

# *Solid Edge Surfacing*

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# Contents

|   |            |
|---|------------|
| <b>Introduction to Surface Modeling</b> .....           | <b>1-1</b> |
| Course Overview .....                                   | 1-1        |
| Introduction to Surface Modeling .....                  | 1-2        |
| Surface Modeling Overview .....                         | 1-2        |
| Surface Modeling Workflow .....                         | 1-5        |
| Surface Modeling Workflow Summary .....                 | 1-6        |
| Summary .....   | 1-17       |
| <b>Creating and Editing Curves</b> .....                | <b>2-1</b> |
| Objectives .....  | 2-1        |
| Overview of Splines .....                               | 2-1        |
| Curve Command .....                                     | 2-1        |
| Curve Definition .....                                  | 2-4        |
| Curve Edit .....  | 2-6        |
| Simplify .....  | 2-9        |
| Element Properties .....                                | 2-11       |
| BlueDot .....   | 2-12       |
| Convert to Curve .....                                  | 2-17       |
| Additional Curve Creation Methods .....                 | 2-19       |
| Curvature Comb .....                                    | 2-30       |
| Review .....  | 2-31       |
| Activity 1 – Drawing and Editing a Curve .....          | 2-33       |
| Activity 2 – Creating and Editing BlueDots .....        | 2-57       |
| Activity 3 – Creating Keypoint Curves .....             | 2-67       |
| Activity 4 – Additional Curve Creation Methods .....    | 2-83       |
| Summary .....   | 2-104      |
| <b>Surface Creation</b> .....                           | <b>3-1</b> |
| Objectives .....  | 3-1        |
| Overview of Surfaces .....                              | 3-2        |
| Creating a Simple Surface .....                         | 3-3        |
| Creating a Swept Surface .....                          | 3-7        |
| BlueSurf .....  | 3-9        |
| Bounded Surface .....                                   | 3-26       |
| Review .....  | 3-31       |
| Activity 1 – Creating and Editing Simple Surfaces ..... | 3-33       |
| Activity 2 – Creating a Swept Surface .....             | 3-39       |
| Activity 3 – Creating a BlueSurf Using Analytics .....  | 3-47       |
| Activity 4 – Creating and Editing a BlueSurf .....      | 3-59       |
| Summary .....   | 3-69       |
| <b>Additional Surfacing Commands</b> .....              | <b>4-1</b> |
| Objectives .....  | 4-1        |
| Overview of Additional Surfacing Commands .....         | 4-1        |

---

## Contents

|  |      |
|--|------|
| Extend Surface . . . . .                                 | 4-2  |
| Offset Surface . . . . .                                 | 4-3  |
| Copy Surface . . . . .                                   | 4-4  |
| Trim Surface . . . . .                                   | 4-5  |
| Delete Faces . . . . .                                   | 4-6  |
| Stitched Surface . . . . .                               | 4-7  |
| Round . . . . .  | 4-10 |
| Replace Face . . . . .                                   | 4-13 |
| Parting Split . . . . .                                  | 4-14 |
| Parting Surface . . . . .                                | 4-15 |
| Split Face . . . . .                                     | 4-16 |
| Review . . . . .   | 4-17 |
| Activity 1 – Surface Manipulation . . . . .              | 4-19 |
| Activity 2 – Putting It All Together . . . . .           | 4-51 |
| Activity 3 – Parting Split and Parting Surface . . . . . | 4-81 |

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## Chapter

# 1 *Introduction to Surface Modeling*

## Course Overview

### Course Description

This course addresses using Solid Edge modeling functions to build free form surfaces. Free form surfaces are those that cannot be constructed using standard solid modeling features. This includes creating bspline curves and other developed curves, building surfaces through one or more sets of curves, sweeping profiles along guides, and applying surface transitions between faces.

### Intended Audience

This course is intended for designers, engineers, manufacturing engineers, application programmers, CAD/CAM managers, and system managers who have the need to use Solid Edge surface modeling.

### Prerequisites

This course is intended for people with experience using Solid Edge. You should not take this course unless you have:

- Attended the Solid Edge Fundamentals course
- Basic understanding of parametric modeling
- Basic understanding of profile/sketch construction.

### Objectives

After successfully completing this course, you should be able to perform the following activities in Solid Edge:

- Create and edit bspline curves and developed curves.
- Create and edit surfaces using curves.
- Inspect surfaces and curves.
- Create and edit transition surfaces.
- Create a solid having one or more surface faces.

## Introduction to Surface Modeling

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

The solid modeling style is characterized by 2D sketches/profiles, extrusions, revolutions, and lofts of those 2D sketches to form solids, blends on the edges of solids, and Boolean operations between solids. 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, and any of the properties set when a feature is created can be edited at any time.

The surface modeling style is characterized by control points used to define 2D and 3D curves by a variety of methods, and these curves can be used to define surfaces, again by a variety of methods. Solid Edge extends this style by making each point, curve, and surface an entity that knows how it was created. Like the solid modeling features, any surface entity properties set when an entity is created can be edited at any time.

### What is surfacing, and why use it?

With solid-based features, faces drive the model. Holes are used for alignment. Faces are used for mating and aligning. Edges are rounded for safety and strength. Edges and faces are mainly analytic based. The solid modeling approach is typically an addition or removal of material. The product's function is the primary concern. Aesthetics is a secondary requirement.

With surface-based features, edges drive the model. Curves are a major part of model definition. The highlight lines, silhouette edges and flow lines of a model are important. Surface shape is still important. Edges and faces are mainly bspline based. The designer typically starts with a wire frame and then adds surfaces. Aesthetics is the primary concern and key element in designing consumer products. The product's function is secondary.

## Surface Modeling Overview

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

An analytic entity type consists of:

- 2D: Lines, arcs, circles, ellipse, parabola, hyperbola
- Intersection of plane and cone
- 3D: Cubes, spheres, cylinders, cones, torus

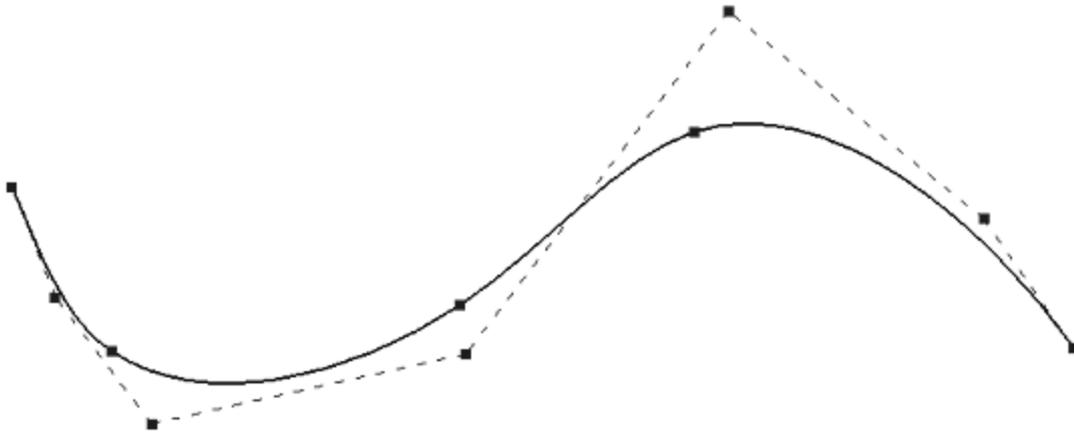
A bspline entity type consists of:

- 2D: constructed bspline curves, derived curves
- 3D: derived bspline curves

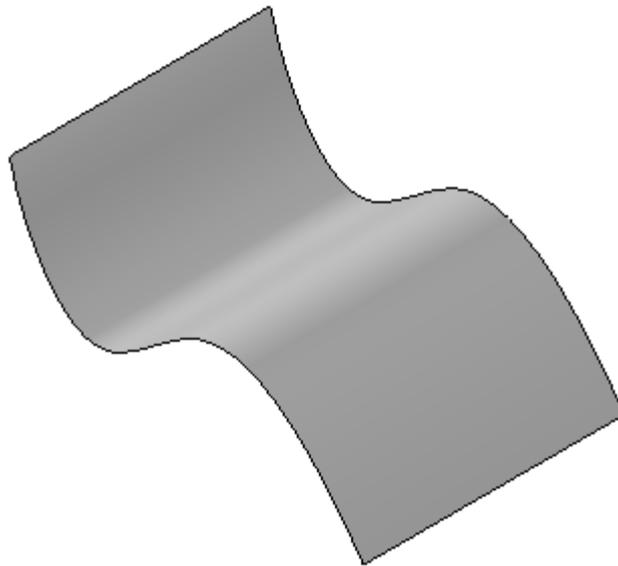


Originally, a spline was a stylist's tool made from wood or thin metal and used to draw a curve through points.

Shown below is a 2-D bspline.

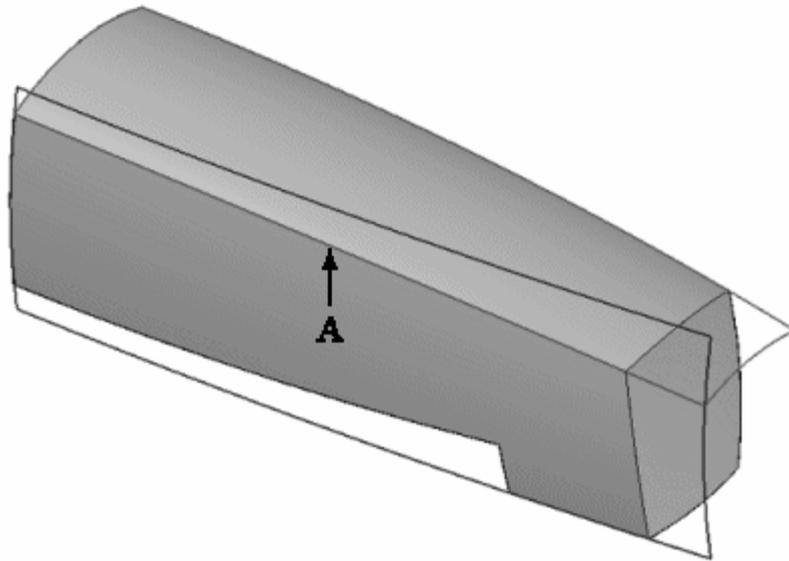


Shown below is a 3-D surface.

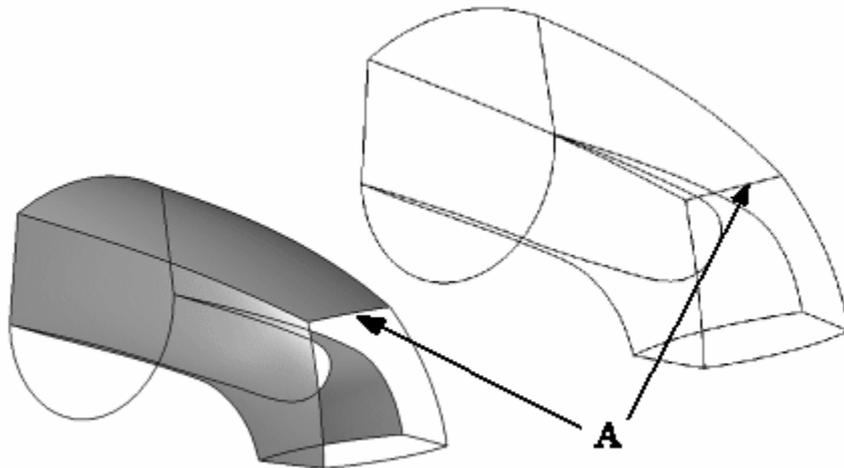


3D: bspline surfaces  
Auxiliary parameter:  $u, v$

A solid modeling method using revolved features results in no edge control and difficult edits. Edge (A) is a result of the intersection of two revolved surfaces. You get what you get.

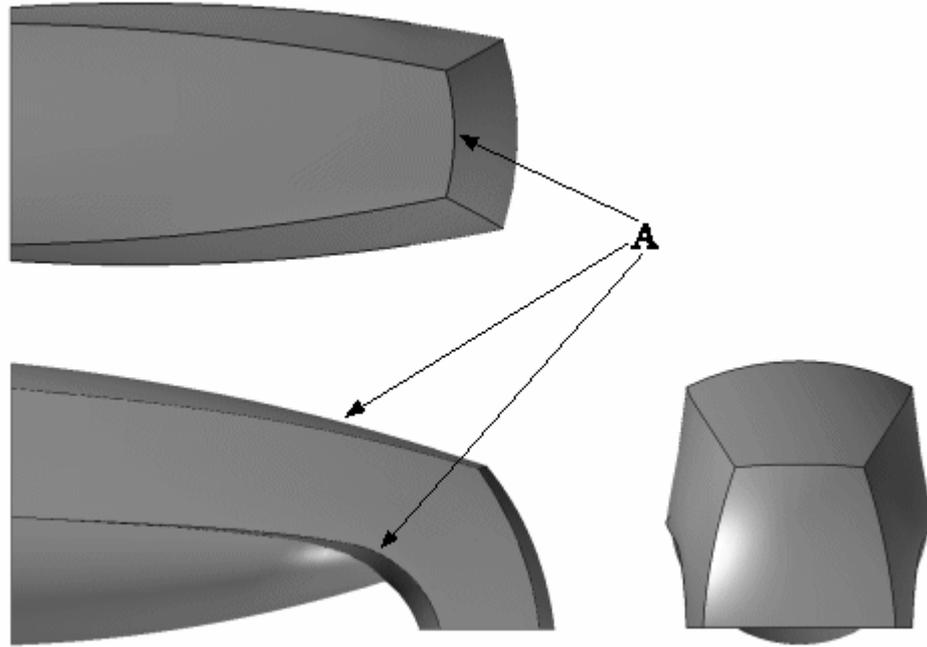


A surface modeling method results in exact edge control and edges are through character curves. User has control over edges such as (A).



## Definition: Character Curves

Character curves generally have no structural value. The model contains “Hard Edges”, which are actual edges used to help define the flow of a surface. These edges are typically of importance for aesthetic definition. The model contains “Soft Edges”, which are horizon edges, typically visible from front, top, and end views. These edges are important in defining the overall shape of the model. Modifying character curves (A) modifies the shape of the model.



## Surface Modeling Workflow

Step 1: Create Control Drawings

- Draw or import 2D sketches on reference planes
- Draw 2D sketches in draft and copy/paste
- Insert graphic images and trace over

Step 2: Use 2D geometry to develop 3D curves

- Project curves to find their intersections

Step 3: Use 3D curves to develop surfaces

- Some additional 3D curves are obtained from surfaces

Step 4: Create a solid and add appropriate solid based features such as holes, stiffening ribs, rounds, thin wall operations, etc.

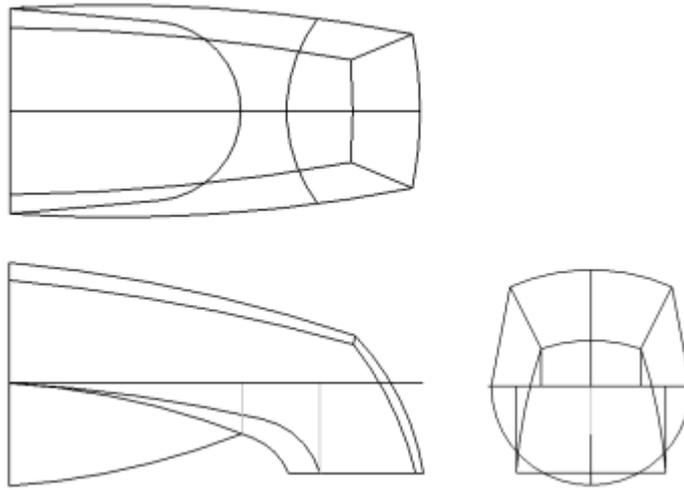
Step 5: Tweak

- Analyze curvature and edge continuity

- Modify character curves, surface tangency, etc.

### Definition: Control Drawing

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).



You can create control drawings directly in the Part environment by drawing directly on reference planes. Pierce points help you in connecting curves.

You can create control drawings in the Draft environment. Copy and Paste the geometry from Draft into Part. You can use Create 3D or import sketches and use existing geometry.

### Definition: Pierce Points

A pierce point is the point of intersection with profile element and active sketch plane. Pierce points are extremely valuable in aligning curves.



## Surface Modeling Workflow Summary

### Step 1: Control Drawings

- Use the Line Color command to help distinguish edges, construction edges, etc. in the control drawing. Control drawings can end up with a lot of elements, thus the line color command can help to eliminate confusion of what each element is.



- 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 bsplines with few Edit Points.
- Make sure view sketches register.
- Build edge continuity into your sketches.

### **Definition: Register**

Ensuring respective curves from different views are connected or geometrically aligned.

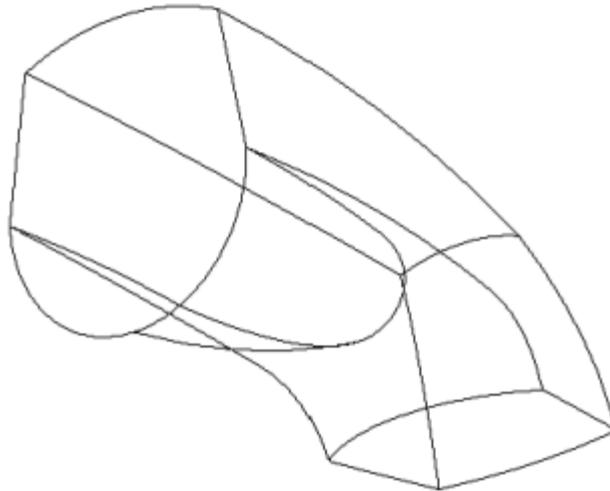
It is critically important that 3D curves are connected to ensure an accurate design.

You can also insert an image as an alternate control drawing process. This process would be used for reverse engineering. The inserted image is traced over to create the control drawing elements.

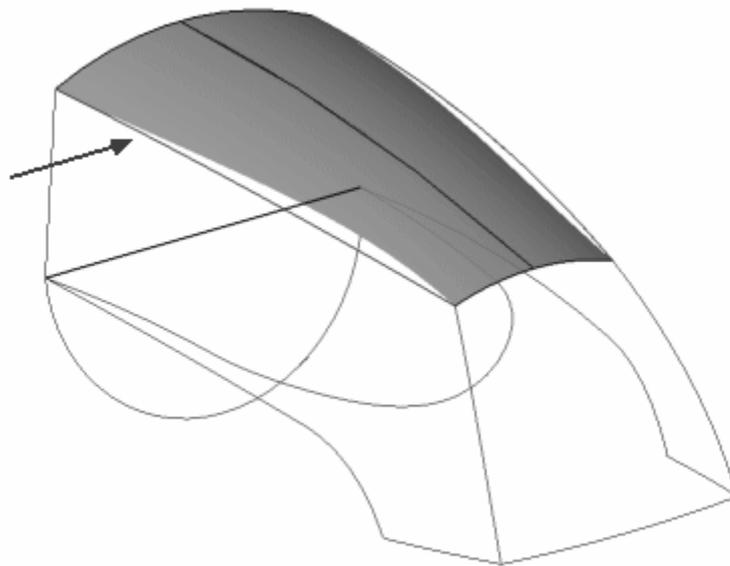


## Step 2: Creating 3D Curves

- 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.

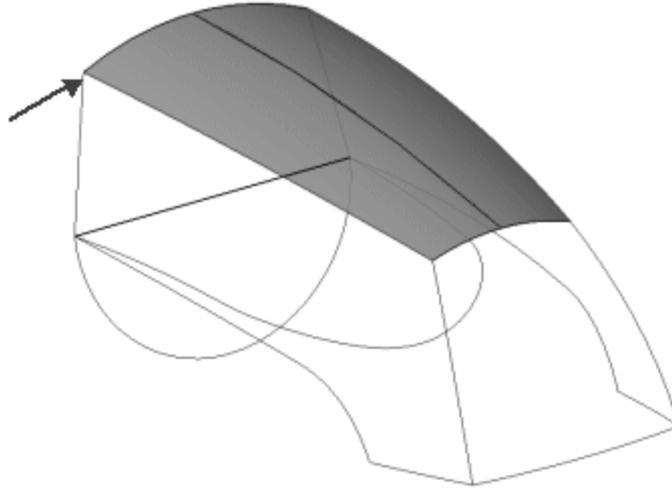


- Without 3D curves, character edges may not be captured.
- Lack of 3D edges eliminates design intent and adds more modeling.



- 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.

#### **Project Curve Method**

- Create a surface using character curves from one view.
- Use the Project Curve command to project the appropriate curves from another view.
- Hide auxiliary surface (these rarely are the actual surface).

#### **Intersection Curve Method**

- Create surfaces from two views.
- Use the Intersect Curve command to generate the intersection curve.
- Hide auxiliary surface (these rarely are the actual surface).

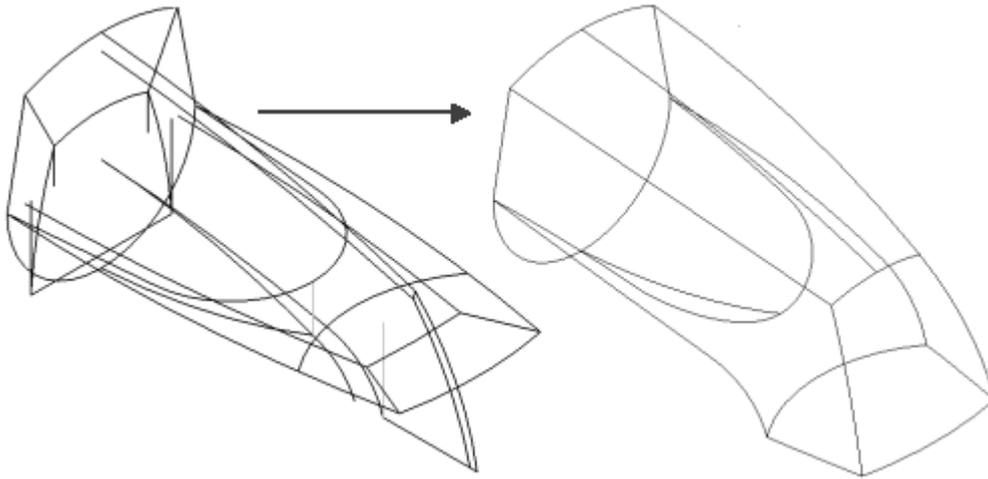
#### **Cross Curve Method**

- Select curves from two different views to create the projected curve.
- Simple and fast!

#### **Sketch Tear Off**

- Fast way of replicating sketches.
- New sketch can be parallel, along curves, perpendicular, angular.
- Options - associative, copy, or a move.

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



### Step 3: Surface Creation

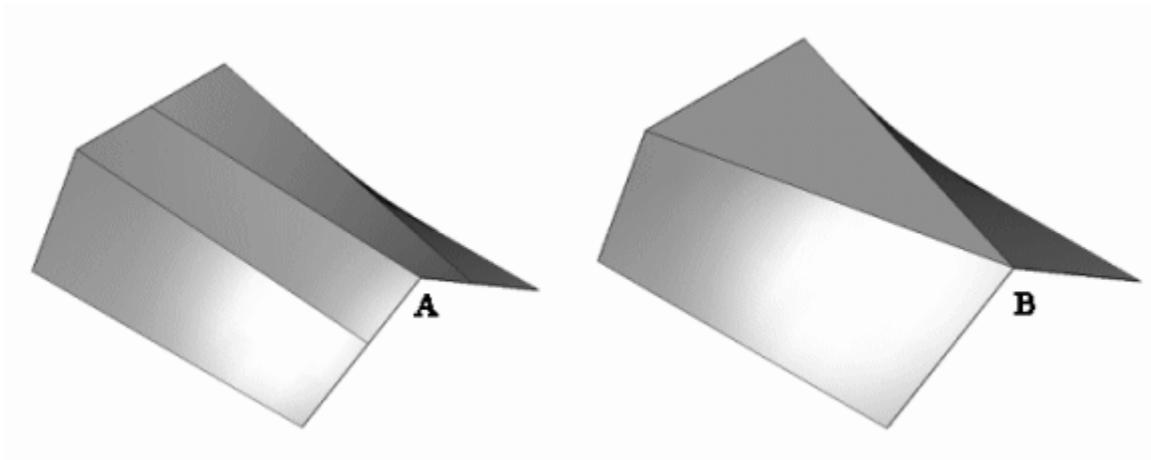
- BlueSurf
  - Inputs are guides and sections.
    - ◇ 1 section, 1 guide.
    - ◇ 2 sections n guides.
  - Guides and sections can be automatically generated from the surface.
  - Automatic creation of BlueDots.
  - Section and guide tangency control.
  - Replaces the Loft command.
  - Edges without guide definition are interpolated from section ends.
  - Single face is created.
  - Option for vertex mapping.
- Swept Surface
  - Inputs are guides and sections.
    - ◇ 1 section / 1 guide (typical use).
    - ◇ 3 guide maximum.
  - Edges without guide definition use input guide for shape.
- Surface By Boundary (Bounded Surface)

- N-sided patch.
  - ◇ 3 or more sides.
- Tangency control between selected adjacent faces.
- Fast easy way to patch holes in models.
- Can be used for legacy data.

### **Definition: Vertex Mapping**

Mapping technique that helps create flow control between section vertices.

If there is a vertex count mismatch between sections, equally spaced vertices are used on each section.

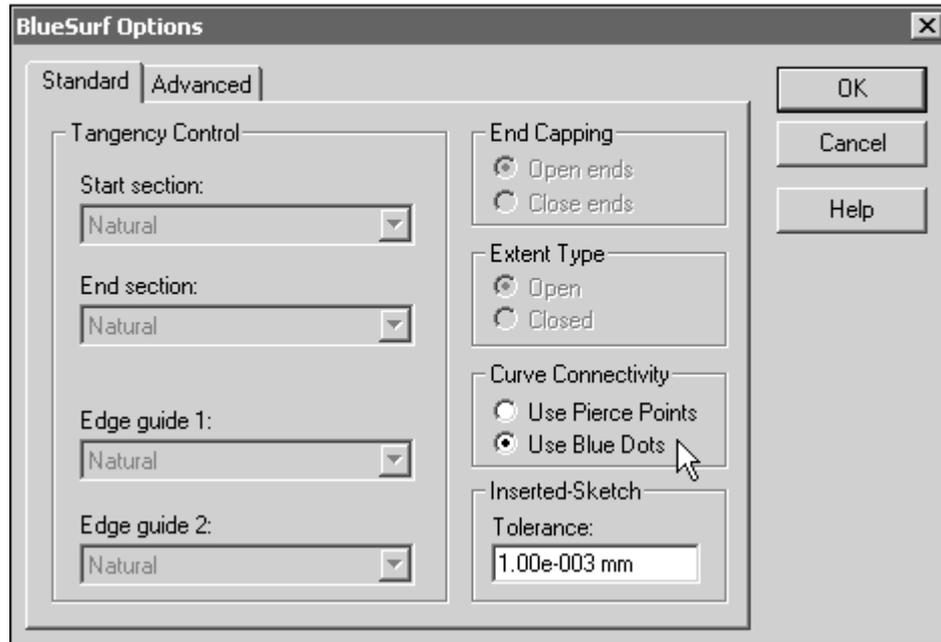


(A) Default vertex map.

(B) User defined vertex maps. Improved flow control.

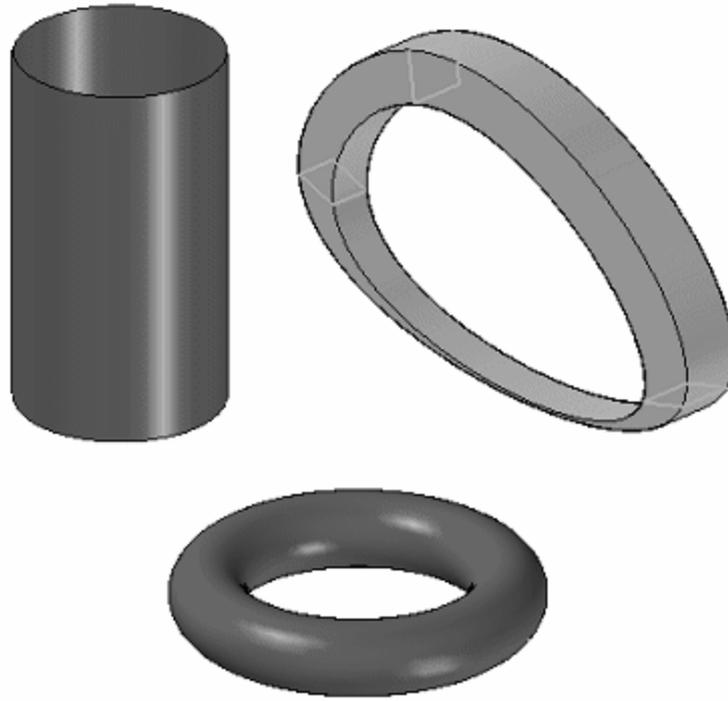
- Difference in results from BlueSurf and Swept Surface
  - For undefined Edges, BlueSurf edges are interpolated.
  - For undefined Edges, sweep uses input guide(s).
  - BlueSurf allows periodics.

The BlueSurf command can add BlueDots automatically. On the BlueSurf Options dialog box, you must set the Curve Connectivity to the Use BlueDots option. The Use Pierce Points option is the default setting.



### **Definition: Periodic**

Closed surface or curve



### **Definition: BlueDot**

- Highly editable point where two curves are connected.
- Each curve has control option for edit type (shape, local or rigid).
- Connectivity of curves is order independent.
- Edit of curves is order independent.
- History-free curve editing.
- You can see changes in real-time.

### **Step 4: Create A Solid**

- Stitch
- Add solid-based features
  - Thinwall
  - Stiffening ribs
  - Holes
  - Rounds

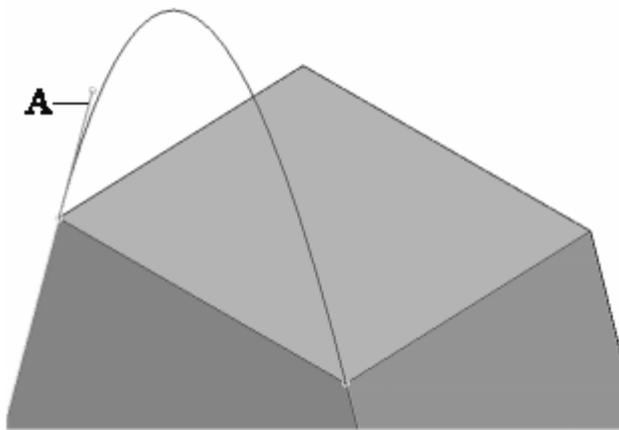
- Web network
- Lip / Groove

### Step 5: Tweak

- Analyze edge continuity
  - Curvature Comb
  - Zebra Stripes
- Edit character curves
- Edit tangent vectors
- Edit vertex mapping

### Definition: Tangent Vectors

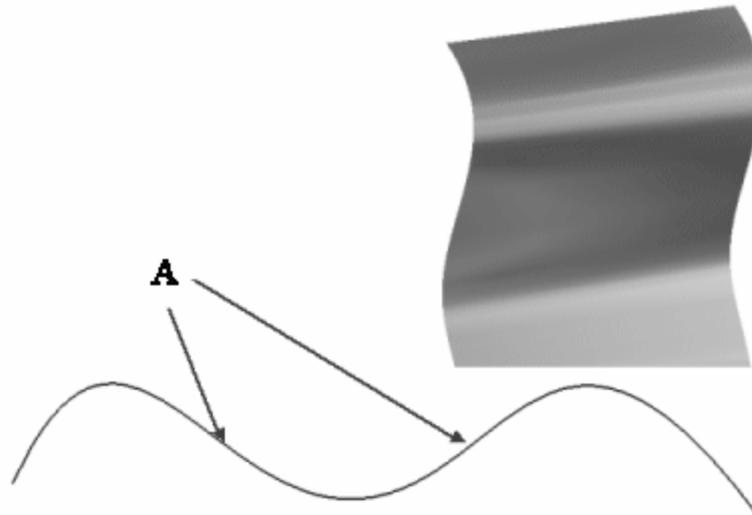
Drag handles that let you change shape of curve or surface at the point of tangency with adjacent curves or surfaces.



- Curvature Comb
  - Quick visual indication of sketch continuity.
  - Numerical reporting on radius of curvature.
  - Adjustable magnitude and density.
  - Illustrates points of inflection.
  - Determined by magnitude = scale x 1/radius.
  - Lines have infinite radius so magnitude = 0.

## Definition: Points of Inflection

Point where slope of curve changes in sign (positive to negative).



### (A) Inflection points

- Zebra Stripes

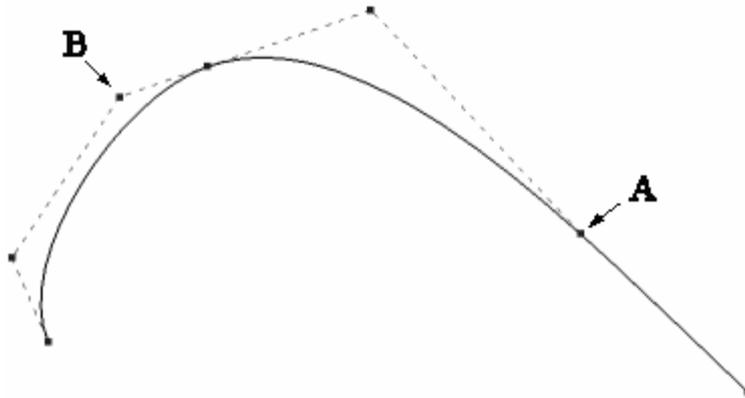
Striping gives quick indication of continuous edges between faces.

- Edit curves

- Edit Points
- Edit through BlueDots
- Control Vertices
- Change Curve type
- Analytics to bsplines
- Shape Edit/Local Edit/Rigid



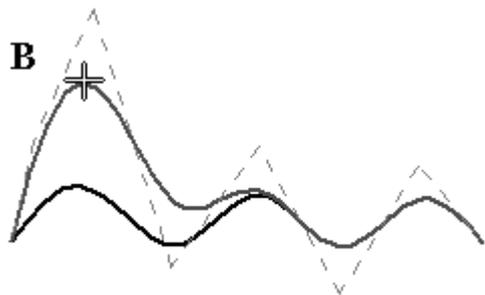
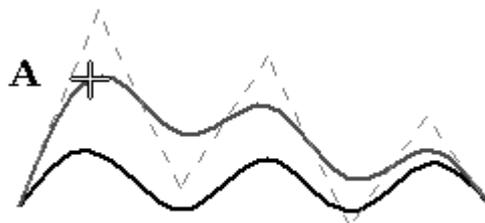
Surfaces are only as good as the underlying curves.

**Definition: Edit Point, Control Vertex**

(A) edit points lie on the curve

(B) control vertex points lie on the control polygon

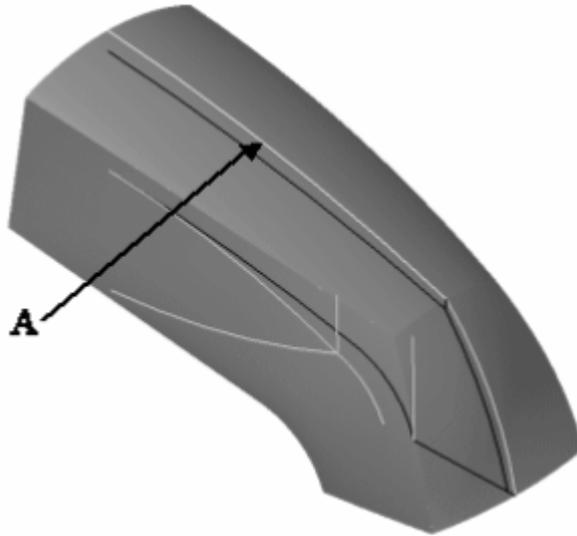
- Curve Edit Control Options
  - Shape Edit: A method of preserving curve shape.
  - Local Edit: Modifies a local portion of the curve.
  - Rigid: Transformation of the curve (move).



(A) Shape Edit

(B) Local Edit

- Convert analytics to bsplines
  - easy initial approach with analytics.
  - add cool curvature later.
  - dimensions and constraints preserved.
  - history tree recomputes after you convert.



(A) Simple initial analytic arc can be converted to a bspline for improved shape control.

## Summary

You should know the differences in solid modeling and surface modeling. You should also have a clear understanding of when to use a surface modeling approach and when to use a solid modeling approach. In surface modeling, curves control the model. In solid modeling, faces control the model.

Understanding the surface modeling workflow is important for success in creating and editing surfaces.

- Step 1:** Create Control Drawings
- Step 2:** Use 2D geometry to develop 3D curves
- Step 3:** Use 3D curves to develop surfaces
- Step 4:** Create a solid and add appropriate solid based feature
- Step 5:** Tweak



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## Chapter

# 2 *Creating and Editing Curves*

## Objectives

After completing this module, you will be able to:

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

## Overview of Splines

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

For the remainder of this course, the term curve will be used instead of splines. Just remember that curves are splines. There will be two types of curves discussed: constructed and derived. You have direct control of constructed curves but derived curves are controlled by the method used to create them. Derived curves cannot be edited directly.

## Curve Command

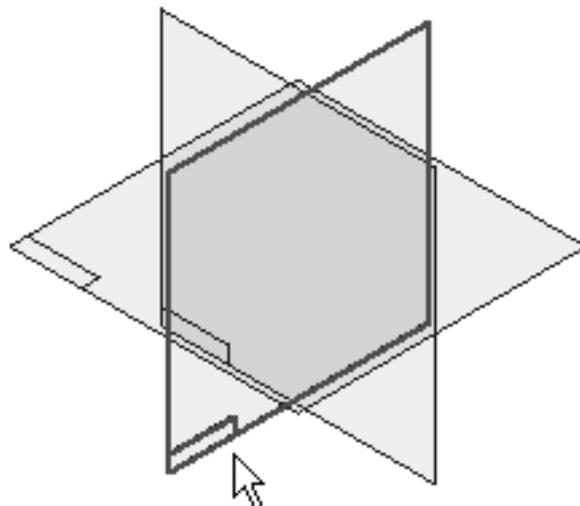


The Curve command is where we will begin. We will cover how to create and edit a curve. To create a curve you will need to select a reference plane to draw a sketch on.

On the command menu, click the Sketch command.



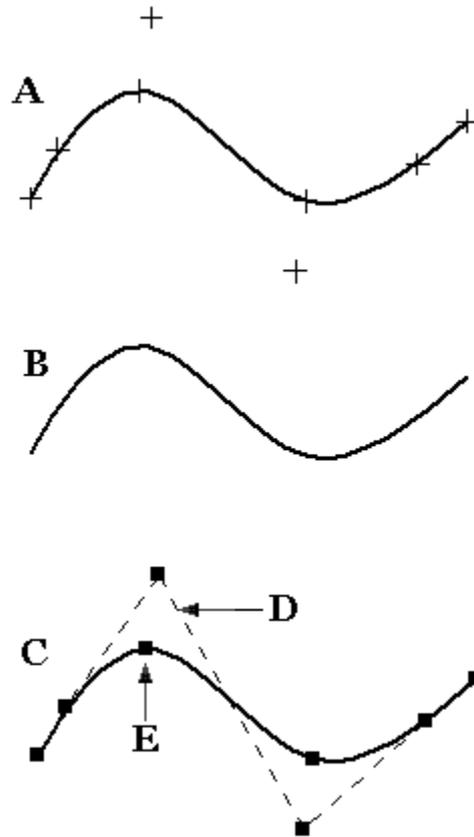
Select a reference plane.



On the draw menu, click the Curve command.



You will need to left-click at least three points to define a curve. The image below shows a curve with four points.



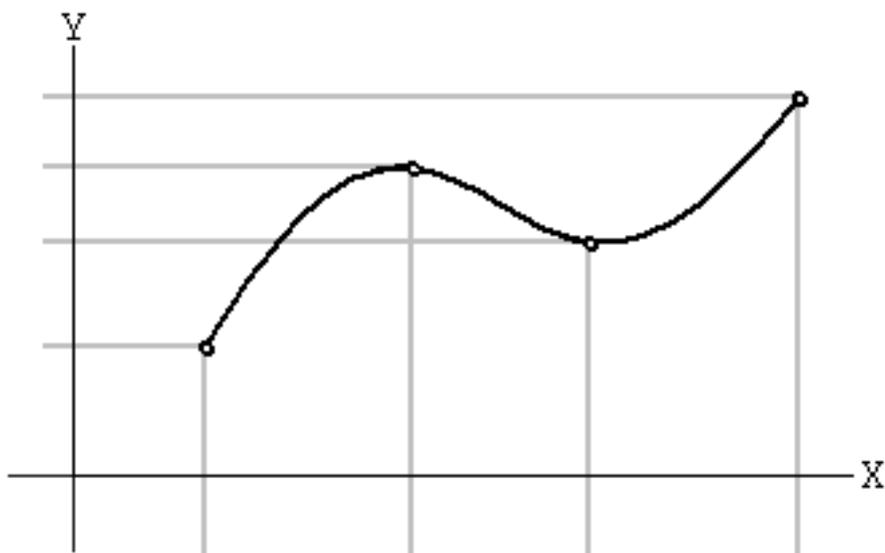
- (A) Curve with control points (edit points) displayed
- (B) Resulting curve
- (C) Edit Curve Display
- (D) Control Polygon
- (E) Edit Point

## Curve Definition

### Defining a Curve

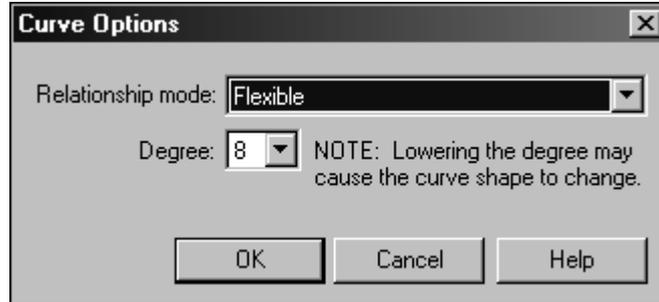
The order of a curve is equal to the degree of the curve, plus 1 (Order = Deg +1).

A polynomial curve is defined as:  
 $x(t) = x_0 + x_1(t_1) + x_2(t_2) + x_3(t_3)$   
 $y(t) = y_0 + y_1(t_1) + y_2(t_2) + y_3(t_3)$



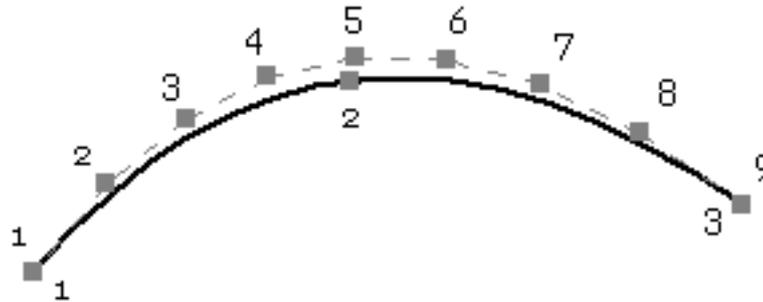
## Determining Control Vertices

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

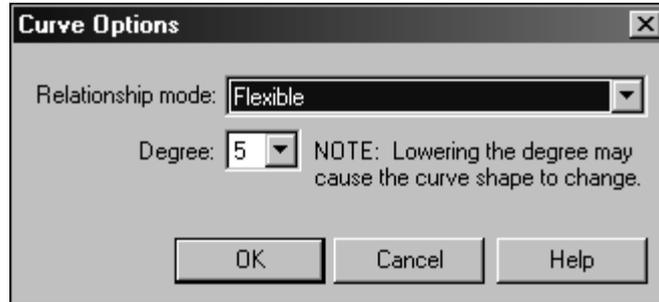


Example:

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

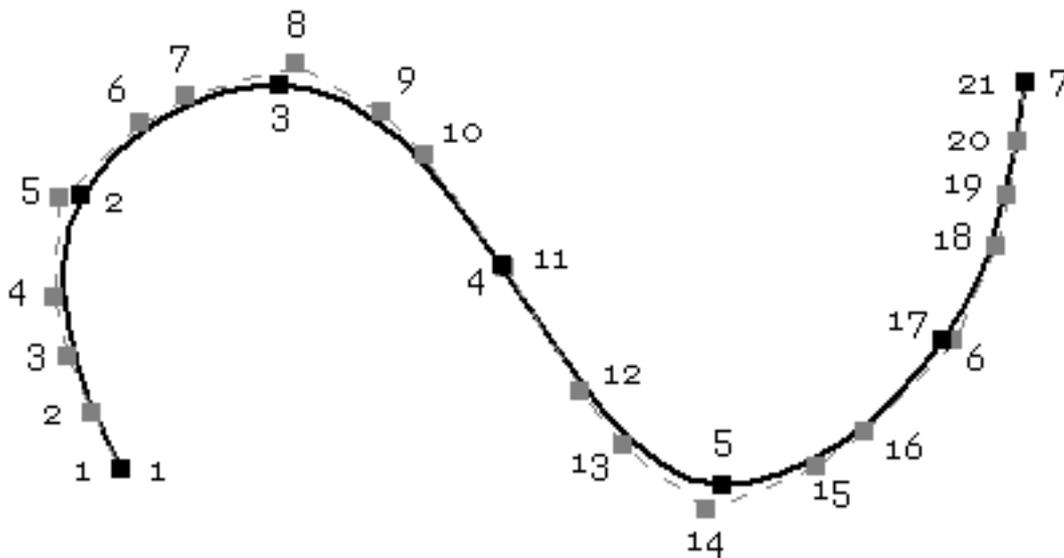


If the number of edit points is  $\geq 4$ , the number of control vertices is  $(n+2) + \{(n-1) \times (k-4)\}$ .  
Where  $n$  = Edit Points, and  $k$  = Order.



Example:

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



## Curve Edit

You can edit b-spline curves at any time. There are two methods for editing curves.

Method 1: Edit Profile mode

To edit a curve in the sketch mode, select a curve in the Part window and then click the Edit Profile button on the ribbon bar. You can now edit the curve with all curve edit options available.

Method 2: Dynamic Edit mode

To edit a curve in dynamic mode, select a curve in the Part window and then click the Dynamic Edit button on the ribbon bar. Select the curve and the control polygon and edit points will display. 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 Add/Remove Points and Curve Options buttons are disabled in dynamic edit mode. These options are only available in Edit Profile mode.



Add/Remove Points



Show Polygon



Show Curvature Comb



Shape Edit

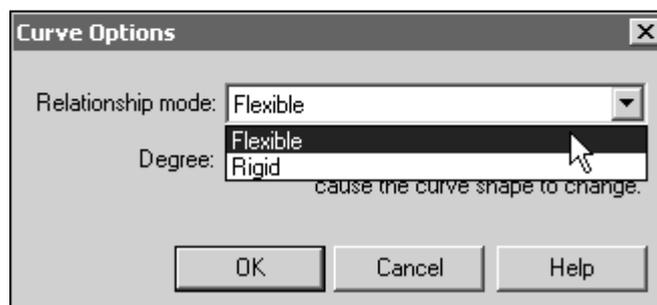
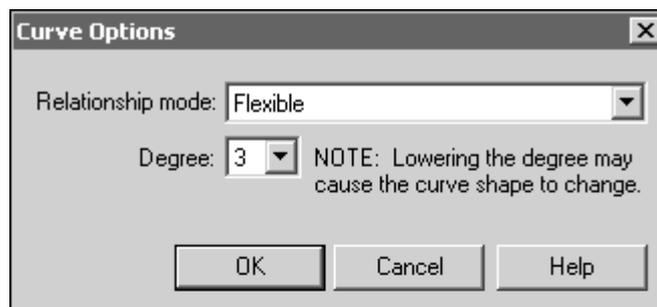


Local Edit



Curve Options

Curve options are shown below.



**Relationship modes:**

Flexible allows you to modify the curve shape during BlueDot edits or an edit to a curve connected to another curve with a connect relationship applied.

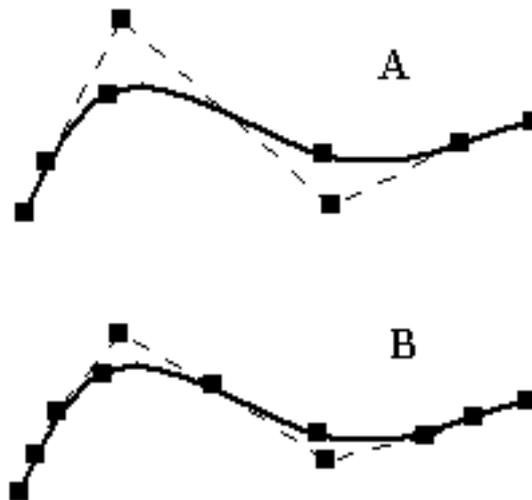
Rigid locks the curve shape. Curve shape will not change during BlueDot edits or an edit to a curve connected to another curve with a connect relationship applied.

These relationships are applied to external edits (BlueDot or dynamic edit). In Edit Profile mode, a curve set to rigid can have its shape modified.

**Degree:**

The Curve Options dialog box allows control of curve degree and relationship mode. The default degree is three. The higher the degree, the more control you have in making local curve edits.

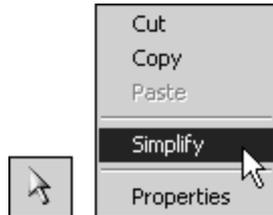
Notice in the illustration below that curve (A) shows the default degree of 3. Curve (B) has degree increased to 4. The higher the degree, the more control you have for editing the curve.



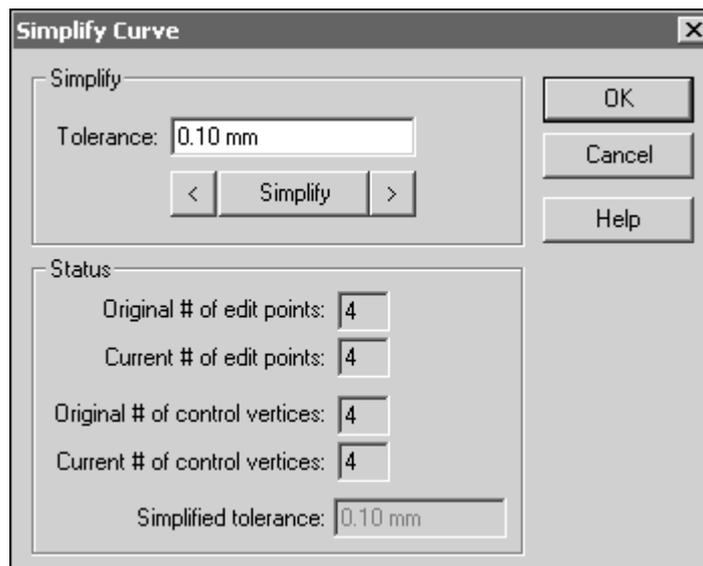
## Simplify

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.

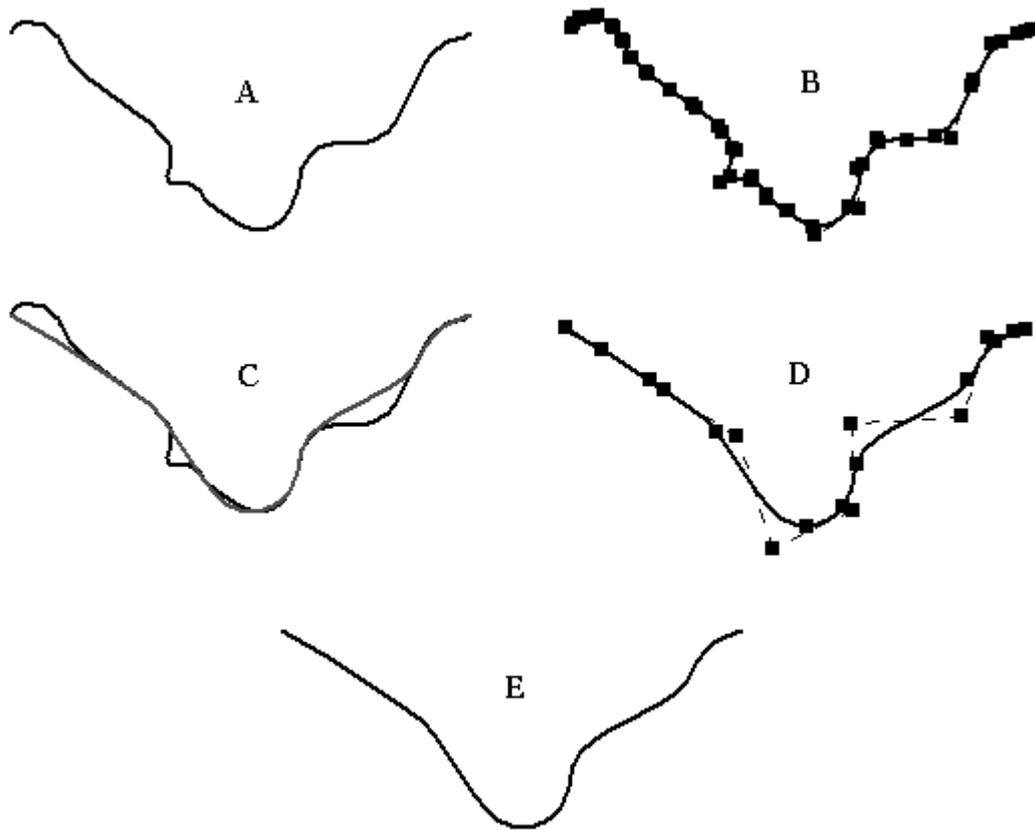
Simplify curve is a tool that allows you to define a tolerance to reduce the number of edit points and control vertices.



Visualize tolerance as a tube. The original curve is at the center of a zero diameter tube. As the tube diameter (tolerance) increases, edit points are reduced as the curve shape is confined to the tube diameter. As the tolerance increases the curve is simplified. You can visually observe the curve simplification process as the tolerance increases using the right arrow in the dialog box.



The following is an example of a curve with a large number of edit points and control vertices. Simplify curve was used to reduce the number of points. The curve shape changed slightly. You can visually observe the curve changes as the tolerance increase.



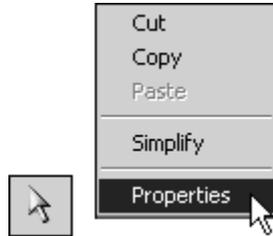
- (A) Original curve
- (B) Original curve in edit mode with 25 edit points and 27 control vertices
- (C) Dynamic display as curve simplify tolerance is increased
- (D) Resulting simplified curve reduced to 7 edit points and 9 control vertices
- (E) Simplified curve



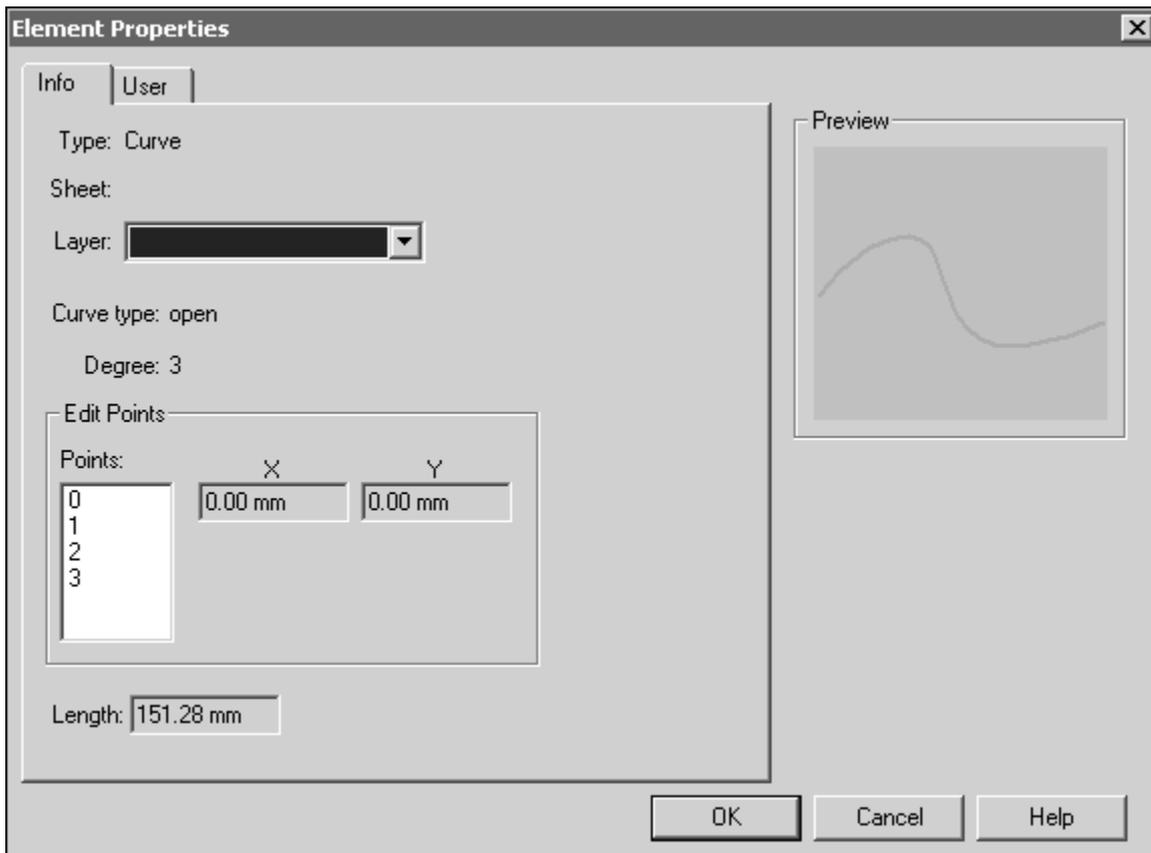
If the number of edit points is equal to 2 then the simplification continues by removing control vertices. Simplification ends when the number of control vertices is equal to the curve order.

## Element Properties

Element properties of a curve can be displayed while in the Edit Profile mode. Right-click on the curve and then on the pop-up menu, click Properties.



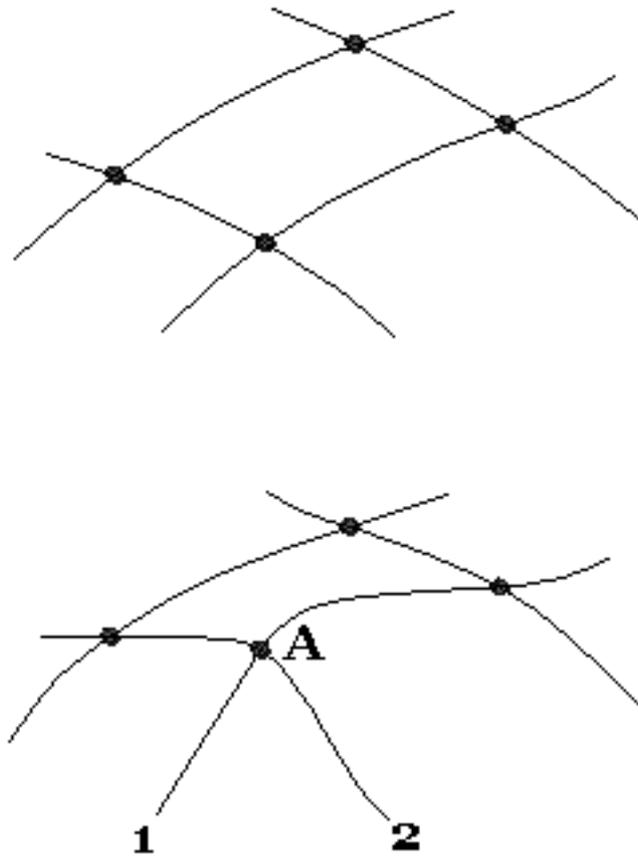
The Element Properties form is informational only. You cannot modify an element using the form.



### BlueDot



A BlueDot is a point where two curves or analytics (or a combination of the two) connect. No more than two curves or analytics (or a combination of the two) can be BlueDot connected. The BlueDot is an editable point. As the BlueDot location is changed, the connected curves/analytics follow the BlueDot location. Curves have a control option for edit type (shape, local or rigid), which is applied as the BlueDot location is edited. An analytic has no shape control option. Connectivity of the two curves/analytics is order independent. Edit of curves/analytics is order independent. BlueDot editing is a history-free type of curve editing. You can see changes in real-time.



Notice in the previous illustration that as BlueDot (A) changes location, curves 1 and 2 remain connected. The shape of curves 1 and 2 change. This example illustrates the edit curve option set to “shape edit”. Both curves maintain their shape.



## BlueDot Creation

You create a BlueDot by identifying the two curves to connect. No more than two curves can be BlueDot connected. The first curve you select will move to intersect the second curve. The second curve is unchanged (it maintains its location and shape).

There are several solutions possible when creating a BlueDot. The solution is dependent on how you select the curves. Each curve has four select zones (two endpoints, a midpoint, curve). The endpoints and midpoint display a red dot for selection. If you select a midpoint or endpoint, that point on the curve will move to the zone you select on the second curve. Experiment with the BlueDot command to best understand the several solutions.

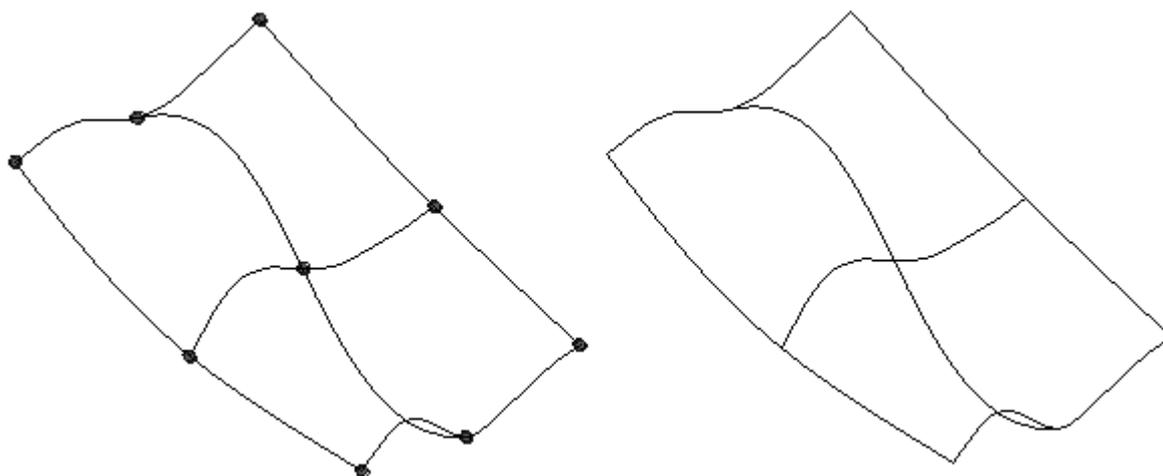
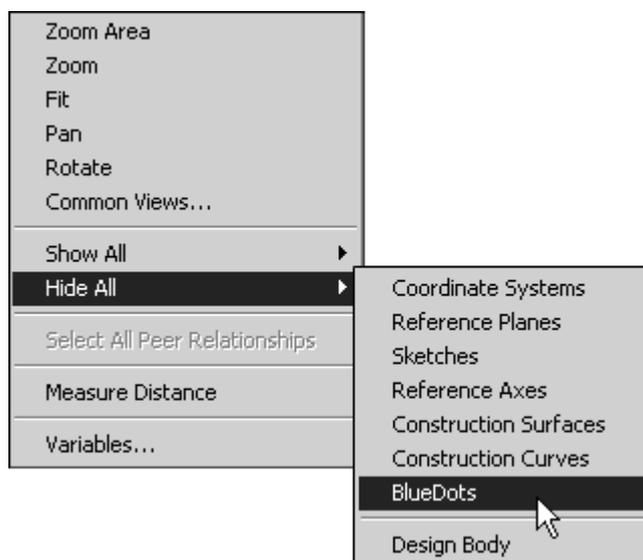
Shown below are the select zones on a curve.



You can only select analytics at their endpoints.

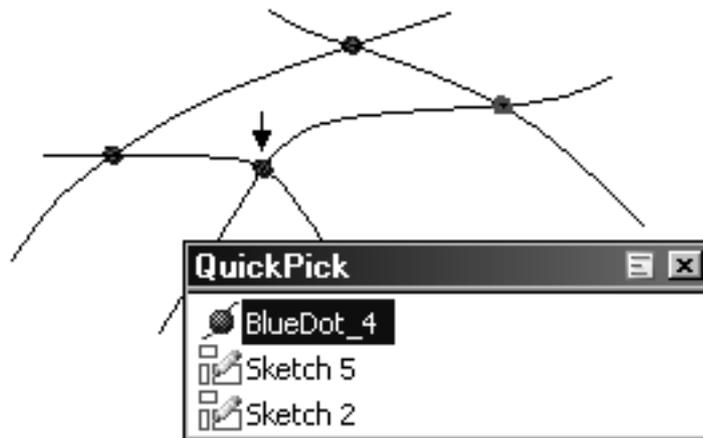
### BlueDot Display

1. Click on Tools>Hide All>BlueDots.
2. Right-click in the part window and click on Hide All>BlueDots.

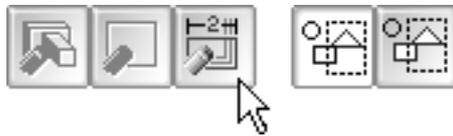


## BlueDot Edit

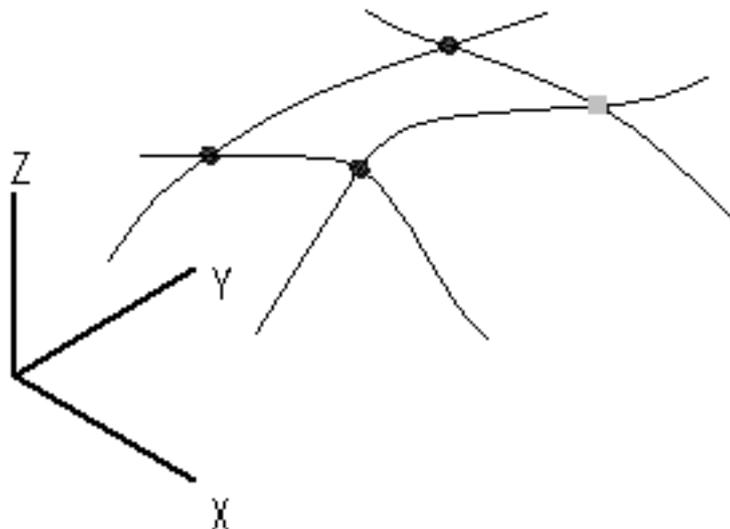
To edit a BlueDot, click the Select tool and pause the cursor over the BlueDot. Use QuickPick because there are normally three elements to choose from (two curves and the BlueDot). From the QuickPick select box, click the BlueDot.



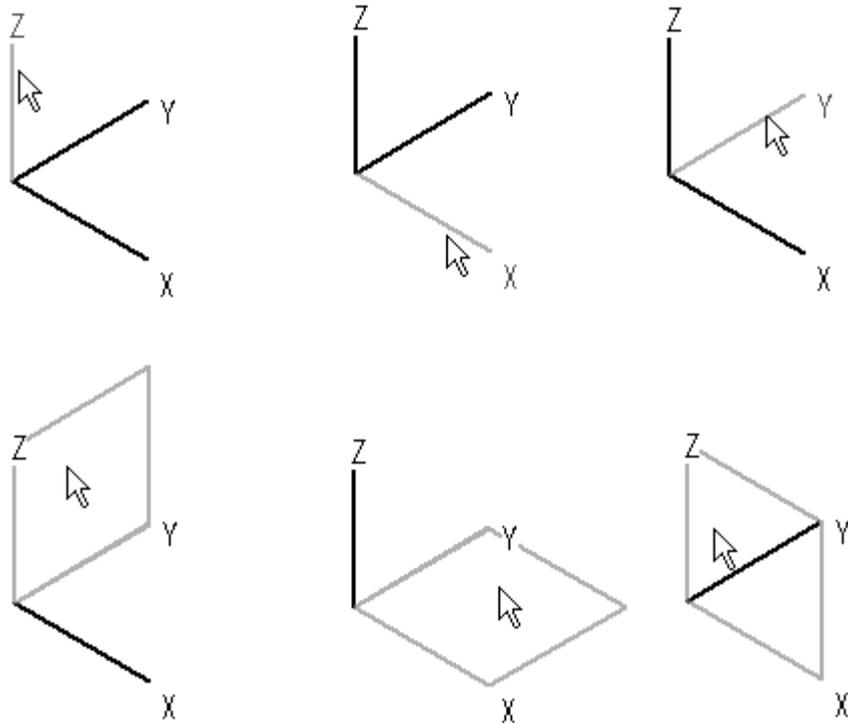
On the ribbon bar, click Dynamic Edit.



A 3D triad displays that allows you to define the BlueDot edit direction. The choices are to lock the movement in the X, Y or Z direction by clicking on the appropriate axis. You can also choose to lock movement to the XY, YZ or XZ planes. If you do not select any axis or plane from the triad, the movement is free in all directions.

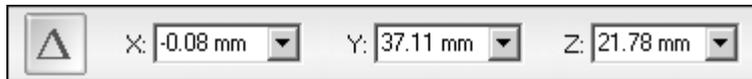


The illustration below shows possible selections from the 3-D triad.

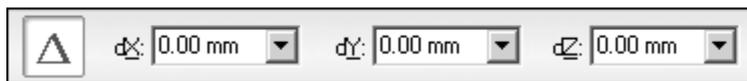


You can also lock to an axis by pressing Z until the preferred axis highlights. Lock to a plane by pressing X until the preferred plane highlights. Press C to clear all locks.

Notice that the coordinates of the BlueDot are displayed on the ribbon bar. You can drag the BlueDot to new locations or enter the coordinate values. If you select a direction on the 3D triad, focus on the ribbon bar will be on that direction field. For example if you select the Z-axis, you can only enter a new Z coordinate. If you select a plane, you can only enter coordinates in the 2 directions of the plane.

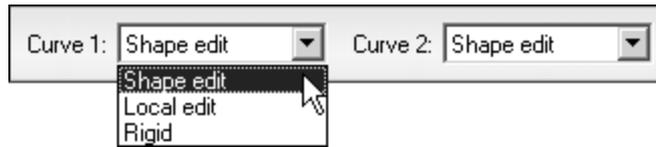


To move a BlueDot a specified distance, on the ribbon bar click the Relative/Absolute Position button.



