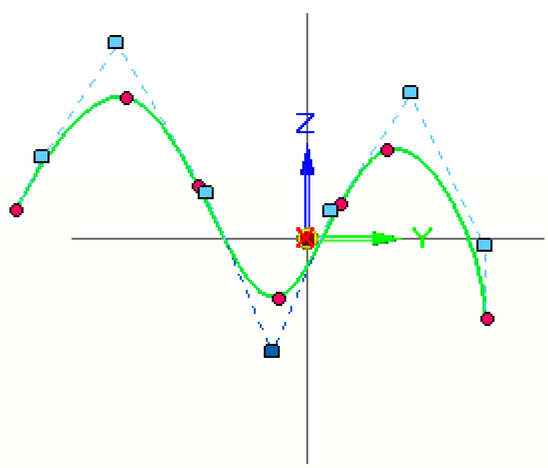
- In the **Curve Options** dialog box, ensure the **Degree** is set to 3 and click **OK**.

- On the **Edit Curve** command bar, click the **Add/Remove Points** button .

- Add edit points at the two locations shown below.

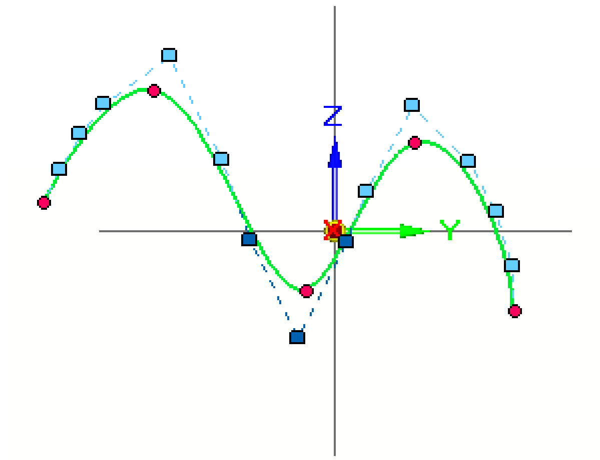
> **Note**
>
> You can only insert one edit point at a time with the **Add/Remove Points** button. You can either select the button again, or you can press and hold the Alt key while you click the curve to place as many points you need.





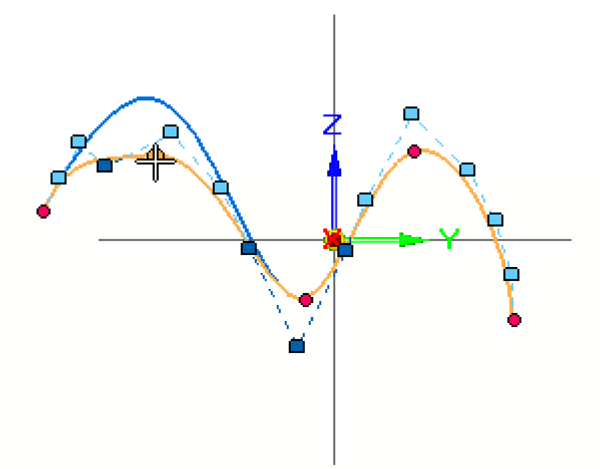
- Edit the curves again to observe how the shape changes. Undo to return the curve to its original shape.

- In the **Curve Options** dialog box, change the **Degree** from 3 to 5 and click **OK**. Observe the change to the control polygon.

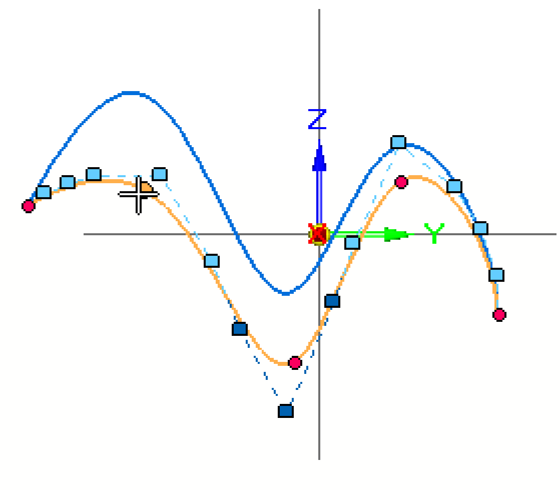


- Edit the curve again with both **Local Edit** and **Shape Edit** options to see how the curve shape changes now with the higher degree. Be sure to undo any changes you make to the curve.

Local Edit



Shape Edit

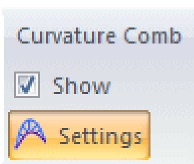


Lesson 16: Inspect the curve using Curvature Comb

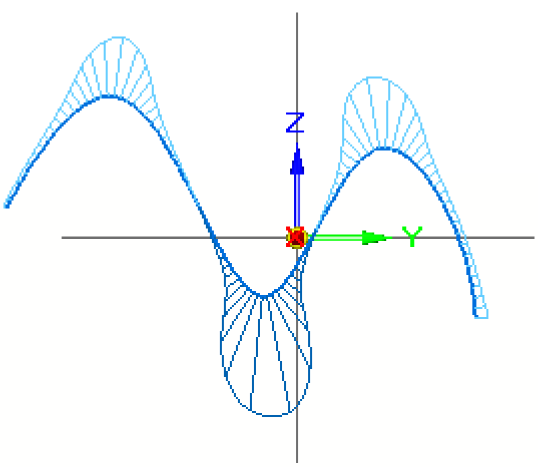
- Select the curve. On the **Edit Curve** command bar, select the **Show Curvature Comb** button .

> **Note**
>
> You can adjust the curvature comb display using the **Curvature Comb Settings** dialog box.

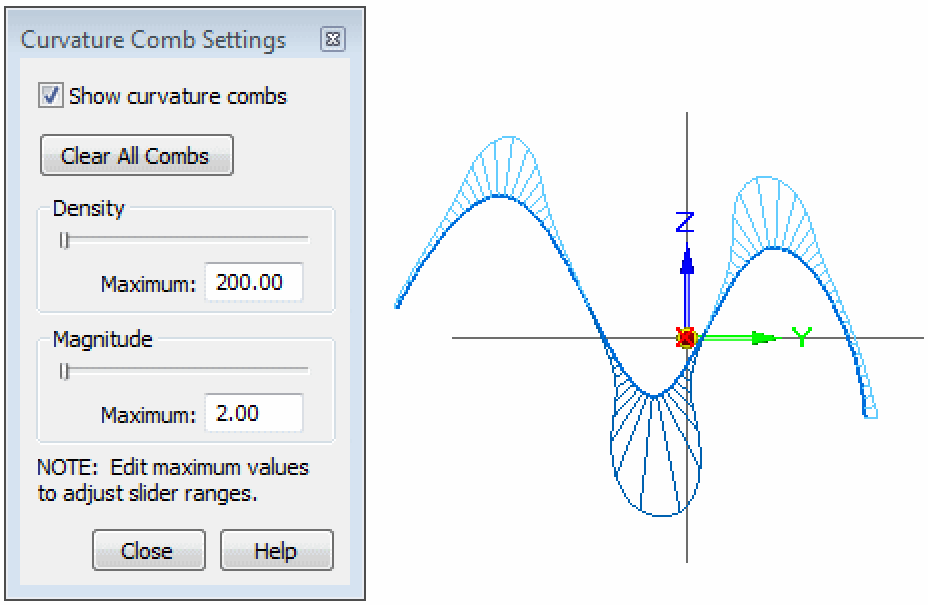
- Select **Inspect** tab→**Analyze** group→**Curvature Comb Settings**.



> **Note**
>
> Density controls the number of normal vectors. Magnitude controls the length of the vectors.



- Move the slider bars and observe the curvature comb display.



- In the **Curvature Comb Settings** dialog box, clear the **Show curvature combs** box and click **Close**.

- Click **Home** tab→**Close** group→**Close Sketch** to complete the sketch.

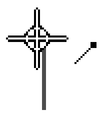
- On the **Sketch** command bar, select **Finish**.

- In PathFinder, turn off the sketch display.

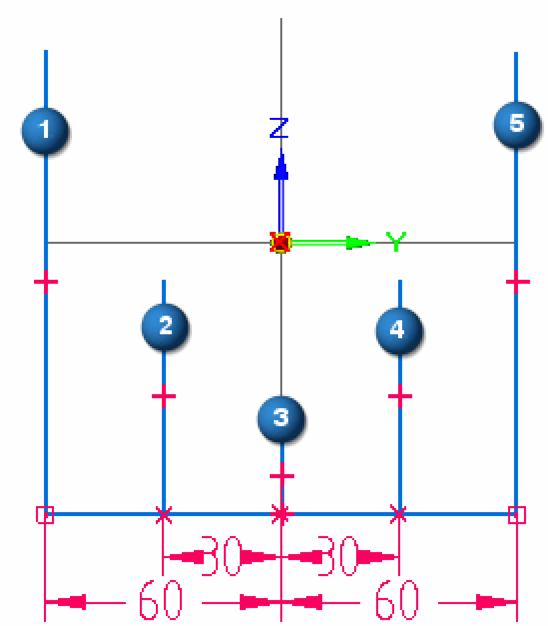
Lesson 17: Draw a curve connected to elements

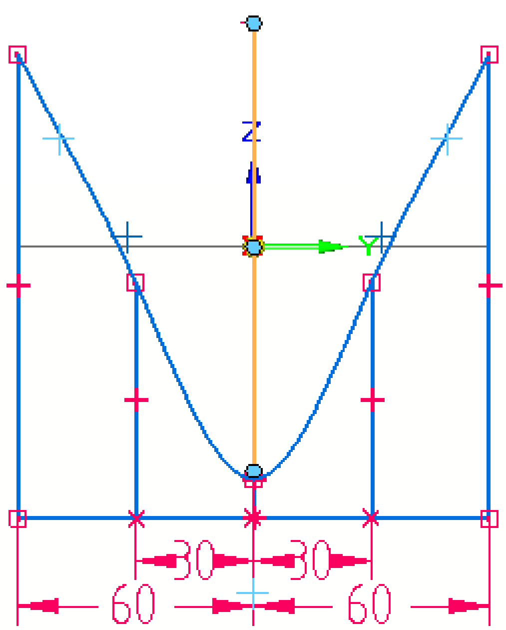
- In PathFinder, turn on the display of Sketch B.
- Select Sketch B and click **Edit Profile**.
- Click **Home** tab→**Draw** group→**Curve** .

- Draw a curve with edit points at the endpoints of the lines (1-5) shown below. Make sure to get the endpoint connect symbol before clicking. After placing the last edit point, right-click to complete the curve.

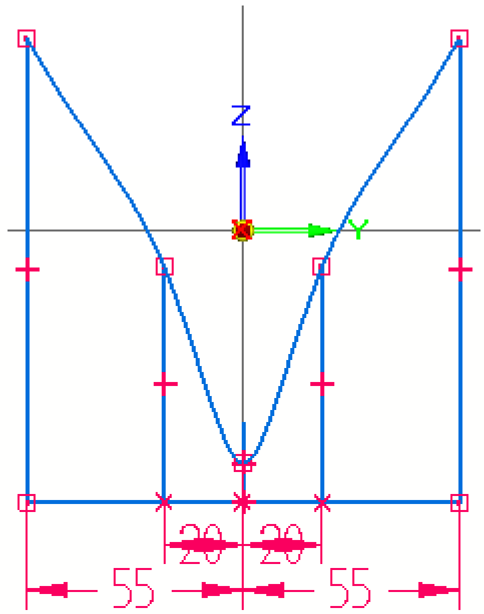


Endpoint Connect Symbol



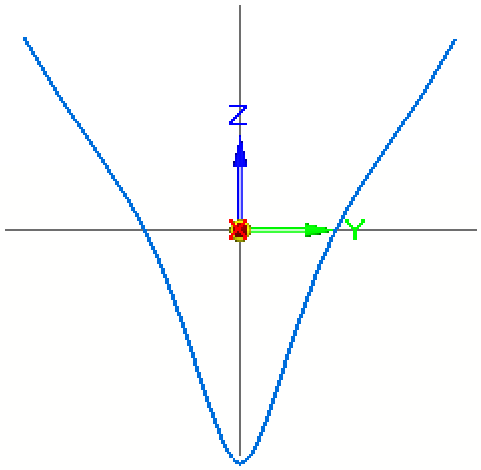


- Select the dimensions and edit their values as shown to observe how the curve is constrained to the dimensioned elements.

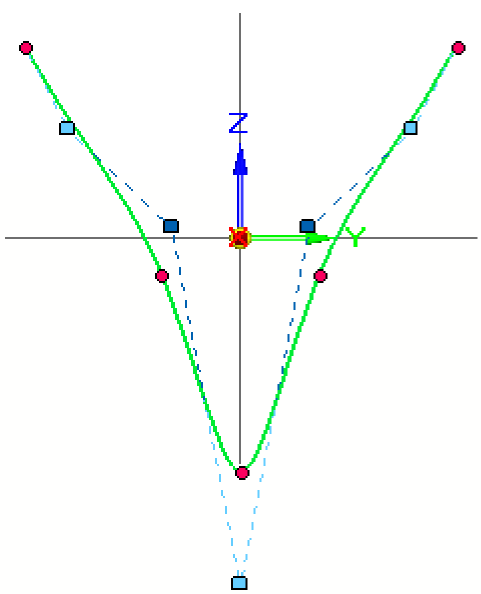


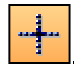
Lesson 18: Constrain the curve

- Delete all elements in the sketch except for the curve.



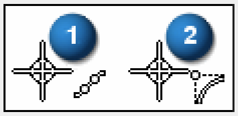
- Select the curve.



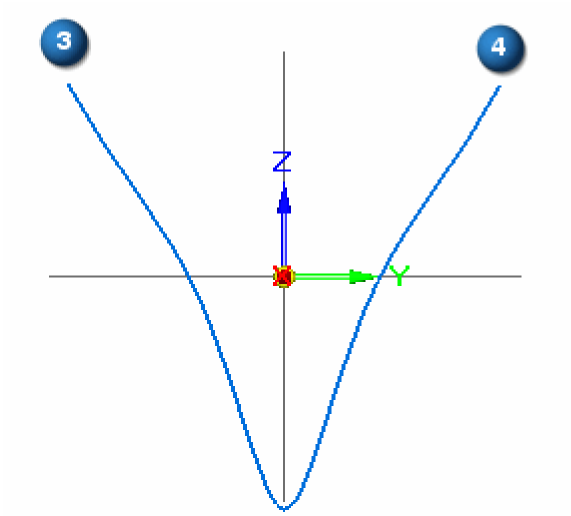
- Click **Home** tab→**Relate** group→**Horizontal/Vertical** .

Note

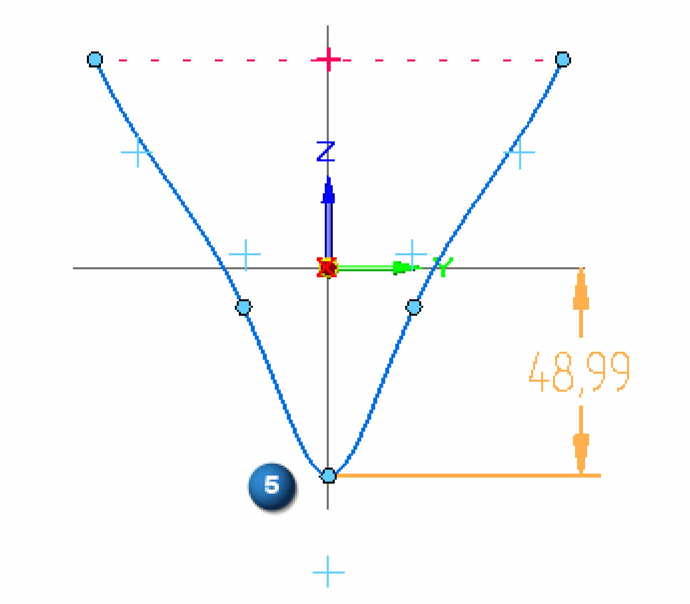
Notice that the edit points and control vertices display as crosses. If you position the cursor over a cross, you will see the following denoting if it is an edit point (1) or control vertex (2).



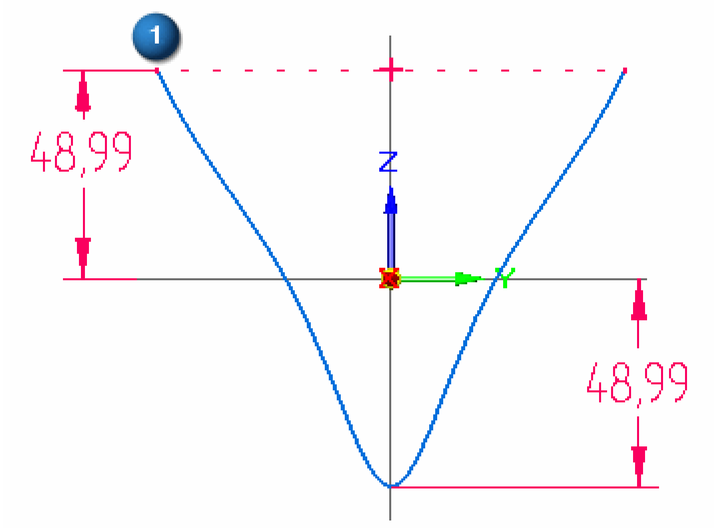
Click point (3) and then click point (4). Points (3) and (4) will remain aligned horizontally.



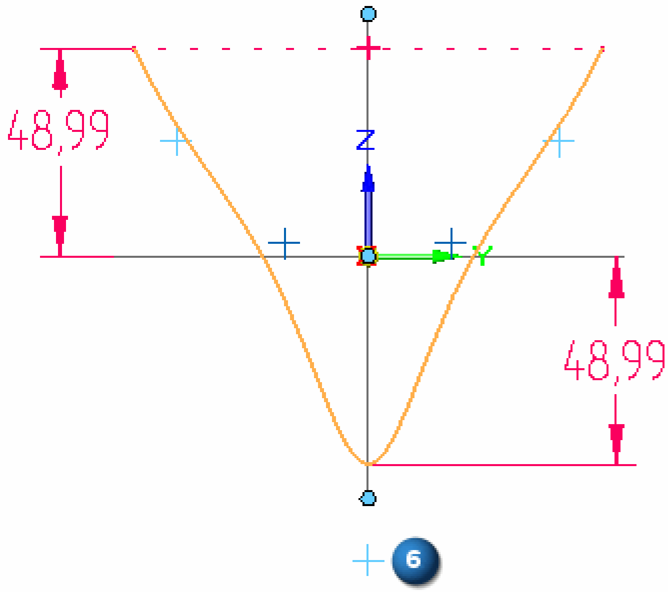
- Place a dimension as shown between the horizontal reference plane and edit point (5).



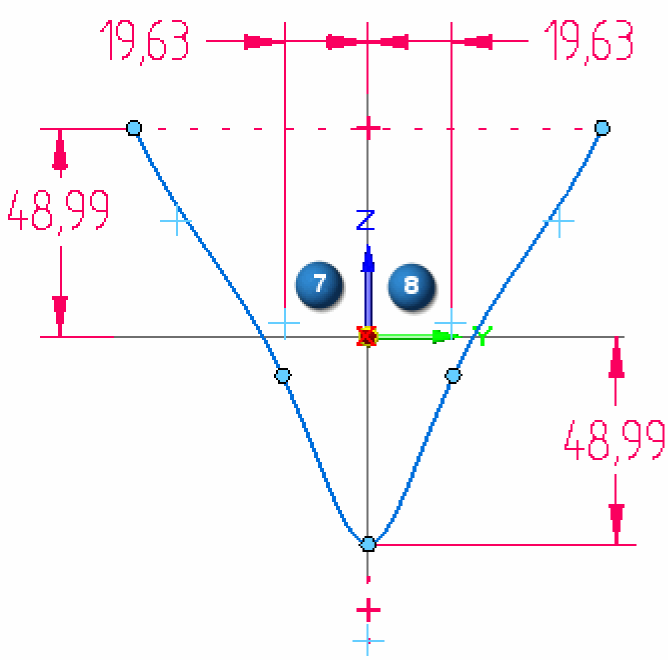
- Place a dimension as shown between the horizontal reference plane and edit point (1).



- Add a vertical relationship between control vertex (6) and the center of the reference planes.



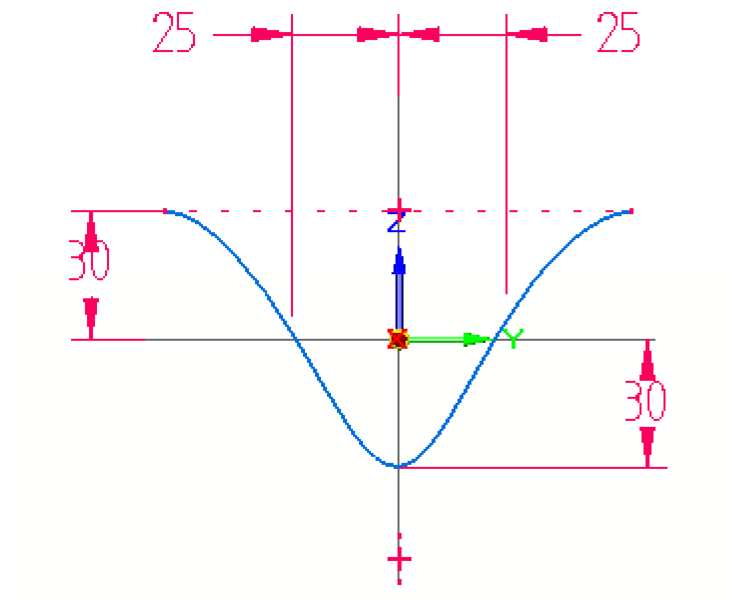
- Apply a final constraint to control vertices. Place two dimensions as shown between the vertical reference plane and control vertices (7) and (8).



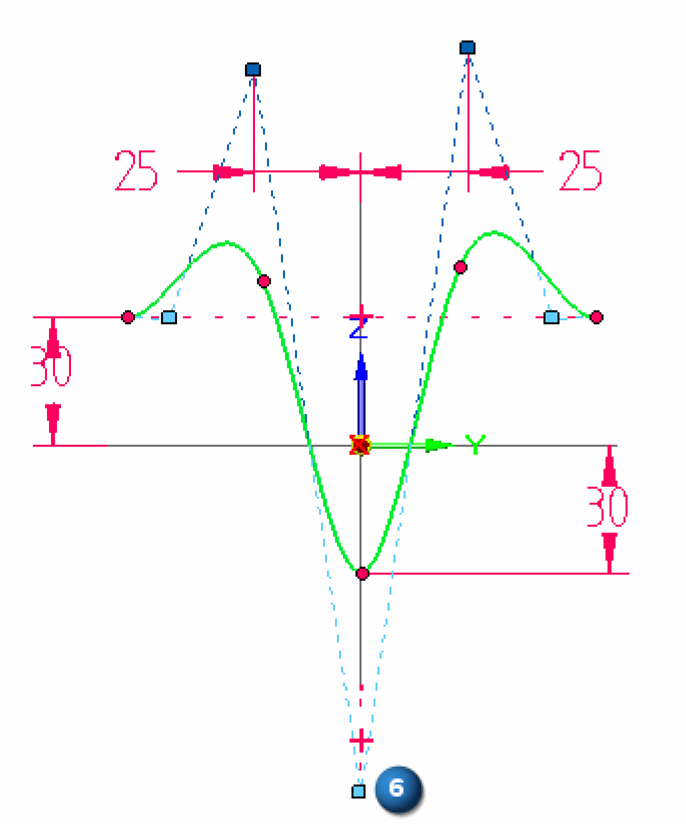
Note

More constraints are needed to make the curve symmetric about the vertical reference plane. For this activity stop adding constraints at this point.

- Edit the dimensions as shown and observe the curve shape constraints.



- Drag control vertex (6) down and observe how the curve shape changes while maintaining the relationships you applied.



- Save and close the part file.

Lesson 19: Summary

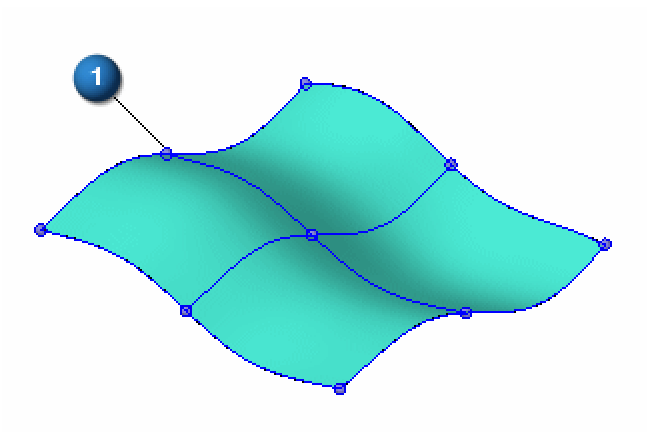
In this activity you learned how to draw and edit curves based on edit points and lines.

Lesson 20: BlueDot command (ordered modeling)

> **Note**
>
> BlueDots are only available in the ordered modeling environment

A *BlueDot* is a control point where two curves or analytics connect, or where one curve and one analytic connect, thereby providing a control point between the curves. It is a point which can edited to suit design or styling needs.

Use the **BlueDot** command (ordered modeling)  to create a control point (1) between two sketch elements. You can connect the elements at their keypoints or at a point along the elements. The BlueDot overrides any existing associativity of the elements. This allows you to edit the location of the BlueDot or the elements it connects without regard to the order in which the elements were constructed.

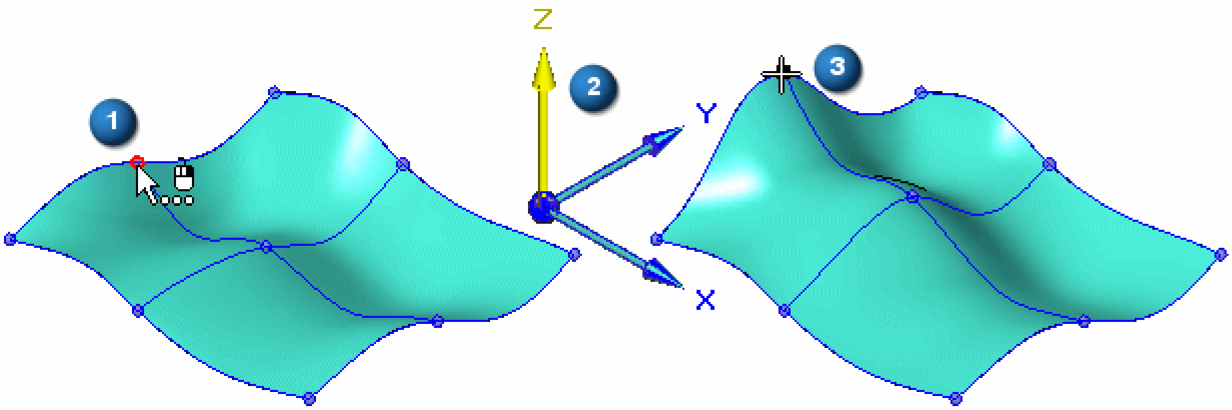


After you connect the keypoints of two elements with a BlueDot, you can edit the position of the BlueDot to change the shape of the elements. Surfaces that were constructed using the elements also update.

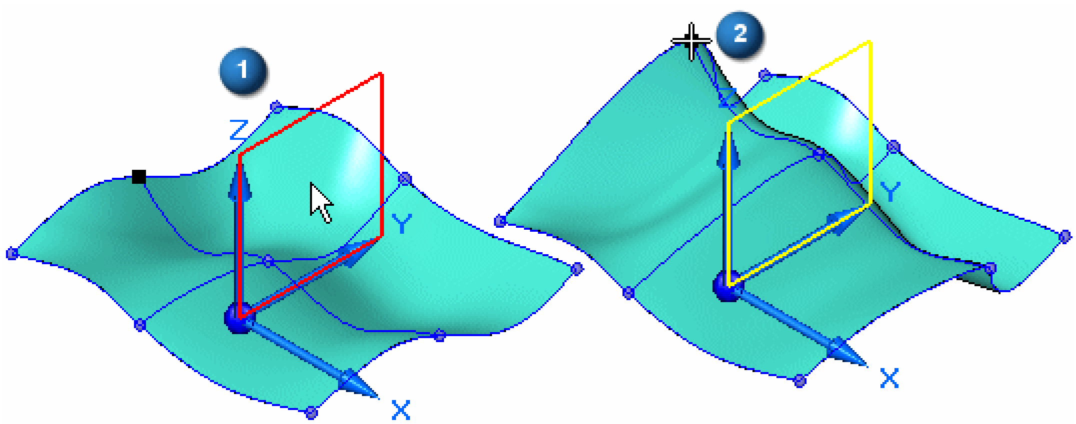
Refer to Connect sketch elements with a BlueDot for more info on BlueDot creation.

Edit a BlueDot

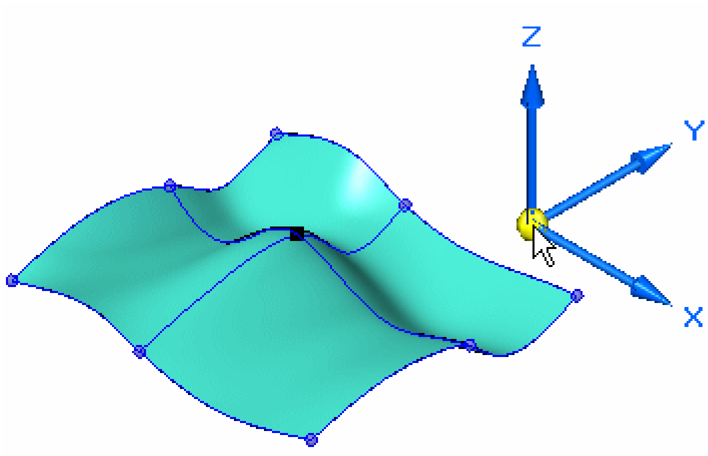
To edit the position of a BlueDot, use the Select Tool to select a BlueDot (1), then click the **Dynamic Edit** button on the **Select** command bar. When you edit the position of a BlueDot, you can use the OrientXpres tool (2) to restrict the movement to be parallel to a particular axis or plane. You can then drag the BlueDot to a new position (3). The wireframe elements and the surface also update.



When you use **OrientXpres** to restrict movement to a plane (1), you can move the BlueDot along two axes simultaneously (2).



You can also reposition the **OrientXpres** tool by selecting the origin of the X, Y, and Z axes, and then dragging **OrientXpres** to a new position.



You can use the **BlueDot Edit** command bar to specify whether the edit value is relative to its current position or its absolute position with respect to the global origin of the document. The global origin is the point where the three default reference planes intersect (the exact center of the design space).

When you apply a BlueDot to b-spline curves, you can also control how the curves react to the edit by setting options on the Curve 1 and Curve 2 controls on the command bar.

Note

When you use a BlueDot to connect two elements, it affects the associative relationship of the reference planes on which the elements lie. For example, if one of the elements lies on a reference plane that was created parallel to another reference plane, the dimensional offset value for the reference plane is deleted. When you edit the position of the BlueDot, the reference plane can be moved to a new position to facilitate the repositioning of the elements.

Refer to BlueDot Edit command bar for more information.

Lesson 21: BlueDot Edit command bar (ordered modeling)

> **Note**
>
> BlueDots are only available in the ordered modeling environment.

Relative/Absolute Position

Specifies whether the value you type is relative to the BlueDot's current position or is based on the global origin of the document. The global origin is the point where the three default reference planes intersect (the exact center of the design space).

X

Sets the position for the x axis.

Y

Sets the position for the y axis.

Z

Sets the position for the z axis.

Curve 1

Specifies the edit method you want for curve 1. This option is only available for b-spline curves. When you edit the position of a BlueDot that connects a curve to another element, you can set the following options to control how the curve is modified.

Shape Edit—Affects the shape of the entire curve when you move a point on the curve.

Local Edit—Affects the shape of a limited portion of the curve around the edit point.

Rigid—Prevents the curve from being modified.

> **Note**
>
> This option is not available in Wire Harness.

Curve 2

Specifies the edit method you want for curve 2. This option is only available for b-spline curves. This option is not available when editing BlueDots in Wire Harness. When you edit the position of a BlueDot that connects a curve to another element, you can set the following options to control how the curve is modified.

Shape Edit—Affects the shape of the entire curve when you move a point on the curve.

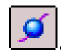
Local Edit—Affects the shape of a limited portion of the curve around the edit point.

Rigid—Prevents the curve from being modified.

Note

This option is not available in Wire Harness.

Lesson 22: Connect sketch elements with a BlueDot

1. Choose **Surfacing** tab→**Surfaces** group→**BlueDot** .
2. Select a keypoint on the first element.
3. Select a keypoint on the second element.

Note

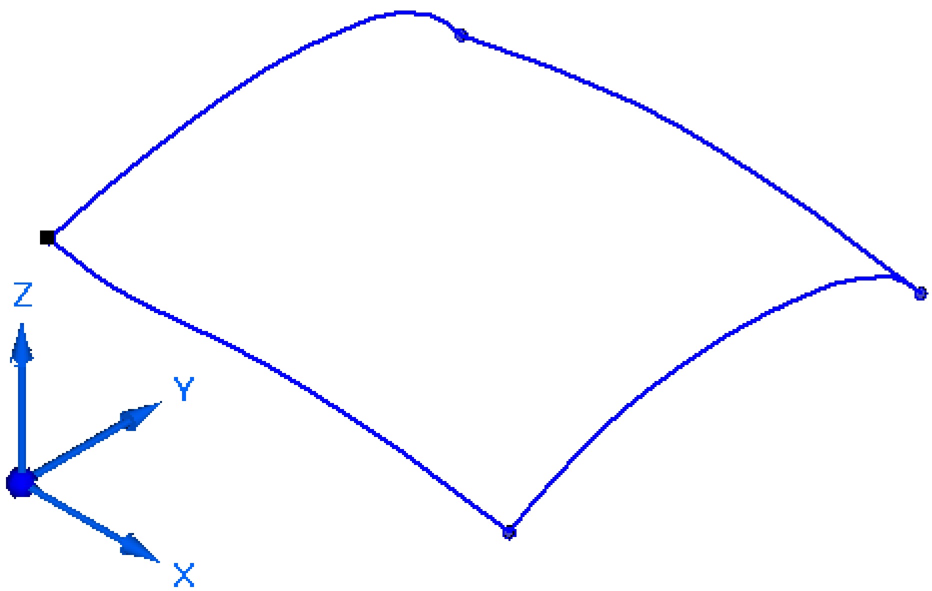
The first curve moves to intersect the second curve. Also, the shape and location of the first curve may change, but the second curve maintains its initial shape and location.

Tip

- Each curve has four select zones: two endpoints, a midpoint and the curve itself.

- You can also use a BlueDot to connect elements at a point along the elements.
- You can edit the position of a BlueDot using the Select Tool and the **BlueDot Edit** command bar.
- You can use the OrientXpres tool to limit the edit to be parallel an axis or plane you select.
- When using the **BlueDot** command in Wire Harness, you can connect the end point of more than two harness elements.

Lesson 23: Activity: Creating and editing BlueDots



Overview

In this activity, learn to manually create and edit BlueDots.

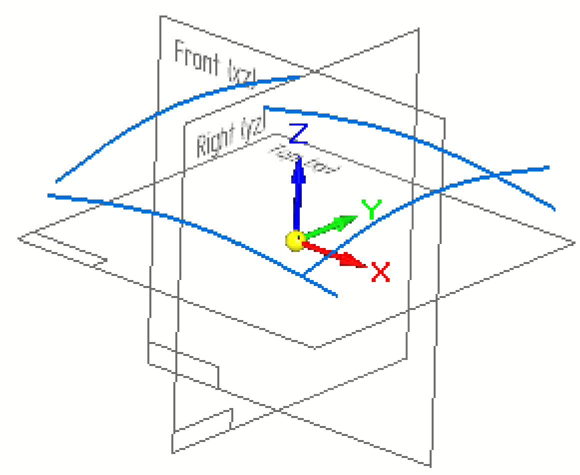
Objectives

After completing this activity you will be able to:

- Create BlueDots.
- Edit BlueDots and curves based on them.

Lesson 24: Open part file

- Open **surface lab 2-02.par**.



Note

Curves must be connected in order to use them to create surfaces. Only the **Swept Surface** command does not require input curves to be connected. You will learn more about this in the next lesson.

Note

The order in which you select curves determines which curve will change location. The first curve you select moves to connect to the second curve. The first curve sketch plane changes to the connected location. The second curve you select does not change.

Note

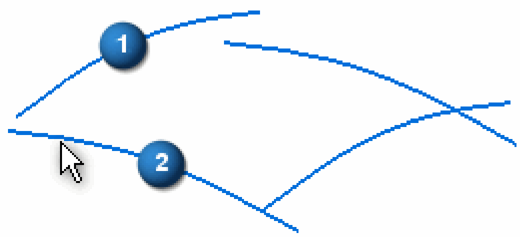
There are several curve selection locations. See the topic on BlueDot Creation in the theory section of this lesson.

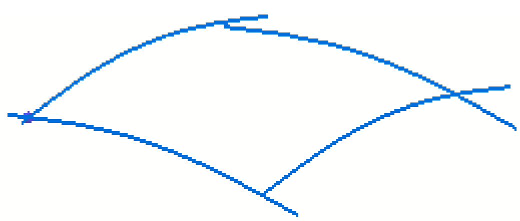
Lesson 25: Use BlueDots to connect two curves

Experiment with connecting two curves using different selection locations. Remember to undo after each connection to return the curves to their original location.

- Click **Surfacing** tab→**Modify Surfaces** group→**BlueDot** command .

- Select curve 1 at the location shown and then select curve 2 at the location shown. Notice the resulting connection and then click Undo.



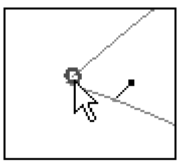


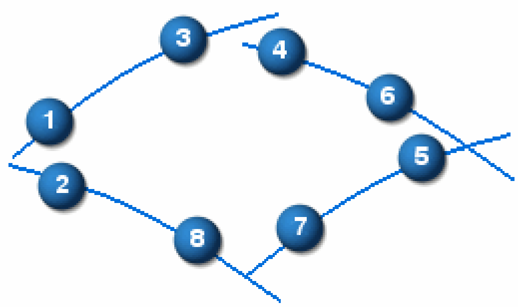
Lesson 26: Connect the four curves at the end points

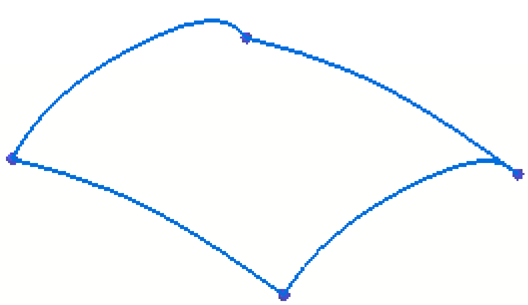
After having experimented with different curve selection possibilities, connect the four curves at the end points.

- Click the **BlueDot** command and connect the curves in the sequence shown below (1-2, 3-4, 5-6, and 7-8).

> **Note**
>
> Make sure the endpoint connect symbol displays before you click.





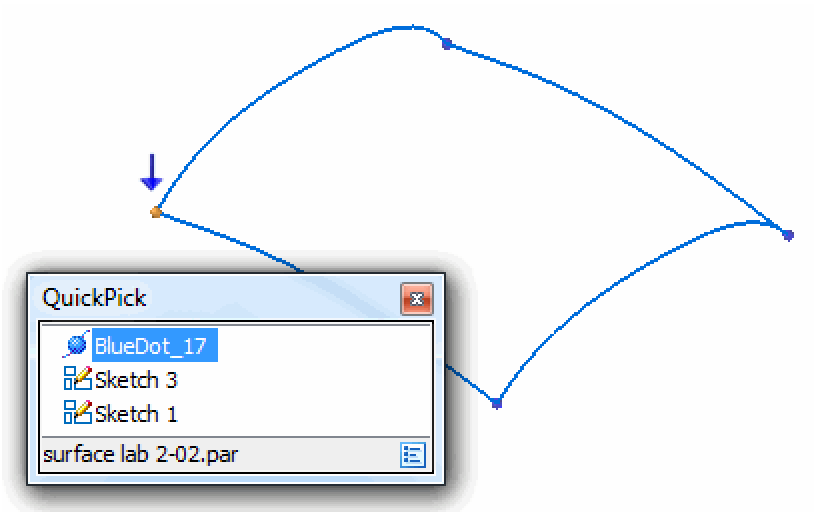


Right-click to finish.

Lesson 27: Edit a BlueDot

The four curves are now BlueDot connected. Edit a BlueDot to observe how the curves behave.

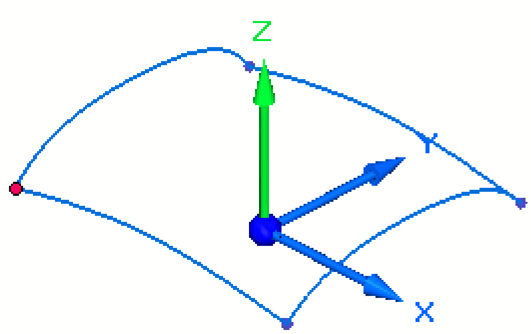
- Select the BlueDot shown. Use **QuickPick** as an aid to select it.



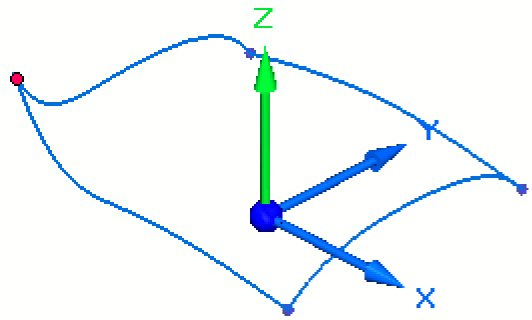
- On the command bar, select **Dynamic Edit** .

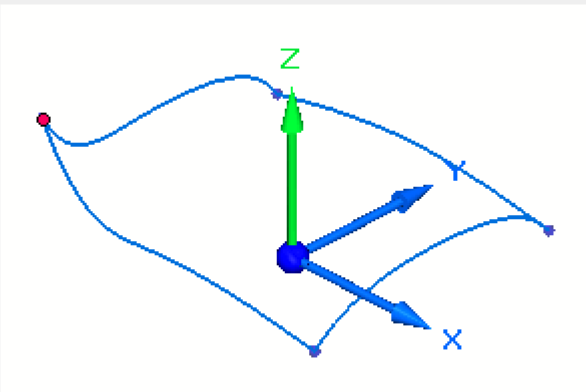
- Click the Z-direction axis on the 3D triad as shown.

  This locks the BlueDot movement to the Z-direction.



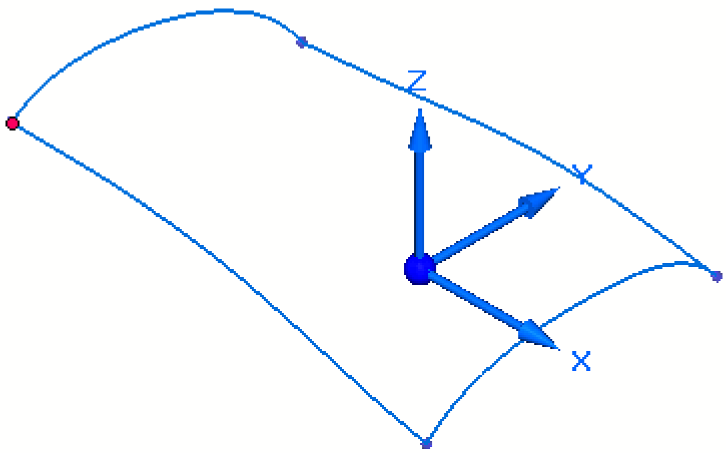
- Edit the BlueDot by dragging it in the graphics window or by typing a new Z-coordinate value. Drag the BlueDot a small distance as shown and observe the behavior of the connected curves.



> **Note**
>
> Both curves are set to Local Edit.
>
> 
>

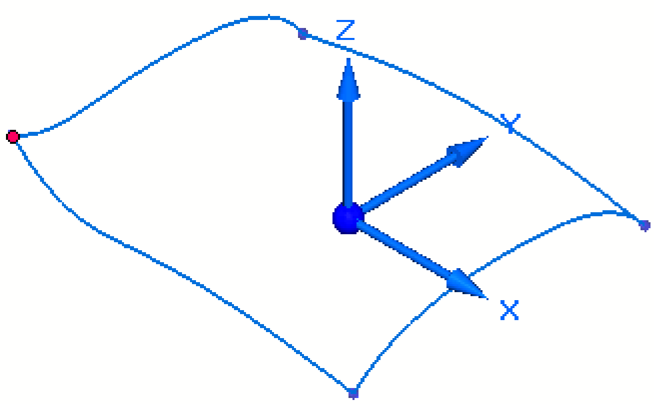
- Click **Undo** to return the BlueDot to its original location.
- Set both curves to Shape Edit.

- Select the BlueDot. On the command bar, select **Dynamic Edit** again. Notice the different result.



- Click **Undo** to return the BlueDot to its original location.
- On the command bar, select **Dynamic Edit** again.
- Edit a BlueDot by specifying a delta distance. Repeat the previous step, but this time click the **Relative/Absolute Position** option on the **BlueDot Edit** command bar.

- Notice that the command bar changes to dX, dY and dZ. Type **20** in the dZ box and Enter.

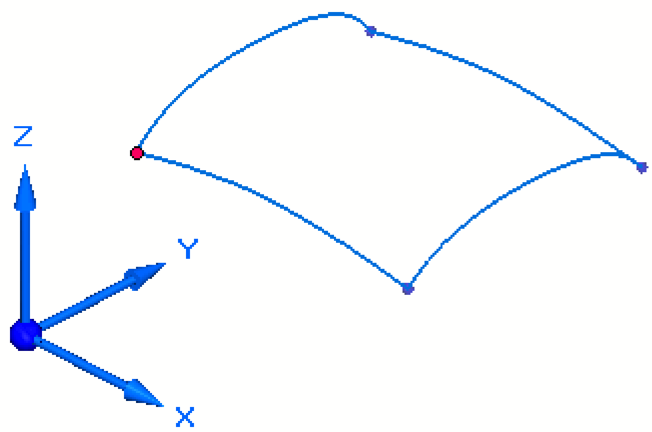


Note

If you Enter a second time, a delta value of 20 is applied again.

- Click **Undo** to return the BlueDot to its original location.

- The 3D triad can be moved if it gets in the way. Click the 3D triad as shown and drag to a new location.



- The activity is complete.

Lesson 28: Summary

In this activity you learned how to draw and edit curves based on BlueDots.

Lesson 29: Lesson review

Answer the following questions:

1. What is an edit point on a curve?
2. How do you display the control polygon of a curve?
3. Explain the differences in **Shape Edit**, **Local Edit**, and **Rigid** when moving a point on the curve.
4. How do you change the degree of a curve?
5. Explain what a BlueDot is and what impact it has on curves.
6. How do you convert analytics to B-splines?
7. What is the Curvature Comb used for?
8. On the **BlueDot** command bar, what does the **Relative/Absolute Position** option do?

Lesson 30: Lesson summary

Surface shapes are directly tied to the curves defining those surfaces. Therefore, the control of curves is crucial in modifying surface topology.

A curve:

- Can be edited by moving its edit points and control points.
- Can be further controlled by increasing its degree.
- Can be drawn directly by first defining edit points. Direct methods include:
  - Curve
  - Curve by table
  - Contour curve
- Can be created indirectly from existing curves and surfaces, making them dependent on underlying parent curves and surfaces. As the parent changes, so do the indirect curves.

  Indirect curve methods are covered in the next chapter.

Lesson 31: Indirect curve creation techniques

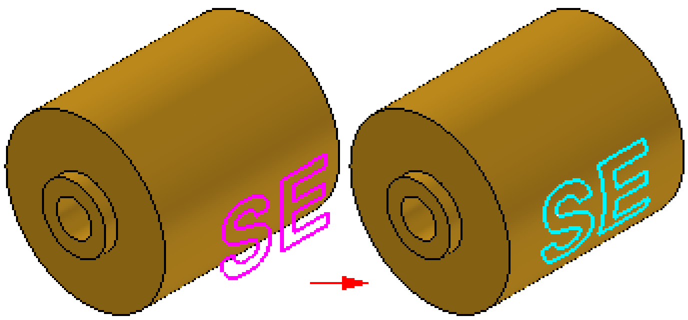
Objectives

After completing this lesson, you will be able to:

- Use the following commands to create curves derived from other geometry:
  - Project curves
  - Intersection curves
  - Cross curves
  - Contour curves
  - Derived curves
  - Offset edge
  - Split curves
  - Keypoint curves
  - Curve by table
- Define and edit pierce and silhouette points.
- Draw curves overtop of a raster image.

Lesson 32: Project Curve command

Use the **Project Curve** command  to project one or more curves (2D or 3D) onto a surface or set of surfaces. You can project the curve along a vector or along surface normals. You can also use this command to project a point onto a surface.



Use the command bar to specify that you want to project a single element, a chain of elements, a point, or an entire sketch.

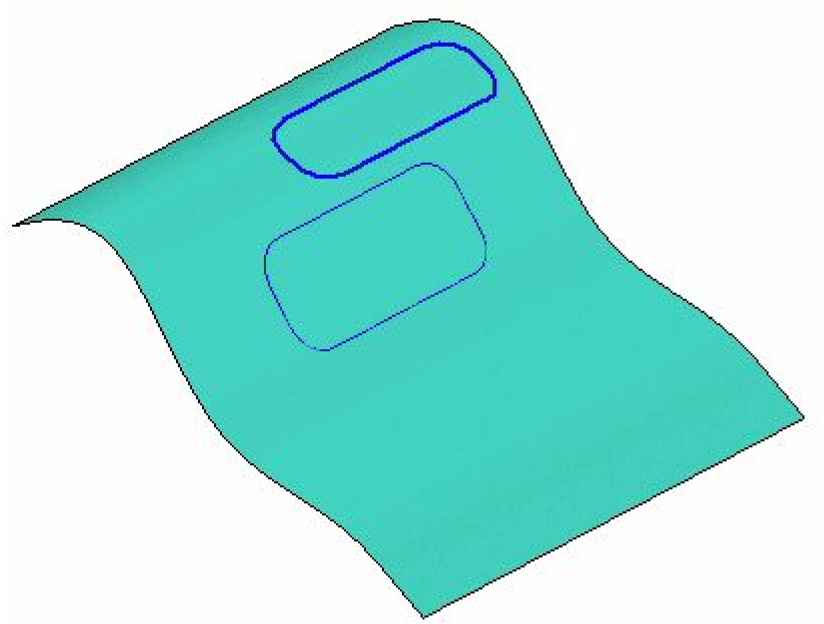
Wireframe elements selected from multiple Parasolid bodies remain associative.

> **Note**
>
> When projecting a curve onto a cylinder, make sure that the curve endpoints do not lie on a silhouette edge of the cylinder when viewed from the projection plane normal. Extend the edges of the curve past the cylinder edge.

Lesson 33: Project Curve Options dialog box

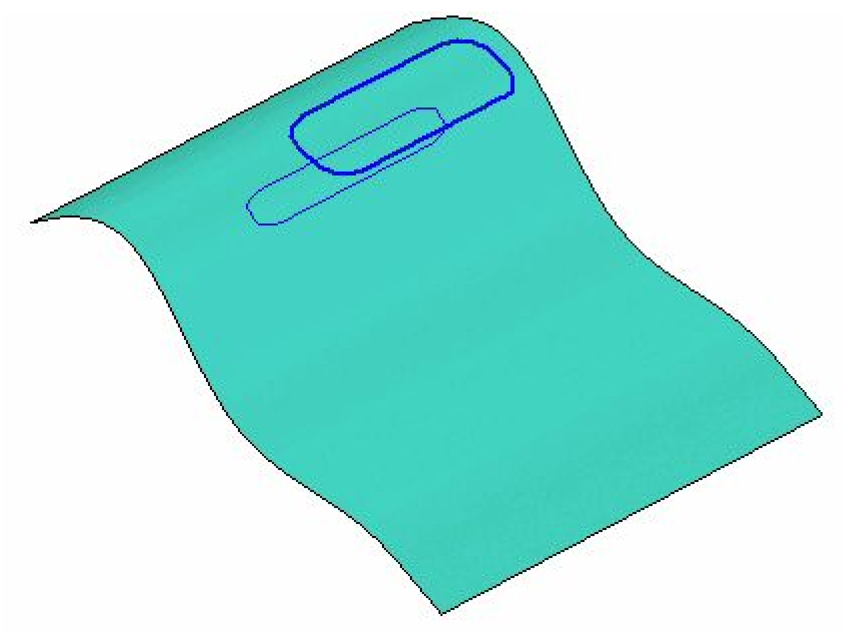
Along Vector

Specifies that the curve or point be projected along a vector you define.



Normal to Selected Surface

Specifies that the curve or point be projected along surface normals.

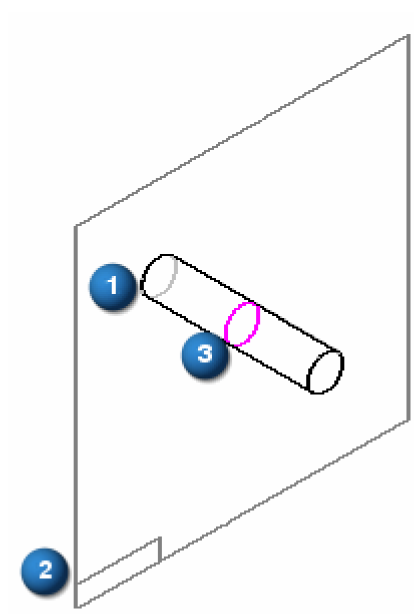


Lesson 34: Intersection Curve command

Use the **Intersection Curve** command  to create an associative curve at the intersection of two sets of surfaces. The surface sets can be any combination of reference planes, model faces, or construction surfaces.

An intersection curve is associative to the surfaces it is based on, so the curve updates if either set of surfaces changes.

For example, you can intersect a cylinder (1) with a reference plane (2). The resulting intersection curve (3) can then be used as input for constructing a feature or in a surface trimming operation.

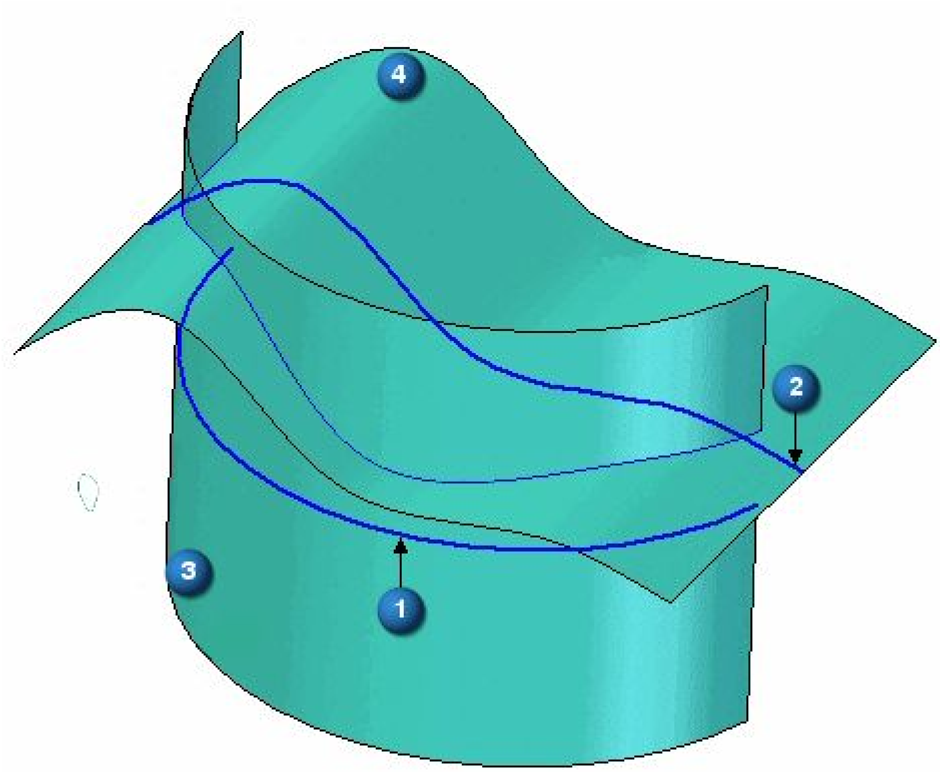


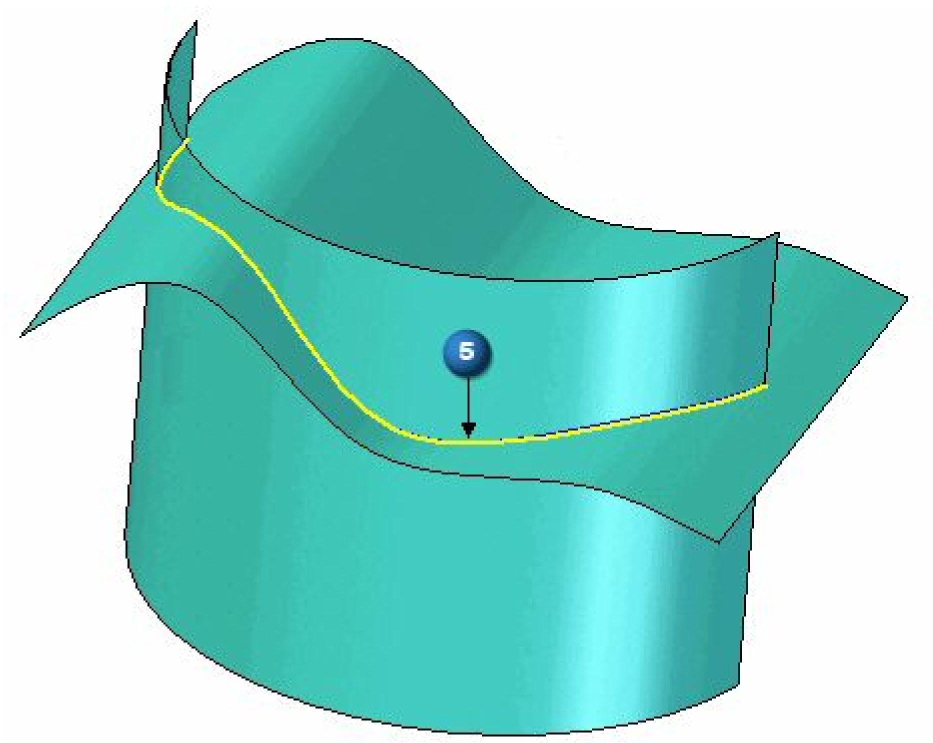
Lesson 35: Cross Curve command

Use the **Cross Curve** command  to create a 3-D curve at the intersection of two curves.

