CATIA V5 Training
Foils

Generative Shape Optimizer

Version 5 Release 19
January 2009

EDU_CAT_EN_GSO_FI_V5R19

Instructor Notes:

About this course

Objectives of the course

Upon completion of the course you will learn to:

- **Develop Shapes**
- **Morph Shapes**
- **Create Junctions (BIW application) between surfaces**
- **Work with Volumes**

Targeted audience

Surface Designers

Prerequisites

Students attending this course should have knowledge of:

CATIA Surface Design.

8 hrs

Instructor Notes:

Table of Contents (1/2)

Instructor Notes:

Table of Contents (2/2)

Instructor Notes:

Introduction to Generative Shape Design Optimizer

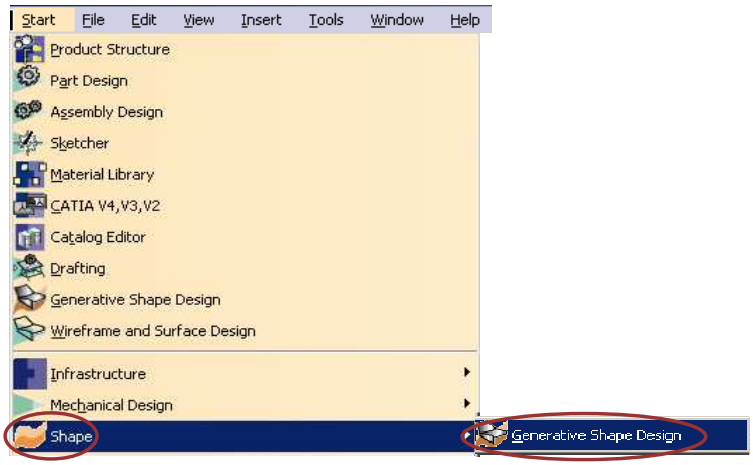
In this lesson you will be introduced to the working environment of Generative Shape Design Optimizer Workbench.

- Accessing Generative Shape Design Optimizer Workbench
- The User Interface

Instructor Notes:

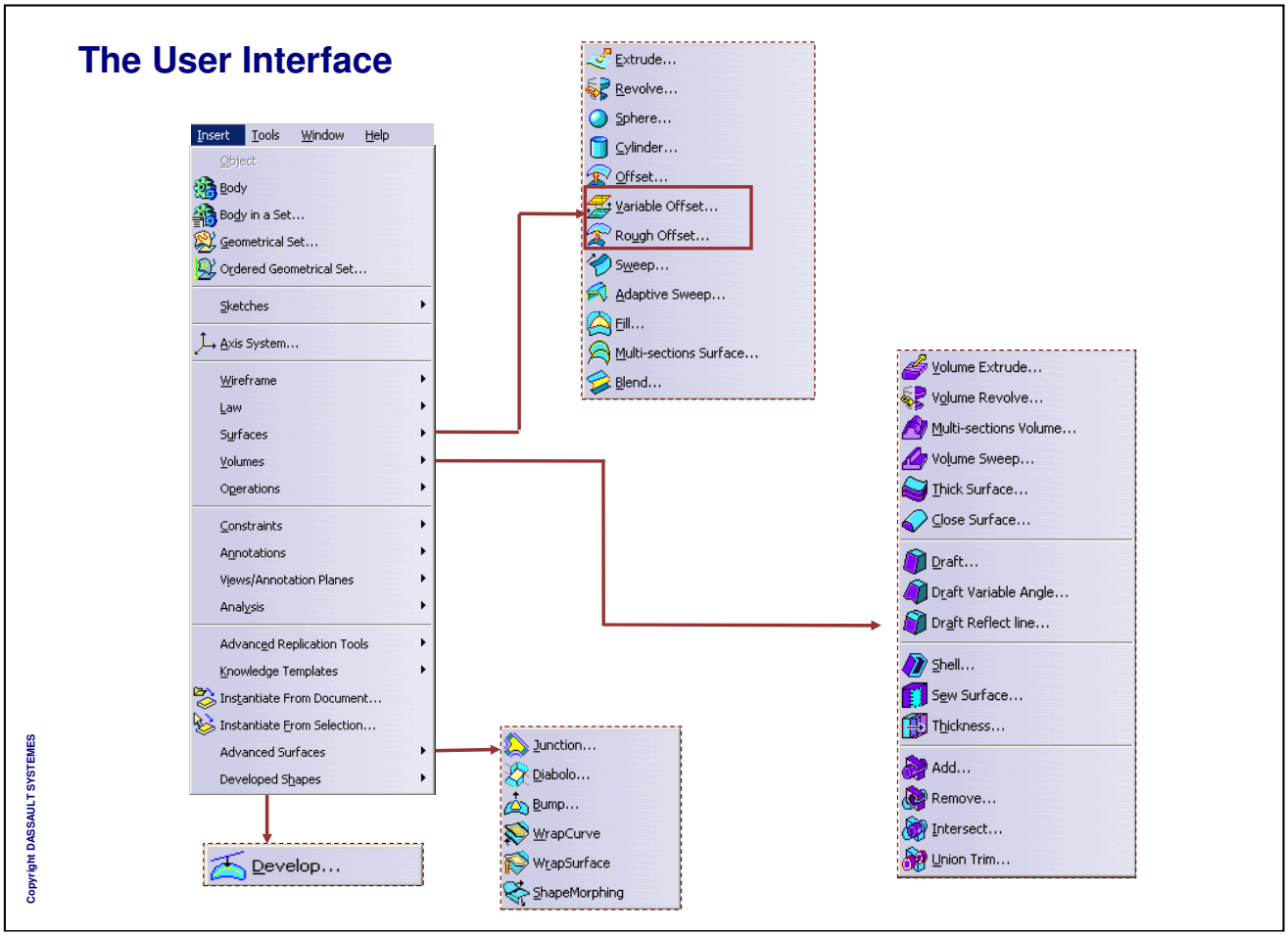
Accessing Generative Shape Design Optimizer Workbench

Generative Shape Design Optimizer is available through the Generative Shape Design workbench:



Toolbars specific to Generative Shape Design Optimizer workbench are:

Instructor Notes:



Instructor Notes:

Creating Offset Surfaces

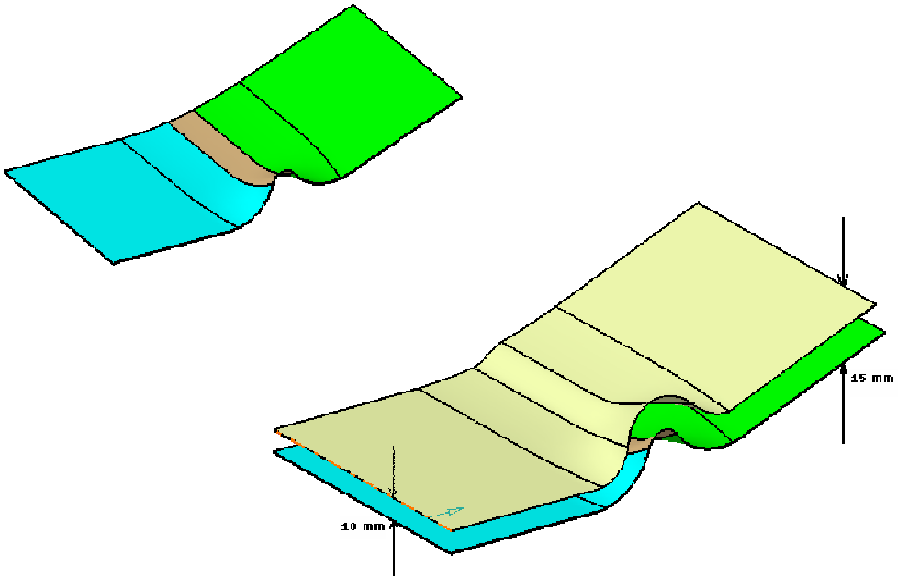
In this lesson, you will learn how to create offsets.

- Creating Variable Offsets
- Creating Rough Offsets

Instructor Notes:

Creating Variable Offsets

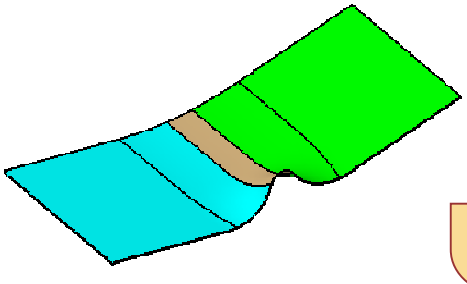
In this Skillet you will learn to create Variable offsets.

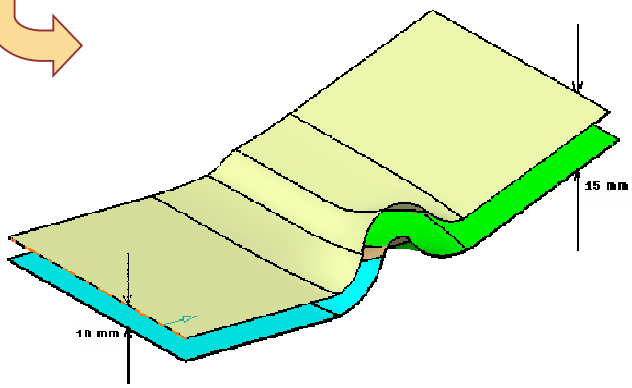


Instructor Notes:

What is Variable Offset

Variable offset allows you a higher degree of control during an offset operation and allows you to offset different sub-elements of a surface by different amount:



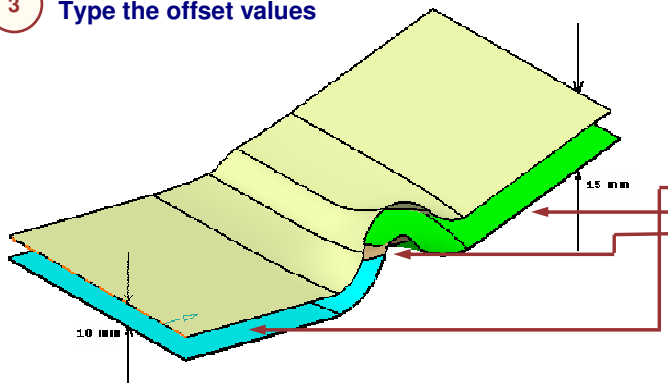


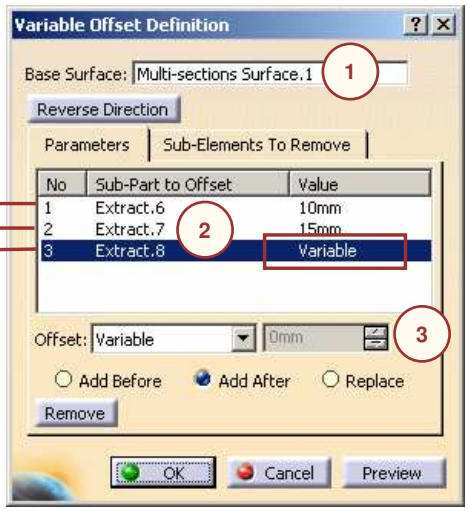
Surface shown above has three patches. With variable offset, the offset can be done by different values for the blue and green patch. Brown patch will move by a variable amount to match the adjacent surfaces.

Instructor Notes:

How to Create Variable Offset

1. Select the surface to offset.
2. Select the patches from the surface
3. Type the offset values

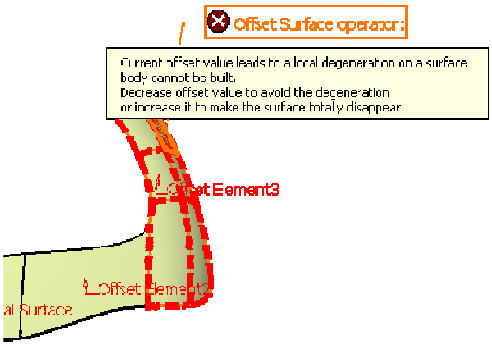




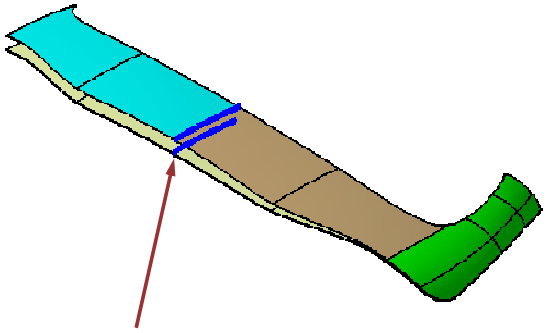
Instructor Notes:

Additional Information on Variable Offset

In this illustration, we are going to show you how to develop the extremum point on a wire



In case of error, there is an accurate error diagnosis for every element in error.

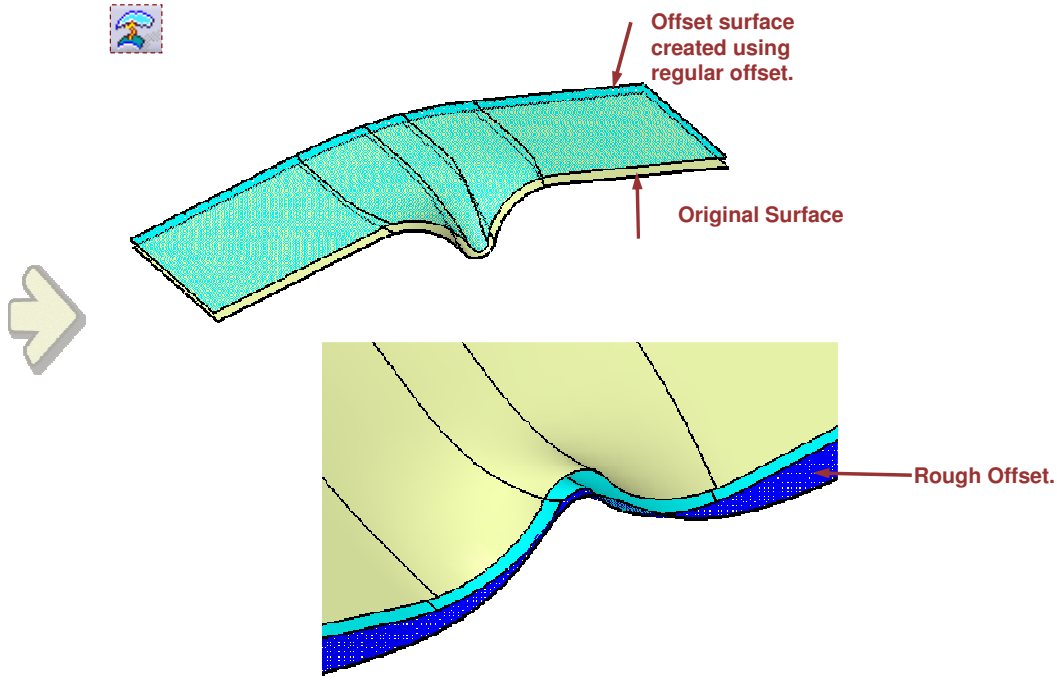


The sub-element that is variable offset matches up with the constant offset elements with the same type of continuity as the original sub-elements.

Instructor Notes:

Creating Rough Offsets

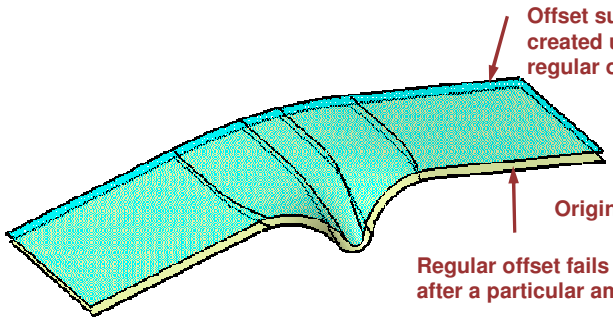
In this skillet you will learn how to create Rough offsets.



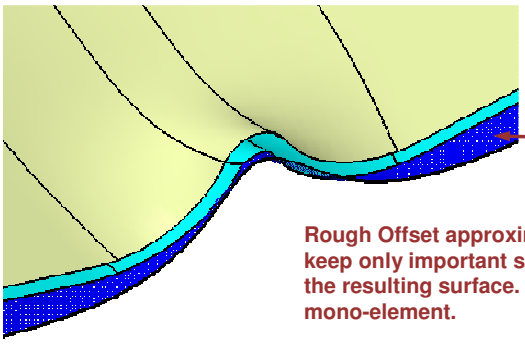
Instructor Notes:

Why Create a Rough Offset

Rough offset simplifies the offset, which results in a robust offset. Rough offset can be used in die face design



Regular offset fails in some situations after a particular amount of offset.

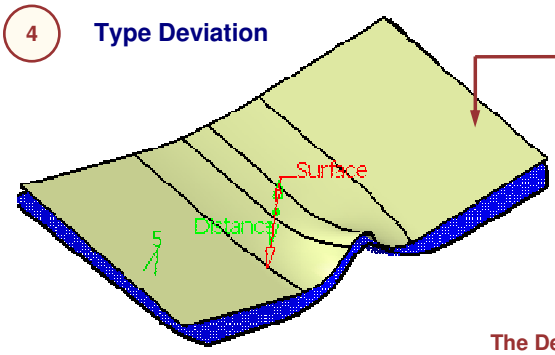


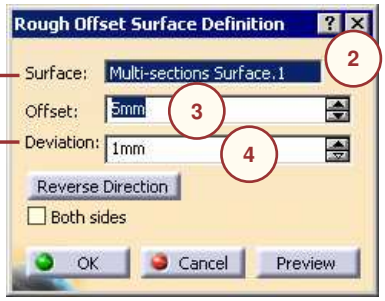
Rough Offset approximates the initial shape to keep only important shape characteristics in the resulting surface. The resultant surface is mono-element.