Selecting a direction on the 3D triad controls the focus of the delta fields (dX, dY, dZ) on the ribbon bar. For example, if you select the Z-axis, you can only enter a Z delta distance. Press “Enter” to apply the delta distance. Another “Enter” applies the delta distance again.

You can set the shape control of the two curves connected to the BlueDot on the ribbon bar.



## Convert to Curve

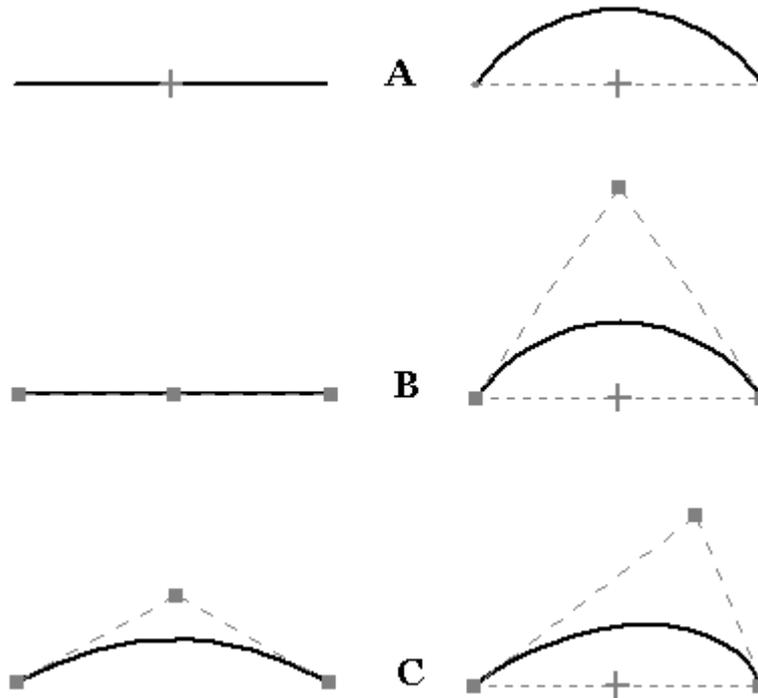


Sometimes you might create cross sections and guide paths used for generating surfaces with analytics (lines and arcs). The resulting surface created from analytic elements has limitations on how it can be edited. Lines remain linear and arcs remain circular. Convert analytic cross sections and guides to bspline curves to provide flexibility and control over the surfaces they create.



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

After you convert an analytic to a curve, you can modify its curve properties. The Convert to Curve command defaults to a degree of 2. You can increase the degree and edit points added for more control.



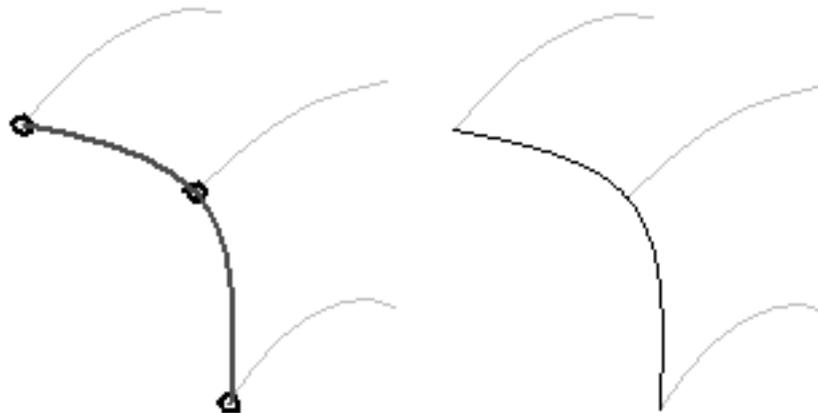
- (A) Analytic line and arc element
- (B) Analytic elements converted to curves
- (C) Curve edits

## Additional Curve Creation Methods

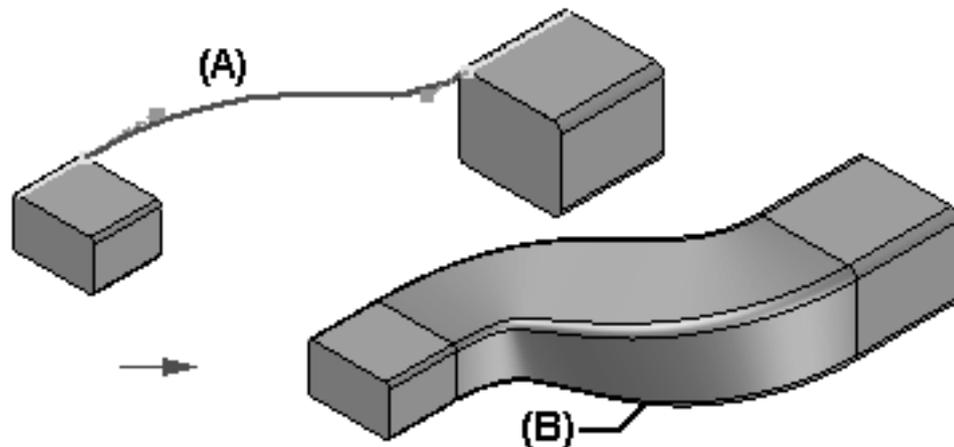
### Keypoint Curve



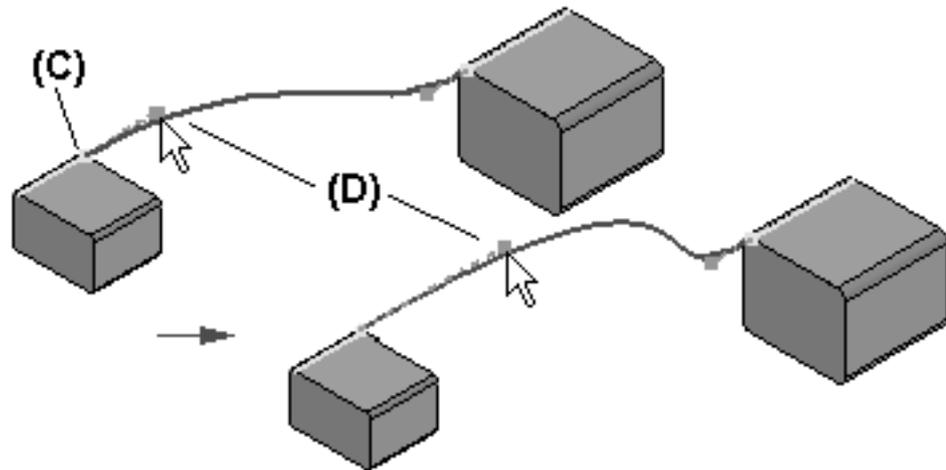
A keypoint curve is a 3D curve that uses keypoints on wireframe elements, edges, and curves to control its shape. You can also use point elements to control a keypoint curve. You can define points in space for the curve definition. You cannot edit a keypoint curve the same way you edit curves. The control comes directly from the keypoints or 3D coordinates for points in space.



An example of the use of Keypoint Curve is shown below. You can use this command to create a bridge curve (A), which can be used as a path for a swept feature (B).



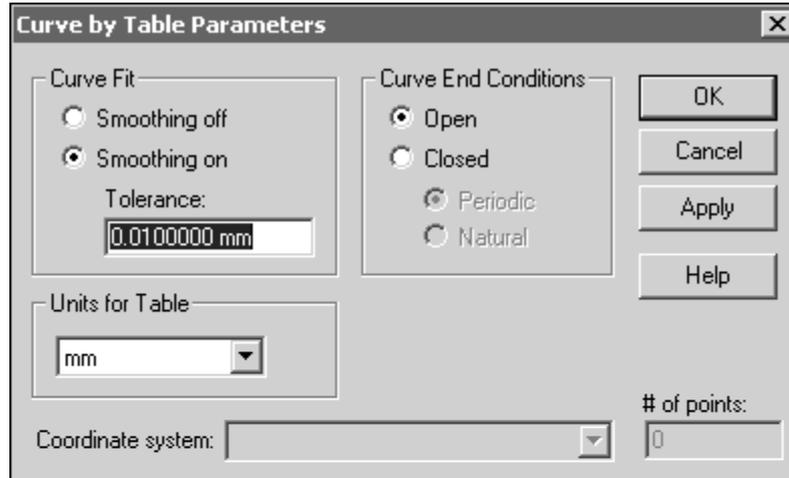
When you select a keypoint on a wireframe element or edge as the endpoint (C) of the curve, the curve is created tangent to the wireframe element or edge you selected. You can also modify the radius of curvature of the curve by dragging the curve handle (D) to a new location. If the modified curve was used as a path for a swept feature, the swept feature will also update.



## Curve by Table



Curve by Table 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.



| Worksheet in Part 1 |    |     |   |   |
|---------------------|----|-----|---|---|
|                     | A  | B   | C | D |
| 1                   | 0  | 0   | 0 |   |
| 2                   | 10 | 20  | 0 |   |
| 3                   | 15 | 40  | 0 |   |
| 4                   | 20 | 60  | 0 |   |
| 5                   | 25 | 80  | 0 |   |
| 6                   | 20 | 100 | 0 |   |
| 7                   | 15 | 120 | 0 |   |
| 8                   | 10 | 140 | 0 |   |
| 9                   | 0  | 160 | 0 |   |
| 10                  |    |     |   |   |

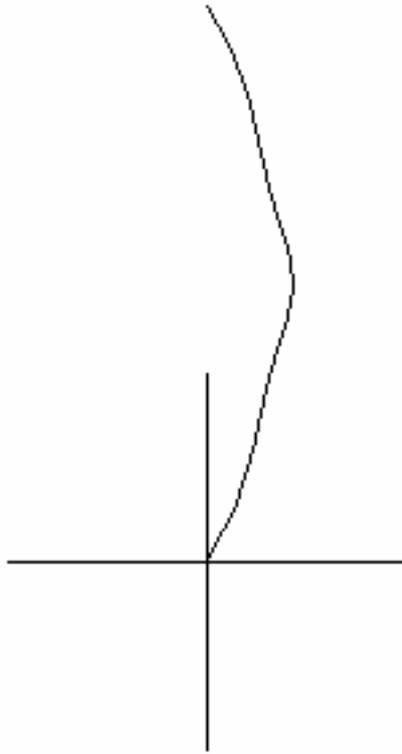
Column A = Global X Coordinate Value

Column B = Global Y Coordinate Value

Column C = Global Z Coordinate Value

Rows 1 – N = Curve Points in sequence

The curve below is the result of data shown in the previous spreadsheet.



The following are possible error messages resulting from Curve by Table:

**Invalid Geometry: Edit Feature Inputs**

The data points in the spreadsheet are incorrect. Check the spreadsheet and make certain you have input at least two rows of X,Y,Z coordinates, have not skipped any cells, and have not entered data points that define a curve that runs back through itself either in 2-D or 3-D.

**Curve Is Self Intersecting**

The curve runs back through itself in 2-D or 3-D. Change the data points in the spreadsheet.

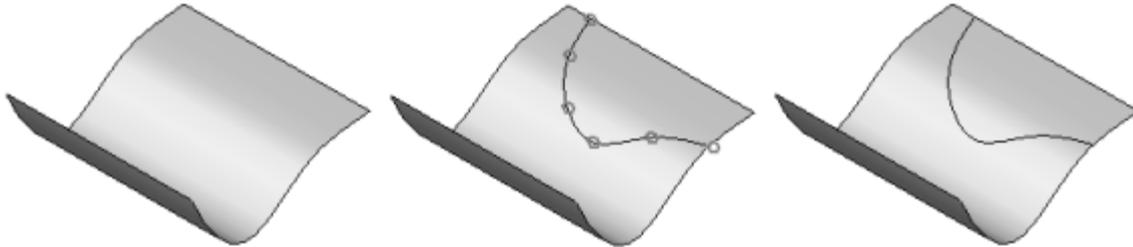
**Units Set Out Of Range**

The units are too large. Make sure the diameter of the model is less than five kilometers.

**Curve By Table Feature Failed**

Closed and Periodic options are set on the Curve Table Parameters dialog box, and the first and last data points in the spreadsheet are coincident. Change one of the coincident data points.

## Contour Curve



The Contour Curve command draws 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 traverse trimmed regions are trimmed.

When defining the points for the curve, you can use existing points that define the surface, such as vertexes, 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.

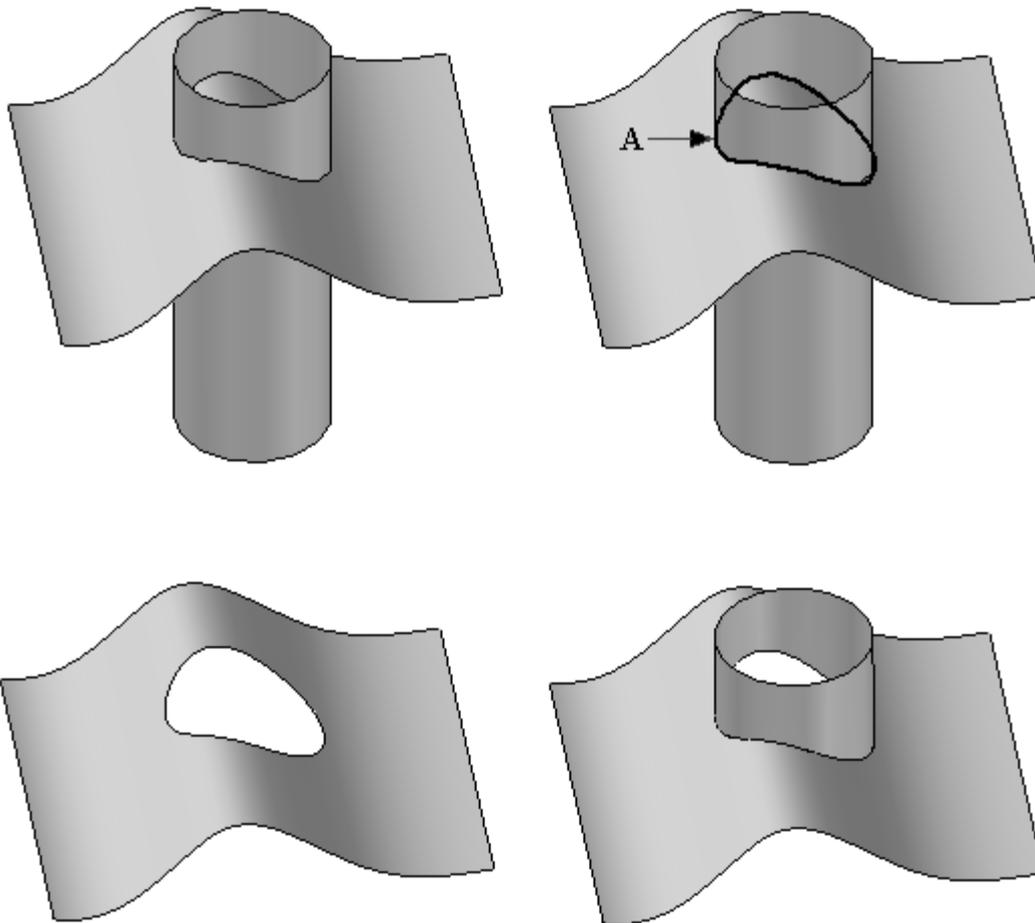


## Intersection Curve



The intersection curve command creates 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. The resulting intersection curve is associative to the two input surfaces. Editing the input surfaces will result in an automatic change to the intersection curve.

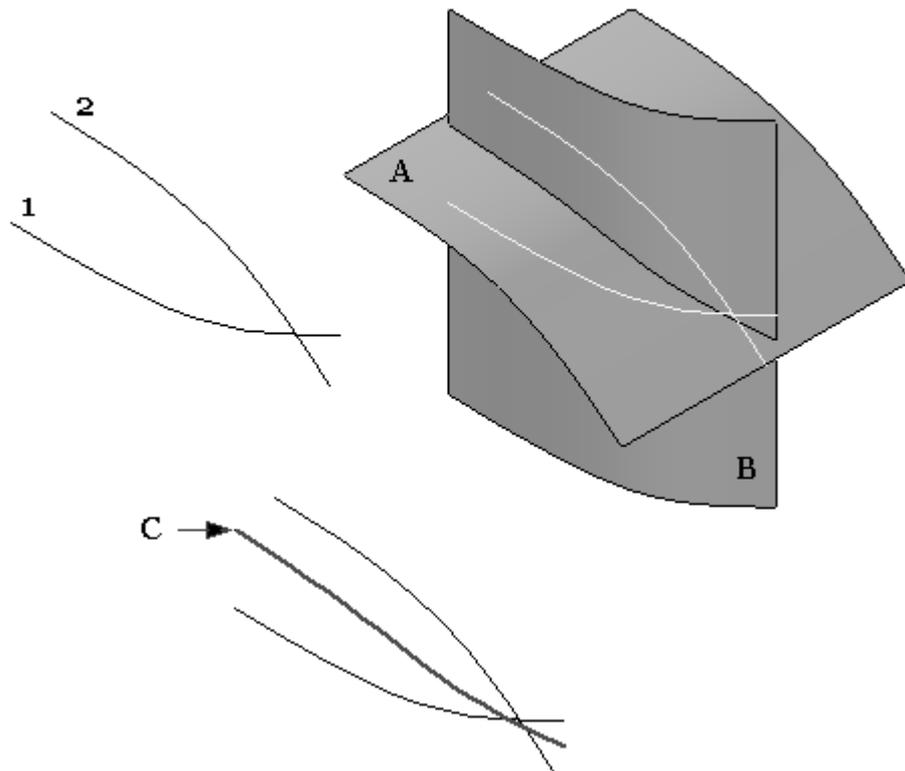
The intersection curve (A) can be used for a surface trimming operation or an edge to be used to create a new construction surface.



## Cross Curve



The Cross Curve command is similar to the intersection curve command. The Cross Curve command does not need existing surfaces to create an intersection 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/analytics.



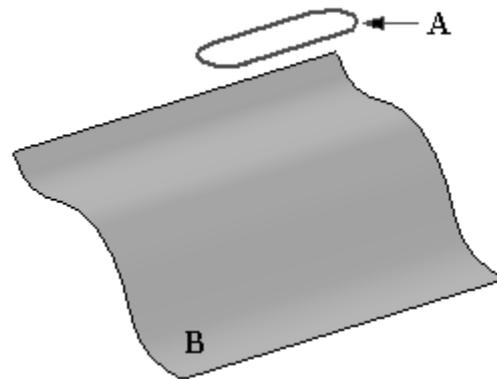
(1) and (2) are the input curves. (A) and (B) are the theoretical extruded surfaces. (C) is the resultant cross curve.

## Project Curve



The Project Curve command projects a set of curves (2-D or 3-D) 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.

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

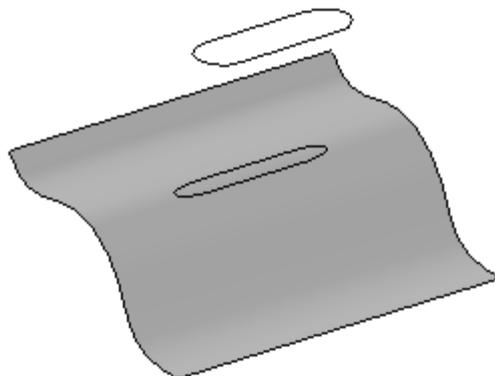
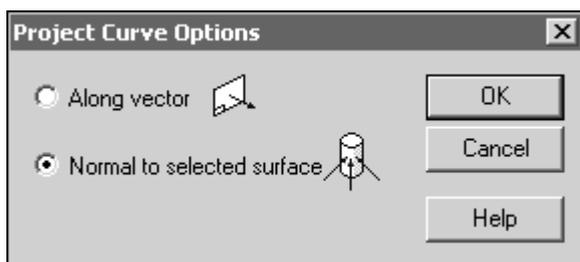
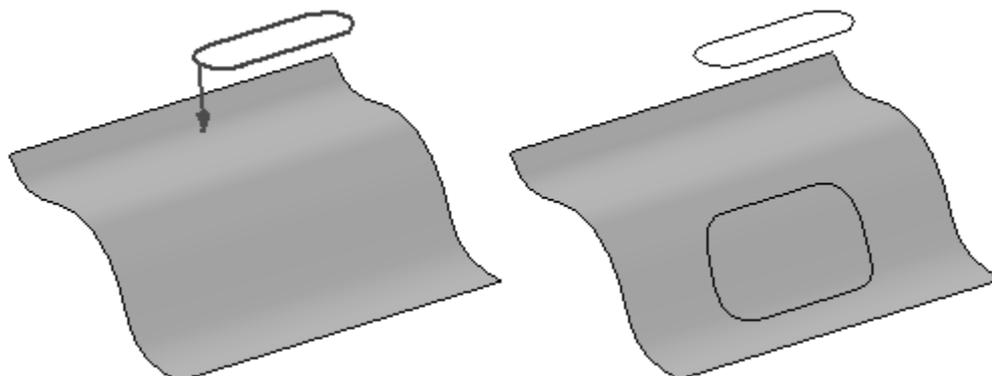
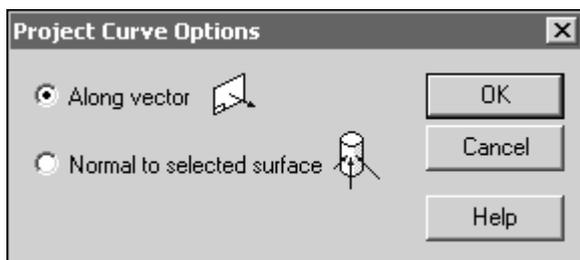


- (A) Projection Curve
- (B) Surface to be projected on to

The following illustrations show the results of options: “Along vector” and “Normal to selected surface”.



Project Curve Option button



## Derived Curve

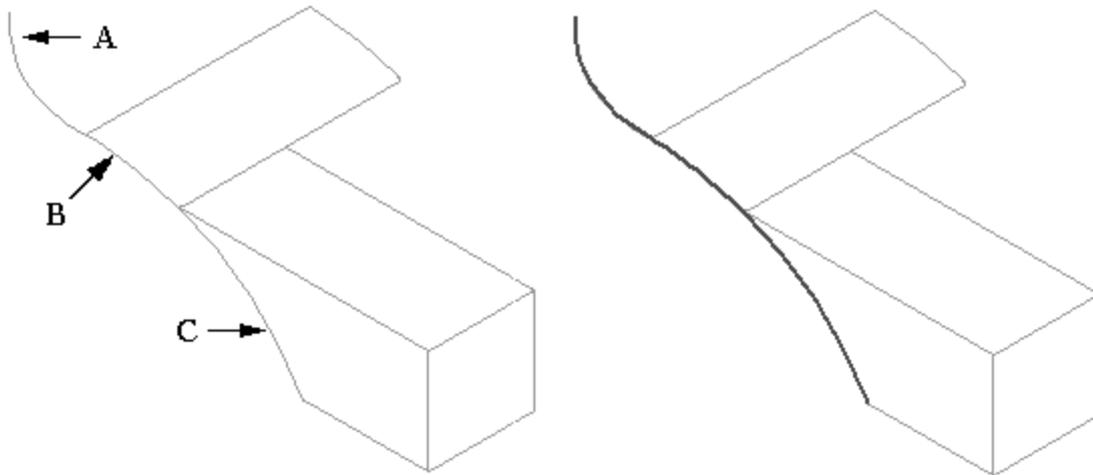


The Derived Curve command constructs a new curve that is derived from one or more input curves or edges. If all the input curves or edges are connected at their endpoints, you can specify that the derived curve be constructed as a single spline curve. If the input curves are connected, but not tangent,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



A derived curve is associative to the edge/curve it was derived from. Changes to the parent curve/edge also changes the derived curve.

You can construct a single derived curve from multiple bodies. In the example shown below, you can construct a derived curve from a sketch (A), edges on a construction surface (B), and edges on a solid (C).



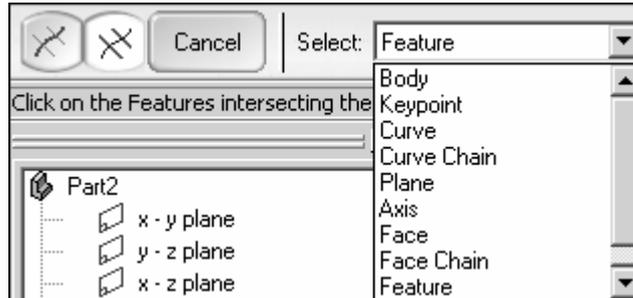
If multiple edges or curves are needed to define a single derived curve, select each curve/edge and then on the ribbon bar, click the Accept button. To create a derived curve from a single edge/curve, select the curve/edge and then click the Accept button.



## Split Curve



The Split Curve command is a tool provided to split a construction curve. Only construction curves can be split (derived curves, intersection curves and projected curves). You can select keypoints, curves, reference planes, solid faces or surfaces as the intersecting features that 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.

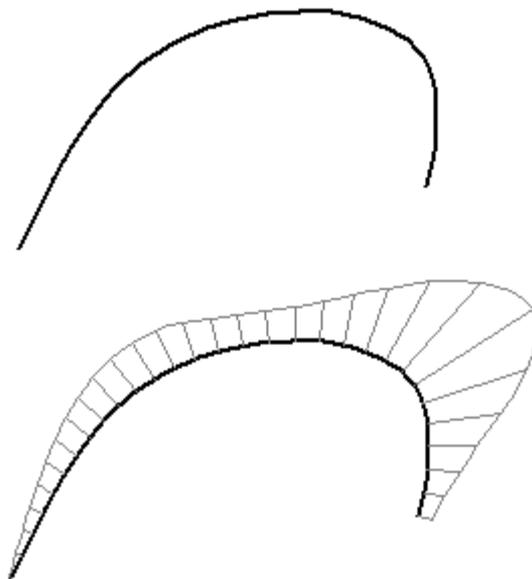
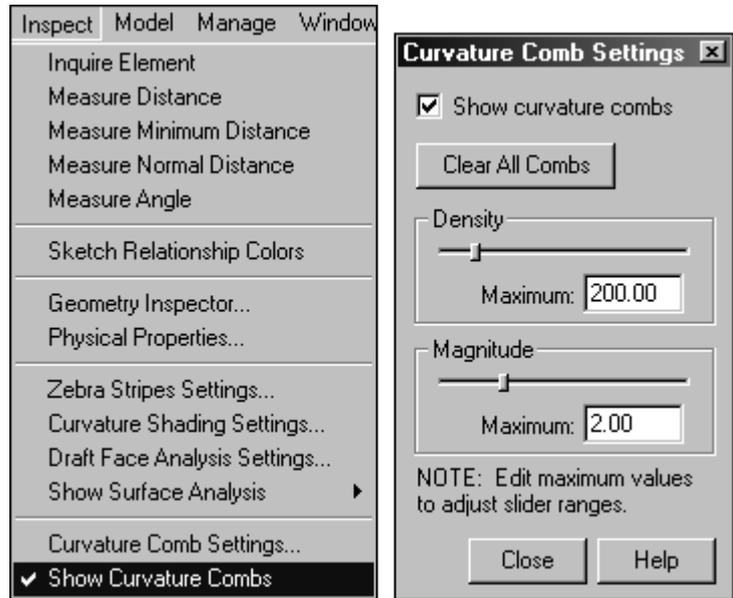
You cannot use the Split Curve command to split an edge on a model. You can use the Derived Curve command to create an associative copy of an edge on the model, and then use the Split Curve command to split the derived curve.

## Curvature Comb

Curvature Comb enables the display of normal vectors on a curve or profile to help illustrate the radius of curvature. The length of the normal vector is a function of the radius of curvature. You can control the number of vectors (density) and the magnitude of the vectors in the Curve Comb settings form.



Curvature comb display is not available for construction curves.



## **Review**

- What is an edit point on a curve?
- What is a control point on a curve?
- What is a control polygon?
- Explain the differences in Shape Edit, Local Edit and Rigid. Define the degree of a curve.
- How do you change the degree of a curve?
- Explain what a BlueDot is and what impact it has on curves.
- Explain the difference in indirected and directed curves.
- How do you convert analytics to bsplines?
- How do you get tangency control on a keypoint curve?
- What is the Curvature Comb used for?



## Activity 1 – Drawing and Editing a Curve

### Overview

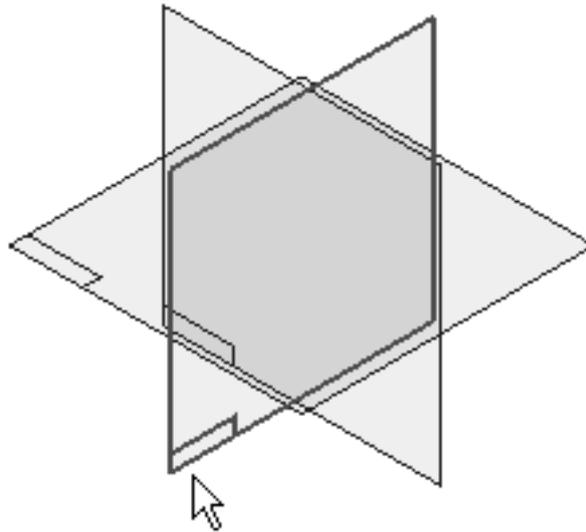
In this activity, you will learn to use the curve creation tools. Curves are the backbone for creating and controlling surface shape.

### Objectives

After completing this activity you will be able to:

- Create curves
- Edit curves
- Analyze curves

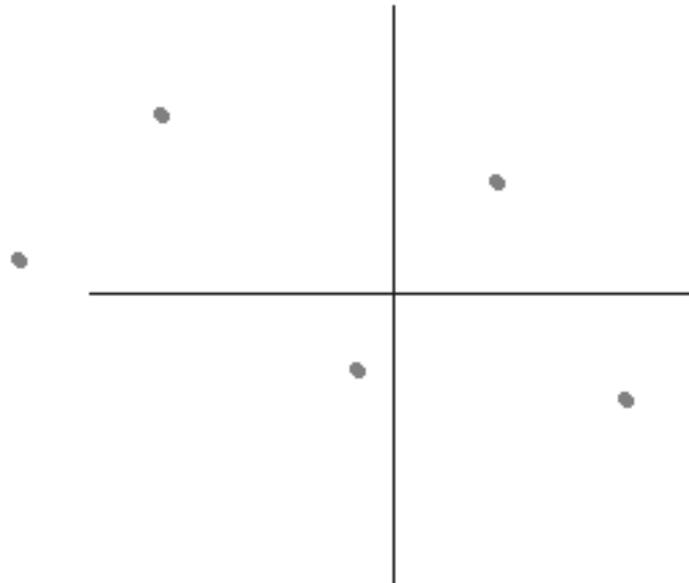
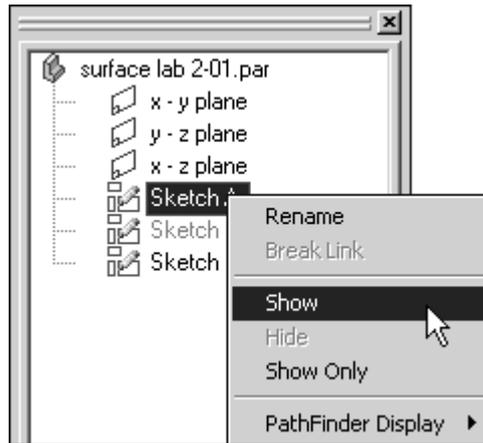
1. Open SURFACE LAB 2-01.PAR in the C:/Surfacing/Parts folder.
2. You will begin the activity by drawing a curve with edit points in space. On the Features toolbar, click the Sketch command. Select the plane shown below.



3. On the Features toolbar, click the Select tool.



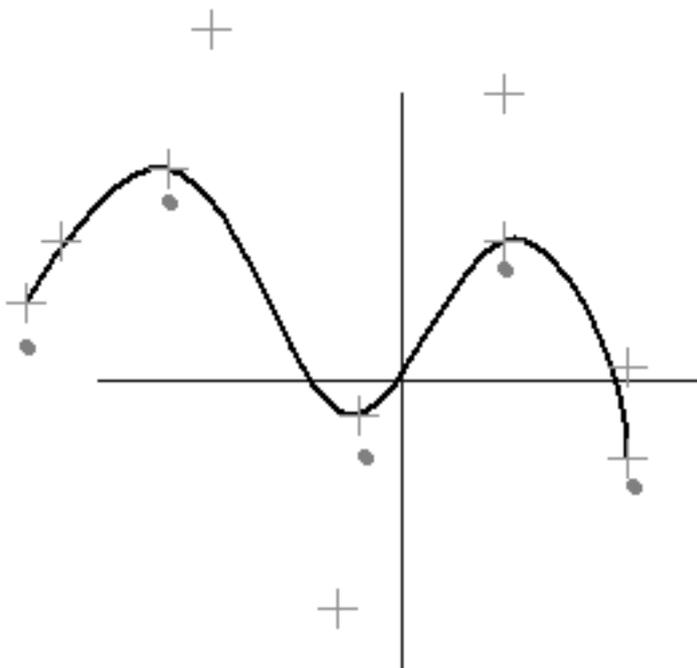
4. In EdgeBar, right-click Sketch A and click Show. You will use the sketch elements in Sketch A as a guide to where to place the edit points.



5. On the Draw toolbar, click the Curve command.



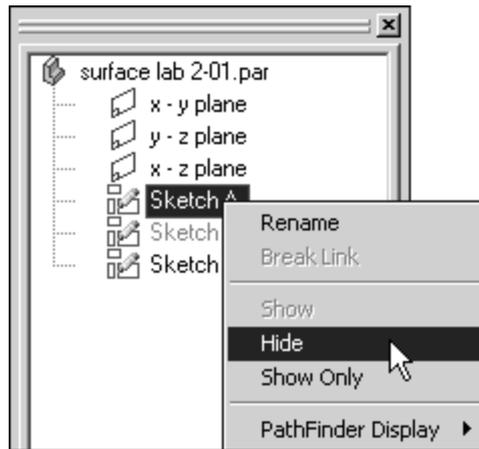
6. Click just above each of the points from left to right as shown. After clicking above the last point, right-click to create the curve.



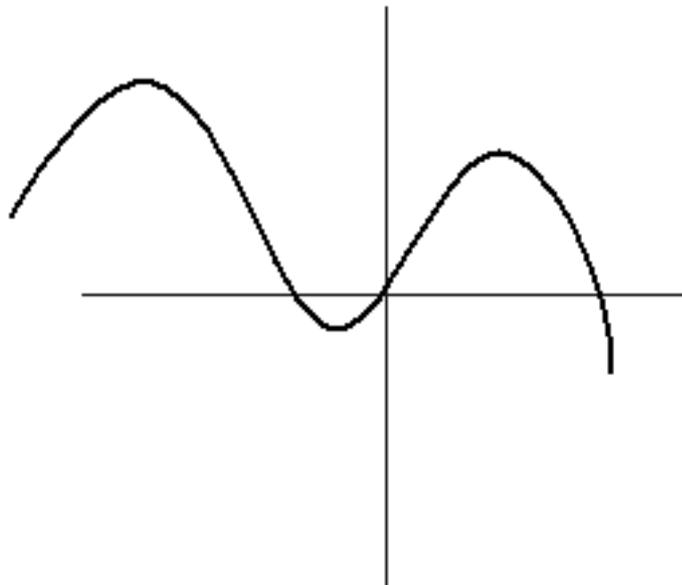
7. On the Draw toolbar, click the Select tool.



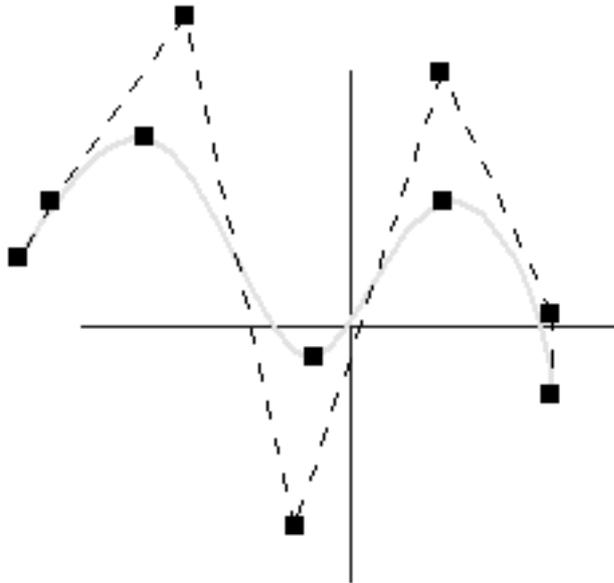
8. In EdgeBar, right-click Sketch A and select Hide.



The image below shows the curve you just created.



9. Select the curve and notice the display of the edit points and control polygon.

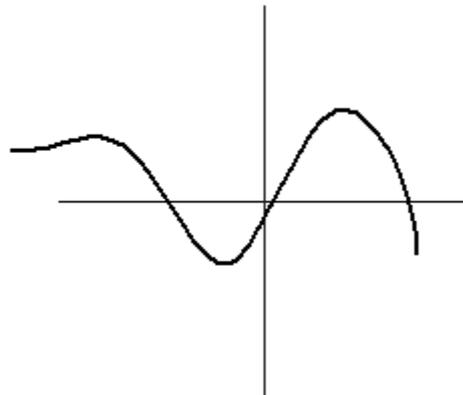
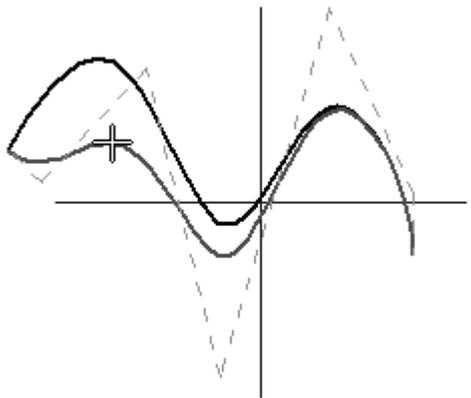
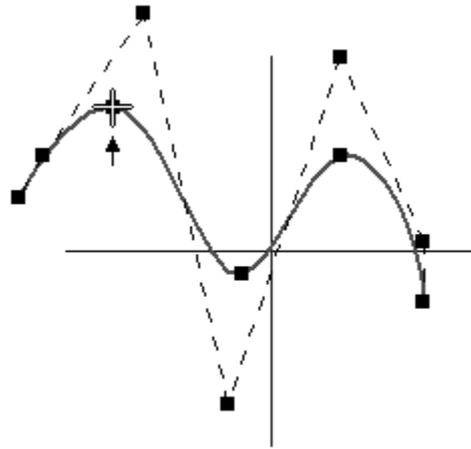


10. At this point you can edit the curve shape by dragging edit points or control points. Notice the Edit Curve ribbon bar displays when you select the curve. The Local Edit option is on.



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



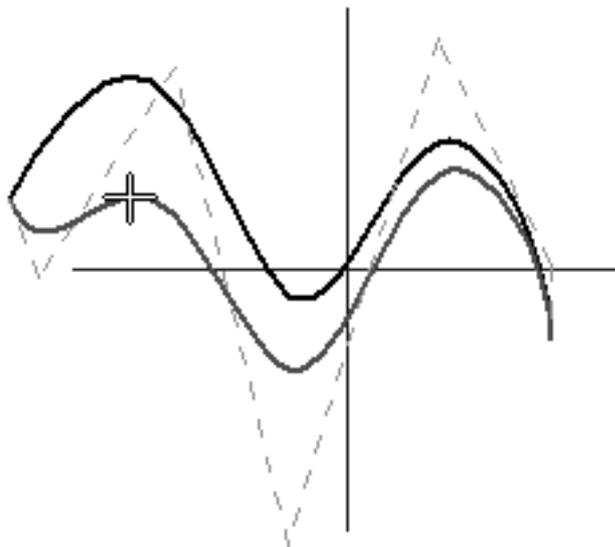
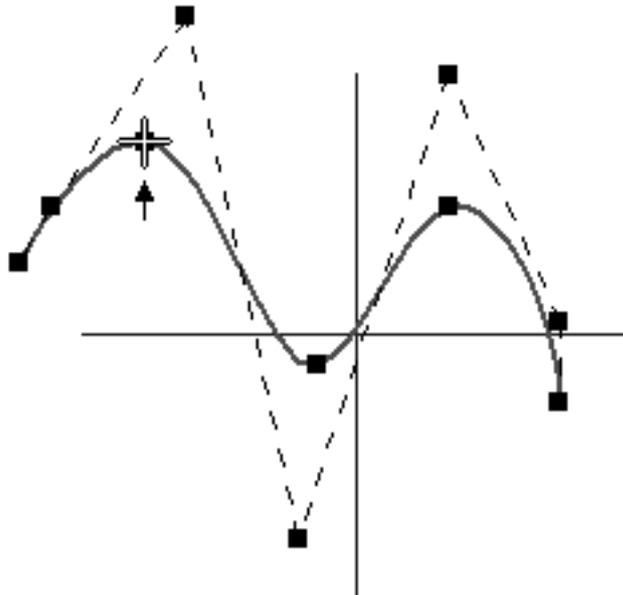
- After you edit the curve, on the Main toolbar, click the Undo command. This returns the curve its original shape.



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



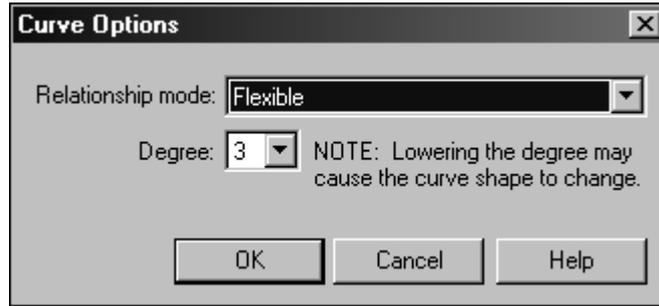
- With Shape Edit option selected, drag the edit point shown and observe how the curve shape changes.



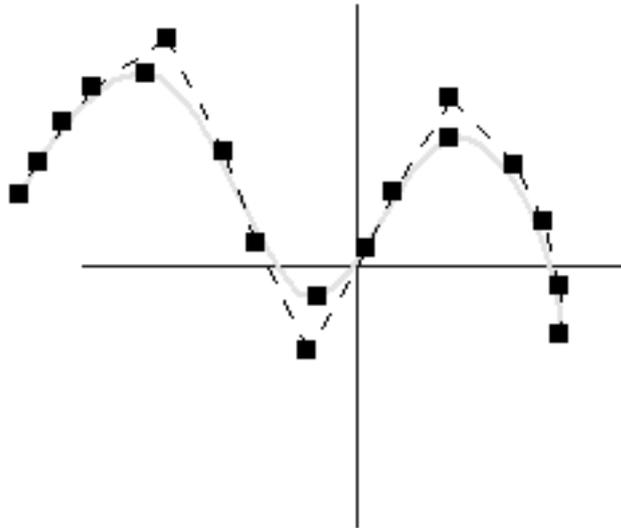
- After you edit the curve, on the Main toolbar, click the Undo command. This returns the curve its original shape.



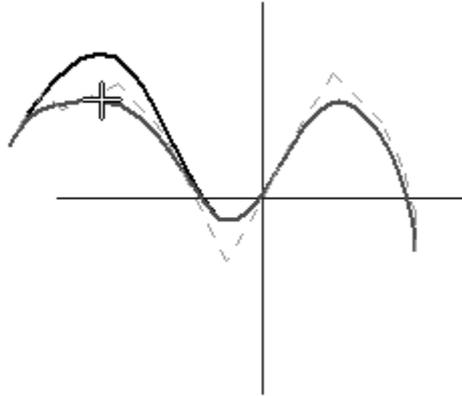
11. To add more control to the curve, you can increase the degree of the curve. The curve was created with a default degree of 3.
  - Select the curve. On the Edit Curve ribbon bar, click the Curve Options button.



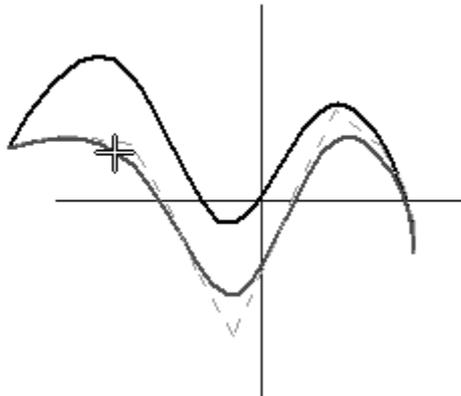
- Change the degree from 3 to 5 and click OK. Observe the change to the control polygon.



- Edit the curve again with both Local 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

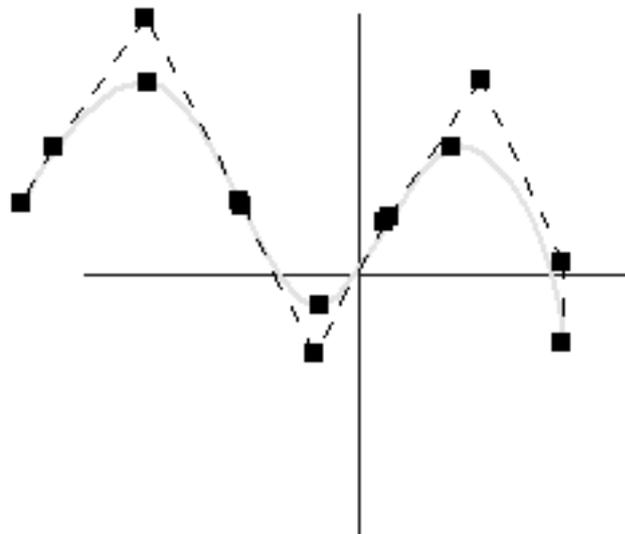
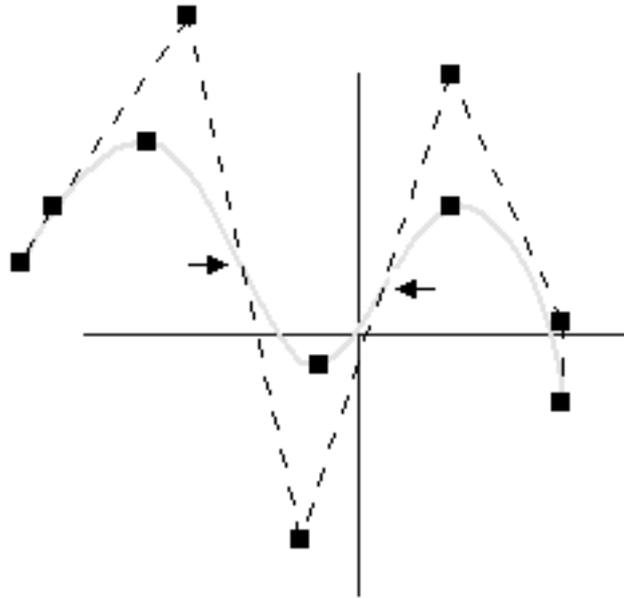
12. You can also add more control to the curve by inserting additional edit points.
  - Select the curve and on the Curve Options, set the degree back to 3 and click OK.
  - On the Edit Curve ribbon bar, click the Add/Remove Points button.



- Add additional edit points to the two locations shown below.



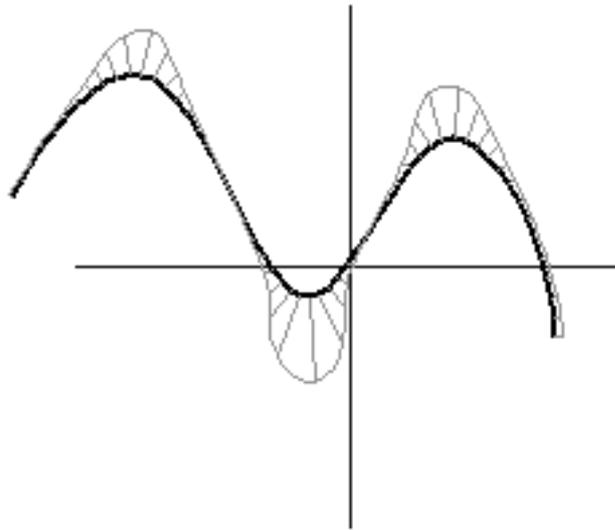
You can only insert one edit point with the Add/Remove Points command. Reselect the command to place another point. However, you can also hold down the Alt key on the keyboard and click on the curve to place as many points you need.



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

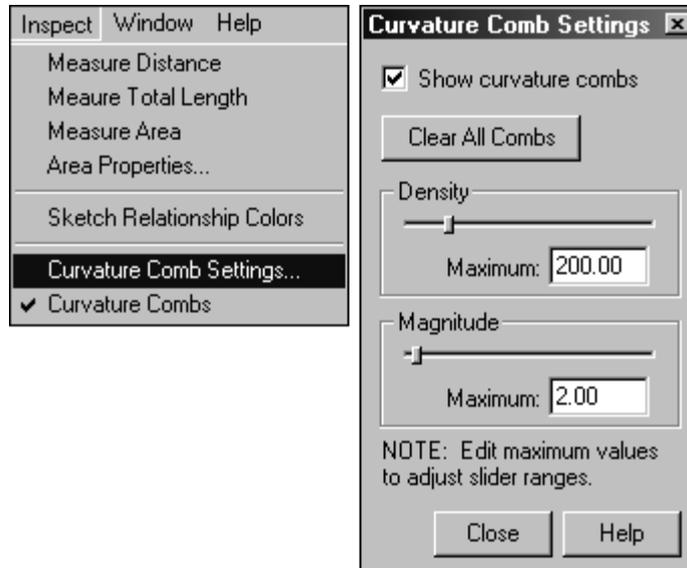
13. You will now use a curve inspection tool called the Curvature Comb.

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



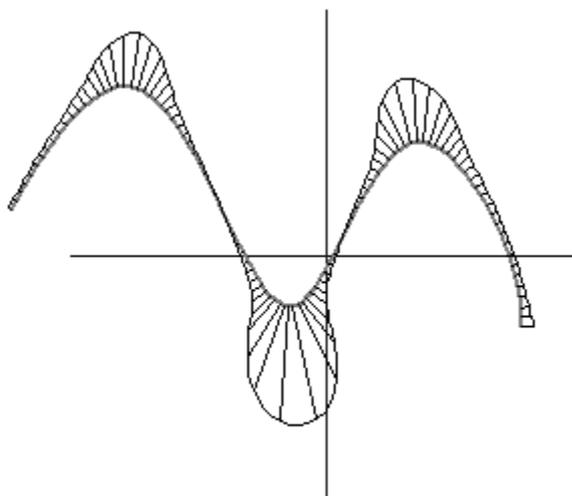
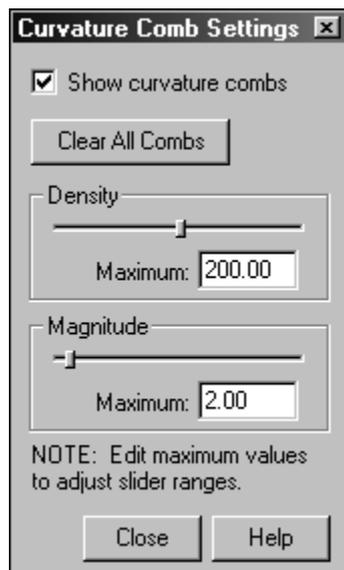
You can adjust the curvature comb display with the Curvature Comb Settings dialog box.

- On the Main menu, click Inspect>Curvature Comb Settings.



Density controls the number of normal vectors. Magnitude controls the length of the vectors.

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



- On the Curvature Comb Settings dialog box, uncheck the “Show curvature combs” box and click Close.

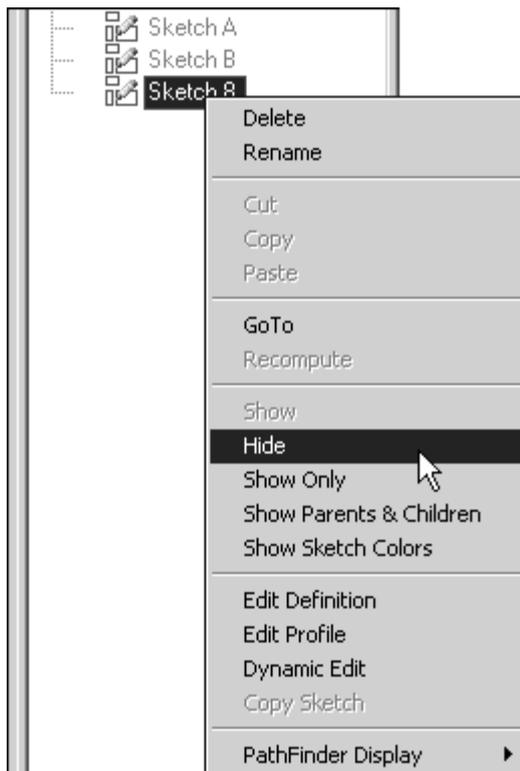
14. Click Finish to complete the sketch.



15. Click Finish again to exit the sketch mode.

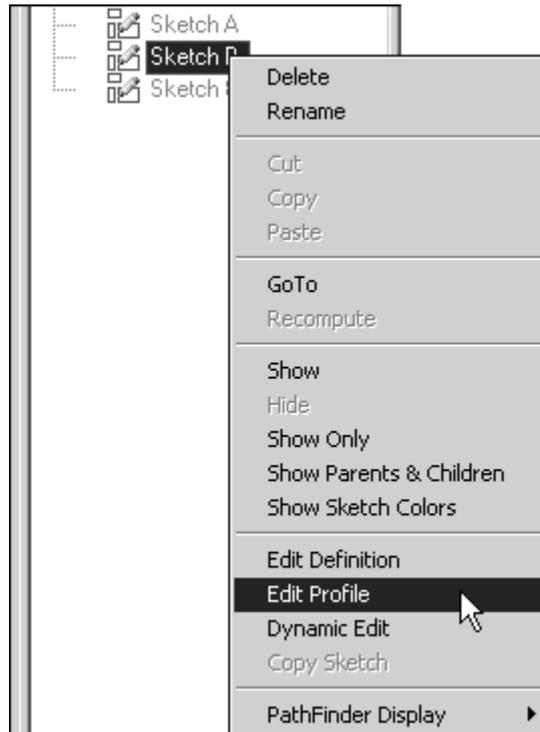


16. In EdgeBar, right-click the sketch you just completed and select Hide.



17. You will now draw a curve with the edit points connected to elements. You will edit dimensions to observe how the curve shape is controlled by the sketch elements.

- Right-click in EdgeBar on Sketch B and click Edit Profile.



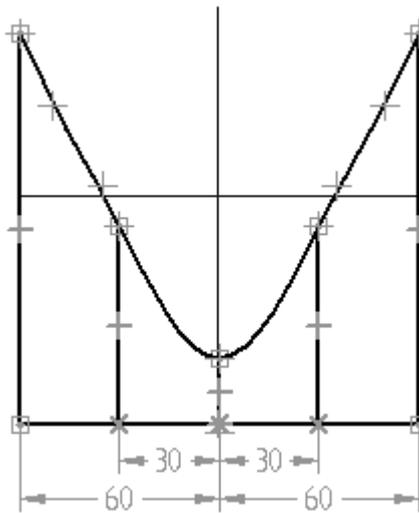
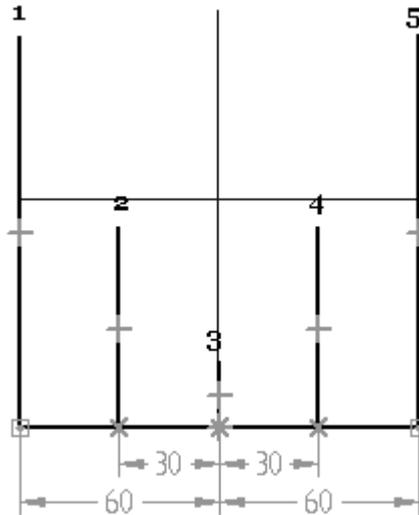
- On the Features toolbar, click the Curve command.



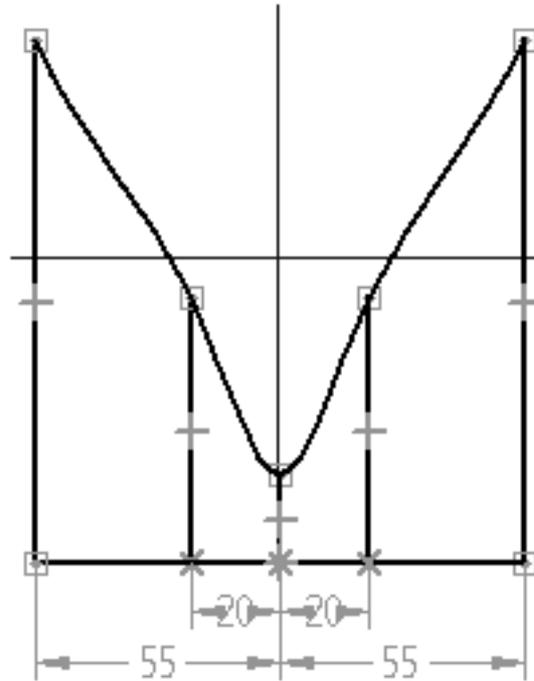
- Draw a curve with edit points at the endpoints of the lines (1-5) shown below. Make sure you get the endpoint connect symbol before clicking. After placing the last edit point, right-click to complete the curve.



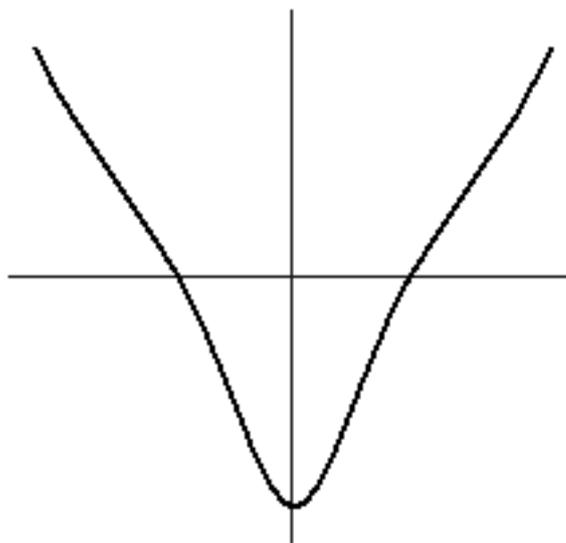
Endpoint Connect Symbol



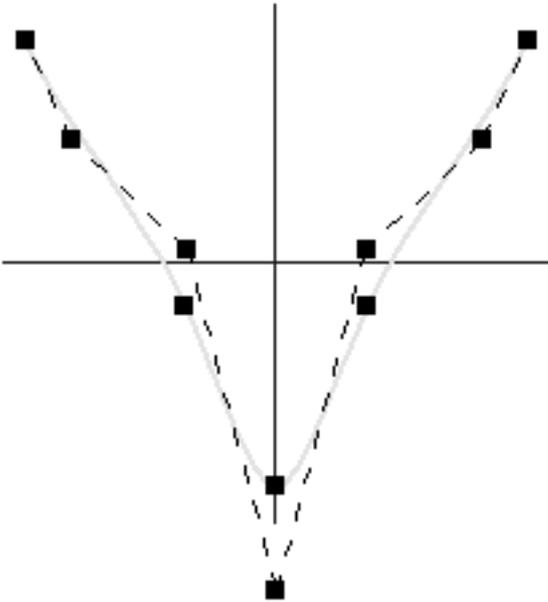
- Edit the dimensions as shown to observe how the curve is constrained to the dimensioned elements.



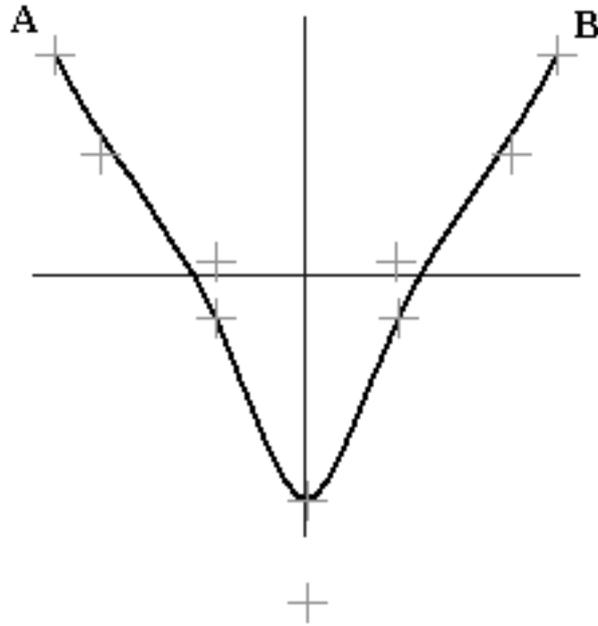
18. You can also add constraints to a curve using the edit points and control vertices. You will now use the last curve created to apply constraints to.
- Delete all elements in the sketch except for the curve.



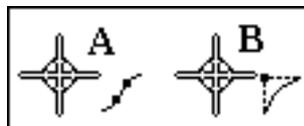
- Select the curve.



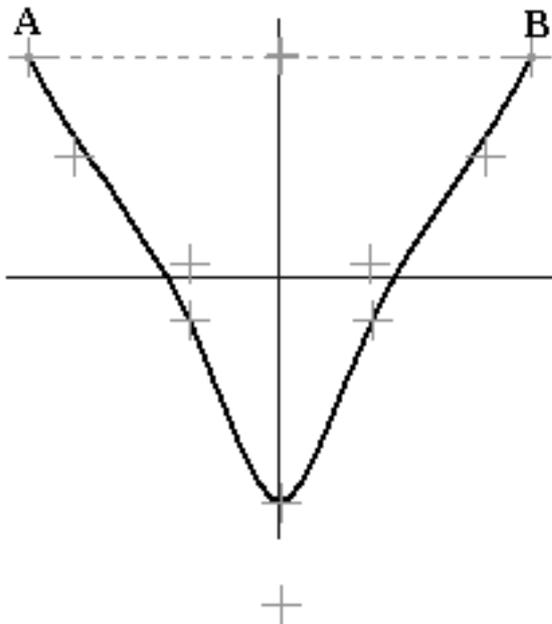
- You will now add constraints to the curve. The first constraint will be a horizontal relationship between edit points A and B. On the Drawing toolbar, click the Horizontal/Vertical command.



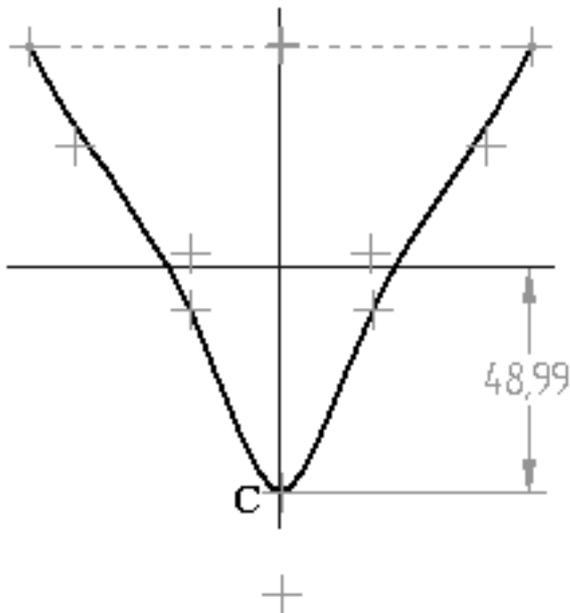
Notice in the image above that the edit points and control vertices display as crosses. If you pause the cursor over a cross, you will see the following denoting if it is an edit point (A) or control vertex (B).



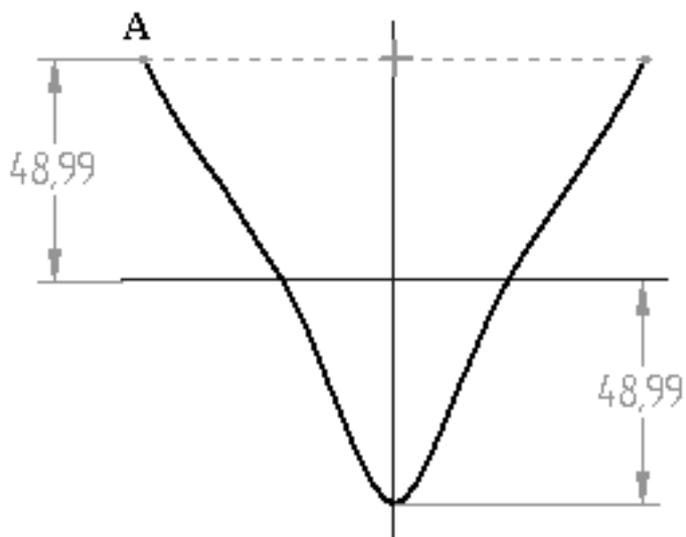
- Click point A and then click point B. Points A and B will always remain aligned horizontally.



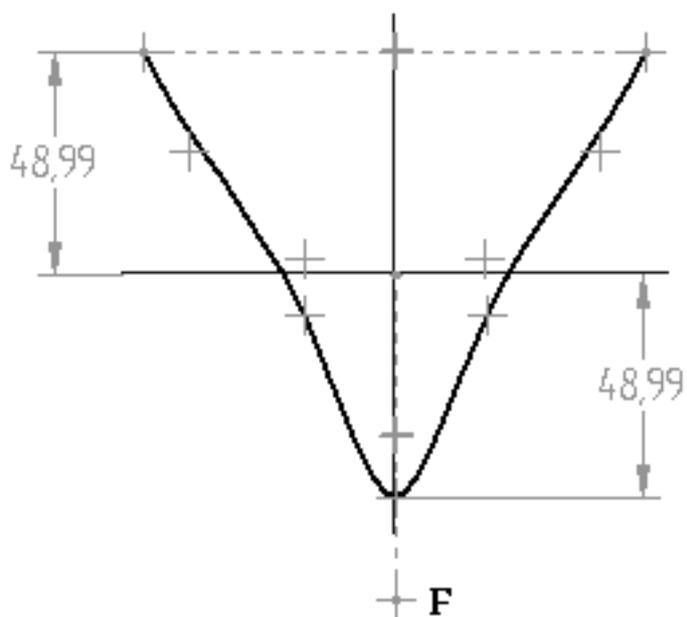
- Place a dimension as shown between the horizontal reference plane and edit point C.



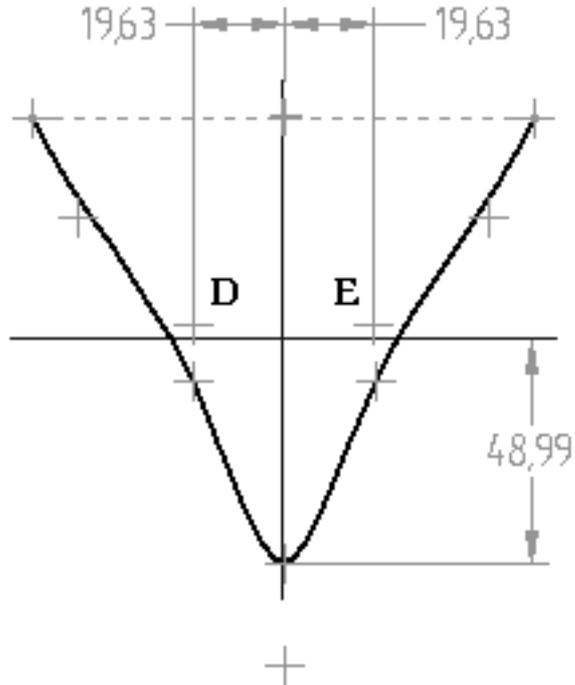
- Place a dimension as shown between the horizontal reference plane and edit point A.



- Add a vertical relationship between control vertex F and the center of the reference planes.

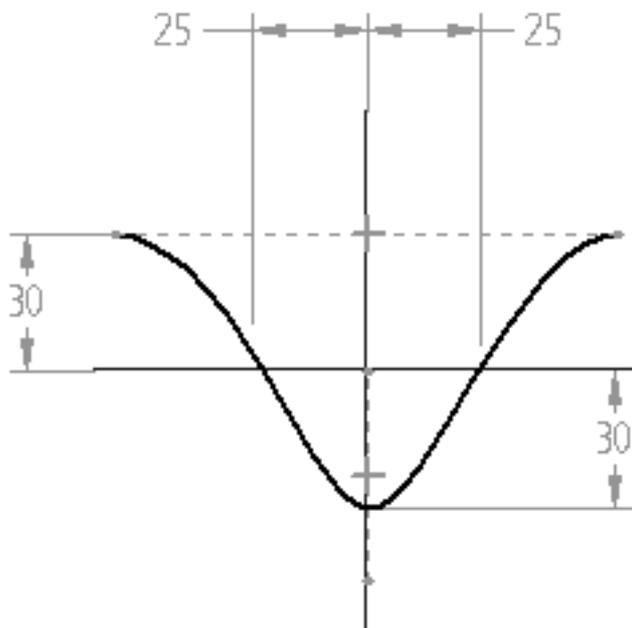


- Apply a final constraint to control vertices. Place two dimensions as shown between the vertical reference plane and control vertices D and E.

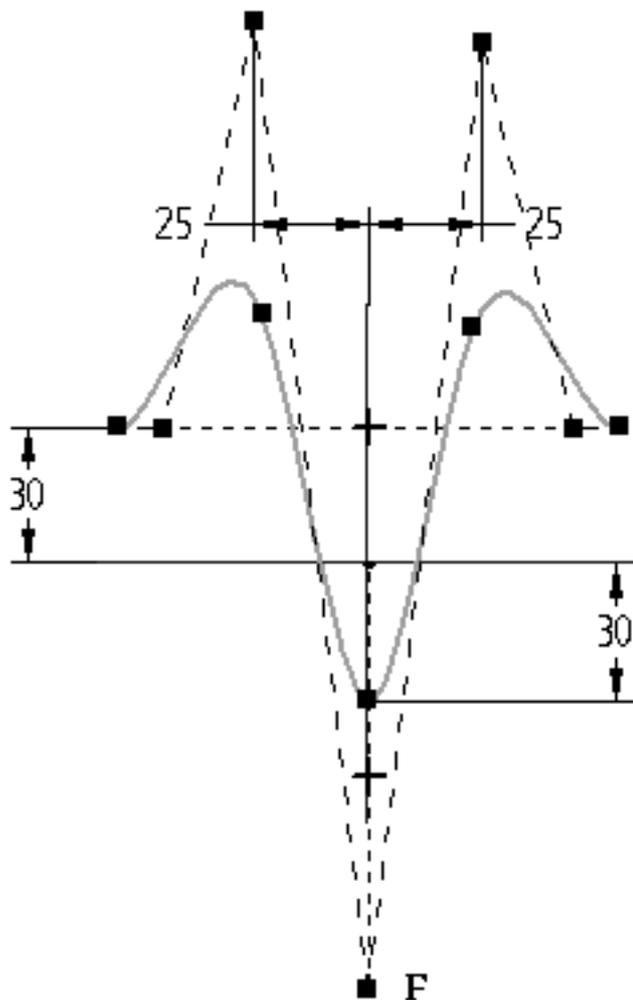


More constraints are needed to make the curve symmetric about the vertical reference plane. But for this activity you will stop adding constraints at this point.