- The command works much like the **Intersection Curve** command, yet it does not need existing surfaces to create a curve.
- The only input required is two curves/analytics or a combination of the two.
- An intersection curve is created with the theoretical extruded surfaces resulting from the two input curves or analytics.

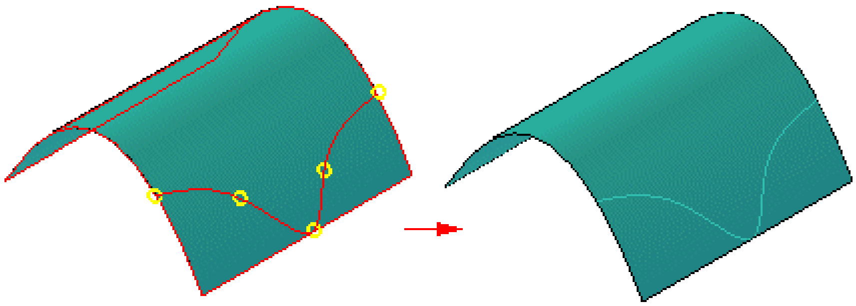
(1) and (2) are the input curves. (3) and (4) are the theoretical extruded surfaces. (5) is the resultant cross curve.



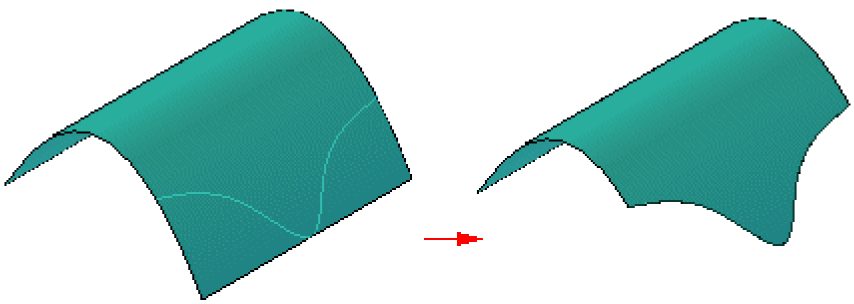


Lesson 36: Contour Curve command

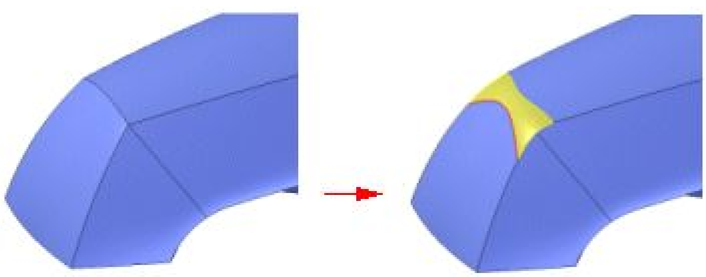
Use the **Contour Curve** command  to draw a curve directly on a surface.



You can then use the curve for such things as a border in trimming operations



or as a tangent hold line in rounding operations.



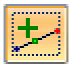
You can select a single face or multiple faces when defining the faces on which you want to draw the curve. You can only draw within the bounded region and the curve will only lie within the bounded region. Curves that fall off the surface or surfaces or traverse trimmed regions are trimmed.

When defining the points for the curve you can use existing points that define the surface, such as vertices, line midpoints, and edges of the surface.

You can add and delete points for the curve to follow and you can drag the points anywhere on the surface.

Tips for creation and manipulation of contour curves.

- Choose **Contour** command bar→**Draw Points** step→**Insert Point** to insert additional points into the curve. To delete a point from the curve, hold the Shift key and click on the point or right-click on the point.

- You can connect a keypoint and an existing keypoint. To do this, right-click the existing keypoint and select **Connect**; follow the prompts to identify the other keypoint.

- You can delete the connect relationships on a keypoint so that you can drag the keypoint on a face. To delete the relationship, right-click the relationship and follow the prompts.

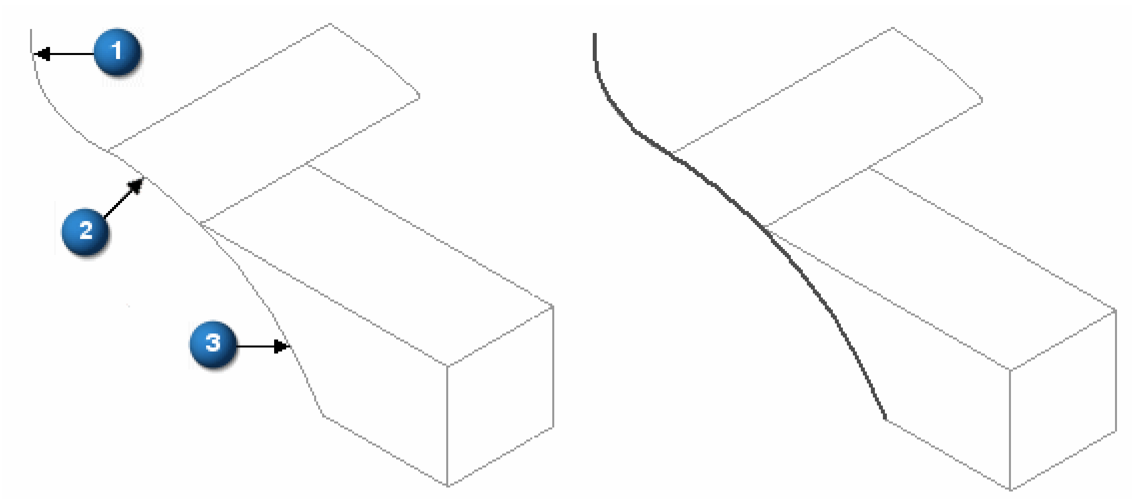
- You can drag an existing point to a new location on the face.

- When drawing a curve across faces that are not tangent, you must place a point at the shared edge.

Lesson 37: Derived Curve command

Use the **Derived Curve** command  to construct a new curve that is derived from one or more input curves or edges. If all the input curves or edges are connected at their endpoints, you can specify that the derived curve is constructed as a single b-spline curve. If the input curves are connected, but not tangent, the output curve will have a minimal amount of curvature added so that a single, smooth b-spline curve is constructed.

You can construct a single derived curve from multiple bodies. For example, you can construct a derived curve from a sketch (1), edges on a construction surface (2), and edges on a solid (3).



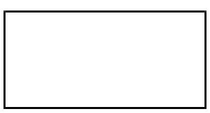
Lesson 38: Offset Edge command

Use the **Offset Edge** command  to offset the selected edges to create an imprint of them on a part or surface by a given distance and direction. You can use this command in synchronous and ordered environments in part and sheet metal models.

Eligible selected edges must form a closed loop on the same plane, or a tangentially continuous chain of edges that do not lie on a planar face. You can select multiple edges from the same solid or surface, or edges across multiple solids and surfaces.

- A closed loop that is tangentially connected:

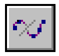
- A closed loop that is not tangentially connected:

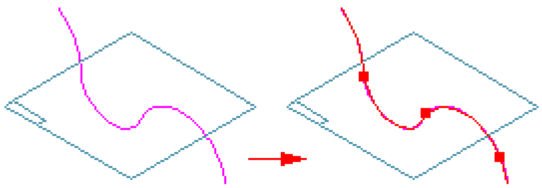


For finite element analysis models that contain representations of bolts, you can use the **Offset Edge** command to produce better meshing results around bolt holes. In this application, the command generates offset faces to represent where each bolt, nut, and washer come in contact with a hole. This produces more nodes for the spider mesh to connect to and a better representation of the bolt.

> **Note**
>
> You can use the **Derived Curve** command to produce a new curve that is derived from one or more input curves.

Lesson 39: Split Curve command

Use the **Split Curve** command  to split a construction curve. You can select keypoints, curves, reference planes, or surfaces as the elements which split the curve.



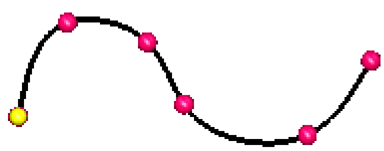
Splitting a construction curve can make it easier to construct other features, such as a bounded surface, a trimmed surface, a normal protrusion, or a normal cutout.

> **Note**
>
> You cannot use the **Split Curve** command to split an edge on a model. You can use the **Derived Curve** command to create an associative copy of an edge on the model, then use the **Split Curve** command to split the derived curve.

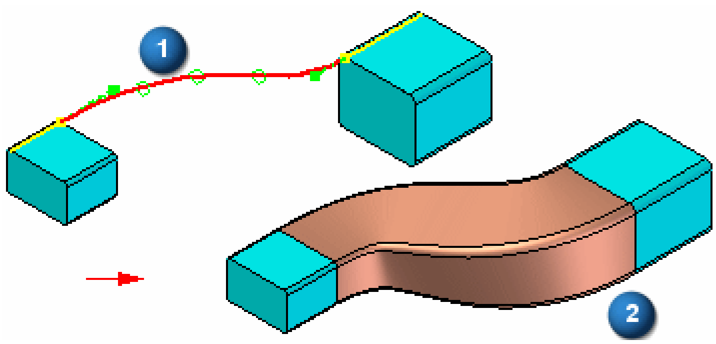
Split Curve command bar

Lesson 40: Keypoint Curve command

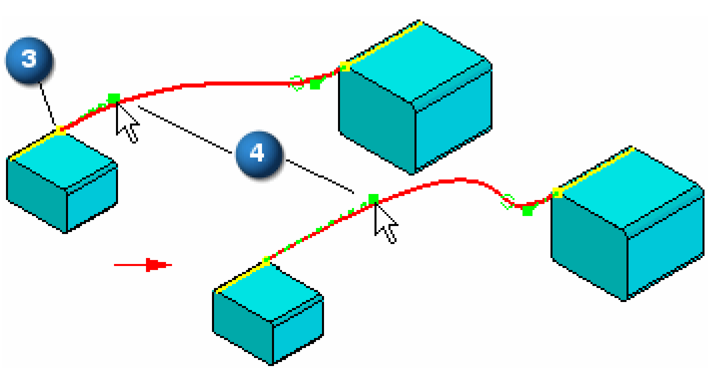
Use the **Keypoint Curve** command to create a 3-D curve through a set of three or more points. The points can be points you create with the Point command, keypoints on wireframe elements and edges, or points in free space.



You can use this command to create a bridge curve (1), which can be used as a path for a swept feature (2).



When you select a keypoint on a wireframe element or edge as the endpoint (3) of the curve, the **End Conditions** Step allows you to specify whether the curve is created tangent to the wireframe element or edge you selected. When you specify that the curve is tangent to an element at its endpoint, you can also modify the magnitude of the tangent vector by dragging the tangent vector handle (4) to a new location. When you modify the magnitude of the tangent vector, you also may change the radius of curvature of the curve. If the modified curve was used as a path for a swept feature, the swept feature will also update.

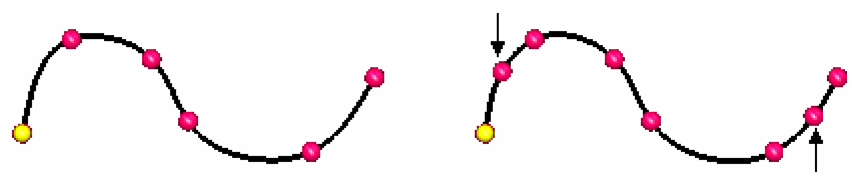


You can use the OrientXpres tool to help you define the location of a point on a keypoint curve. For example, you can use **OrientXpres** to lock input to a particular axis or plane when creating or editing a keypoint curve.

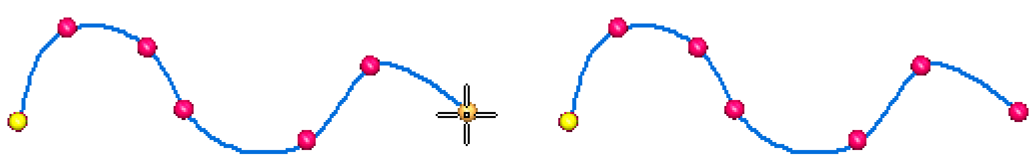
Inserting points to a curve

You can add new points along a curve or add a point in free space to add a new segment to the end of the curve.

To add a point along a path, while editing the curve, hold the Alt key and click the location along the curve where you want to add the point.



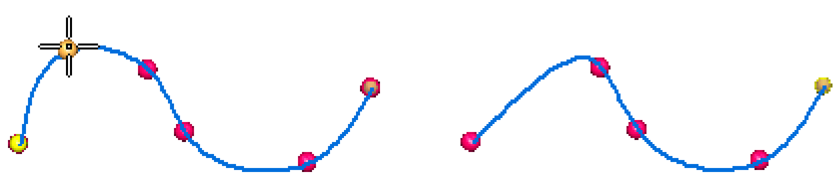
To add a point to the end of the path, while editing the curve, hold the Alt key and click a location in free space where you want to add the point.



Removing points from a curve

You can remove a point from a curve.

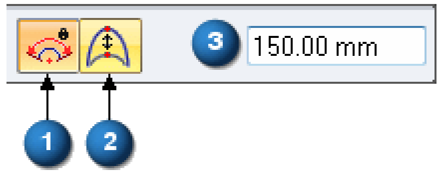
To remove a point, while editing the curve, hold the Alt key and click the point you want to remove. When you remove edit points, the control vertex points move and the shape of the curve changes.



If you remove the start or end point of a curve, the path truncates to the next control on the curve and the tangency of the next point remains the same.

Defining curve length

You can set the **Curve Length**  options for the keypoint curve definition. When this option is set, the curve length remains fixed during an edit. You click the **Fixed Length** (1) button to lock the curve length (3). The **Constrain Direction** (2) button defines the direction that the control points move when you edit the curve length You can constrain along the X, Y and Z axis or any linear edge/curve or axis. The increase or decrease in length bows in that direction.

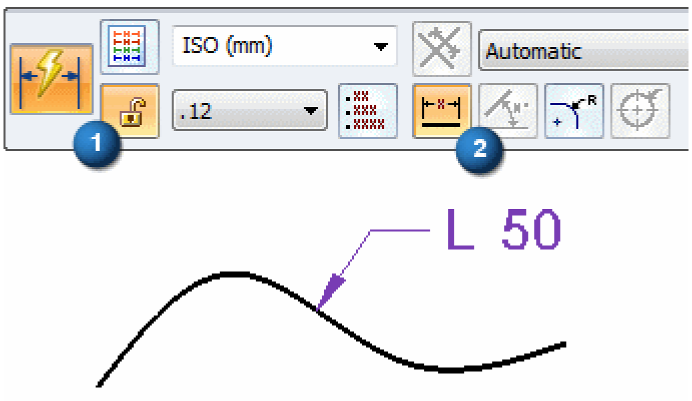


Note

Synchronous keypoint curves cannot be edited after creation.

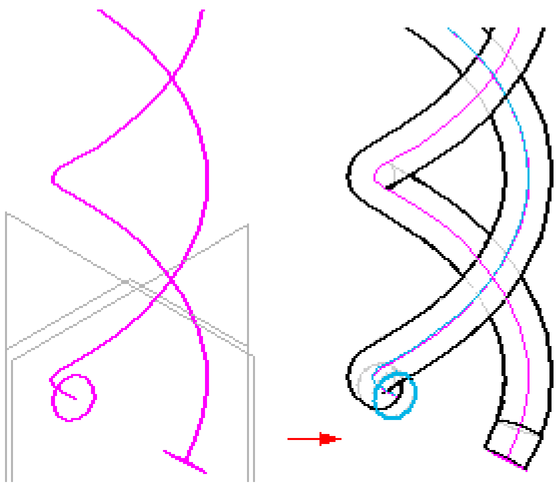
Dimensioning a keypoint curve

When placing a **Smart** dimension on a keypoint curve, the dimension type is set to Length (2), unlocked (1), and cannot be edited.



Lesson 41: Curve By Table command

The **Curve By Table** command  uses an Excel spreadsheet to define a construction curve. The spreadsheet, which is embedded in the Solid Edge document, allows you to better import and manage engineered curves. You can generate a curve by creating a new spreadsheet or by opening an existing spreadsheet. For example, you can create two helix curves using a spreadsheet with the **Curve By Table** command. You can then use these curves as paths to construct swept protrusions.



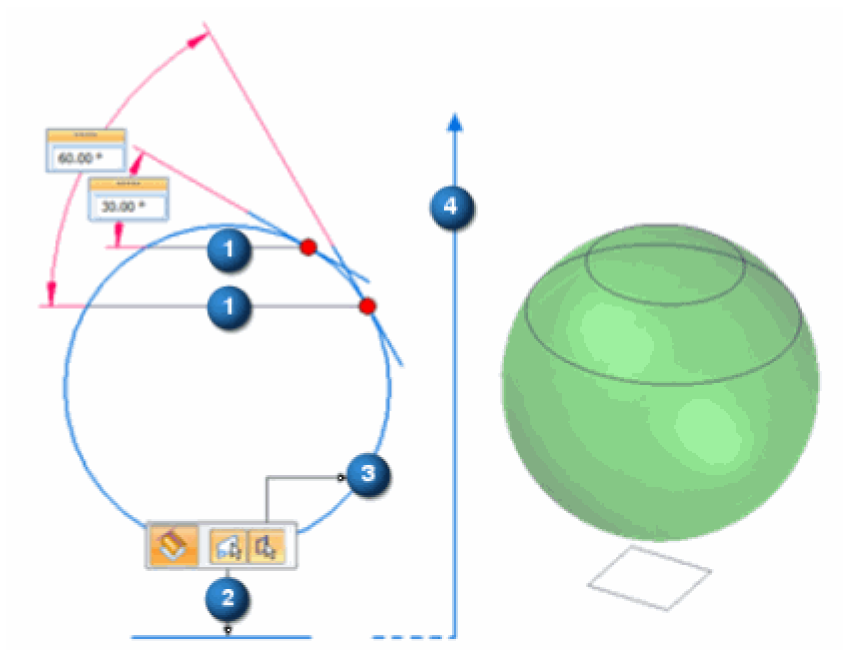
Note

To create a curve using the **Curve By Table** command, or to edit an existing curve, Microsoft Excel must be loaded on your machine.

Lesson 42: Isocline command

Use the **Isocline** command  to place an isocline curve on one or more selected faces. An *isocline* curve is a curve connecting points on a surface whose surface normal makes a constant angle to the direction of pull (4).

The required inputs are: a reference plane, surface body, and an angle.



The above example shows two single isocline curves (1) created on a spherical surface (3), at 30 and 60 degrees from the reference plane (2).

You can see the relationship between the angle and where the isocline curve is created. The direction vector (4) is normal to the reference plane.

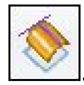
You can create isocline curves on any selected face, or on multiple faces. You may need to flip the direction vector to get corresponding isocline curves on adjacent faces.

The input angle range is 0.00 < 90.00 degrees.

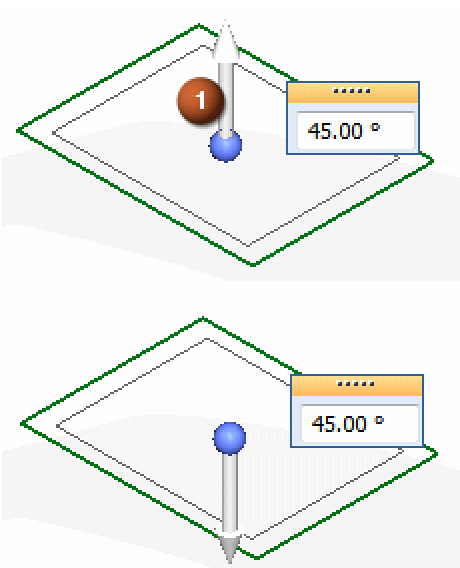
Use isocline curves to:

- Split a surface.
- Construct parting lines and parting surfaces on a mold or casting.
- Analyze a face/surface using draft angle map (family of isoclines).
- create new surfaces.

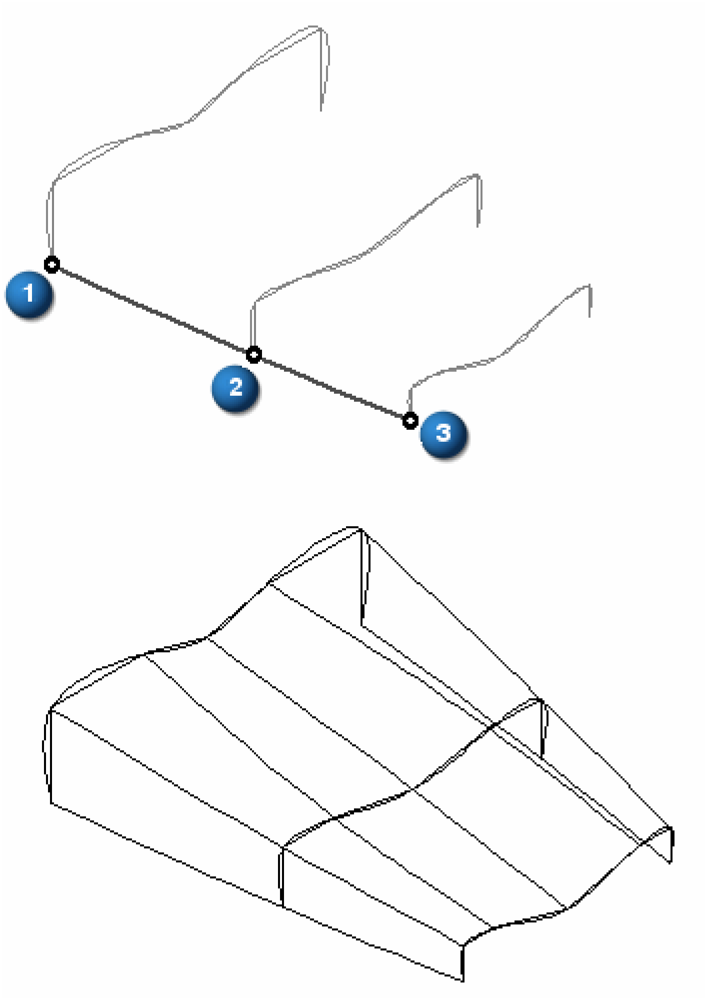
Lesson 43: Create an isocline curve

1. Choose **Surfacing** tab→**Curves** group→**Isocline** .
2. Select a reference plane or planar face.
3. Select a surface body.
4. In the **Angle** edit box, type an angle within the range of 0.00 to less than 90.00 degrees.
5. Click the **Accept** button.

> **Note**
>
> You can click the direction arrow (1) or press **F** to get another possible isocline curve result.



Chapter 44: Activity: Creating keypoint curves



Overview

In this activity, you learn to create a keypoint curve. A keypoint curve is a 3D curve. The curve is defined by connecting to keypoints from existing geometry.

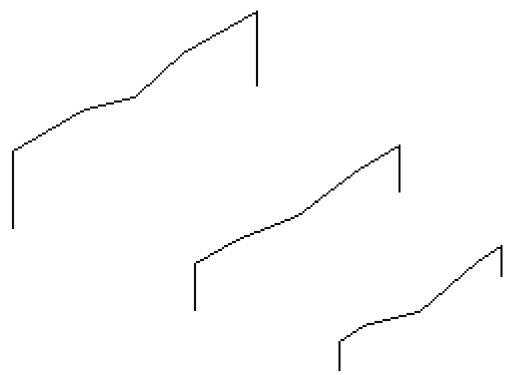
Objectives

After completing this activity you will be able to:

- Create a keypoint curve.
- Modify tangency vectors.

Chapter 45: Open the part file

- Open **surface lab 2-03.par**.

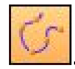


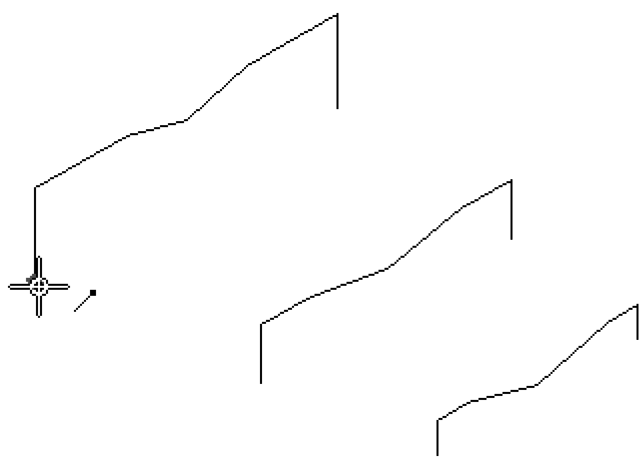
Note

The part file contains three sketches that you use to create keypoint curves. Each sketch has seven keypoints.

Chapter 46: Create a keypoint curve

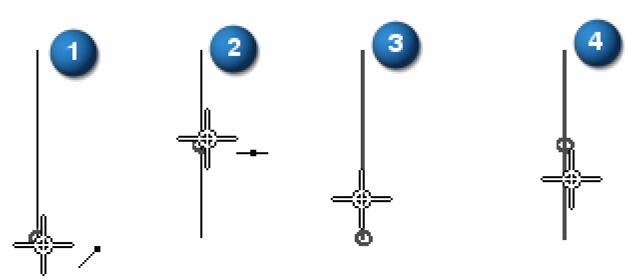
Create the first keypoint curve using geometry from Sketch A.

- Click **Surfacing** tab→**Curves** group→**Keypoint Curve** .
- Click the endpoint shown. Make sure the endpoint connect symbol displays.



Note

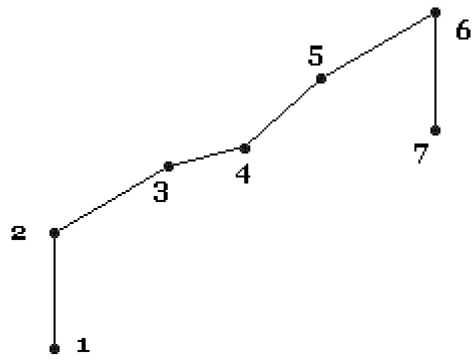
There are other keypoint select locations possible on a line. You can select endpoint (1), midpoint (2), line and endpoint (3) or line and midpoint (4). If you select a line and endpoint or a line and midpoint, the curve becomes tangent to the line at that point. You can modify the tangent vector. For this activity, only select endpoints.



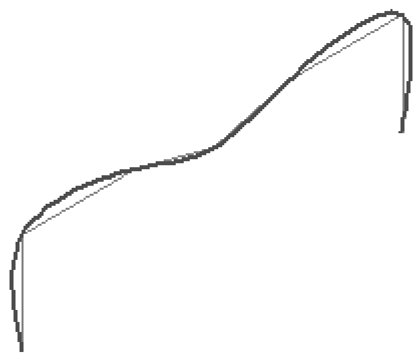
- To make it easier to only select endpoints, click the Keypoints button on the command bar.

  Select the **Endpoint** option .

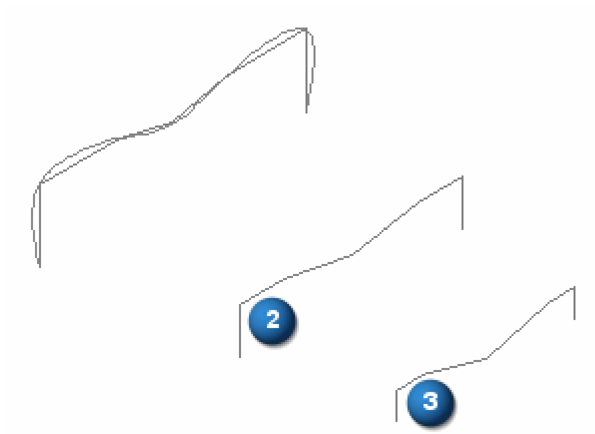
- Click the remaining endpoints in the following order.



- After clicking the last endpoint, click the **Accept** button, and then click **Finish**.

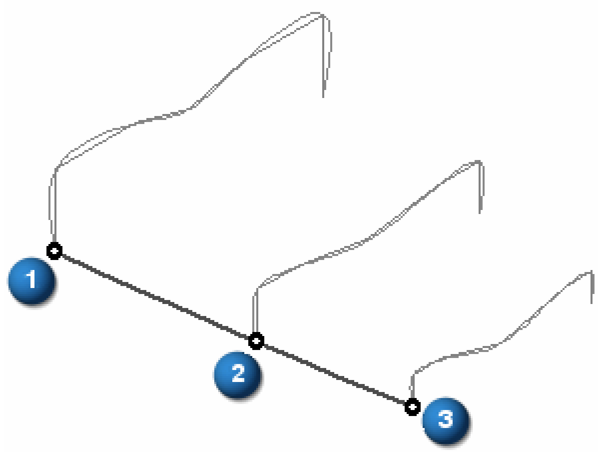


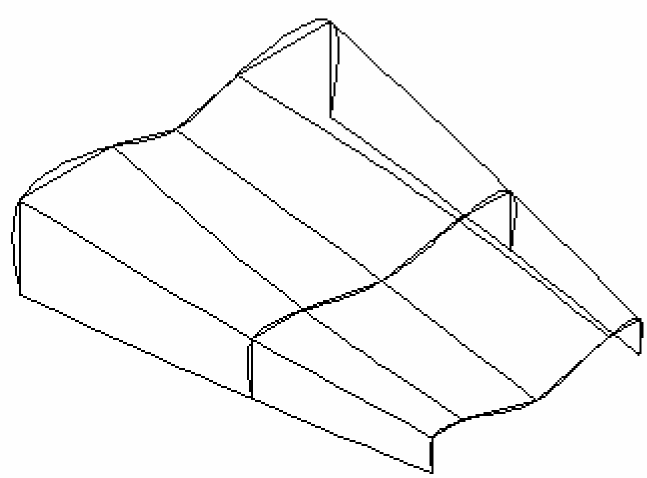
- Repeat the previous step to create keypoint curves using Sketches (2) and (3).



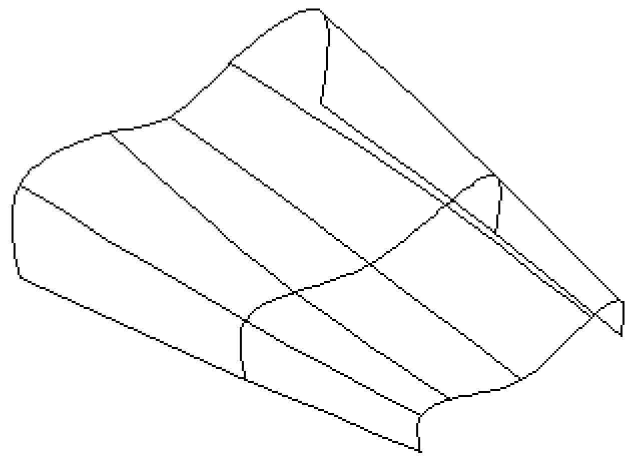
Chapter 47: Create keypoint curves between the sketches

Create seven keypoint curves between the sketches. The first curve is shown below and also the completed curves.





- Press Escape. Right-click in space, and then select **Hide All→Sketches**.



Note

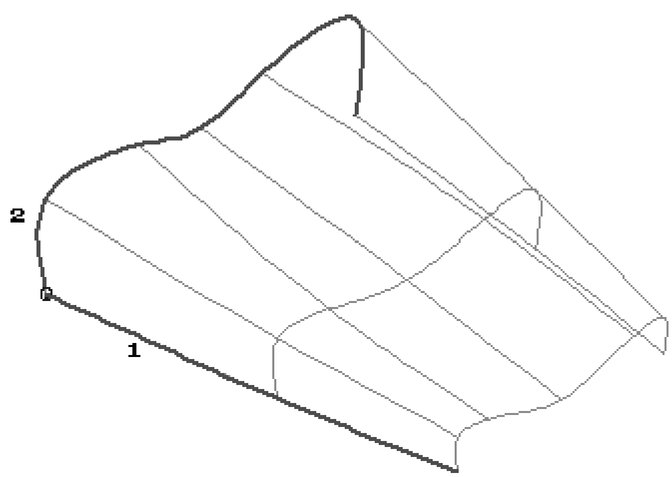
The keypoint curves are not connected to each other. They are only connected to the sketch elements. If you edit one of the sketches used to keypoint connect to, the keypoint curve changes with edits made to the sketch.

Chapter 48: Connect the keypoint curves with BlueDots

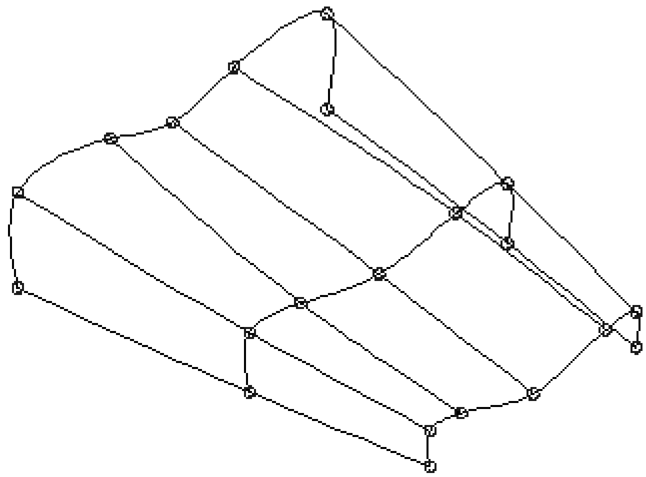
Connect the keypoint curves with BlueDots. Once a BlueDot is added, the creation history of the curves is lost.

- Click the **BlueDot** command .
- Click keypoint curve 1, and then click keypoint curve 2 as shown.

> **Note**
>
> Make sure no keypoints display when selecting the curves to BlueDot connect. Just click the curve away from any possible keypoint.



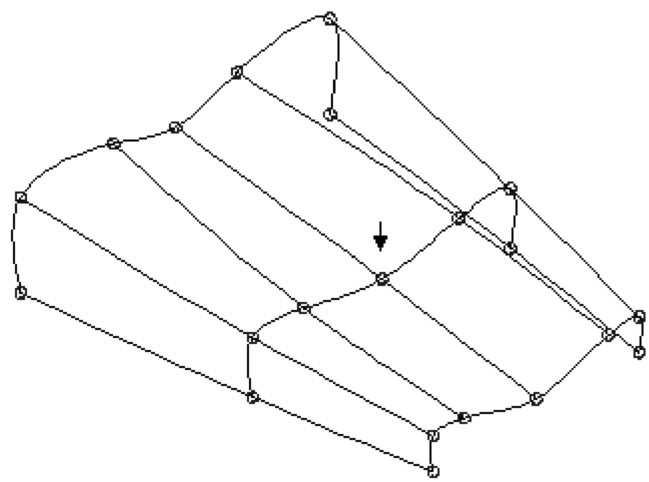
- Continue placing the remaining BlueDots. There are a total of 21 BlueDots. In case of a mistake, click the **Undo** command.



Chapter 49: Edit a BlueDot

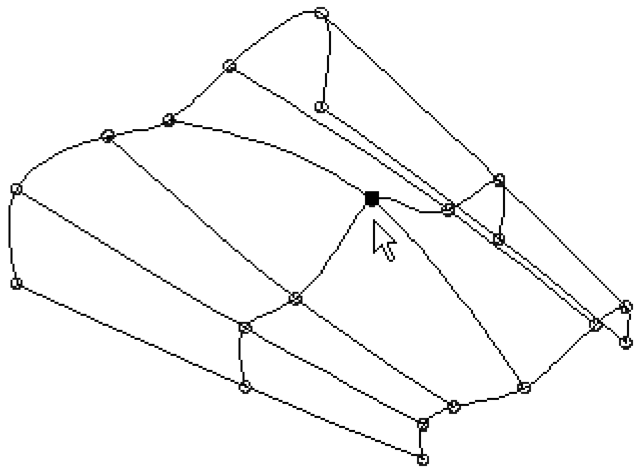
Edit a BlueDot and observe how the keypoint curves behave.

- Click the Select tool.
- Select the BlueDot shown.



- Click **Dynamic Edit** on the command bar.
- Click the Z-direction on the 3D triad.

- Drag the BlueDot up and notice how the two keypoint curves remain connected.



Note

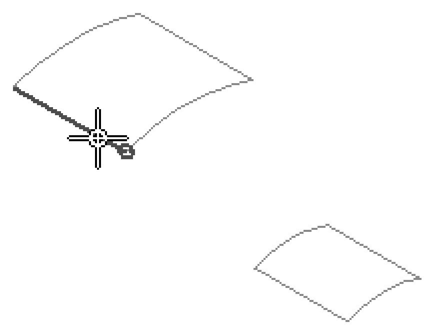
Notice on the BlueDot Edit command bar that curve edit fields are not available. Keypoint curves cannot be controlled with local or shape edits.

- Click the Select tool. Right-click in the graphics window. Choose **Hide All→BlueDots** and **Hide All→Curves**.

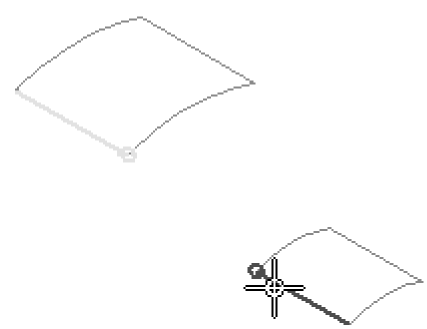
Chapter 50: Include tangency on keypoint curves

Create two keypoint curves that include a tangency vector.

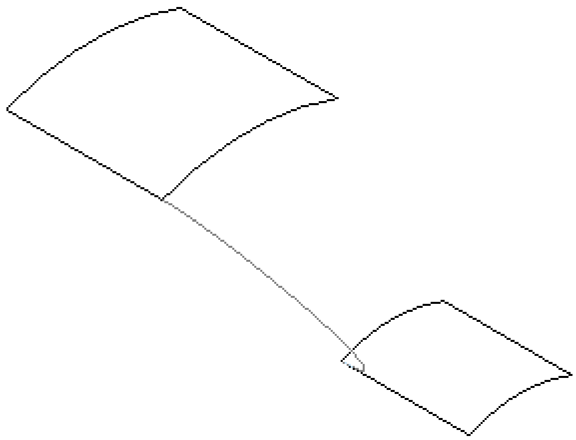
- Click the Select tool.
- In **PathFinder**, select the check boxes next to features Extrude 4 and Extrude 5.
- Create keypoint curves between these two surfaces that are tangent to an edge of each surface. Choose the **Keypoint Curve** command to begin.
- Select the surface edge as shown. Make sure the line and endpoint highlight.



- Select the surface edge as shown. Make sure the line and endpoint highlight.



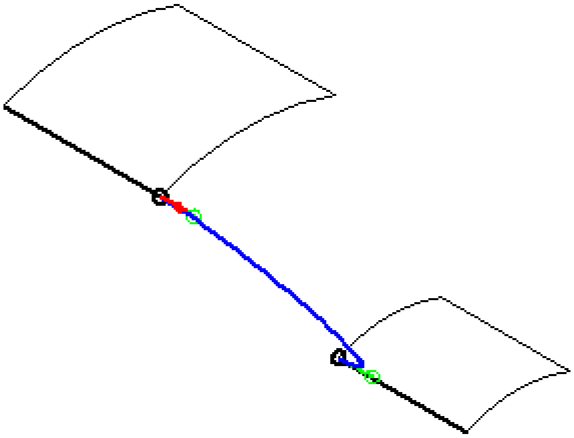
- Click the **Accept** button.



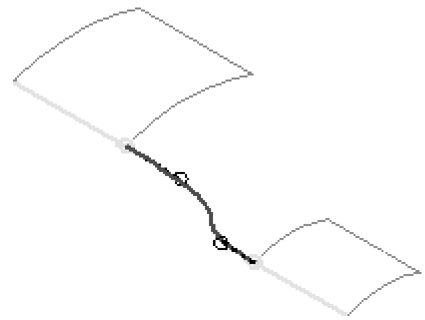
- Click the **End Conditions** Step.

- Set the start and end tangent conditions to **Tangent**.

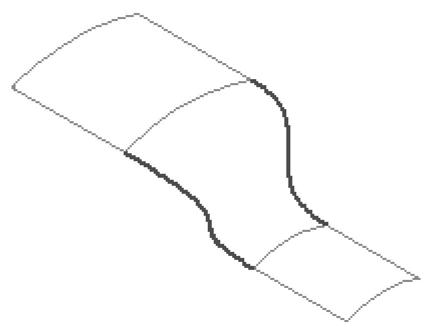
> **Note**
>
> Notice the green dot and line on each surface edge. These are the tangency vectors. The keypoint curve is tangent to the surface edge. By dynamically dragging the green dot, the curve changes shape while remaining tangent.



- Drag the tangency vectors as shown.



- Click **Preview** and then click **Finish**.
- Create another keypoint curve on the opposite edges of the surfaces.



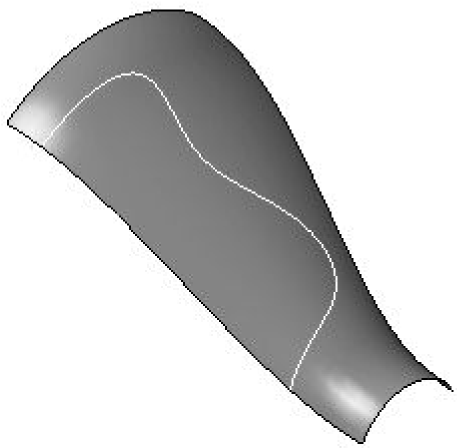
Note

In the next lesson, you learn that the two keypoint curves you just created are a step in the process of constructing a tangent transition surface between two surfaces. The activity is complete.

Chapter 51: Summary

In this activity you learned how to create and edit keypoint curves.

Chapter 52: Activity: Additional curve creation methods



Overview

In this activity, you will learn additional methods of creating curves. So far, you have learned to draw curves directly, point by point. Now you will learn to create curves indirectly, by combining inputs from existing curves and surfaces.

Objectives

After completing this activity you will know how to use:

- Intersection curves
- Cross curves
- Projected curves
- Contour curves
- Derived curves
- Split curves

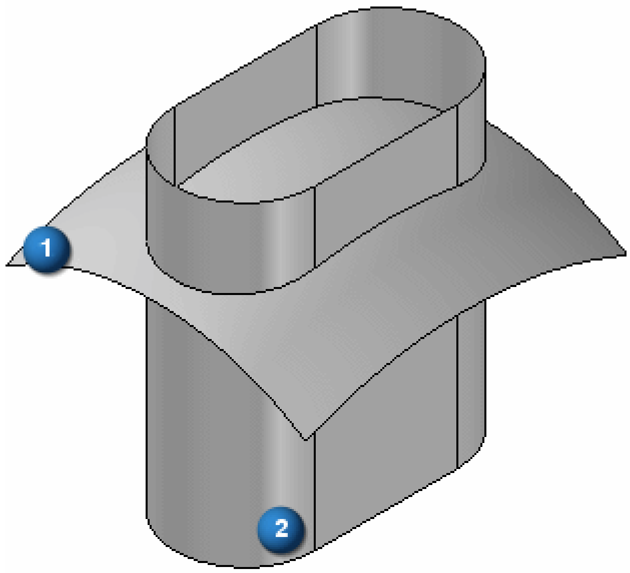
Chapter 53: Open the part file

- Open **surface lab 2-04.par**.

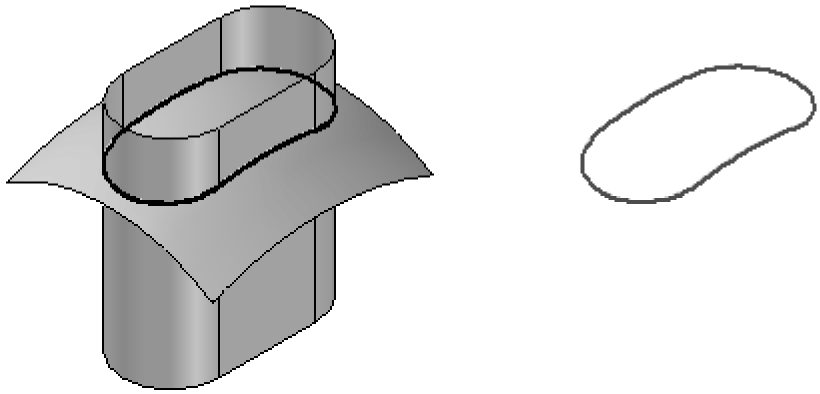
> **Note**
>
> In order to create curves in this activity, existing construction surfaces are needed. Since you have not yet learned how to create surfaces, the surfaces needed in the activity are provided.

Chapter 54: Create an Intersection curve

- In **PathFinder**, select the check boxes next to features BlueSurf 1 and Extrude 1.
- Right-click in the graphics window and choose **Hide All**→**Reference Planes**.



- Create a curve where construction surfaces (1) and (2) intersect. Choose **Surfacing** tab→**Curves** group→**Intersection** .
- On the command bar, set the Select filter to **Feature**.
- Select surface (1) and **Accept**.
- Select surface (2) and **Accept**.
- Click **Finish**.



Note

In **PathFinder**, notice the intersection curve just created is named Intersection 2.

Note

The intersection curve is associative to the two input surfaces it was created from. These surfaces are the parents of the intersection curve. If a parent is edited, the intersection curve updates automatically.

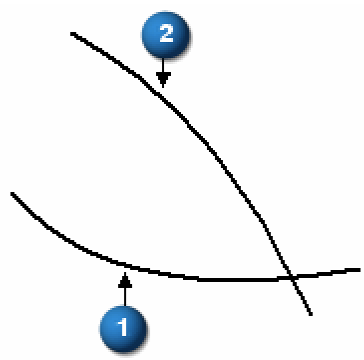
In the next lesson you learn how you can use the intersection curve in surface editing operations.

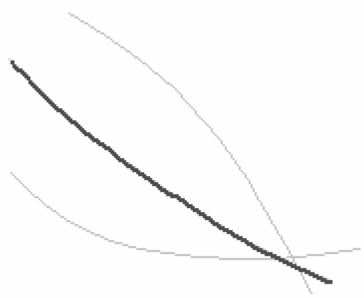
- Hide the following features in **PathFinder**: *BlueSurf1*, *Extrude1* and *Intersection 2*

Chapter 55: Create a cross curve

A cross curve is an intersection curve that is created with the theoretical extruded surfaces resulting from the two input curves or analytics.

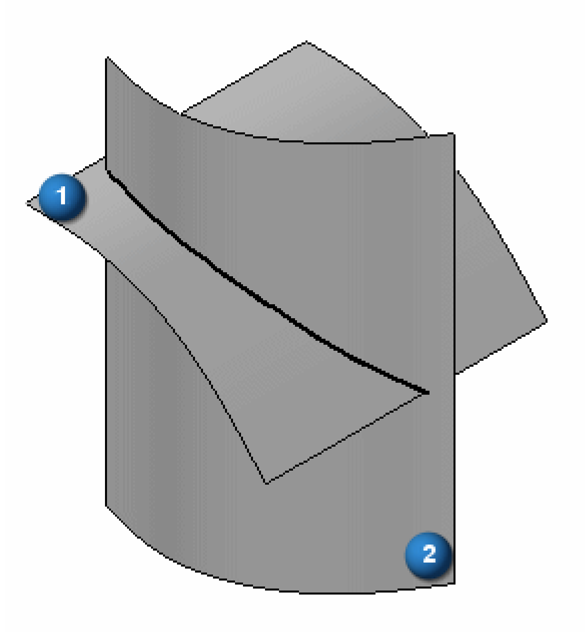
- In **PathFinder**, show the following sketches: *Sketch 2a* and *Sketch 2b*.
- Select **Surfacing** tab→**Curves** group→**Cross** .
- Click sketch (1) and then click **Accept**. Click sketch (2) and then click **Accept**.





- Click **Finish**.

  The cross curve is the result of the intersection of the two theoretical extruded surfaces (1) and (2).



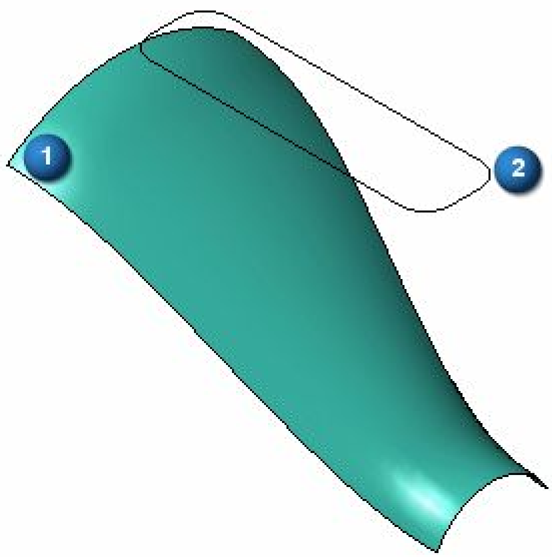
> **Note**
>
> The **Cross** command eliminates the need to construct extruded surfaces from curves and then find the intersection between the two surfaces.

- Hide the sketch curves and cross curve: *Sketch 2a*,*Sketch 2b* and *Cross Curve 8*.

Chapter 56: Project a curve

The **Project** command projects a curve onto a surface.

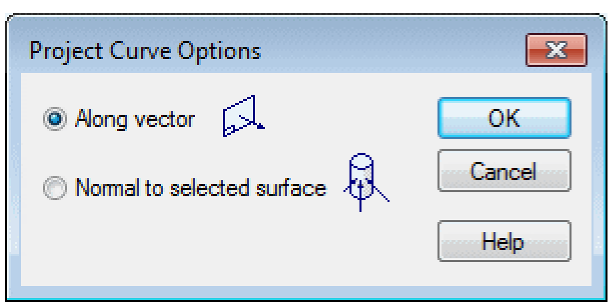
- Show the following features in **PathFinder**: *BlueSurf 2* and *Sketch 3c*.



- Project curve (2) onto surface (1). Choose **Surfacing** tab→**Curves** group→**Project** .

- Click the **Options** button on the command bar.

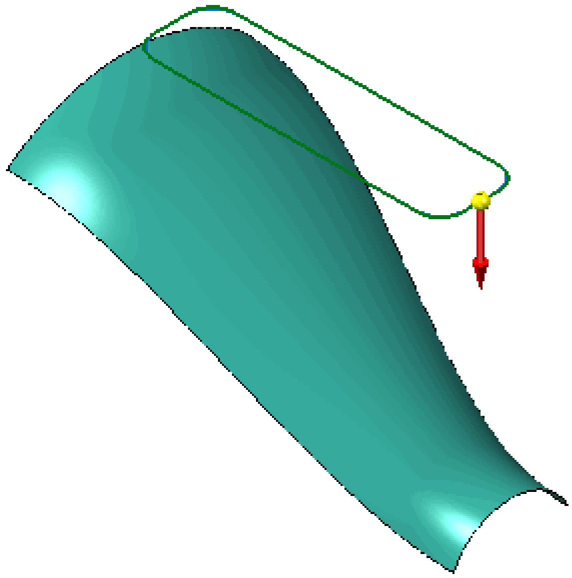
- The default option is **Along vector**. This projects a curve along its normal vector. Click **OK**.



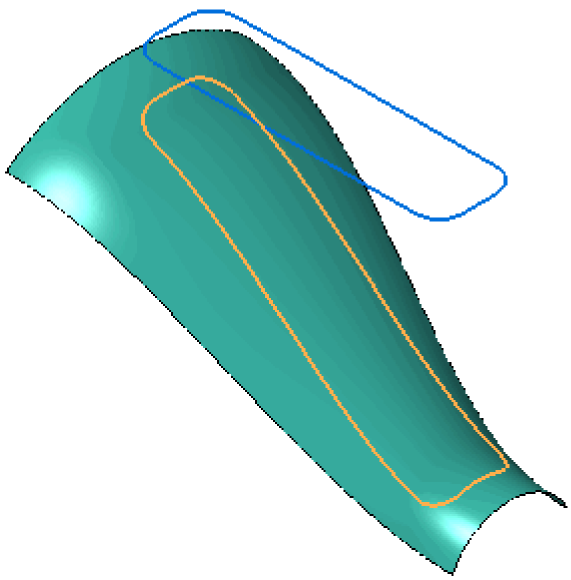
- Select curve (2) and then click **Accept**.

- Select surface (1) and then click **Accept**.

- For the direction vector, point the direction arrow down as shown.



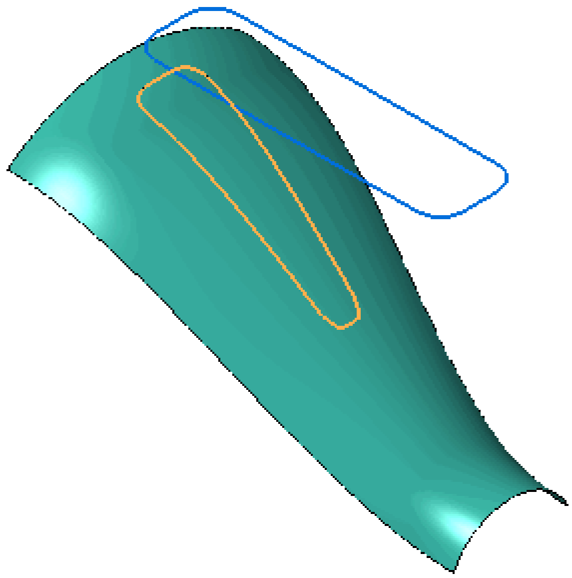
- Click **Finish**.



- Click the Select tool. In **PathFinder**, select the Projection feature and press **Delete**.
- Project the curve normal to the surface. Select the **Project** command again.
- Click the **Options** button.

- Select the **Normal to selected surface** option and click **OK**.

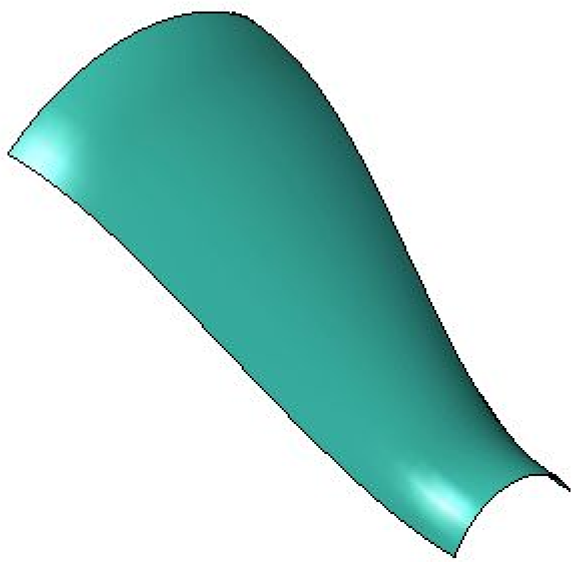
- Click the curve and then click **Accept**.
- Click the surface and then click **Accept**. Click **Finish**. Notice the different results



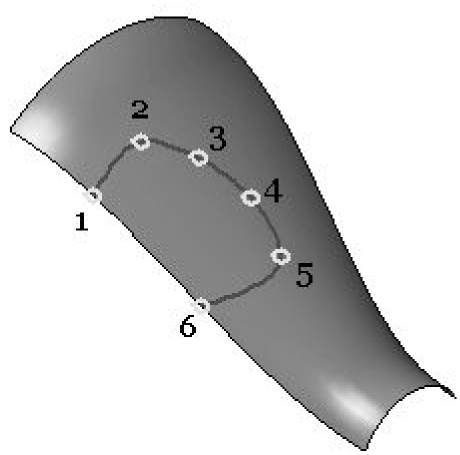
- Hide the features *BlueSurf 2* , *Sketch 3c* and *Projection 9*.

Chapter 57: Create a contour curve

- In **PathFinder**, show the feature *BlueSurf 2*.



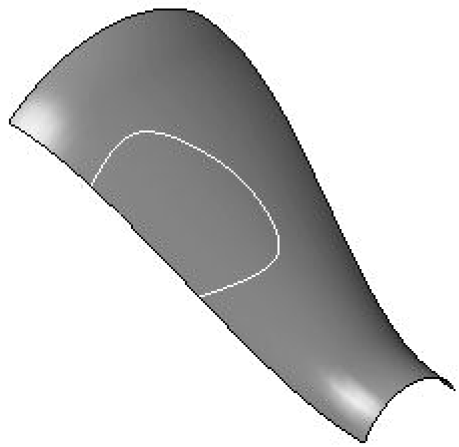
- Select **Surfacing** tab→**Curves** group→**Contour** .
- Click the surface and then click **Accept**.
- Click the surface to place the six contour curve points approximately as shown. Points 1 and 6 are on the edge. Points 2 through 5 are on the face.



Note

To insert points on an edge, set the command bar **Select** box to **Edges**. To insert points on the face, set the **Select** box to **Face**.

- After placing the last point , click **Accept**. Click **Finish**.