Instructor Notes:

How to Create a Rough Offset

1. Click the icon
2. Select the surface to offset
3. Type Offset value
4. Type Deviation





The Deviation parameter is a driving parameter for the approximation. Therefore, the higher the deviation, the higher the approximation and the smoother the result is produced.

Instructor Notes:

Developing Shapes

In this lesson, you will learn how to develop wires using the Generative Shape Design Optimizer workbench.

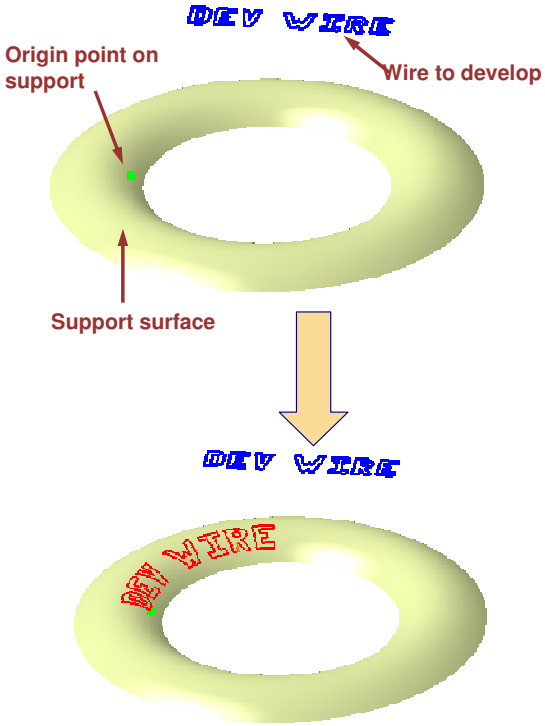
- What is Developing a Wire
- Developing Wires
- Additional Information on Developing Wire

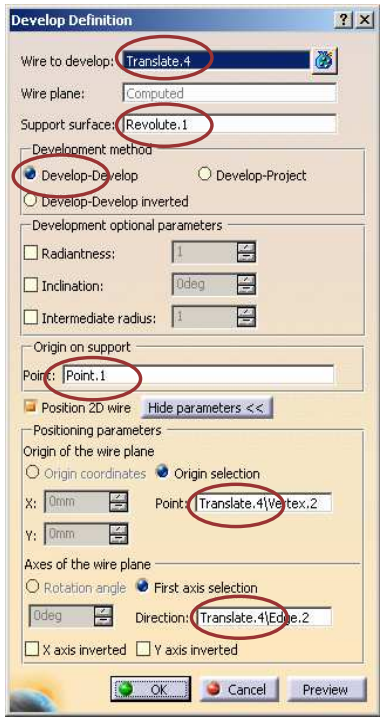
Instructor Notes:

What is Developing a Wire

Load:Develop_plane-revol.CATPart

Developing Wire allows you to develop a wire lying on a plane into a revolution surface

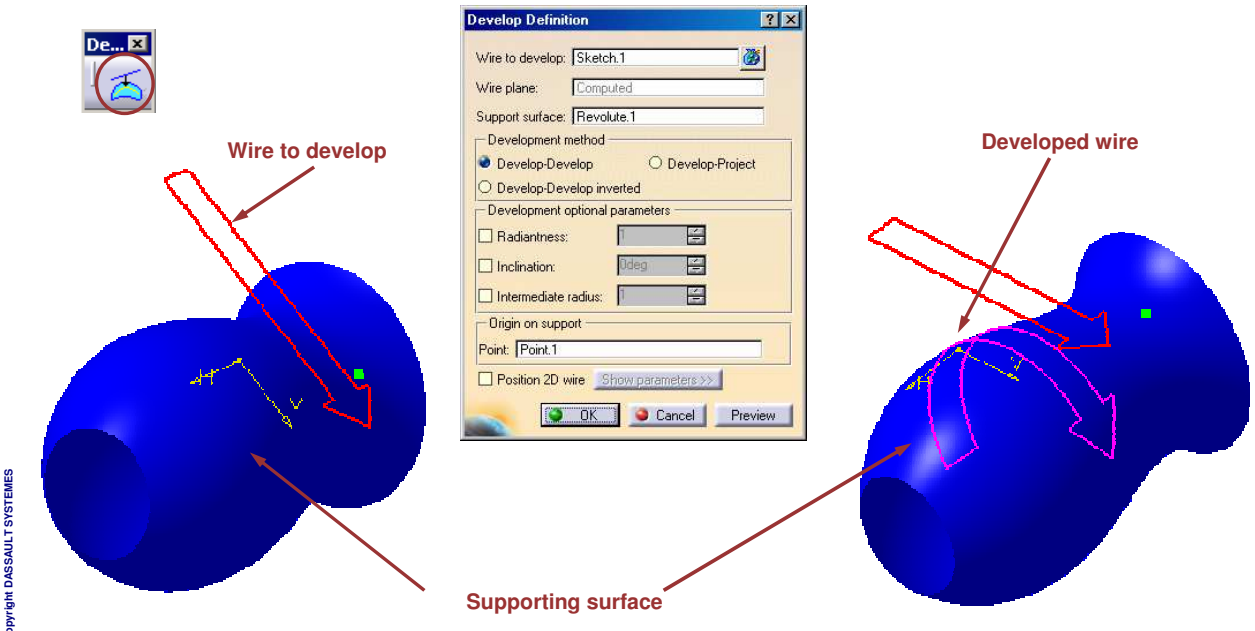




Instructor Notes:

Developing Wires (1/2)

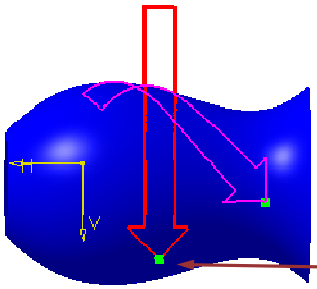
The tool is used to develop wires, and points, onto a revolution surface, that is to create a new wire by mapping a wire's planar parameters onto a surface.

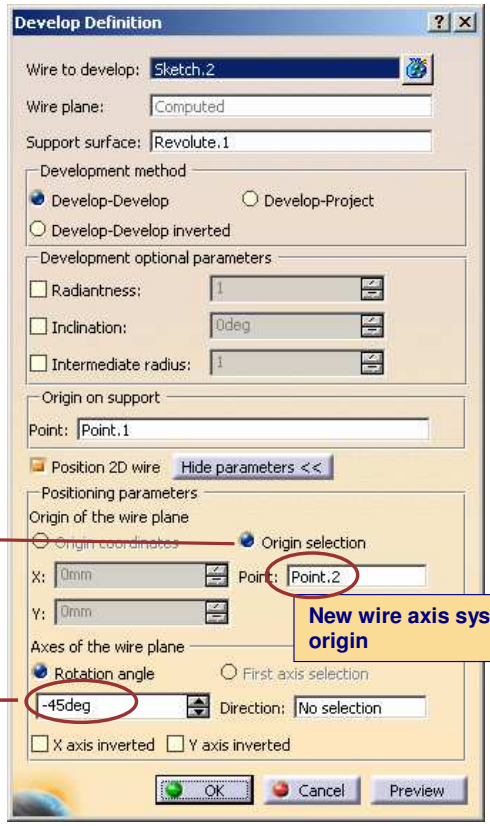


Instructor Notes:

Developing Wires (2/2)

Positioning the 2D wire
(modifying the wire axis system positioning)





New wire axis system origin

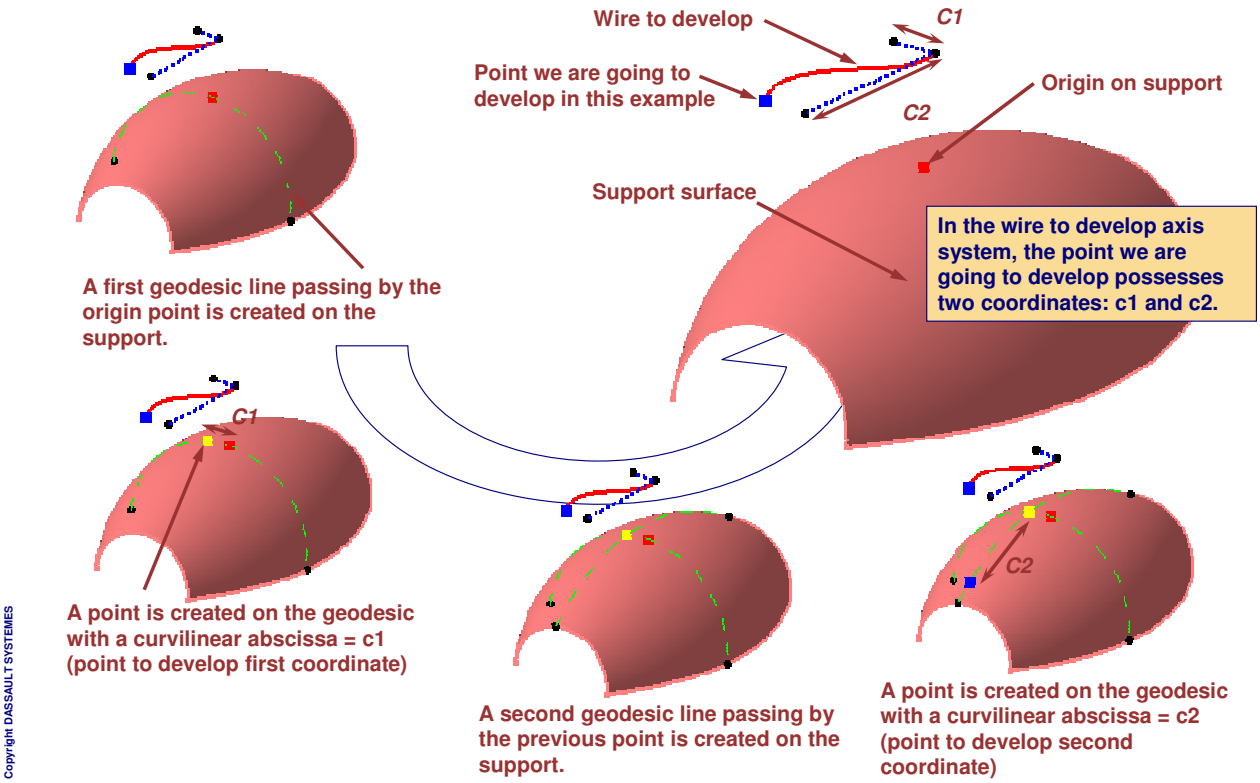
Rotation of the wire axis system positioning

See the Result: develop_simple.CATPart

Instructor Notes:

Additional Information on Developing Wire (1/2)

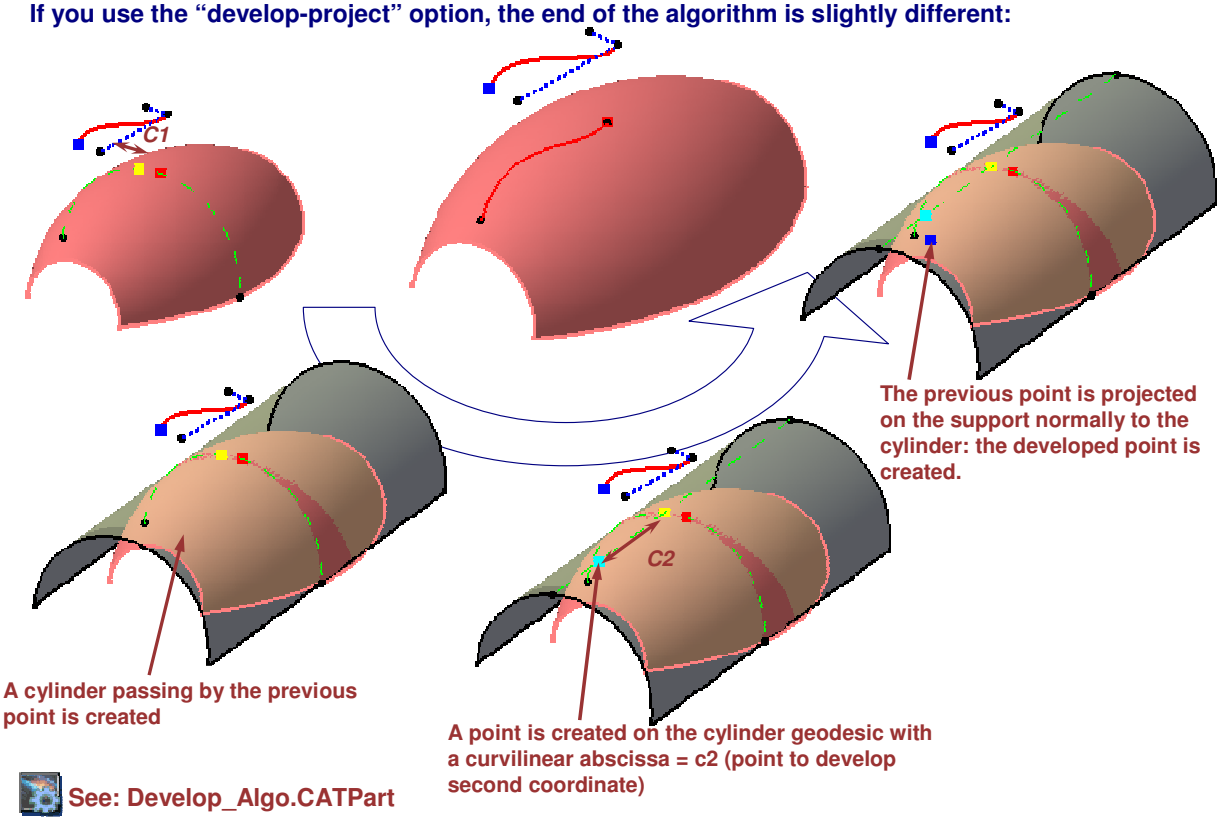
In this illustration, we are first going to show you to develop the extremum point of the wire



Instructor Notes:

Additional Information on Developing Wire (2/2)

If you use the "develop-project" option, the end of the algorithm is slightly different:



See: Develop_Algo.CATPart

Instructor Notes:

Using BIW Tools

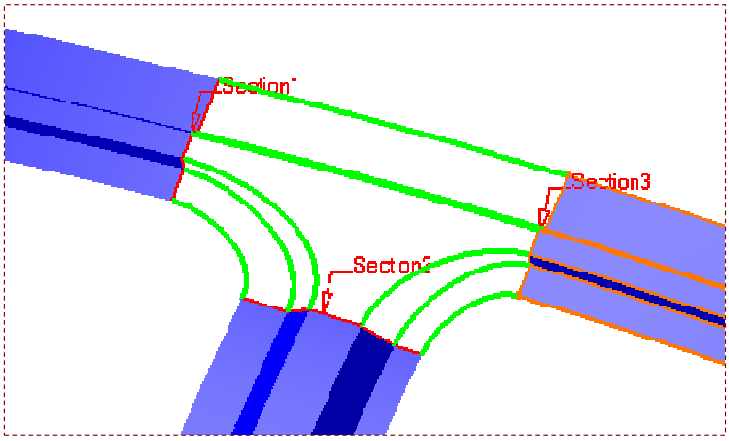
In this Lesson you will learn to create Junctions and Diabolo Seat

- Creating Junctions
- How to Create a Diabolo Seat

Instructor Notes:

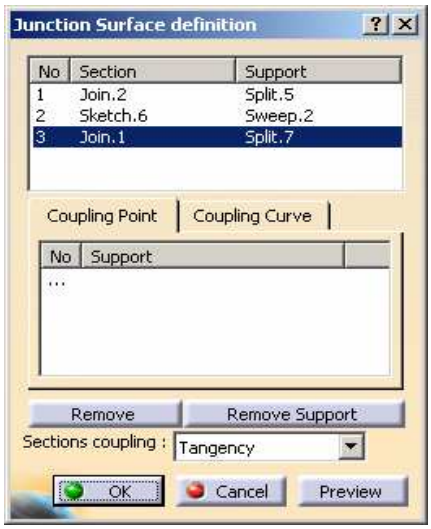
Creating Junctions

In this skillet you will learn how to create Junctions

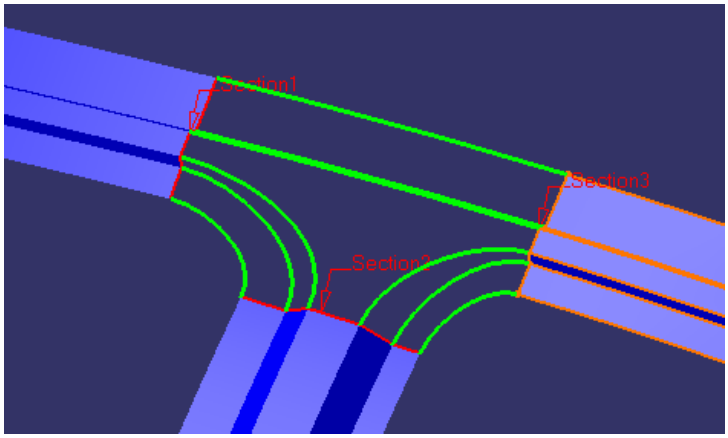


Instructor Notes:

How to Create a Junction



A junction is a surface created between the boundaries of 3 surfaces. It is equivalent to a GSD blend surface except that it involves a 3rd surface. The junction is a "T-shape" surface connecting the 3 surfaces:



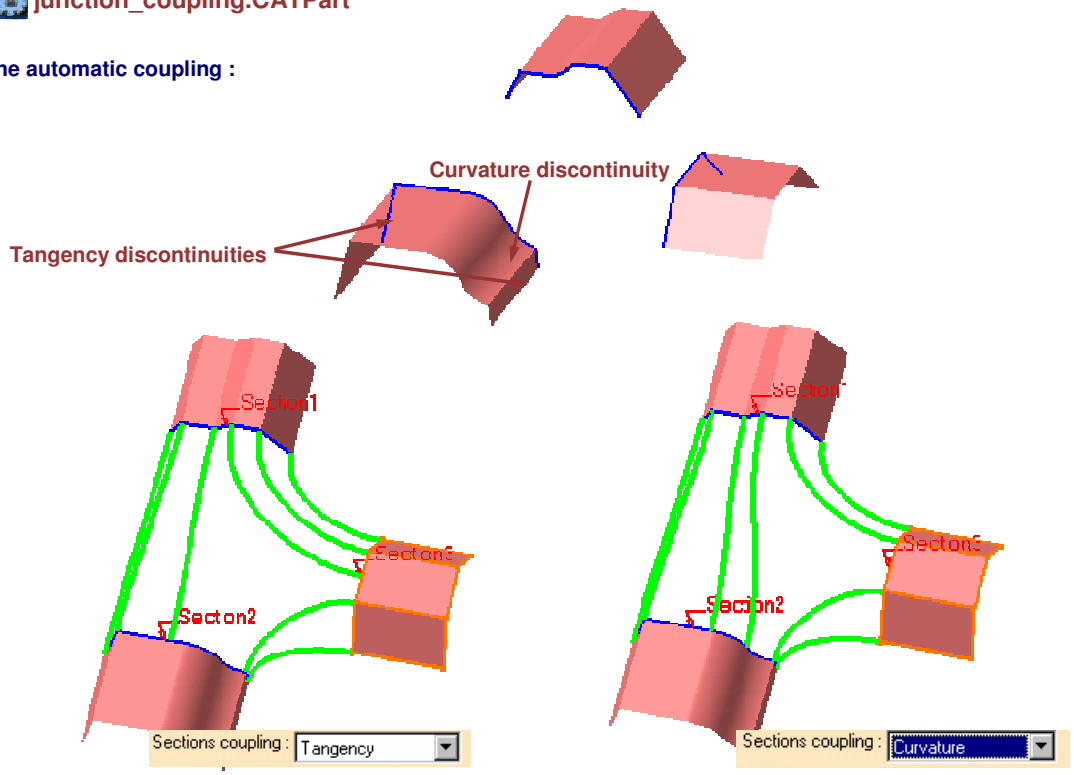
 See the Result: junction.CATPart

Instructor Notes:

How to Create a Junction: Automatic Coupling

junction_coupling.CATPart

The automatic coupling :



Tangency discontinuities detection

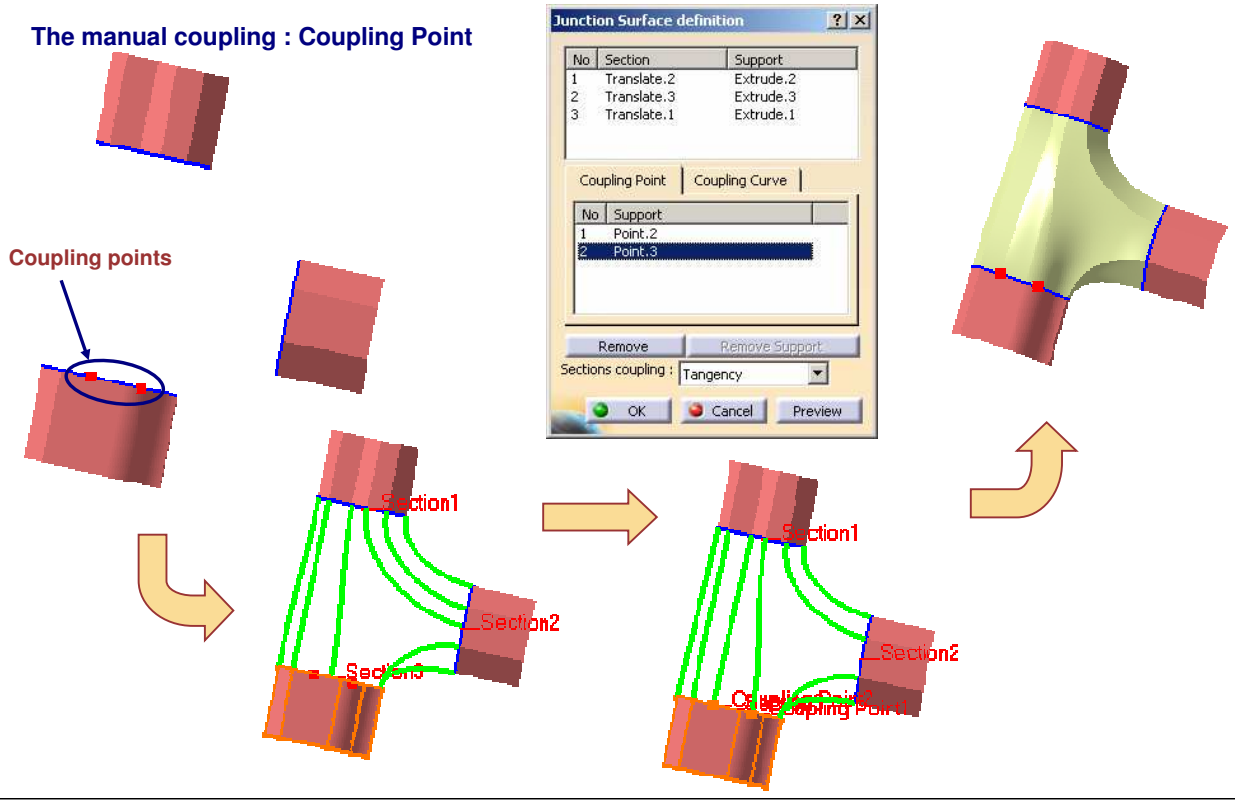
Curvature discontinuities detection

Instructor Notes:

How to Create a Junction: Manual Coupling (1/2)

Load: Junction_Coupling_Point_New.CATPart

The manual coupling : Coupling Point

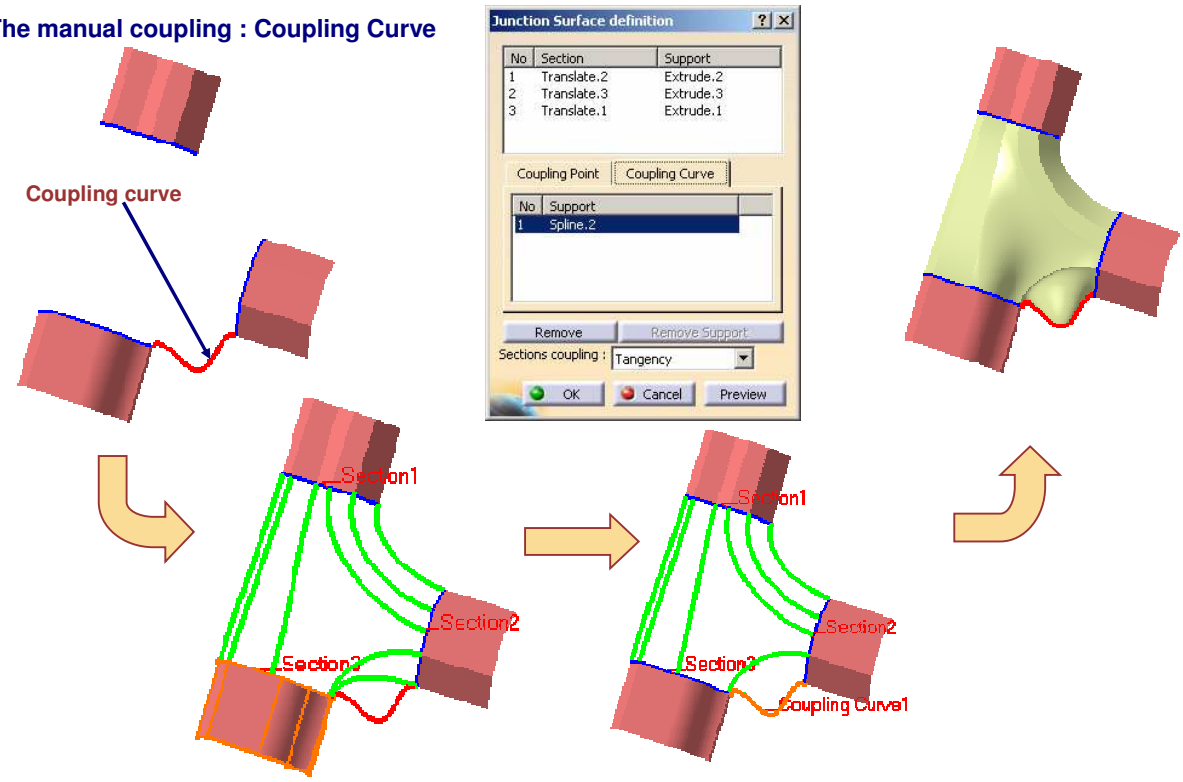


Instructor Notes:

How to Create a Junction: Manual Coupling (2/2)

Load: Junction_Curve_Coupling.CATPart

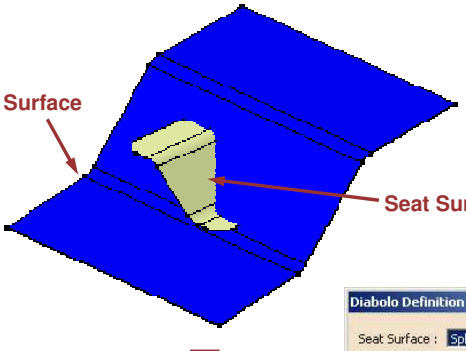
The manual coupling : Coupling Curve

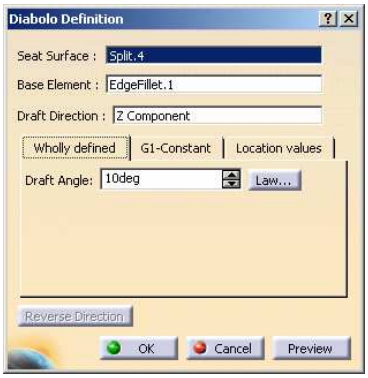


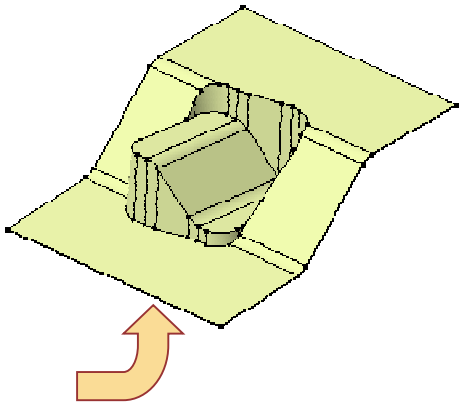
Instructor Notes:

How to Create a Diabolo Seat

This tool allows you to create a Diabolo Seat surface including a draft angle:







Instructor Notes:

Creating Advanced Surfaces

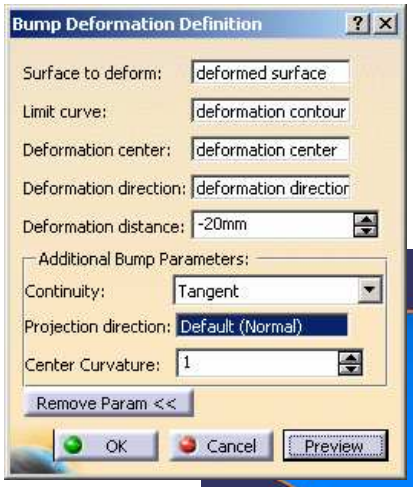
In this lesson you will get familiar with the use of Advanced Surfaces

- How to Create a Bumped Surface
- Deforming Surfaces by Wrapping Curve
- How to Create a Wrap Surface
- How to Morph a Shape
- How to Morph a Shape: Limit Curve

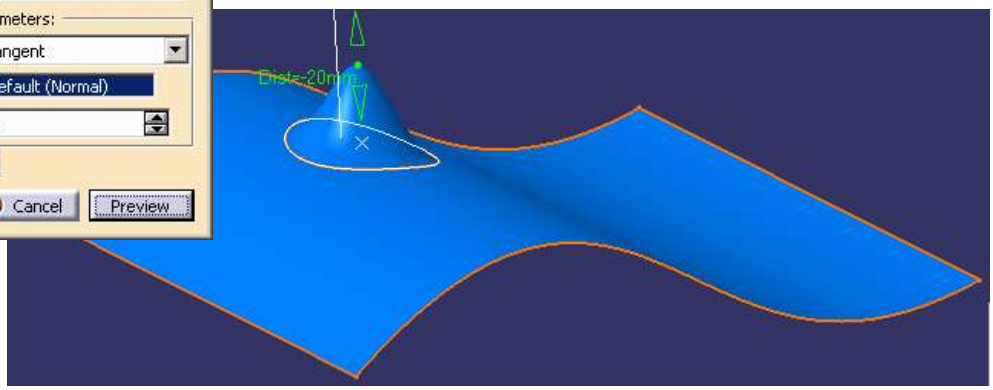
Instructor Notes:

How to Create a Bumped Surface (1/3)

Bump_Start.CATPart



A bumped surface is a surface deformed inside a given area:

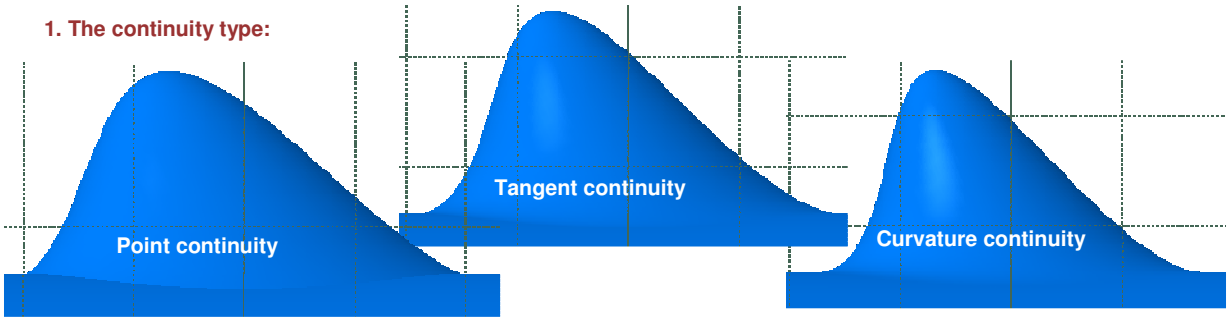


Instructor Notes:

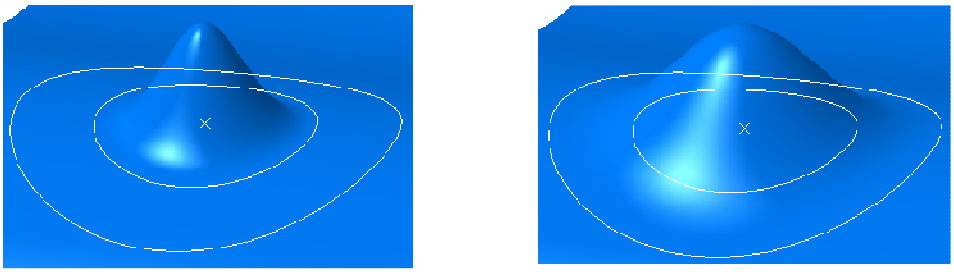
How to Create a Bumped Surface (2/3)

Influence of the bumped surface parameters:

1. The continuity type:



2. The deformation area:

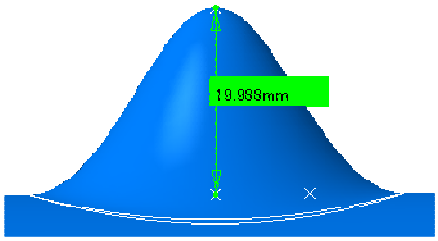


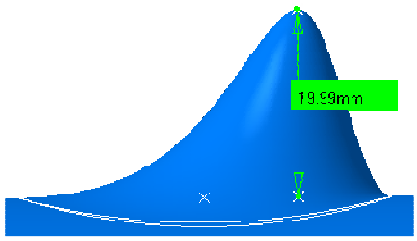
Instructor Notes:

How to Create a Bumped Surface (3/3)

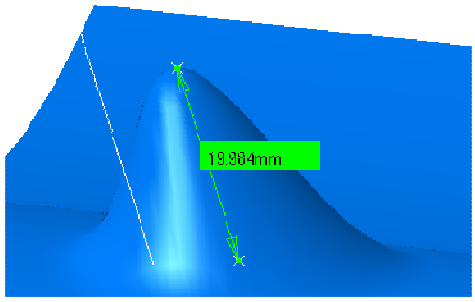
Influence of the bumped surface parameters:

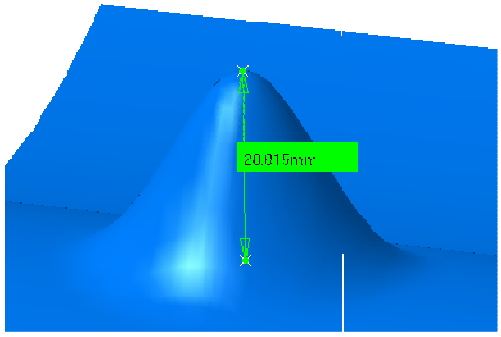
3. The deformation center:





4. The deformation direction:

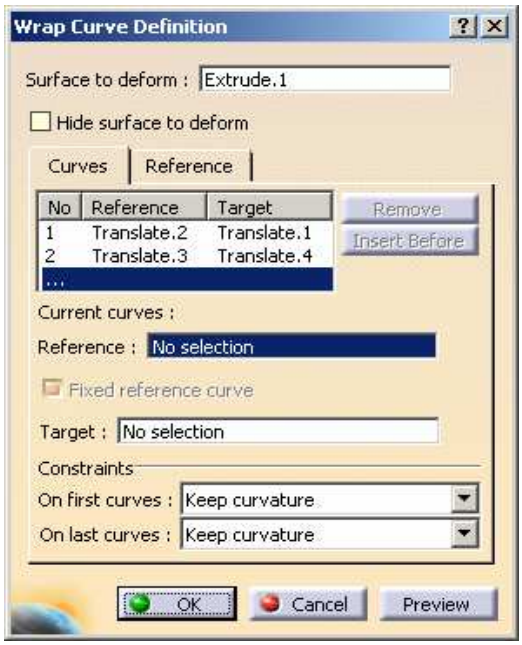


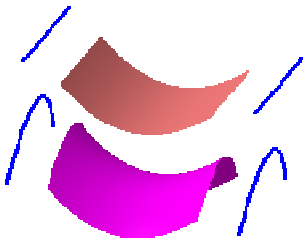


Instructor Notes:

Deforming Surfaces by Wrapping Curve

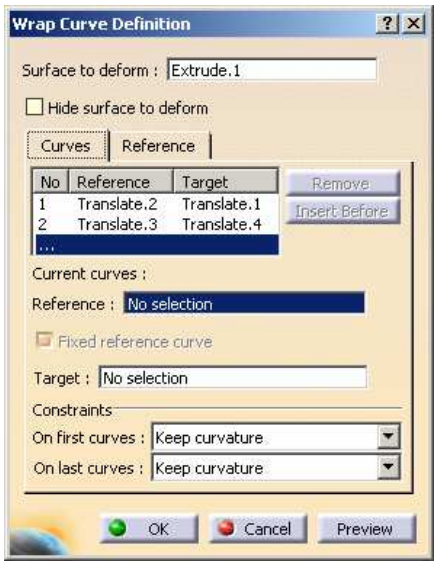
In this skillet you will learn how to create a Wrap curve.