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



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



19. The activity is complete. Exit and save the file.



## Activity 2 – Creating and Editing BlueDots

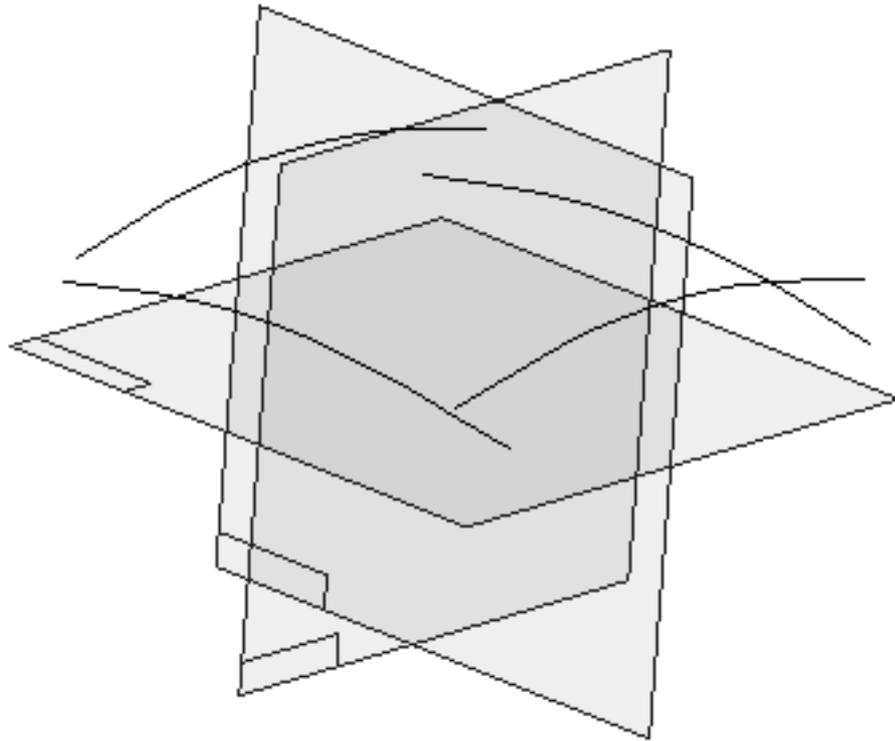
### Overview

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

### Objectives

After completing this activity you will be able to:

- Create BlueDots
  - Edit BlueDots
1. Open SURFACE LAB 2-02.PAR in the C:/Surfacing/Parts folder. The part file contains four curve sketches. You will connect these curves using the BlueDot command.



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 module.



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

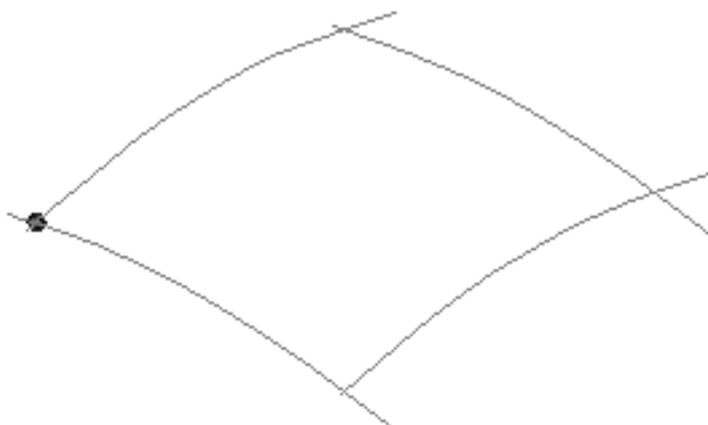
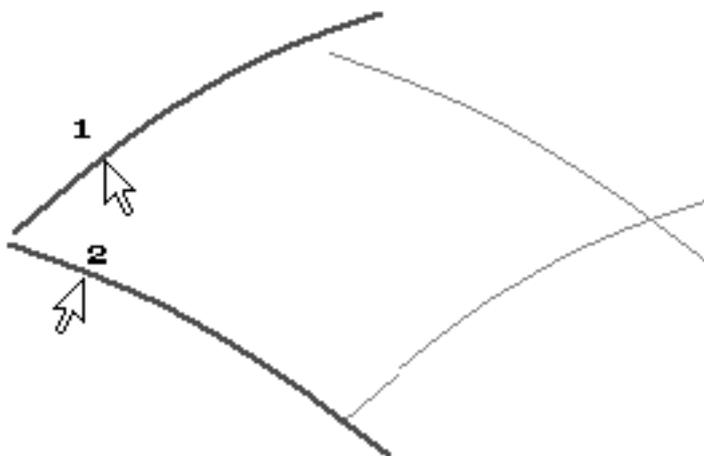


There are several curve selection locations. See the topic on BlueDot Creation in the theory section of this module.

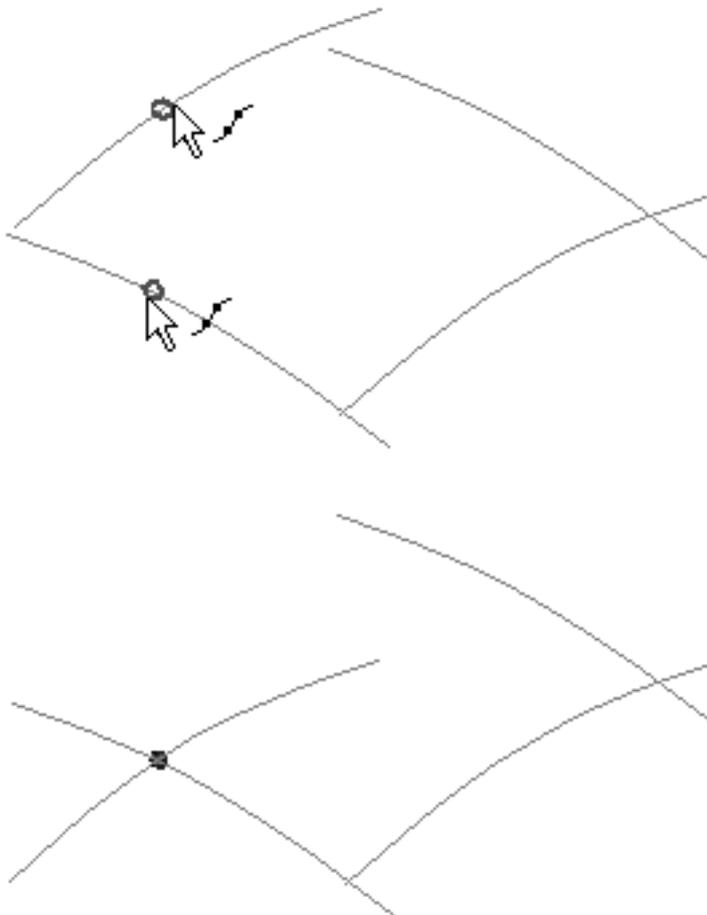
2. On the Surfacing toolbar, click the BlueDot command.



3. In the next few steps, you will experiment with connecting two curves using different selection locations. Remember to undo after each connection to return the curves to their original location.
  - Select curve 1 at the location shown and then select curve 2 at the location shown. Notice the resulting connection and then click Undo.



- Click the BlueDot command again. Select curve 1 at the location shown and then select curve 2 at the location shown. You are selecting edit points on the curves. Notice the resulting connection and then click Undo.

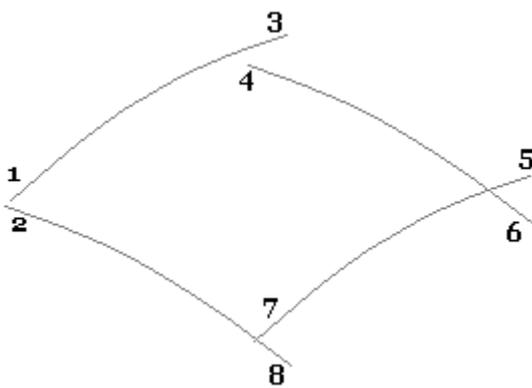
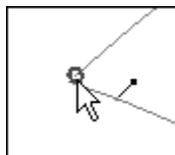


- Now that you have experimented with different curve selection possibilities, you will connect the four curves at the end points. Click the BlueDot

command again. Connect the curves in the sequence shown below (1-2, 3-4, 5-6 and 7-8).

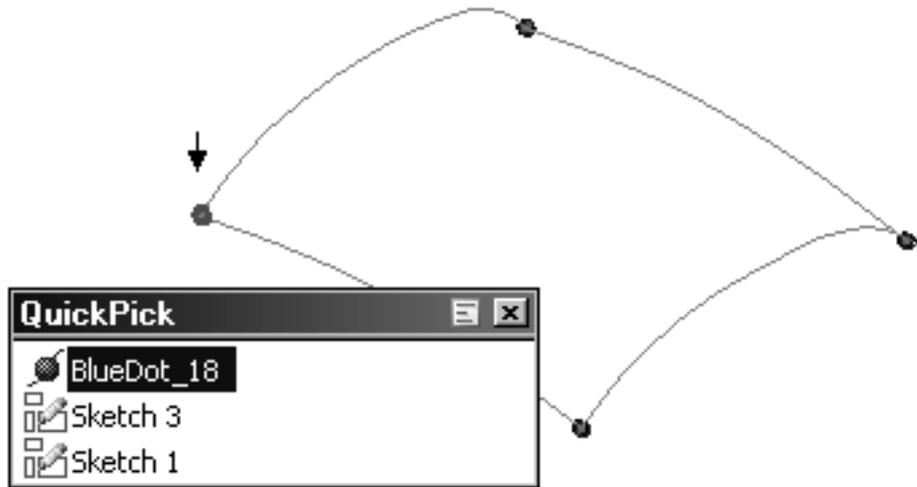


Make sure the endpoint connect symbol displays before you click.



4. The four curves are now BlueDot connected. You will now edit a BlueDot to observe how the curves behave.

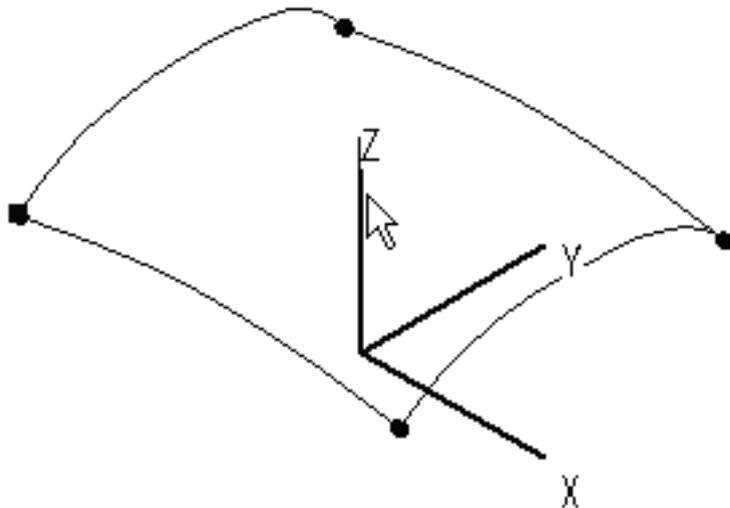
- Click the Select tool and then select the BlueDot shown.



- On the Select ribbon bar, click Dynamic Edit.



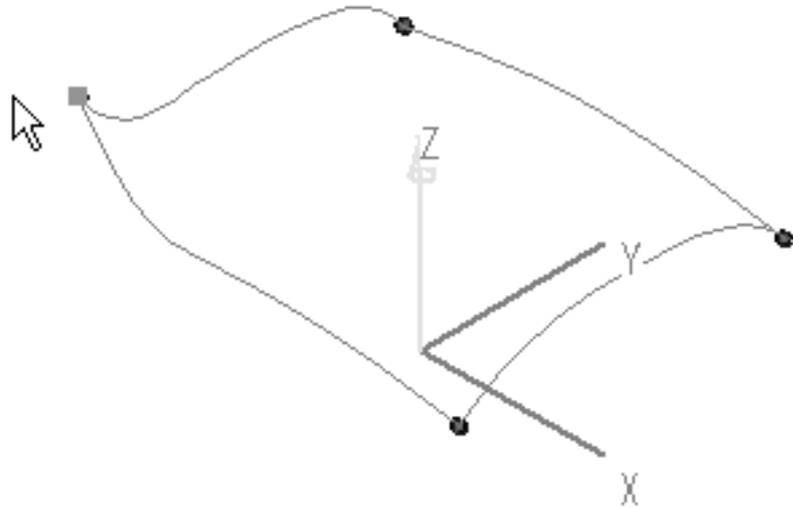
- Click the Z-direction axis on the 3-D triad as shown. This locks the BlueDot edit in the Z-direction.



Notice on the Edit BlueDot ribbon bar that the focus is locked to the Z value. This value is the actual Z-coordinate value.



- You can now edit the BlueDot by dragging it on the screen or by keying in a new Z-coordinate value. Drag the BlueDot a small distance as shown and observe the connected curves' behavior.

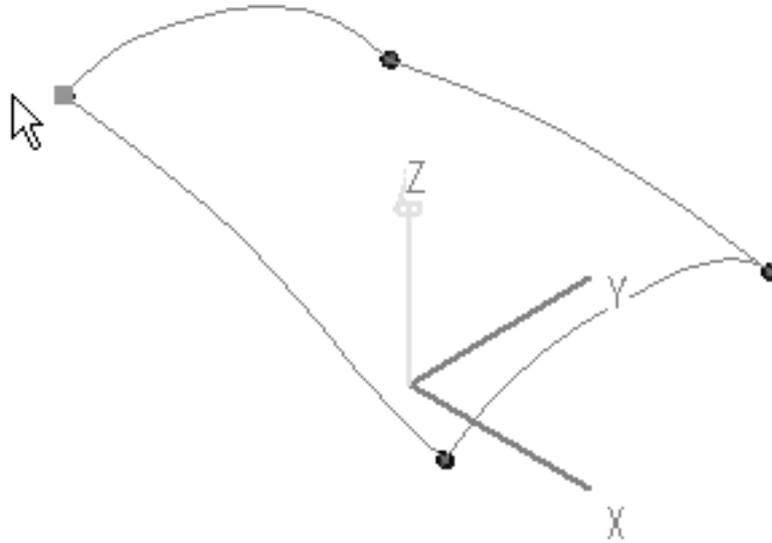


The BlueDot edit you just made had both curves connected to the BlueDot set to Local edit.



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

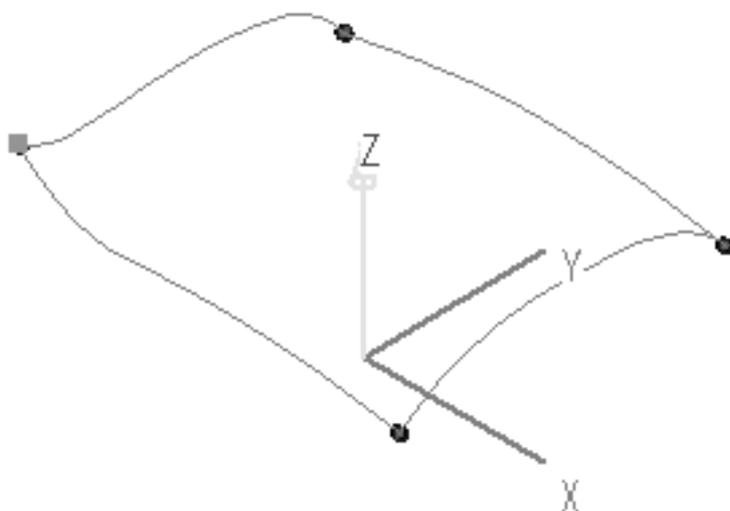
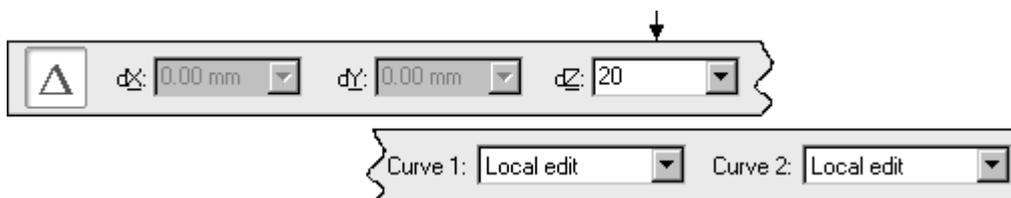
- On the BlueDot Edit ribbon bar, set both curves to Shape edit and then repeat the previous step. Notice the different result.



- Click Undo to return the BlueDot to its original location.
- You can also 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 ribbon bar.



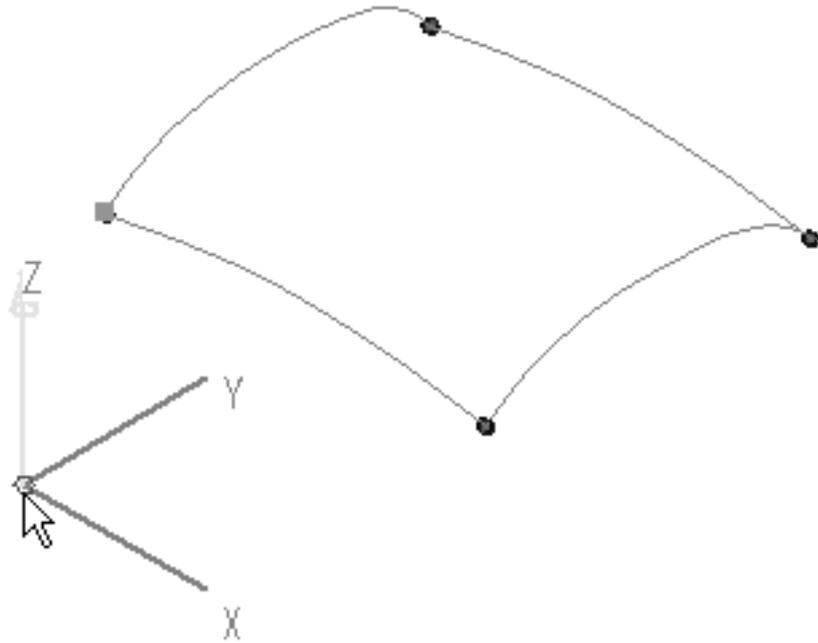
- Notice that the ribbon bar changes to dX:, dY: and dZ:. Enter 20 in the dZ: field and press Enter on the keyboard.



If you press Enter again, a delta value of 20 will be applied again.

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

5. You can move the 3-D triad if it gets in the way. Click the 3-D triad as shown and drag to a new location.



6. The activity is complete.



## Activity 3 – Creating Keypoint Curves

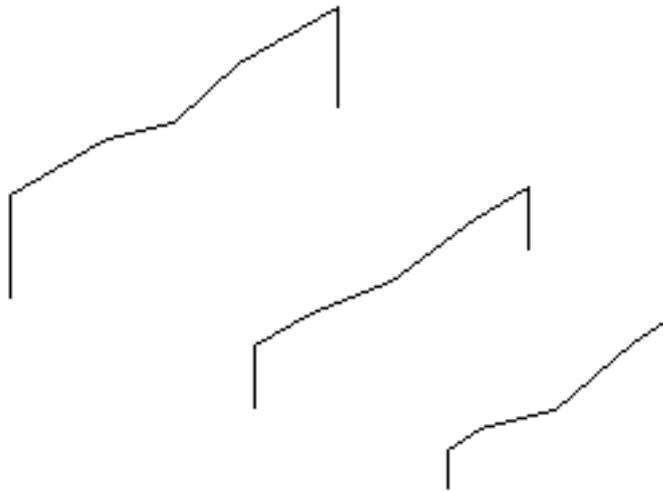
### Overview

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

### Objectives

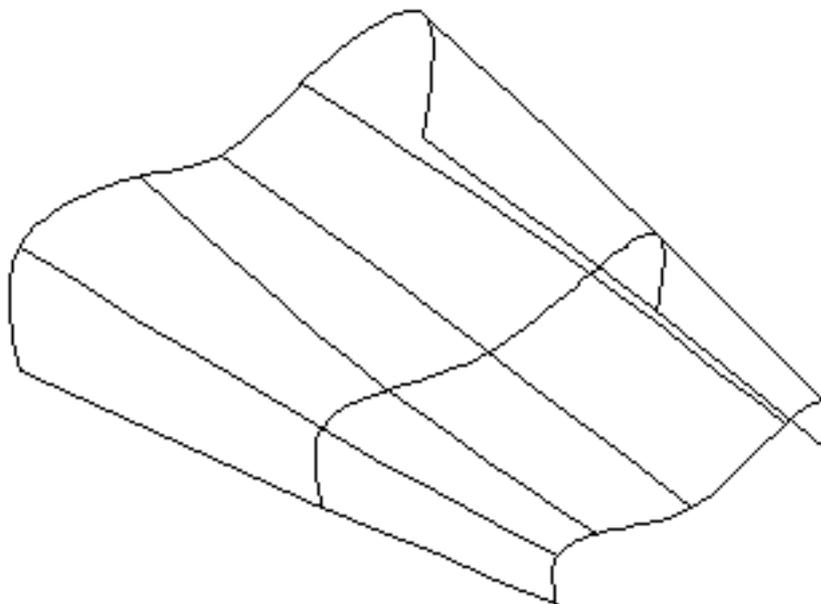
After completing this activity you will be able to:

- Create a keypoint curve
  - Modify tangency vectors
1. Open SURFACE LAB 2-03.PAR in the C:/Surfacing/Parts folder.



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

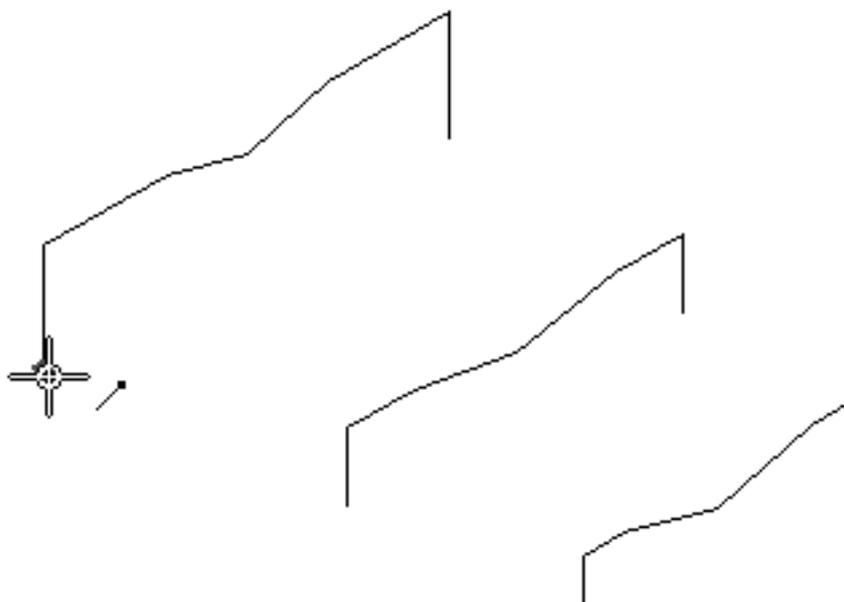
2. Create a keypoint curve on each sketch and then create keypoint curves connecting each sketch. The final result is shown below.



- You will now create the first keypoint curve using geometry from Sketch A. On the Surfacing toolbar, click the Keypoint Curve command.

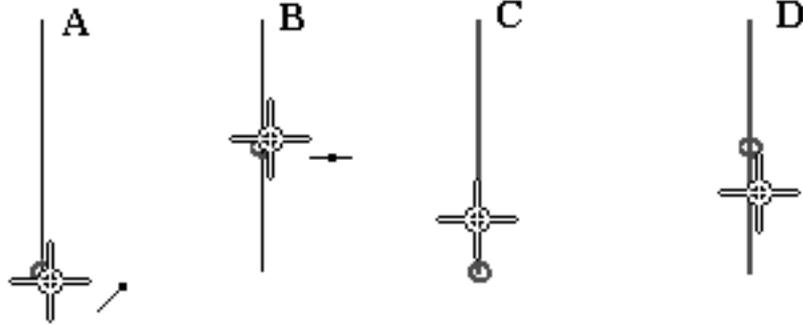


- Click the endpoint shown. Make sure the endpoint connect symbol displays.





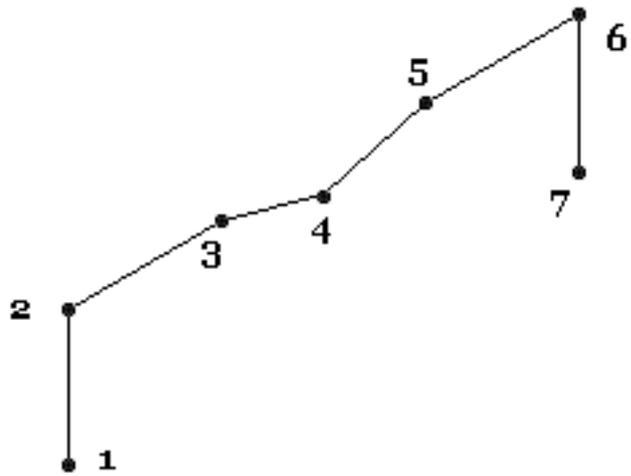
There are other keypoint select locations possible on a line. You can select the endpoints (A), midpoint (B), line and endpoint (C) or line and midpoint (D). If you select a line and endpoint or line and midpoint, the curve will be tangent to the line at that point. You will have the opportunity to modify the tangent vector. For this lab activity, only select endpoints.



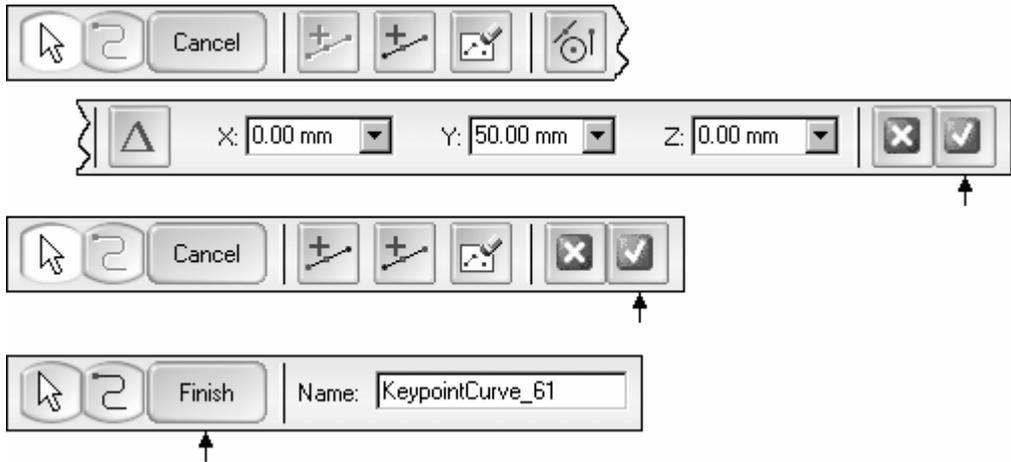
To make it easier to only select endpoints, on the ribbon bar click the Keypoints button. Select the endpoint option as shown.



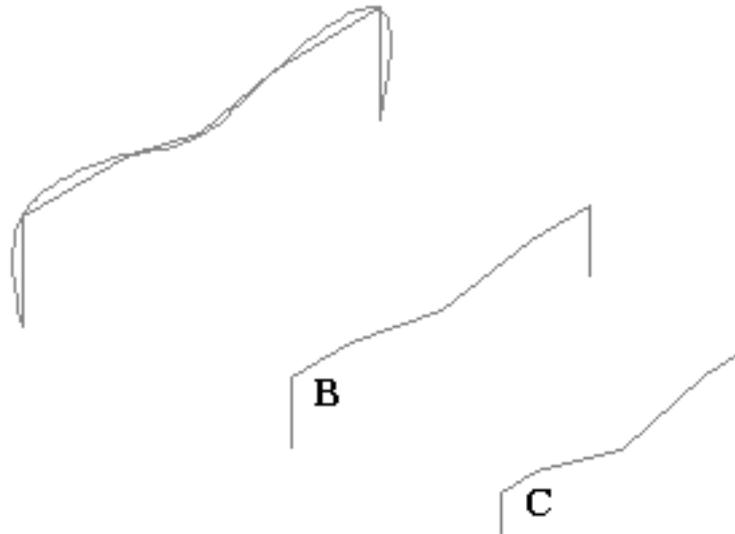
- Click the remaining endpoints in the following order.



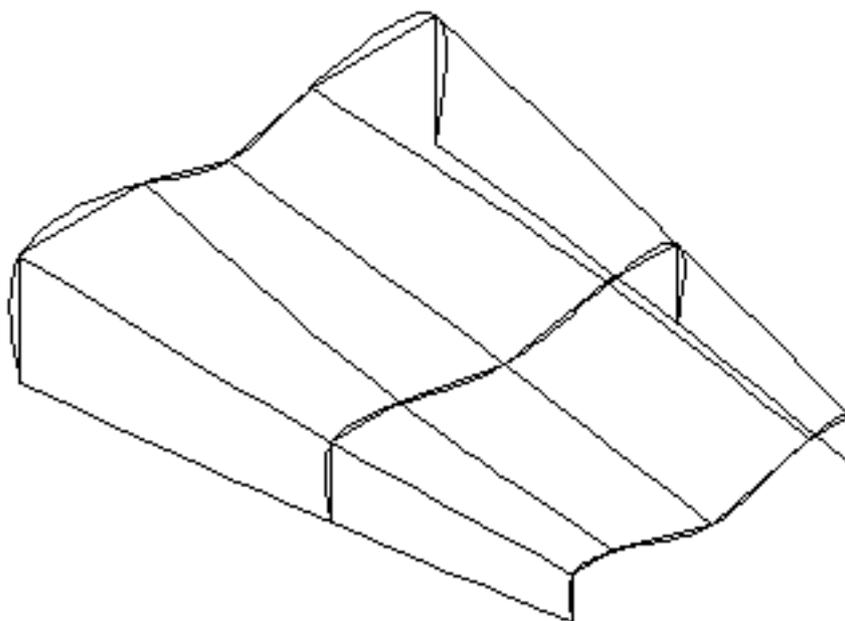
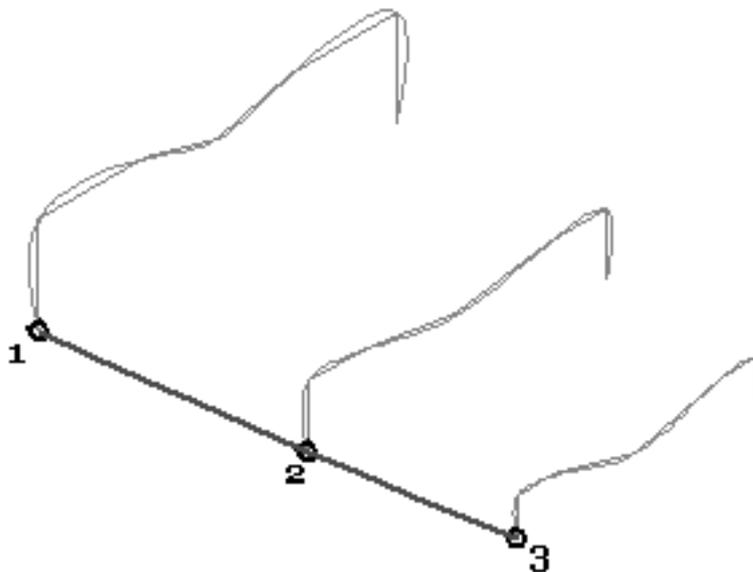
- After you click the last endpoint, on the Keypoint Curve ribbon bar, click the Accept button. Click the Accept button again and then click Finish.



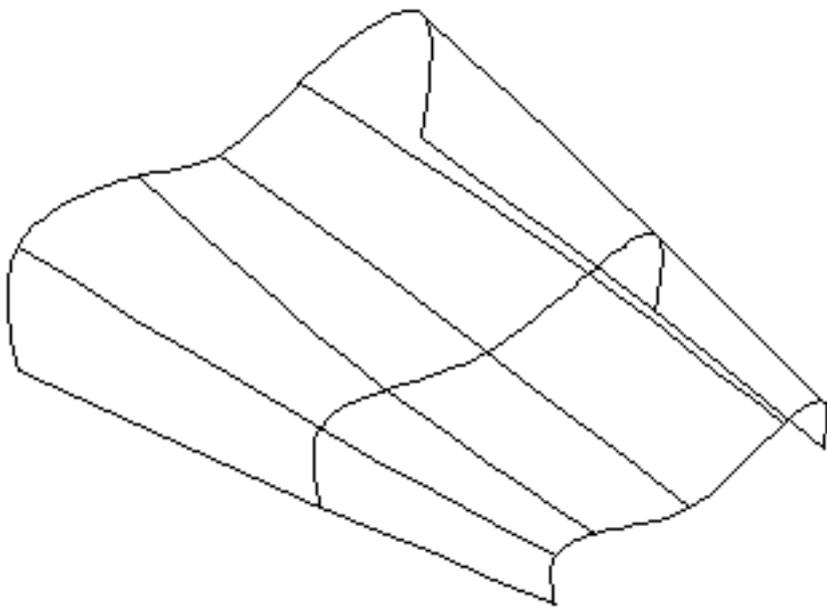
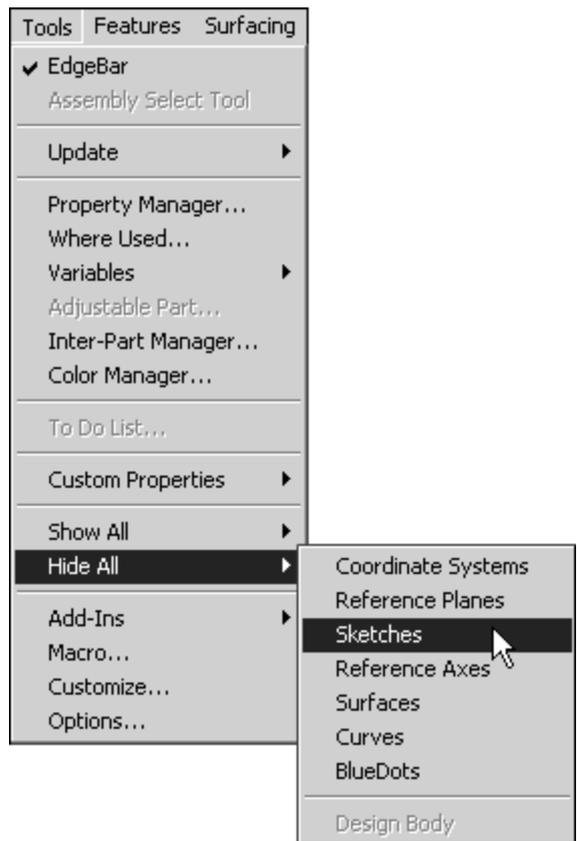
- Repeat the previous step to create keypoint curves using Sketches B and C.



- You will now create keypoint curves between the sketches. You will create seven keypoint curves. The first curve is shown below and also the completed curves.



- On the Main menu, click Tools>Hide All>Sketches.





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 will change with the edit made to the sketch.

3. You will now connect the keypoint curves with BlueDots. Once you add a BlueDot, the history of how the curves were created is lost.

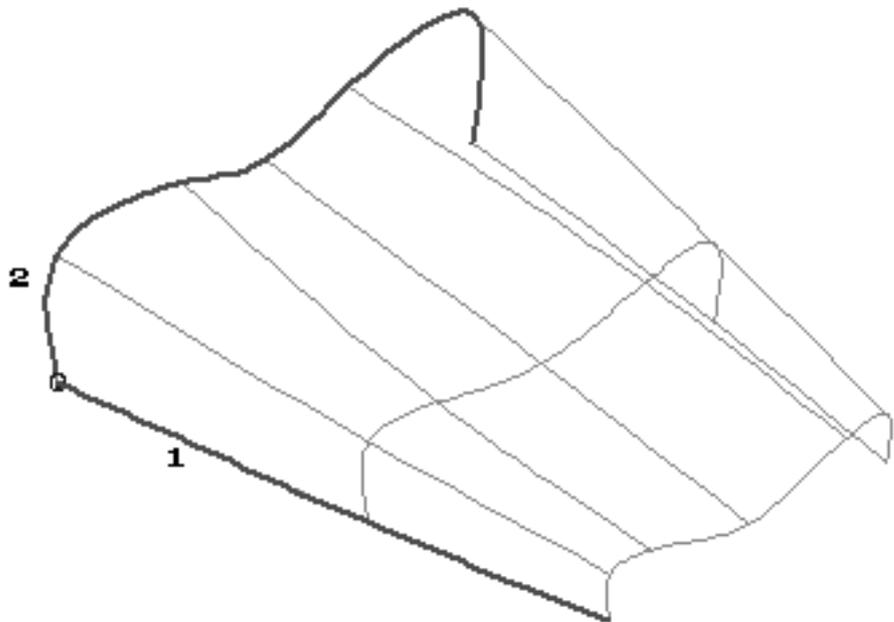
- On the Surfacing toolbar, click the BlueDot command.



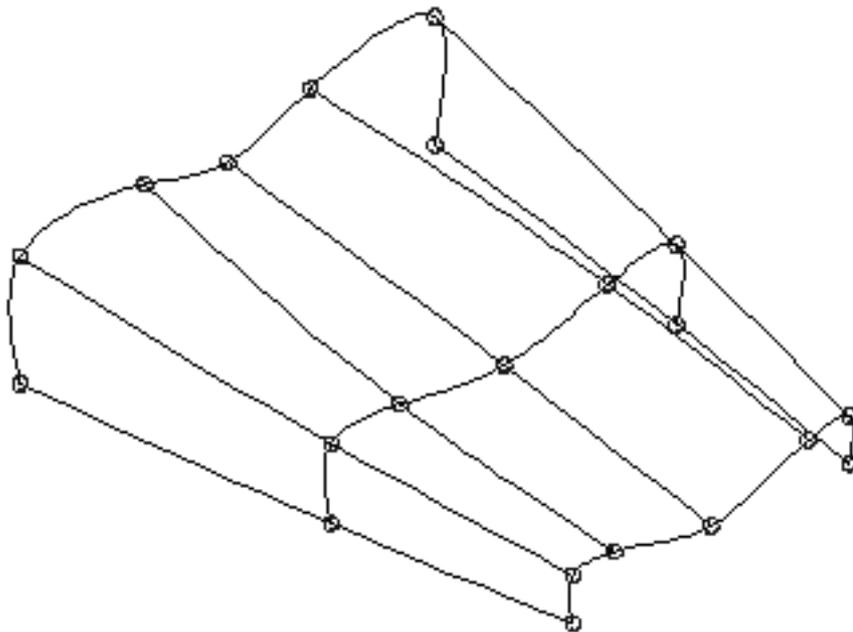
- Click keypoint curve 1 and then click keypoint curve 2 as shown.



Make sure no keypoints display when you select the curves to BlueDot connect. Just click the curve away from any possible keypoint.

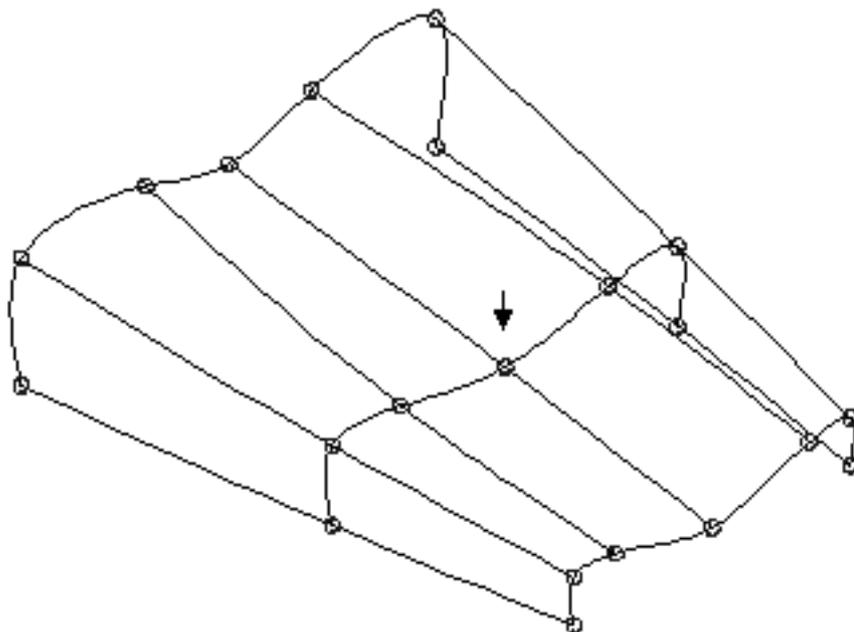


- Continue placing the remaining BlueDots. There will be a total of 21 BlueDots. If you make a mistake, click the Undo command.

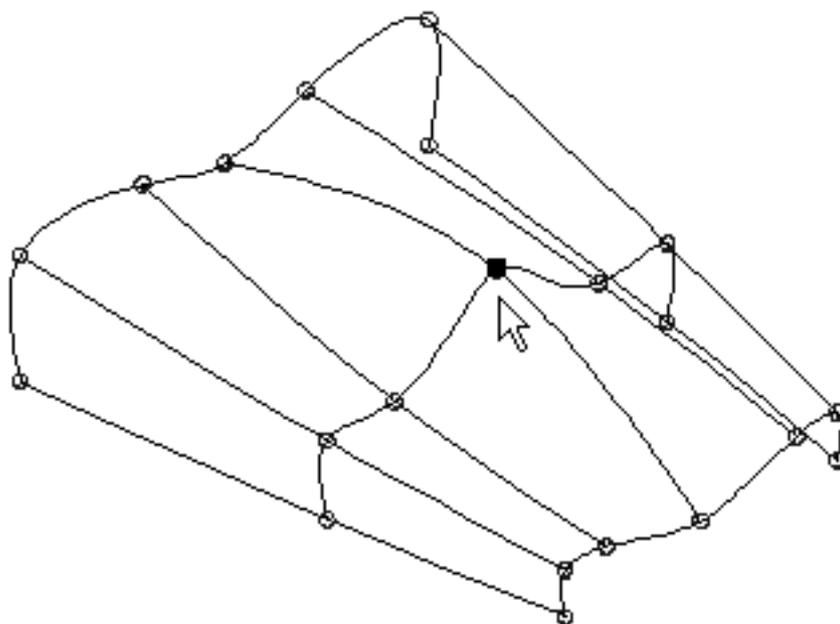


4. You will now edit a BlueDot and observe how the keypoint curves behave.
  - On Features toolbar, click the Select tool.

- Select the BlueDot shown.



- On the Select Tool ribbon bar, click Dynamic Edit.
- Click the Z-direction on the 3-D triad.
- Drag the BlueDot up and notice how the two keypoint curves remain connected.

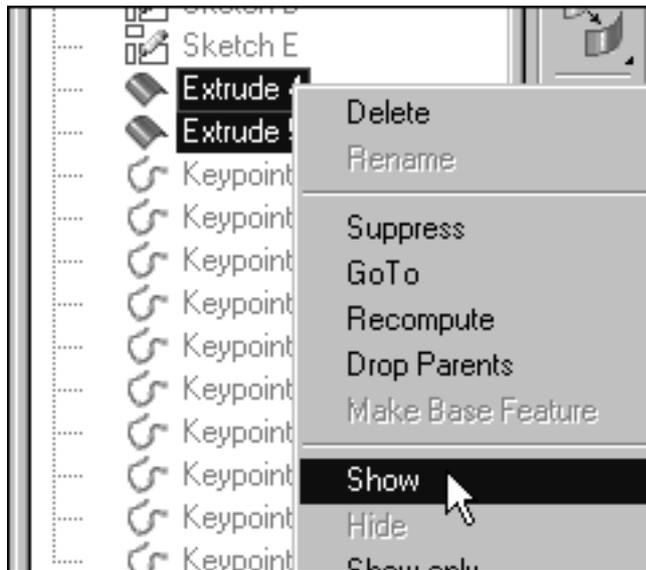




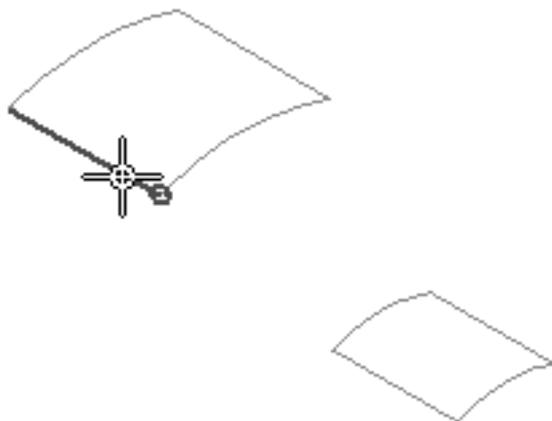
Notice in the BlueDot Edit ribbon bar that curve edit fields are grayed out. Keypoint curves cannot be controlled with local or shape edits.



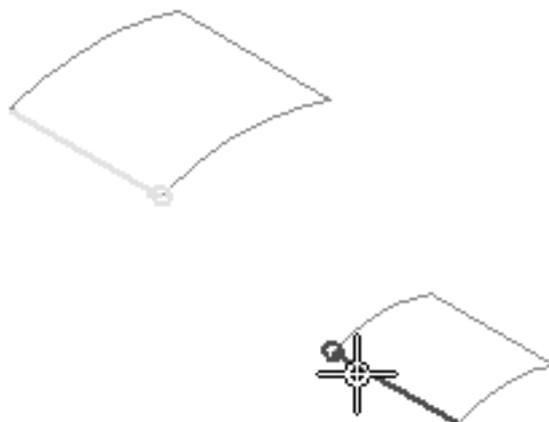
- Click the Select tool. Right-click in the part window and Hide All>BlueDots and Hide All>Construction Curves.
5. You will now create two keypoint curves that include a tangency vector.
- Click the Select tool.
  - In EdgeBar, hold down the Ctrl key and select features Extrude 4 and Extrude 5. Right-click these selections and click Show.



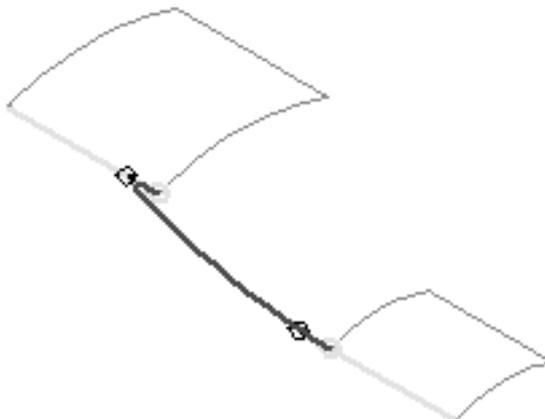
- You will now create keypoint curves between these two surfaces that are tangent to an edge of each surface. On the Surfacing toolbar, click the Keypoint Curve command.
- Select the edge of the surface as shown. Make sure the line and endpoint are highlighted.



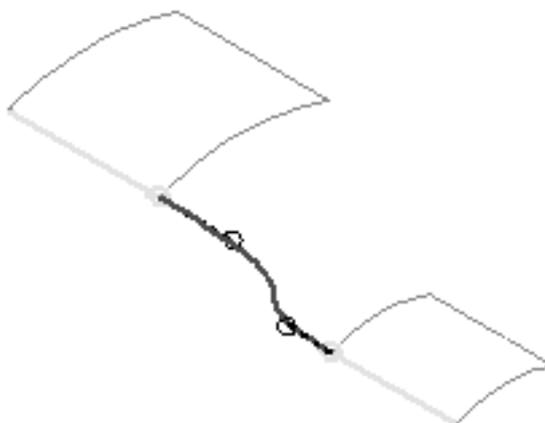
- Select the edge of the surface as shown. Make sure the line and endpoint are highlighted.



- 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.

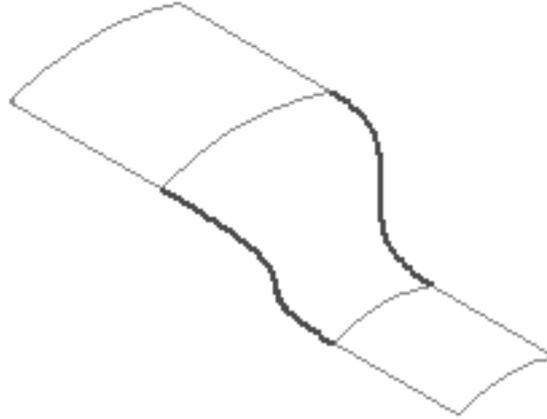


- Drag the tangency vectors as shown.



- On the Keypoint Curve ribbon bar, click the Accept button. Click the Accept button again and then click Finish.

- Create another keypoint curve on the opposite edges of the surfaces.



You will learn in the next module that the two keypoint curves you just created are a step in the process of constructing a tangent transition surface between two surfaces.

6. The activity is complete.



## Activity 4 – 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 be able to create the following:

- intersection curve
- cross curve
- projected curve
- contour curve
- derived curve
- split curve

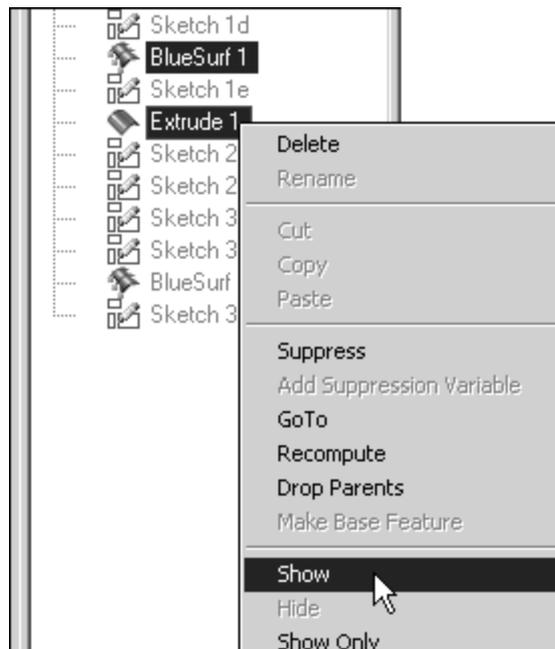
1. Open SURFACE LAB 2-04.PAR in the C:/Surfacing/Parts folder.



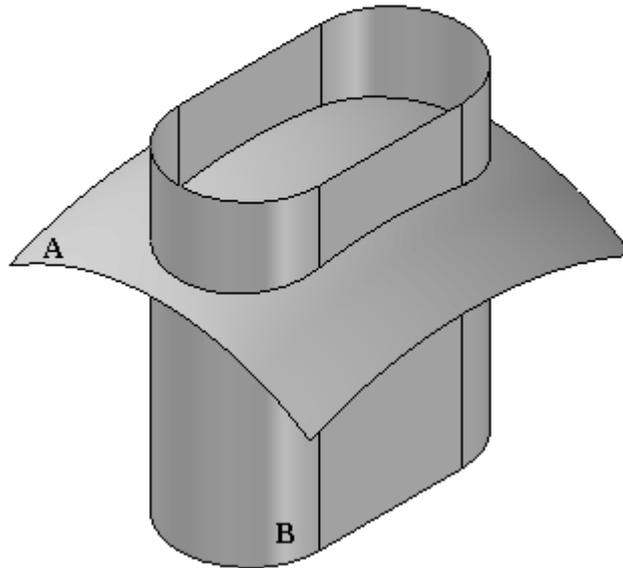
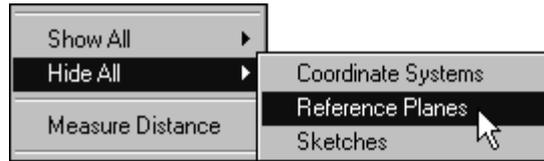
In order to create curves in this activity, you need existing construction surfaces. Since you have not yet learned how to create surfaces, the surfaces needed in the activity have already been created for you.

2. The first curve creation method you will use is Intersection Curve.

- In EdgeBar, select features BlueSurf 1 and Extrude 1. Right-click the selected features, and click Show.



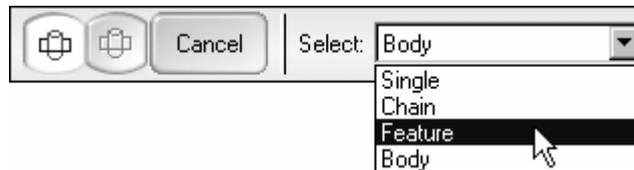
- Right-click in the part window and Hide All>Reference Planes.



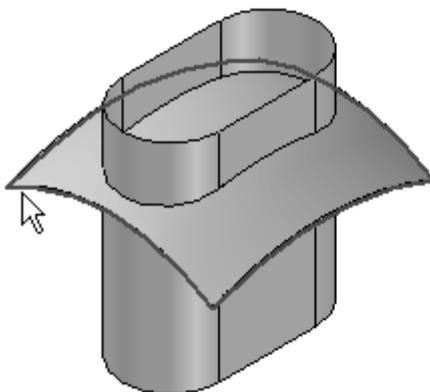
- You will now create a curve where construction surfaces A and B intersect.
- On the Surfacing toolbar, click the Intersection Curve command.



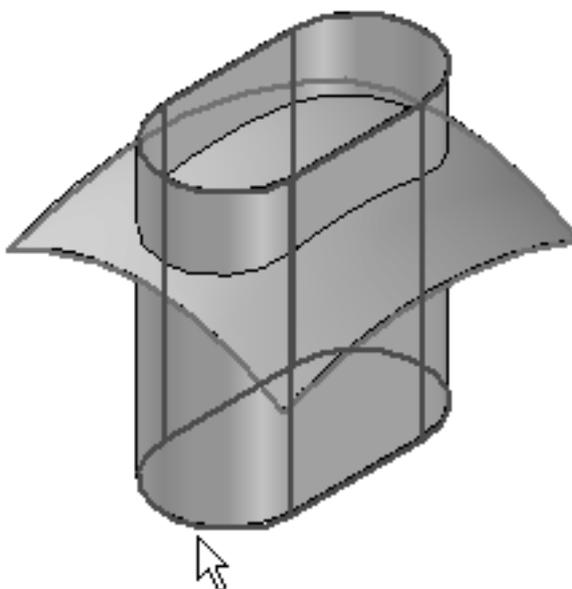
- On the Intersection Curve ribbon bar, set the select filter to Feature.



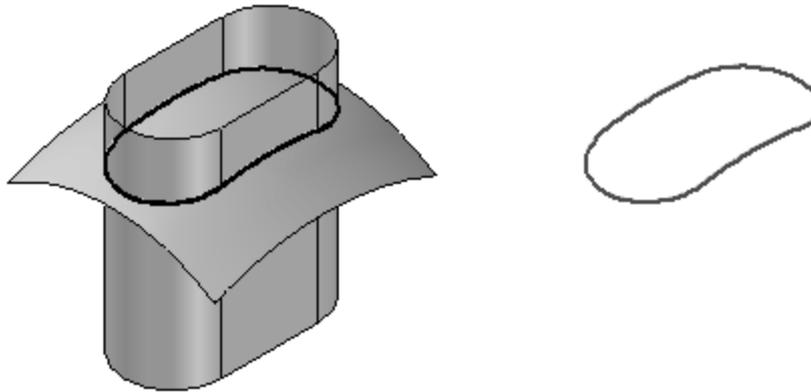
- Select the surface as shown, and then on the ribbon bar, click the Accept button.



- Select the next surface as shown, and then on the ribbon bar, click the Accept button.



- On the ribbon bar, click Finish.



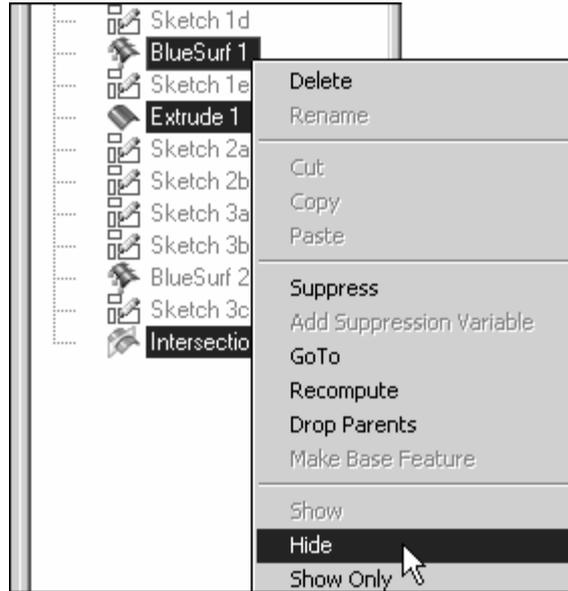
Notice in EdgeBar the intersection curve you just created is named Intersection 2.



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

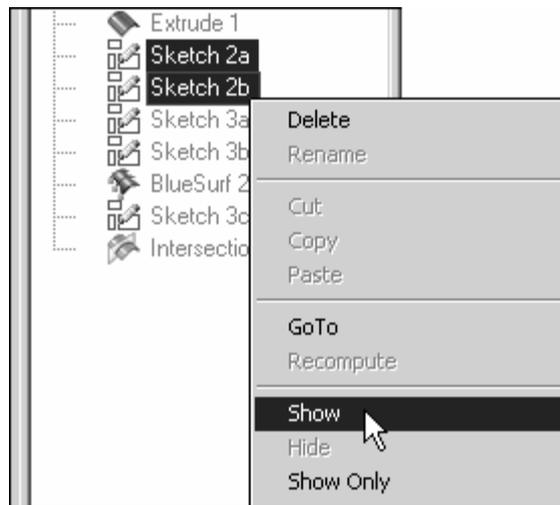
You will see in the next module how you can use the intersection curve in surface editing operations.

- Hide the following features in EdgeBar.



3. The next curve creation method you will use is Cross Curve. A cross curve is an intersection curve that is created with the theoretical extruded surfaces resulting from the two input curves/analytics.

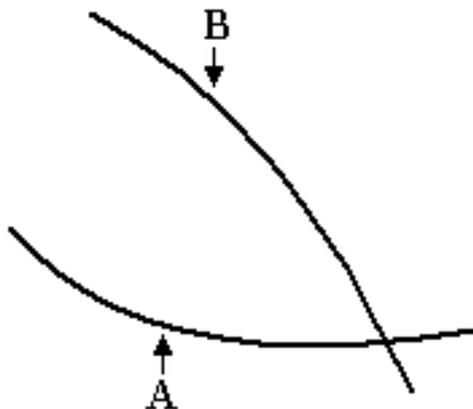
- In EdgeBar, show the following sketches.



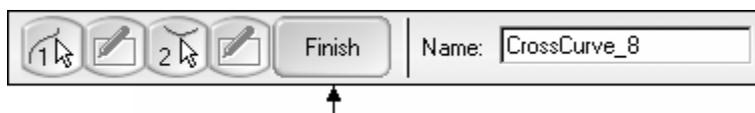
- On the Surfacing toolbar, click the Cross Curve command. The command is on the fly out of the Project Curve command.



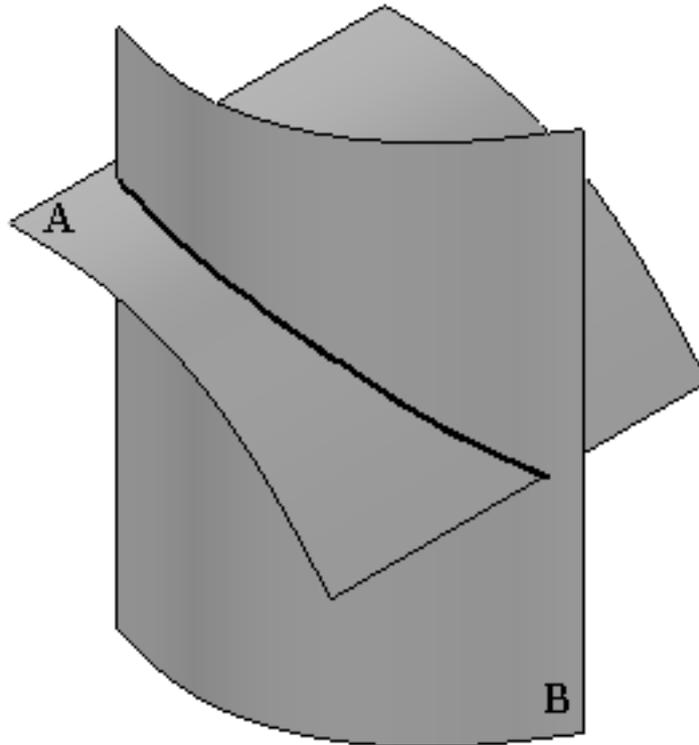
- Click sketch A and then on the Cross Curve ribbon bar, click the Accept button. Click sketch B and then on the Cross Curve ribbon bar, click the Accept button.



- On the Cross Curve ribbon bar, click Finish.

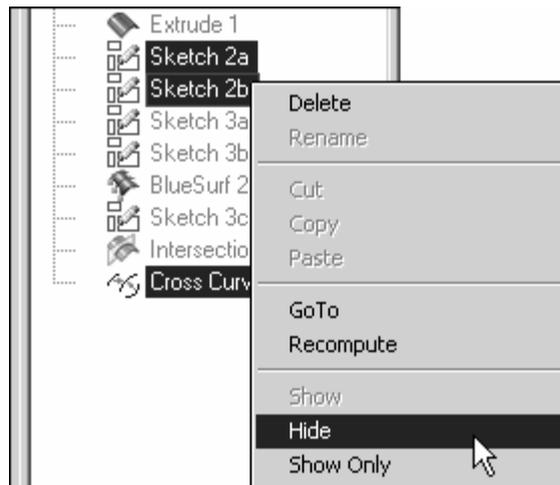


- The cross curve is the result of the intersection of the two theoretical extruded surfaces A and B.



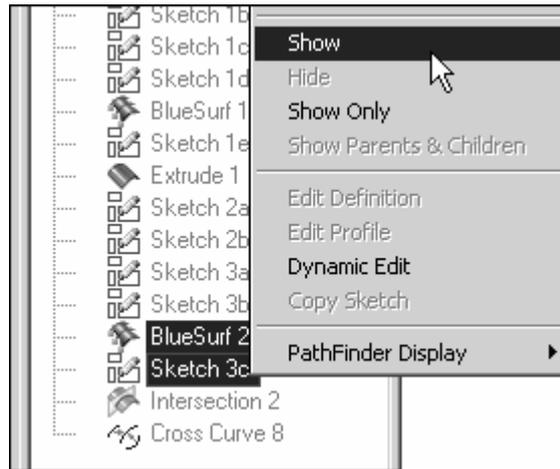
The Cross Curve command eliminates the need to construct extruded surfaces from curves and then find the intersection between the two surfaces.

- Hide the sketch curves and cross curve shown.



4. The next curve creation method you will use is Project Curve. This command takes a curve and projects it onto a surface.

- Show the following features in EdgeBar.



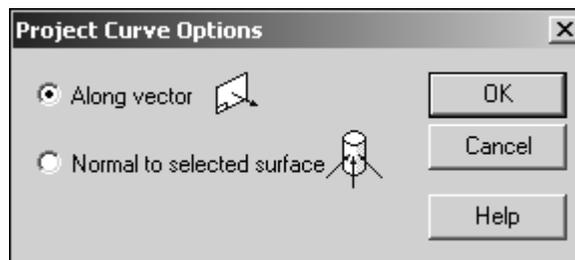
- You will now project curve B onto surface A.
- On the Surfacing toolbar, click the Project Curve command.



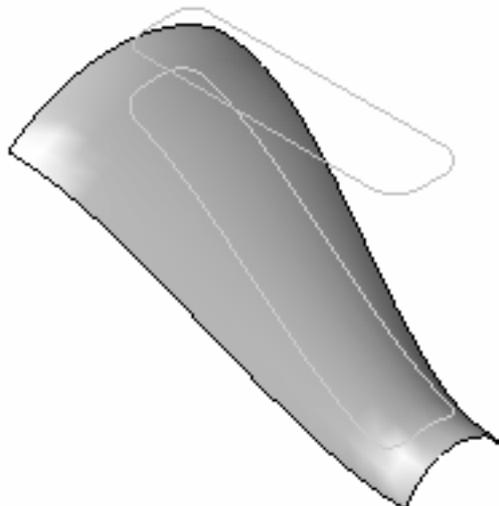
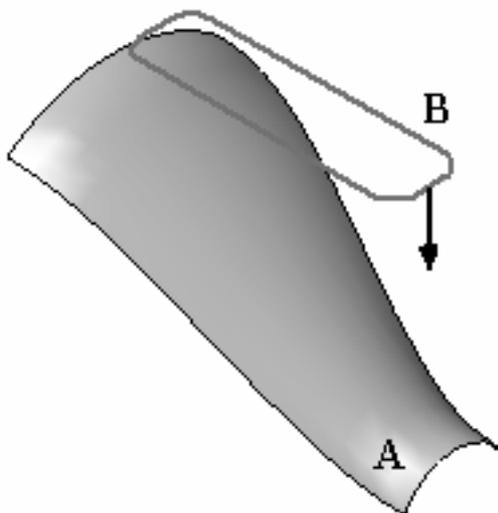
- On the Project Curve ribbon bar, click the Options button.



- The default option is Along vector. The curve will be projected along its normal vector. Click OK.



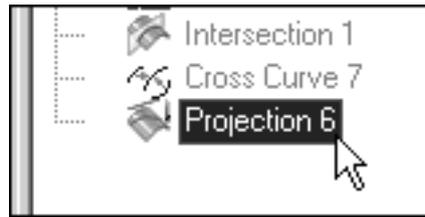
- Click curve B, and then click the Accept button.
- Click surface A, and then click the Accept button. Then point the direction arrow down as shown.



- Click Finish.



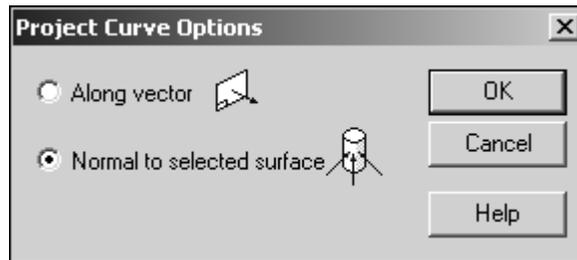
- Click the Select tool. In EdgeBar, select the Projection feature and press the Delete key on the keyboard.



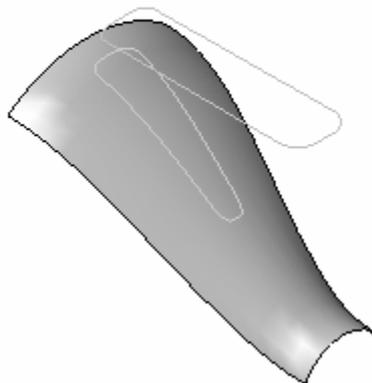
- You will now project the curve normal to the surface. On the Surfacing toolbar, click the Project Curve command.
- On the Project Curve ribbon bar, click the Options button.



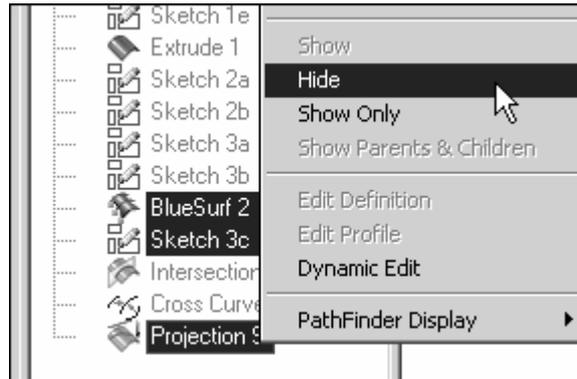
- Click the Normal to selected surface option and click OK.



- Click the curve, and then click the Accept button.
- Click the surface, and then click the Accept button. Click Finish.

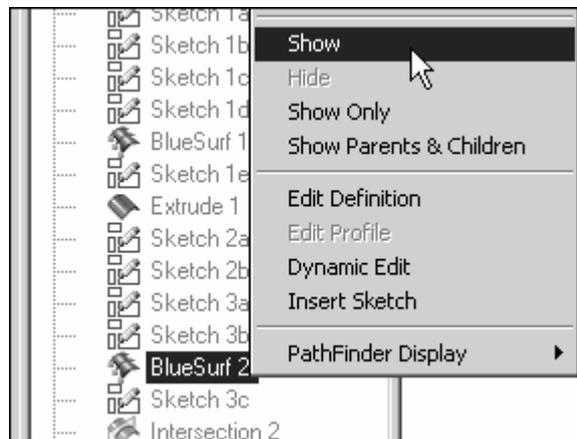


- Notice the different results. Hide the features shown.



5. The next curve creation method you will use is Contour Curve. You will draw a contour curve directly on a surface.

- In EdgeBar, show the feature shown below.

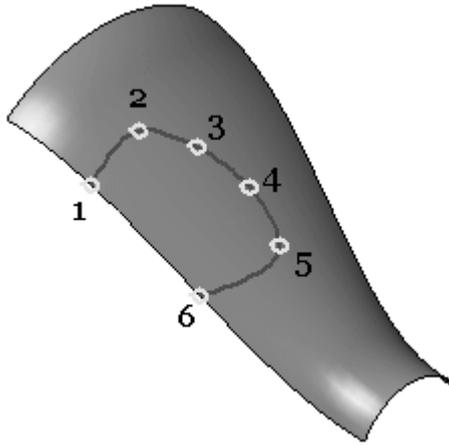


- On the Surfacing toolbar, click the Contour Curve command.