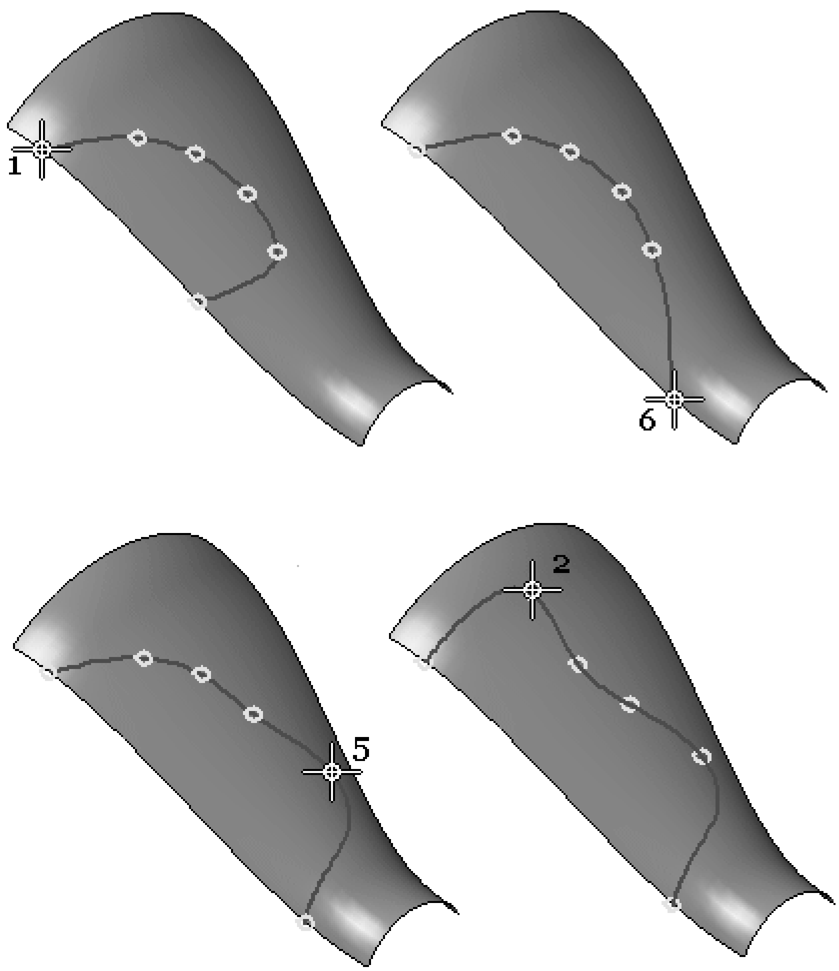


Chapter 58: Edit the shape of the contour curve

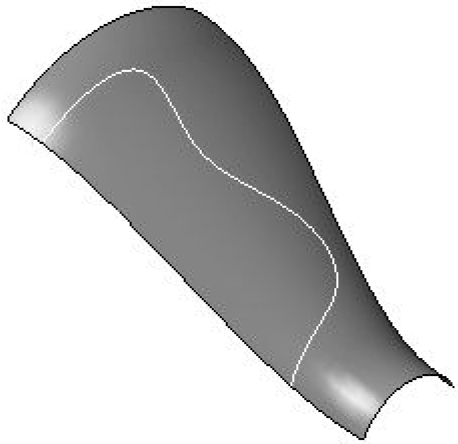
- Click the Select tool.
- In **PathFinder**, right-click the contour curve feature and select **Edit Definition**.

> **Note**
>
> The shape can be edited while creating the contour curve.

- Click **Draw Points** Step .
- Click the points shown and drag to edit the shape approximately as shown. Points 1 and 6 will remain attached to the edge. Points 2–5 can be moved anywhere along the face.



- Click **Accept** and then click **Finish**.

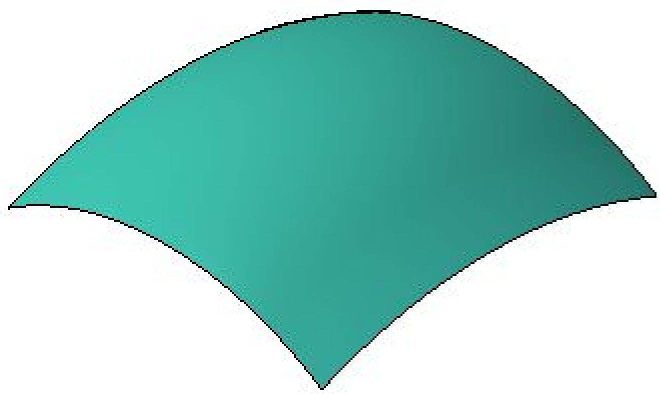


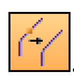
- Hide the two features *BlueSurf 2* and *Contour Curve 2*.

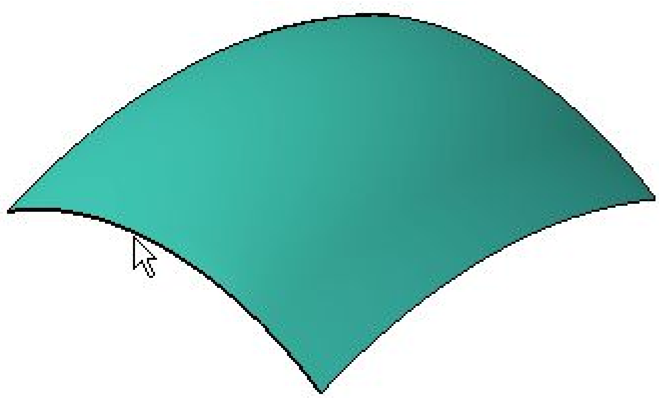
Chapter 59: Use the derived curve and split curve creation methods

You can create derived curves from the four edges of a surface. Then you can split the derived curves that would be used to create additional surfaces. No surfaces are created in this activity.

- In **PathFinder**, show *BlueSurf 1*.

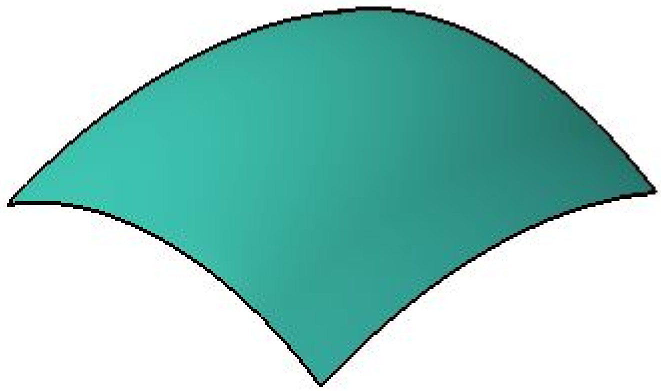


- Select **Surfacing** tab→**Curves** group→**Derived** .
- Click the edge shown, and then click **Accept**.

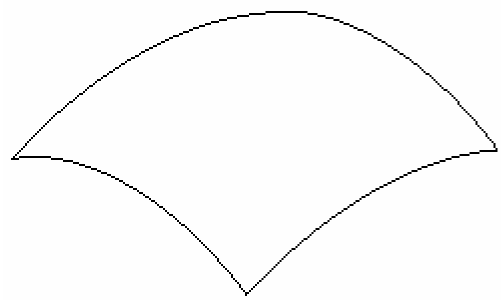


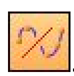
- Click **Finish**.

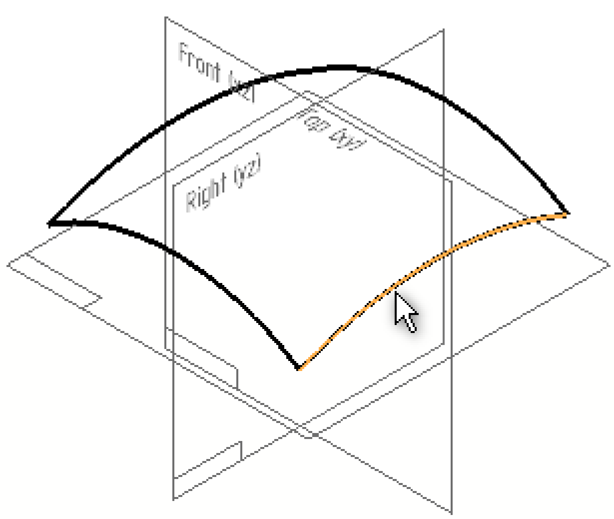
- Repeat the above step to create derived edges for the three remaining surface edges.



- Click the Select tool. Hide the surface.

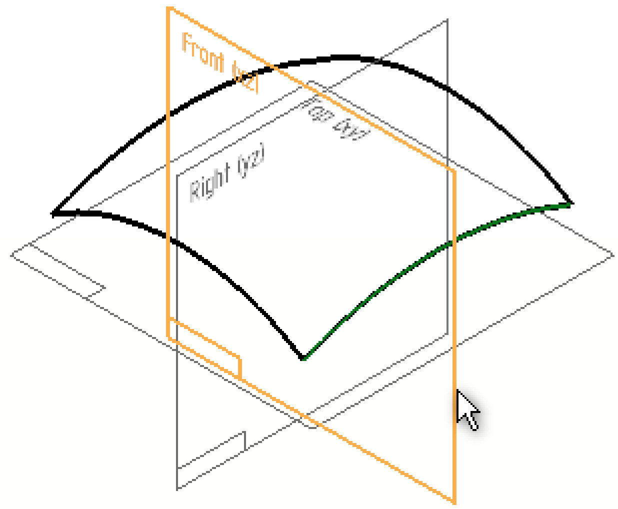


- In **PathFinder**, show the **Base Reference Planes**.
- Select **Surfacing** tab→**Curves** group→**Split** .
- Select the derived curve shown, and then click **Accept**.

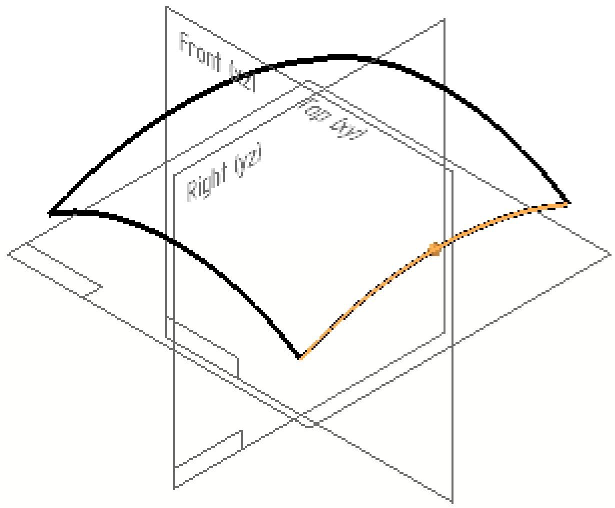


- On the **Split Curve** command bar, set the Select filter to **Body**.

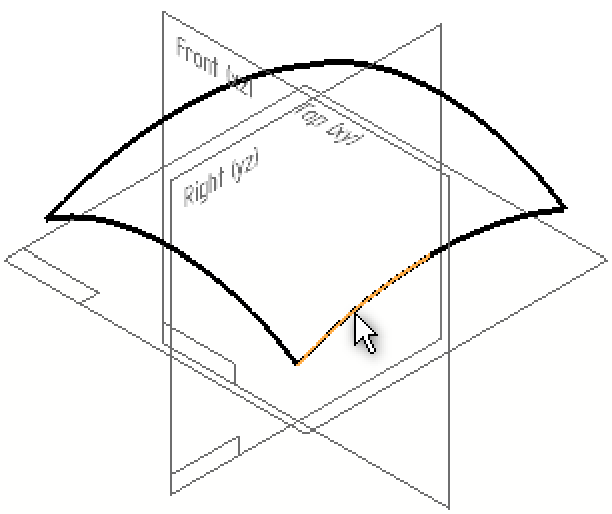
- Click the reference plane shown.

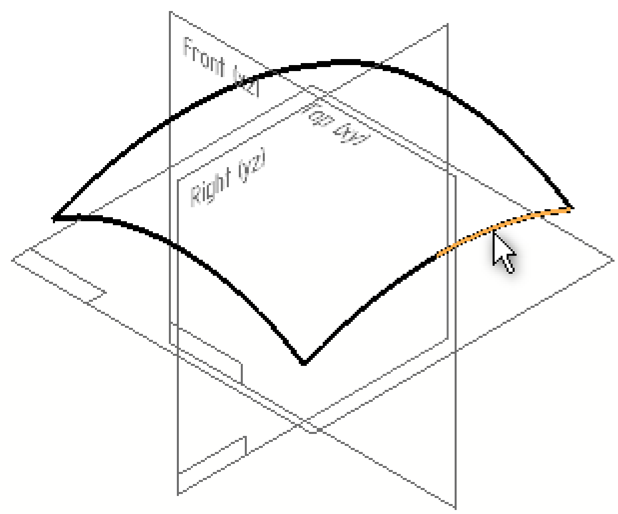


- Click **Accept**, and then click **Finish**.



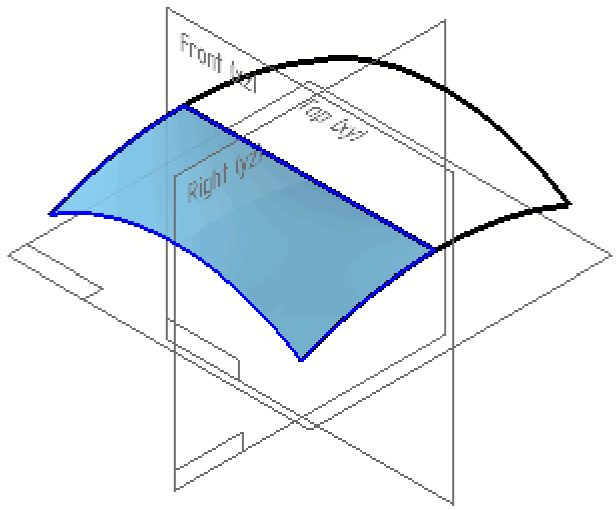
Notice the original derived curve is now split and there are two new curves that can be used for surfacing operations.





- Repeat the previous step to split the remaining three derived curves.

  The following image shows a new surface created using the split curves.



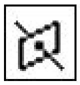
> **Note**
>
> The BlueSurf construction method was used to create the surface above. You will learn how to create BlueSurfs in an upcoming lesson.

- The activity is complete. Exit and Save the file.

Chapter 60: Summary

In this activity you learned how to create curves using several additional methods.

Lesson 44: Pierce points

A *pierce point*  is the point of intersection between a profile element and the active sketch plane.

> **Example**
>
> You can use a connect relationship to position the element you are drawing to where a profile element on another reference plane pierces the current profile plane.

Pierce points:

- Are extremely valuable in aligning curves.
- Recognize where a 3D curve, a sketch, or an edge passes through (pierces) the active profile plane.
- Connect geometry to curves intersecting a profile plane.
- Are useful for creating guide paths for BlueSurf and Sweep operations.

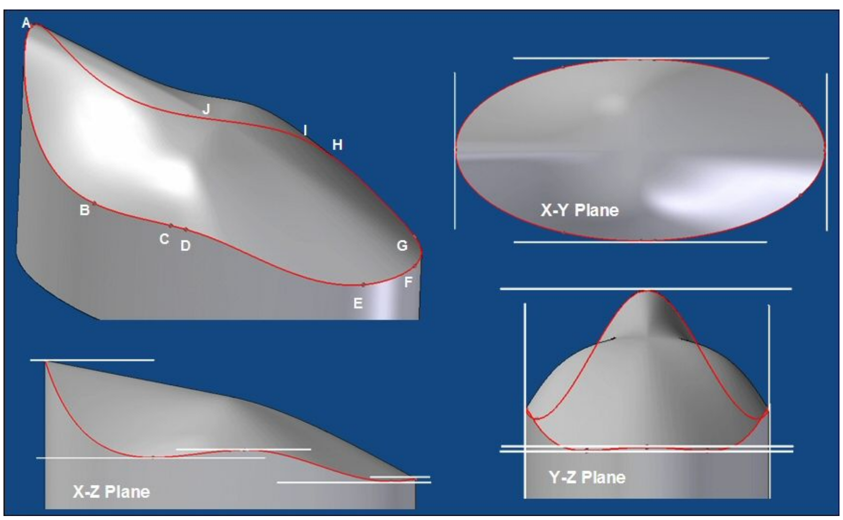
Lesson 45: Silhouette points

Silhouette points are keypoints that occur on an arc, circle, or ellipse.

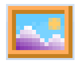
> **Example**
>
> When you draw a new line, you can touch the silhouette point on a circle. When you click, the new line is connected to the silhouette point on the existing circle.

Silhouette points:

- Are defined relative to the horizontal and vertical axis of the draft sheet or profile/sketch plane.
- Constitute any point where a plane parallel to the base reference plane passes tangent to a given curve.
- Can be used to connect dimensions.
- Act like endpoints.



Lesson 46: Insert Image command

Use the **Insert Image** command  to insert an image into a document. Inserted images can contribute to your modeling workflow in several ways. For example, you can sketch geometry over an image to create features based on it. Or you can use an image as a label or decal on a plane or planar face in the model.

You can insert image files of the following types:

- Windows Bitmap (*.bmp)
- Portable Network Graphics (*.png)
- Graphics Interchange Format (*.gif)
- Joint Photographic Experts Group (JPEG) (*.jpg, *.jpeg, *.jpe)
- Tagged Image File Format (TIFF) (*.tif, *.tiff)

> **Note**
>
> JPEG image files must be in RGB format. CMYK format is not supported.

You can either link or embed the image, and you can control its display, including height, width, and aspect ratio.

In a draft document, another way to insert an image is to drag it from your desktop onto the working sheet, or copy and paste it from an external application, such as Microsoft Paint. Pictures inserted in this manner are created as image objects rather than as symbols.

Lesson 47: Points, curves (and surfaces) as construction elements

For some model types, you may not use the solid modeling commands until very late in the modeling process. Complex, freeform parts often require that you begin the modeling process by defining points and curves, which are used to define and control the surfaces that comprise the model. Surfaces are then generated, and in the final steps, the surfaces are stitched together to form a solid model.

- These construction element types can be created within Part, Sheet Metal, and Profile or Sketch environments:
  - Points
  - Curves
  - Surfaces

Construction elements that drive other features have a parent-child relationship with the features they drive. If you delete a construction element that is a parent to another feature, you can invalidate the other feature.

Displaying construction elements

- Use the **View** tab→**Show** group→**Construction Display**  command to control display.
- Construction elements are listed in the **Feature PathFinder**.

> **Note**
>
> When hidden, a construction element entry in **PathFinder** changes to indicate that it is hidden.

- Color control is available for construction elements using either of the following:
  - The **View** tab→**Color Manager** command
  - The **Colors** tab in the Solid Edge **Options** dialog box.
- Specific display considerations:
  - Construction elements used to create new features are not consumed by the new feature, and are hidden by default.
  - It can be useful to hide the design body while you are working with construction surfaces. Under **View** tab→**Show** group→**Construction Display**, use either the **Show Design Body** or **Hide Design Body** commands.

Methods of creating construction elements

- Use existing geometry on the model. Intersection curves, Keypoint curves, Derived curves, Project curves, Split curves, and relevant point creation commands can be used.
- Create construction elements from scratch using the Solid Edge construction surface creation commands (extruded, revolved, and swept surfaces).
- Use **Curve By Table** command to generate a curve based on input points.
- Use an external file. For example, you can create a helix curve using coordinate data in a spreadsheet.
- Import them from another CAD system. For example, you can import surfaces and solids from a third-party CAD system.
- Generate them as a part copy from another Solid Edge part. For example, you can create construction geometry using the **Part Copy** command on the Insert menu.

Using construction elements

- Points can be used in several ways:
  - To create other features:
    - Use a construction point or curve as a path or cross section for lofted and swept features.
    - Use the **Intersection** command to create cross sections for lofted features.
  - To define the extents of another feature:
    - Use keypoints of construction curves to define the extent for a feature.
    - Use the **Intersection** command to create associative points as input to define the extents of a feature
- Curves can be used in two distinct ways:
  - Curves can be used to create other features, such as:
    - Paths and cross sections for lofted and swept features using Intersection curves, Keypoint curves and Derived curves.
    - Profiles for profile-based features using the **Project** (useful for creating embossed text on a curved surface) and **Split** (divide one into multiple curves to create a normal protrusion) commands:
    - Construction surfaces—the **Split** command can divide one construction surface into multiple curves to create a surface by boundary.
  - Use a construction curve as input to the reference plane **Normal To Curve** command.
- Surfaces can be used as well; creation methods are covered in other training modules. Some general uses of surfaces are:

- To define the projection extents when extruding a feature. For example, a construction surface can be used as input during the **Extent** step when constructing a protrusion.
- To replace existing part faces.
- To divide a part into multiple parts.
- To create a new surface or solid by stitching together separate surfaces. Use the **Offset Surfaces** command to offset a new surface.
- To repair a model imported from a third-party CAD system.
- Construction surfaces are commonly used as projection extents when extruding a feature.

Lesson 48: Lesson review

Answer the following questions:

1. How do you get tangency control on a keypoint curve?
2. Intersection curves are considered to be associative. What does this mean?
3. From what types of elements can a derived curve be formed?
4. Generally, why would you define construction elements (curves and points) as a first step?

Lesson 49: Lesson summary

- You can create various curves indirectly from existing curves and surfaces. These curves are controlled by the parent curves and surfaces. As the parents change, so do the indirect curves.
- Surface shapes are directly tied to the curves defining those surfaces. Therefore, the control of curves is crucial in modifying surface topology.
- Pierce and silhouette points can assist in connecting curves to off-plane geometry.

Lesson 50: Surface creation

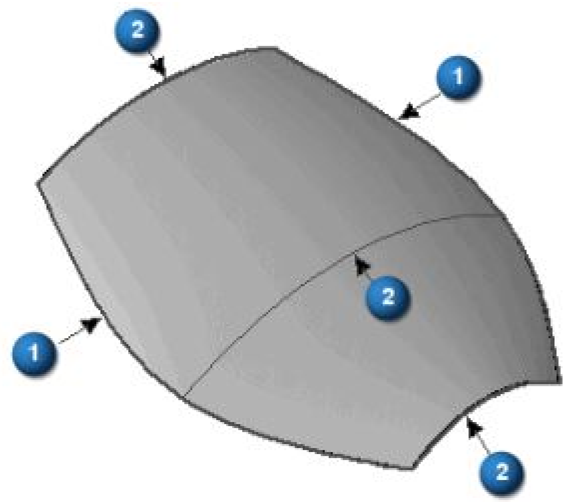
Objectives

After completing this lesson, you will be able to:

- Create simple surfaces.
- Create a BlueSurf.
- Edit a BlueSurf.
- Create a Bounded surface.

Overview of surfaces

A surface is a 3D element that is controlled by curves. Surfaces have no thickness and therefore can be visualized as a thin sheet. The complexity of a surface is directly proportional to the number of curves used to define it. A small, underlying curve set produces a relatively simple surface, while a complex face consists of a large number of curves. In Solid Edge modeling, a surface consists of cross-sections and guide curves. Guide curves may be preexisting or interpolated from the cross-section elements.



(1)	Guide curves
(2)	Cross Section curves

Curves form the mathematical basis of a surface. As your understanding of how to control curves increases, your mastery of surfaces grows.

There are two basic ways curve manipulation affect an associated surface:

1. Editing underlying cross-sections and guide curves directly modifies surface shape.

2. A surface can be trimmed and extended using curves and edges.

Once its shape is finalized, a surface can be used in the creation of additional faces via the following commands (covered in Lesson 5):

- Offset

- Copy

- Mirror

A surface also can be stitched together with other faces to form a solid, or it can include rounds between adjoining surfaces.

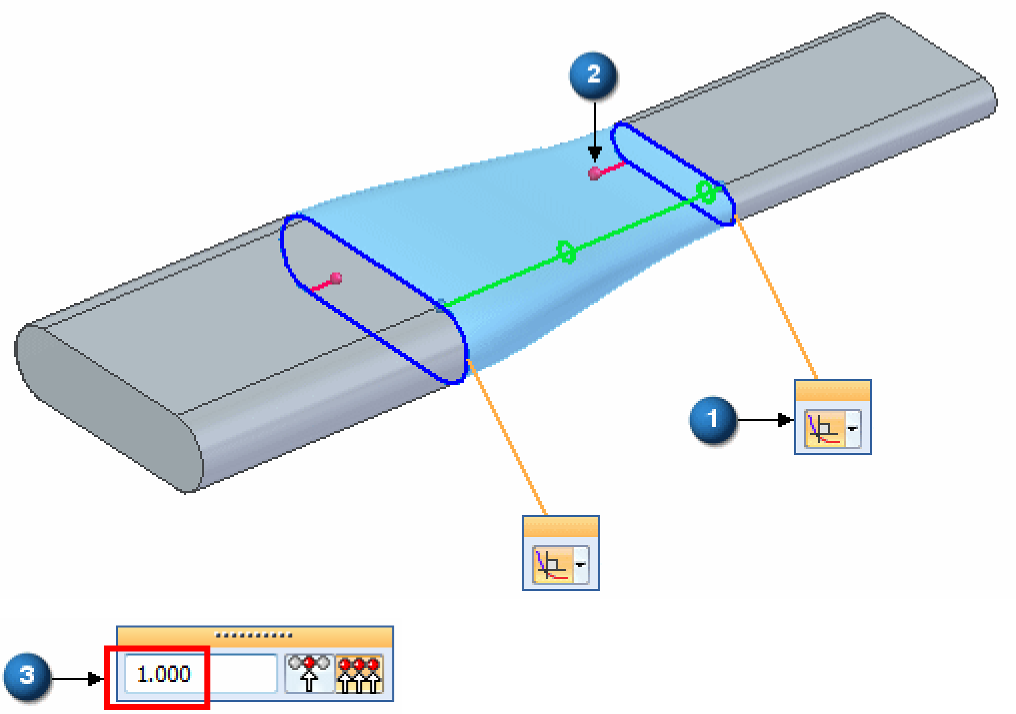
Tangency control handles

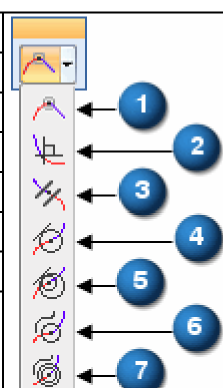
Use the tangency control handles to specify the tangency condition at curve and surface boundaries. These handles provide a direct method to manipulate the shape of a surface by specifying different tangency conditions.

The **BlueSurf**, **Bounded**, **Redefine**, **Lofted**, and **Keypoint Curve** commands support various continuity and tangency conditions. The tangency control handles provide a common method for each of these commands to control the end conditions.

The handle has two elements:

- Tangency control handle (1) – Drop-list to control the type of tangent condition
- Tangent magnitude handle (2) – Drag handle which also supports a magnitude dynamic input box (3).

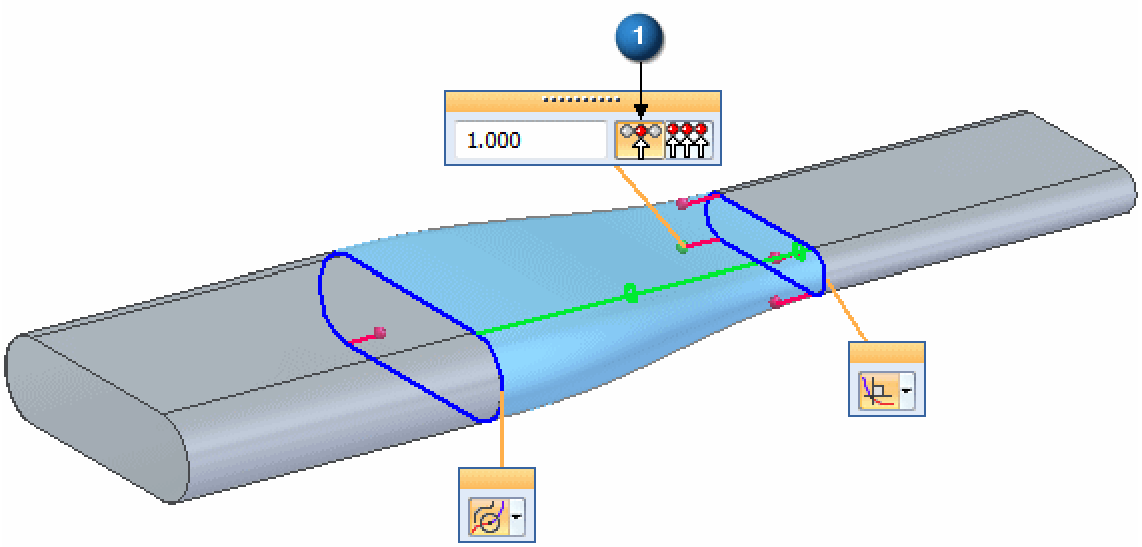


Tangent conditions	
natural (1)	
normal to section (2)	
parallel to section (3)	
tangent continuous (4)	
alternate tangent continuous (5)	
curvature continuous (6)	
alternate curvature continuous (7)	

There are two tangent magnitude handle options that are available when creating and editing a bluesurf surface, loft protrusion, or keypoint curve.

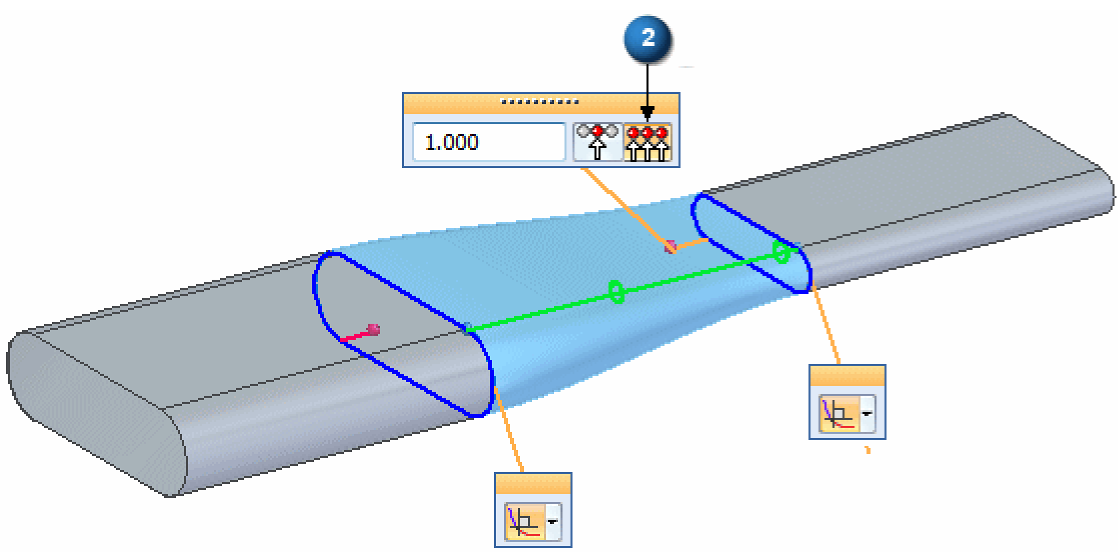
- Multiple tangent magnitudes (1)

  This option exposes all magnitude handles for the section. You can edit any magnitude handle and the remaining handles are unchanged.



- Single tangent magnitude (2)

  This option exposes a common magnitude handle for the section. You edit a common magnitude handle and the remaining section magnitude handles have the same value.



You can drag the handles or type a magnitude value in the dynamic input box.

Tangency handle display

Tangency handle display can be turned on or off in each command.

- **BlueSurf** command — Use the on or off button in the **BlueSurf-Surface Visualization** dialog box.

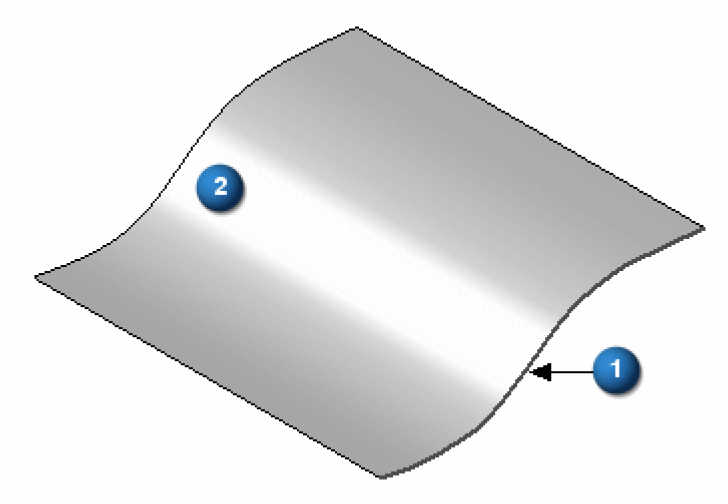
- **Bounded** command — Use the **Hide Tangency Control Handles** button  on the **Bounded** command bar. The button appears while in the **Face Tangency** step.

- **Redefine** command — Use the **Hide Tangency Control Handles** button  on the **Redefine** command bar. The button appears while in the Tuning step.

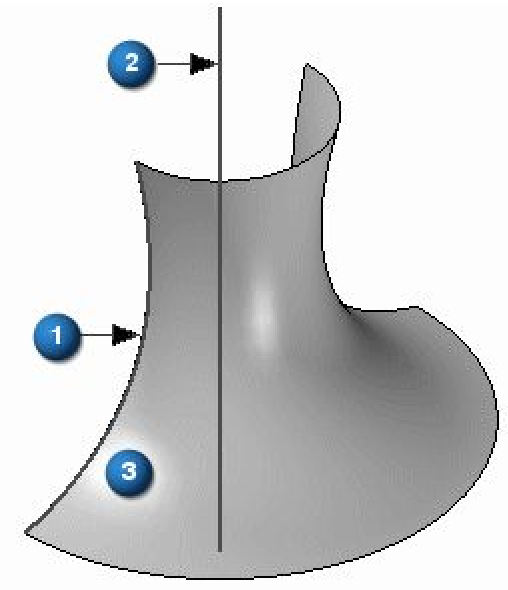
- **Keypoint Curve** command — Use the **Hide Tangency Control Handles** button  on the **Keypoint Curve** command bar. The button appears while in the **End Conditions** step.

Creating a simple surface

The two most basic surface creation techniques utilize the **Extruded**

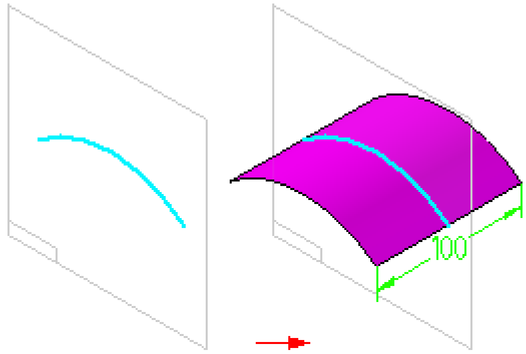


and **Revolved** surface commands.

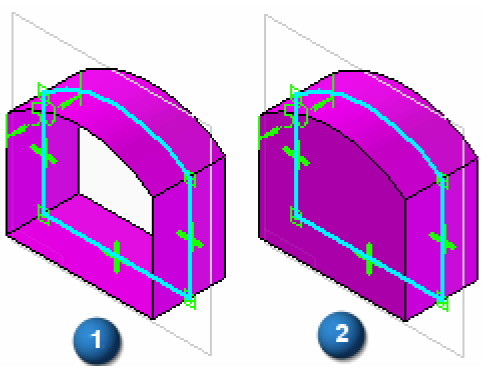


Extruded command

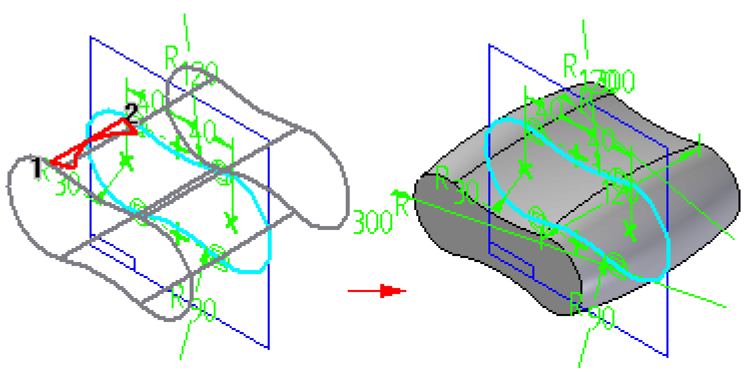
Use the **Extruded** surface command to create a construction surface by projecting a profile along a straight line. Options are available to control the extents of the surface.



When you create an extruded surface using a closed profile, you can use the **Open Ends** and **Close Ends** options on the command bar to specify whether the ends of the surface are open (1) or closed (2). When you set the **Close Ends** option, planar faces are added to the ends of the feature to create a closed volume.

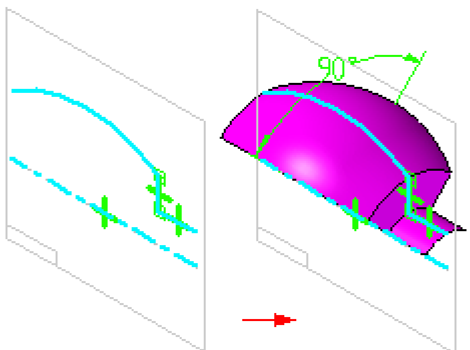


When constructing extruded surface features, you can also apply draft angle or crowning to the faces on the feature that are defined by profile elements. For more information, see the Applying Draft Angle and Crowning to Features Help topic.

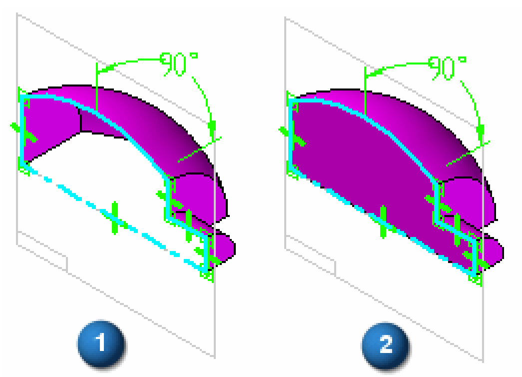


Revolved command

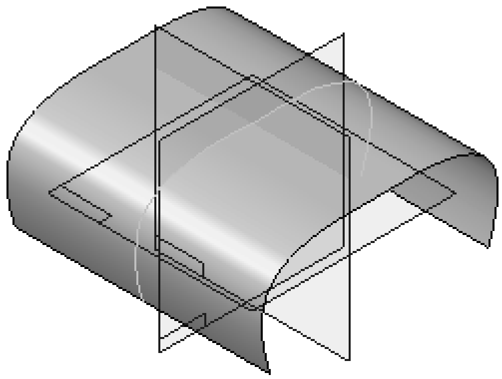
Use the **Revolved** surface command  to create a construction surface by revolving a profile around an axis of revolution.



When you create a revolved surface using a closed profile that is revolved less than 360 degrees, you can use the **Open Ends** and **Close Ends** options on the command bar to specify whether the ends of the surface are open (1) or closed (2). When you set the **Close Ends** option, planar faces are added to the ends of the feature to create a closed volume.



Activity: Creating and editing simple surfaces



In this activity, you will learn to create and edit simple surfaces. You will use sketches in a training file to create an extruded surface and a revolved surface. After completing the surface, you will edit the sketch curve to observe the surface shape changes.

Objectives

After completing this activity you will be able to:

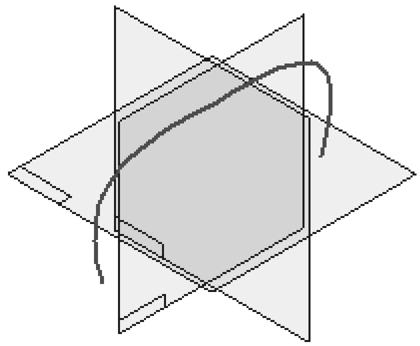
- Create and edit an extruded surface.
- Create and edit a revolved surface.

Open part file

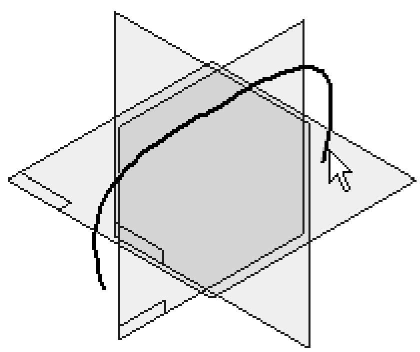
- Open **surface lab 3-01.par**.

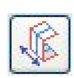
Create an extruded surface

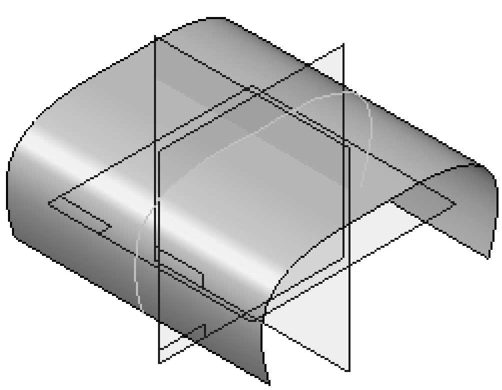
- In **PathFinder**, show *Sketch A*.



- Select **Surfacing** tab→**Surfaces** group→**Extruded** .
- On the command bar, from the Create-From list, select *Select from Sketch*.
- Click the sketch curve shown and then click **Accept**.

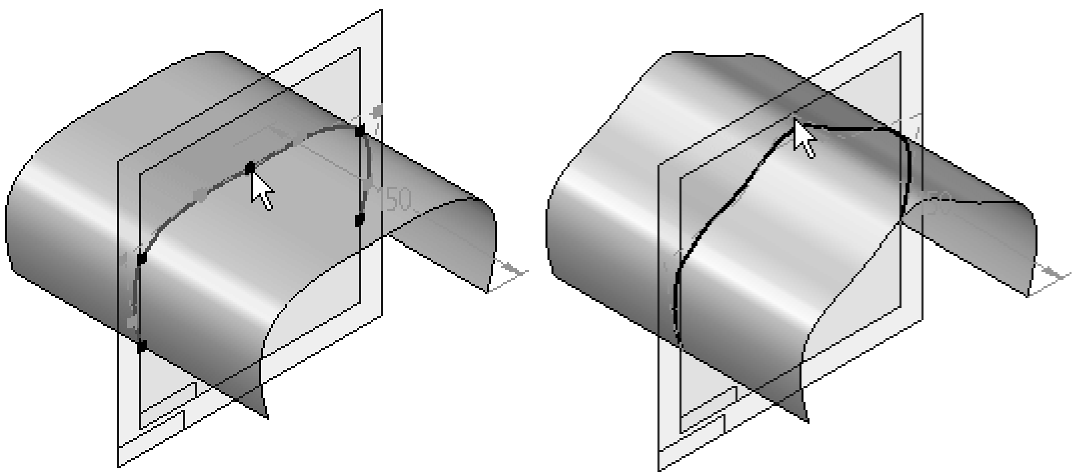


- Click the **Symmetric Extent** button  and type **150** in the **Distance** box.
- Click **Finish**.



Modify the shape of the extruded surface

- Click the Select tool.
- In **PathFinder**, hide **Base Reference Planes**.
- Select the extruded surface and then click the **Dynamic Edit** button  on the command bar.
- Click the sketch curve. Use the curve **Local Edit** option , and drag the edit point shown. Drag the edit point around slightly and notice how the surface shape changes.

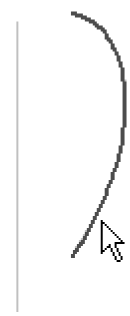


- Click the Select Tool to end the dynamic edit, and then press Escape.
- In **PathFinder**, hide the features *Sketch A* and *Extrude 3*.

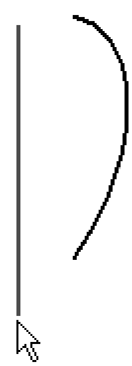
Create a revolved surface

> **Note**
>
> The **Revolved** surface command has the same steps as the part **Revolve** command.

- Show *Sketch B*.
- Select **Surfacing** tab→**Surfaces** group→**Revolved** .
- On the command bar, from the Create-From list, select **Select from Sketch**.
- Select the sketch curve shown and then click **Accept** on the command bar.

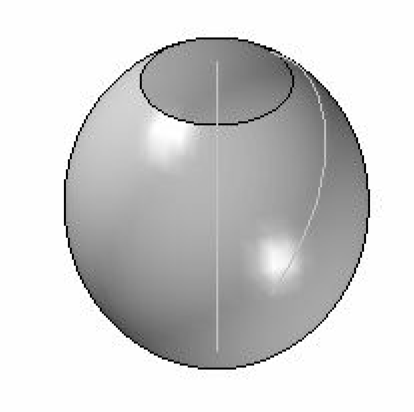


- Notice on the **Revolved** surface command bar that the next step is to define the axis of revolution. Click the line as shown.

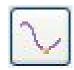


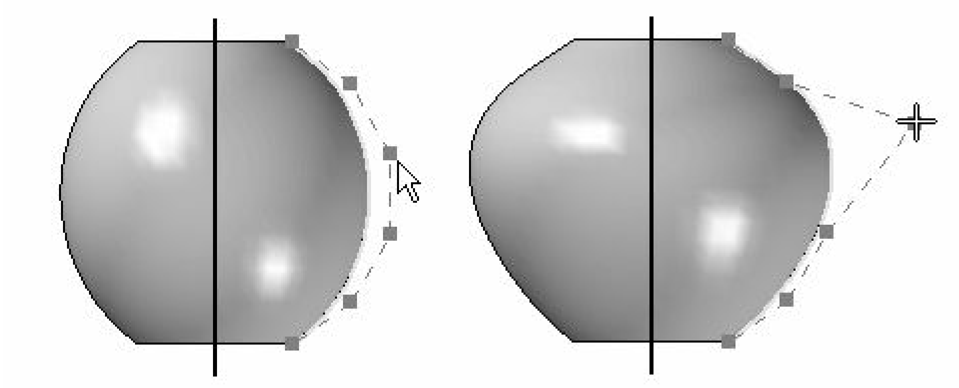
- For the extent step, click the revolve 360° button .

- Click **Finish**.



Edit the shape of the revolved surface

- Click the Select tool.
- Press Ctrl+R to rotate the view to a right view.
- Select the revolved surface and then click the **Dynamic Edit** button .
- Select the sketch curve. Use the curve **Local Edit**  option and drag the edit point shown. Drag the control vertex point around slightly and notice how the surface shape changes.



- Drag the control vertices around to come up with your own surface shape.
- This concludes the activity.

Summary

In this activity you learned how to create simple surfaces using Extrude and Revolve, and how to edit the surfaces by manipulating their parent curves.

Using simple surfaces as construction surfaces

Creating simple construction surfaces

Besides representing a very simple method of creating needed faces, the extruded and revolved surface commands can be used to build construction surfaces necessary for generating intersection curves with other faces. In this situation, the surfaces can be hidden after the operation is complete. This is preferred to deleting the faces, because they are the parents of the intersection curves.

Deleting construction surfaces

If you need to delete a surface, and if that face has children (the intersection curve, possibly others), use the **Drop Parents** command to permit the curves to remain after the surface is removed. However, those curves are no longer associated, and cannot be edited. Therefore, caution is advised when using the **Drop Parents** command.

To hide the display of surfaces, right-click in the part window and choose **Hide All→Surfaces**.

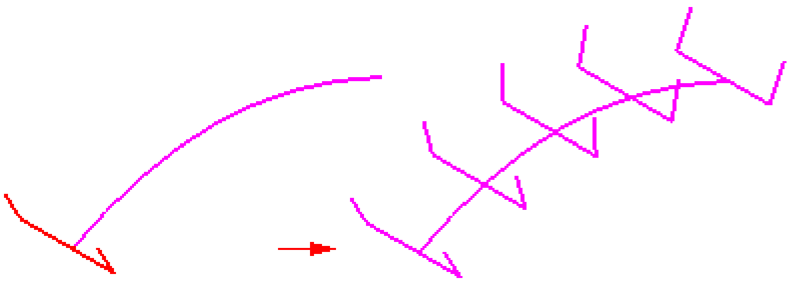
Tear-Off Sketch

Typical surface design methodology creates several curves on three base planes. As new planes are created, adding and copying profiles can be tedious, if not impossible. The **Tear-Off Sketch** command provides a clear modeling advantage in that it:

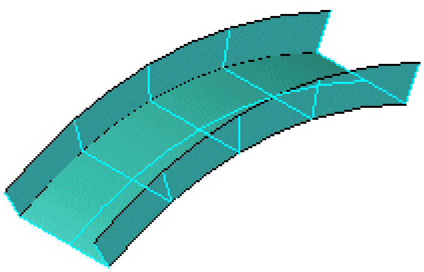
- Transfers or copies sketches from one plane onto another.
- Quickly creates new cross sections without having to define a plane and include geometry.
- Replicates sketches quickly for use in swept or lofted protrusions.
- Creates new sketches which are parallel or perpendicular, along curves, angular associative, copied or moved.

Tear-Off Sketch command

Use the **Tear-Off Sketch** command  to copy or move a sketch and layout elements from one reference plane to another. This allows you to divide a large sketch into a series of smaller sketches, which can make it easier to complete the part or assembly you are documenting. For example, you can associatively copy a single sketch to a series of sketches using reference planes normal to a curve.



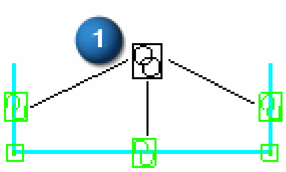
You can then use the resulting sketches as cross sections for constructing a feature such as a swept surface.



You can use the **Tear-Off Sketch** Options dialog box to:

- Copy sketch elements associatively
- Copy sketch elements non-associatively
- Move sketch elements

When you copy sketch elements associatively, a special symbol (1) is added to the copied sketch elements to indicate that the copied elements are associatively linked to the original sketch elements. If you modify the original elements, the associative elements also update.



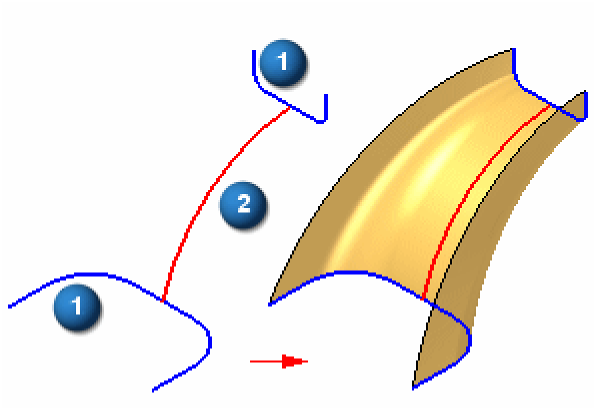
When selecting the sketch to tear off, you can select a single sketch element or a chain of sketch elements. You can only tear off sketch elements within the same sketch. If you select multiple sketch elements, all the elements are copied or moved either associatively or non-associatively. You cannot copy some of the elements associatively and some of them non-associatively.

After you copy or move the elements to the new sketch, you can use the Reposition button on the **Tear-Off Sketch** command bar to connect keypoints of an element profile to a pierce point that passes through the target reference plane. In the Assembly environment, the pierce point option is not available.

You can connect multiple keypoints on a torn off sketch to multiple keypoints. For example, you can connect keypoints on a sketch to multiple guide curves. You have to select the **Reposition** button for each new position definition.

Swept command

Use the **Swept** surface command to construct a surface by extruding one or more cross sections (1) along a path you define (2).



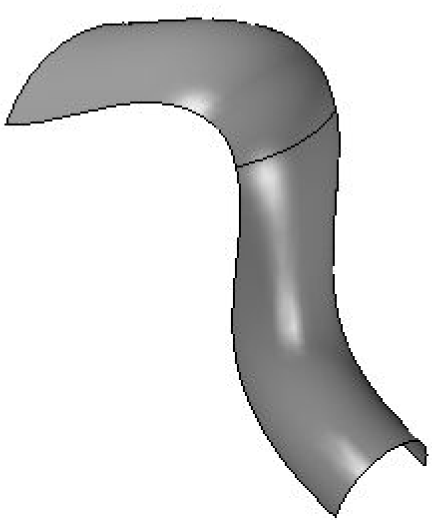
You can define up to three paths and many cross sections. After you define the third path, the command automatically proceeds to the cross section step.

The cross sections can be open or closed and can be planar or non-planar. You can place them anywhere along the path. For predictable results, it is best if the cross sections intersect all paths. The sweep paths can be either tangent or non-tangent.

When you create a swept surface using a closed sketch, you can use the **Open Ends** and **Close Ends** options on the command bar to specify whether the ends of the swept surface are open or closed. When you set the **Close Ends** option, faces are added to the ends of the feature to create a closed volume.

You can select wireframe elements from multiple Parasolid bodies or sketches and the elements will remain associative.

Activity: Creating a swept surface



Overview

In this activity, you will learn to create and edit a swept surface. You will use provided sketches to create a swept surface. After completing the surface, you will edit the sketch path and cross sections to observe the surface shape changes.

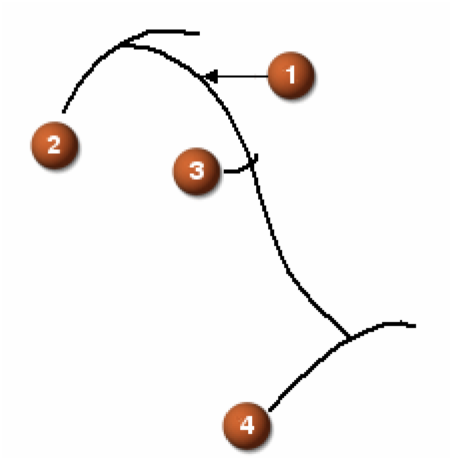
Objectives

After completing this activity you will be able to create and edit a swept surface.

Open the part file

- Open **surface lab 3-02.par**.

The part file contains four sketches. Sketch element (1) is the guide path (curve) and sketch elements (2–4) are the cross sections (arcs).



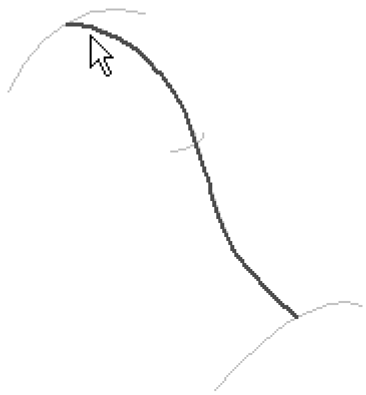
> **Note**
>
> The **Swept** surface command has the same steps as the part **Swept Protrusion** command.

Create a swept surface

- Select **Surfacing** tab→**Surfaces** group→**Swept** .

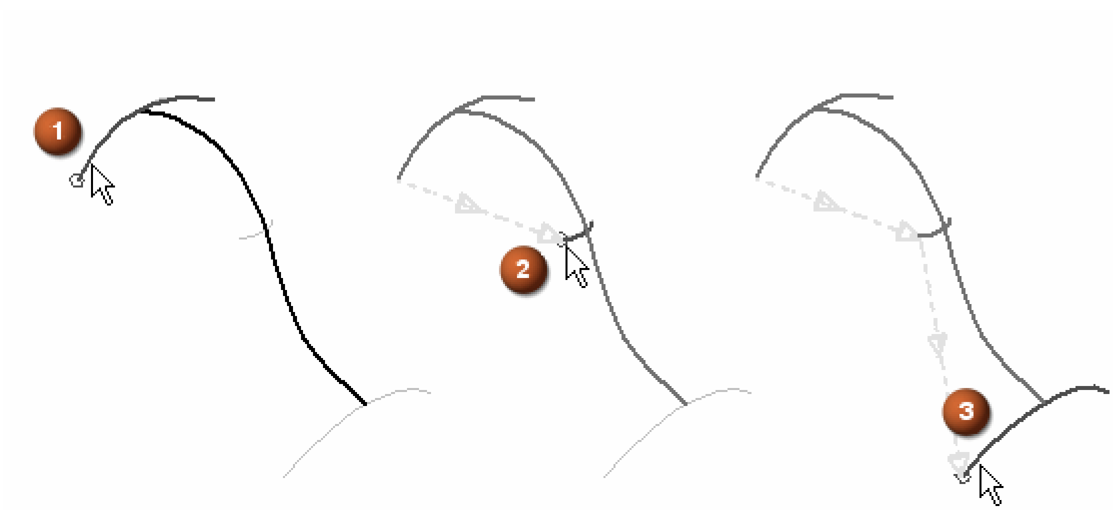
  In the **Sweep Options** dialog box, select *Multiple paths and cross sections*. Click **OK**.

- For the **Path** step, select the curve shown and then click **Accept**.

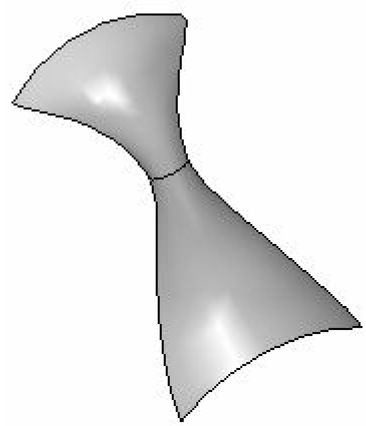


- Since there is only one path, click the **Next** button on the command bar to proceed to the **Cross Section** step.

- Select cross section 1, and then click Accept. Select cross section 2 and click Accept. Select cross section 3 and click Accept.

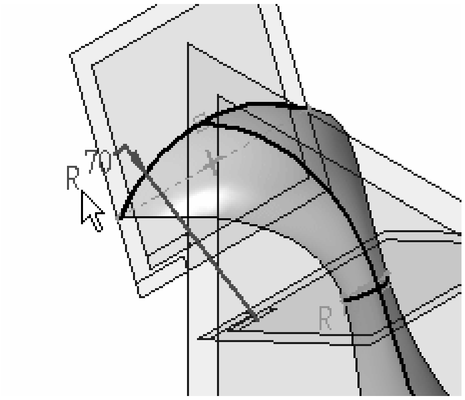


- On the command bar, click **Preview** and then **Finish**.

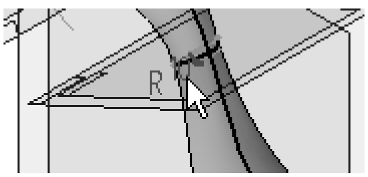


Modify the shape of the surface

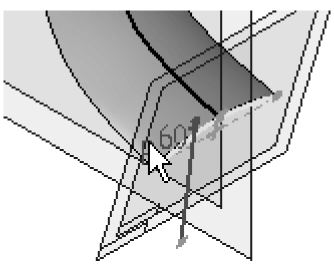
- Click the Select tool. Select the surface, and then click **Dynamic Edit** .
- Click the 70 mm radius dimension on cross section 1. Type **50** and press Enter.



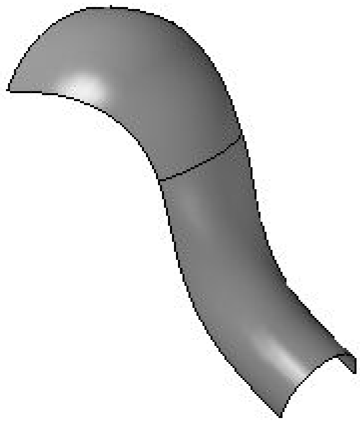
- Click the 10 mm radius dimension on cross section 2. Type **40** and press Enter.



- Click the 60 mm radius dimension on cross section 3. Type **20** and press Enter.

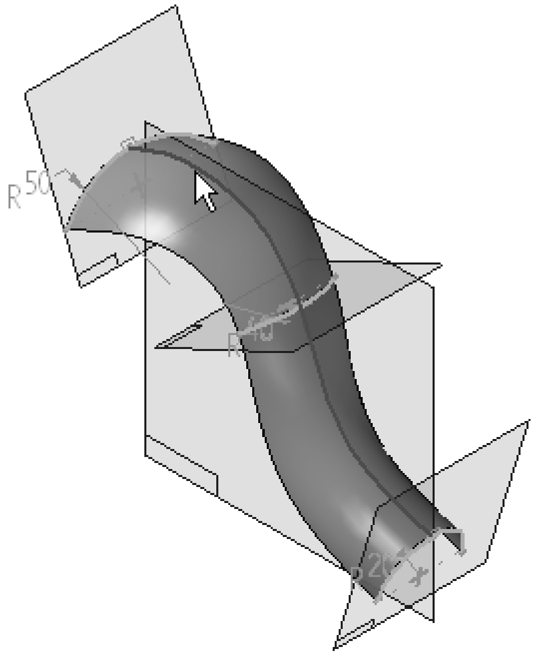


- Click the Select tool to end the dynamic edit. Press Escape to finish.