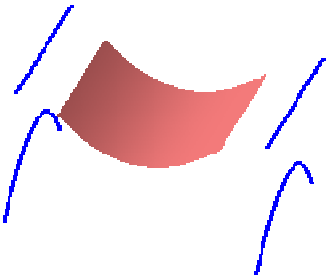


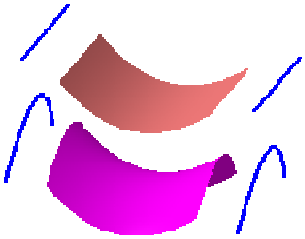


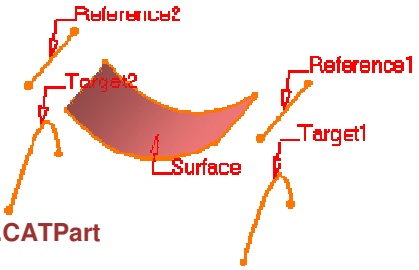
Instructor Notes:

How to Create a Wrap Curve



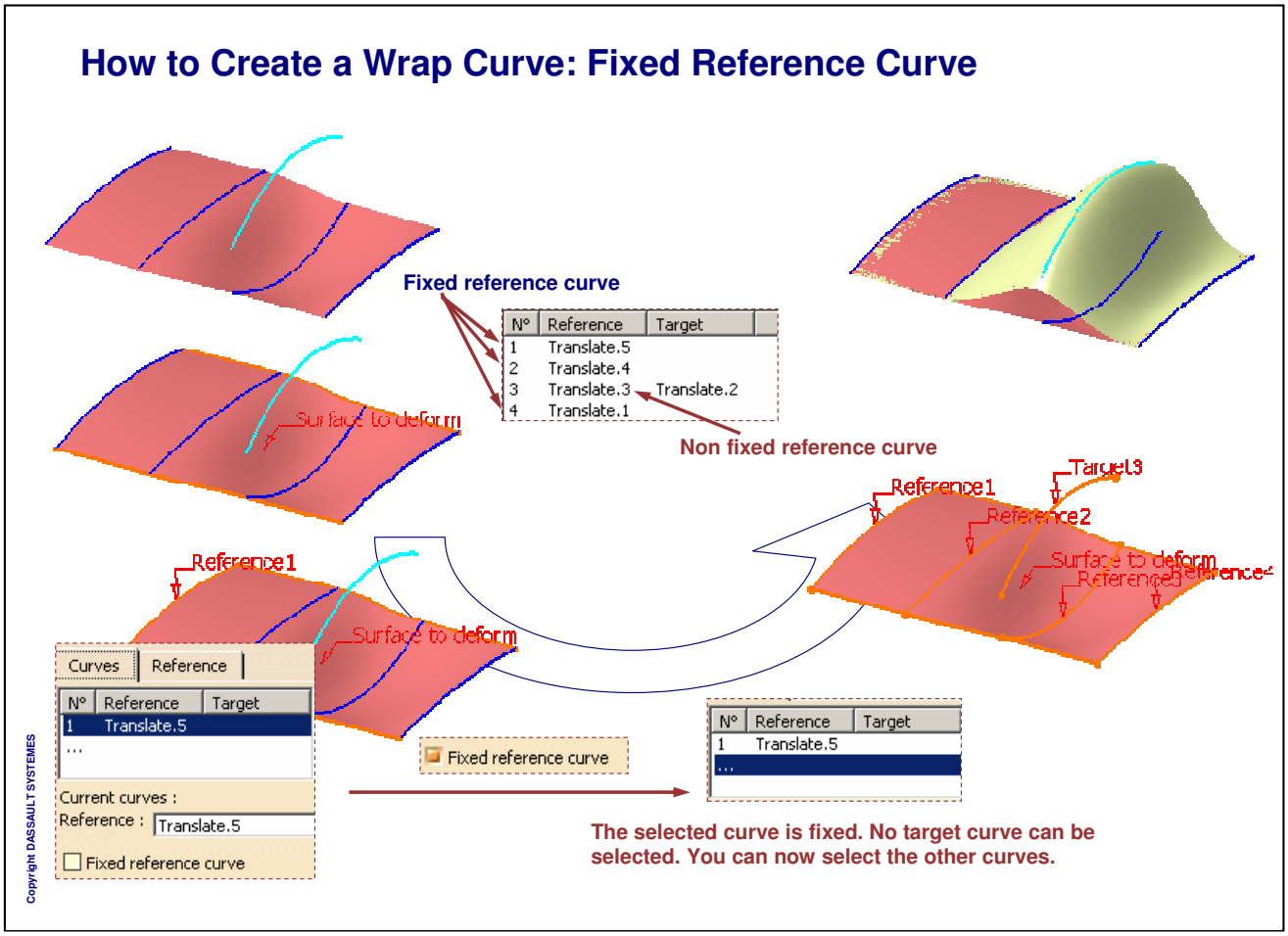






See Result: wrap_curve.CATPart

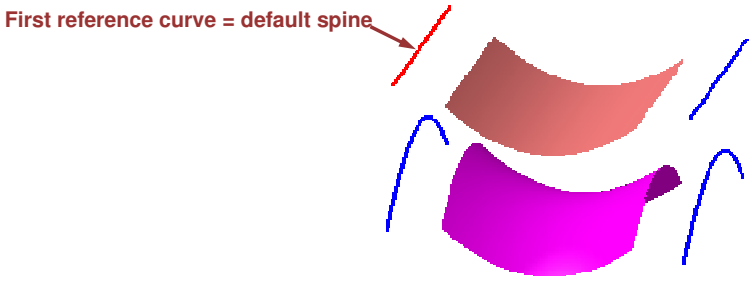
Instructor Notes:



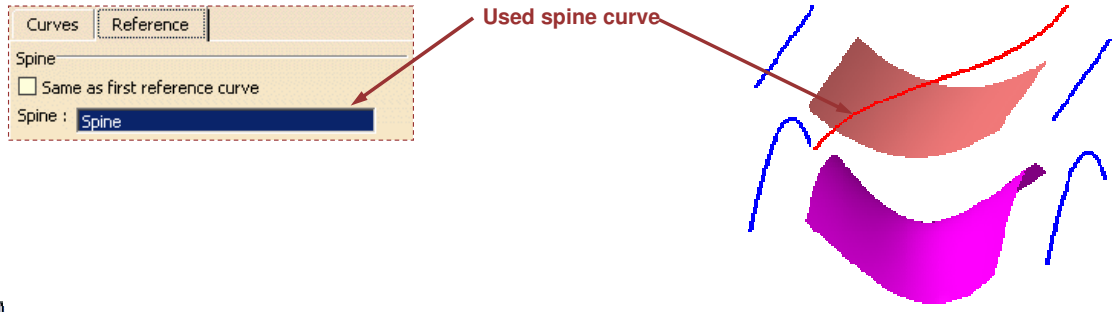
Instructor Notes:

How to Create a Wrap Curve: Spine

By default, the spine used is the first selected reference curve:



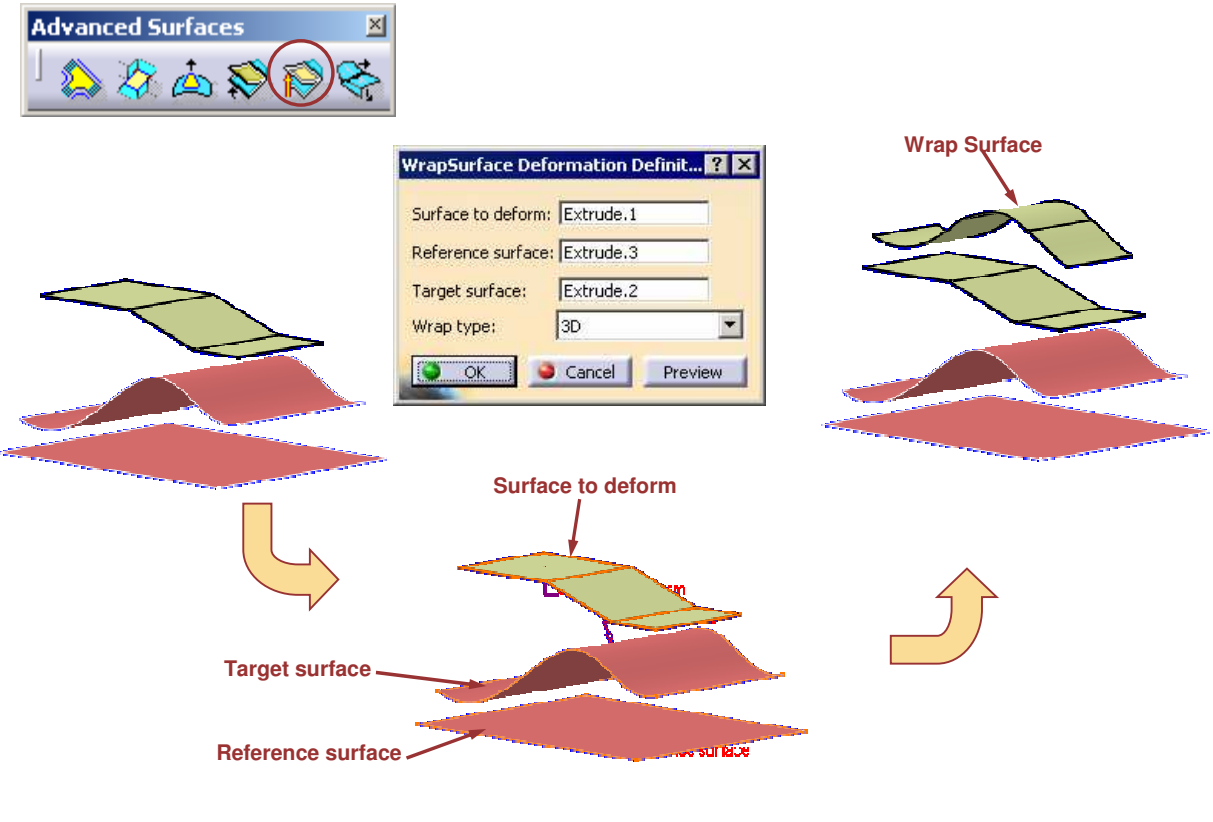
You can choose another spine to compute the new surface :



See Result: wrap_curve2.CATPart

Instructor Notes:

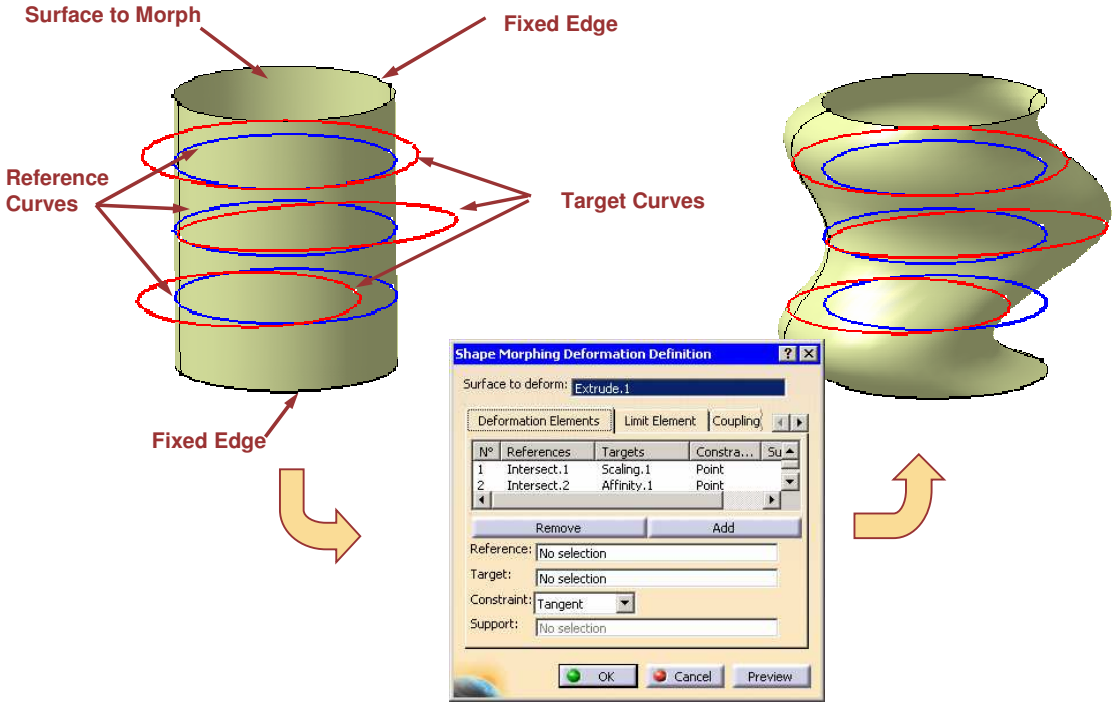
How to Create a Wrap Surface



Instructor Notes:

How to Morph a Shape

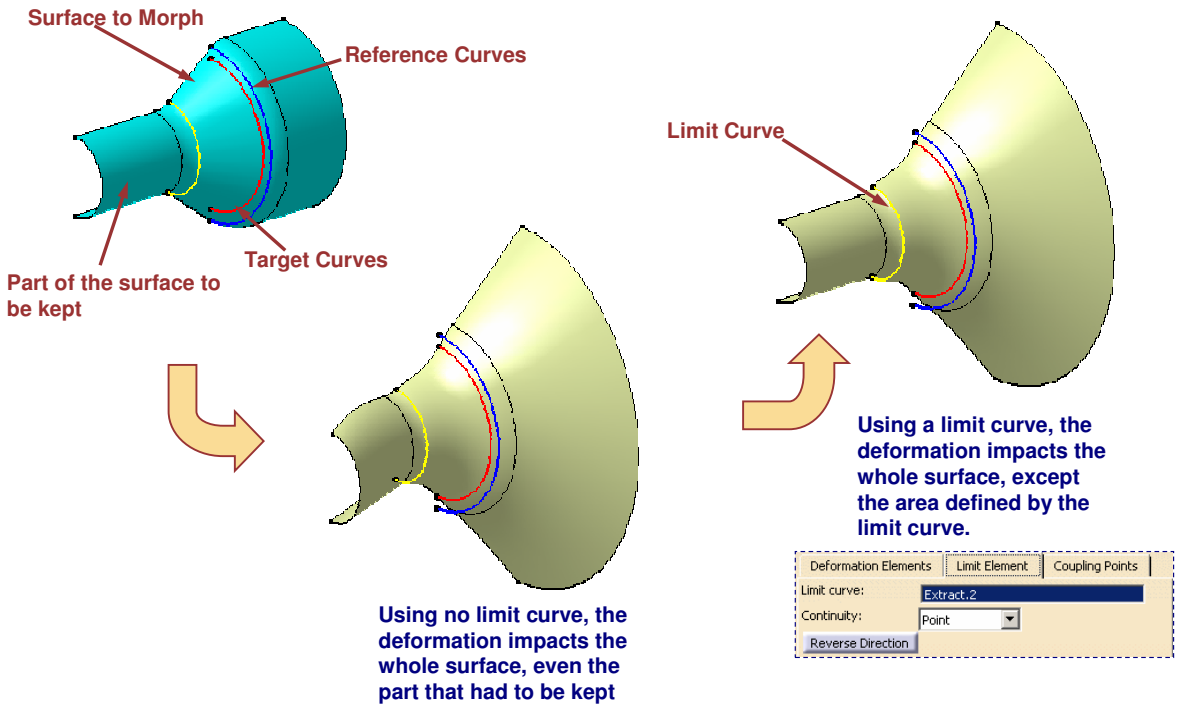
Shape Morphing is a surface deformation defined by reference and target curves:



Instructor Notes:

How to Morph a Shape: Limit Curve

You can define the area impacted by the shape morphing operation by selecting a limit curve:



Instructor Notes:

Creating Advanced Operations

In this lesson you will get familiar with the use of Advanced Operations

- What is an Auto Fillet?
- Auto Fillet User Interface
- How to Create an Auto Fillet

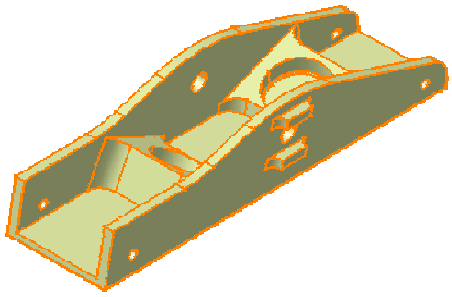
Instructor Notes:

What is an Auto Fillet?