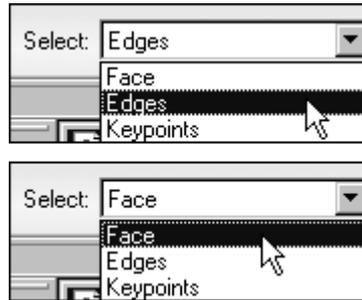


- Click the surface and then click the Accept button.

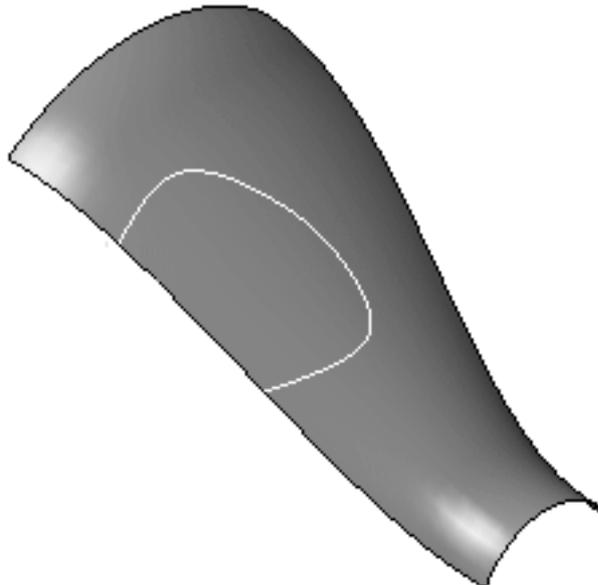
- Click on the surface to place the six contour curve points approximately as shown. Points 1 and 6 are on the edge. Points 2–5 are on the face.



To insert points on the edge, in the Select box click Edges. To insert points on the face, in the Select box click Face.



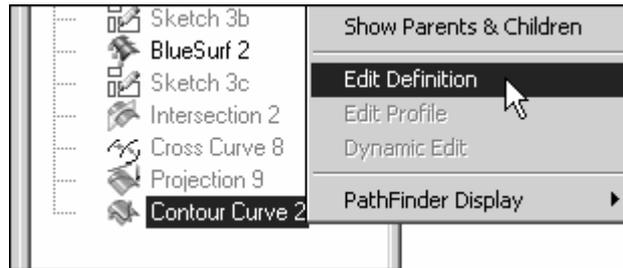
- After you place the last point, click the Accept button.
- Click Finish.



- Click the Select tool.
- You will now edit the shape of the contour curve. In EdgeBar, right-click the contour curve feature, and click Edit Definition.



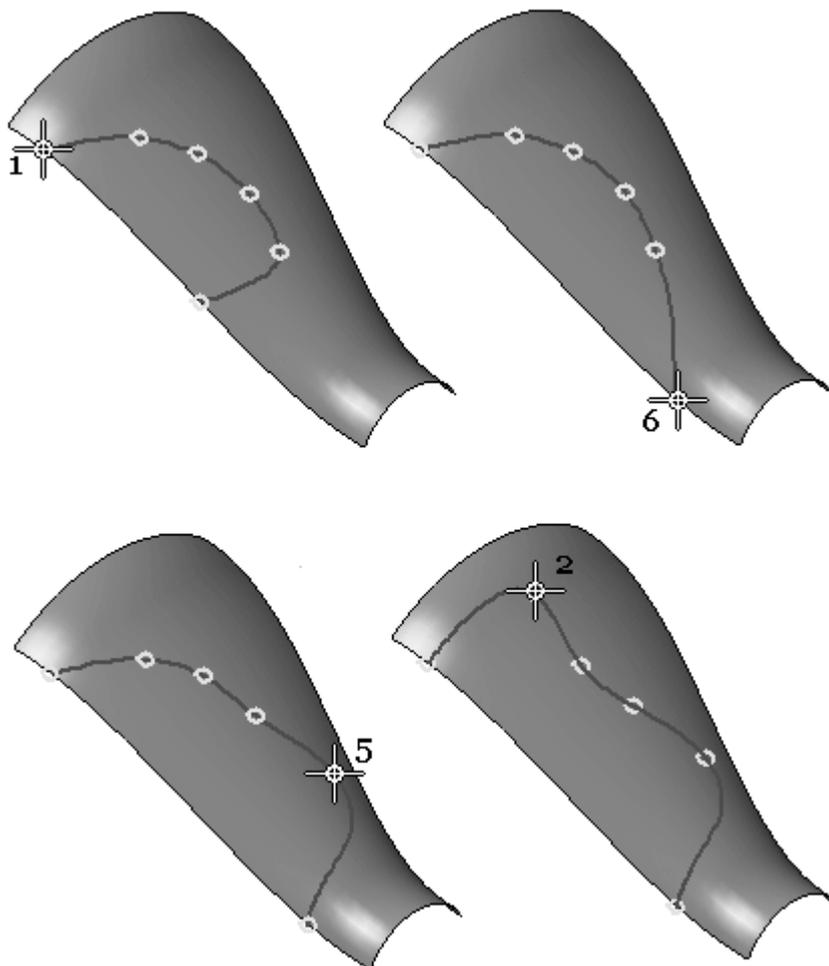
You can edit the shape while creating the contour curve.



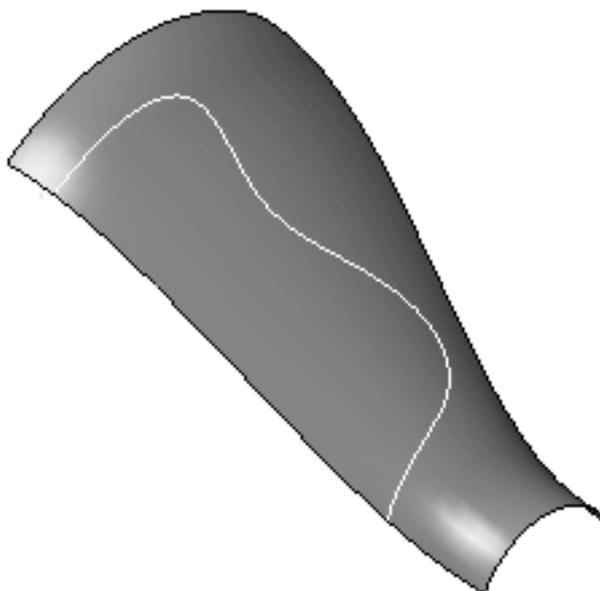
- On the Contour Curve ribbon bar, click the Draw Points Step.



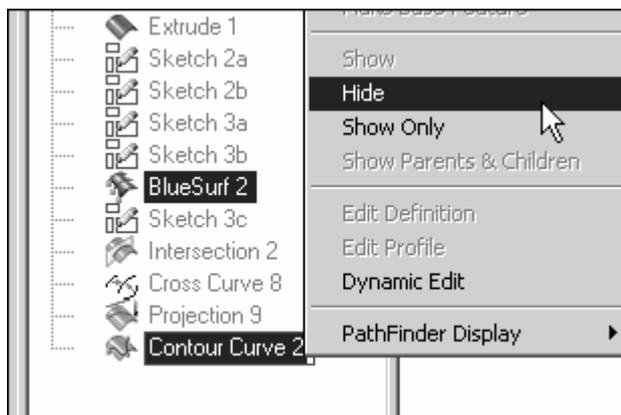
- Click the points shown and drag to edit the shape approximately as shown. Points 1 and 6 will remain attached to the edge. Points 2–5 can be moved anywhere along the face.



- Click the Accept button and then click Finish.

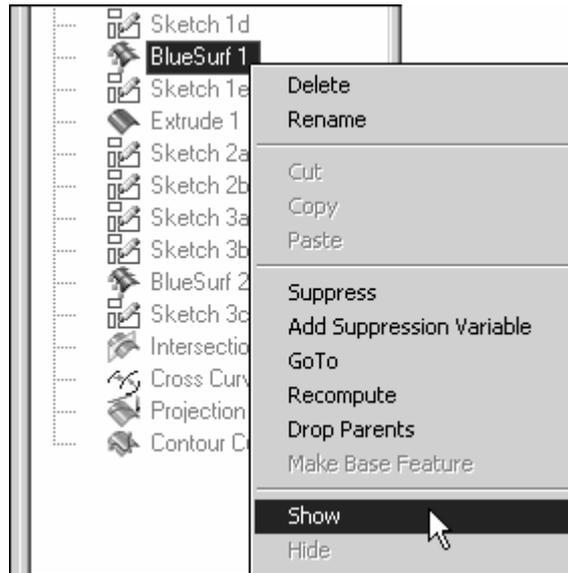


- Hide the two features shown.



6. The next curve creation methods you will use are Derived Curve and Split Curve.

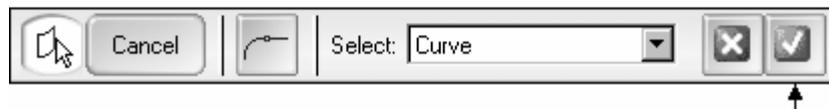
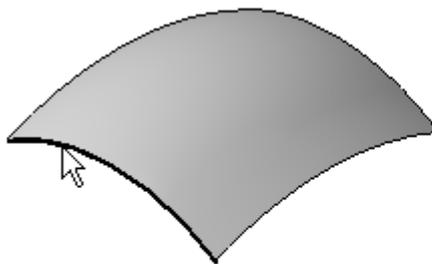
- In EdgeBar, show the feature shown below.



- You will now create derived curves from the four edges of the surface. You will split the derived curves that would be used to create additional surfaces. You will not create the surfaces in this activity.
- On the Surfacing toolbar, click the Derived Curve command.

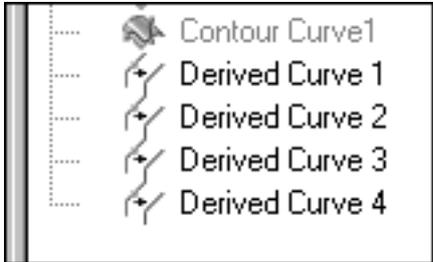
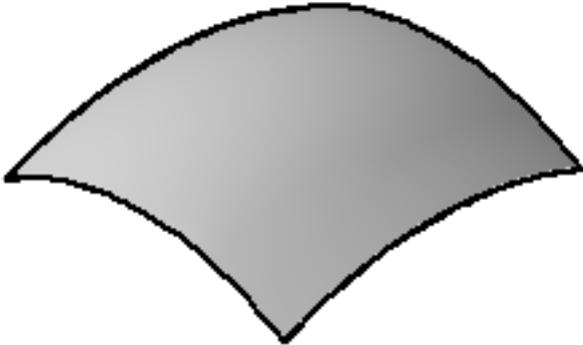


- Click the edge shown, and then click the Accept button on the Derived Curve ribbon bar.

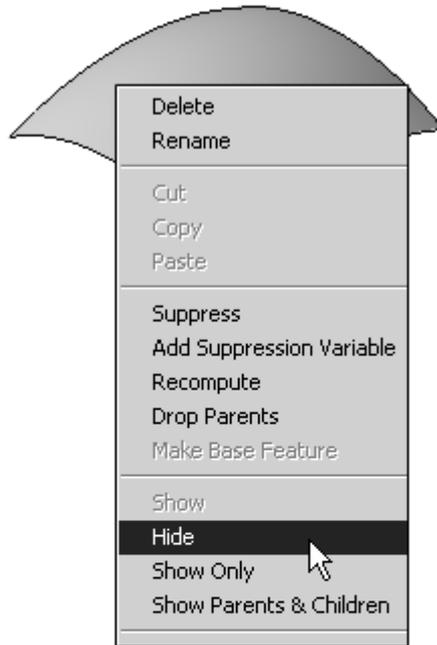


- On the ribbon bar, click Finish.

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



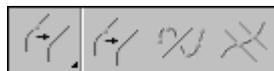
- You will now split these derived curves. Click the Select tool and then right-click the surface in the part window and click Hide.



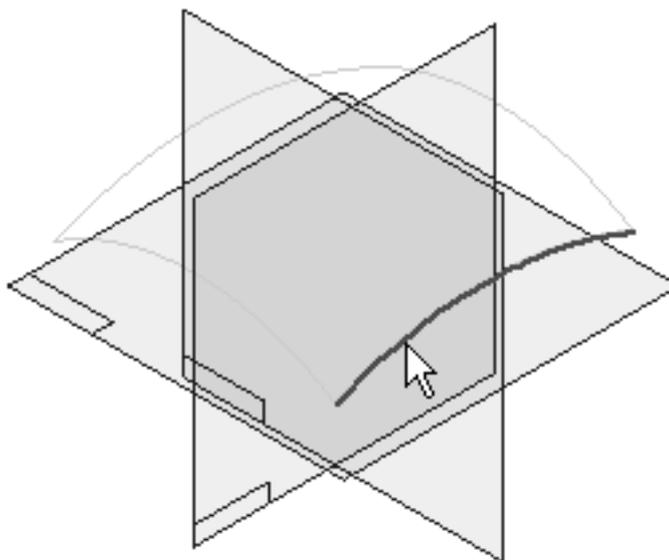
- Right-click in an open area of the part window and click Show All>Reference Planes.



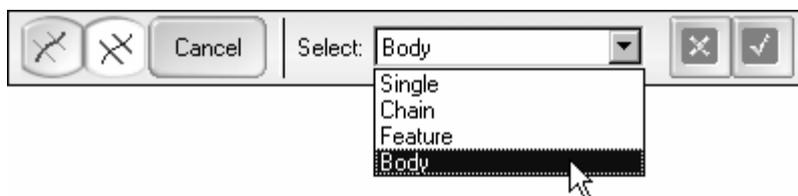
- On the Surfacing toolbar, click the Split Curve command. The command is located on the fly out of the Derived Curve command.



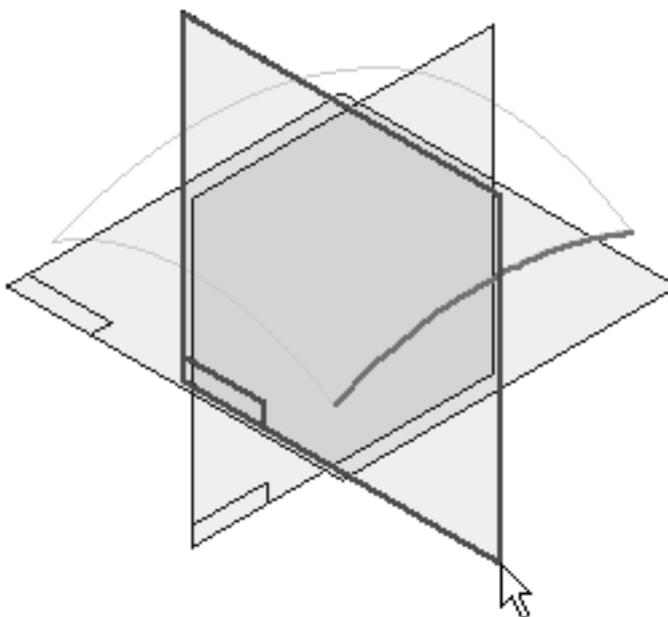
- Click the derived curve as shown, and then click the Accept button.



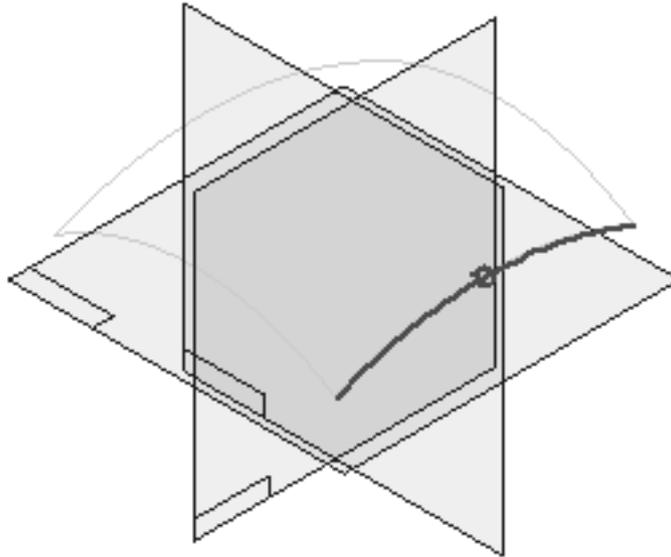
- On the Split Curve ribbon bar, set the Select filter to Body.



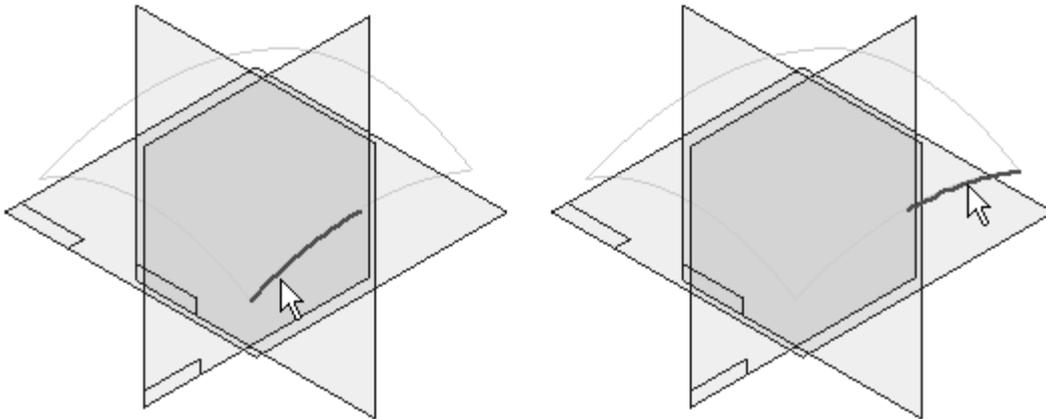
- Click the reference plane shown.



- Click the Accept button, and then click Finish.

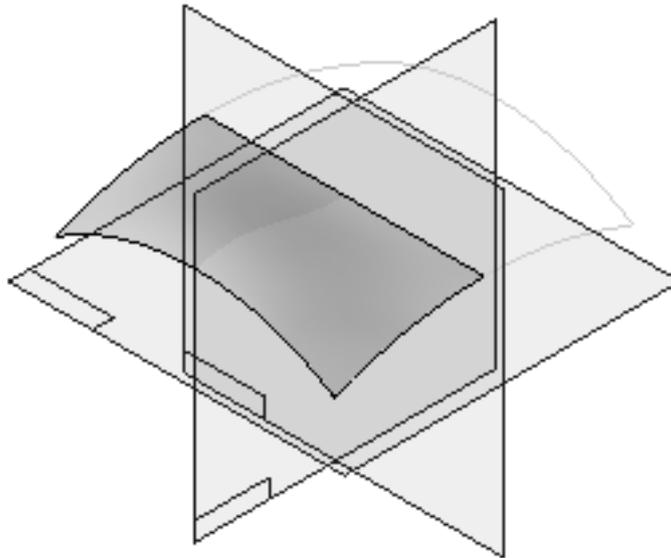


- Notice the original derived curve is now split and there are two new curves that can be used for surfacing operations.



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

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



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

## Summary

Curves are crucial for controlling surfaces. You can edit curves by moving edit points and control points. You can add more control to a curve by increasing the degree of the curve. You can draw curves directly by defining the edit points. The direct curve methods are Curve command, Curve by Table and Contour Curve command. You can also create curves indirectly from existing curves and surfaces. These curves are controlled by the parent curves and surfaces. As the parents change, so will the indirected curves. These curve creation methods are Derived Curve, Cross Curve, Intersection Curve, Project Curve and Keypoint Curve.

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## Chapter

# 3 *Surface Creation*

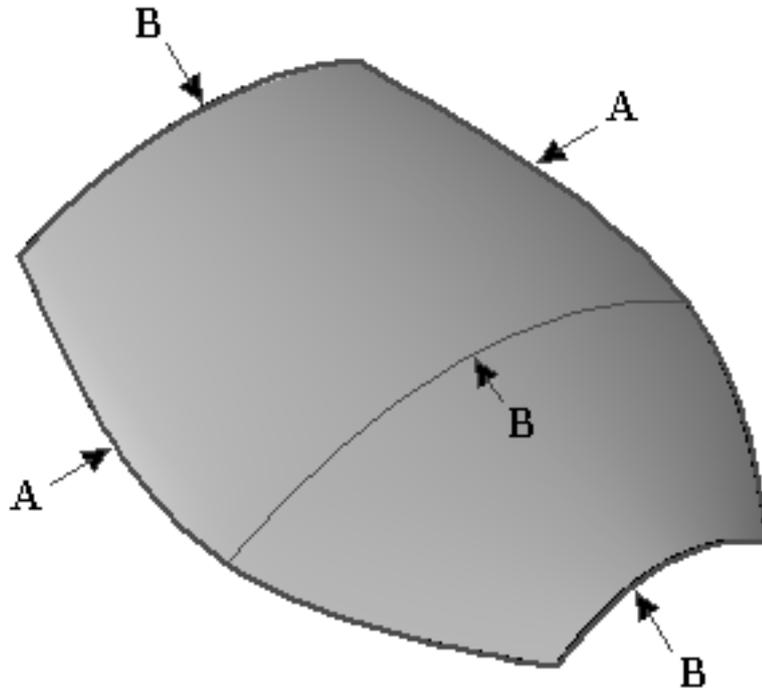
## Objectives

After completing this module, 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. Visualize a surface as a thin sheet. A surface has no thickness. A surface can be simple or complex. The more curves used to define the surface the greater the surface complexity. In a mathematical definition, a surface has two directions (u and v). In modeling terms, a surface consists of cross-sections curves and guide curves. Guide curves can exist or be interpolated by the system from the cross-section elements.



|     |                      |
|-----|----------------------|
| (A) | Guide curves         |
| (B) | Cross Section curves |

You can modify a surface shape by editing the underlying cross section and guide curves. You can trim a surface, extend, offset, copy and mirror while maintaining its shape. You can stitch together multiple surfaces to form a solid or to include rounds between adjoining surfaces.

Curves form the mathematical basis for a surface. As you increase your understanding of how to control curves, you are taking a major step towards understanding how to control surfaces.

## Creating a Simple Surface

You will begin the surface creation module with the simplest methods. An extruded surface is the simplest surface creation method. The only input required is a sketch or profile containing curves or analytic elements. The Extruded Surface command works similarly to the Protrusion command.

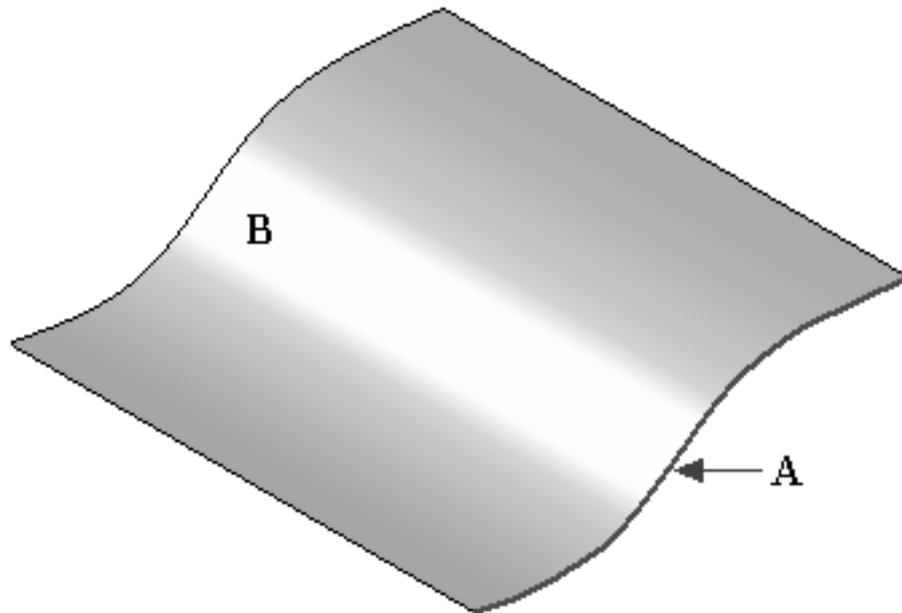


Extruded Surface Creation Steps:

- Start the Extruded Surface command.
- Select a profile plane.
- Draw the cross section. You can use analytics or curves.
- Finish the profile step.
- Define the extruded distance dynamically or with a value on the ribbon bar. The symmetric extent option is available.
- Finish extruded surface command.

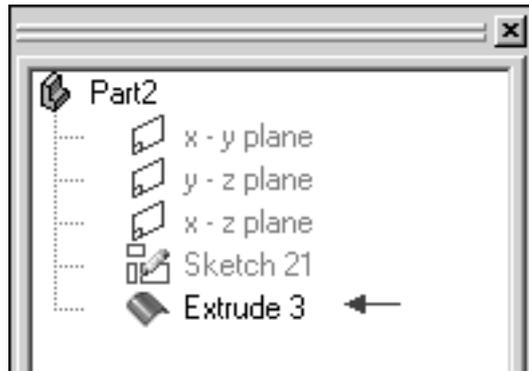


You can create a sketch to draw the cross-section. Use the Select from Sketch option to create an extruded surface from a sketch.



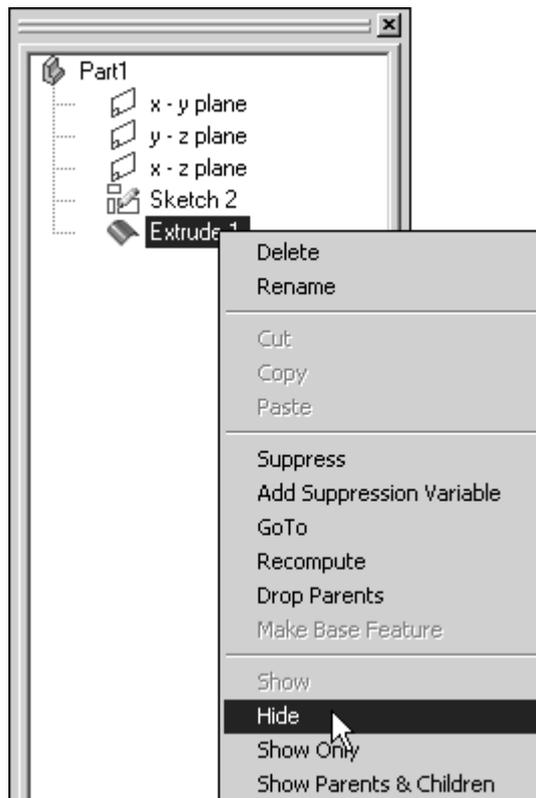
|     |                  |
|-----|------------------|
| (A) | profile/sketch   |
| (B) | extruded surface |

Notice the extruded surface feature in EdgeBar. The extruded surface was created from a sketch. The sketch is shown in EdgeBar as Sketch21.



You can use extruded surfaces as construction surfaces to create intersection curves with other surfaces. An extruded surface may also be the surface you actually need. If you use an extruded surface as a construction surface to generate intersection curves, you can turn off its display. It may no longer need to be displayed. You should not delete it because it is a parent to the intersection curves generated from it. If you want to delete a surface that has children, you must go to the children and drop their parents. The surface could then be deleted but the children are no longer associative and cannot be edited. Caution is highly recommended when using the drop parents command. As you become more familiar with the surfacing process, you will understand when to use drop parents.

To hide the display of the surface, in EdgeBar right-click the surface and click Hide.



The next simple surface creation method is a revolved surface. The only input required is a sketch or profile containing curves or analytic elements and an axis of revolution. The Revolved Surface command works similarly to the Revolved Protrusion command.

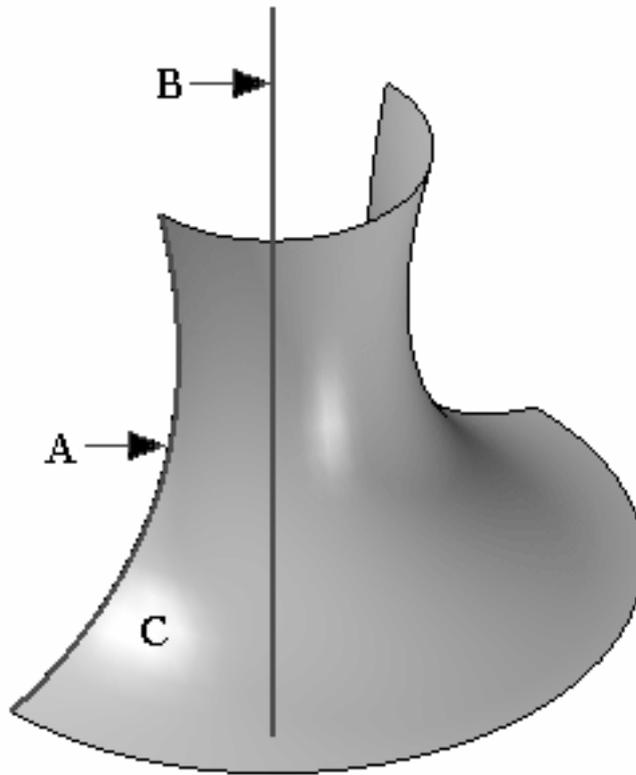


**Creation Steps:**

- Start the Revolved Surface command.
- Select a profile plane.
- Draw the cross section. The cross-section can be open or closed. You can use analytics or curves.
- Define the Axis of Revolution. You can select a reference plane for the axis or select a line drawn in the profile as the axis of revolution.
- Finish the profile step.
- On the ribbon bar, enter a revolved angle or dynamically drag the revolved surface to an angular distance. The symmetric extent and 360 degree options are available.
- Finish the revolved surface command.

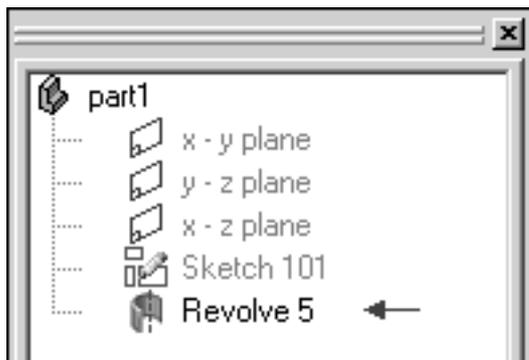


You can draw the cross section and axis of revolution in a sketch. Use the Select from Sketch option to create a revolved surface from a sketch.



|     |                    |
|-----|--------------------|
| (A) | profile/sketch     |
| (B) | axis of revolution |
| (C) | revolved surface   |

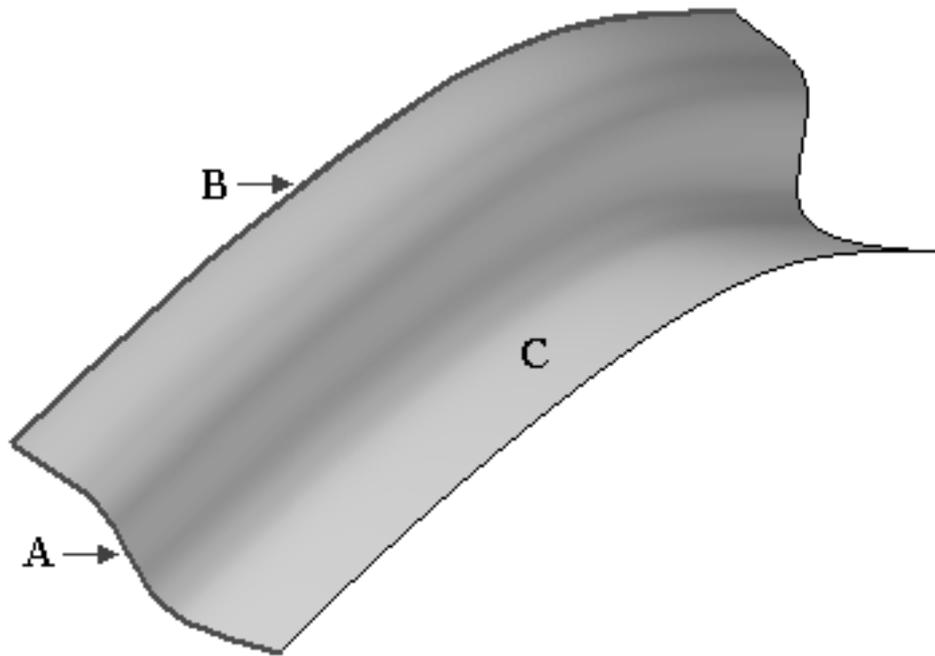
Notice the revolved surface feature in EdgeBar. The revolved surface was created from a sketch. The sketch is shown in EdgeBar as Sketch101.



## Creating a Swept Surface



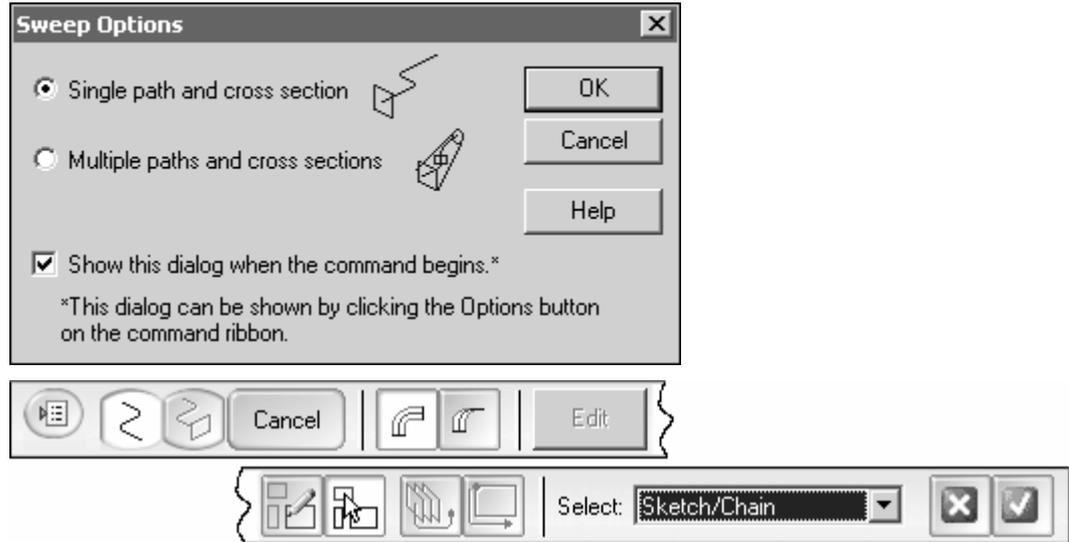
The Swept Surface command creates a construction surface by extruding a profile along a path.



An example of a swept surface using a single path and cross-section

|     |                |
|-----|----------------|
| (A) | profile/sketch |
| (B) | guide path     |
| (C) | swept surface  |

You must specify the Sweep Options before the command begins.

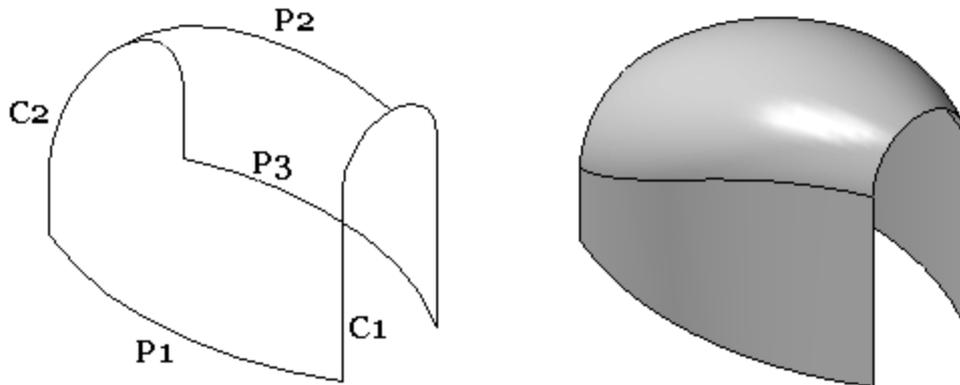


The “single path and cross-section” option allows you to only define one path and one cross section. The path and cross section can be open or closed.

The “multiple paths and cross-sections” option allows you to define up to three paths and an unlimited number of cross sections. After you define one or two paths, click the Next button on the ribbon bar to proceed to the cross-section step. After you define the third path, the command automatically proceeds to the cross-section step. The cross sections can be all open or all closed and can be planar or non-planar. You cannot mix open and closed cross-sections. You can place them anywhere along the path. A sweep path can consist of either tangent or non-tangent elements.

When you create a swept surface using a closed profile, you can use the Open Ends and Close Ends options on the SmartStep ribbon 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.

For cross sections and paths, you can select wireframe elements from multiple Parasolid bodies or sketches and the elements will remain associative.



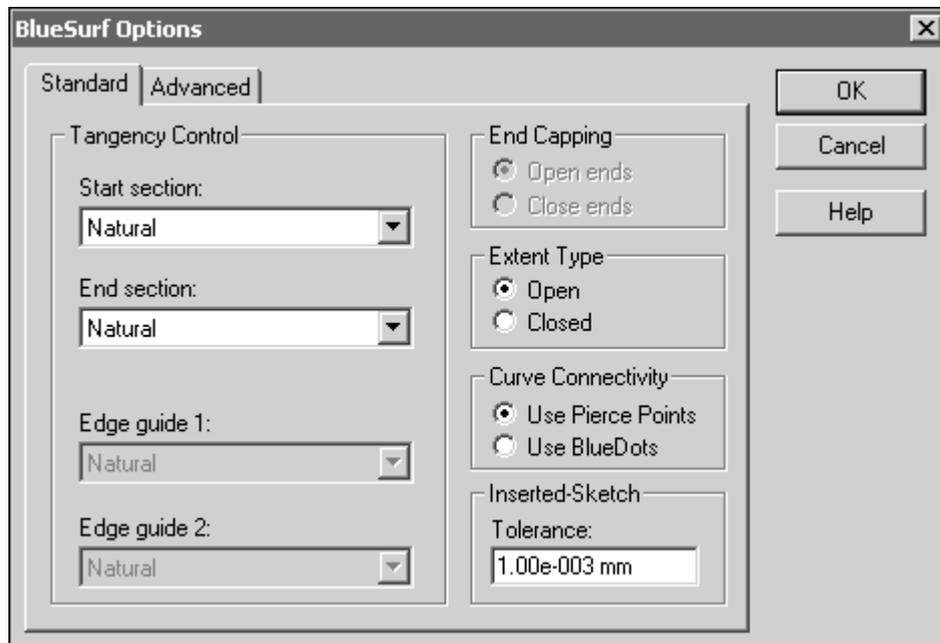
|            |                |
|------------|----------------|
| P1, P2, P3 | path curves    |
| C1, C2     | cross sections |

## BlueSurf

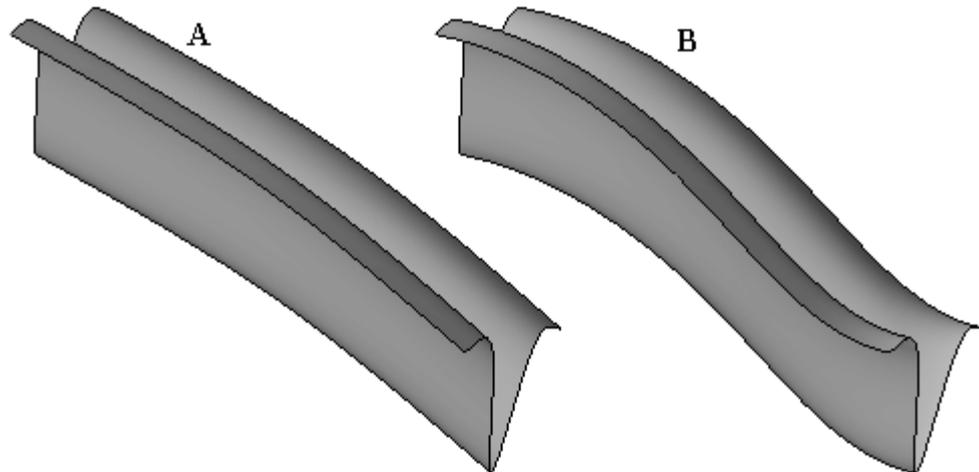


BlueSurf is a surface creation command for developing complex surfaces. A BlueSurf is a highly editable surface. It behaves similarly to the loft and sweep commands, where numerous sections and guides can be used. You can insert new sections or guides on a BlueSurf with intersection planes, which will be added to the surface for additional control. You can manipulate the surface by moving BlueDot Edit points. You can use the Shape Edit or Local Edit option when you move BlueDots on a BlueSurf. As you add sections or guides, the amount of edit points may need to be reduced or added, through another concept called (Edit Point Data Manipulation).

To begin the overview of the BlueSurf command, look at the BlueSurf Options dialog box.

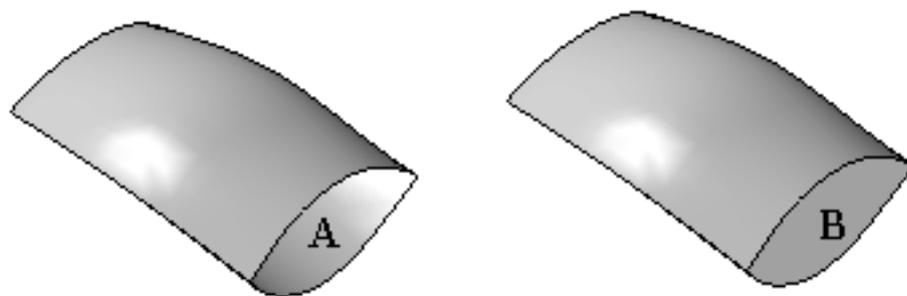


On the Standard tab, there are options to control tangency for cross-sections and edge guides. The default setting for cross-section tangency is natural. The following example shows the results from a natural (A) and normal to section (B) tangency control setting.

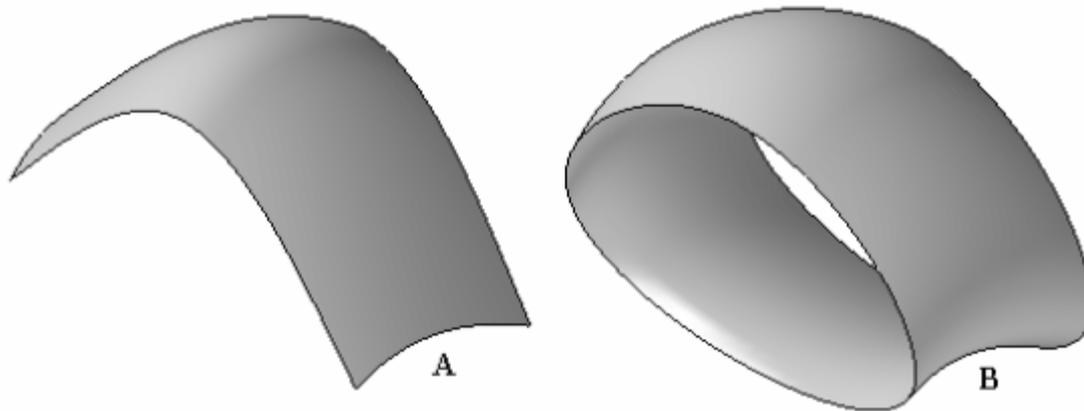


The Edge Guide option specifies the tangency control for the guide curves. The options available for defining guide curve tangency conditions depend on the type of element you select for the guide curve. For example, if you want to be able to control the tangency of the BlueSurf feature with respect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

The End Capping option is available when you use closed cross sections. The default option is Open. The resulting surface will have both ends open. The Close ends option caps the ends of the surface, creating a closed volume. The following example shows Open ends (A) and Closed ends (B) results.

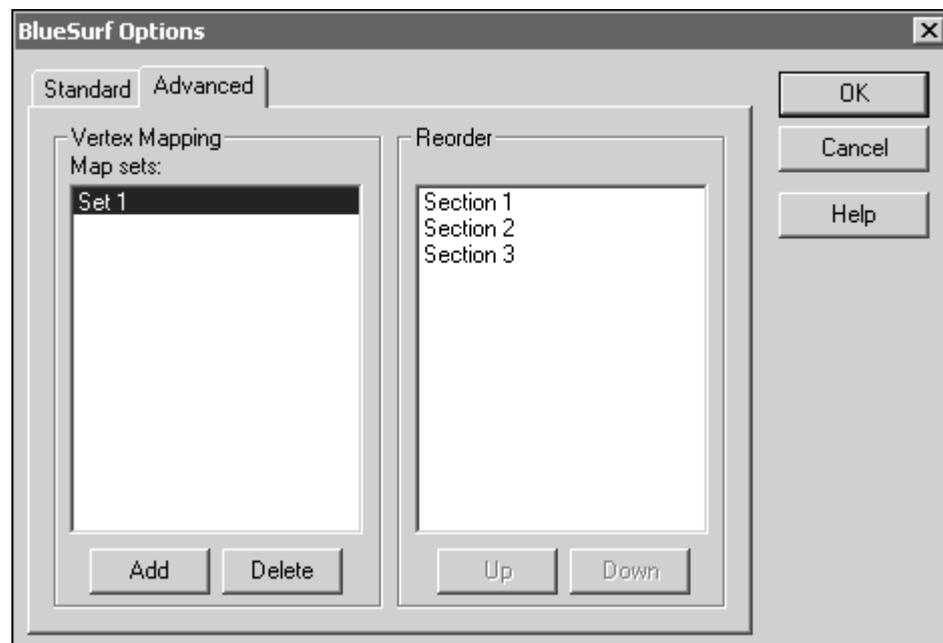


The Extent Type control allows Open or Closed extents. Open extent creates a surface that starts at the first cross-section and ends at the last cross section. A closed extent starts and ends at the first cross section, to form a closed loop. The following example shows the two extents: (A) Open, (B) Closed.



The Curve Connectivity option applies to inserted sketches. You will learn about inserting sketches later in this module. The inserted sketches can be either connected with pierce points or with BlueDots.

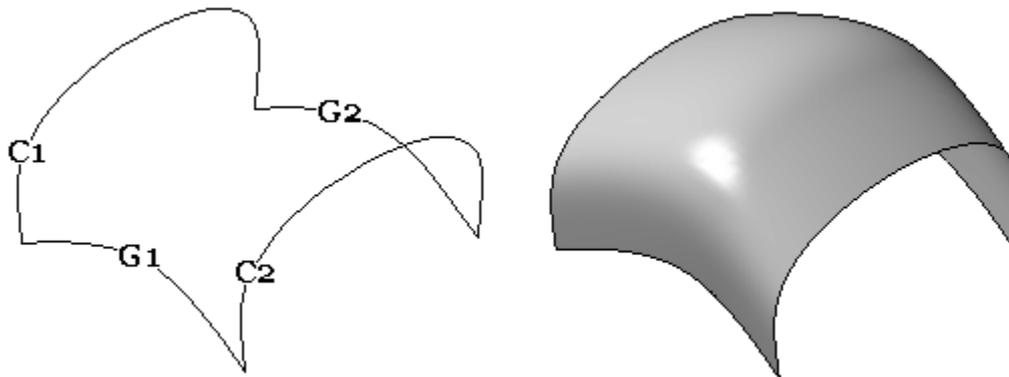
The Advanced tab lets you apply vertex mapping. This is covered later in this module.



Examine the BlueSurf SmartStep ribbon bar.

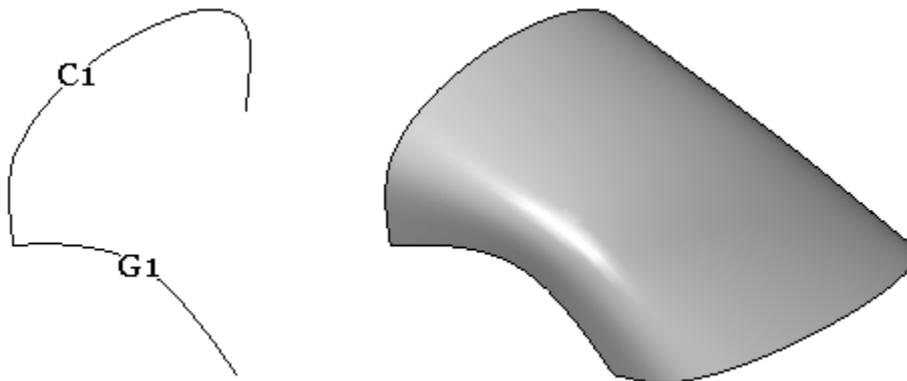


The first step in creating a BlueSurf is selecting cross sections. The Cross Section Step (B) activates automatically. At least two cross sections are required. After the cross-section step, you can select a guide curve if needed. Click the Guide Curve Step (C) and select the guide curve(s). Once you select the cross sections and guide(s), click Preview and then Finish.



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

A BlueSurf can also consist of a single cross-section and a single guide curve.



This 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.

## Inserting Sketches



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 section(s) or guide curve(s). You can use the BlueSurf Options dialog box to specify whether a connect relationship or a BlueDot is used to connect the sketch to the guide curves. The option you specify also affects how you can edit the feature later.

On the BlueSurf Options dialog box, there are two methods for curve connectivity (Use Pierce Points or Use BlueDots).

### 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. When you connect the new sketch using the Use Pierce Points option, you can modify the cross section or guide curve the new sketch intersects and the bspline curve for the inserted sketch will update. 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. The Use Pierce Points option maintains the existing parent/child history of the model.

### Use BlueDots

Specifies that a BlueDot be 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 on models that are driven by aesthetic 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 aesthetic 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.

### Insert Sketch Option

You can also 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. As the tolerance value decreases, more control points are added to the inserted sketch curve.

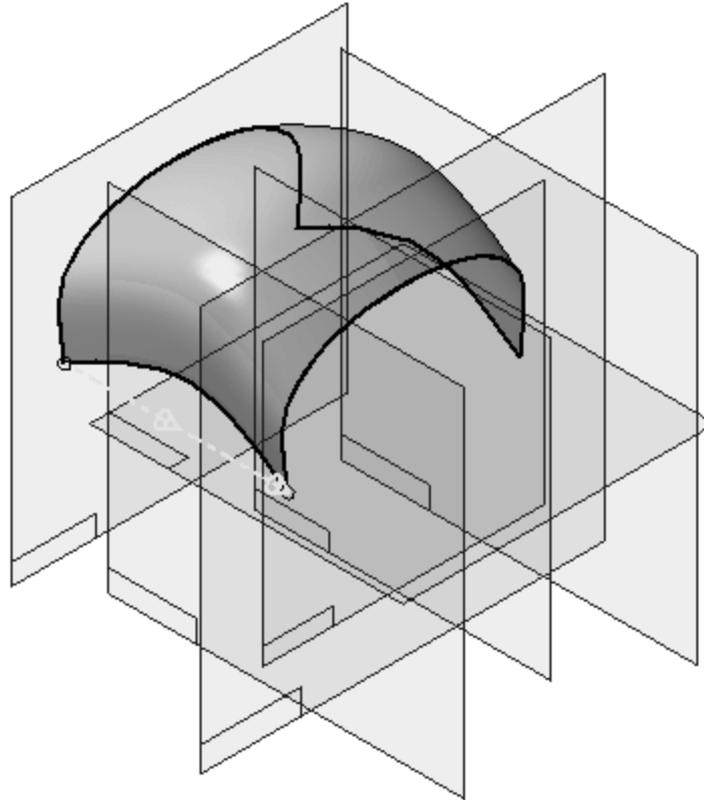
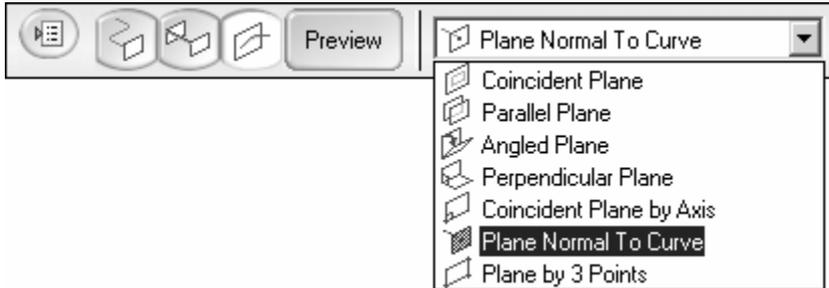
You can select the Insert Sketch Step while in the BlueSurf command. If the BlueSurf has already been created, you can select the BlueSurf with the Select tool and then click the Edit Definition command on the Select tool ribbon bar.



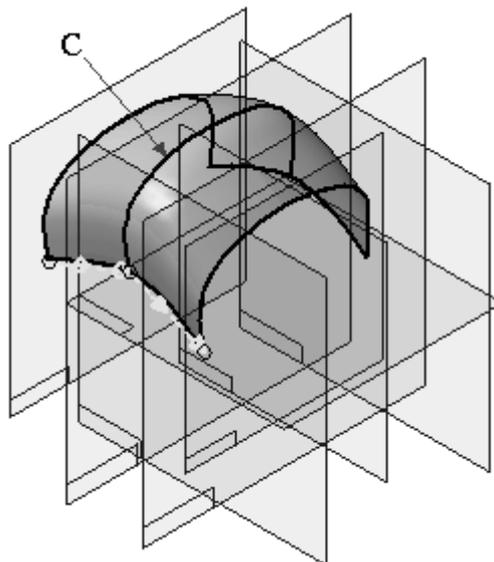
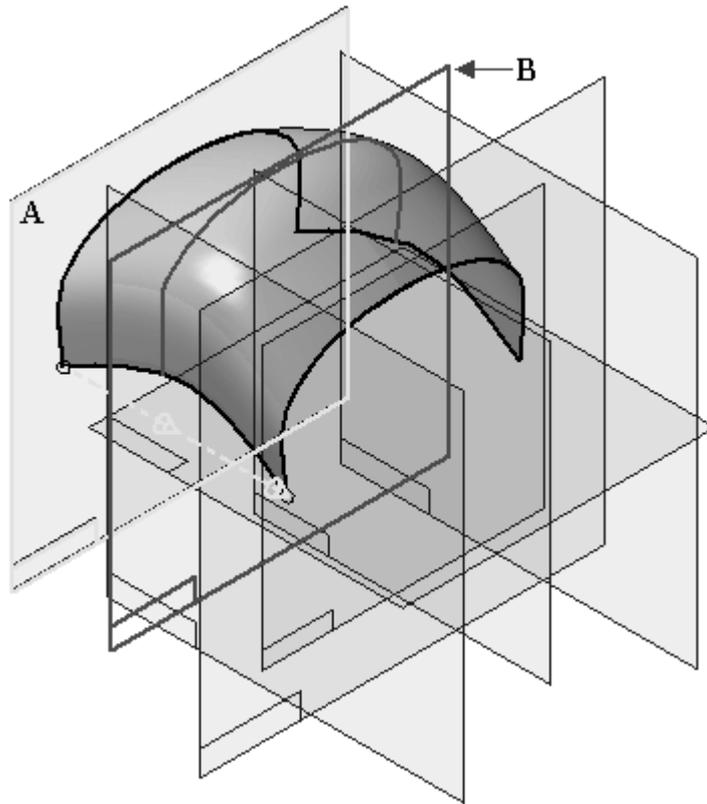


In the following example, the Use BlueDots option is used for curve connectivity.

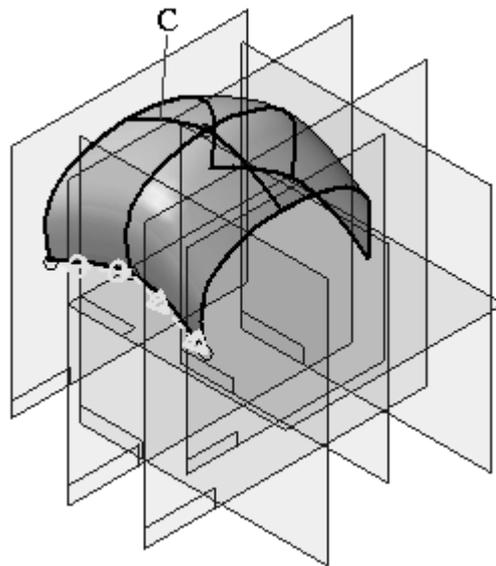
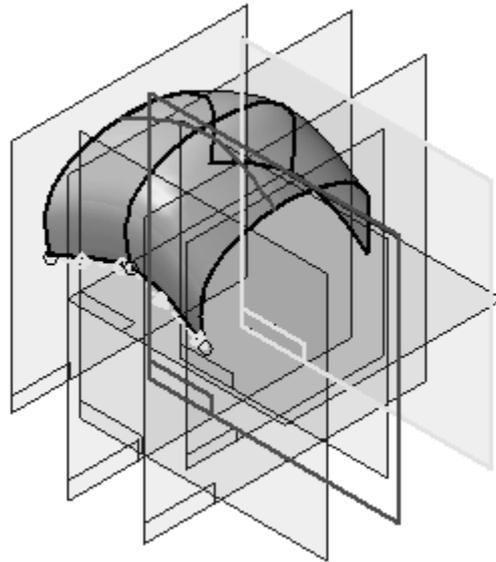
On the BlueSurf ribbon bar, click the Insert Sketch Step. Notice all reference planes are displayed. You must select a plane to insert a sketch on. All of the plane creation methods are available.



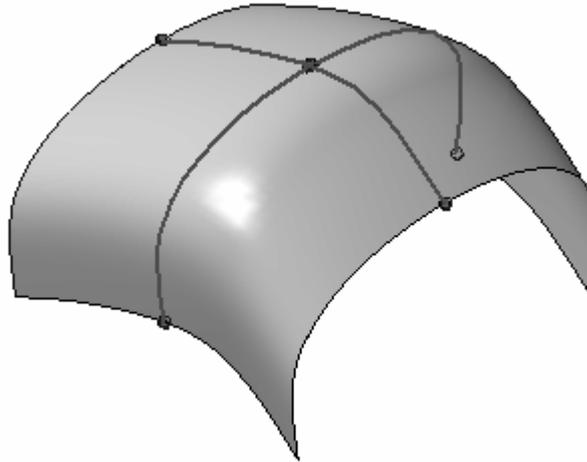
In the following example, the parallel plane option was selected. Reference plane (A) was selected as the plane to be parallel to. Reference plane (B) can be dynamically dragged to the location to insert a sketch. You can also key in a distance. Click location to insert a sketch (C).



Now consider inserting a sketch (C) in the guide curve direction and notice the results. The parallel plane is used again.

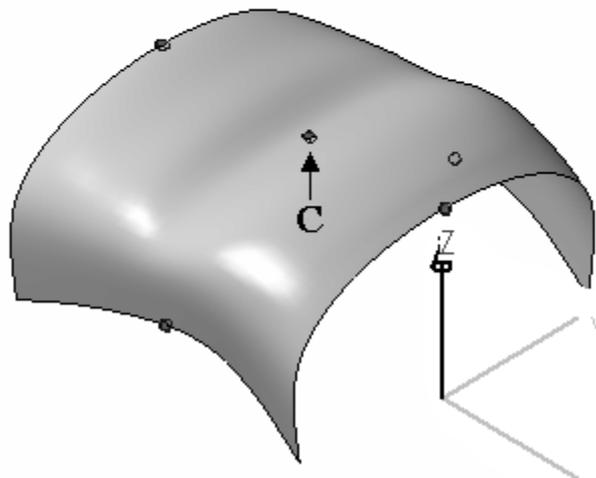
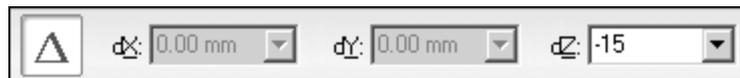


Now turn off the reference planes and observe the results.

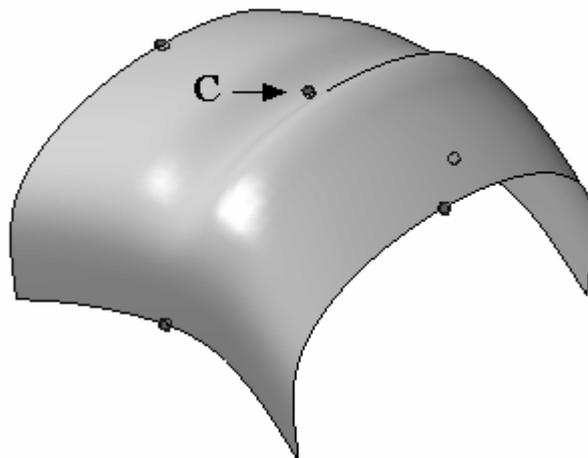
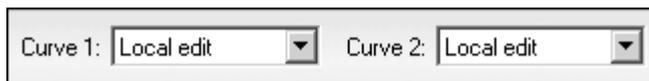


When the guide curve direction sketch was inserted, it crossed another sketch. The BlueSurf command automatically inserts BlueDots at the intersection of the curves. If there were several sketches in the cross-section direction, the inserted sketch in the guide curve direction would be connected with BlueDots to all of the sketches it crosses. The above example now has surface shape control at the BlueDot locations.

The following example shows BlueDot (C) edited with a delta distance of -15 in the Z direction. The two curves connected to the BlueDot were set for Shape edit.



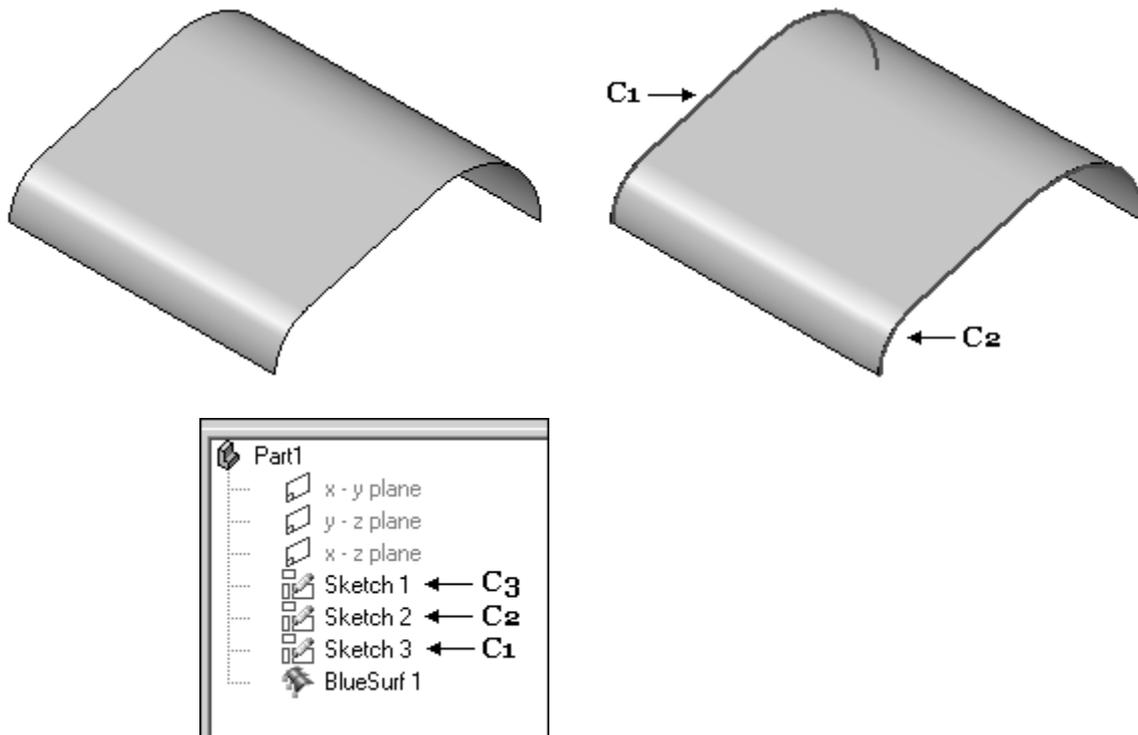
The following example shows BlueDot (C) edited with both curves set to Local edit -5 in the Z direction. Notice the different results.



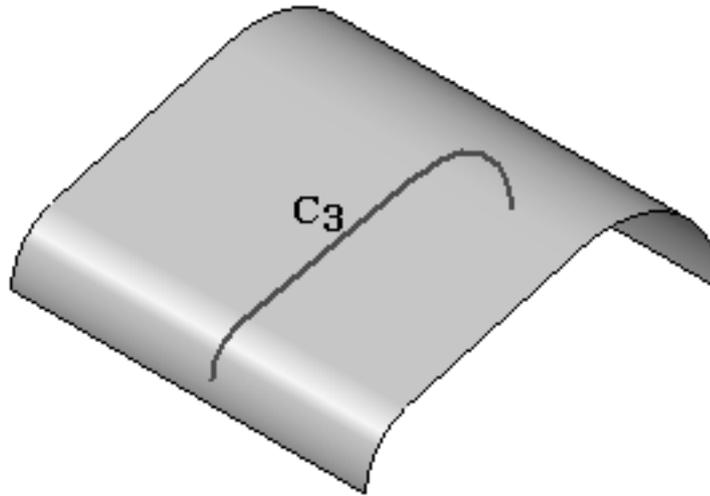
## Adding Additional Cross Sections

You can add additional cross sections to an existing BlueSurf. An added cross section redefines the surface shape definition. The insert sketch command inserts a sketch that follows the shape of the BlueSurf and does not alter the surface shape. Editing an inserted sketch alters the BlueSurf shape. Any cross section sketch created after the BlueSurf is created will not be seen by the BlueSurf feature. When you edit a BlueSurf, it only recognizes sketches created before it was created.

The following is an example of how to add a new cross section. The BlueSurf feature below was created with two cross sections (C1, C2).



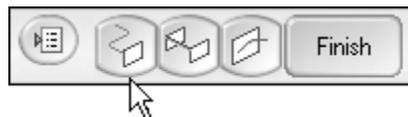
First, add a new cross section (C3) that was created before the BlueSurf feature.



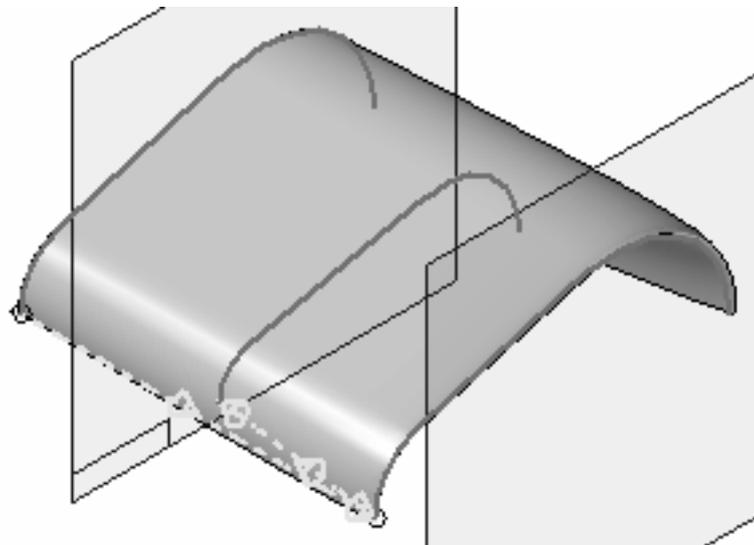
Click the Select Tool and then select the BlueSurf feature. On the ribbon bar, click Edit Definition.



On the BlueSurf ribbon bar, click the Cross Section Step.



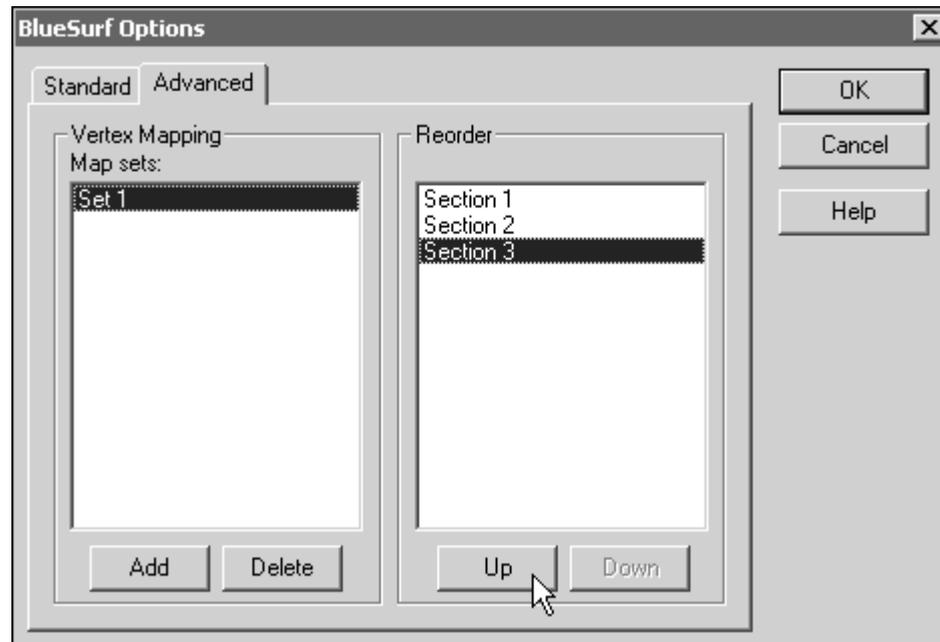
Identify the new cross section (C3). Notice that cross section C3 is placed last in the cross section order, which causes the BlueSurf feature to reverse direction. The cross section order below is C1, C2 and then C3. You can reorder the cross sections to make C3 be defined between C1 and C2.



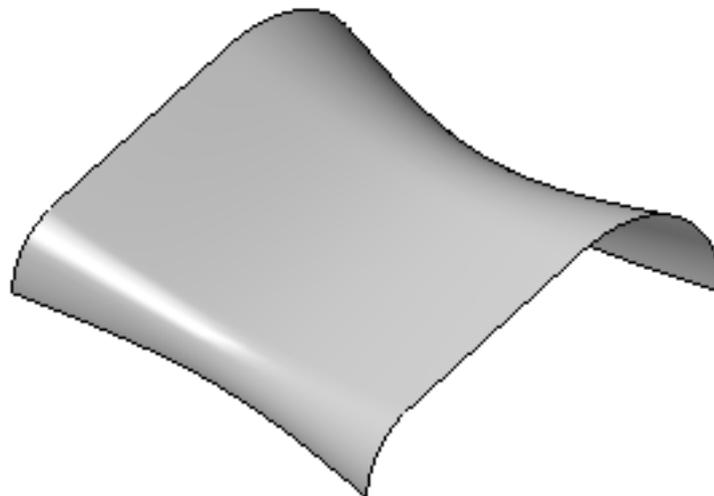
On the BlueSurf ribbon bar, click the options button. Click the Advanced tab.