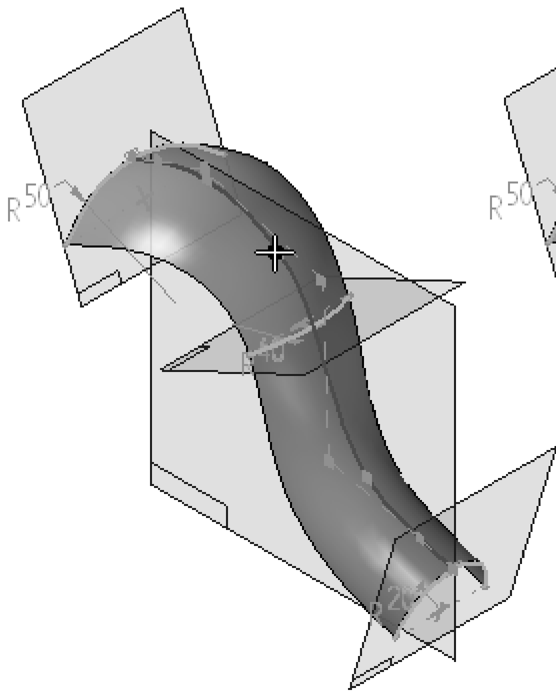


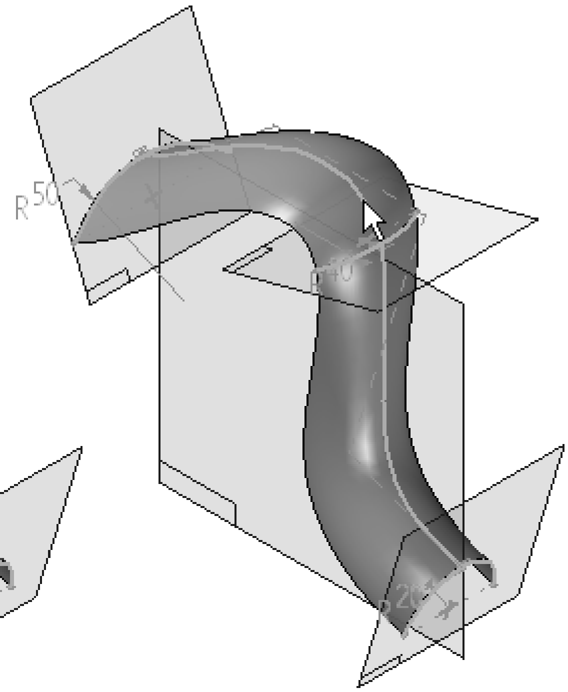
Dynamically edit the path curve

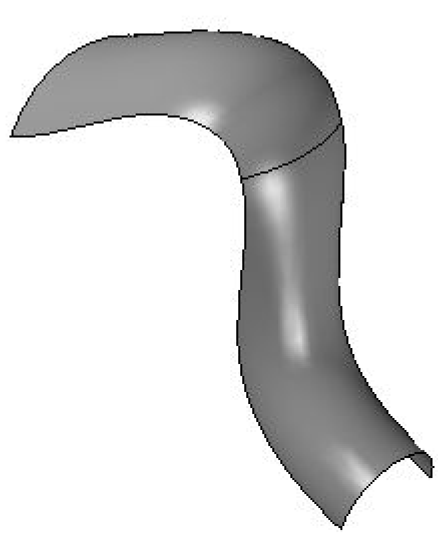
- Click the Select tool. Select the surface and then click **Dynamic Edit** .
- Click the path curve as shown.



- Select the edit point shown and drag it to the right.







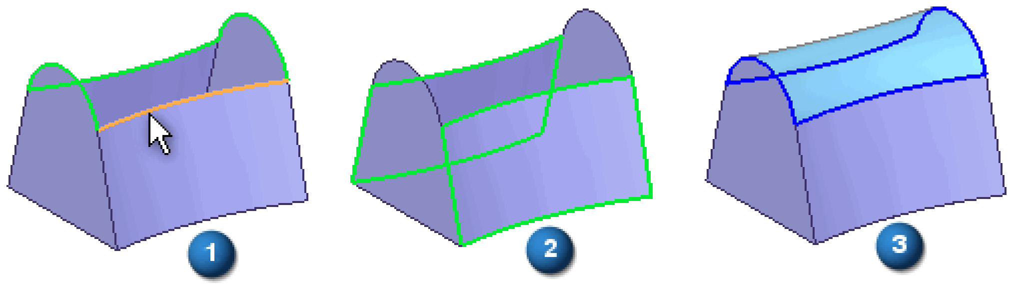
- The activity is complete.

Summary

In this activity you learned how to create and edit a swept surface.

Bounded Surface command

Use the **Bounded** surface command  to create a construction surface using boundary elements you define. The boundary elements can be curves or edges and they must define a closed area (1). You can also specify whether any adjacent faces (2) are used to control tangency on the new bounded surface (3).

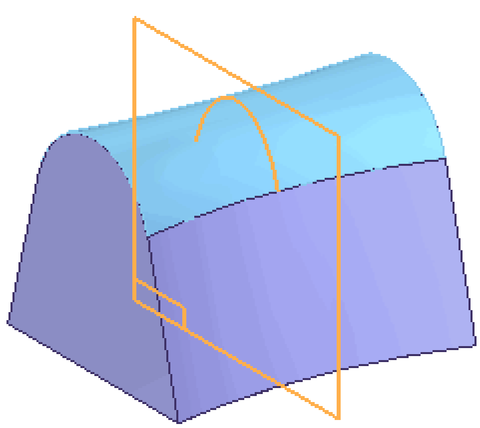


- The curve/edge set must form a closed loop.
- Adjacent faces can be used to control tangency on the new bounded surface.
- The preparation of edges/curves to be utilized may require the use of the derived curve and split curve commands.
- The keypoint curve command can be used to generate a boundary curve.

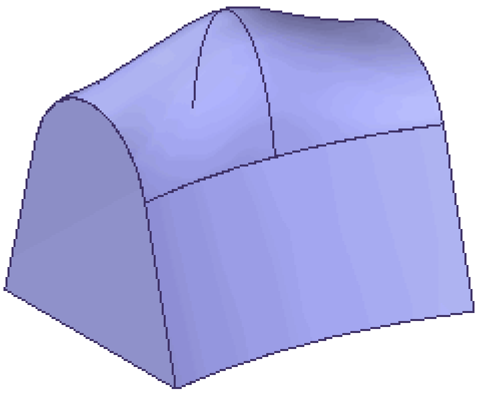
Inserting sketches

You can use the **Insert Sketch** Step on the command bar to add new sketches to a Bounded feature. The geometry for the new sketch is created by intersecting a reference plane you define with the Bounded feature. You do not have to create the sketch geometry yourself. When you insert a sketch, the new geometry is a created as a B-spline curve. If you want the new geometry to consist of lines, arcs, or circles, you must create the new sketch manually outside of the **Bounded** surface command.

When you click the **Insert Sketch** button on the command bar, plane creation options are added to the command bar so you can define the position for the new reference plane. For example, you can use the **Parallel Plane** option to define an offset reference plane where you want additional control over the resultant surface.



You can then edit the sketch to change the surface shape.



When you add a cross section or guide curve to an existing Bounded Surface feature using the **Insert Sketch** option, the new sketch is connected to the cross sections or guide curves. You can use the **Bounded Surface Options** dialog box to specify whether Pierce Points or BlueDots are used to connect new section to the surface.

> **Note**
>
> BlueDots are only available in the ordered modeling environment

Pierce points

When you set the **Use Pierce Points** option, connect relationships are used to tie the inserted sketch to the cross sections or guide curves it intersects. When you set the **Use BlueDots** option, BlueDot elements are used to tie the inserted sketch to the cross sections or guide curves it intersects. The option you specify also affects how you can edit the feature later.

When you connect the new sketch using the **Use Pierce Points** option, you can modify the cross sections or guide curves the new sketch intersects and the B-spline curve for the inserted sketch will update. The **Use Pierce Points** option is most suitable for models that must conform to engineering data or dimension-driven criteria, such as turbine blades, fan housings, and so forth. The **Use Pierce Points** option maintains the existing parent/child history of the model.

BlueDots

If you are working with a Bounded Surface in the ordered environment, and you insert a sketch using the **Use BlueDots** option, you can also modify the Bounded Surface feature by editing the position of the BlueDots using the Select Tool and the **BlueDot Edit** command bar. When you move a BlueDot, the portion of the sketches that are controlled by the BlueDot update, and that portion of the BlueSurf also updates.

The **Use BlueDots** option is most suitable for ordered models that are driven by esthetic requirements, such as consumer electronics products, bottle and container design, and so forth. When you use BlueDots to connect an inserted sketch, moving a BlueDot can also change the location of the reference planes of the sketches it connects.

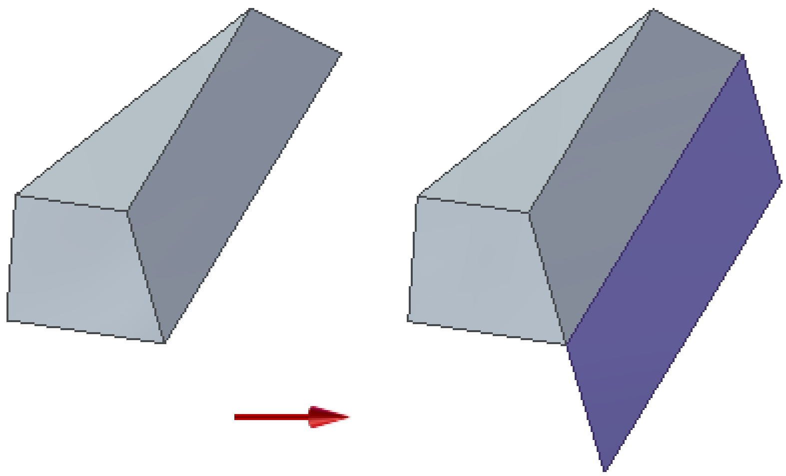
This is because a BlueDot allows you to override the existing parent/child history of the model. For example, if you insert a sketch using a parallel reference plane with an offset value of 25 millimeters, editing the location of the BlueDot can also change the offset value of the reference plane.

This behavior can be preferable when exploring the esthetic possibilities of a surface, but can be counter-productive when working with engineered surfaces. In some cases, using BlueDots can also cause a model to take longer to update, because moving a BlueDot may require more of the model to recompute than a connect relationship would.

> **Note**
>
> When you set the **Use BlueDots** option on the **Bounded Surface Options** dialog box, but existing constraints prevent BlueDots from being created, then pierce points are created instead.

Ruled Surface command

Use the **Ruled** surface command to create a surface that extends out in a specified direction from a selected edge.

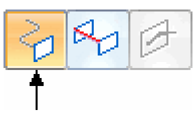


BlueSurf overview

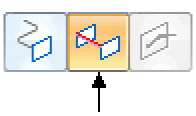
BlueSurf is a surface creation command used to generate complex, but highly editable surfaces. Like loft and sweep, a BlueSurf utilizes cross sections and guide curves, and these parent curves drive the behavior of the resultant surface. Several techniques can be applied to further edit a BlueSurf.

- New sections and guides can be incorporated, providing additional control over the BlueSurf topology.
- As sections and guides are added, the number of edit points can be increased or reduced through the concept called Edit Point Data Management.
- BlueDot edit points can be moved in order to manipulate the surface; both Shape and Local Edits are available.

The first step in creating a BlueSurf is selecting cross sections. The **Cross Section** Step activates automatically. At least two cross sections are required.

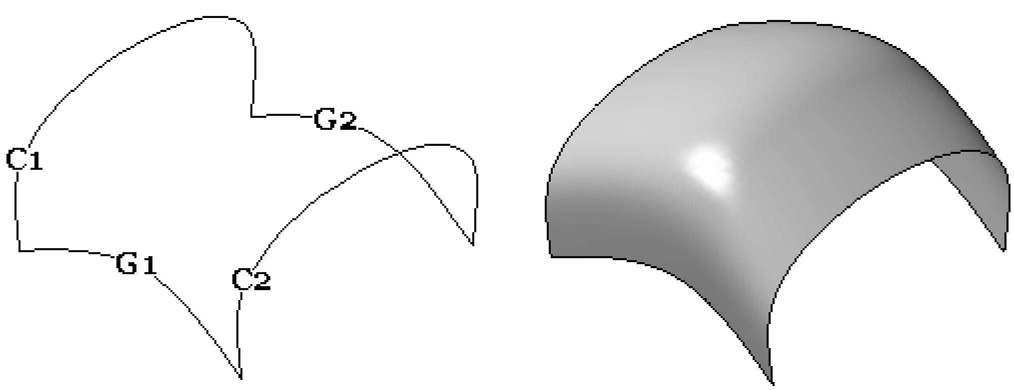


Next, you can select guide curves if needed. Click the **Guide Curve** Step and select the guide curve(s).

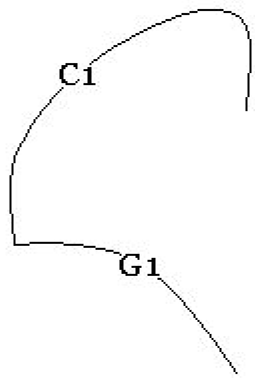


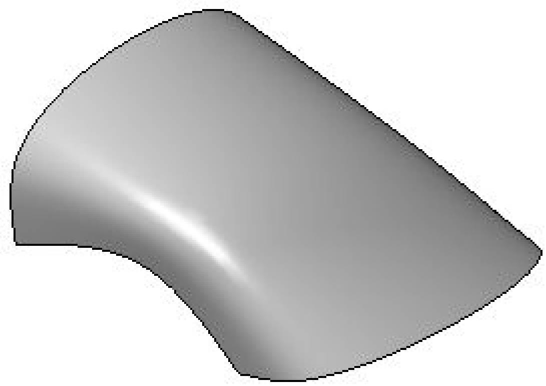
Click **Preview** and then **Finish**.

The example below shows the BlueSurf result of two cross sections (C1, C2) and two guide curves (G1, G2).



A BlueSurf may also consist of a single cross section and a single guide curve. The following example shows the BlueSurf result of using cross section (C1) and guide curve (G1) from the previous example.





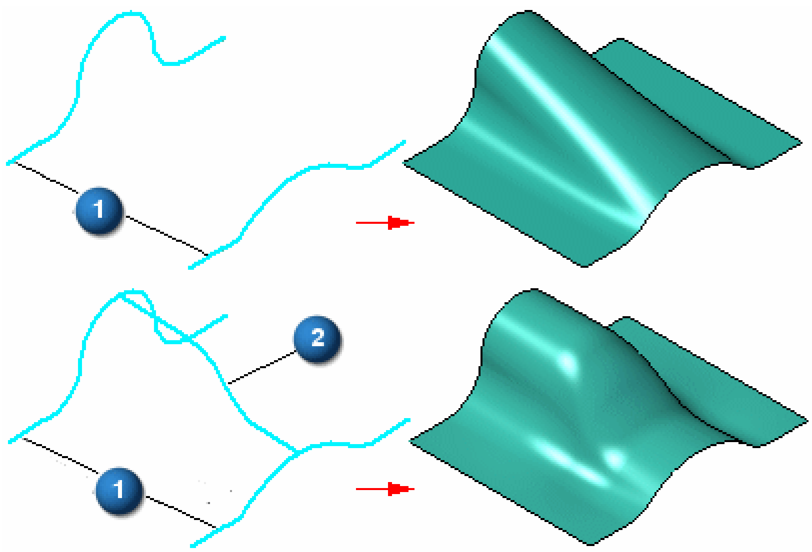
At this point, editing any of the cross sections or guide curves modifies the shape of the BlueSurf. If you need additional surface shape control, the **BlueSurf** command provides a step to insert additional sketches.

BlueSurf command

Use the **BlueSurf** command  to construct a surface using existing sketches or part edges. You can use the **BlueSurf** command to construct complex surfaces that provide many editing options.

Input requirements

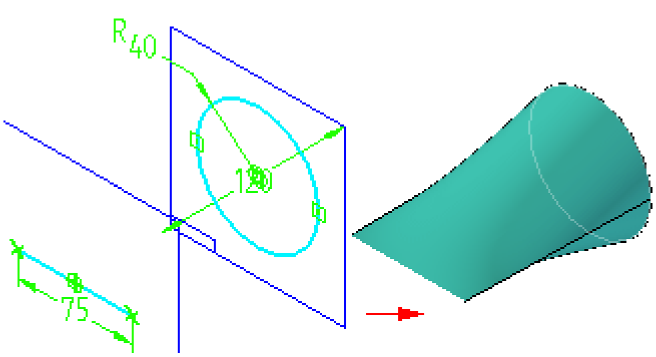
The sketches or edges can represent cross sections only (1) or cross sections (1) and guide curves (2). At a minimum, you must define two cross sections or one cross section and one guide curve.



The sketches or part edges can be open or closed.

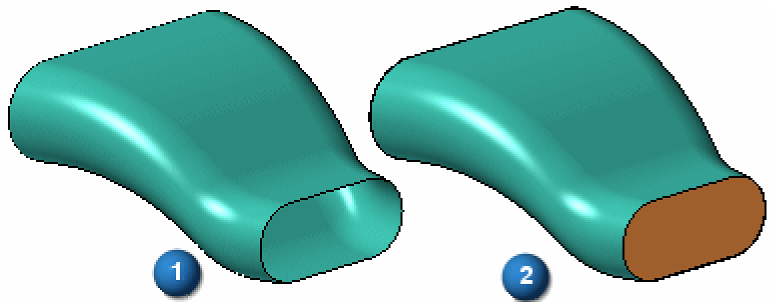
Mixing open and closed elements

When constructing a BlueSurf feature, you can use both open and closed elements in a single feature. For example, you can construct a BlueSurf feature that uses a line and a closed element, such as a rectangle or a circle, as cross sections. In some situations, you may need to split elements or define vertex mapping parameters to construct the surface you want. For example, to construct a BlueSurf feature using a line and a circle, you must split the circle into two arcs. You can use the **Split** command to split the circle into two connected arcs.



Closing ends

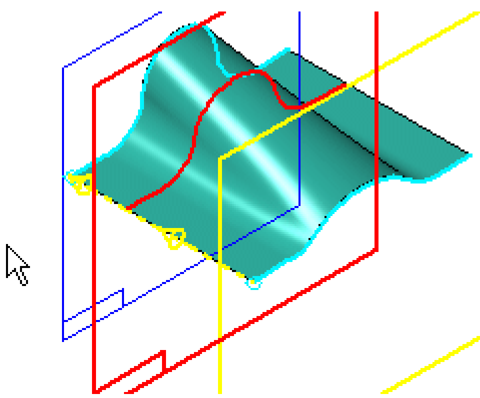
When you create a BlueSurf feature using closed cross sections, you can use the **End Capping** options on the **BlueSurf Options** dialog box to specify whether the ends of the feature are left open (1) or closed (2). When you set the **Close Ends** option, a solid construction body is created.



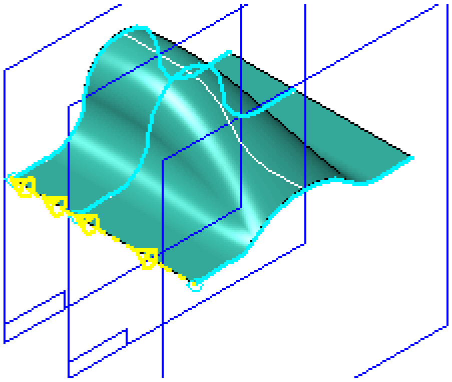
Inserting sketches

You can use the **Insert Sketch** Step on the command bar to add new sketches to a BlueSurf feature. The geometry for the new sketch is created by intersecting a reference plane you define with the BlueSurf feature. You do not have to create the sketch geometry yourself. When you insert a sketch, the new geometry is a created as a B-spline curve. If you want the new geometry to consist of lines, arcs, or circles, you must create the new sketch manually outside of the **BlueSurf** command.

When you click the **Insert Sketch** button on the command bar, plane creation options are added to the command bar so you can define the position for the new reference plane. For example, you can use the **Parallel Plane** option to define an offset reference plane where you want additional control over the resultant surface.



You can then edit the sketch to change the surface shape.



When you add a cross section or guide curve to an existing BlueSurf feature using the **Insert Sketch** option, the new sketch is connected to the cross sections or guide curves. You can use the **BlueSurf Options** dialog box to specify whether Pierce Points or BlueDotsare used to connect new section to the surface.

> **Note**
>
> BlueDots are only available in the ordered modeling environment

Pierce points

When you set the **Use Pierce Points** option, connect relationships are used to tie the inserted sketch to the cross sections or guide curves it intersects. When you set the **Use BlueDots** option, BlueDot elements are used to tie the inserted sketch to the cross sections or guide curves it intersects. The option you specify also affects how you can edit the feature later.

When you connect the new sketch using the **Use Pierce Points** option, you can modify the cross sections or guide curves the new sketch intersects and the B-spline curve for the inserted sketch will update. The **Use Pierce Points** option is most suitable for models that must conform to engineering data or dimension-driven criteria, such as turbine blades, fan housings, and so forth. The **Use Pierce Points** option maintains the existing parent/child history of the model.

BlueDots

If you are working with a BlueSurf in the ordered environment, and you insert a sketch using the **Use BlueDots** option, you can also modify the BlueSurf feature by editing the position of the BlueDots using the Select Tool and the **BlueDot Edit** command bar. When you move a BlueDot, the portion of the sketches that are controlled by the BlueDot update, and that portion of the BlueSurf also updates.

The **Use BlueDots** option is most suitable for ordered models that are driven by esthetic requirements, such as consumer electronics products, bottle and container design, and so forth. When you use BlueDots to connect an inserted sketch, moving a BlueDot can also change the location of the reference planes of the sketches it connects.

This is because a BlueDot allows you to override the existing parent/child history of the model. For example, if you insert a sketch using a parallel reference plane with an offset value of 25 millimeters, editing the location of the BlueDot can also change the offset value of the reference plane.

This behavior can be preferable when exploring the esthetic possibilities of a surface, but can be counter-productive when working with engineered surfaces. In some cases, using BlueDots can also cause a model to take longer to update, because moving a BlueDot may require more of the model to recompute than a connect relationship would.

Note

When you set the **Use BlueDots** option on the **BlueSurf Options** dialog box, but existing constraints prevent BlueDots from being created, then pierce points are created instead.

Creating new sketches manually (ordered modeling)

Alternately, you can also create new sketches for a BlueSurf feature using the **Sketch** command, or you can copy an existing sketch using the **Tear-Off Sketch** command. You can then edit the BlueSurf feature and add the new sketches as cross sections or guide curves.

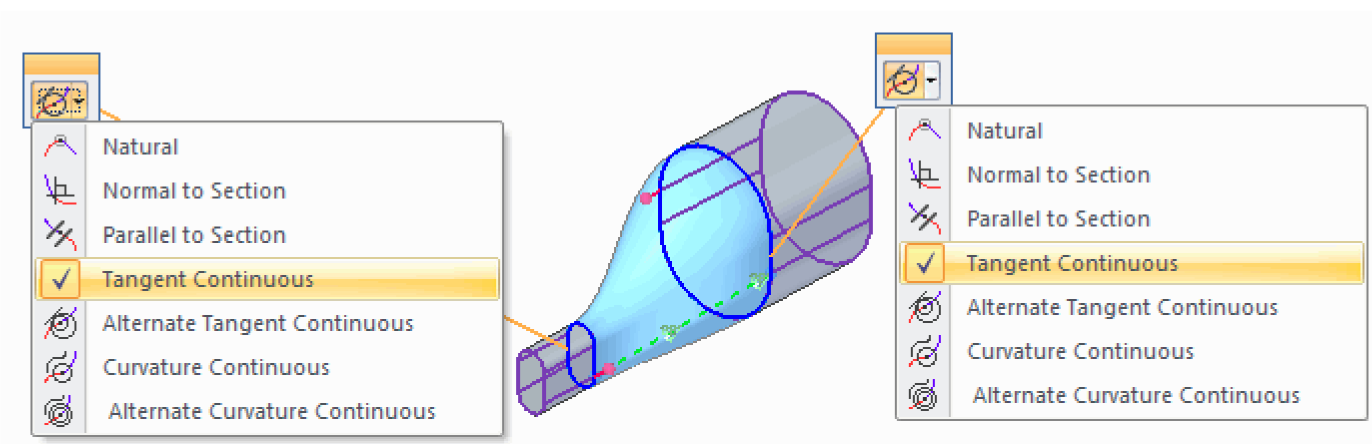
For example, when you add new cross sections, the system adds them after the existing cross sections, regardless of their physical orientation with respect to the existing cross sections. You can use the **Reorder** option on the **Advanced** tab on the **BlueSurf Options** dialog box to define the cross section sequence.

Cross section and guide curve connectivity

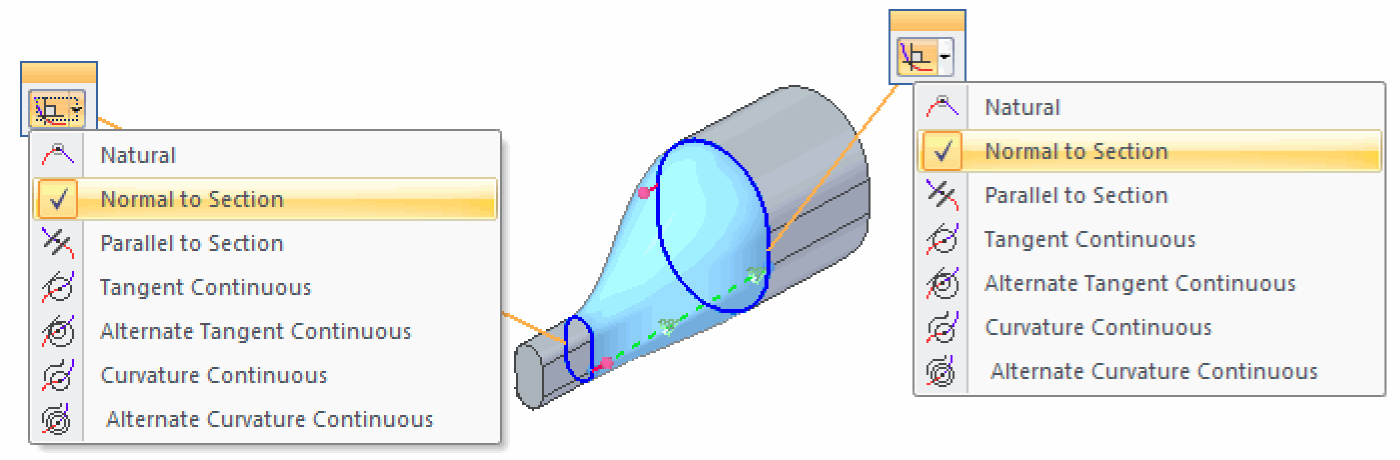
If you use a guide curve to construct a BlueSurf feature, the guide curve must intersect each cross section and be tangent continuous (the curve cannot have any sharp corners). To ensure that the guide curve stays intersected to the cross sections, you should add a connect relationship or a BlueDot at each intersection point.

End condition control

On-screen graphic handles will be displayed during placement and edit workflows to quickly change the tangency controls for cross sections and guide curve. You can use the **Tangency Control** options on the **BlueSurf Options** dialog box to define the end condition options you want for the resultant surface. For example, you can specify that the surface is tangent to the adjacent surfaces.



Many of the end condition options allow you to dynamically adjust the surface using a graphic handle (1) or by modifying a variable in the variable table. For surfaces that have several graphic handles or variables for a single cross section, you can create a master variable for all the variables that control the cross section, and then use a formula to drive all the variables for that cross section simultaneously.



BlueSurf features and lofted features

In many respects, a BlueSurf feature is constructed and behaves similarly to a lofted feature, such as a lofted surface or a lofted protrusion. For example, you can reorder cross sections, define vertex mapping rules, and define the end section conditions for a BlueSurf feature and a lofted feature.

BlueSurf Options dialog box

Standard Tab Options

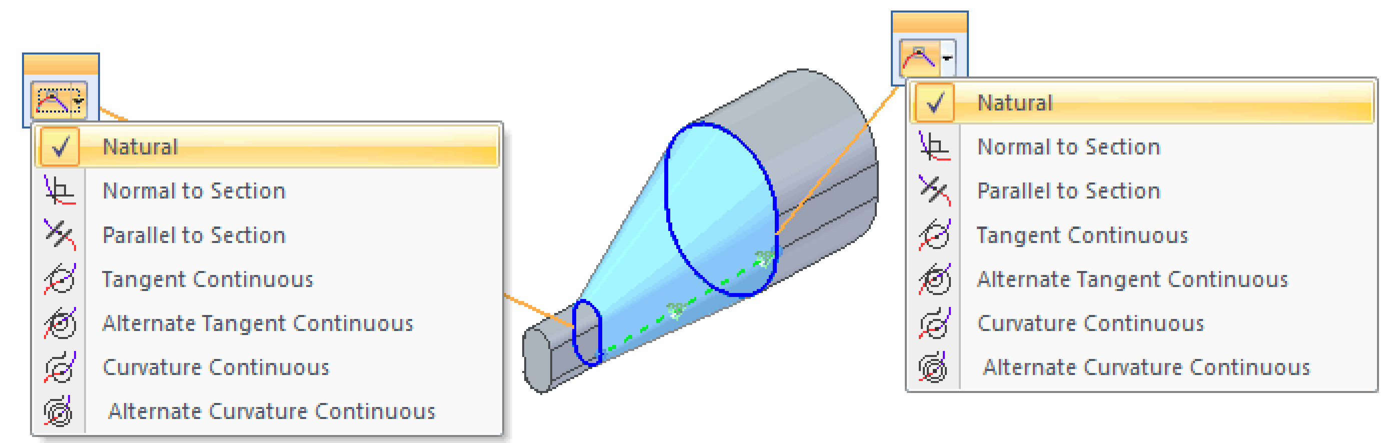
Tangency Control

Specifies the options you want for controlling the shape at the ends of the feature. For example, when you are creating a BlueSurf feature that must blend smoothly with adjacent surfaces, you can set the **Normal to Section** option to ensure a smooth blend between the existing surfaces.

The following options are available, depending on the geometry you select for the cross section or guide curve:

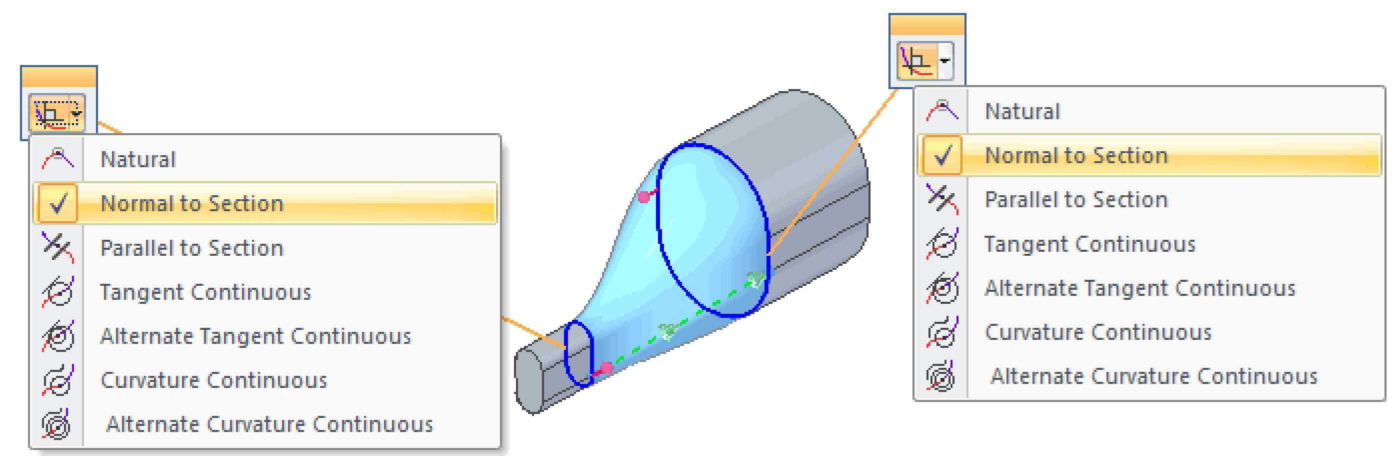
Natural

There are no constraining condition enforced at the end sections. This is the default end condition and is valid for any cross section type.



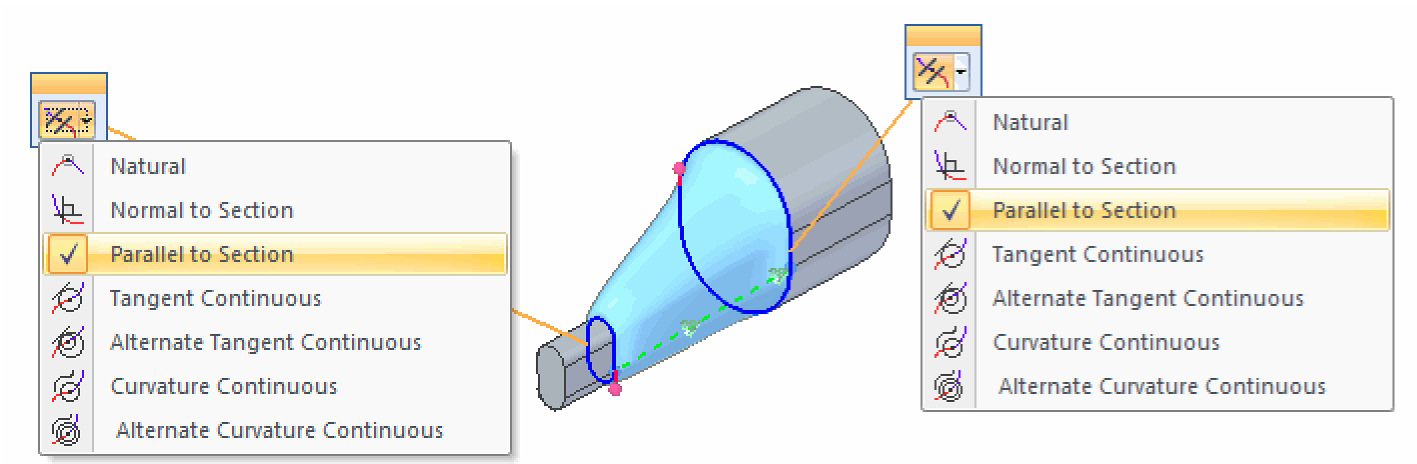
Normal to Section

End cross sections that are planar support a normal to section end condition. You can control the length of the vector using the variable table or by modifying the vector handle in the graphics window. In this example, the resultant surface illustrates the graphic handles (1) that you can use to modify the surface.



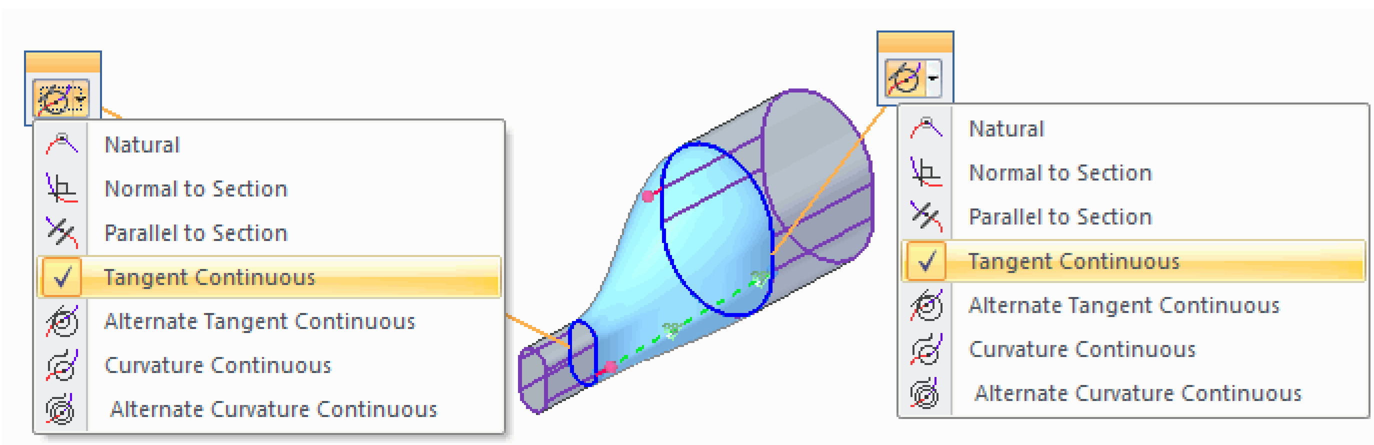
Parallel to Section

End cross sections that are planar support a parallel to section end condition. You can control the length of the vector using the variable table or by modifying the vector handle in the graphics window. To see the effect of this setting, compare the following illustration of Parallel to Section with the Normal to Section example.



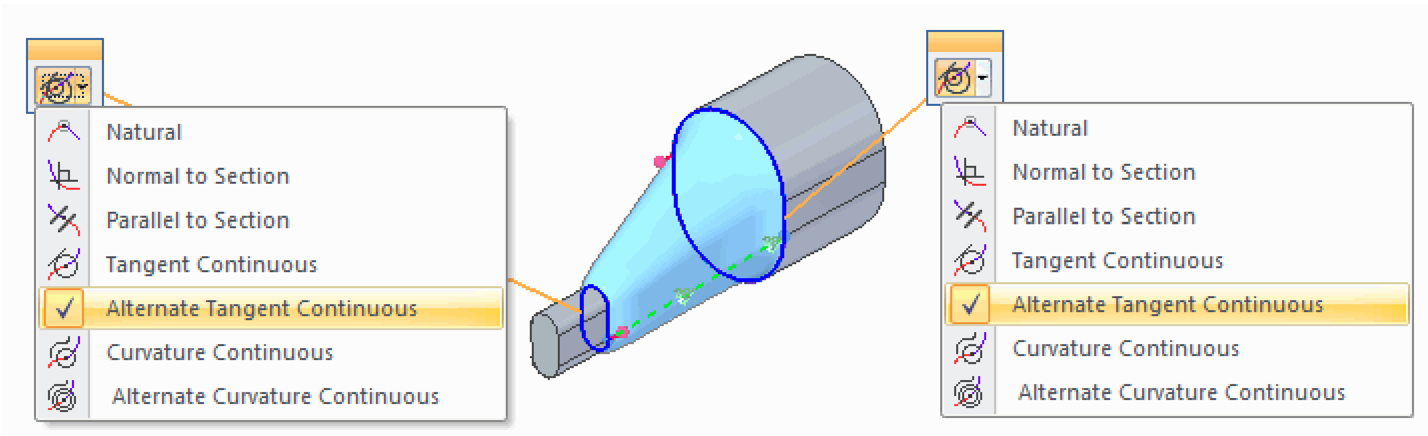
Tangent Continuous

End cross sections defined using part edges and construction curves support a tangent condition. The tangent vector for the surface is determined by the adjacent surfaces. You can control the length of the vector using the variable table or by modifying the vector handle in the graphics window.



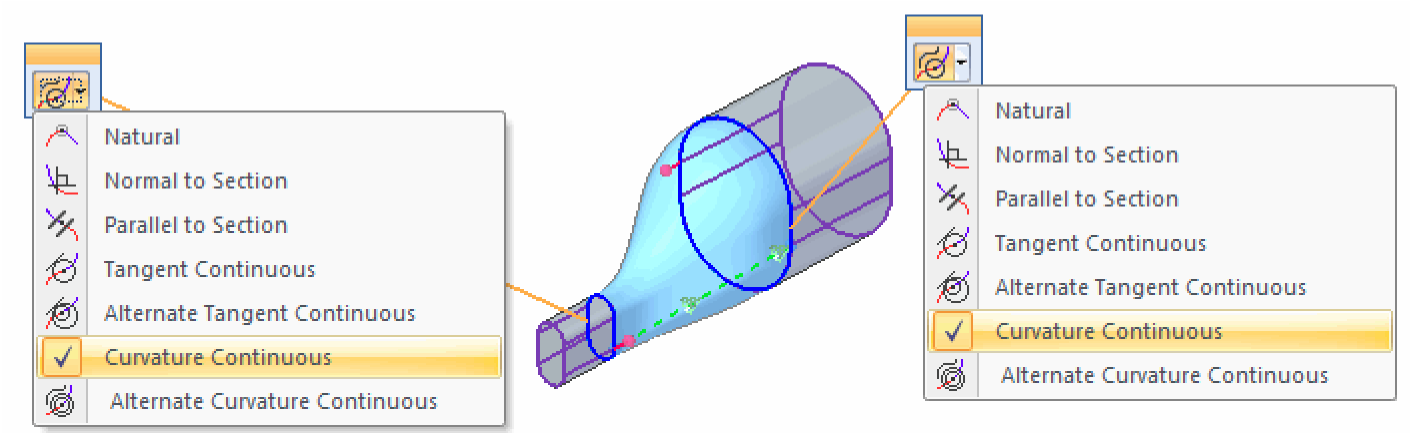
Alternate Tangent Continuous

End cross sections defined using part edges and construction surfaces support a tangent interior condition. Tangent Interior forces the surface to be tangent to the inside faces. For example, the surface below has the **Alternate Tangent Continuous** option applied to both cross section. The resulting surface is constructed tangent to the planar face.



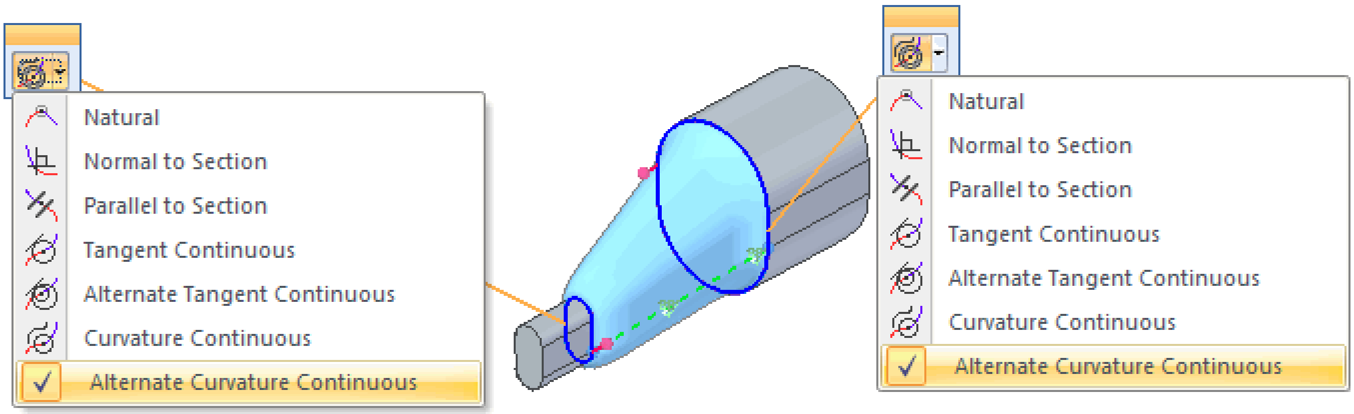
Curvature Continuous

End cross sections defined using part edges and construction surfaces support a curvature continuous condition. The tangent vector for the surface is determined by the adjacent surfaces. You can control the length of the vector using the variable table or by modifying the vector handle in the graphics window.



Alternate Curvature Continuous

End cross sections defined using part edges and construction surfaces support a alternate curvature continuous condition. The tangent vector for the surface is determined by the alternate face at that end. You can control the length of the vector using the variable table or by modifying the vector handle in the graphics window.



For more information and illustrations which show you how you can control the surface shape at the ends of BlueSurf and lofted features, see the End Conditions section in the Constructing Lofted Features (ordered) or Constructing Lofted Features (synchronous) Help topics.

Start Section

Specifies the tangency control option you want for the first cross section.

End Section

Specifies the tangency control option you want for the last cross section.

Edge Guide 1

Specifies the tangency control option you want for the first guide curve. The options available for defining guide curve tangency conditions depend on the type of element you select for the guide curve. For example, if you want to be able to control the tangency of the BlueSurf feature with respect to an adjacent surface, use an edge on the surface as the guide curve rather than, for example, the sketch that was used to construct the adjacent surface.

Edge Guide 2

Specifies the tangency control option you want for the last guide curve. The options available for defining guide curve tangency conditions depend on the type of element you select for the guide curve. For example, if you want to be able to control the tangency of the BlueSurf feature with respect to an adjacent surface, use an edge on the surface as the guide curve rather than, for example, the sketch that was used to construct the adjacent surface.

End Capping

Specifies the end capping options you want. This option is available only when the cross section profiles are closed.

Open Ends

Specifies that no planar end caps are added to the feature.

Close Ends

Specifies that planar end caps are added to the feature to create an enclosed volume.

Extent type

Controls whether or not the feature closes on itself.

Open

Specifies that the feature begins with the first cross section and ends with the last cross section. The feature does not close on itself.

Closed

Specifies that the surface will close on itself. When you set this option, the first cross section is also used for the last cross section.

Curve Connectivity

Specifies how a cross section and a guide curve are connected. These options only apply to new sketches you add using the **Insert sketch** button on the command bar.

Use Pierce Points

Specifies that a connect relationship is used to connect the cross section and guide curve where they intersect. The position of the connect relationship is calculated using the **Pierce Point** option on the **IntelliSketch** dialog box. The **Use Pierce Points** option is typically used when constructing engineered surfaces, such as the surfaces for a fan or turbine blade, where engineering data or dimension-driven criteria must be maintained.

Use BlueDots

Specifies that a BlueDot is used to connect the cross section and guide curve where they intersect. When you connect a cross section and a guide curve with a BlueDot, you can use the BlueDot as a handle to dynamically modify the shape of the cross section and guide curve. The **Use BlueDots** option is typically used when constructing esthetic surfaces, such as the surfaces for consumer electronics product, where a more free-form approach to surface design is desired.

Note

The **Use BlueDots** option is available only in the ordered modeling environment. The BlueDots functionality is not available in the synchronous environment.

Inserted-Sketch

Allows you to define a tolerance value for sketches you insert. The tolerance value you specify is used to control the complexity of the curve that is created.

Tolerance

Specifies the tolerance value you want to use.

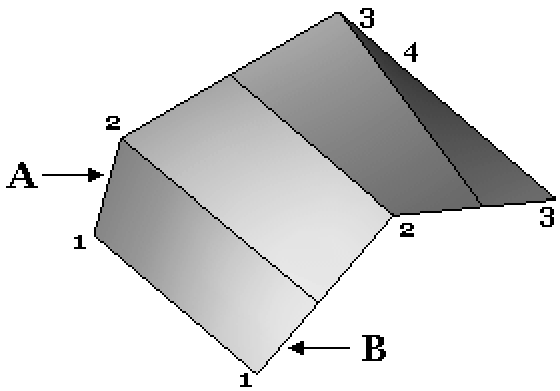
Advanced Tab Options

Vertex Mapping

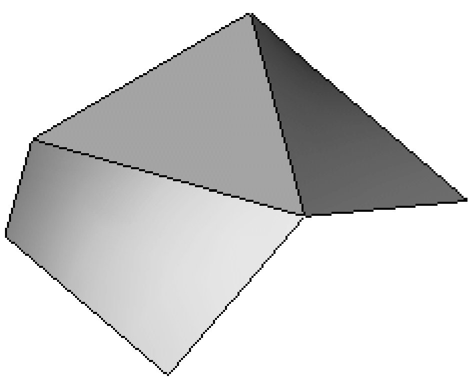
Vertex mapping is a technique to help create flow between section vertices; you can map a vertex or point on one cross section to a vertex or point on another cross section. Vertex mapping is useful for controlling or eliminating twists and discontinuities in a surface. If there is a vertex count mismatch between sections, equally spaced vertices are used on each section.

You can add vertex maps while creating a BlueSurf or by editing an existing BlueSurf.

Notice in the first image below that section (A) has four vertices and section (B) has three vertices. The BlueSurf command automatically inserts vertices equally spaced on each section. Notice the surface flow is not smooth.



The result of vertex mapping.



Map Sets

Lists the sets of mapped vertices you have defined. You can add vertex map sets to create a smooth surface flow; to add a new set of mapped vertices, click the **Add** button, then click a point on each cross section curve.

Add

Allows you to add a new mapped vertex set.

Delete

Allows you to delete an existing mapped vertex set.

Reorder

Allows you to reorder cross sections that were defined out-of-sequence. This option is useful when you modify an existing feature by adding a new cross section. You cannot use the reordering capability to create a feature that intersects itself.

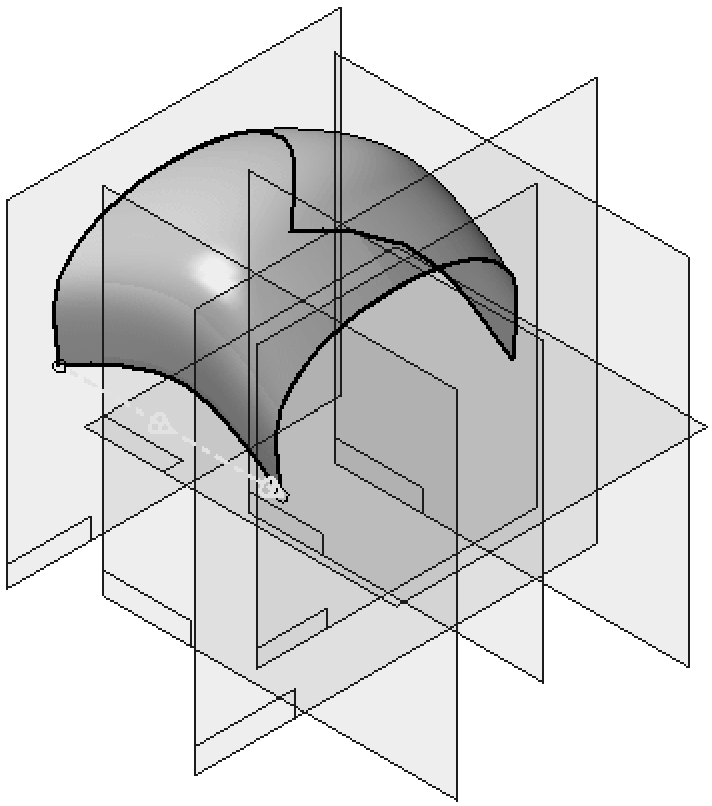
To reorder a cross section, select the cross section in the list, then click the **Up** or **Down** buttons to move the cross section entry in the list.

Insert sketches into a BlueSurf

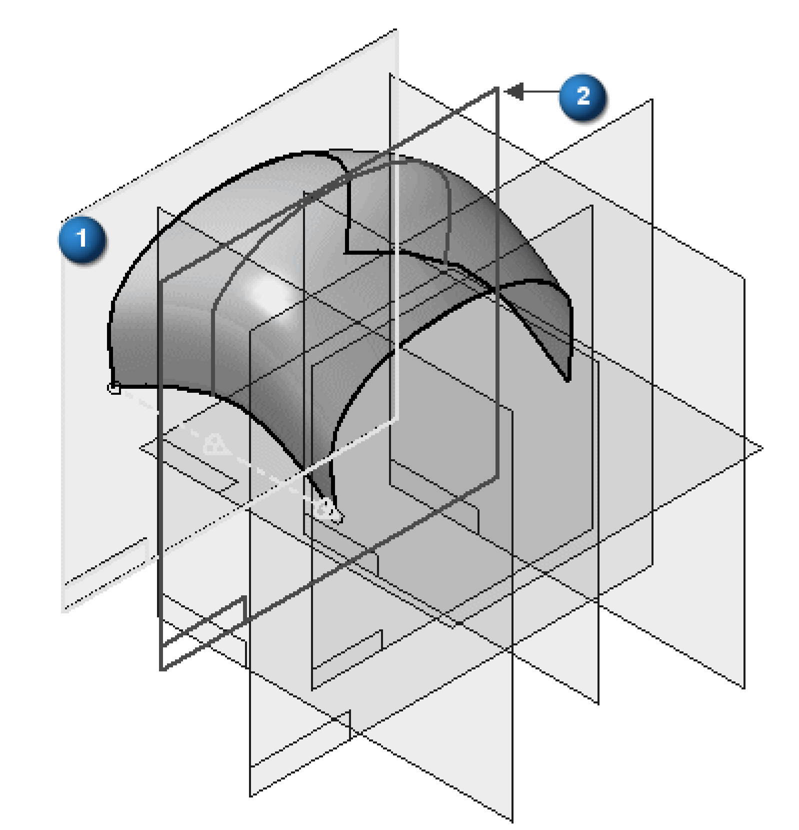
In the following example, the **Use BlueDots** option is selected in the **BlueSurf Options** dialog box for curve connectivity.

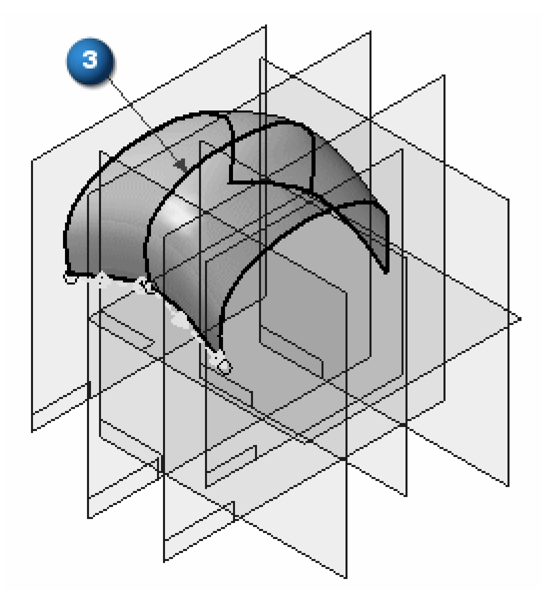
Inserting a sketch.

1. On the **BlueSurf** command bar, click the **Insert Sketch** Step. You must select a plane to insert a sketch on. All of the plane creation methods are available.

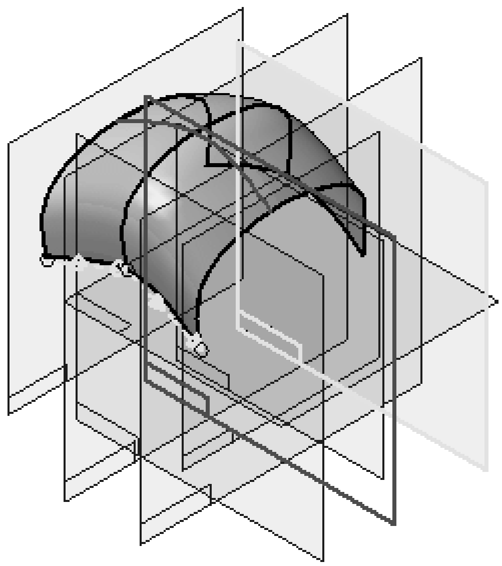


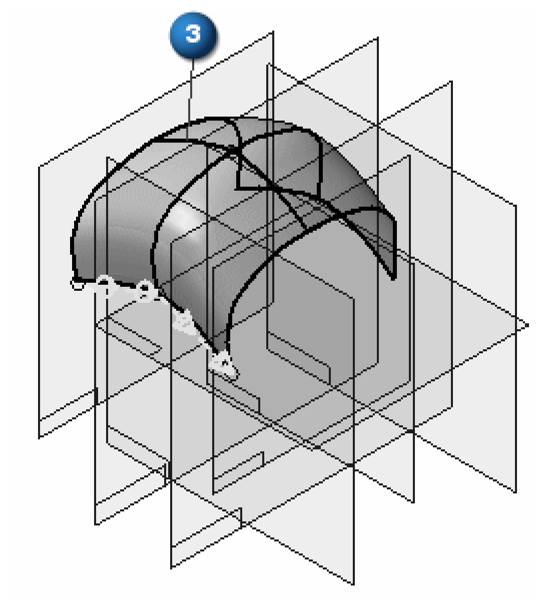
In the following example, the parallel plane option was selected. Reference plane (1) was selected as the plane to be parallel to. Reference plane (2) can be dynamically dragged to the location to insert a sketch. You can also key in a distance. Click location to insert a sketch (3).



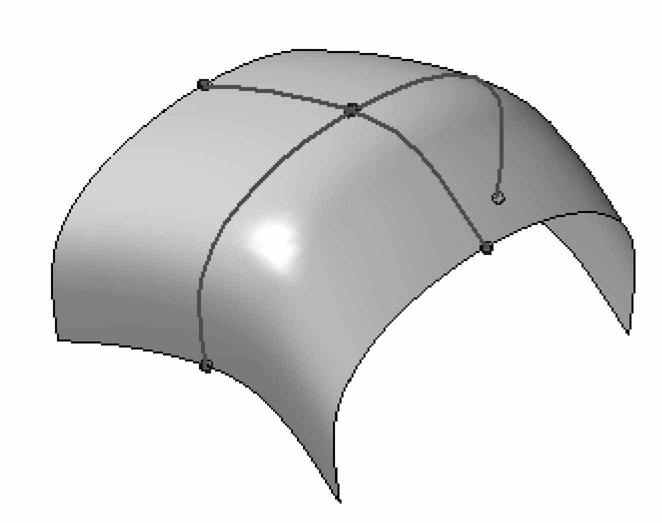


2. Insert a sketch (3) in the guide curve direction and notice the results. The parallel plane is used again.





3. Now turn off the reference planes and observe the results.



When the guide curve direction sketch was inserted, it crossed another sketch. The **BlueSurf** command automatically inserts BlueDots at the intersection of the curves. If there were several sketches in the cross-section direction, the inserted sketch in the guide curve direction would be connected with BlueDots.