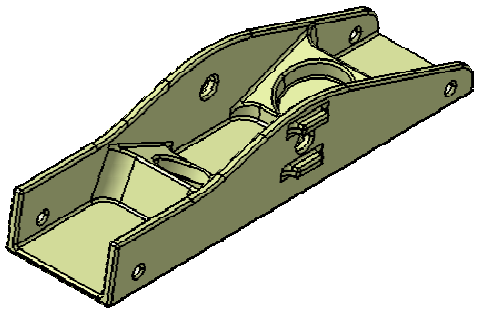
An Auto Fillet is a new tool added in V5R19 Generative Shape Design workbench. Using this tool you can remove the sharp edges of a part in one shot hence reducing the time spent in applying fillets with the existing features.

For an Auto Fillet you need to select the faces to be filleted and optionally select the functional faces which you do not want to fillet.

In the illustration given below, you can observe that on selecting the joined surface at least 90% of the sharp edges get automatically filleted.



Whole part is selected



Final result

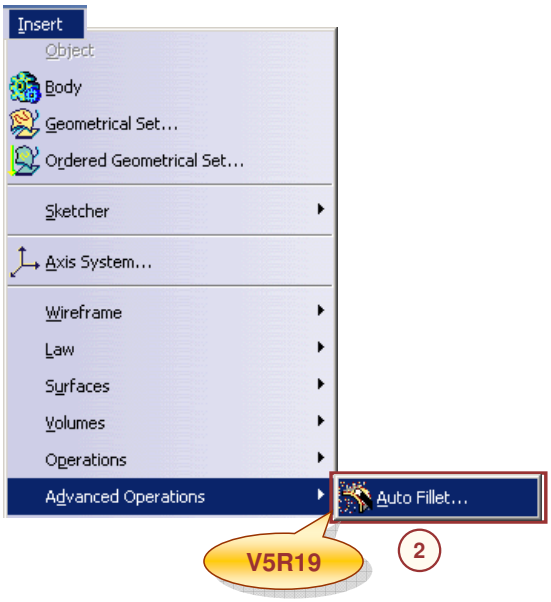
Instructor Notes:

Auto Fillet User Interface (1/3)

You can use any of the following methods to access the Auto Fillet command in the Generative Shape Design workbench:

1. You can access it from the 'Advanced Operation' toolbar.

2. You can also access it from the Insert > Advanced Operations menu.



Instructor Notes:

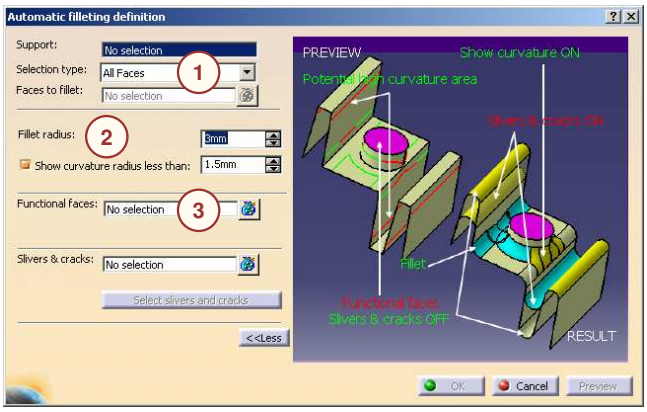
Auto Fillet User Interface (2/3)

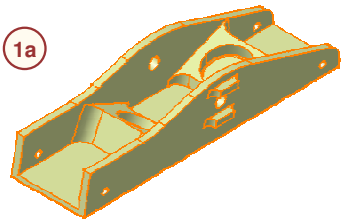
The user interface of the Auto Fillet dialog box is explained below.

1. **Selection type: You can select the computation mode from the two available modes.**
   a. **All Faces: It allows you to select the whole surface.**
   b. **Faces Selection: It allows you to select the faces to fillet individually.**

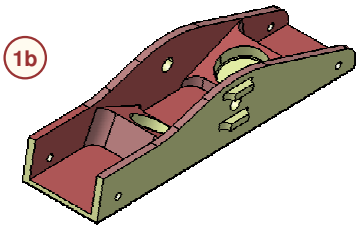
2. **Fillet radius: It is the radius of the surface.**

3. **Functional faces: You can specify the faces which you do not want to fillet.**





Whole part is selected

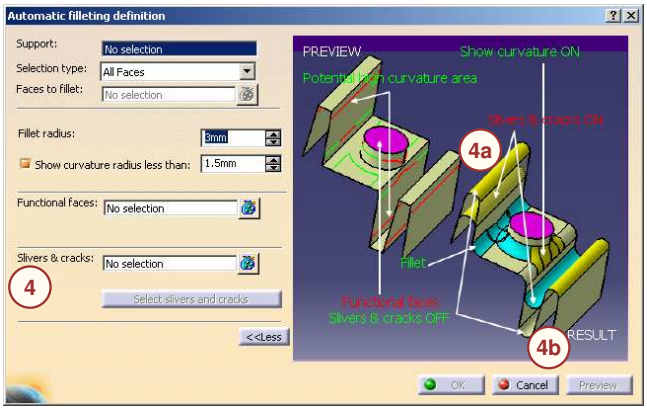


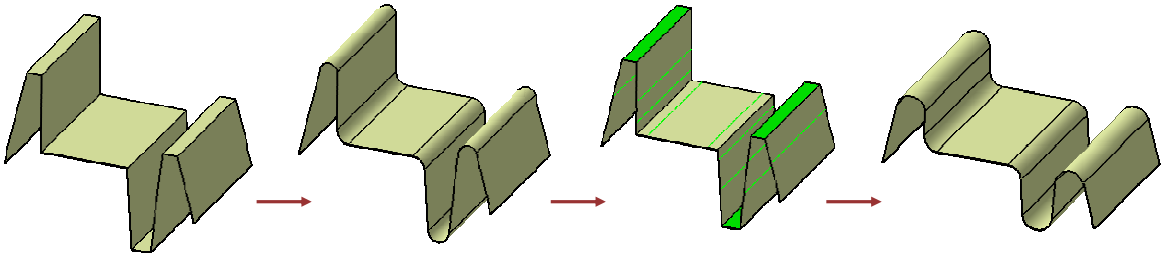
Some faces are selected

Instructor Notes:

Auto Fillet User Interface (3/3)

4. Slivers & cracks:
   a. Silver: It is an unwanted thin wall which should be smoothed and partially removed by the automatic filleting operation.
   b. Crack: It is an unwanted slot which should be smoothed and partially filled by the automatic filleting operation.





Part to Auto fillet | Result without silver and crack | Faces selected for silver and crack | Final result

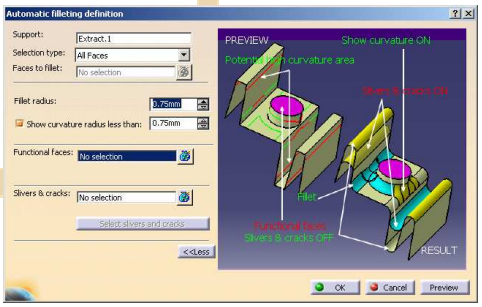
Instructor Notes:

How to Create an Auto Fillet

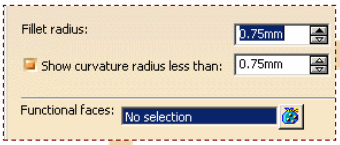
You will perform the following steps to create an Auto Fillet

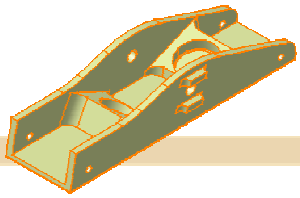
1 Click on the Auto Fillet tool.

2 Select the surface to be filleted.

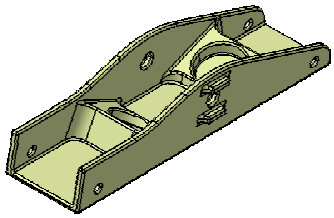


3 Specify the Fillet radius.





4 Click OK to create the Auto fillet.



Instructor Notes:

Creating Volumes

In this lesson, you learn how to create Volumes

- What is a Volume ?
- Different types of volumes
- Volumes Made From Sketches
- Creating Volumes From Surfaces
- Applying Dress-up Features on Volumes
- Transformations and Operations on Volumes
- Performing Boolean Operations on Volumes
- To Sum Up

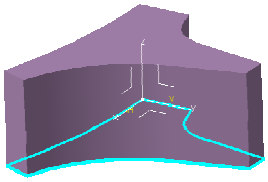
Instructor Notes:

What is a Volume ?

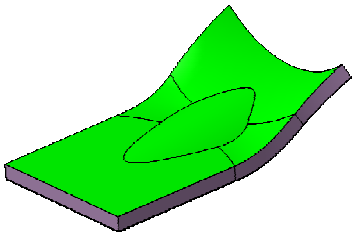
A volume is a closed skin that can be generated by different methods.
It can be created from a sketch or from a surface. It may also result from an operation or a dress up feature.

Some of the Volumes that can be created are:

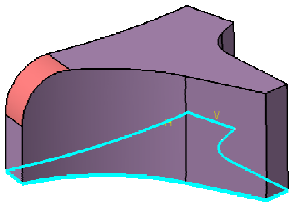
Volume Extrude



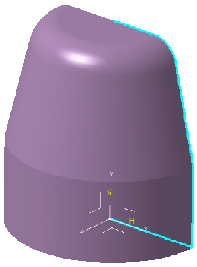
Thick Surface



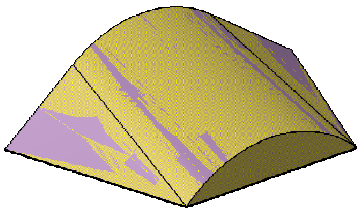
Edge Fillet



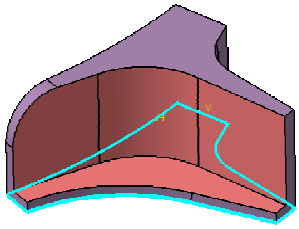
Volume Revolve



Close Surface



Remove



Instructor Notes:

Different Types of Volumes

Generative Shape Design provides several volumes creation or modification tools .

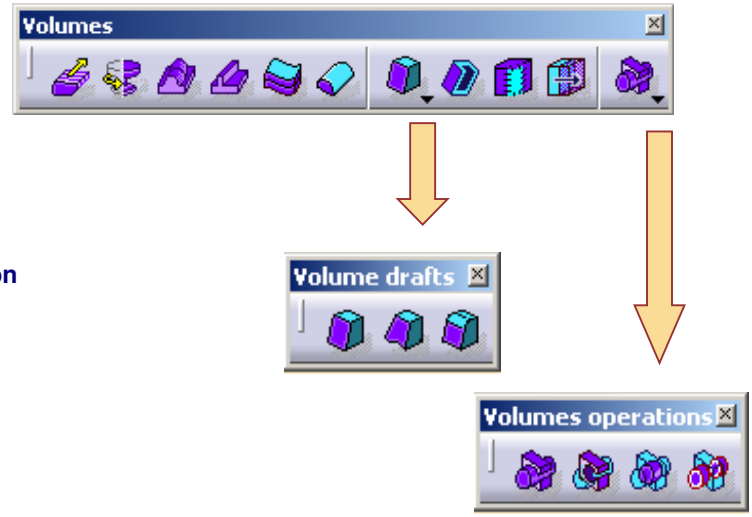
Here are the 4 types of volumes you can create:

Volumes made from a sketch
- **-Volume Extrude**
- **-Volume Revolve**
- **-Mullti-sections Volume**
- **-Volume Sweep**

Volumes made from a surface
- **-ThickSurface**
- **-CloseSurface**
- **-Sew surface**

Volumes made from a Boolean operation
- **-Add**
- **-Remove**
- **-Intersect**
- **-Union Trim**

Volumes made from an operation
- **-EdgeFillet**
- **-Split**
- **-Transformation**

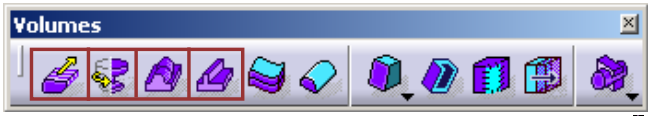


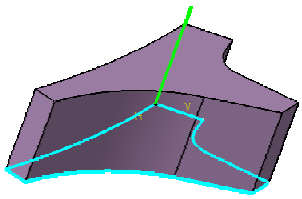
Dress-Up features can be applied on Volumes using Shell and Drafts

Instructor Notes:

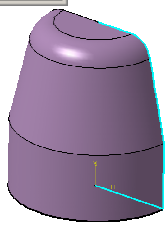
Volumes Made From Sketches

In this Skillet you will learn how to create Volumes from Sketches.

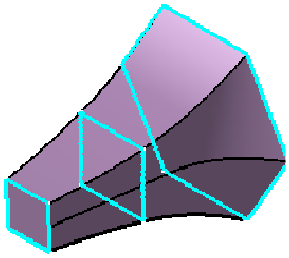




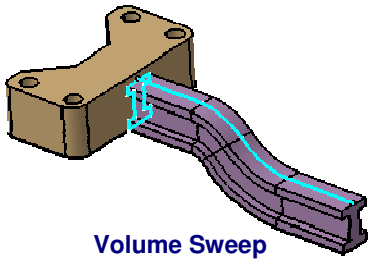
Volume Extrude



Volume Revolve



Multi-sections Volume



Volume Sweep

Instructor Notes:

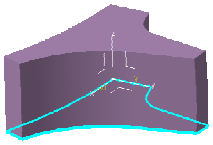
How to Create Volume From Sketches

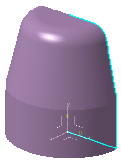
A volume made from a sketch can be generated by four different methods.
It can be an extruded volume, a revolution volume, Multi-sections volume,Volume sweep

These kinds of volumes are generated from a profile sketch. The resulting volume respects the sketch's shape around an axis or following a direction.

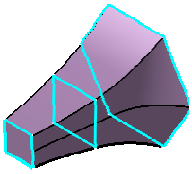
 Volume Extrude

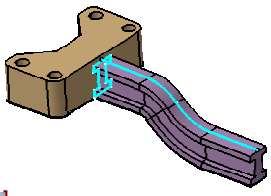
 Volume Revolve





When creating Multi-sections Volume and Volume Sweep more than one sketch is required to define different section profiles and guide curves respectively.





 Multi-sections Volume

 Volume Sweep

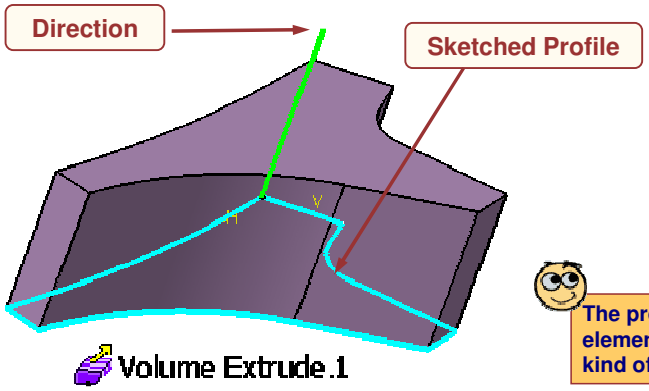
Instructor Notes:

How to Create a Volume Extrude

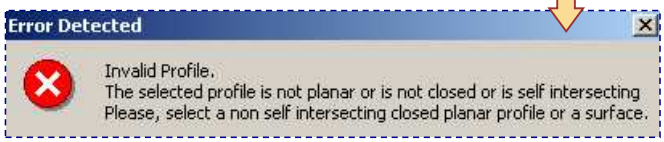
Volume extrude is a sketch based type of feature.

1

2 **Set the parameters**



The profile has to be closed and must not contain overlaid elements and standard points, otherwise, you could get this kind of error:



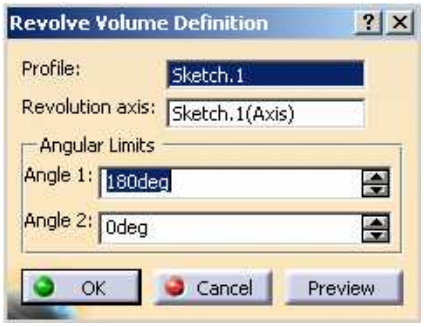
Instructor Notes:

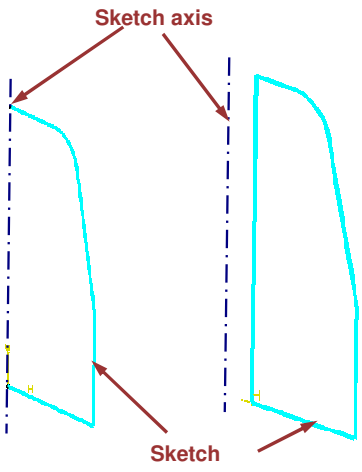
How to Create a Volume Revolve

Volume Revolve is a sketch based type of feature.

1 

2 **Define the Revolve parameters**

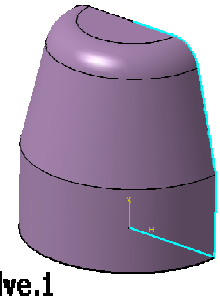




The revolution axis has to belong to the sketch's plane.