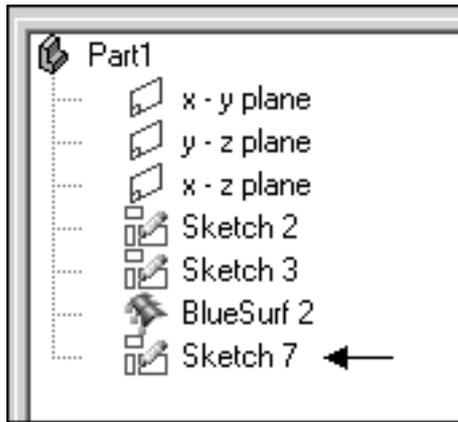
Cross section C3 is shown as Section 3. To reorder C3 to be between C1 and C2, click Section 3 and then click Up. Click OK to apply the reorder.



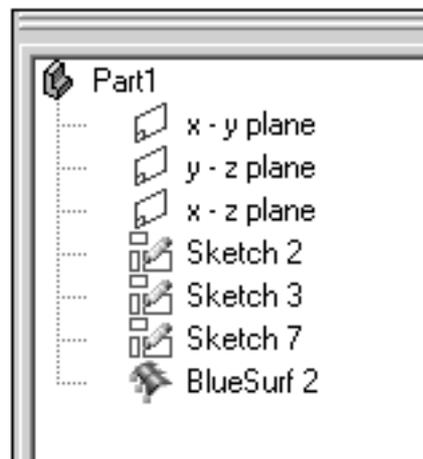
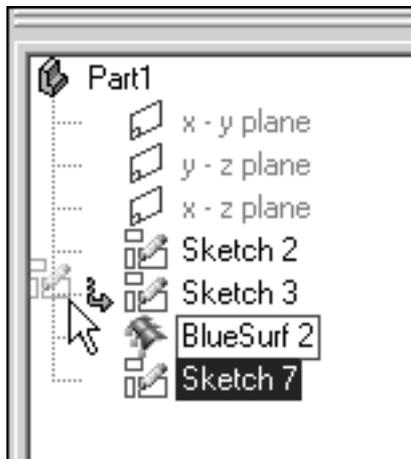
The following shows the result with cross-sections ordered C1, C2 and C3.



If you create a cross section after the BlueSurf feature, 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 EdgeBar, click and hold the Sketch and drag the sketch above the BlueSurf feature as shown.

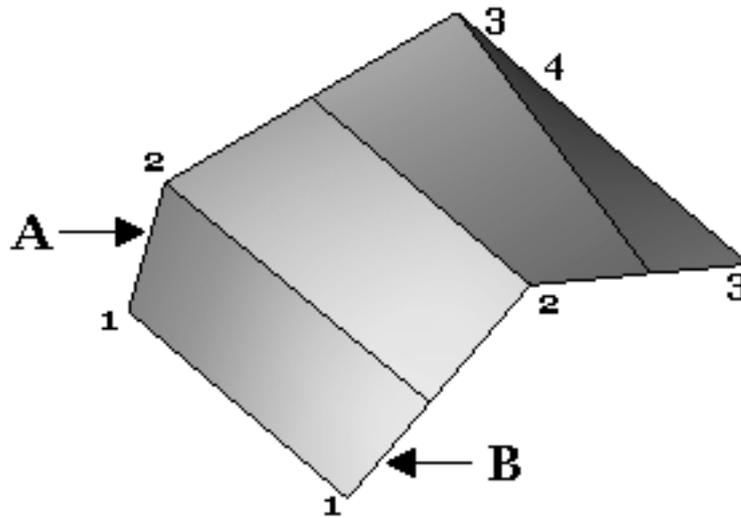


The sketch can now be seen by the BlueSurf feature.

## Vertex Mapping

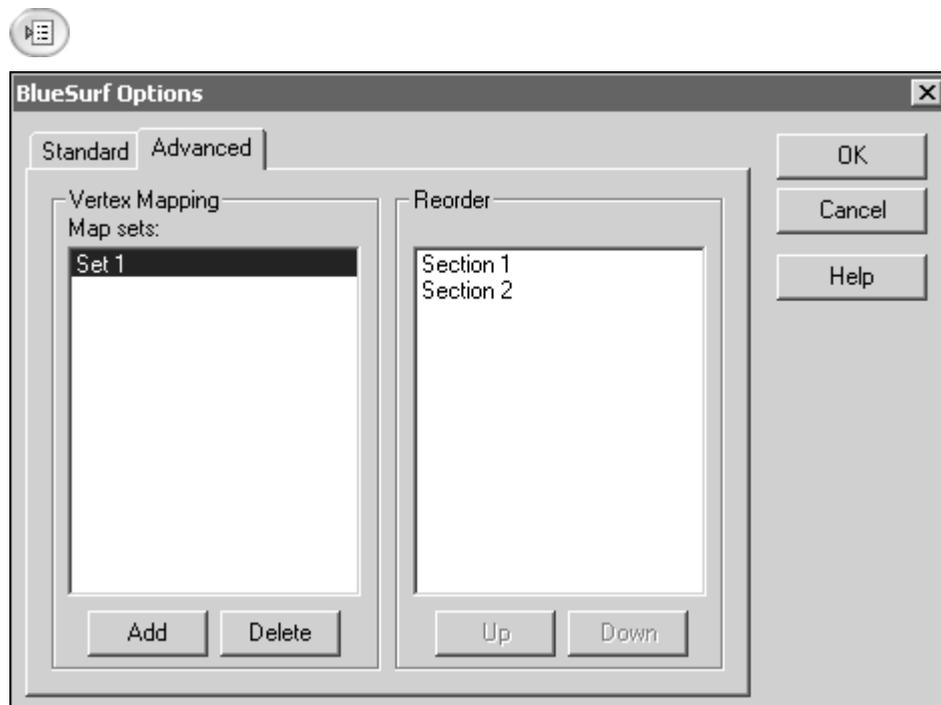
Vertex mapping is a technique to help create flow between section vertices. If there is a vertex count mismatch between sections, equally spaced vertices are used on each section.

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.

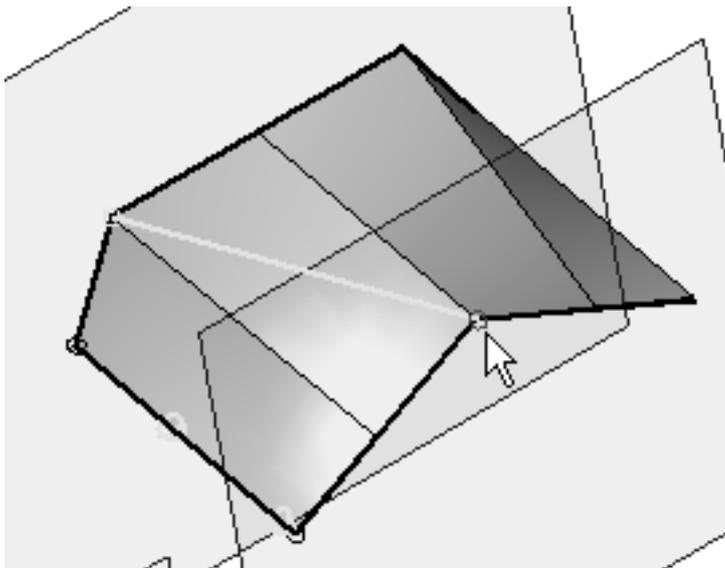


You can add vertex map sets to create a smooth surface flow. You can add vertex maps while creating a BlueSurf or by editing an existing BlueSurf.

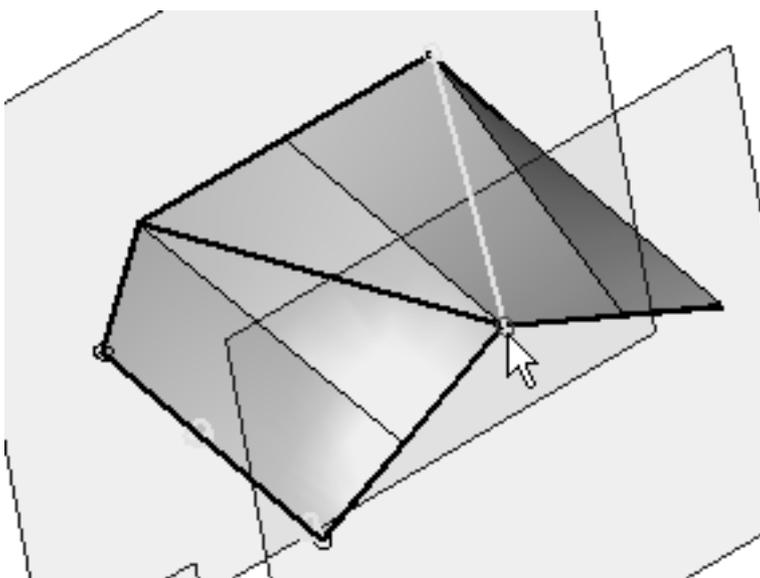
On the BlueSurf ribbon bar, click the options button. On the BlueSurf Options dialog box, click the Advanced tab.



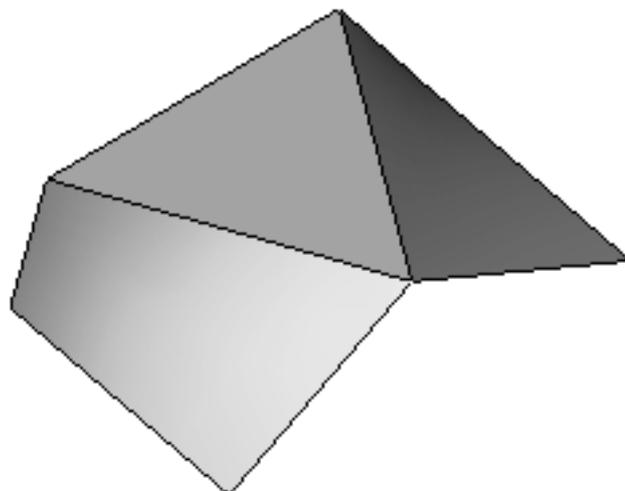
Click Add and then select two vertices to be mapped together as shown.



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



Click OK in the dialog box and then Finish on the ribbon bar. The result is shown below.

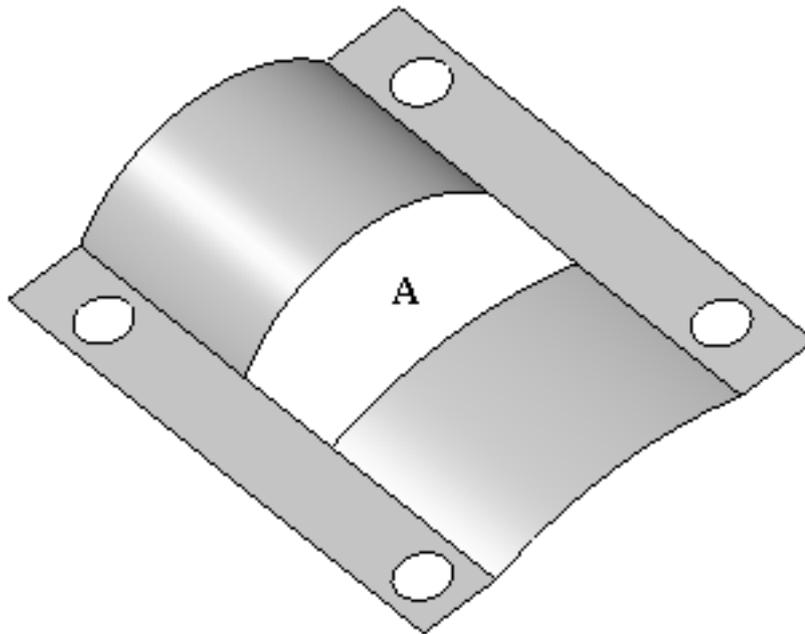


## Bounded Surface

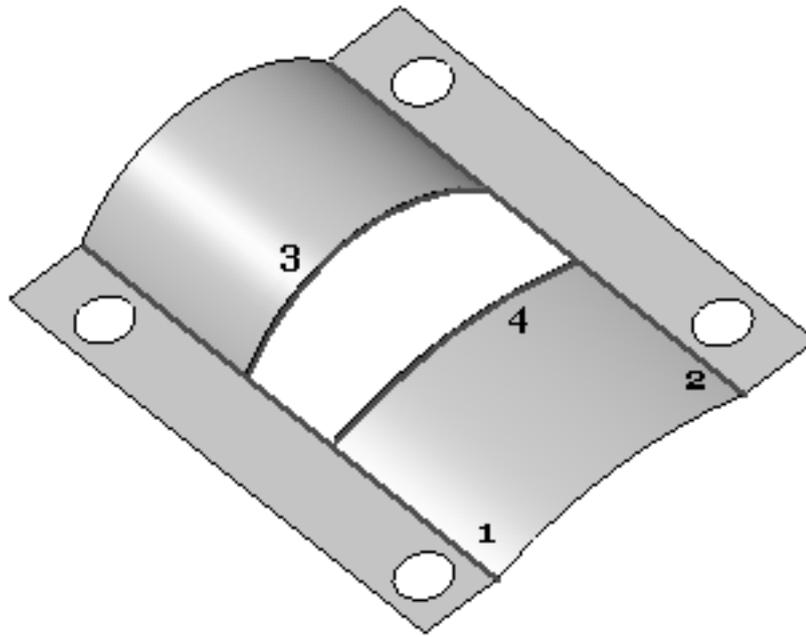


The Bounded Surface command creates a surface between curves or edges. The curve/edge set must form a closed loop. You can also specify whether any adjacent faces are used to control tangency on the new bounded surface. This command is useful when you want to fill in gaps between other adjacent surfaces. In preparing the edges/curves to be used to create a bounded surface, you may need to use the Split Curve and Derived Curve commands. If the edge/curve you need does not exist, you can use the Keypoint Curve command to construct a curve for the bounded surface.

In the following example, you will create a bounded surface in the open area (gap) A.



The image below shows four edges that can be used to create a bounded surface. However, these edges do not form a closed loop. Curves are needed along edges 1 and 2 that are split at the endpoints of edges 3 and 4.



Since an edge cannot be split, you will need to create a derived curve from edges 1 and 2.

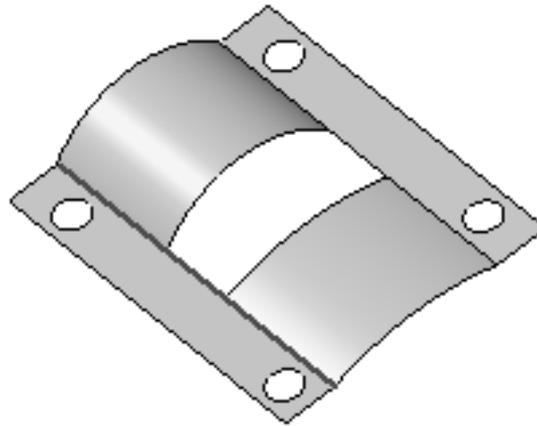
On the Surfacing toolbar, click the Derived Curve command. Select edge 1 and 2, and then click the Accept button. You can now split the derived curve.



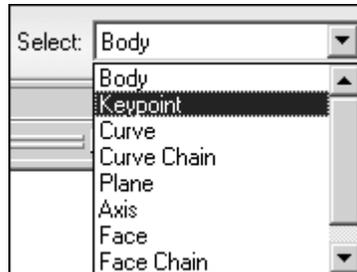
On the command menu, click the Split Curve command.



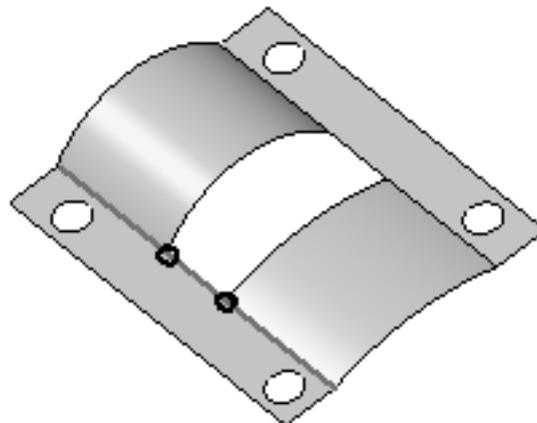
Select the derived curve as shown.



On the Split Curve ribbon bar, set the Select option to Keypoint.



Select the two keypoints shown. Click the accept button and then Finish.

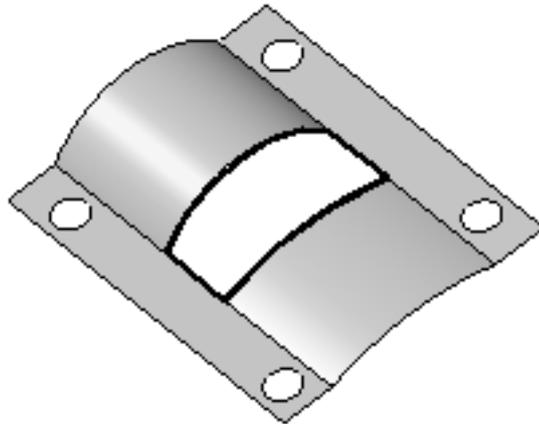


Repeat the process for the edge on the other side. You now have the four curves needed to construct the bounded surface.

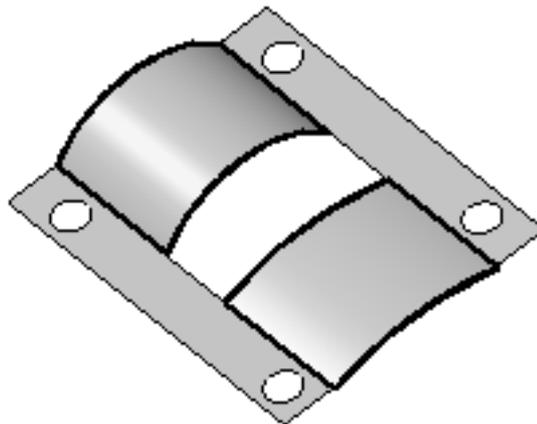
On the command menu, click the Bounded Surface command.



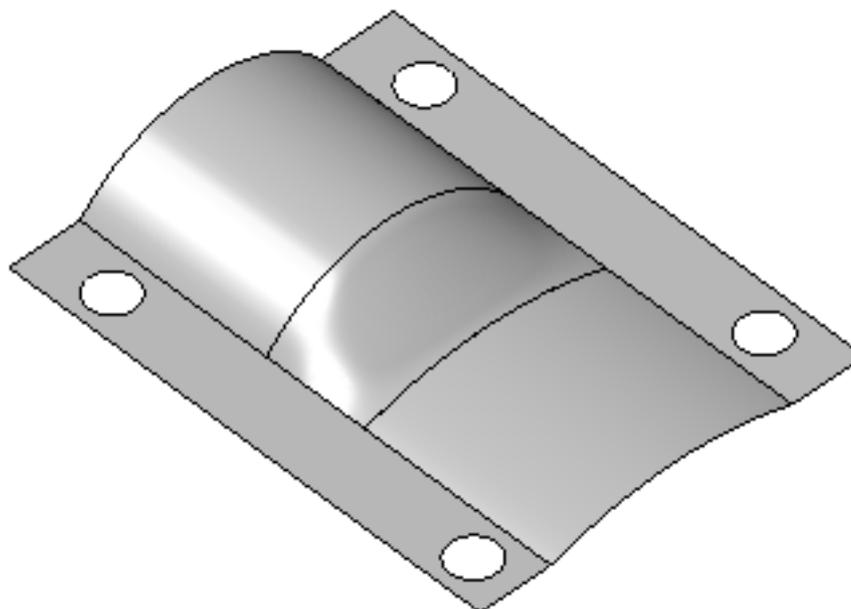
Select the four edges shown and click the accept button.



Notice the two surfaces that are highlighted. These faces will be used as tangent faces. If you do not want the bounded surface to be tangent to these faces, just click on them to remove them from the tangency set.



Click Preview and then Finish.



## **Review**

- What is a cross section used for in creating a surface?
- What are guide curves?
- When do cross sections and guide curves have to be connected?
- Name the two ways to edit a cross section or guide curve?
- How do you add more cross sections to a BlueSurf?
- How do you add more guide curves to a BlueSurf?
- What happens to inserted sketches on a BlueSurf when the BlueSurf is deleted?
- How do you get BlueDots on inserted sketches on a BlueSurf?
- How do you turn off the display of BlueDots?
- On the BlueDot ribbon bar, what does the Relative/Absolute Position option do?



## Activity 1 – Creating and Editing Simple Surfaces

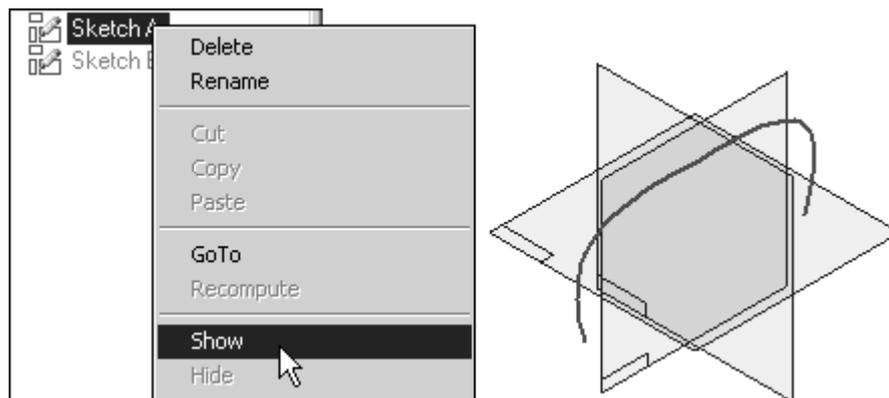
### Overview

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
1. Open SURFACE LAB 3-01.PAR in the C:/Surfacing/Parts folder.
  2. You will now create an extruded surface. In EdgeBar, show Sketch A.



The Extruded Surface command has the same steps as the Protrusion command.

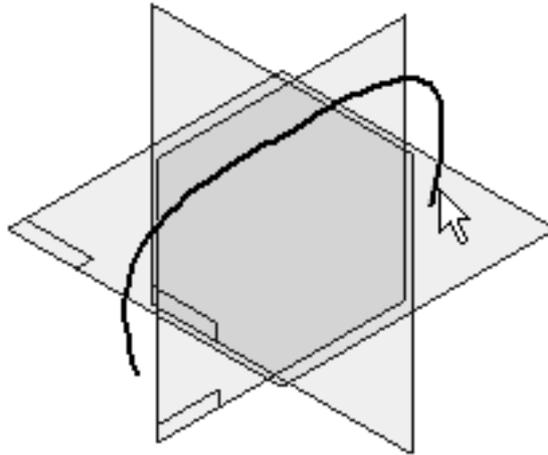
- On Surfacing toolbar, click the Extruded Surface command.



- On the Extruded Surface ribbon bar, click the Select From Sketch button.



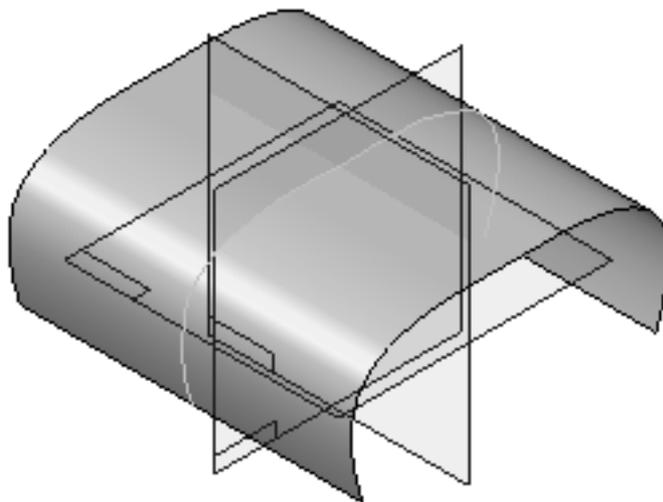
- Click the sketch curve shown below, and then click the Accept button on the ribbon bar.



- On the ribbon bar, click the Symmetric Extent button and type 150 in the Distance field.



- Click Finish.

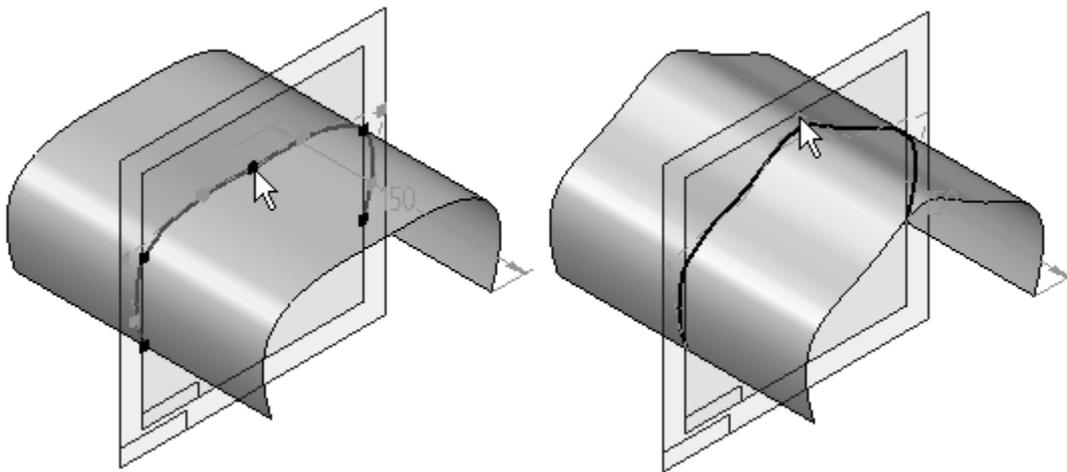


3. You will now modify the shape of the extruded surface by dynamically editing the sketch curve.
  - On the Features toolbar, click the Select tool.
  - Right-click in the part window and click Hide All>reference Planes.

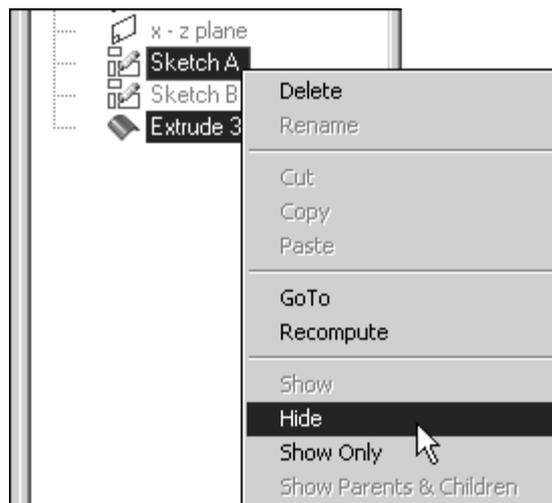
- Select the extruded surface and on the Select Tool ribbon bar, click the Dynamic Edit button.



- 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.



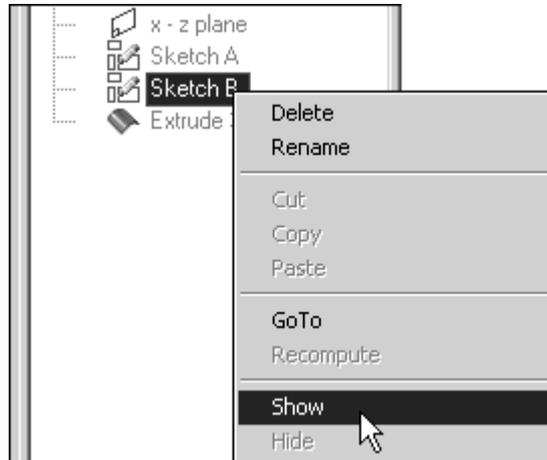
- Click the Select Tool to end the dynamic edit.
- In EdgeBar, hide the features shown.



- 4. You will now create a revolved surface. In EdgeBar, show Sketch B.



The Revolved Surface command has the same steps as the Revolved Protrusion command.



- On the Surfacing toolbar, click the Revolved Surface command. The command is located on the fly out of the Extruded Surface command.



- On the Revolved Surface ribbon bar, click the Select From Sketch option.



- Select the sketch curve shown below and then click the Accept button on the ribbon bar.



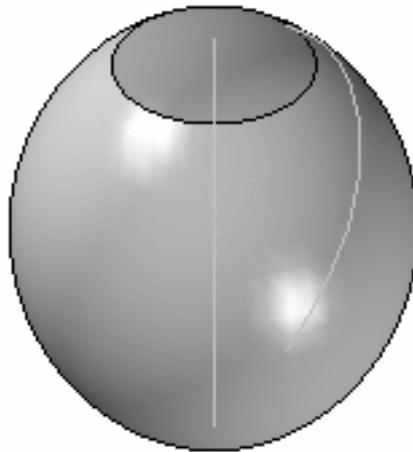
- Notice on the Revolved Surface ribbon bar that the next step is to define the axis of revolution. Click the line as shown.



- On the ribbon bar, for the extent step click the revolve 360 degrees button.



- On the ribbon bar, click Finish.

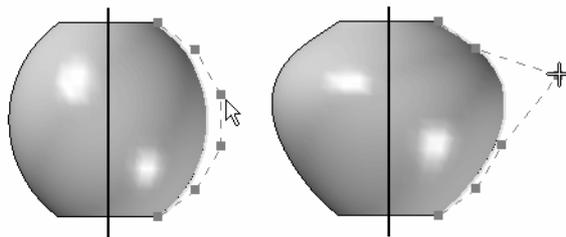


5. You will now edit the shape of the revolved surface by dynamically dragging an edit point on the sketch curve.
  - On the Features toolbar, click the Select tool.
  - Press Ctrl-R to rotate the view to a right view.

- Select the revolved surface and on the Select Tool ribbon bar, click the Dynamic Edit button.



- Select 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.



- Drag the control vertices around to come up with your own surface shape.
6. This concludes the activity. Exit and save the file.

## Activity 2 – Creating a Swept Surface

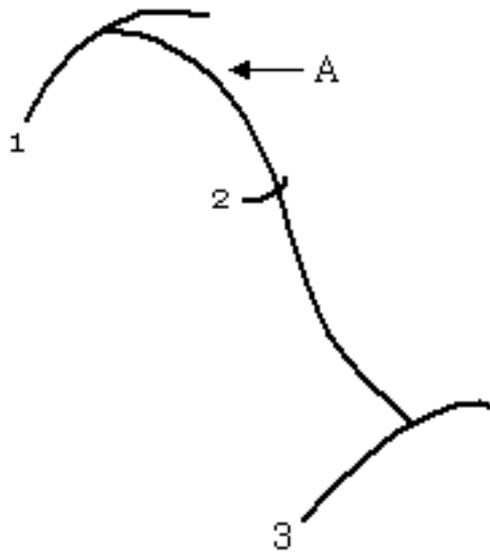
### Overview

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
1. Open SURFACE LAB 3-02.PAR in the C:/Surfacing/Parts folder. The part file contains four sketches. Sketch A is the guide path (curve) and sketches 1-3 are the cross sections (arcs).

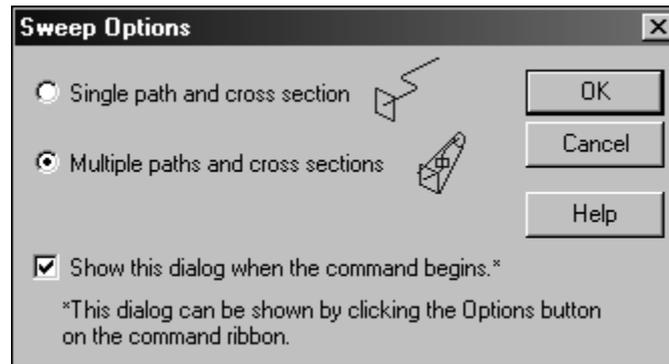


The Swept Surface command has the same steps as the Swept Protrusion command.

2. You will now create a swept surface using the sketches provided.
  - On the Surfacing toolbar, click the Swept Surface command.



- On the Sweep Options dialog box, click the Multiple paths and cross sections button. Click OK.



- For the Path step, select the curve shown and then click the Accept button on the ribbon bar.



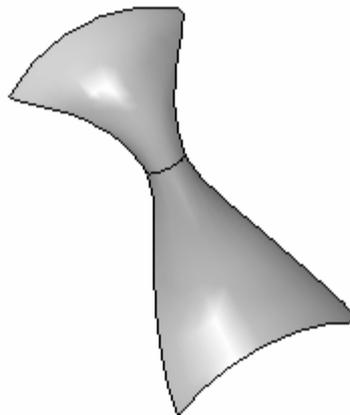
- Since there is only one path, click the Next button on the ribbon bar to proceed to the Cross Section step.



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



- On the Swept Surface ribbon bar, click Preview and then click Finish.

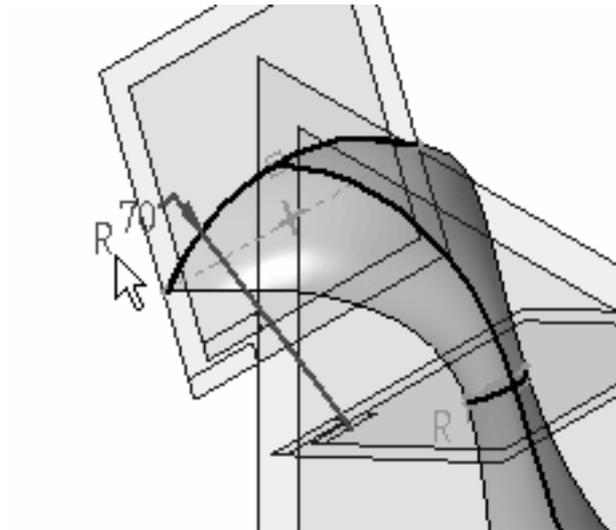


3. The swept surface is complete. You will now edit the cross sections to modify the shape of the surface.

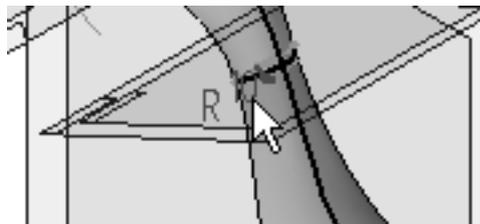
- Click the Select Tool. Select the surface and on the Select Tool ribbon bar, click Dynamic Edit.



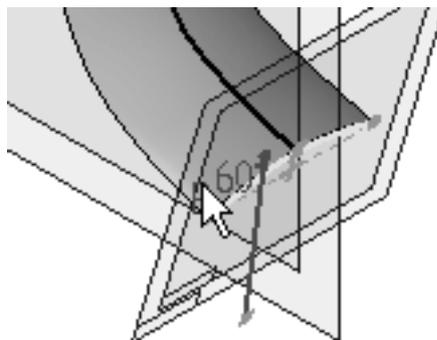
- Click the 70 mm radius dimension on cross section 1 and on the ribbon bar, type 50 and press enter on the keyboard.



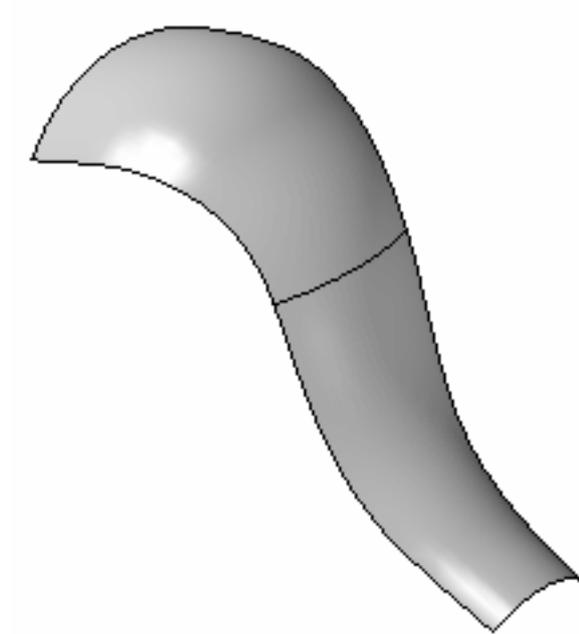
- Click the 10 mm radius dimension on cross section 2 and on the ribbon bar, type 40 and press enter on the keyboard.



- Click the 60 mm radius dimension on cross section 3 and on the ribbon bar, type 20 and press enter on the keyboard.



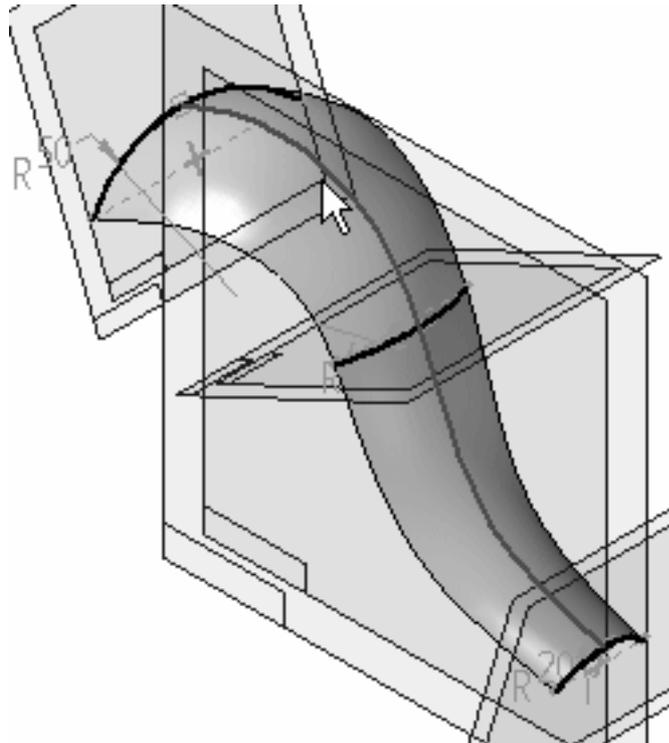
- Click the Select tool to end the dynamic edit.



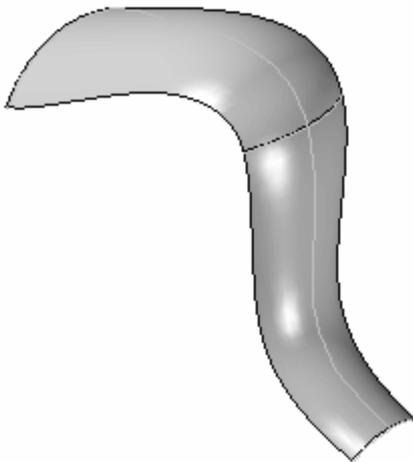
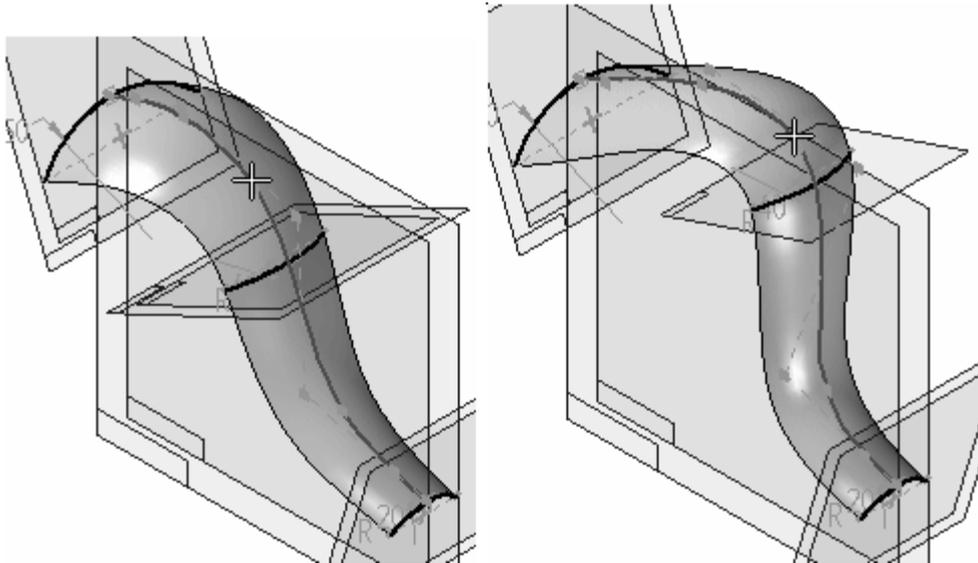
4. You will now dynamically edit the path curve.
  - Click the Select Tool. Select the surface and on the Select Tool ribbon bar, click Dynamic Edit.



- Click the path curve as shown.



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



5. The activity is complete.



## Activity 3 – Creating a BlueSurf Using Analytics

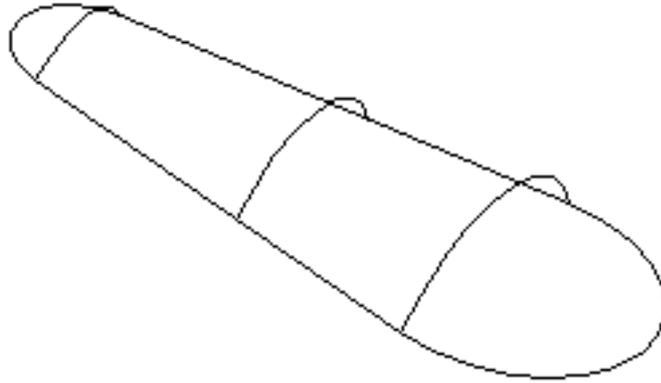
### Overview

In this activity, you will 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
1. Open SURFACE LAB 3-03.PAR in the C:/Surfacing/Parts folder.



2. You will create several BlueSurf features using the provided sketches. The sketches contain all analytic elements (arcs and lines). You will not edit the surface in this activity.

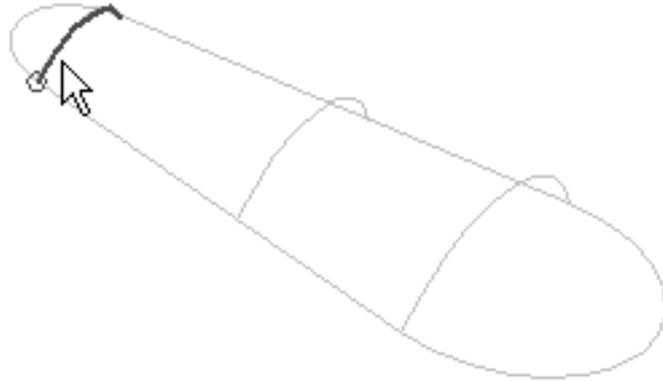
- On the Surfacing toolbar, click the BlueSurf command.



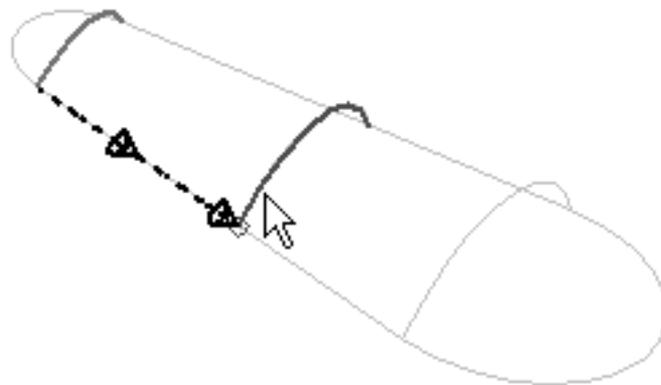
- Notice on the BlueSurf ribbon bar that you are in the Cross Section Step.



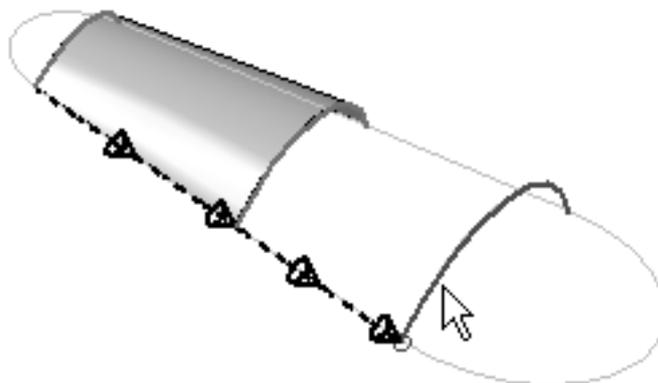
- To define the first cross section, click the arc shown and then right-click. You can also click the Accept button on the BlueSurf ribbon bar.



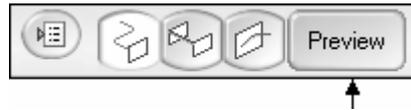
- For the next cross section, click the arc shown and then right-click.



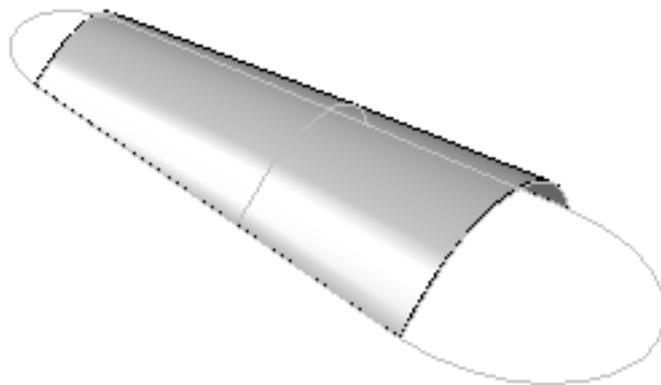
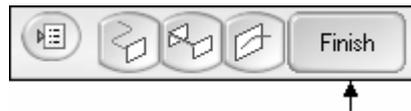
- For the last cross section, click the arc shown and then right-click.



- On the ribbon bar, click Preview.



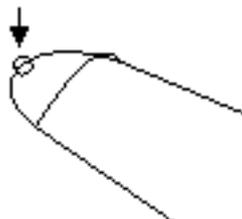
- On the ribbon bar, click Finish.



Notice the BlueSurf feature shown in EdgeBar.



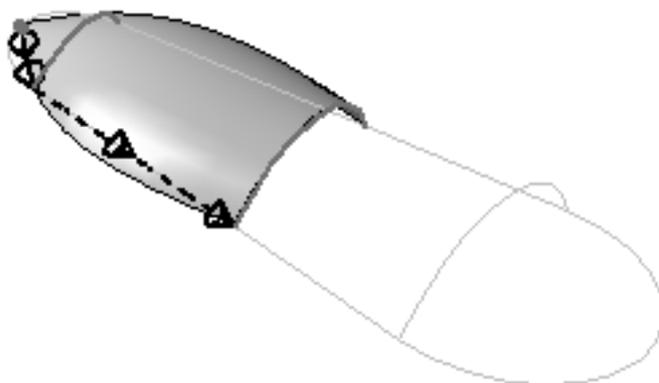
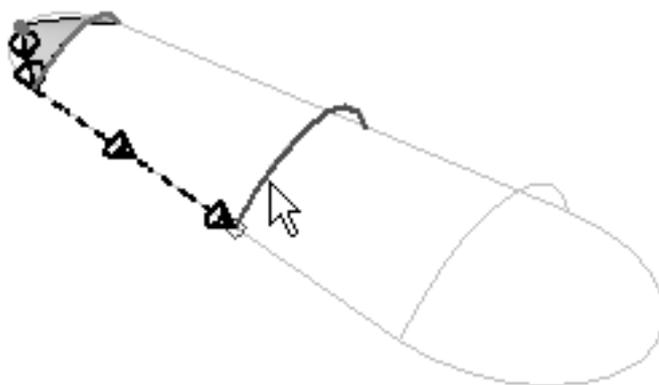
- Delete the BlueSurf feature.
3. You will now create another BlueSurf using guide curves.
- Click the BlueSurf command.
  - For the first cross section, select the point shown.



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



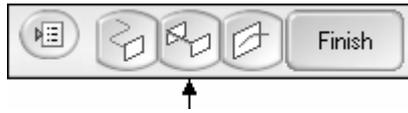
- Click the arc shown for the last cross section and then right-click.



- On the ribbon bar, click Preview. Do not click Finish.



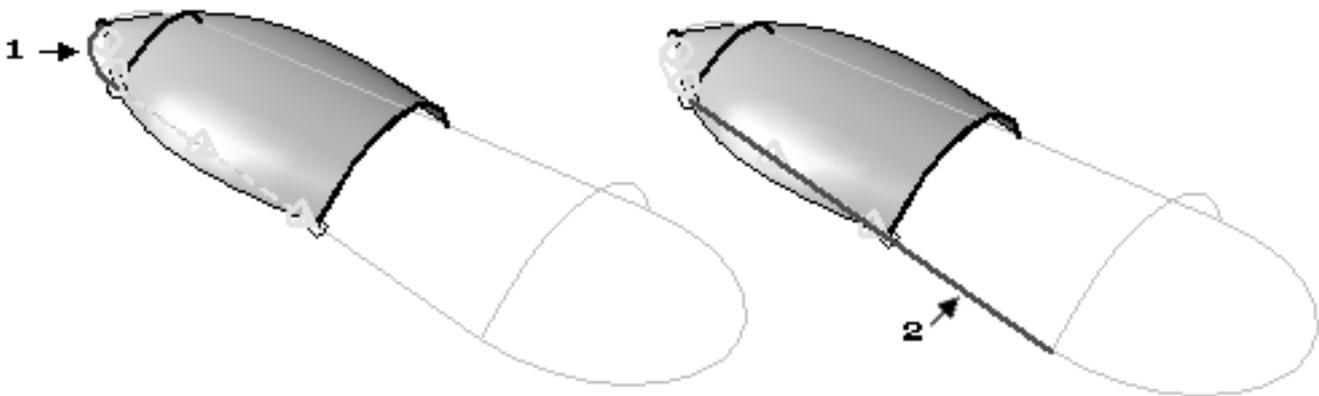
- You will now apply guide curves to the BlueSurf. Click the Guide Curve Step on the ribbon bar.



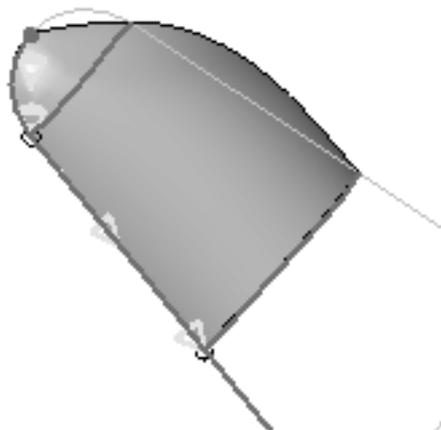
- In the Select drop down list, click Single. This allows you to select single sketch elements for the guide curve.



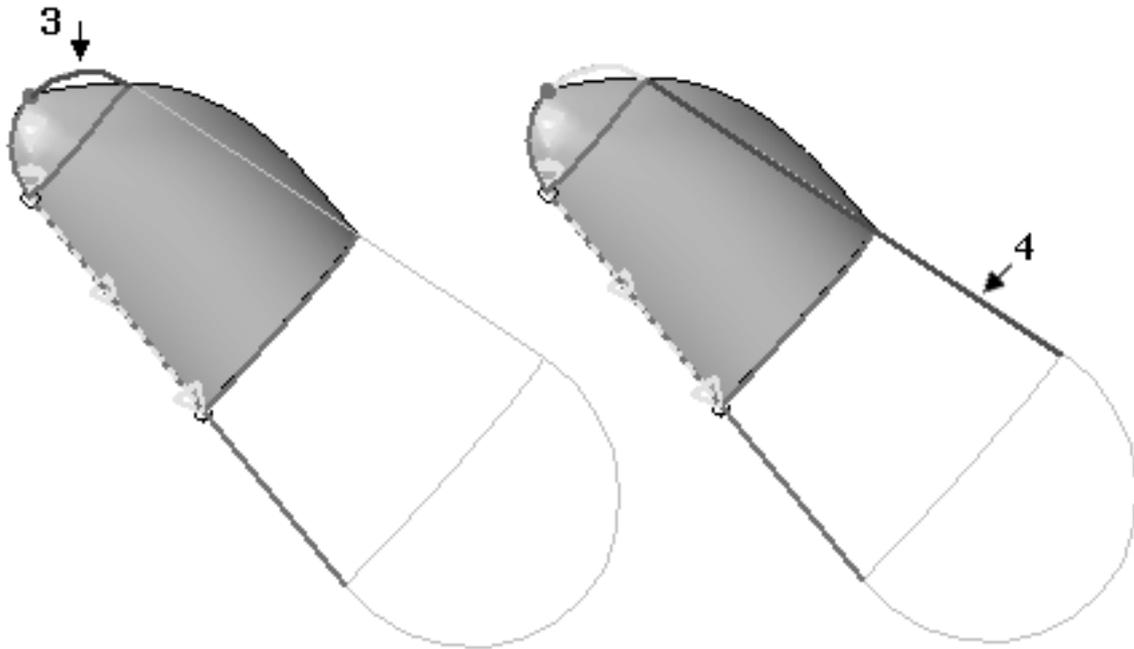
- Select sketch elements 1 and 2 as shown and then right-click to complete the first guide curve.



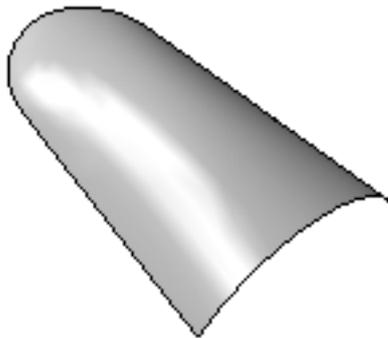
Notice how the BlueSurf follows the guide curve.



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



- On the ribbon bar, click Preview and then Finish.



- Delete the BlueSurf feature.
4. You will now create a BlueSurf with two points, a cross section and two guide curves.
- Click the BlueSurf command.

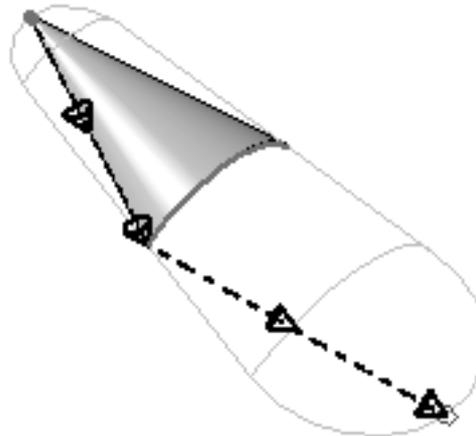
- Click the point shown.



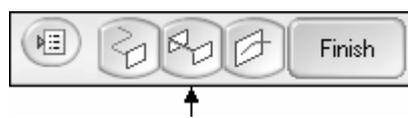
- Click the cross section shown and right-click.



- Click the point shown.



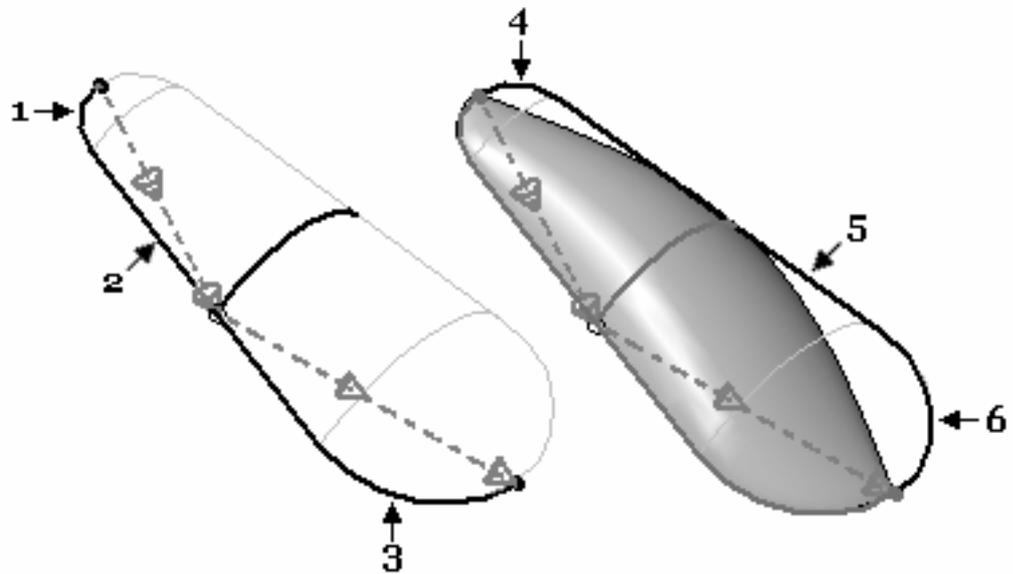
- On the ribbon bar, click the Guide Curve Step.



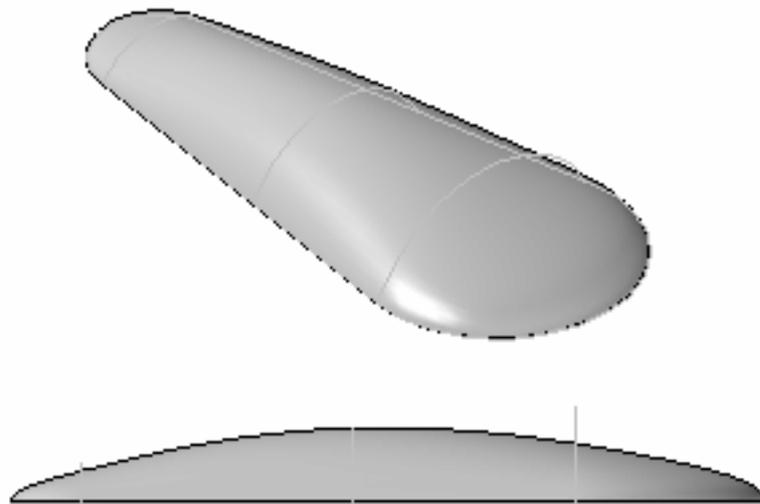
- In the Select drop down list, click Single. This allows you to select single sketch elements for the guide curve.



- Select sketch elements 1 –3 as shown and then right-click. Select sketch elements 4-6 as shown and then right-click.



- On the ribbon bar, click Preview and then click Finish.

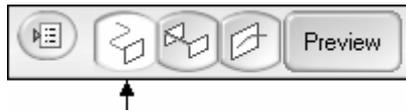


5. You will now add cross sections to BlueSurf just created.

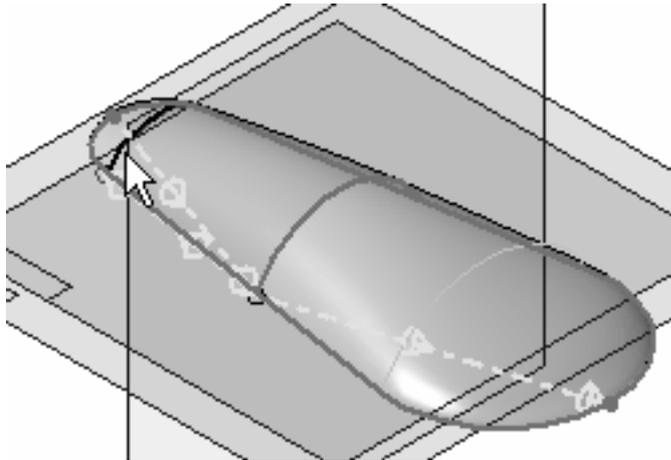
- Click the Select tool and then select the BlueSurf feature.
- On the Select tool ribbon bar, click Edit Definition.



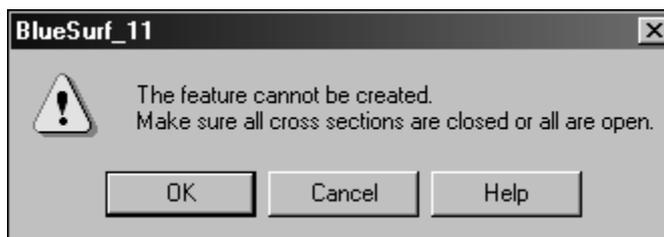
- On the BlueSurf ribbon bar, click the Cross section Step.



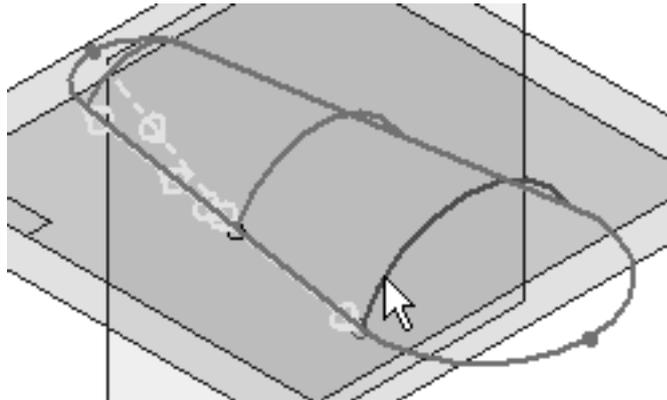
- Click the cross section shown and right-click.



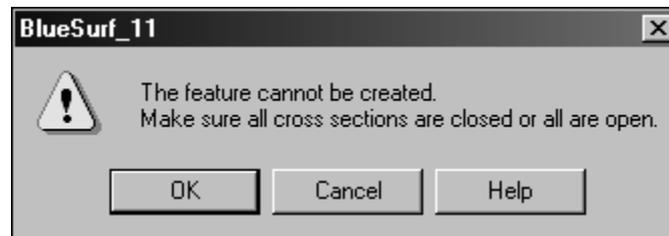
- Click OK on the error dialog box. The cross section order is the cause for the error. You will correct this later.



- Click the cross section shown and right-click.



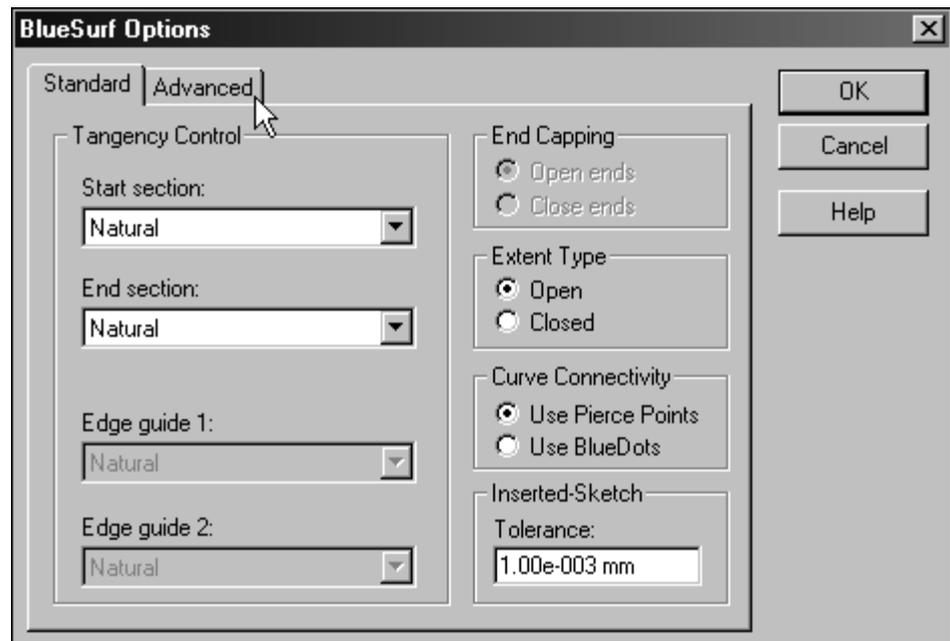
- Click OK on the error dialog box. The cross section order is the cause for the error. You will correct this later.



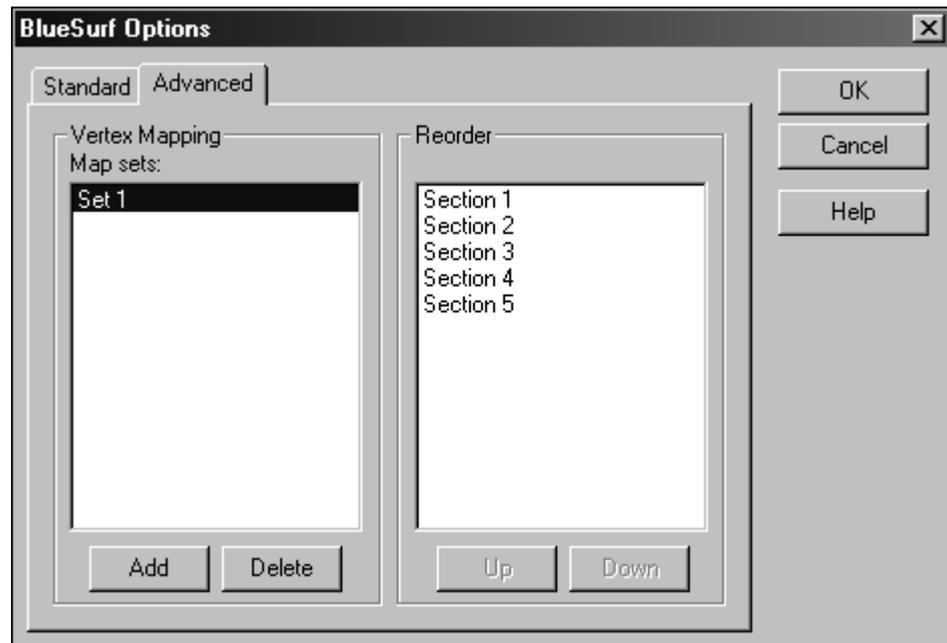
- On the ribbon bar, click the BlueSurf Options button.



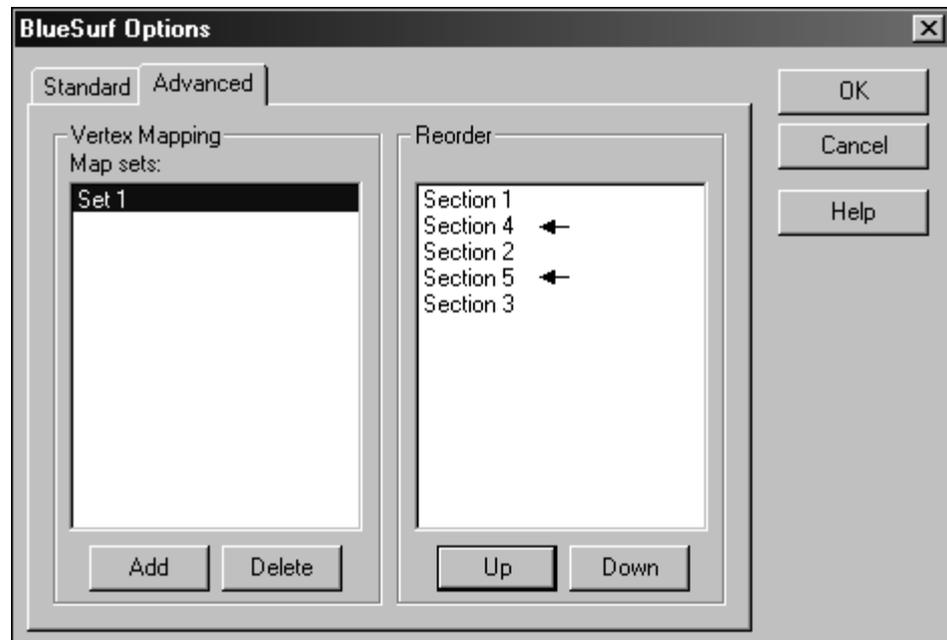
- Click the Advanced tab on the BlueSurf Options dialog box.



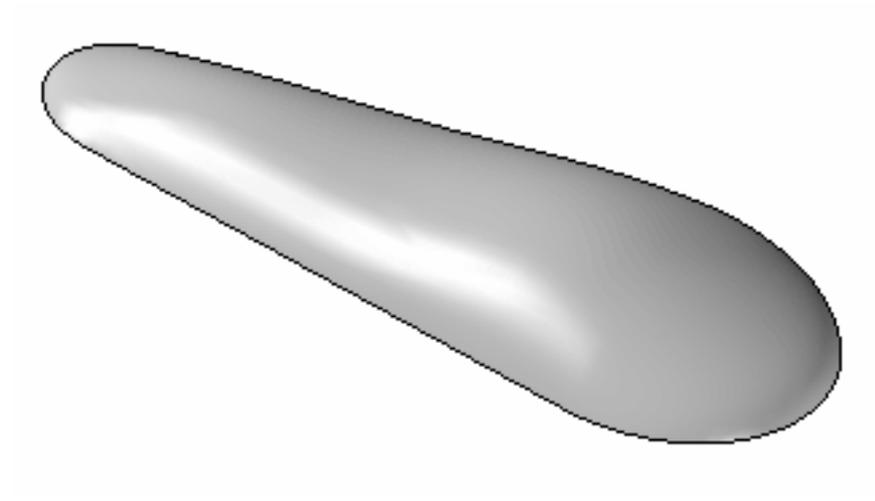
- Notice the order of the Sections in the dialog box. You can pause your cursor over a section and it will highlight in the part window.



- 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.



- On the ribbon bar, click Finish.



6. The activity is complete.

## Activity 4 – Creating and Editing a BlueSurf

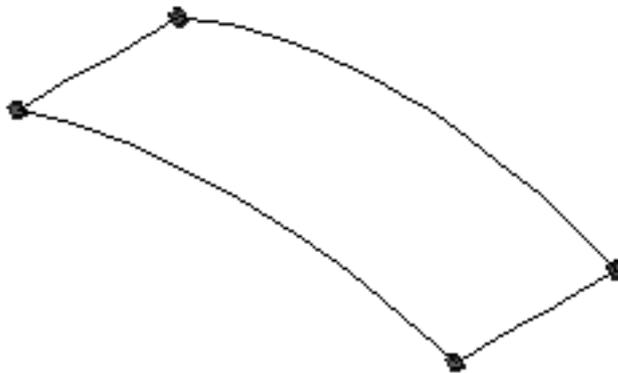
### Overview

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
1. Open SURFACE LAB 3-04.PAR in the C:/Surfacing/Parts folder.

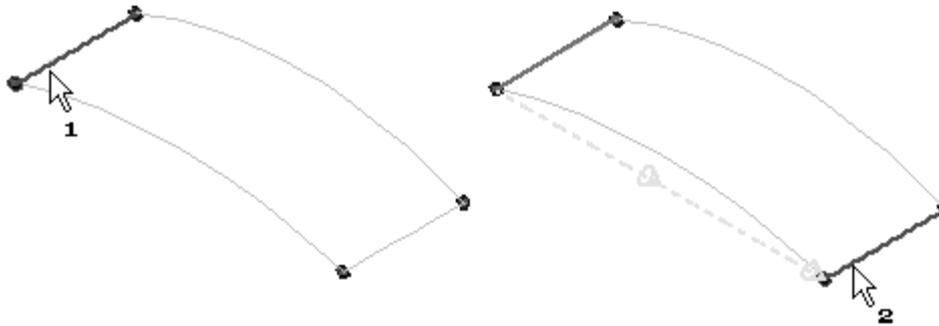


The part file contains four curves that are BlueDot connected.

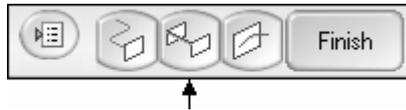
2. You will now create a BlueSurf with guides using the curves provided.
  - On the Surfacing toolbar, click the BlueSurf command.



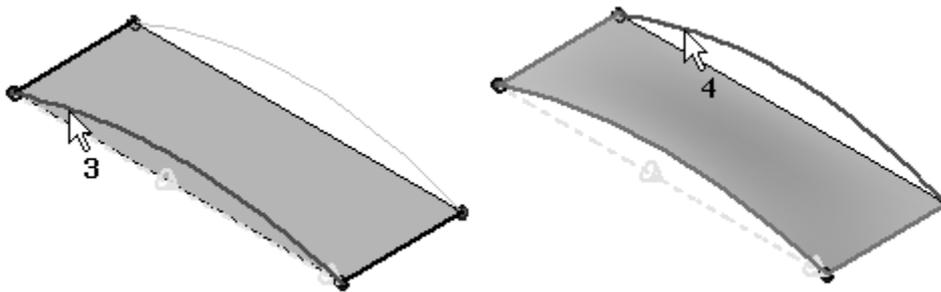
- Click curve 1 as shown for the first cross section and then right-click. Click curve 2 as shown for the second cross section and then right-click.



- On the BlueSurf ribbon bar, 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.



- On the ribbon bar click Preview and then click Finish.



3. You will now insert sketches on the BlueSurf to provide additional shape control.
  - Click the Select tool and then select the BlueSurf feature.

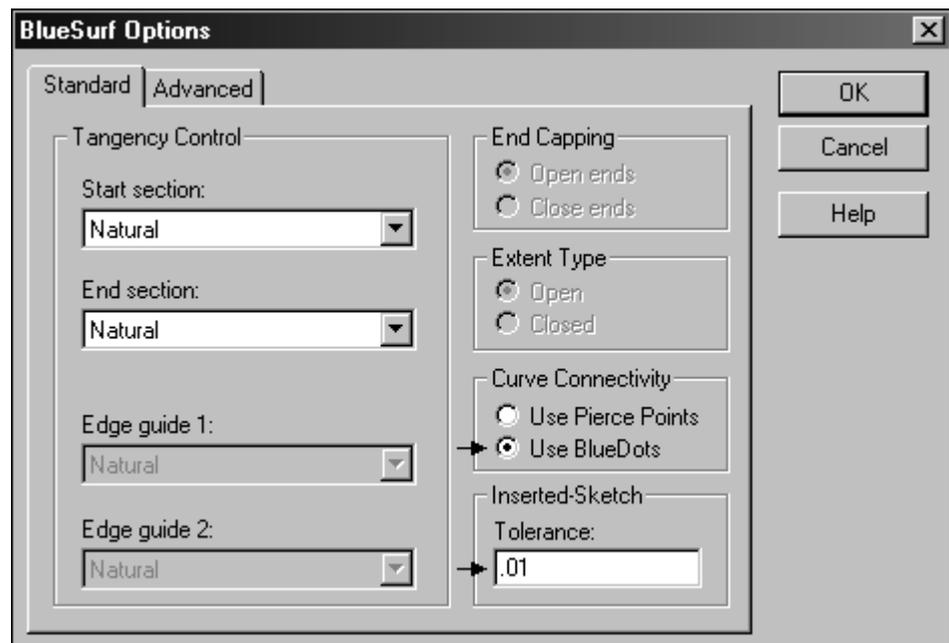
- On the Select tool ribbon bar, click Edit Definition.



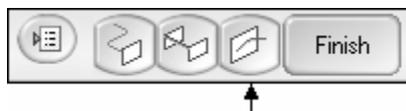
- On the BlueSurf ribbon bar, click the Options button.



- On the BlueSurf Options dialog box, under Curve Connectivity click Use BlueDots. Under Inserted-Sketch type .01. The tolerance controls the number of edit points used on inserted sketches. Click OK.



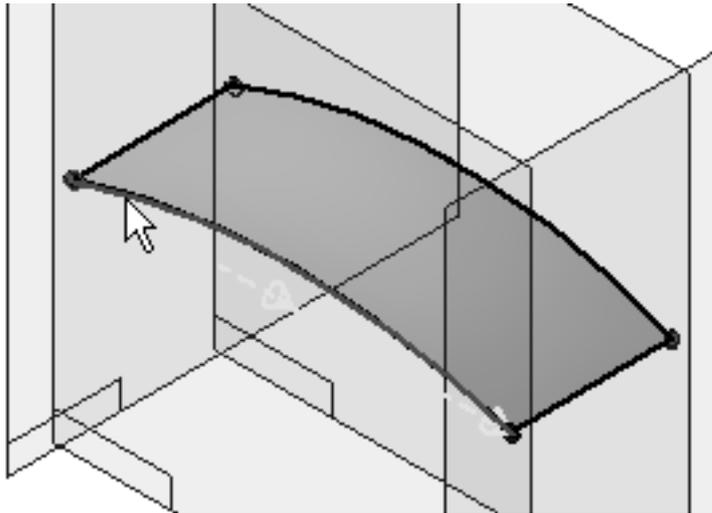
- On the ribbon bar, click the Insert Sketch Step button.



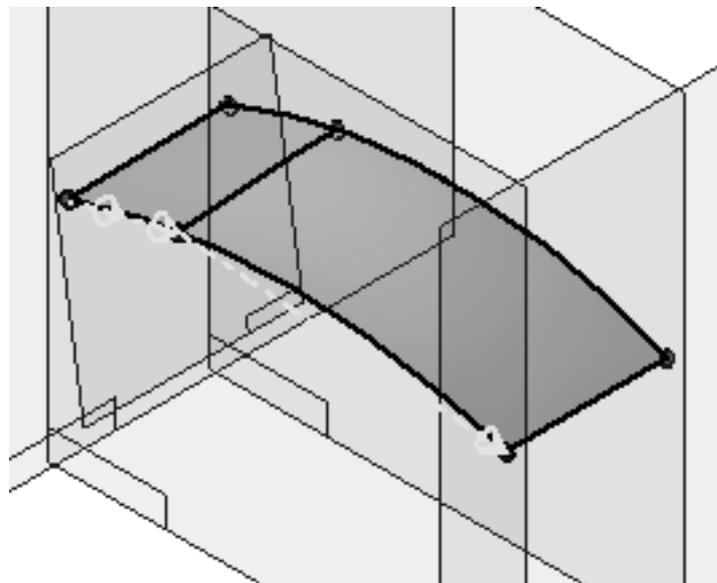
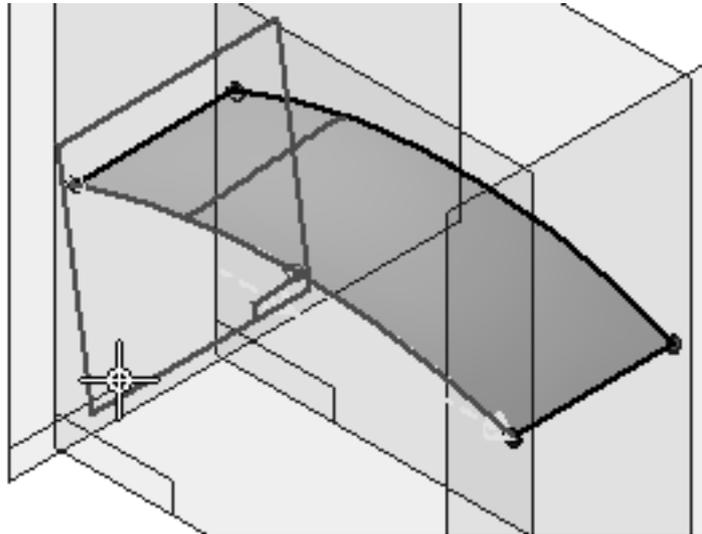
- On the ribbon bar, click the Plane Normal to Curve option.



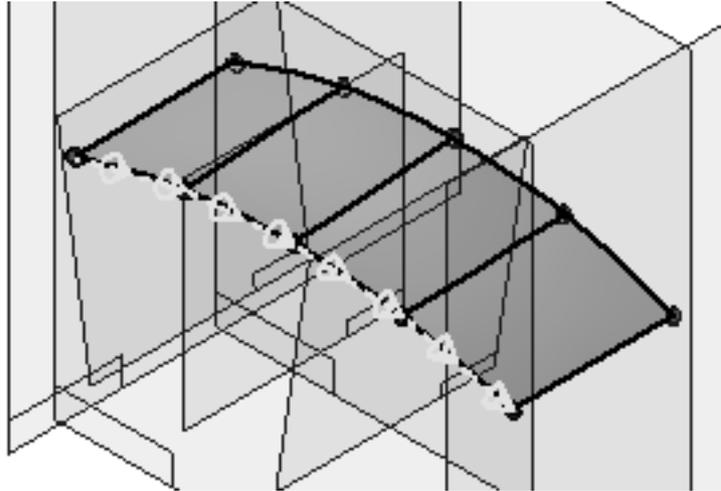
- Click the curve shown.



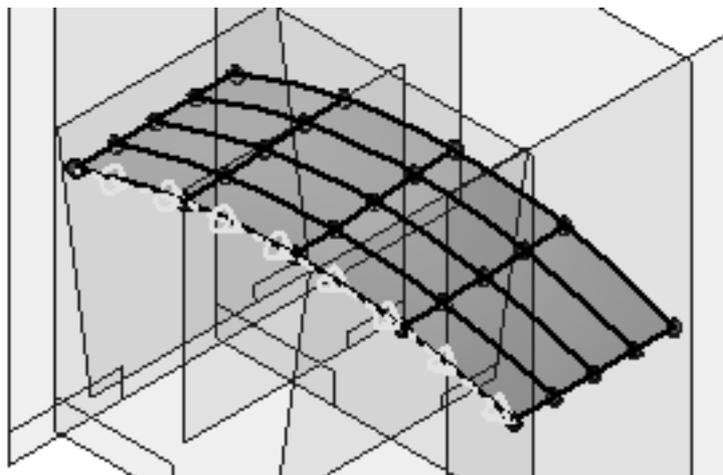
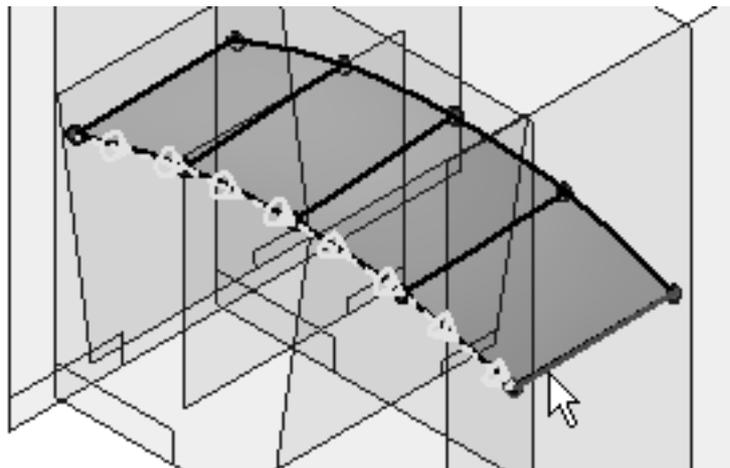
- Notice a normal plane is attached to the curve, which can be dynamically dragged along the curve. Drag the plane until the Position value is 0.25. You can also enter .25 on the ribbon bar. Click to place the plane.



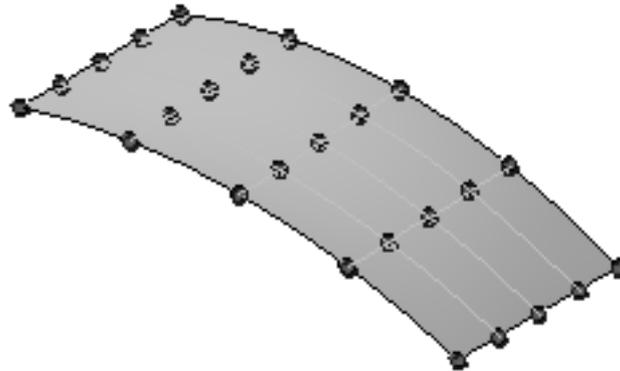
- Repeat the previous step to insert sketches at positions .50 and .75 as shown.



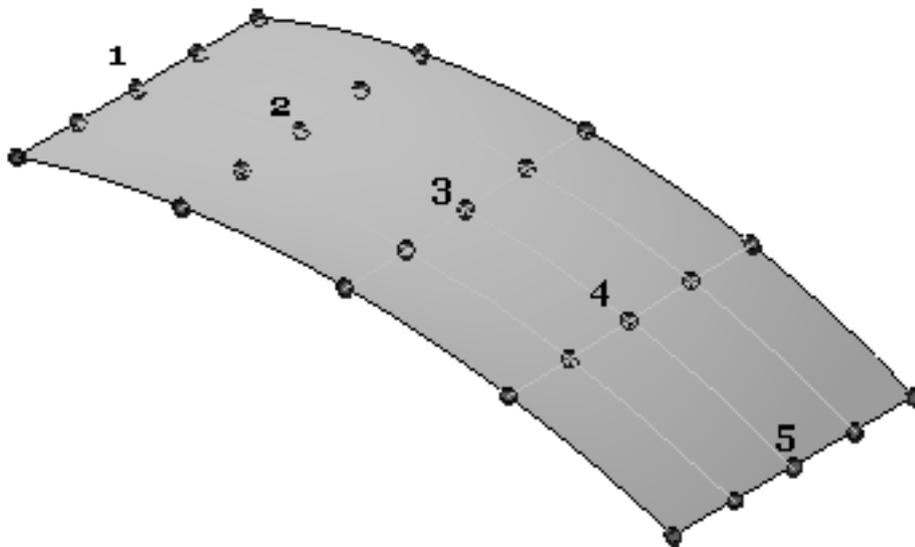
- You will now insert sketches normal to the cross section curve. Click the curve shown and insert sketches at positions .25, .50 and .75.



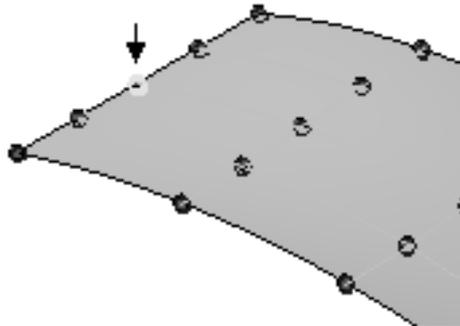
- On the ribbon bar, click Finish and then click Finish again.



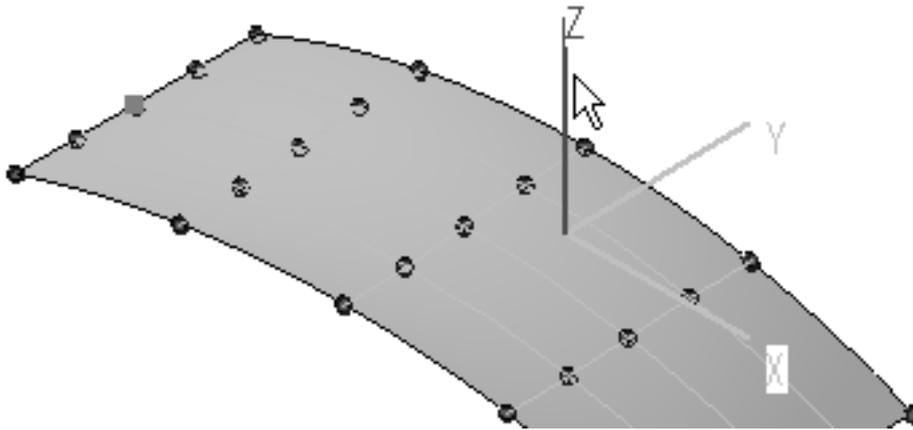
4. You will now perform BlueDot edits to change the shape of the surface. You will edit the BlueDots along the center by changing their position in the Z-direction.
  - You will edit BlueDots 1 and 5 with a delta distance of 5. You will edit BlueDots 2-4 with a delta distance of 10.



- Click the Select tool. Select BlueDot shown below and on the Select tool ribbon bar, click Dynamic Edit.



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



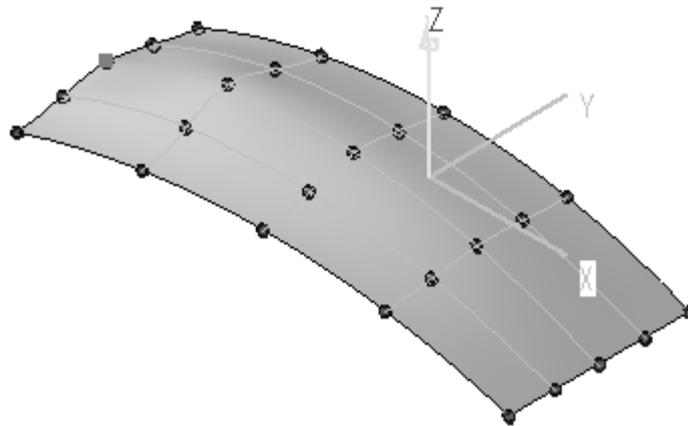
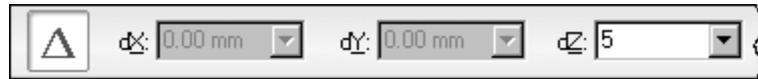
- On the ribbon bar, click the Relative/Absolute Position button.



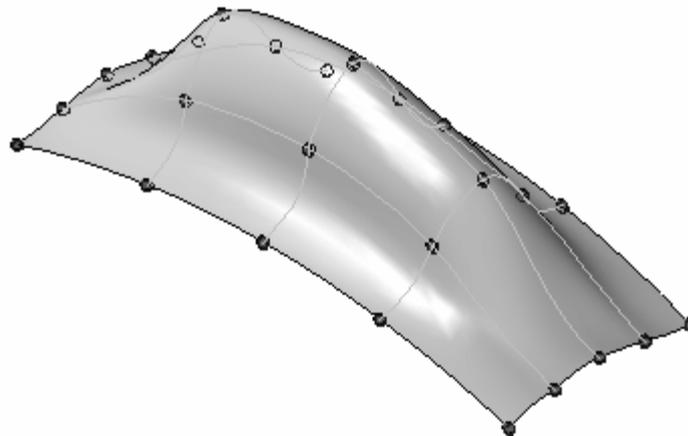
- In the dZ: field, type 5. Make sure Curves 1 and 2 are set to Shape edit. Press the Enter key on the keyboard.



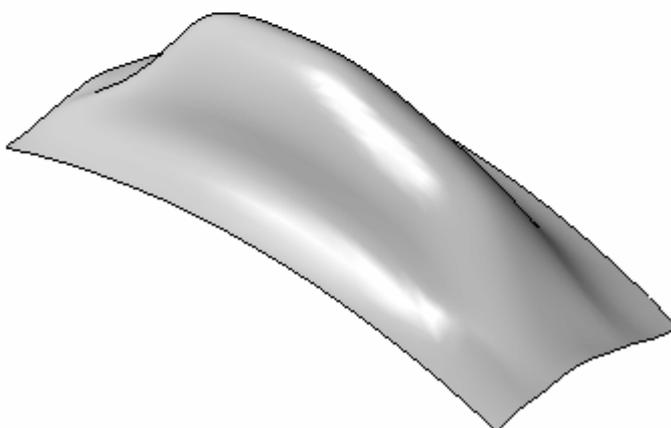
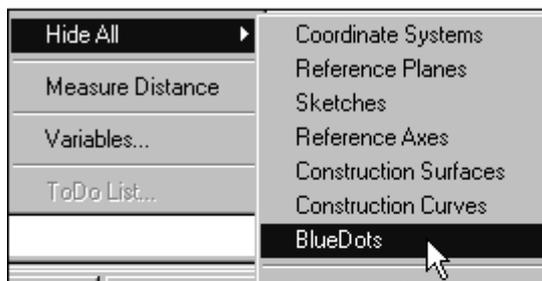
If you press Enter again, the value will be applied again.



- Click in an open space in the part window to exit the BlueDot edit.
- Repeat the previous step to edit BlueDots 2-5.



- Right-click in the part window. Turn off the display of BlueDots and sketches. The activity is complete.



## Summary

You control surfaces by curve definitions. You change the surface shape by editing the underlying curves. Edit curves by using Dynamic Edit or by editing the curve sketch/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 additional 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.



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## Chapter

# 4 *Additional Surfacing Commands*

## Objectives

After completing this module, you will be able to:

- Learn the use of the surface manipulation commands.

## Overview of Additional Surfacing Commands

This module covers the remaining surfacing commands available in Solid Edge. These commands are all used to manipulate existing construction surfaces.

Extend Surface

Offset Surface

Copy Surface

Trim Surface

Delete Faces

Stitched Surface

Round

Replace Face

Parting Split

Parting Surface

Split Face

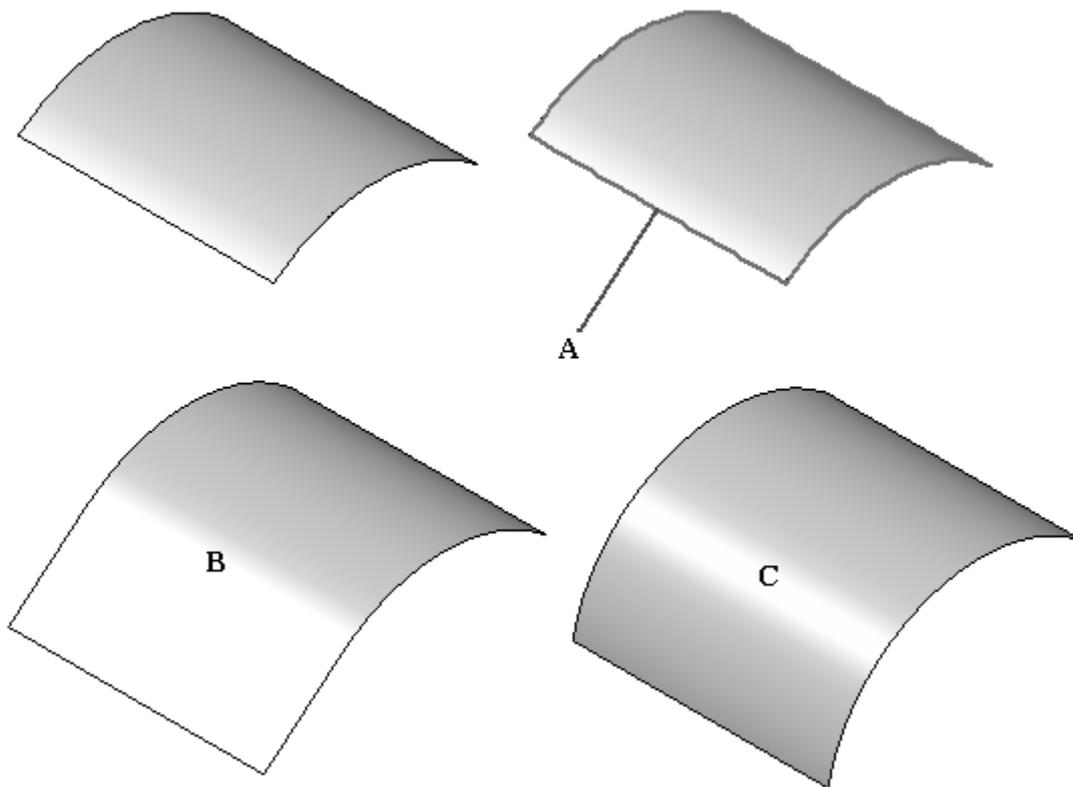
## Extend Surface



Extends a construction surface along an edge. The extended edge can either follow a natural or linear path. You can step back in the command to try different extent options and distances until you achieve the results you want.



The following illustration shows a construction surface extended with distance (A) with a linear extent (B) and a natural extent (C).



## Offset Surface

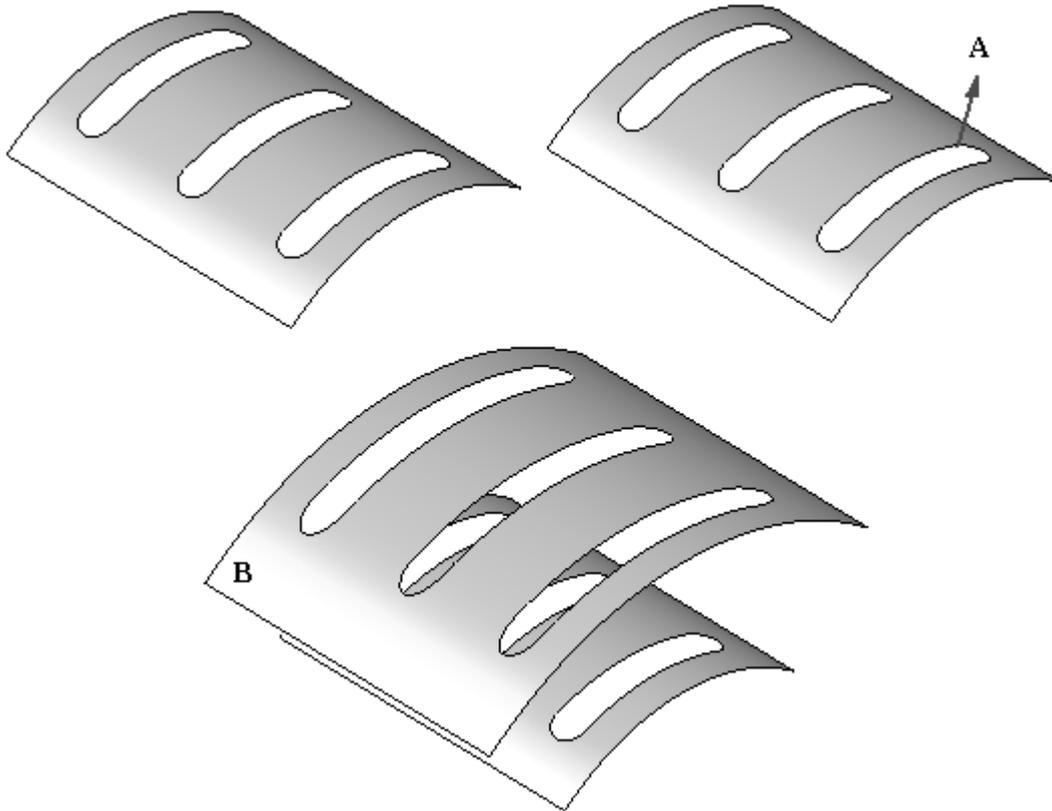


Creates a construction surface by offsetting a solid body, solid face, a construction surface face or chain of faces, or a construction surface. The new surface is offset along a normal vector at a specified distance from the original surface, and is associative to it.

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



The following illustration shows an offset surface B offset in direction A with the show boundaries option on.

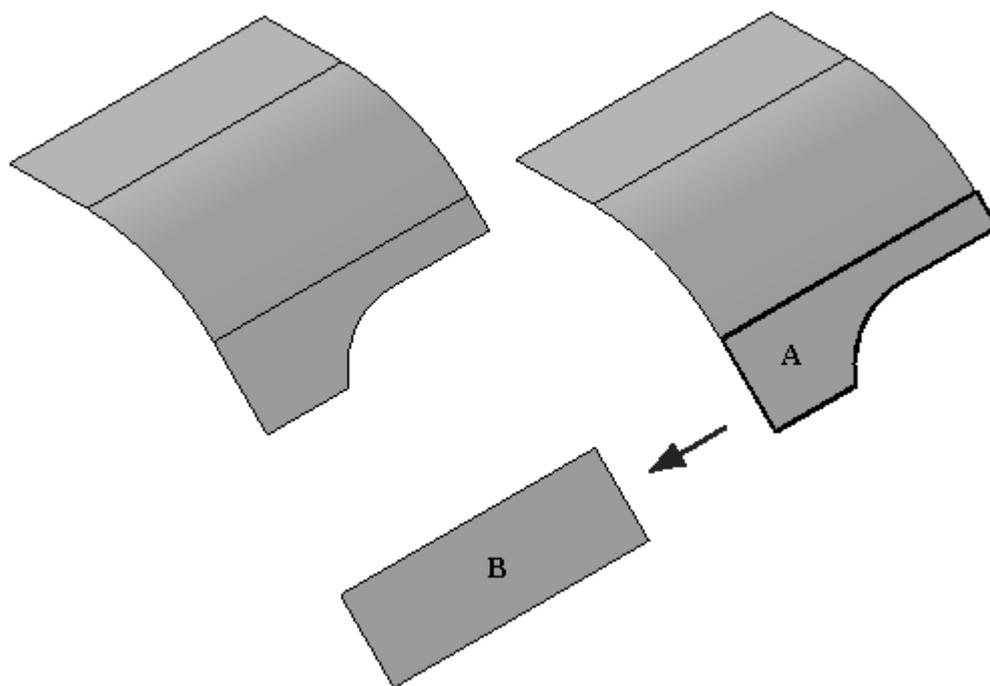


## Copy Surface



Creates 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.

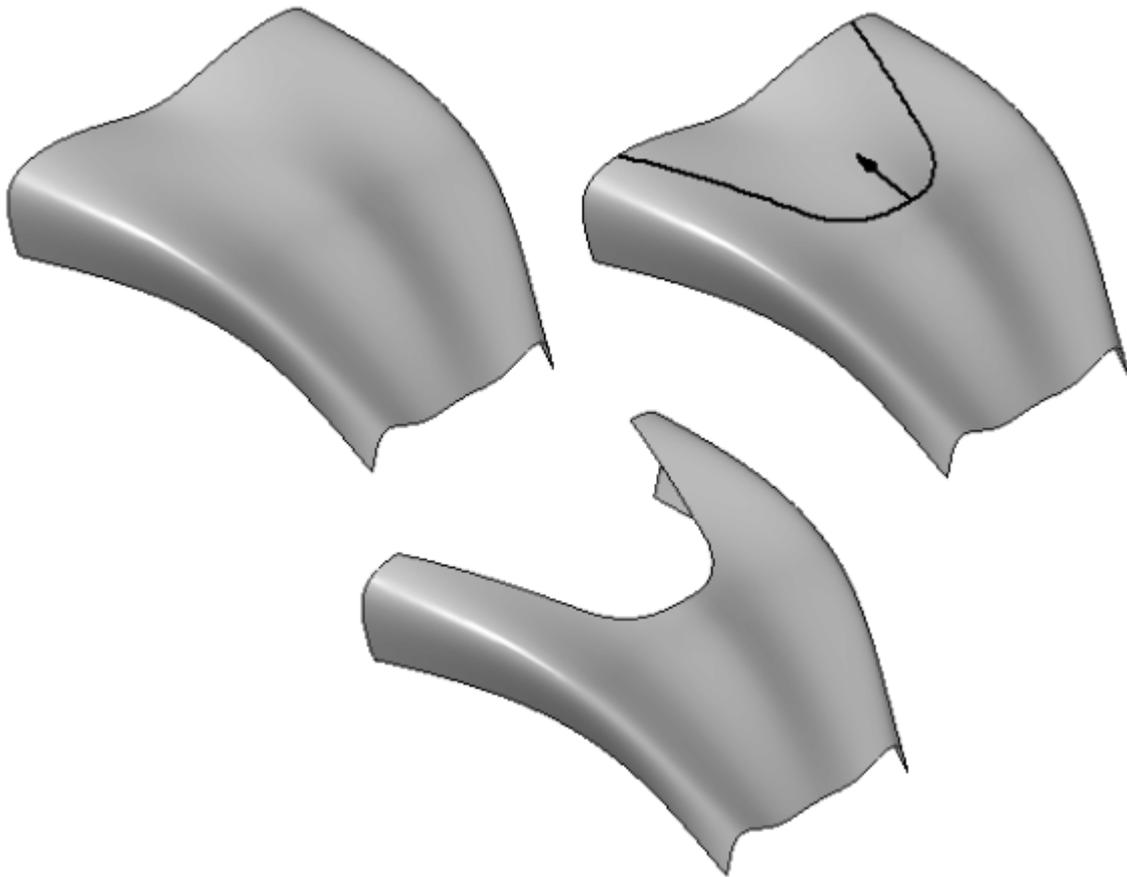
The following illustration shows surface face A copied with boundaries removed B.



## Trim Surface



Trims a surface along input curves. The curves must lie on the surface you are trimming. When trimming non-planar surfaces, you can use the Project Curve command to project the curve onto the surface first.



## Delete Faces

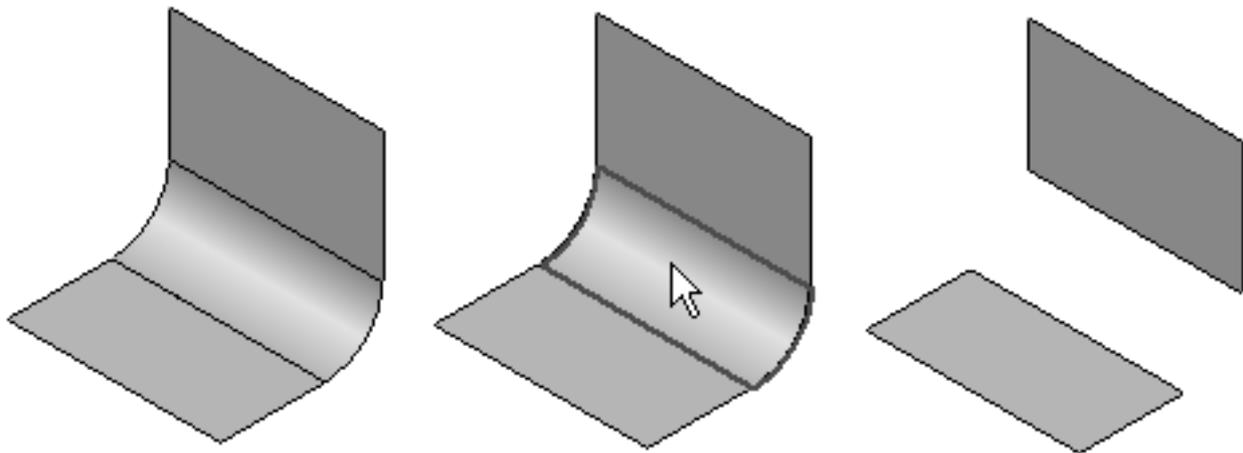


Deletes faces from a model. You can use this command to simplify a model so that it processes faster when used in an assembly, to remove faces from a sheet metal part when working in the Flatten environment, or to delete faces from a construction body.

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 ribbon 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.



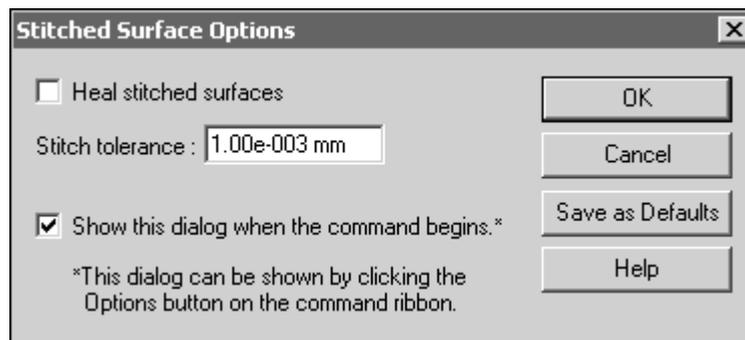
## Stitched Surface



Stitches 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.

Notice the default tolerance on the Stitched Surface Options dialog box. You can change this value if the edges of two surfaces being stitched together do not meet the default tolerance.



If you need to place a round at a common edge of two surfaces, you must stitch the two surfaces together first. You will learn more about rounding in the next section.

### Tips:

- You can set the stitched surface options for tolerance and surface healing on the Stitched Surface Options dialog box.
- To remove surfaces from the select set, select the surfaces while pressing the SHIFT key.
- 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.

### Possible errors:

**Parent Surface Lost**

If a parent surface is lost, Solid Edge presents the warning message, "The stitch operation was unsuccessful. One or more of the input construction surfaces no longer exists." If this message displays, you can re-select the input surfaces.

**Gap Too Great**

If the gap between the edges is greater than the gap tolerance, the surfaces will not stitch. This error can occur on creation or recompute. Fill the gap between the surfaces by increasing the gap tolerance in the Stitch Options dialog box.

**Non-Manifold Condition**

If the stitch surface results in a non-manifold condition, wherein three or more surfaces of the input set share a common edge, Solid Edge presents the warning message: "The stitch operation was unsuccessful. Three or more faces share a common edge resulting in a non-manifold condition." Redefine the input set to eliminate the non-manifold condition.

**Solid Construction Body Given As Input**

If a solid construction body is given as input, Solid Edge will not attempt the stitch.

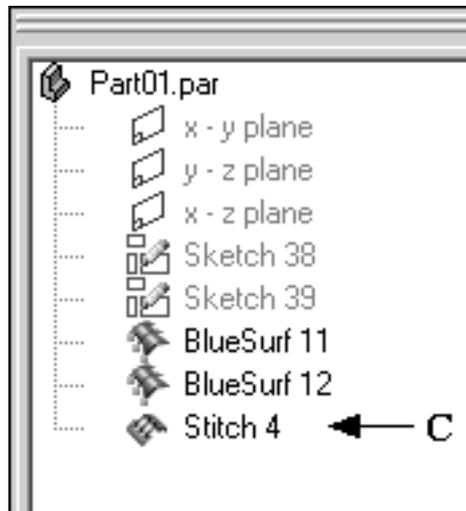
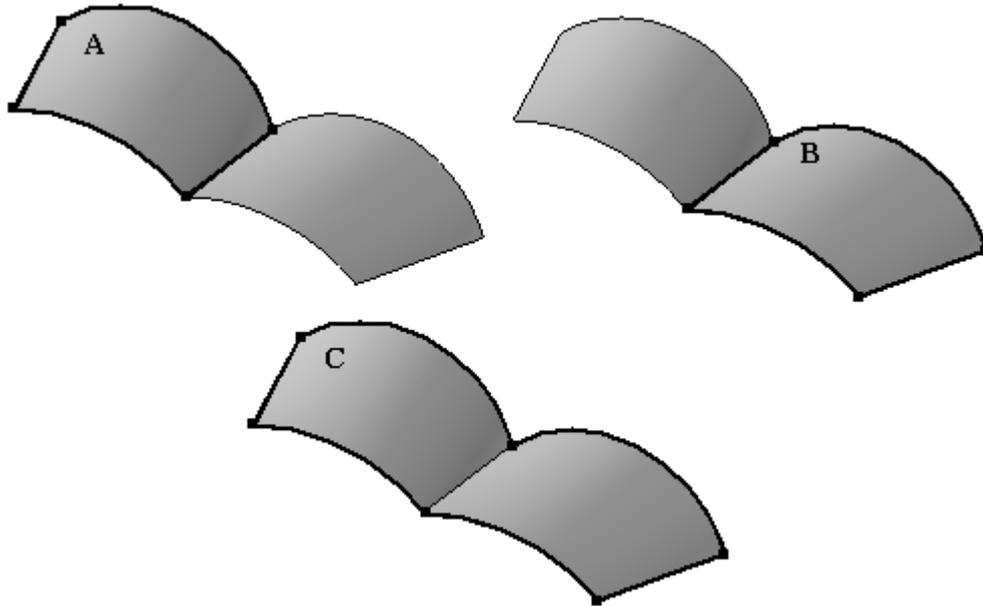
**Stitch Failed: Unable to Stitch to a Single**

Sheet Solid Edge is unable to create the stitch. This can be caused by a number of reasons, such as attempting to stitch surfaces that are not adjacent or the surfaces are beyond the limits of the stitch tolerance.

To show the stitchable edges on construction surfaces, click the Show Non-Stitched Edges command located on the flyout from the Stitched Surface command.



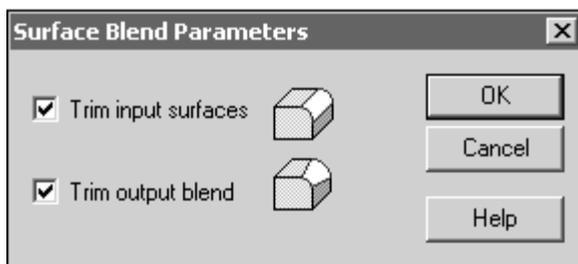
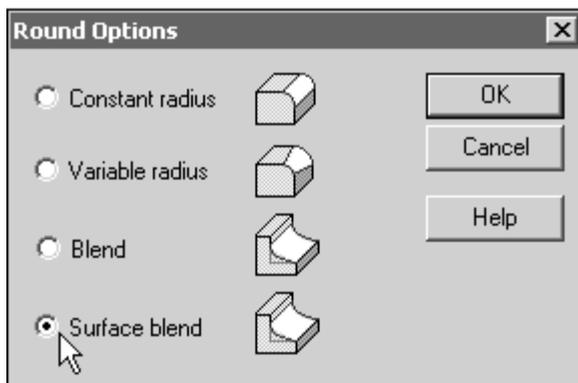
The illustration below shows the stitchable edges for surface A and surface B. Surfaces A and B were stitched together to produce C and the stitchable edges are shown.

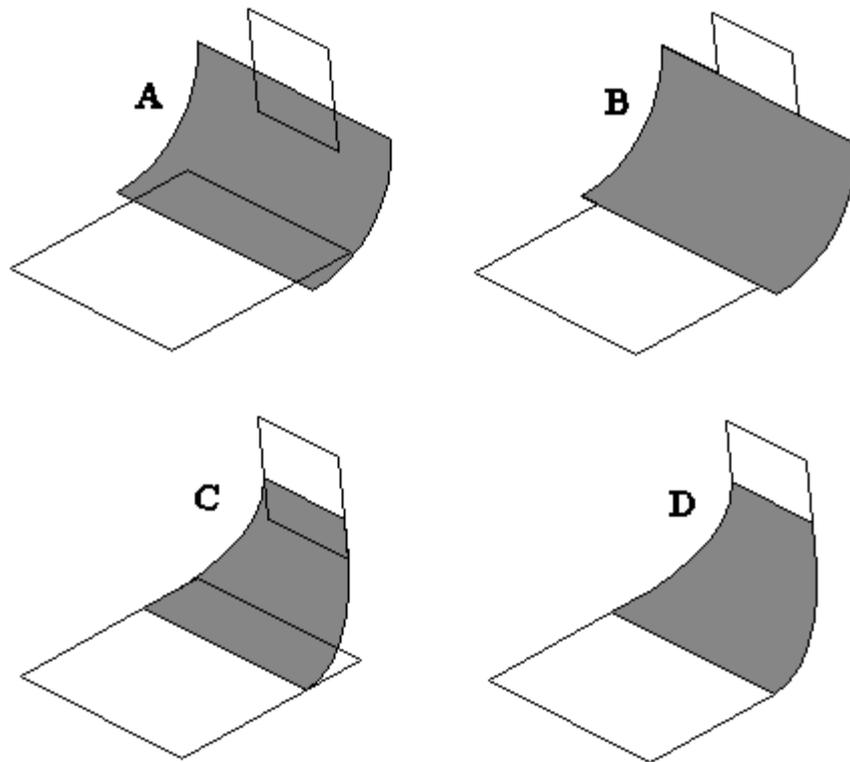


## Round



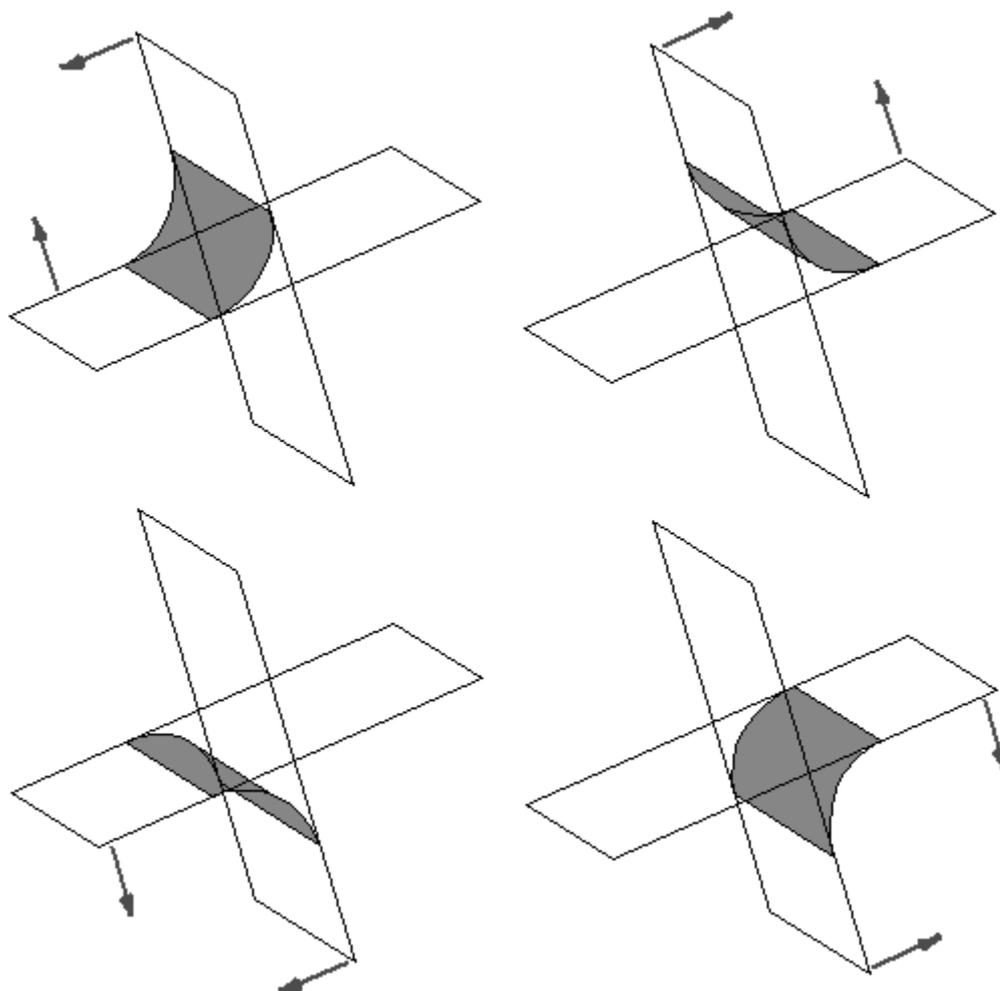
You can use the Round command to place fillets and rounds on surface edges or between two adjoining surfaces.





- (A) no trim
- (B) trim input surfaces only
- (C) trim blend surface only
- (D) trim both input and blend surfaces

Possible solutions of intersecting surfaces:

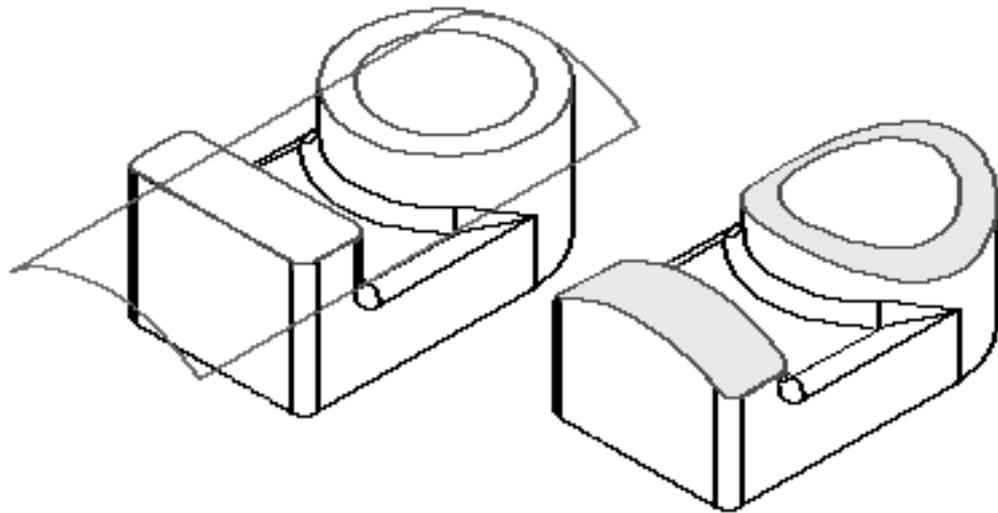


## Replace Face



The Replace Face command replaces 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.

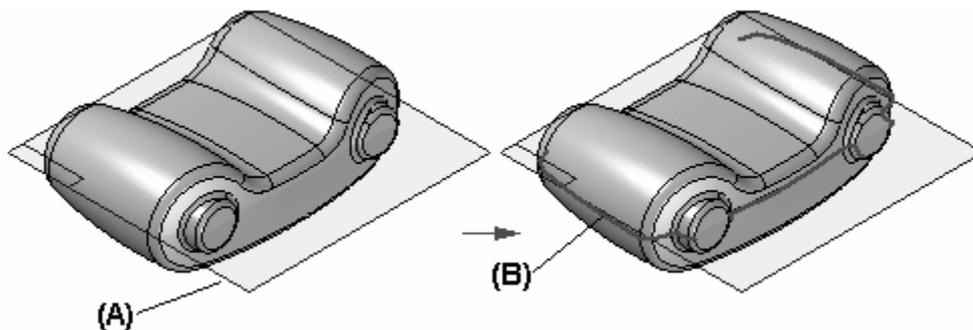
When you replace a face using a construction surface, the construction surface is hidden automatically when you finish the feature.



## Parting Split



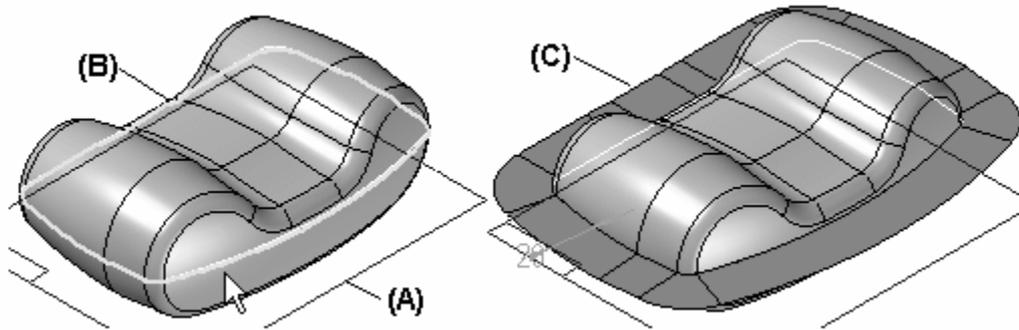
The parting Split command splits a set of faces along the silhouette edges of the part, which can be useful when working with a part that will be molded or cast. Parting lines are the same as silhouette lines for a given face. You define the vector direction for calculation of the parting lines by defining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



## Parting Surface



Constructs a parting surface along a parting curve you select. You construct a parting surface by selecting a reference plane (A) to define the orientation of the linear cross section curve, and a 2-D or 3-D parting curve (B), which defines the sweep path for the parting surface (C).

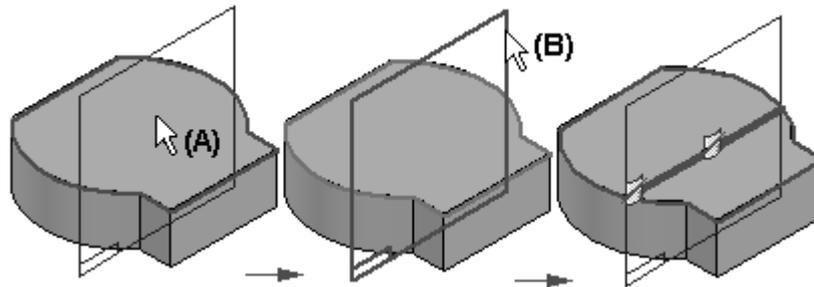


You create the parting curve in a separate operation. For example, you can use the Intersection Curve command or the Parting Split command to create the parting curve.

## Split Face

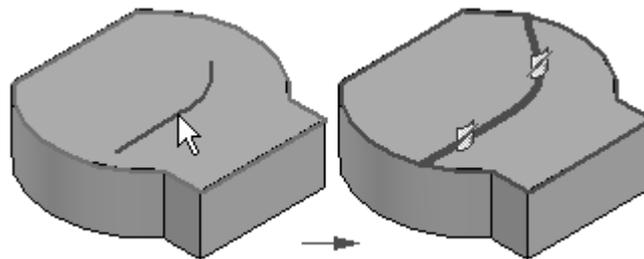


Splits one or more surfaces (A) using an element (B) you define. You can select curves, edges, surfaces, reference planes, and design bodies as the elements that split the face.

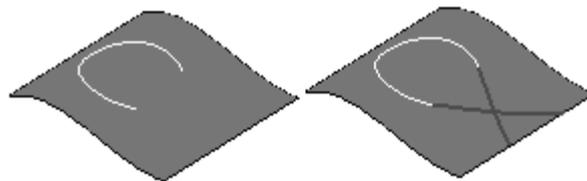


Splitting a face can be useful when constructing a model that you want to use for finite element analysis purposes or when you want to isolate a portion of a face so you can to apply a decal or image in a specific location.

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.

## **Review**

- What are the two extent options available on the Extend Surface ribbon bar?
- How do you create an offset surface without boundaries from an input surface that has boundaries?
- Name the methods of creating a boundary curve on a surface. · Can you trim a surface with multiple open curves in one step?
- Can you trim a surface with multiple closed curves in one step?
- Explain what stitch tolerance is.
- After stitching surfaces together, what happens when there are no stitchable edges remaining?
- How do you round a common edge of two separate surfaces?
- What is a surface blend?
- Do two surfaces have to be stitched together to insert a surface blend?
- Can multiple faces of a solid be replaced in one step?



## Activity 1 – Surface Manipulation

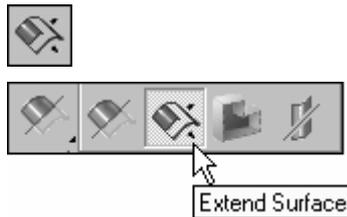
### Overview

In this activity, you will learn to use the surface manipulation commands. These commands let you manipulate existing surfaces.

### Objectives

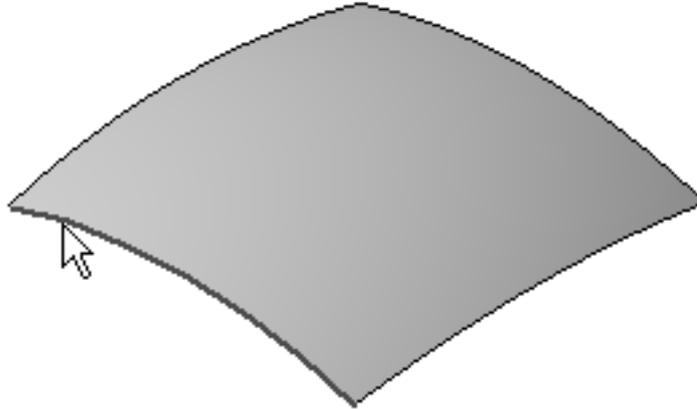
After completing this activity you will be able to:

- 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
1. Open SURFACE LAB 4-01.PAR in the C:/Surfacing/Parts folder.
  2. You will begin the activity by using the Extend Surface command.
    - On the Surfacing toolbar, click the Extend Surface command. The command is on a pull out menu.



- Click on the surface. In EdgeBar the feature name is Sweep A.

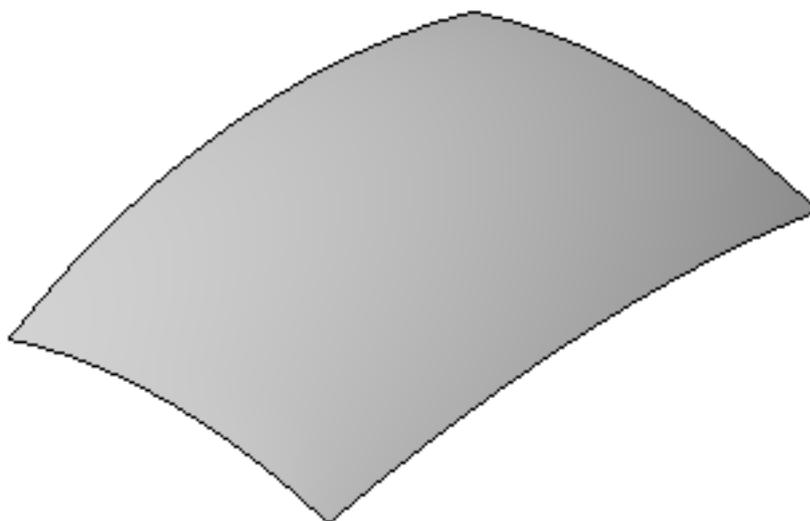
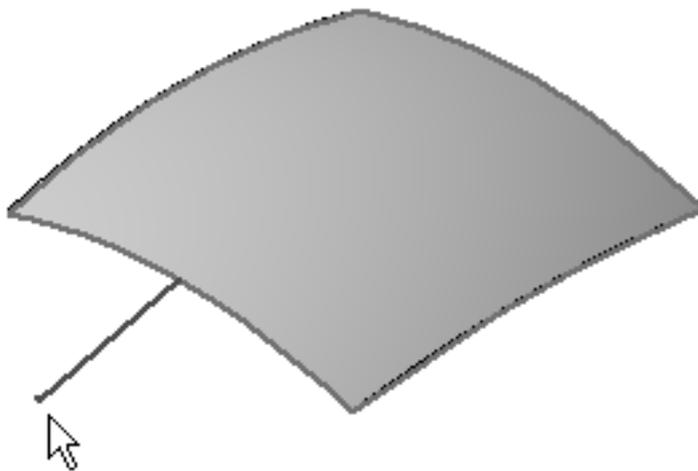
- Click the edge shown and click the Accept button on the ribbon bar.



Notice on the Extend Surface ribbon bar the options for Natural or Linear Extent. The natural option extends the surface to follow the curvature of the surface. The linear option extends the surface in a linear direction. For this example, the linear extent option is grayed out because the surface cannot be extended linearly.



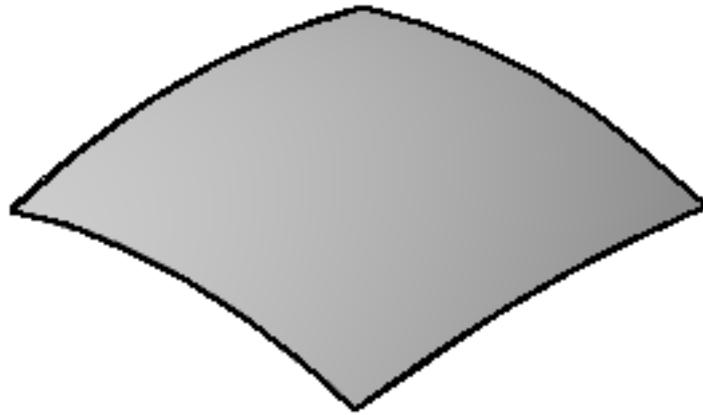
- Drag the distance vector approximately as shown, and click.



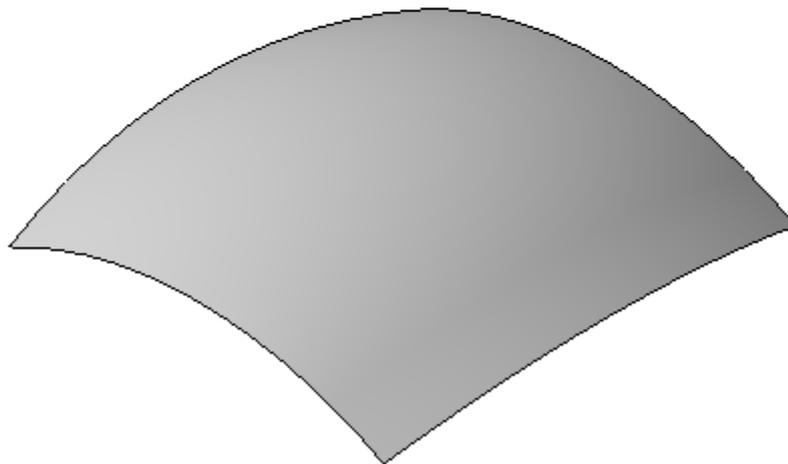
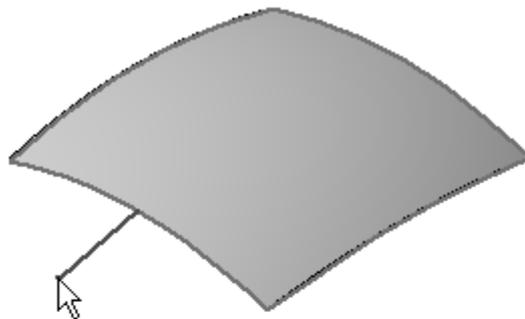
- On the ribbon bar, click Finish.
- Click the Select tool.
- In EdgeBar, right-click on the Extend feature and click delete. This is the last feature listed in EdgeBar.



- You can also extend multiple edges. Repeat the previous step. Select all four edges and then click the Accept button.



- Drag the distance vector approximately as shown and click.



- On the ribbon bar, click Finish.
- In EdgeBar, delete the Extend feature.

3. You will now use the Offset Surface command.

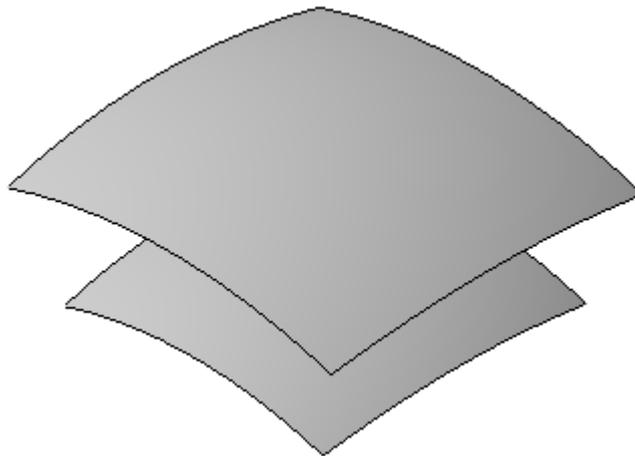
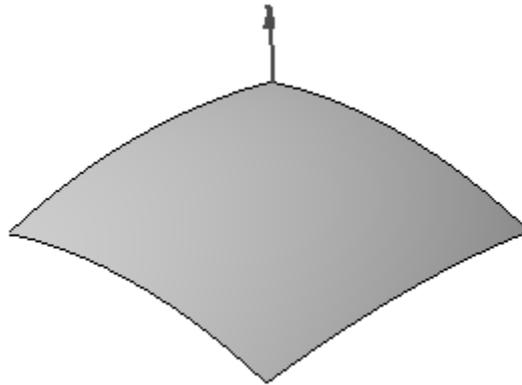
- On the Surfacing toolbar, click the Offset Surface command.



- Click the surface, and then click Accept.
- On the Offset Surface ribbon bar, type 50 for the offset distance and press ENTER.



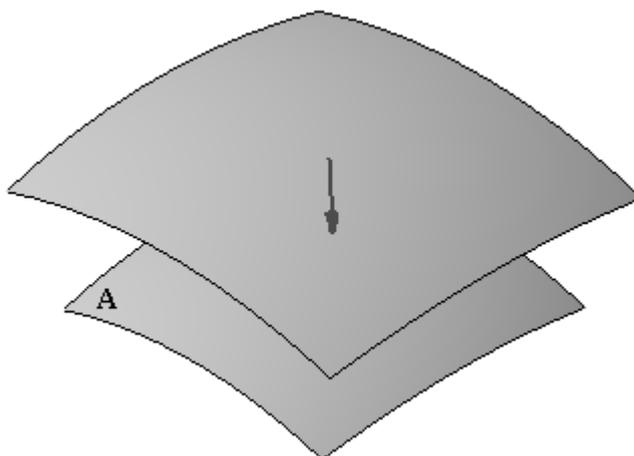
- Position the direction arrow as shown, and click.



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

- On the ribbon bar, click Finish.

- Repeat the previous step. Use 50 for the offset distance and position the direction arrow shown. Offset surface A.

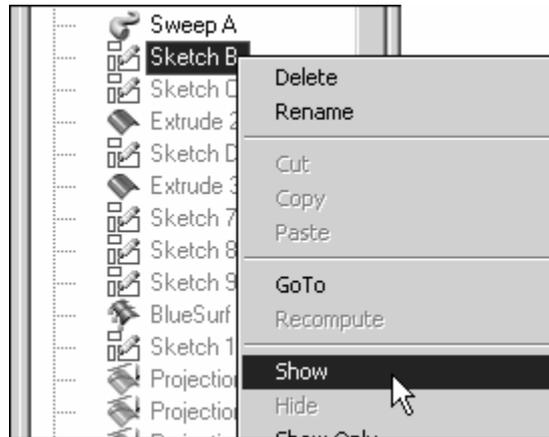


- On the ribbon bar, click Finish.



- In EdgeBar, delete the two offset surfaces.
4. You will now use the Trim Surface command. This command is used extensively in surface modeling.

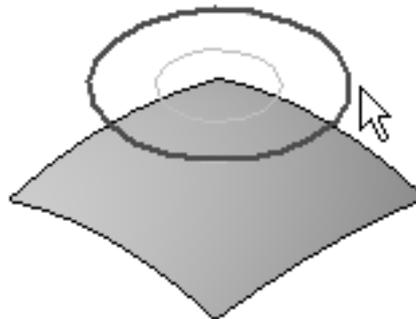
- In order to trim a surface, a trim curve needs to lie on the surface. You will now project two sketches onto a surface to use as trim curves. In EdgeBar, show Sketch B.



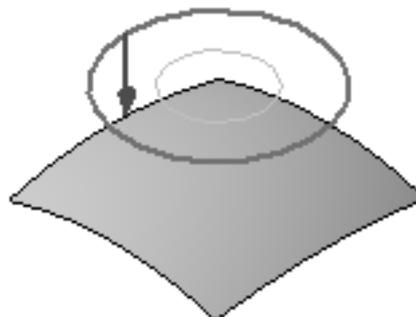
- On the Surfacing toolbar, click the Project Curve command.



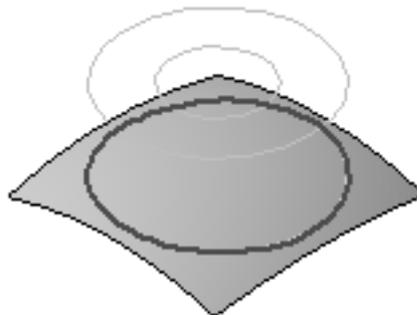
- Click the circle shown below and then click the Accept button.



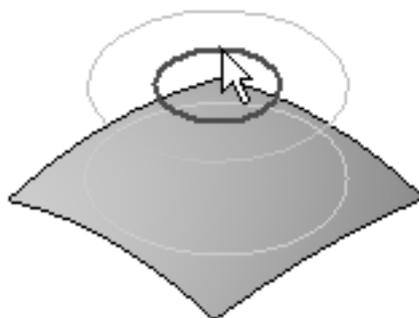
- Click the surface, click Accept and then position the direction arrow as shown, and click.



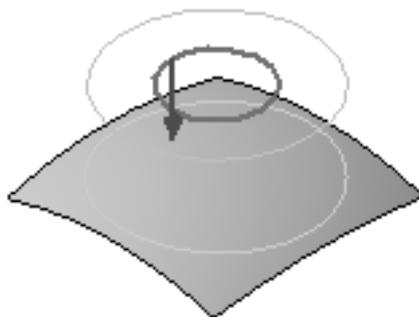
- Click Finish.



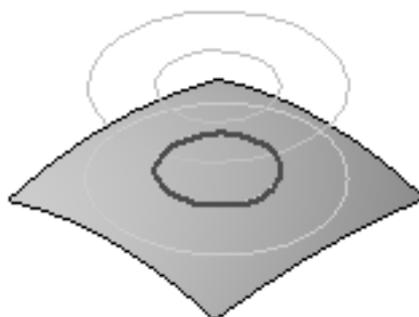
- Click the circle shown below and then click the Accept button.



- Click the surface, click Accept and then position the direction arrow as shown, and click.



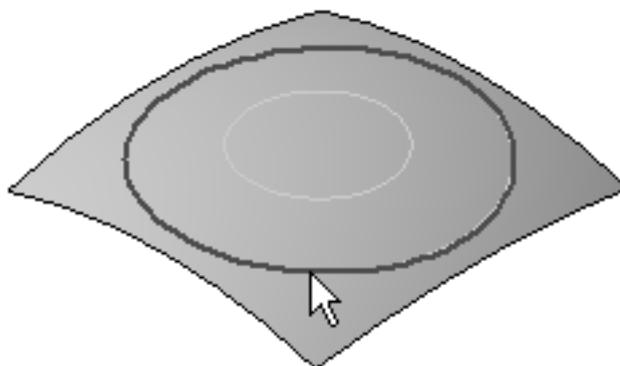
- Click Finish.



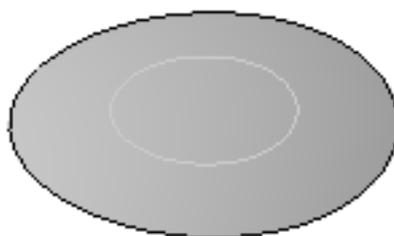
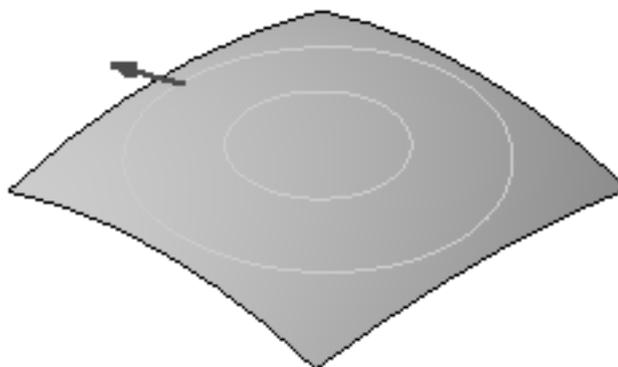
- In EdgeBar, hide Sketch B.
- On the Surfacing toolbar, click the Trim Surface command.



- Click the surface and click Accept.
- Click the projection curve as shown and click the Accept button.

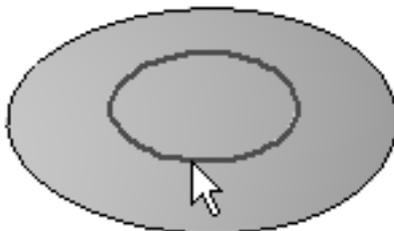


- Position the direction arrow as shown to trim the surface outside the projection curve.

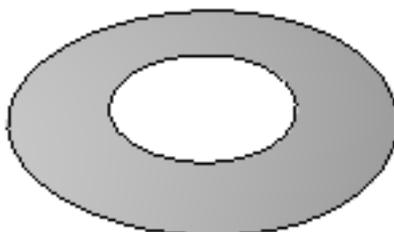
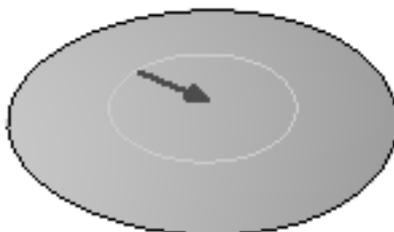


- Click Finish.

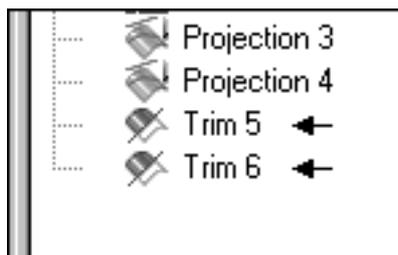
- Click the surface and click Accept.
- Click the projection curve as shown and click the Accept button.



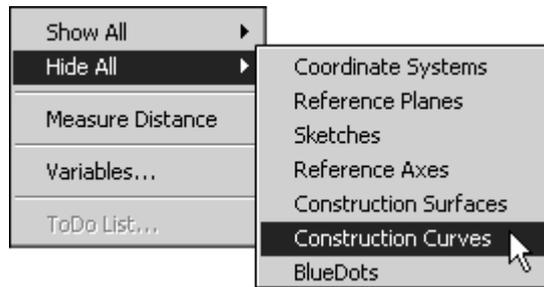
- Position the direction arrow as shown to trim the surface inside the projection curve.



- Click Finish.
- Notice the Trim features in EdgeBar.



- Hide all construction curves.



5. You will now use the Copy Surface command.

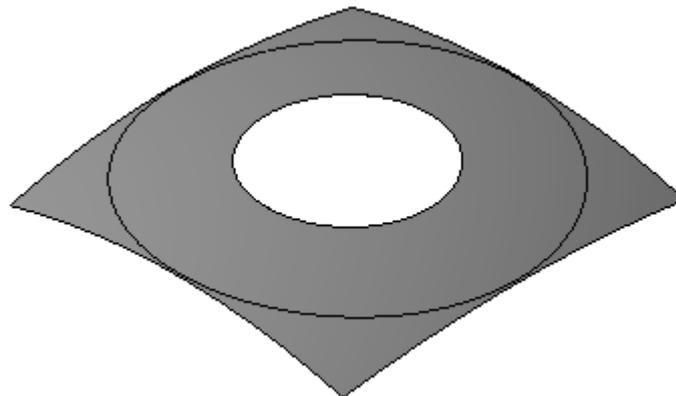
- On the Surfacing toolbar, click the Copy Surface command. The Copy Surface command is located on the fly out of the Offset Surface command.



- Notice on the Copy Surface ribbon bar the two options for removing boundaries. To remove internal boundaries on the copied surface click option A. To remove external boundaries on the copied surface click option B. If neither option is selected, the copied surface will maintain all boundaries.



- On the ribbon bar, click the Remove External Boundaries button B.
- Click the surface and then click the Accept button.



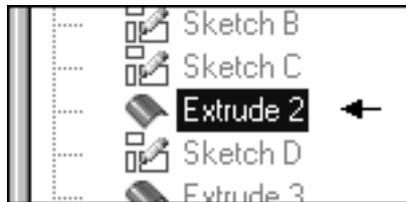
- Click Finish.
- Notice the Copy feature in EdgeBar.



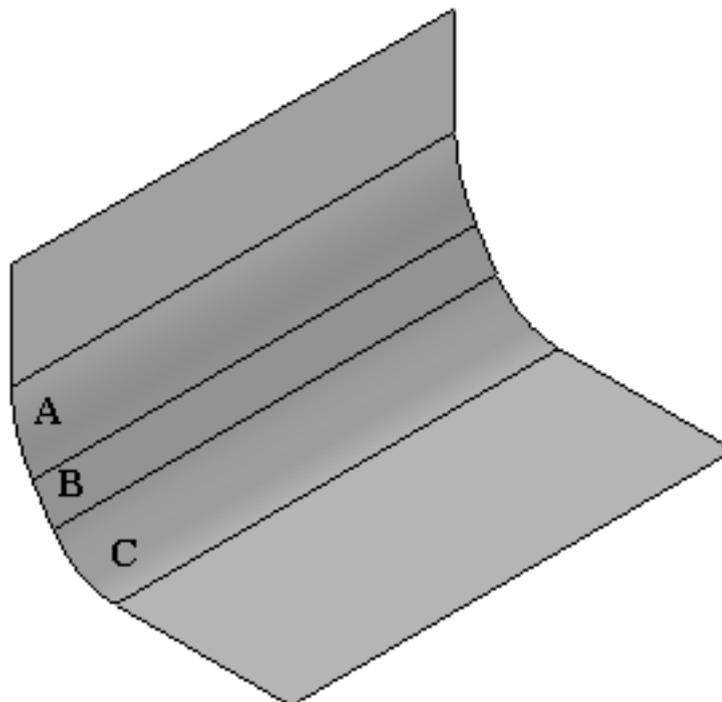
- In EdgeBar, delete the Copy feature, the two Projection features and the two Trim features.



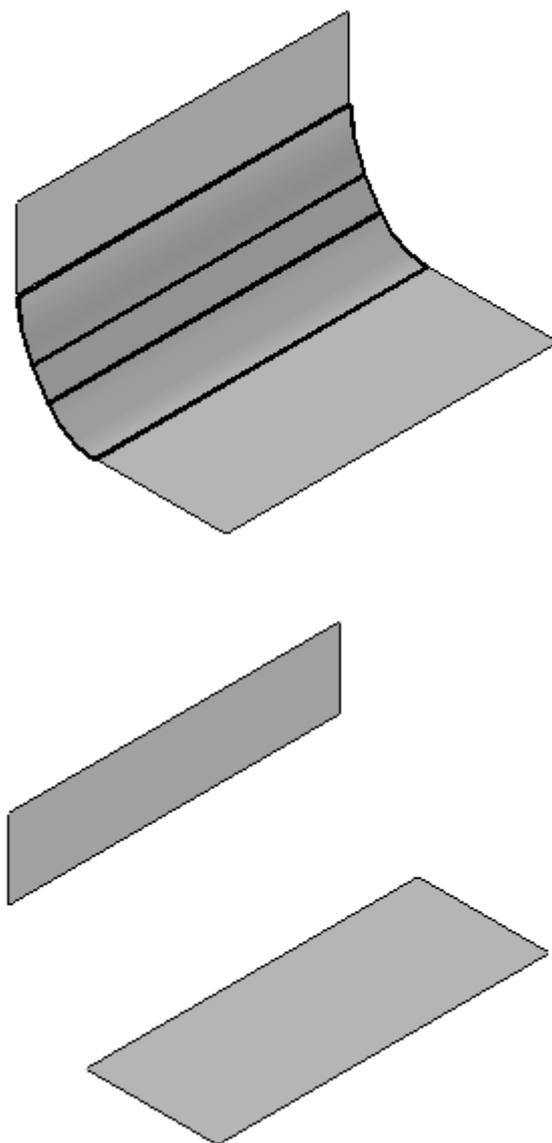
- Hide feature Sweep A.
6. You will now use the Delete Faces command. You can delete faces on a construction body and replace the deleted faces with a new surface.
- In EdgeBar, show feature Extrude 2. Click the Fit command.



- You will delete faces A, B and C. On the Surfacing toolbar, click the Delete Faces command. The Delete Faces command is located on the fly out of the Extend Surface command.



- Select the three faces shown and click the Accept button.

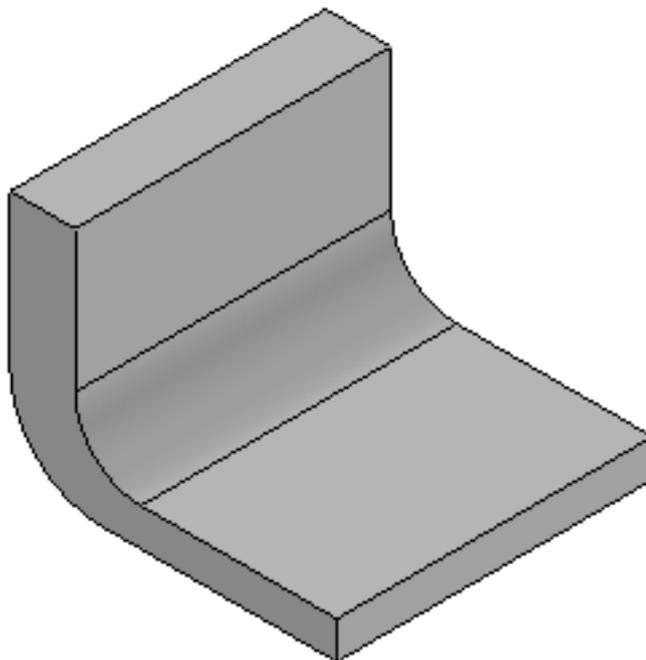


- On the ribbon bar, click Finish.
- Notice the Delete Face feature in EdgeBar.



- Hide feature Extrude 2. Show feature Extrude 3. You will now use the Heal option on the Delete Faces ribbon bar. In order for the heal option to work,

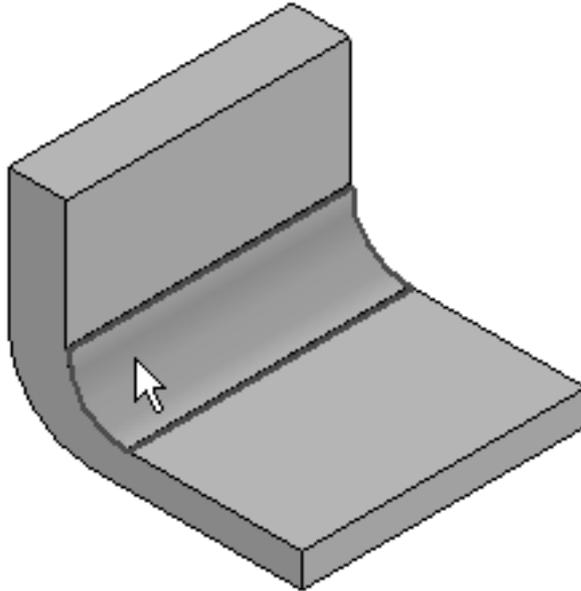
the construction body must be closed. The Extrude 3 feature was constructed with a closed profile and the ends were capped.



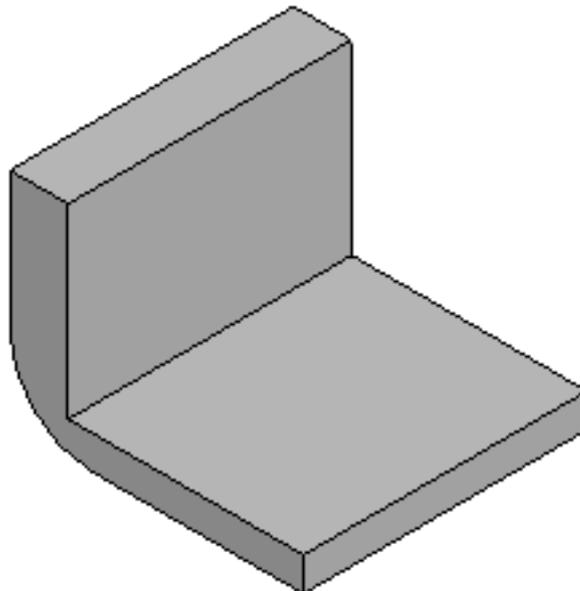
- On the Surfacing toolbar, click the Delete Faces command.
- On the Delete Faces ribbon bar, click the Heal Option button.



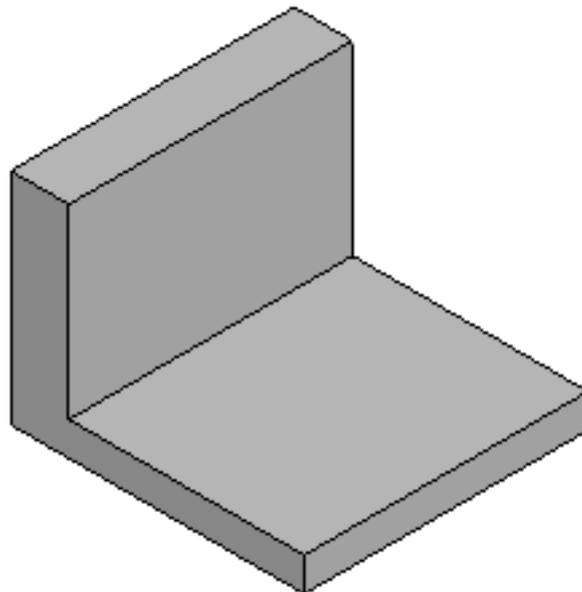
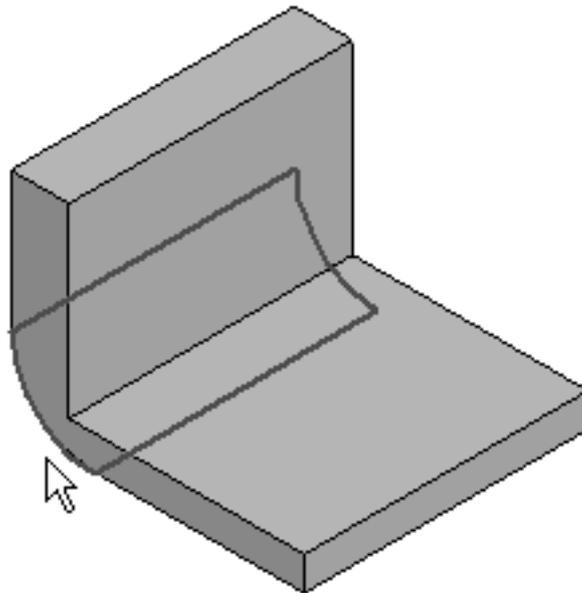
- Click the surface shown and then click the Accept button on the ribbon bar.



- 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.

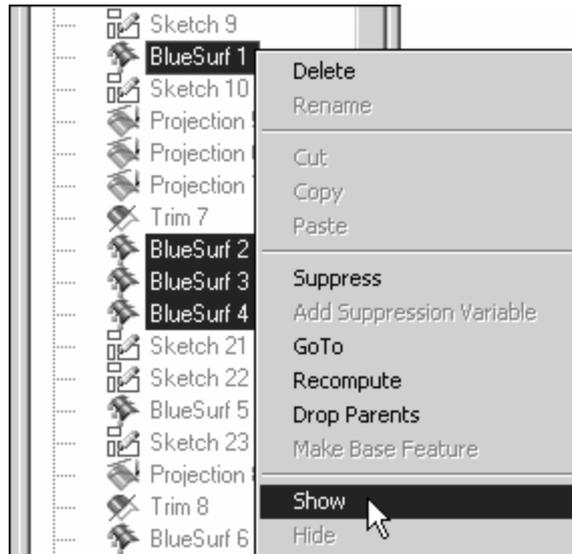


- Delete the face shown.



- Click Finish.
- In EdgeBar, hide the Extrude 3 feature.

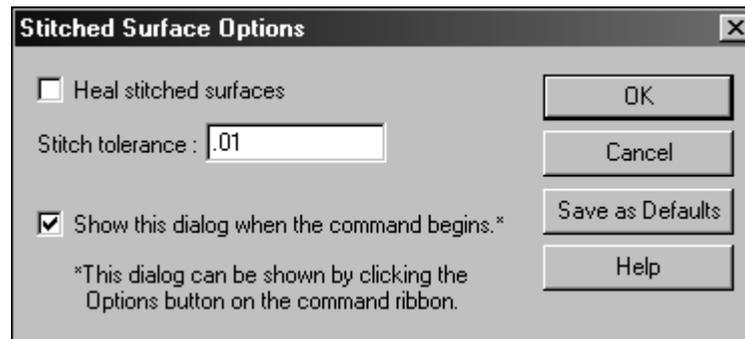
- 7. You will now use the Stitched Surface command. In EdgeBar, show features BlueSurf 1, BlueSurf 2, BlueSurf 3, and BlueSurf 4.



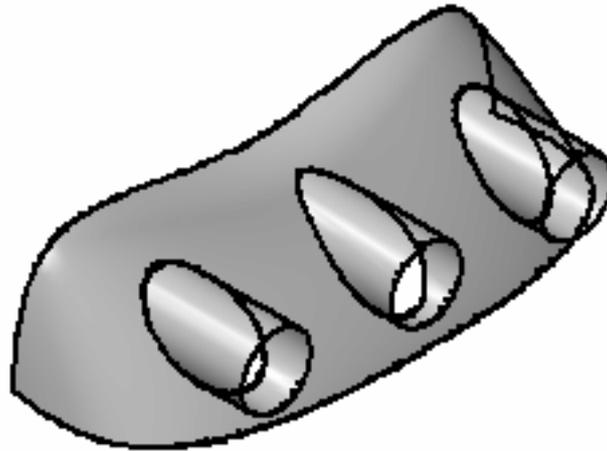
- You will now stitch these BlueSurf features together. On the Surfacing toolbar, click the Stitched Surface command.



- Type .01 in the Stitch tolerance field and then click OK.



- Select all four surfaces and then click the Accept button.



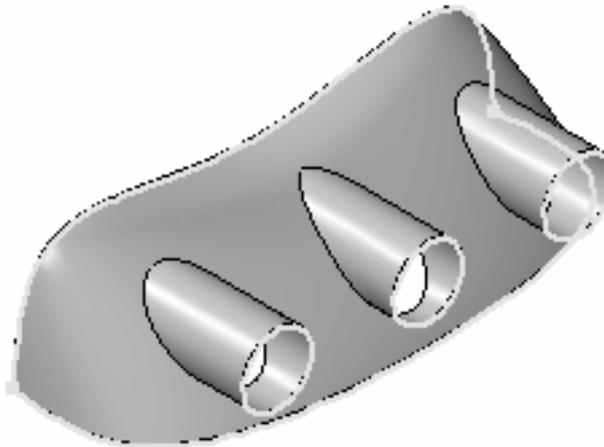
- Click Finish.
- Notice in EdgeBar the Stitch feature.



- On the Surfacing toolbar, click the Show Non-Stitched Edges command. The command is located on the Stitched Surface fly out.



- Click the stitched surface. Notice the yellow edges. These are the un-stitched edges.



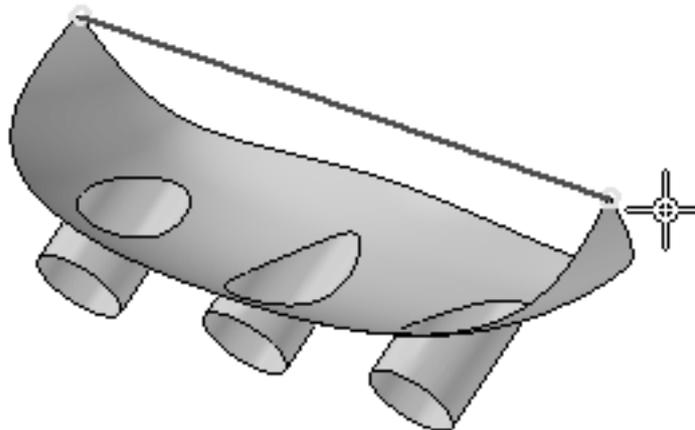
- On the ribbon bar, click Close.



- In order to make a solid feature from these stitched surfaces, you must have surfaces stitched to all of the non-stitched edges. You will now create surfaces needed to make a solid feature. On the Surfacing toolbar, click the Keypoint Curve command.



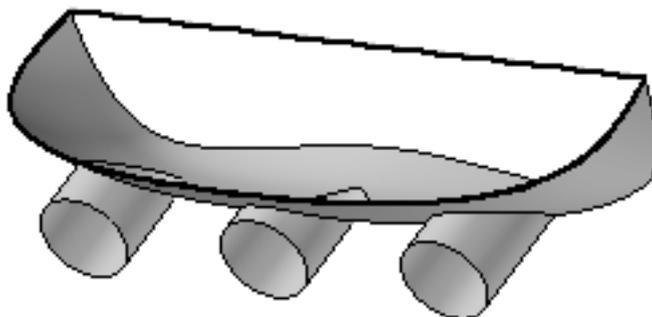
- Draw a keypoint curve as shown. The curve has two points.



- You will now create five bounded surfaces. On the Surfacing toolbar, click the Bounded Surface command.



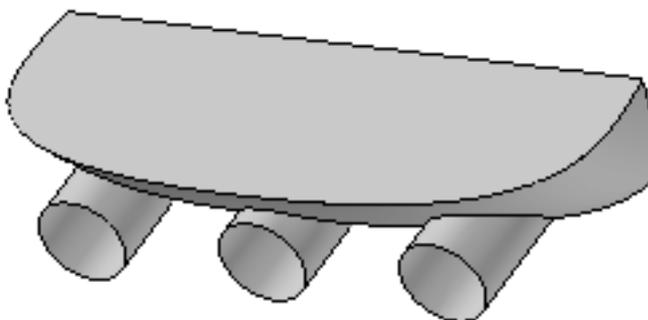
- Click the edges shown and click the Accept button.



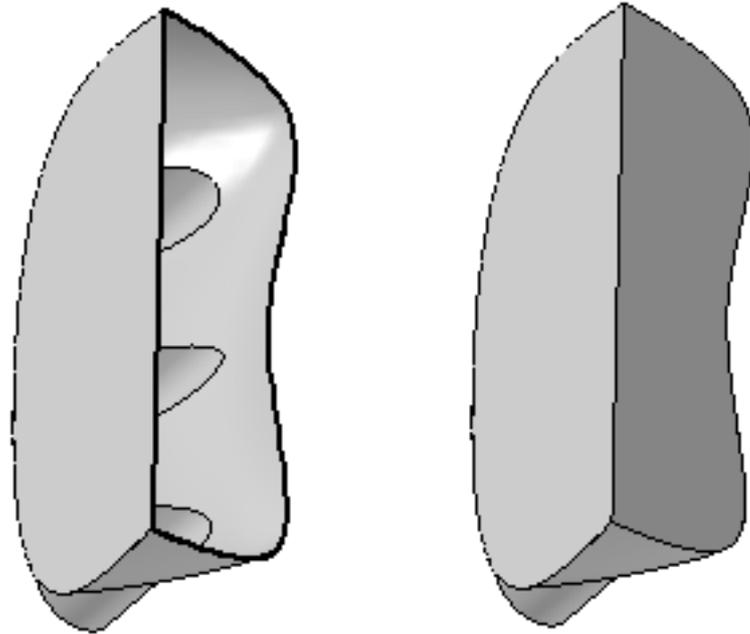
- Turn off the Face Tangency option.



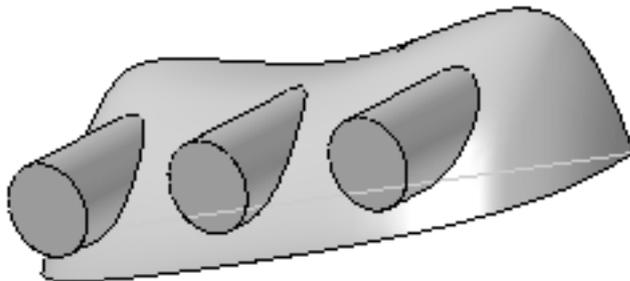
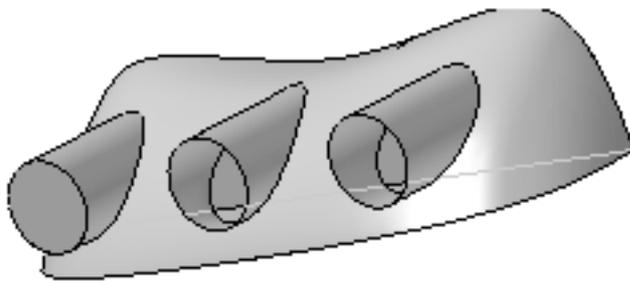
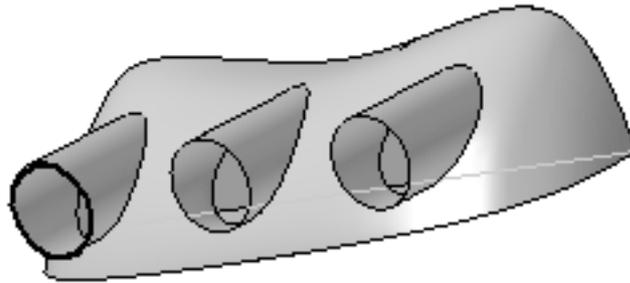
- Click Preview and then click Finish.



- Repeat the same steps to create the remaining bounded surfaces.



- Repeat the steps for the three circular edges.

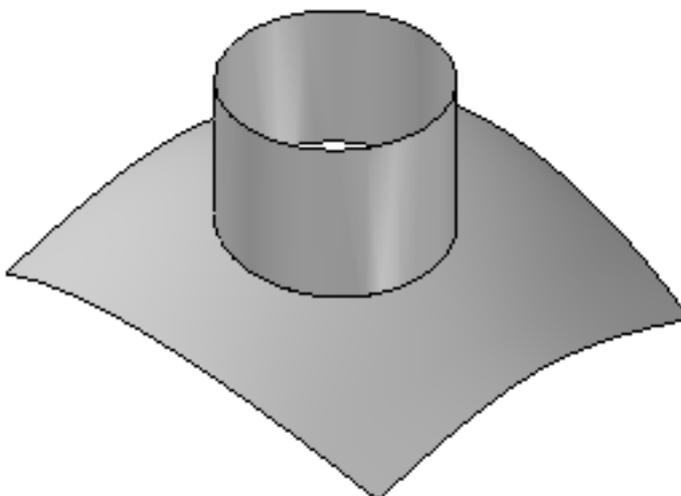


- Click the Stitched Surface command. Type .01 in the Stitch tolerance field and click OK.
- Click the stitched surface and then click the five bounded surfaces.
- Click the Accept button. Since there are no non-stitched edges remaining, the stitched surfaces result in a solid body. Click OK.



- Click Finish.

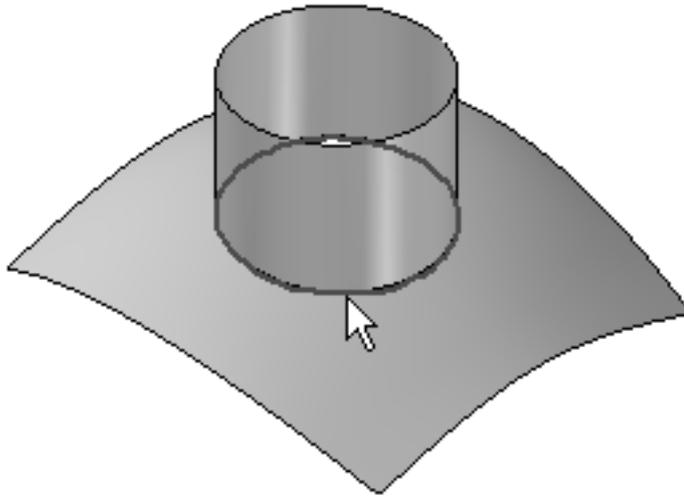
- If there is no base feature (solid) in the file, you can right-click on the stitched surface and click Make Base Feature to create a solid of the stitched surface.
  - Hide features BlueSurf 1-4 and the keypoint curve.
8. You will now use the Round command. You will round an edge between two surfaces and also perform a surface blend between two surfaces.
- In EdgeBar, show features BlueSurf 5 and BlueSurf 6. Click the Fit command.



- In order to insert a round between these two surfaces, the two surfaces must be stitched together. Stitch the two surfaces together.
- On the Features toolbar, click the Round command.



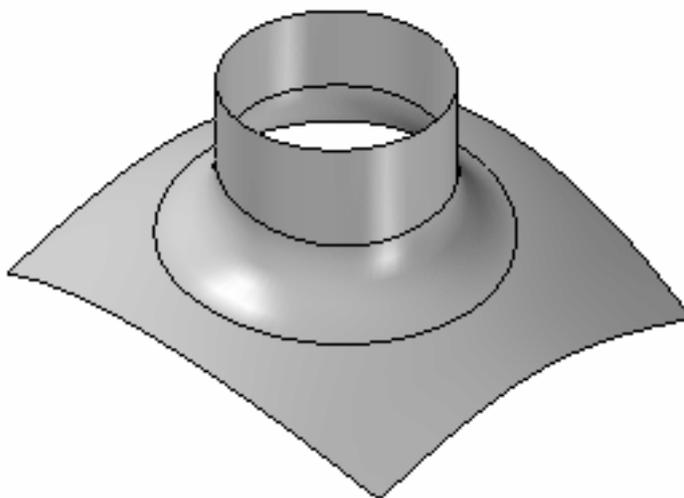
- Click the edge shown.



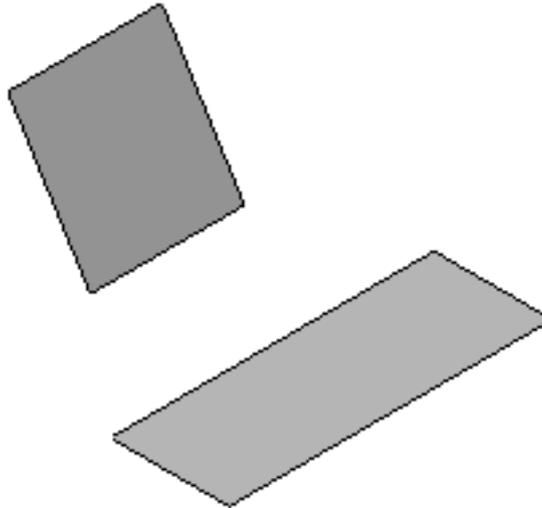
- On the Round ribbon bar, type 25 in the radius field and then click the Accept button.



- Click Preview and then click Finish.



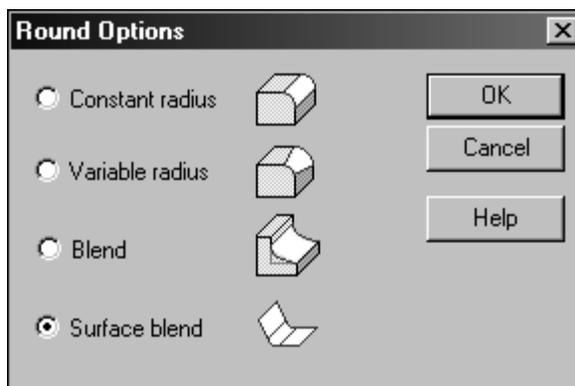
- In EdgeBar, hide features BlueSurf 5 and BlueSurf 6. Show features Extrude 4 and Extrude 5.



- You will now insert a surface blend between these two extruded surfaces. On the Features toolbar, click the Round command.
- On the Round ribbon bar, click the Round Options button.

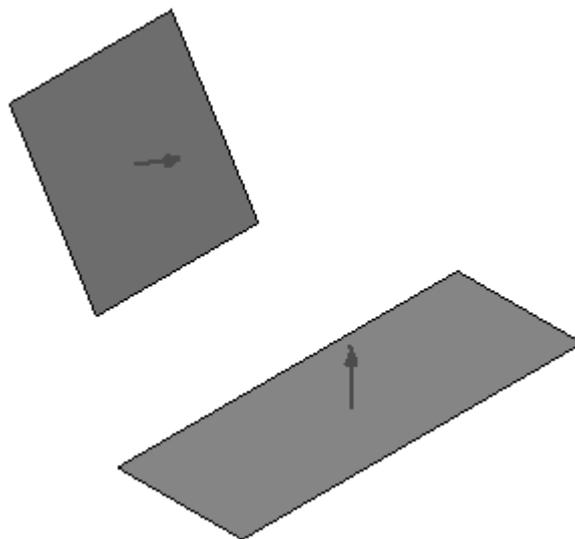


- Click the Surface blend option and click OK.

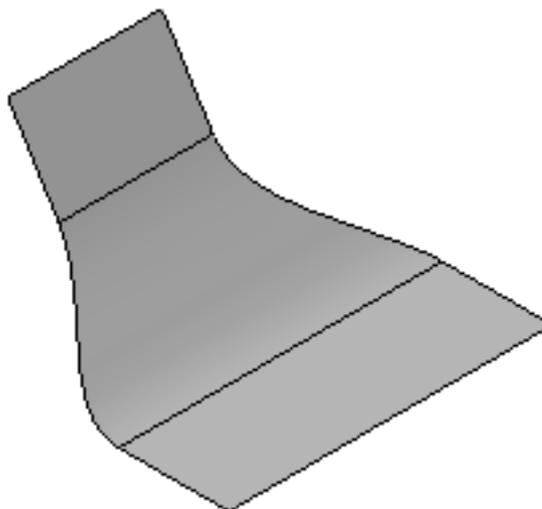


- Click the two surfaces and type 70 in the radius field on the ribbon bar. Click the Accept button.

- Position the direction arrows as shown. This assigns which side of the surface to place the blend.

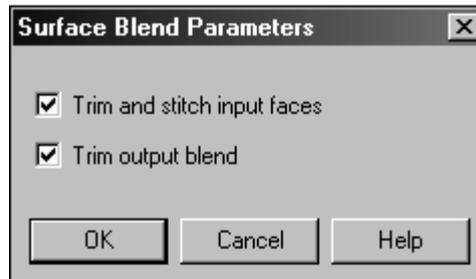


- Click Preview and then click Finish.

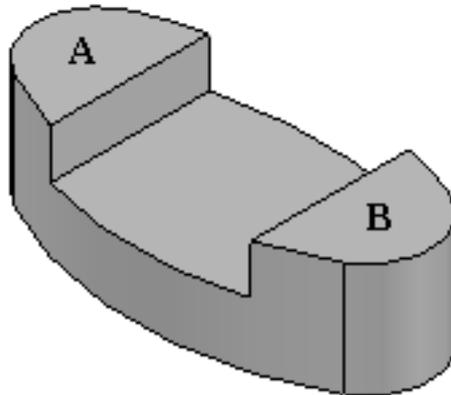




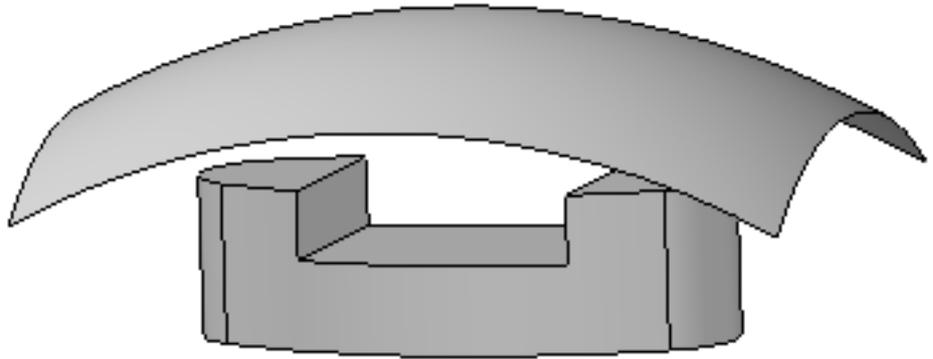
Located on the ribbon bar is a Surface Blend Parameters option button. The default setting trims the input faces and output blend. Try each option and observe the results.



- In EdgeBar, hide features Extrude 4 and Extrude 5.
9. You will now use the Replace Face command. In EdgeBar, show feature Protrusion A. You will replace faces A and B on Protrusion A with a construction surface.



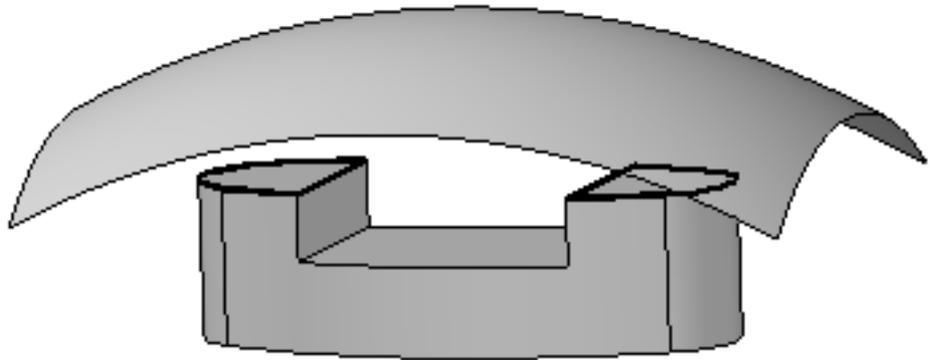
- In EdgeBar, show feature BlueSurf 7.



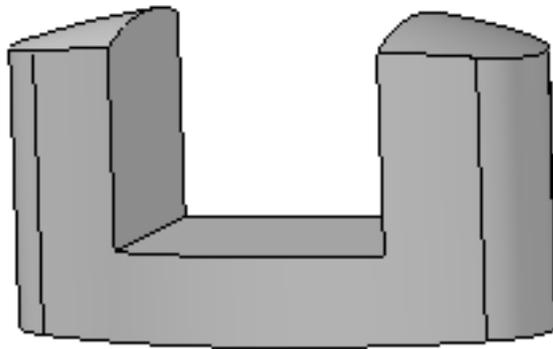
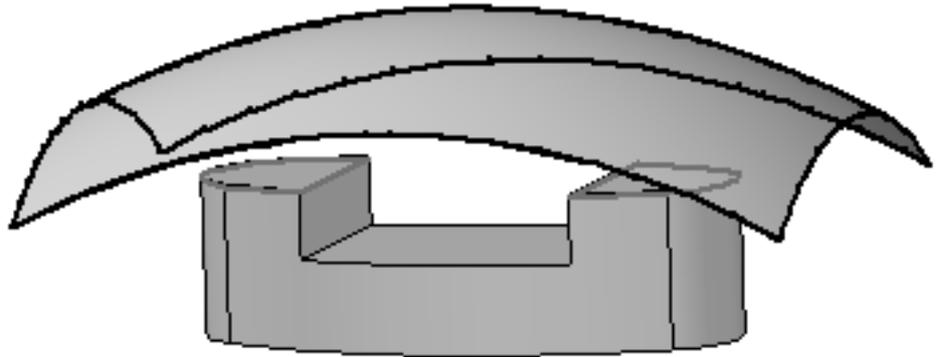
- On the Surfacing toolbar, click the Replace Face command.



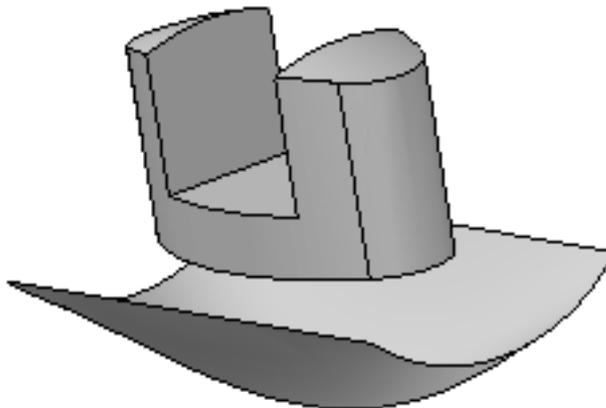
- Click the faces shown and click the Accept button.



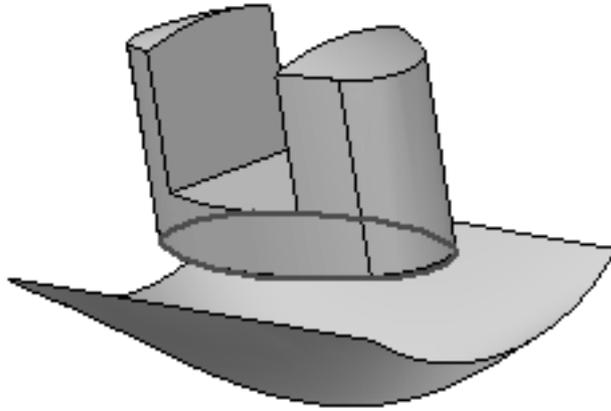
- Click the surface shown for the replacement surface.



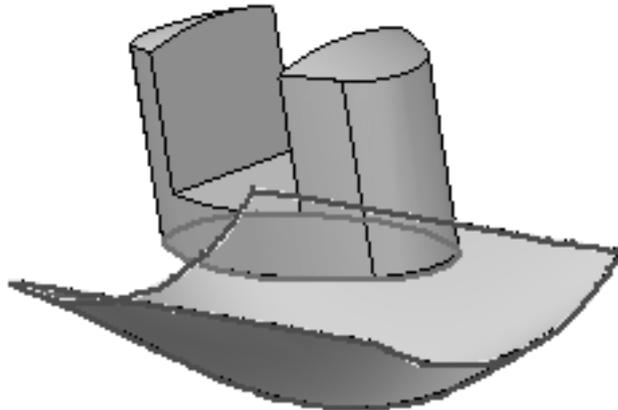
- Click Finish.
- You will now replace the bottom face on the protrusion. In EdgeBar, show BlueSurf 8.



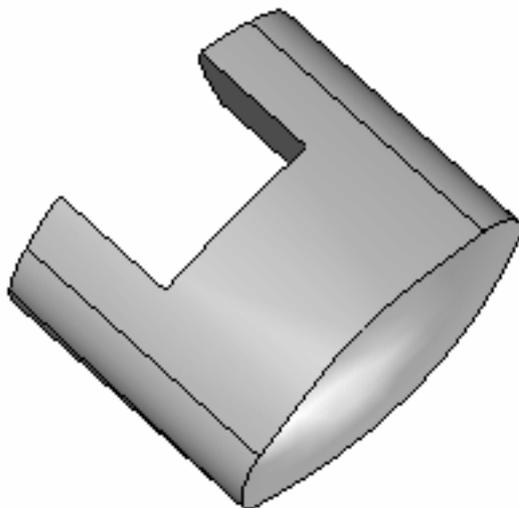
- Click the Replace Face command.
- Click the face as shown and click the Accept button.



- Click the surface shown for the replacement surface.



- Click Finish.



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

## Activity 2 – Putting It All Together

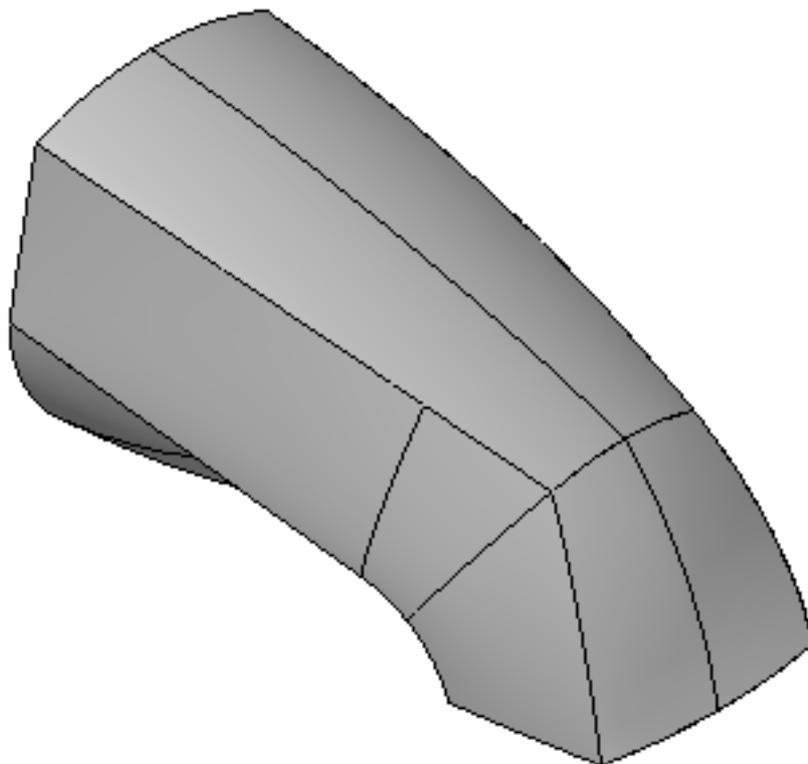
### Overview

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

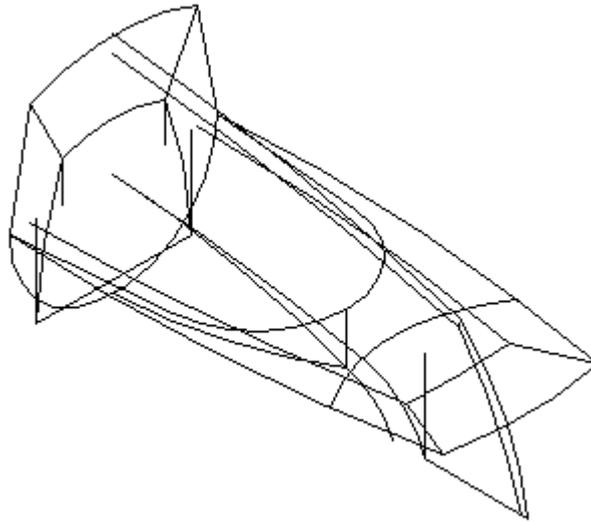
### Objectives

After completing this activity you will be able to:

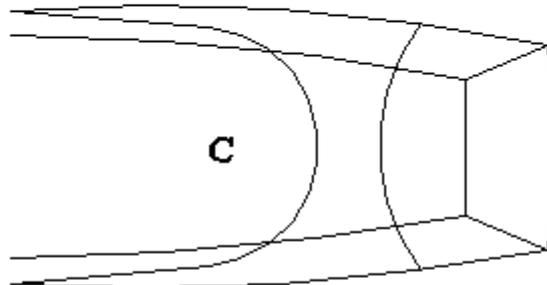
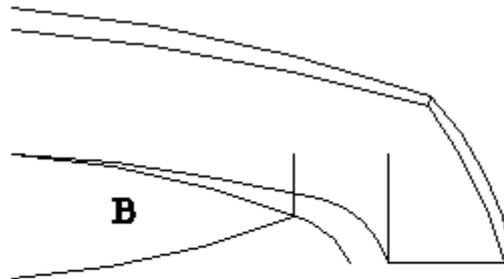
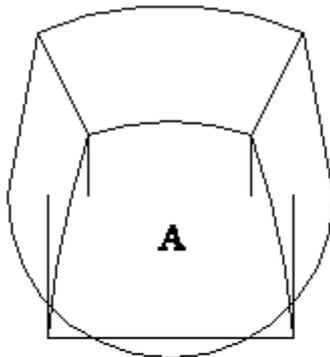
- Read a control drawing
- Create and edit curves
- Create and edit surfaces
- Make a solid feature



1. Open SURFACE LAB 4-02.PAR in the C:/Surfacing/Parts folder.



Control Drawings were discussed in module 1. For this activity the control drawing is provided. You will use the sketches in the control drawing to build curves and surfaces.



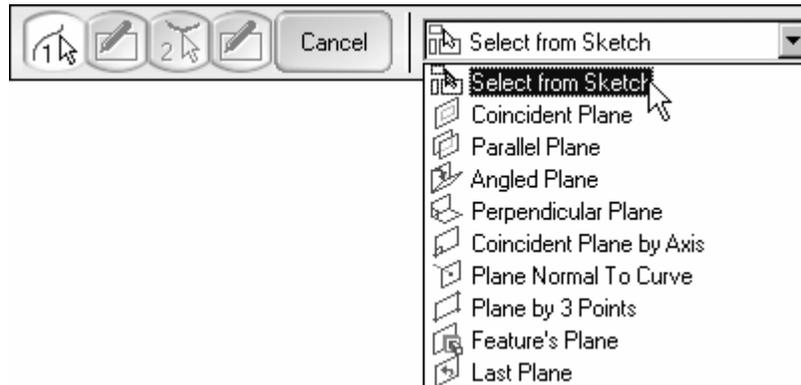
Control Sketches (A=Right View, B=Front View, C=Top View)

2. You will begin by constructing the top surface.

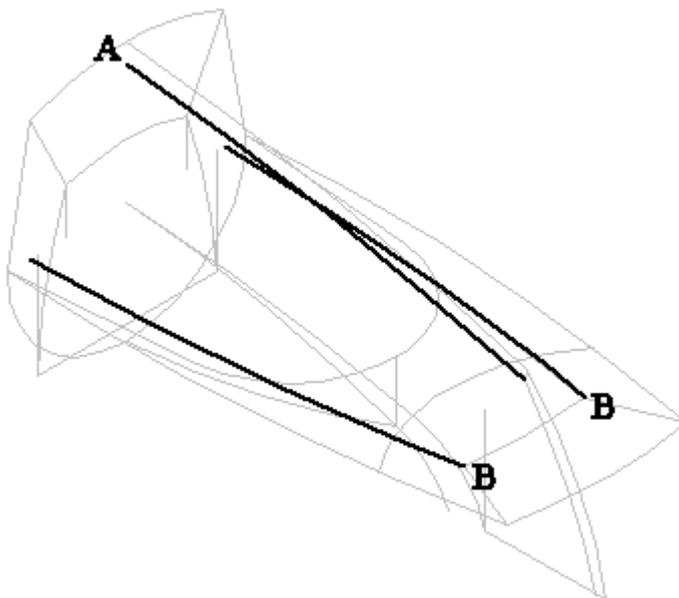
- Create intersection curves to be used to develop the top surface. On the Surfacing toolbar, click the Cross Curve command.



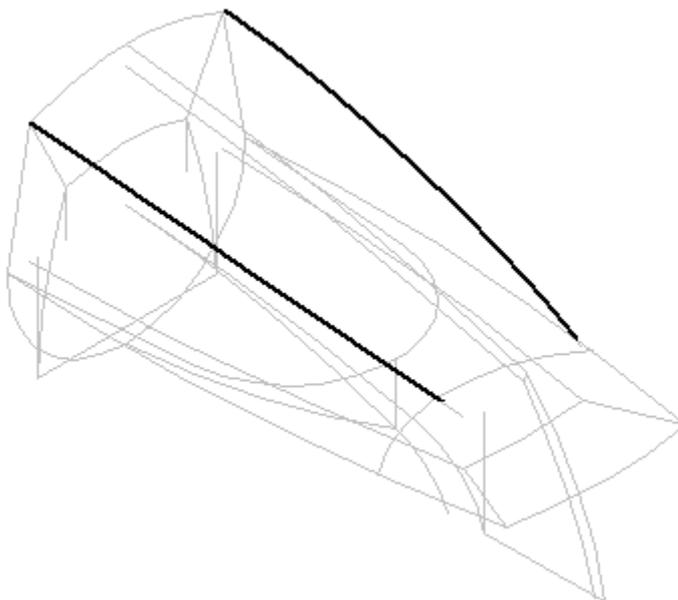
- On the ribbon bar, click the Select from Sketch option.



- Select sketch element (A) and then click the Accept button. Select both sketch elements (B) and then click the Accept button. Click Finish.

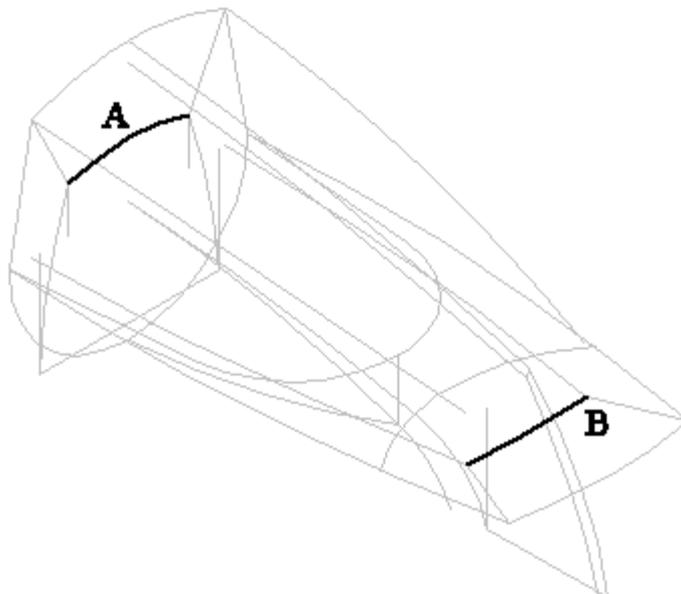


**Cross Curve result**

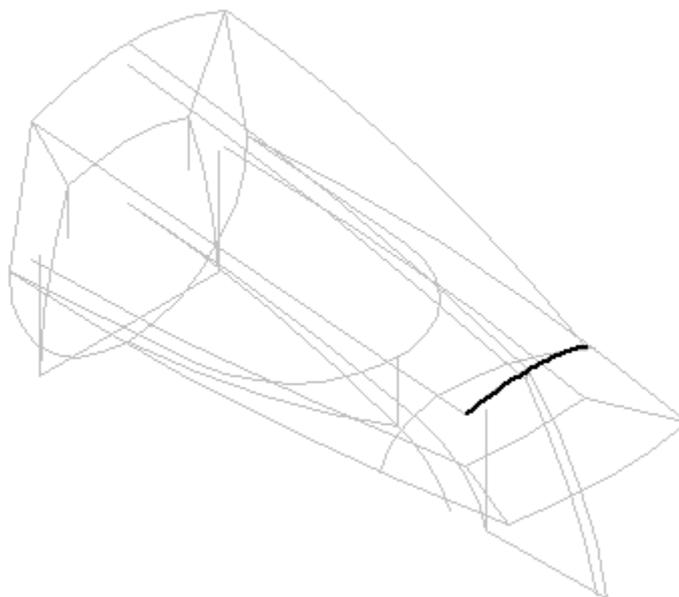


- Construct another cross curve. Click the Cross Curve command.

- Click **Select from Sketch**. Select sketch element (A) and then click the **Accept** button. Select sketch element (B) and then click the **Accept** button. Click **Finish**.



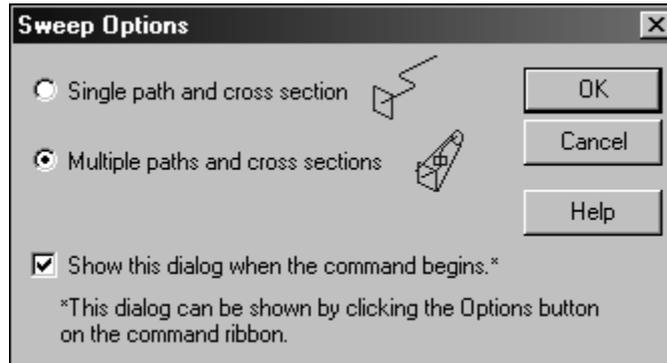
**Cross Curve result**



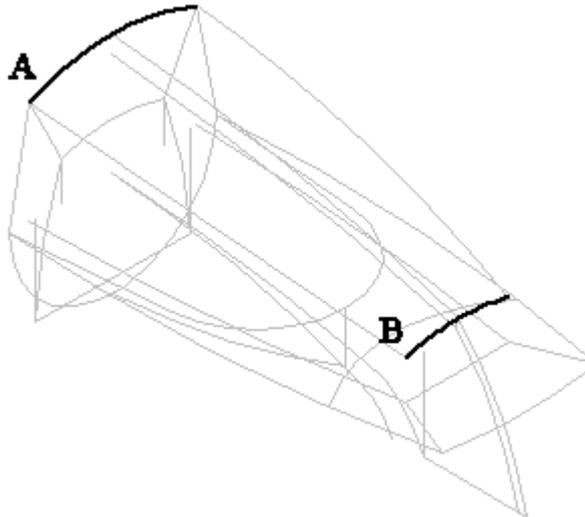
- Create the top surface. On the Surfacing toolbar, click the **Swept Surface** command.



- Select the Multiple paths and cross sections option.



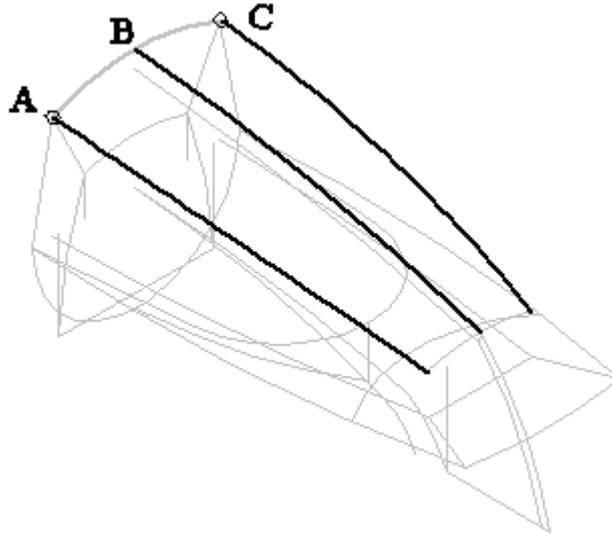
- For the Path Step, select path (A) and click the Accept button or click the right mouse button. Select path (B) and click the Accept button or click the right mouse button.



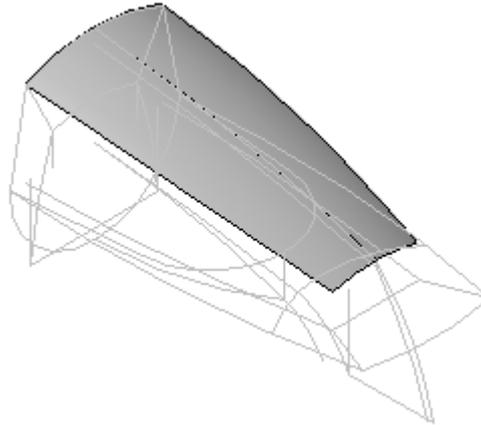
- On the Swept Surface ribbon bar, click Next to proceed to the Cross Section Step.



- Select cross section (A) and click the right mouse button. Select cross section (B) and click the right mouse button. Select cross section (C) and click the right mouse button.

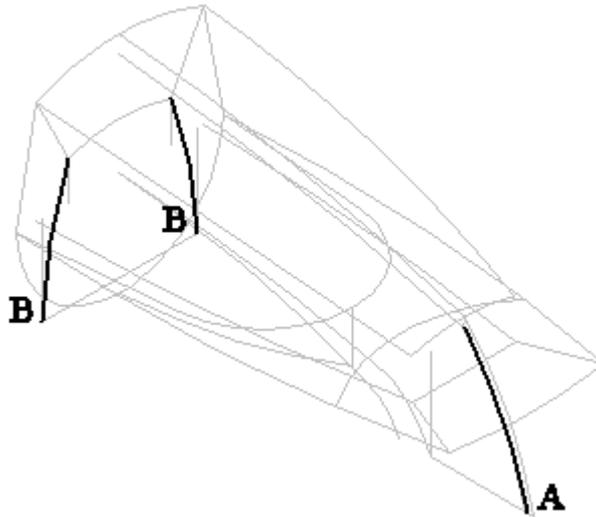


- On the ribbon bar, click Preview and then Finish.

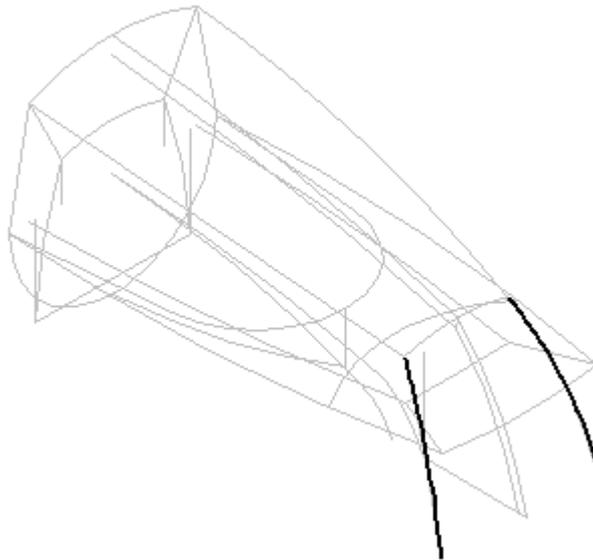


3. You will now create intersection curves to be used to develop the front surface.
  - Click the Cross Curve command.
  - On the ribbon bar, click Select from Sketch.

- Select sketch element (A) and then click the Accept button. Select both sketch elements (B) and then click the Accept button. Click Finish.

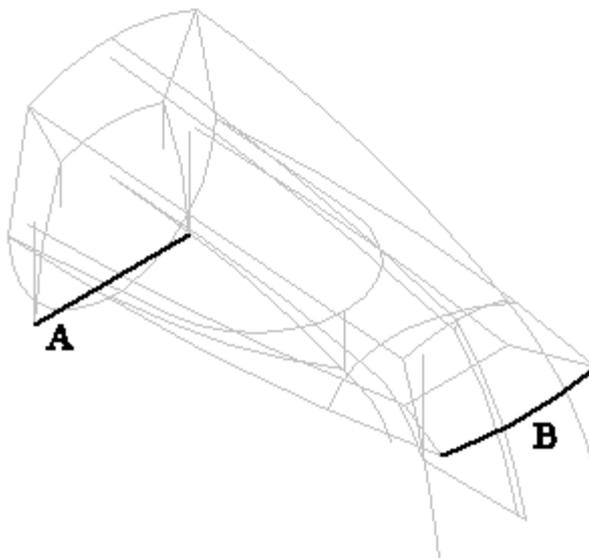


**Cross Curve result**

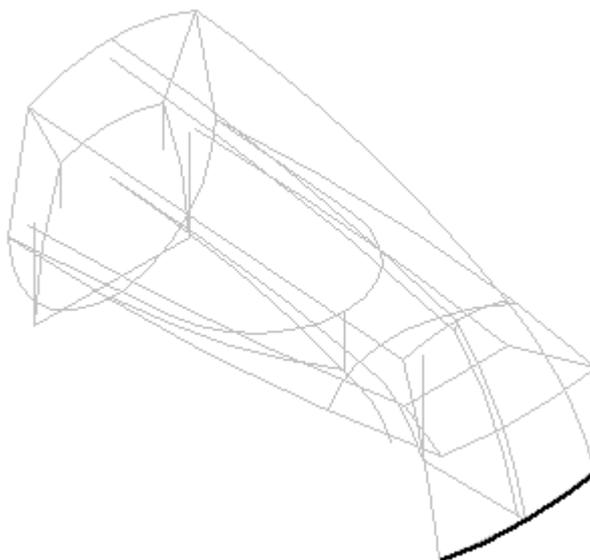


- Construct another cross curve. Click the Cross Curve command and on the ribbon bar, click Select from Sketch.

- Select sketch element (A) and then click the Accept button. Select sketch element (B) and then click the Accept button. Click Finish.

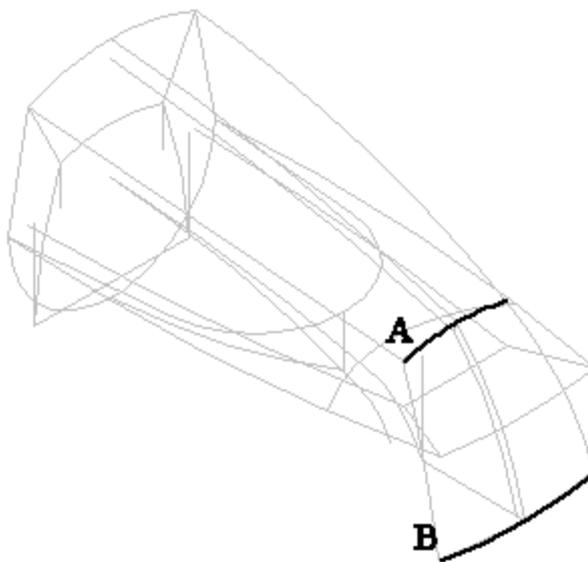
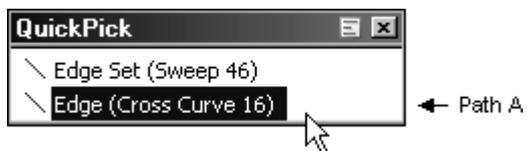


**Cross Curve result**



- Create the front surface. Click the Swept Surface command and select the Multiple paths and cross sections option.

- Select path (A) and click the right mouse button. Use QuickPick to select path (A). Make sure you select the cross curve element. Select path (B) and click the right mouse button.

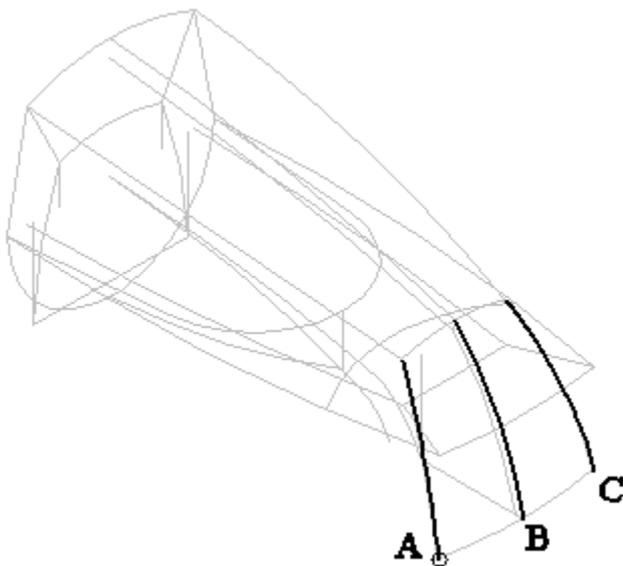


- Click Next to define the cross sections.

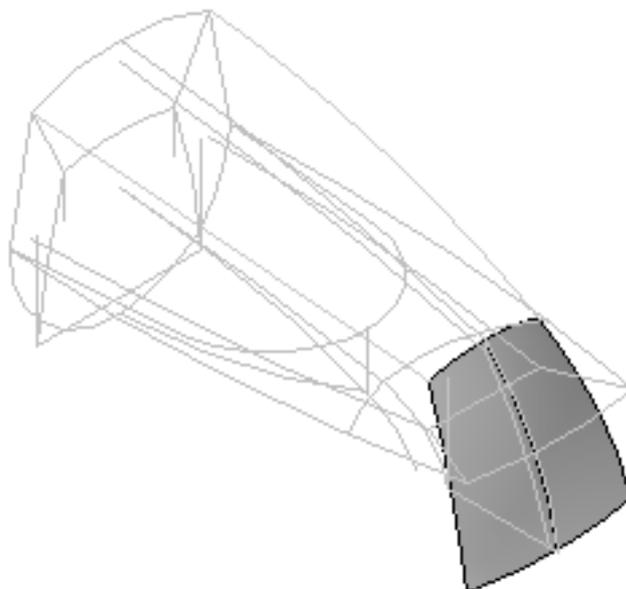
- Set the Select filter to Single. Select cross section (A) and click the right mouse button.



- Select cross section (B) and click the right mouse button.
- Select cross section (C) and click the right mouse button.



- Click Preview and then Finish.

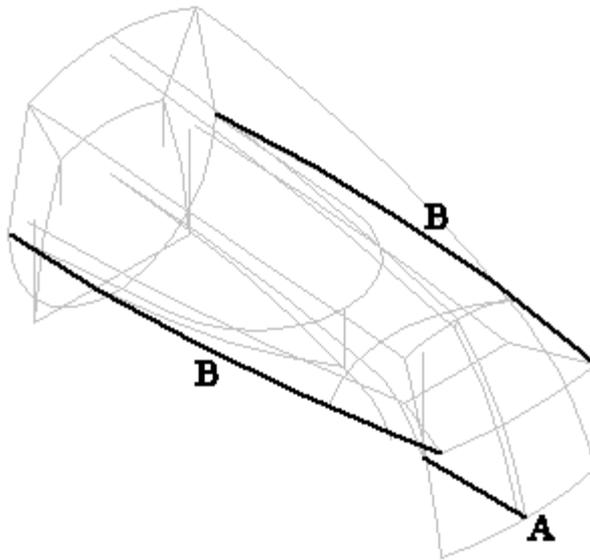


4. You will now create intersection curves to be used to develop the side surfaces.

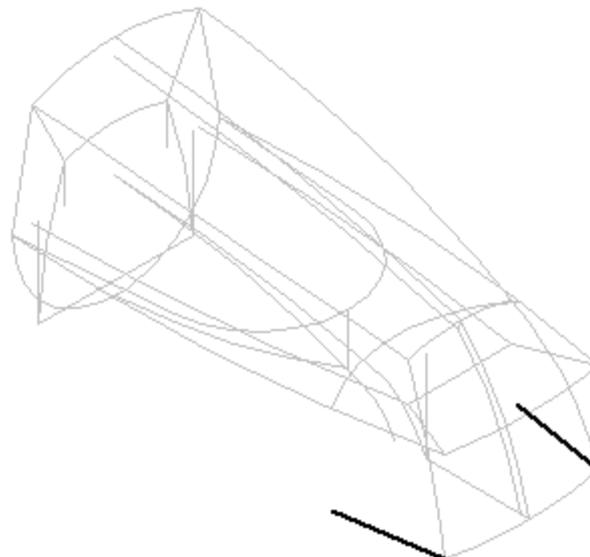
- Click the Cross Curve command and on the ribbon bar click Select from Sketch.
- Toggle the Select field to Single.



- Select sketch element (A) and then click the Accept button. Toggle the Select field to Single. Select both sketch elements (B) and then click the Accept button. Click Finish.



**Cross Curve result**

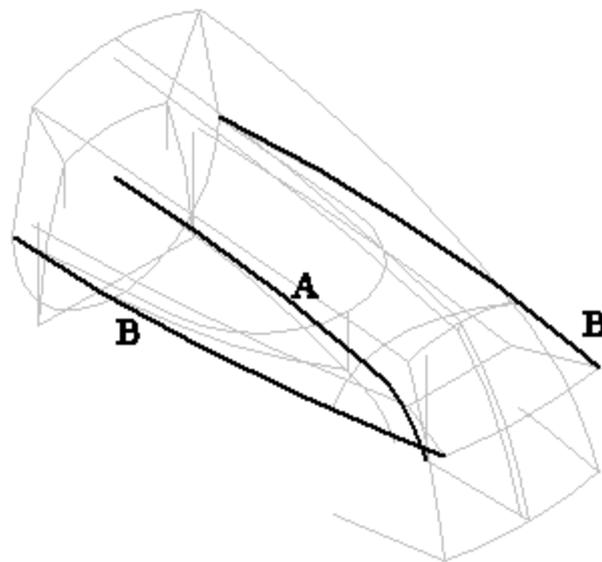


- Construct another set of cross curves. Click the Cross Curve command. Click Select from Sketch.
- On the ribbon bar, set the Select field to Single. Select sketch elements (A) and then click the Accept button.

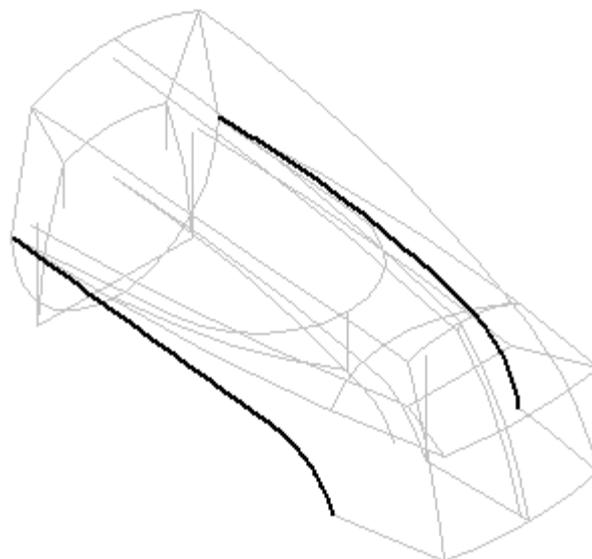


There are two elements in (A).

- On the ribbon bar, set the Select field to Single. Select both sketch elements (B) and then click the Accept button. Click Finish.



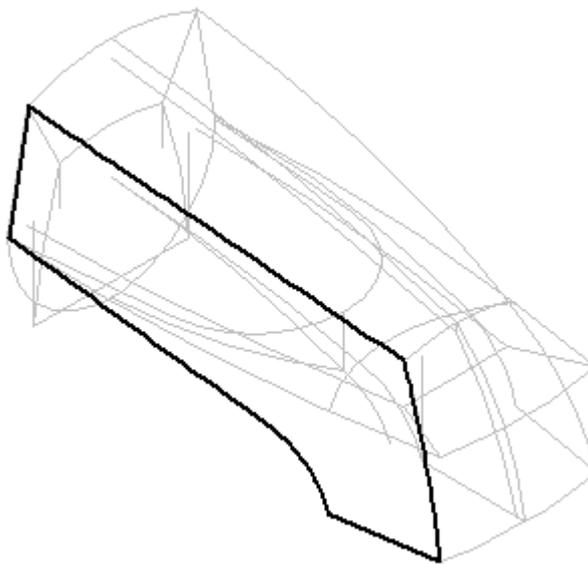
**Cross Curve result**



- Create the side surfaces using the cross curves generated in the previous steps. Click the Bounded Surface command.



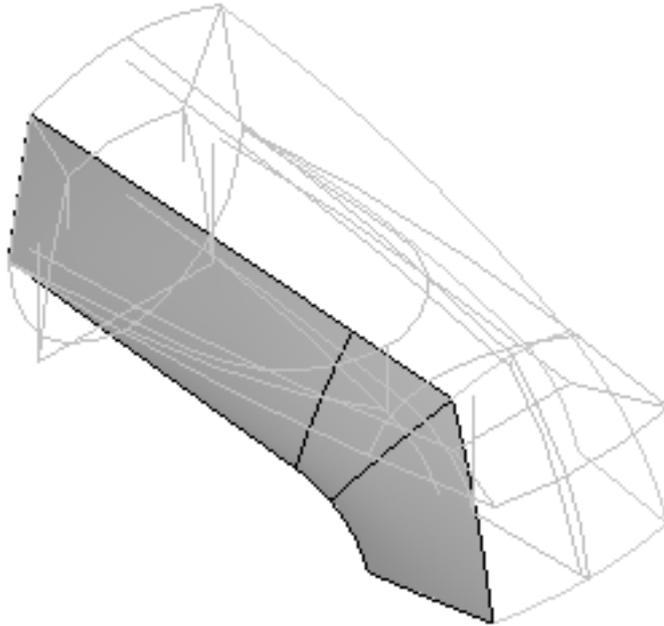
- Select the six edges as shown.



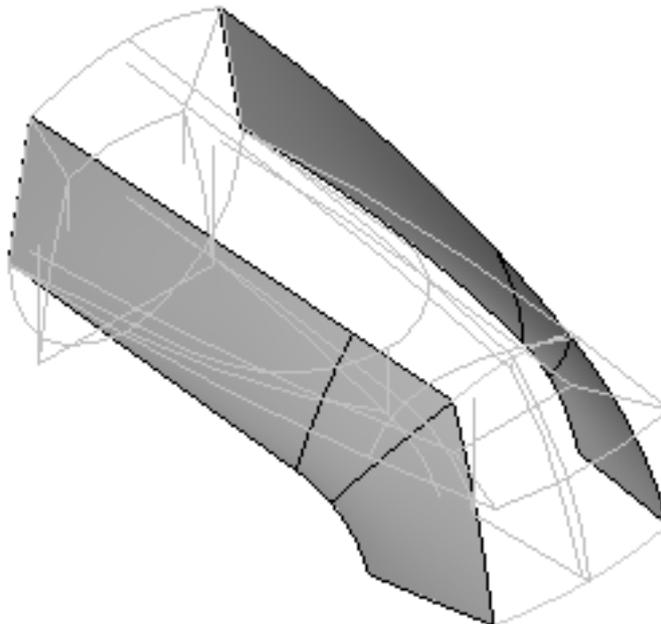
- On the ribbon bar, click the accept button. Click the Face Tangency option and make sure it is off.



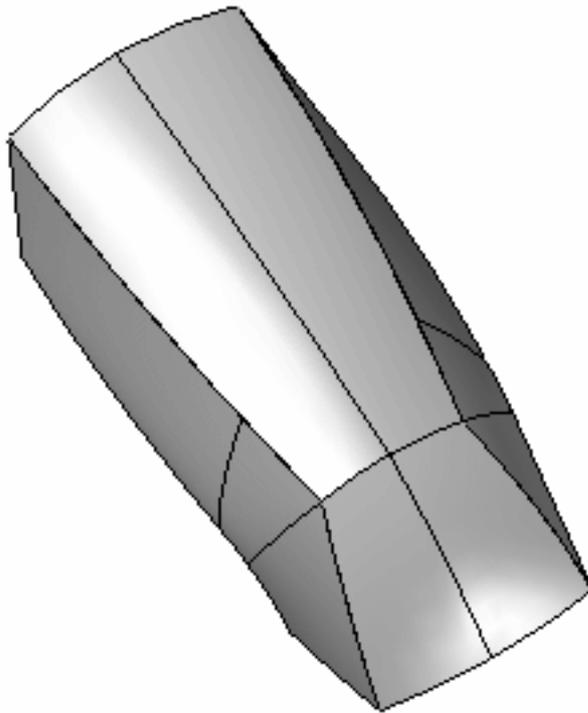
- Click Preview and Finish.



- Repeat the previous step to create the other side surface.



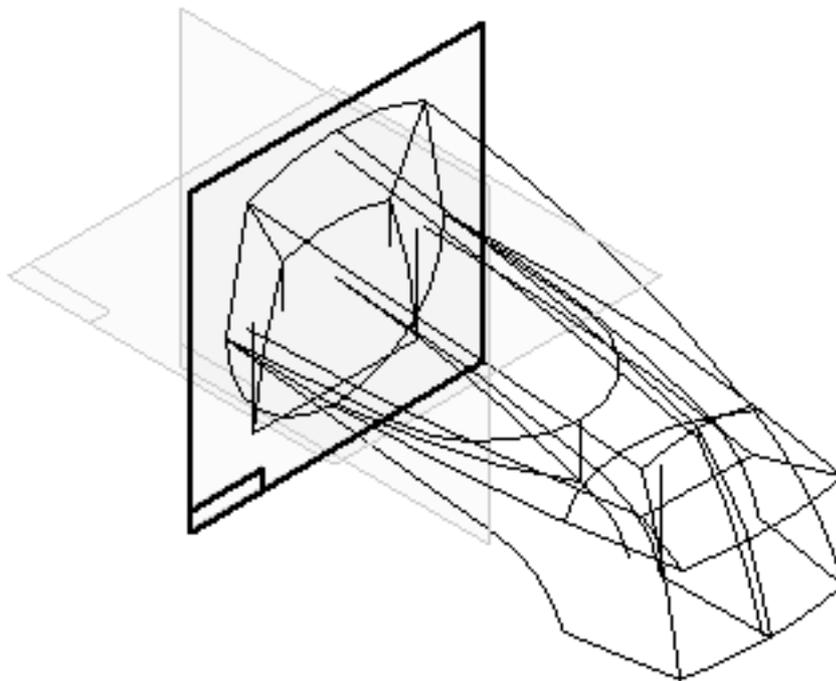
- Shown are the surfaces created so far.



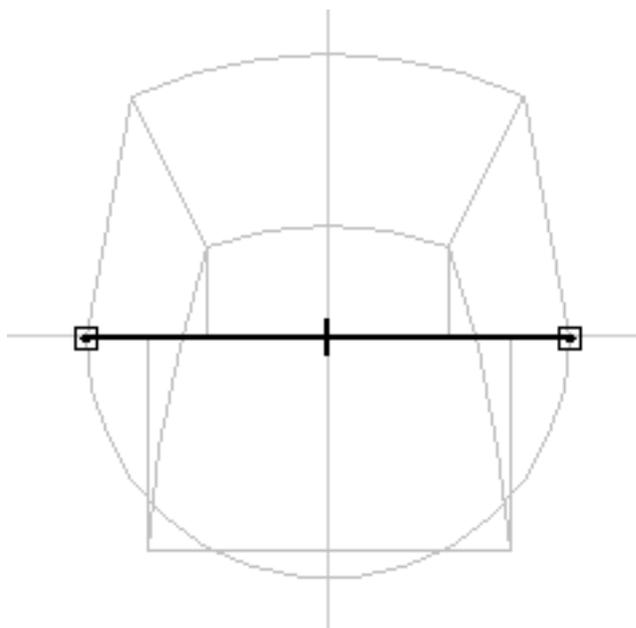
5. You will now create the bottom surface. To begin, you will need to create two sketches.
  - Show all reference planes. On the Features toolbar, click the Sketch command.



- Select the plane shown for the first sketch.

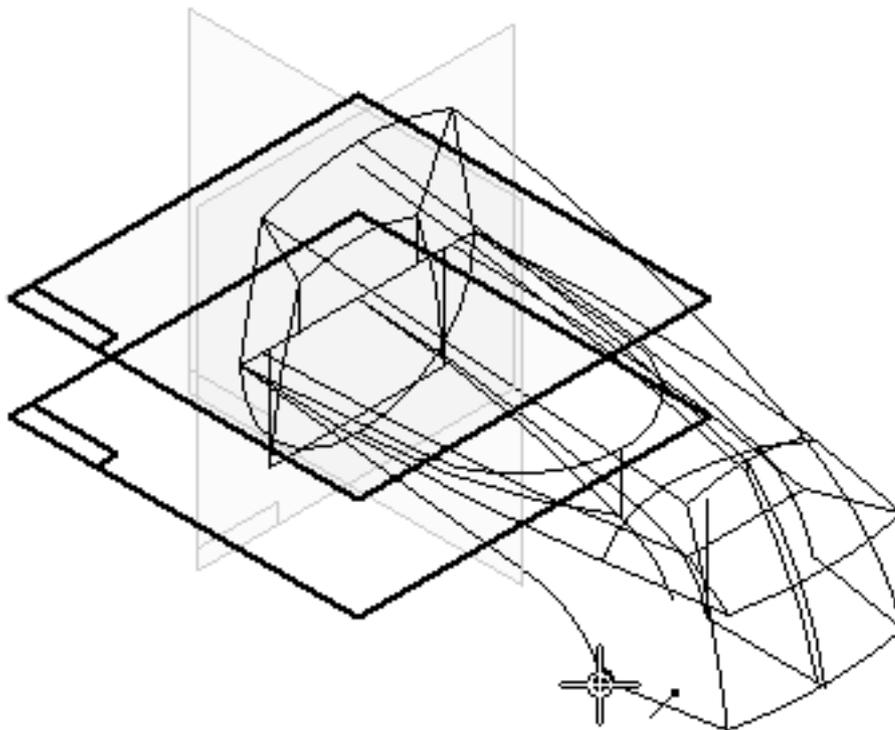


- Hide all construction surfaces.
- Draw the following sketch.

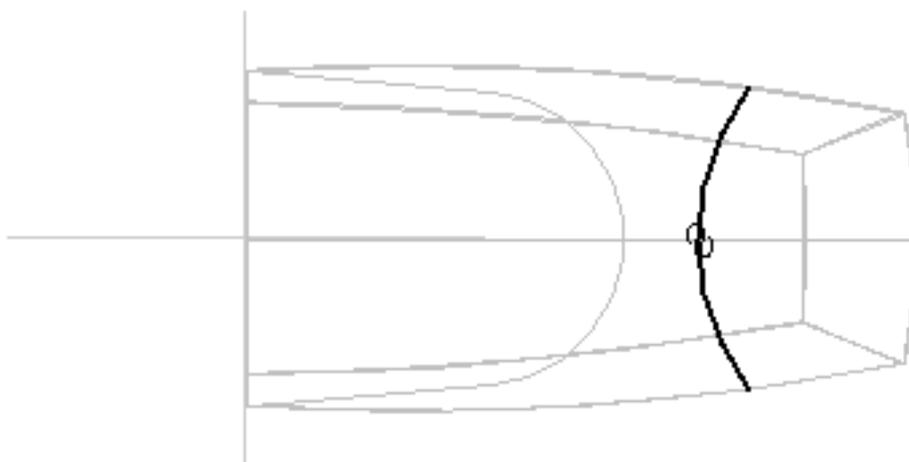


- Finish the sketch.

- For the second sketch, create the sketch on a parallel plane as shown. To define the distance, click the keypoint as shown.

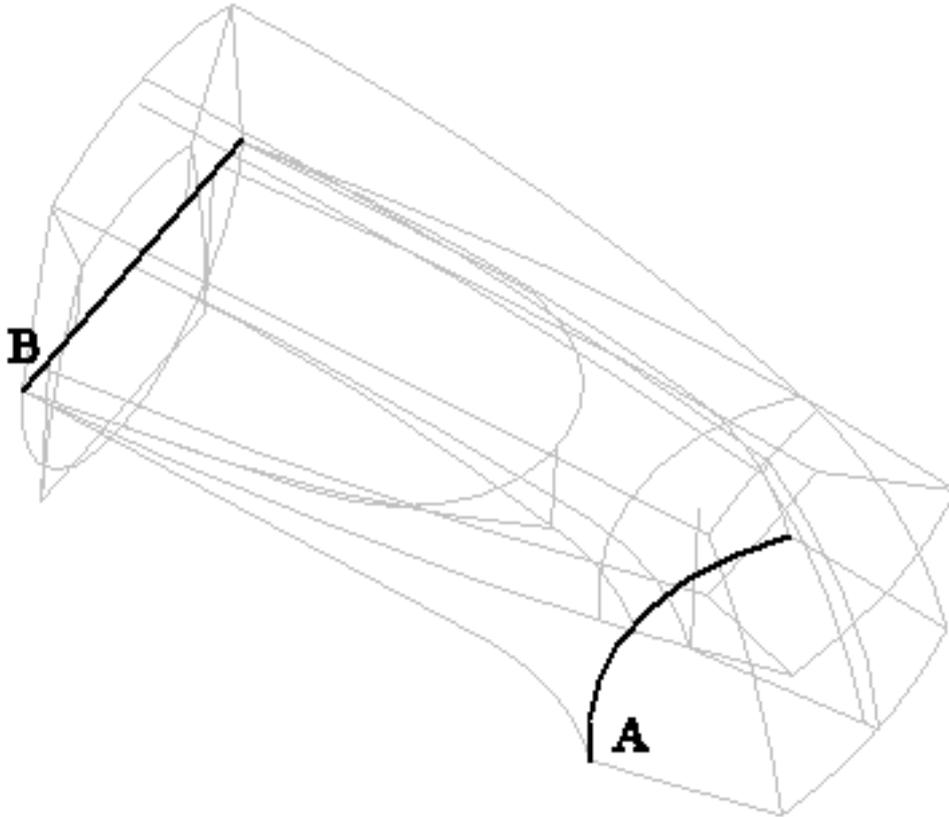


- Click the Include command and select the arc shown.



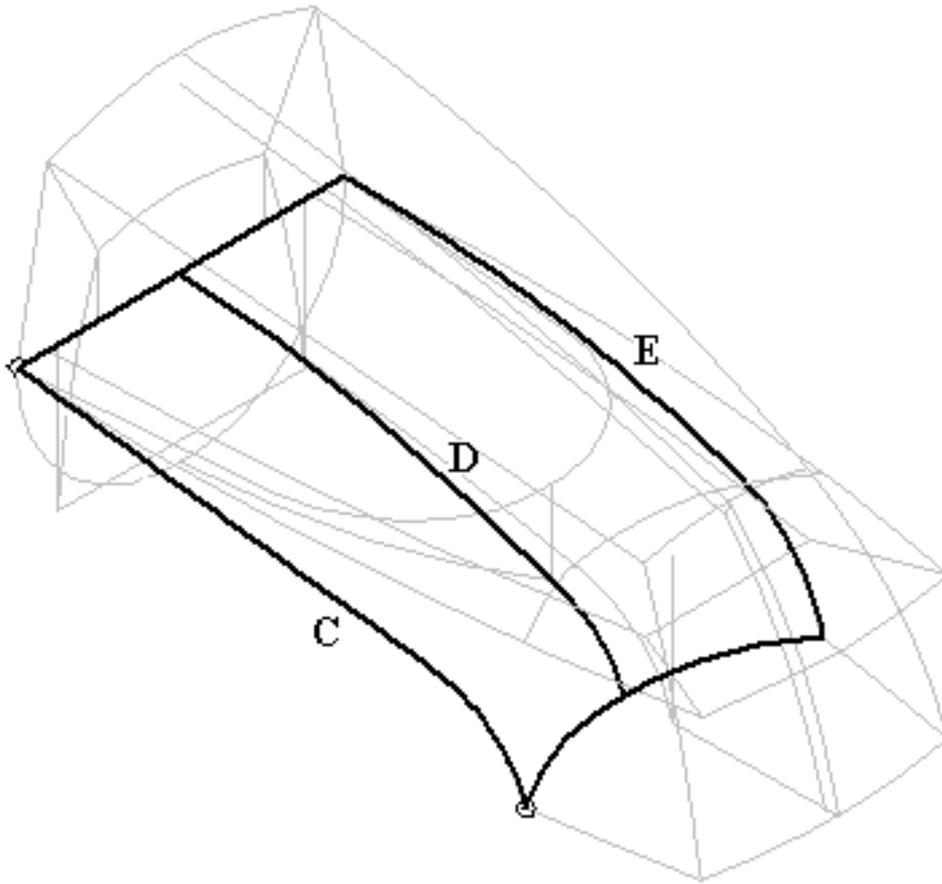
- Finish the sketch.
- To create the bottom surface, you will use the BlueSurf command. Click the BlueSurf command.

- Select cross section (A) and click the right mouse button. Select cross section (B) and click the right mouse button.

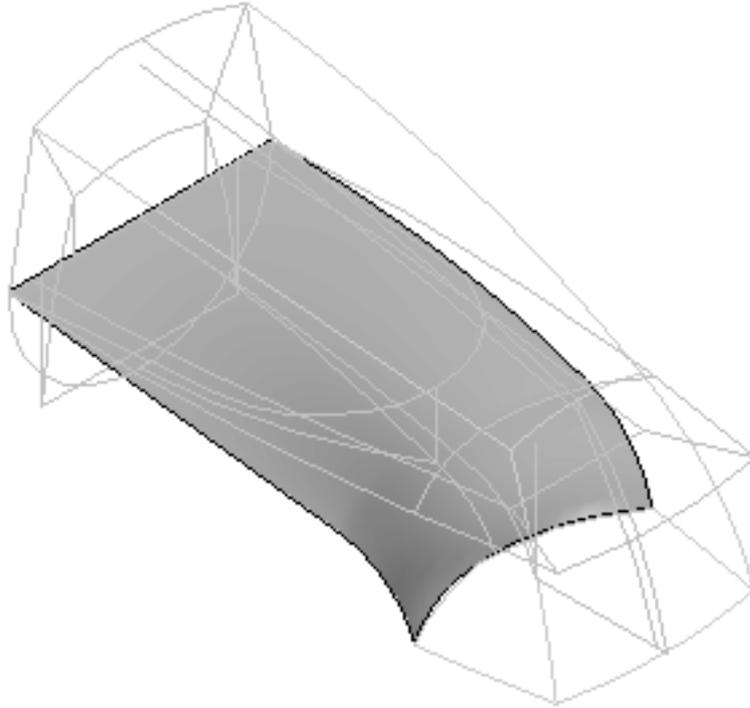


- On the ribbon bar, click the Guide Curve Step.
- Select guide curve (C) and click the right mouse button.
- Set the Select filter to “Single” to select the two elements in guide curve (D). Select guide curves (D) and click the right mouse button.

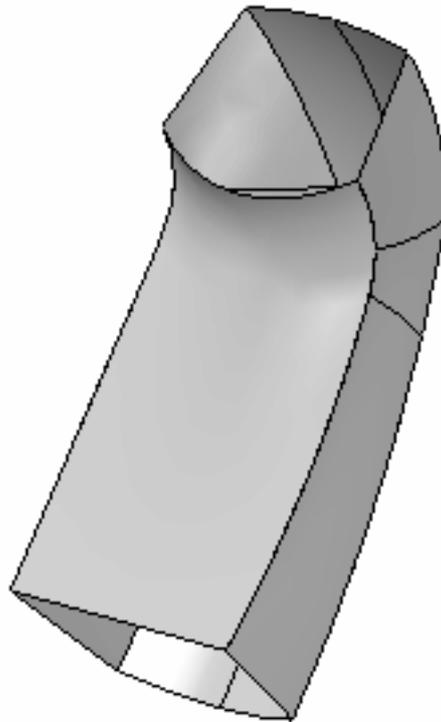
- Set the select filter to “Chain” to select guide curve (E). Select guide curve (E) and click the right mouse button.



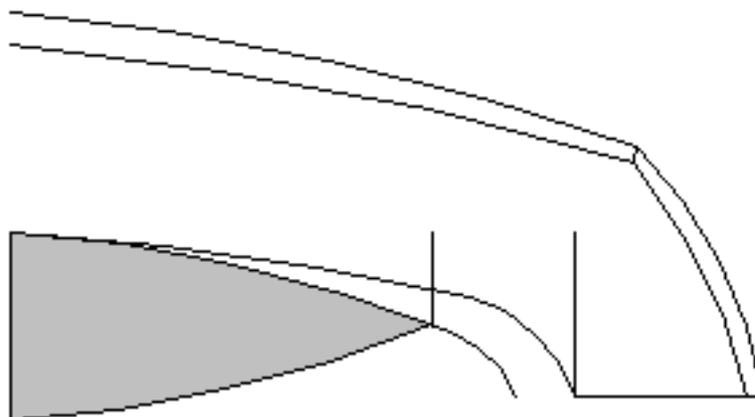
- Click Preview and Finish.



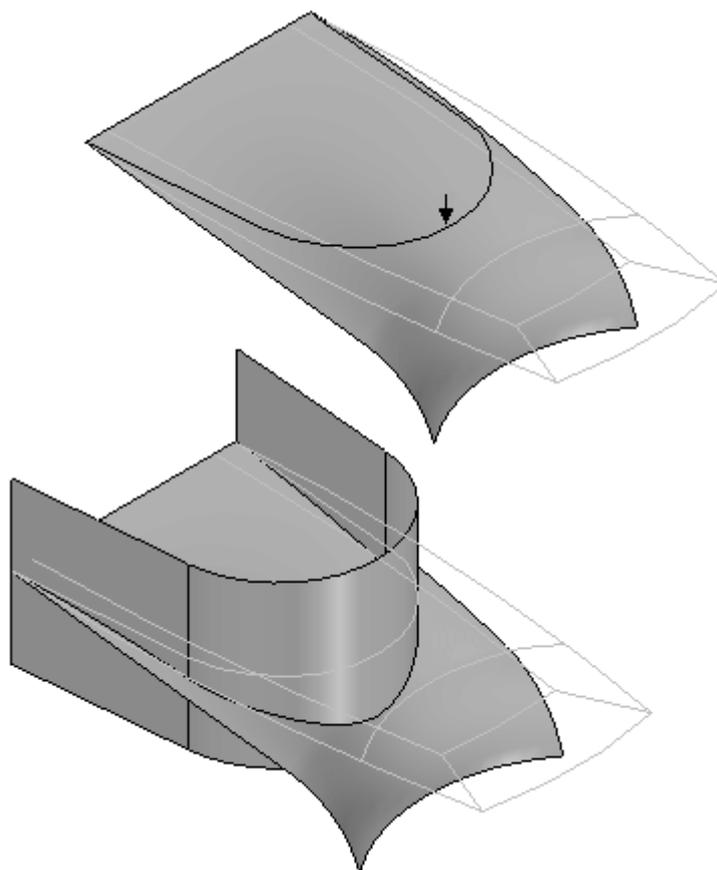
- Shown below are the surfaces created so far.



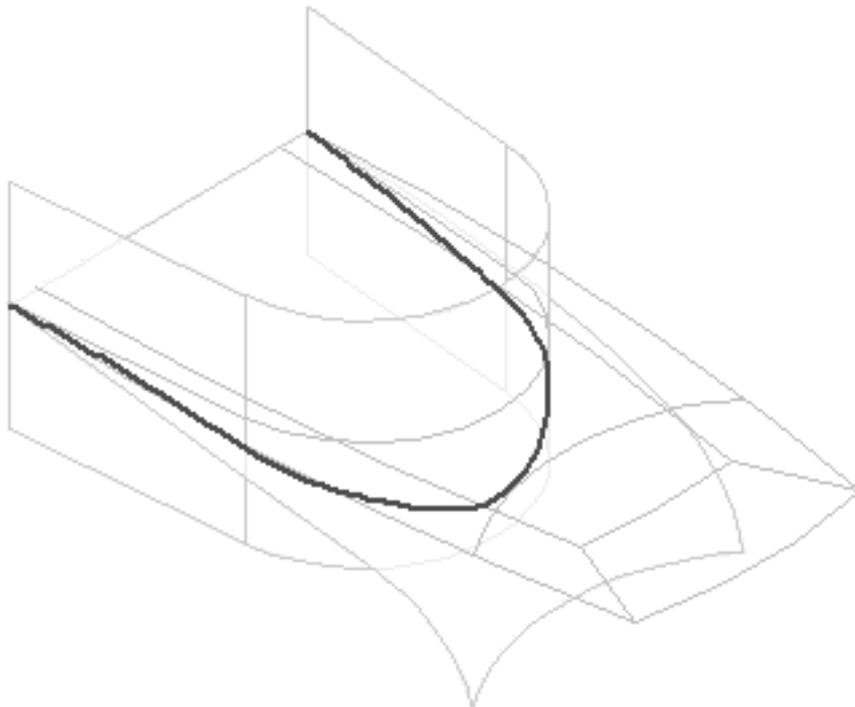
6. You will now add the surface as shown in the control sketch below.



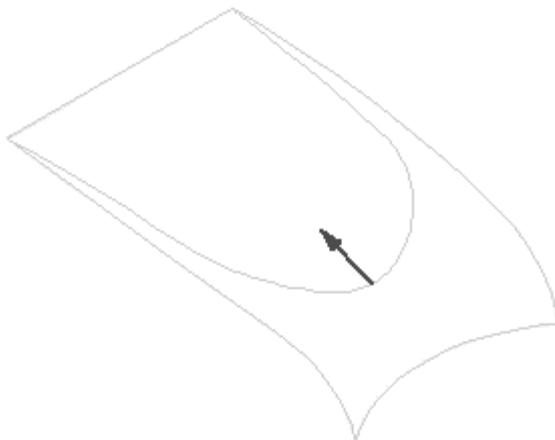
- To create this surface, an intersection curve is needed between the bottom bluesurf and an extruded surface created from the element shown in the control sketch. Create an extruded surface from the element shown in the image below.



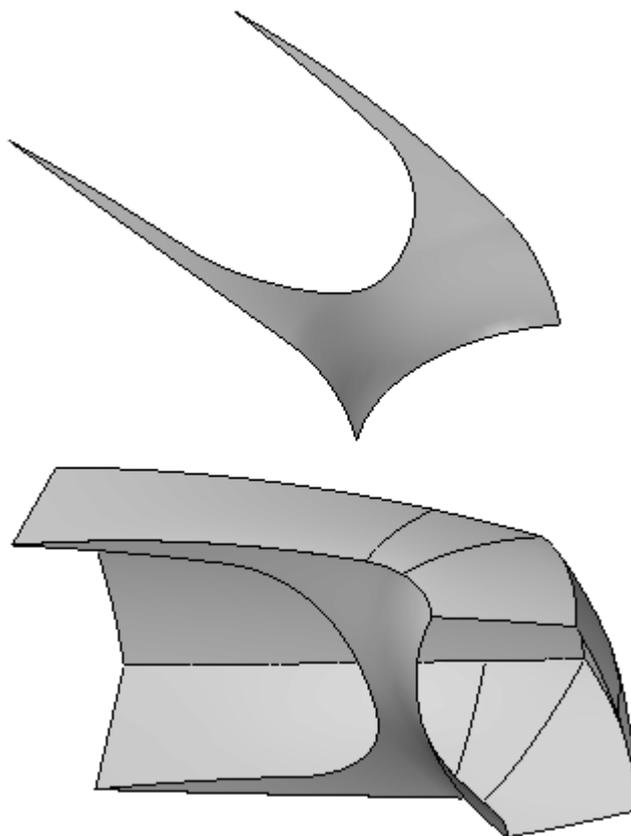
- Click the Intersection Curve command. Select the bluesurf and the extruded surface as the surface set to intersect. The intersection curve is shown in the image below.



- You will now trim the bottom surface using the intersection curve. Click the Trim Surface command. Select the bluesurf. Select the intersection curve and make sure the arrow points in the direction shown.

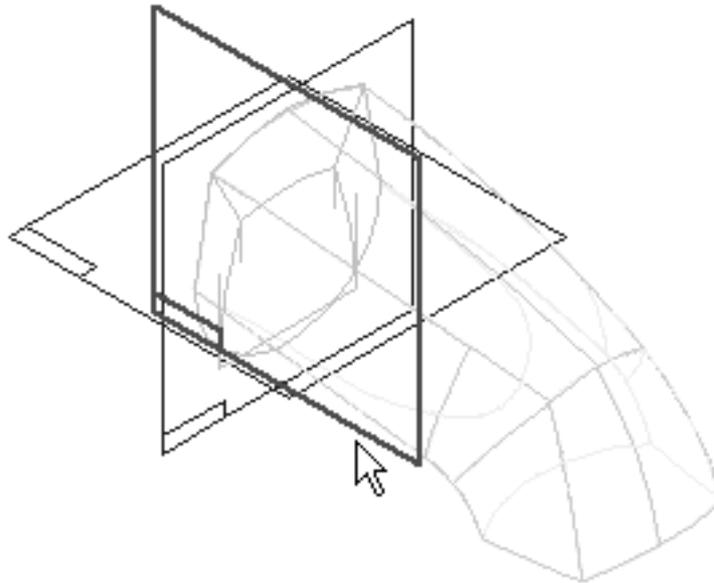


**Trim result**

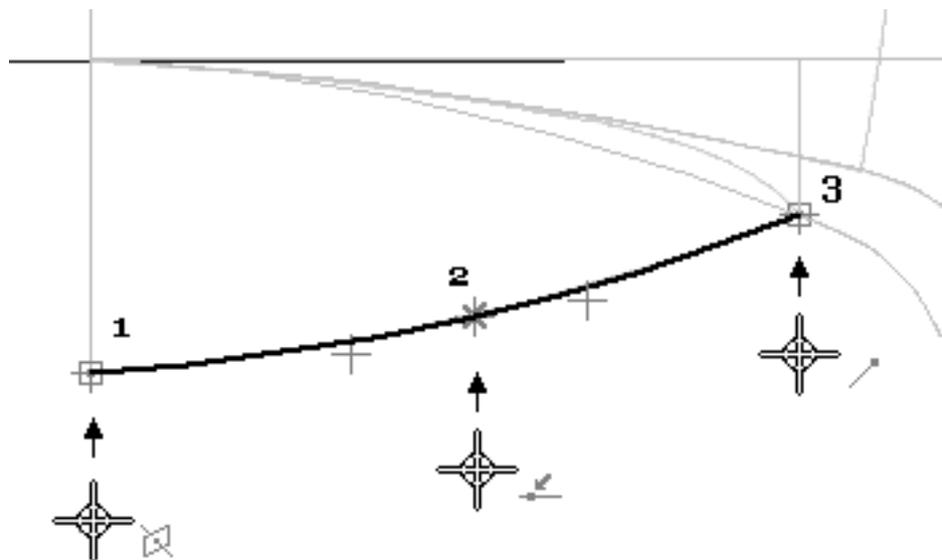


7. You will now create the final surface. You will now create a curve to control the shape of the final surface.

- Create a sketch on the plane as shown.

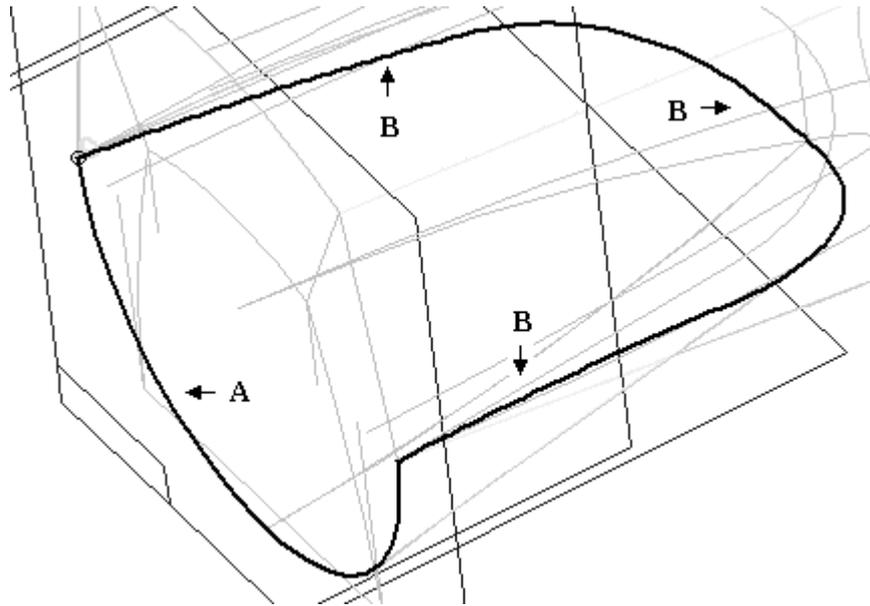


- Click the Curve command.
- Position cursor over area (1) until you get the pierce point display, and then click. This makes the first point of the curve connected to this pierce point.
- Click the second curve point with the point on display, and click.
- Position cursor over area (3) until you get the keypoint display, and then click. This makes the third point of the curve connected to this keypoint. Click the right mouse button to complete the curve.

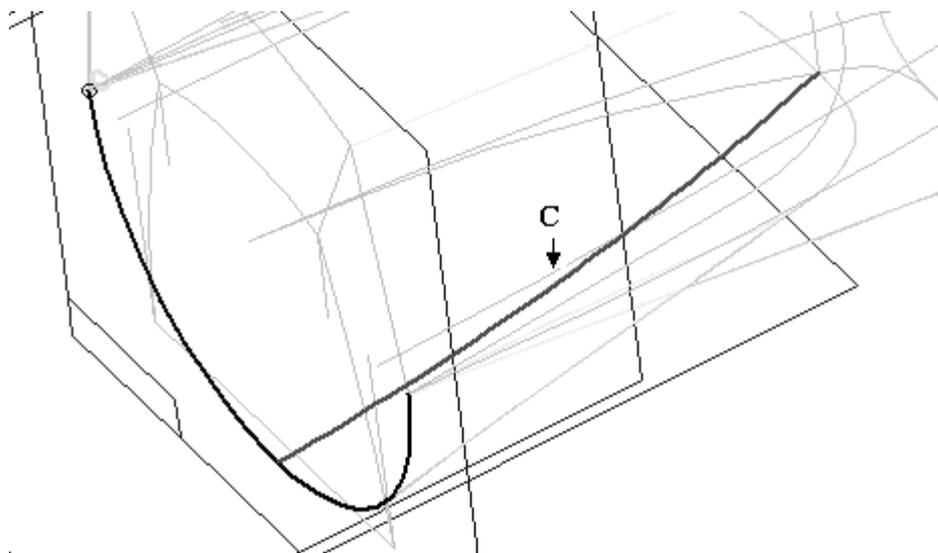


- Finish the sketch.

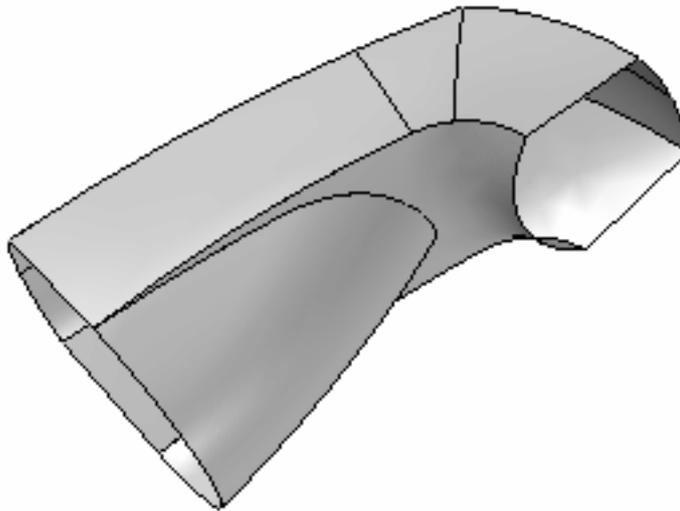
- You will now create an additional surface. Click the BlueSurf command.
- Select the cross sections shown. Set the select filter to “Single” in order to select the first cross section (A). Select cross section A and right-click. For the second cross section (B), set the select filter to “Single” in order to select the cross section. The second cross section (B) has (3) segments. Select cross section B and then right-click.



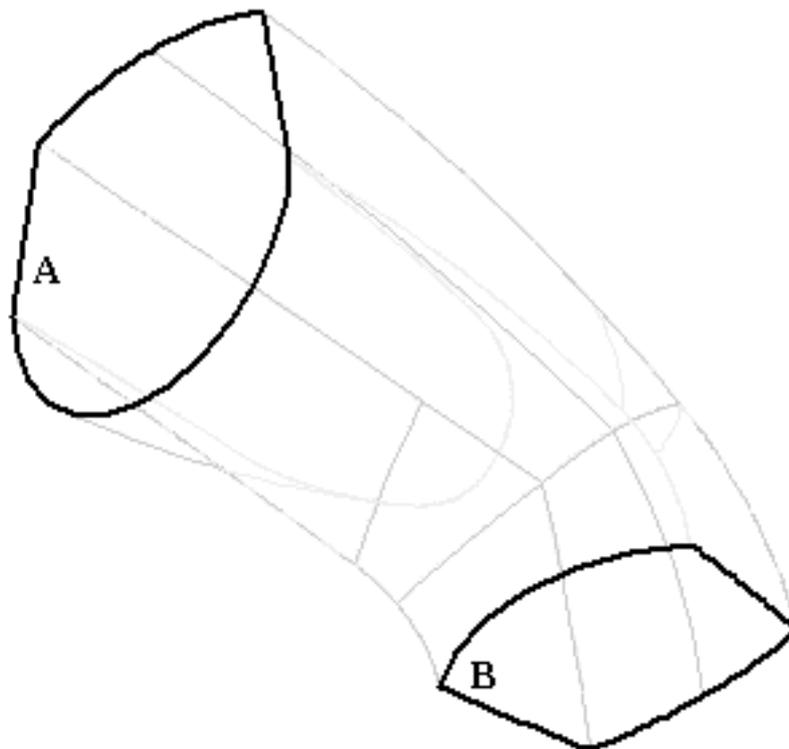
- After selecting cross sections (A) and (B), click the Guide Curve Step. Select the curve (C) created in the previous step and then right-click. Make sure you select the correct cross section C. Observe the status in the upper right portion of the part window. The name should match the curve name in EdgeBar of the last sketch created.



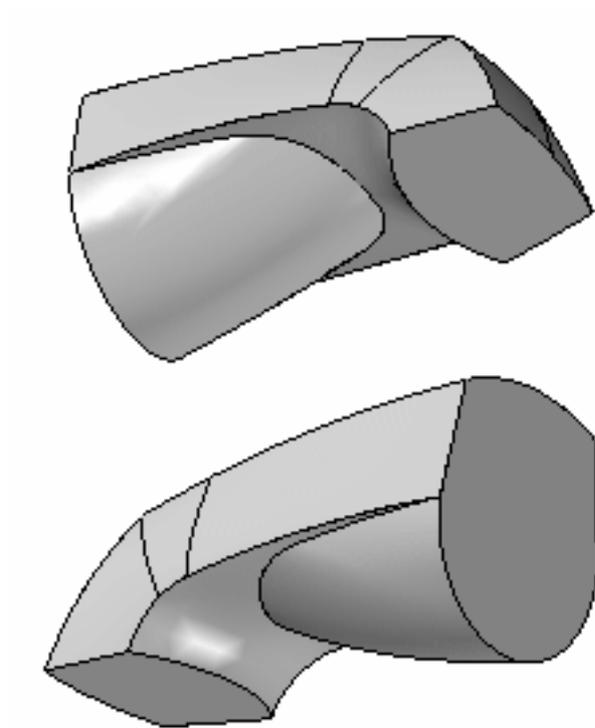
- Click Preview and then Finish.



8. The spout is finished except for capping the ends. You will now cap the ends. Click the Bounded Surface command. Select the edges shown for (A) and make sure face tangency is turned off. Select the edges shown for (B) and make sure face tangency is turned off.

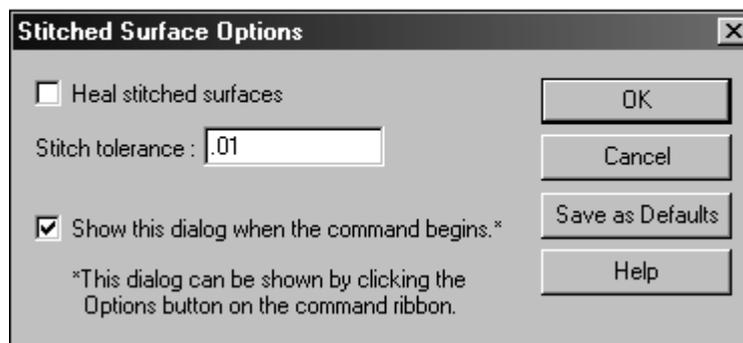


- The surface model is complete.

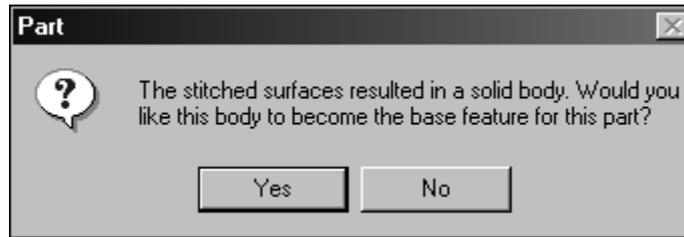


9. The surface model can now be converted to a solid base feature. The first step is to stitch all surfaces together.

- Click the Stitched Surface command and type .01 for the Stitch tolerance.



- Select all surfaces and then click the Accept button. A dialog box is displayed as shown.



- Click Yes.
- The stitched surface entry in EdgeBar is now grayed out and a Solid Body Feature is created.



10. This completes the spout lab activity.

You rarely get the surface you want from the initial surface creation method. You will probably need to manipulate the surface by: adding boundaries, creating an offset, extending, rounding, etc.

Solid Edge provides several commands for you to use to develop the final surface shape. A good understanding of these tools will help you to master surface modeling.



## Activity 3 – Parting Split and Parting Surface

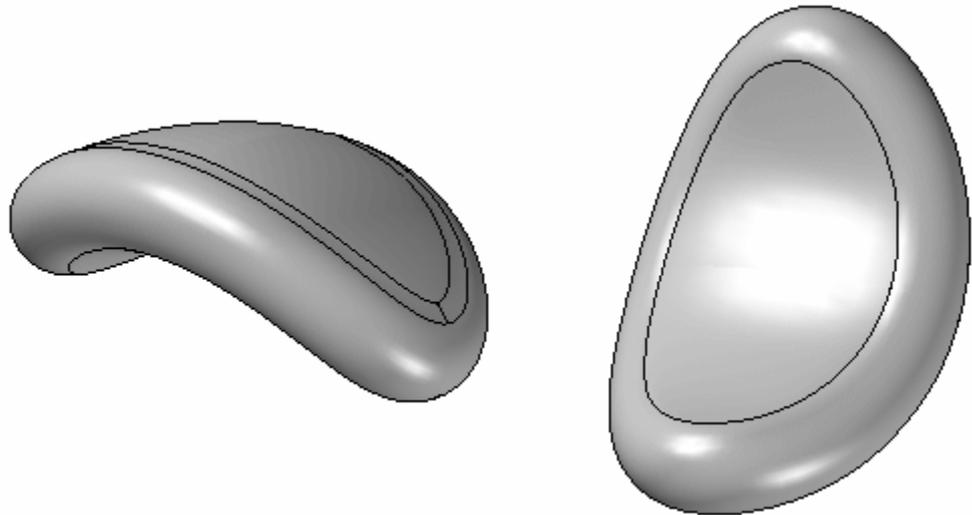
### Overview

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

### Objectives

In this activity, you will create a simple mold. You will use the existing model PAD.PAR to develop two mold halves. This activity covers the following topics:

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



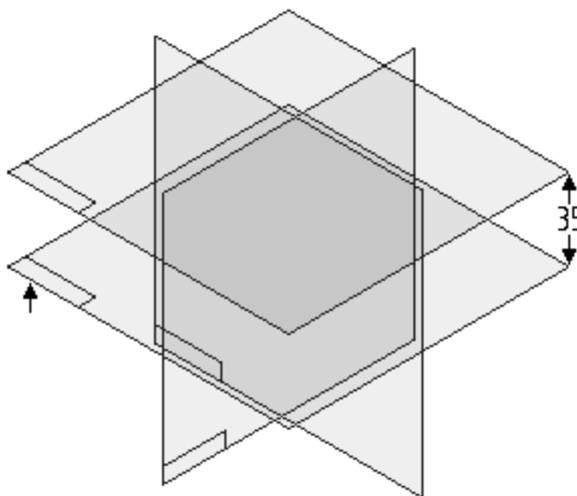
### PAD.PAR



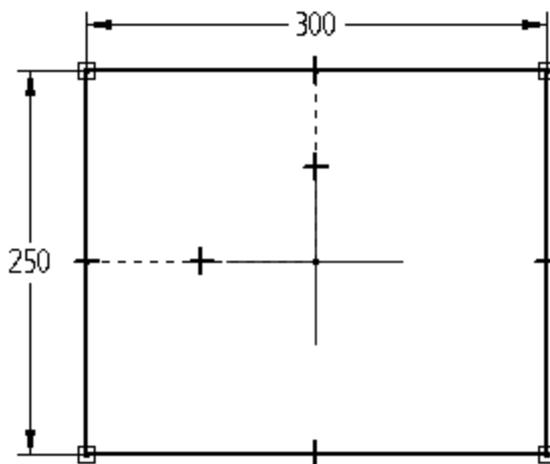
Files for this activity are located in the C:/Surfacing/Parts folder.

1. Create a new metric part file. 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.
2. Create the following sketch.

- Select the plane shown by the arrow. Create a parallel plane 35mm as shown.

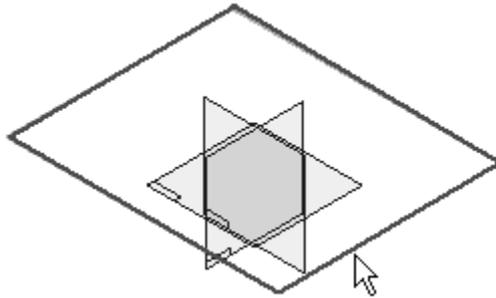


- Draw the following profile. Make sure the midpoints of the line elements are horizontal/vertical aligned to the center of the reference planes.

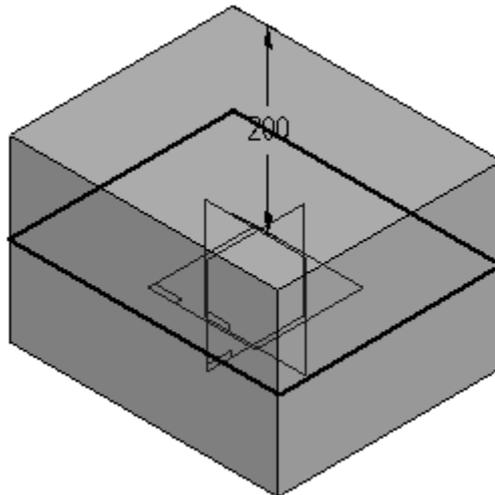


- Click Finish and then click Finish again.
3. You will now create the core part.
- Click the Protrusion command.
  - Click the Select from Sketch option.

- Select the sketch shown and click the Accept button.



- Click the Symmetric Extent button and type 200. Click Finish.

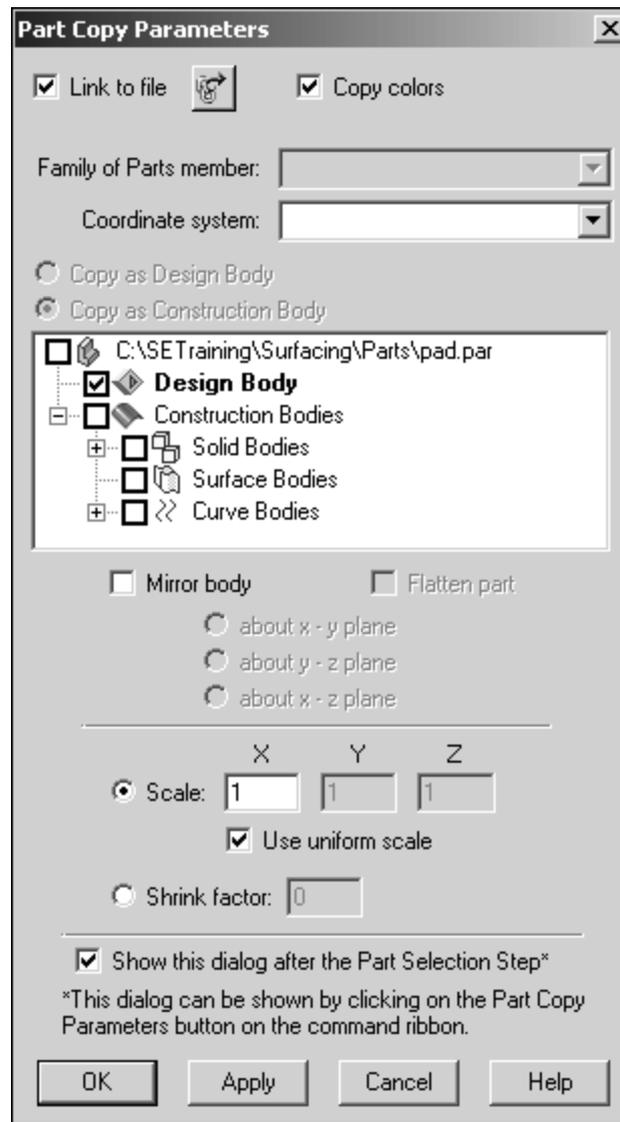


4. You will now create the cavity.

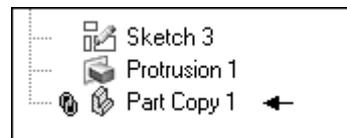
- On the Main menu, click Insert>Part Copy.



- In the Select Part Copy dialog box, set the folder to C:/Surfacing/Parts. Select PAD.PAR and click Open.
- On the Part Copy Parameters dialog box, make sure Design Body is checked and then click OK.



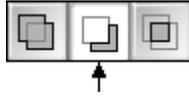
- On the ribbon bar, click Finish. Notice the Part Copy is now listed in EdgeBar.



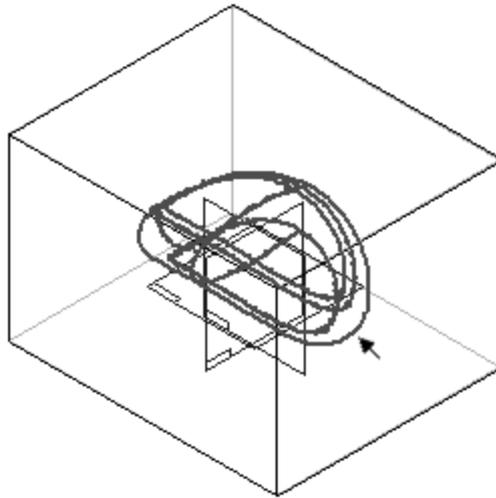
- To create the cavity, you will now perform a Boolean difference to remove PAD.PAR from the core. On the Surfacing toolbar, click the Boolean Feature command.



- On the ribbon bar, click the Subtract button.



- Click the Part Copy and then click the Accept button.



- Click Finish. Notice the Boolean listed in EdgeBar.



- The core now has a cavity of PAD.PAR.

5. You will now create a parting split curve on the cavity.

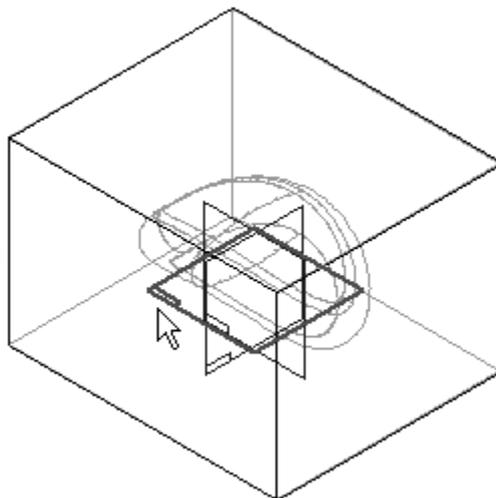


A parting split curve is derived from silhouette edges as viewed normal to a selected plane. The parting split curve is used to define where a part needs to split to allow for removal from a mold.

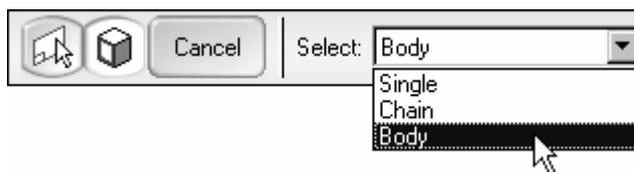
- On the Surfacing toolbar, click the Parting Split command.



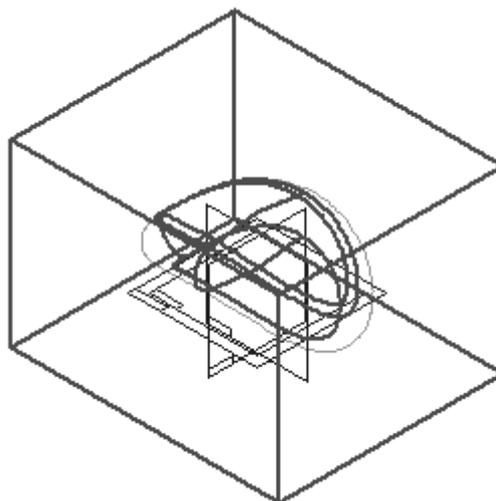
- Select the plane shown.



- On the ribbon bar, set the Select filter to Body.

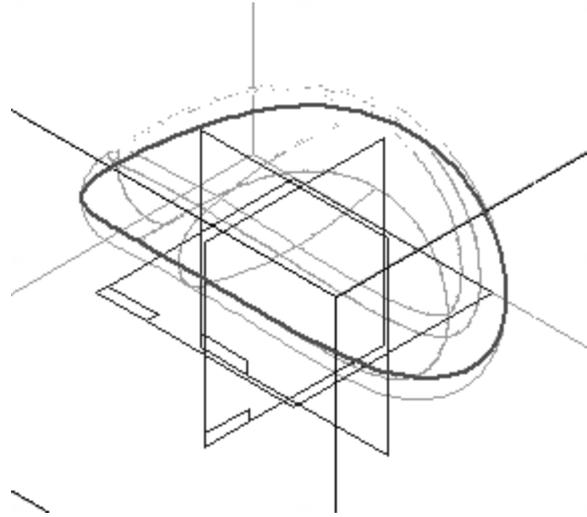


- Select the body as shown.



- Click the Accept button and then click Finish.

- The parting split is highlighted for you to observe.

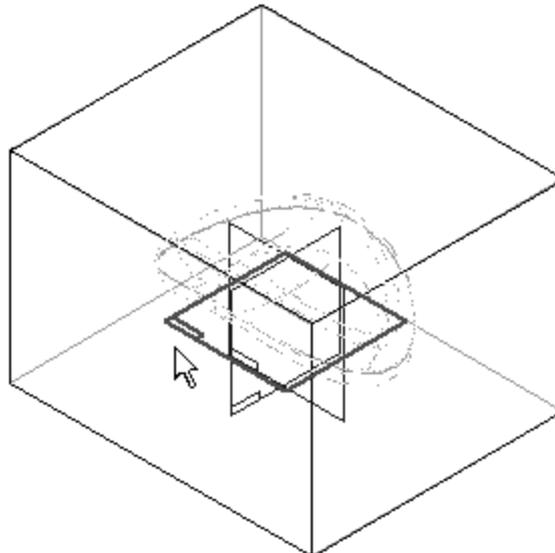


6. You will now create a parting surface using the parting split curve as input. The resulting parting surface will be used to split the core in a later step.

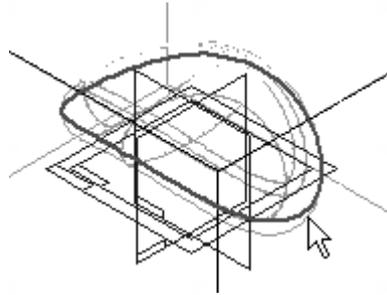
- On the Surfacing toolbar, click the Parting Surface command.



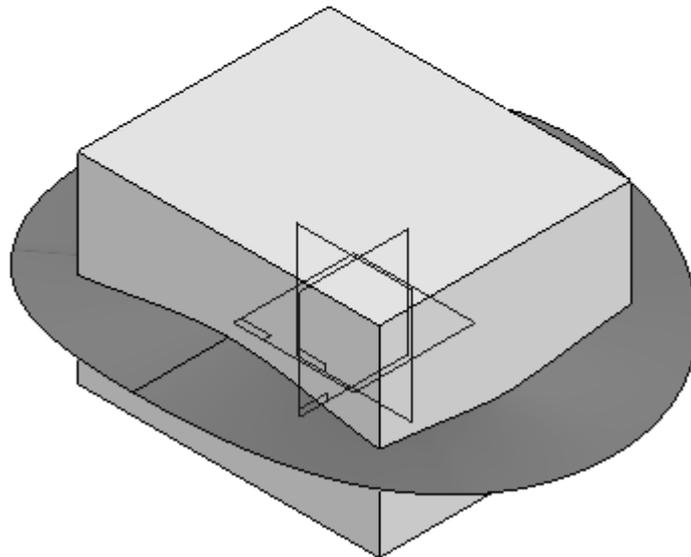
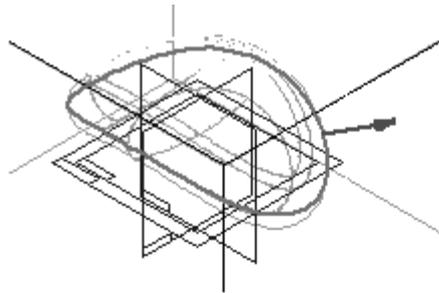
- Select the plane shown. The parting surface will use this plane as reference for all normal vectors in creating the surface.



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



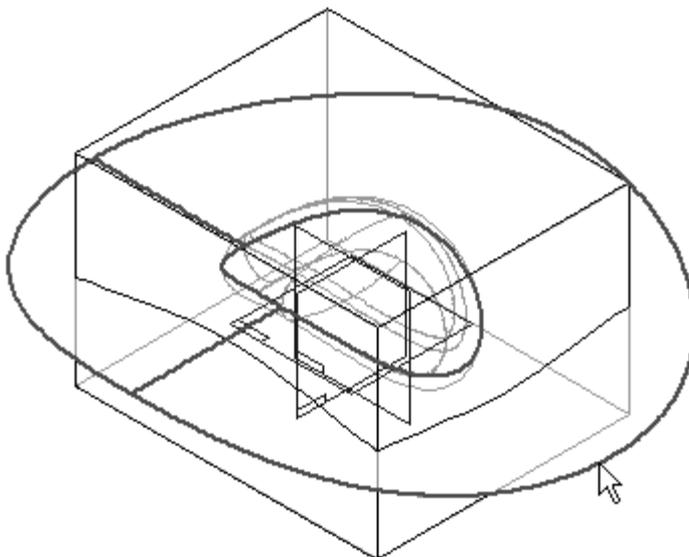
- On the ribbon bar, type 150 in the distance field and press ENTER.
- Position arrow (pointing outward) as shown and click. Click Finish.



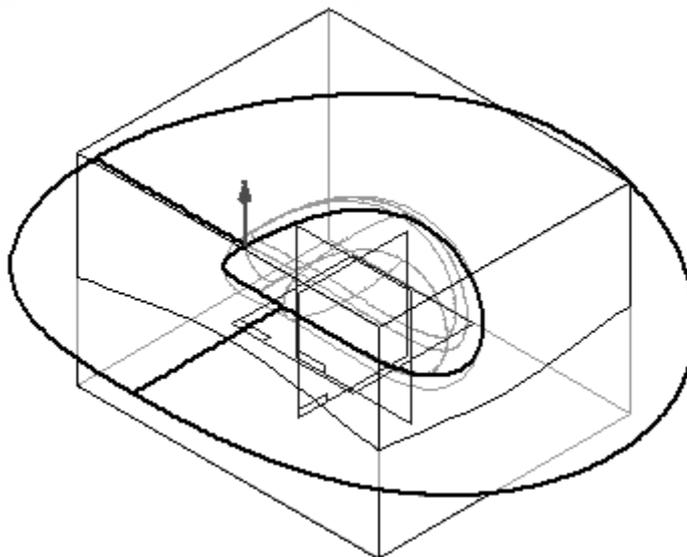
- Save the file as PAD CORE.PAR in the C:/Surfacing/Parts folder.
7. You will now split the core along the parting surface to create two mold halves. You will use the Divide part command.
- On the Surfacing toolbar, click the Divide Part command.



- Select the parting surface shown as the surface to be used to divide the core.



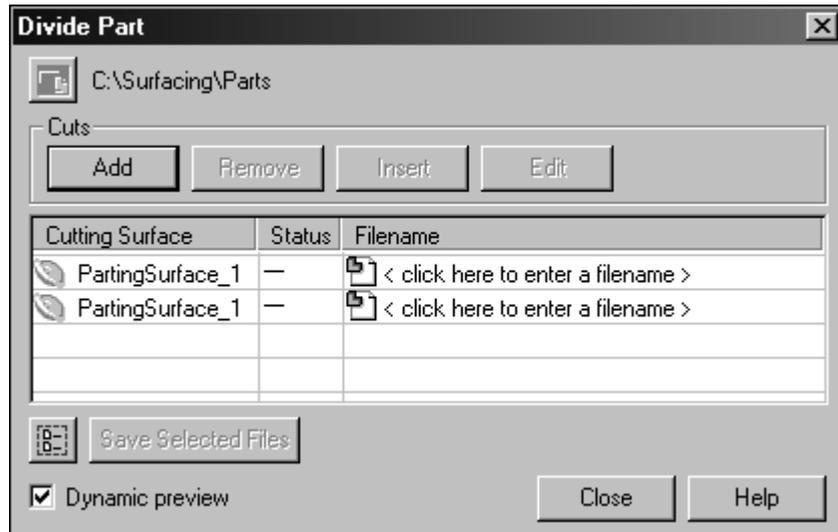
- Position the direction arrow as shown as the side to be divided into a new file.



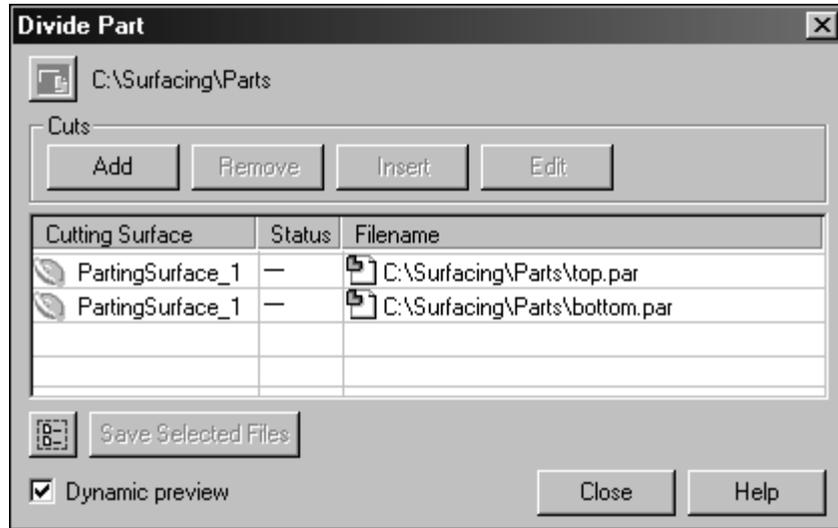
- On the ribbon bar, click Finish.



- On the Divide Part dialog box, you will enter filenames for the two new divided parts.



- Click on the first < click here to enter a filename > and type TOP. Click on the second < click here to enter a filename > and type BOTTOM. Do not click Close.



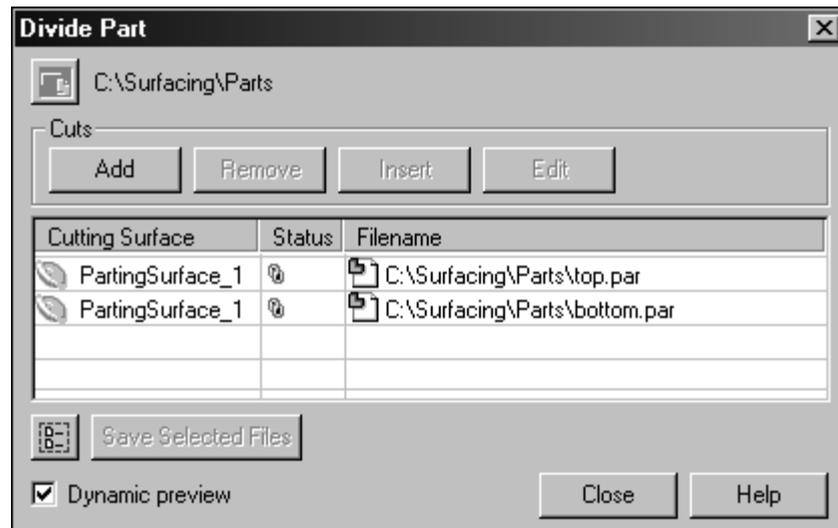
- You will now save the files. Click the Select All button as shown.



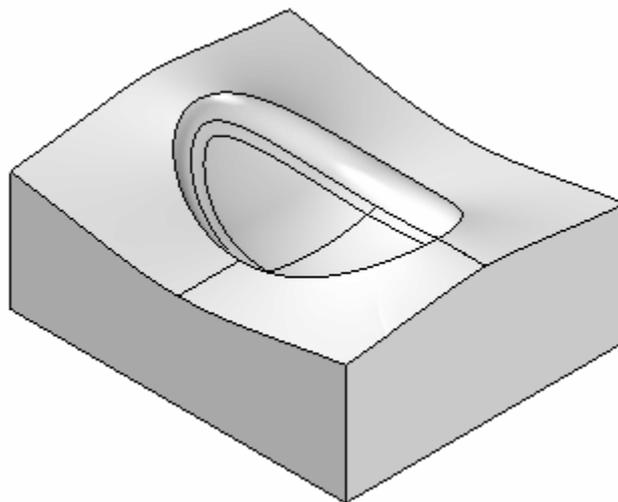
- Click the Save Selected Files button.



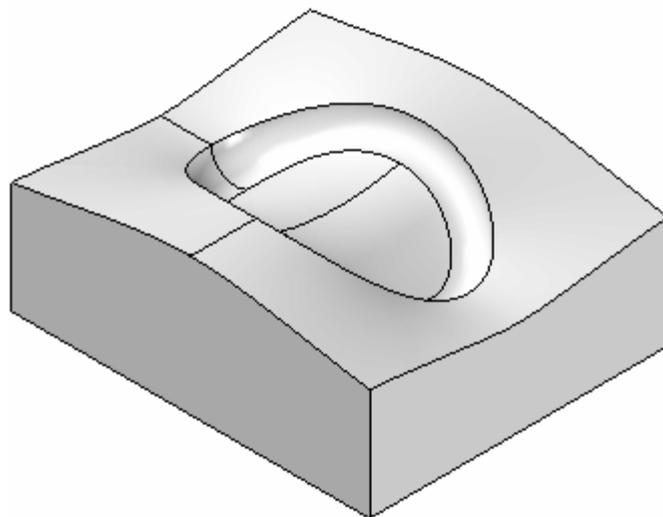
- The two new files are created and notice that they are linked to the parting surface. Click Close.



8. Close PAD CORE.PAR.
9. You will now open each of the two mold halves to observe the results.
  - Open TOP.PAR in the C:/Surfacing/Parts folder. In the image below, the part was flipped 180 degrees to show the cavity.



- Close TOP.PAR and open BOTTOM.PAR in the C:/Surfacing/Parts folder.



10. The activity is complete.