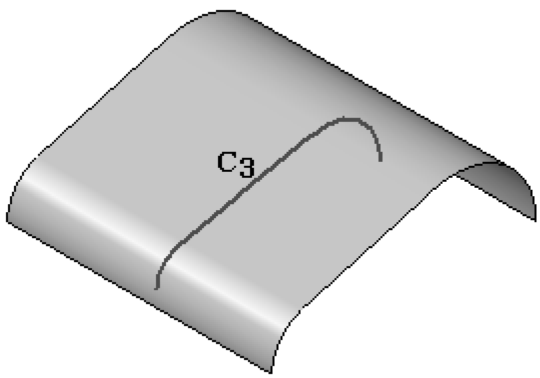
Adding cross sections into a BlueSurf (ordered modeling)

Any cross section sketch created after the BlueSurf is created will not be seen by the BlueSurf feature. When you edit a BlueSurf created in the ordered modeling environment, it only recognizes sketches created before it was created.

To add a new cross section:

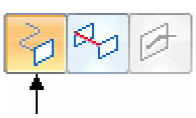
1. The BlueSurf feature below was created with two cross sections (C1, C2).

   First, add a new cross section (C3) that was created before the BlueSurf feature.

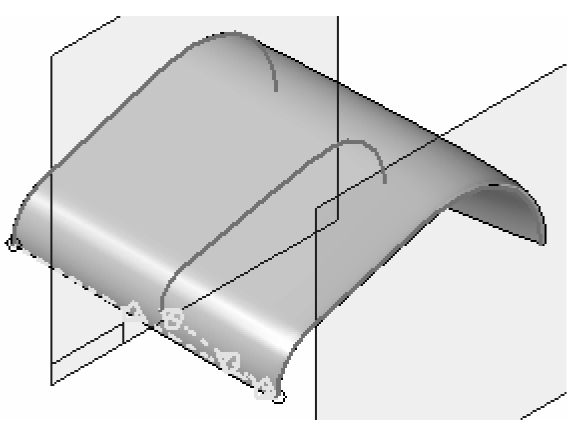


2. Click the Select Tool and then select the BlueSurf feature. On the ribbon bar, click **Edit Definition** .

3. On the **BlueSurf** command bar, click the **Cross Section** Step.

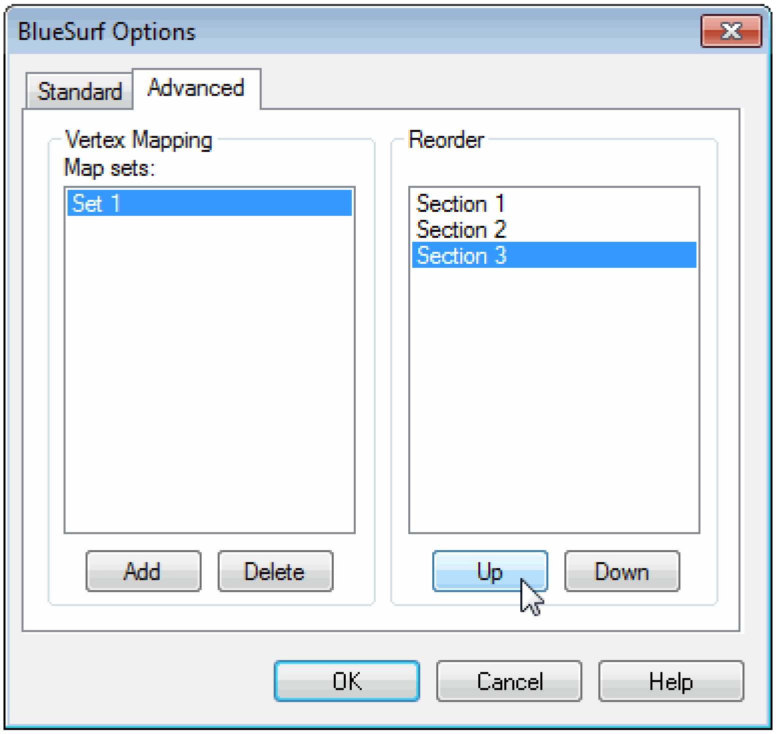


4. Identify the new cross section (C3). Notice that cross section C3 is placed last in the cross section order, which causes the BlueSurf feature to reverse direction. The cross section order below is C1, C2 and then C3. You can reorder the cross sections to make C3 be defined between C1 and C2.

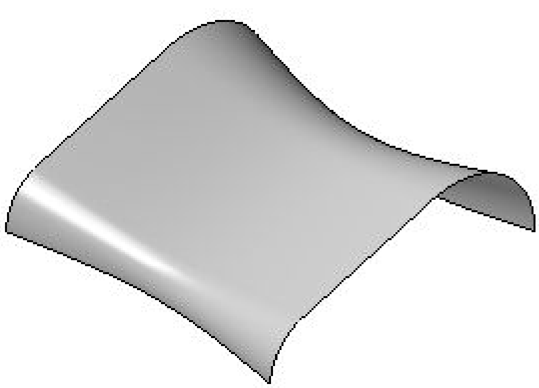


5. On the **BlueSurf** command bar, click the options button. Click the **Advanced** tab.

   Cross section C3 is shown as Section 3. To reorder C3 to be between C1 and C2, click Section 3 and then click Up. Click **OK** to apply the reorder.

   The following shows the result with cross-sections ordered C1, C2 and C3.

Adding cross sections created after the BlueSurf feature

If you create a cross section (1) after the BlueSurf feature (2), the cross section will have to be moved up in the feature tree to be recognized by the BlueSurf feature.

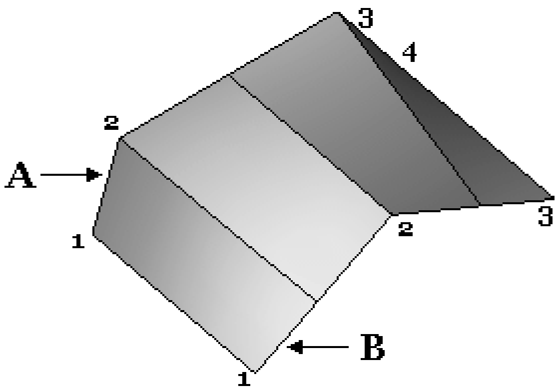
To move the cross section up in the feature tree, click the Select tool. In **PathFinder**, click and hold the Blue Surf and drag it below the latest sketch as shown.

The sketch can now be seen by the BlueSurf feature.

Vertex mapping

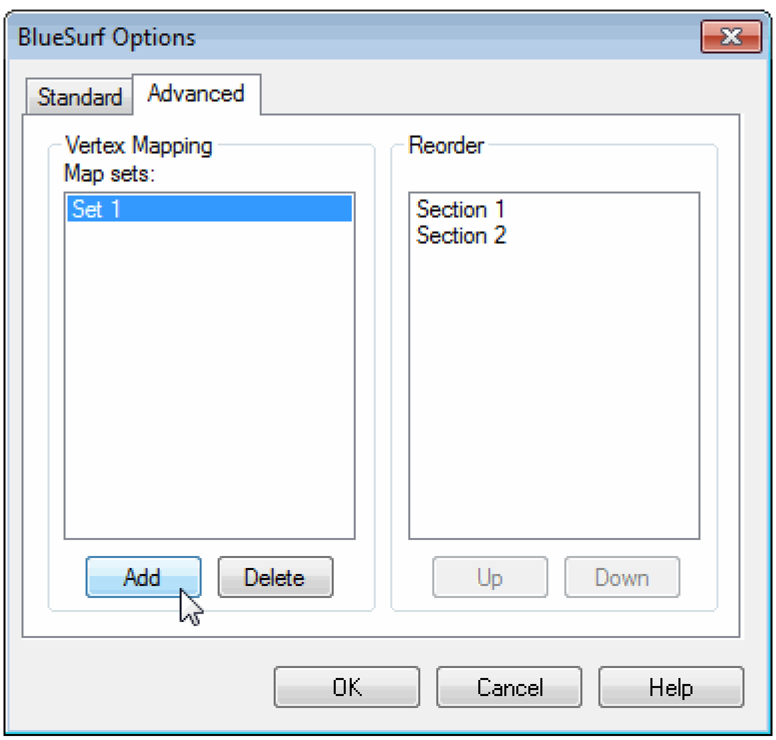
Vertex mapping is a technique to help create flow between section vertices. If there is a vertex count mismatch between sections, equally spaced vertices are used on each section.

Notice in the image below that section (A) has four vertices and section (B) has three vertices. The **BlueSurf** command automatically inserts vertices equally spaced on each section. Notice the surface flow is not smooth.

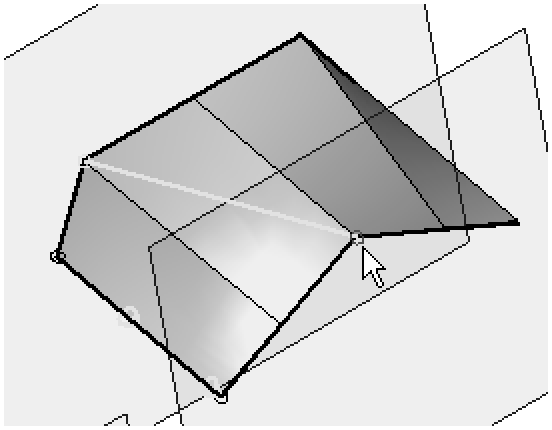


You can add vertex map sets to create a smooth surface flow. You can add vertex maps while creating a BlueSurf or by editing an existing BlueSurf.

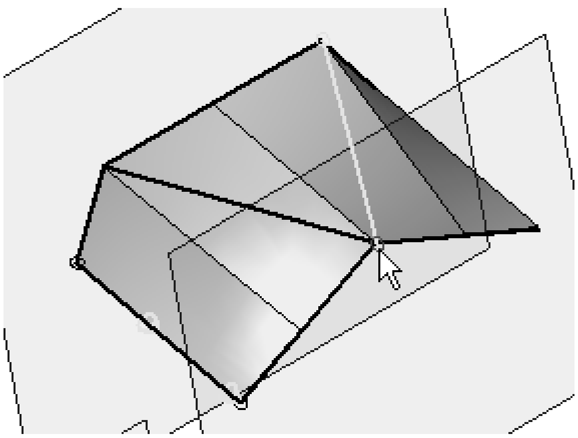
On the **BlueSurf** command bar, click the **Options** button. On the **BlueSurf Options** dialog box, click the **Advanced** tab.



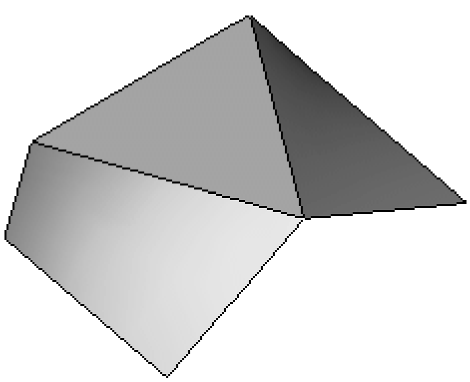
Click **Add** and then select two vertices to be mapped together as shown.



Click **Add** again and select the next two vertices to be mapped together as shown.



Click **OK** in the dialog box and then **Finish**. The result is shown below.



BlueSurf command bar

Main Steps

BlueSurf Options

Displays the BlueSurf Options dialog box.

Surface Visualization Options

Displays the Surface Visualization Options dialog box.

Cross Section Step

Selects the cross sections to which the feature will be fitted. You can select any number of cross sections for a BlueSurf feature, using any combination of cross sections created from sketches and cross sections created from part edges.

Guide Curve Step

Selects the guide curve for the feature to follow. To be valid, guide curves must touch each cross section.

Insert Sketch Step Options

Allows you to insert a sketch as a new cross section or guide curve. Inserting sketches between existing cross sections or guide curves can give you more localized control over the resulting surface. When you click the **Insert Sketch** Step button, the **Defining a Plane** options are displayed, so you can dynamically define a new reference plane. When you click to define the position for the new reference plane, Solid Edge creates the sketch by intersecting the reference plane with the current surface. The inserted sketch is created as a B-spline curve and is connected to the existing cross sections or guides curves using BlueDots (ordered environment only) or connect relationships, depending on the options you set on the BlueSurf Options dialog box.

Plane Options

Sets the method of defining the plane for the inserted sketch. Depending on the model you are constructing, some of the options listed may not be available.

Coincident Plane

Specifies that you want to define a plane that is coincident to an existing reference plane or a planar face on the part. When you set this option, a default X-axis and direction is applied to the new reference plane. You can use keyboard accelerators to define a different X-axis and direction for the new reference plane.

Parallel Plane

Specifies that you want to define a plane that is parallel to an existing reference plane or a planar face on the part. When you set this option, you can specify the parallel offset distance. When you set this option, a default X-axis and direction is applied to the new reference plane. You can use keyboard accelerators to define a different X-axis and direction for the new reference plane.

Angled Plane

Specifies that you want to define a plane that is at an angle to an existing reference plane or planar face on the part. When you set this option, you can specify the angle value you want.

Perpendicular Plane

Specifies that you want to define a plane that is perpendicular to an existing reference plane or planar face on the part.

Coincident Plane by Axis

Specifies that you want to define a plane that is coincident to an existing reference plane or a planar face on the part. When you set this option, you define the X-axis and direction for the new reference plane using a linear edge, a planar face, or another reference plane.

Plane Normal to Curve

Specifies that you want to define a plane that is perpendicular to a curve you select. This is the default option when constructing a helix using the Perpendicular option.

Plane by 3 Points

Specifies that you want to define a plane by three keypoints you select.

Tangent Plane

Specifies that you want to define a plane that is tangent to a curved face on the part. You can select a cylinder, cone, sphere, torus, or b-spline surface. When you set this option, you can also specify the angular rotation value. When you set this option, a default X-axis and direction is applied to the new reference plane. You can use keyboard accelerators to define a different X-axis and direction for the new reference plane.

Feature’s Plane

Specifies that you want to define a plane that is coincident to a reference plane used to define an earlier feature. You can select the feature you want using **Feature PathFinder** or in the graphic window. This option is not available when constructing the base feature.

Preview/Finish/Cancel

This button changes function as you move through the feature construction process. The **Preview** button shows what the constructed feature will look like, based on the input provided in the other steps. The **Finish** button constructs the feature. After previewing or finishing the feature, you can edit it by re-selecting the appropriate step on the command bar. The **Cancel** button discards all input and exits the command.

Command Tangent condition

Applies a command tangent condition to the boundary edges.

Show/Hide Tangency Control Handle

Show or hide the tangency control handle.

Deselect All

Deselects all inputs in the current step.

Select

Sets the edge selection method for defining the cross sections and guide curves. You can use any combination of selection methods to select a set of edges. Hold the Ctrl key or the Shift key to deselect an edge.

Single

Selects a single edge or sketch element.

Chain

Selects a sketch or a tangentially continuous chain of edges.

Loop

Selects all the edges of individual loops of a face by selecting the face and then choosing a loop.

Face

Selects all the edges of a face by selecting the face.

Accept (check mark)

Accepts the edge selection criteria and selects all edges that meet the criteria. Right-click to accept the selection or press Enter.

Deselect (x)

Clears any selected edges and the edge selection criteria.

Other command bar Options

Open Ends

Creates the BlueSurf feature with open ends.

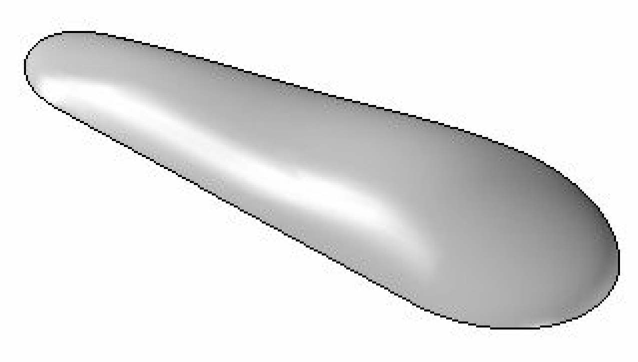
Closed Ends

Creates the BlueSurf feature with closed or capped ends

Name

Displays the feature name. Feature names are assigned automatically. You can edit the name by typing a new name in the box on the command bar or by selecting the feature and using the **Rename** command on the shortcut menu.

Activity: Creating a BlueSurf using analytics



Overview

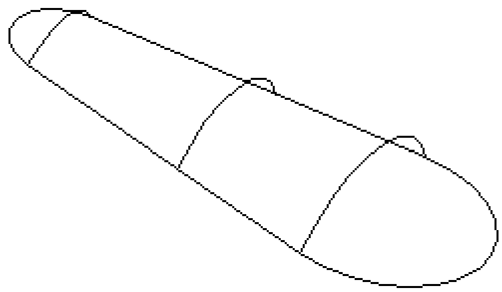
In this activity, you learn to create a BlueSurf feature. You will use provided sketches to create a BlueSurf surface.

Objectives

After completing this activity you will be able to create a BlueSurf feature.

Open the part file

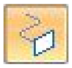
- Open **surface lab 3-03.par**.



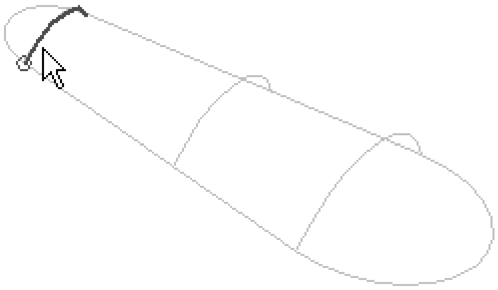
Create several BlueSurf features

- Select **Surfacing** tab→**Surfaces** group→**BlueSurf** .

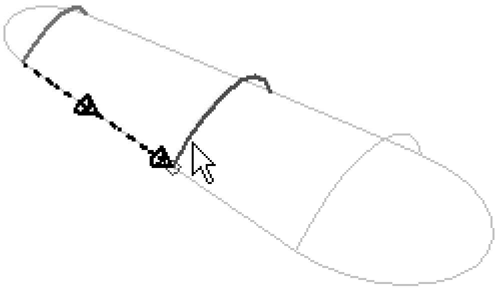
  Notice on the **BlueSurf** command bar that the **Cross Section** Step is active.

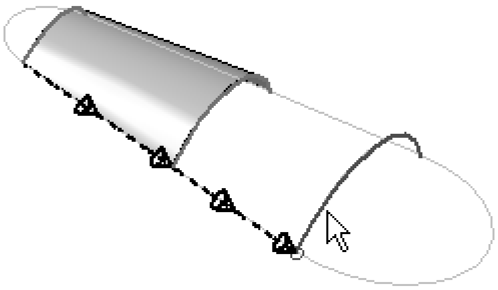
- To define the first cross section, click the arc shown and then right-click (or click Accept).



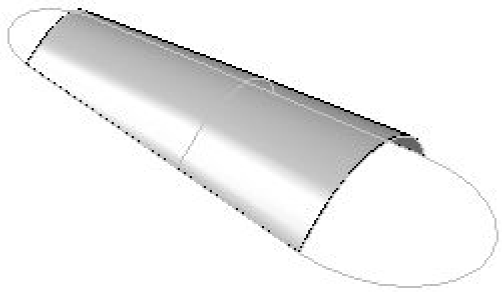
- For the next cross section, click the arc shown and then right-click.



- For the last cross section, click the arc shown and then right-click.



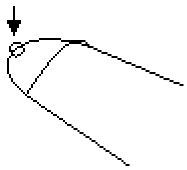
- Click **Next**, and then click **Finish**.



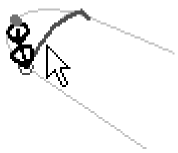
Notice the BlueSurf feature shown in **PathFinder**. Hide this feature.

Create another BlueSurf

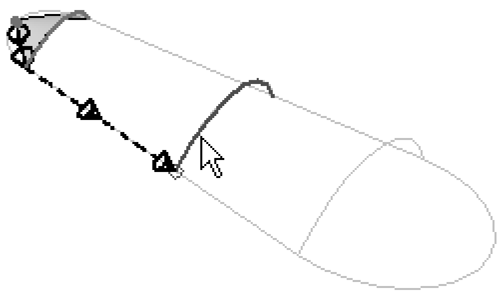
- Click the **BlueSurf** command.
- For the first cross section, select the point shown. Use **QuickPick** to select the vertex point.

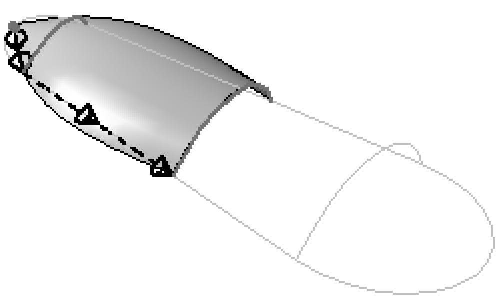


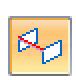
- Click the arc shown for the second cross section, and then right-click.



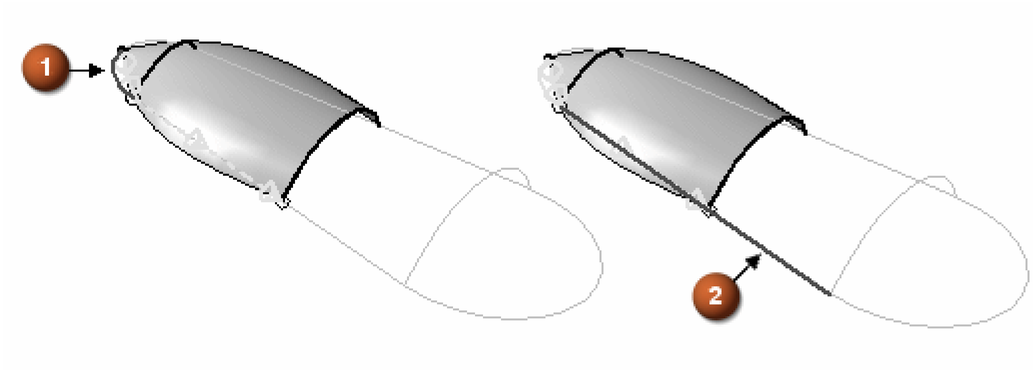
- Click the arc shown for the last cross section, and then right-click.



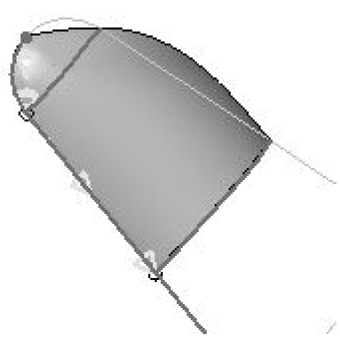


- Click **Next**. Do not click **Finish**.
- Apply guide curves to the BlueSurf. Click the **Guide Curve Step**  on the command bar.
- In the command bar Select list, click **Single**. This allows you to select single sketch elements for the guide curve.

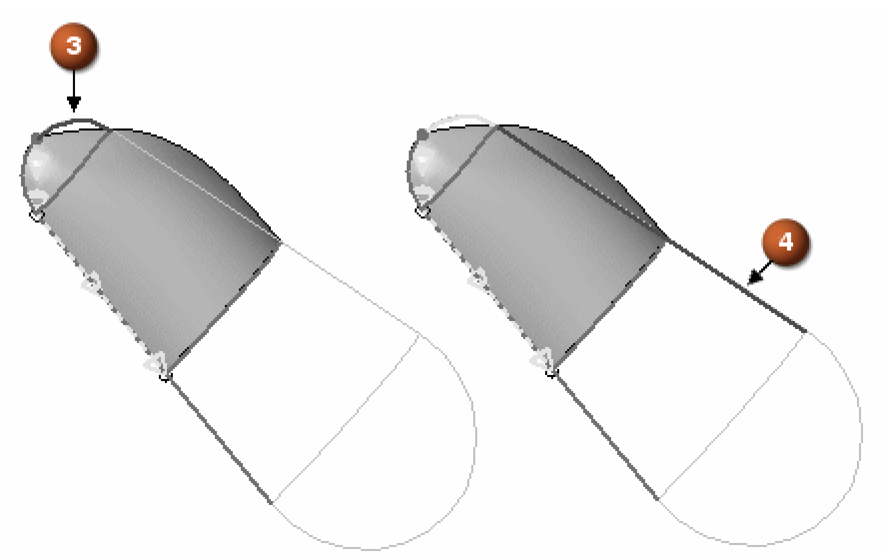
- Select sketch elements (1) and (2) as shown, and then right-click to complete the first guide curve.



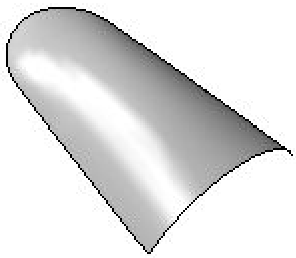
> **Note**
>
> Notice how the BlueSurf follows the guide curve.



- Select sketch elements (3) and (4) as shown, and then right-click to complete the second guide curve.



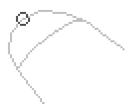
- Click **Next** and then **Finish**.



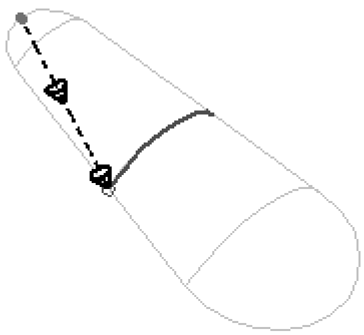
- Hide this second BlueSurf feature.

Create a third BlueSurf

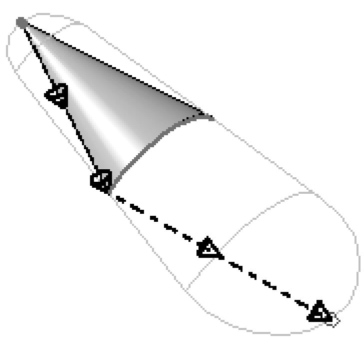
- Click the **BlueSurf** command.
- Click the point shown. Use **QuickPick** to select the vertex point.

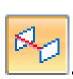


- Click the cross section shown and right-click.

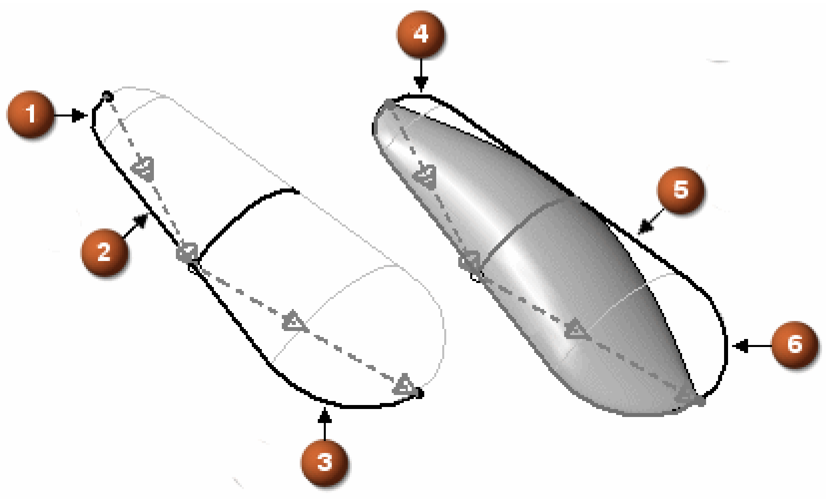


- Click the point shown and right-click. Use QuickPick to select the vertex point.

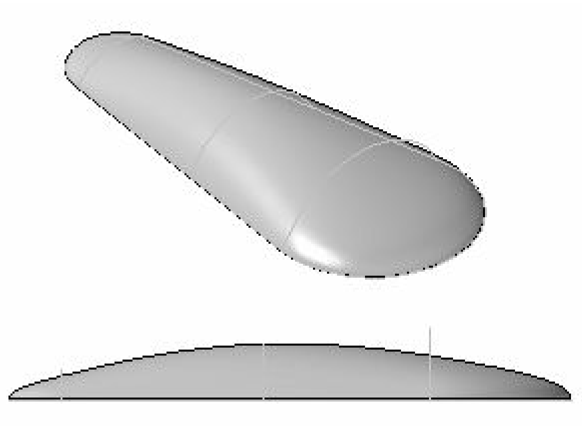


- Click the **Guide Curve Step** .
- From the command bar Select list, select **Single**.

- Select sketch elements 1, 2, and 3 as shown, and then right-click. Select sketch elements 4, 5, and 6 as shown, and then right-click.

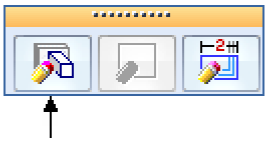


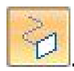
- Click **Next** and then click **Finish**.

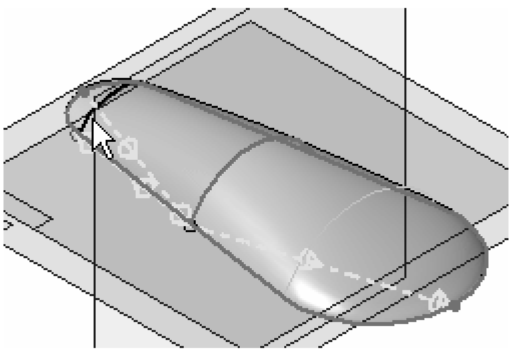


Add cross sections to the BlueSurf

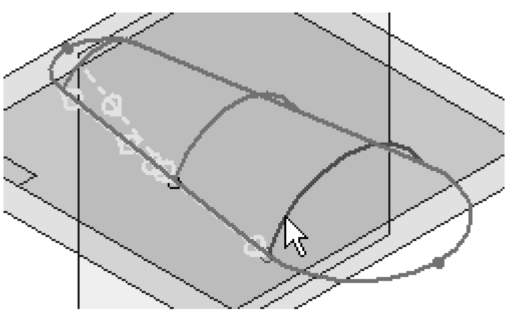
- Click the Select tool, and then select the BlueSurf feature.
- Click **Edit Definition**.



- Click the **Cross section** Step .
- Click the cross section shown and right-click.

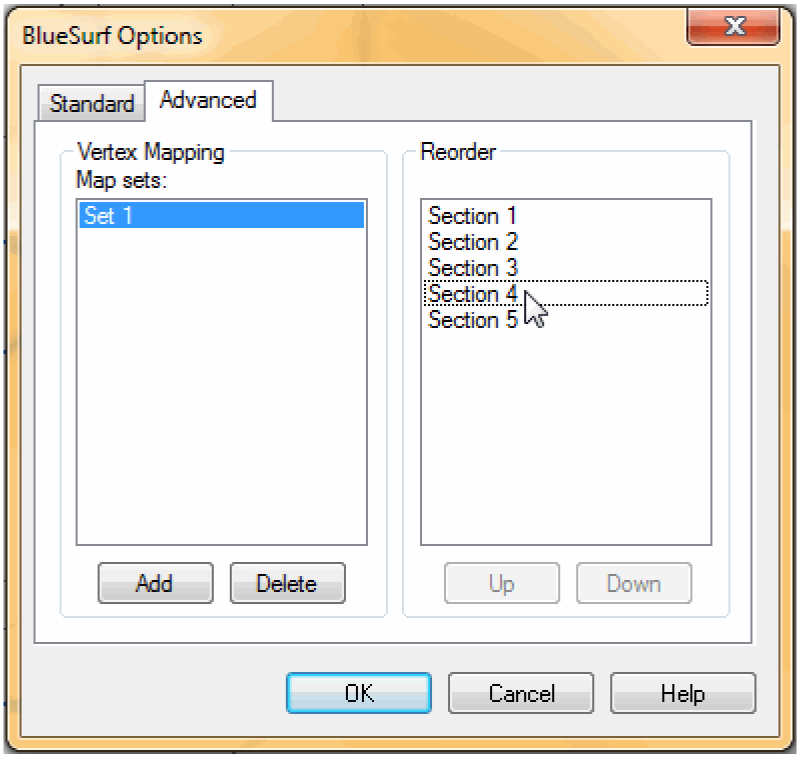


- Click **Edit** on the error dialog box. The cross section order is the cause for the error. This will be corrected later.
- Click the cross section shown and right-click.

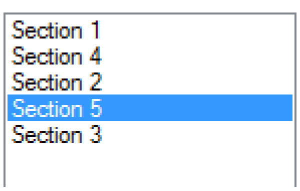


- Click **Edit** on the error dialog box. The cross section order is the cause for the error. This will be corrected later.
- Click the **BlueSurf Options** button.
- Click the **Advanced** tab in the **BlueSurf Options** dialog box.

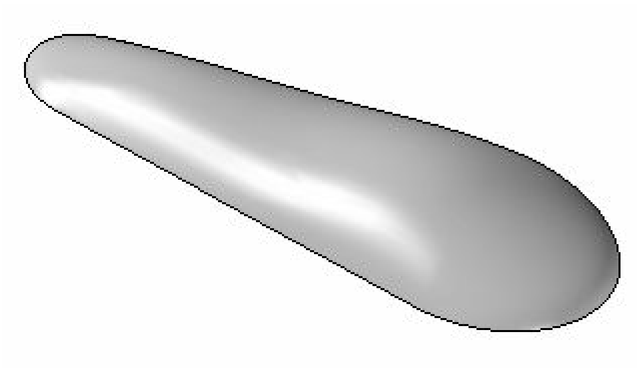
Notice the order of the Sections in the dialog box. Position the cursor over a section and it highlights in the graphics window.



- Select Section 4, and then click the **Up** button to reorder the section between Sections 1 and 2.
- Select Section 5, and then click the **Up** button to reorder the section between Sections 2 and 3.
- Click **OK**.



- Click **Finish**.

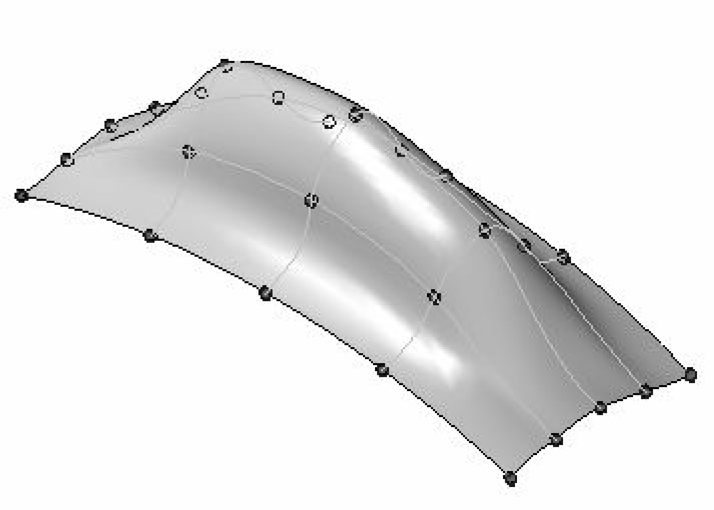


- The activity is complete.

Summary

In this activity you learned how to create and edit BlueSurf surfaces.

Activity: Creating and editing a BlueSurf



Overview

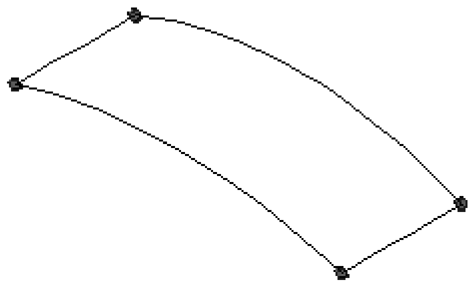
In this activity, you will learn to create and edit a BlueSurf. You will use provided curve sketches to create a BlueSurf.

Objectives

After completing this activity you will be able to:

- Create a BlueSurf.
- Insert sketches.
- Edit BlueDots.
- Dynamically edit curves.

Open the part file



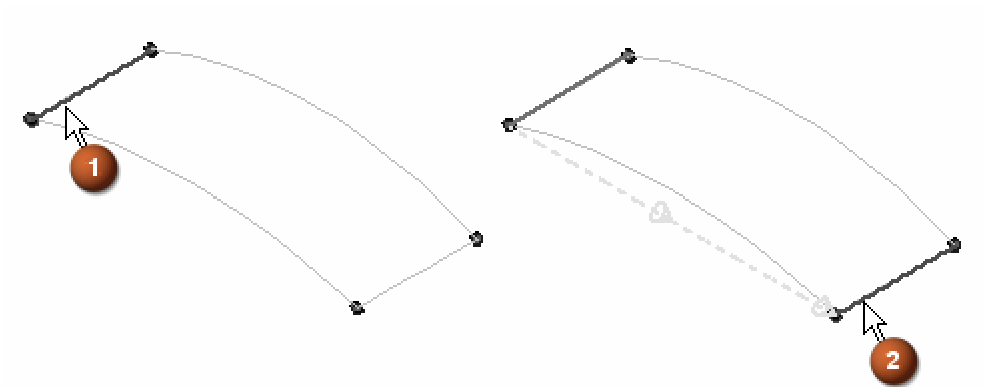
Note

The part file contains four curves that are BlueDot connected.

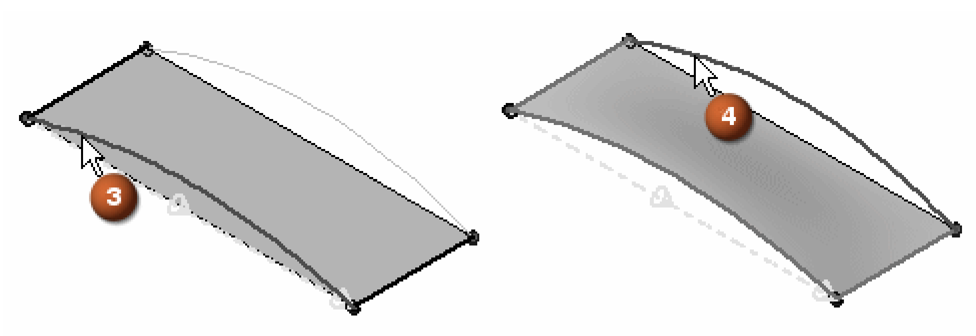
- Open **surface lab 3-04.par**.

Create a BlueSurf with guides

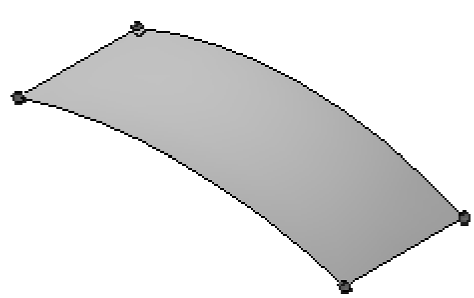
- Select **Surfacing** tab→**Surfaces** group→**BlueSurf** .
- Click curve (1) as shown for the first cross section and then right-click. Click curve (2) as shown for the second cross section and then right-click.



- Click the **Guide Curve** Step .
- Click guide curve (3) as shown and then right-click. Click guide curve (4) as shown and then right-click.



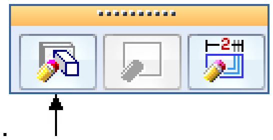
- Click **Next**, and then click **Finish**.



Insert sketches on the BlueSurf

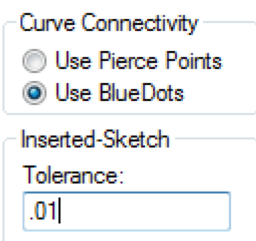
You can insert sketches to provide additional shape control.

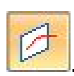
- Click the Select tool, and then select the BlueSurf feature.

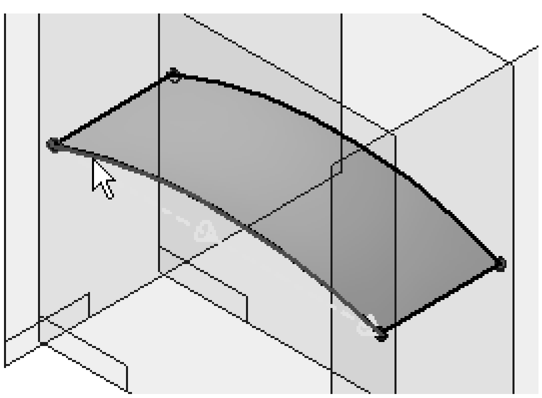


- Click **Edit Definition**.
- Click the **Options** button on the command bar.
- In the **BlueSurf Options** dialog box, under **Curve Connectivity** click **Use BlueDots**. Under **Inserted-Sketch**, in the **Tolerance** box, type **.01**. Click **OK**.

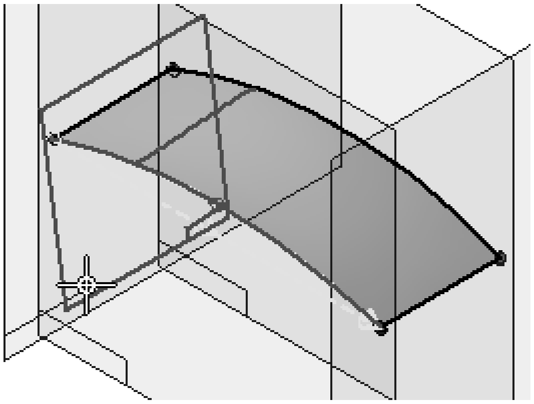
> **Note**
>
> The tolerance controls the number of edit points used on inserted sketches.

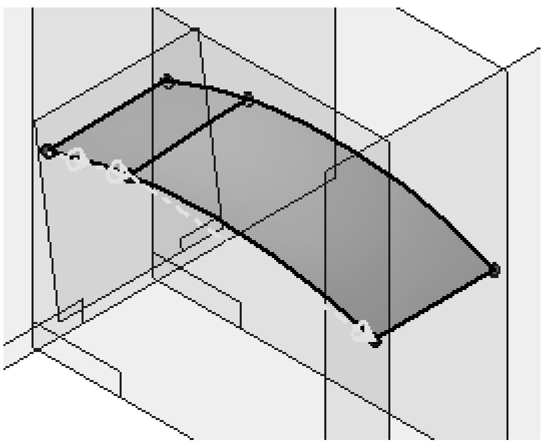


- Click the **Insert Sketch** Step button .
- Click the **Plane Normal to Curve** option.
- Click the curve shown.

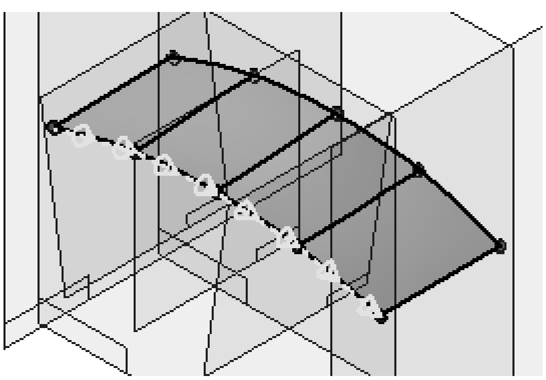


- Notice a normal plane is attached to the curve, which can be dynamically dragged along the curve. Drag the plane until the **Position** value is 0.25. You also can enter .25 on the command bar. Click to place the plane.

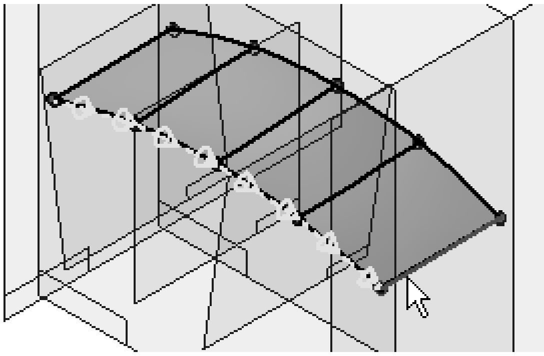


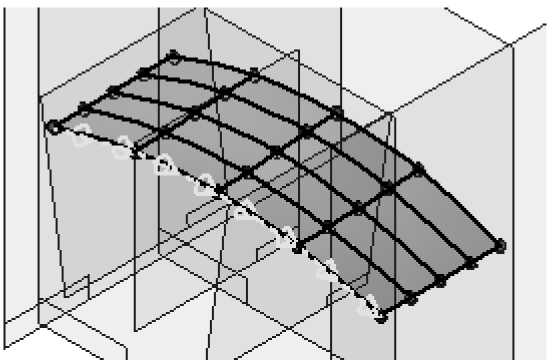


- Repeat the previous step to insert sketches at positions .50 and .25 (from the opposite end) as shown.

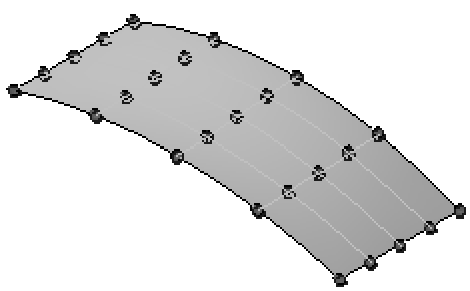


- Insert sketches normal to the cross section curve. Click the curve shown and insert sketches at positions .25, .50 and .25 (from the opposite end).



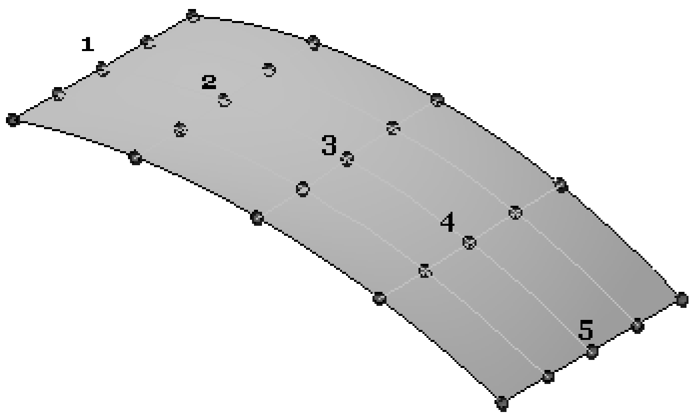


- Click **Finish** twice.

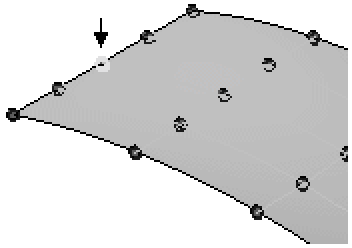


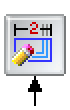
Perform BlueDot edits to change the shape of the surface

Edit the BlueDots along the center by changing their position in the Z-direction.

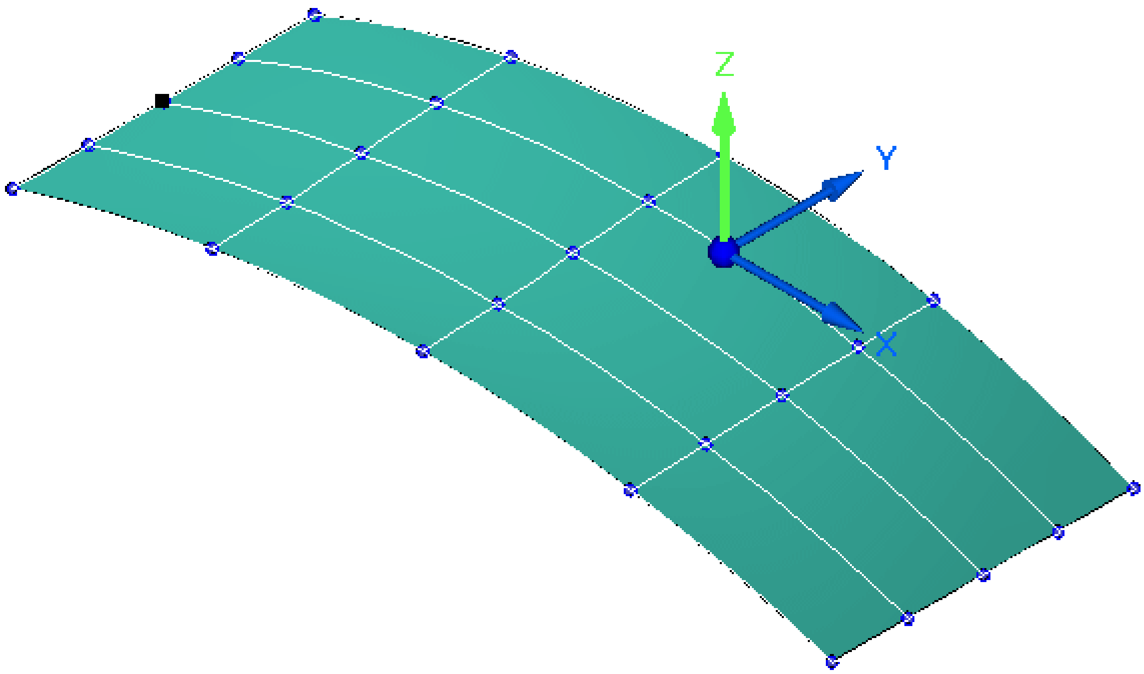


- Click the Select tool. Select the BlueDot shown and then click **Dynamic Edit**.





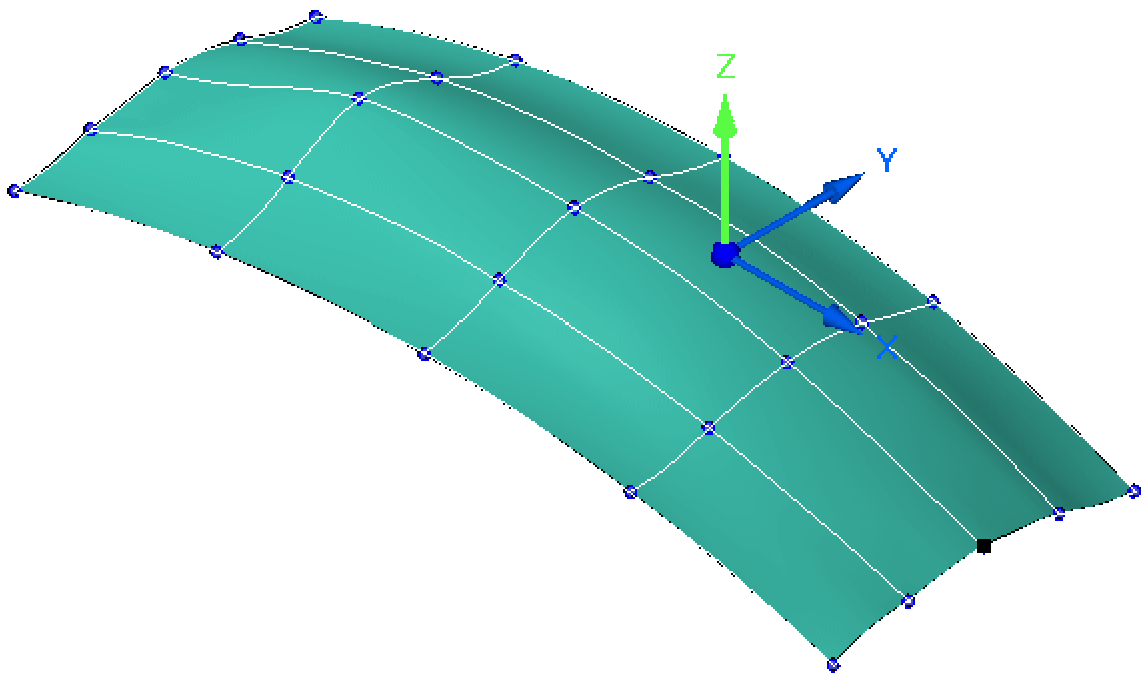
- Click the Z-axis on the 3D triad.



- Click the **Relative/Absolute Position** button .

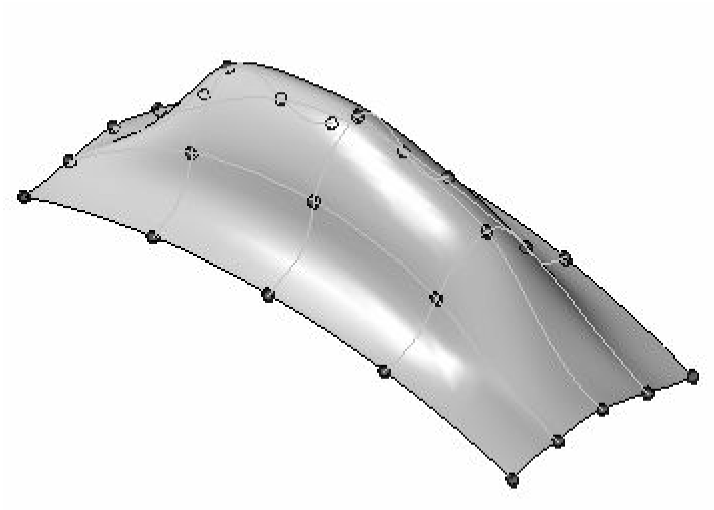
- In the dZ box, type **5**. Make sure Curves 1 and 2 are set to **Shape Edit**. Press Enter.

> **Note**
>
> If you press Enter again, the value is applied again.

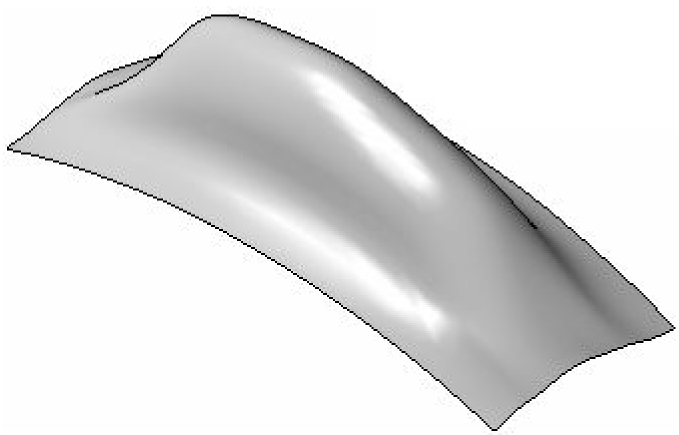


- Click in open space in the graphics window to exit the BlueDot edit.

- Repeat the previous step to edit BlueDots 2 through 5. Edit BlueDot 5 with a delta distance of 5. Edit BlueDots 2 through 4 with a delta distance of 10.



- Right-click in the graphics window. Turn off the display of sketches and BlueDots.



- The activity is complete. Save and Close this file.

Summary

In this activity you learned how to create and edit a BlueSurf.

Lesson review

Answer the following questions:

1. When do cross sections and guide curves have to be connected?
2. Name the two ways to edit a cross section or guide curve?
3. How do you add more cross sections to a BlueSurf?
4. How do you add more guide curves to a BlueSurf?
5. What happens to inserted sketches on a BlueSurf when the BlueSurf is deleted?
6. How do you get BlueDots on inserted sketches on a BlueSurf?
7. How do you turn off the display of BlueDots?

Lesson summary

You control surfaces by curve definitions. You change the surface shape by editing the underlying curves. You edit curves using **Dynamic Edit** or by editing the curve sketch or profile.

The extruded and revolved surfaces creation methods work similarly to the solid protrusion and revolved protrusion commands. These surfaces are useful in the development of more complex surfaces.

BlueSurf provides you the same results of a swept or lofted surface. However, BlueSurf provides much more control and editing capability. You can add cross sections and guide curves. You can control tangency at the start and end cross sections. You can use BlueDots or Pierce points to connect the inserted cross section and guides. Editing the BlueDots gives you real-time surface shape updates as they are moved.

Bounded surfaces are used to fill in gaps in a model. A bounded surface is created by selecting edges (curves) that form a closed loop. You have the option to make the resulting surface tangent to adjacent surfaces.

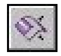
Lesson 51: Surface manipulation tools

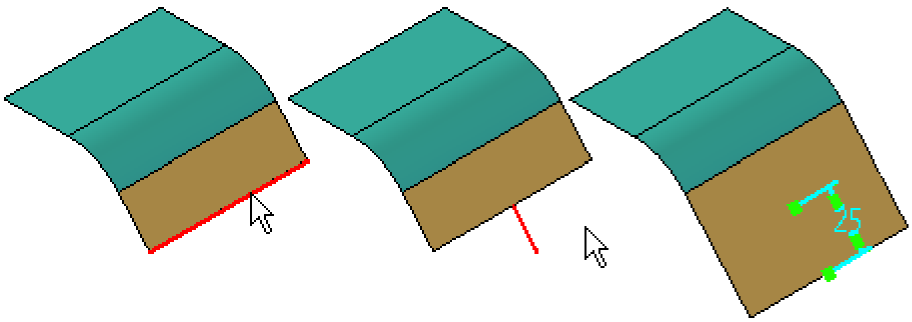
Objectives

After completing this lesson you will be able to use the surface manipulation commands:

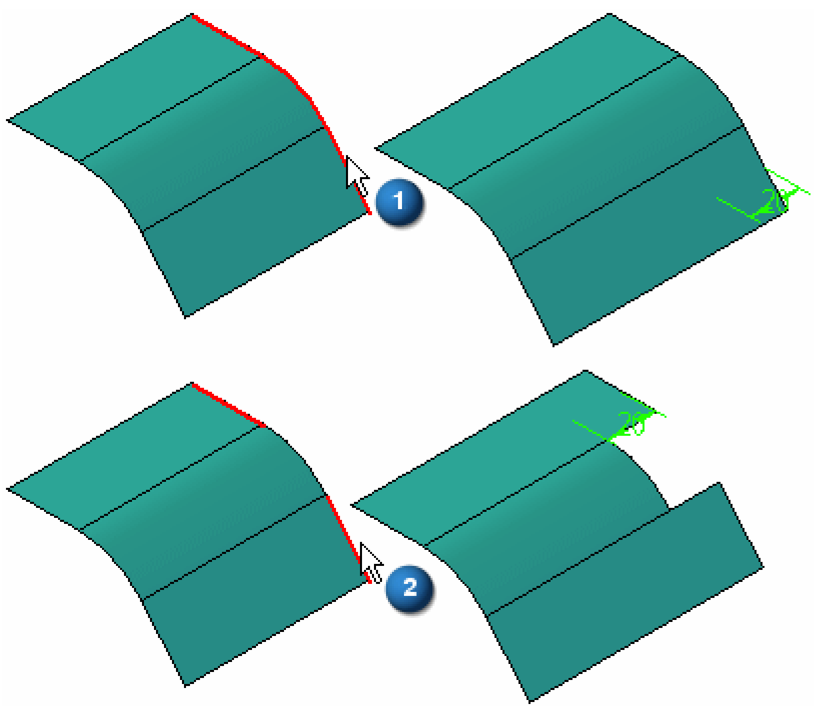
- Extend Surface
- Offset Surface
- Copy Surface
- Trim Surface
- Intersect Surface
- Delete Faces
- Stitched Surface
- Round
- Replace Face
- Split Face
- Parting Split
- Parting Surface

Lesson 52: Extend command

Use the **Extend** surface command  to extend a surface along one or more selected edges.



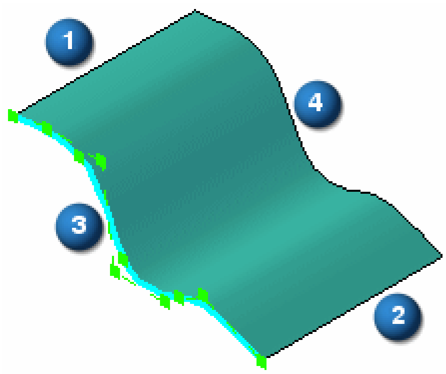
The selected edges can form a continuous chain (1) or be interrupted (2).



The extend options which are available depend on whether the surface is an analytic surface or a non-analytic surface. Examples of analytic surfaces include planes, partial cylinders, cones, spheres, and tori. Sweeping or extruding a b-spline curve creates non-analytic surfaces. Constructing lofted, swept or BlueSurf features using b-spline curves also creates non-analytic surfaces.

When extending a non-analytic surface, specify whether the extension is, Linear, Curvature Continuous or Reflective along certain types of edges. For example, when extending an extruded surface constructed using a b-spline curve, specify the Linear Extend, Curvature Continuous Extend, or Reflective Extend options for the two edges which are parallel to the input b-spline curve (1, 2).

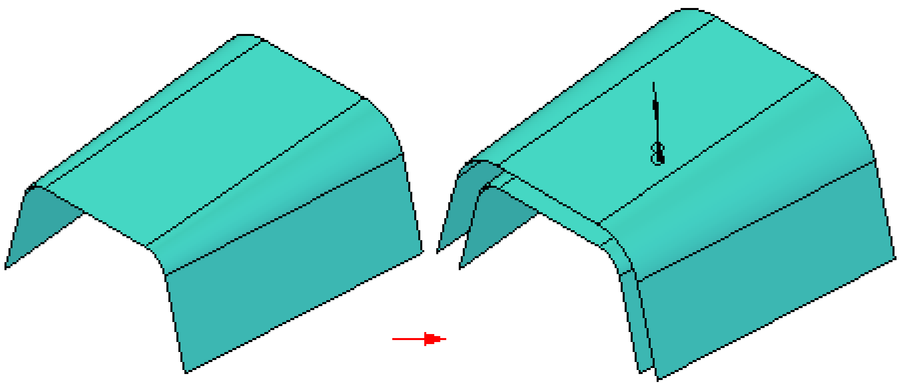
For the two edges which are perpendicular to the input b-spline curve (3, 4), only the **Curvature Continuous** option is mathematically possible.



Additional examples are illustrated in the Extend Surface command bar topic.

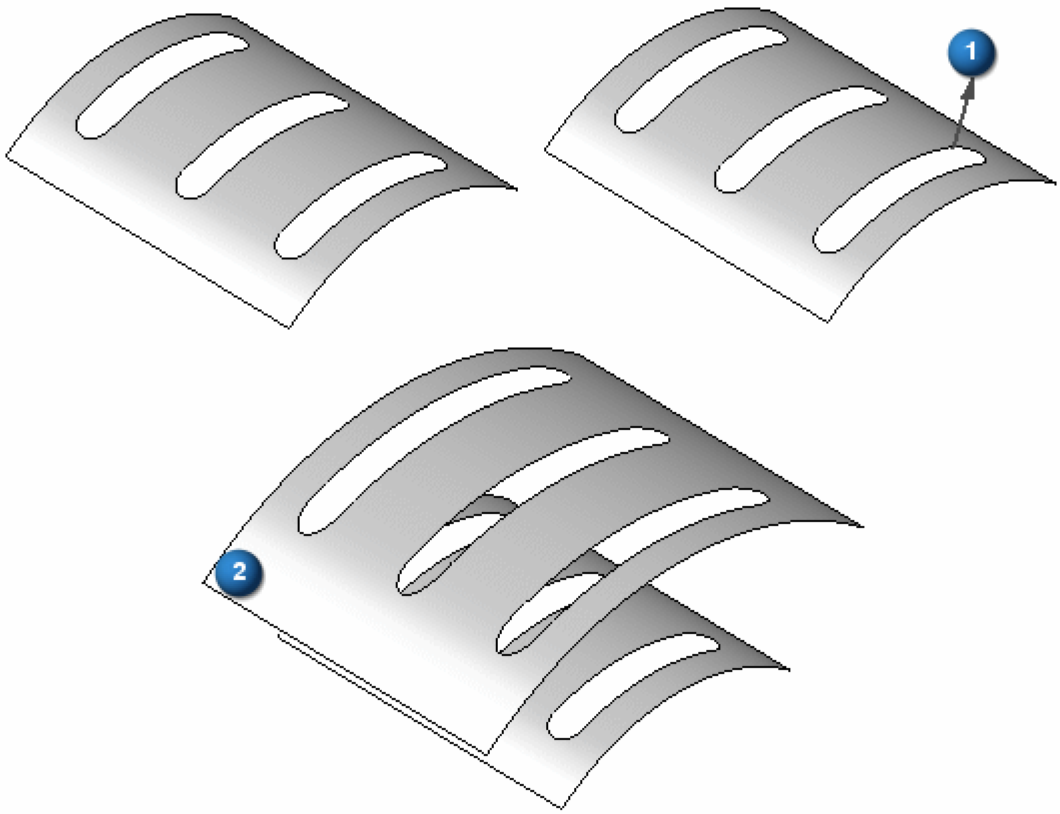
Lesson 53: Offset Surface command

Use the **Offset** surface command  to create a construction surface by offsetting a model face, a reference plane, or another construction surface. The new surface is offset a specified distance from the original surface, and is associative to it.

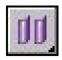


If the face or surface has boundaries, Offset Surface has options to remove or keep the boundaries on the offset surface.

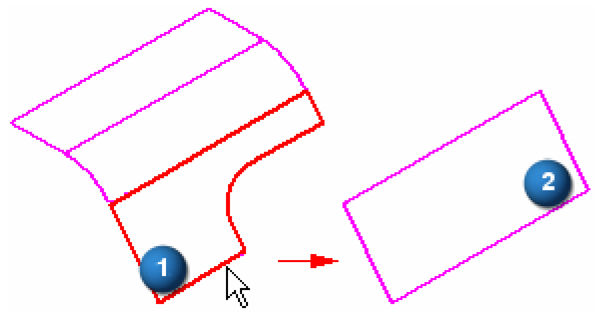
The following illustration shows an offset surface (2) offset in direction (1) with the show boundaries option on.



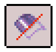
Lesson 54: Copy command

Use the **Copy** surface command  to create a construction surface feature that is derived from one or more input faces. The faces you select do not need to be adjacent to each other. You can specify whether any internal or external boundaries are removed on the new copy of the surface.

The following illustration shows surface face (1) copied with boundaries removed (2).

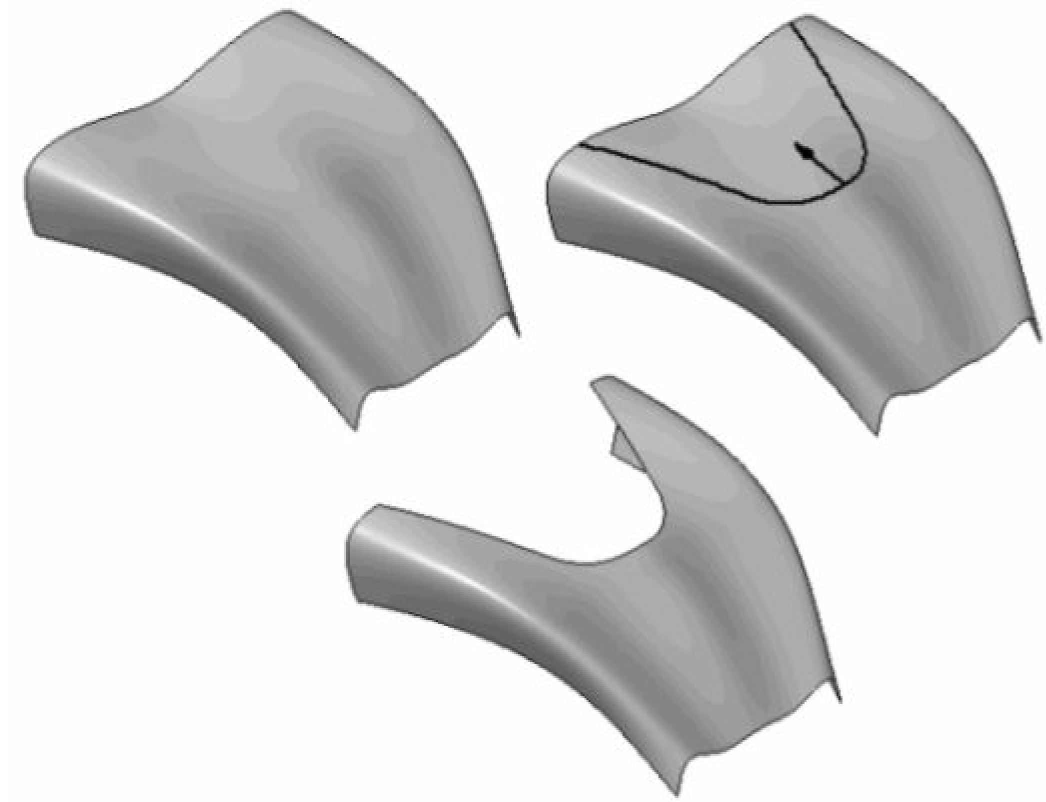


Lesson 55: Trim command

Use the **Trim** surface command  to trim one or more surfaces along the input element you define. You can use a curve, reference plane, or another surface as the input element.

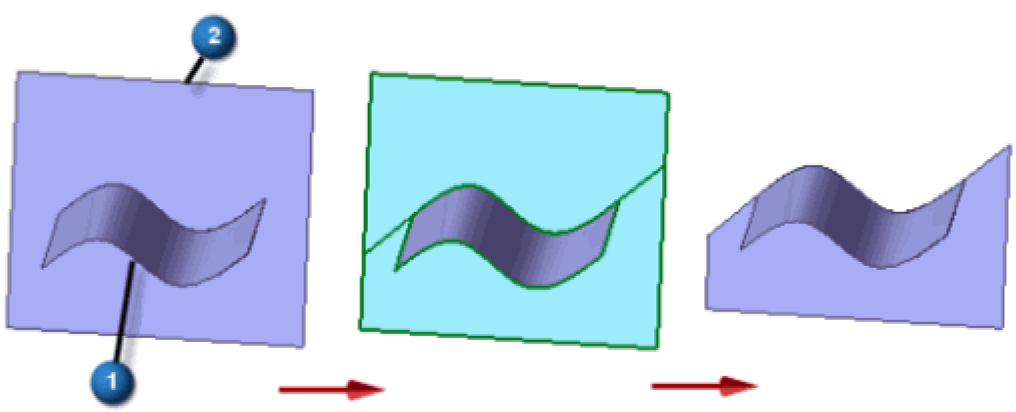
- If using curves,
  - They must lie on the surface you are trimming. Use the **Project** command to project the curve onto the surface first.
  - Closed curves that do not completely lie on the surface are not supported.
  - If using a surface as the trimming element the surface must touch or intersect the target surface.

•



- o If the curve or surface boundary does not extend to the edges of the target surface, the trim boundary element is extended linearly and tangent to the input element.

> **Note**
>
> Surface 1 is used to trim surface 2. Since surface 1 does not extend to the edges of surface 2, linear extensions are added to the trim boundary element. The input element you select as the trimming tool surface 1 is not modified.

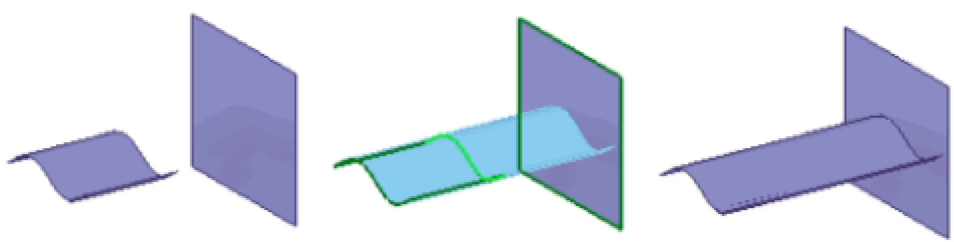


Lesson 56: Intersect command

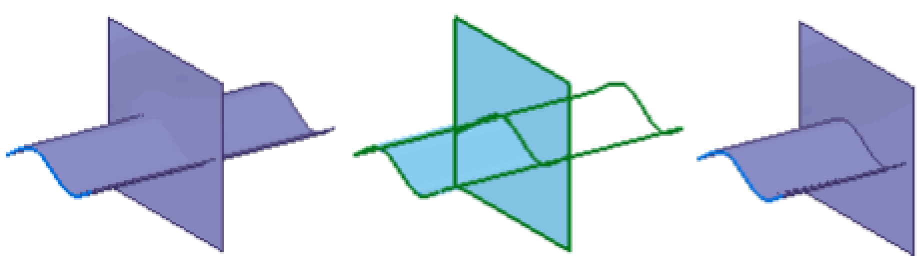
Use the **Intersect** surface command  to mutually extend or trim surfaces to a common intersection. You can extend surfaces up to other surface bodies. The trim capabilities support trimming multiple bodies. The intersect command can select only surface bodies, and all surface bodies act as both a target and tool.

> **Note**
>
> When selecting surface bodies, selecting only two bodies enables both Extend and Trim operations in the next step of the command. Selecting more than two bodies enables only the Trim operation.

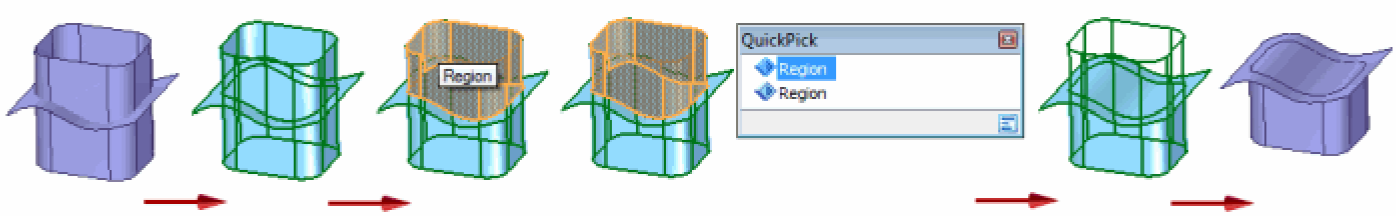
The example below shows a surface being extended.



The example below shows a surface being trimmed.



The example below shows a surface being trimmed via the region selection.

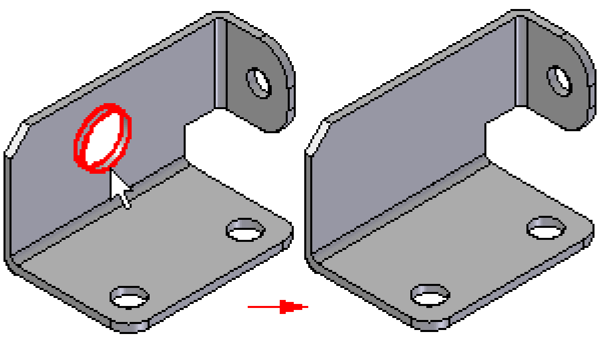


Lesson 57: Delete Faces command

> **Note**
>
> To access the **Simplify Model** environment, choose **Tools** tab→**Model** group→**Simplify**.

> **Note**
>
> To access the **Flatten Model** environment, choose **Tools** tab→**Model** group→**Flat Pattern**.

Use the **Delete Faces** command  to delete faces from the model.



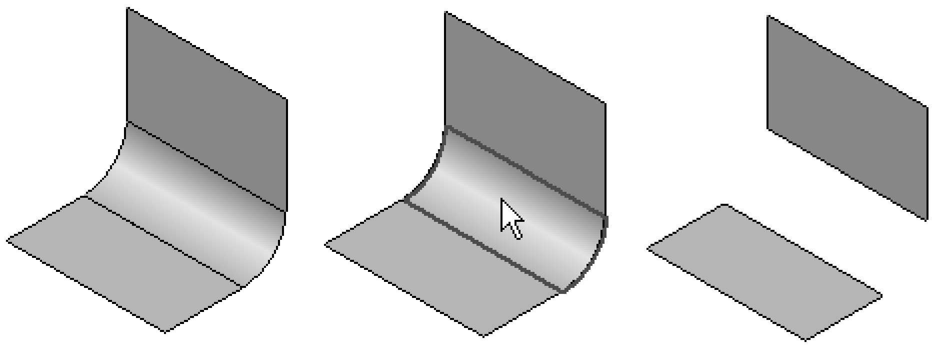
You can use this command to do the following:

- To remove faces from a design model to make design changes.
- To simplify a model in the Simplify Model environment so that it processes faster when used in an assembly.
- To remove faces from a sheet metal part when working in the Flat pattern environment.
- To remove faces from a construction body.

When you delete a face on a part body, which must always be a solid body, the gap created by the deleted surface is automatically closed.

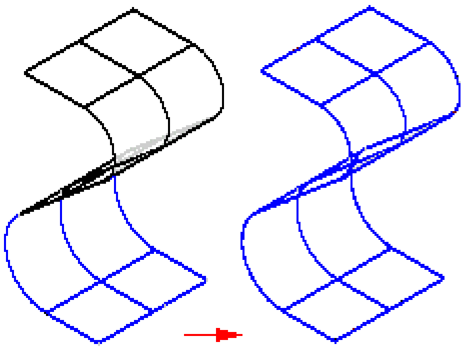
When you delete a face on a construction body, which is not required to be a solid body, you can specify whether the gap is closed or left open using the **Heal** option on the command bar.

When you clear the Heal option, the gap is not closed and you can construct another surface to close the gap. This can be useful when working with foreign data which cannot be converted into a solid body when you import it.



Lesson 58: Stitched command

Use the **Stitched** surface command  to stitch together multiple, adjacent construction surfaces to form a single construction surface feature.



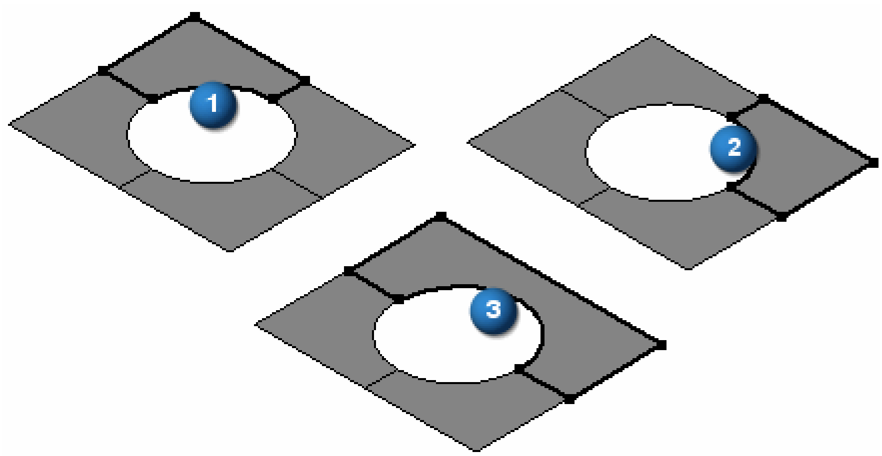
- This command is useful for joining imported surfaces.
- If the stitched surfaces form a closed volume, you have the option to designate the solid body as a base feature.
- You can set the stitched surface options for tolerance and surface healing on the **Stitched Surface Options** dialog box.
- Notice the default tolerance on the **Stitched Surface Options** dialog box. Once you turn on the **Heal** option, you can change this value if the edges of two surfaces being stitched together do not meet the default tolerance.

Tips:

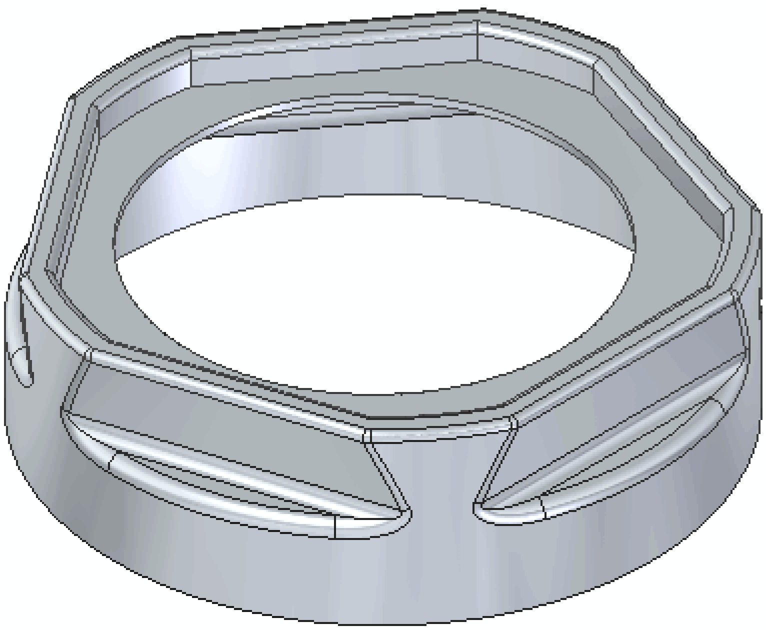
- To remove surfaces from the select set, select the surfaces while pressing Shift.
- To delete the link between the stitched surface feature and its parents, use the **Drop Parents** command on the shortcut menu. This command reduces the amount of data in the file. Once you drop the parent information, the stitch surface feature can no longer be edited.
- You can use the commands on the shortcut menu to display, hide, edit, rename, or recompute the stitched surfaces.
- If the output forms a closed volume, a solid body will be created. Otherwise, the stitch surface will be a sheet body with free edges that can be stitched to other surfaces.
- If the stitched surfaces result in a solid body and there is no base feature in the file, the **Make Base Feature** command becomes available on the shortcut menu, and you can make the stitched body the base feature for the part.

To show the stitchable edges on construction surfaces, choose **Surfacing** tab→**Surfaces** group→**Show Non-Stitched Edges**  located on the **Stitched Surface** command list.

The illustration below shows the stitchable edges for surface (1) and surface (2). Surfaces (1) and (2) were stitched together to produce (3) and the stitchable edges are shown.



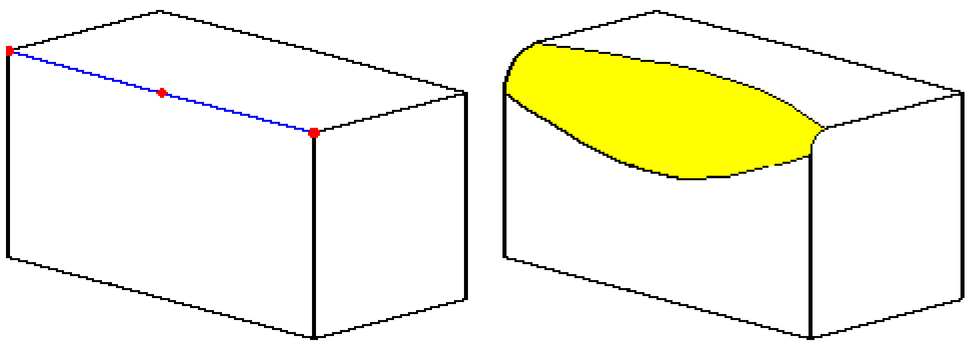
Lesson 59: Round



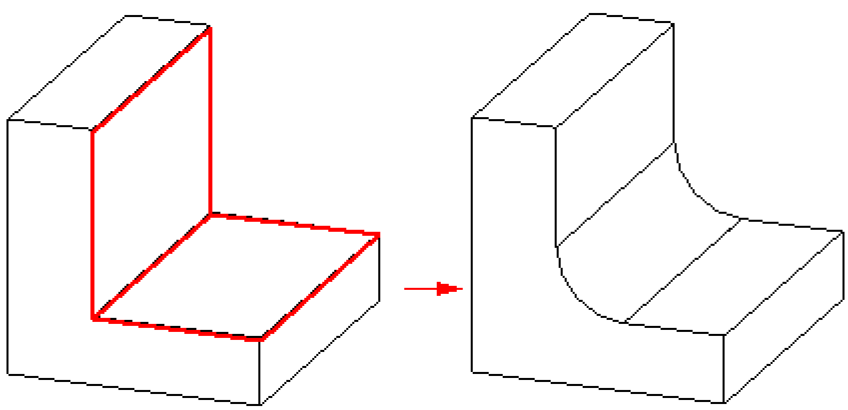
You can use the **Round** command to place fillets and rounds on surface edges or between two adjoining surfaces.

Lesson 60: Blend command

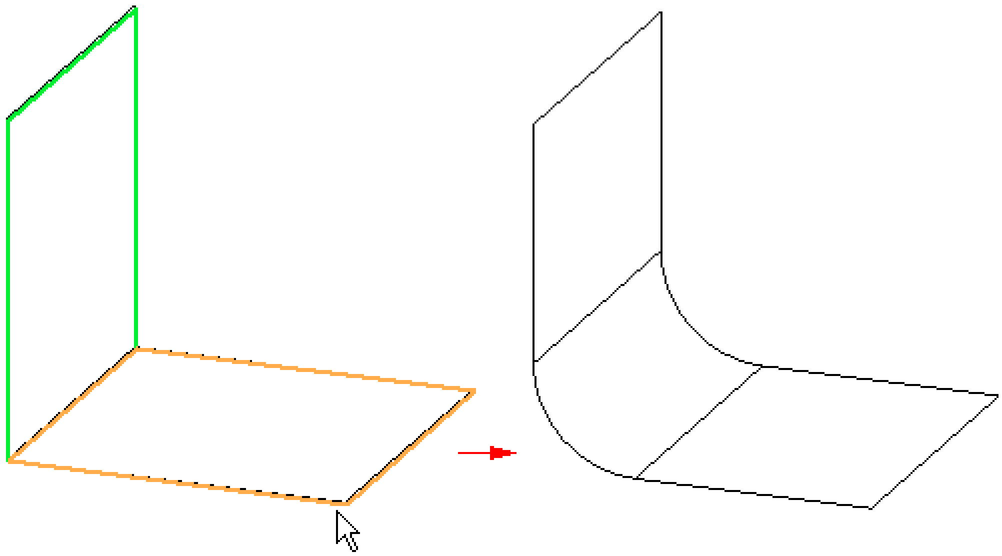
Use the **Blend** command  to create a variable radius round,



a blend between faces,



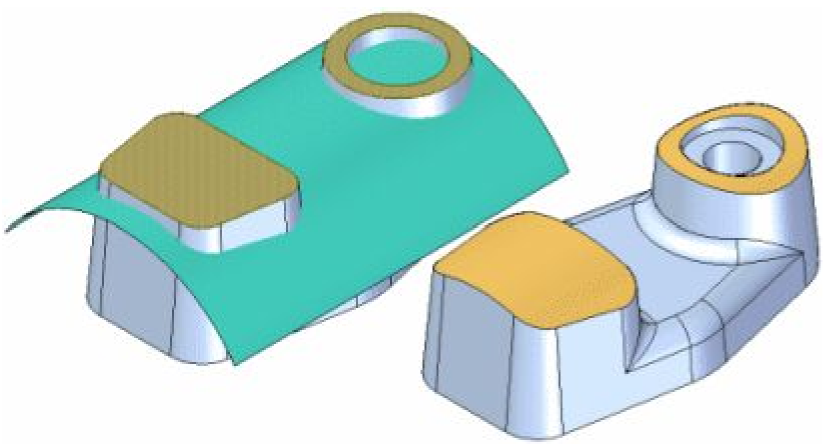
or a blend between surface bodies.



Lesson 61: Replace Face command

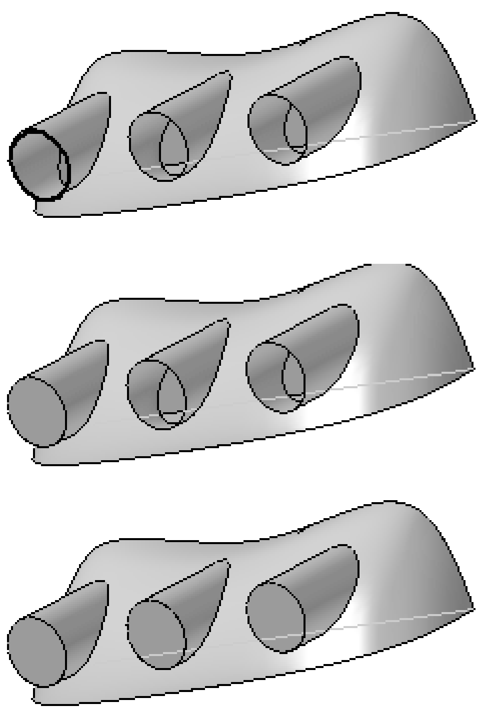
Use the **Surfacing** tab→**Modify Surfaces** group→**Replace Face** command  to replace selected faces on a part. The replacement face can be a construction surface, a reference plane, or another face on the part. When replacing more than one face, the faces being replaced cannot touch each other.

When you replace a face using a construction surface, the construction surface is hidden automatically when you finish the feature.



If edges on the face you are replacing have rounds applied, the rounds are reapplied after you complete the replace face operation.

Lesson 62: Activity: Surface manipulation



Overview

In this activity you will learn to use the surface manipulation commands.

Objectives

After completing this activity you will be able to:

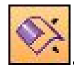
- Extend a surface.
- Offset a surface.
- Trim a surface.
- Copy a surface.
- Delete faces of a surface.
- Stitch surfaces together.
- Round surfaces.

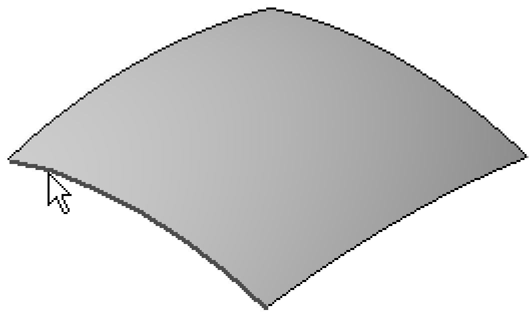
- Replace a face on a solid body.

Lesson 63: Open the part file

- Open **surface lab 4-01.par**.

Lesson 64: Extend a surface

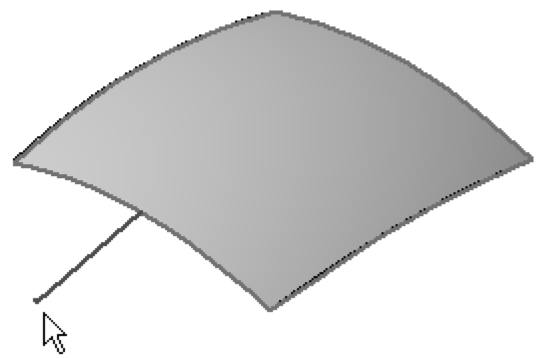
- Select **Surfacing** tab→**Surfaces** group→**Extend** .
- Select the edge shown and click **Accept**.

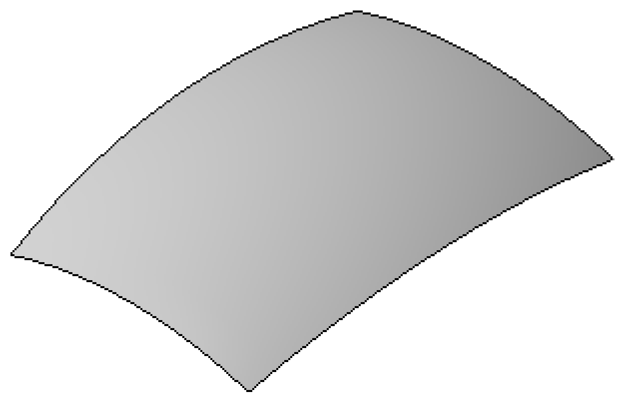


Note

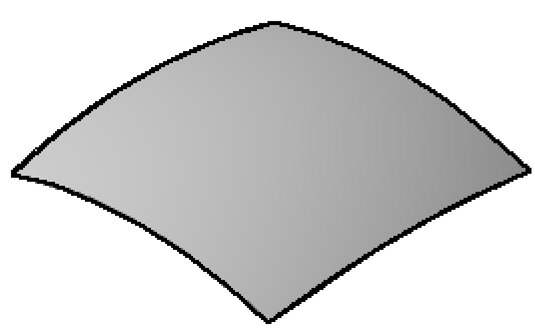
Notice on the **Extend** surface command bar the options for **Linear**, **Curvature Continuous**, and **Reflective Extent**. The **Linear** option extends the surface in a linear direction. The **Curvature Continuous** option extends the surface to follow the same curvature of the select edge. The **Reflective Extent** option specifies that the extended portion of the surface is a reflection of the input surface. This option is not available for analytic surfaces.

- Use the **Curvature Continuous** extent option and drag the distance vector approximately as shown, and click.

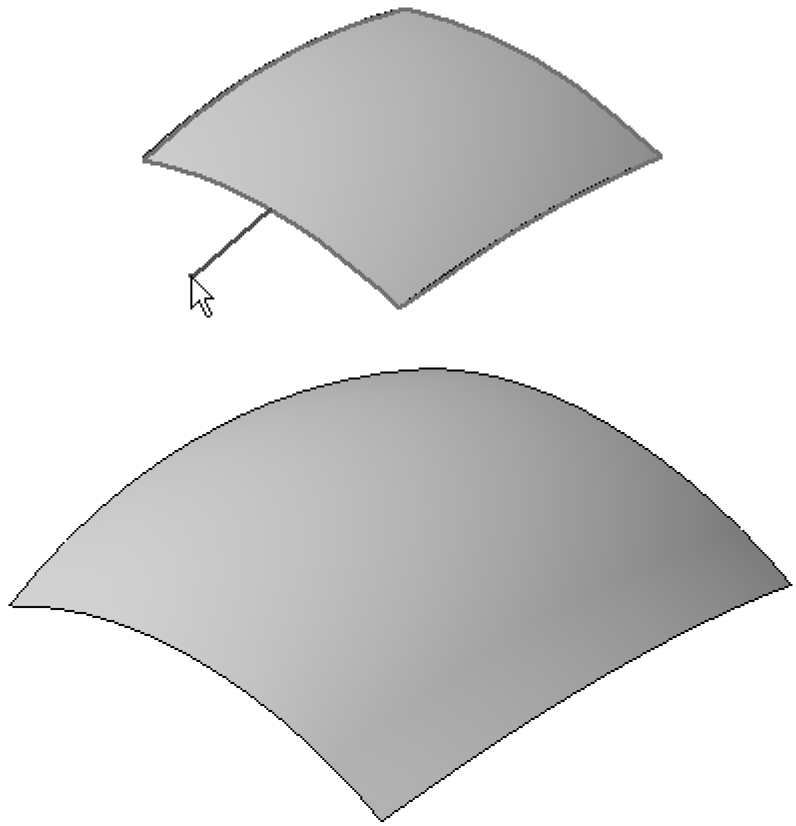




- Click **Finish**.
- Click the Select tool.
- In **PathFinder**, select the new Extend feature and press Delete.
- Multiple edges can be extended. Select the **Extend** command again, select all four edges, and click **Accept**.



- Drag the distance vector approximately as shown and click.

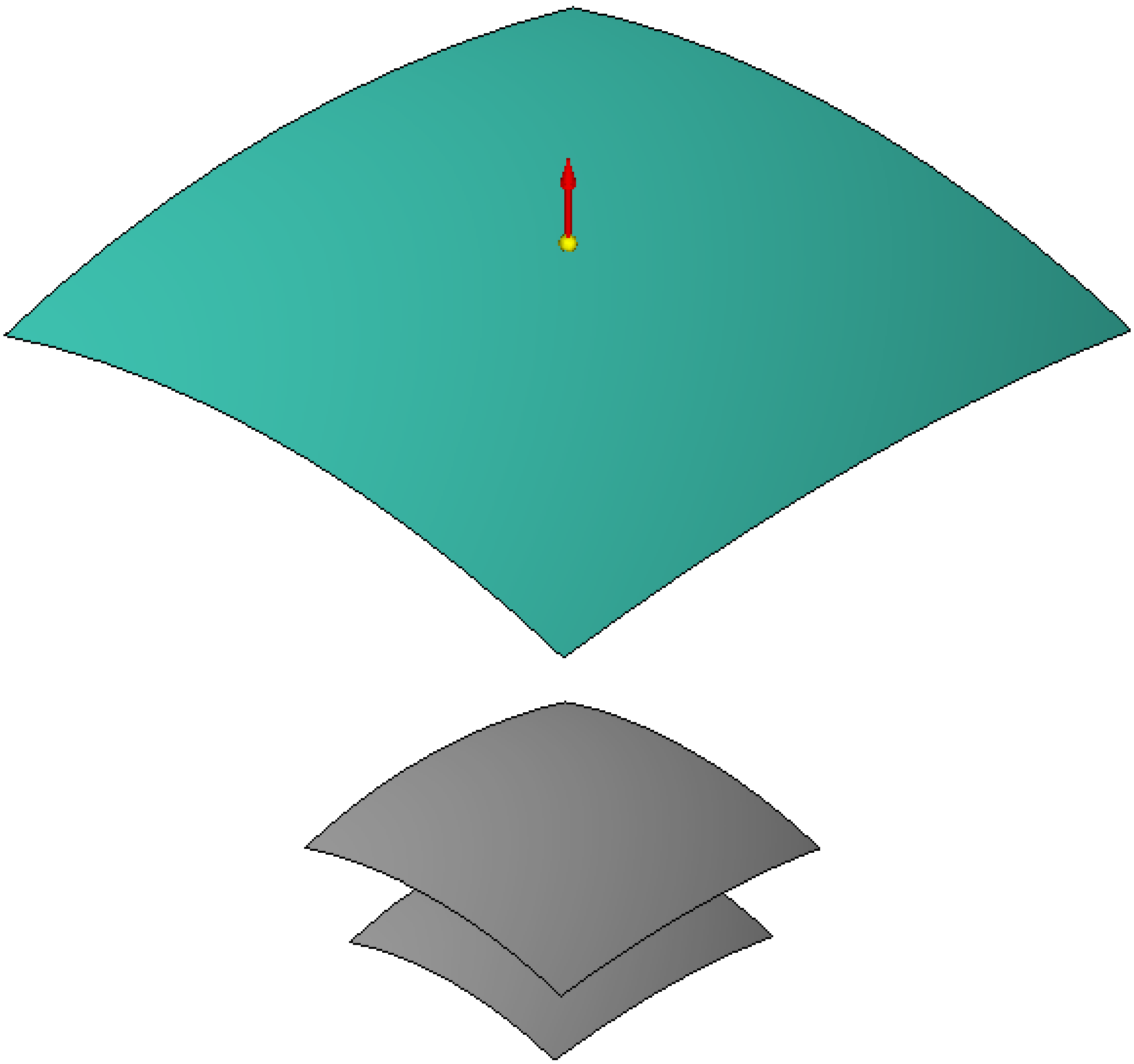


- Click **Finish**.
- Click the Select tool.
- In **PathFinder**, delete the Extend feature.

Lesson 65: Offset a surface

- Select **Surfacing** tab→**Surfaces** group→**Offset** .
- Click the surface, and then click **Accept**.
- Type **50** in the **Distance** box and press Enter.

- Position the direction arrow as shown, and click.

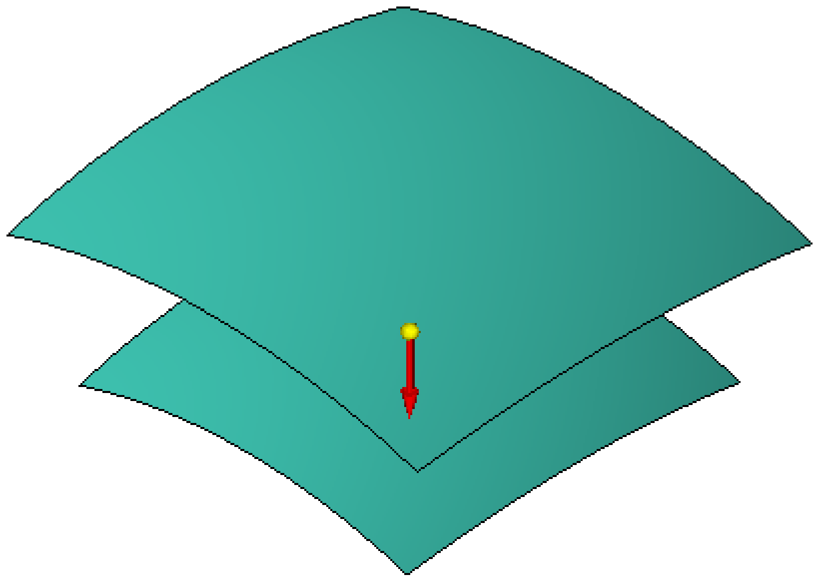


Note

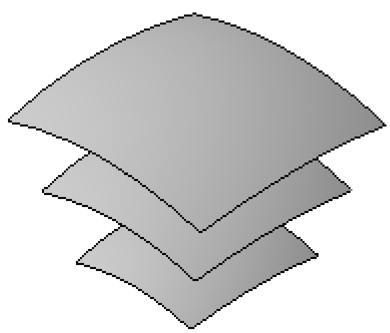
The offset surface is offset along normal vectors from the input surface a distance of 50 mm.

- Click **Finish**.

- Create another surface offset from the original (bottom) surface. Use 50 for the offset distance, and position the direction arrow downward as shown.

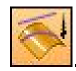


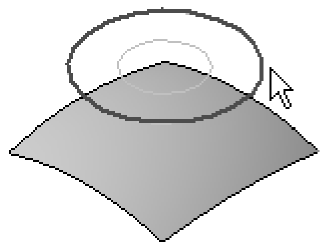
- Click **Finish**.



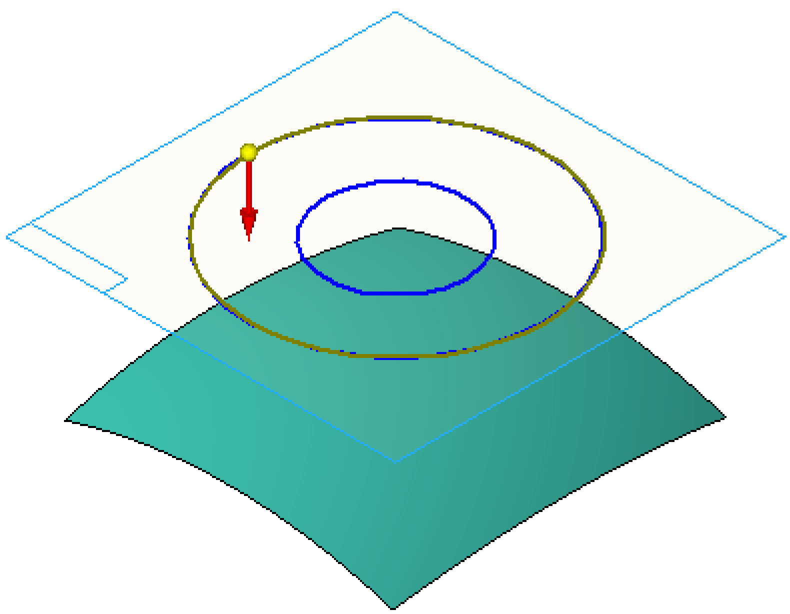
- Click the Select tool.
- In **PathFinder**, delete the two offset surfaces.

Lesson 66: Project a curve onto a surface

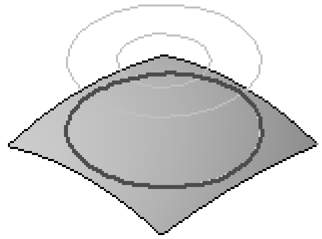
- In **PathFinder**, show *Sketch B*.
- Select **Surfacing** tab→**Curves** group→**Project** command .
- Select the circle shown below and then click the **Accept** button.



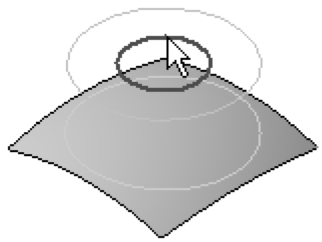
- Select the surface and click **Accept**. Position the direction arrow as shown and click.



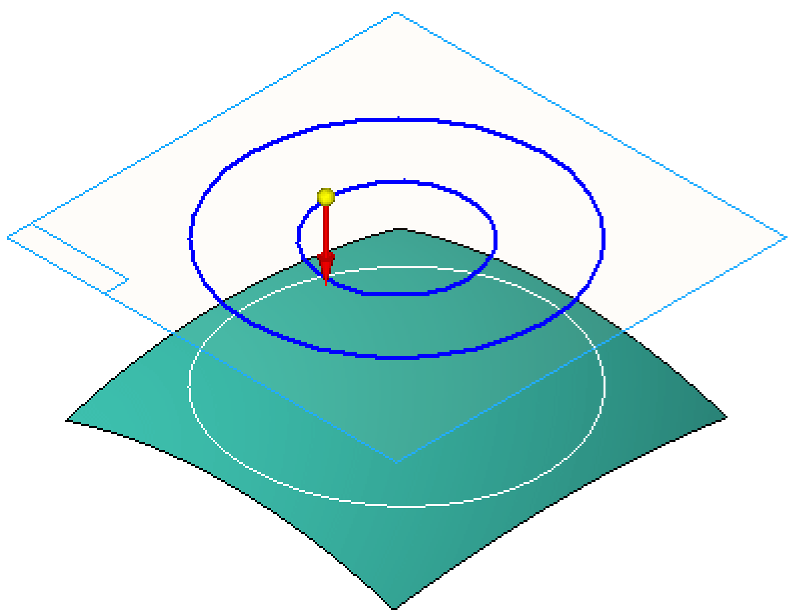
- Click **Finish**.



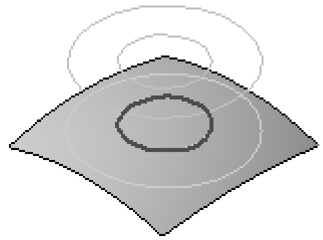
- The **Project** command is still active. Select the circle shown below and then click **Accept**.



- Select the surface and click **Accept**. Position the direction arrow as shown and click.



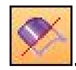
- Click **Finish**.

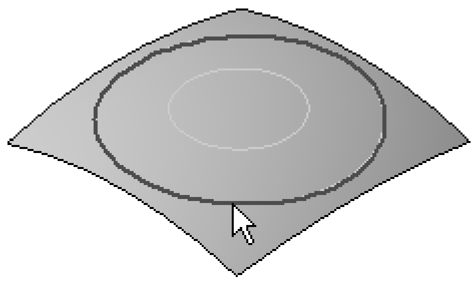


- In **PathFinder**, hide Sketch B.

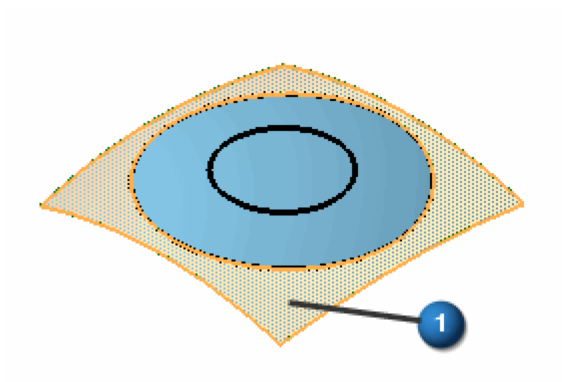
Lesson 67: Trim a surface

Trim surface is used extensively in surface modeling.

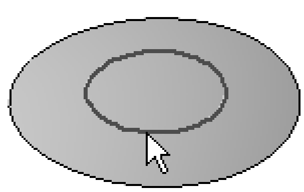
- Select **Surfacing** tab→**Modify Surfaces** group→**Trim** .
- Select the surface and click **Accept**.
- Select the projection curve as shown and click **Accept**.



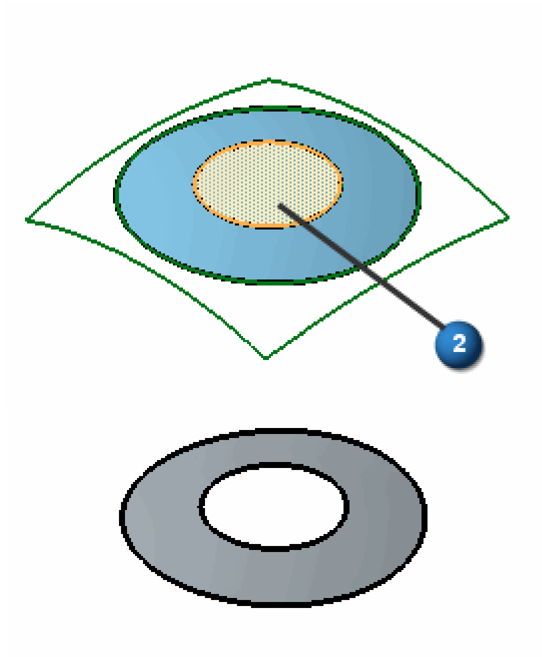
- Select the region labeled 1 as shown to trim the surface outside the projection curve.



- Click **Finish**.
- Select the surface and click **Accept**.
- Select the projection curve as shown and click **Accept**.



- Select the region labeled 2 as shown to trim the surface inside the projection curve.

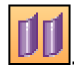


- Click **Finish**.

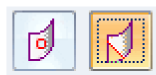
  Notice the two Trim features in **PathFinder**.

- Right-click and **Hide All→Curves**.

Lesson 68: Copy a surface

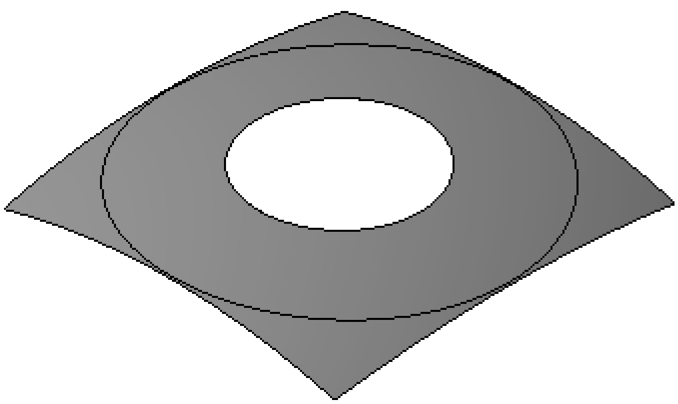
- Select **Surfacing** tab→**Surfaces** group→**Copy** .

  Notice on the **Copy** surface command bar the two options for removing boundaries. To remove internal boundaries on the copied surface, select the left button. Right-click to remove external boundaries on the copied surface. If neither option is selected, the copied surface maintains all boundaries.

- Click the **Remove External Boundaries** button.

- Select the surface and then click **Accept**.

- Click **Finish**.

  Notice the Copy feature in PathFinder.

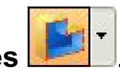
- Click the Select tool.

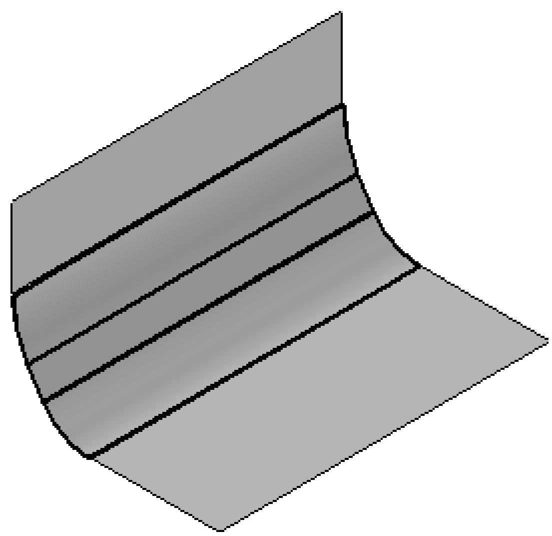
- In **PathFinder**, delete the following features: *Projection 11*, *Projection 12*, *Trim 11*, *Trim 12*, and *Copy 7*.

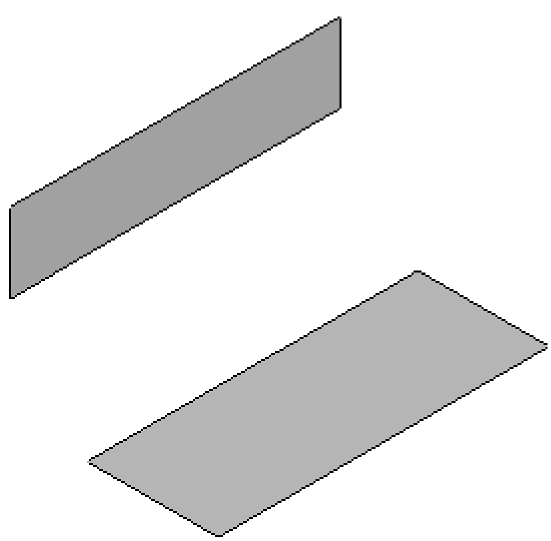
- Hide feature *Sweep A*.

Lesson 69: Delete faces

. Faces on a construction body can be deleted and replaced with a new surface.

- In **PathFinder**, show feature *Extrude 2*. Choose the **Fit** command.
- Select **Home** tab→**Modify** group→**Delete Faces** .
- Select the three faces shown and click **Accept**.

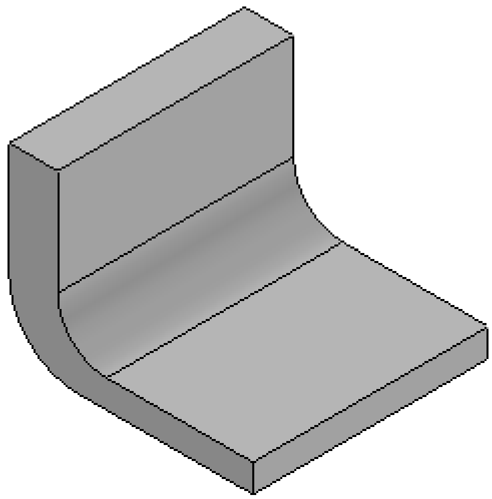




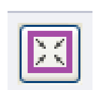
- Click **Finish**.

  Notice the Delete Face feature in **PathFinder**.

- Hide feature *Extrude 2*. Show feature *Extrude 3*. The Extrude 3 feature was constructed with a closed profile and the ends were capped.

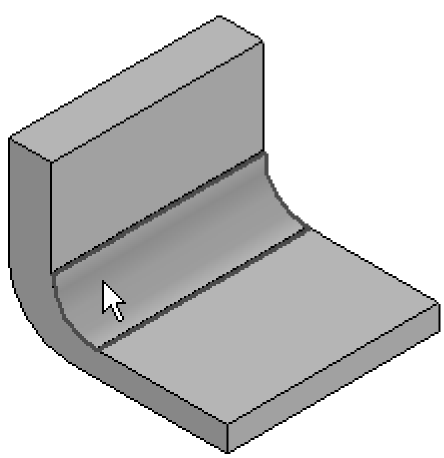


- Click the **Delete Faces** command.

- Click the **Heal Option** button  on the command bar.

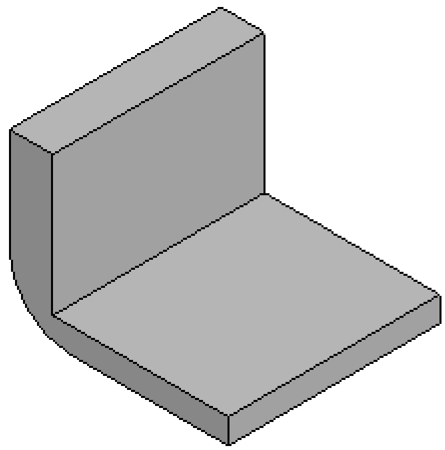
> **Note**
>
> For the heal option to work, the construction body must be closed.

- Select the surface shown and then click **Accept**.

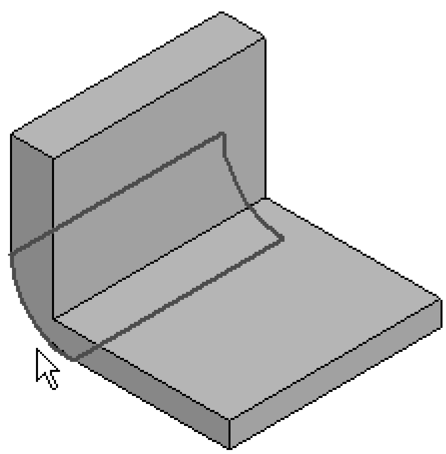


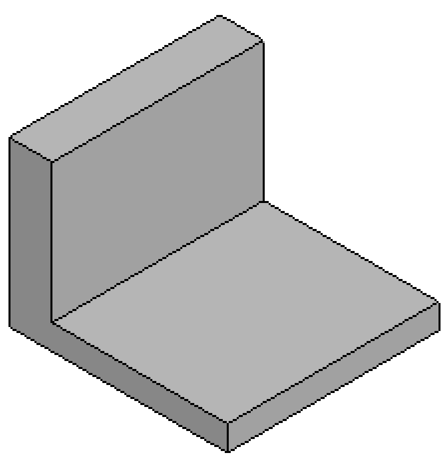
- Click **Finish**.

  Notice the result. The face was deleted and the two adjoining faces adjust to fill the gap. The two end caps were also modified.



- Repeating the previous step, delete the face shown.

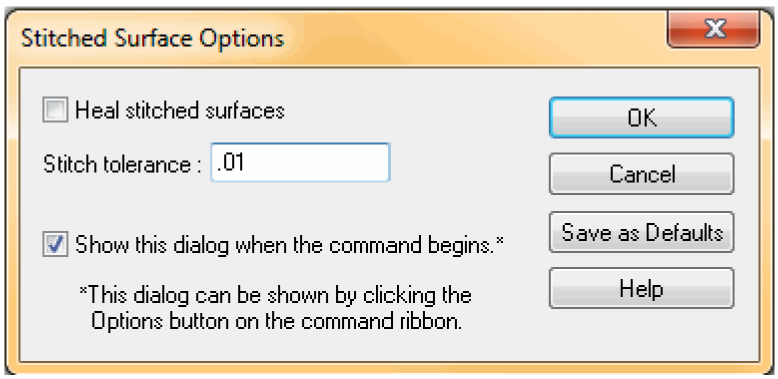




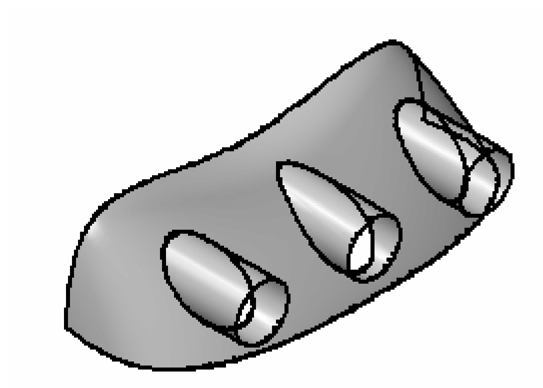
- Click **Finish**.
- Click the Select tool.
- Hide *Extrude 3*.

Lesson 70: Stitch surfaces

- In **PathFinder**, show features *BlueSurf 1, BlueSurf 2, BlueSurf 3*, and *BlueSurf 4*.
- Select **Surfacing** tab→**Modify Surfaces** group→**Stitched** .
- Type **.01** in the **Stitch tolerance** box and then click **OK**.



- Select all four surfaces and then click Accept.

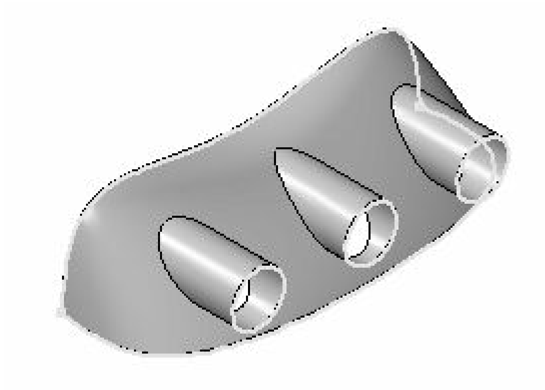


- Click **Finish**.

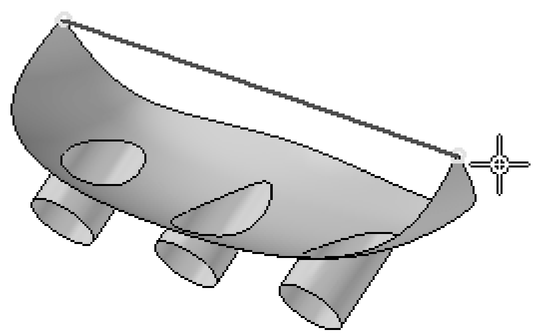
  Note in PathFinder the Stitch feature.

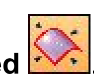
- Select **Surfacing** tab→**Modify Surfaces** group→**Stitched** list→**Show Non-Stitched Edges** .

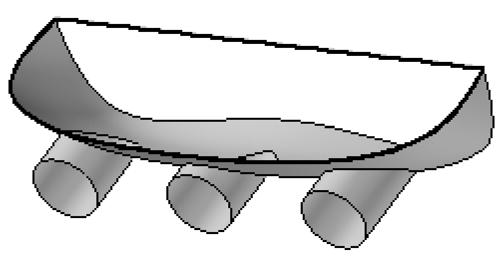
- Click the stitched surface. Notice the highlighted edges. These are the unstitched edges.



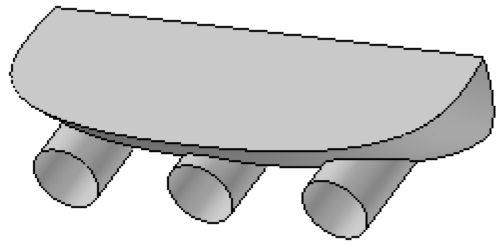
- Click **Close** on the command bar.
- In order to make a solid feature there must be surfaces stitched to all of the non-stitched edges. Create surfaces needed to make a solid feature. Select **Surfacing** tab→**Curves** group→ **Keypoint Curve** .
- Draw a keypoint curve as shown. The curve has two points.



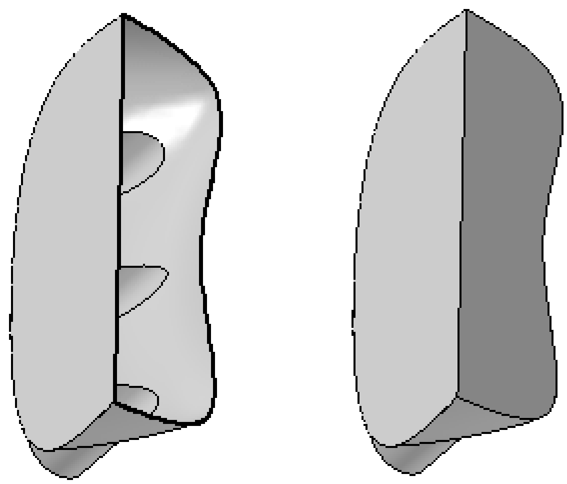
- Create five bounded surfaces. Select **Surfacing** tab→**Surfaces** group→ **Bounded** .
- On the command bar, click the Select: **Single** option.
- Select the edges shown and click Accept.



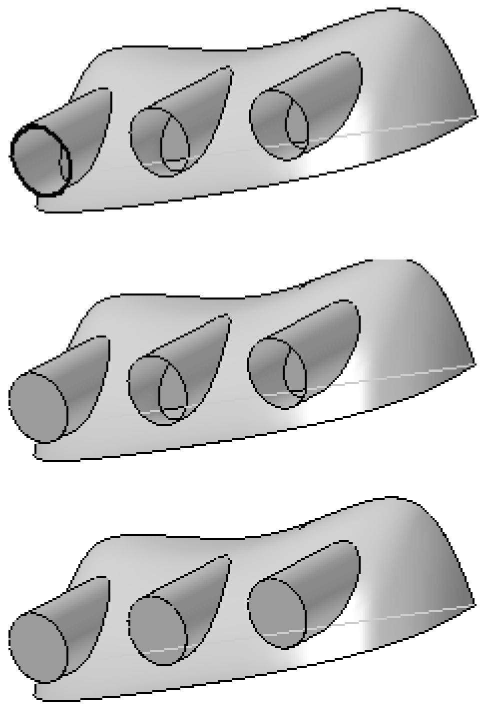
- Click **Finish**.



- Repeat the same steps to create the bounded surface shown.



- Repeat the steps for the three circular edges.



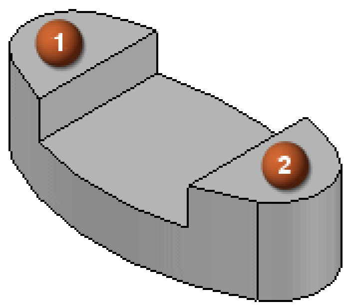
- Select the **Stitched** command. Type **.01** in the **Stitch tolerance** box and click **OK**.
- Select the stitched surface and then select the five bounded surfaces.
- Click the **Accept** button. Since there are no non-stitched edges remaining, the stitched surfaces result in a solid body. Click **OK** in the message dialog box.
- Click **Finish**.

> **Note**
>
> If there is no base feature (solid) in the file, you can right-click on the stitched surface and click **Make Base Feature** to create a solid of the stitched surface.

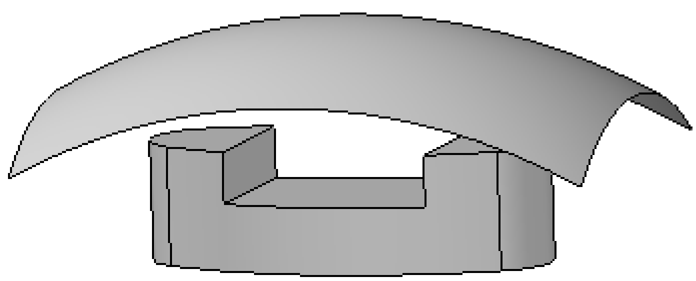
- Click the Select tool.
- Hide features BlueSurf 1-4 and the keypoint curve.

Lesson 71: Replace a face

- Select **View** tab→**Show** group→**Construction Display**. In the **Show All/Hide All** dialog box, choose **Show All**→**Design body**. *Protrusion A* displays. Replace faces (1) and (2) on Protrusion A with a construction surface.

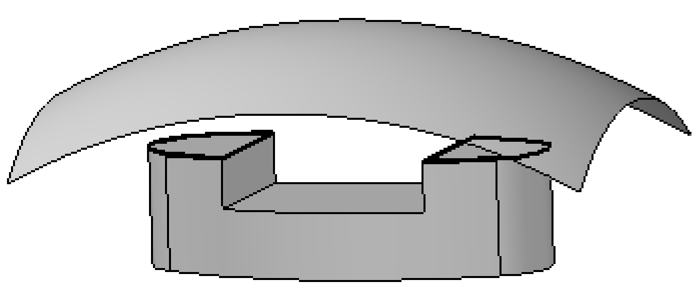


- Show feature *BlueSurf 7*.

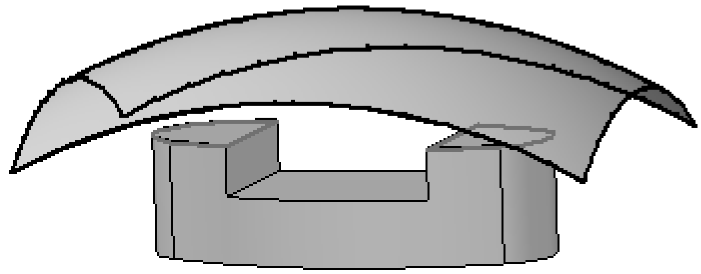


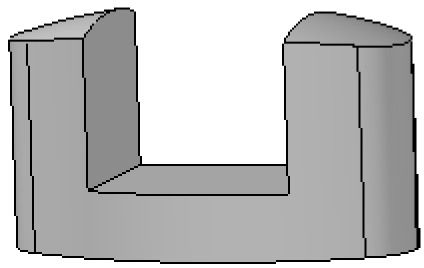
- Select **Surfacing** tab→**Modify Surfaces** group→**Replace Face** .

- Select the faces shown and click **Accept**.

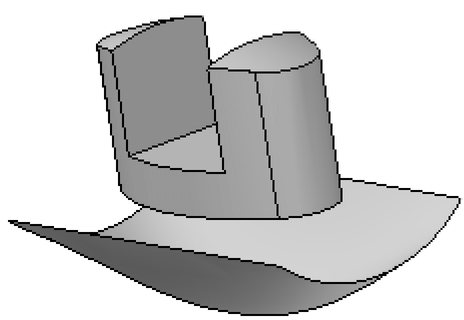


- Select the surface shown for the replacement surface.



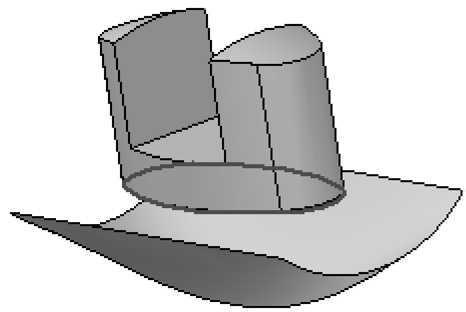


- Click **Finish**.
- Click the Select tool.
- Replace the bottom face on the protrusion. Show *BlueSurf 8*.

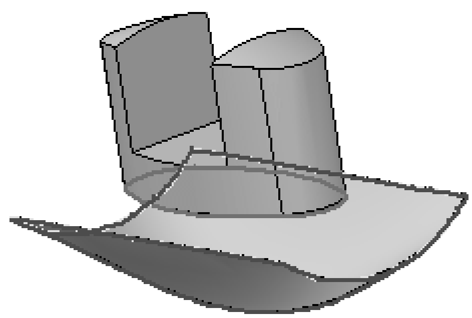


- Click the **Replace Face** command.

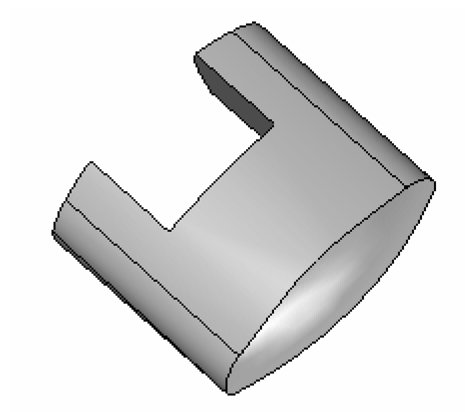
- Select the bottom face as shown and click **Accept**.



- Select the surface shown for the replacement surface.



- Click **Finish**.

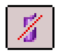


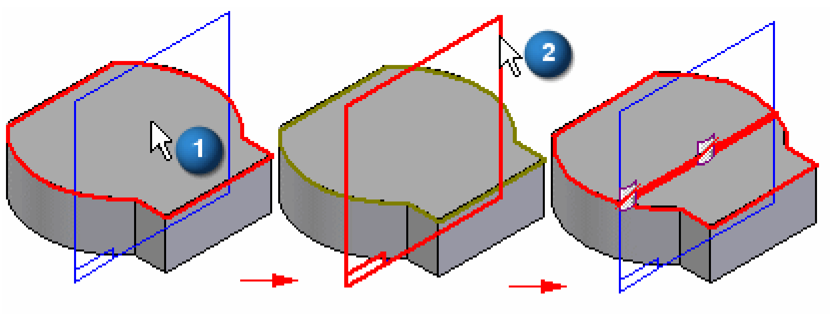
- This completes the activity. Exit and save the file.

Lesson 72: Summary

In this activity you learned how to use several surface manipulation commands.

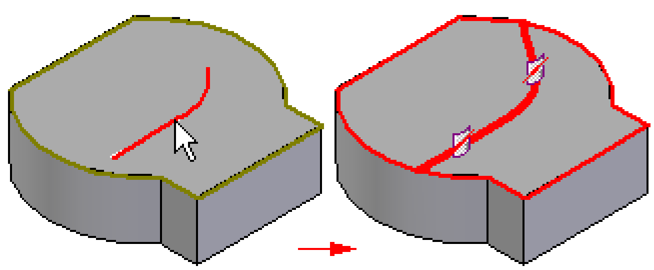
Lesson 73: Split Face Command

Use the **Split Face** command  to split one or more surfaces (1) using an element (2) you define. You can select curves, edges, surfaces, reference planes, and design bodies as the elements that split the face.

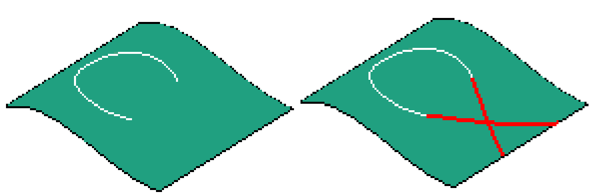


Splitting a face can be useful when constructing a model that you want to use for finite element analysis purposes or when you want to isolate a portion of a face so you can to apply a decal or image in a specific location.

If the element you are using to define the split location does not extend to the boundary of the face you are splitting, the **Split Face** command will extend the imprinted splitting curve tangentially. The original element you selected is not extended. For example, if you split a face using a sketch that consists of a line and an arc, the imprinted curve is extended linearly and tangent to the original line and arc.



If the imprinted curves intersect when they are extended, the split face feature will not succeed.



When you use a surface as the splitting element, the surface must physically intersect the surface you want to split. When you use a reference plane as the splitting element, the reference plane must theoretically intersect the surface you want to split (the reference plane is considered to be infinite in size).

When you use curves or edges as the splitting elements, such as a sketch to split a face, the splitting elements must lie on the face you are splitting. You can use the **Project** curve command to project the elements onto the 3-D face.

What can go wrong-split face features

This topic gives you solutions to problems you might have when constructing split face features.

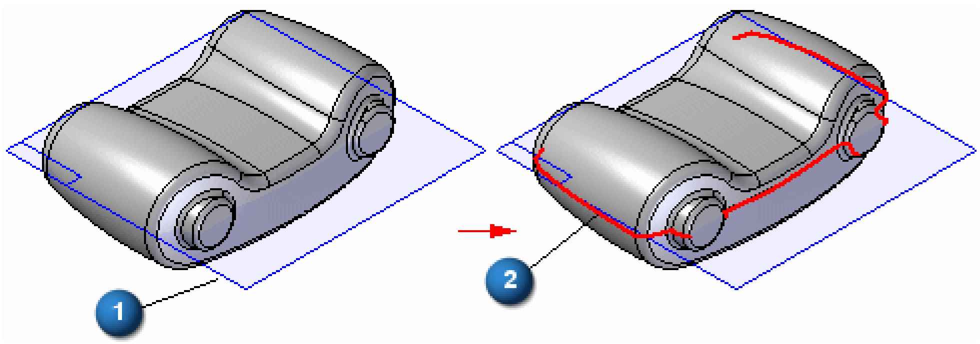
Missing Parent

Target faces must be from the same body: When constructing split face features, the faces you want to split must be from the same body.

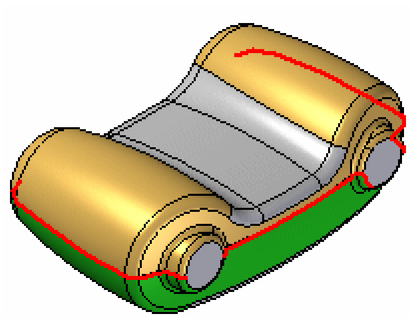
Targets and tools do not intersect. The splitting element does not intersect with the surfaces you want to split.

Lesson 74: Parting Split command

Use the **Parting Split** command  to split a set of faces along the silhouette edges of the part, which can be useful when working with a part that will be molded or cast. Parting lines are the same as silhouette lines for a given face. You define the vector direction for calculation of the parting lines by defining a reference plane (1). A parting split feature (2) is represented by curve.

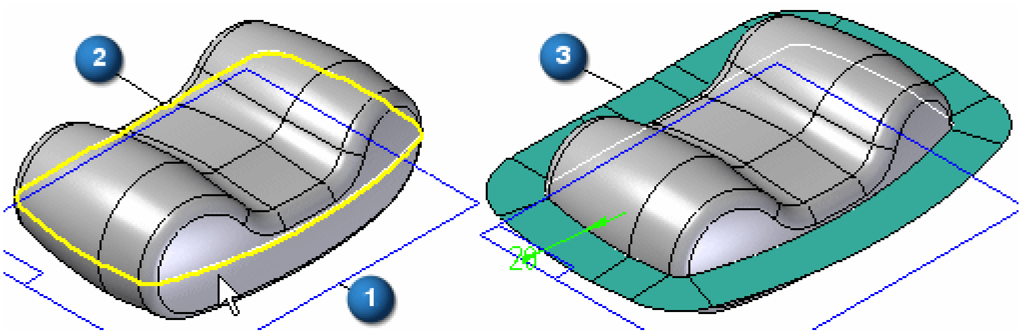


To better illustrate the results, the surfaces which are split by the parting split feature are shown in green and gold below. The surfaces shown in gray were not split. Surfaces which do not cross the parting line and planar faces are not split by this command.



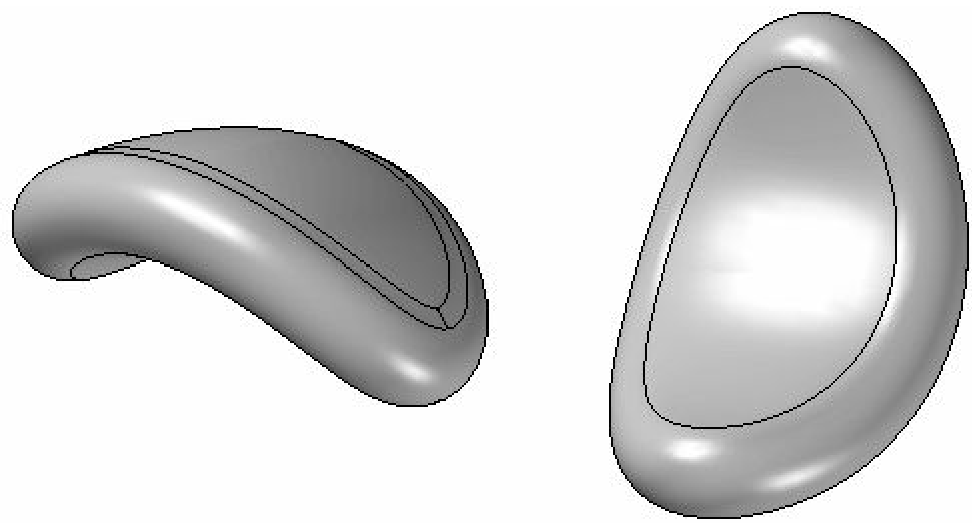
Lesson 75: Parting Surface command

Use the **Parting Surface** command to construct a parting surface along a parting curve you select. You construct a parting surface by selecting a reference plane (1) to define the orientation of the linear cross section curve, and a 2-D or 3-D parting curve (2), which defines the sweep path for the parting surface (3).



You create the parting curve in a separate operation. For example, you can use the Intersection Curve command or the Parting Split command to create the parting curve.

Lesson 76: Activity: Parting split and parting surface



Overview

When you complete this activity, you will be able to use the **Parting Split** and **Parting Surface** commands.

Objectives

After completing this activity you will be able to use the following commands:

- Insert Part Copy
- Boolean
- Parting Split
- Parting Surface

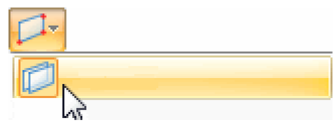
Lesson 77: Create a new part file

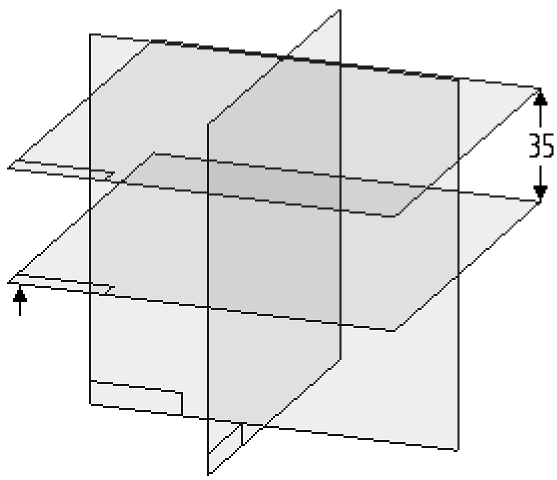
- Create a new ISO Meric Part.
- In **PathFinder**, right-click the Synchronous header and select **Transition to Ordered**.

Note

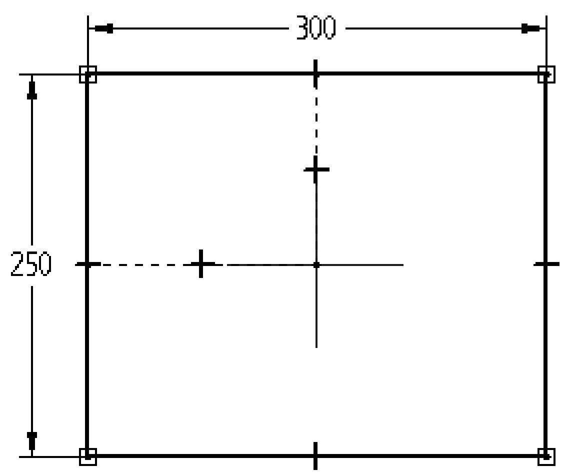
You will begin by constructing a core for the mold. The core needs to be sized to accommodate the file **pad.par** that will be the cavity.

Lesson 78: Create a sketch

- Turn on the display of the **Base Reference Planes**.
- Select **Home** tab→**Planes** group→**More Planes**→**Parallel**. 
- Select the **Top (xy)** base reference plane indicated by the arrow. Create a parallel plane 35 mm above it as shown.



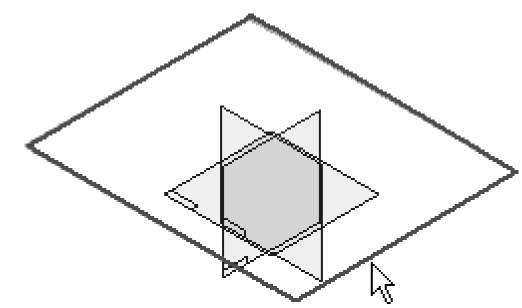
- Sketch the following profile on the parallel plane you just created. Make sure the midpoints of the line elements are horizontal/vertical aligned to the center of the reference planes.



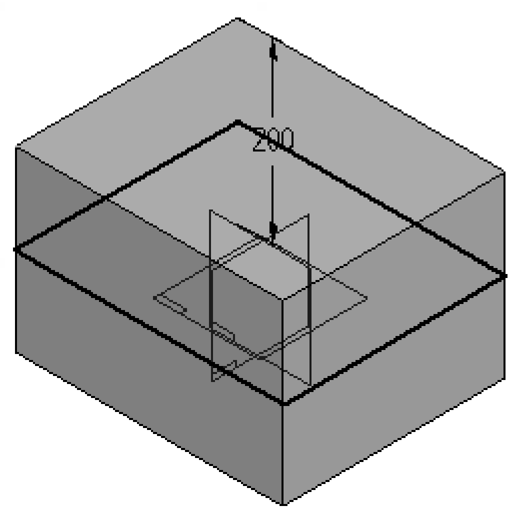
- Click **Close Sketch** and then click **Finish**.

Lesson 79: Create the core part

- Select **Home** tab→**Solids** group→**Extrude**.
- Click the **Select from Sketch** option on the command bar.
- Select the sketch shown and click **Accept**



- Click the **Symmetric Extent** button and type **200** for **Distance**. Click **Finish**.

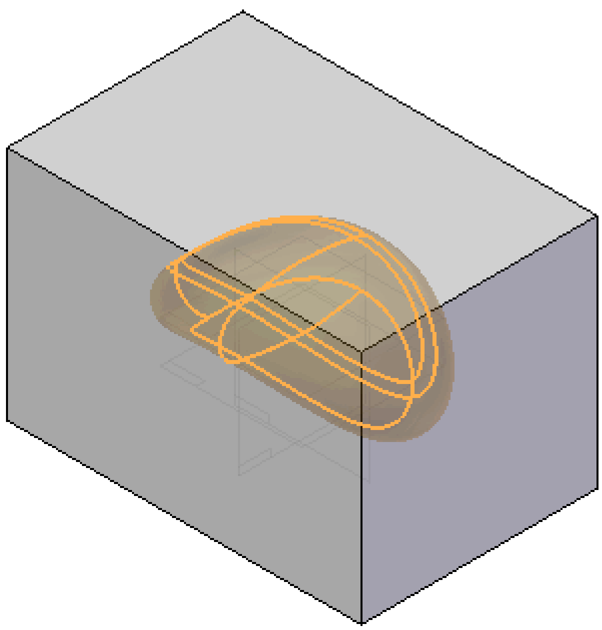


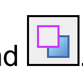
- In **PathFinder**, hide *Sketch1*.

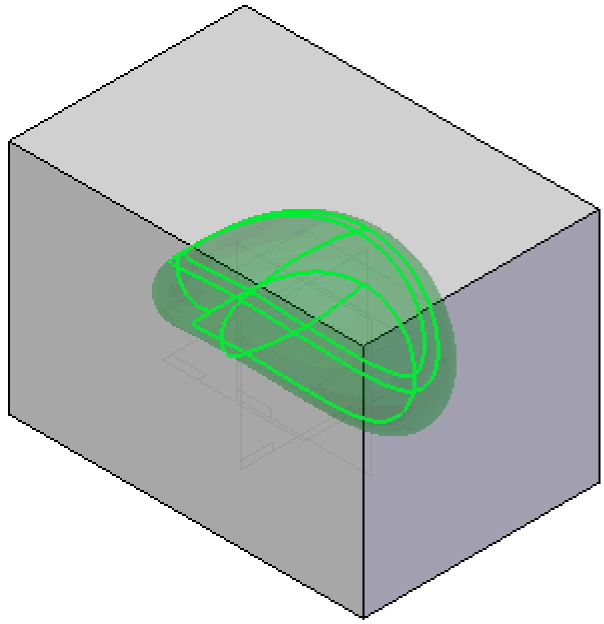
Lesson 80: Create the cavity

To create the cavity, use the Boolean difference to remove pad.par from the core.

- Select **Home** tab→**Clipboard** group→**Part Copy** .
- In the **Select Part Copy** dialog box, set **Look in** to the folder in which you installed the training parts for this course. Select **pad.par** and click **Open**.
- In the **Part Copy Parameters** dialog box, make sure **Copy as Design Body** is checked and uncheck the **Merge Solid Bodies** option. Click **OK**.
- Click **Finish**. Notice the feature *Part Copy 1* is now listed in PathFinder. Use your cursor to highlight this entry in **PathFinder**, and you can see the pad highlight in the graphics window.



- On the **Home** tab→**Solid** group→**Add body** list, choose the **Subtract** command .
- Select the extrusion (block) and then click **Accept**.



- In the part window, select the part copy (pad.par) and then click **Accept**.
- Click **Finish**. Notice the *Subtract* feature listed in PathFinder.

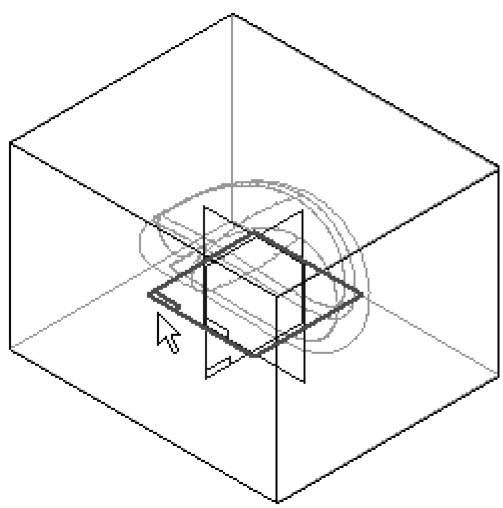
The core now has a cavity of pad.par.

Lesson 81: Create a parting split curve on the cavity

A parting split curve is derived from silhouette edges as viewed normal to a selected plane. The parting split curve defines where a part needs to split to allow for removal from a mold.

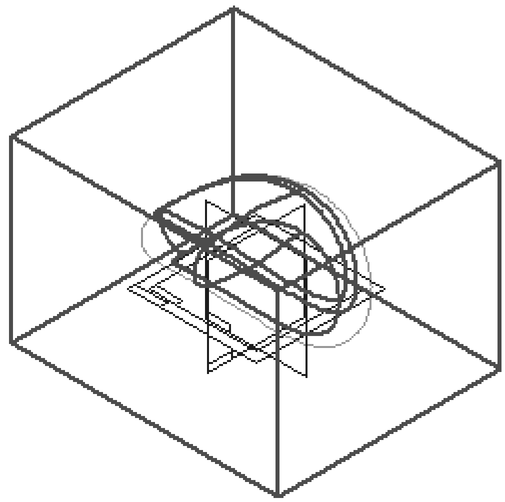
- In **PathFinder**, in the **Design Bodies** collector, uncheck Design Body_2. This turns off the display of the body.
- Right-click on Design Body_1 and choose the **Activate Body** command.
- On the **Surfacing** tab→**Modify Surfaces** group→choose the **Parting Split** command .
- Select the plane shown (**Top (xy)** base reference plane).

> **Note**
>
> You may want to change to a Wireframe view style for better visibility.

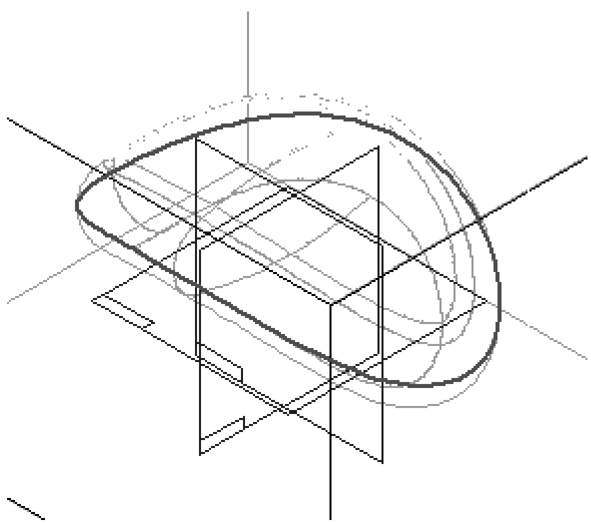


- On the **Parting Split** command bar, set the Select filter to **Body**.

- Select the body (block) as shown.

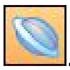


- Click **Accept** and then click **Finish**.
- Position your cursor on *Parting Split 1* in **PathFinder** and notice the parting split highlights in the graphics window.

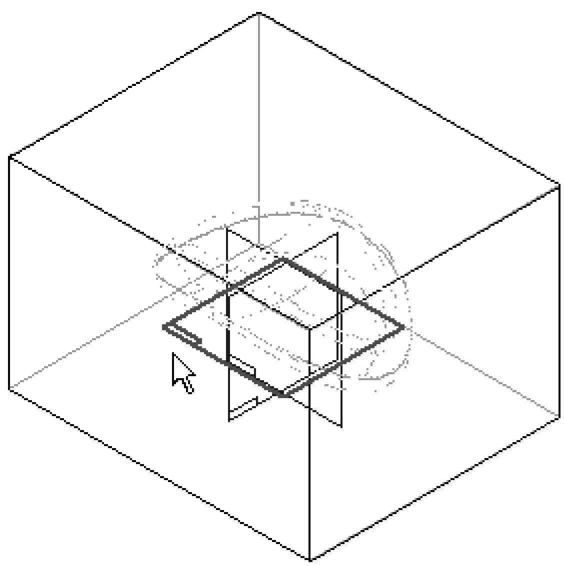


Lesson 82: Create a parting surface

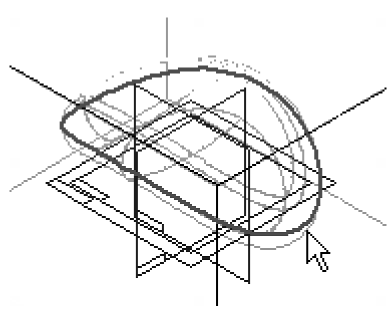
This surface will be used to split the core in a later step.

- On the **Surfacing** tab→**Modify Surfaces** group→**Parting Split** list, choose the **Parting Surface** command .
- Select the **Top (xy)** base reference plane.

> **Note**
>
> The parting surface will use this plane as reference for all normal vectors in creating the surface.

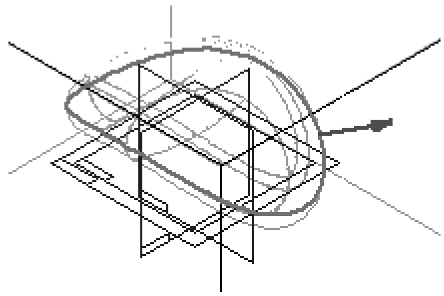


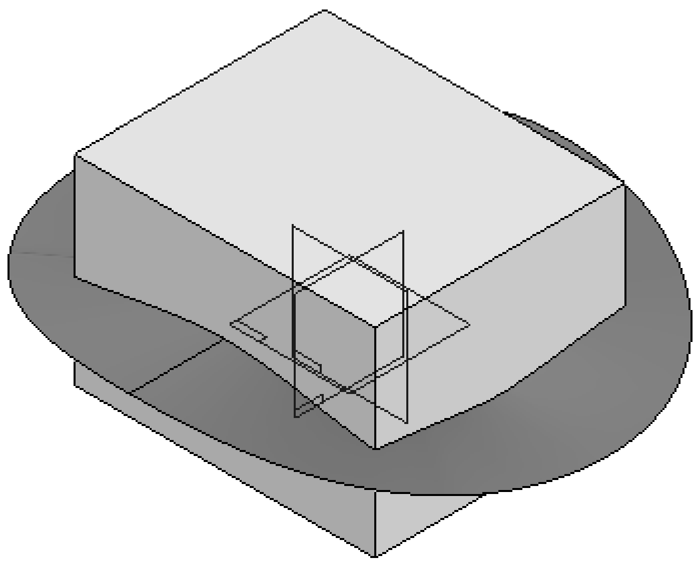
- Select the parting split curve as shown and click **Accept**.



- Type **150** in the **Distance** field and press Enter.

- Position the arrow to point outward as shown and click. Click **Finish**.





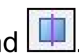
Lesson 83: Split the core

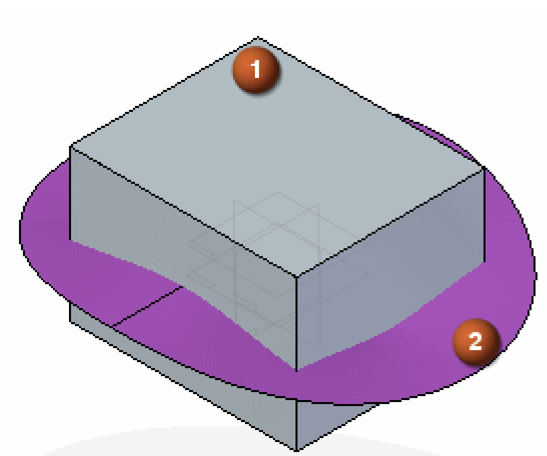
Note

Notice in Pathfinder that there are two solid bodies in the file.

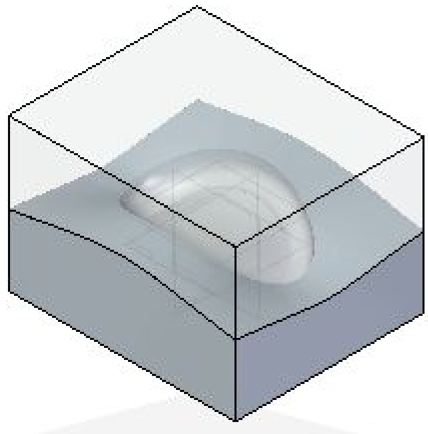
Design Body_1 is the core and Design Body_2 is the pad.par that was subtracted from the core.

Split the core along the parting surface to create mold halves. The core will be split into to solid bodies.

- On the **Home** tab→**Solid** group→**Add Body** list, choose the **Split** command .
- Select the core (1) and then select the parting surface (2).



- On the command bar, click **Accept** and then click **Finish**.

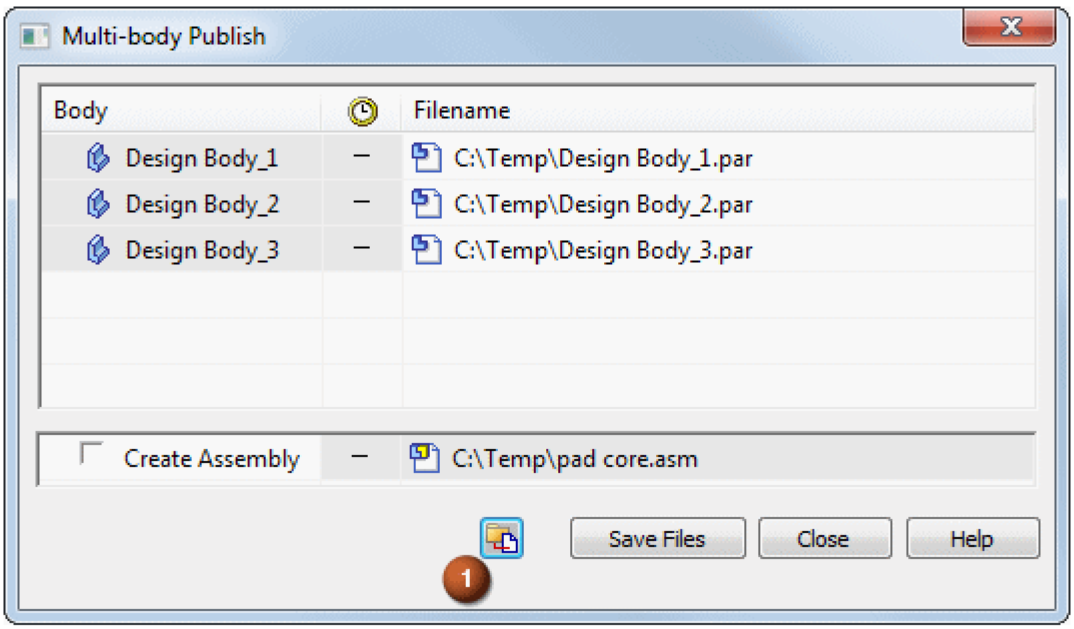


- Click **Finish**.

> **Note**
>
> Only the active solid body shades. The other bodies are opaque. You can pause your cursor over a solid body in **Pathfinder** to highlight it.

Lesson 84: Create separate core parts

- Save the file.
- On the **Home** tab→**Solid** group→**Add Body** list, choose the **Multi-body Publish** command.
- On the **Multi-body Publish** dialog box, you can click the **Set Path** button (1) for the published part files. The default path is the path of the active file. Click **Save Files** to create the part files.

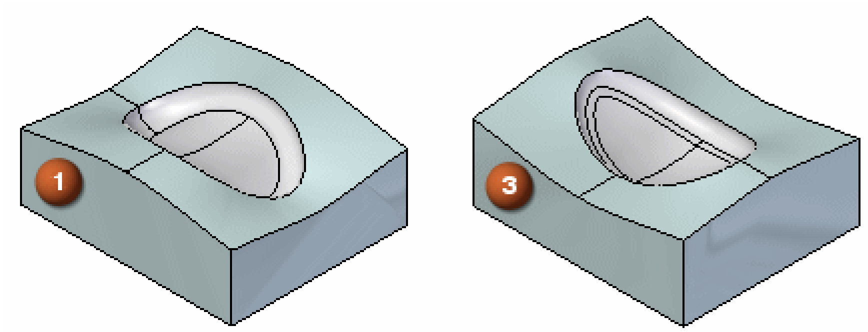


- On the **Multi-body Publish** dialog box, click **Close**.
- Close the file.

Lesson 85: Open core parts

- Navigate to the folder where the core parts were published.
- Open each core part and observe the results.

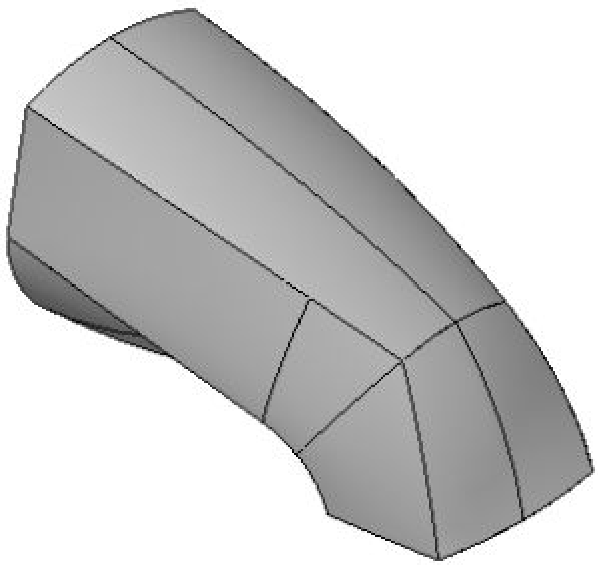
(1) Design Body_1.par, (3) Design Body_3.par.



Lesson 86: Summary

In this activity you learned how to construct two halves of a mold from a single core part.

Lesson 87: Activity: Putting it all together



Overview

In this activity, you will use the surfacing tools and workflows learned in this course to build a bathtub spout.

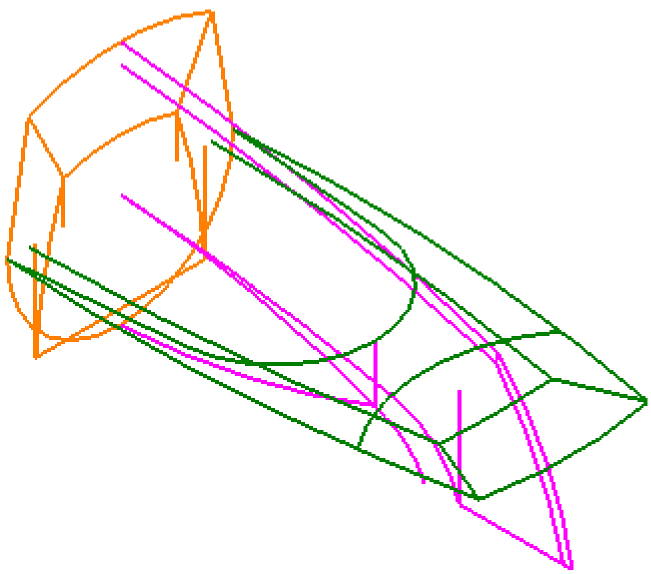
Objectives

After completing this activity you will be able to:

- Read a control drawing.
- Create and edit curves.
- Create and edit surfaces.
- Make a solid feature.

Lesson 88: Open the part file

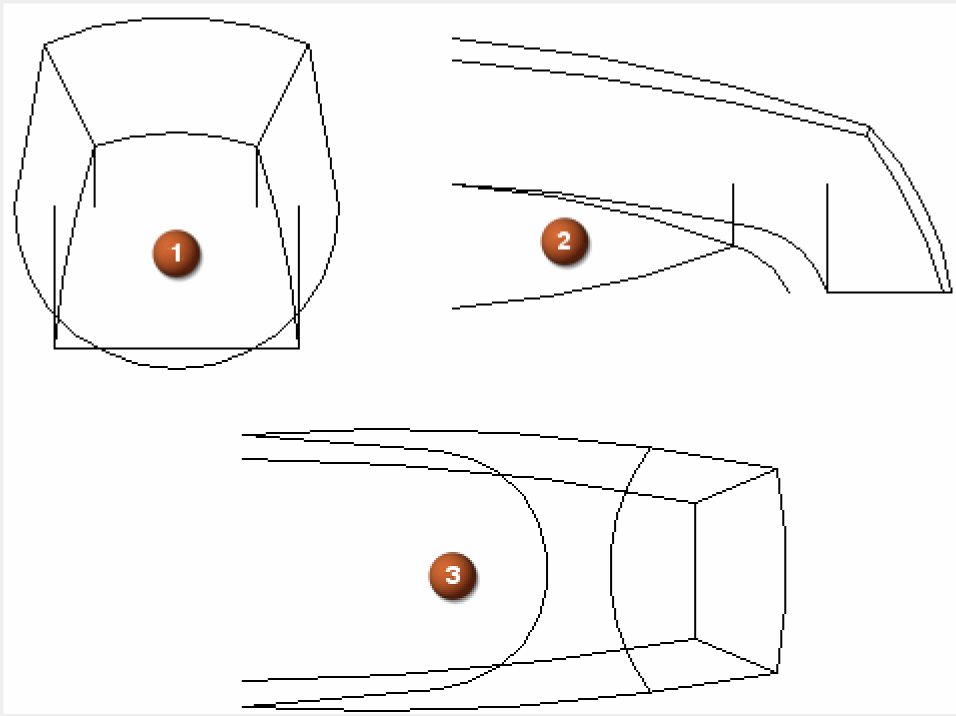
- Open **surface lab 4-02.par**.



Note

Control Drawings were discussed in the previous lesson *Surface Modeling*. For this activity the control drawing is provided. For ease of viewing, the curves in each sketch are color-coded, and color references will be made when appropriate.

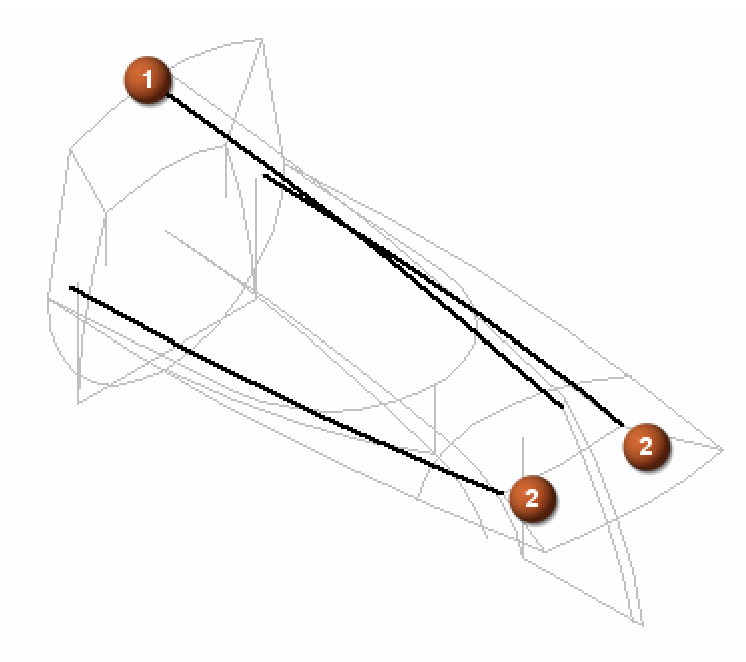
Control sketches (1)=Right View-ORANGE, (2)=Front View-PURPLE, (3)=Top View-GREEN



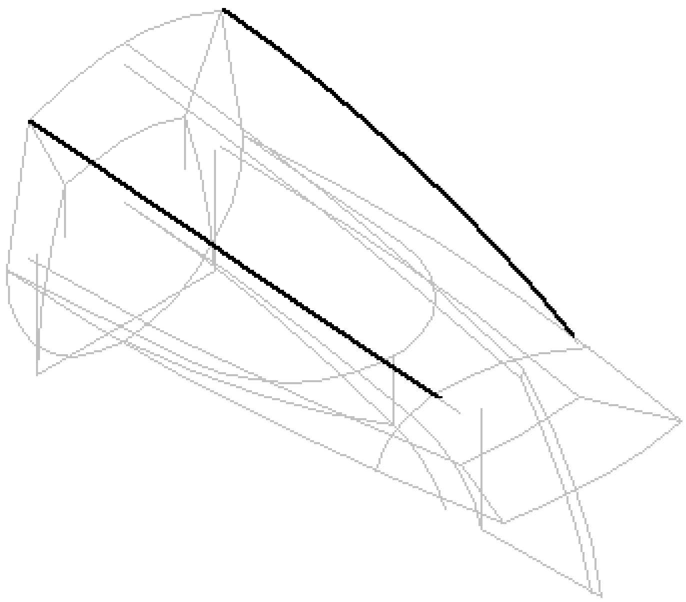
Lesson 89: Construct the top surface

Create intersection curves to be used to develop the top surface.

- On the **Surfacing** tab→**Curves** group, choose the **Cross** command .
- Click the **Select from Sketch** option.
- Select sketch element (1)—PURPLE—and then click **Accept**. Select both sketch elements (2)—GREEN—and then click **Accept**.

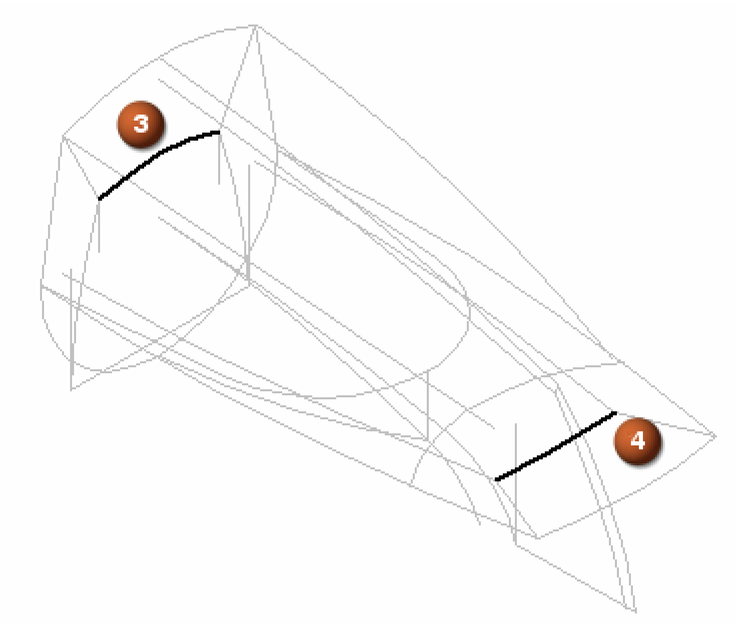


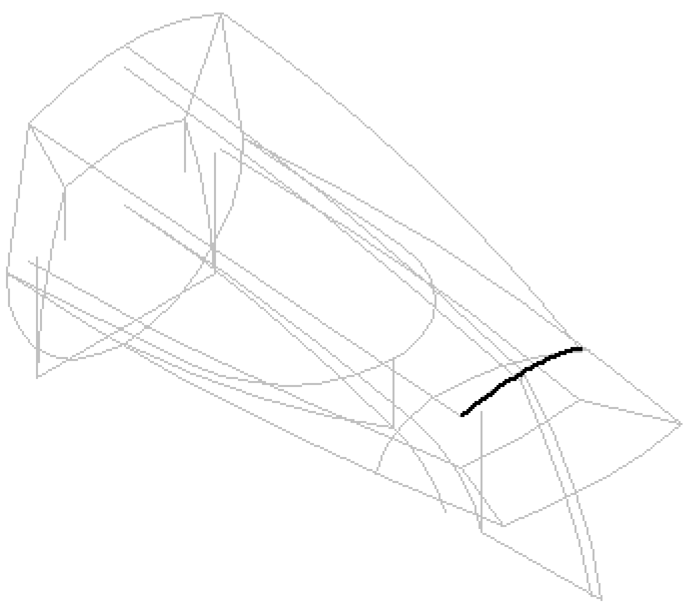
- Click **Finish**.



- Choose the **Cross** command.

- Select sketch element (3)—ORANGE—and then click **Accept**. Select sketch element (4)—GREEN—and then click **Accept**. Click *Finish*.

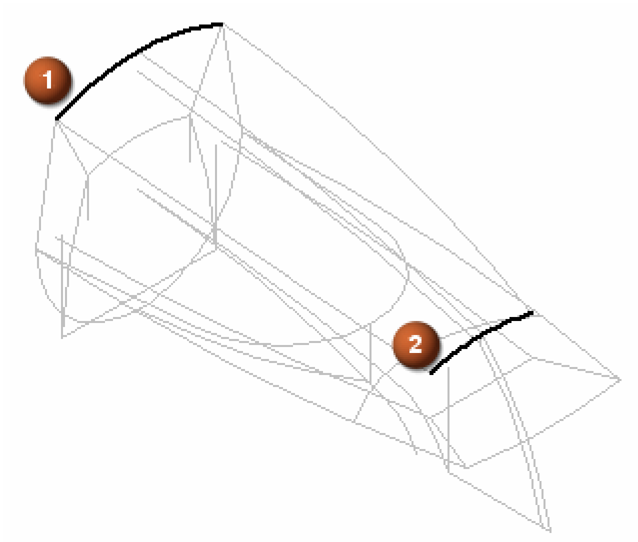




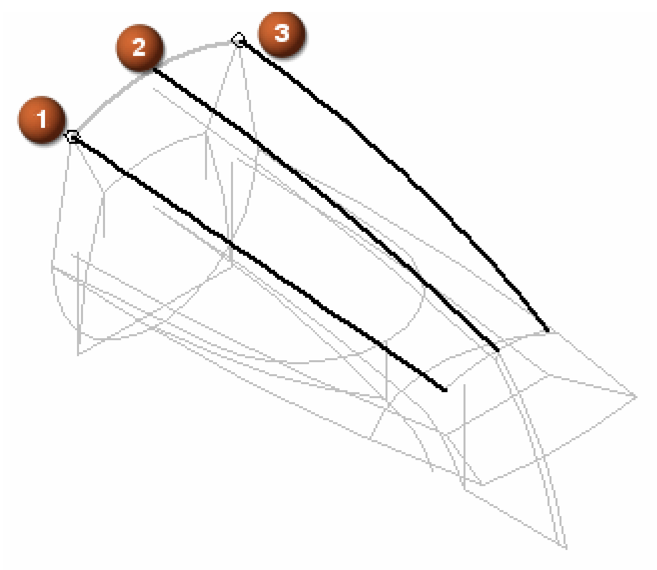
- To create the top surface, on the **Surfacing** tab→**Surfaces** group, choose the **Swept** command .

- In the **Sweep Options** dialog box, select the **Multiple paths and cross sections** option.

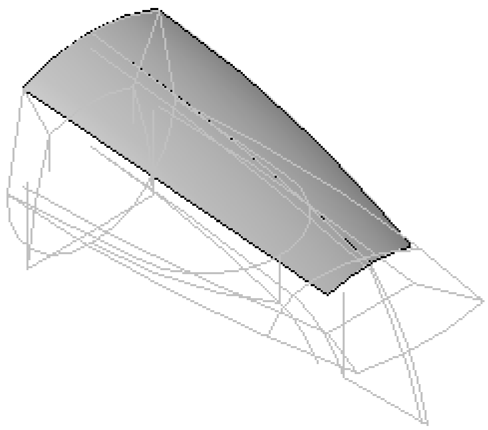
- For the **Path** Step, select path (1) and click **Accept** or right-click. Select path (2) and click **Accept** or right-click.



- Click **Next** to proceed to the **Cross Section** step.
- On the command bar, click the Select: **Single** option.
- Select cross section (1) and right-click. Select cross section (2) and right-click. Select cross section (3) and right-click.

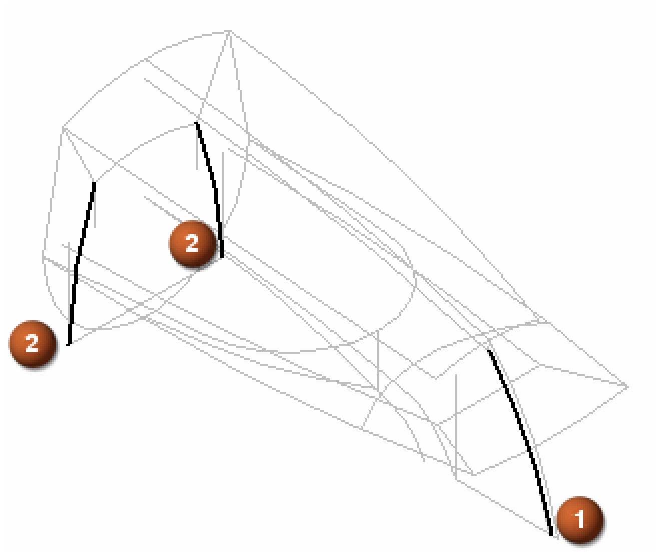


- Click **Preview** and then **Finish**.

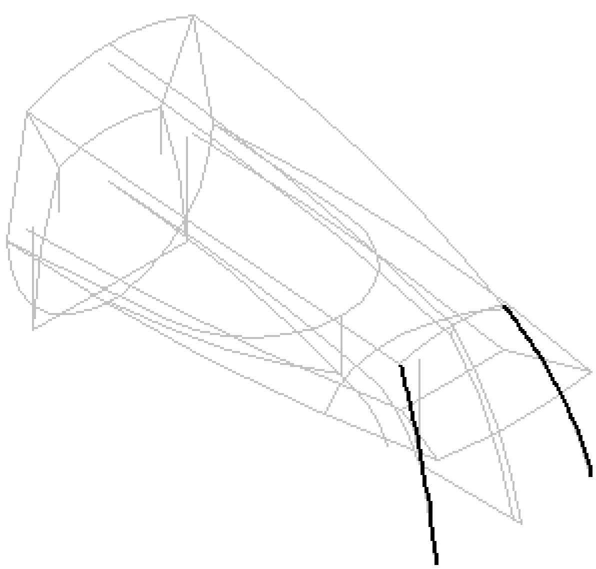


Lesson 90: Create intersection curves to develop the front surface

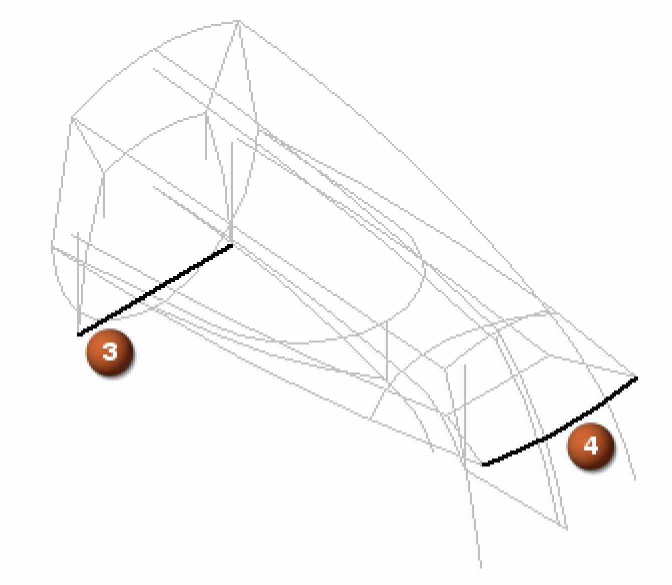
- Hide the swept surface you just created.
- Choose the **Cross** command.
- Select sketch element (1)—PURPLE—and then click **Accept**. Select both sketch elements (2)—ORANGE—and then click **Accept**.



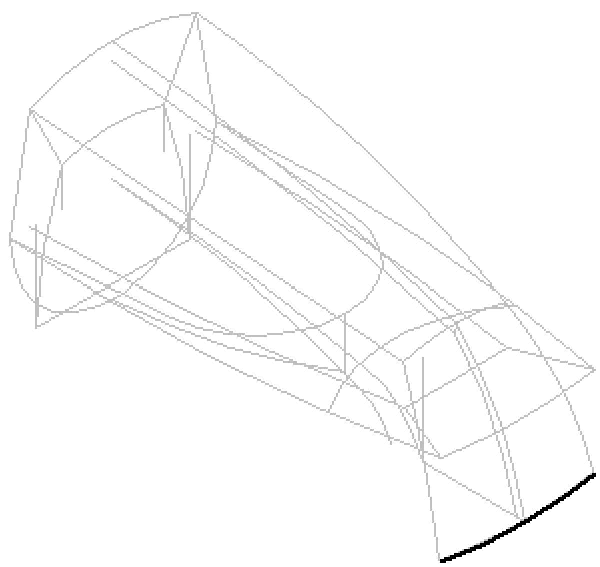
- Click **Finish**.



- Choose the **Cross** command.
- Select sketch element (3)—ORANGE—and then click **Accept**. Select sketch element (4)—GREEN—and then click **Accept**. Click **Finish**.

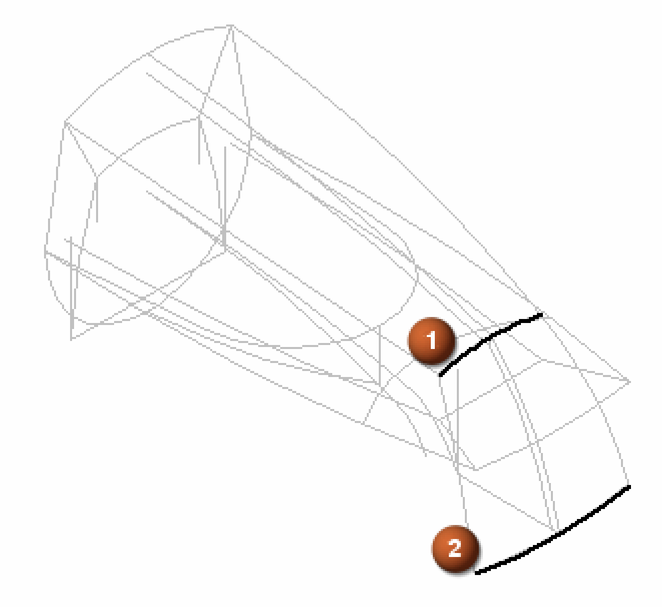


- Click **Finish**.

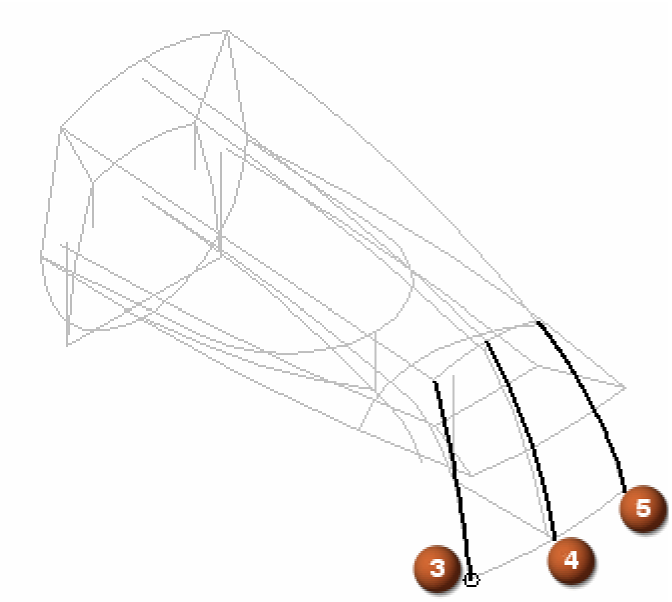


- Save the file.
- Choose the **Swept** command. Click the **Multiple paths and cross sections** option and set the **Section Alignment** to Parallel.

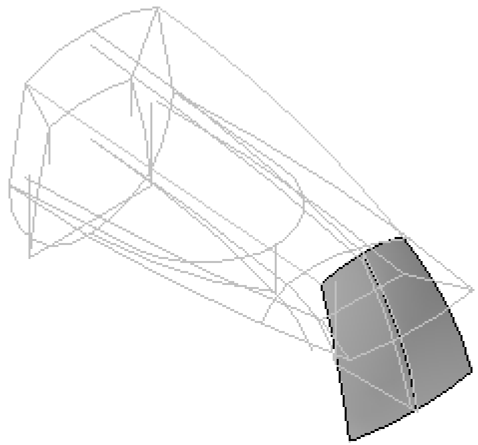
- Use **QuickPick** to select path (1). Make sure you select the cross curve element and then click **Accept**. Select path (2) and right-click.



- Click **Next** to define the cross sections.
- Set the Select filter to **Single**. Select cross section (3) and right-click.
- Select cross section (4) and right-click.
- Select cross section (5) and right-click.



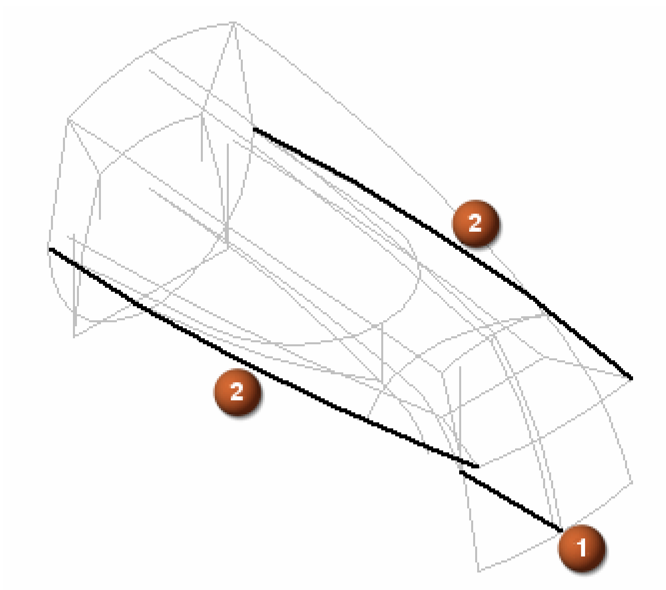
- Click **Preview** and then **Finish**.



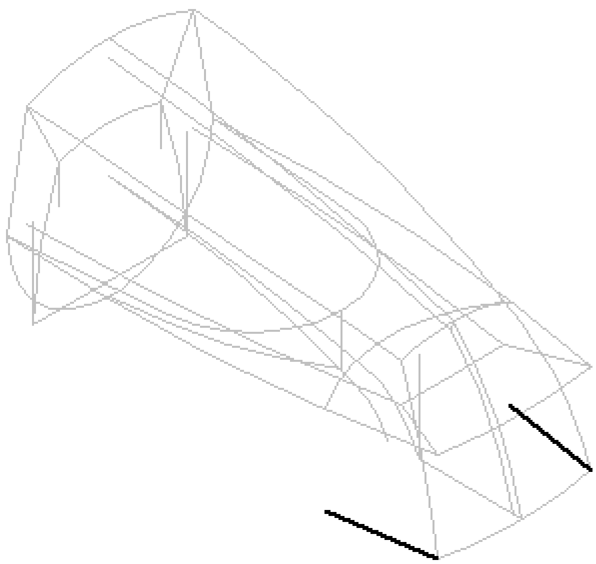
- Save the file.

Lesson 91: Create intersection curves to develop the side surfaces

- Hide the swept surface you just created.
- Choose the **Cross** command.
- Change the Select option to **Single**.
- Select sketch element (1)—PURPLE—and then click **Accept**. Change the Select option to **Single**. Select both sketch elements (2)—GREEN—and then click **Accept**.



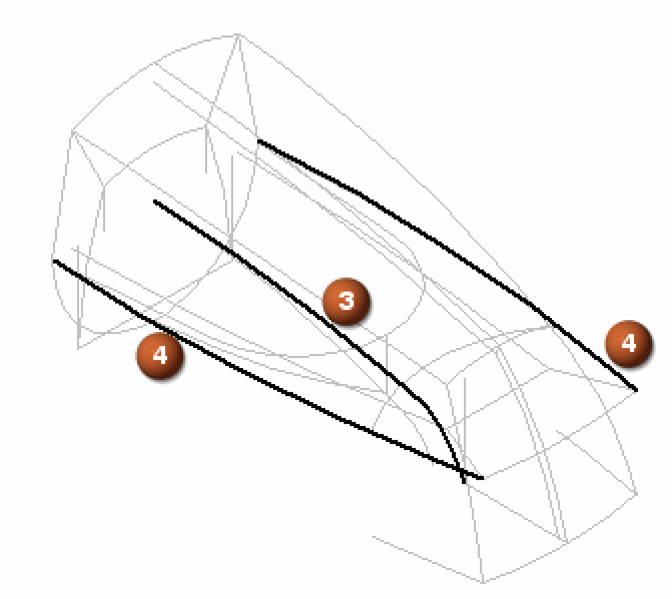
- Click **Finish**.



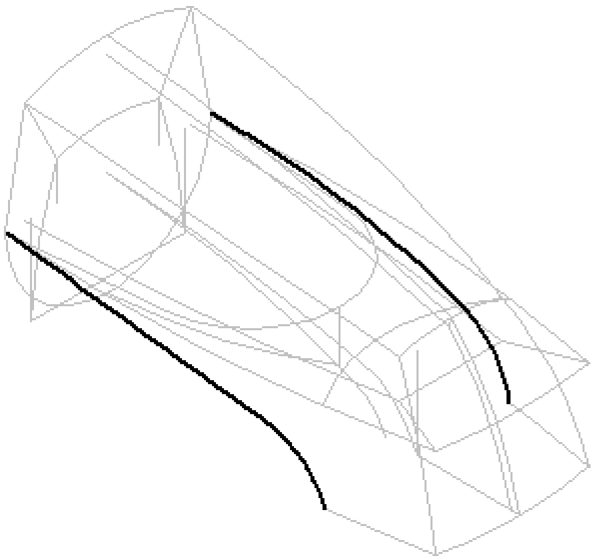
- Choose the **Cross** command.
- Set the Select option to **Single**. Select sketch elements (3)—PURPLE—and then click **Accept**.

> **Note**
>
> There are two elements in (3).

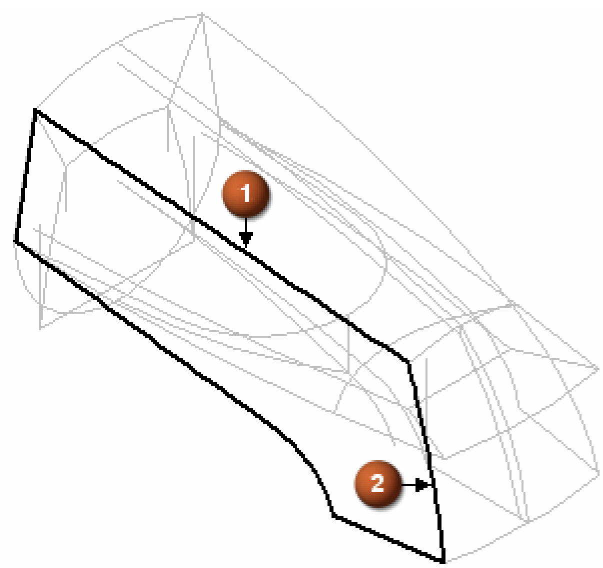
- Set the Select option to *Single*. Select both sketch elements (4)—GREEN—and then click **Accept**. Click **Finish**.



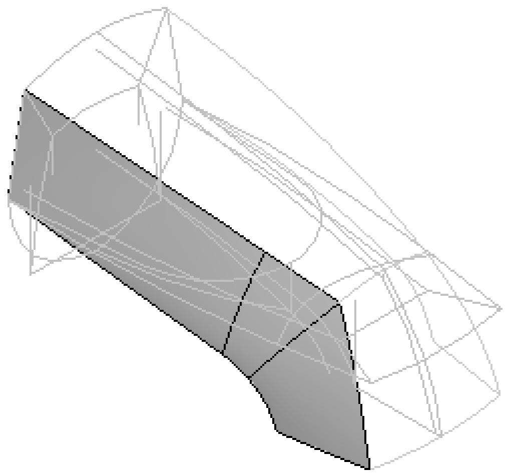
- Click **Finish**.



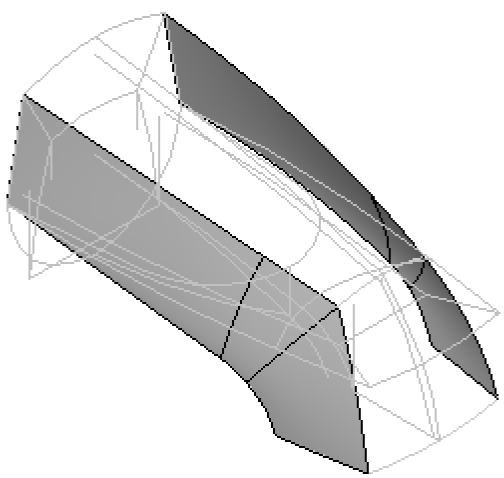
- On the **Surfacing** tab→**Surfaces** group, choose the **Bounded** command .
- Select the six edges as shown. Use **QuickPick** for edges (1) and (2) to ensure you select the cross curve edges.



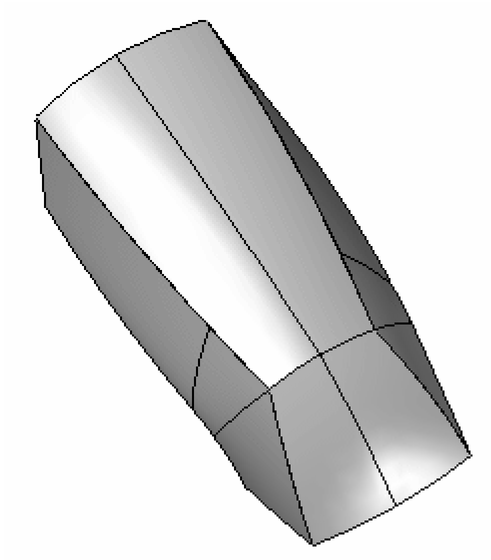
- Click **Finish**.



- Create another bounded surface on the other side.



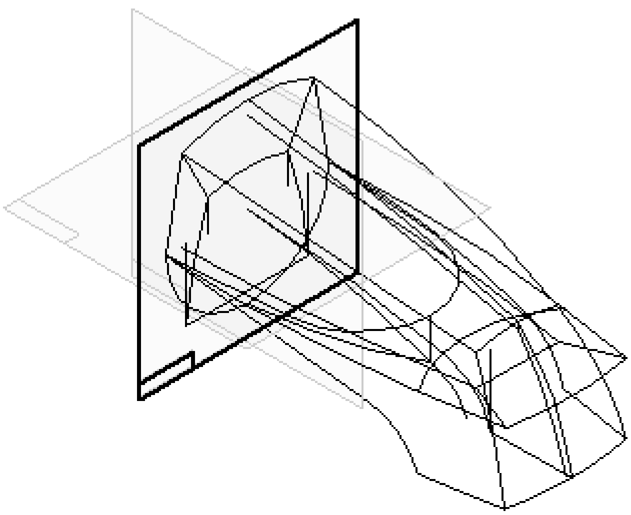
- Save the file.
- Show all of the surfaces created so far.



Lesson 92: Create the bottom surface

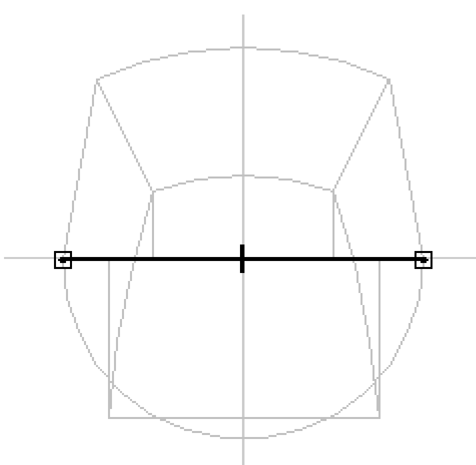
- Show all base reference planes. On the **Home** tab→**Sketch** group, choose the **Sketch** command .

- Select the **Right (yz)** plane for the first sketch.



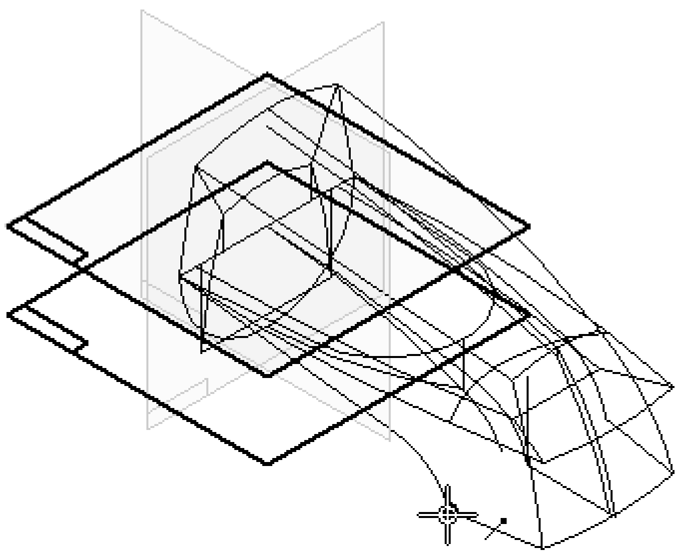
- Hide all surfaces.

- Draw the following sketch.



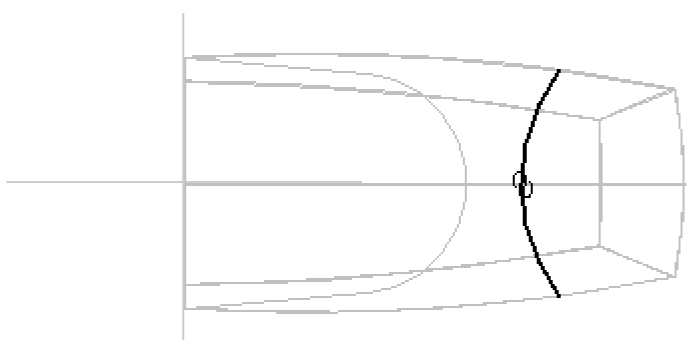
- Select **Close Sketch**.

- For the second sketch, create the sketch on a parallel plane as shown. To define the distance, click the keypoint as shown.



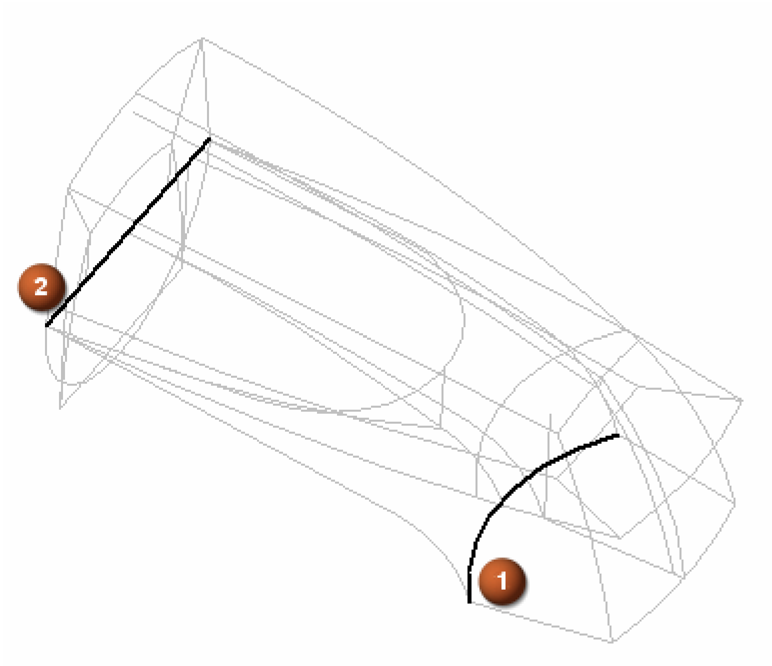
- On the **Home** tab→**Draw** group, choose the **Project to Sketch** command. Select the arc shown.

> **Note**
>
> Click **OK** in the **Include Options** dialog box.

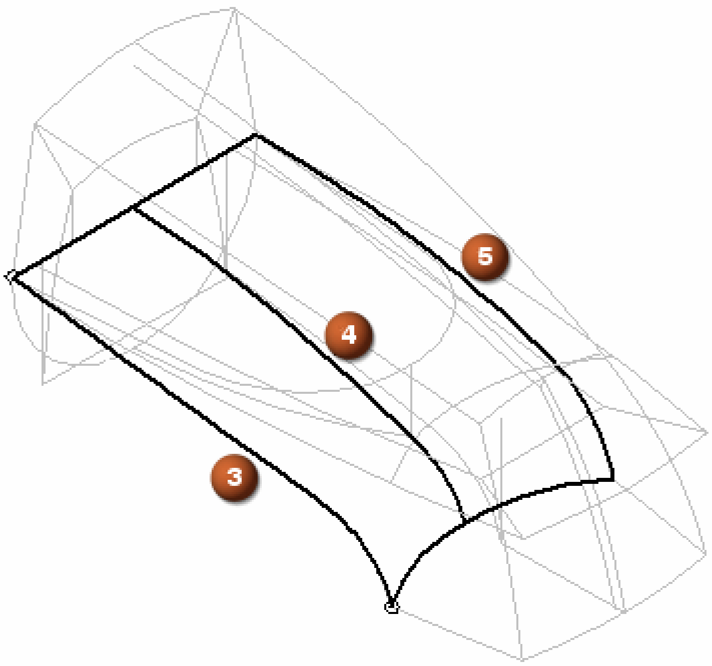


- Select **Close Sketch**.
- Choose the **BlueSurf** command.

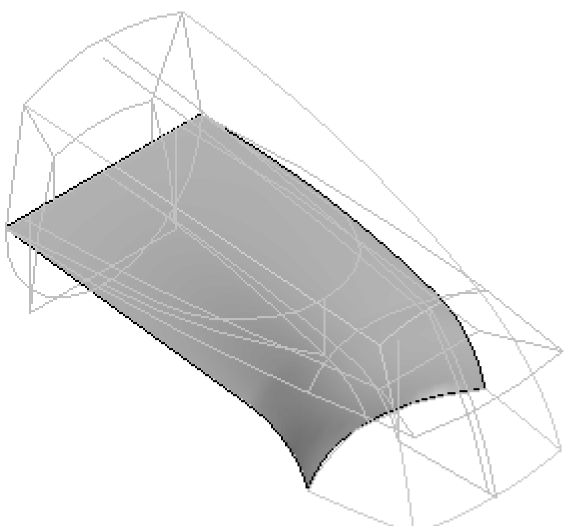
- Select cross section (1) and right-click. Select cross section (2) and right-click.



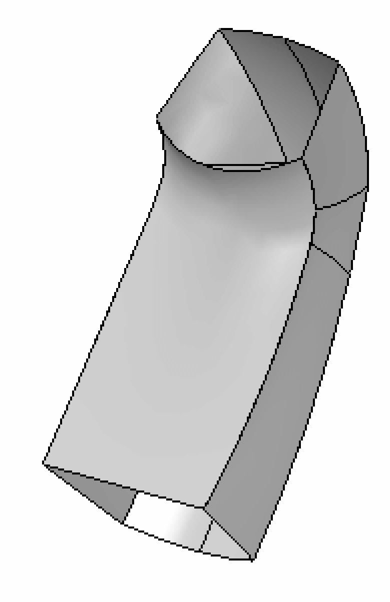
- Hide all of the base reference planes.
- Click the **Guide Curve** Step.
- Select guide curves (3) and right-click.
- Select guide curves (4) and right-click.
- Select guide curves (5) and right-click.



- Click **Next** and **Finish**.

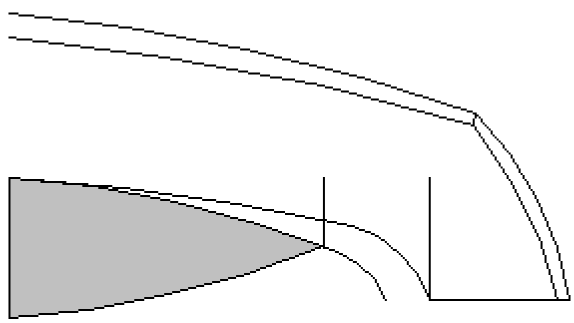


- Save the file.
- Show all of the surfaces created so far.

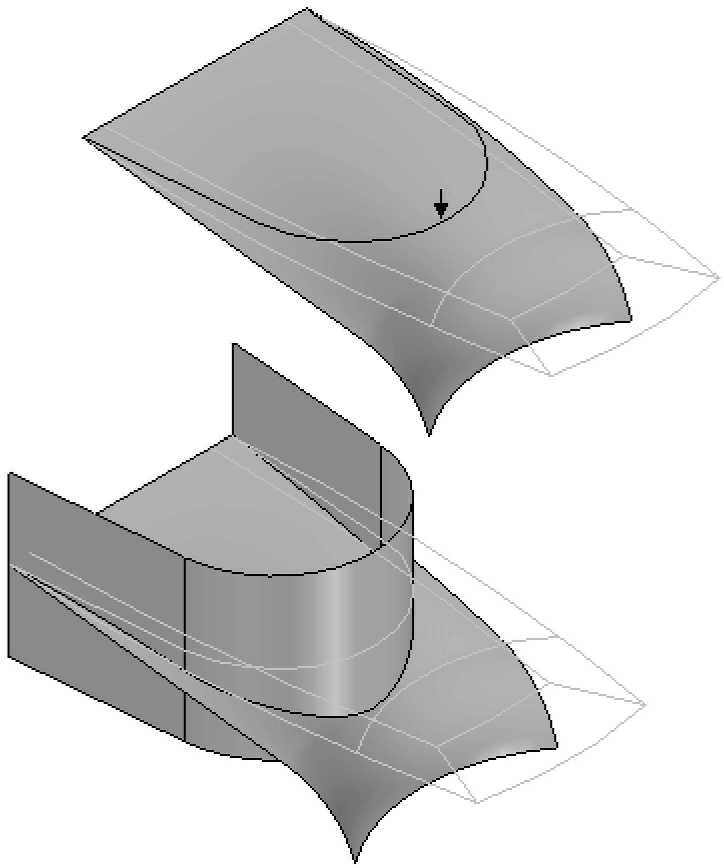


Lesson 93: Add another surface

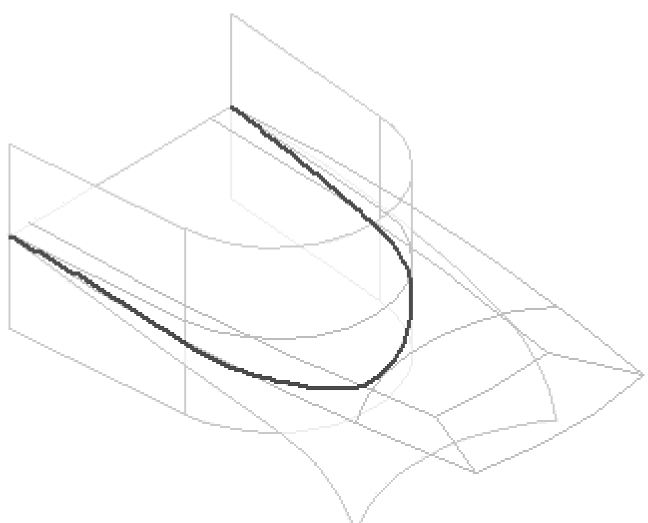
You will add the surface as shown below.



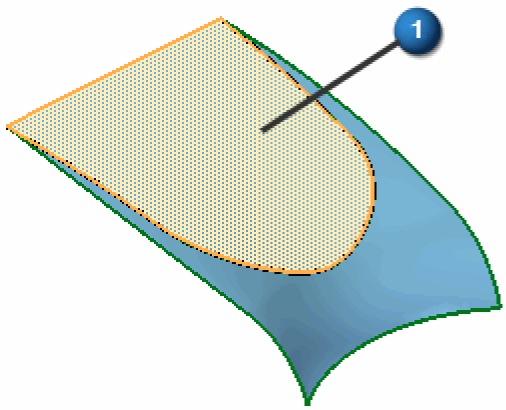
- Hide all of the surfaces except the BlueSurf just created.
- To create this surface, an intersection curve is needed between the bottom BlueSurf and an extruded surface created from the element shown in the control sketch. Create an extruded surface with a **symmetric** extent from the element shown in the image below. Do not be concerned with an exact distance.



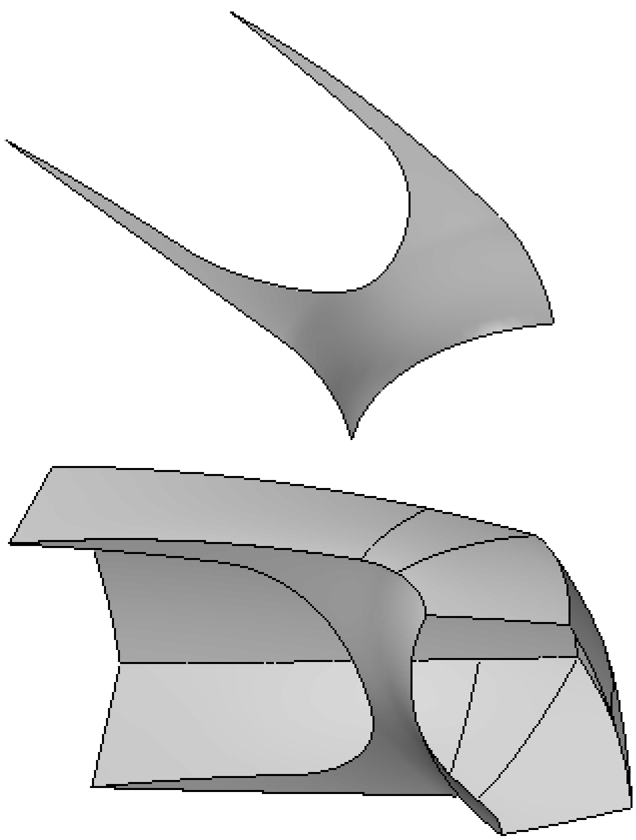
- On the **Surfacing** tab→**Curves** group, choose the **Intersection** command. Select the BlueSurf and the extruded surface as the surface set to intersect. The intersection curve is shown in the image below. Hide the extruded surface after the intersection curve is created.



- Hide all sketches. On the **Surfacing** tab→**Modify Surfaces** group, choose the **Trim** command. Select and accept the BlueSurf. Change the Select option to **Chain**. Select the intersection curve and select the region labeled 1 to remove as shown.

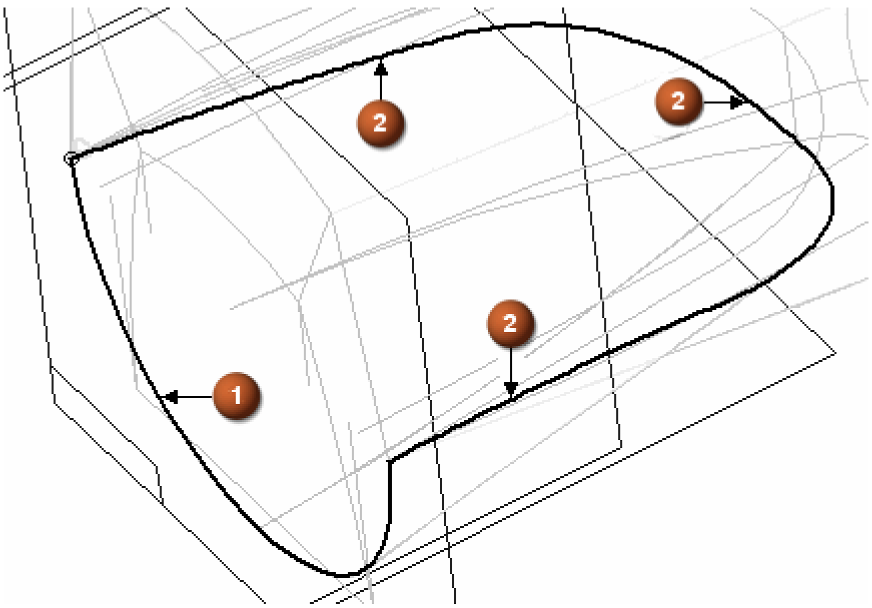


- Click **Accept** and **Finish**.

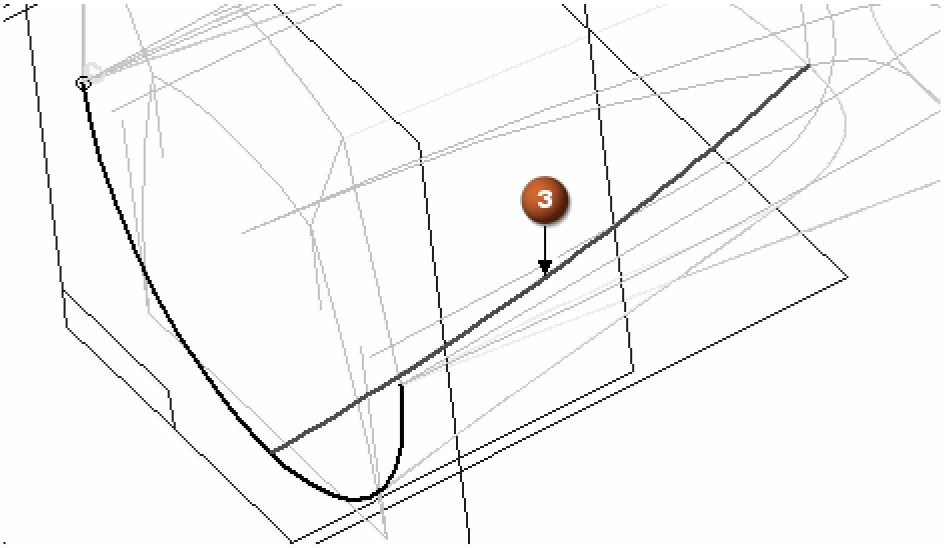


Lesson 94: Create the final surface

- Show all sketches and curves.
- Choose the **BlueSurf** command.
- Select the cross sections shown. Set the Select option to **Single** in order to select the first cross section (1). Select cross section (1) and right-click. For the second cross section (2), set the Select option to **Single** in order to select the cross section. The second cross section (2) has 3 segments. Use **QuickPick** to ensure you pick the intersection edges. Select cross section (2) and then right-click.

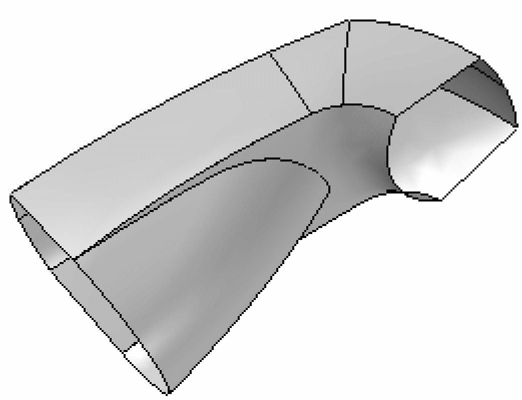


- After selecting cross sections (1) and (2), click the **Guide Curve** Step. Set the Select option to **Single**, select the curve (3) and then right-click.



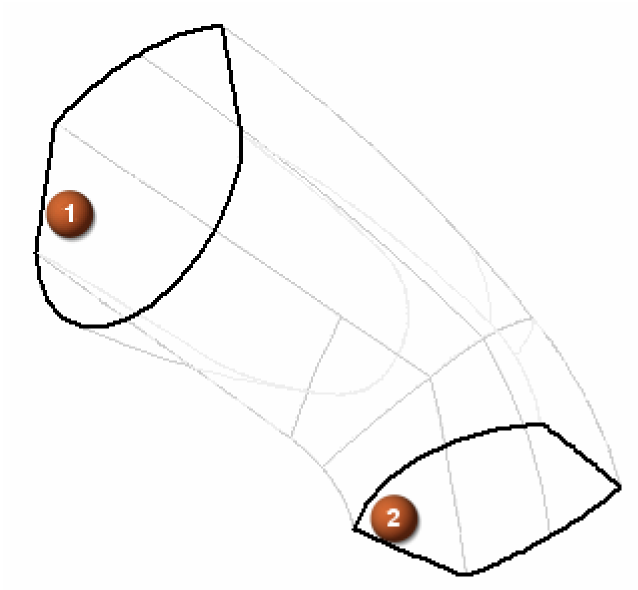
- Click **Next** and **Finish**.
- Choose the **Select** command and right-click. Select:
  - **Hide All→Sketches**
  - **Hide All→Curves**
  - **Show All→Surfaces**

> **Note**
>
> Hide the extruded surface you created as a construction surface.

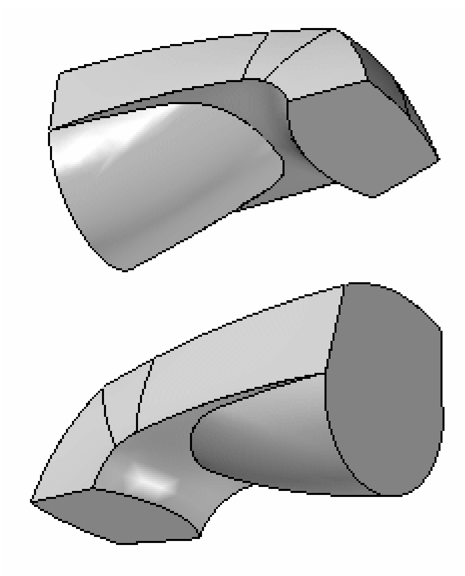


Lesson 95: Cap the ends

- Choose the **Bounded** command. Select the edges shown for (1). Select the edges shown for (2).



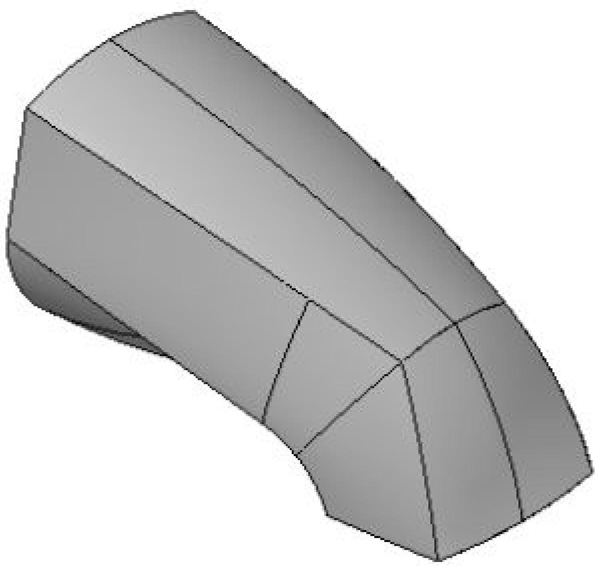
The surface model is complete.



Lesson 96: Stitch together the surfaces

The surface model now can be converted to a solid base feature.

- Choose the **Stitched Surface** command and type **.01** in the **Stitch tolerance** box. Click **OK**.
- Select all surfaces and then click **Accept**. Click **Yes** in the resulting solid message dialog box.
- This completes the bathtub spout lab activity.



Note

You rarely get the surface you want from the initial surface creation method. You also need to manipulate the surface by adding boundaries, creating an offset, extending, rounding, and so on.

Solid Edge provides several commands for you to use to develop the final surface shape. A good understanding of these tools will help you master surface modeling.

Lesson 97: Summary

In this activity you learned how to create a solid model from control curves using multiple curve and surface manipulation techniques.

Lesson 98: Lesson review

Answer the following questions:

1. What are the three extent options available on the Extend Surface command bar?
2. How do you create an offset surface without boundaries from an input surface that has boundaries?
3. Can you trim a surface with multiple open curves in one step?
4. Can you trim a surface with multiple closed curves in one step?
5. How do you round a common edge of two separate surfaces?
6. Can multiple faces of a solid be replaced in one step?

Lesson 99: Lesson summary

Many tools exist to modify surfaces. These commands greatly increase your ability to create and control complex shapes.

After stitching surfaces together, Solid Edge automatically creates a solid body.

You can delete and replace faces as needed to optimize your design.

The **Parting Split** and **Parting Surface** commands facilitate the development of parts to be manufactured in molds or dies (often called plastic parts).

Lesson 100: Curve and surface inspection tools

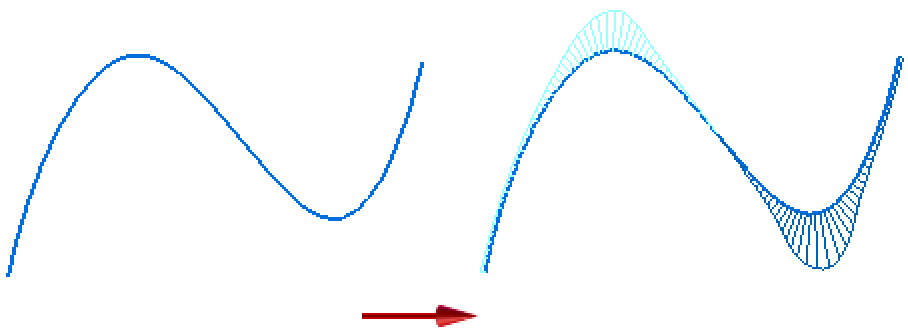
Objectives

After completing this lesson, you will be able to:

- Understand and use curvature combs.
- Use Draft Face Analysis.
- Use Curvature Shading.
- Use Zebra Stripes.

Lesson 101: Show Curvature Comb command

Use the **Curvature Comb** command to switch on or off the display of the curvature comb for a curve. The curvature comb shows the smoothness of a curve.



Curvature combs help you determine how quickly or gradually curves change and where they change direction. You can use the curvature comb to quickly determine the feasibility of machining and to predict the aesthetic qualities of surfaces generated from a curve.

If you have a curvature comb displayed and use dynamic edit to make changes to the curve geometry, the comb updates immediately to reflect the changes.

Lesson 102: Surface inspection tools

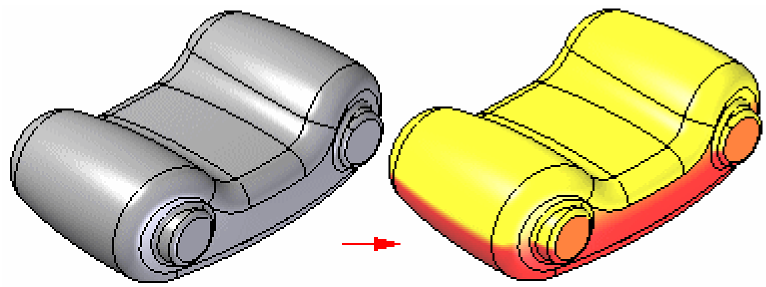
Surface inspection tools are available in the **Inspect** tab→**Analyze** group.

- Draft Face Analysis
- Curvature Shading
- Zebra Stripes

Lesson 103: Show Draft Face Analysis command

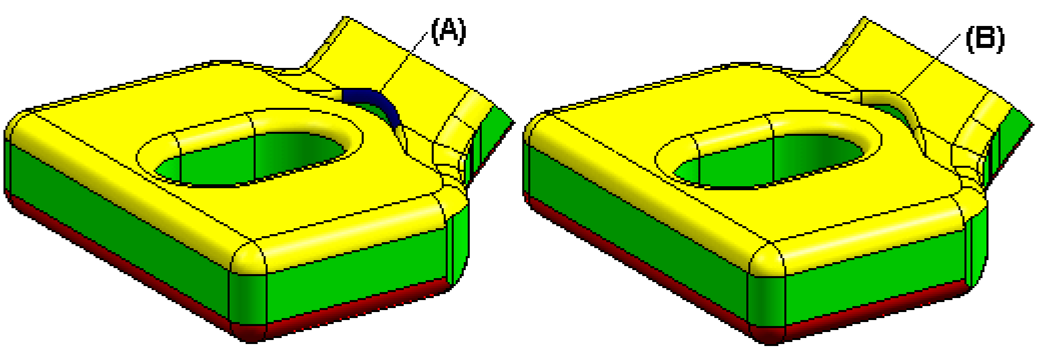
Displays colors on the model based on the surface angles with respect to a draft plane you define. This is a visualization of whether a part can be removed from a mold or die. To display draft face analysis colors, shade the active window using the **Shaded** or **Shaded With Visible Edges** commands.

Use the Draft Face Analysis Settings command to specify the draft plane, draft angle, and assign the colors to use.



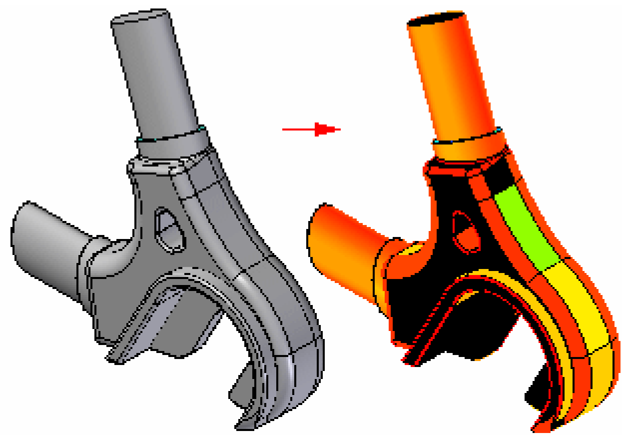
Draft face analysis and view quality

The results of a draft face analysis depends on the current view quality. You might find that the draft face analysis result changes if you modify the view quality. For example, if increasing the view quality using the **Sharpen** command from 2 to 4, the results for the face shown in the illustration are changed from a crossover face (A) to a positive face (B).



Lesson 104: Show Curvature Shading command

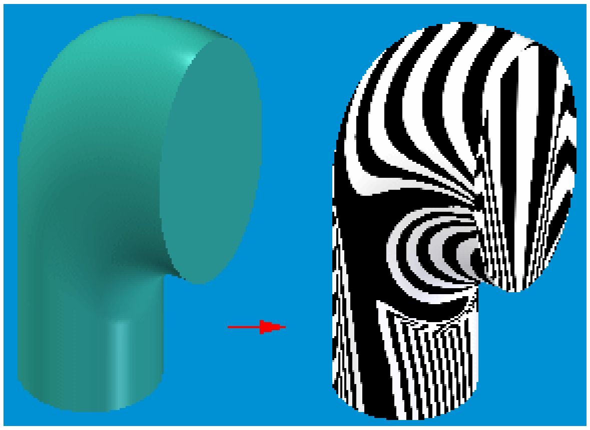
Displays colors on the model based on the radius of curvature of the model's surfaces. Use the command to graphically visualize the radius of curvature of a model. You must also shade the active window using the **Shaded** or **Shaded With Visible Edges** commands to display curvature shading colors.



Lesson 105: Show Zebra Stripes command

Displays zebra stripes on the model. Zebra stripes are useful for visualizing the curvature of surfaces to determine if there are surface discontinuities and inflections.

> **Note**
>
> You must also shade the active window using the **Shaded** or **Shaded With Visible Edges** commands to display zebra stripes.



- Zebra Stripes are solid bands of color overlaid on top of a single face or set of surfaces:
  - Displayed at regular spacing, controlled by the user.
  - Follow the contour of the relevant faces.
- One might ask: "How do these 'stripes' help?"
  - Smooth stripes are manifested by smooth, continuous surfaces (ie., no cusps or "wrinkles").
  - Stripes with sharp bends would indicate abrupt changes in surface curvature (ie., a discontinuity).
  - Discontinuities will make manufacturing more difficult.
    - Metallic Parts: Machining will be more complex.
    - Molded Parts: Injection of plastic may be difficult into discontinuous areas.
  - You can control colors, spacing and the method of mapping the stripes using the Zebra Stripes Settings.

Benefits

- Striping gives quick indication of continuous edges between faces.

- Dynamic; users can see changes in real-time.
- Non-rollback edit method.

Lesson 106: Show Reflective Plane command

Turns the display of active reflective planes on or off. The reflective planes display a reflection of a symmetrical part instead of an actual mirror feature. This is a quick way of resolving symmetrical form and studying the volume of a sculpted model.

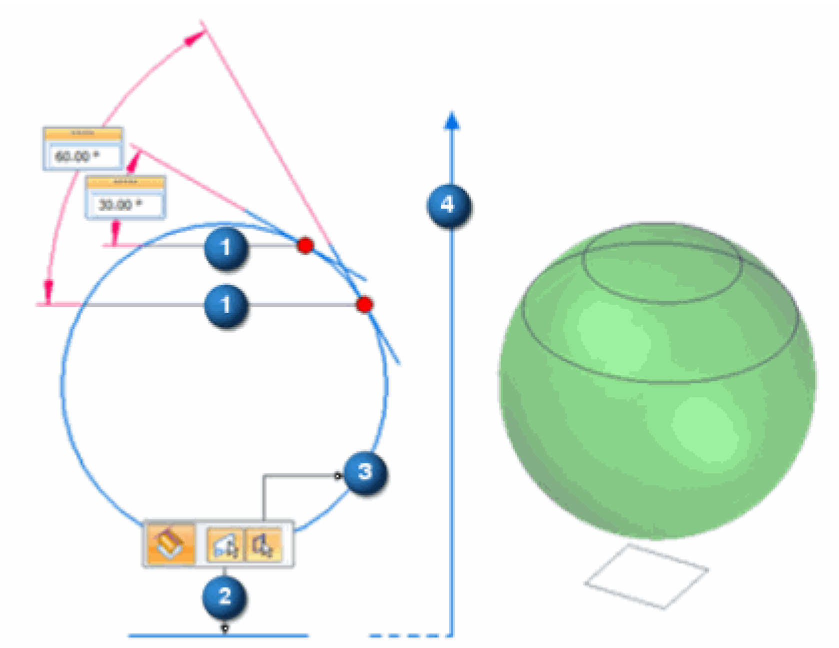
Reflective planes are used for display only. Reflective planes reflect model faces and curves. Reflected faces include both construction and solid faces. Face styles are also reflected. Reflective planes do not reflect coordinate systems, reference planes, sketches, or dimensions.

Use the **Reflective planes** dialog box for the reflectivity settings.

Lesson 107: Isocline command

Use the **Isocline** command  to place an isocline curve on one or more selected faces. An *isocline* curve is a curve connecting points on a surface whose surface normal makes a constant angle to the direction of pull (4).

The required inputs are: a reference plane, surface body, and an angle.



The above example shows two single isocline curves (1) created on a spherical surface (3), at 30 and 60 degrees from the reference plane (2).

You can see the relationship between the angle and where the isocline curve is created. The direction vector (4) is normal to the reference plane.

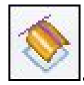
You can create isocline curves on any selected face, or on multiple faces. You may need to flip the direction vector to get corresponding isocline curves on adjacent faces.

The input angle range is 0.00 < 90.00 degrees.

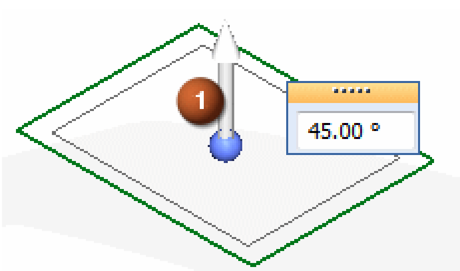
Use isocline curves to:

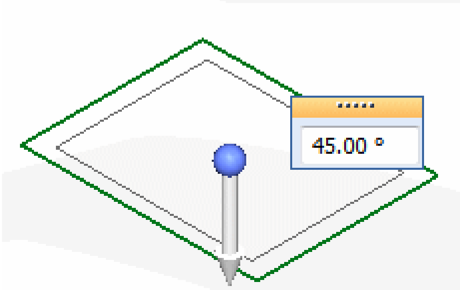
- Split a surface.
- Construct parting lines and parting surfaces on a mold or casting.
- Analyze a face/surface using draft angle map (family of isoclines).
- create new surfaces.

Lesson 108: Create an isocline curve

1. Choose **Surfacing** tab→**Curves** group→**Isocline** .
2. Select a reference plane or planar face.
3. Select a surface body.
4. In the **Angle** edit box, type an angle within the range of 0.00 to less than 90.00 degrees.
5. Click the **Accept** button.

> **Note**
>
> You can click the direction arrow (1) or press **F** to get another possible isocline curve result.

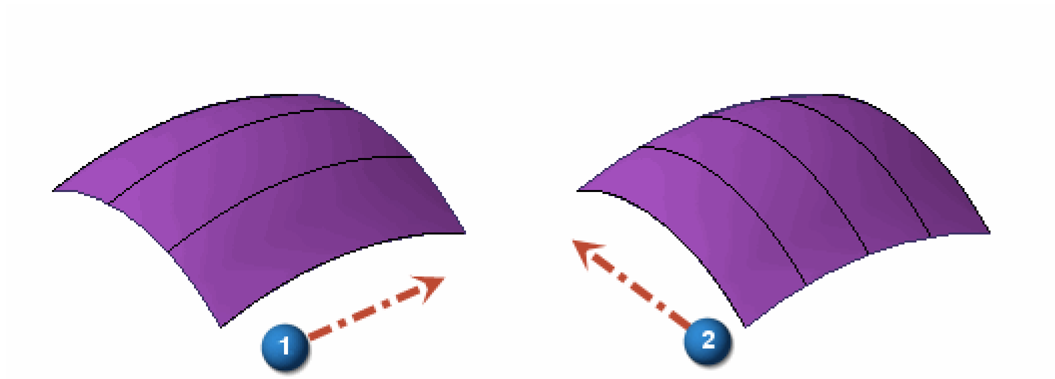




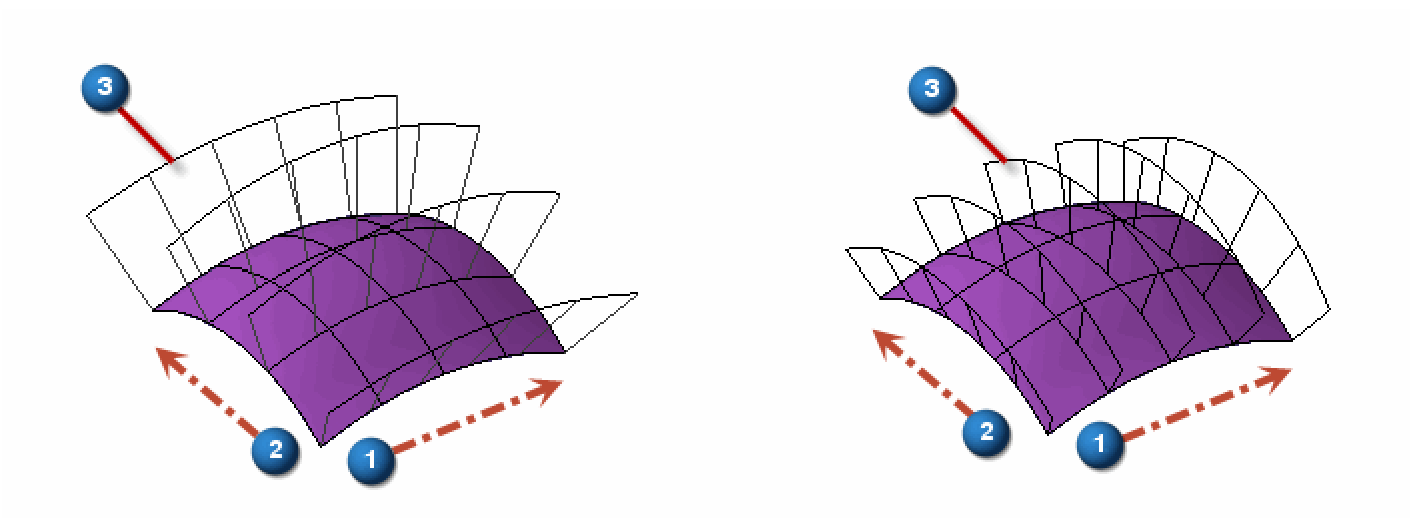
Lesson 109: Surface Visualization command

Use the **Surface Visualization** command  to display the UV Mesh and curvature combs on a selected surface as defined in the dialog settings box. The UV Mesh or Isoparameter curve mesh are lines running along the surface in the U and V directions, showing the shape of the surface. The curvature comb shows the smoothness of a curve on the surface.

The following illustration shows U direction labeled as 1 and V direction labeled as 2.



The following illustration shows the curvature combs labeled as 3 for direction 1 and 2 respectively.



Lesson 110: Lesson review

Answer the following questions:

1. What is the Curvature Comb used for?
2. In what situations would the Draft Face Analysis command be useful?
3. Explain the difference between Curvature Shading and Zebra Stripes.

Lesson 111: Lesson summary

You have learned what curvature combs show, and how to modify their output. Methods for visualizing the quality of surfaces have also been covered here.

Siemens Industry Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

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