It may also be part of the sketch.

The profile has to be closed or end on the revolution axis.



Instructor Notes:

How to Create a Multi-Sections Volume

Multi-Sections_Volume.CATPart

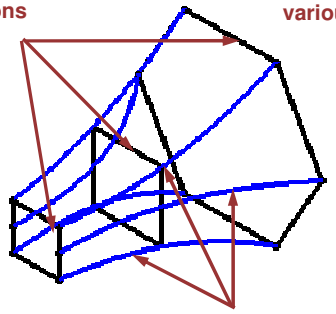
Multi-Sections volume sketch based type of feature.

1 

2 Select the section sketches and guide curves.

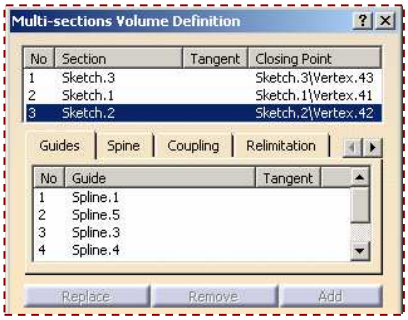
Sections

Check the orientation of various closing points

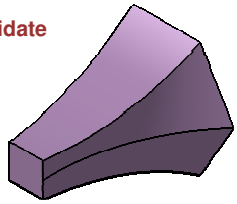


Guides

3 The parameters are defined in the dialog box



4 Click OK to Validate



Check various tabs: Spine,coupling, Relimitation.Also check various coupling options(Ratio,Tangency,Tangency then curvature and vertices.

Instructor Notes:

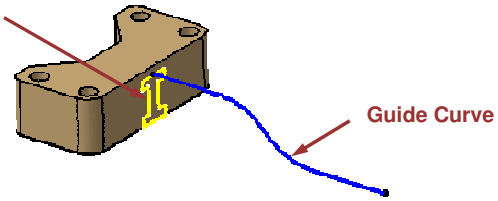
How to Create a Swept Volume – Explicit Type

Swept_Volume_Explicit.CATPart

Swept volume is a sketch based type of feature.

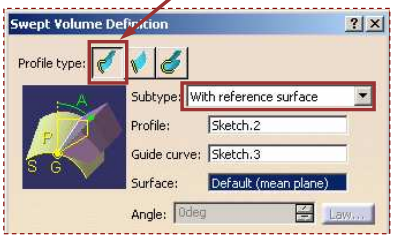
1 

2 Select the profile and the Guide curve.

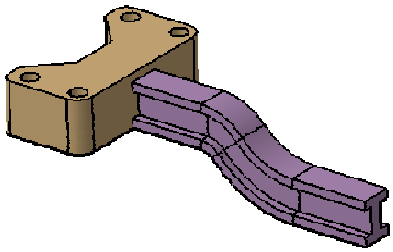


3 The parameters are defined in the dialog box.

Select Explicit type of sweep definition



4 Click OK to Validate



Check various subtypes of explicit sweep

Instructor Notes:

How to Create a Swept Volume – Line Type

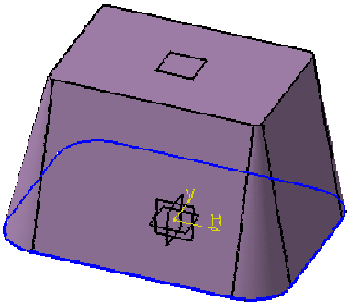
Swept_Draft_Line.CATPart

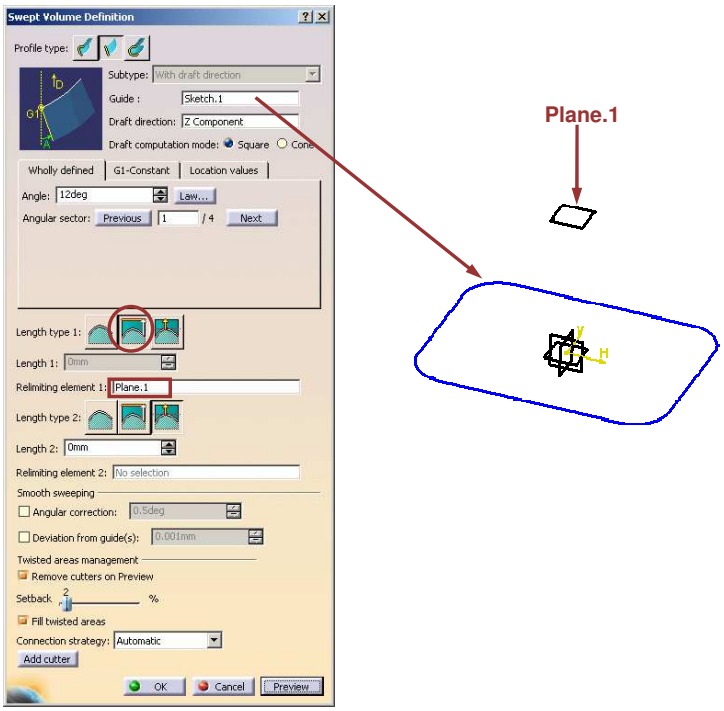
To create this type of Volume sweep you have to specify a Guide curve and a draft direction.

1. Select Volume Sweep

2. Select 'Line' Draft. Enter Draft as 12 deg. Select Plane.1 as the element upto which the volume will be created

3. The Result you get is





Instructor Notes:

How to Create a Swept Volume – Circle Type

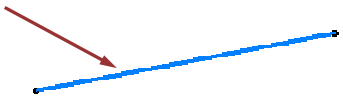
Swept_Volume_Circle.CATPart

Swept volume is a sketch based type of feature.

1 

2 **Select the profile type as circle.**

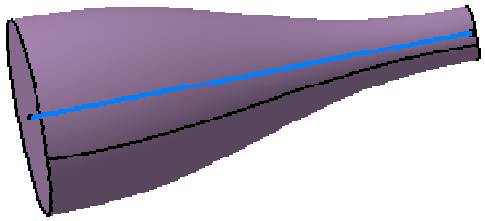
Center Curve



3 **The parameters are defined in the dialog box.Select Circle sweep.**

Here S- Type LAW is defined for the Radius value.

4 **Click OK to Validate**



Check various subtypes of circle sweep

Instructor Notes:

Creating Volumes From Surfaces

You will learn how to create Volumes from Surfaces

Instructor Notes:

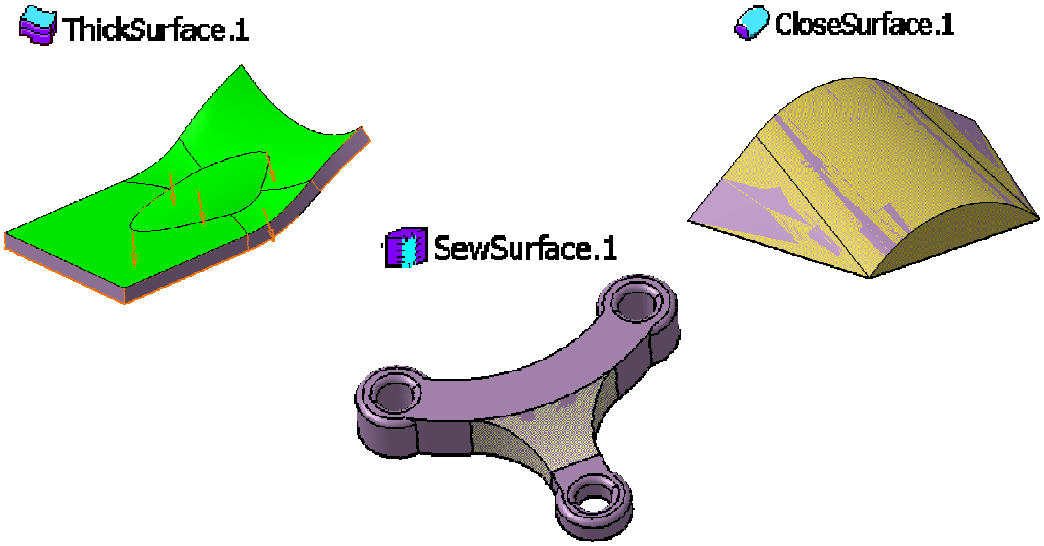
Volumes Made From Surfaces

A volume made from a surface can be generated by three different methods.
It can be either a ThickSurface or a CloseSurface or a Sewsurface.

These kinds of volumes can be the result of a constant offset (ThickSurface).

They can also fill a surface (CloseSurface).

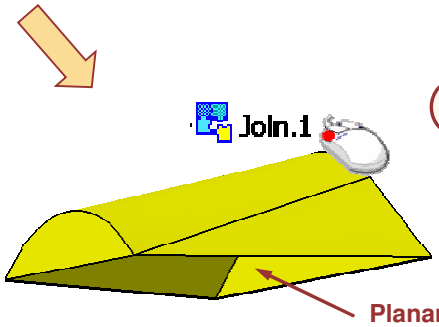
Sewing is an operation by which we combine a surface with a body.

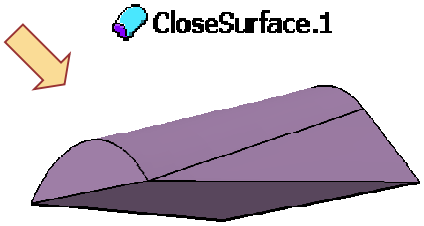


Instructor Notes:

How to Create Volumes by Closing Surfaces

CloseSurface are volumes based on a surface



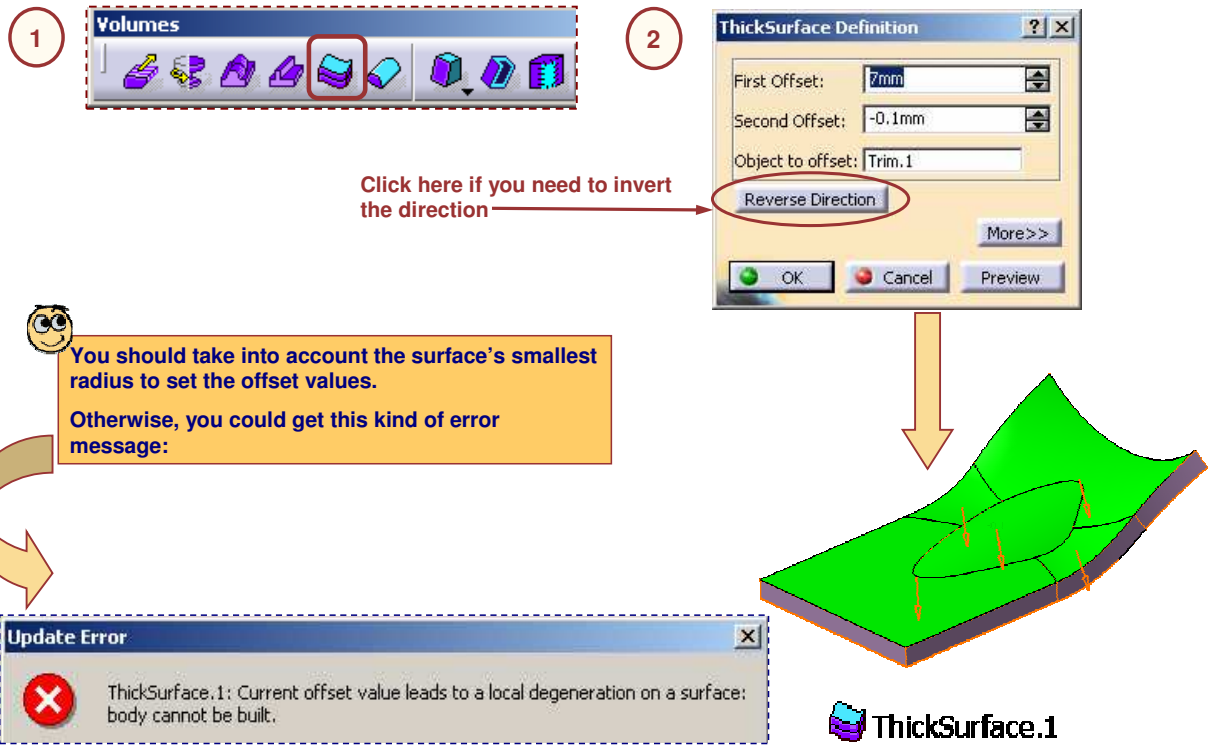


You should use a closed surface or an open surface that can be closed by a plane.

Instructor Notes:

How to Create a Volume by Thickening a Surface

A ThickSurface is a constant offset of an existing surface



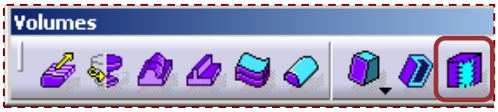
Instructor Notes:

How to Create a Volume by Sewing a Surface

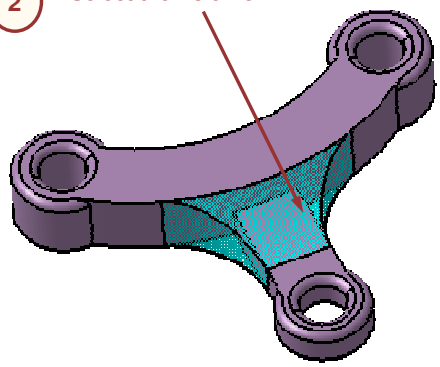
Sew_surface.CATPart

A SewSurface is an operation by which we combine a surface with a volume

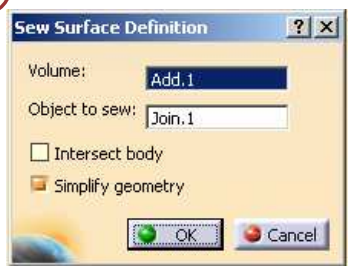
1



2 Select the Volume



3 Select the surface to combine with the volume

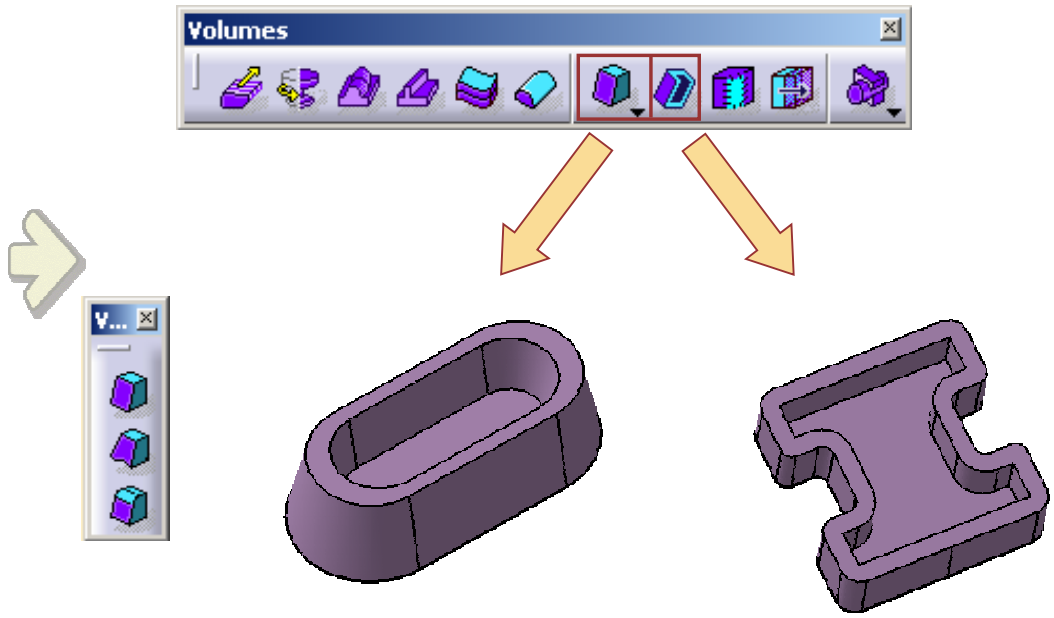


4 Click OK to Validate

Instructor Notes:

Applying Dress-up Features on Volumes

You will learn how to apply Dress-Up Features on Volumes



Instructor Notes:

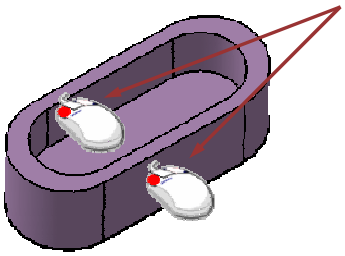
How to Apply Draft to a Volume

Draft_Volume.CATPart

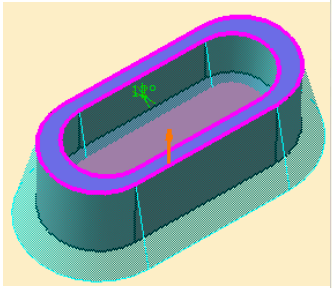
Drafts are defined on molded parts to make them easier to remove from molds

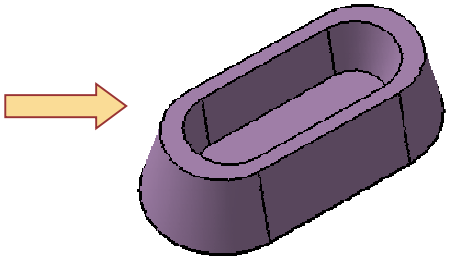
1

2 Select the faces to be drafted



3 Enter the parameters





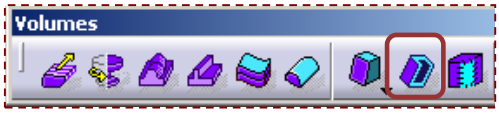
Instructor Notes:

How to Shell a Volume

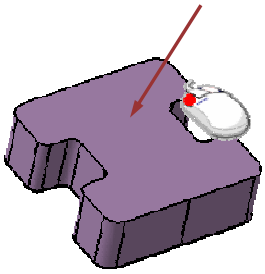
Shell_Volume.CATPart

Shelling a feature means emptying it, while keeping a given thickness on its sides

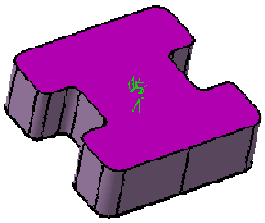
1

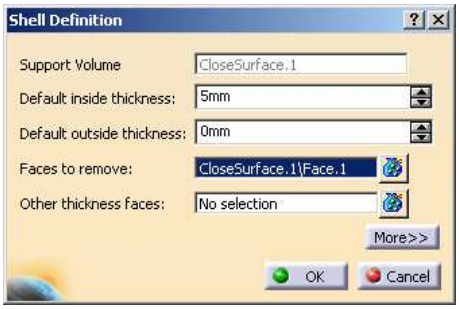


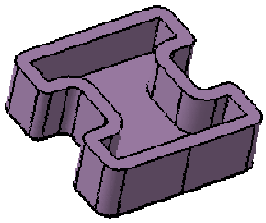
2 Select the faces to be shelled



3 Enter the parameters



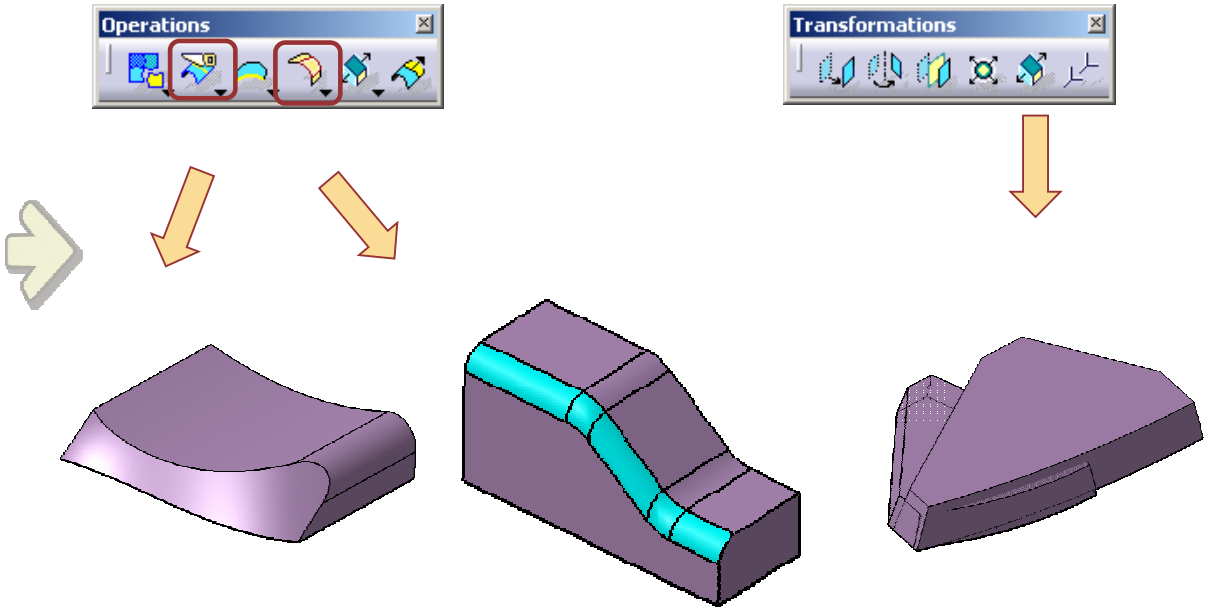




Instructor Notes:

Transformations and Operations on Volumes

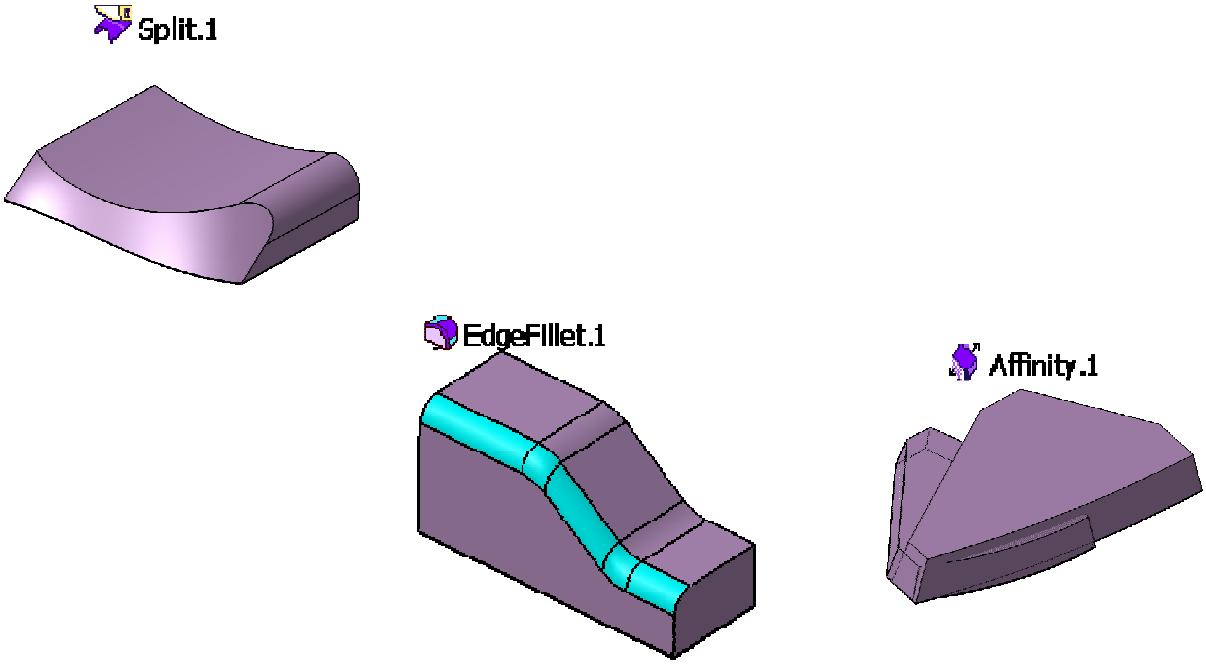
In this skillet you will learn how to perform Operations and Transformations on Volumes.



Instructor Notes:

Volumes by Performing Operations and Transformations

A volume can be dressed up with fillets and splits. It can also be transformed using translations, rotations, symmetry, scaling, affinity and axis to axis.



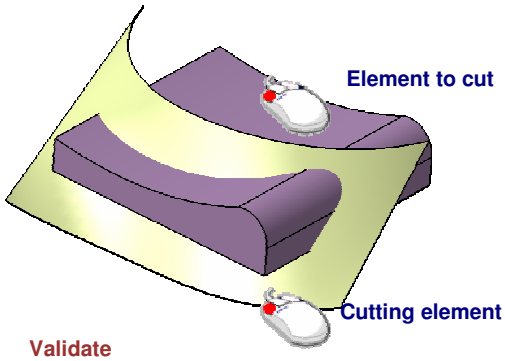
Instructor Notes:

How to Split Volumes

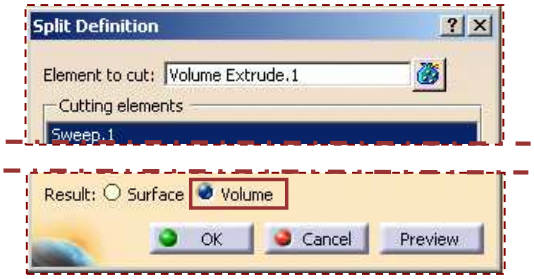
You will learn how to split a volume using a surface or a plane

1. Click on Split icon

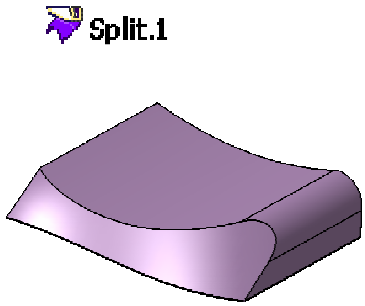
2. Select the operation elements



3. Select the "volume" option



4. Validate



Instructor Notes:

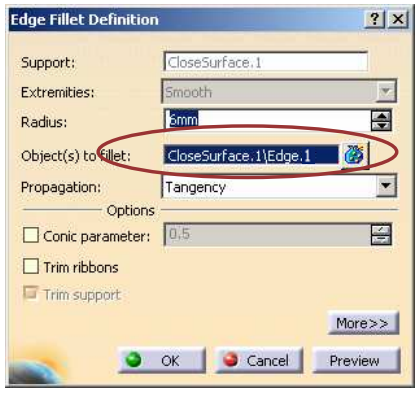
How to Apply Fillet on Volumes

Fillet_Volumes.CATPart

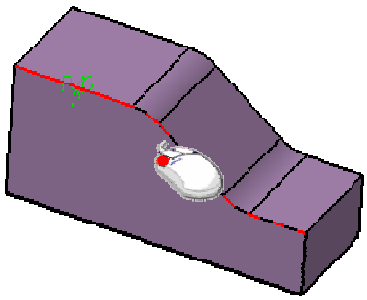
You will learn how to create volumes using operations

1 Select Edge Fillet

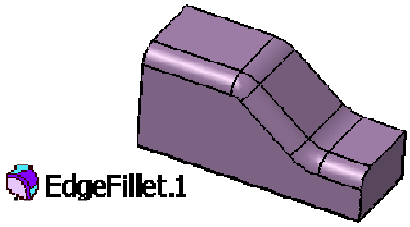
2 Set the Fillet parameters



3 Select the edges to fillet



4 Validate



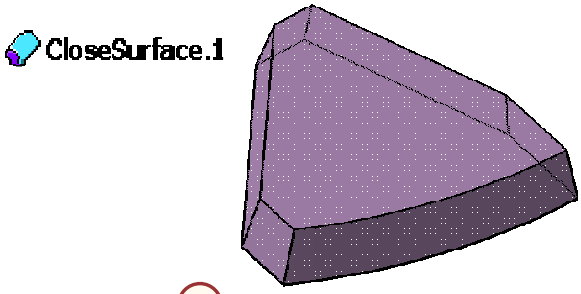
Instructor Notes:

How to Transform a Volume

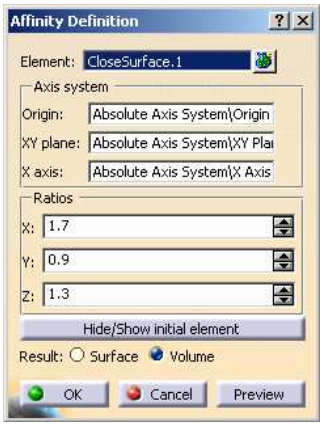
You will learn how to create volumes using transformations

1 Example: Affinity 

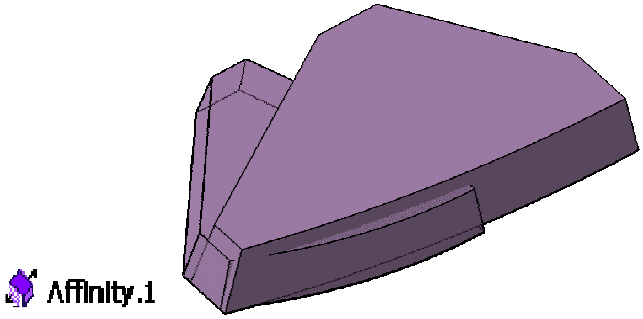
2 Select the volume



3 Set the affinity parameters



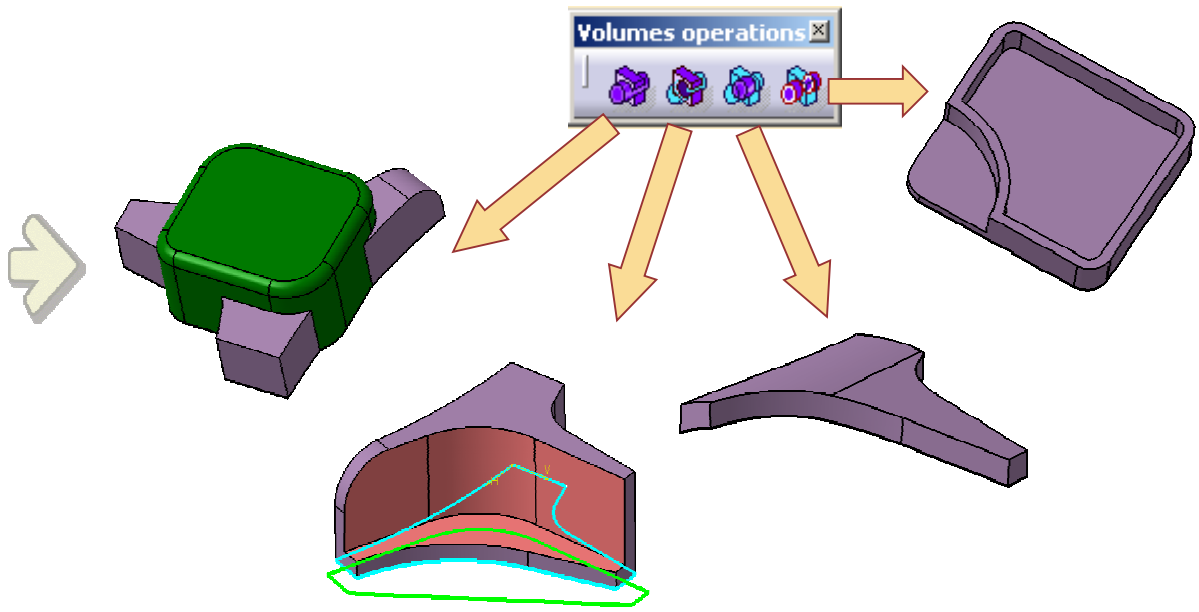
4 Validate



Instructor Notes:

Performing Boolean Operations on Volumes

You will learn how to perform boolean operations between two Volumes



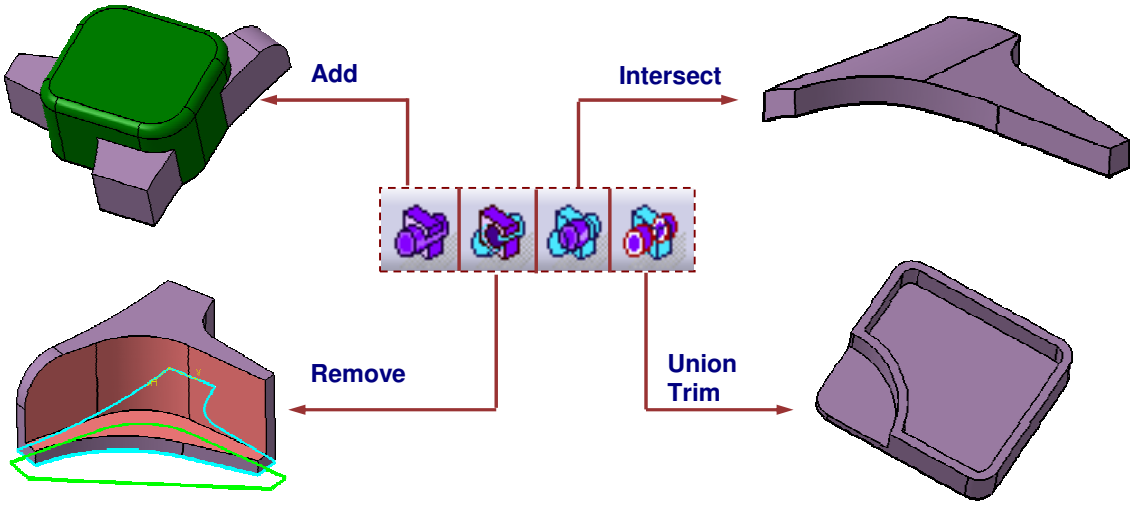
Instructor Notes:

What is a Volume Made From a Sketch

A volume made from a sketch can be generated by two different methods.
It can be either an extruded volume or a revolution volume.

Volumes made from a Boolean operation enable to create a shape following several steps. In each step, the part is built by adding or removing volumes.

You can even combine two volumes using the Intersect. You can also trim (union trim) one volume with respect to other and select the faces to remove (or keep)



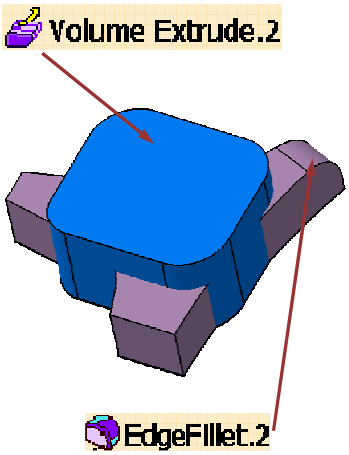
Instructor Notes:

How to Add Volumes

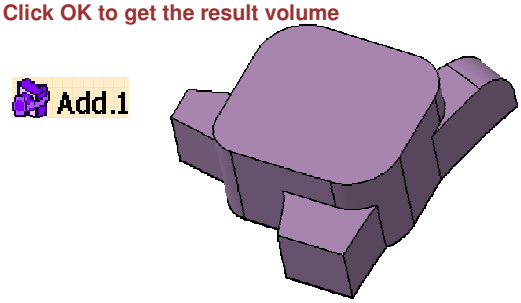
Adding a volume to another volume is uniting the two volumes.

1 

2 Select the two volumes



3 Click OK to get the result volume



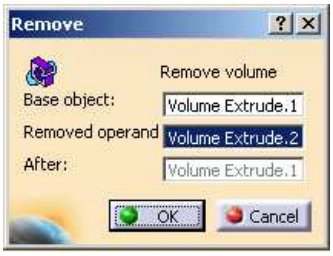
Instructor Notes:

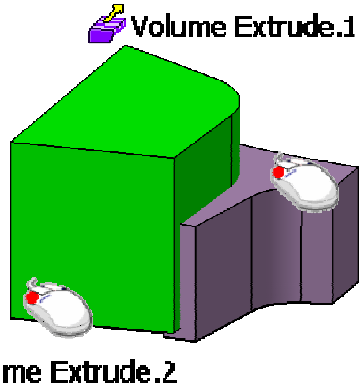
How to Remove A Volume from another

Removes a volume from another volume.

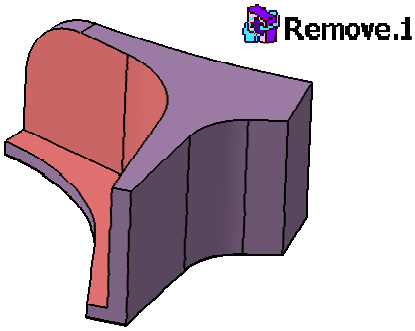
1 

2 Select the volume to remove, then the volume to be cut





3 Click OK to get the result volume



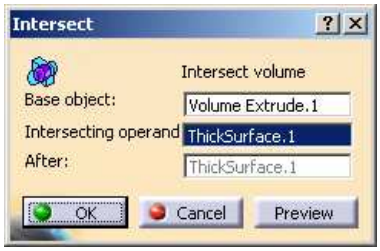
Instructor Notes:

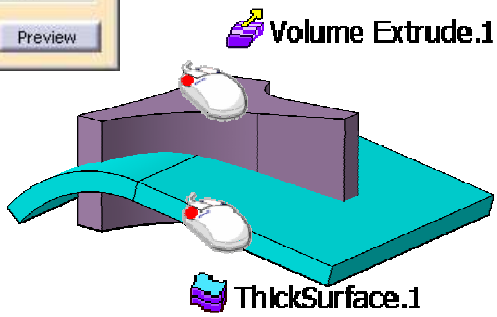
How to Intersect Two Volumes

Intersected Volumes are equivalent to the common volume between two volumes

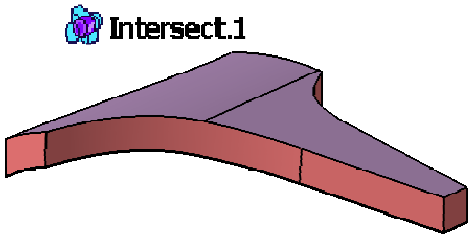
1

2 **Select the volumes to intersect**





3 **Click OK to get the result volume**



Instructor Notes:

How to Trim Volumes

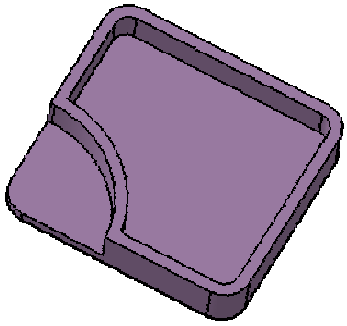
Union_Trim_Using_Boolean_Operations.CATPart

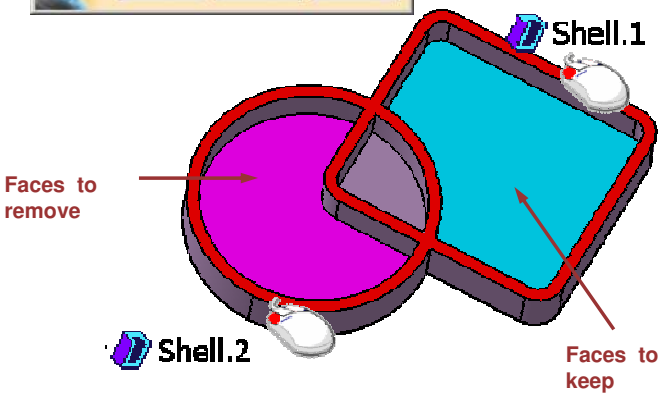
Trims a volume with respect to another by defining elements to be kept or removed while performing the union operation.

1 

2 Select the volumes to intersect

3 Click OK to get the result volume





Instructor Notes:

To Sum Up

In this section, you have been introduced to volumes.
You have seen:

- **The volume creation tools**
- **The different ways to create volumes**
- **How to combine several volumes using Boolean operations**
- **How to modify existing volumes using operations or transformations**

Instructor Notes:

Generative Shape Design Optimizer Exercises

This lesson provides you with a collection of exercises to practice.

- Door Junction Exercise
- Holding Arm - Volumes Exercise 1
- Torch – Volumes Exercise 2
- Steering Wheel Exercise

Instructor Notes:

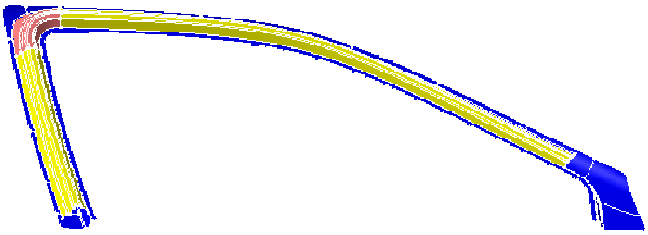
Door Junction

Door Junction Exercise Presentation

40 min

In this exercise you will see how to create a Door Junction using the Generative Shape Optimizer tools:

- Creating reference and target curves for Wrap Curve surfaces :
  - Extracted curves
  - Boundary
  - Parallel curve
  - Split operations
- Creating the Wrap Curves surfaces
- Creating the Sections and Coupling Curves for Junction :
  - Extracted curves
  - Boundary
  - Spline curve
  - Join and Split operations
- Creating the Junction

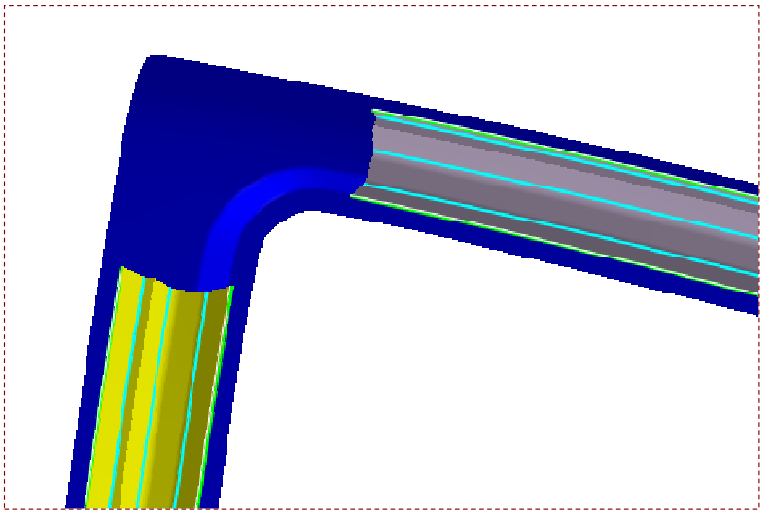


Instructor Notes:

Wrap Curves – Step 1

Creating the reference and target curves for Wrap Curve surfaces

15 min

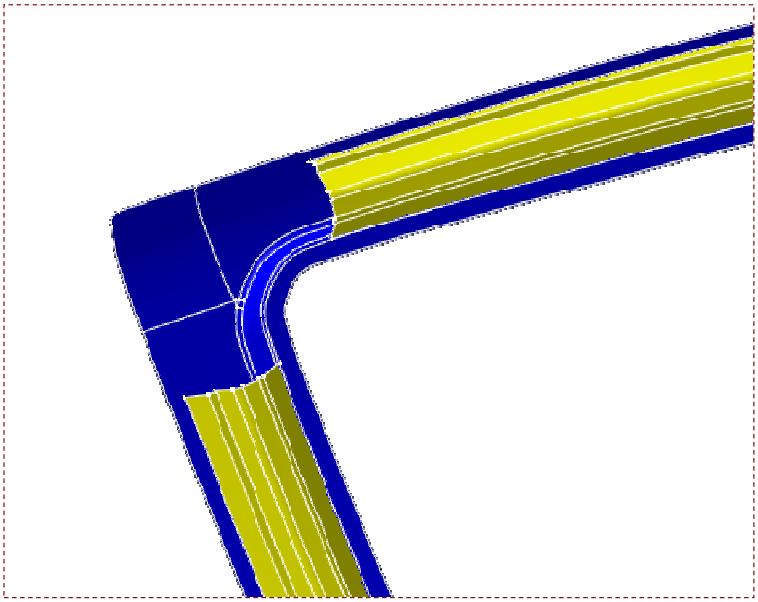


Instructor Notes:

Wrap Curves – Step 2

Creating the Wrap Curve surfaces

10 min

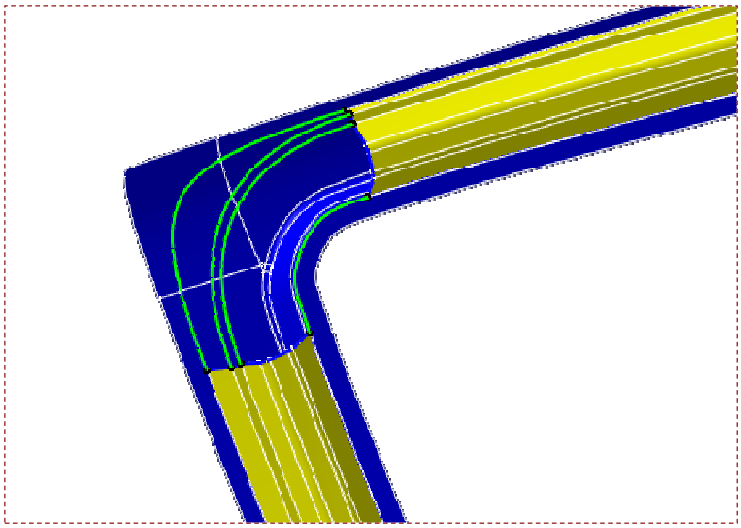


Instructor Notes:

Junction – Step 1

Creating the sections and coupling curves for Junction

10 min

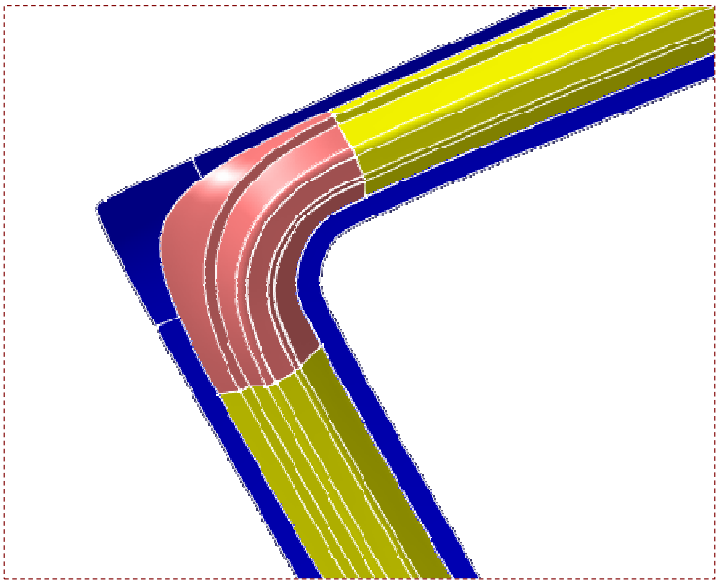


Instructor Notes:

Junction – Step 2

Creating the Junction

5 min



Instructor Notes:

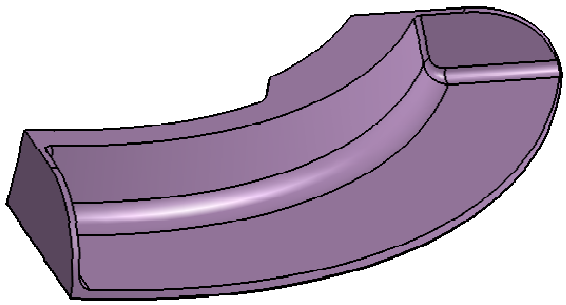
Holding Arm – Volumes Exercise 1

Holding Arm Exercise Presentation

30 min

In this step you will:

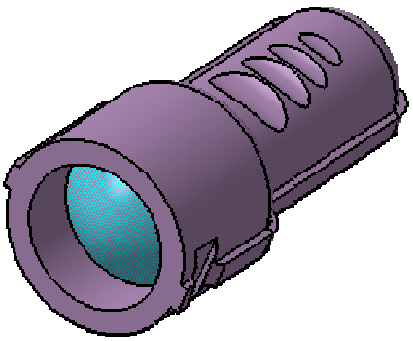
- Create three basic Revolve Volumes
- Prepare these volumes to be assembled
- Perform Boolean Operations



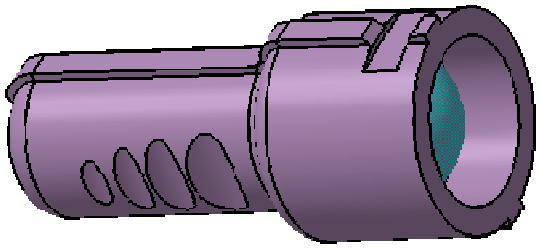
Instructor Notes:

Torch – Volumes Exercise 2

Torch Exercise Presentation



In this exercise you will design the Torch using 'Volumes' in the Generative Shape Design Optimizer workbench.



Instructor Notes:

Steering Wheel

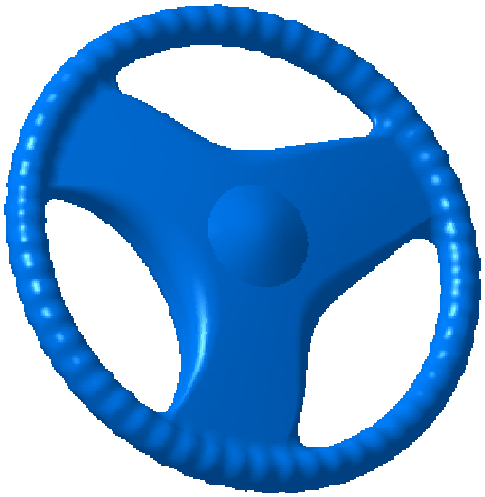
Generative Shape Design Optimizer exercise

90 min

In this exercise you will build the Steering Wheel by following a recommended process.

Here you will :

- **Create the Steering Ring and Handle Area using advanced surfaces**
- **Convert these advanced surfaces to volumes**



Instructor Notes:

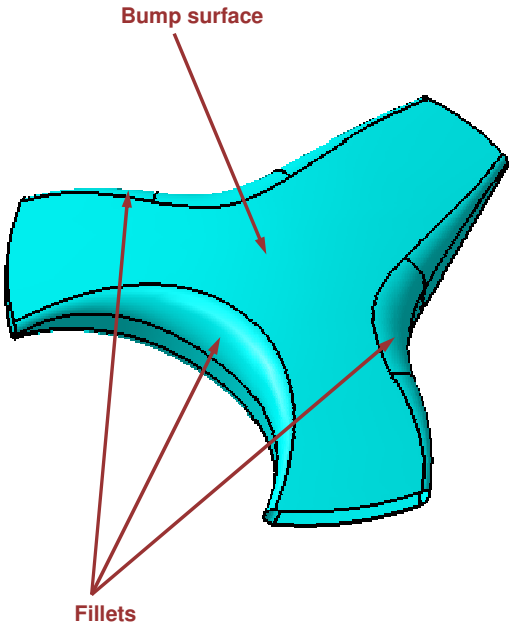
Steering Wheel Exercise

Step 1: Design the Handle Area

30 min

In this step you will create :

- **Bump Deformation**
- **Symmetry of Bump**
- **Multi-section surface to define the side surfaces**
- **Fillets**



Instructor Notes:

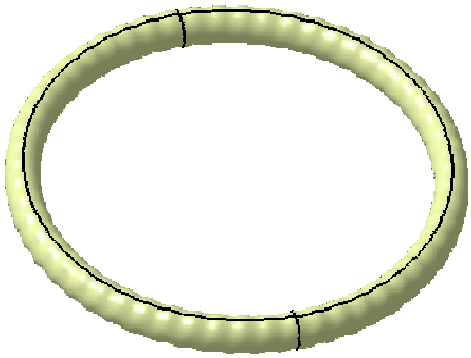
Steering Wheel Exercise

Step 2: Design the Steering Ring

30 min

In this step you will create :

- **Surface of Revolution**
- **Developing Wire**
- **Surface Morphing**



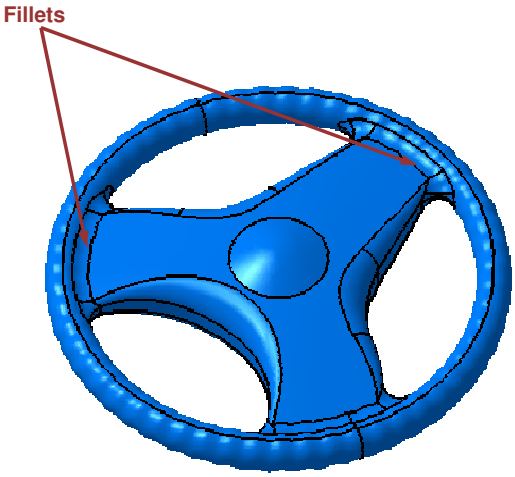
Instructor Notes:

Steering Wheel Exercise

Step 3: Completing the Model

In this step you will create :

- **Close volumes**
- **Fillets**
- **Volumes using Boolean operations**



Instructor Notes:

To Sum Up

In this course you have learnt:

- **How to create offsets, develop shapes**
- **How to use BIW tools**
- **How to create advanced surfaces like Bumped surface**
- **How to create Volumes**

Instructor Notes: