STUDENT GUIDE

Student Notes:

CATIA V5 Training
Foils

# CATIA Generative Shape Design

Version 5 Release 19
September 2008

EDU_CAT_EN_GSD_FF_V5R19

Student Notes:

# About this course

## Objectives of the course

**Upon completion of this course, you will be able to:**

**-Correctly recall the tools of the Generative Shape Design workbench that are common to MD2 and HD2 licenses**

**- Identify and use the Generative Shape Design tools that are specific to the HD2 license**

**- Create advanced and parameterized swept surfaces**

**- Perform advanced surfaces analysis and gap correction**

**- Improve geometry quality and stability**

## Targeted audience

**Mechanical Surface Designers**

*2 days*

## Prerequisites

**Students who are familiar with the basics of wireframe and surfaces creation**

Student Notes:

# Table of Contents (1/3)

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Student Notes:

# Introduction to Surface Design

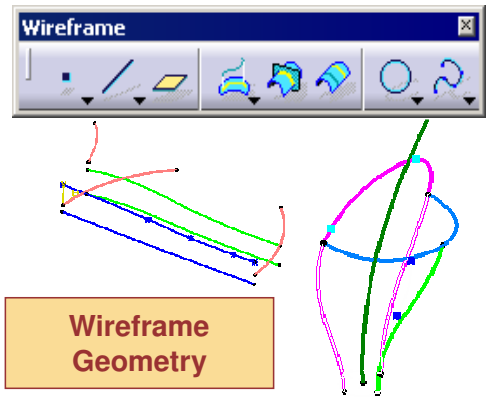
*In this lesson you will revise the concepts learnt in wireframe and surfaces course. You will later learn about managing features and Geometrical sets.*

- Surface Design Workbench Overview
- Managing Features and Geometrical Sets
- Geometrical Sets - Recommendations
- Recap Exercise: Managing Features
- Recap Exercise: Shampoo Bottle

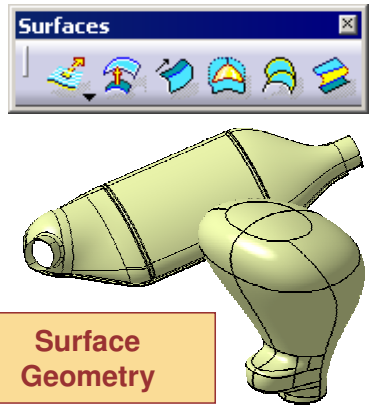
Student Notes:

# Surface Design Workbench Overview

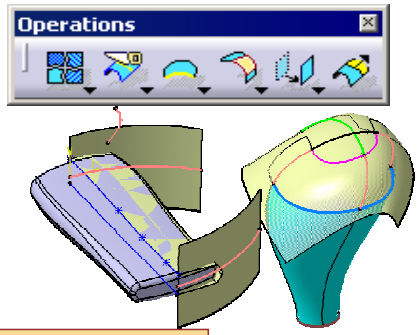
*You will revise the concepts learnt in Wireframe and Surfaces course.*




Wireframe Geometry




Surface Geometry




Operations

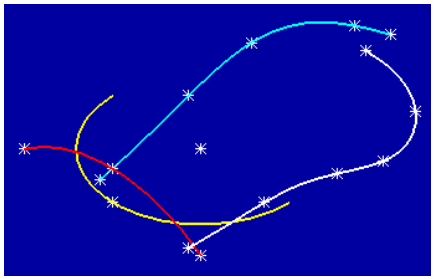

Student Notes:

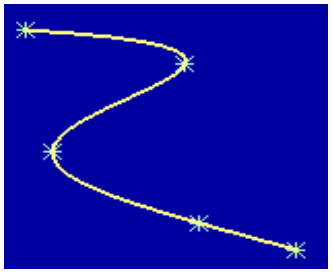
# Review of Surface Design Basics (1/2)

**In the Surface Design Fundamentals, you have learnt the basics of creation, modification & completion of geometry which is in the form of wireframes and surfaces. You have also learnt about Shape design Common tools.**

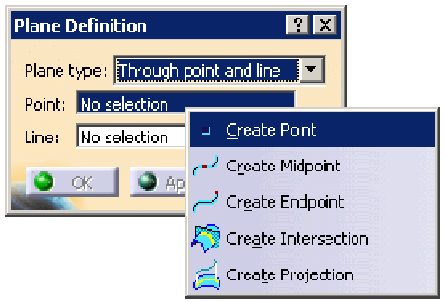
**Here you will review the concepts learnt in wireframe and surface design course**

- **Creating wireframe Geometry**
  - **Creating Points**
  - **Creating Lines**
  - **Creating Planes**
  - **Creating Curves**



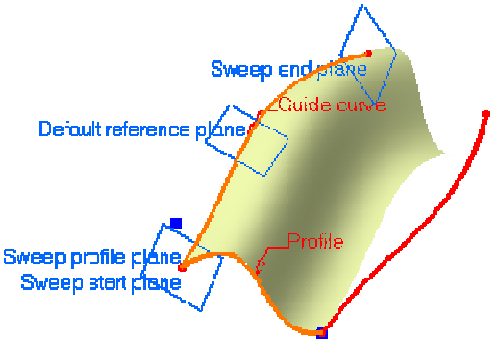


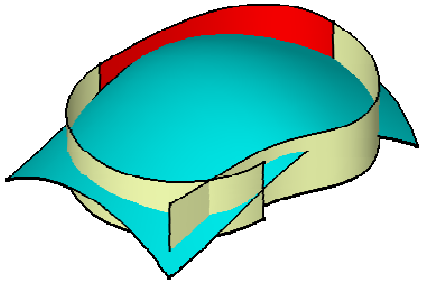
- **Shape Design Common Tools**
  - **About Stacking Commands**
  - **Managing Geometrical Sets**




# Review of Surface Design Basics (2/2)

- Creating Surfaces
  - Creating Basic surfaces
  - Swept surfaces
  - Multi-Section surfaces



- Modifying the Geometry
  - Editing Elements,
  - Imposing value ranges for parameters
  - Identifying Parent Child relations

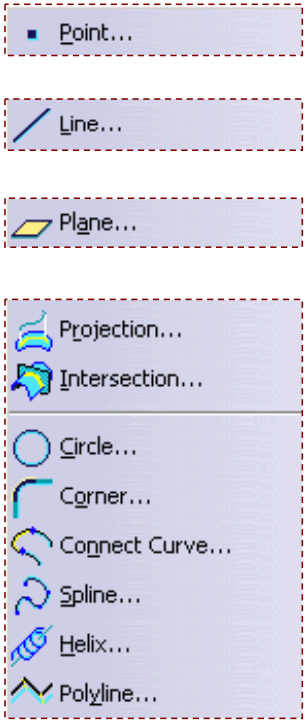
- Using Tools
  - Creating Datum features
  - Checking Connection between elements
  - Updating a Part
  - Working on Support
  - Manipulating Elements.

Student Notes:

Student Notes:

# Review of Wireframe Covered in the Surface Design course

*You can review the tools in the GSD Workbench already covered in the Surface Design Course.*




Creating Points in 3D

Creating Lines in 3D

Creating Planes in 3D

Creating Curves in 3D

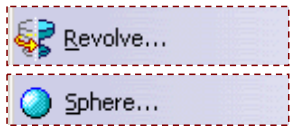
Student Notes:

# Review of Surfaces covered in the Surface Design course

***You can review the tools from the Generative Shape Design workbench already covered in the Surface Design Course.***

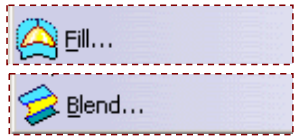

**Creating a Surface from a profile**

***Creating an Extruded Surface***




***Creating a Surface of Revolution***

***Creating a Sphere***




**Creating a Surface from Boundaries**

***Creating a Fill Surface***

***Creating a Blend Surface***


**Creating a Surface from another Surface**

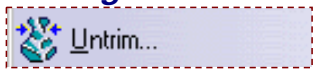
***Creating an Offset Surface***


**Creating a Lofted Surface (Multi-sections surface)**

Student Notes:

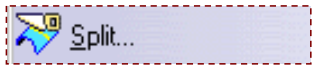
# Review of Operations covered in the Surface Design course

***You can review the tools from the Generative Shape Design workbench already covered in the Surface Design Course.***

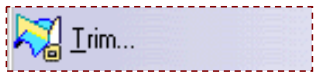



**Restoring Surfaces**


**Disassembling Surfaces**




**Splitting Elements**

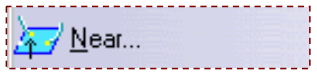



**Trimming Elements**


**Transforming Elements**

- *Translating an Element*
- *Rotating an Element*
- *Applying a Symmetry to an Element*
- *Scaling an Element*
- *Creating an Affinity*
- *Performing an Axis-to-Axis transformation*


**Extrapolating Elements**

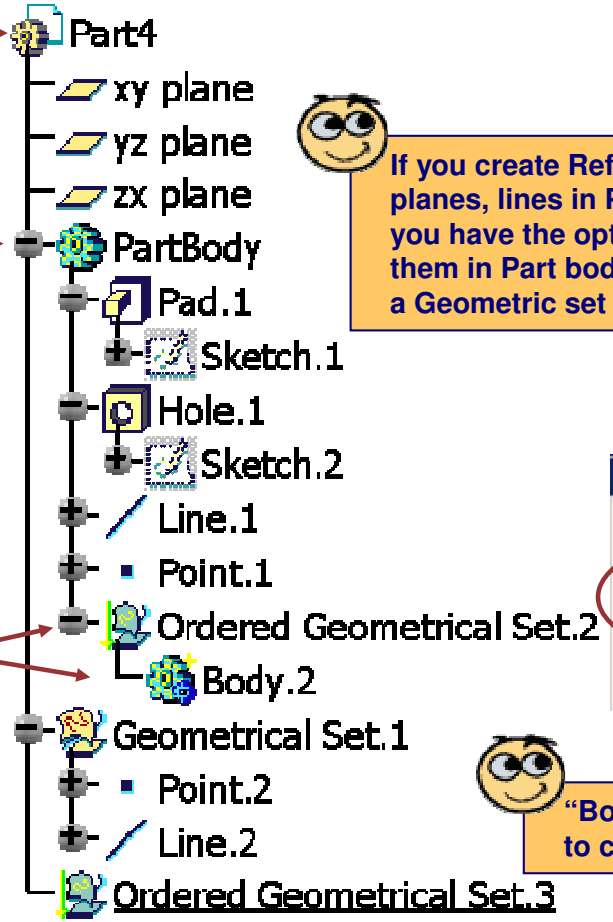



**Creating Near Elements**

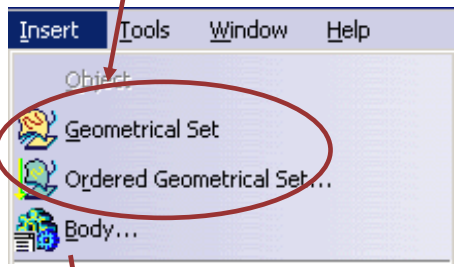
Student Notes:

# Surface Design Workbench Terminology

- **Part** is a combination of Part Body and Geometrical Sets.
- **PartBody** basically contains the features used to create a solid. It can contain surfacic and wireframe elements also.
- **Geometrical Sets** contain the features used to create surface and wireframe elements.
- **Ordered Geometrical Sets(OGS)** contain surface and wireframe . The elements in this body are created in a linear manner. OGS can also contain "Body" . Body allows creation of Part Design Solids within an OGS.




If you create Reference Elements ; points, planes, lines in Part Design Workbench , you have the option of directly containing them in Part body / Body , or you can insert a Geometric set and place these elements.



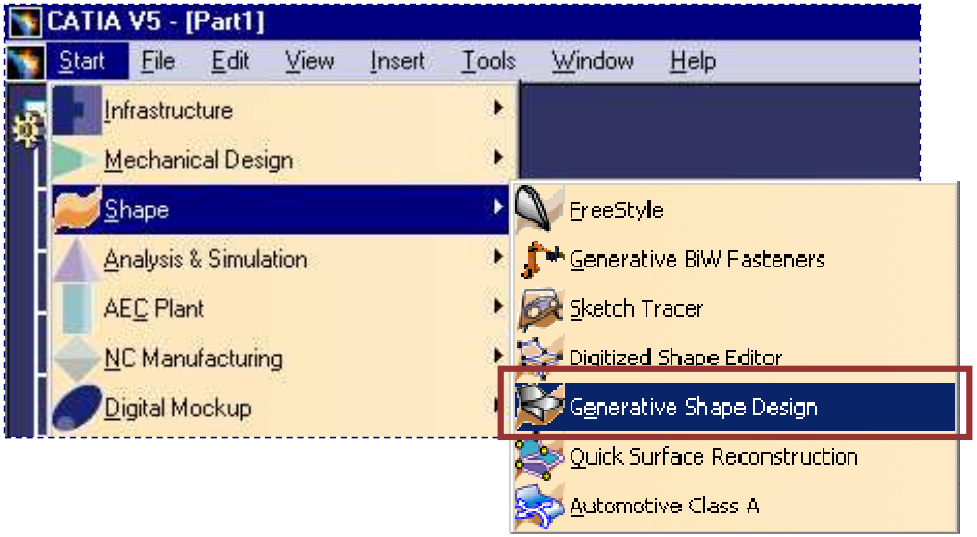

"Body" can be inserted in OGS to contain Part Design Solids.

When you enter the Generative Shape Design workbench Part Body is the default body available. "Geometric Set", "Ordered Geometric Set" can be inserted manually.

Student Notes:

# Generative Shape Design Workbench

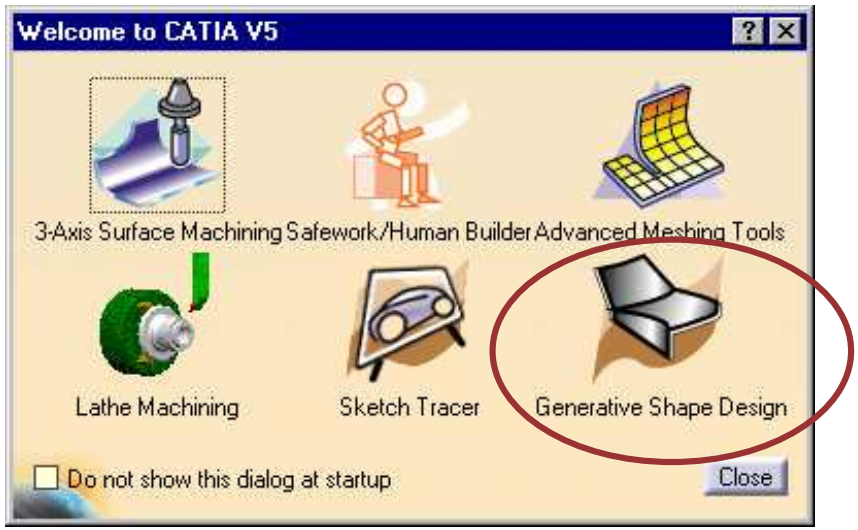
1




From the MENUBAR

Start/Shape/Generative Shape Design

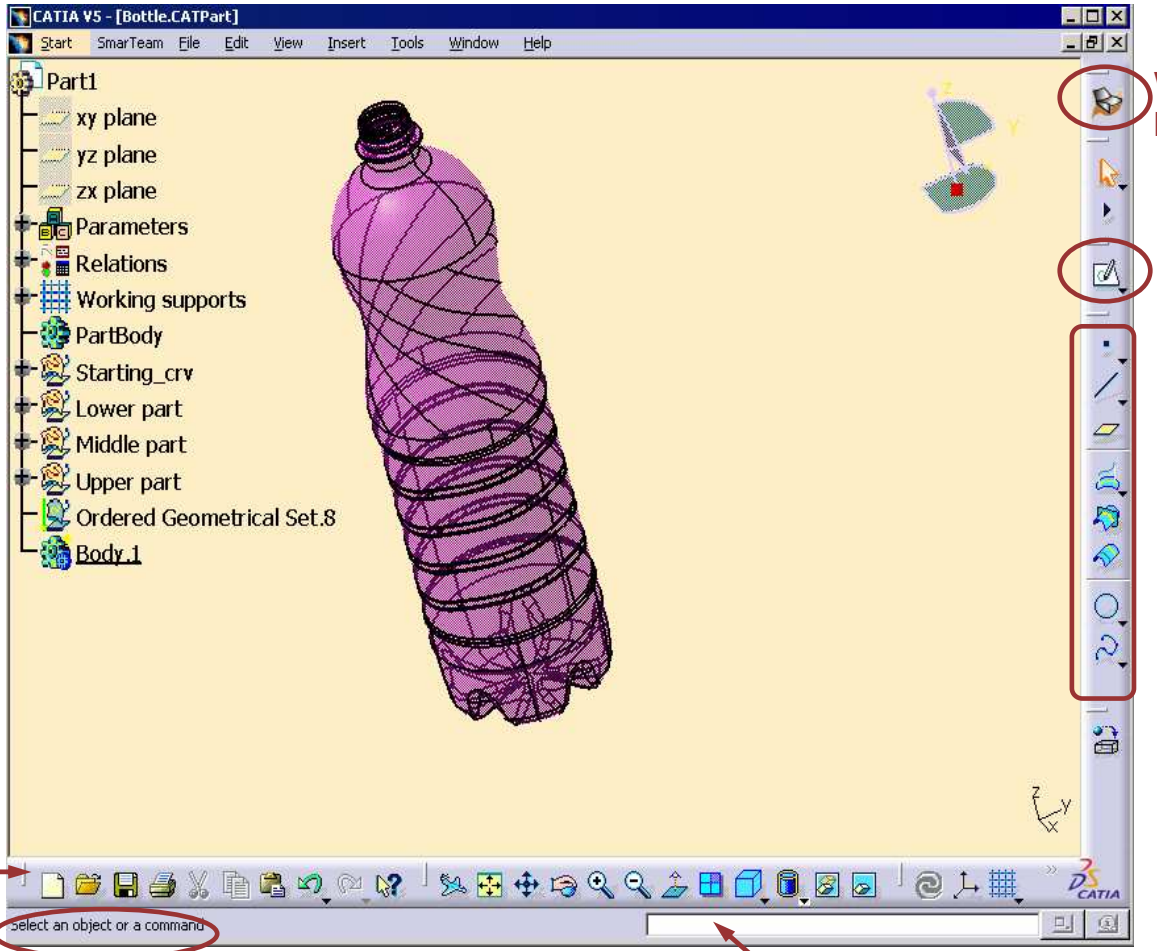
2




By clicking on the current Workbench icon (top right) to access the Favourite Workbenches window.

Student Notes:

# Generative Shape Design Workbench User Interface




Specification Tree

Containers of type Geometric Set , Ordered Geometric Set and Body

Standard tools

Workbench Icon

Sketcher access...

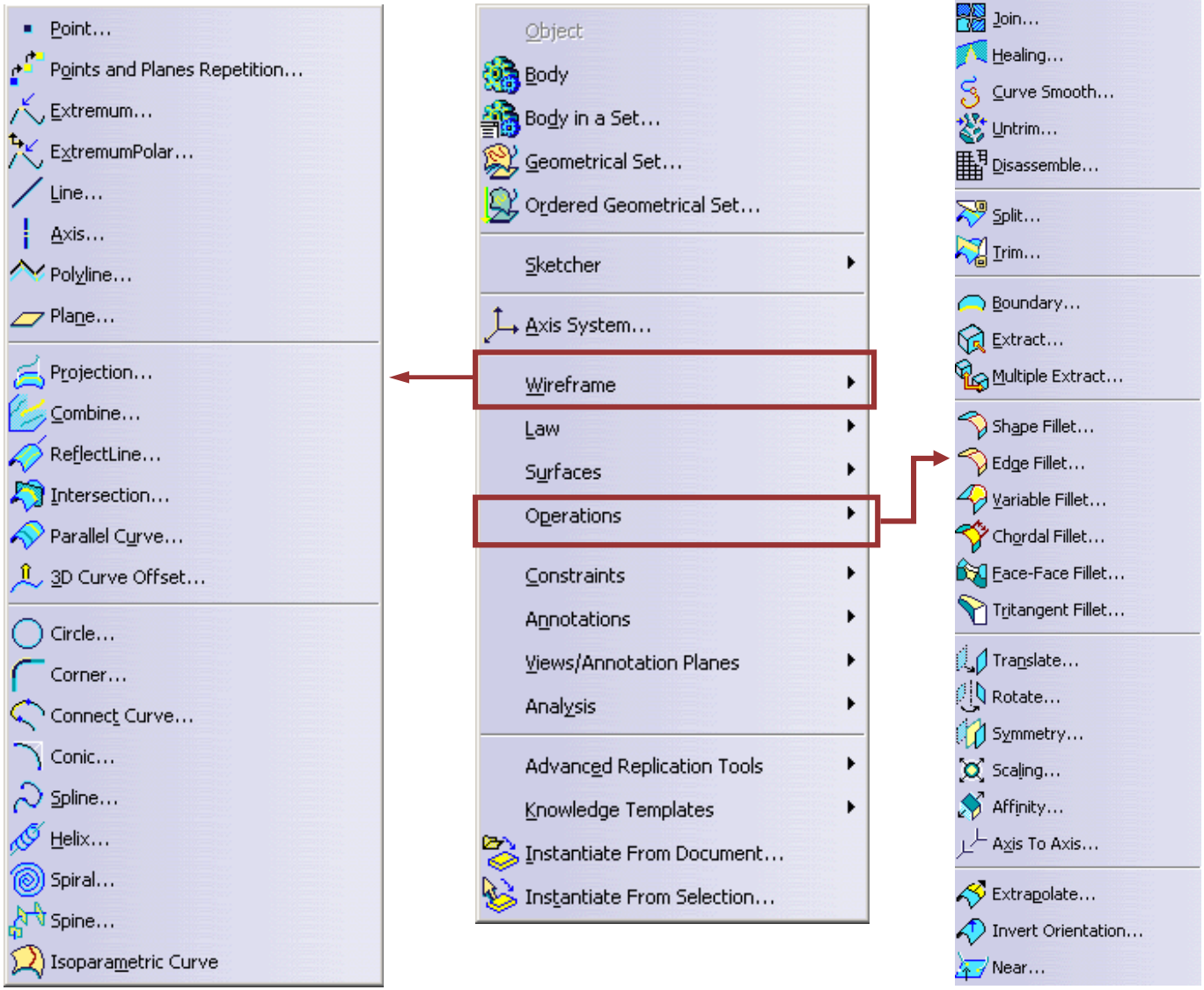
Shape Design tools...

This area shows current status

Command Bar

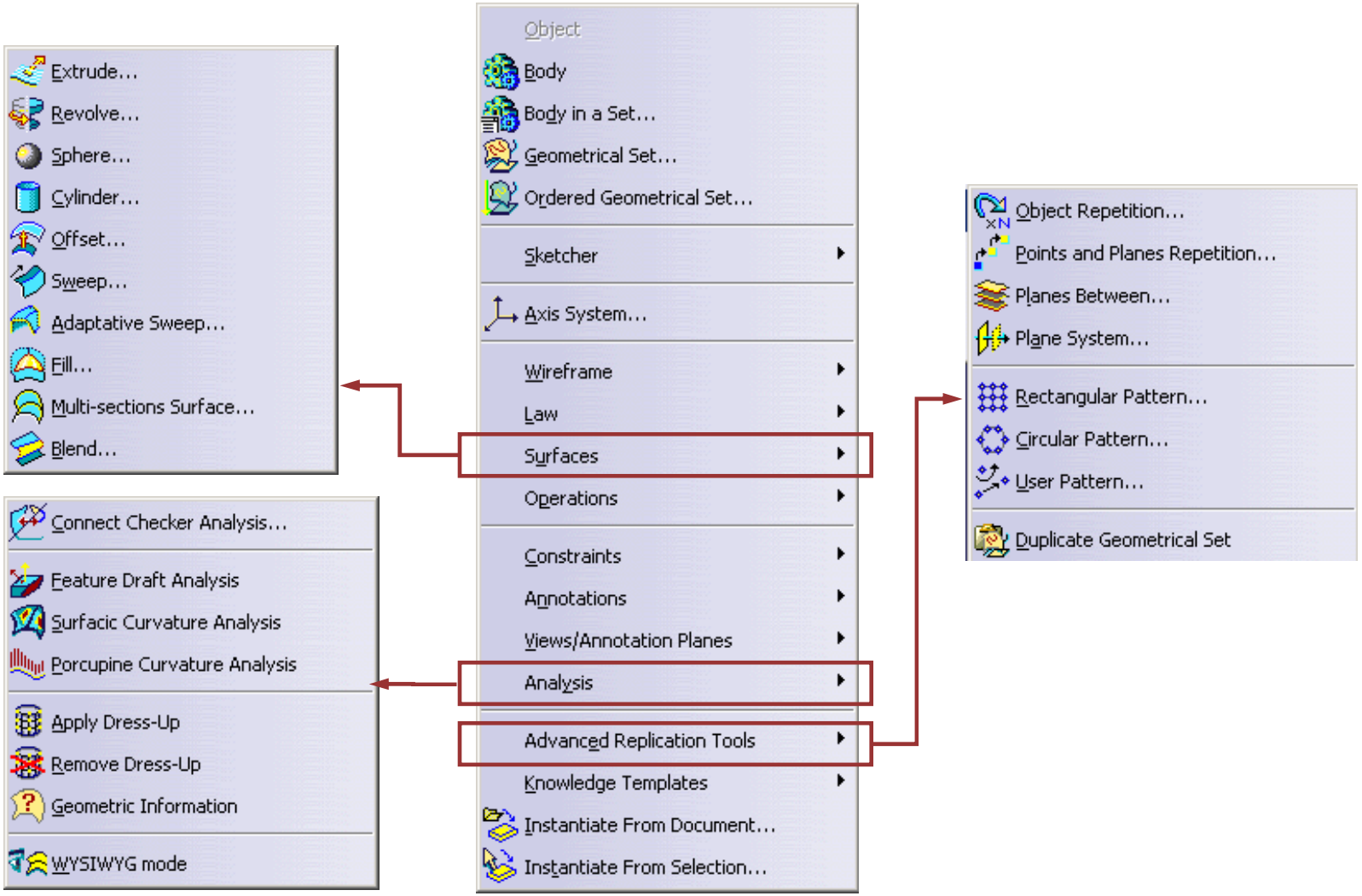
Student Notes:

# Generative Shape Design Workbench User Interface (1/2)



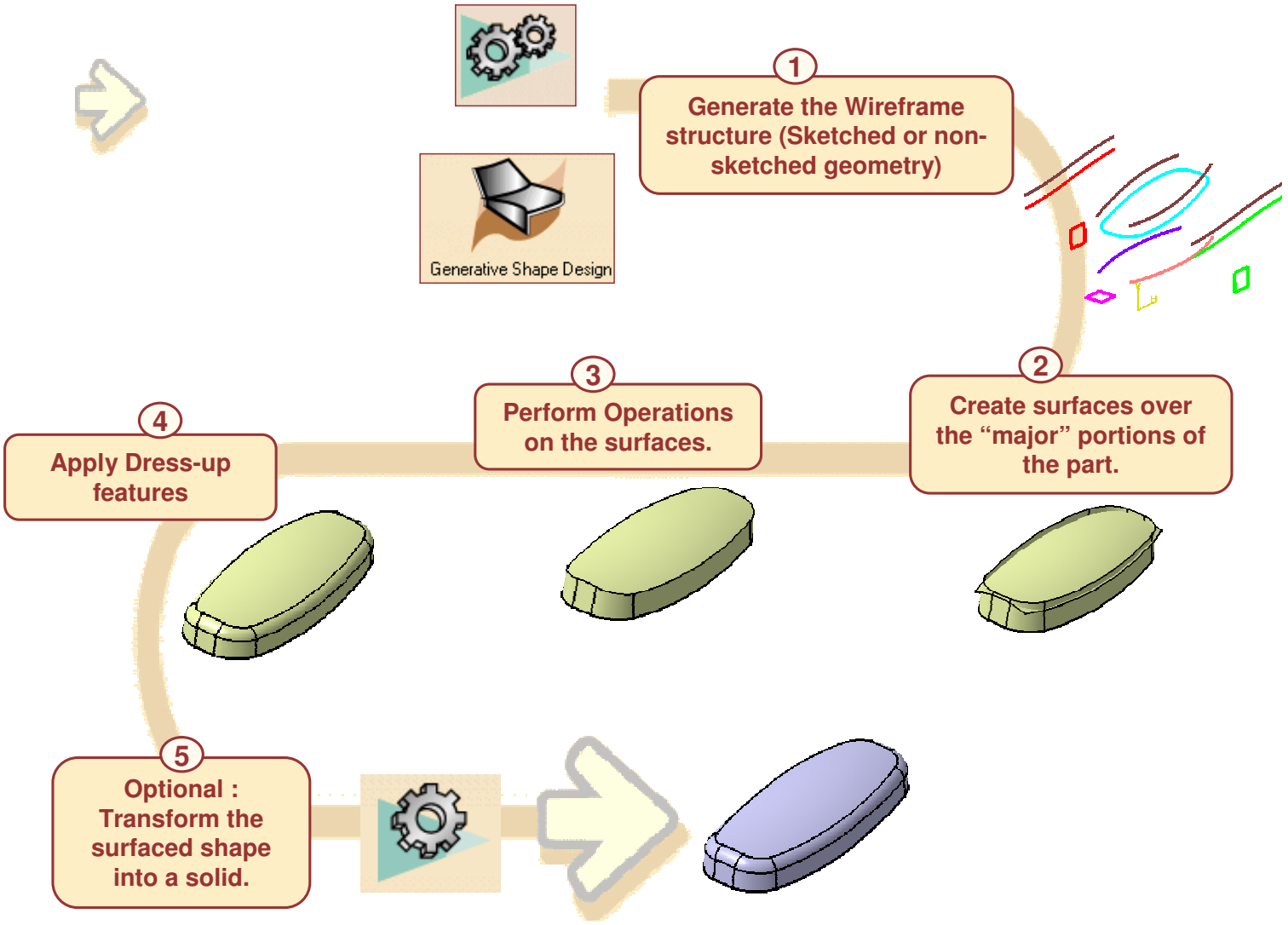
# Generative Shape Design Workbench User Interface (2/2)

Student Notes:



Student Notes:

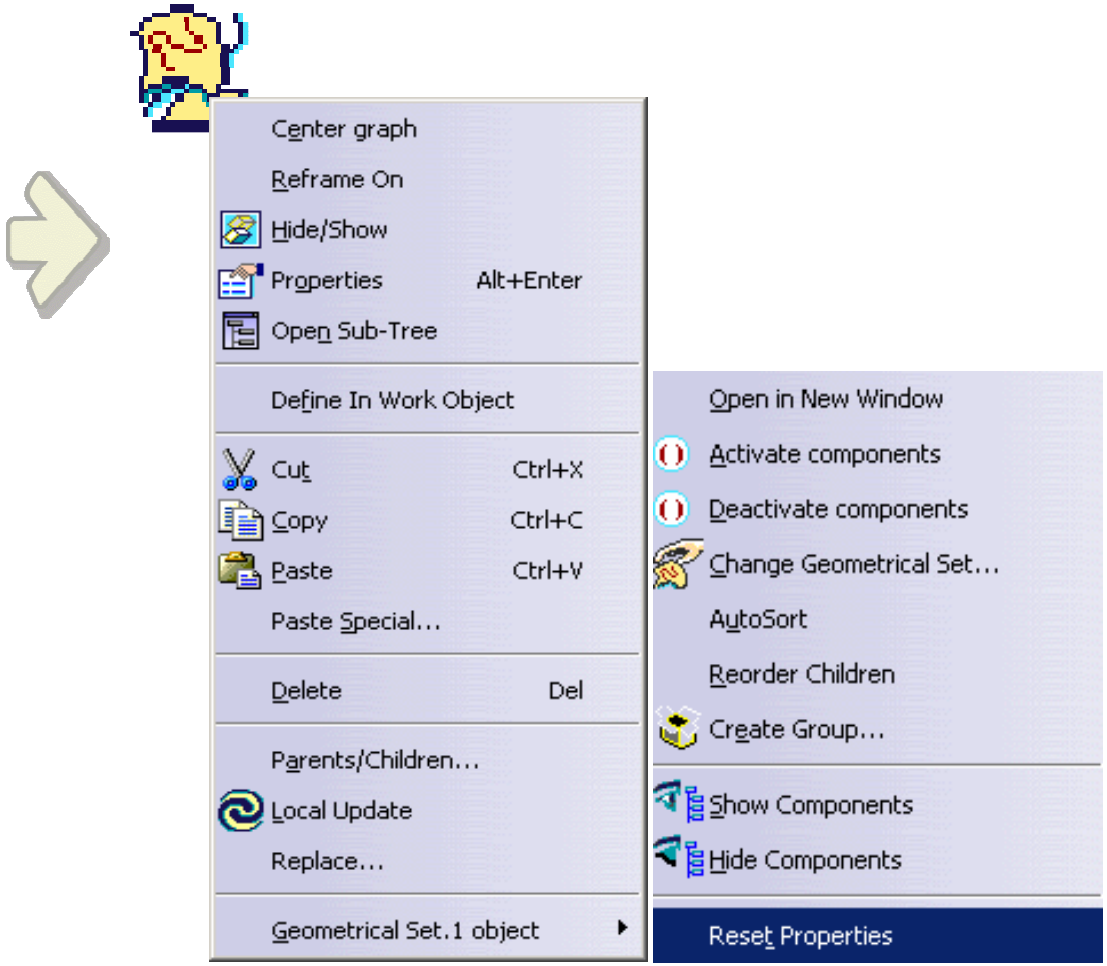
# General Surface Design Process



Student Notes:

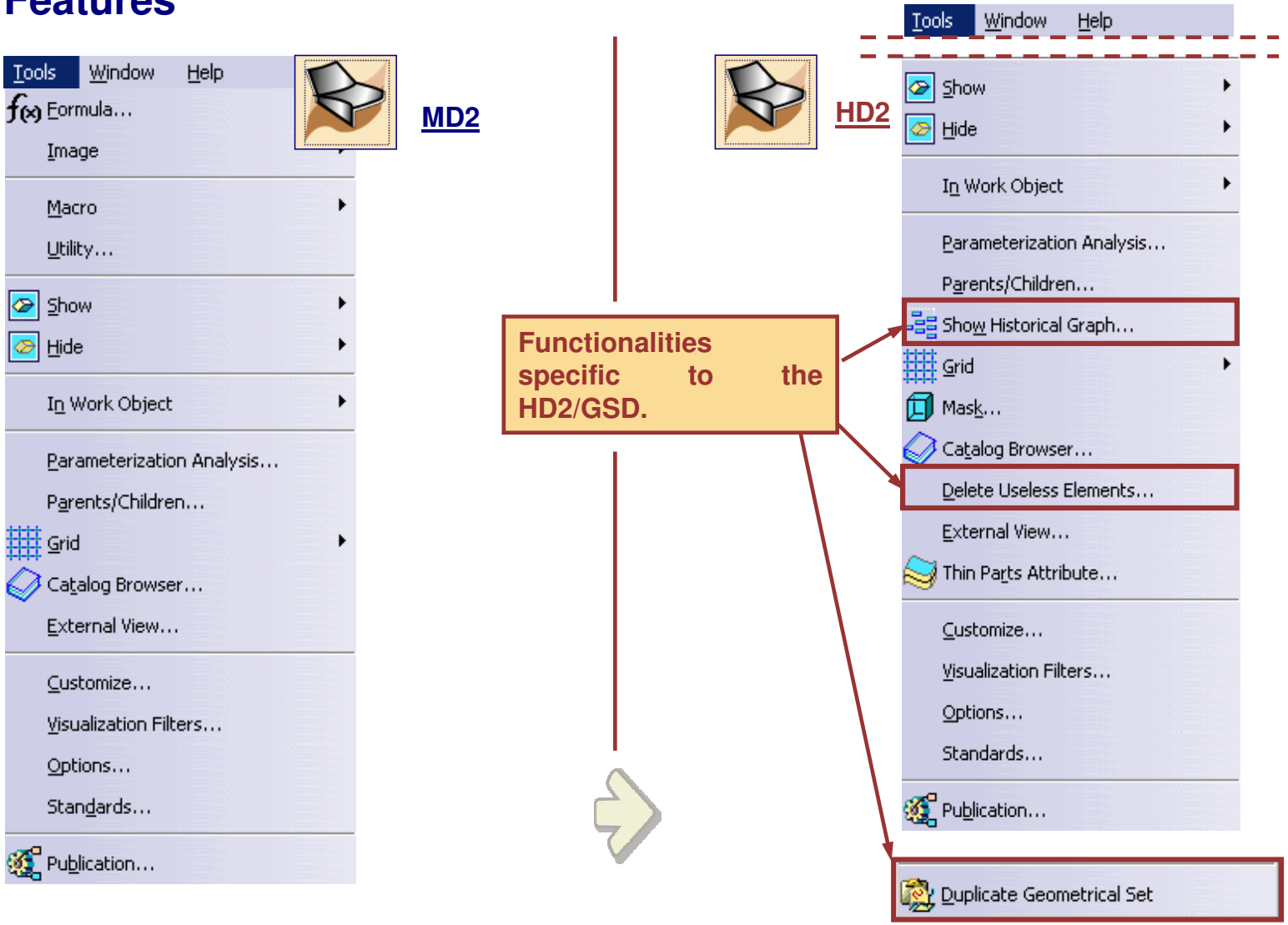
# Managing Features and Geometrical Sets

*You will see the different tools to manage features and Geometrical sets.*



Student Notes:

# MD2/GSD Management Features versus HD2/GSD Management Features



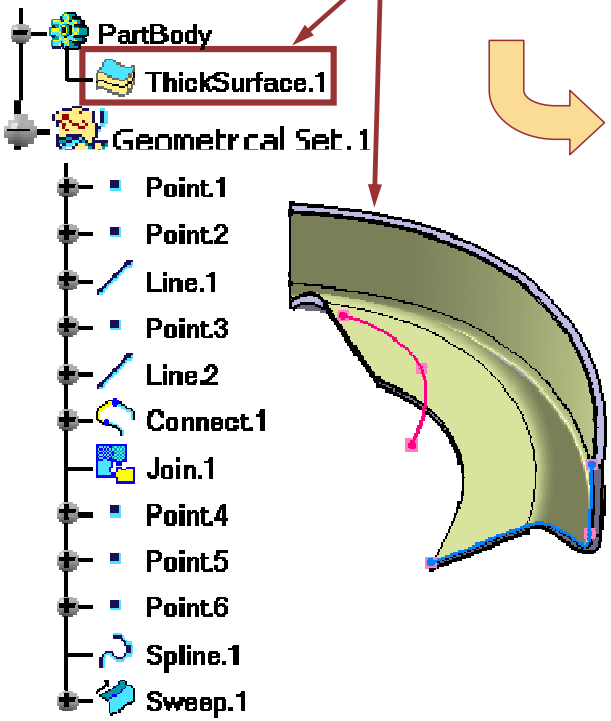
Student Notes:

# Using the Historical Graph (1/2)

**The Historical Graph allows you to display the hierarchical links between the different features of a part.**

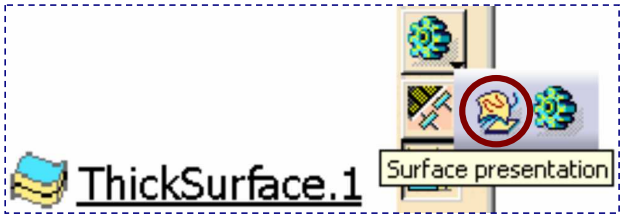
1 Select the feature from which you want to know the hierarchy.

2 Select the Historical Graph icon.



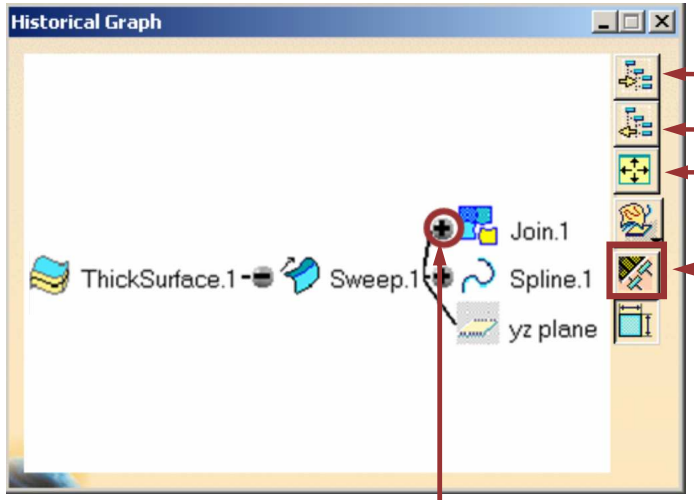



3 Select the Surface Presentation to display the surfacic hierarchical elements.



Student Notes:

# Using the Historical Graph (2/2)



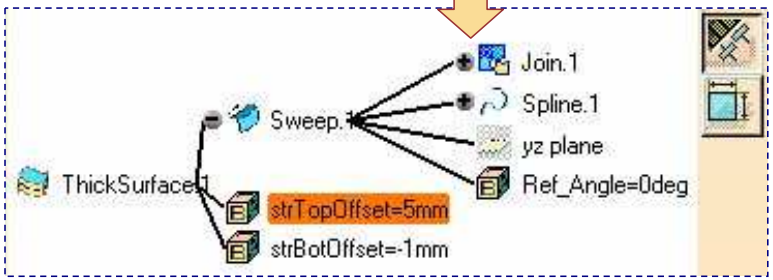

To add a Graph

To remove the Graph

Reframe the Graph

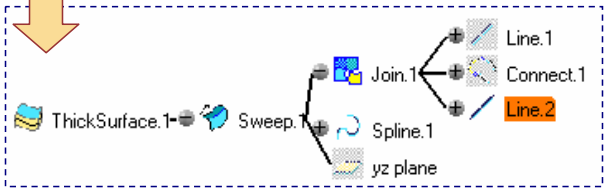
4b

Select the Parameter Filter button.




4a

Click on plus to expand the tree.




Double click a feature to edit and modify it.

You can Edit and modify a Parameter directly by double click on it

Student Notes:

# Why Geometrical Set Management Tools?

**In V5, during the creation and trimming of surfaces, the history of parent surfaces is kept in its entirety in order to allow for automatic update of downstream geometry following a modification of any parent surface. Due to this fact, the specification tree can get large and often confusing. The tools listed below help manage this tree.**

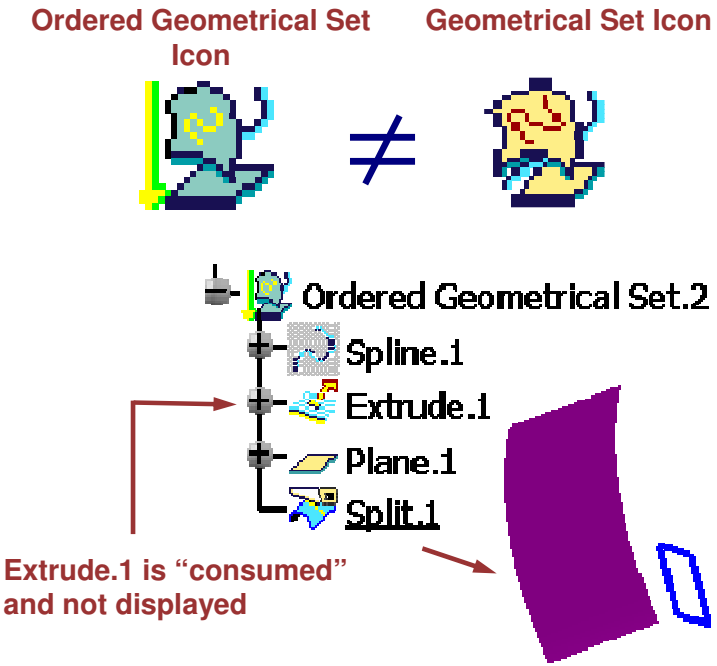
- **New Geometrical Set : Creates a new Geometrical Set branch in the specification tree with the option of putting nodes from existing Geometrical Sets into it. (Allows for multiple groups containing related elements)**
- **Duplicating a Geometrical Set : One of the modes of this tool duplicates the Geometrical Set in its entirety. This allows the user to edit nodes in the copied Geometrical Set without affecting the original Geometrical Set.**
- **Changing the Father node of a Geometrical Set : Allows the user to change the position of a Geometrical Set in the specification tree.**
- **Group : Hides all the nodes of a Geometrical Set except for specific nodes the user chooses to see.**

Student Notes:

# Why Do You Need Ordered Geometrical Sets ?

**Ordered Geometrical Sets are containers like Geometrical Sets but they have additional behaviours and allow feature creation using Linear Methodology.**

- **Ordered Geometrical Sets allow the user to visualize the model progressing after each feature operation. User can 'Define in work Object' to any of the feature in Ordered Geometrical Set to study the model up to that stage.**
- **In an Ordered Geometrical set, intermediary steps are not shown in 3D Display. This results in better visualization management.**
- **By Reordering elements in Ordered Geometrical Sets, you can achieve different results based on Linearity.**
- **In ordered Geometrical sets Graphical properties are inherited from parent elements.**

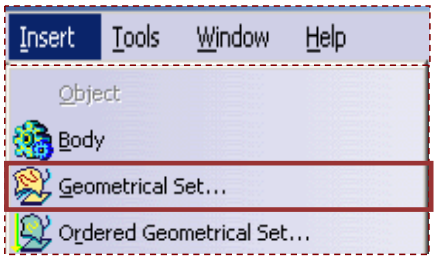



In example above Split.1 will inherit properties of Extrude.1.

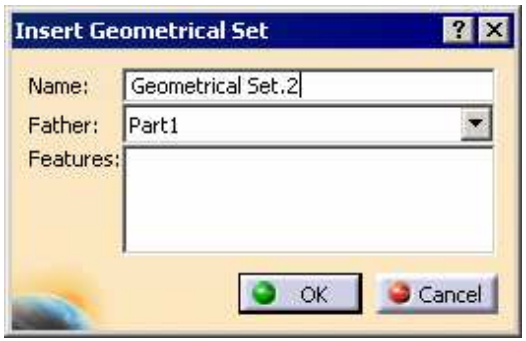
Student Notes:

# Creating a New Geometrical Set

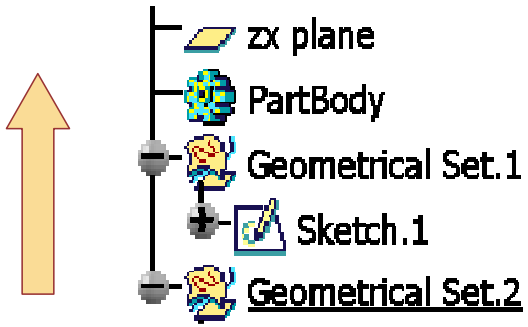
1 Insert/Geometrical Set from the Menubar.




2 Specify the node under which the new Geometrical Set will be inserted.




3 Click on OK to confirm. The new Geometrical Set is added to the specification tree.



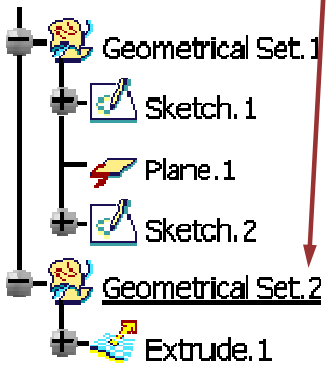

If Geometrical Set.1 was selected as the Father, the new Geometrical Set will be created under this node.

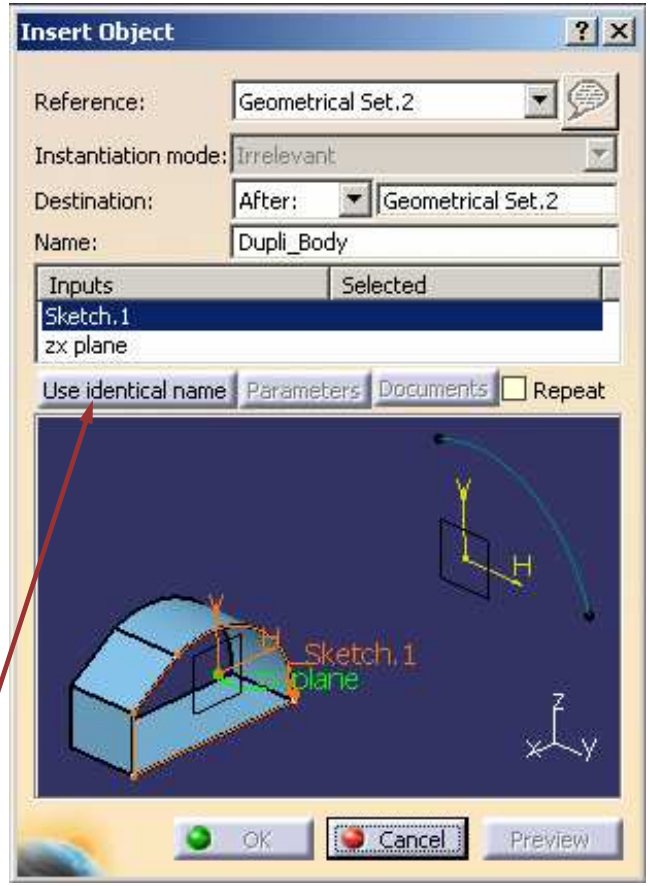
Student Notes:

# Duplicating a Geometrical Set (1/2)

1. Select the Duplicate Geometrical Set icon in the Replication toolbar

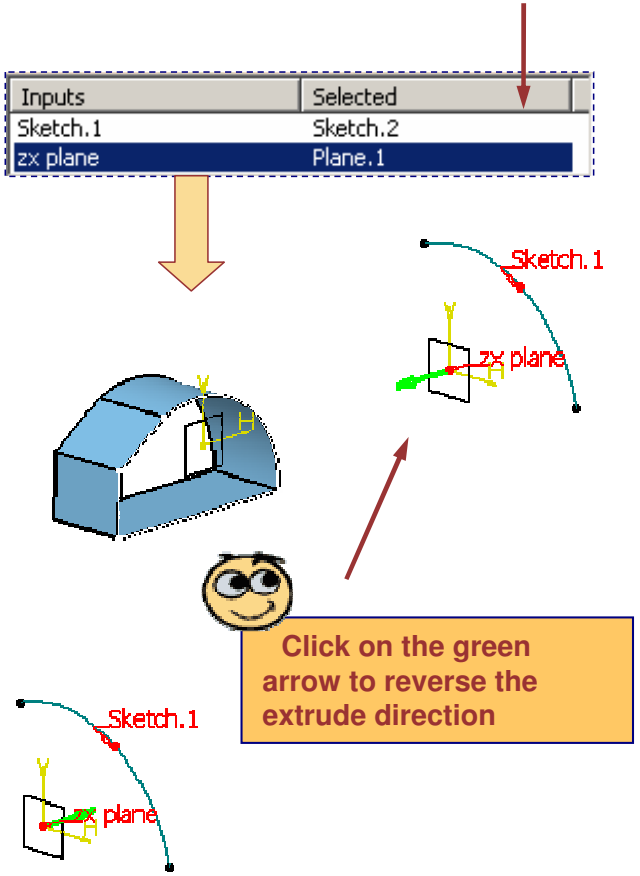

2. Select the Geometrical Set to be duplicated




Click on "Use identical name" to just create an identical second instance of the selected Geometrical Set.

3. Select the corresponding generating features as shown below

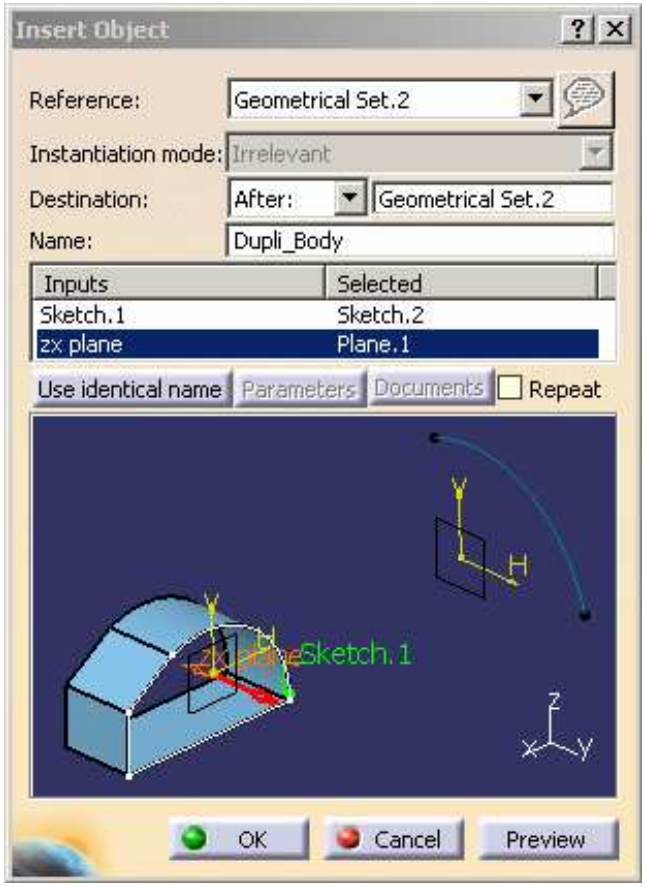


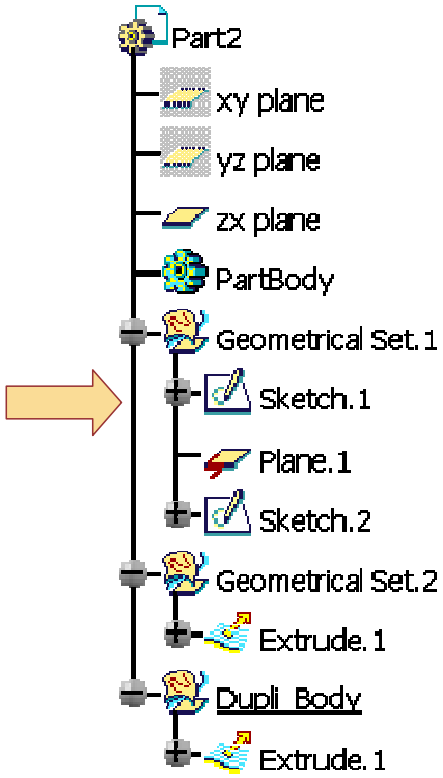

Click on the green arrow to reverse the extrude direction

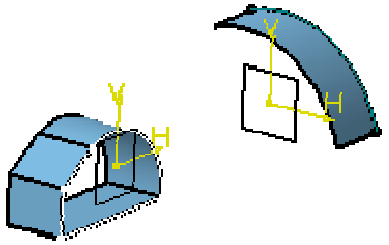
Student Notes:

# Duplicating a Geometrical Set (2/2)

4 **Click on OK to confirm the duplication**



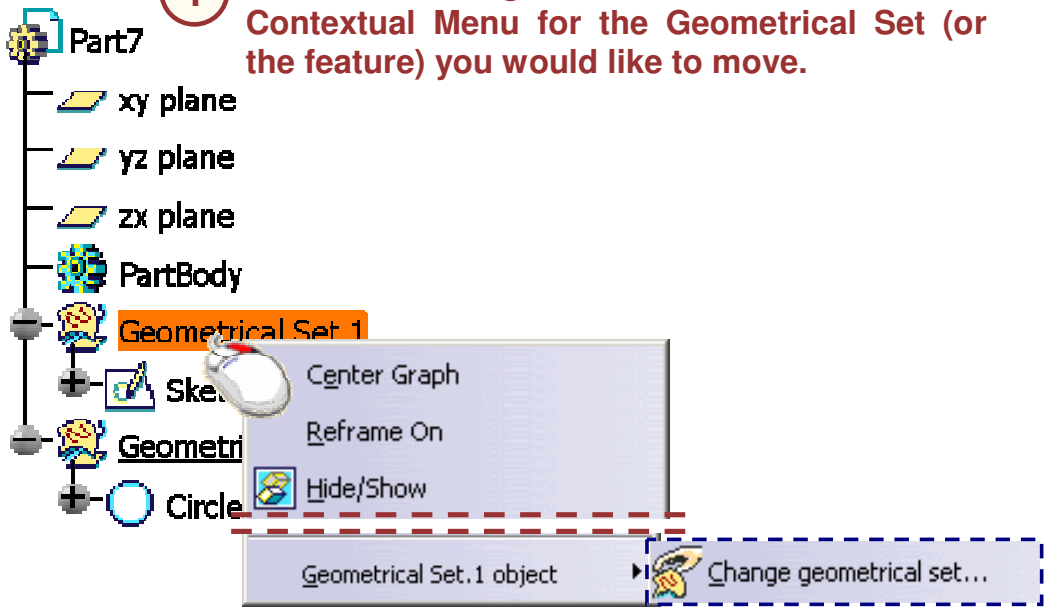



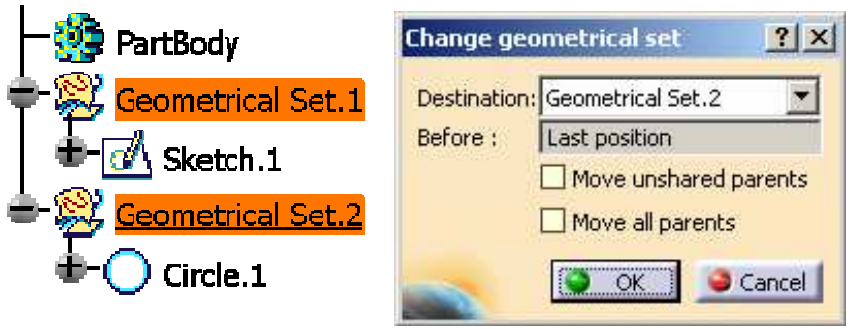
Student Notes:

# Changing the Father Node of a Geometrical Set

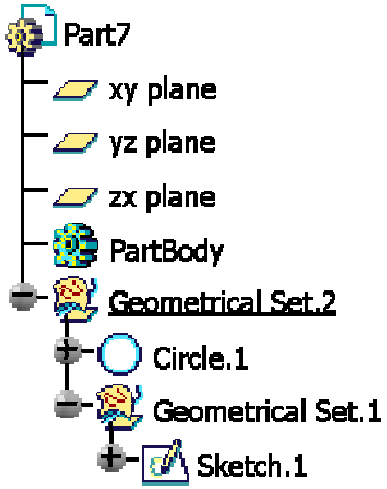
1 Activate "Change Geometrical set" in the Contextual Menu for the Geometrical Set (or the feature) you would like to move.




2 Select the destination node (new Father node) for your Geometrical Set (or your feature)




3 Click on OK to confirm.



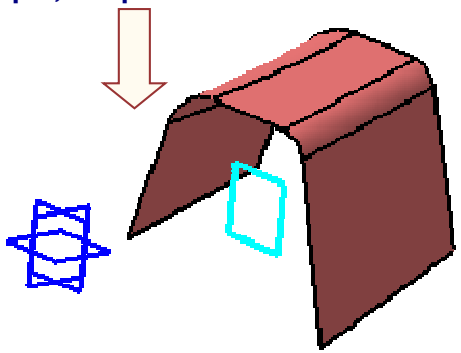

The Geometrical Set moved to its new location.

Student Notes:

# Scanning a Geometrical Set

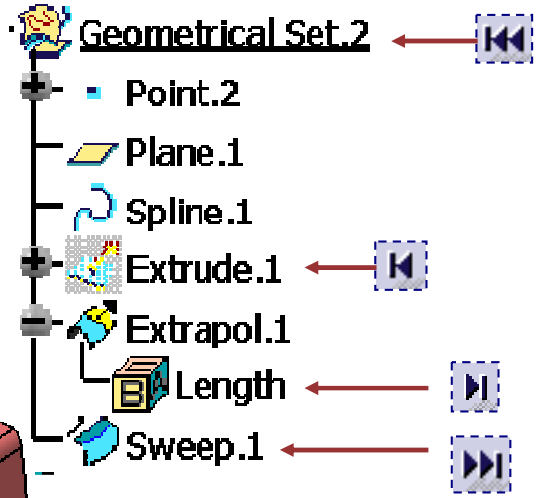
**In an Geometrical set, you can take advantage of the elements creation order to study the part's history:**

**In our example, the part looks like this :**



**The exit icon makes the selected feature the "in work object"**

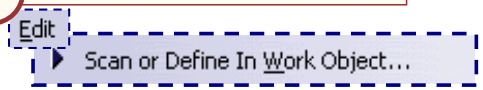
**3** As you click the scanning icons, the in work object changes and so does the displayed part.




**In work object**

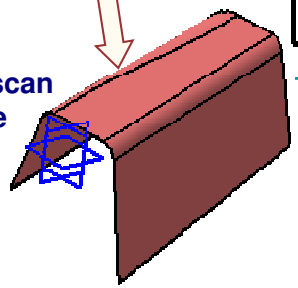
**Displayed part**

**1** Let's have a look at the design order using "Scan or define in work object":




**2** This "Scan" window appears:

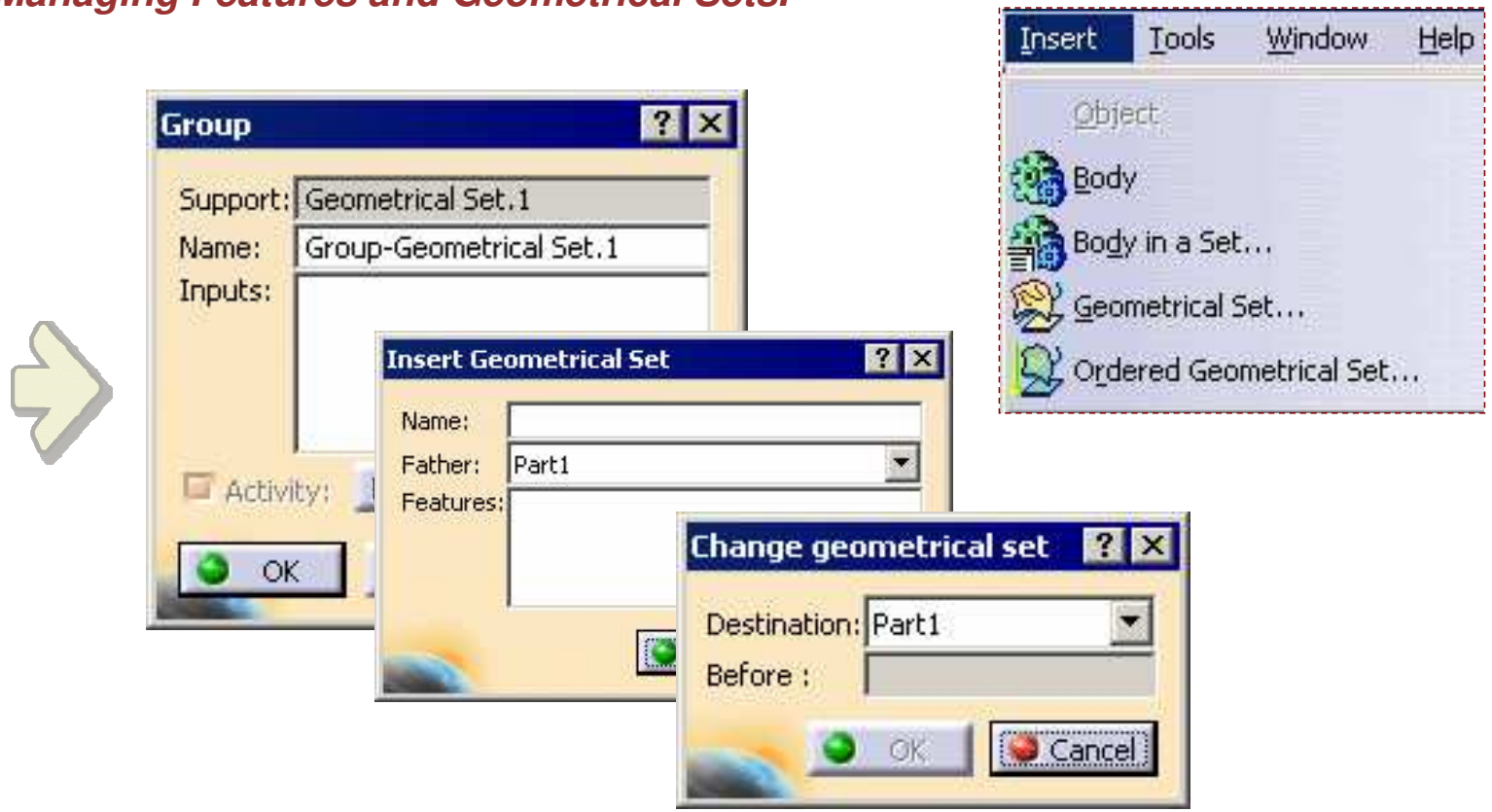

**You may use it to scan the geometry in the specification tree.**



Student Notes:

# Geometrical Sets – Recommendations

*You will learn about specific methods and recommendations concerning Managing Features and Geometrical Sets.*

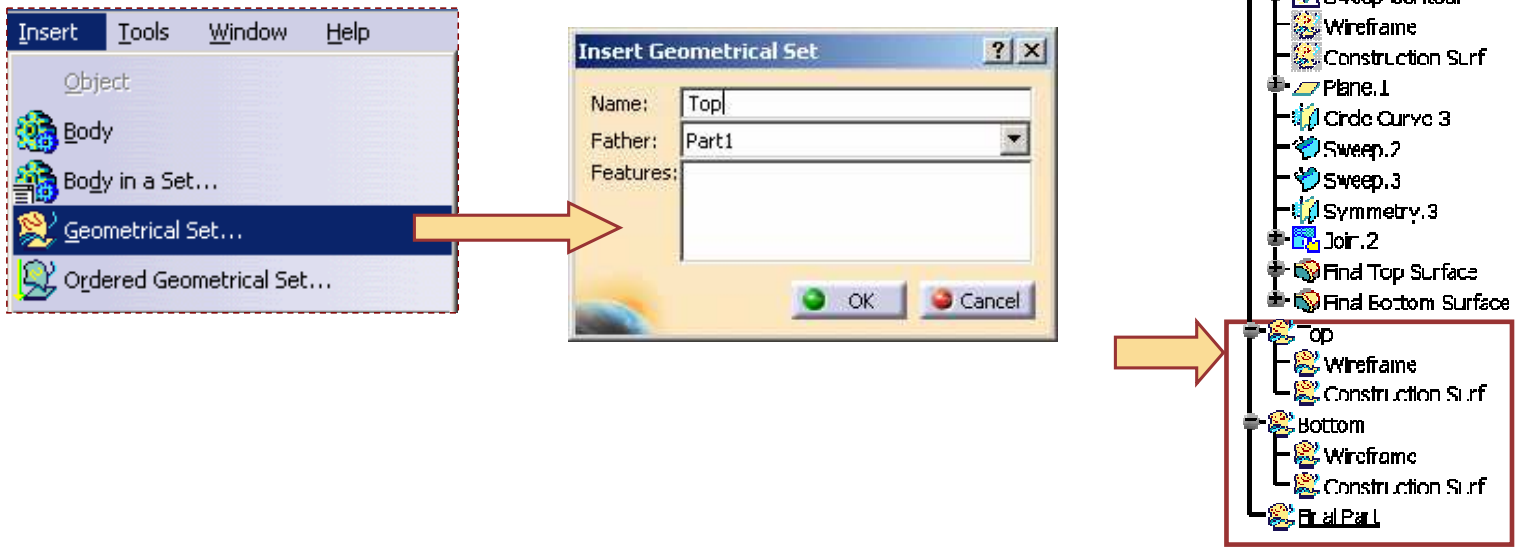


Student Notes:

# Using Geometrical Sets to organize Geometry (1/4)

**Geometrical Sets can be used to not only organize the geometry in a part, but can also be used to manage the visualization of those geometric features. Here is how that works:**

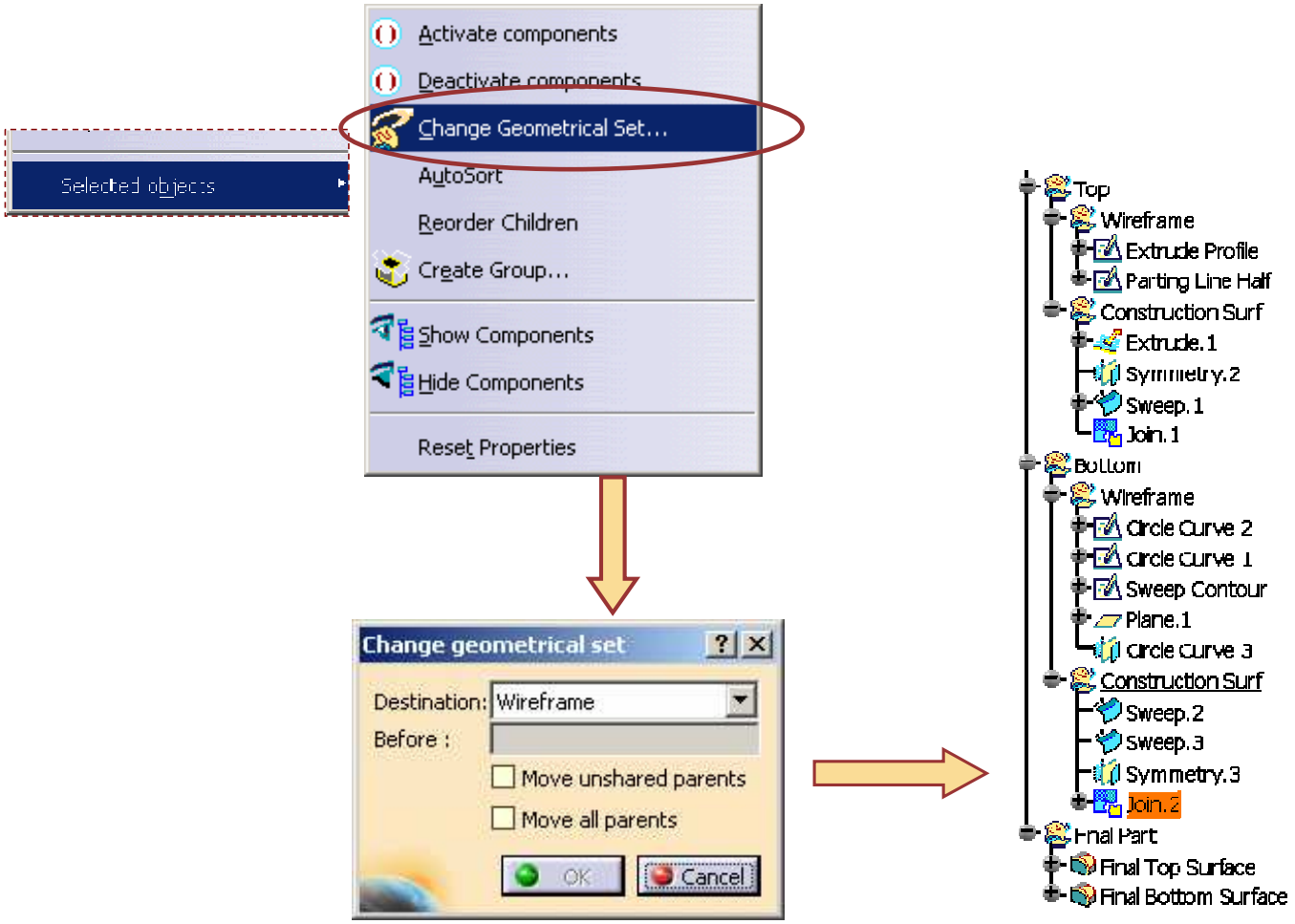
- **Create a Geometrical Set structure. Consider this step similar to creating a folder structure. Make sure you rename the Geometrical Set in a meaningful way.**



Student Notes:

# Using Geometrical Sets to organize Geometry (2/4)

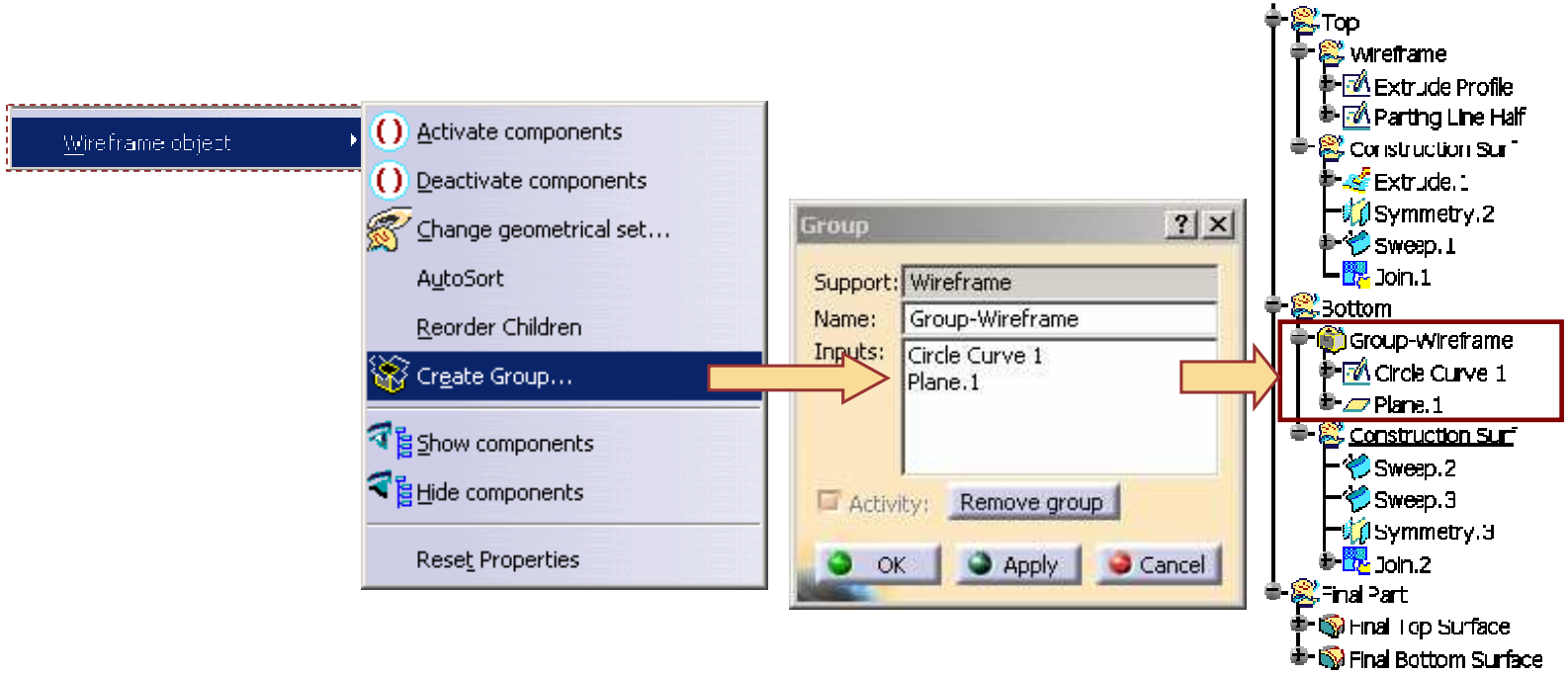
- Move features into the appropriate Geometrical Set.



Student Notes:

# Using Geometrical Sets to organize Geometry (3/4)

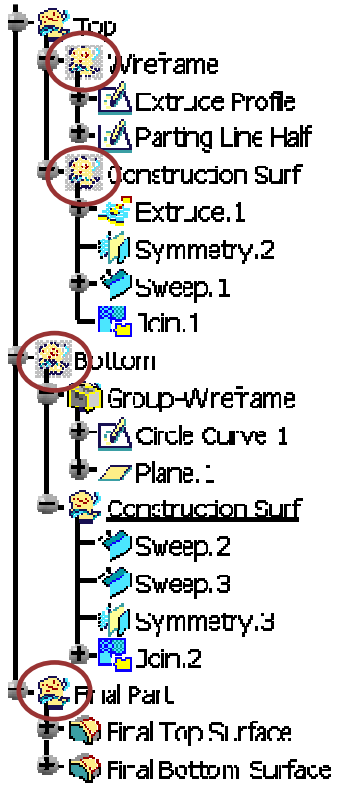
- Use groups to reduce the volume of features shown for any Geometrical Set.



Student Notes:

# Using Geometrical Sets to organize Geometry (4/4)

- HIDE/SHOW various combinations of Geometrical Sets to visualize different areas of the design.




Student Notes:

# Ordered or Standard Geometrical Set (1/2)

**GS and OGS are independent entities having different capabilities. Designer has an option to choose GS or OGS based on the design requirement.**

**Here are some facts about GS and OGS listed in the following table:**

| SL.No | Geometrical set | Ordered Geometrical set |
|---|---|---|
| 01 | Elements in this set can be shuffled irrespective of their sequence of creation | Elements in this set maintain the linearity with respect to their order of creation |
| 02 | Consists of only wireframes and surfaces | Consists of wireframes, surfaces and solid bodies |
| 03 | The parent element in this set is not absorbed after any operation. Hence an element can be used & re-used at different levels | The parent in this set is absorbed after performing an operation |
| 04 | Features in this set cannot be set as “in work object” where as the body can be set as “in work object” | Any feature in this set can be set as “in work object” and the features located after it are neither accessible nor visible |

Student Notes:

# Ordered or Standard Geometrical Set (2/2)

| SL.No | Geometrical set | Ordered Geometrical set |
|---|---|---|
| 05 | Maintains better flexibility | Maintains better linearity in design flow understanding |
| 06 | Geometrical sets cannot be switched to Ordered geometrical sets | Ordered Geometrical sets can be switched to Geometrical sets |
| 07 | Two or more Geometrical sets can be grouped to form a “Grouped Geometrical Set” | These cannot be grouped |
| 08 | As it consists only wireframe elements & surfaces it is not compatible for Boolean operations with solid bodies | This supports the volumes & can hold volume features this makes OGS compatible with boolean operations with Solid bodies |

This set can be recommended for use when working with non-hybrid environments and also when linearity is not a criteria

This set can be recommended for use when working with hybrid environments where a model is a combined result of GSD & Part Design. (Also GSO sometimes).

Student Notes:

# Hybrid or Non-Hybrid Bodies

| SL.No | Hybrid | Non-Hybrid |
|---|---|---|
| 01 | Hybrid Body is recommended, when a designer requires a combination of solid and surface based modeling under a single body. | Non-Hybrid body is recommended when designer opts for a method of modeling the solid features & surface features independently in separate bodies. |
| 02 | This is used to maintains the linearity in sequence of feature creation. | In this the sequence of creation is divided in different bodies based on surfacic OR solid features. |

Hybrid bodies are recommended when the modeling requires surfacic parents or volumes

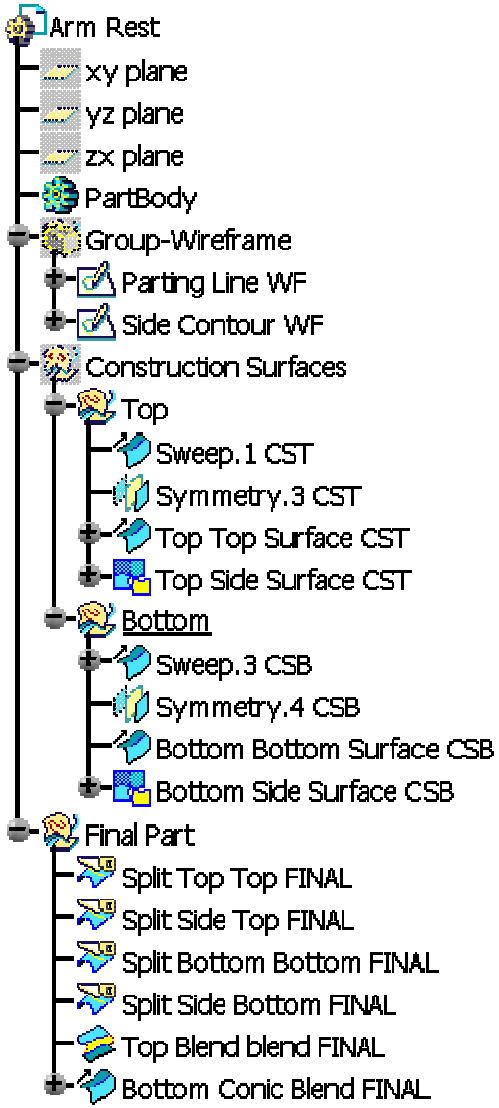
Non Hybrid bodies are recommended when the model can be built using only solids OR only surfaces.

# Managing Features

## Recap Exercise

15 min

- Create the necessary Geometric Sets
- Relocate features into the appropriate Geometric Set
- Hide/Show combinations of Geometric Sets



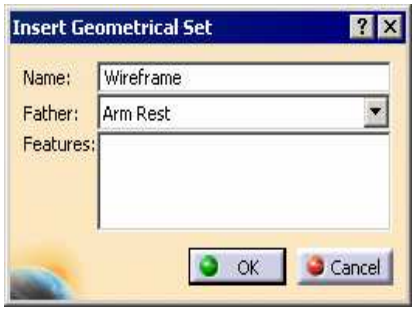

Student Notes:

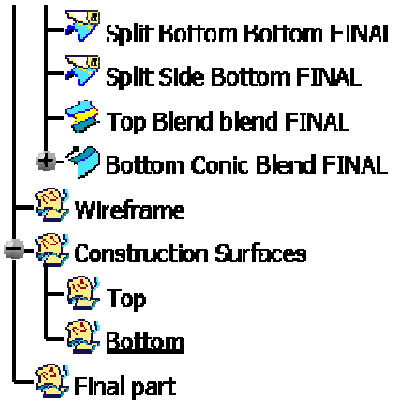
Student Notes:

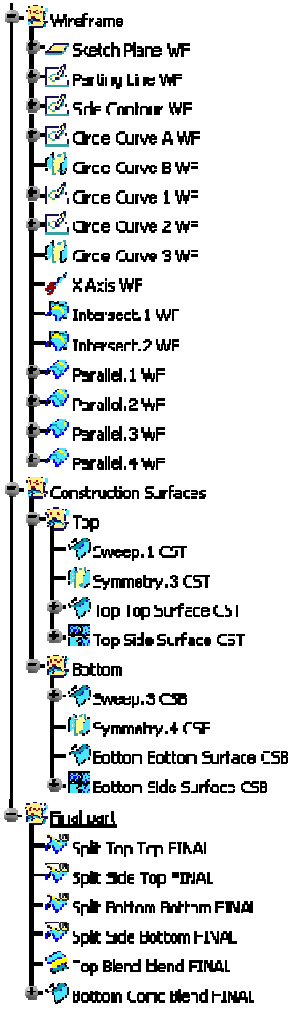
# Do It Yourself (1/2)

**Part used:**
**CATGSD_F_Managing_Features_and_Geometrical_Sets_Recap_begin.CATPart**

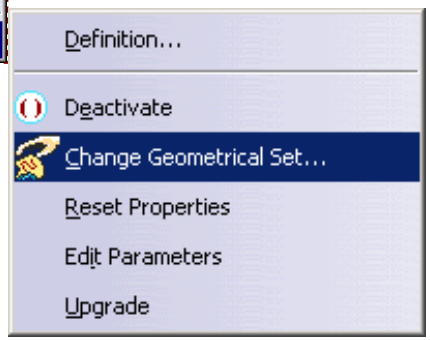
**1. Create new Geometrical Sets to form a logical feature organizational structure.**

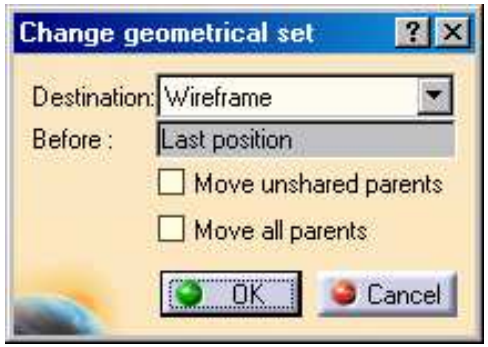



**2. Relocate features into the appropriate Geometric set. For clarity, you will be relocating features according to the name the feature has been provided.**

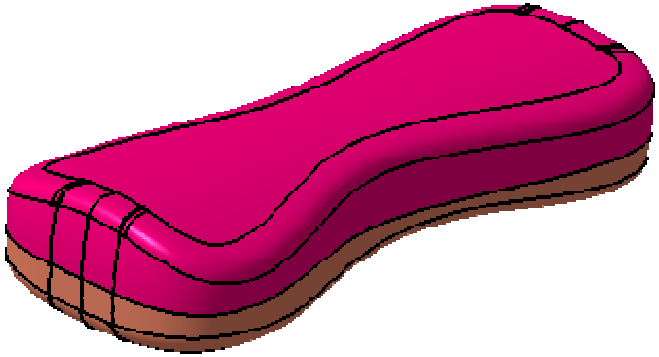



Student Notes:

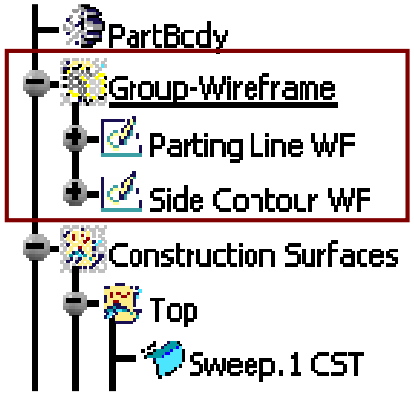
# Do It Yourself (2/2)

3. Visualize only the FINAL PART.



4. Create a GROUP for the Wireframe Geometric Set.






End Part:CATGSD_F_Managing_Features_and_Geometrical_Sets_Recap_end.CATPart

STUDENT GUIDE

Student Notes:

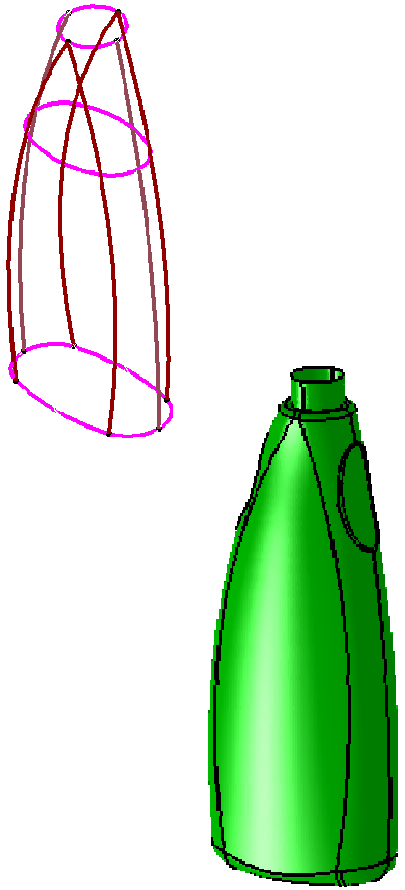
# Shampoo Bottle

## Recap Exercise: Surface Design Overview

40 min

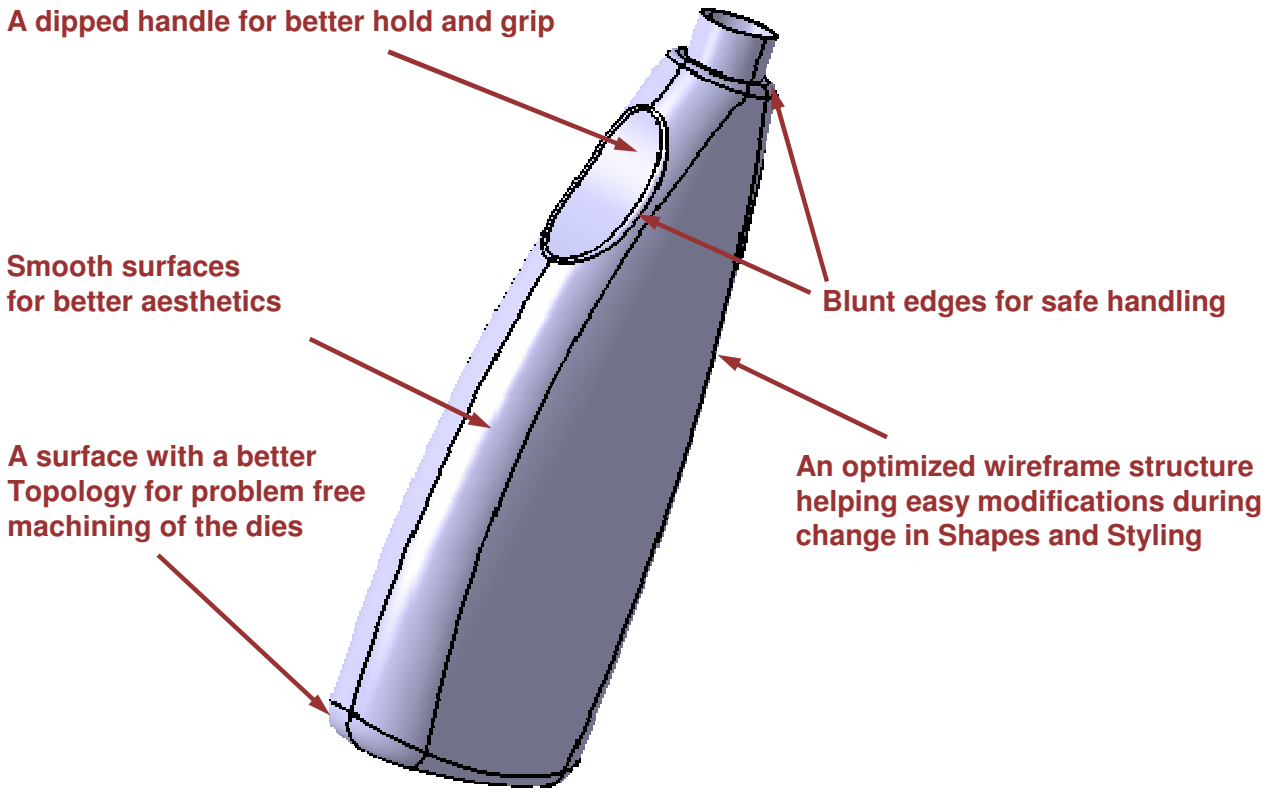
**In this exercise you will:**

- Create, analyse and modify the wireframes and surfaces using advance tools of Generative Shape Design
- Learn that, high quality surface can be achieved using advance tools of Generative Shape Design



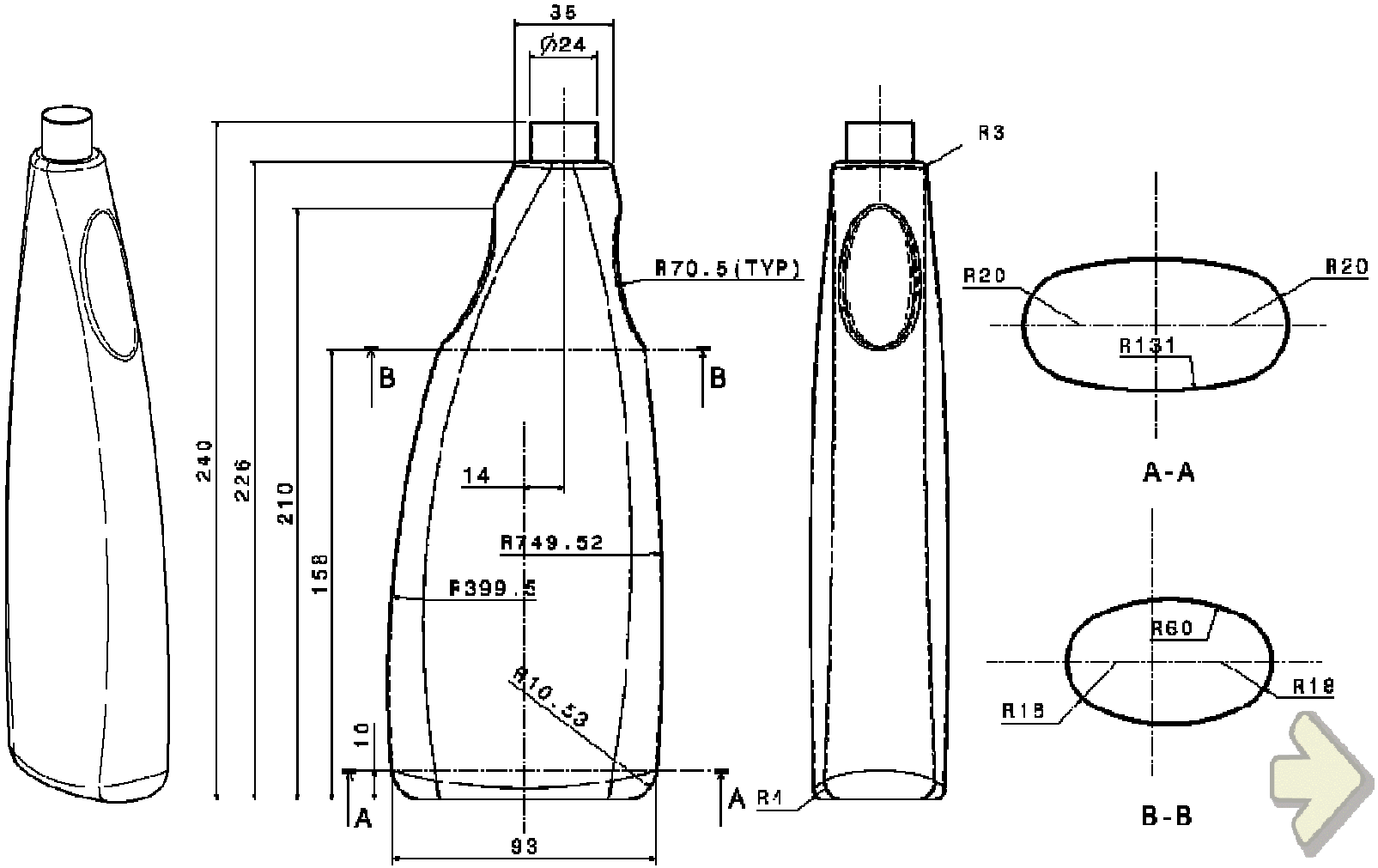
Student Notes:

# Design Intent: Shampoo Bottle (1/2)



# Design Intent: Shampoo Bottle (2/2)

**Build the Shampoo Bottle geometry using the shown specifications**

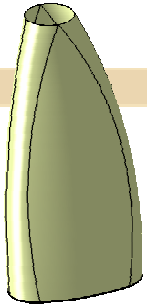



Student Notes:

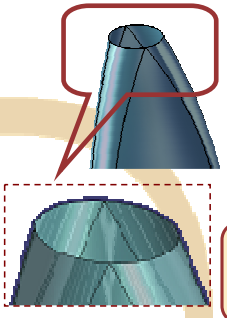
Student Notes:

# Design Process: Shampoo Bottle (1/2)

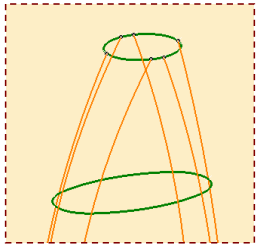
1. Create Basic Multi-Section Surface from given Wireframe



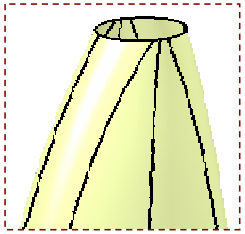
2. Analyze the Surface



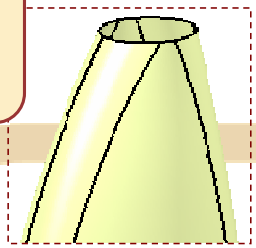
3. Modify the wireframe



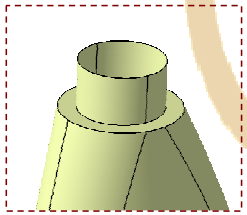
4. Create the Multi-Section surface using the modified wireframe

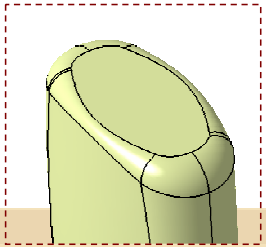


5. Smoothen the Multi-Section surface specifying the Deviation



6. Create the Neck of the bottle



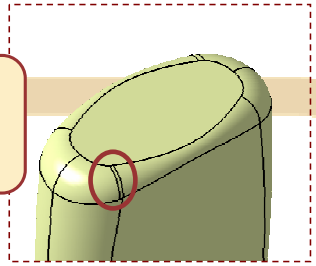


7. Create a Bitangent fillet at the bottom without specifying Hold curve and complete the model

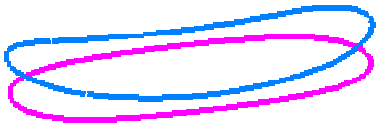
Student Notes:

# Design Process: Shampoo Bottle (2/2)

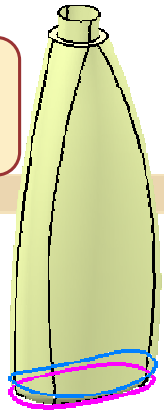
8 Analyze the surface for broken fillet at the bottom



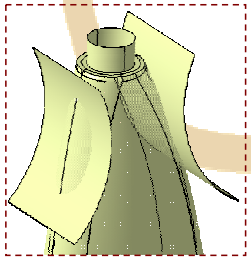
9 Create a 'Hold curve' using 'intersection' and spine using 'curve Smooth'

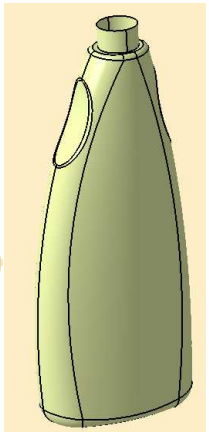


10 Modify the Bitangent Fillet using Hold curve and spine



11 Create the Bottle Handle



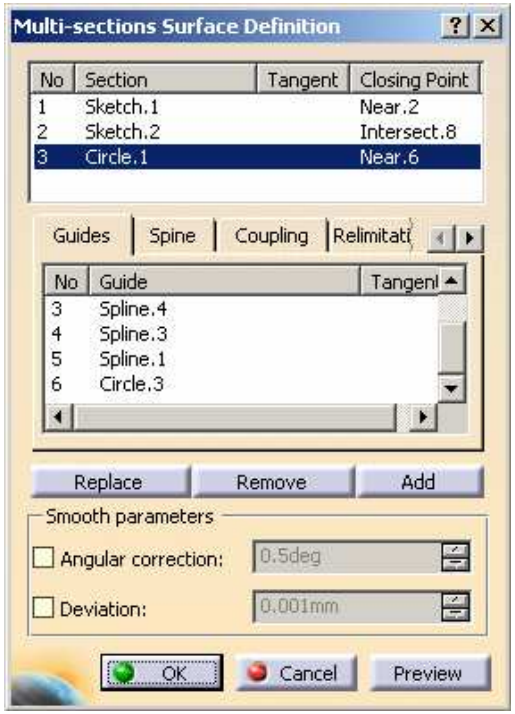


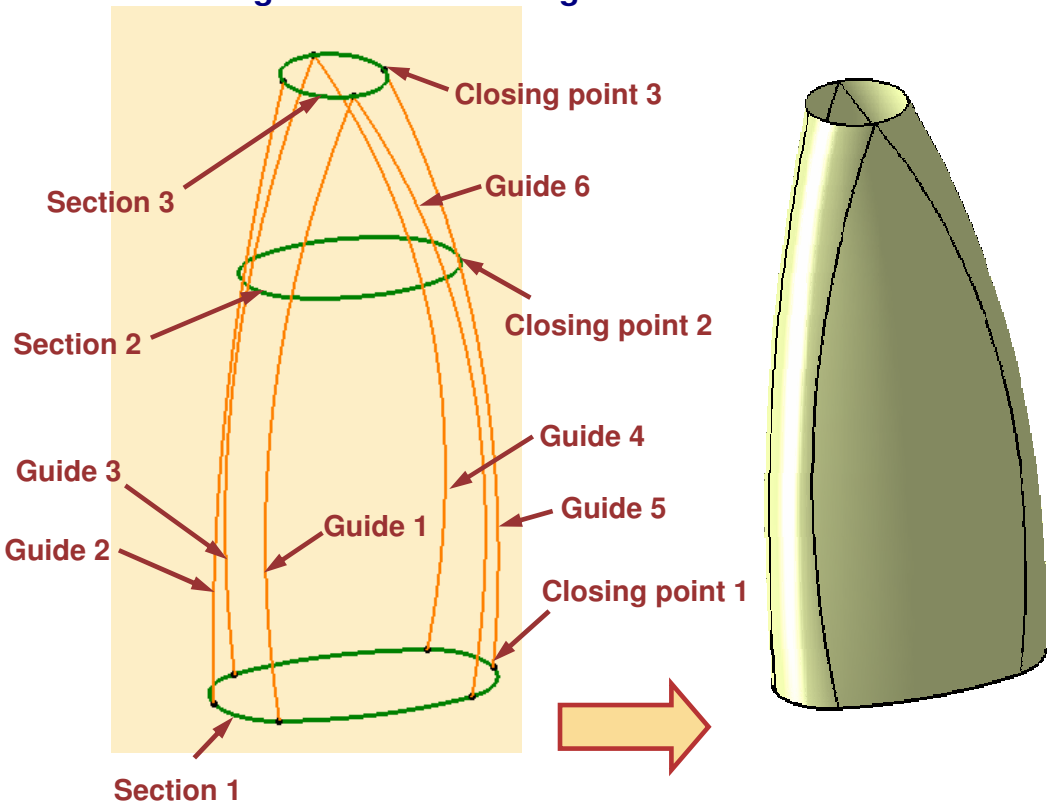
Student Notes:

# Do it Yourself (1/11)

Part Used: CATGSD_F_Shampoo Bottle_Start.CatPart

- Create Basic surfaces using the given wireframe. This surface will then be analyzed for quality
  - Create a Multi-Section surface from the given sections and guide curves



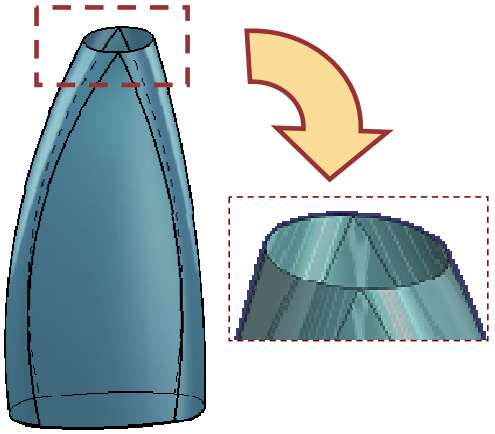
Student Notes:

# Do it Yourself (2/11)

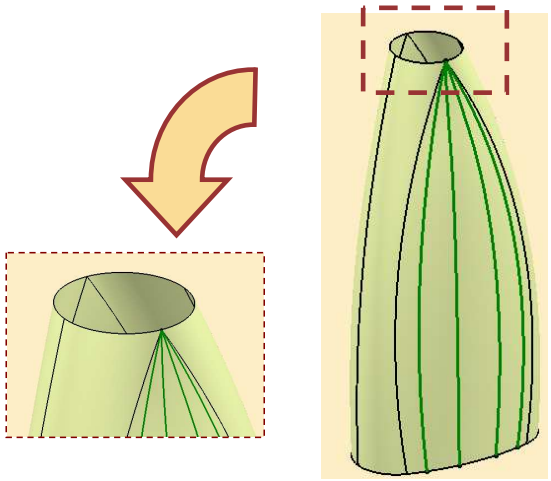
- Analyze the surface quality visually
  - Apply the material specifications to the surfaces-
    - DS Light Blue

    You will find the distortion on the surface formed by three edges, at the converging point

  - Change the View to “Shading with Edges” mode.



- You will Create the Isoparametric curves to see the segmentation of the surface
  - Invoke the Isoparametric curves function
    - Insert > Wireframe > Isoparametric curves
  - Select a point on the surface(support) where you would require the curve. Swap U V if required contextually, and confirm OK to create the curve
  - Similarly extract the few more Isoparametric curves.



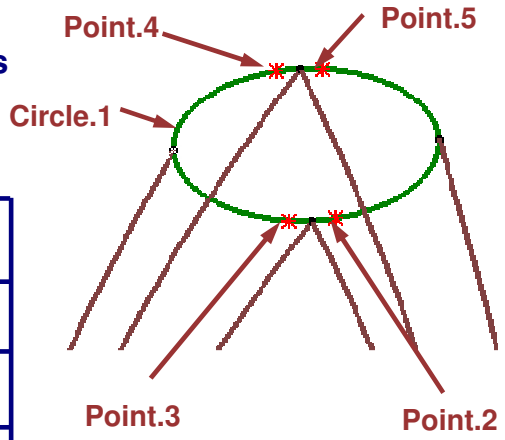
You will observe that the Isoparametric curves intersect each other at the same converging point. This signifies that the surface is NOT of good quality.

Student Notes:

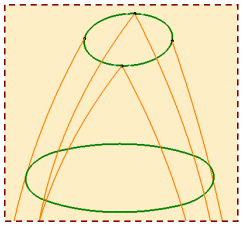
# Do it Yourself (3/11)

To improve the surface quality you will improve the quality of wireframe.

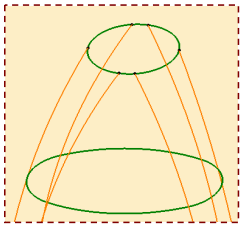
- Modify the wireframe to achieve a good quality surface
  - Deactivate all the Isoparametric curves before modifications
  - Create Four points using “Point on Curve”




| Name | Curve | Ref Point | Length |
|---|---|---|---|
| Point 2 | Circle 1 | Near 3 | 3 mm |
| Point 3 | Circle 1 | Near 3 | 3 mm |
| Point 4 | Circle 1 | Near 4 | 3 mm |
| Point 5 | Circle 1 | Near 4 | 3 mm |



Wireframe with guide curves converging at same point.



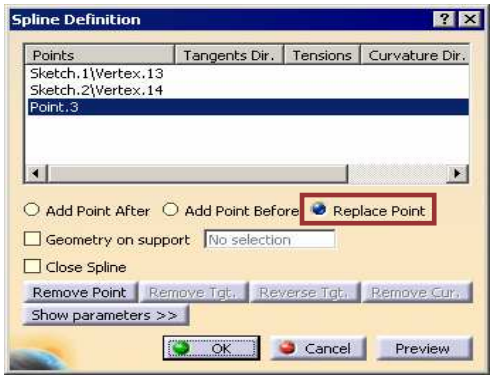
Wireframe modified to achieve a good quality surface.

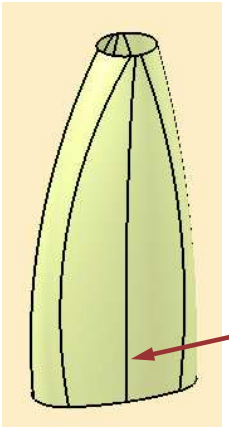
Once you finish the modification of wireframe. You can activate the Isoparametric curves to visualize the changes.Deactivate or delete these curves after your study.

Student Notes:

# Do it Yourself (4/11)

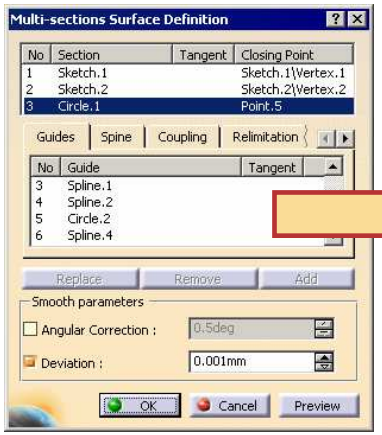
- Modify Guide curves using the new points to overcome the problem of point converged surface
  - Replace the end points(Near.3 and Near.4) of Splines 1,2,3 and 4 by new points created

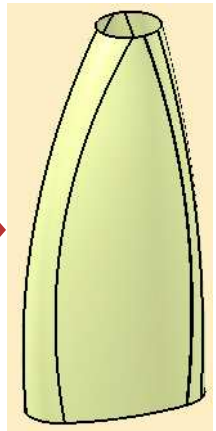



Multi-section surface gets updated according to the modified wireframe. A smooth edge is created on the surface forming two faces which would affect the aesthetics of the bottle.

- Specify deviation value in smooth parameter box to achieve smooth surface
  - Edit Multi-section surface and select the Deviation checkbox in smooth parameters
  - Specify the value as 0.001mm (default value) and click OK



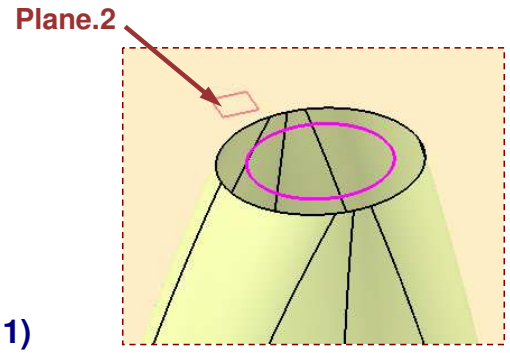



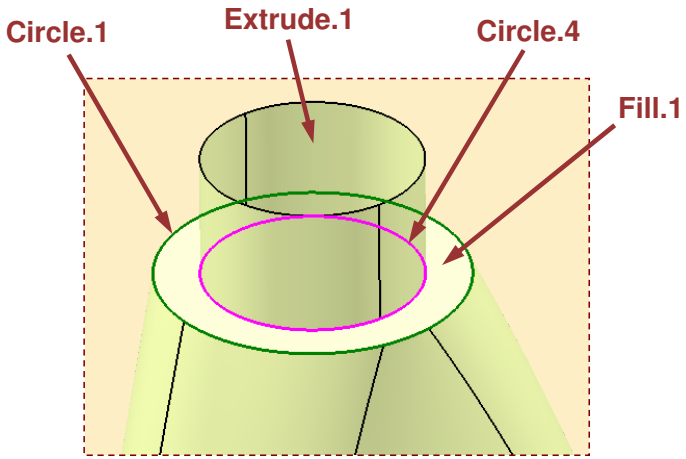
With the smoothing parameters the two faces are converted into a single face. This is achieved by specifying the Deviation parameter.

Student Notes:

# Do it Yourself (5/11)

- Create the top portion of the bottle
  - Create Circle.4 of type “Center and Radius”
    - Center: Point.1, Support: Plane. 2
    - Radius: 12mm
  - Create fill.1 by filling Circle.1
  - Extrude Circle.4 to create neck portion of the bottle (Extrude.1)
    - Direction : XY Plane
    - Limit 1 : Up to plane.1

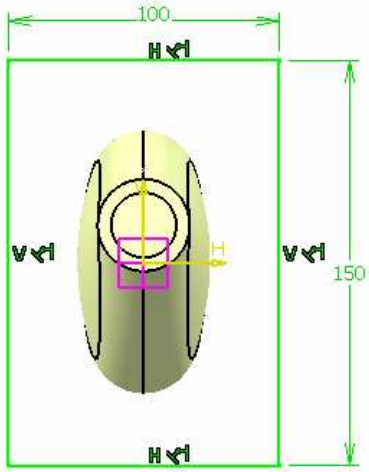


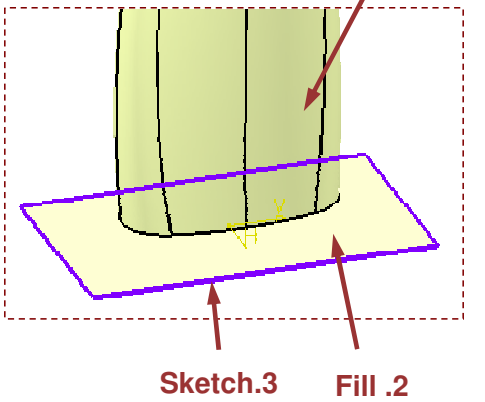



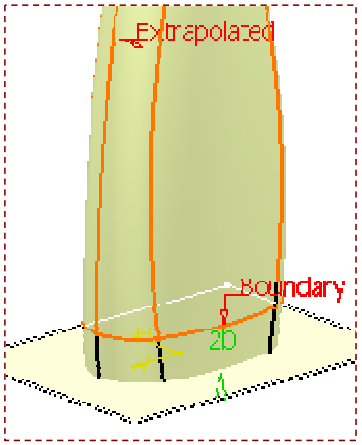
Student Notes:

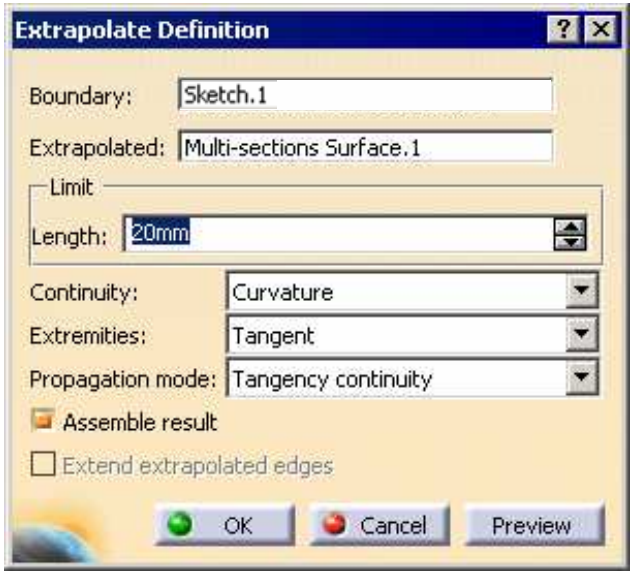
# Do it Yourself (6/11)

- Create a fillet at the bottom of the bottle
  - Sketch a Rectangular Profile into Sketch.3 on XY Plane.
    - Profile :100X150mm
  - Create fill.2 by filling Sketch.3
  - Extrapolate Multi-Section Surface.1 up to 20mm using Curvature Continuity and Tangent continuity as Propagation .



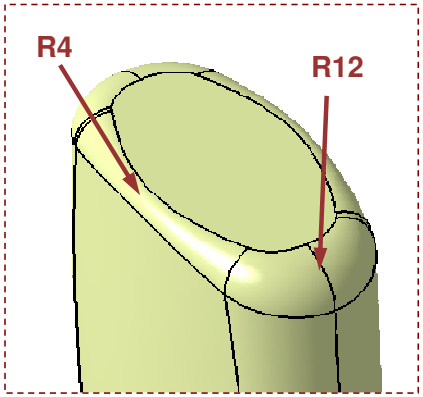



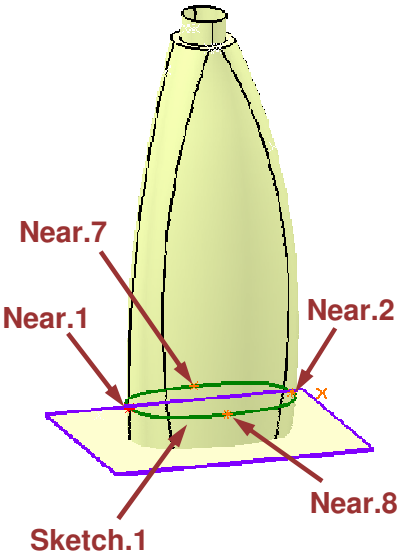



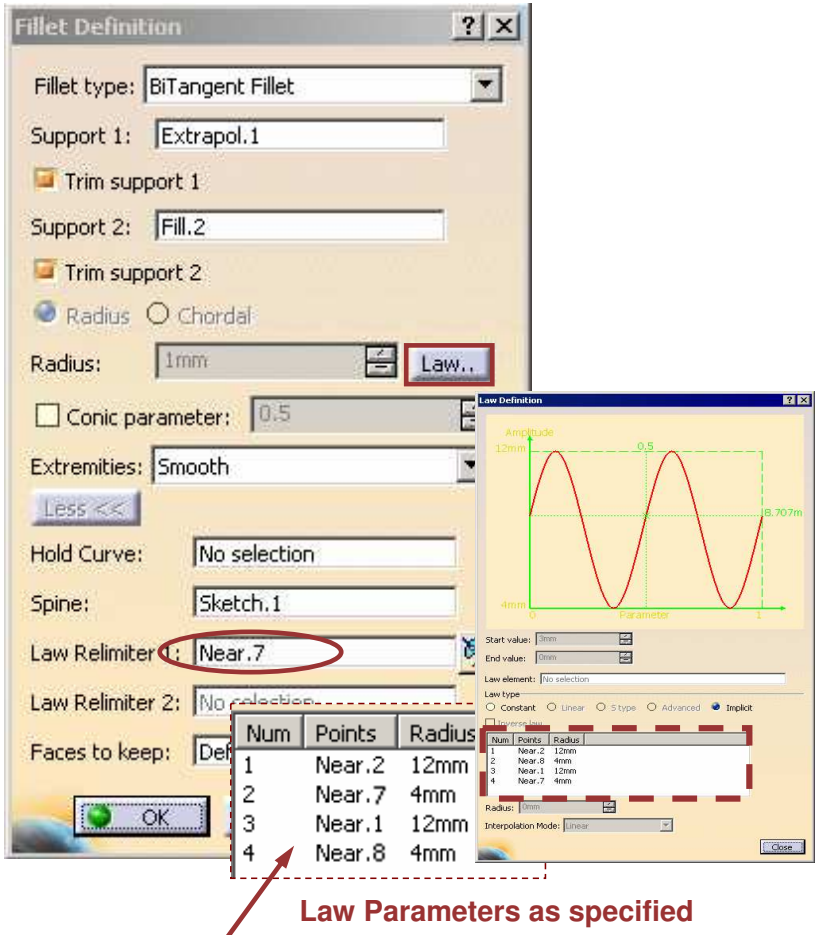
Student Notes:

# Do it Yourself (7/11)

- Create Intersection.9 between Sketch.1 and ZX Plane.creating the points Near.7 and Near.8 from Intersection.9. (Use Plane.7 and Plane.8 as reference)
- Create Bitangent Fillet between Fill.2 and Extrapol.1, specify sketch.1 as a Spine. Use Implicit Law to define the radius.




Law Parameters as specified

Clear the default selection in the box if any. Select all 4 points manually as specified.

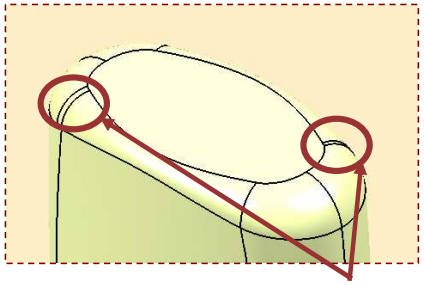
Student Notes:

# Do it Yourself (8/11)

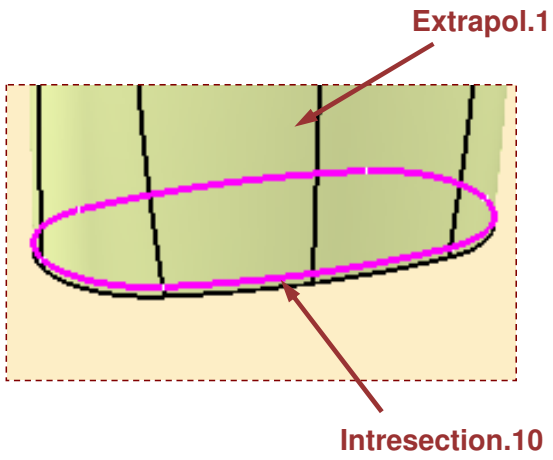
**Observe that fillet.1 has a broken faces at few of areas.To overcome this, you will have to use the advance options of Bitangent fillet to achieve a good quality surface.**

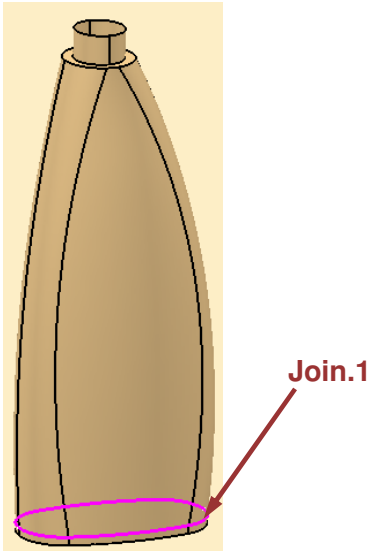
**Create a smoother Spine**

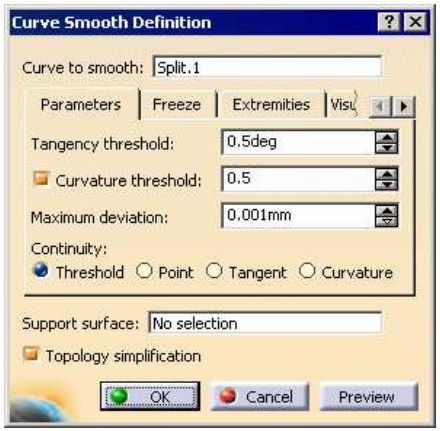
- **Create Intersection.10 between Extrapol.1and XY Plane**
- **Split the Intersection.10 by YZ Plane**
- **Smooth the resultant curve(Split.1) using 'Curve Smooth'**
- **Symmetry Curve Smooth.1 about YZ plane**
- **Join Curve Smooth.1 and Symmetry.1 to form Join.1**



Broken faces



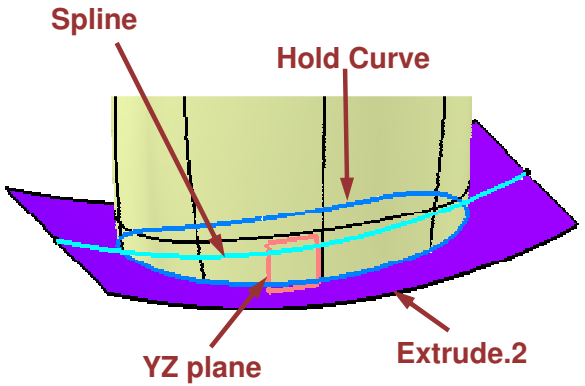



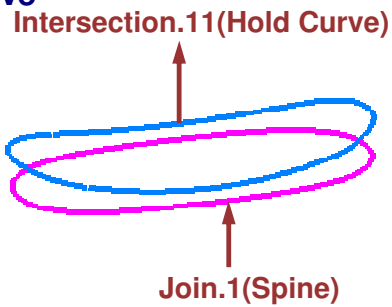
Student Notes:

# Do it Yourself (9/11)

- Create a 'Hold Curve'
  - Create a Spline on YZ plane passing through co-ordinates
    - H= 65, V =16
    - H= 0, V = 4
    - H= -65 , V =16
    - Keeping 'origin' as reference point.
  - Using Spline as input curve, create an Extrude(Extrude.2) of length 40mm each side
  - Create an Intersection.11 between Extrude.2 and Extrapol.1




- Modify the Bitangent Fillet.1
  - Replace the spine by Join.1 and select Intersection.11 as a Hold Curve
  - Observe the modified fillet surface getting improved after using advance options

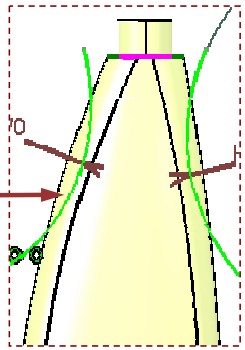


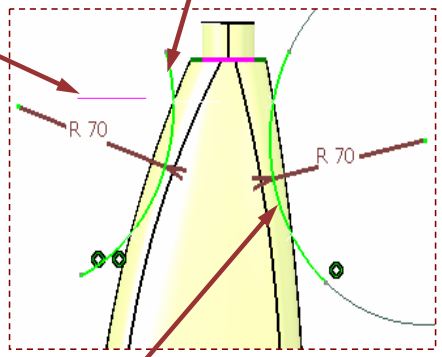
Student Notes:

# Do it Yourself (10/11)

- Create the Handle portion of the bottle
  - Create a plane.11 at distance of 210mm from XY Plane using offset Plane function
  - Create an intersection.12 and Intersection.13
    - First Element : Plane.11
    - Second Element : Circle.2 + Circle.3
  - Create two Arcs of Radius 70 mm in Sketch.4 on YZ plane. Create a sketch output for one of the arcs


Create Sketch Output of this Arc





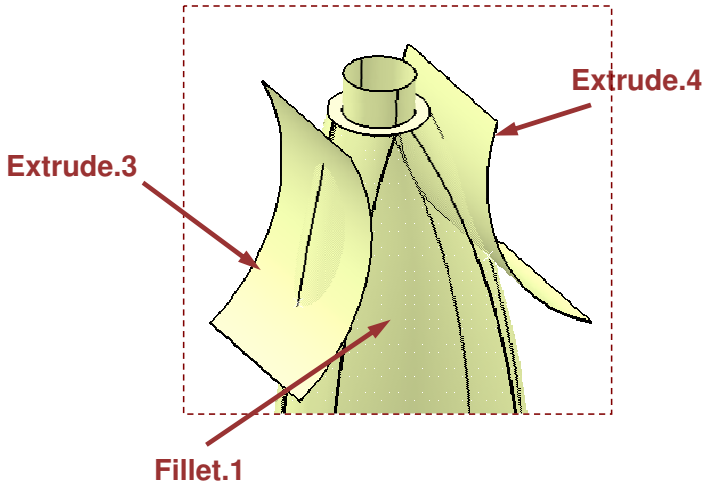

  - Create Extrude.3 and Extrude.4 surface using sketch.4 along YZ direction with the length 30mm both sides.

Student Notes:

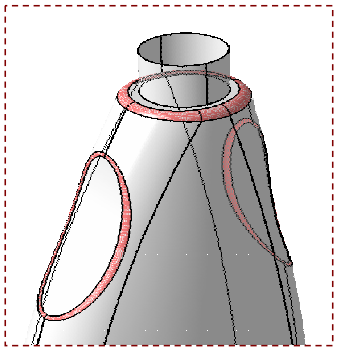
# Do it Yourself (11/11)

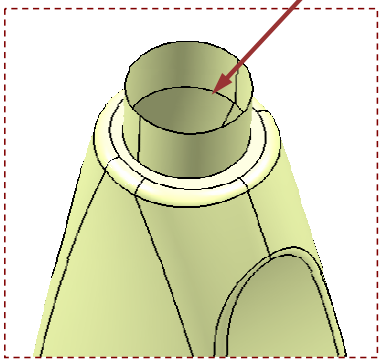
- Create the Fillets(Blunt Edges) at Handle and neck area of the bottle
  - Create a BiTangent fillet of radius 3 mm between:

| First Element | Second Element |
|---|---|
| Extrude.3 | Fillet.1 |
| Extrude.4 | Fillet.2 |
| Fillet.3 | Fill.1 |




  - Create a Trim.1 between Fillet.4 and Extrude.1 to achieve hole neck opening






End Part: CATGSD_F_Shampoo_Bottle_End.CATPart

This exercise will demonstrate that surfaces of high quality can be achieved using advance tools of Generative Shape Design

# Creating Advanced Wireframe Features

*This lesson will cover the following advanced wireframe geometry topics:*

- About this Lesson
- MD2/GSD Wireframe versus HD2/GSD Wireframe
- Creating an Extremum
- Creating a Connect Curve
- Recap Exercise: Advanced Wireframe Geometry

Student Notes:

Student Notes:

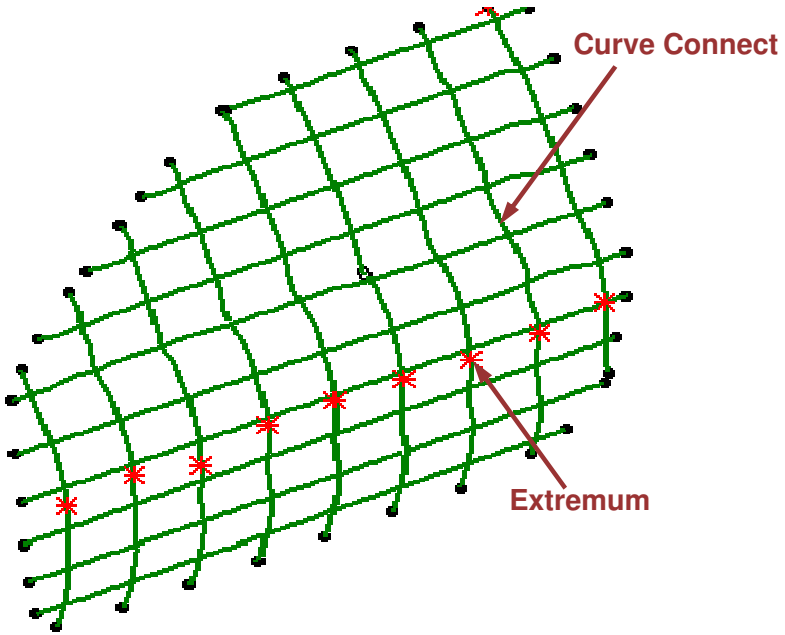
# About this Lesson

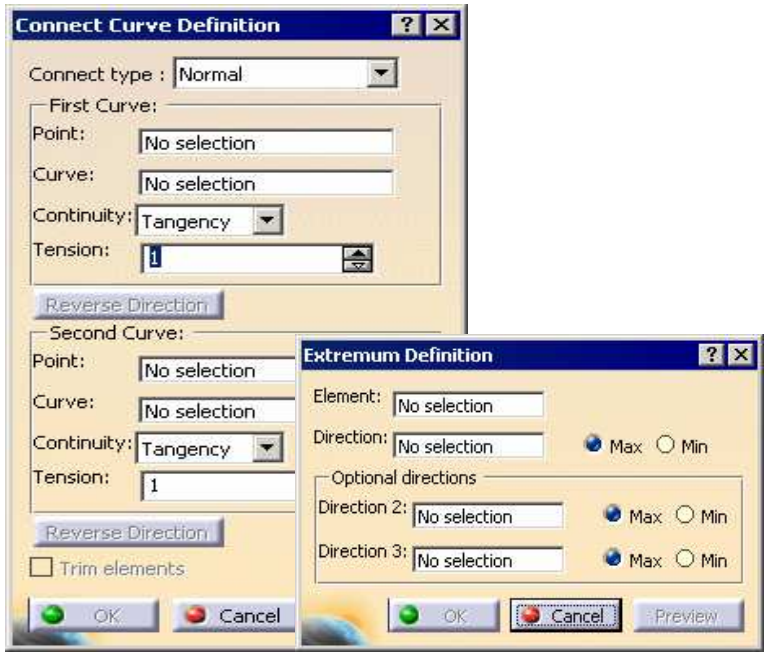
**In this lesson you will learn and practice the advanced wireframe tools of Generative Shape design workbench .You will also perform an exercise at the end of the lesson in which you will practice these tools.**

**You will learn the following tools in this lesson:**

- **Extremum points**
- **Curve Connects**

**Along with this you will also learn a few Recommendations and Tips to achieve better quality wireframes.**

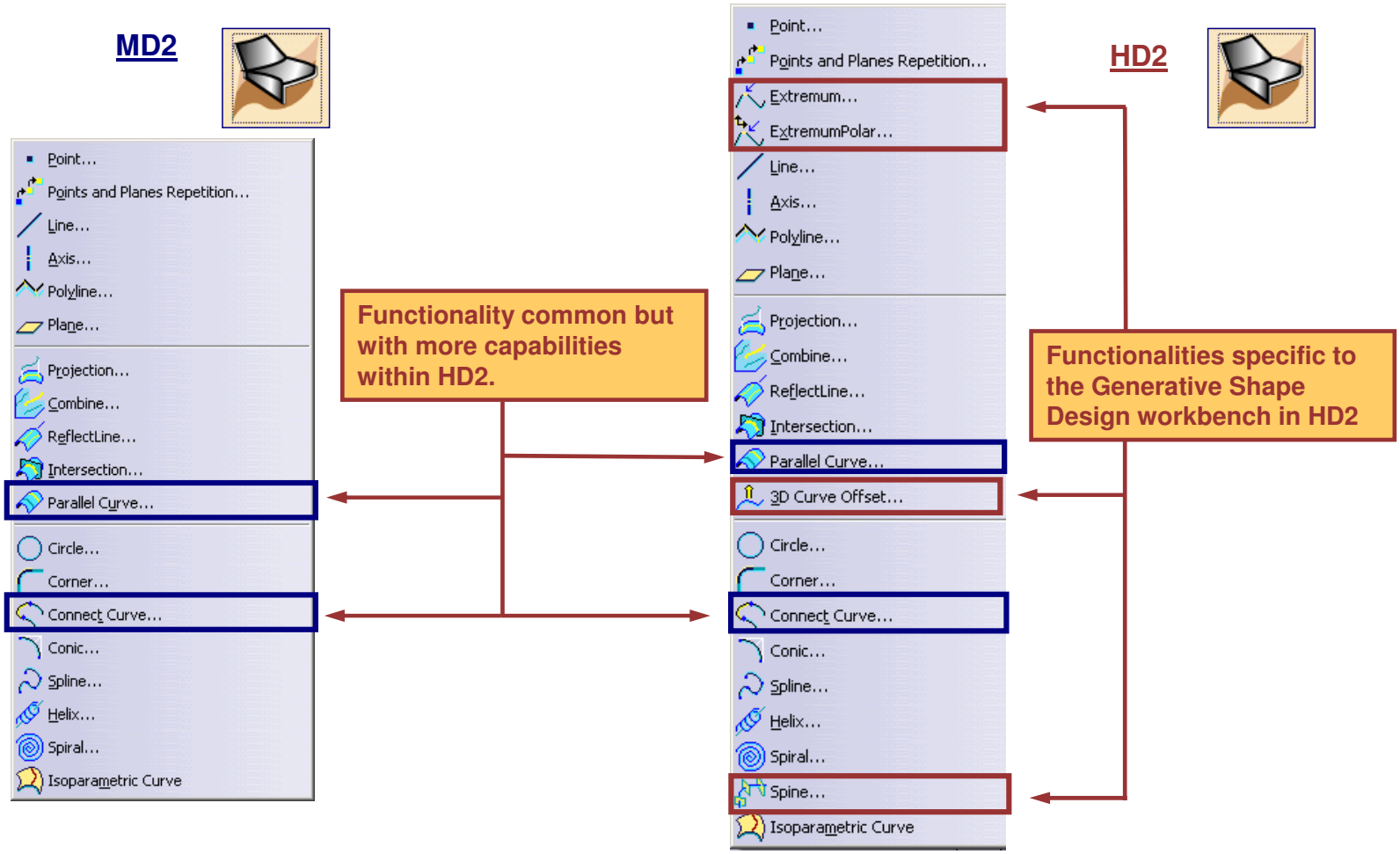


Student Notes:

# MD2/GSD Wireframe versus HD2/GSD Wireframe

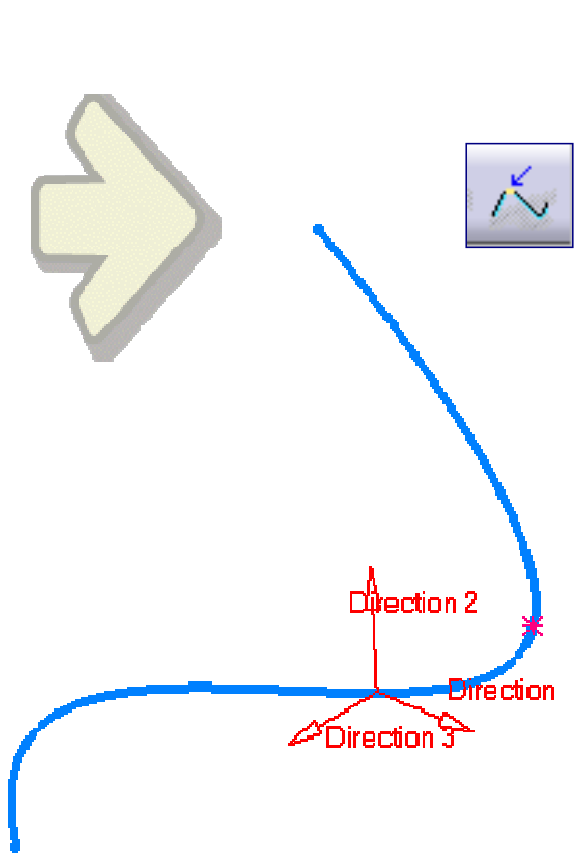
**The Generative Shape Design workbench while in MD2 and HD2 configurations have many common functionalities. Within HD2 you will discover new functionalities that are not in MD2 and also advanced capabilities in some functions that exist in both workbenches.**

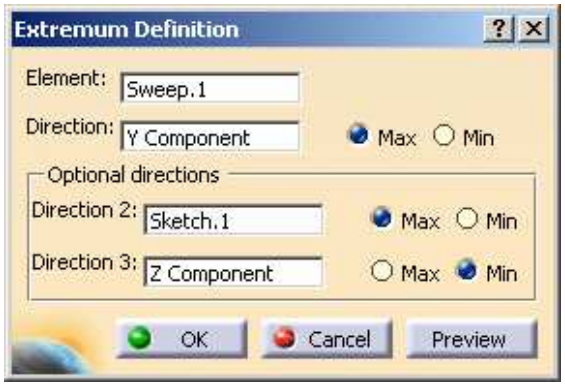


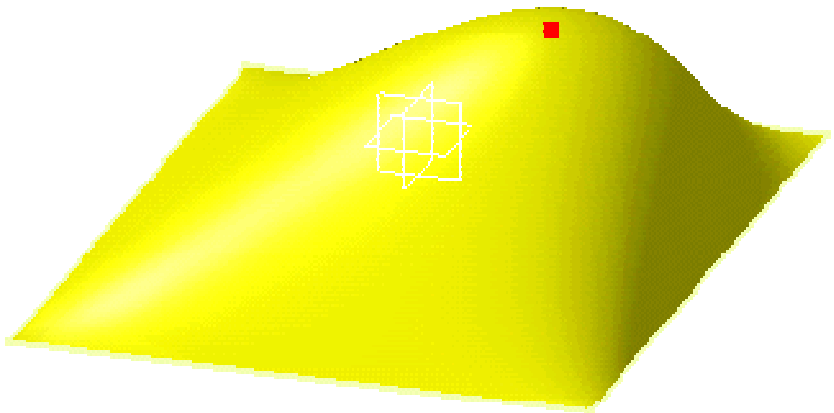
Student Notes:

# Creating an Extremum

*You will learn what an Extremum is and how to create it.*



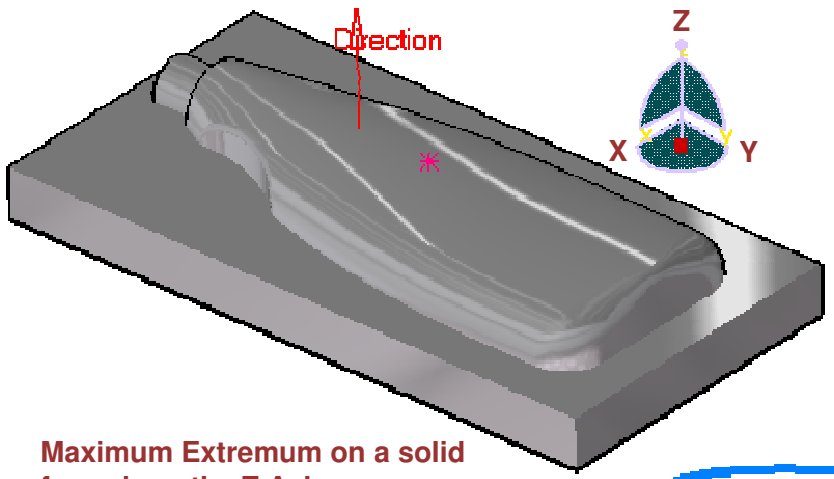



Student Notes:

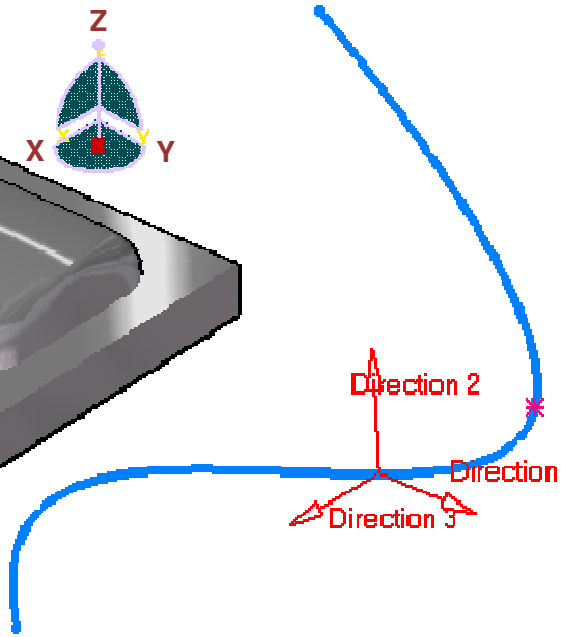
# Why Create an Extremum?

**In order to help CATIA find the maximum or minimum point of a curve or surface along any direction chosen by the user.**

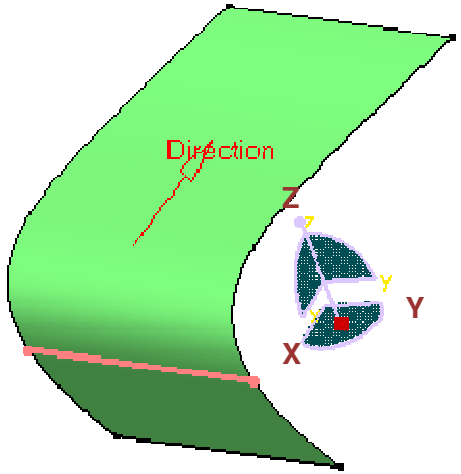
**The element might be a sketch, a 3D curve or line, a surface or a solid face.**




Maximum Extremum on a solid face along the Z Axis




Maximum Extremum on a Curve along the Z Axis

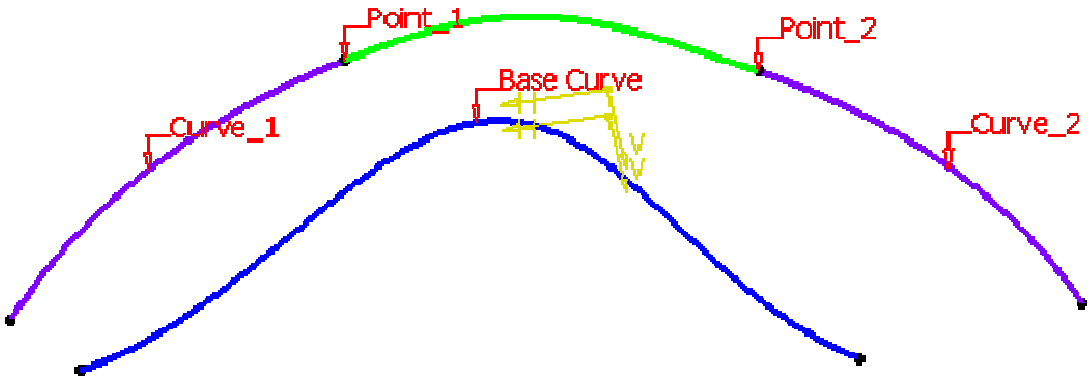



Minimum Extremum on a Surface along the X Axis

Student Notes:

# Creating a Connect Curve

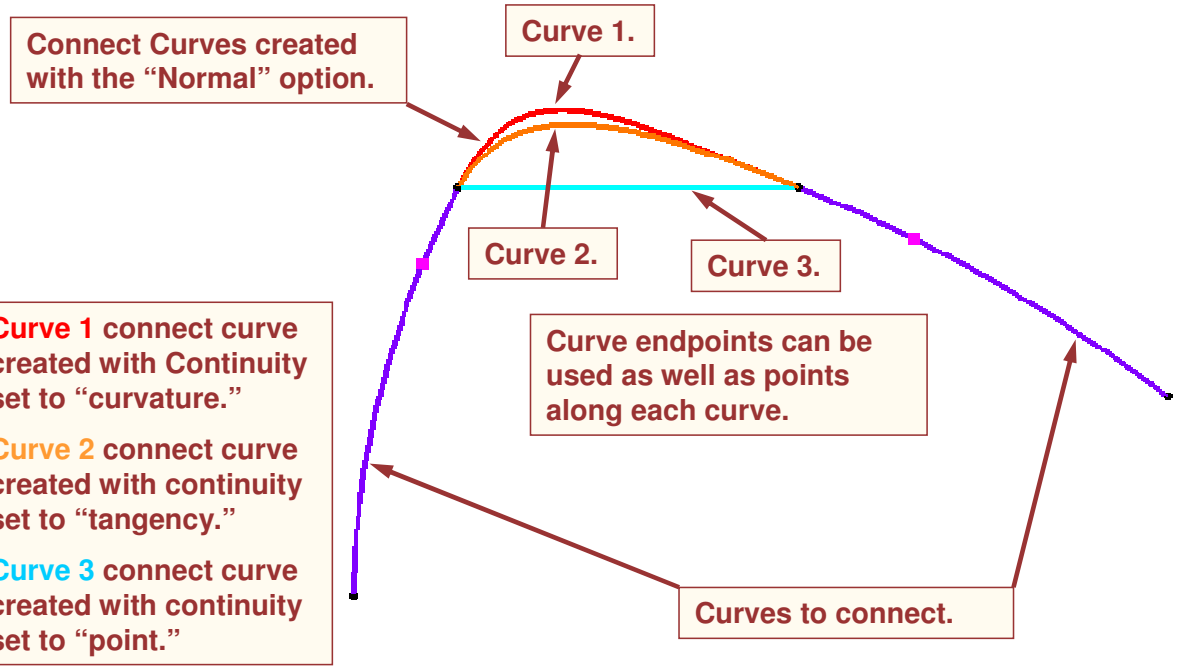
*You will learn how to create a connect curve within the Generative Shape Design workbench.*

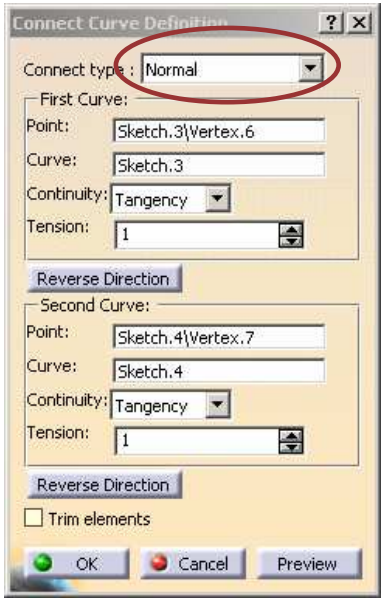



Student Notes:

# Review of Connect Curve from Surface Design Course

**Recall that in the Surface Design course, you created a Connect Curve that connects two points on two curves allowing you to control the continuity and shape as shown using type: Normal.**

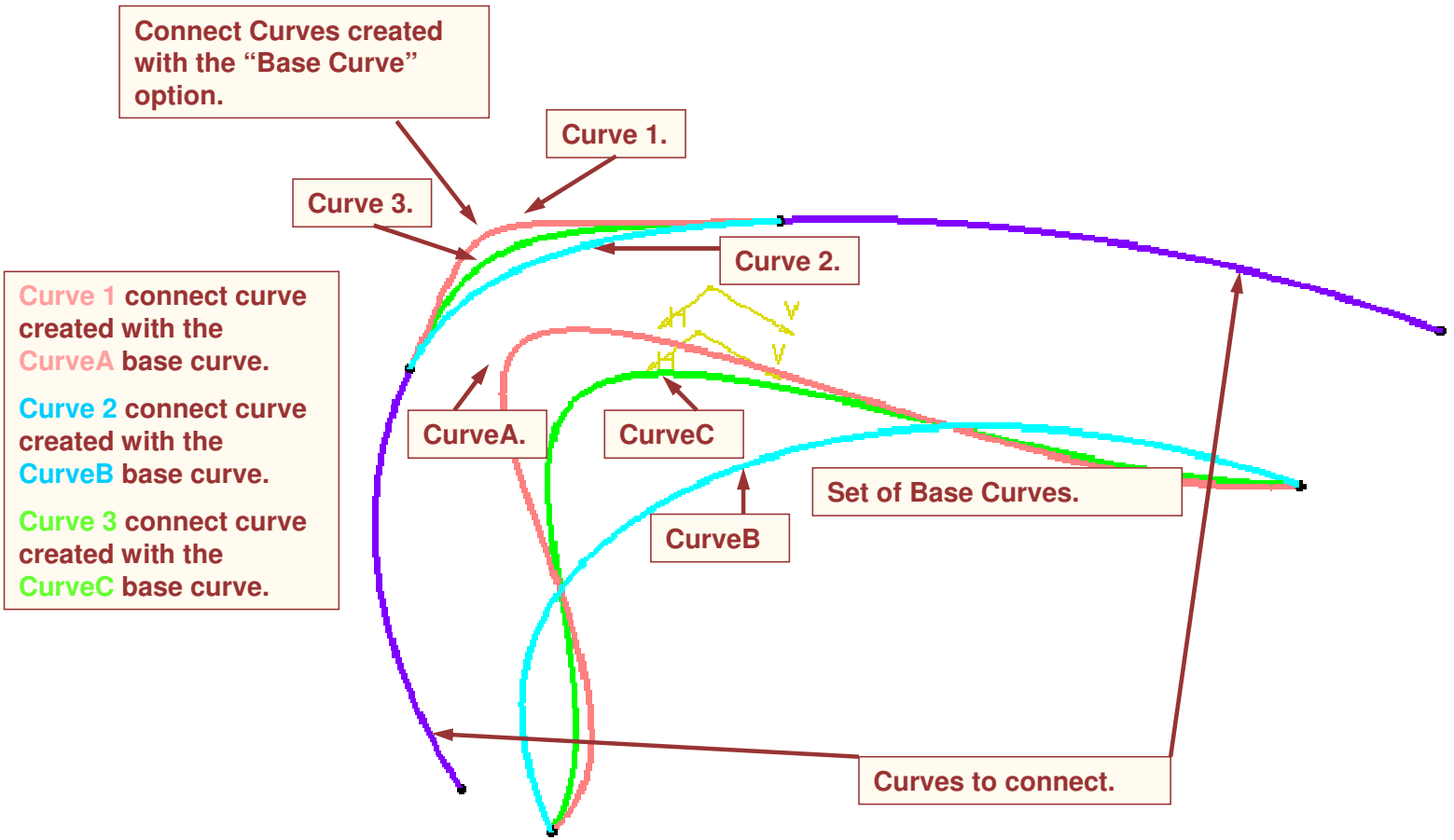


Student Notes:

# What is a connect curve created with “Base Curve”?

**In the Generative Shape Design workbench (under HD2 configuration), you can create a Connect Curve that connects two points or two curves by keeping nearly the same shape as a reference curve: the Base Curve.**



Student Notes:

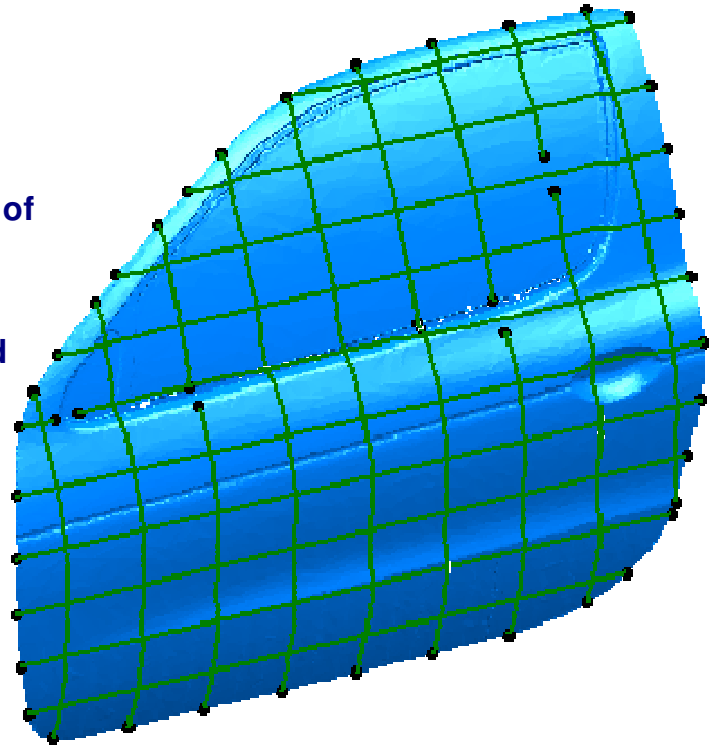
# Advanced Wireframe Geometry

## Recap Exercise

15 min

In this exercise you will work on the wireframe mesh.

- You will be creating a fine wireframe from a given set of curves using connect curves.
- You will also create a extremums on these curves and use them to determine the maximum distance of the door from the absolute axis system.

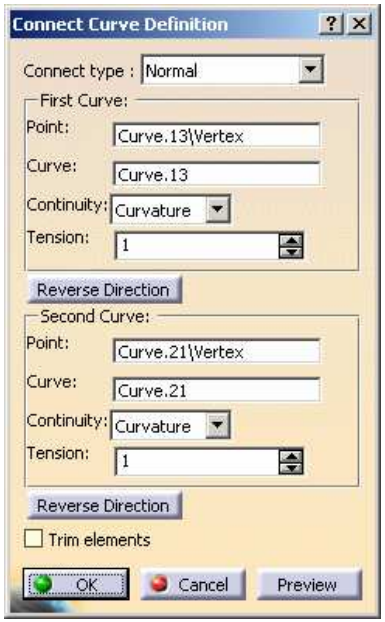


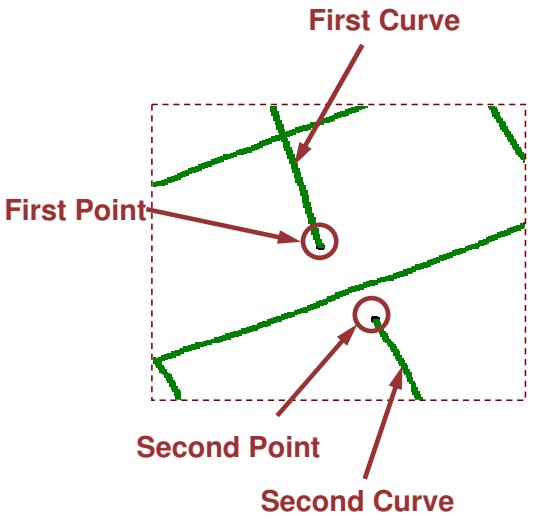
Student Notes:

# Do It Yourself (1/2)

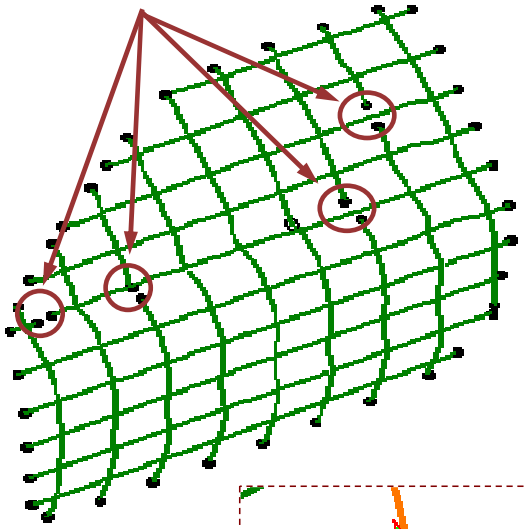
Part used: CATGSD_F_Connect_Curve_Start.CATPart

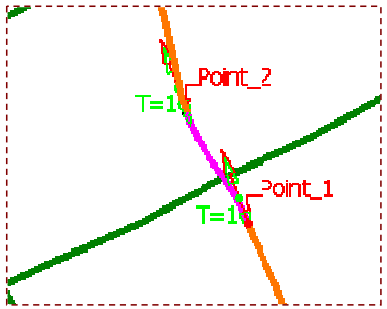
- Identify the broken curves in the given curve mesh and create a connecting curve using connect curve.
  - Select the parameters as shown in the image.
    - With Curvature continuity
    - Tension: 1
- Join each of the 'connect curve' created with its parent curves to achieve single curve.




Broken areas on the curves

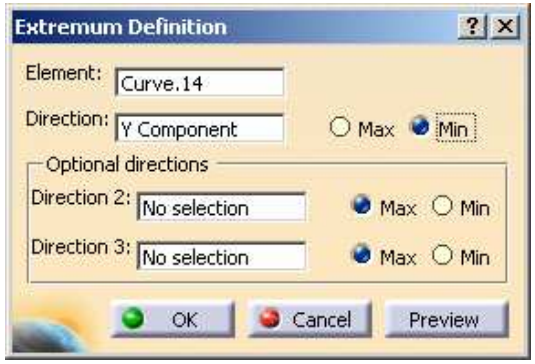


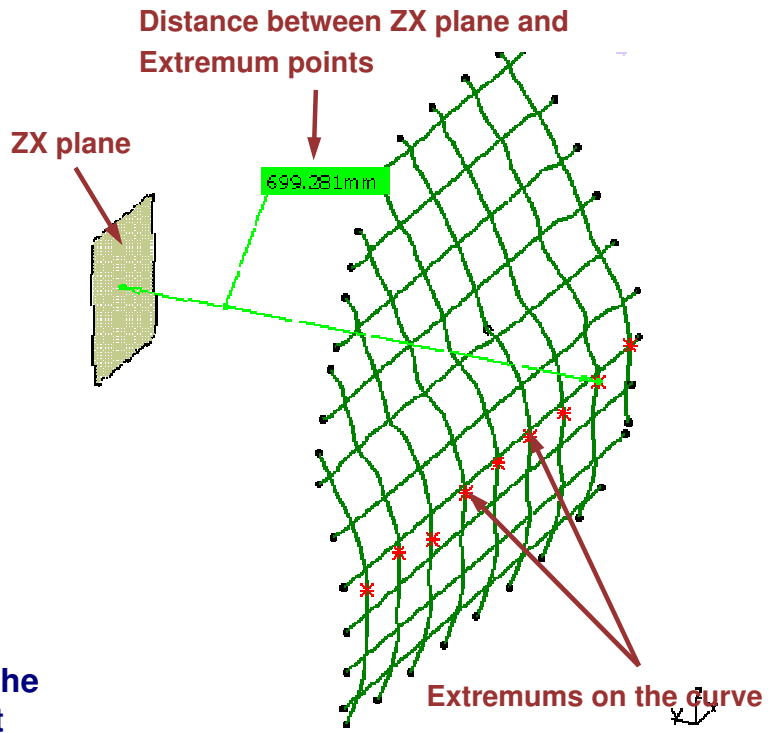


Student Notes:

# Do It Yourself (2/2)

- Create an Extremum on the curves in ZX direction.
  - Check the 'Min' box.
  - Select the input parameters as shown in the panel.
  - Repeat this for all the curves




You can determine the maximum distance of the door from absolute axis by measuring highest extremum point.

End Part: CATGSD_F_Connect_Curve_End.CATPart

Student Notes:

# Wireframe Analysis and Repair

*This lesson will cover the following Wireframe Analysis and Repair topics:*

- About this Lesson
- Why do you need to Analyze Wireframe Geometry?
- Curve Connect Checker
- Smoothing Curves
- Wireframe Analysis -Recommendations
- Recap Exercise: Wireframe Analysis and Repair

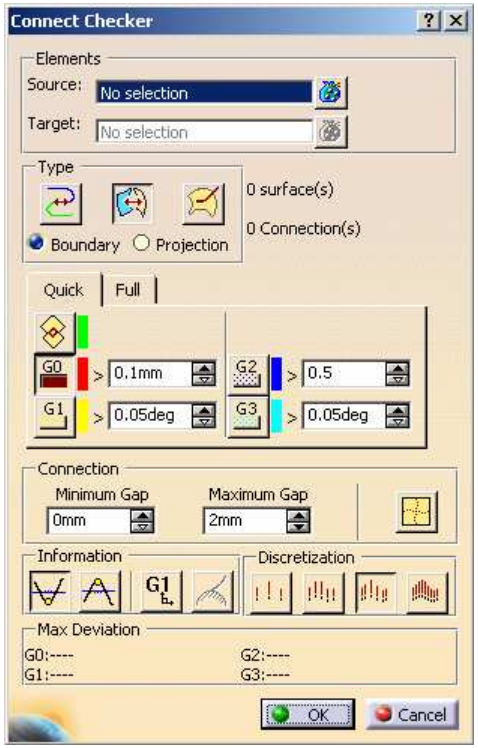
Student Notes:

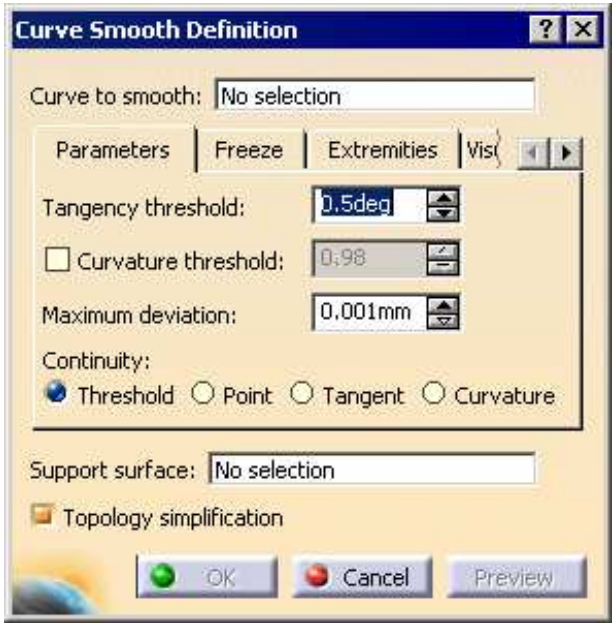
# About this Lesson

**In this lesson you will learn and practice the advanced wireframe analysis tools of Generative Shape Design workbench .You will also perform an exercise at the end of the lesson in which you will practice these tools.**
**Along with understanding what is wireframe analysis, you will also see the recommendations for performing the wireframe.**

**In this Lesson you will learn tools like,**

- **Connect Checker**
- **Smoothing Curves**



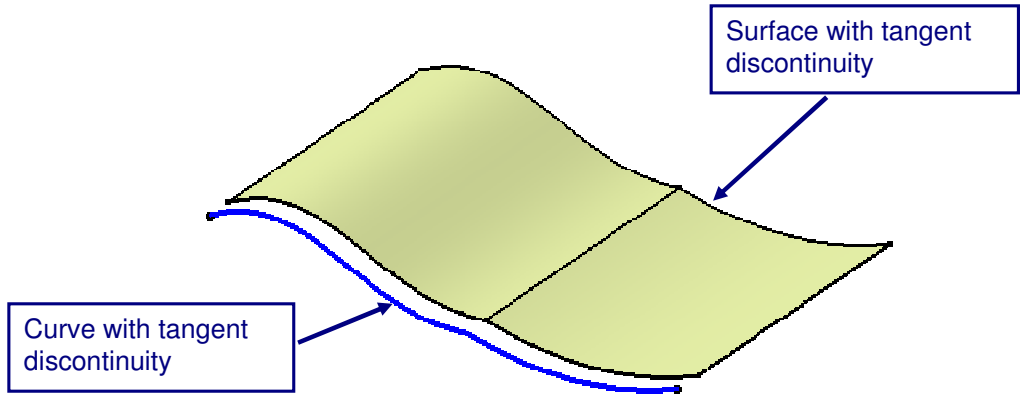
Student Notes:

# Why do You need to Analyze Wireframe Geometry?

**When a surface is created from wireframe, many of the surface's characteristics are derived from the wireframe's characteristics.**

**If a curve has an inflection, the surface will have an inflection.**
**If a curve has a tangency discontinuity, the surface will have a tangency discontinuity.**
**If a curve has a curvature discontinuity, the surface will have a curvature discontinuity.**
**If a curve has a geometric flaw, the surface will have a geometric flaw.**

**You probably get the idea……..**

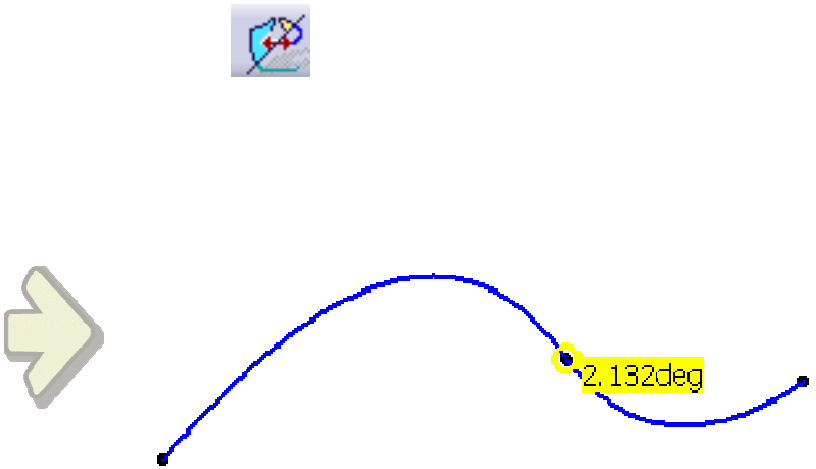


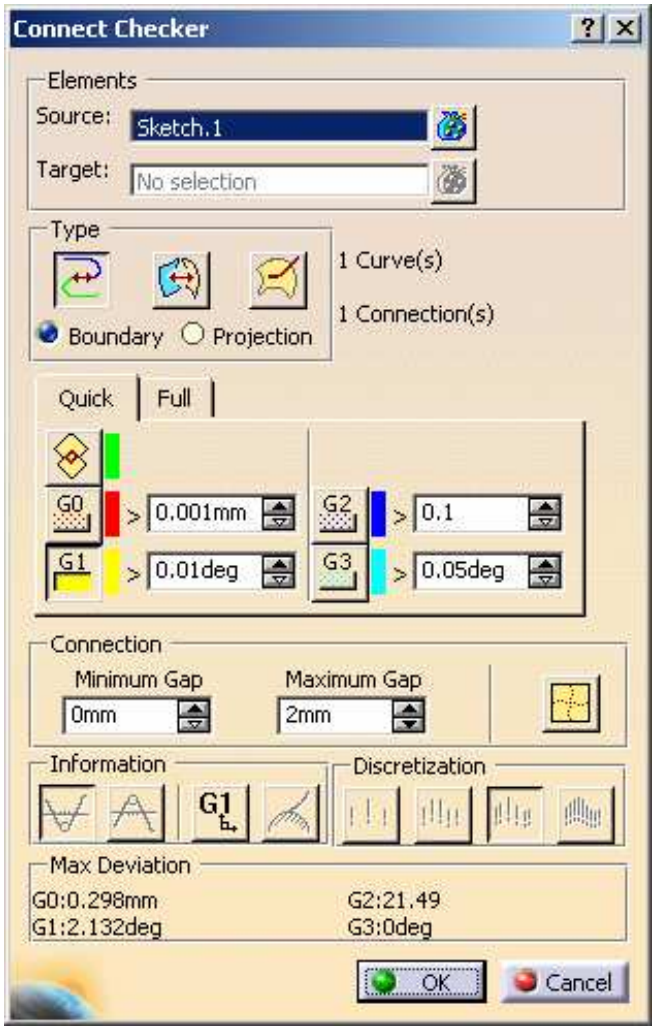

**Many times the issue with the curve cannot be seen by the naked eye. Therefore, the tools covered in this lesson are very important in assuring a high quality surfaced part.**

Student Notes:

# Connect Checker

*You will learn how to use the Connect Checker tool to analyze the curvature discontinuities on curves.*



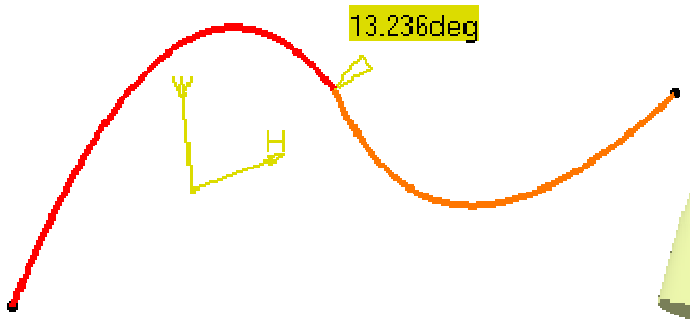



Student Notes:

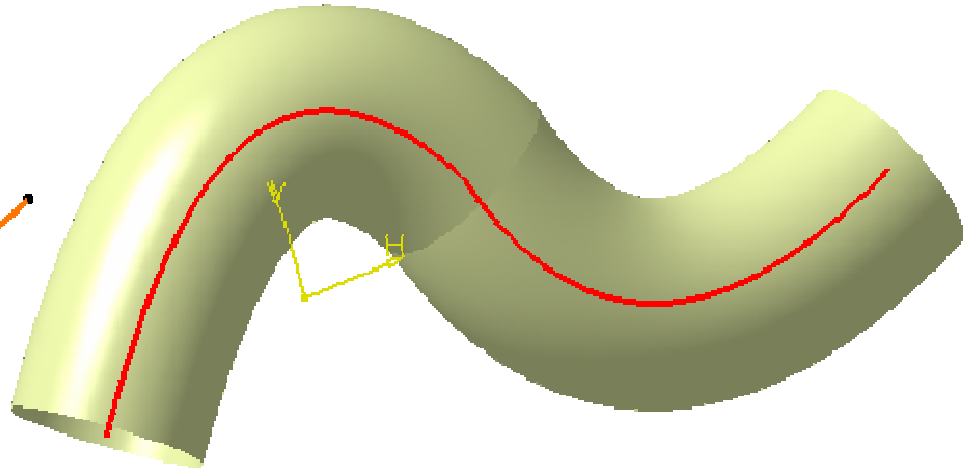
# Why the Connect Checker ?

**For wireframe based surface modeling, it is necessary to use curves that are continuous in tangency and in curvature. The connect checker allows you to detect the point, tangency or curvature discontinuities in order to smooth the non-continuous curves :**

- **G0 (mm)**
- **G1 (deg)**
- **G2 (%)**
- **G3 (deg)**
- **Overlap Defect.**




**This curve is discontinuous in tangency.**



**Building a circle sweep on it, you get a surface that is not continuous in tangency.**

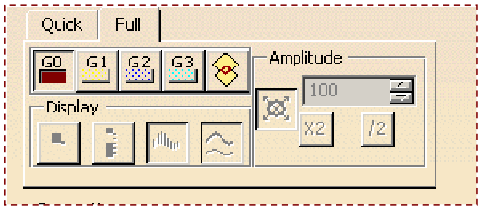
Student Notes:

# How to use the Connect Checker ? (1/2)

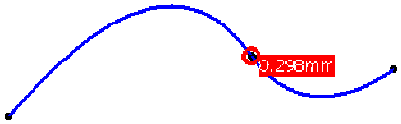
**This tool allows you to detect the G0, G1, G2 and G3 discontinuities on curves.**

1 **Select the Connect Checker icon and the curve to analyse.**


2 **Select the Analysis Type you want to process.**

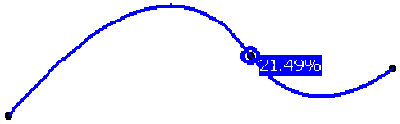



**G0 analysis**



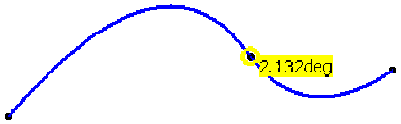

**The G0 discontinuities are displayed on the analyzed curve.**

**G2 analysis**



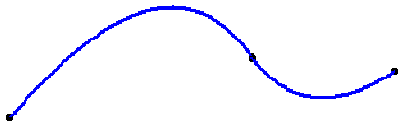

**The G2 discontinuities are displayed on the analyzed curve.**

**G1 analysis**




**The G1 discontinuities are displayed on the analyzed curve.**

**G3 analysis**

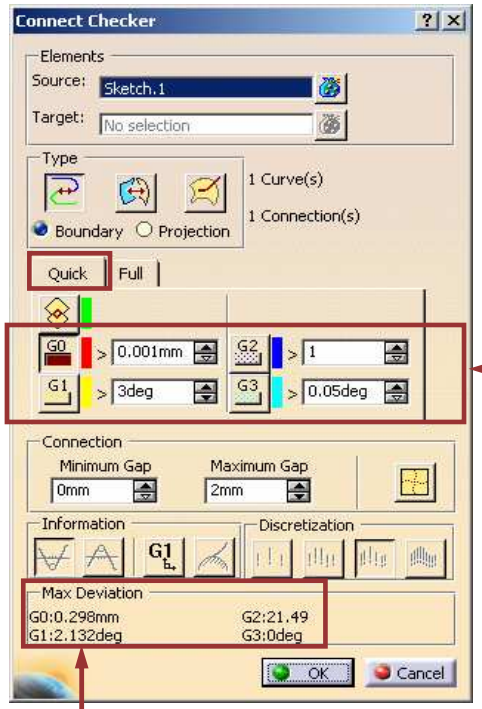


**The G3 discontinuities are not displayed on the analyzed curve.**

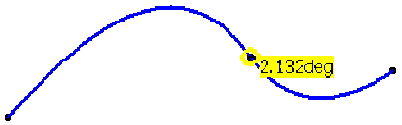
Student Notes:

# How to use the Connect Checker ? (2/2)

3 Select the Quick tab.

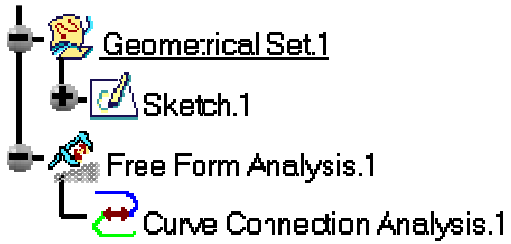



This option allows the user to give thresholds below which the discontinuity is not detected.




Display of the maximum discontinuity values on the curve.

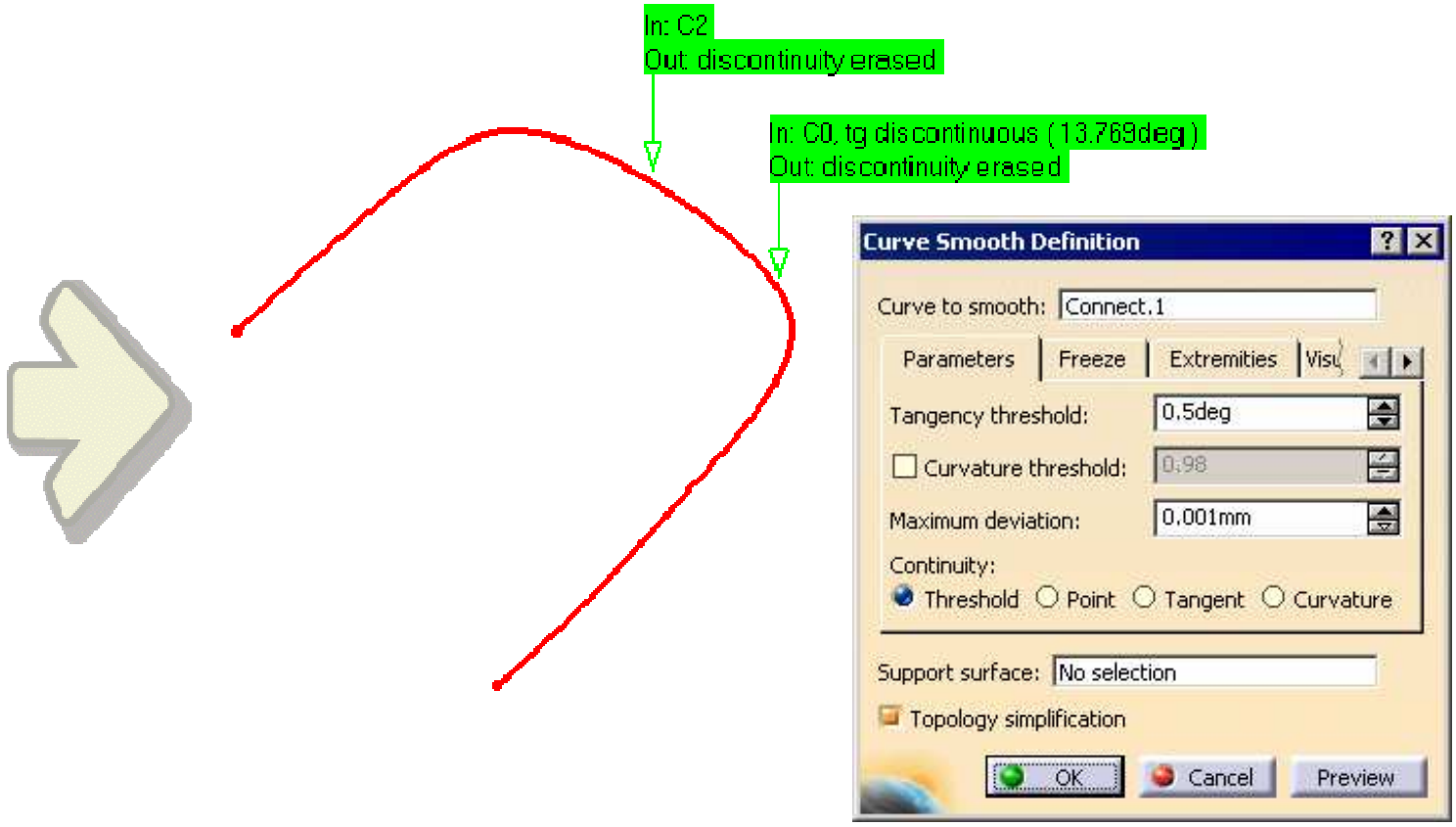
4 Click OK to confirm. The Connect Checker Analysis is added to the specification tree :



Student Notes:

# Smoothing Curves

*You will learn how to smooth curves.*

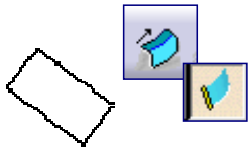


Student Notes:

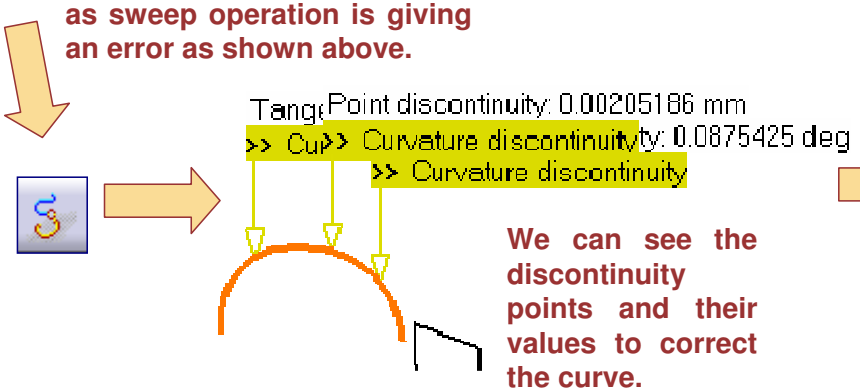
# Why Smoothing Curves?

**Sometimes when you want to create a sweep for instance, CATIA warns you that the profile curve is not continuous in tangency and that it could not build the geometry as you wish. The Smoothing Curve function allows you to clean these curves from distance,Curvature and tangency discontinuity.**

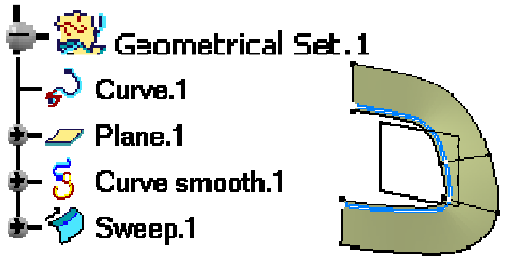
We want to create a Line-type sweep from this curve using the plane as reference surface.




We need to smooth the curve before generating the sweep as sweep operation is giving an error as shown above.




We can see the discontinuity points and their values to correct the curve.




Using the smoothed curve, we can create the swept surface.

STUDENT GUIDE

Student Notes:

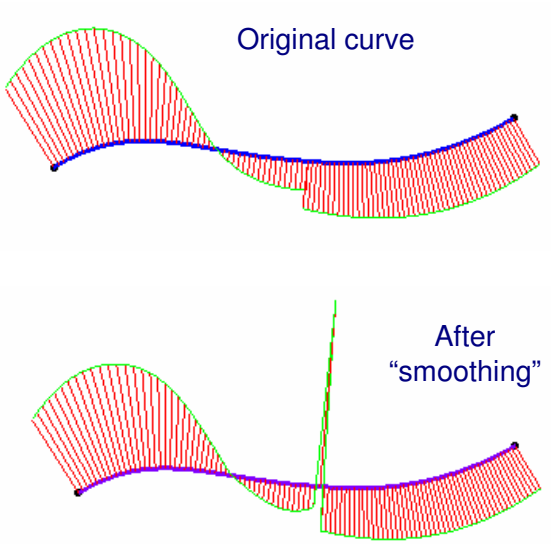
# Wireframe Analysis –Recommendations

*You will learn about specific methods and recommendations concerning the use of wireframe analysis and repair.*

Student Notes:

# Cautions for using Curve Smooth

**Curve Smooth repair internal curve flaws such as:**
**- Tangent discontinuities**
**- Curvature discontinuities**
**by selecting the curve you want to repair.**




**However, you should always use Porcupine Curvature Analysis after performing the operation as the resultant curve may not be as “smooth” as you would wish.**

**This is especially true when the “Maximum deviation” value is small. As a general rule, the larger the flaw, the larger the Maximum deviation value.**

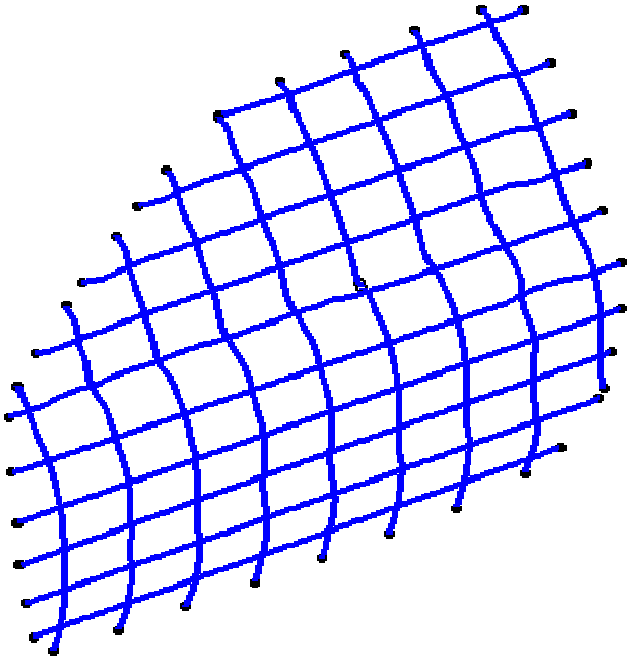
Student Notes:

# Wireframe Analysis and Repair

## Recap Exercise

10 min

- Perform a Connect Checker Analysis
- Repair the curve using Curve Smooth

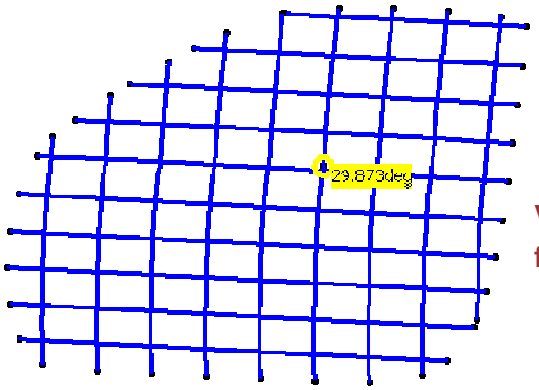


Student Notes:

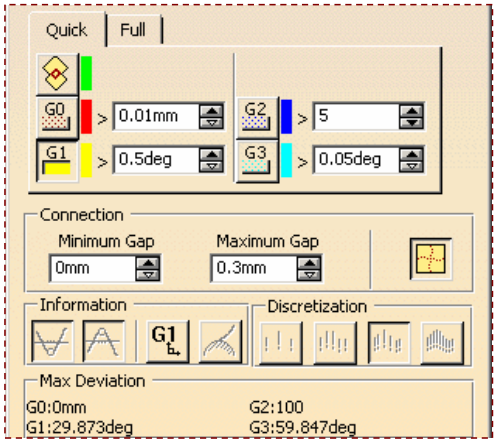
# Do It Yourself…

**Part used: CATGSD_F_Wireframe_Analysis_Recap.CATPart**

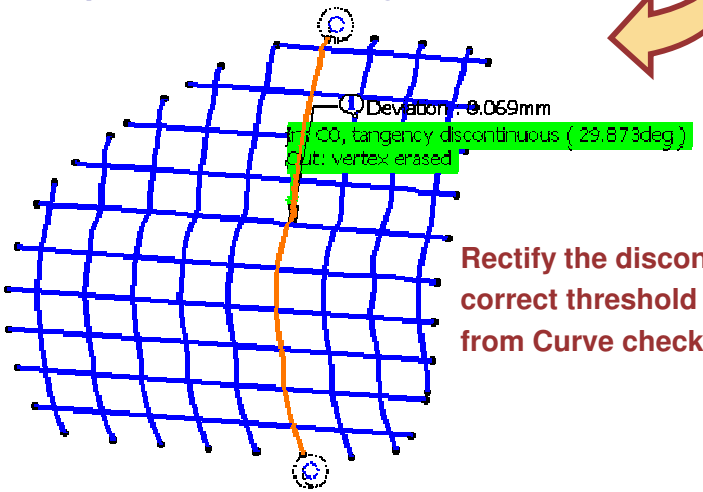
- **Perform a Connect Checker analysis.**



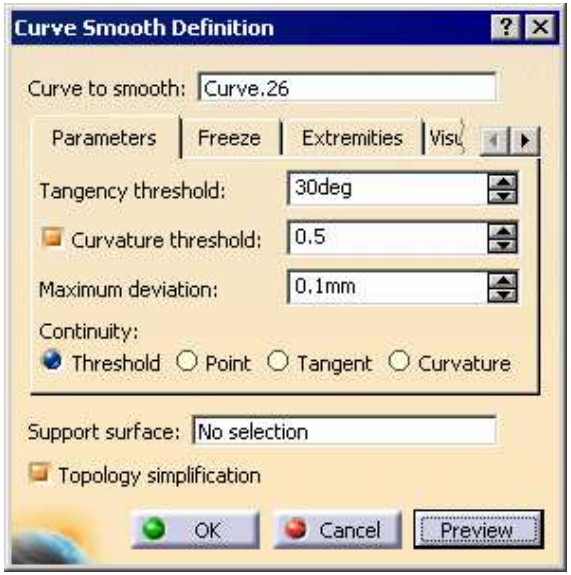

**Validate the discontinuity for all the curves**




- **Repair the curve using Curve Smooth.**




**Rectify the discontinuity Using correct threshold value obtained from Curve checker analysis**



Student Notes:

# Creating Advanced Swept Surfaces

*This lesson will cover the following Advanced Swept Surface topics:*

- About this Lesson
- What is a Swept Surface?
- Inputs for Sweep
- Creating a Swept Surface
- Recap Exercise: Knob
- Creating an Adaptive Swept Surface
- Recap Exercise: Adaptive Swept Surface

Student Notes:

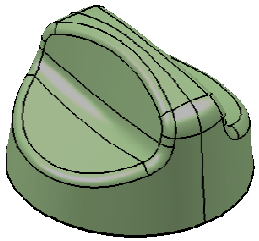
# About this Lesson

**In this lesson you will be learning how to create different types of Swept Surfaces (Line,Circle,Conic surfaces) and Adaptive Swept surfaces.**

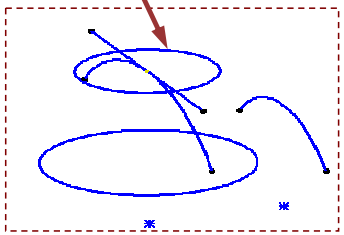
**You will be creating a Parts 'Knob' and 'Housing' at the end of this lesson.**

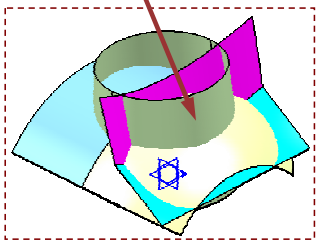
**Along with knowing how to create different type of Swept surfaces, you will learn about the inputs required to create these surfaces such as,**

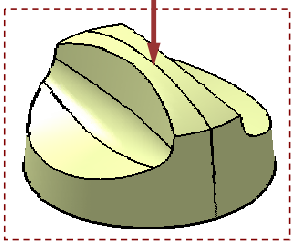
- **Spine**
- **Laws**

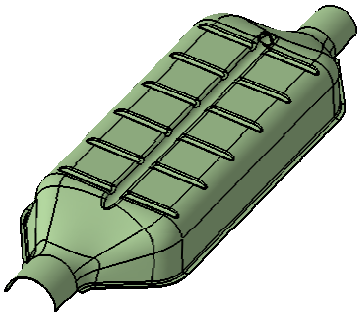


Knob

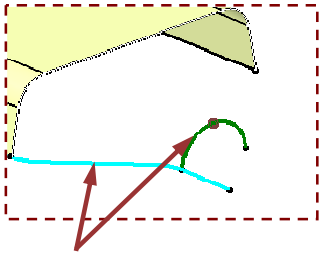


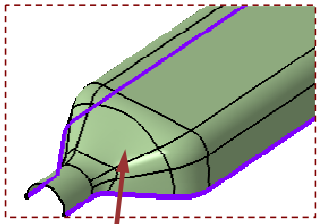



Adaptive Swept Surface



Inputs forAdaptive Swept Surface



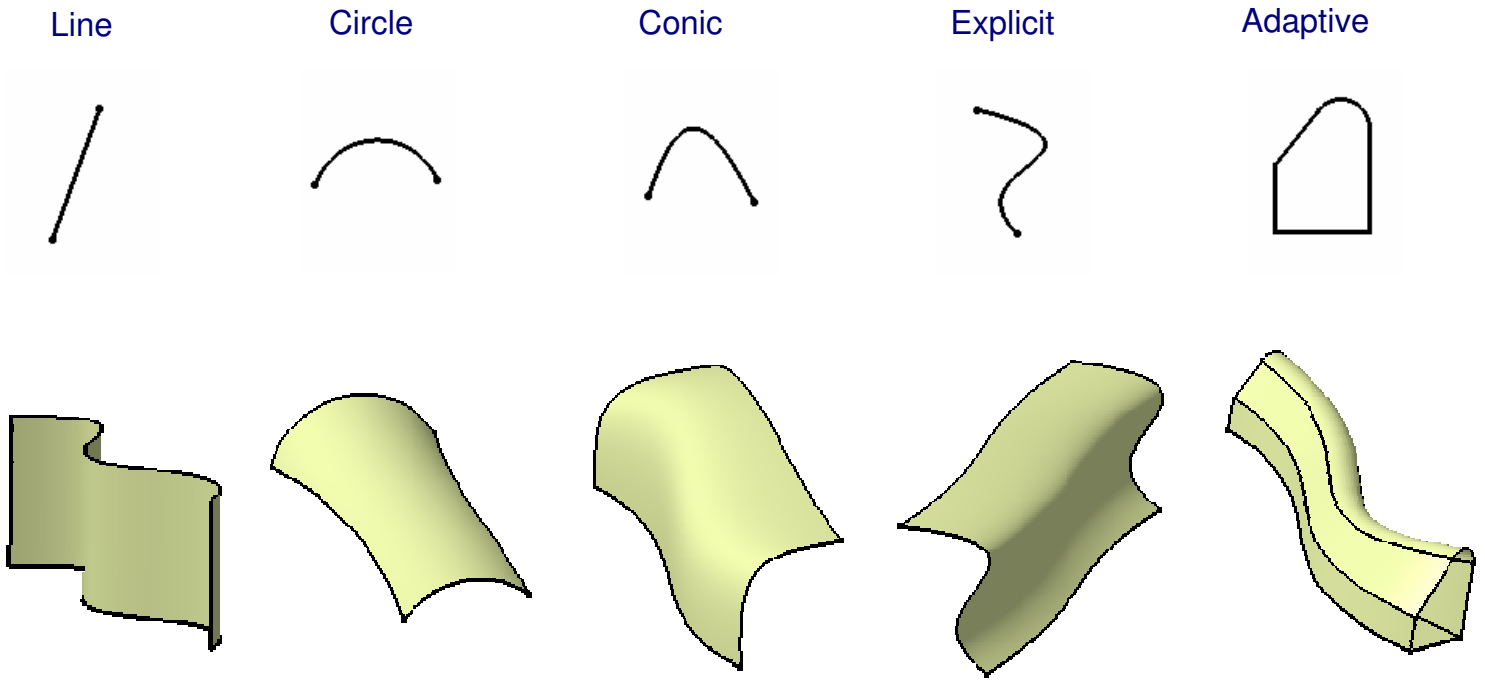
Adaptive Swept Surface

Student Notes:

# What is a Swept Surface?

**A swept surface is one that is created by sweeping a particular type of profile through space to form a surface. Each swept surface will follow along one or more guide curves.**

**There are five types of swept surfaces:**



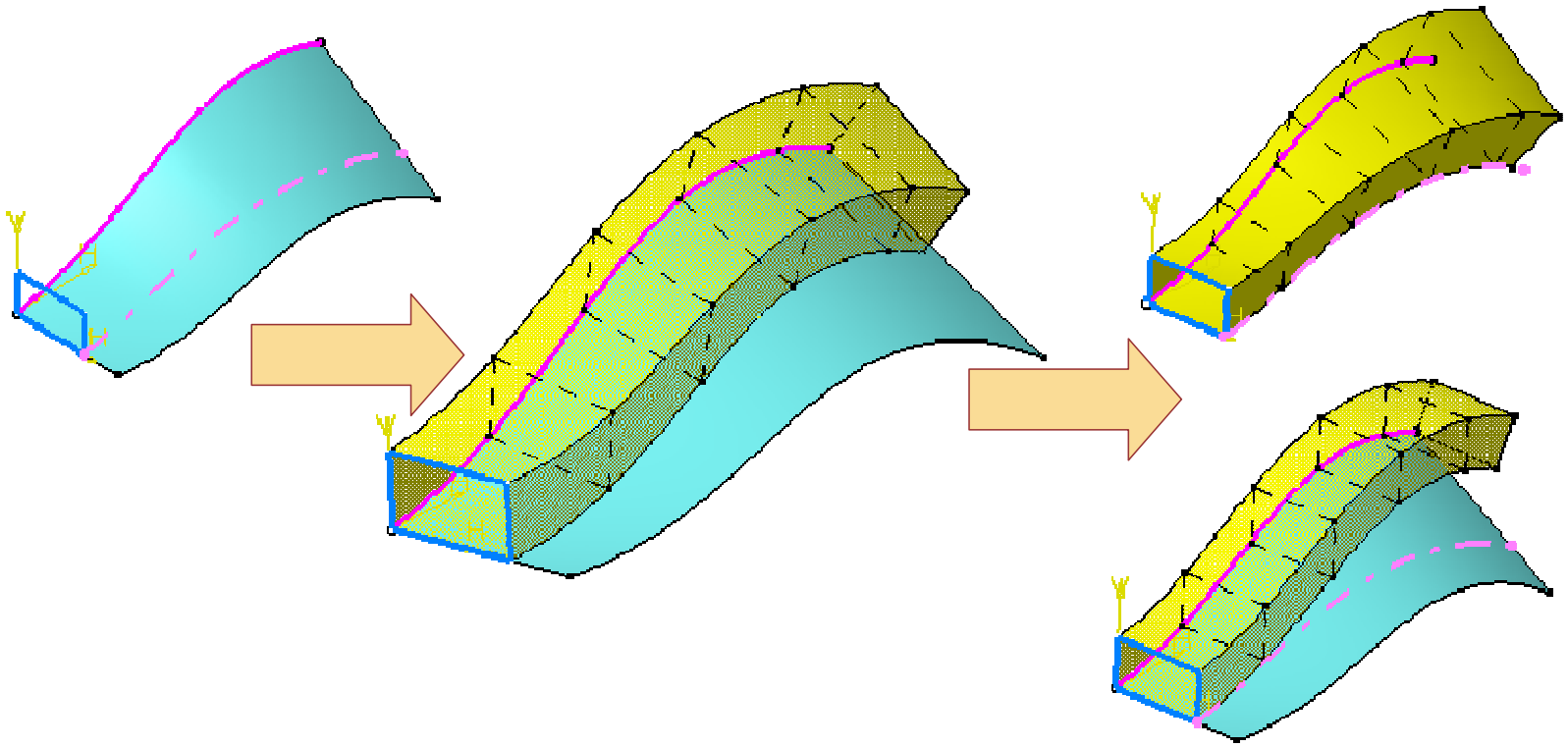

**For Line, Circle, and Conic sweeps, CATIA will generate the profile for you.**

Student Notes:

# Show Me...

## *Scenario: Create an Explicit Sweep.*

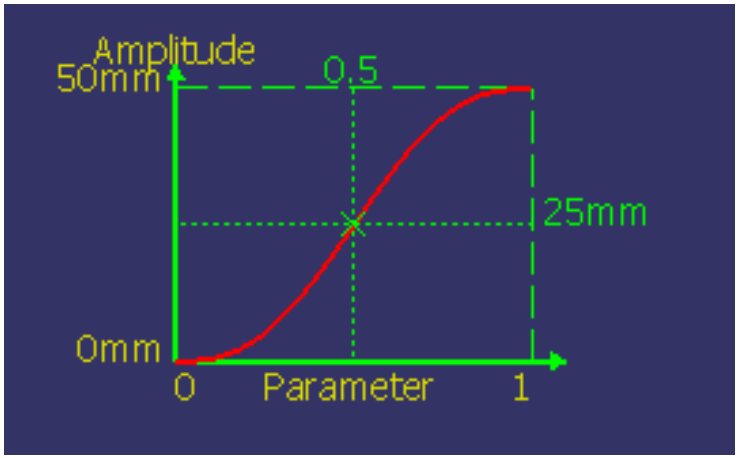
**Let us show you through a simple scenario the general process to create an Explicit Sweep.**



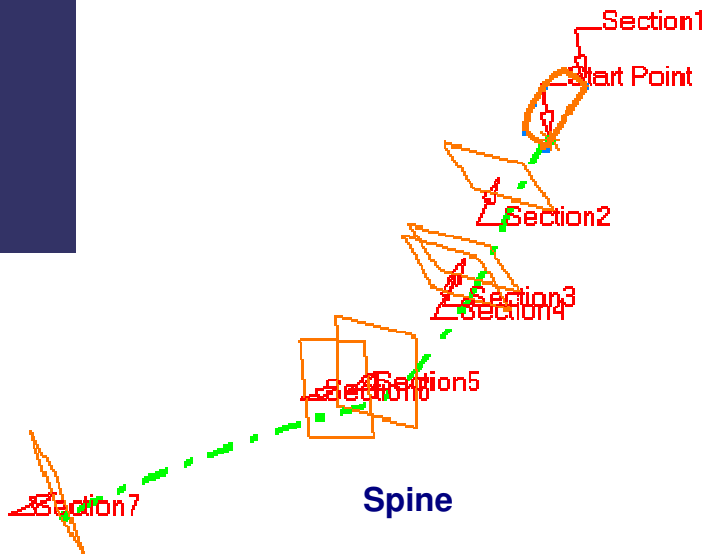
Student Notes:

# Inputs for Sweep

*You will learn about the Inputs used when creating Swept Surfaces*




**Law**

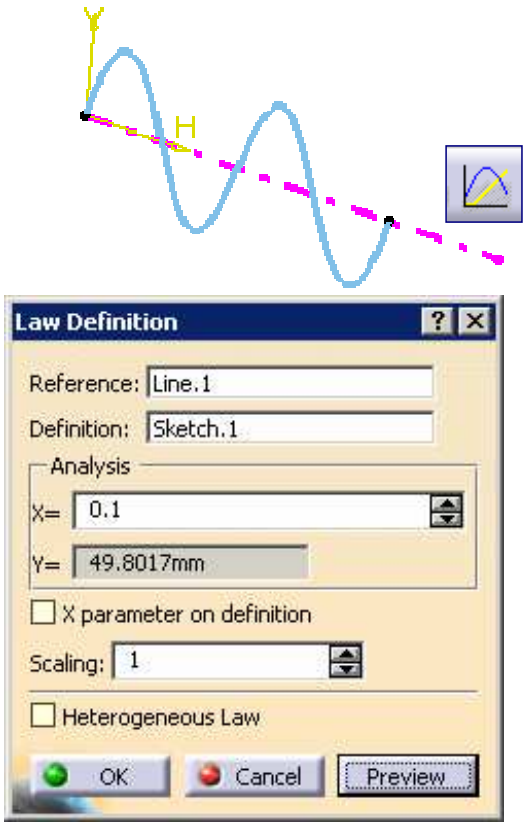


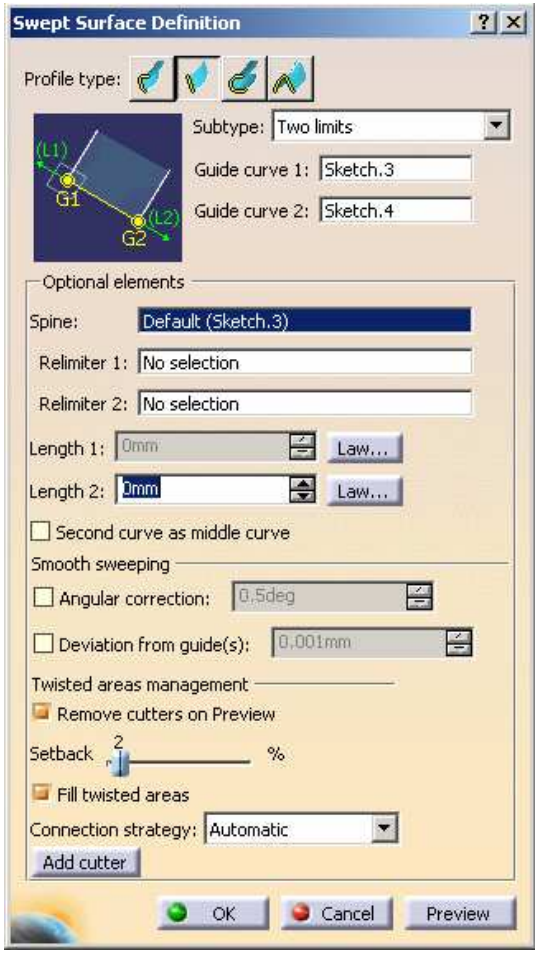

**Spine**

Student Notes:

# Laws

*You will learn how to create laws and use them to Generatate Shape Design Elements, such as Swept surface, fillets.*



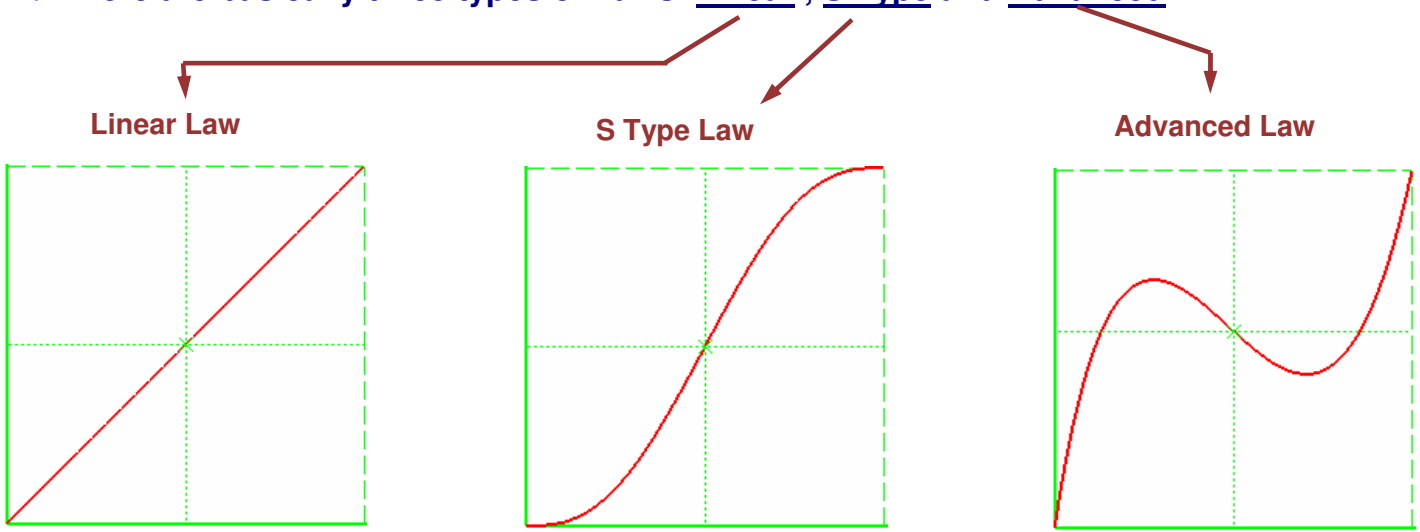



Student Notes:

# What are Laws?

**Parametric surfaces created require certain input parameters to define them. Example of input parameters can be length or angle parameters. When you input a numeric value, the parameter will remain constant throughout the entire development of the surface. When you desire a particular input to vary, you must use a LAW.**

- **Types of Laws.**
  - **There are basically three types of Laws: Linear , S Type and Advanced**

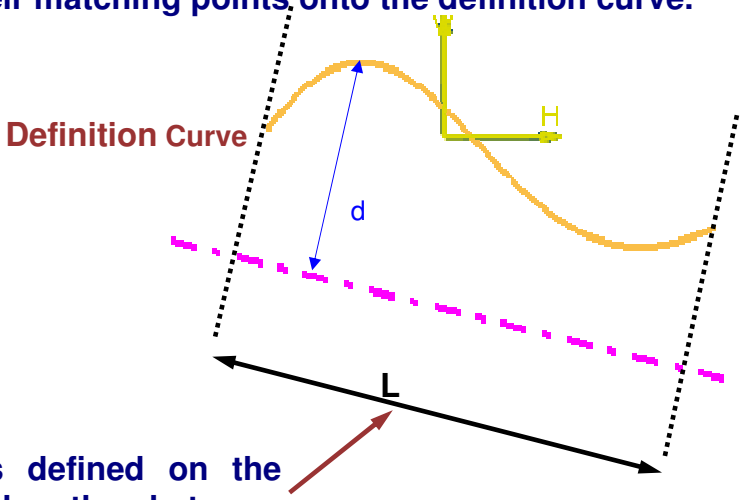



**Linear and S Type laws are defined by simple parameter input in a panel.**
**An Advanced Law requires you to define geometry in the part : a reference and a definition curve.**

Student Notes:

# What are Advanced Laws?

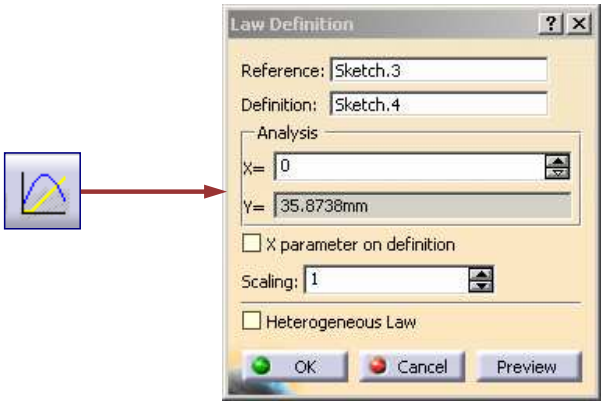
**An advanced law is computed as the distance between points on the reference line and their matching points onto the definition curve.**




**The law define the variations of d along L.**

**Reference Line**

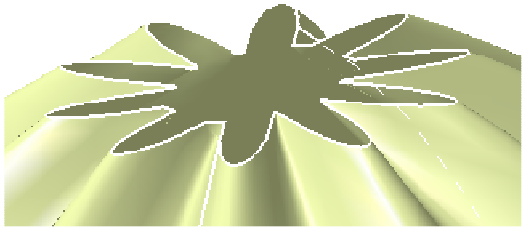
**The law is defined on the common length between both entities.**



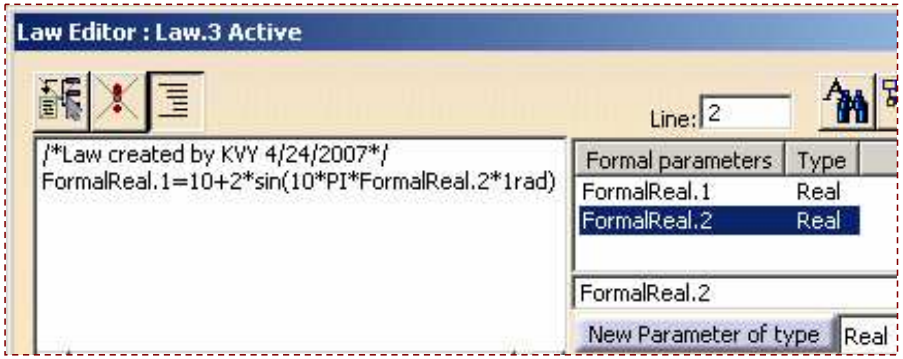
Student Notes:

# Where can Laws be used (1/2)

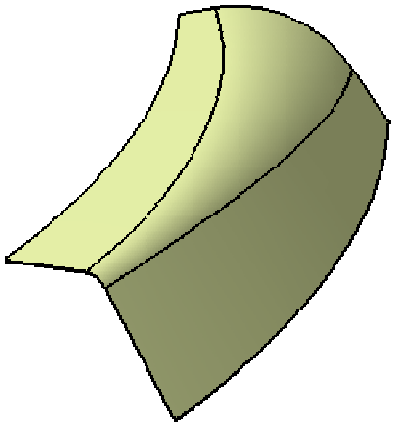
**Laws can be used in situations where the curves or surfaces that are being created vary according to a pattern / mathematical equation.**

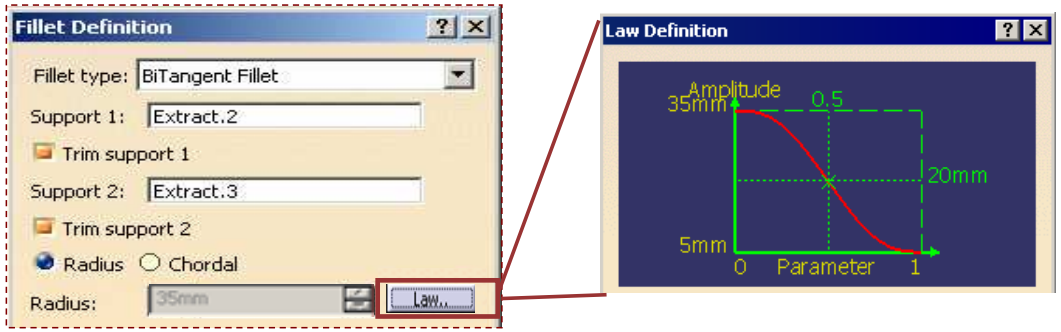
**The curve shown here has been created using Law in the Knowledgeware Law editor**




**Once this law is created, it can be used in GSD commands.**



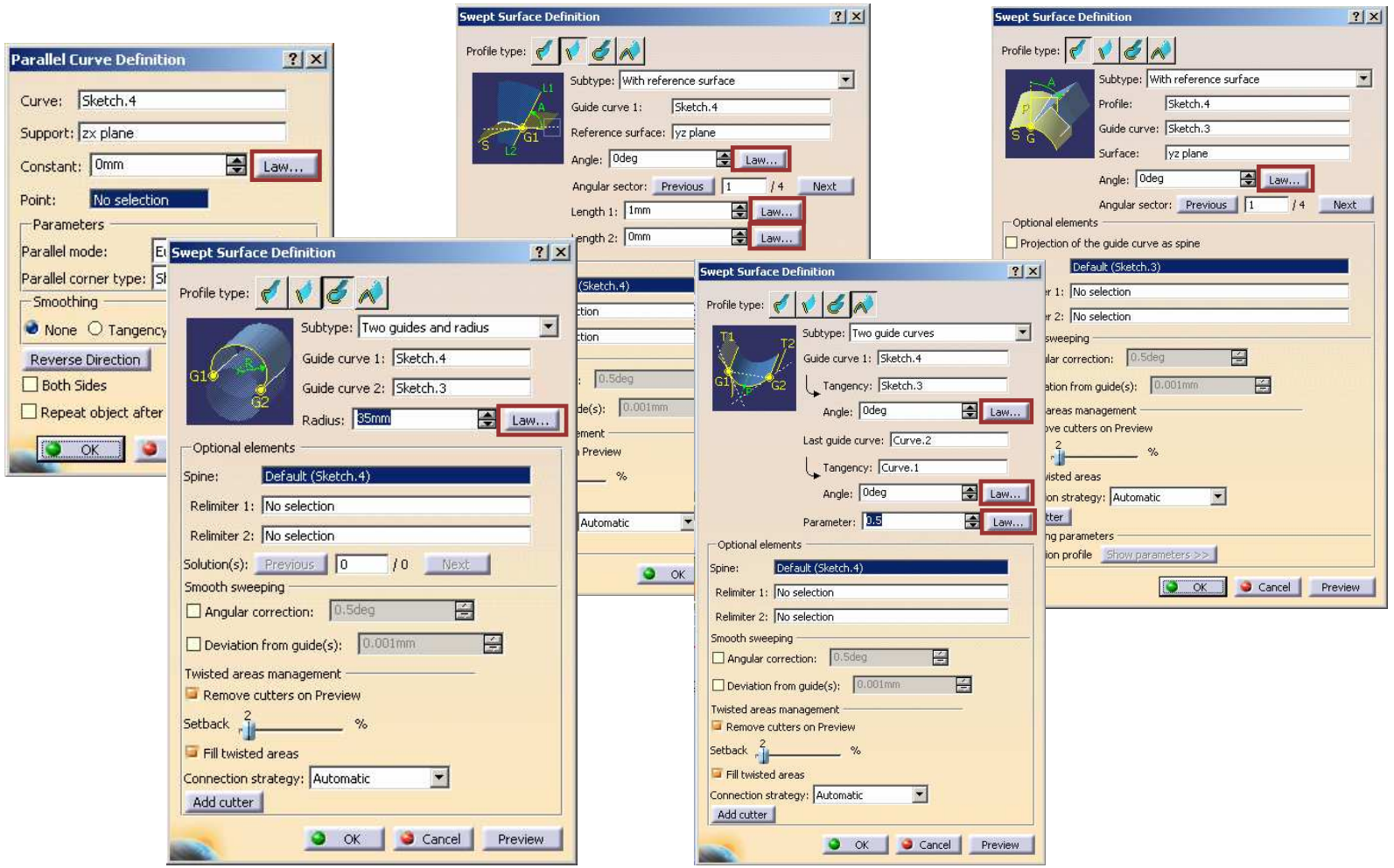
**Radius of the Shape Fillet shown here varies according to the "S" law that can be defined while creating shape fillet.**



Student Notes:

# Where can Laws be used (2/2)

**Laws can be used wherever you see a law button near an input field. As you can see below, there are many places you can use a law.**

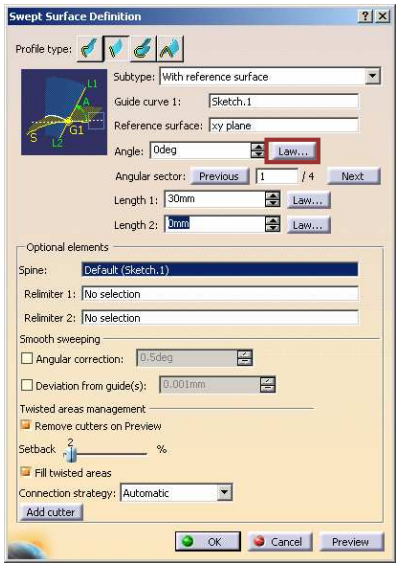


Student Notes:

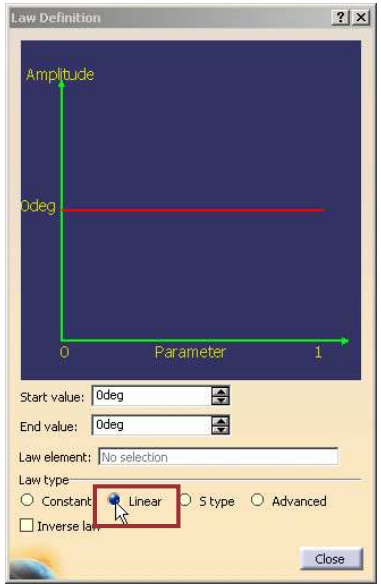
# Creating a Linear Law

**A Linear law is one where the parameter varies between two values in a "linear" fashion. No geometry is created.**

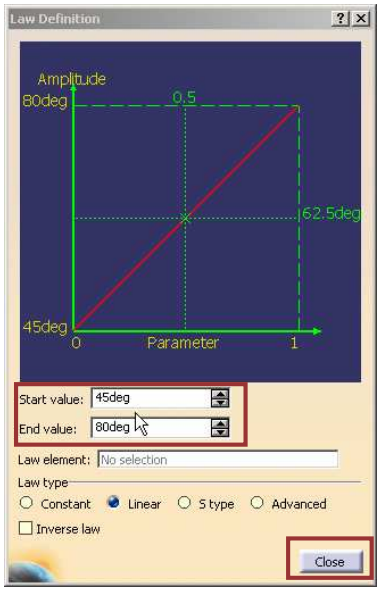
1. Select the Law button next to the desired parameter.




2. Select "Linear" in the law panel.

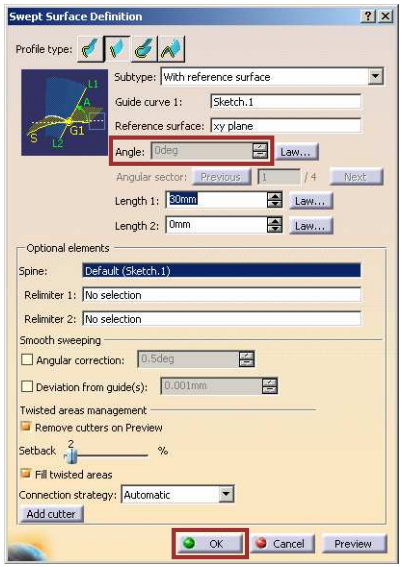



3. Key in the start and end parameters.




4. Select Close.

5. The parameter field is dimmed, indicating that a law is used for that parameter.

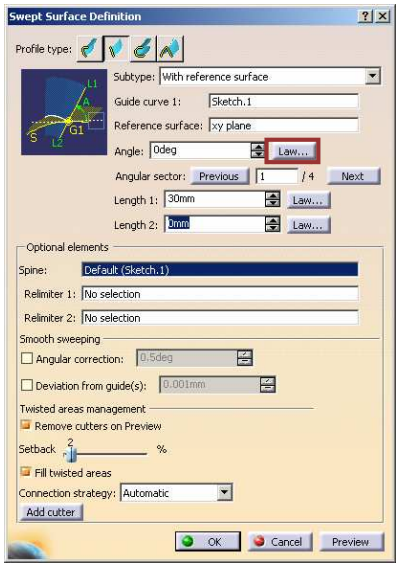



6. Click on OK to create the surface using the law.

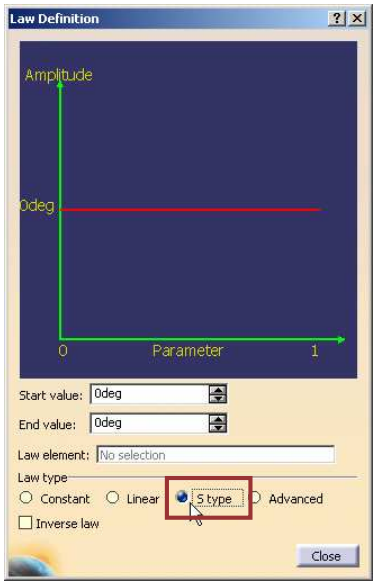
# Creating an S Type Law

**An S Type law is one where the parameter varies between two values in an "s-shaped" fashion. No geometry is created.**

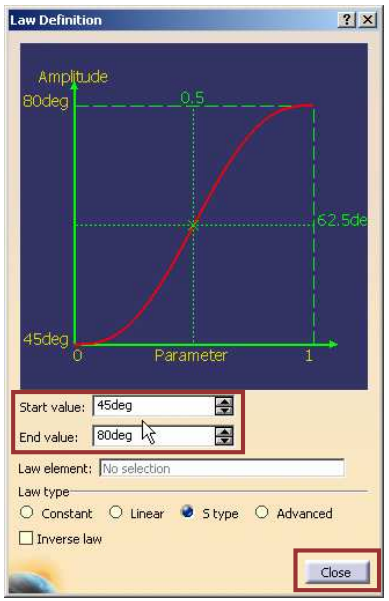
**1** Select the Law button next to the desired parameter.




**2** Select "S type" in the law panel.

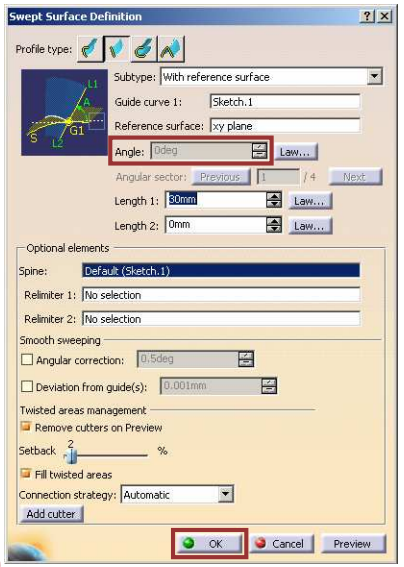



**3** Key in the start and end parameters.




**4** Select Close.

**5** The parameter field is dimmed, indicating that a law is used for that parameter.




**6** Click on OK to create the surface using the law.

Student Notes:

Student Notes:

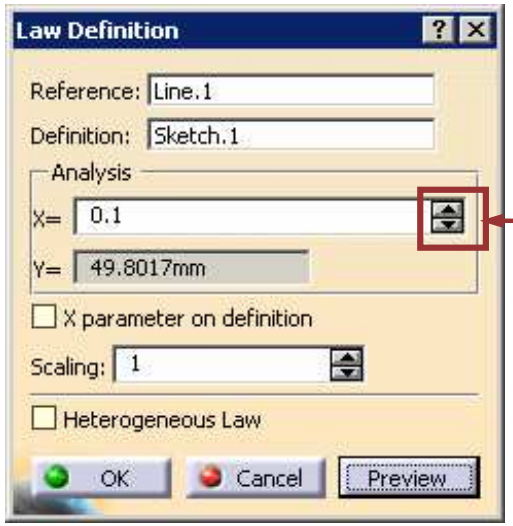
# Creating an Advanced Law

**Create an evolution function from existing geometry.**

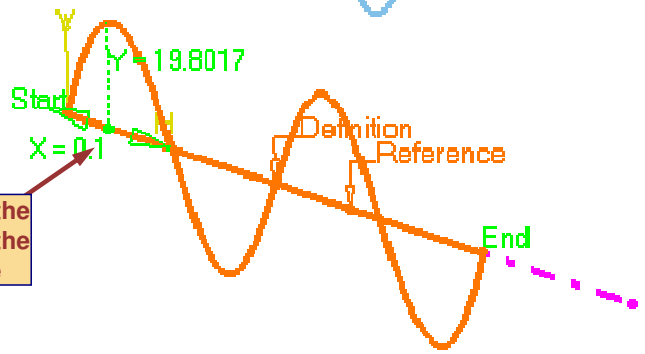
1.  Select the Law Icon.

2. Select the line you want as reference line.

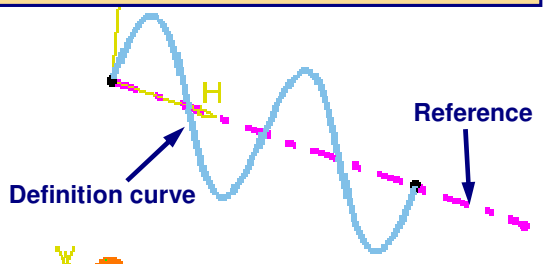
3. Select the line or curve you want as definition curve for the evolution law.




Fix a X value or use the manipulators to see the corresponding Y value




For an Advanced Law, the first thing you must do is create two pieces of geometry in two sketches somewhere in the part. These represent the Reference and Definition curves.




4. Click on OK to confirm. The law is added to the Specification Tree.

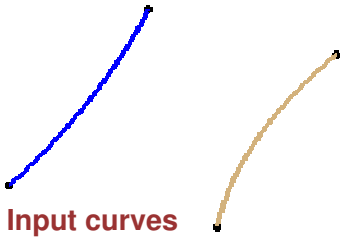

When the reference line and definition curve do not present the same length, only the common area is used to compute the law.

Student Notes:

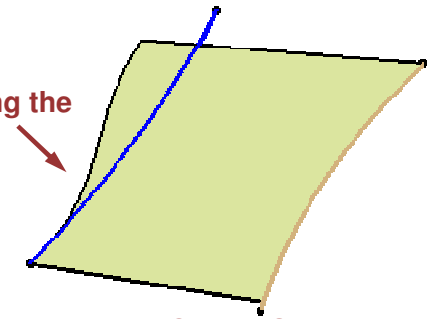
# Do It Yourself

CATGSD_Laws_DoIt_UseLaw.CATPart.CATPart

- Use S type Law in length while creating a Sweep.
  - Create Sweep of type “Two limits”.(Use the curves).
    Use “Law” , of “S” type and “Linear” type(Values : 0 , 50)
    Use Advanced Law inside Law tab and Use the Law.1 available in the part attached.

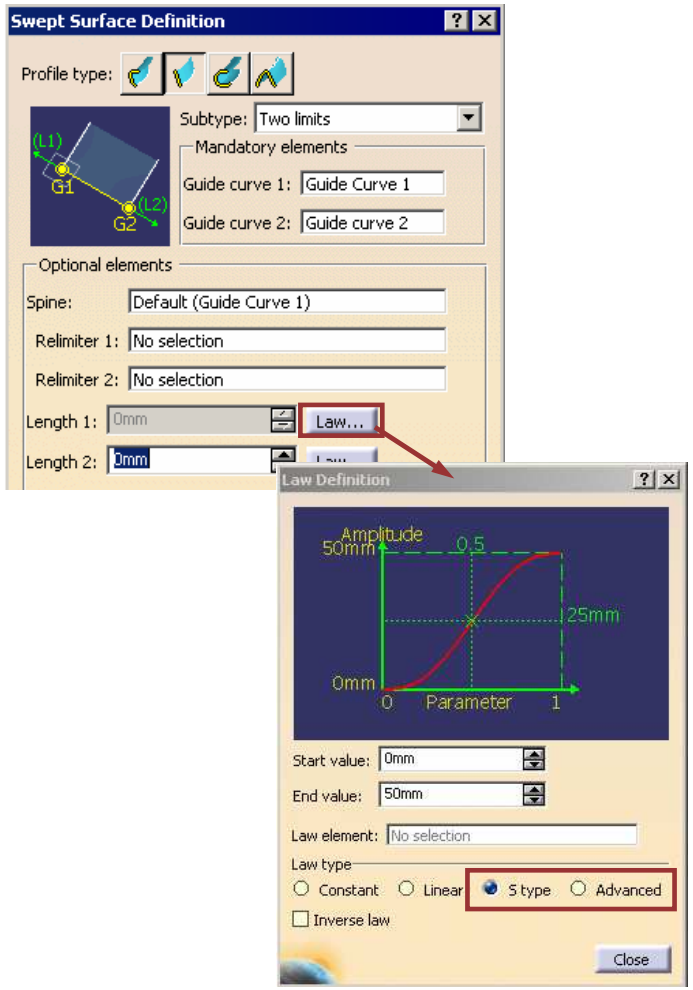


Input curves



Surface following the Law.

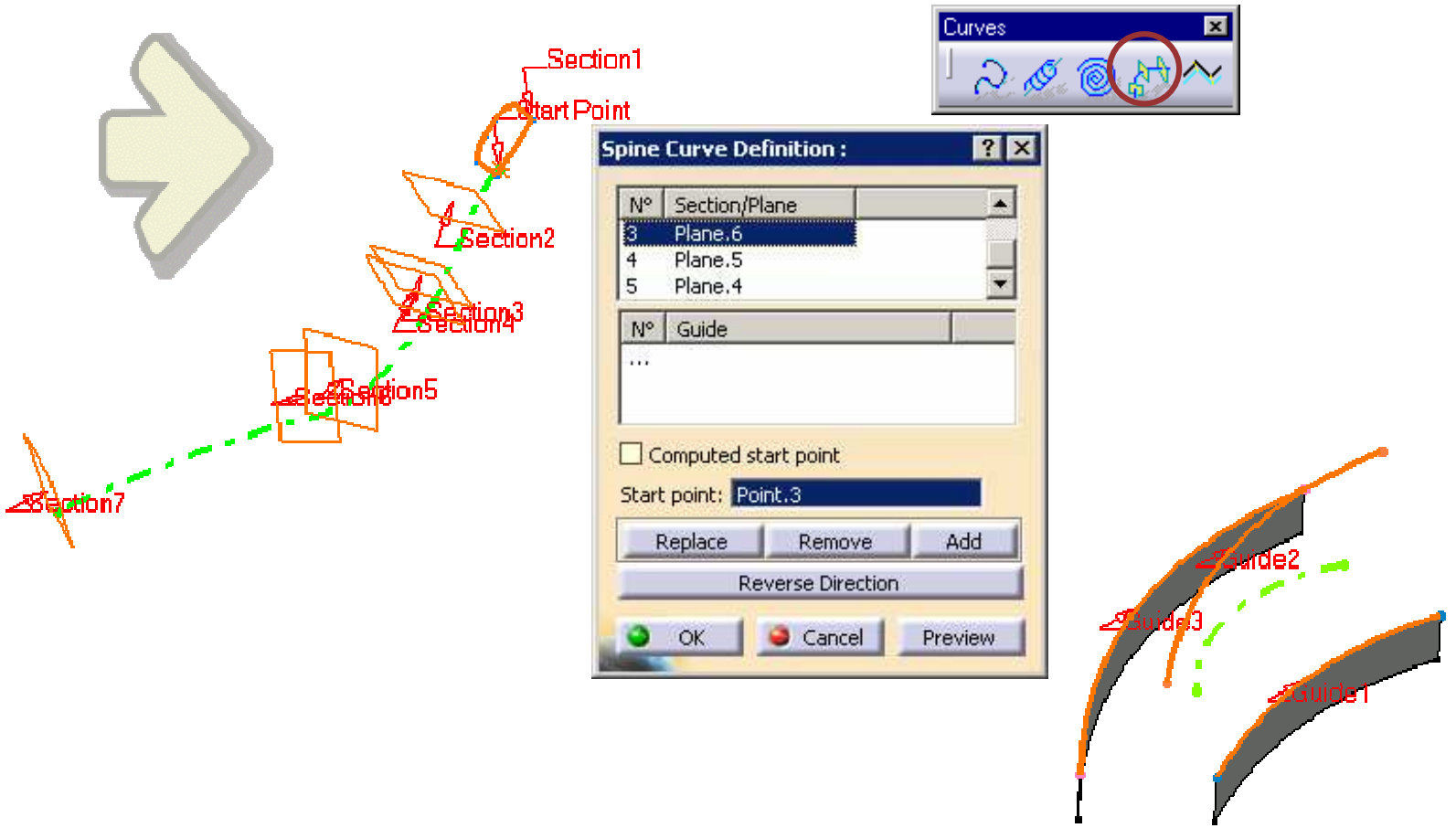
Output Sweep



Student Notes:

# Creating a Spine

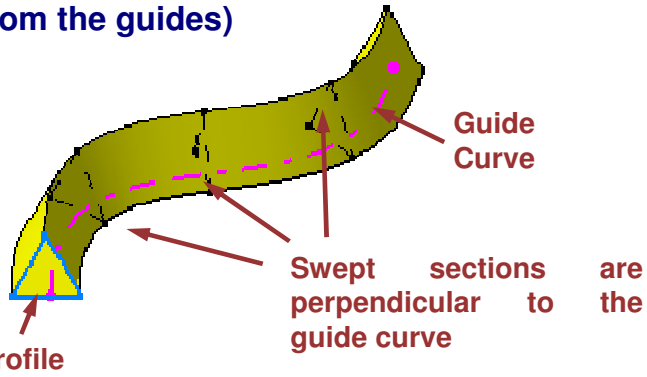
*You will learn what a Spine is and how to create it.*



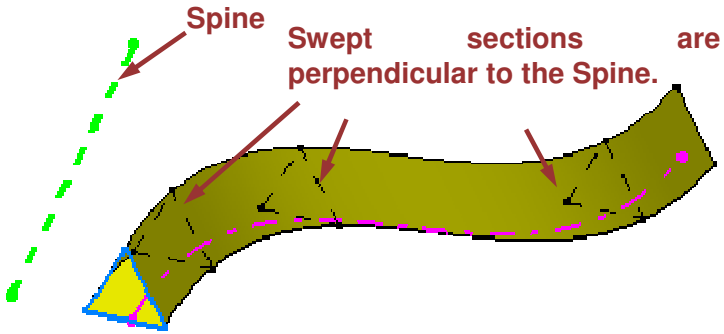
Student Notes:

# What is a Spine ?

If you want to fix an orientation for your surface sections you will have to define a Spine.The surface sections created are perpendicular to the spine defined.For the Swept and Lofted surface, there is a default spine (the guide or a computation from the guides)

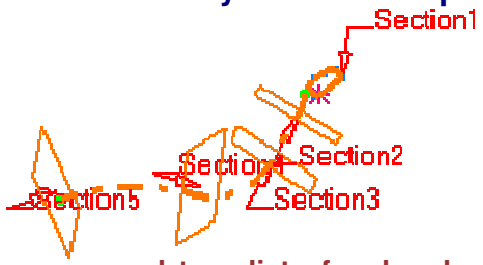



The swept sections may be oriented by another Spine (not the default one). For instance you want to get the swept sections perpendicular to the green spine:

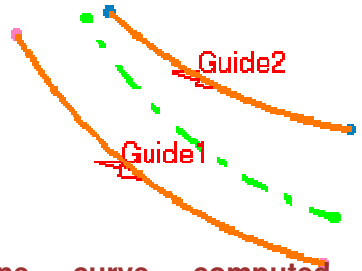



In this Swept surface, the Spine is, by default, the guide curve. Each section of the swept surface is perpendicular to this Guide Curve

The Spine icon will allow you to create a curve that will be used later as a spine
There are two ways to build a spine :


Curve normal to a list of ordered planes or planar curves




Spine curve computed from several guide curves

# What types of Surfaces use Spines ?

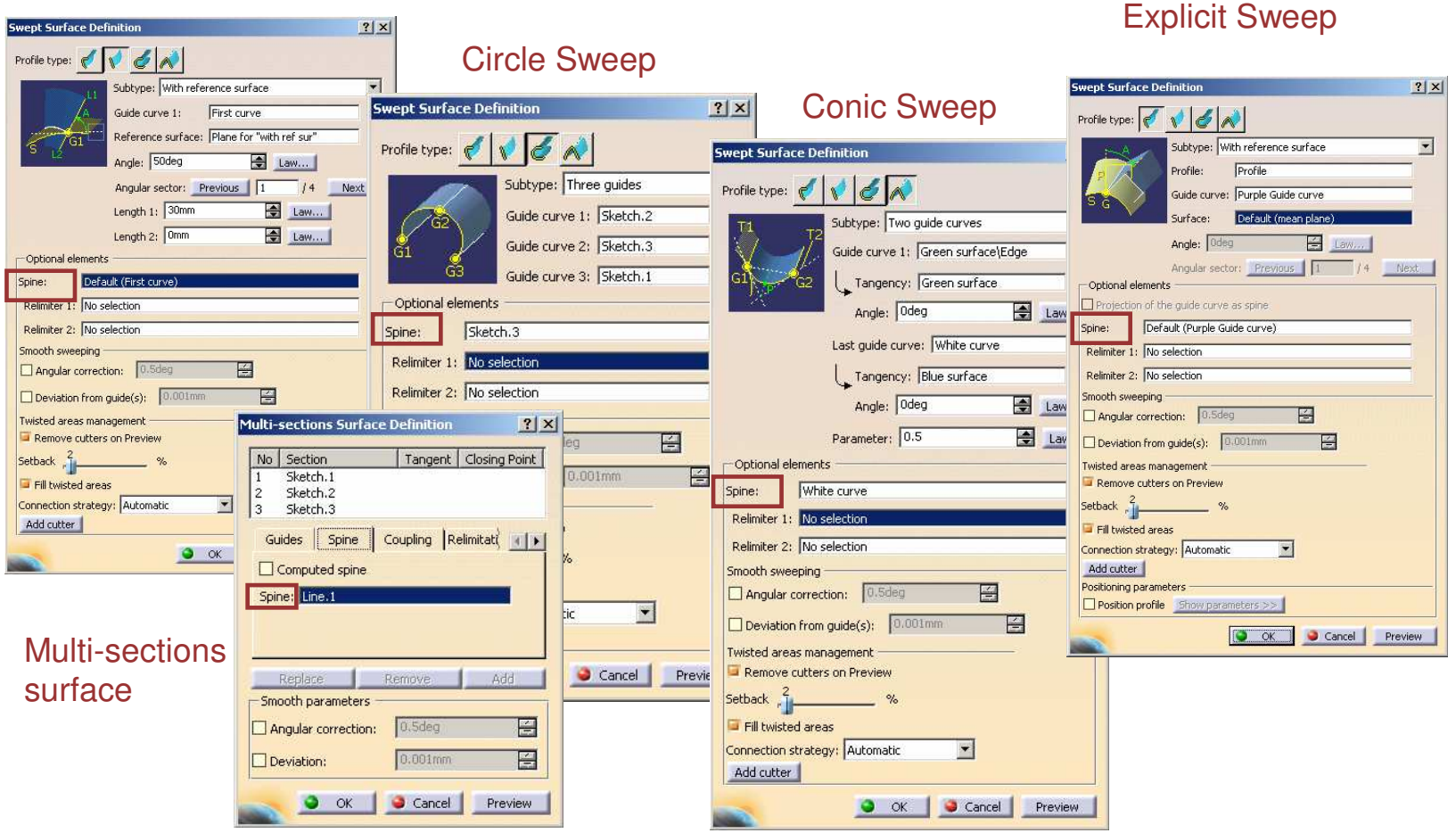
**Spines are used to create the following types of surfaces:**

Line Sweep

Circle Sweep

Conic Sweep

Explicit Sweep




Multi-sections surface

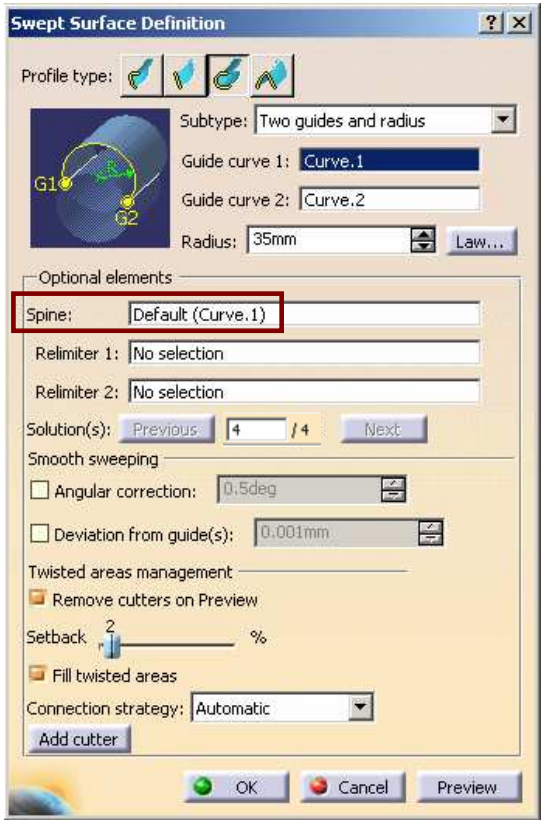
**Student Notes:**

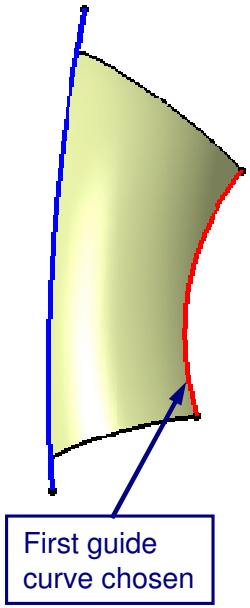
Student Notes:

# Default Spines

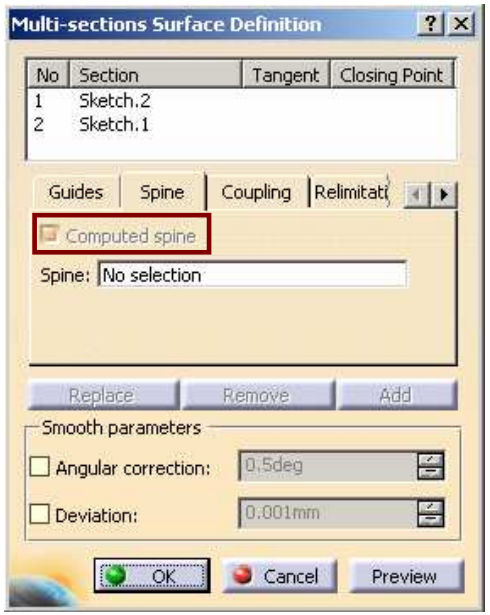
**It is not necessary to explicitly specify a Spine for all of the types of surfaces that require a spine.**

**For Swept surfaces, the first guide curve chosen will be the Spine if nothing else is specified in the Spine field.**



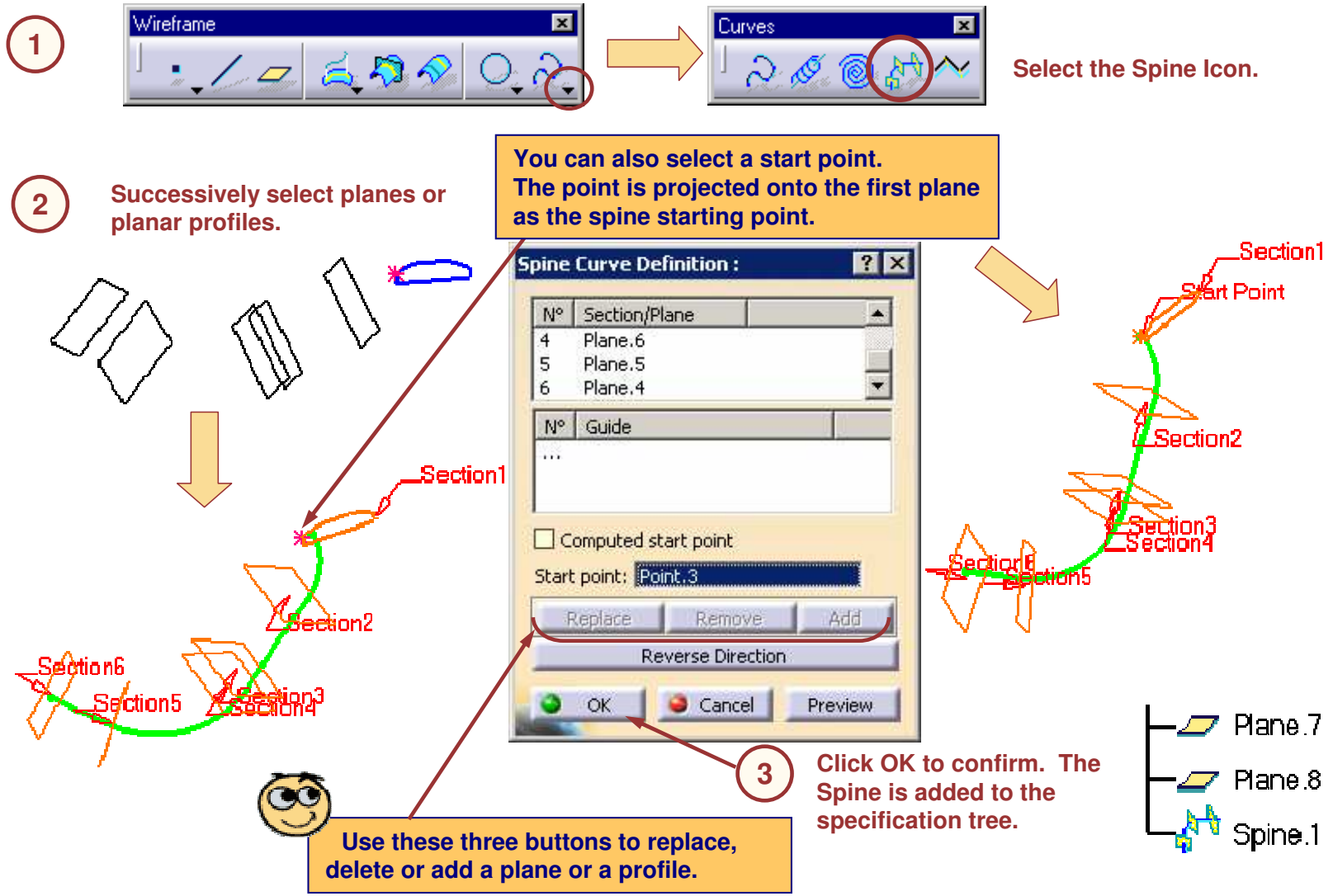




**For Multi-sections surface, if a Spine is not specified, it will be computed for you.**



Student Notes:

# Creating a Spine from Planes and Planar Curves



Student Notes:

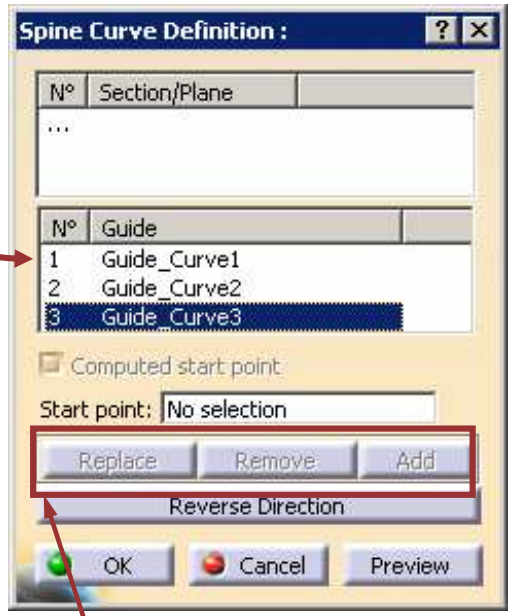
# Creating a Spine from Guide Curves

1



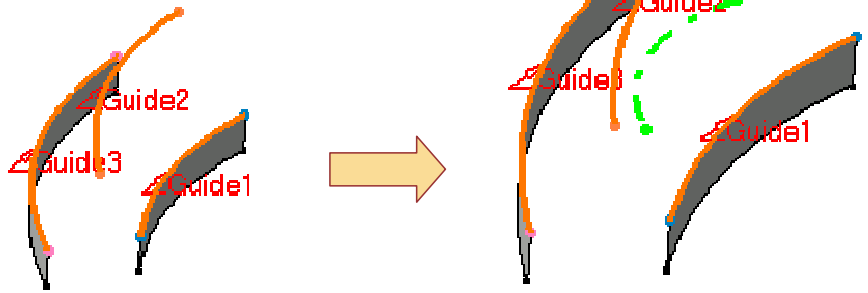
Select the Spine Icon.

2 Click in the field Guide

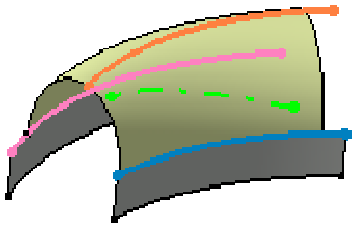



Use these three buttons to replace, delete or add a plane or a profile.

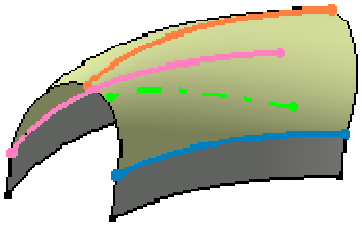
3 Select the Guide Curves




4 Click OK to confirm. The Spine is added to the specification tree.



Sweep using the default spine (guide curve 1)



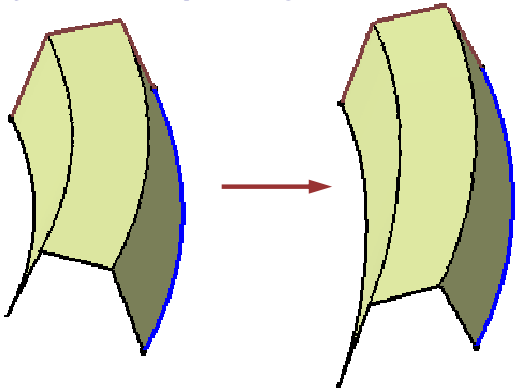
Sweep using the user created spine

Student Notes:

# Do It Yourself

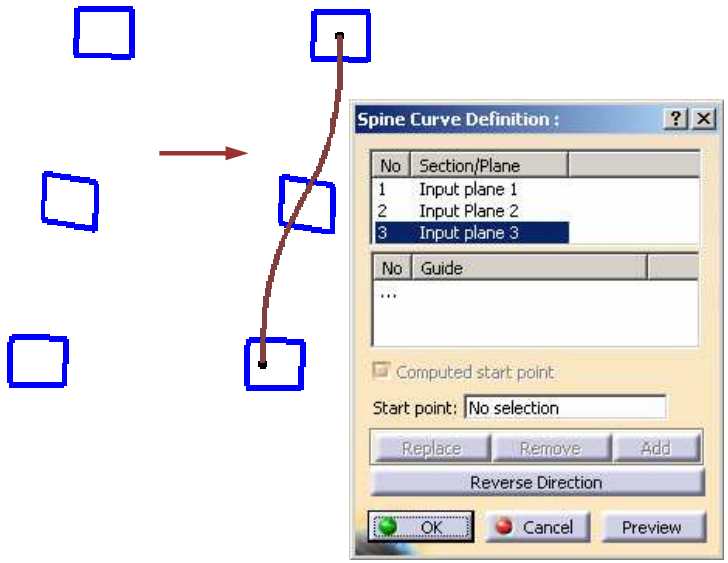
Part used: CATGSD_SpineDoIt_UseSpine.CATPart

- Modify the given Sweep and Multi-section surface by using a new spine.
  - Create the spine with the three planes provided.
  - Modify the Sweep using the new created spine

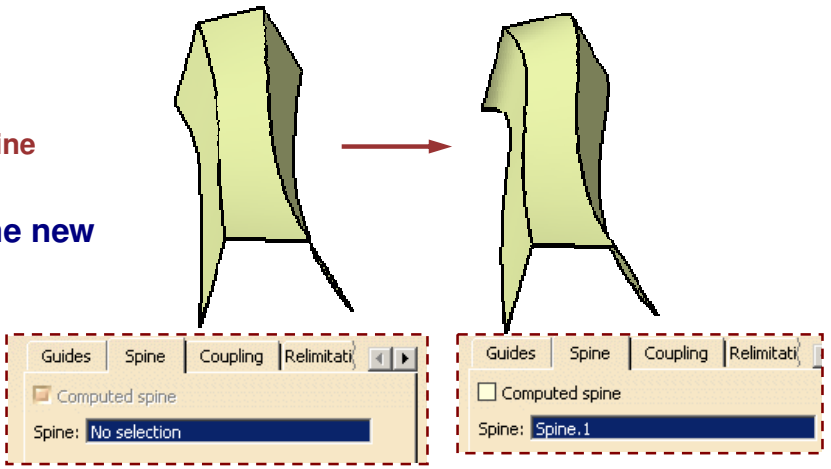


Sweep with default spine

Sweep with new spine



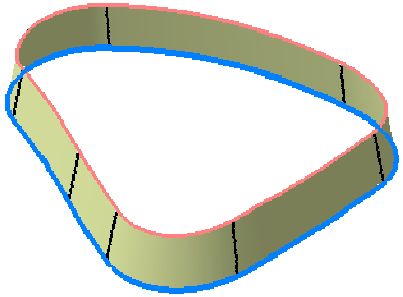

- Modify the Multi-section surface with the new created spine.



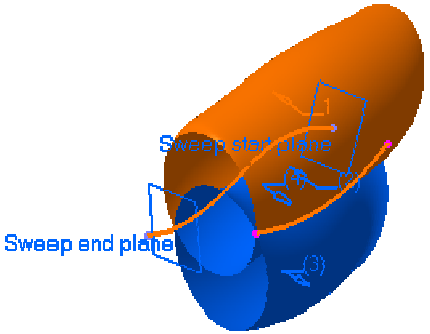
Student Notes:

# Creating a Swept Surfaces

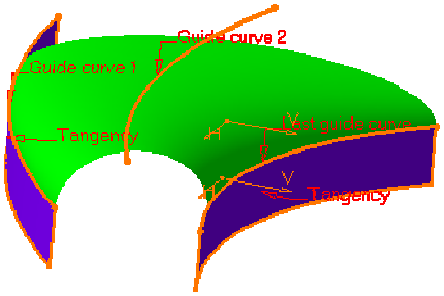
*You will learn about creating different types of swept surfaces*



Line Sweep




Circle Sweep

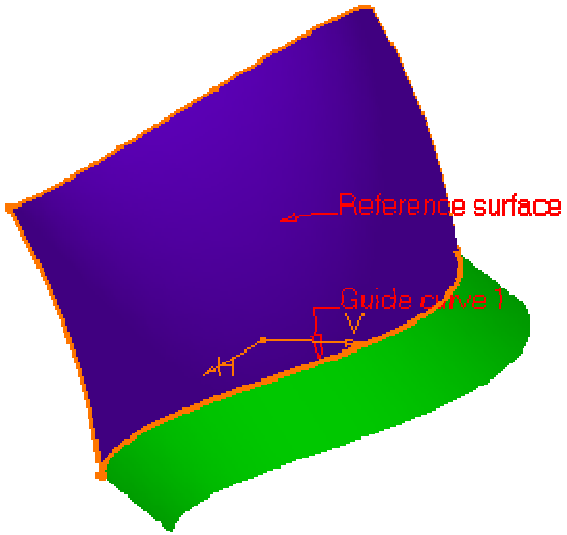


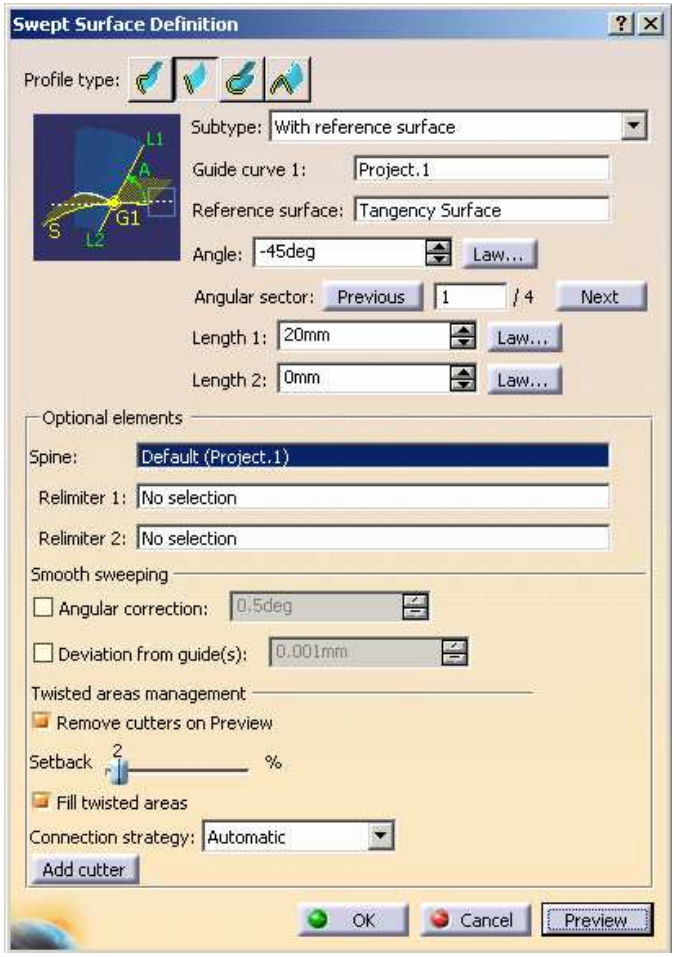

Conic Sweep

Student Notes:

# Creating Line Type Swept Surfaces

*You will learn how to create swept surfaces using Linear Profiles*



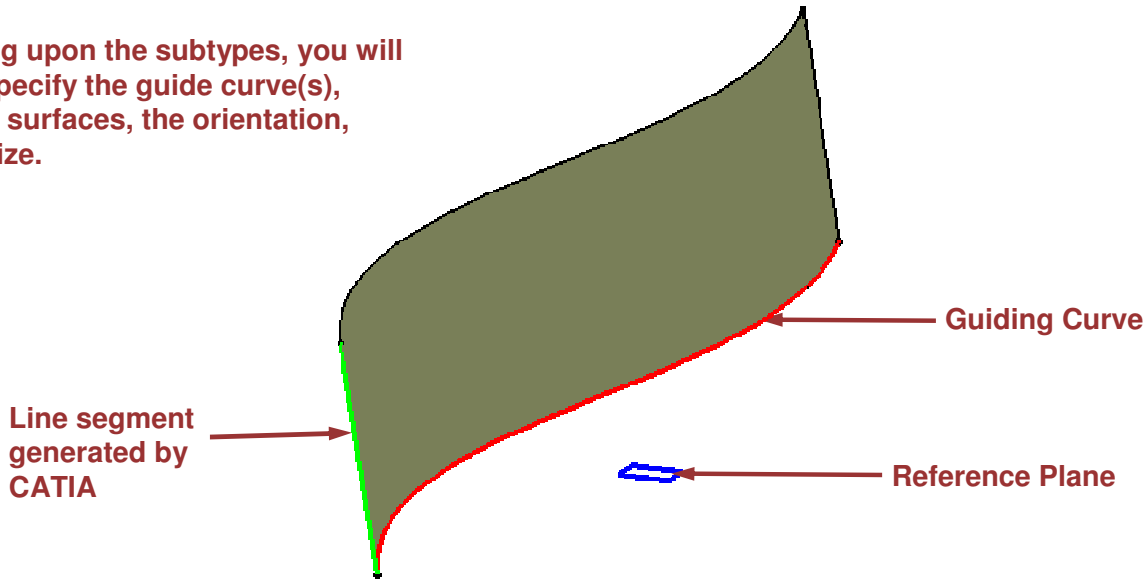



Student Notes:

# What is a Line Sweep?

**A particular surface which is created by sweeping a generated line segment along one or two guide curves**

**Depending upon the subtypes, you will have to specify the guide curve(s), reference surfaces, the orientation, and the size.**



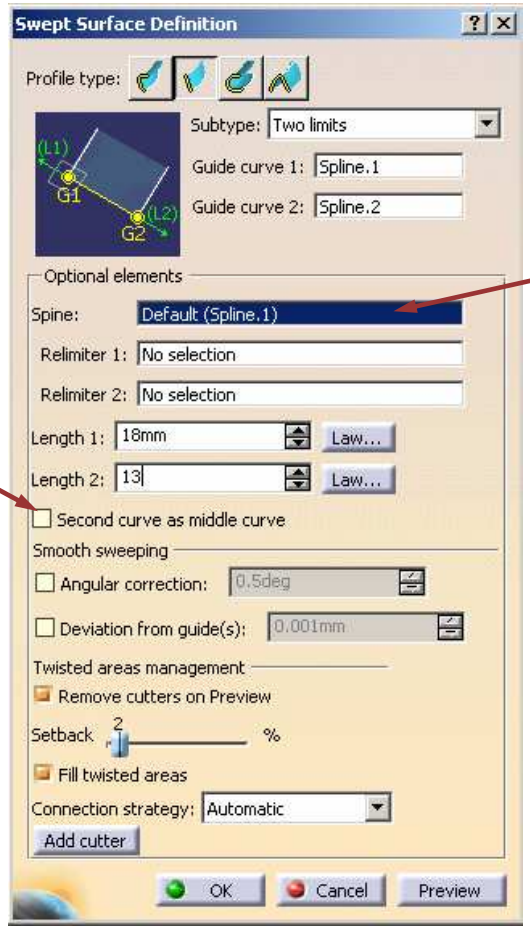
Student Notes:

# Creating a Line-type Swept Surface : Two Limits

1

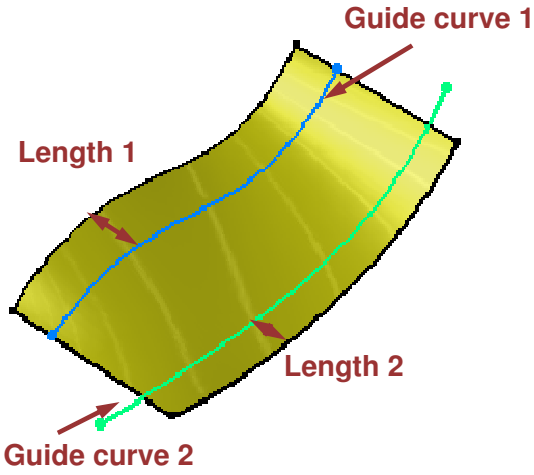

Line type :  Subtype : Two limits

2 Click on the Line icon, then select the Two limits subtype and the two guide curves.




You can select the second guide curve as middle curve instead of entering length values (same as Limit and middle subtype)

If no spine is selected the first guide curve is used as spine.




3 Confirm surface creation

Student Notes:

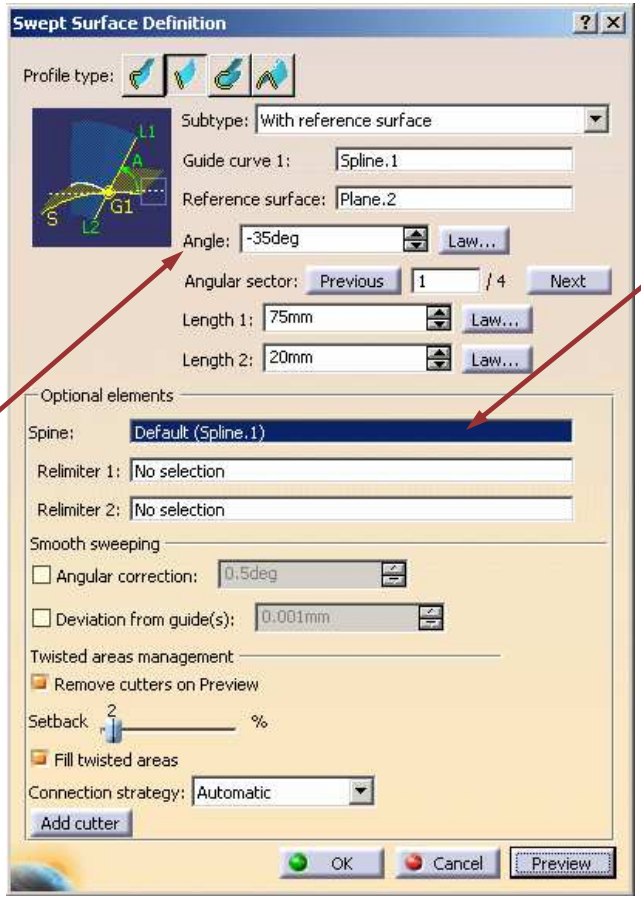
# Creating a Line-type Swept Surface : Reference Surface

1


Line type : 

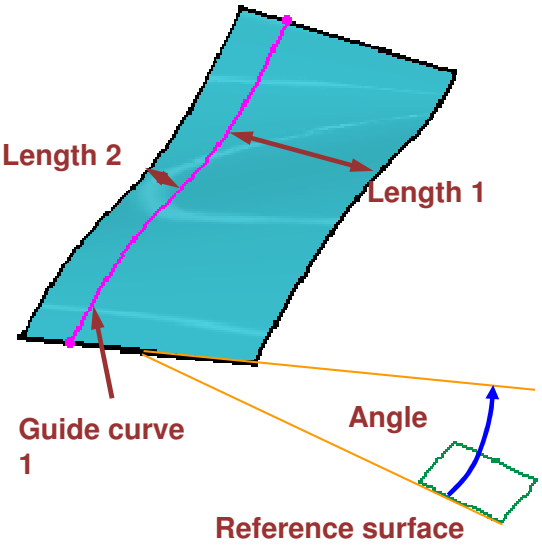
Subtype : With reference surface

2 Click on the Line icon, then select the With reference surface subtype, the guide curve and the reference surface. Key in an angle value and define the length of the surface.




Angle between the sweep and the reference surface.

If no spine is selected the first guide curve is used as spine.




3 Confirm surface creation

Student Notes:

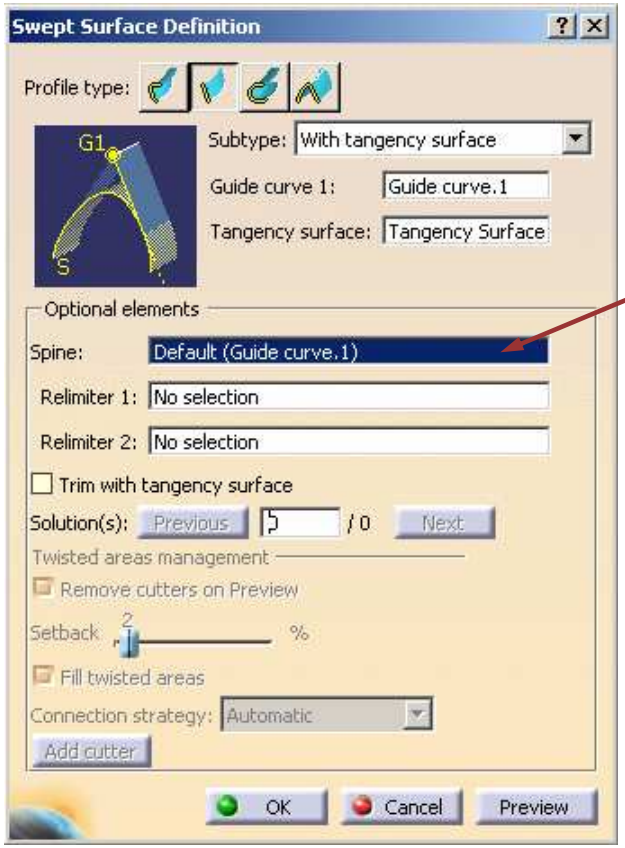
# Creating a Line-type Swept Surface : Tangency Surface

1

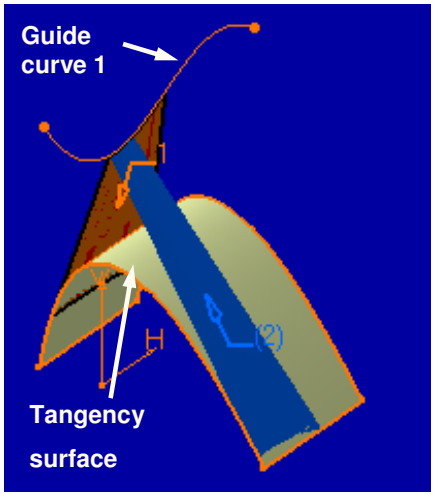

Line type : 

Subtype : With tangency surface

2 Click on the Line icon, then select the With tangent surface subtype, the guide curve and the tangency surface.




If no spine is selected the first guide curve is used as spine.



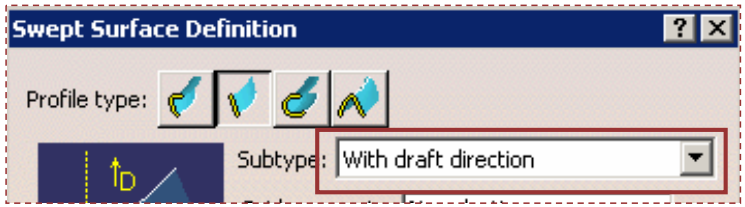

3 Confirm surface creation

# Creating a Line Type Swept Surface : Draft Sweep (1/3)

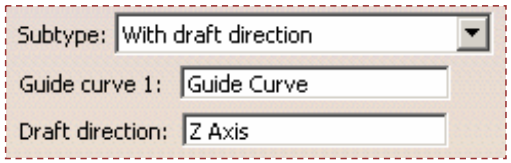
Student Notes:

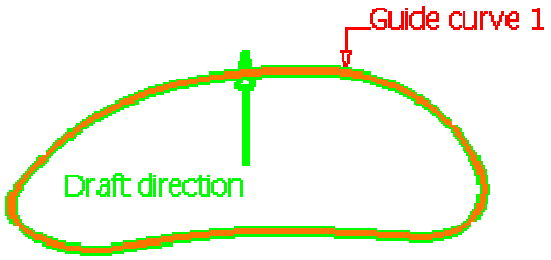
1


2 Select the sweep with draft direction as sweep type :




3 Select a guide curve and a draft direction :



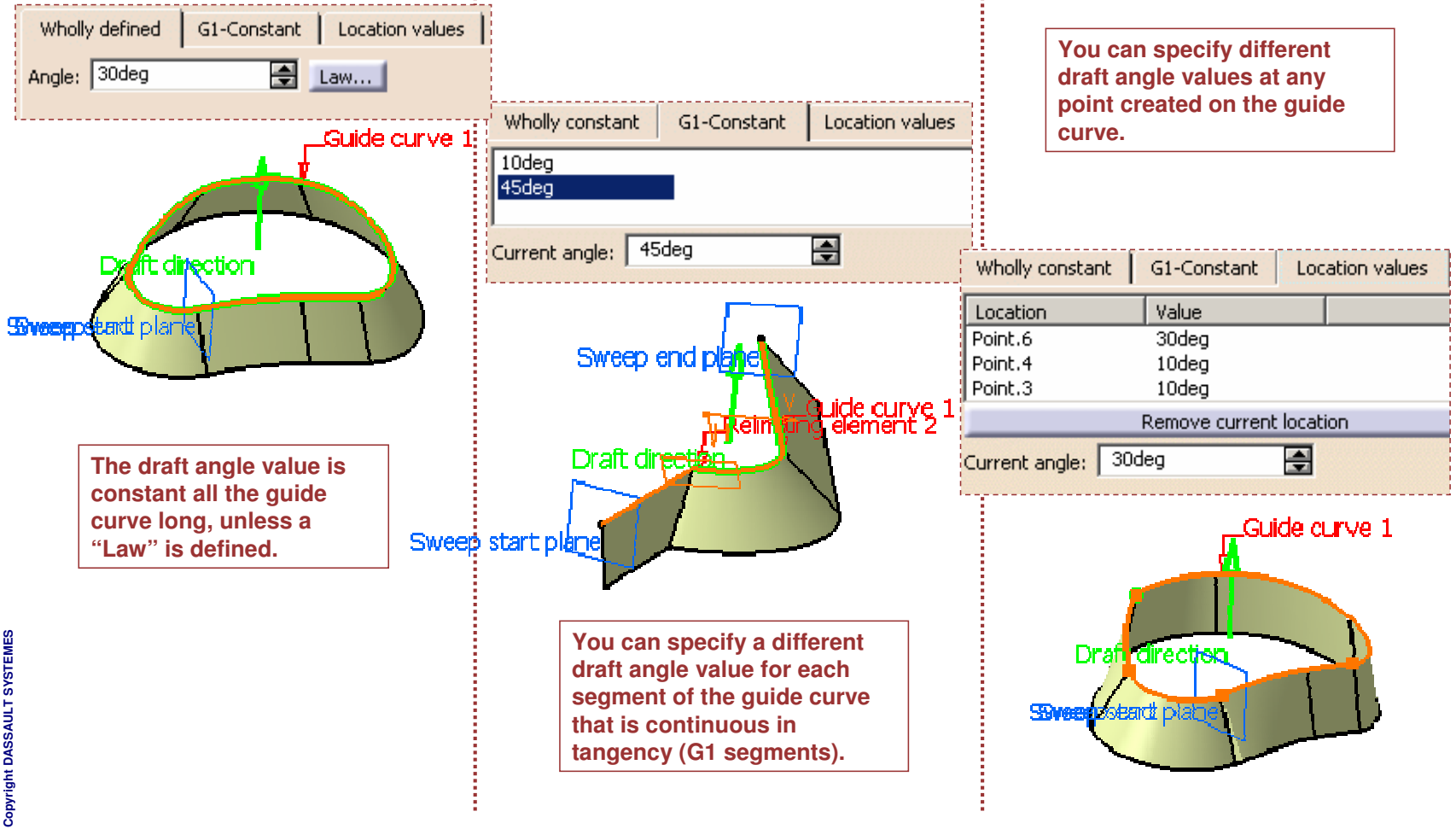




4 Choose a draft computation mode :

Student Notes:

# Creating a Line Type Swept Surface : Draft Sweep (2/3)

5 Select the way you want the draft angle to evolve :




The draft angle value is constant all the guide curve long, unless a "Law" is defined.

You can specify a different draft angle value for each segment of the guide curve that is continuous in tangency (G1 segments).

You can specify different draft angle values at any point created on the guide curve.

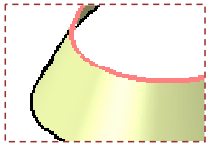
Student Notes:

# Creating a Line Type Swept Surface : Draft Sweep (3/3)

6 Select a relimitation mode for each side of the swept surface :

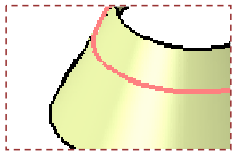
**Implicit 0 value :**

The surface limitation is the guide curve.



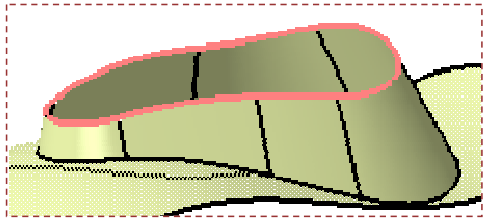
**Length :**

The surface limitation is defined by a length between the guide curve and the surface edge.



**Up to plane or surface :**

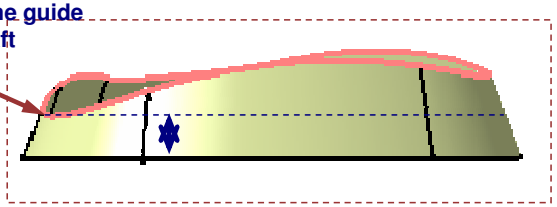
The sweep is relimited by a surface or a plane.

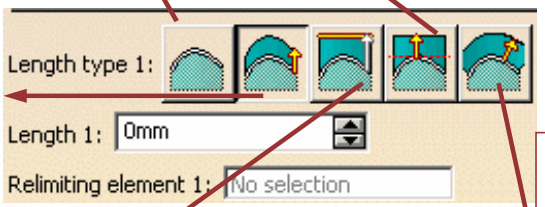


**Length starting from guide curve extremum :**

The surface length is calculated from the guide curve extremum in the draft direction.

Extremum on the guide curve in the draft direction

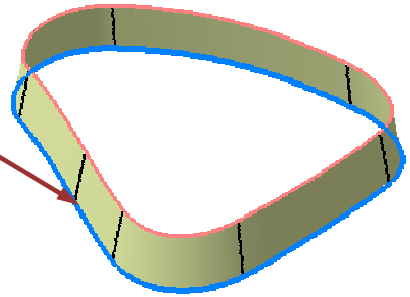





**Length along Surface:**

The length is used as an Euclidean parallel curve distance to relimit the swept surface .

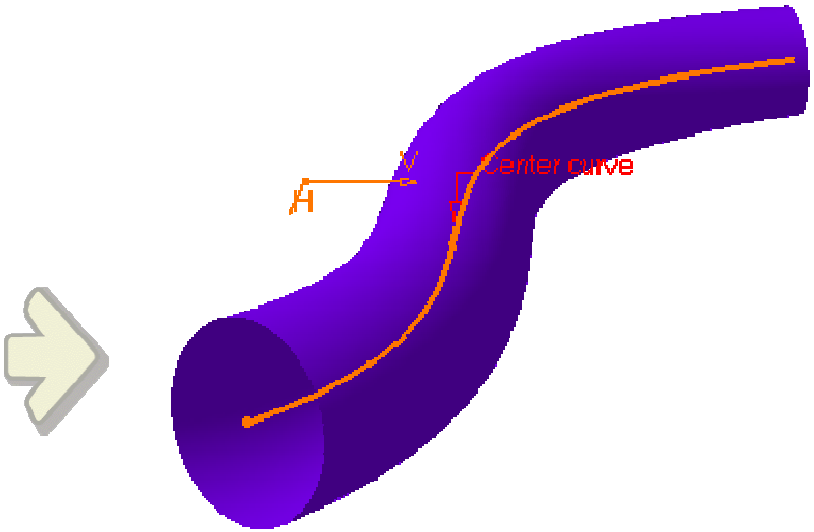
Euclidean parallel curve

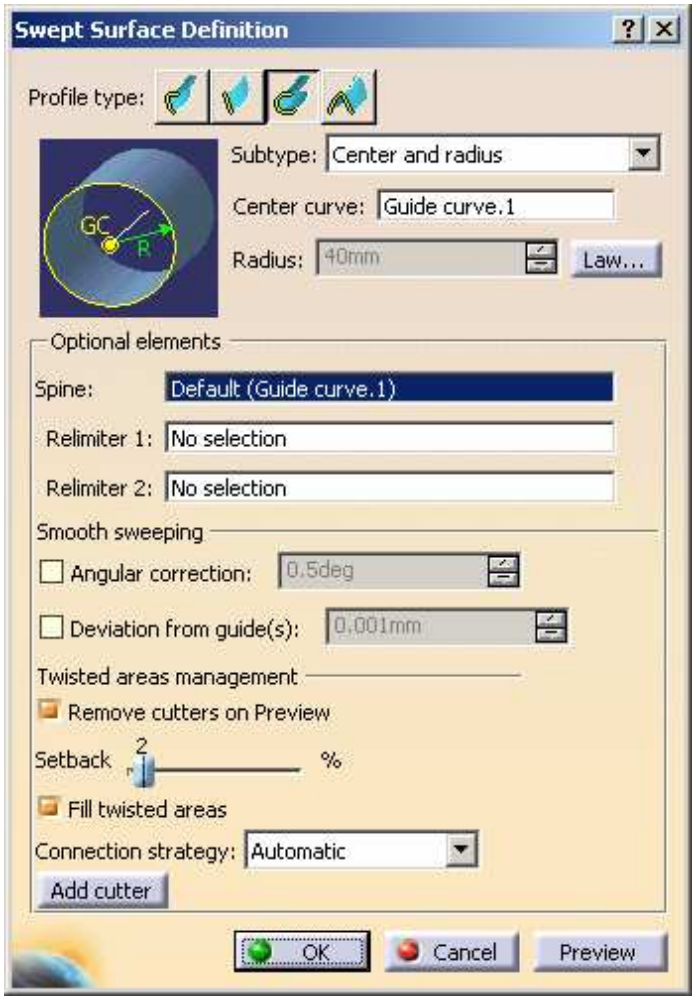


Student Notes:

# Creating Circle Type Swept Surfaces

*You will learn how to create swept surfaces using Circular Profiles*

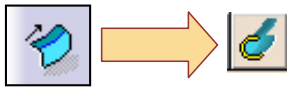


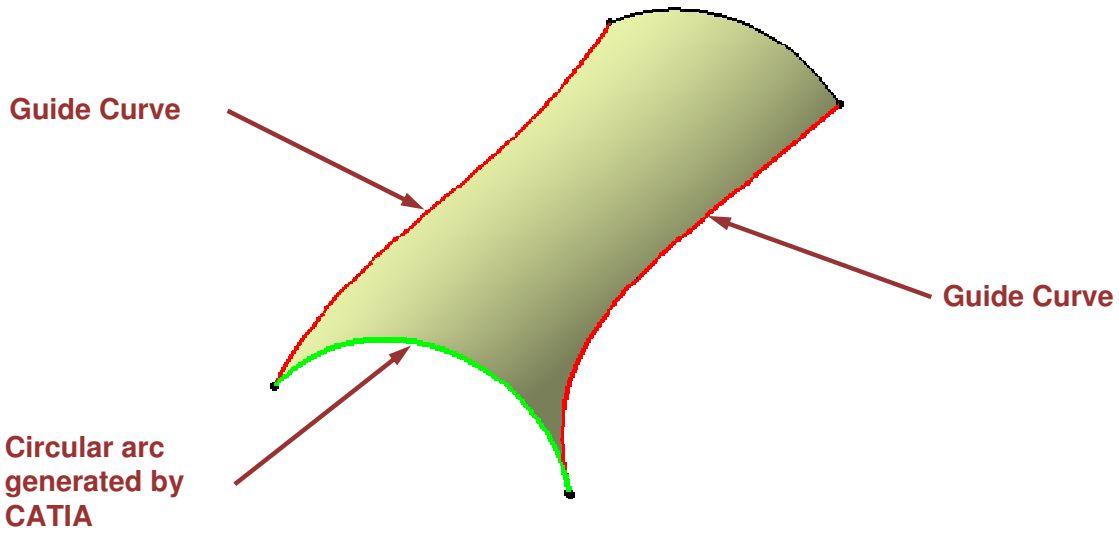



Student Notes:

# What is a Circle Sweep?

**A particular surface which is created by sweeping a generated circular arc along several guiding conditions.**





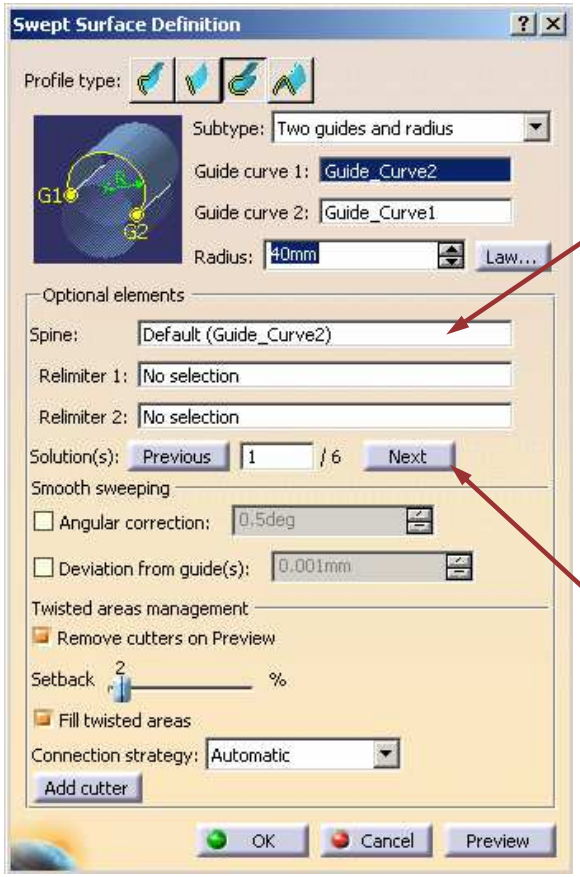
Student Notes:

# Creating a Circle-type Swept Surface : Two Guides and Radius

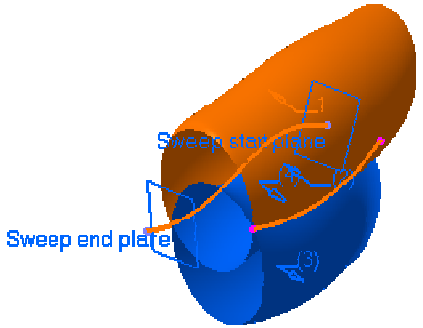
1

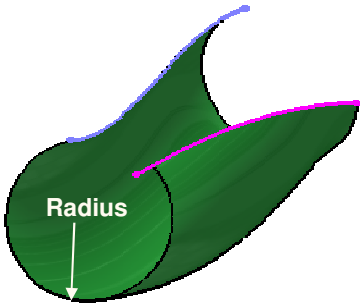

Circle type :  Subtype : Two guides and radius

2 Click on the Circle icon, then select the Two guides and radius subtype, the two guide curves and the radius.




If no spine is selected the first guide curve is used as spine.




In case of several solutions you can check them all and then select one of them (Highlighted color = active solution)

3 Confirm surface creation

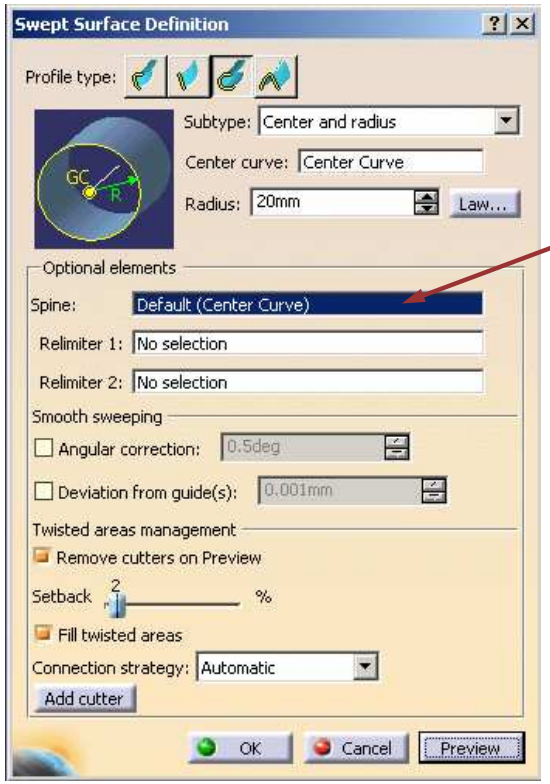
Student Notes:

# Creating a Circle-type Swept Surface : Center and Radius

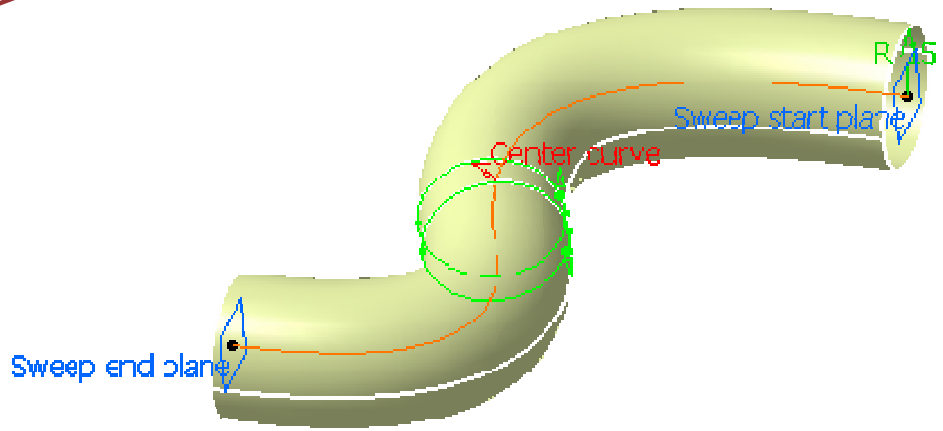
1


Circle type :  Subtype : Center and radius

2 Click on the Circle icon, then select the Center and radius subtype, a center curve and a radius.




If no spine is selected the center curve is used as spine.



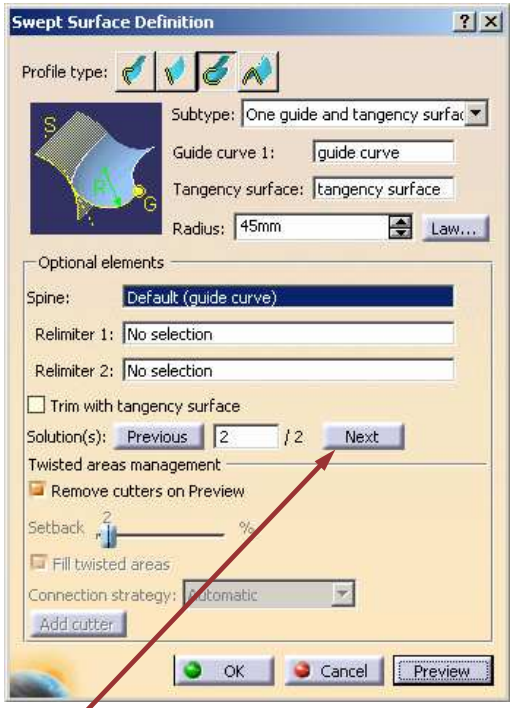

3 Confirm surface creation

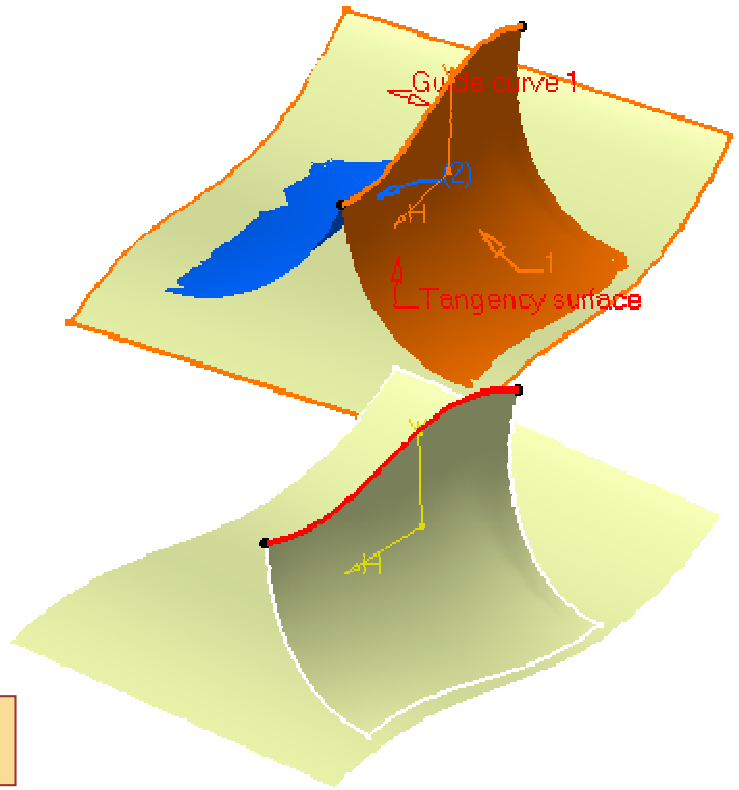
# Creating a Circle-type Swept Surface : One Guide and Tangency Surface

1


Circle type :  Subtype : One Guide and Tangency Surface

2 Click on the Circle icon, then select the one guide and tangency surface as subtype. Select the guide curve, the tangency surface, and key in a radius sufficient to link the guide curve and the tangency surface.



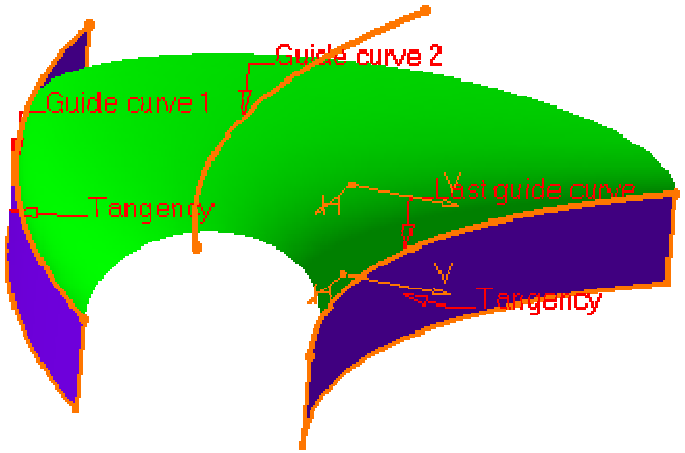

In case of several solutions you can check them all and then select one of them (orange color = active solution)

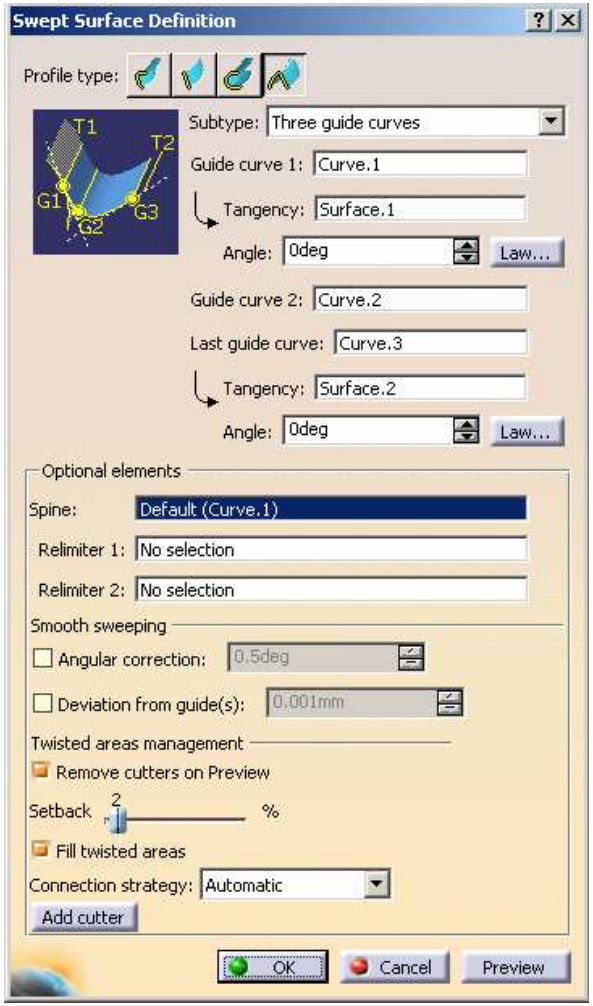
Student Notes:

Student Notes:

# Creating Conic Type Swept Surfaces

*You will learn how to create swept surfaces using Conical Profiles*



Student Notes:

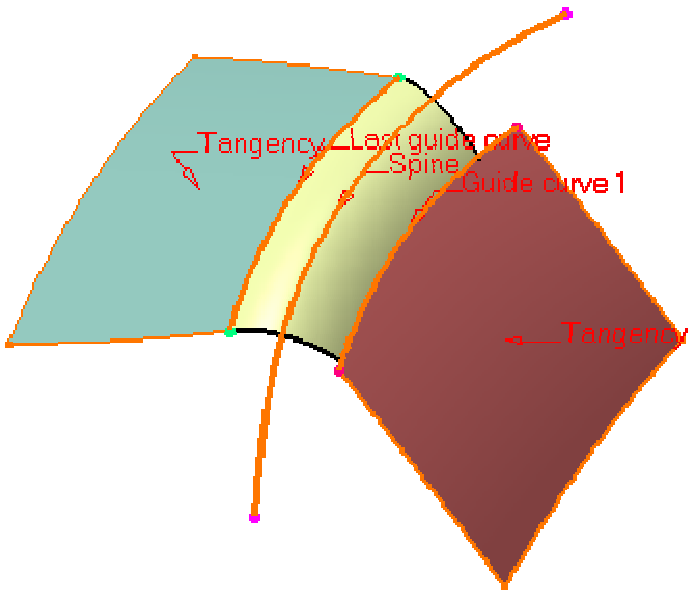
# Creating a Conic Type Swept Surface : Two Guide Curves

1 

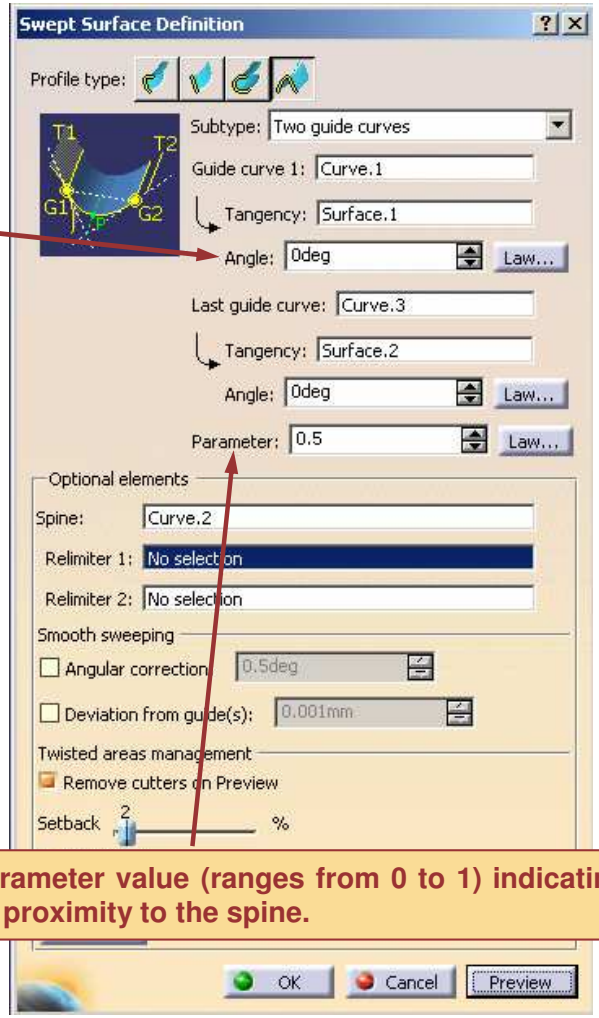

Conical type :  Subtype : Two Guide curves

2 Click on the Conic icon, then select Two guide curves and their tangency supports.

Define an angle between the swept surface and the tangency surface




3 Confirm surface creation




Set the parameter value (ranges from 0 to 1) indicating the sweep proximity to the spine.

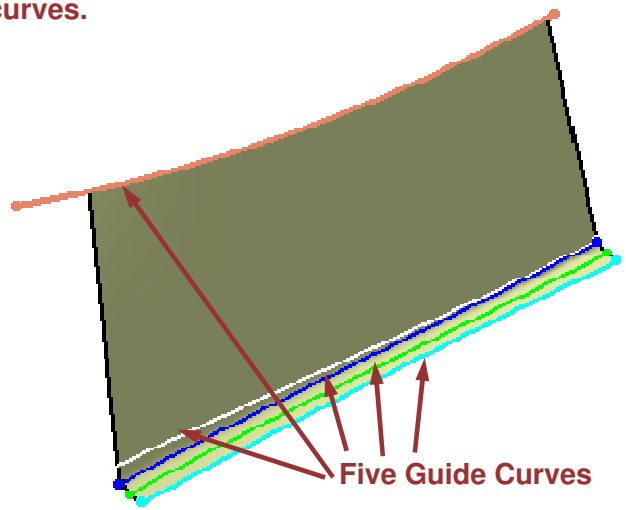
Student Notes:

# Creating a Conical-type Swept Surface : Five Guide Curves

1


Conical type : Subtype : Five Guide curves


2 Click on the Conic icon, then select Five guide curves.




You can specify a Spine curve. The default spine is always the first guide curve.

3 Confirm surface creation

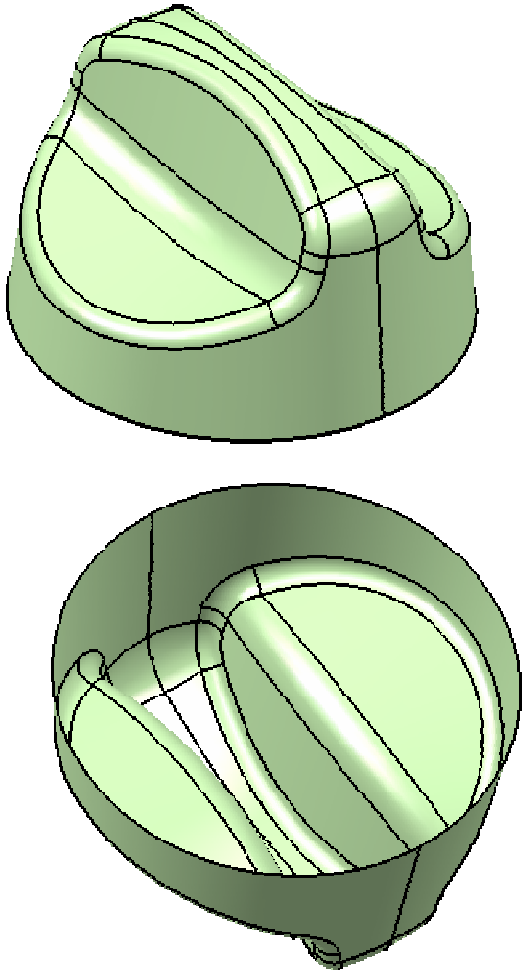
# Knob

## Recap Exercise: Swept Surface

40 min

In this exercise you will:

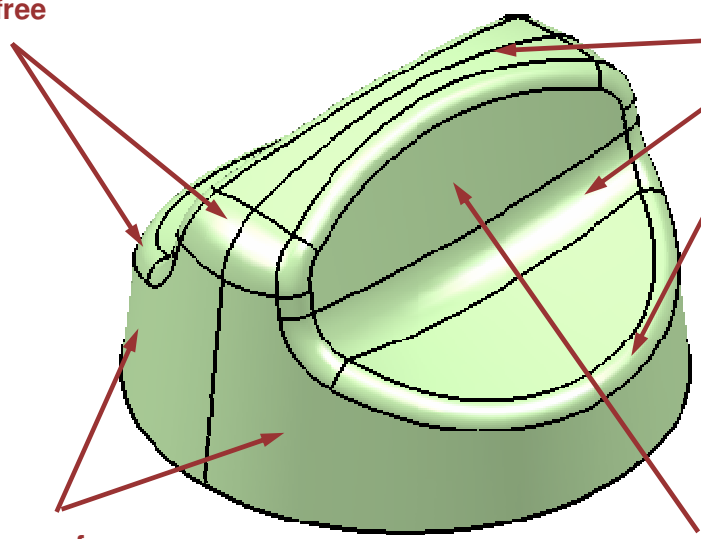
- Build a 'Knob' model using a given wireframe.
- Use Line and Circle Swept Surfaces to generating the 'Knob' shapes.
- Design a law and use it to define the shape of the fillets
- Finally, apply dress up features and complete the model



Student Notes:

Student Notes:

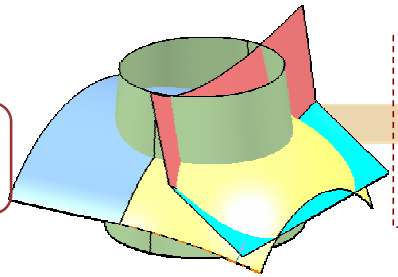
# Design Intent: Knob

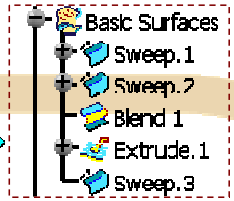


Student Notes:

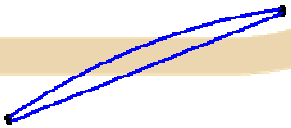
# Design process: Knob

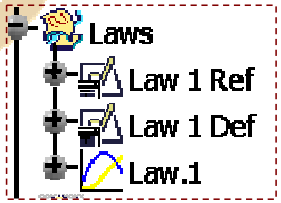
1 Design Basic Surfaces



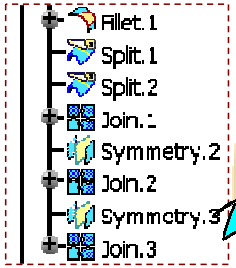


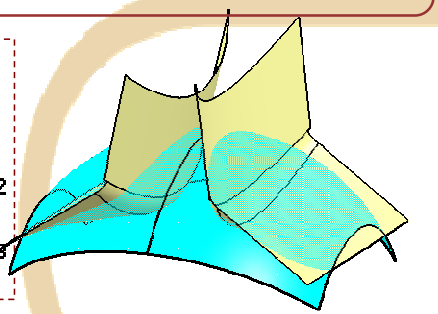

2 Create Laws

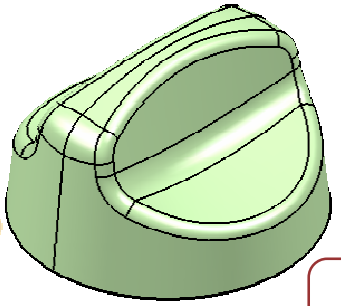





3 Design the Intermediate Surfaces





4 Design the Final Surfaces

Student Notes:

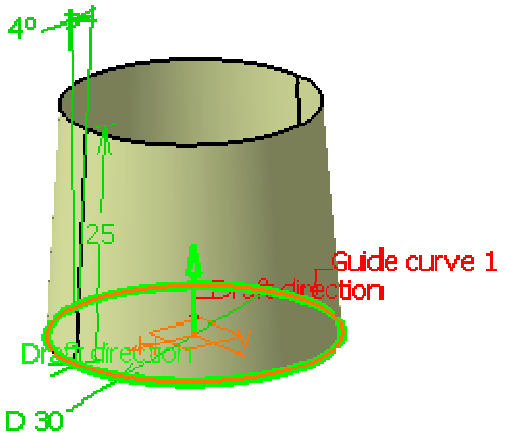
# Do It Yourself (1/11)

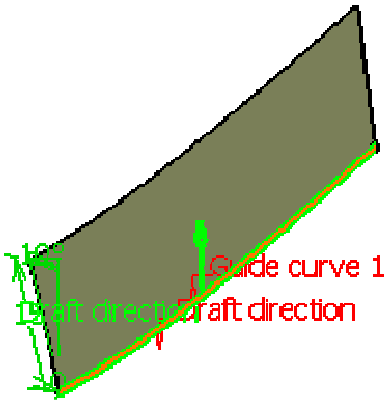
**Part Used: CATGSD_F_Knob_Start.Catpart**

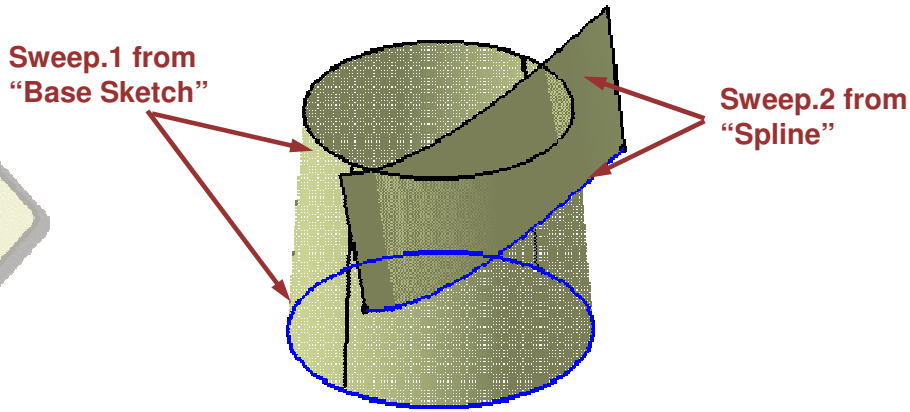
**Create the basic surface of the Knob using the given wireframe**

- **Create the first swept surface using the "Base sketch"**
  - **Sweep type "Line" & Subtype "with draft direction"**
  - **Use "Base sketch" as Guide Curve 1**
  - **Draft direction "XY Plane"**
  - **Draft angle "4 deg" and square wholly defined**
  - **Enter Length.2 as 25 mm**

- **Create the second swept surface using 'Spline.1' as a 'Guide curve.1' with the same specifications as the first sweep.**
  - **Draft angle 10deg**
  - **Enter Length.2 equal to 15 mm.**



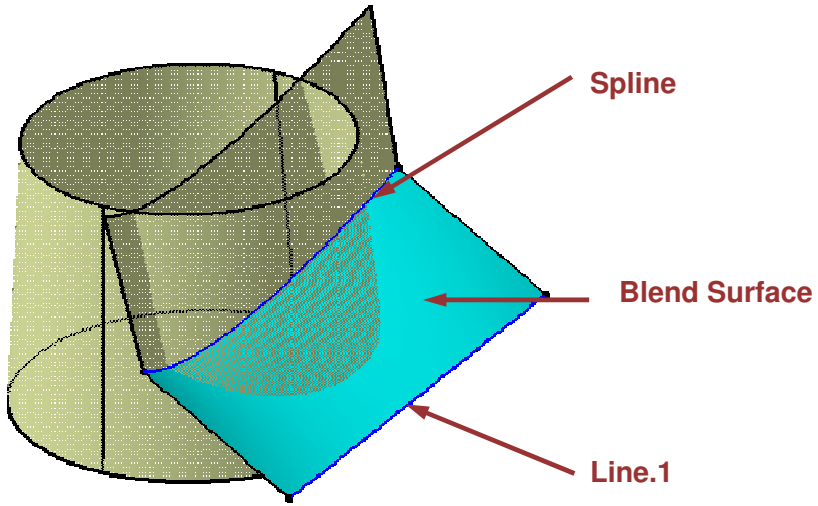



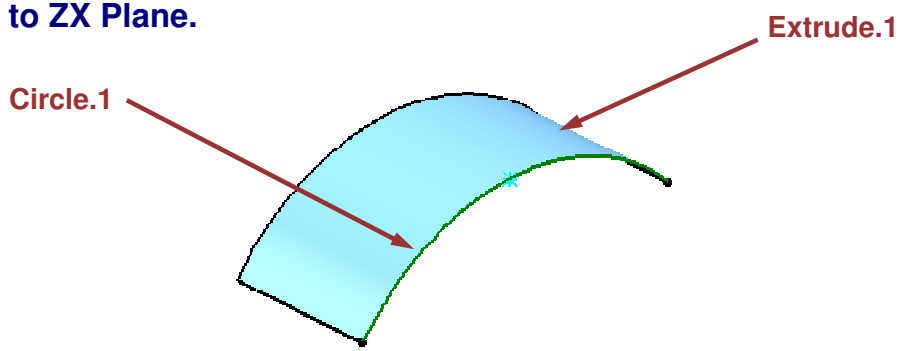
Student Notes:

# Do It Yourself (2/11)

- Create a Blend.1 surface between “Spline.1” and “Line.1”



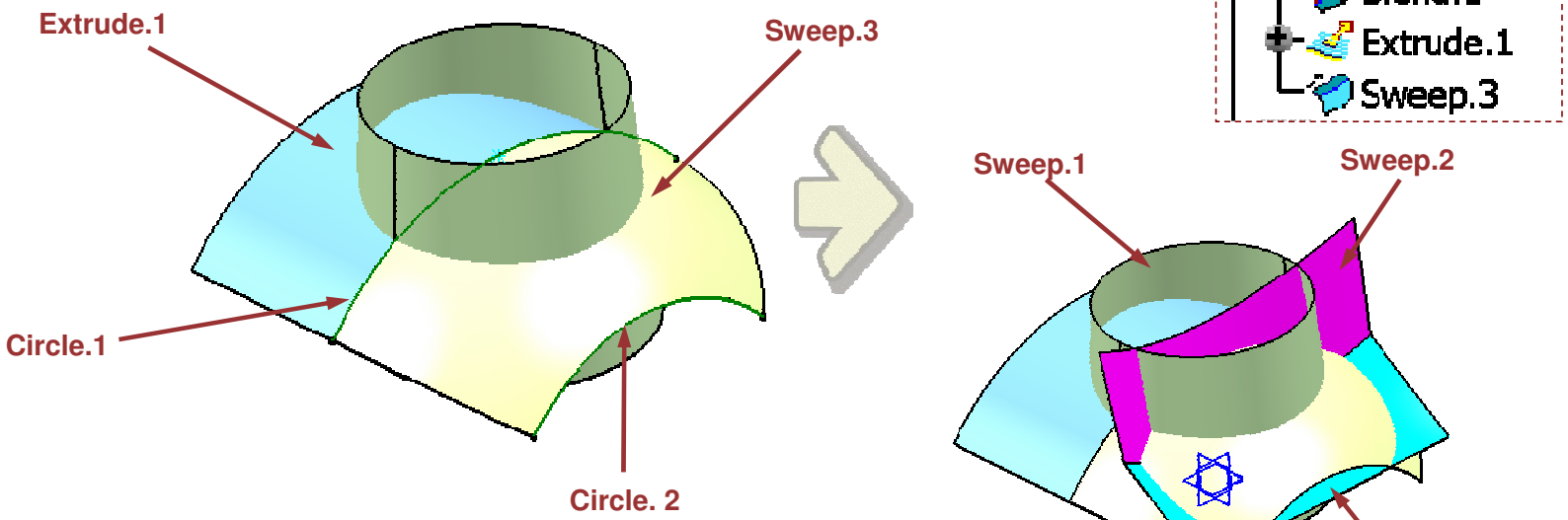

- Create an Extrude from “Circle.1”. This extrude will be used to create “Sweep.3”.
  - Extrude it by 20 mm normal to ZX Plane.



Student Notes:

# Do It Yourself (3/11)

- Create a “Sweep.3”. Use previously created extrude to create it.
  - Type: Circle. Sub type: Two guides and tangency surface
  - Limit curve with tangency: Circle.1
  - Tangency surface: Extrude.1
  - Limit curve: Circle.2



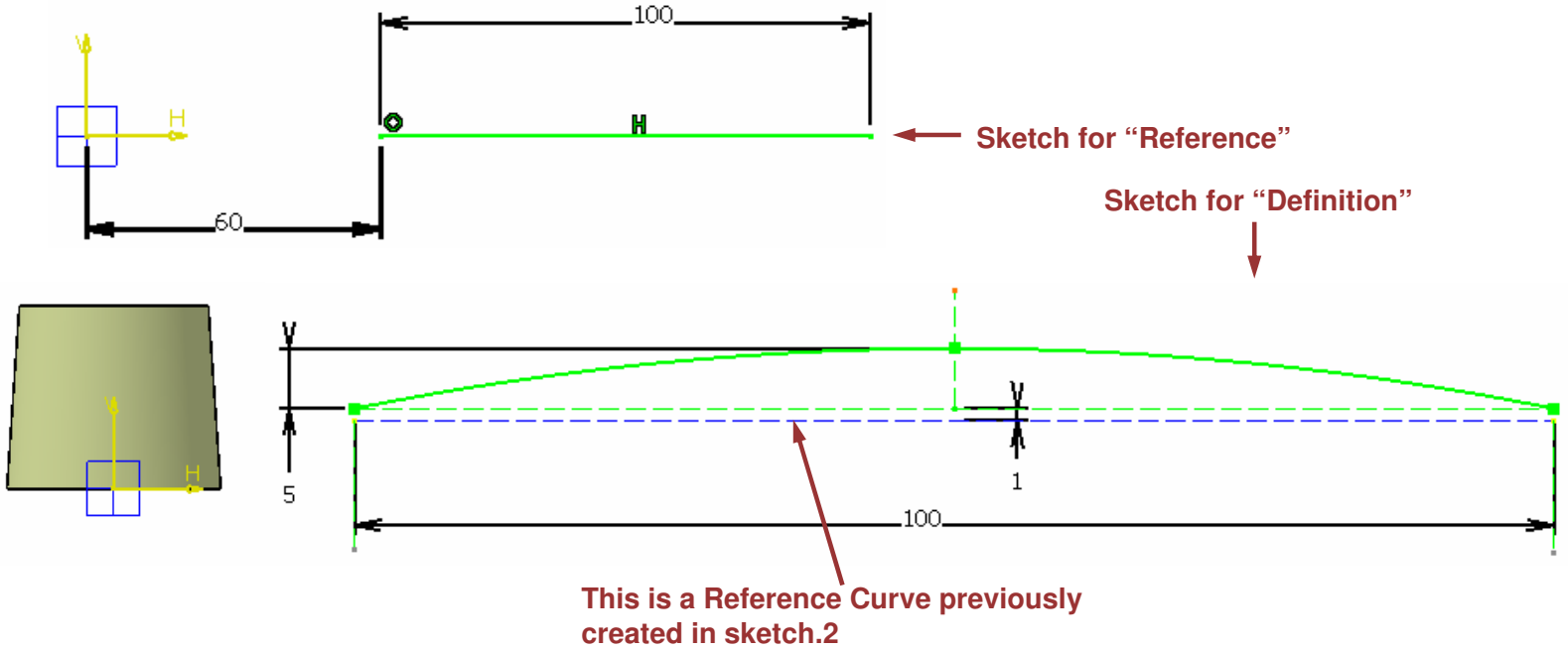

Contents of the “Basic Surfaces” Geometrical Set.

Student Notes:

# Do It Yourself (4/11)

**Create a law to define the shape of the fillet.**

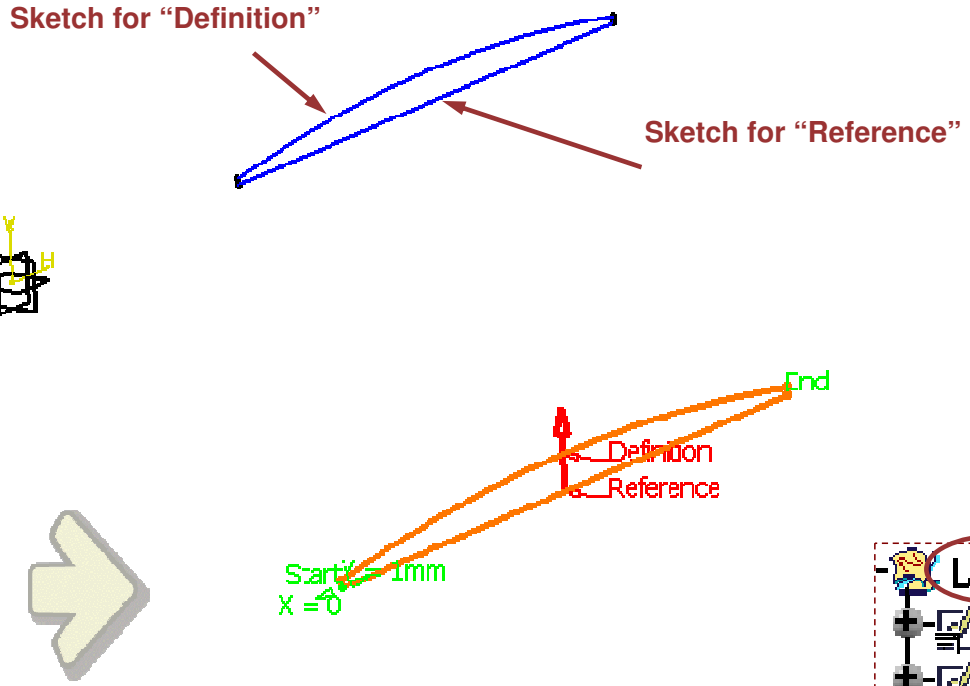
- To create a Law, you need to specify the “Reference” and “Definition” curves.
  - Make Geometrical set “Laws” Active
  - Sketch the “Reference” for the law in to Positioned Sketch.2 (Law.1Ref).
  - Sketch the “Definition” for the law in to Positioned Sketch.3 (Law.2 Def) as shown in the picture. Use following Inputs for both Sketches: Origin: Part Origin. Orientation: X axis, Reference:ZX plane.

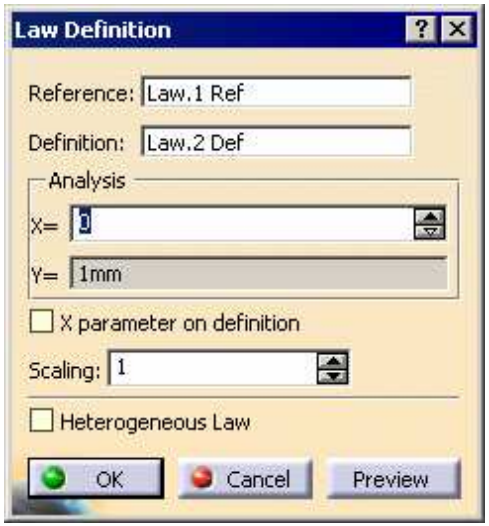


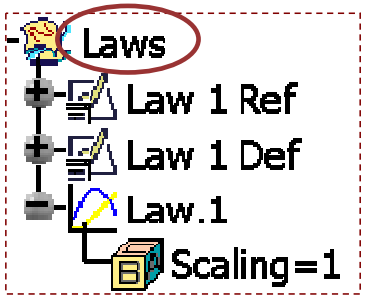
Student Notes:

# Do It Yourself (5/11)

- Access the law command to create the law.
- This Law will then be used to define the shape of the fillet



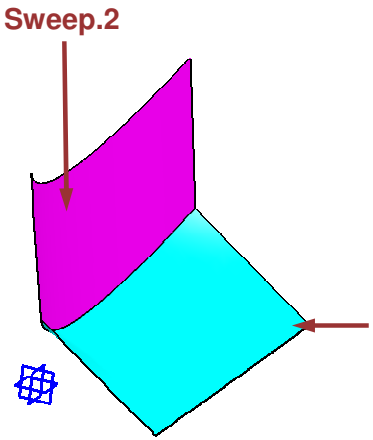



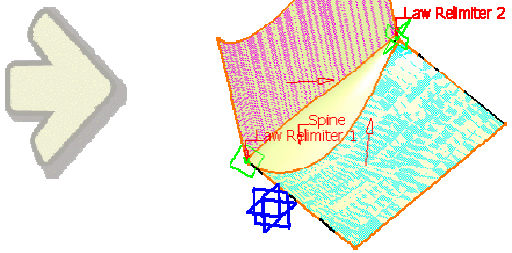
Student Notes:

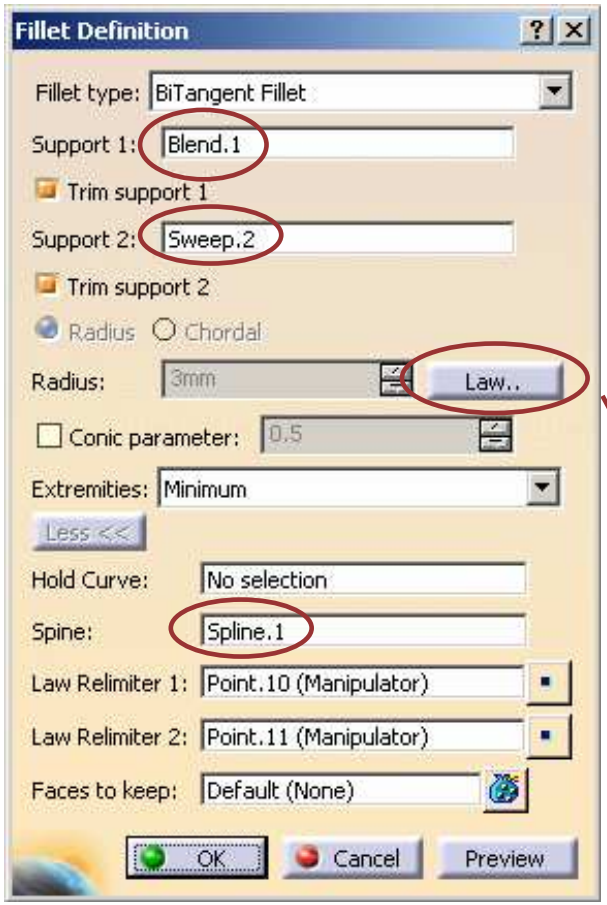
# Do It Yourself (6/11)

**Create Intermediate surfaces which will be used as inputs to create final Knob**
**(Work in Object "Intermediate surfaces")**

- **Create a Bitangent Shape fillet between Blend.1 and Sweep.2**



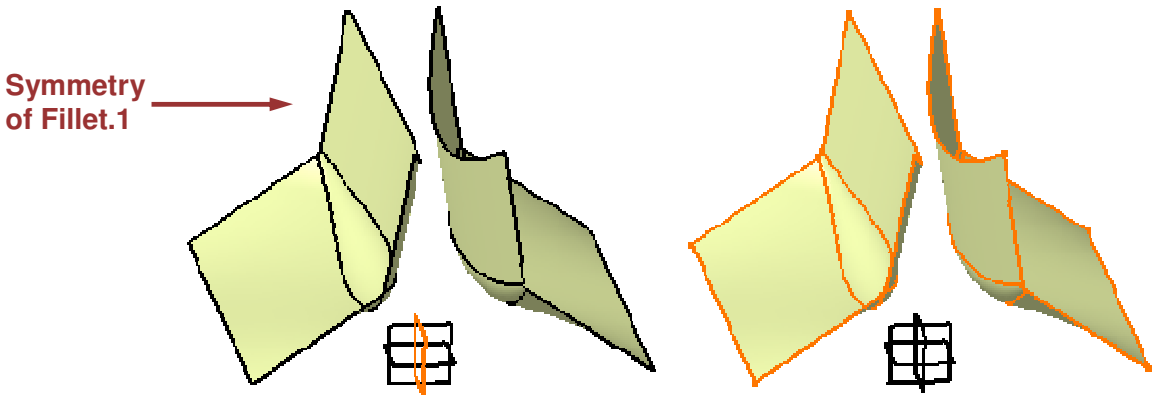



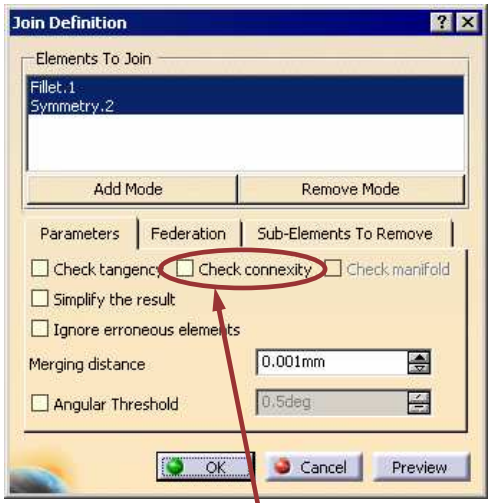

**Specify the law you created here**

Student Notes:

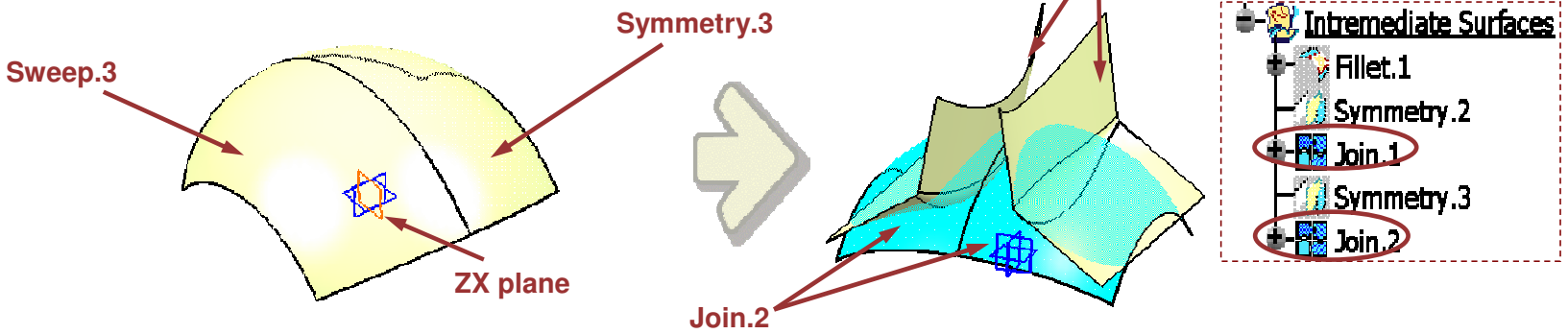
# Do It Yourself (7/11)

- Create a Symmetry.2 of this Fillet.1 about ZX Plane.
- Create a Join.1 between Fillet.1 and Symmetry.2




- Create a Symmetry(Symmetry.3) of Sweep.3 about ZX plane.
- Create a Join.2 between Sweep.3 & and Symmetry.3

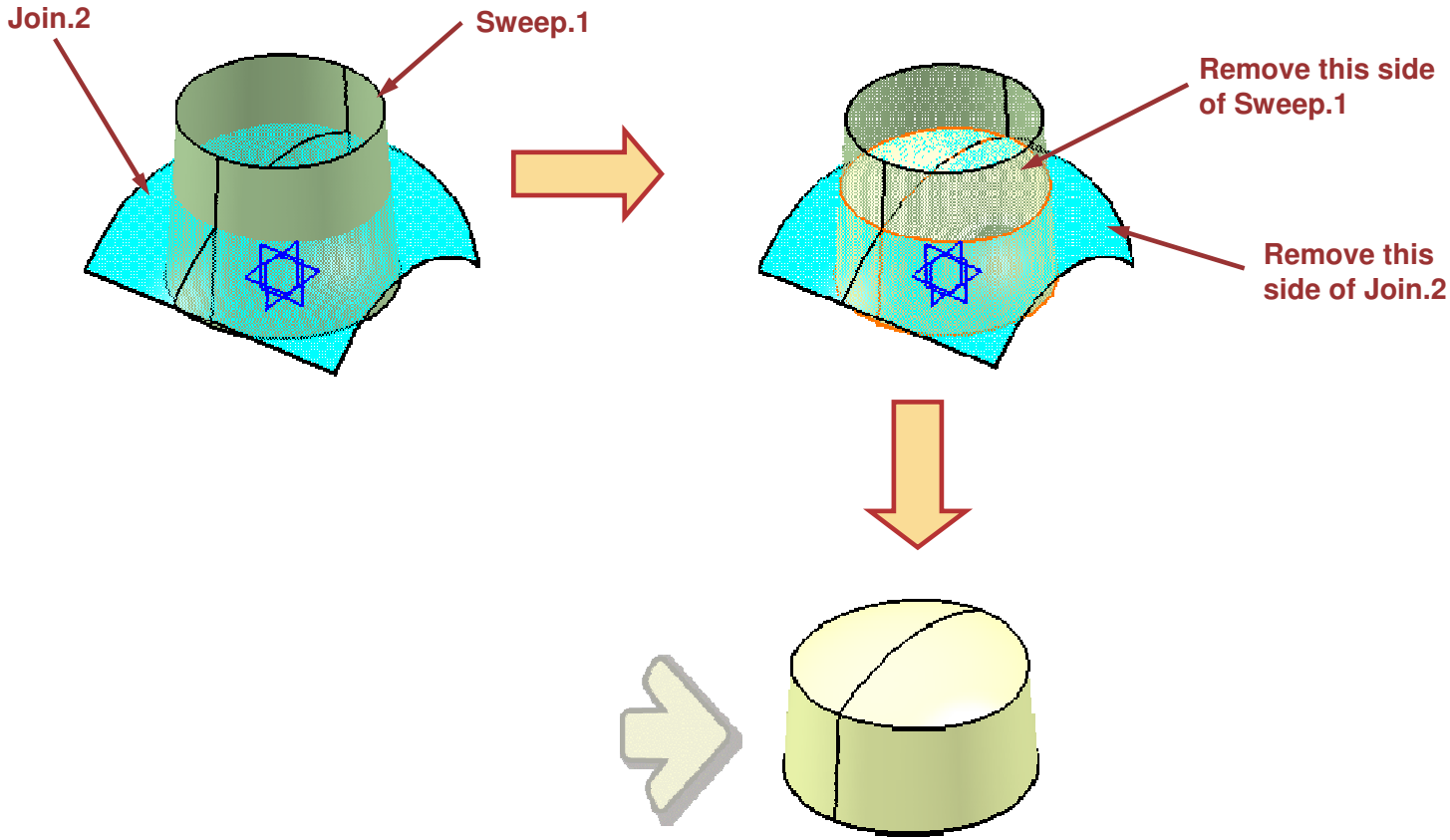


Student Notes:

# Do It Yourself (8/11)

**Create the final Knob using the Basic surfaces and the Intermediate surfaces.**
**(Work in Object "Final surfaces")**

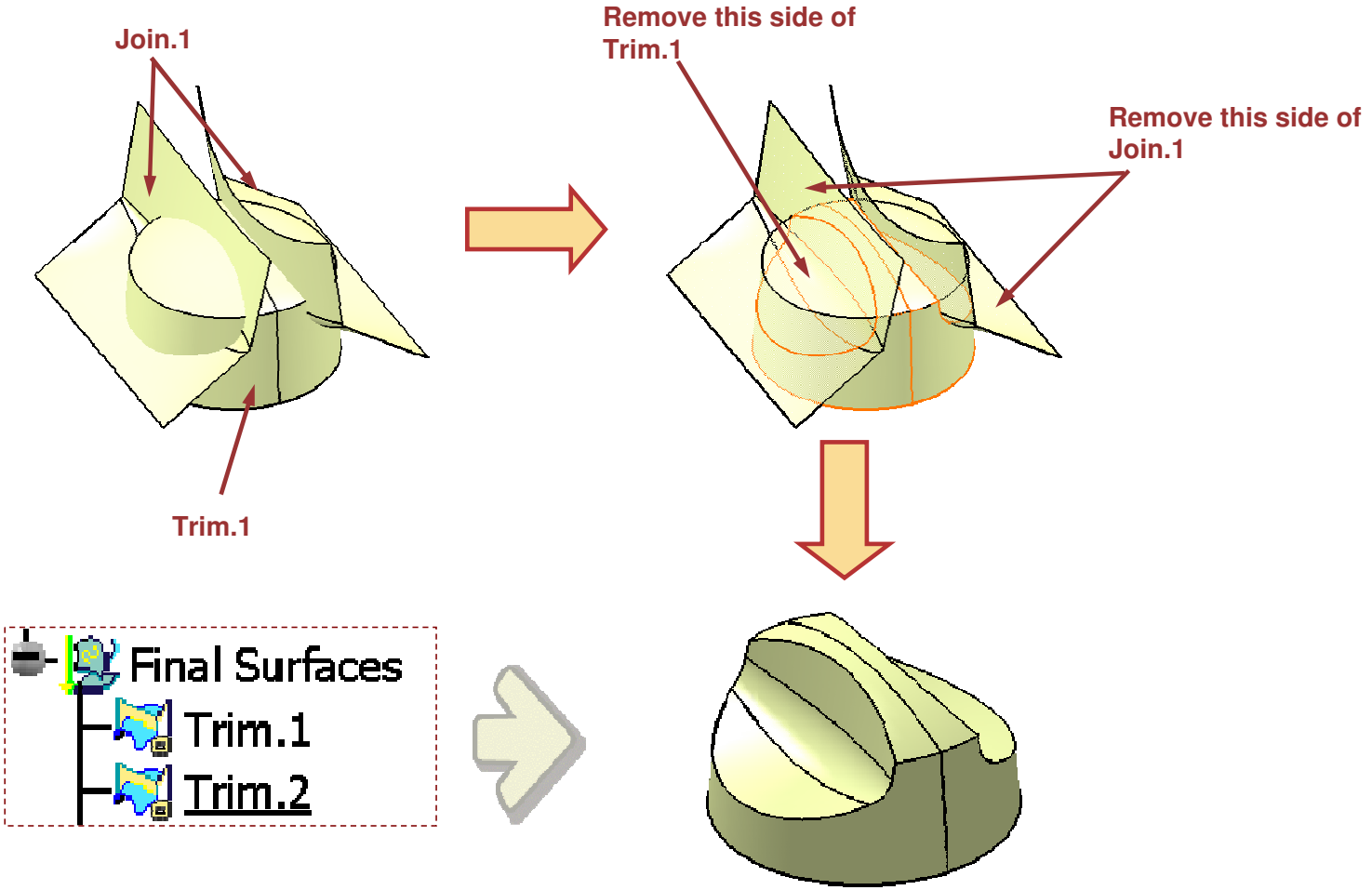
- **Trim "Sweep.1" from "Basic Surfaces" with "Join.2" from "Intermediate Surfaces".**



Student Notes:

# Do It Yourself (9/11)

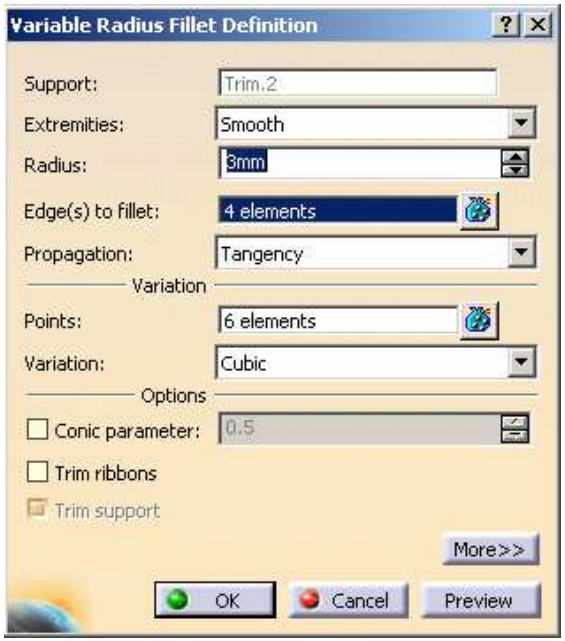
- Show “Join.1”.
- Trim Join.1 with previously created Trim.1

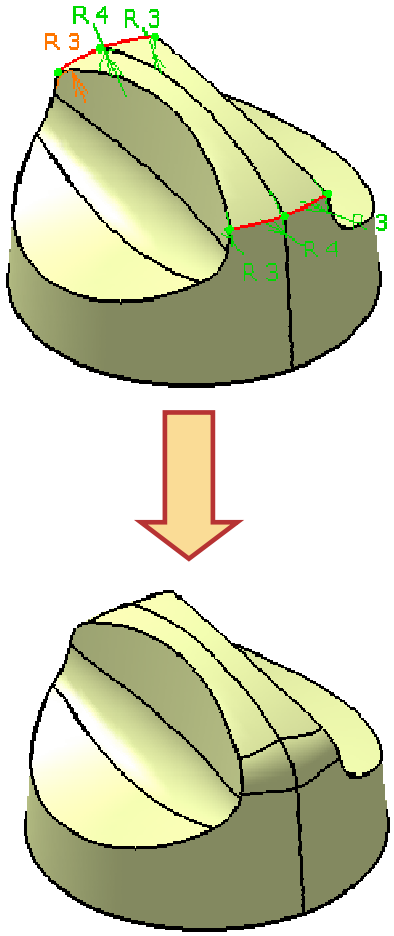


Student Notes:

# Do It Yourself (10/11)

- Apply Variable radius fillet to the edge shown. The radius value at the ends is 3mm and at the middle is 4mm.

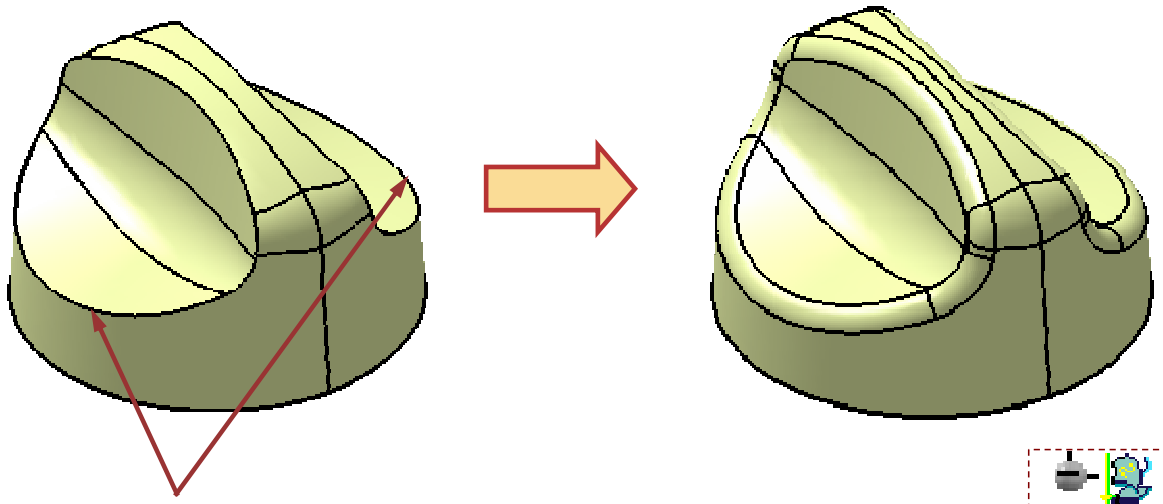


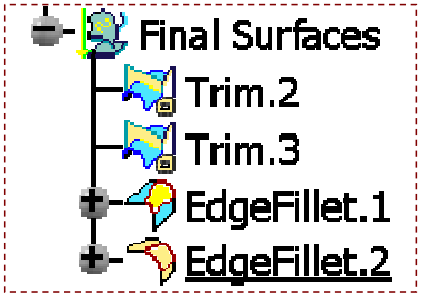
Student Notes:

# Do It Yourself (11/11)

- Apply Edge fillet of 1.5mm radius to the edge shown.



Apply Edge fillet to the two Edges.

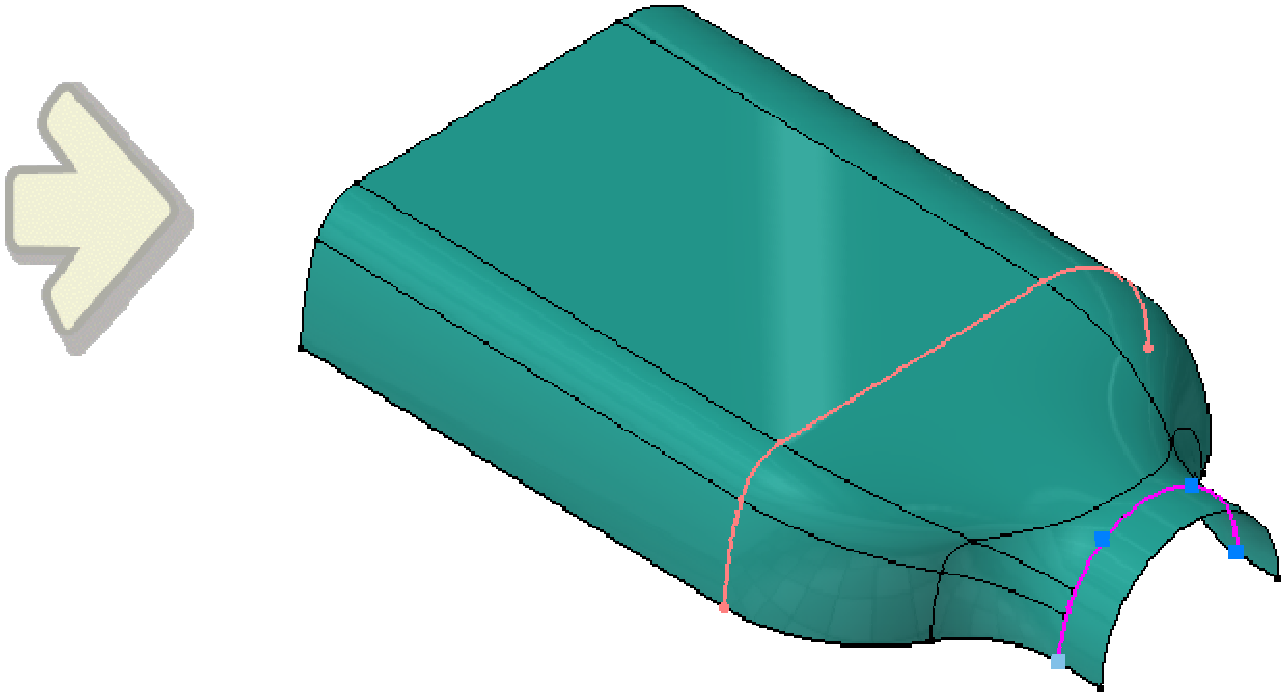



End part: CATGSD_F_Knob_End.CATPart

Student Notes:

# Creating an Adaptive Swept Surface

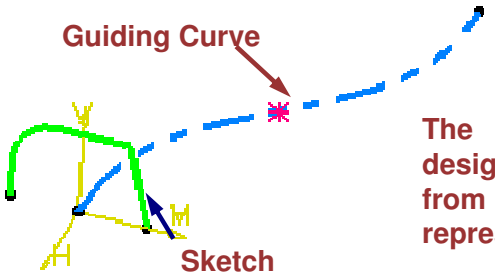
*You will learn about Adaptive Swept Surface and how to create it.*



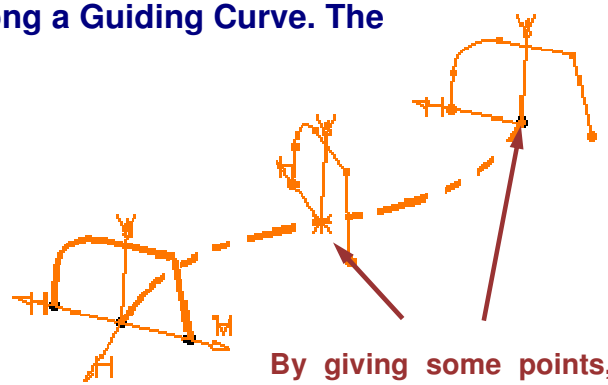
Student Notes:

# What is an Adaptive Swept Surface.

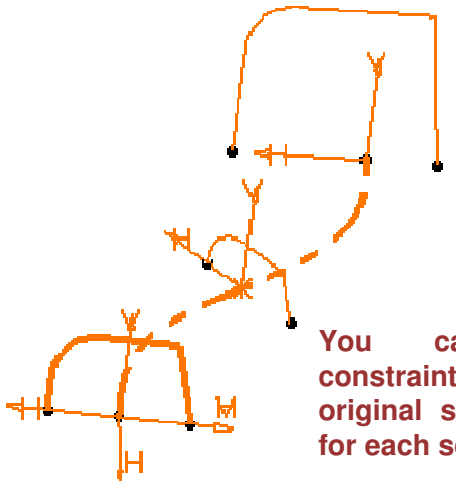
**This particular sweep uses a Sketch as Implicit profile along a Guiding Curve. The guiding curve is used as the default spine.**



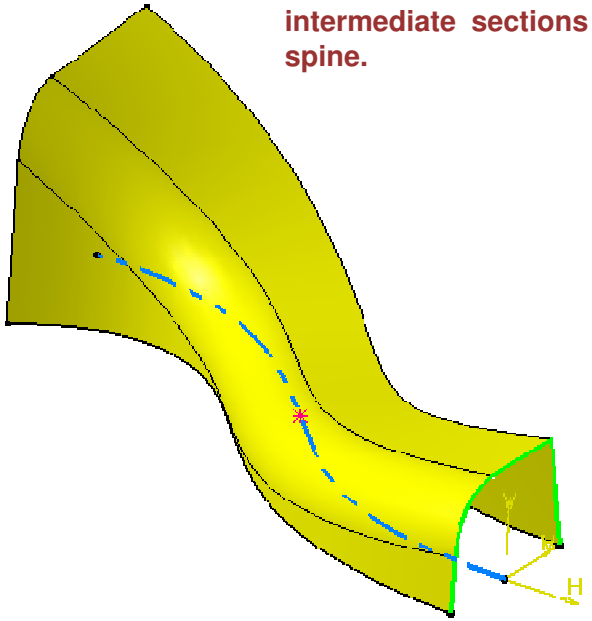

The Sketch has been designed in context directly from the dialog box and represents a connex profile



By giving some points, you will define automatically intermediate sections on the spine.



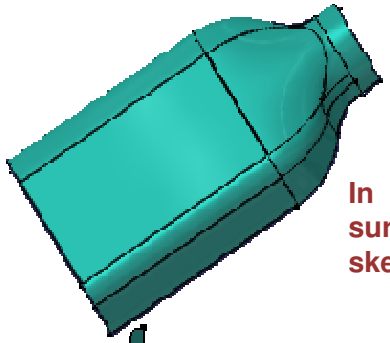
You can modify the constraints defined in the original sketch independently for each section.

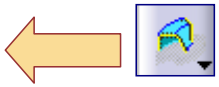


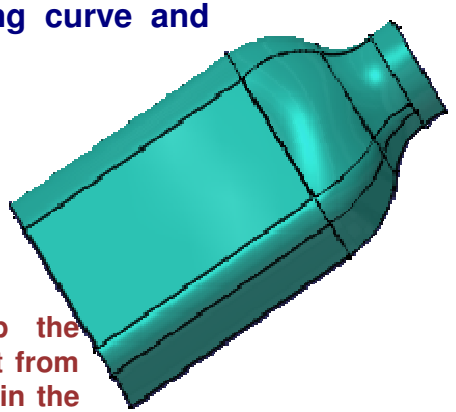
Student Notes:

# What are the differences with the other Swept Surfaces?

**An Adaptive sweep is always defined from a sketch. This leads to build a surface that inherits from the sketch constraints scheme on the whole surface. Besides you can create on the fly intermediate sections along the guiding curve and modify the constraints independently in each section.**



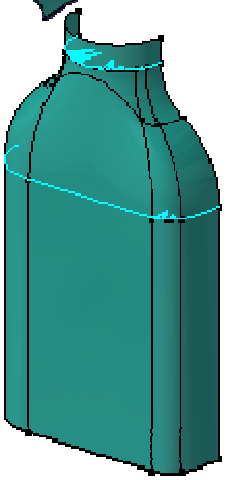




In an adaptive sweep, the surface inherits from the sketch constraints.

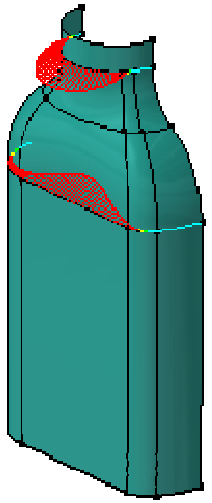
In the Explicit sweep the surface does not inherit from the constraints defined in the sketch.

What does that mean ?



If we analyse the connections between the surfaces, there is a few acceptable tangency discontinuity areas.

If we analyse the connections between the surfaces, there are important tangency discontinuities.



Student Notes:

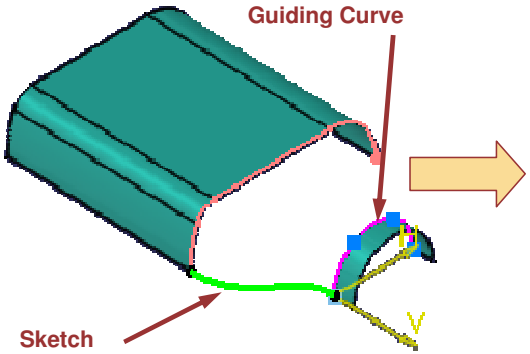
# Creating an Adaptive Swept Surface (1/2)

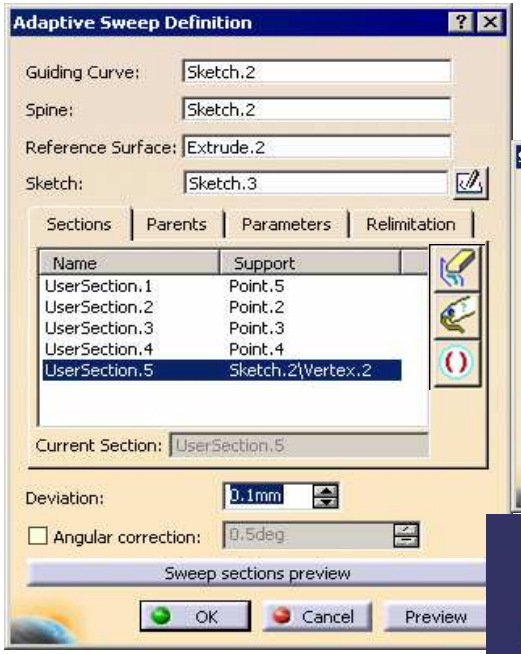
1

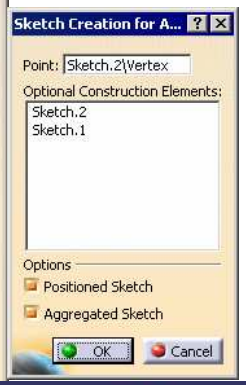


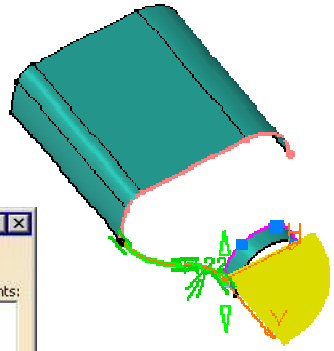
Select the Adaptive Sweep icon.

2 Select the Guide Curve and the Sketch to be swept.

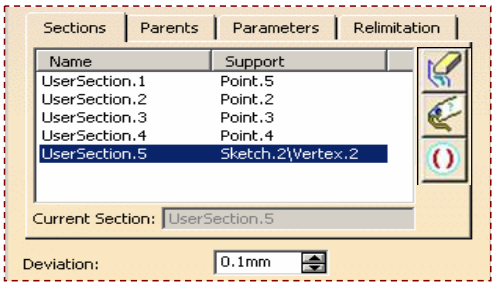


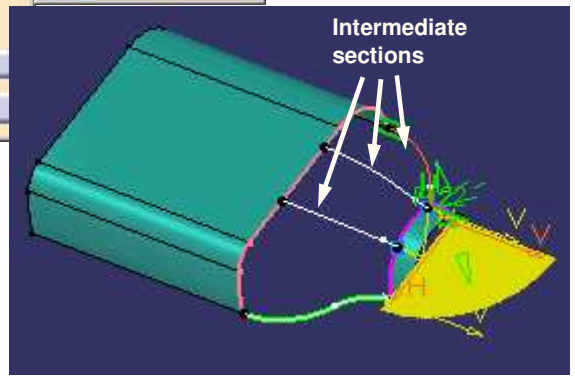



3 Select predefined points or vertices on the guide curve to add intermediate sections.

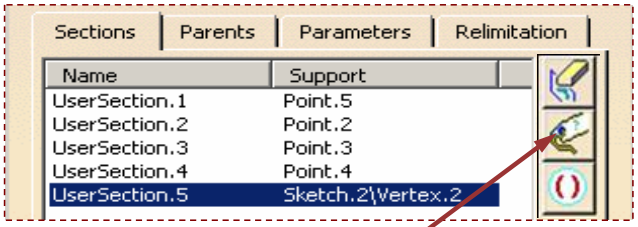


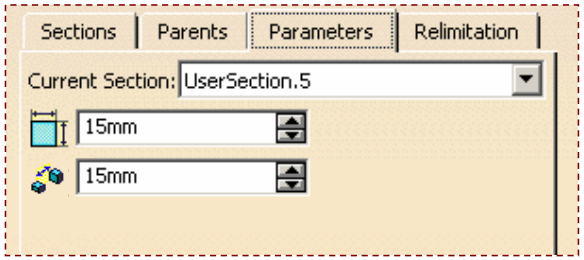



Student Notes:

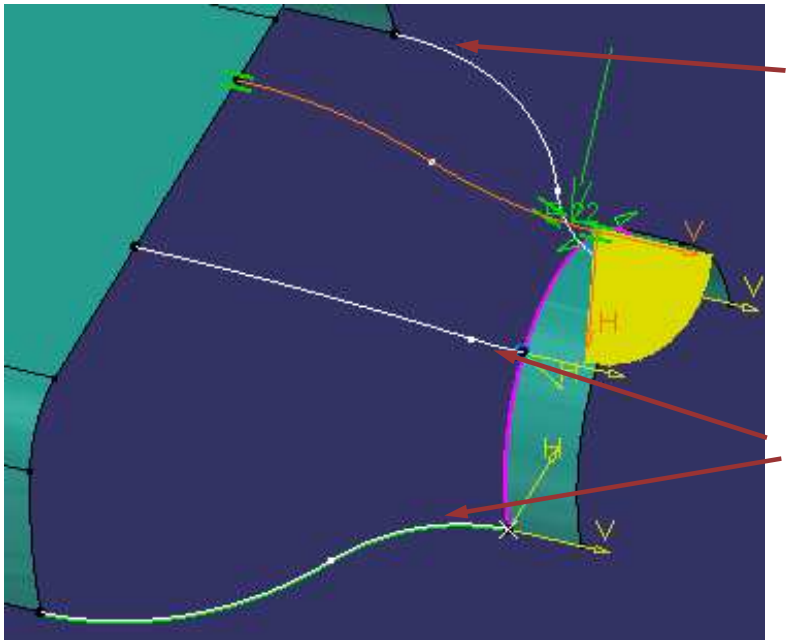
# Creating an Adaptive Swept Surface (2/2)

4 Under the Parameters tab, you can modify the constraints defined in the original sketch for each section independently




Use this icon to remove a section



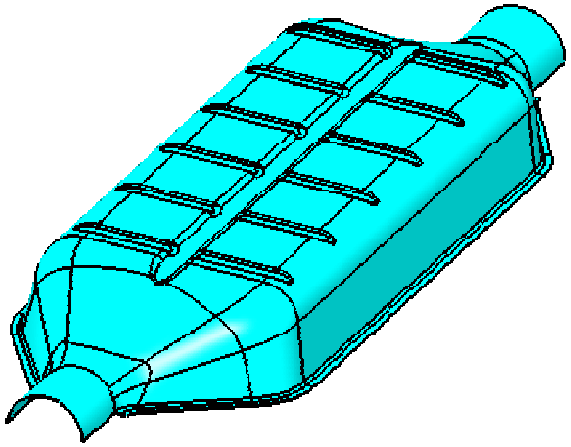

5 Click OK to confirm the surface creation

# Housing

## Recap Exercise: Adaptive Swept Surface

30 min

**In this exercise you will Create a part (Housing) in which you will be practicing Adaptive Swept surface.**

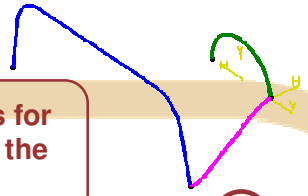


Student Notes:

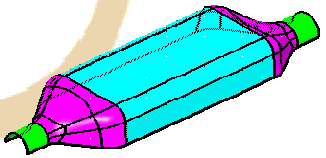
Student Notes:

# Design process: Housing

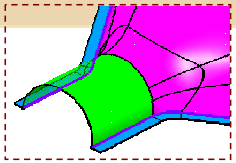
1. Design Basic surfaces for building main body of the housing



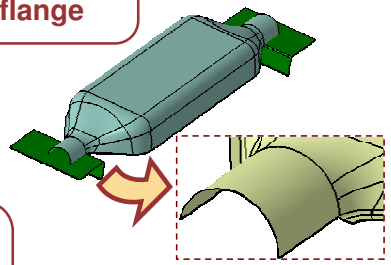
2. Create the main body of the housing using Adaptive swept surface and extrude



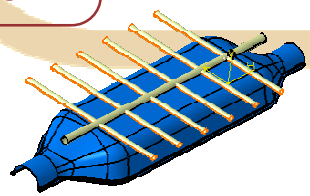
3. Extract the boundaries of the main body and create flange surfaces

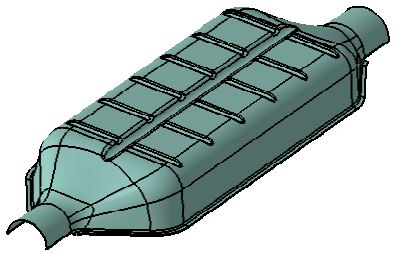


4. Create flange surface and split it from the main body to form the trimmed flange



5. Create tubular serrations on the main body and fillet the sharp edges





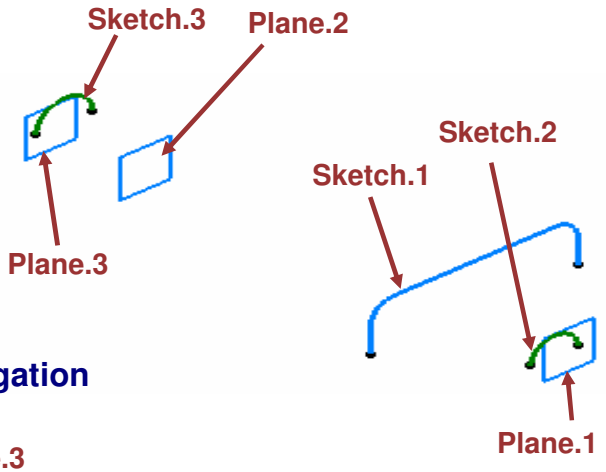
Student Notes:

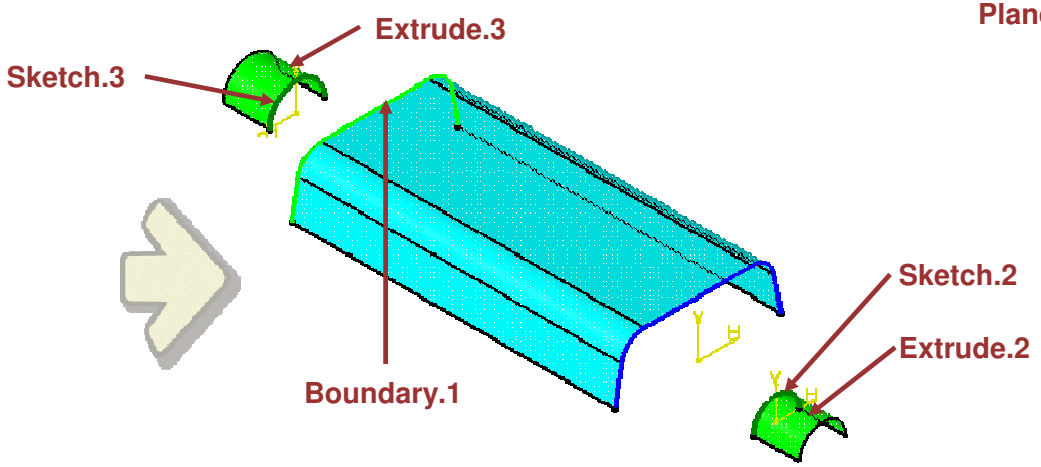
# Do It Yourself (1/11)

**Part Used: CATGSD_F_Adaptive_Sweep_Start.CatPart**

**Create the basic surface of the Housing using the given wireframe**

- **Create an Extrude.1 from Sketch.1 up to Plane .2 in the ZX direction**
- **Create an Extrude.2 from Sketch.2 up to distance 50mm in the ZX direction**
- **Create an Extrude.3 from Sketch.3 up to distance 50mm in the ZX direction**
- **Extract a Boundary.1 of Extrude.1/edge.1,Keep the propogation Tangent Continuity.**

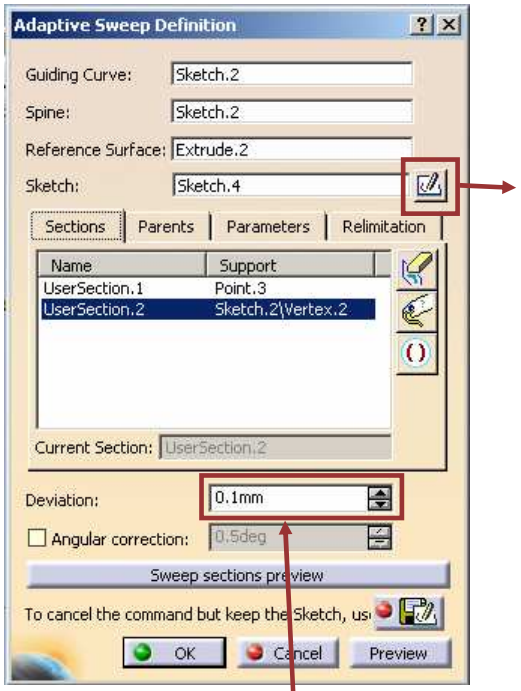


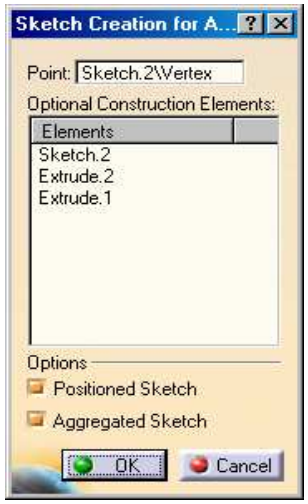
Student Notes:

# Do It Yourself (2/11)

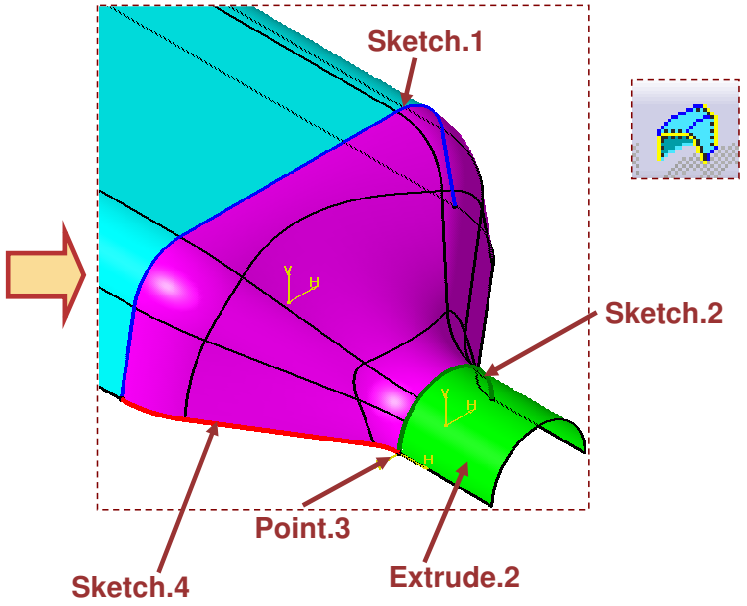
- Create an Adaptive swept surface(Adaptive sweep.1) as per the parameters shown:




Modify the Deviation to 0.01mm



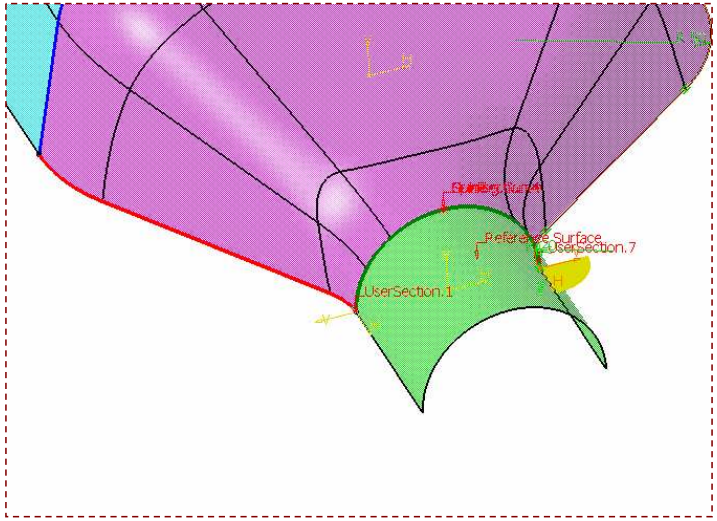

Sketch creation Box



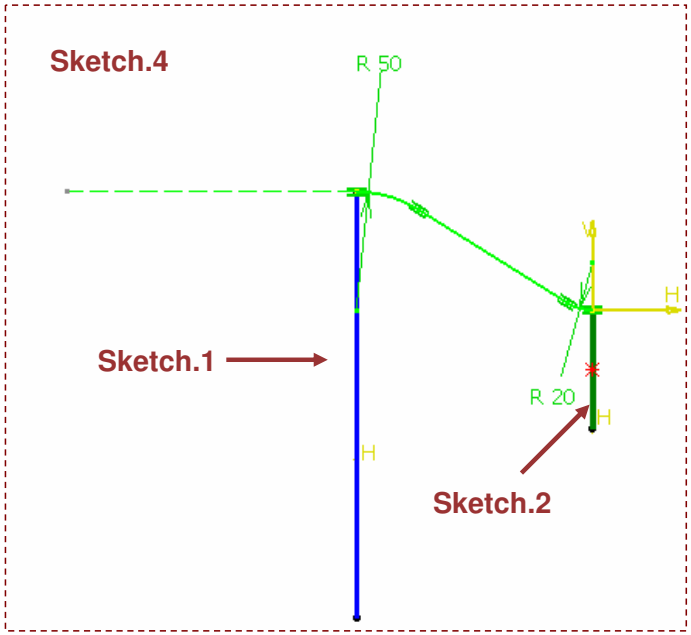

1. Refer to the next slide for detail Sketch and Dimensions.
2. Point.3 is to be created at the extremity of Sketch.2 using contextual menu.

Student Notes:

# Do It Yourself (3/11)




Create a Sketch.4 as shown below



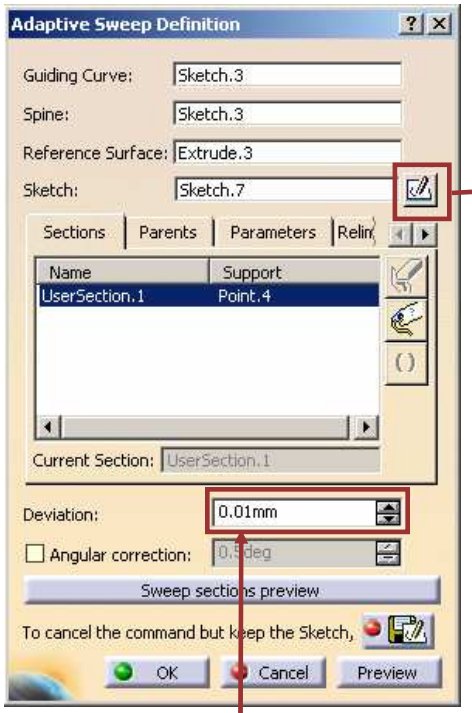

It is recommended that the sketch.4 used for creating Adaptive swept surface is to be created contextually by 'sketch creation box' in Adaptive swept panel

Hide the surfaces and Sketches for convenience while creating the profile in the Sketcher

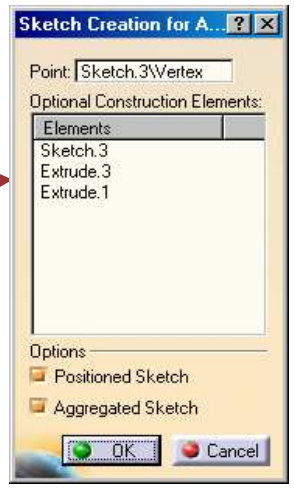
Student Notes:

# Do It Yourself (4/11)

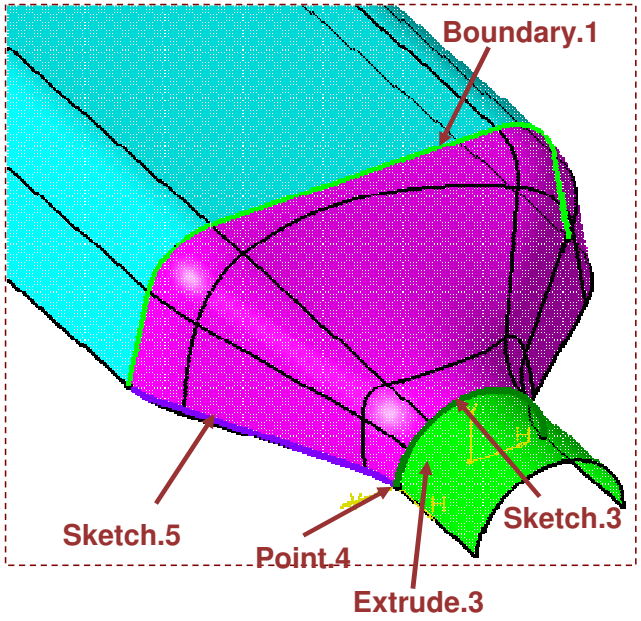
- Create an Adaptive swept surface(Adaptive sweep.2) as per the parameters shown:




Modify the Deviation to 0.01mm




Sketch creation Box

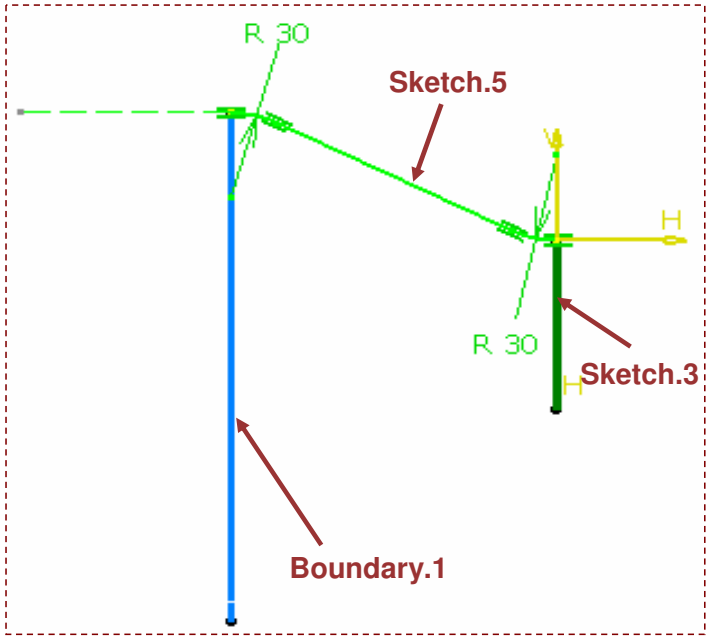


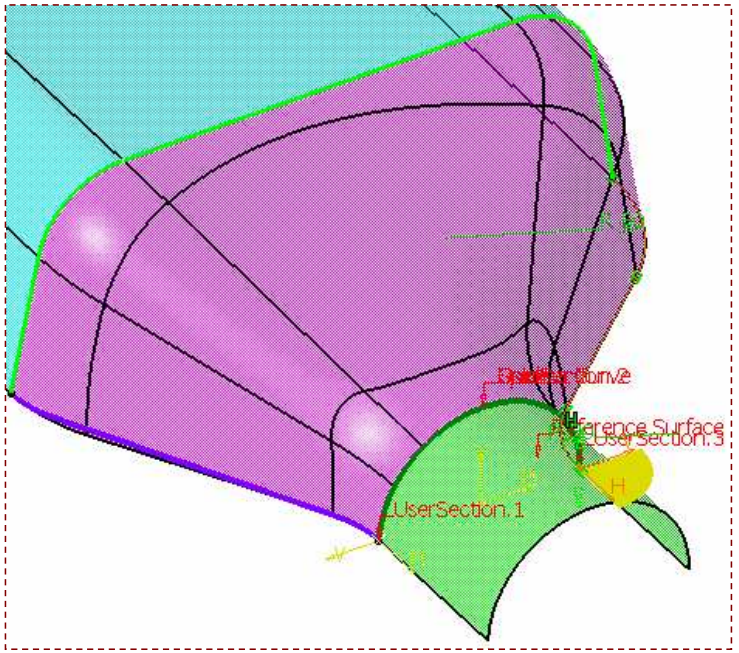

1. Refer to the next slide for detail Sketch and Dimensions.
2. Point.4 is to be created at the extremity of Sketch.3 using contextual menu.

Student Notes:

# Do It Yourself (5/11)

Create a Sketch.5 as shown below




It is recommended that the sketch.5 used for creating Adaptive swept surface is to be created contextually by 'sketch creation box' in Adaptive swept panel

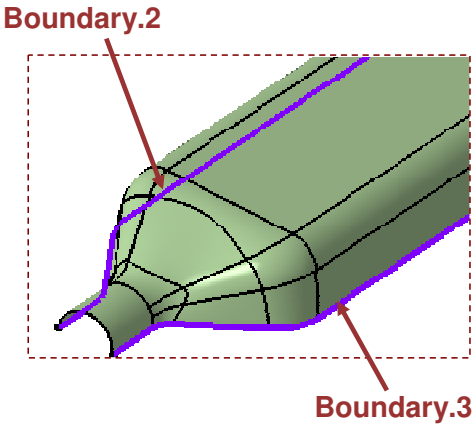
Hide the surfaces and Sketches for convenience while creating the profile in the Sketcher

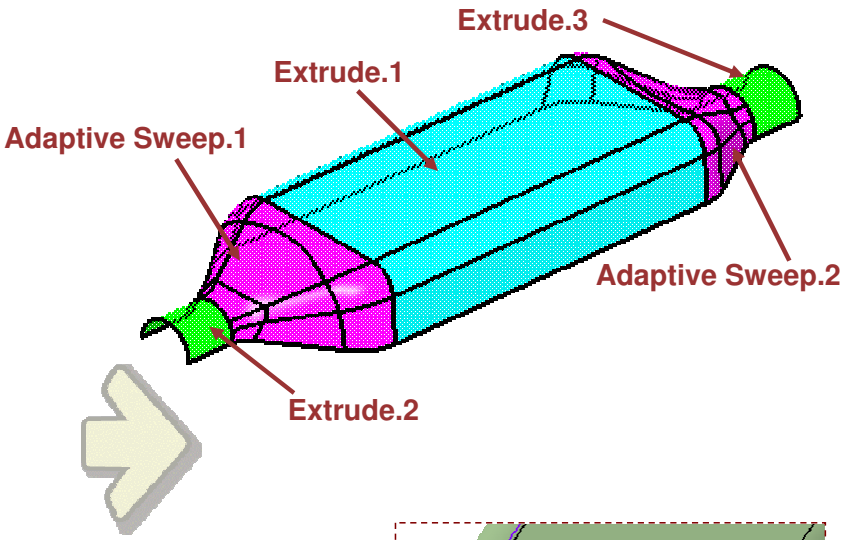
Student Notes:

# Do It Yourself (6/11)

- Create a Join.1 between Extrude.1,Extrude.2, Extrude.3, Adaptive sweep.1 and Adaptive sweep.2

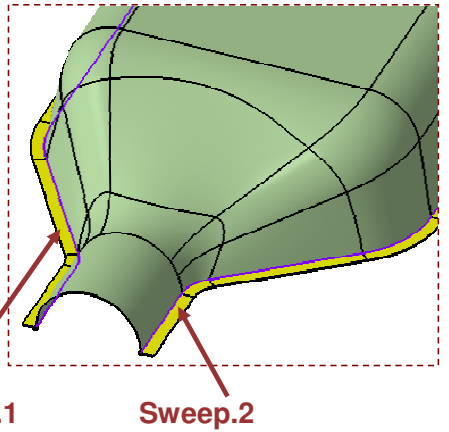
- Extract Boundary.2 and Boundary.3 of join.1




- Create a Line swept surface(Sweep.1) from the following parameters:
  - Sweep subtype:With Draft Direction, Guide Curve 1:Boundary.2 Direction:XY plane, Angle:90deg, Length 1= 6mm

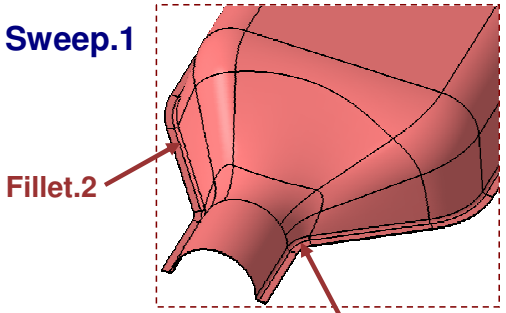
- Create a Line swept surface(Sweep.2) from the following parameters:
  - Sweep subtype:With Draft Direction, Guide Curve 1:Boundary.3 Direction:XY plane, Angle:90deg, Length 1: 6mm

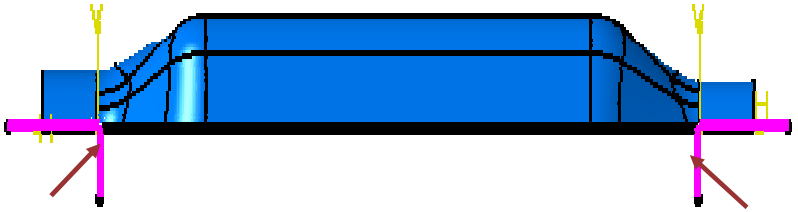


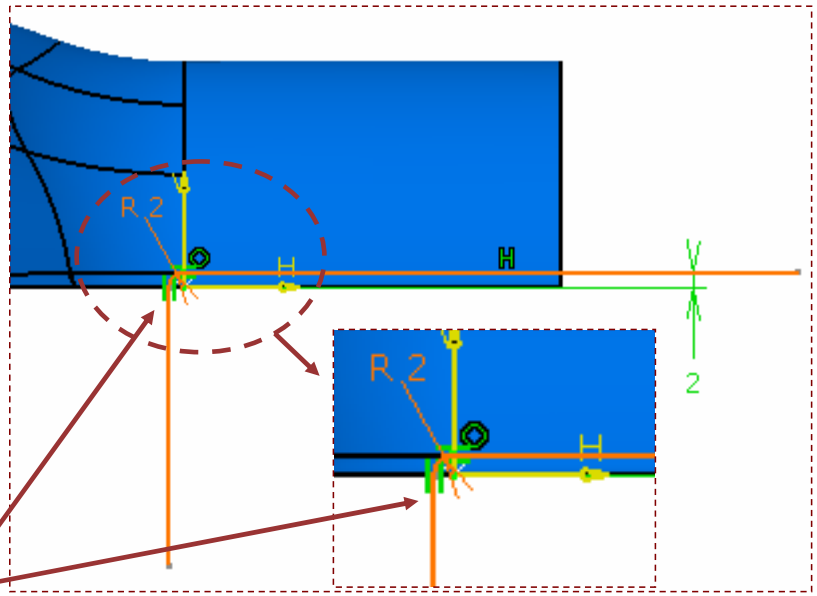
Student Notes:

# Do It Yourself (7/11)

- Create a Bi-tangent fillet (Fillet.1) between Sweep.2 and Join.1
  - Radius:R 2.0mm
- Create a Bi-tangent fillet (Fillet.2) between Fillet.1 and Sweep.1
  - Radius:R 2.0mm
- Create a Point.5 and Point.6 as shown:
  - Point.5: On Plane.3, H=0, V=0
  - Point.6: On Plane.1, H=0, V=0
- Create a Positioned Sketch.6 on YZ Plane, Projection point:Point.5, orientation: Y axis
- Create a Positioned Sketch.7 on YZ Plane, Projection point:Point.6, orientation: Y axis



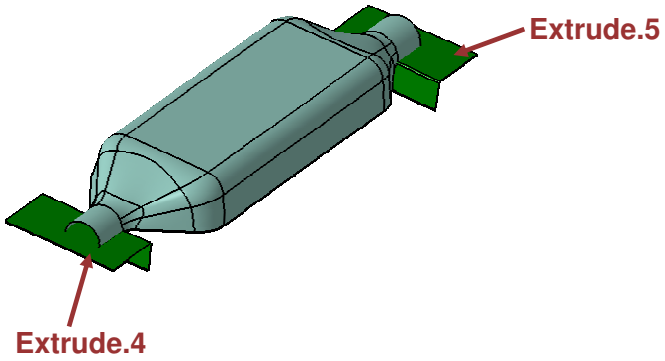



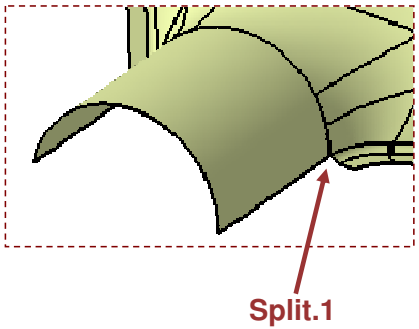
Student Notes:

# Do It Yourself (8/11)

- Create Extrude.4 and Extrude.5 from Sketch.6 and Sketch.7 respectively
  - Direction :YZ plane
  - Distance : 75mm on both side

- Split(Split..1) fillet.2. Specify Extrude.4 and Extrude.5 as cutting elements



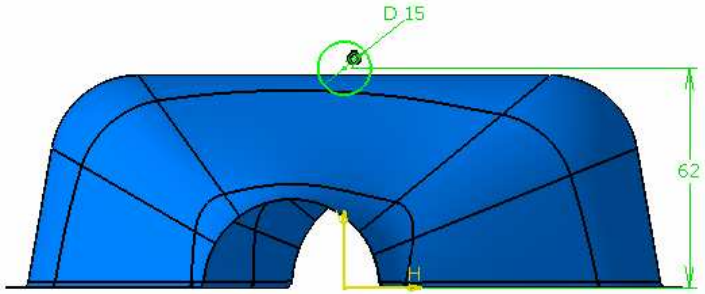



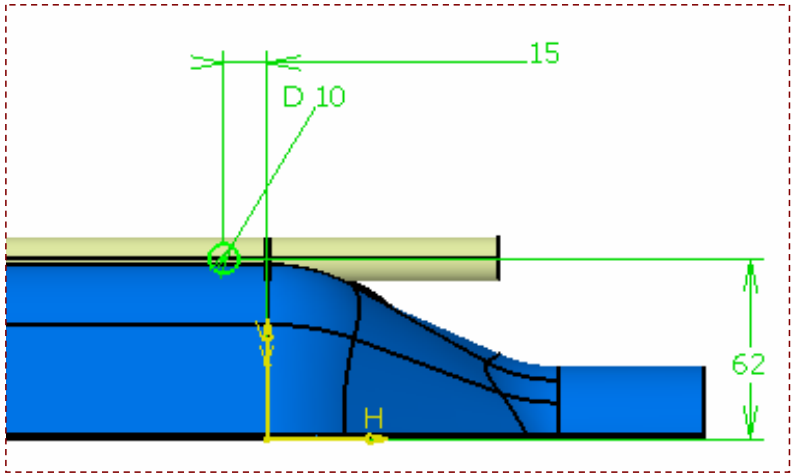
# Do It Yourself (9/11)

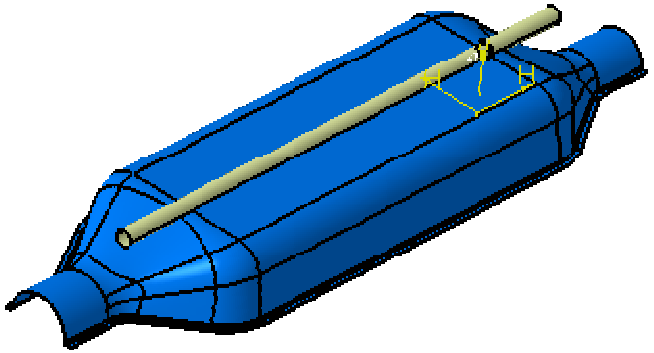
- Create a Positioned Sketch.8 on ZX Plane,
  Projection point:Part origin, orientation: Implicit

- Create an Extrude.6 from sketch.8 in the ZX direction
  - Limit 1:80mm
  - Limit 2:425mm

- Create a Positioned Sketch.9 on YZ Plane,
  Projection point:Part origin, orientation: Implicit



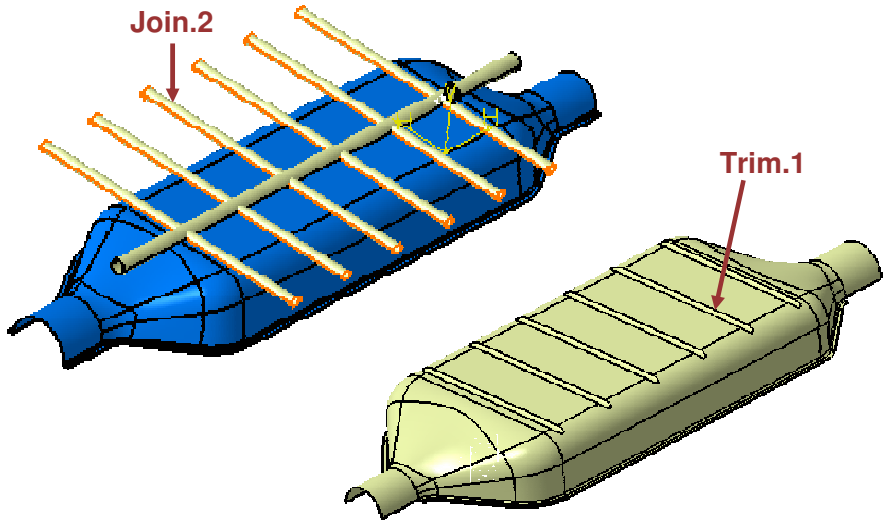




Student Notes:

Student Notes:

# Do It Yourself (10/11)

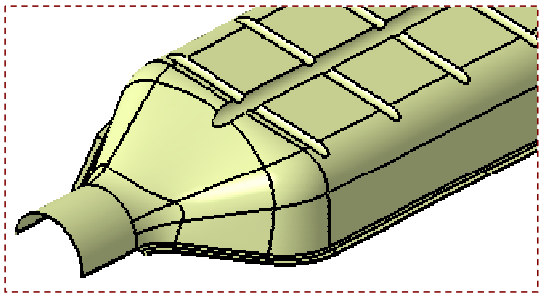
- Create an Extrude.7 from sketch.9 in the YZ direction
  - Limit 1:120mm
  - Limit 2:120mm
- Rectangular Pattern(Rectpattern.1) Extrude.7 as shown.
- Create a Join.2 between Extrude.7 and Rectpattern.1(Check Connexity box off)
- Trim (Trim.1) Join.2 and Split.1

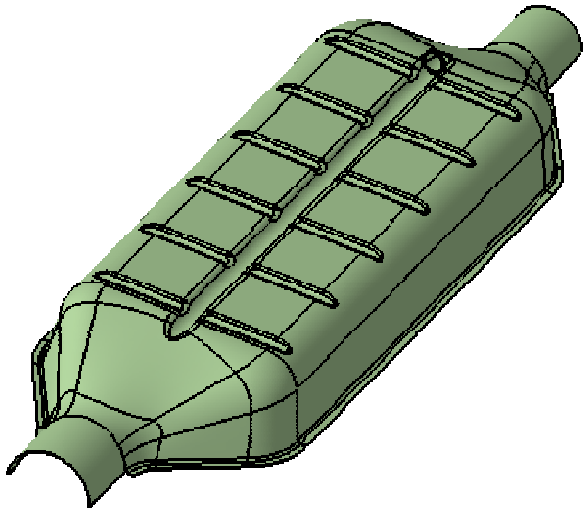



Student Notes:

# Do It Yourself (11/11)

- Trim (Trim.2) Trim.1 and Extrude.6
- Apply edge fillet(Edgefillet.1) of radius 5mm to Trim.2/Face.1
- Apply edge fillet(Edgefillet.2) of radius 3mm to Edgefillet.1/Face.2





End Part: CATGSD_F_Adaptive_Sweep_End.CatPart

Student Notes:

# Creating Advanced Blending Surfaces

*This lesson will cover the following Advanced Blend Surface topics:*

- About this Lesson
- What is a Blend?
- Types of Blends
- Creating Fillets using Hold Curve and Spine
- Creating Fillets using Law
- Creating a Blend Surface
- Blend Surfaces Recommendations
- Recap Exercise: Advanced Blending Surface

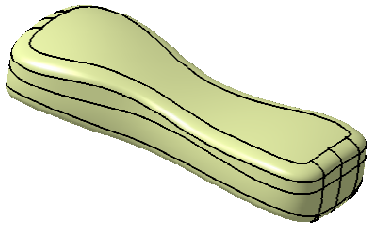
Student Notes:

# About this Lesson

In this lesson you learn and practice the Advance tools for Blending Surfaces.

Along with knowing how to create Blend surfaces, you will learn about the functionalities that will be used to create the part namely:

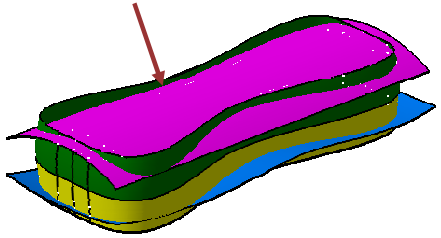
- Fillets using Hold Curve and Spine
- Fillets using Law



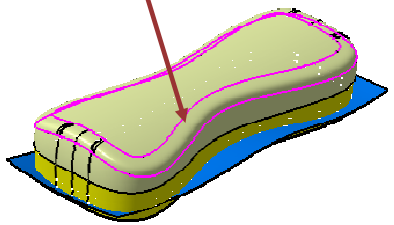
Blended Surface

At the end of this Lesson, you will create a surfacic part in which Blend surface will be used to create the surfaces.

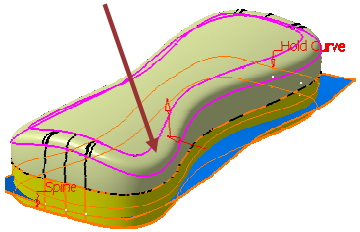
Wireframe Intersections



Blend Surface



Shape Fillet using Hold curve and Spine

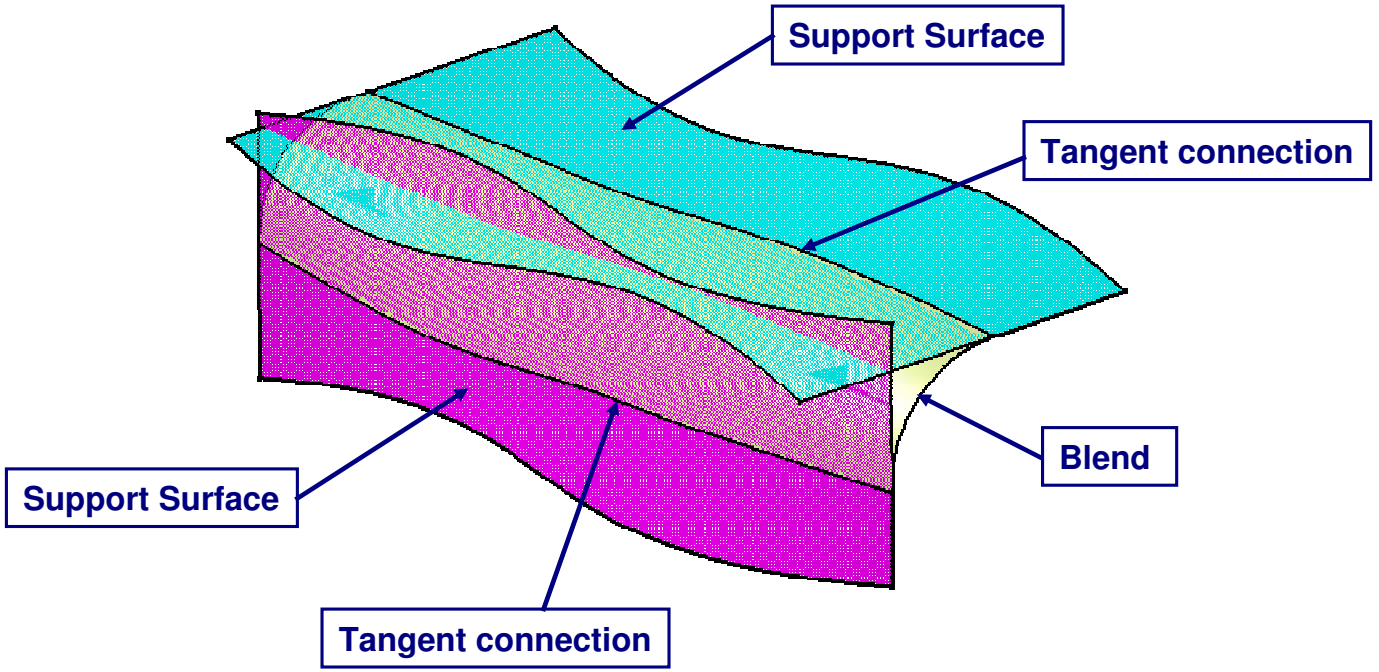



You will also learn a few Blend surface recommendations.

Student Notes:

# What is a Blend?

**A blend is a surface that is created between two support surfaces that forms a smooth transition between the two. At minimum, the connection a each support is tangent. However, curvature continuity is also sometimes possible.**

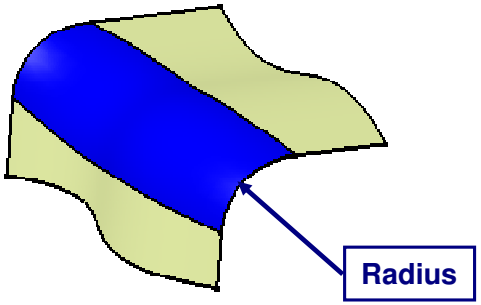


Student Notes:

# Types of Blends.

**There are several types of blends to choose from. Which one to choose is dependant on the aesthetic and functional requirements of the part (see the Methods and Recommendations section for more details). Several radius blends were covered in the Surface Design course.**

## Radius Blends




**Shape Fillet**

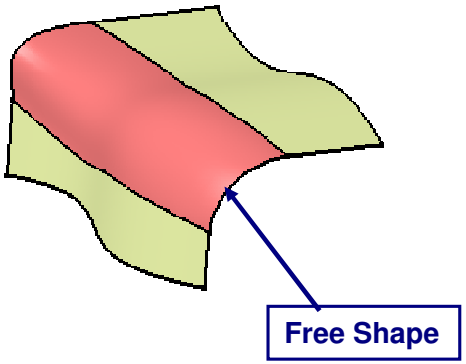
**Edge Fillet**

**Variable Radius Fillet**

**Face-Face Fillet**

**Tritangent Fillet**

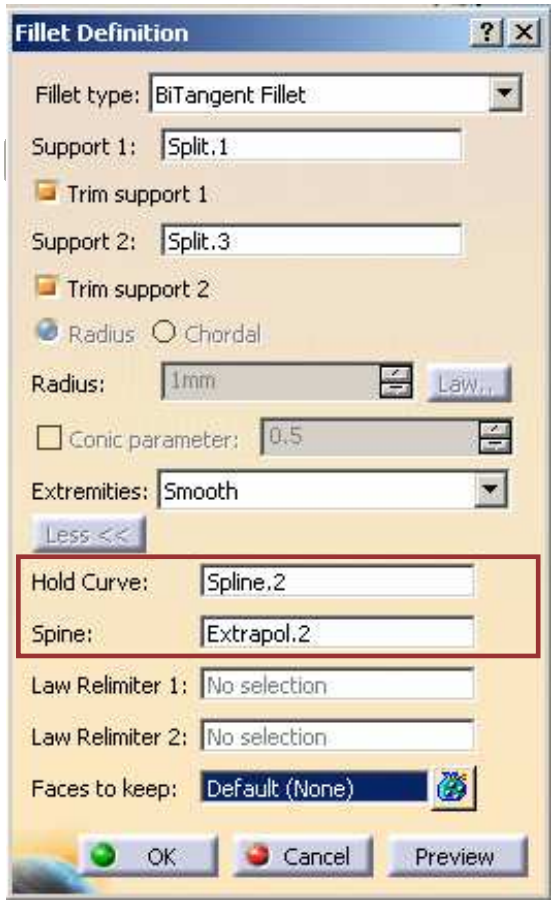
## “Blend” Blends



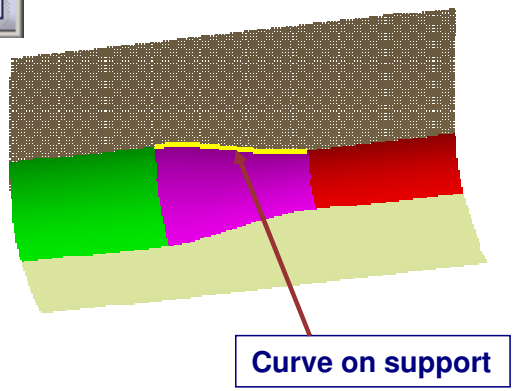
Student Notes:

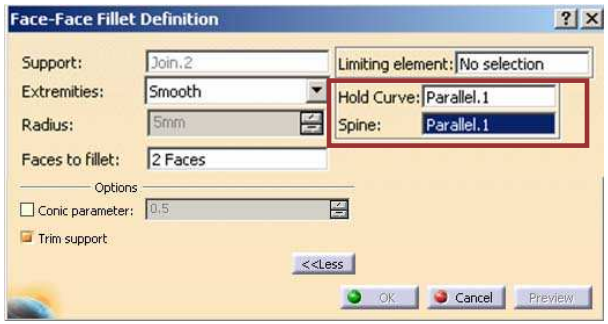
# Creating Fillets using Hold Curve and Spine

*You will see how to easily make variable radius fillets by following a curve lying on one of the supports*







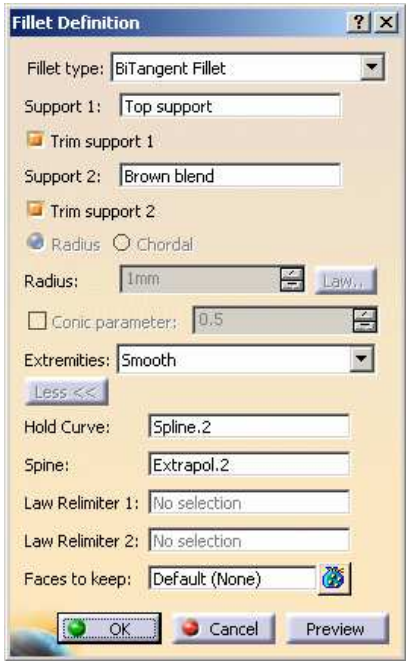



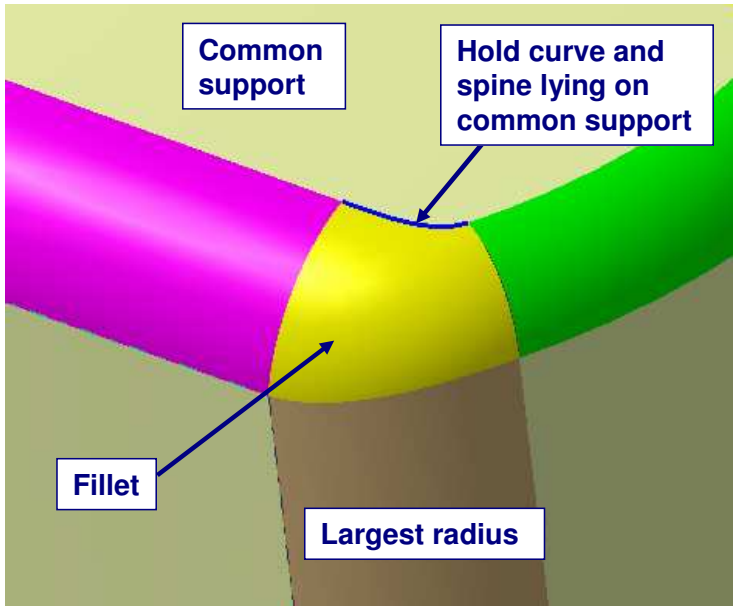
Student Notes:

# Why create a Fillet with Hold Curve and Spine?

**This tool is often useful when creating a complex variable transition blend. For instance, in the example shown here, three fillet radius blend of all different sizes come together in the corner of the part.**

**The rule of thumb here is to create a variable radius fillet blend between the largest radius and the support surface common to the other two. A curve, tangent to the two upper blend boundaries is created on the top support. Using this curve as the hold curve and spine, the transition variable radius blend is automatically created. You also have the option of using law to achieve this.**



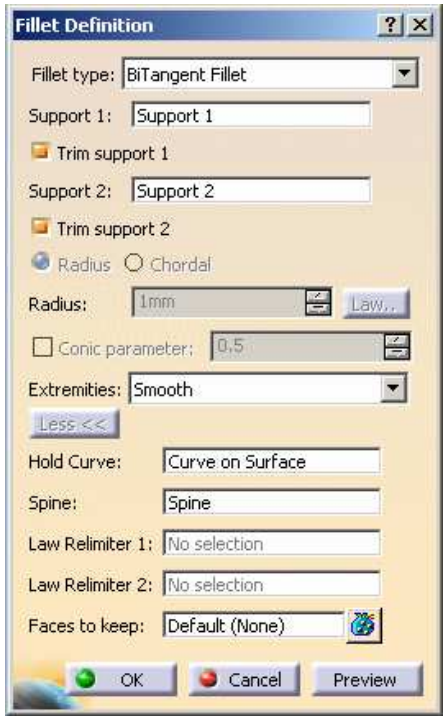

**This is much more reliable than using a Fill surface for this example.**

Student Notes:

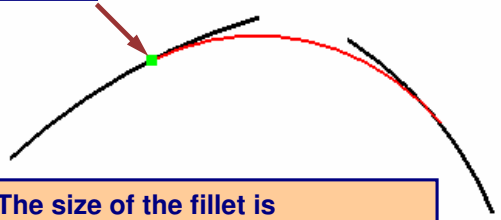
# Creating a fillet using Hold Curve and Spine

**This tool allows you to create a variable radius fillet between two surfaces where the radius is determined by a curve lying on one of those surfaces.**

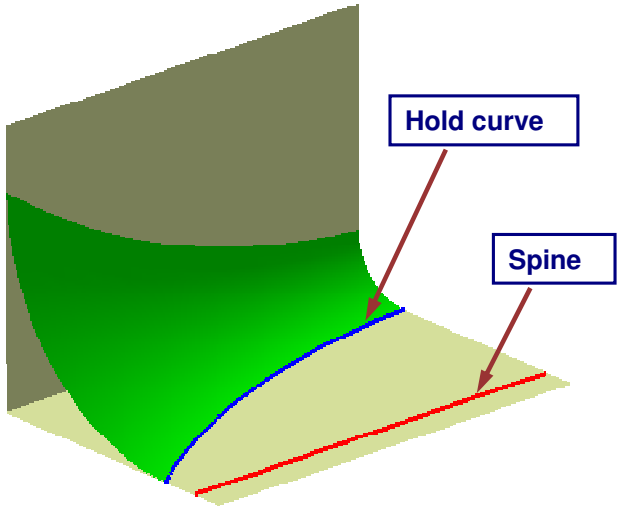
1




Hold curve



The size of the fillet is determined by the relationship of the curve to the intersection of the two surfaces.




2 Select the two surfaces

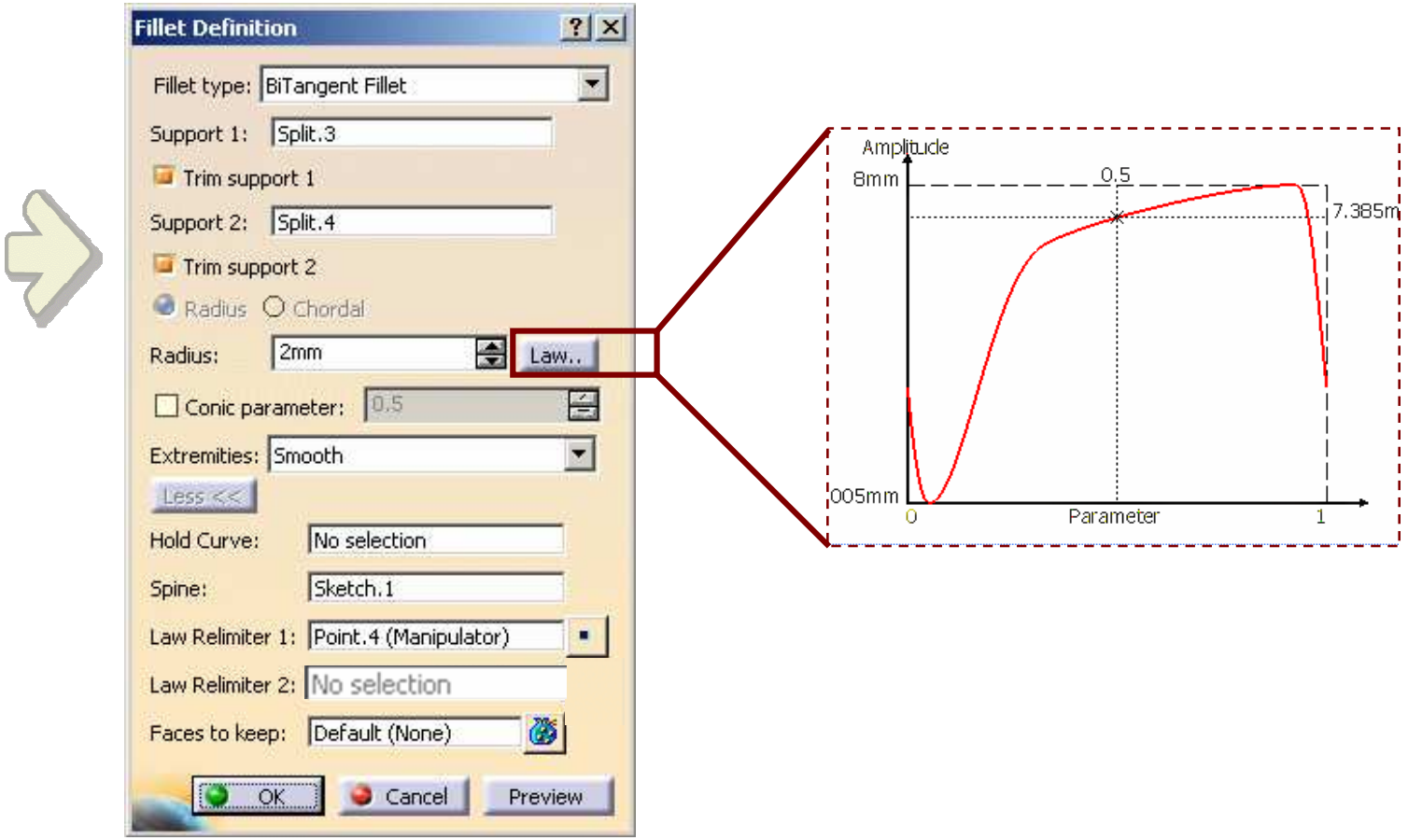
3 Select the hold curve

4 Select the spine

Student Notes:

# Creating Fillets using Law

*You will now see the method of creating fillets using Law.*



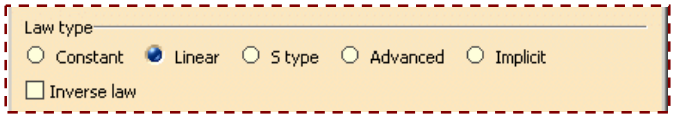
Student Notes:

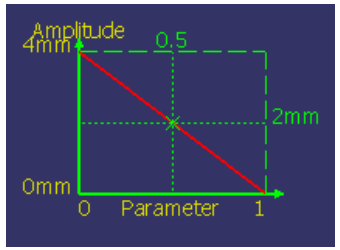
# Why do we need Law in Fillets ?

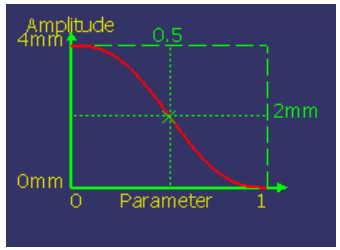
**Creating a hold curve is sometimes difficult. By using law , you can define the variation in the fillet radius separately in a knowledgeware law editor and use it at the time of the fillet creation or you can define simple laws at the time of the fillet creation itself.**

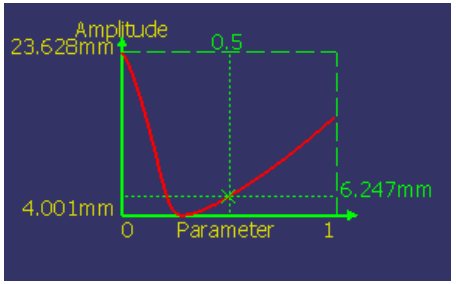
## What about this tool

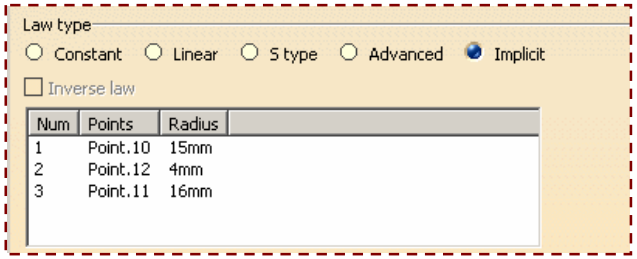
- **You can define various types of laws for the radius.**



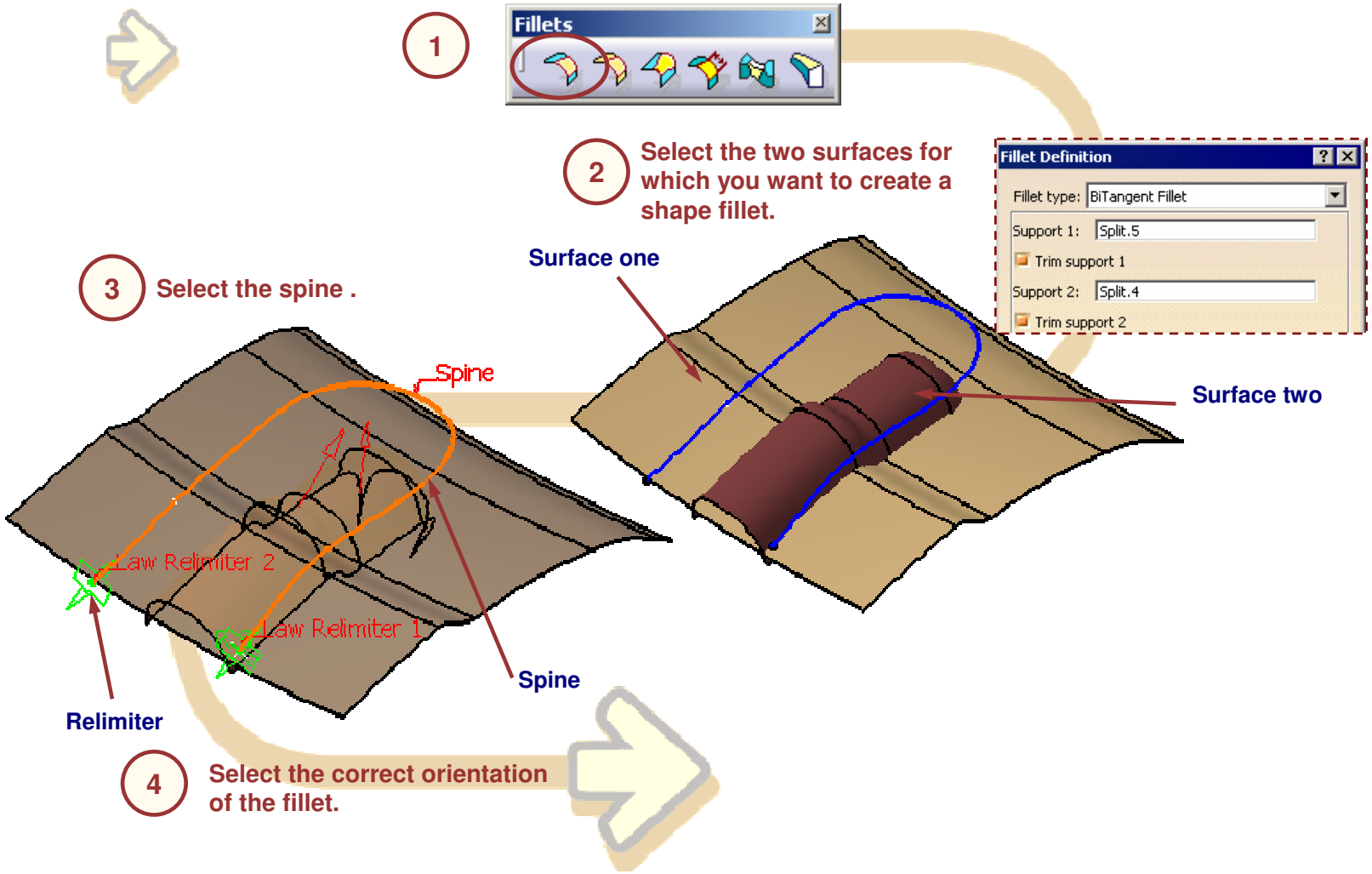

- **By selecting advanced law you are able to use a predefined law.**

- **When a spine is selected the end points of the spine are the default re-limiters for the law. You can modify the position of the re-limiter.**

Student Notes:

# Creating Fillet Using Law (1/2)

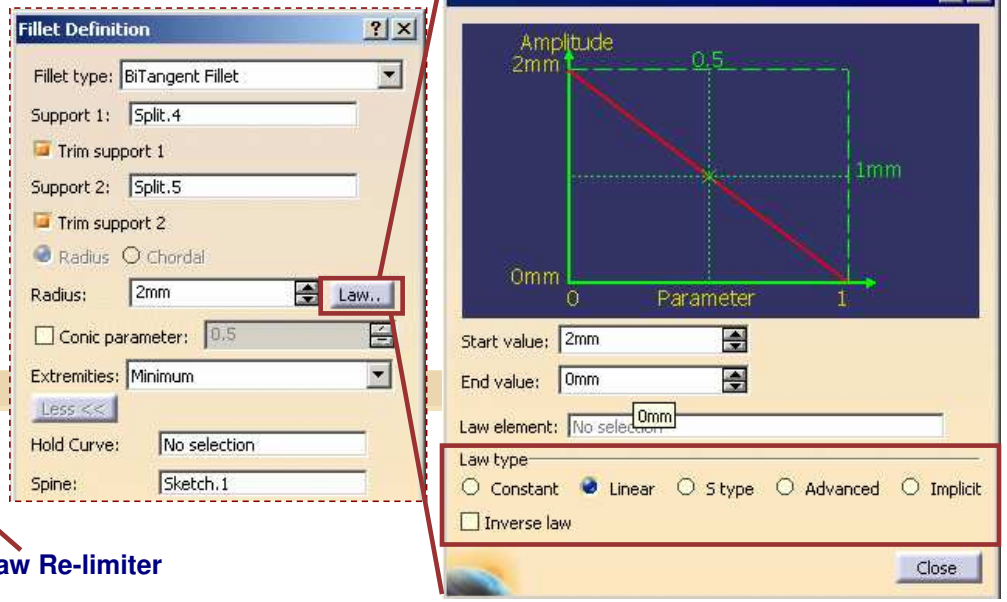
**Shape fillet allows you to define a variable radius path for the fillet.**



Student Notes:

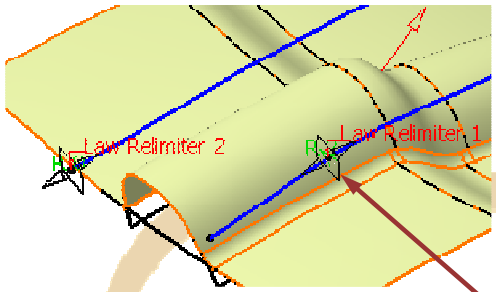
# Creating Fillet Using Law (2/2)

5 Select an appropriate Law for the radius value of the fillet.

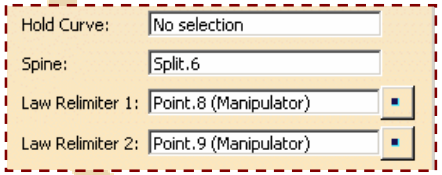



6 Change the relimiters of the law.

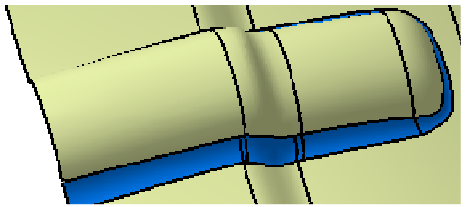
The end points of the spine are initially selected as re-limiters.




Law Re-limiter




7 Confirm OK



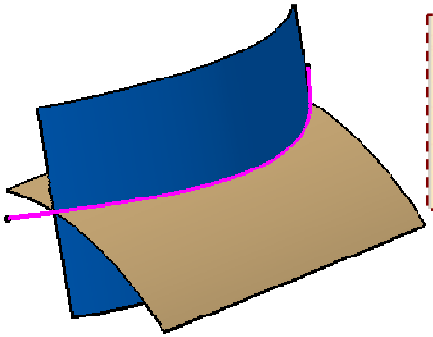
The fillet will follow the law only up to the law relimiter, after that the radius remains constant.

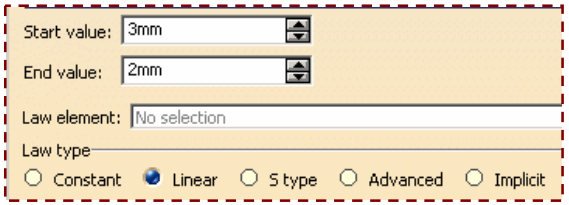
Student Notes:

# Do It Yourself

**Part used: CATGSD_ShapeFilletWithLaw.CATPart**

- **Create a Shape Fillet between the blue surface (First surface) and the brown surface (second surface) by the following methods.**
  - **Using a Linear Radius Law**

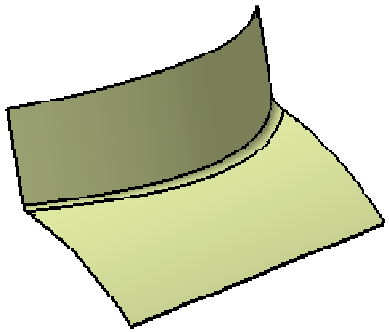




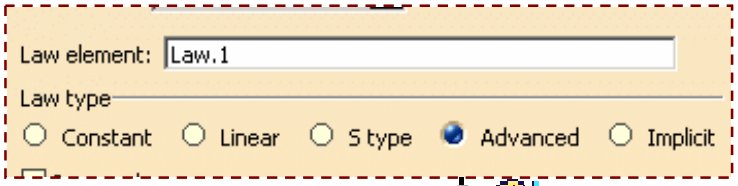

**Law : Linear , Start Value = 3 mm**

**End Value = 2 mm , Select Pink Spine**

**Extremities : Smooth**



  - **Using Advanced Law**

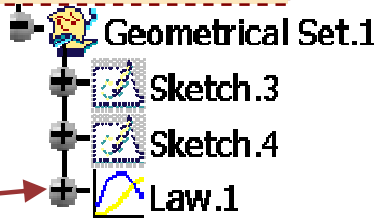


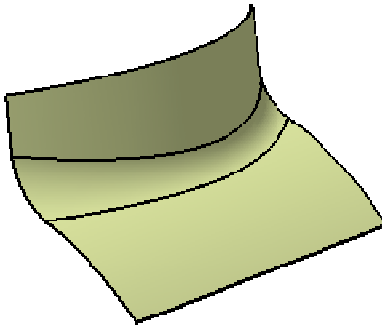

**Law : Advanced , Select Law.1**

**Select Pink Spine**

**Extremities : Smooth**

**Use this Law**

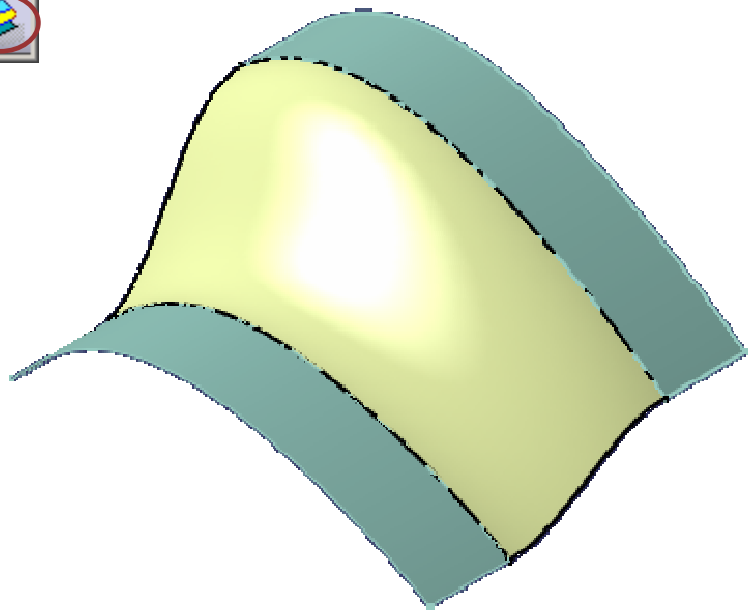


Student Notes:

# Creating a Blend Surface

*You will learn how to create a Blend Surface*

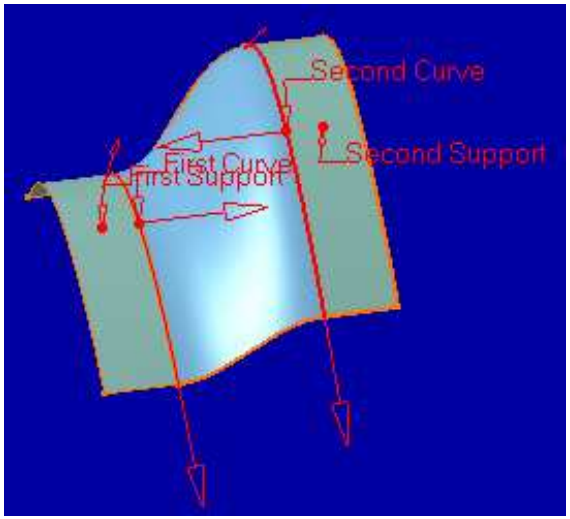



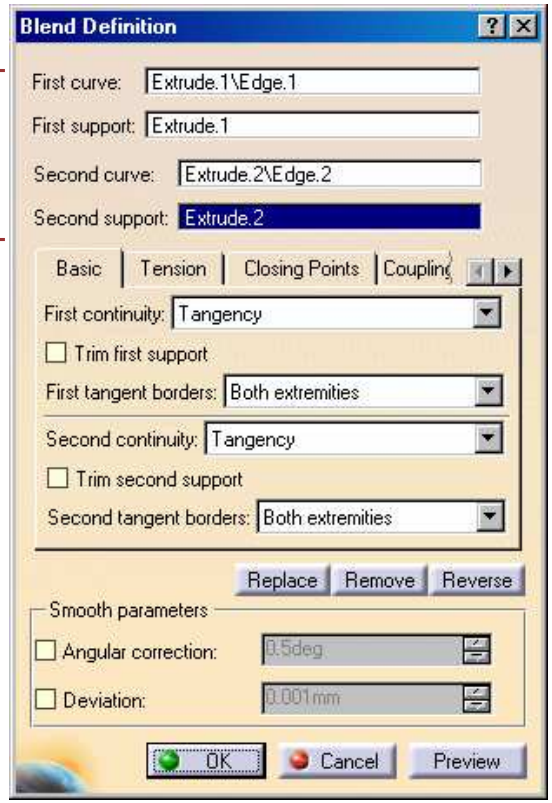
Student Notes:

# Creating a Blend Surface (1/7)

1


2 Select the two curves between which you will create the blend surface and, if needed, the support associated with each curve.

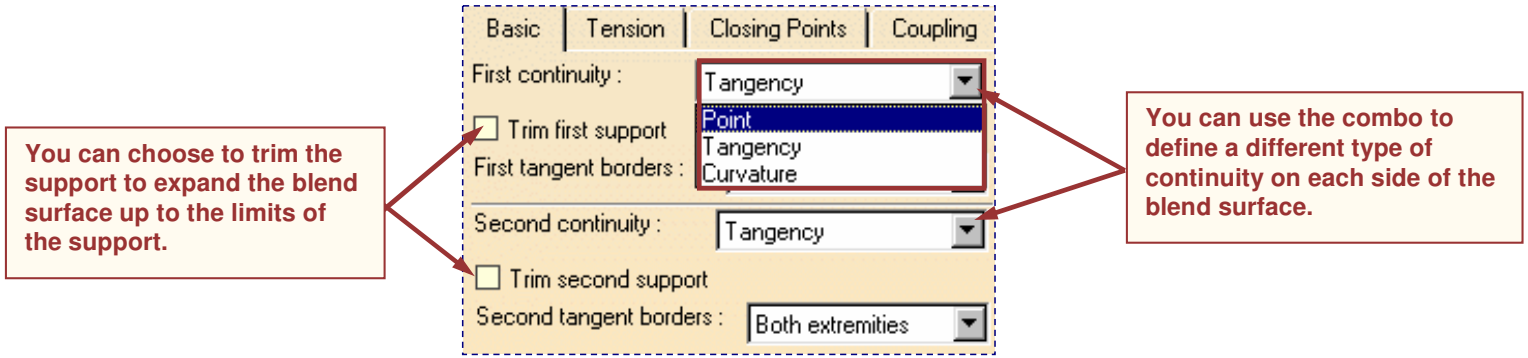


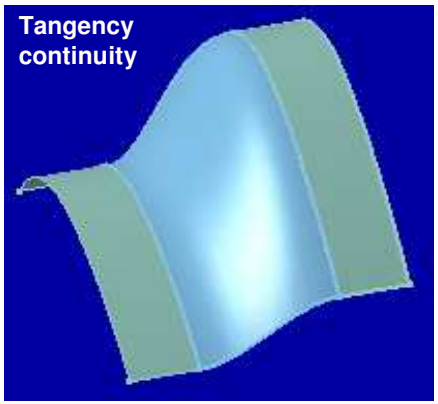



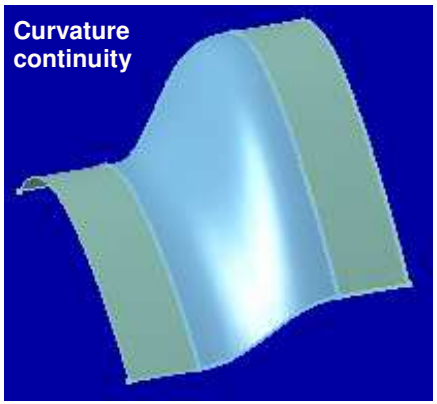
Student Notes:

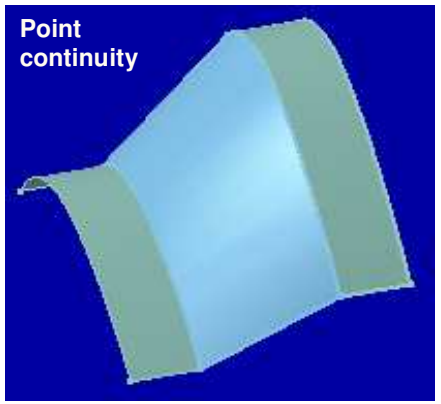
# Creating a Blend Surface (2/7)

3 If you have selected one or more support surface(s) define the type of continuity (Tangency, Curvature or Point) between each support surface and the blend surface.



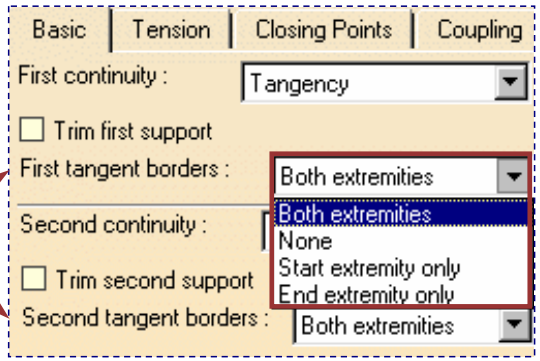



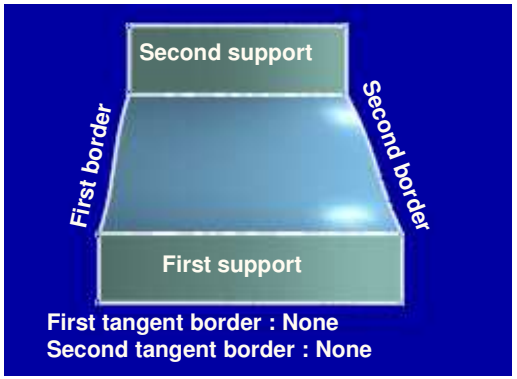
Student Notes:

# Creating a Blend Surface (3/7)

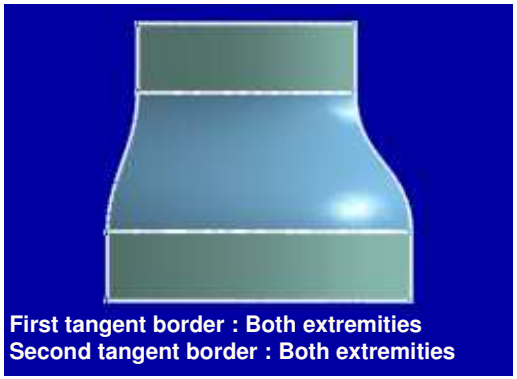
4 If you have selected one or more support surface(s) you can choose to make the borders of the blend surface tangent to the borders of the supports.

For each border of the blend surface you can choose the extremity(ies) that will be tangent to the corresponding border of the support.

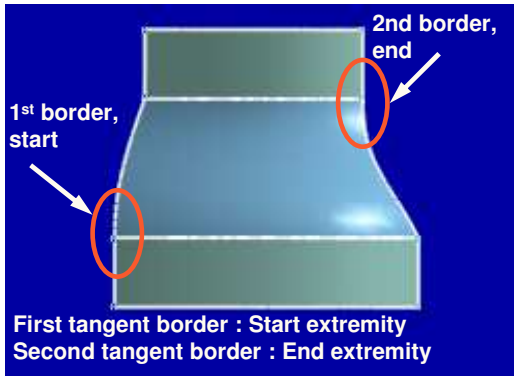



First tangent border : None
Second tangent border : None



First tangent border : Both extremities
Second tangent border : Both extremities

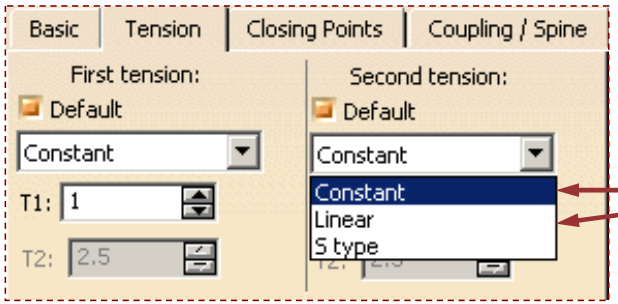



First tangent border : Start extremity
Second tangent border : End extremity

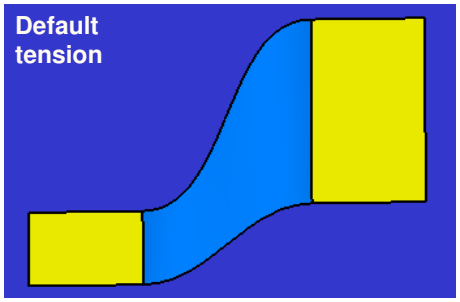
# Creating a Blend Surface (4/7)

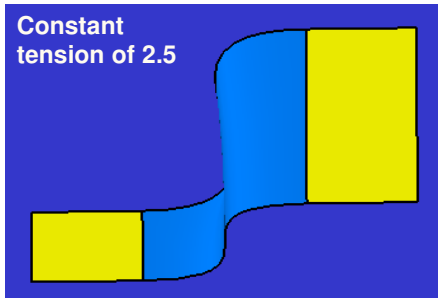
Student Notes:

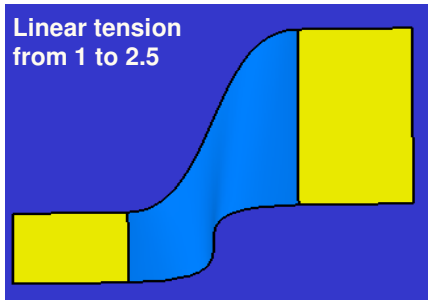
5 Select the Tension tab to define the tension at the limits of the blend surface.

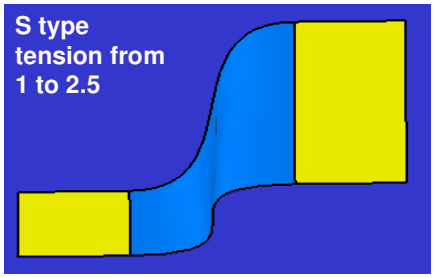



You can keep the default tension or define a constant, linear or S type tension at each limit of the blend surface.



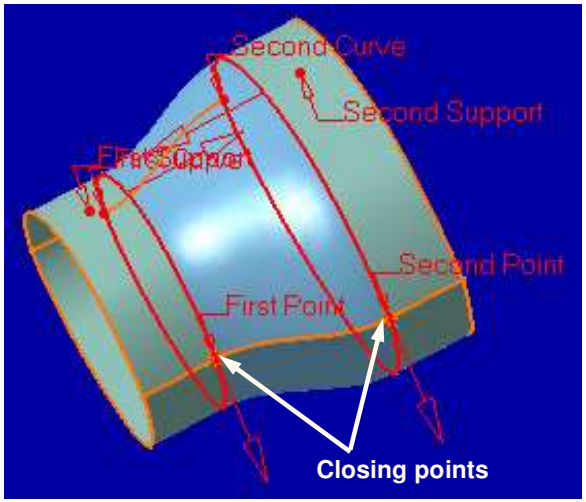



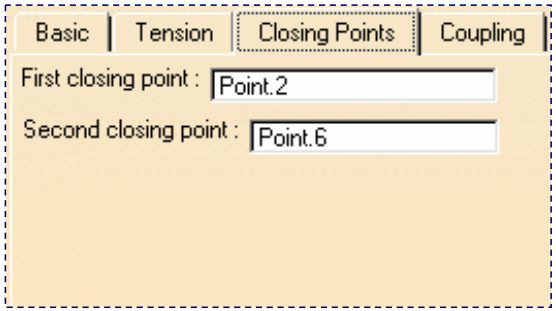
# Creating a Blend Surface (5/7)

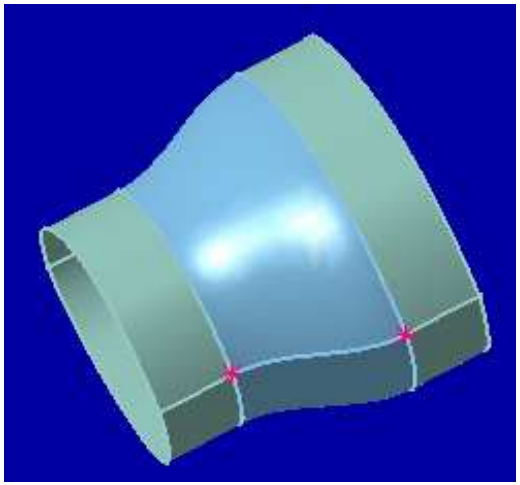
Student Notes:

6 In the case of a closed curve you can select the Closing Points tab and choose the closing point of each curve.




You can define the orientation of the blend surface by clicking on the arrows located on the selected closing points to invert them.

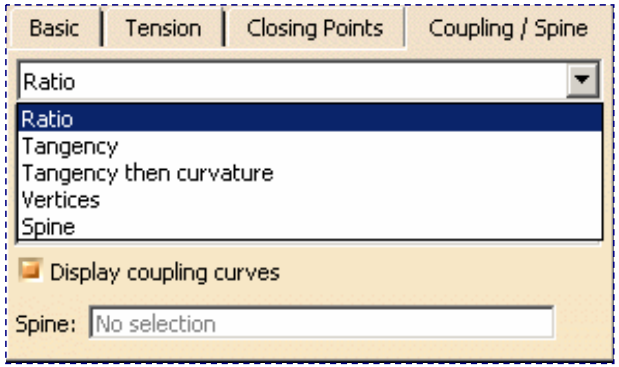


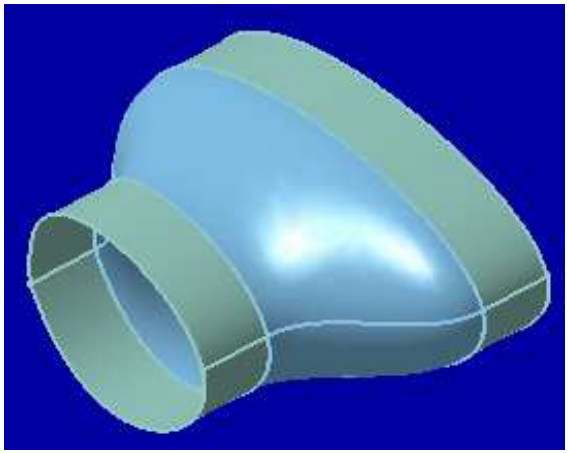



Student Notes:

# Creating a Blend Surface (6/7)

7a Select the Coupling tab to define the type of coupling :
- automatic with five options: Ratio, Tangency, Tangency then curvature, Vertices or Spine



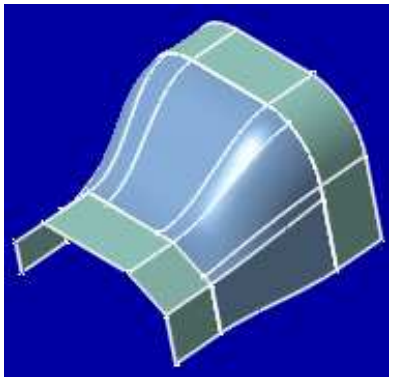
According to the chosen options you will compute the blend surface :
- using the total length of the sections (Ratio),
- between the tangency discontinuity points of the curves (Tangency),
- between the tangency discontinuity points of the curves then between the curvature discontinuity points of the curves (Tangency then curvature)
- between the vertices of the curves (Vertices).
- between the spine of the curves (Spine).

Student Notes:

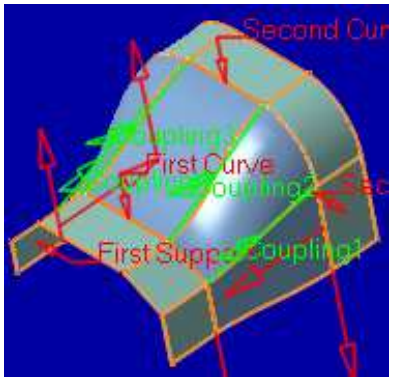
# Creating a Blend Surface (7/7)

**7b** Select the Coupling tab to define the type of coupling :
- manual coupling with definition of the coupling curve(s)

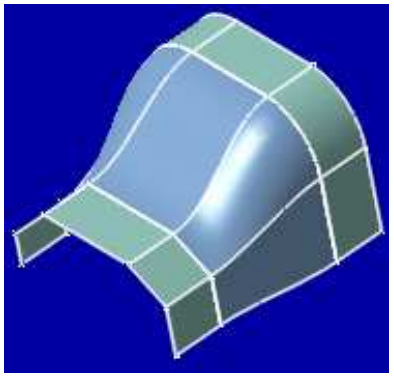

**Automatic coupling**



**Coupling curves**




**Manual coupling**

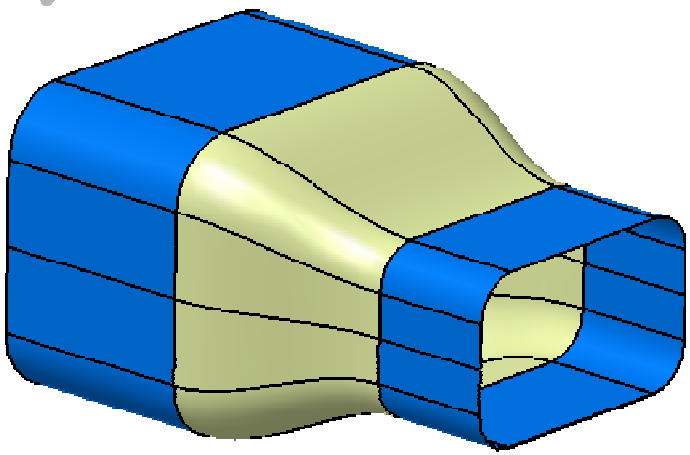


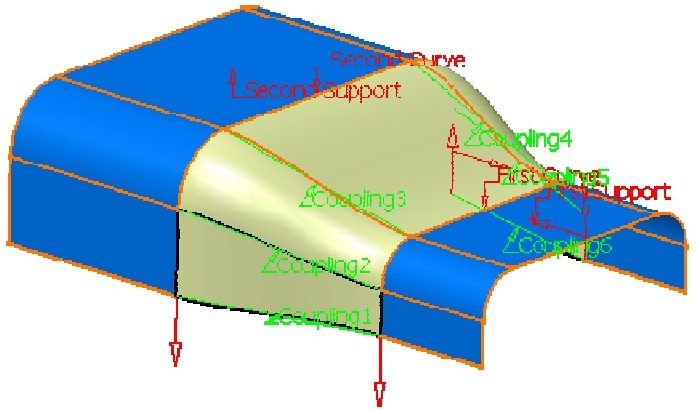
**8** Click OK to confirm blend surface creation.

Student Notes:

# Blend Surfaces – Recommendations

*You will learn about specific methods and recommendations concerning Advanced Blend Surfaces.*





Student Notes:

# How to choose the type of blend to use?

**The type of blend to choose is dependant on the functional and aesthetic requirements of the part being designed.**

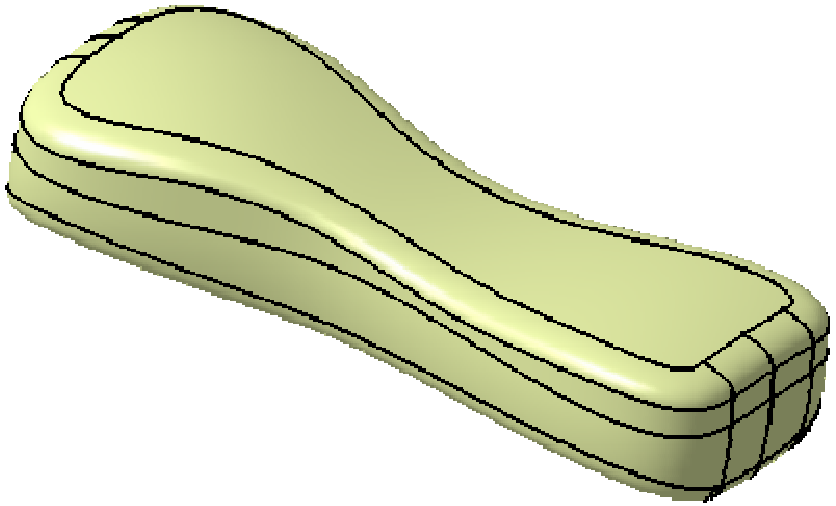
| Blend Type | Advantages | Disadvantages |
|---|---|---|
| Fillet | Simple to create | Cannot impose curvature continuity |
| Blend | Can impose curvature continuity | Very complex shape; can create unwanted features; must analyze |

# Advanced Blending Surface

## Recap Exercise

15 min

- Create the blend boundaries (upper and lower)
- Create the top blend using a Blend Surface
- Create the Shape fillet using Hold Curve and Spine at the bottom



Student Notes:

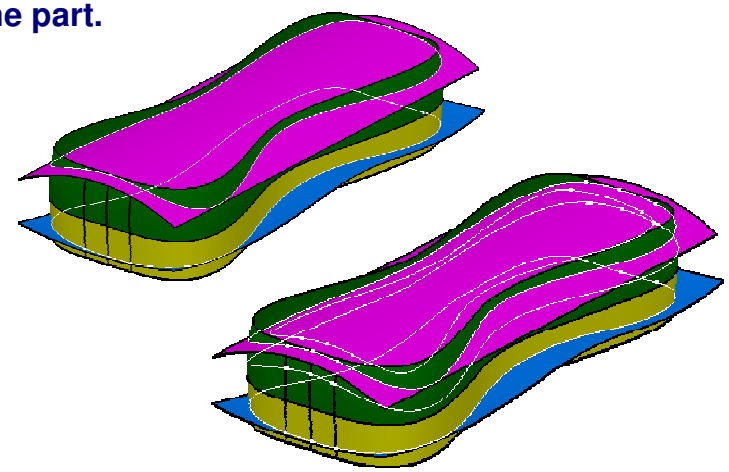
Student Notes:

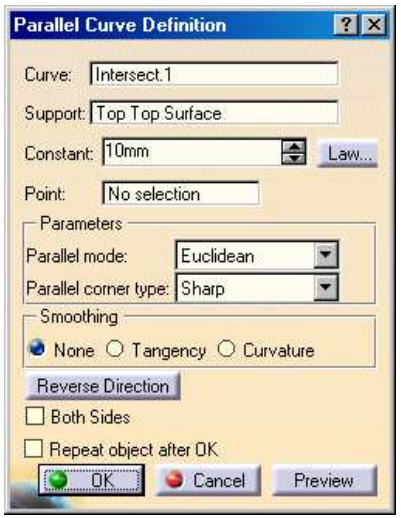
# Do It Yourself (1/3)

Part used: CATGSD_F_Advanced_Sweep_Surface_Recap_begin.CATPart

Create the blend boundaries on the top and bottom of the part.

- Create an Intersect.
  - Select the Pink and Green Surfaces.
  - Select OK. The Intersection curve is created.
- Create an Intersect.
  - Select the Yellow and Blue Surfaces.
  - Select OK. The Intersection curve is created.
- Create a Parallel Curve.
  - Select the Intersection curve created on the top of the part.
  - Specify the pink surface as the support.
  - Specify an offset of 10mm towards inside of the part.
  - Select OK. The curve is created.
- Create a Parallel Curve.
  - Select the Intersection curve created on the bottom of the part.
  - Specify the green surface as the support.
  - Specify an offset of 5mm downward.
  - Select OK. The curve is created.

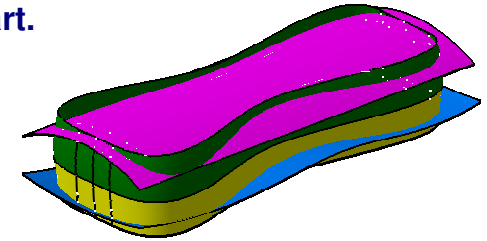




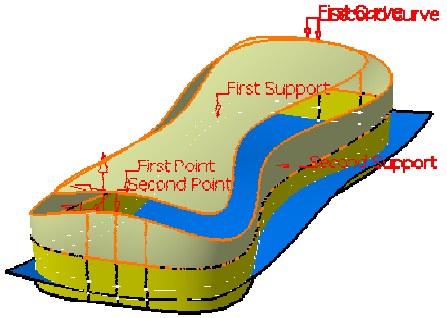
Student Notes:

# Do It Yourself (2/3)

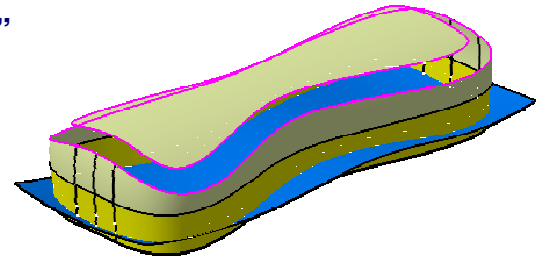
- Create a Parallel Curve.
  - Select the Intersection curve created on the Bottom of the part.
  - Specify the Yellow surface as the support.
  - Specify an offset of 5mm towards upwards.
  - Select OK. The curve is created.



- Create a Split.
  - Specify the pink surface as the “Element to cut.”
  - Specify the parallel curve lying on it as the “Cutting element.”
  - Make sure the proper side is retained.
  - Select OK.The surface is created.




- Create a Split.
  - Specify the Green surface as the “Element to cut.”
  - Specify the parallel curve lying on it as the “Cutting element.”
  - Make sure the proper side is retained.
  - Select OK.The surface is created.

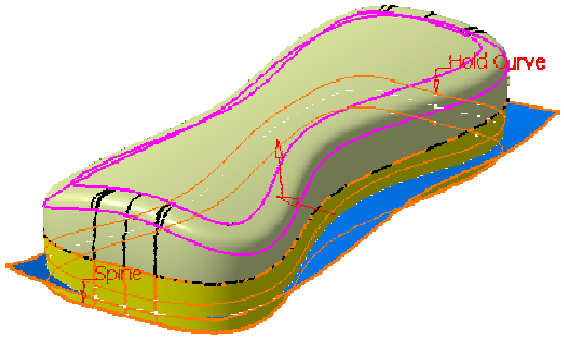


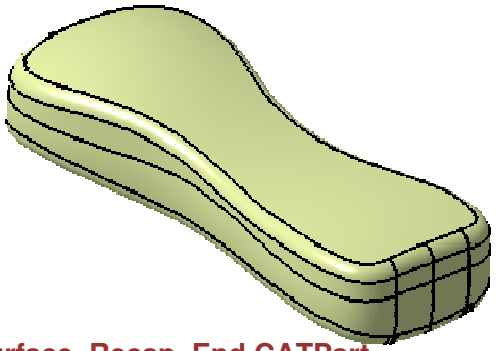
Student Notes:

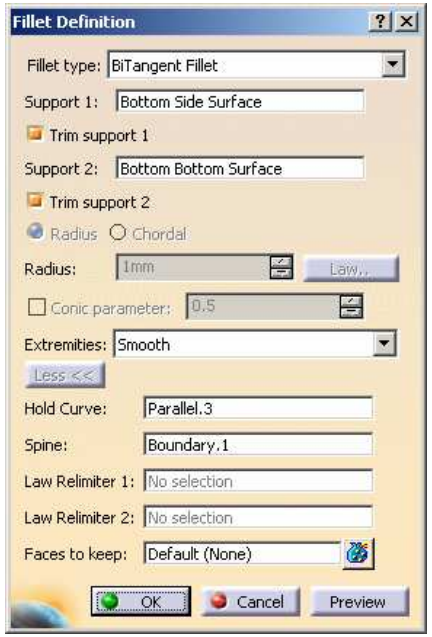
# Do It Yourself (3/3)

- Create a BLEND on the top of the part.
  - Specify the “Curves” and “Supports” as shown.
  - Specify Curvature continuity at each boundary.
  - Specify a Tension of a constant 1.0 at each boundary.
  - Select OK. The surface is created.

- Create a Bi-Tangent Fillet with Hold Curve and Spine at the bottom of the part.
  - Extract the bottom Boundary of Yellow surface as shown.
  - Click on the Shape Fillet icon
  - Specify the parameters as shown.
  - Select OK. The surface is created.






End Part:CATGSD_F_Advanced_Blend_Surface_Recap_End.CATPart

Student Notes:

# Surface Analysis

*This lesson will cover the following Surface Analysis and Repair topics:*

- About this Lesson
- Curvature Analysis
- Surface Analysis -Recommendations
- Recap Exercise: Surface Analysis

Student Notes:

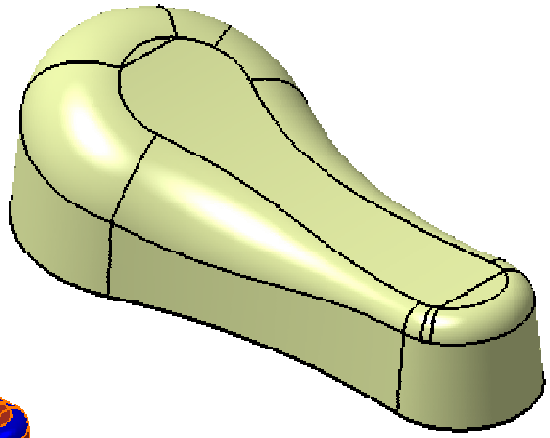
# About this Lesson

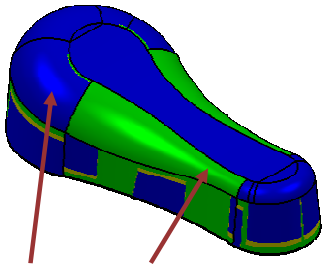
In this lesson you will learn and practice the advance tools of 'Surface Analysis'.

At the end of this Lesson, you will identify and analyze Inflection on surfaces and also find the minimum inside radius on the part.

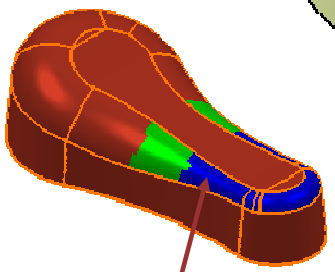
In this lesson you will learn a few functionalities of surface analysis such as,

- Curvature Analysis
  - Measuring mean curvature on a surface
  - Measuring Minimum and Maximum Curvature
  - Checking the inflection area on the Surface





Inflection Analysis on surface



Finding minimum inside radius on the part

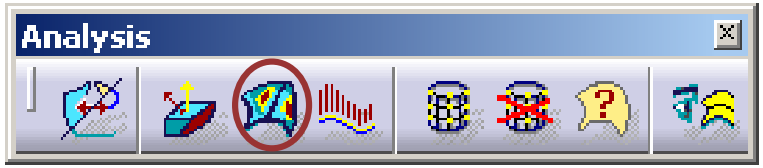
Simple visual inspection of a part will not always uncover surface flaws. Hence the surfaced geometry should always be analyzed using above tools to achieve better quality surfaces.

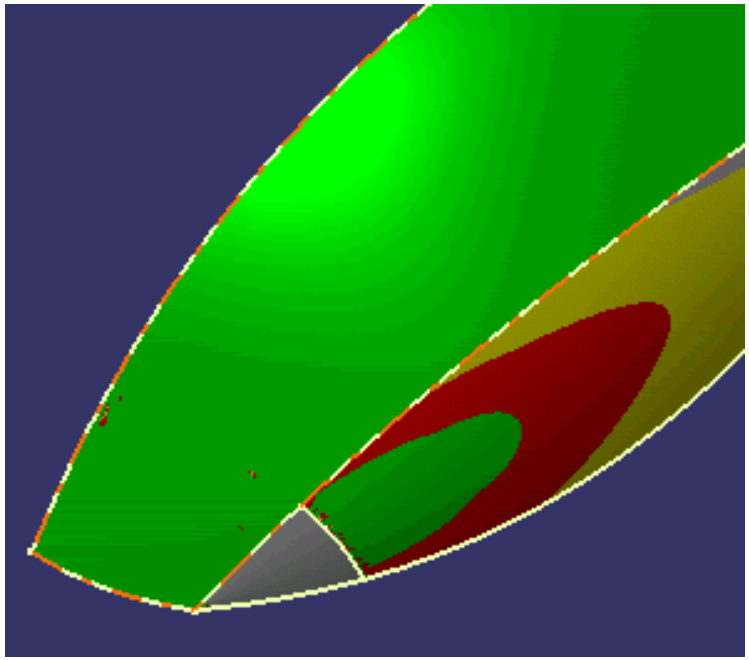
You will also see a few Surface Analysis recommendations at the end of the lesson.

Student Notes:

# Curvature Analysis

*You will learn how to use the Mapping Analysis tool to analyze surface curvature*



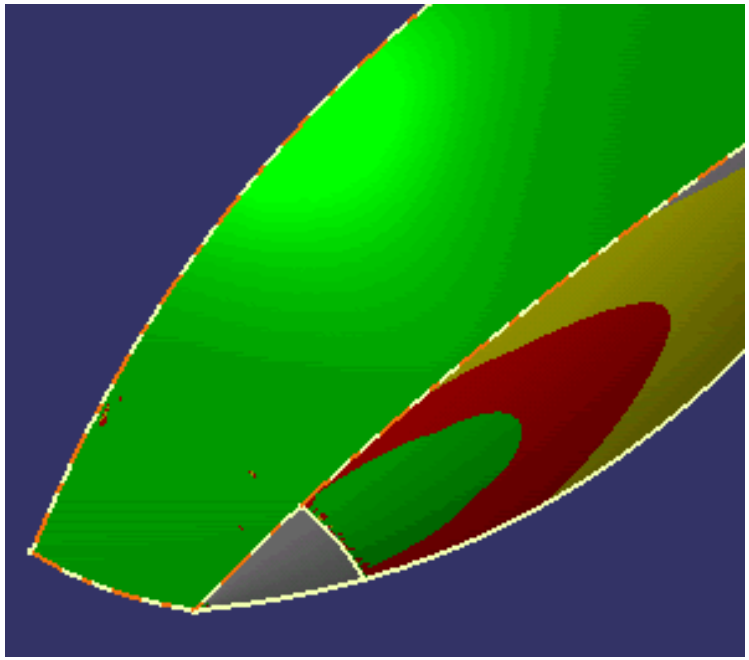



Student Notes:

# Why use Curvature Analysis?

**Curvature analysis of surfaces are generally used to help model high quality surfaces.**

**Abrupt changes of curvature on a surface (for example on a car exterior body) can be easily seen by the naked eye and must be smoothed.**



Student Notes:

# What is a Curvature Analysis? (1/2)

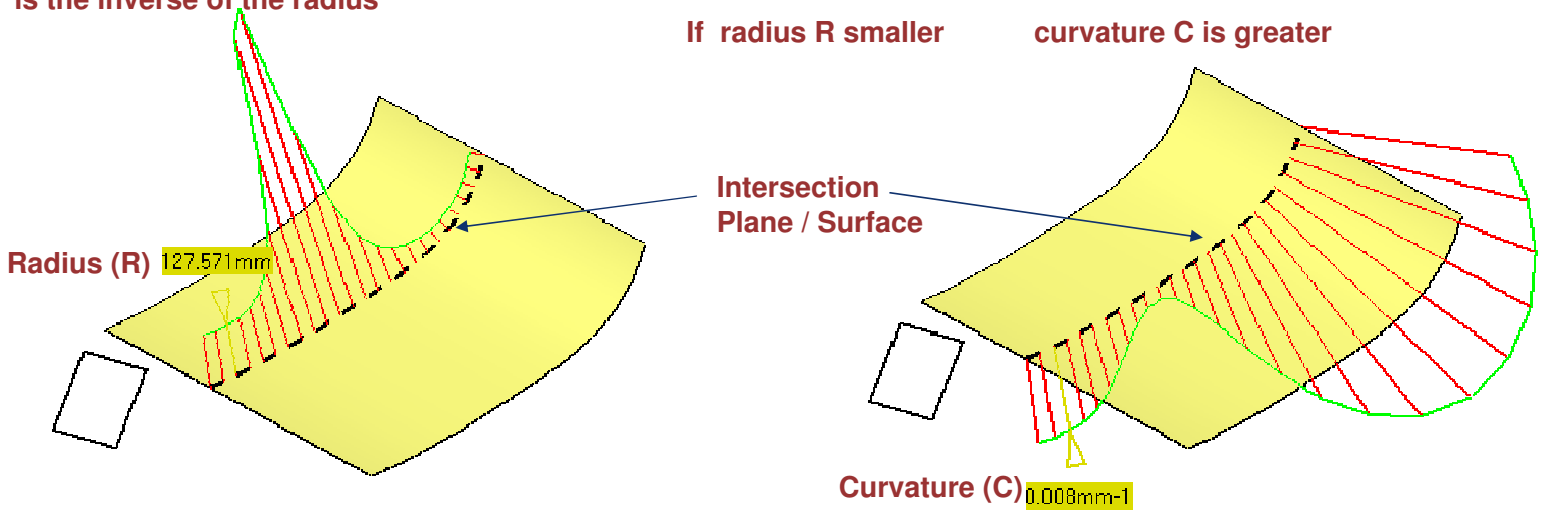
**Curvature analysis of surfaces is used to help detect the imperfections on surfaces. Abrupt changes of curvature on a surface can be easily seen by the naked eye and must be smoothened. The curvature analysis measures the curvature on each point of a surface according to the following method :**

**Curvature radius in one point (R): represents the local convexity of the surface**

**The curvature in one point (C): C = 1 / R**
**is the inverse of the radius**

**If radius R greater curvature C is smaller**

**If radius R smaller curvature C is greater**



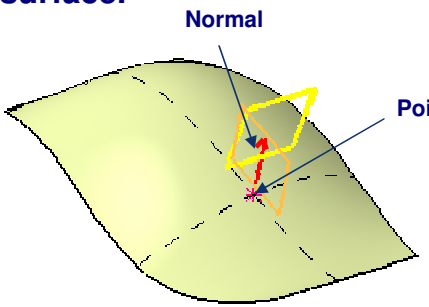

**Radius measure of the surface intersection with a cutting plane**

**Curvature measure of the surface intersection with a cutting plane**

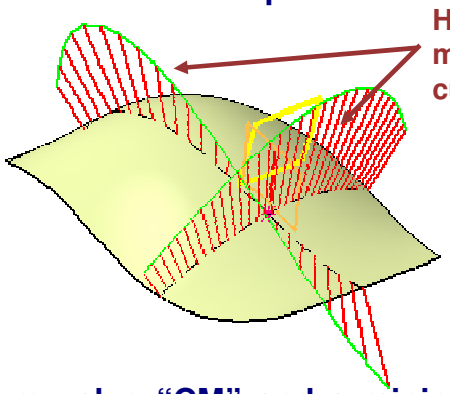
Student Notes:

# What is a Curvature Analysis? (2/2)

If we rotate planes around the normal vector on a point of the surface, we can build the intersection of these planes with the surface.

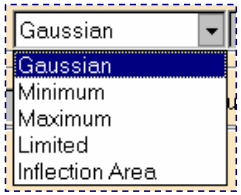


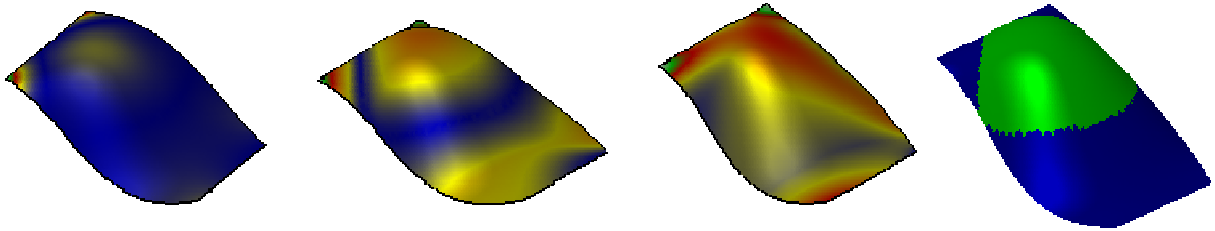

On these intersection curves we can measure an infinite number of curvature values for this point.




At each point we will have a maximum curvature value “CM” and a minimum curvature value “Cm.”

The Mapping analysis tool allows you to measure these minimum and maximum values (Minimum/Maximum analysis), the mean value (Gaussian analysis) and to see the inflection areas.



Gaussian $\sqrt{|C=|}$ CM.Cm

Minimum

Maximum

Inflection area

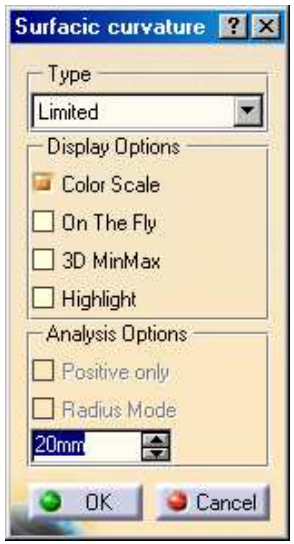
Student Notes:

# Measuring the Mean Curvature on a Surface.

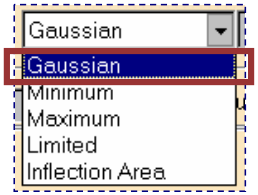
1. Select the customized view render style :

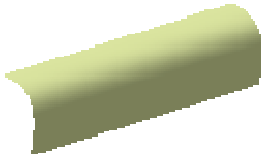
2. 


3. Select the surface where you want to examine the curvature:

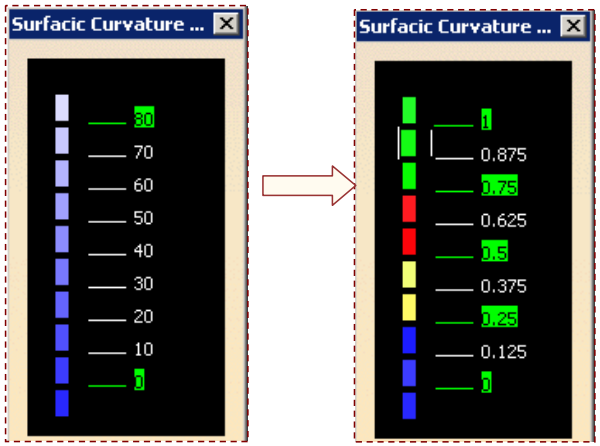



4. Select Gaussian as analysis type :

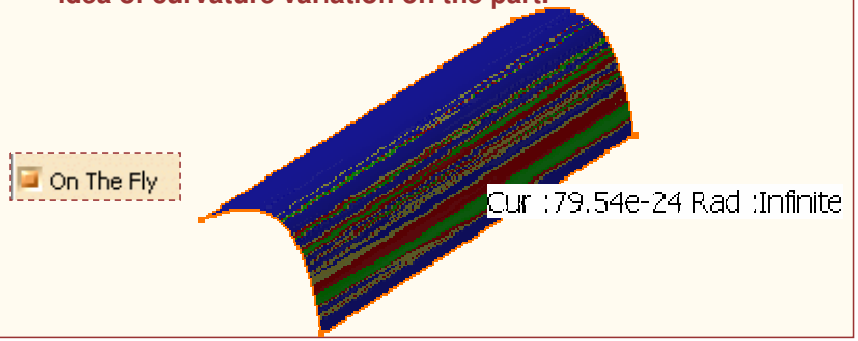


5. Adjust the color range fields by right clicking on the thresholds values.




Pass the mouse over the surfaces and read the curvature values shown in order to get a general idea of curvature variation on the part.



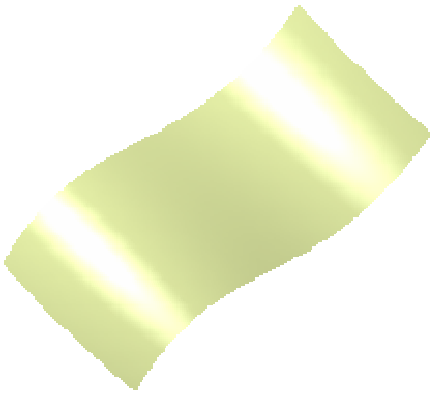
Student Notes:

# Measuring the Minimum or Maximum Curvature on a Surface.

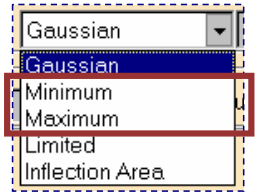
1. Select the customized view render style :

2. 

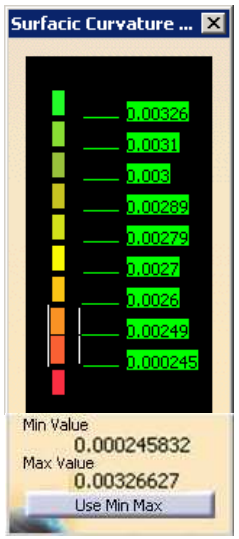

3. Select the surface where you want to examine the curvature:

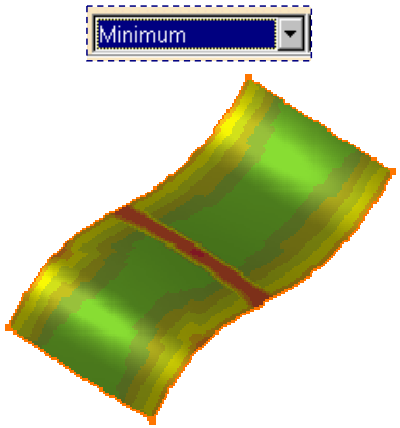


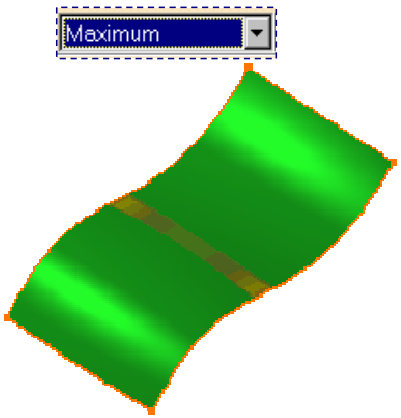
4. Select Minimum or Maximum as analysis type :



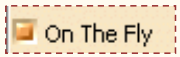

5. Adjust the color range fields right clicking on the thresholds values and on the colours boxes.




Pass the mouse over the surfaces and read the curvature values shown in order to get a general idea of curvature variation on the part.



Student Notes:

# Checking a Surface Using the Limited Radius

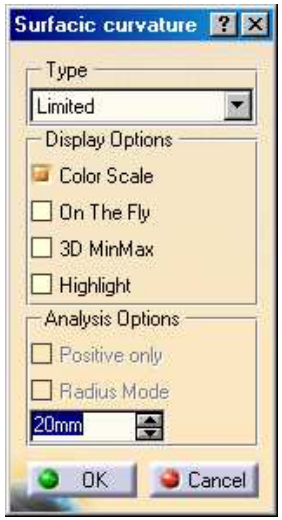
**Use the Limited Radius analysis to check if the surface can be offset or to check if a tool (an end mill) with an end radius can mill the part.**

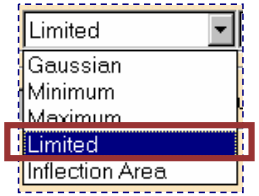
1. Select the customized view render style :

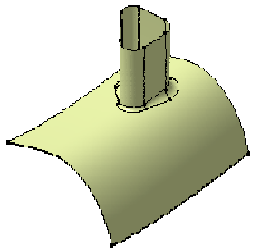
2.


3. Select the surface where you want to examine the curvature:

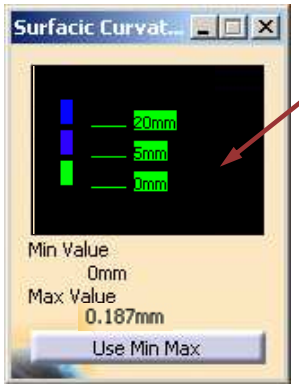
4. Select Limited as analysis type :

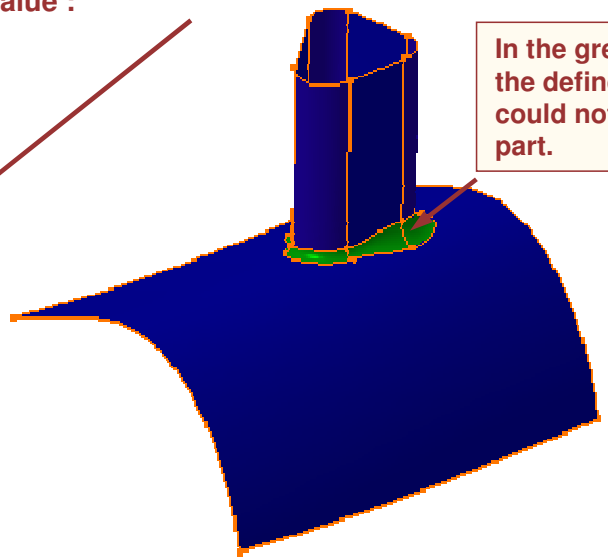


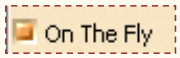
5. Set the limited radius value :



In the green area, the defined tool could not mill the part.

Pass the mouse over the surfaces and read the curvature values shown in order to get a general idea of curvature variation on the part.

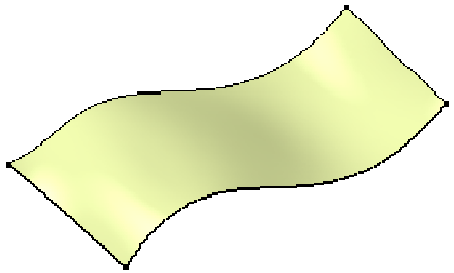


Student Notes:

# Checking the Inflection Areas on Surfaces.

**Using the Inflection Area analysis type you can see where the curvature sign changes.**

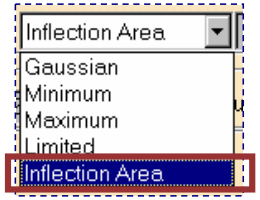
0 Select the customized view render style :

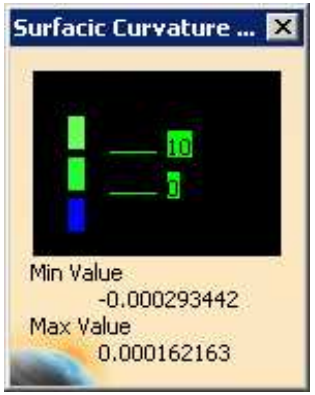


1

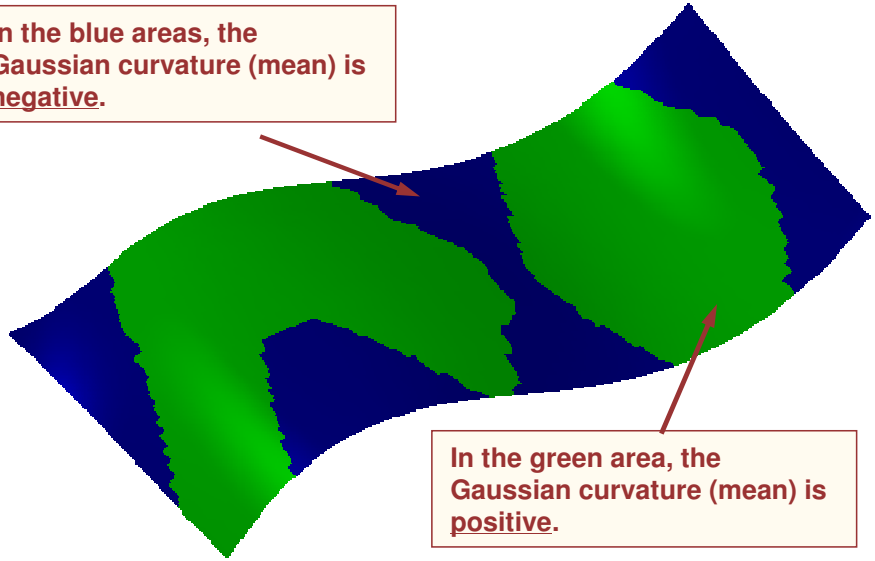

2 Select the surface where you want to examine the curvature:

3 Select Inflection Area as analysis type :




In the blue areas, the Gaussian curvature (mean) is negative.

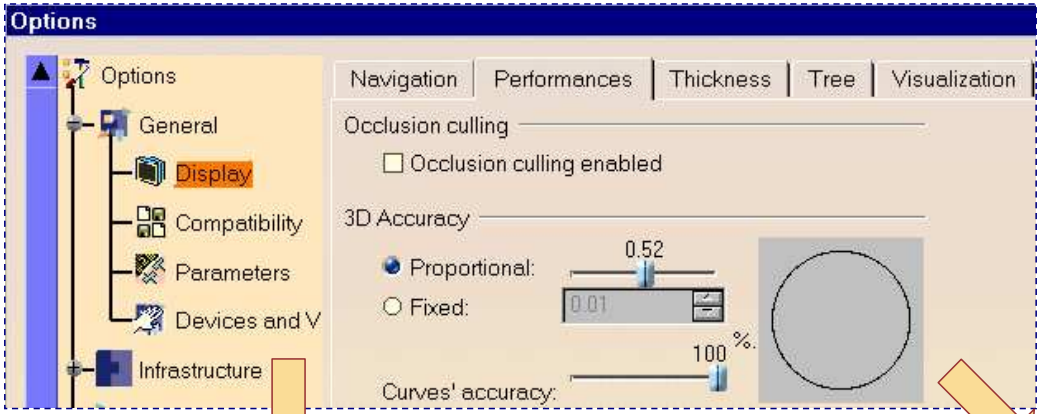


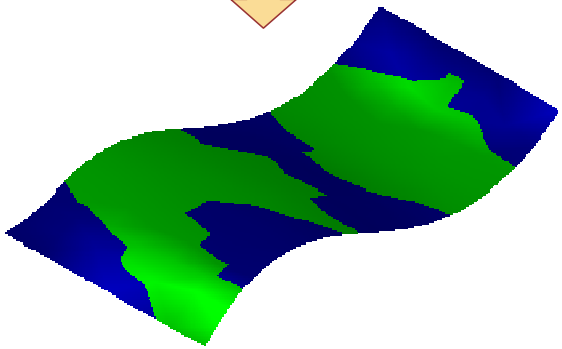
In the green area, the Gaussian curvature (mean) is positive.

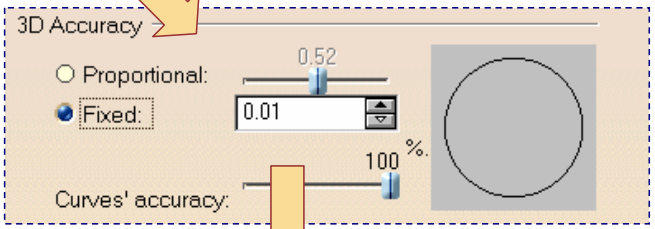
Student Notes:

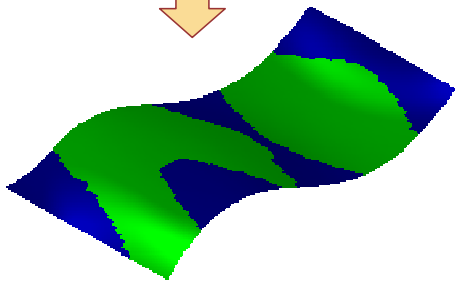
# Additional Information on Curvature Analysis

**The Analysis is calculated on the mesh used to display the object, the precision of the analysis depends on the display settings.**





**Fix the 3D Accuracy to the minimum value to have a better analysis rendering.**

STUDENT GUIDE

Student Notes:

# Surface Analysis – Recommendations

*You will be given specific methods and recommendations concerning Surface Analysis and Repair.*

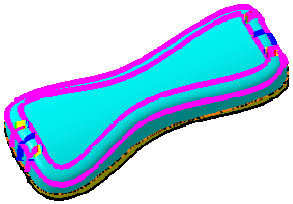
Student Notes:

# Tips on performing Surface Analysis

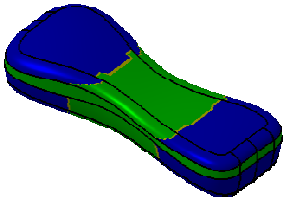
**As a general rule of thumb, the following tools of GSD workbench are useful to obtain an initial assessment of the surface quality of a part.**

1. **Visual – make sure you place a metallic material on the part and shade in material mode. Also reduce the "Accuracy" values in Tools + Options + General + Display + Performances.**

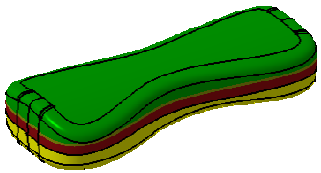
2. **Surface Connect Checker – verify internal connections on the surface. Make sure they conform to specifications.**



3. **Inflection Area Analysis (under Surfacic Curvature Analysis tool) – look for color changes that are not in areas where inflection changes are expected.**



4. **Draft Analysis – make sure the part conforms to all molding or forming criteria.**

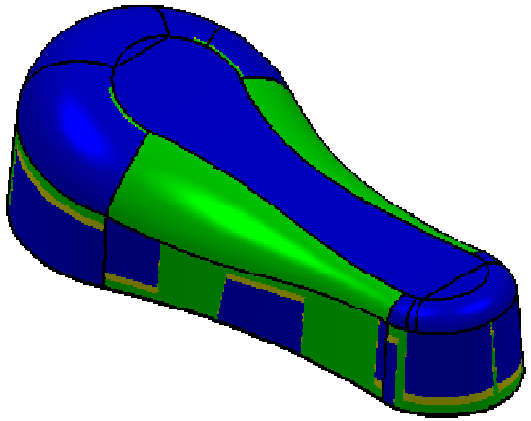


# Surface Analysis

## Recap Exercise

15 min

- Perform an Inflection Analysis
- Find the minimum inside radius



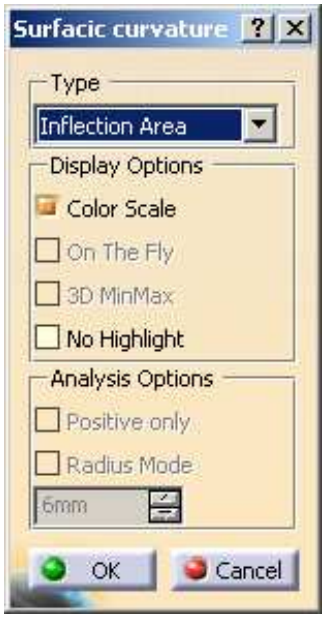
Student Notes:

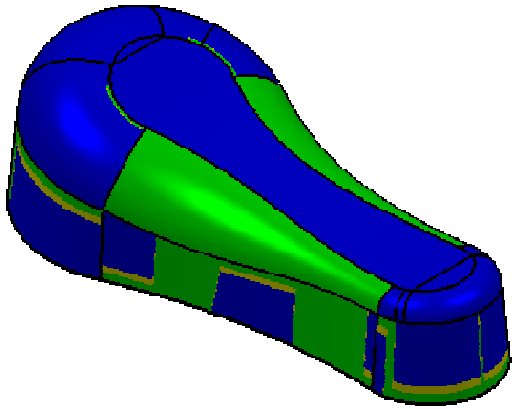
Student Notes:

# Do It Yourself (1/2)

**Part used: CATGSD_F_Surface_Analysis_Recap.CATPart**

- **Perform an Inflection Analysis on the part.**
  - **Click on the Surfacic Curvature Analysis icon.**
  - **Choose type “Inflection” area.**
  - **Select the seat surface.**



**Notice areas of green and blue color. You are looking for is areas of infection that are out of place. On this surface, all the color changes are reasonable**

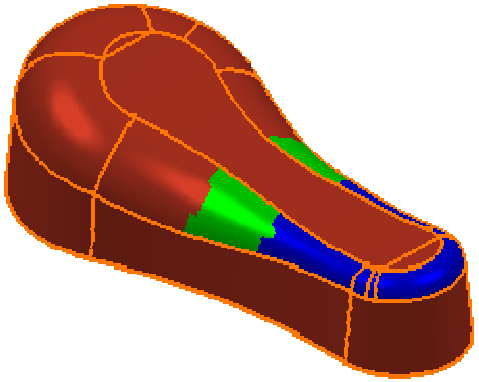
Student Notes:

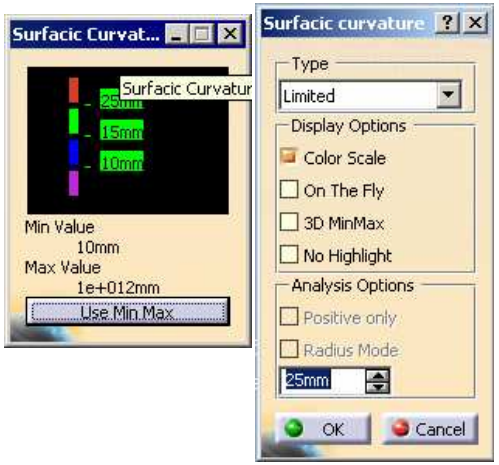
# Do It Yourself (2/2)

- Delete the Inflection Analysis from the specification tree.

- Find the minimum inside radius on the part.
  - Click on the Surfacic Curvature Analysis icon.
  - Choose type “Limited.”
  - Make sure the panels appear as shown.
  - Select the “Whole top surface.”

Notice areas of color indicating minimum radius. Notice the minimum radius in the panel for this surface. This may be critical information for Homologation or for manufacturing.





Student Notes:

# Additional Surface Design Tools

*You will learn the tools which are commonly used to design a surfacic model and Molded Parts.*

- Improving Geometry Stability
- Improving Geometry Quality
- Checking Molded Parts

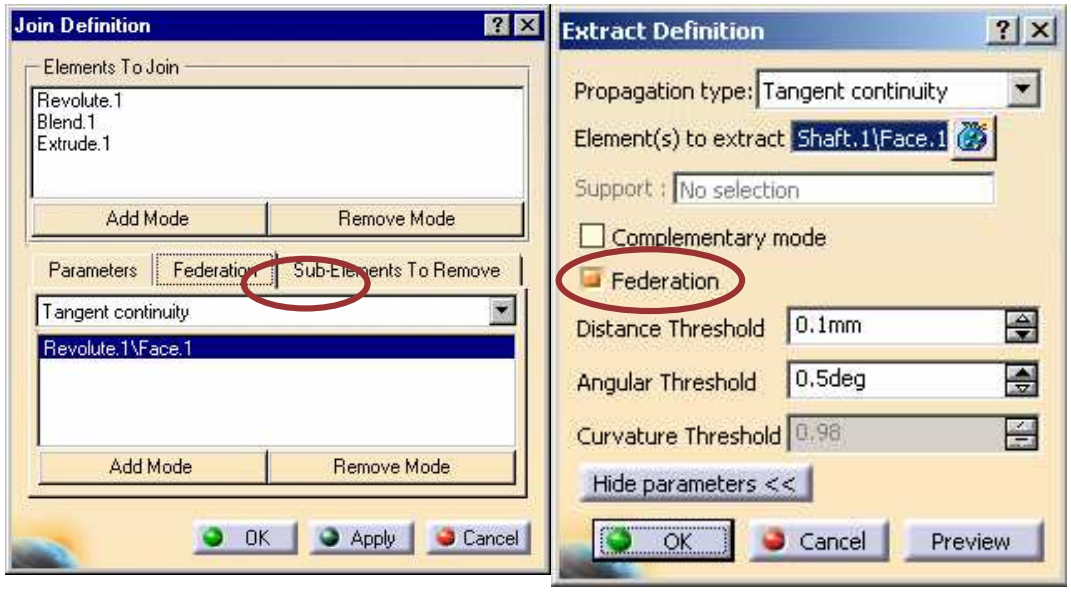
Student Notes:

# Improving Geometry Stability

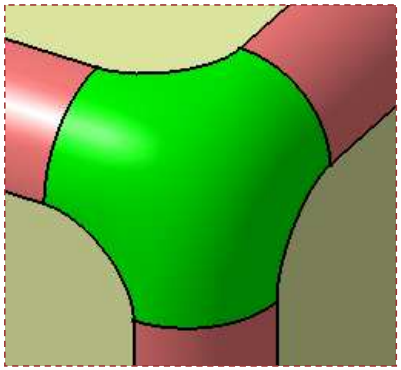
*You will learn about few surfacic tool which are used to improve geometry stability.*

- Federating Elements
- Blending Vertex when making fillets

Federating Elements

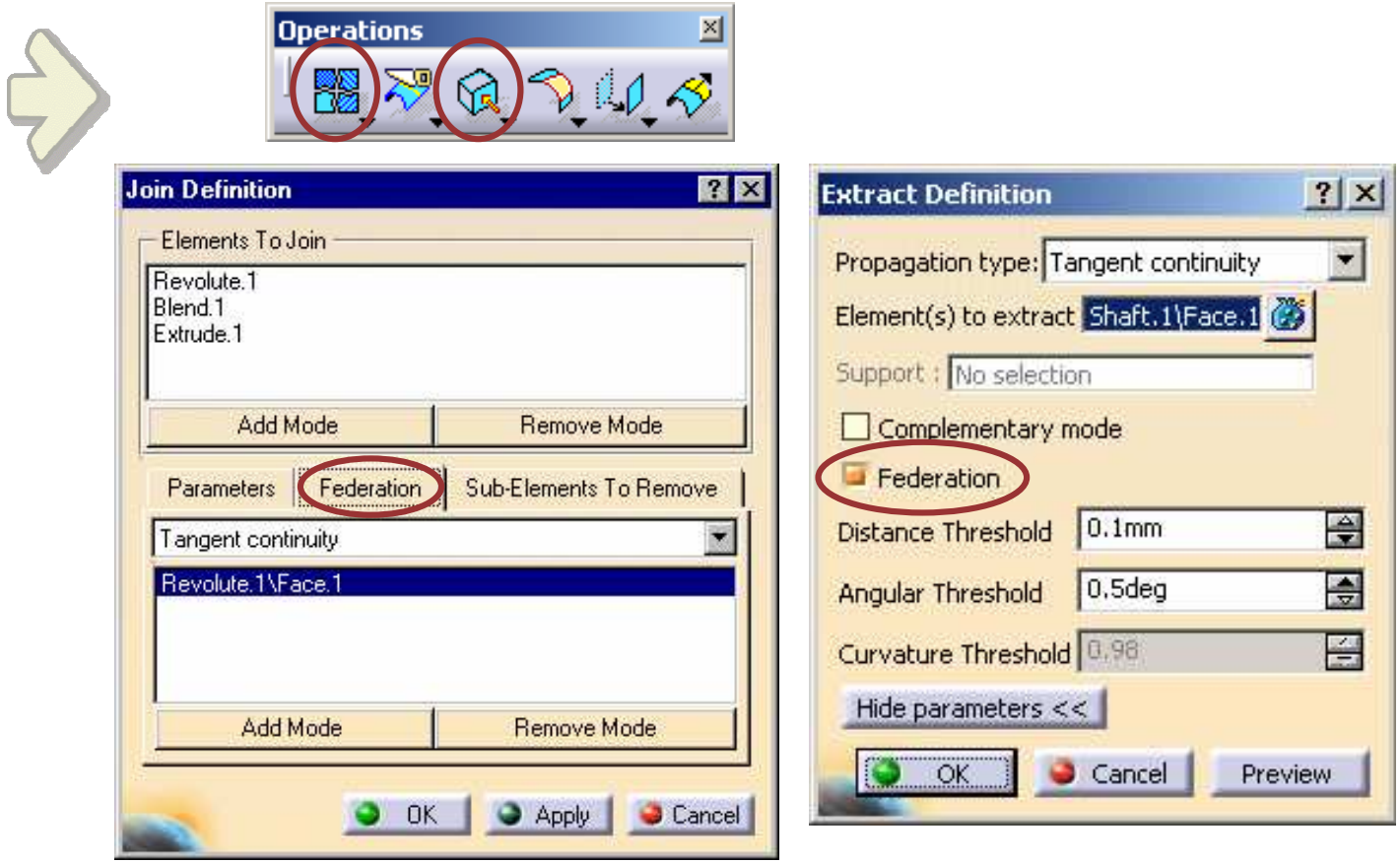

Blending Vertex when making fillets



Student Notes:

# Federating Elements

*You will learn how to federate elements while joining surfaces and extracting faces*

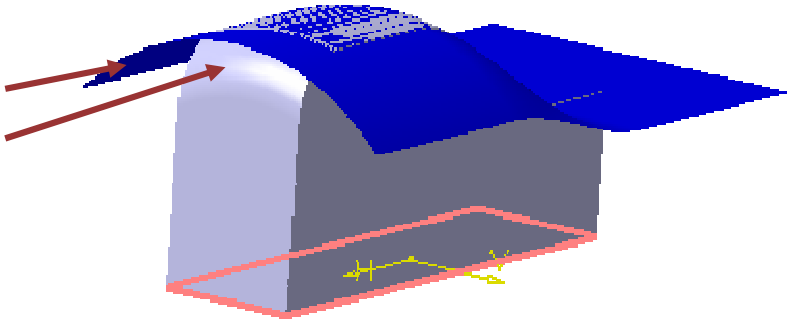


Student Notes:

# Why Federate ? (1/2)

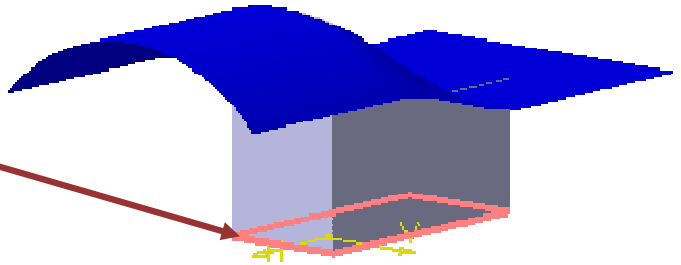
**1- Surfaces are made of several faces.**
**Elements created from a surface are in fact created from its faces.**

The pad has been created with the option “Up to surface”, using the blue surface.
A fillet has been added to the top edge of this pad.
This edge depends on a face of the blue surface.



**2- A modification of the part geometry may lead to a change of the supporting face.**

The sketch supporting the pad has been modified so that the filleted edge does not lie anymore on the same face



Student Notes:

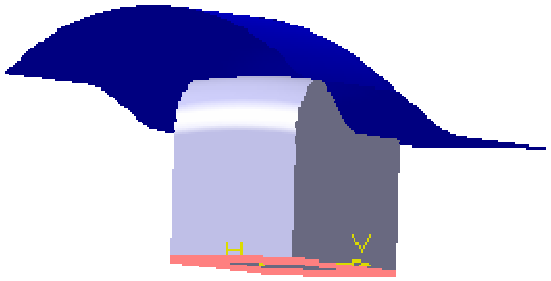
# Why Federate ? (2/2)

**3- This change can lead to an update error because the elements created from these faces are no longer recognized.**


**During the update of the part, an update error occurred : the filleted edge is not recognized :**

**4- Federating the faces of the surfaces, this kind of update error does not occur anymore.**

**To solve the problem, you just have to federate the faces of the blue surface. Then the part is updated without any problem :**

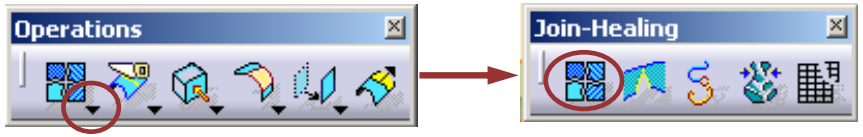


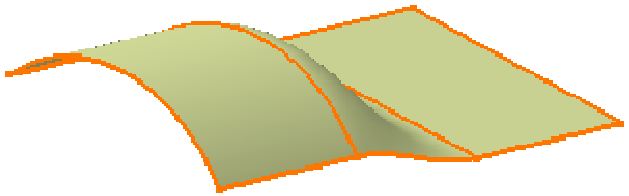
Student Notes:

# Federating Elements while Joining Surfaces

**Joining surfaces, you have the possibility to federate the faces of the resulting surface**

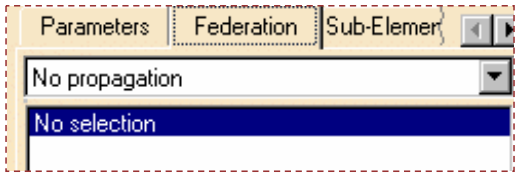
1

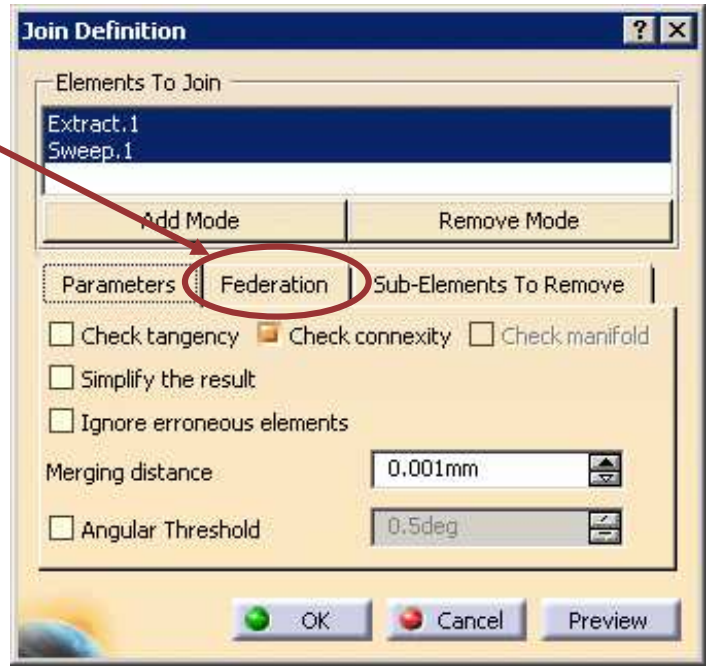


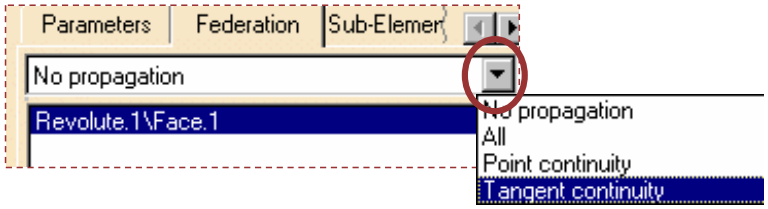
2 **Select one by one the elements to be joined together.**

3 **Expand the new "Federation" panel in the join dialog box.**




4 **Select one face of the join surface and choose a propagation type.**




5 **Click OK to create the federated joined surface.**

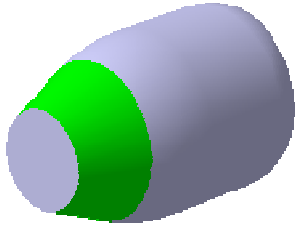
Student Notes:

# Federating Elements while Extracting Faces

**Extracting faces from a solid, you have the possibility to federate the faces of the resulting surface**

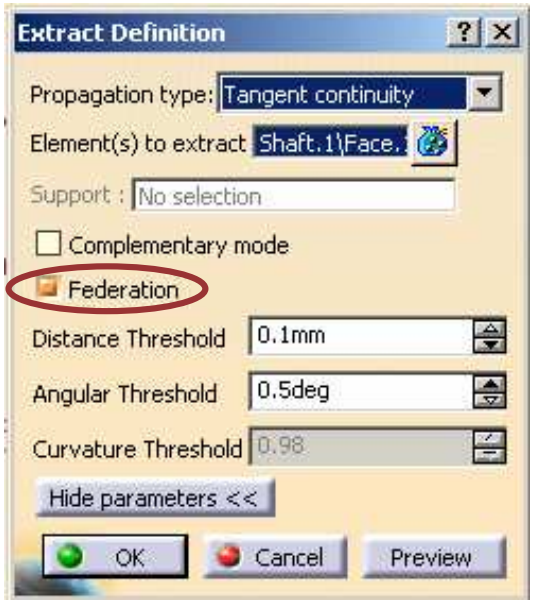
1





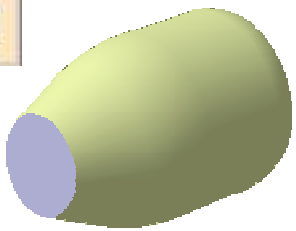
2 Select one face of the solid.

3 Choose a propagation type.




4 Activate the federation switch.

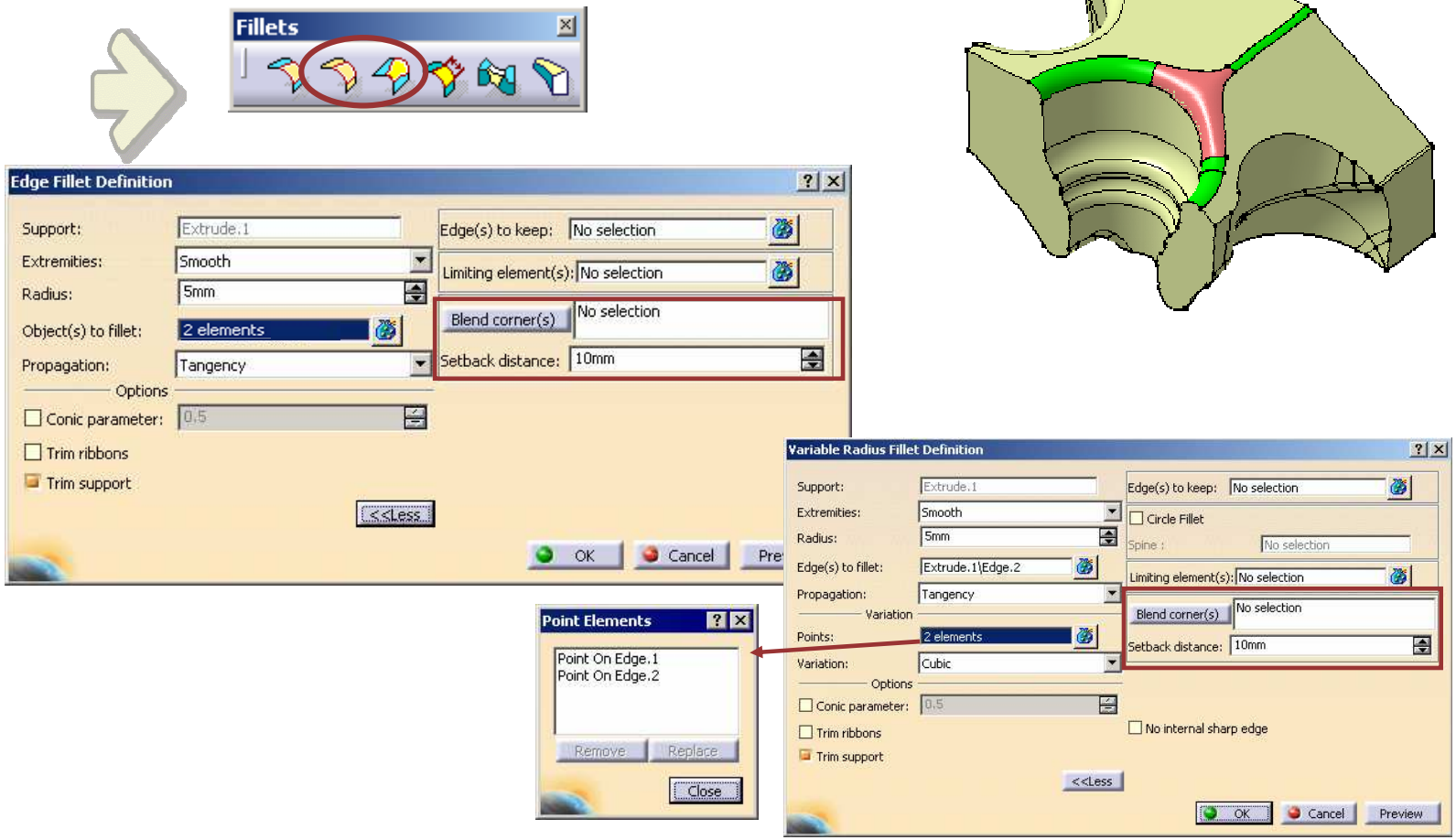
5 Click OK to create the federated extracted surface.



Student Notes:

# Blending Vertex when Making Fillets

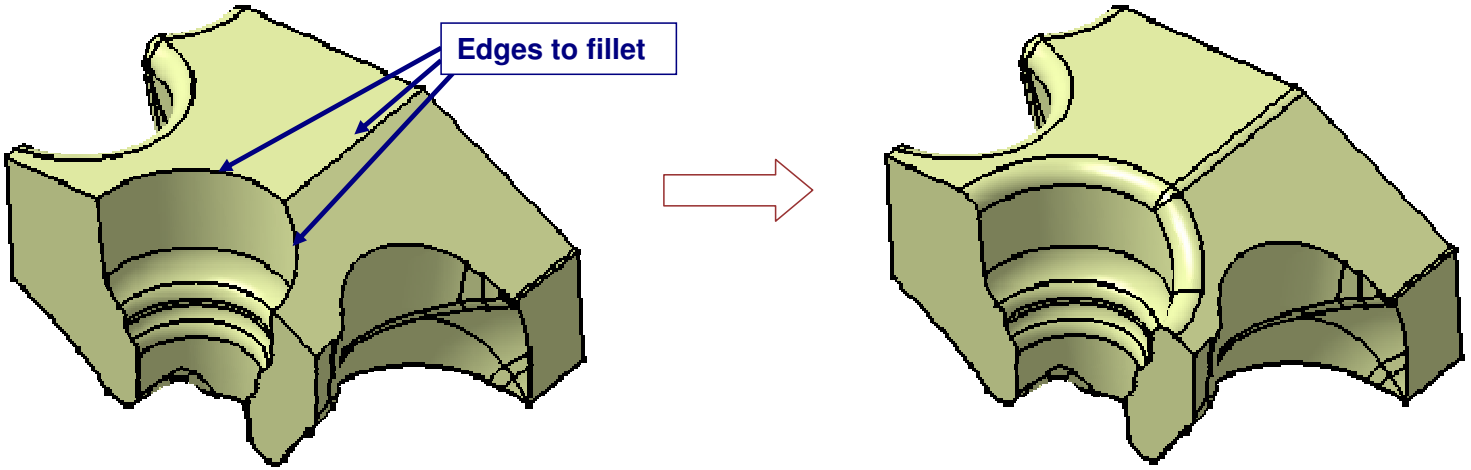
*You will see how to easily make fillets on sharp edges that are incident to a sharp vertex point*



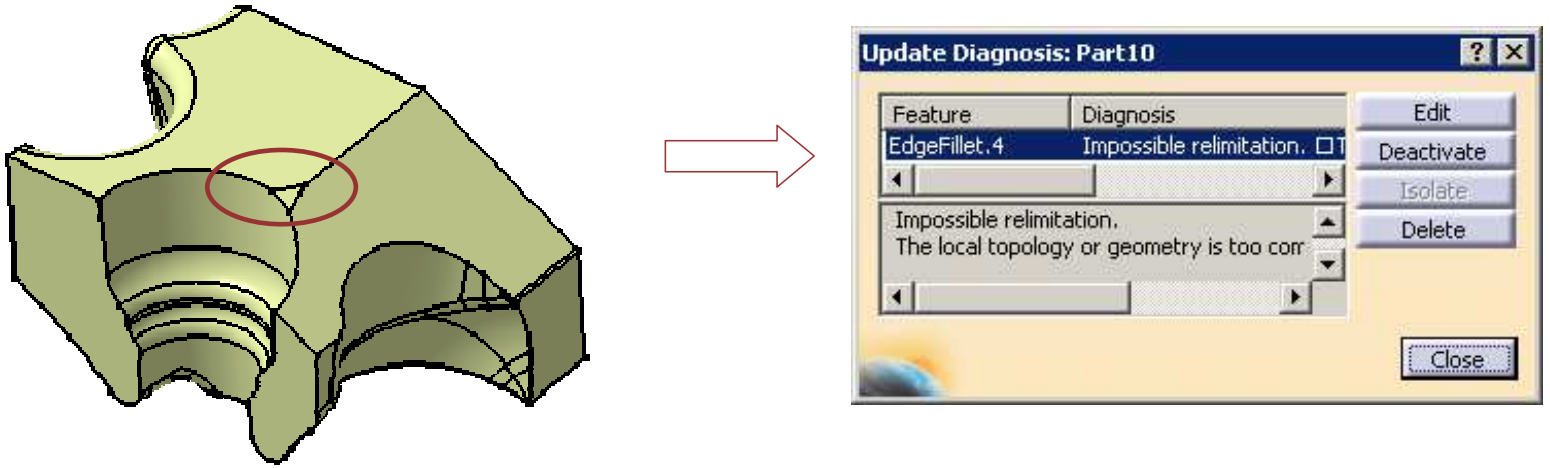
Student Notes:

# Why this Tool ?

**When the initial geometry on which lies a sequence of fillets is modified, the sequence of fillets may fail and the designer may have to have a new fillet sequence :**




**If the initial geometry is modified, the fillet sequence cannot be recalculated :**



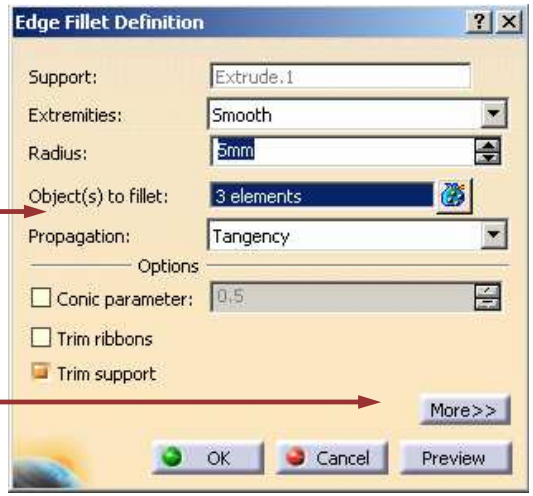

**The Blend Vertex allows you to make fillets that are more stable during the modifications …**

Student Notes:

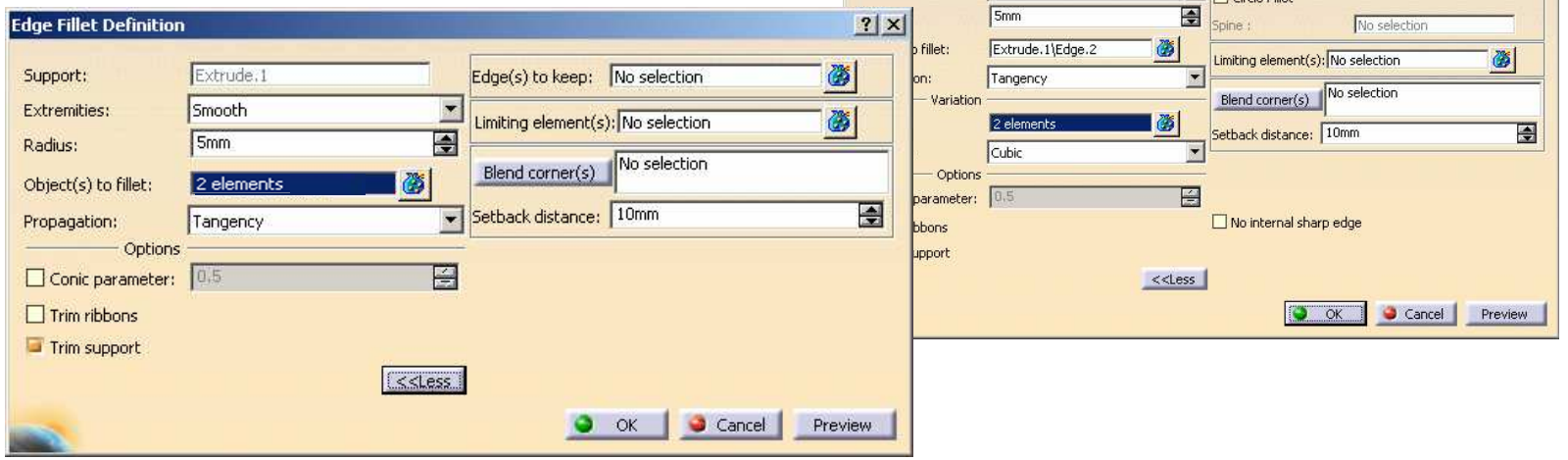
# Blending Vertex when making Fillets (1/2)

1


2 Select the edges on which you want to make fillets :




3 Click on the “More” button to expand the fillet dialog box :

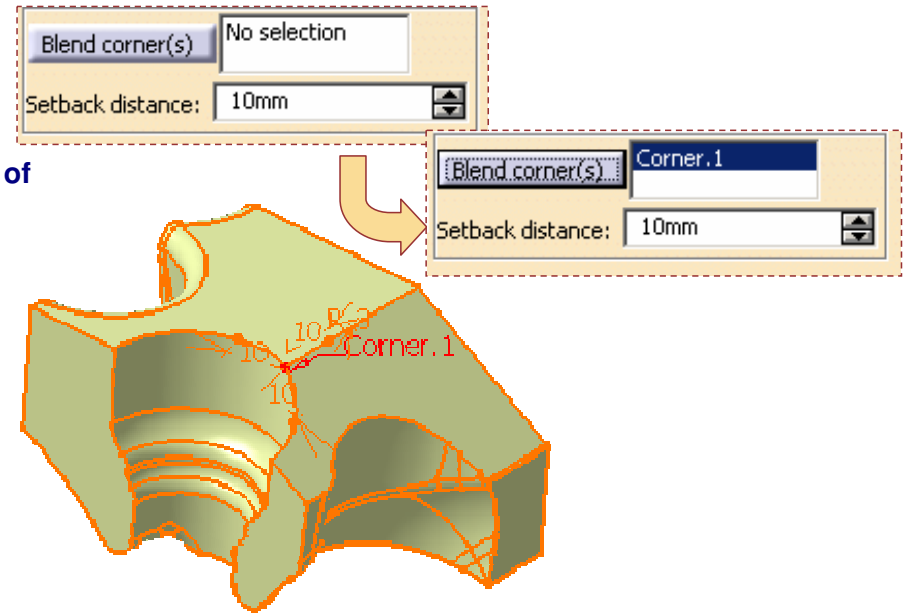


# Blending Vertex when making Fillets (2/2)

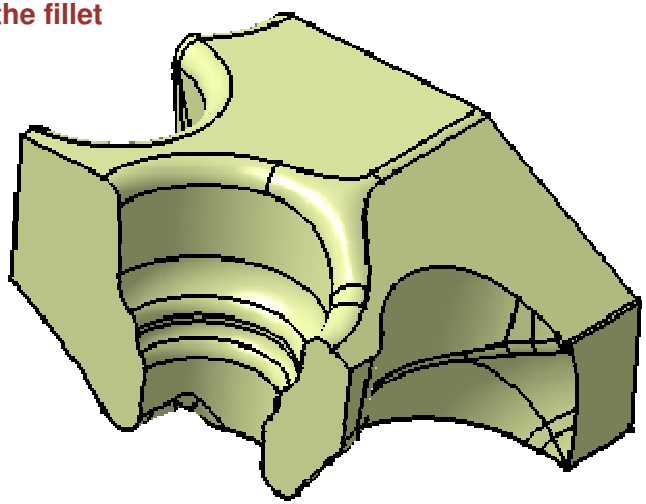
Student Notes:

4 Click on the “Blend Corner” button :

CATIA automatically detects the existence of a sharp vertex point common to the edges you selected :




5 Click on OK to confirm the fillet creation :

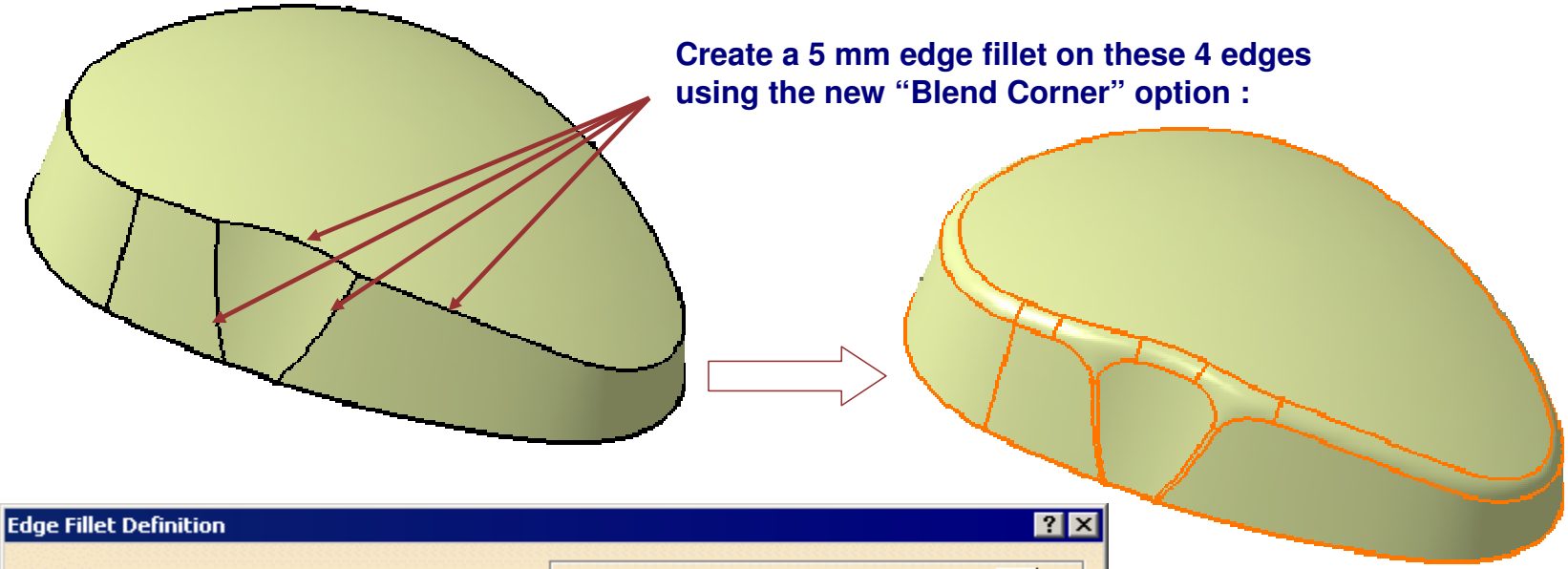


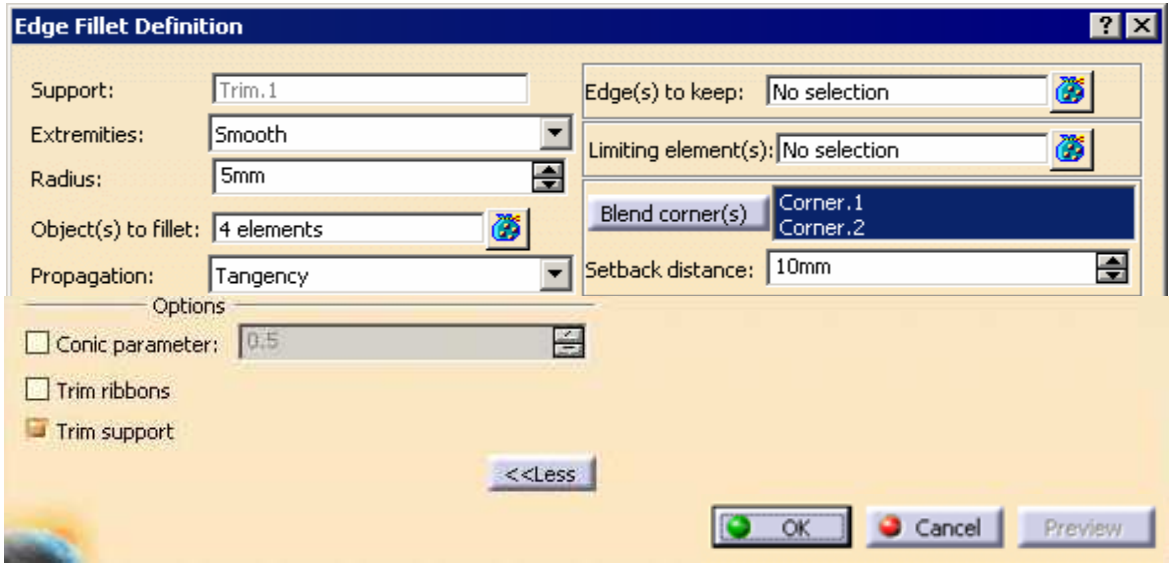
Student Notes:

# Do it Yourself

Part used: FilletBlendVertex.CATPart

**Create a 5 mm edge fillet on these 4 edges using the new "Blend Corner" option :**





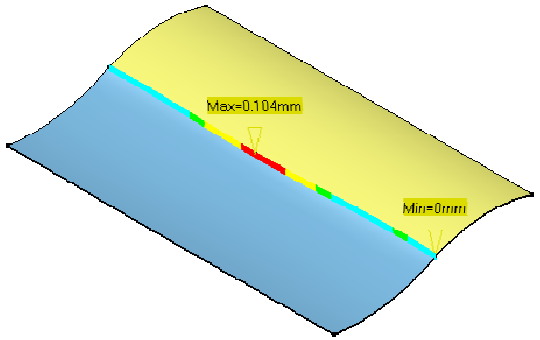
Student Notes:

# Improving Geometry Quality

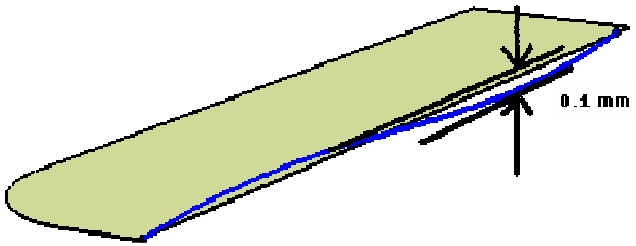
*You will learn about a few surfacic tools which are used to analyze and improve geometry quality.*

- Healing Surface
- Tolerant Modeling
- Connect Checker

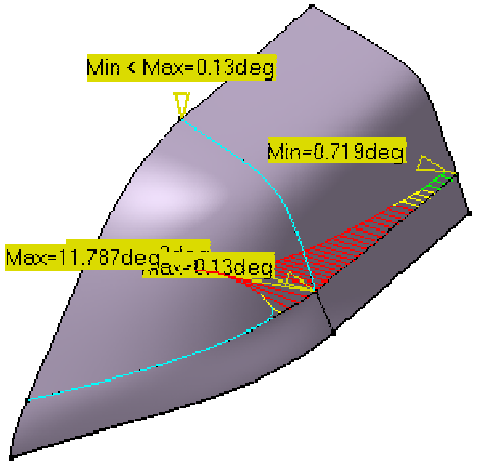
Healing Surfaces




Tolerant Modeling



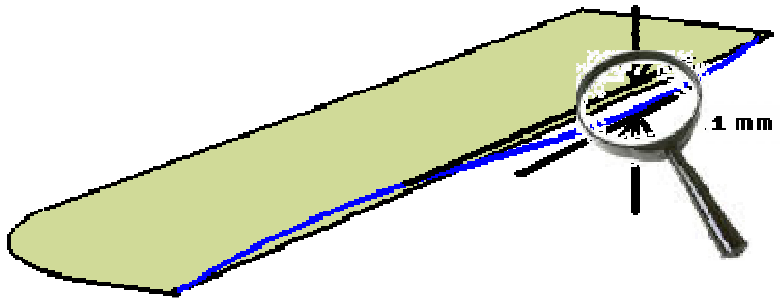

The Connect Checker

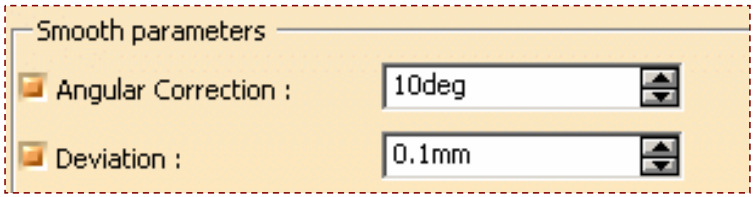


Student Notes:

# Tolerant Modeling

*You will see the tolerant Modeling Concept.*

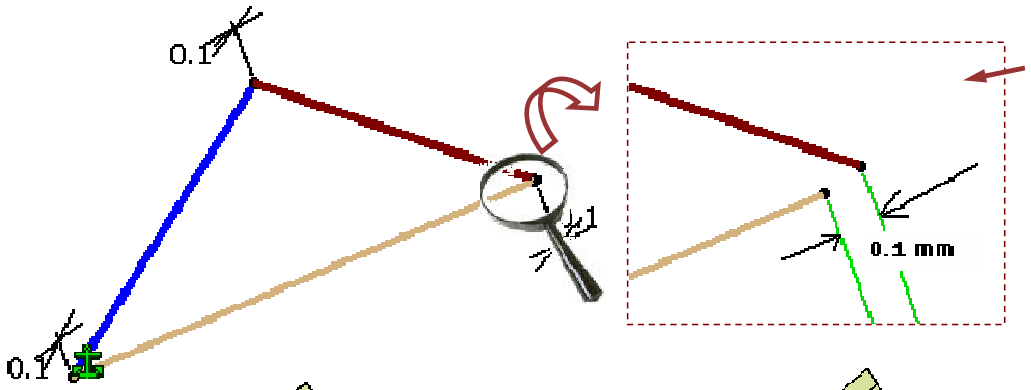


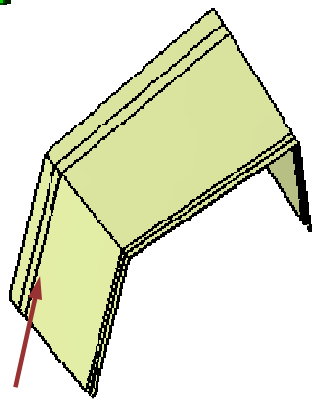
Student Notes:

# Why Tolerant Modeling?

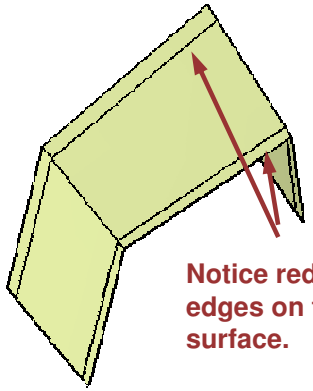
**Sometimes the input wireframe and surfaces are not of desired quality. Using these elements propagates the problem to child elements.**
**Tolerant Modeling aims at creating good results using imperfect inputs.**




As seen in this example there is a gap between the two curves. You will still succeed in creating a fill surface using the three curves , because of Tolerant Modeling.(The gap should not be more than 0.1mm).

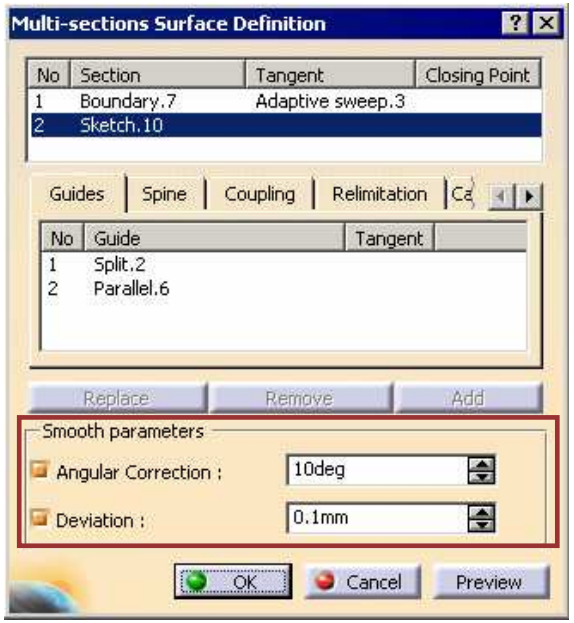


Before Application of Tolerant Modeling Parameters in Multisection Surface there are many edges.



Notice reduced edges on the surface.

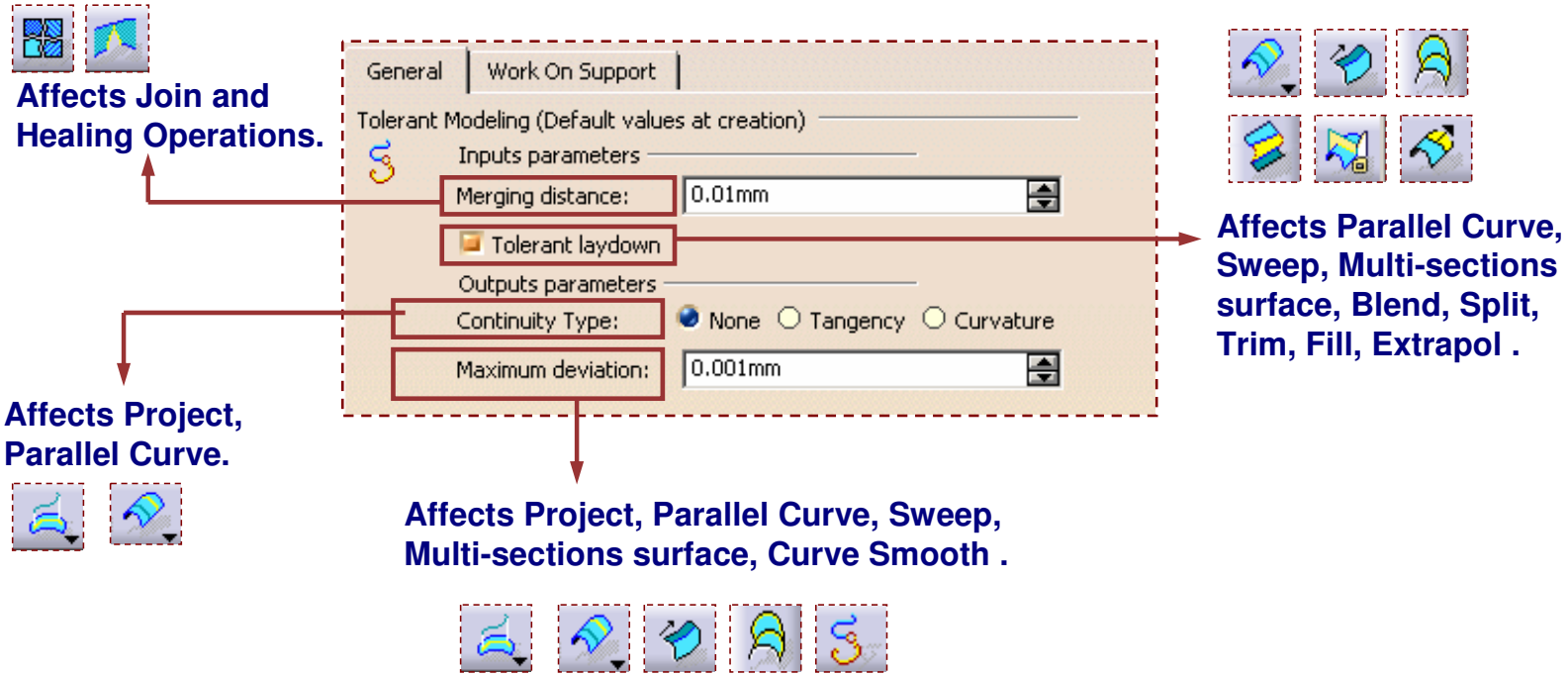
After Application of Tolerant Modeling Parameters the edges is reduced.



Student Notes:

# What About Tolerant Modeling

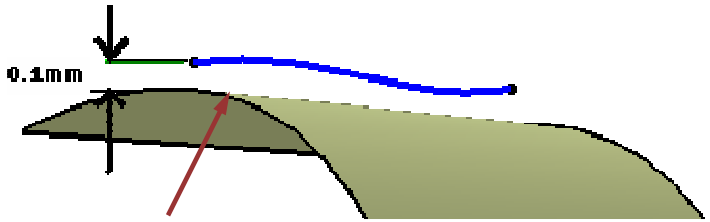
**Some of the tolerant Modeling facilities are inbuilt and no UI is available.**
**There are settings in Tools > Options > Shape >Generative Shape Design >General from where you can set some of the Tolerant Modeling parameters.**



Student Notes:

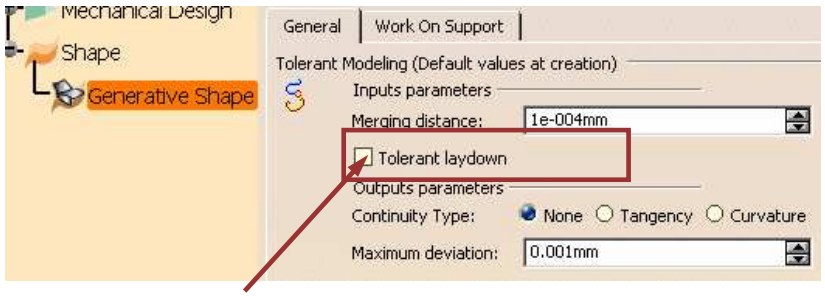
# What is Tolerant Laydown?

**Some features require wires that are laid down on their support shell. When those supports are created within a tolerance (tolerant modeling), this tolerance has to be used to lay down the wire on the support.**

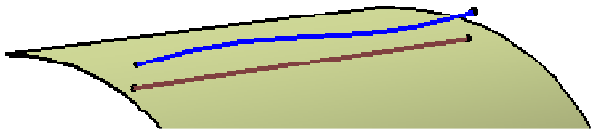



**The wire does not lie on the surface and the distance is 0.1 mm**


**If you try to create parallel curve on the surface, using the curve shown in the figure, CATIA will give an error as it is expected that the curve should lie on the surface.**




**Select this option to make tolerant laydown work. The limit for Laydown is 0.1 mm**



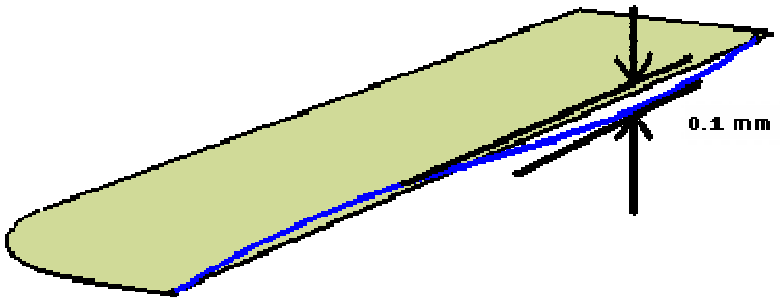
**Tolerant laydown projects the wire on the support within the tolerance.**

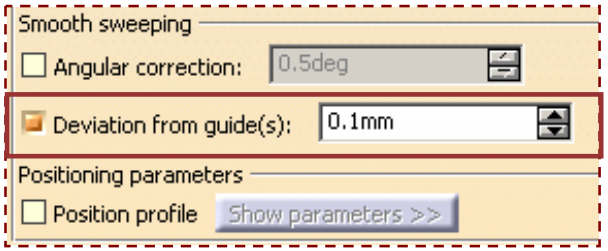
**The support must thus be large enough for the whole wire to be projected**

Student Notes:

# Tolerant Projection for Fill and Extrapolate

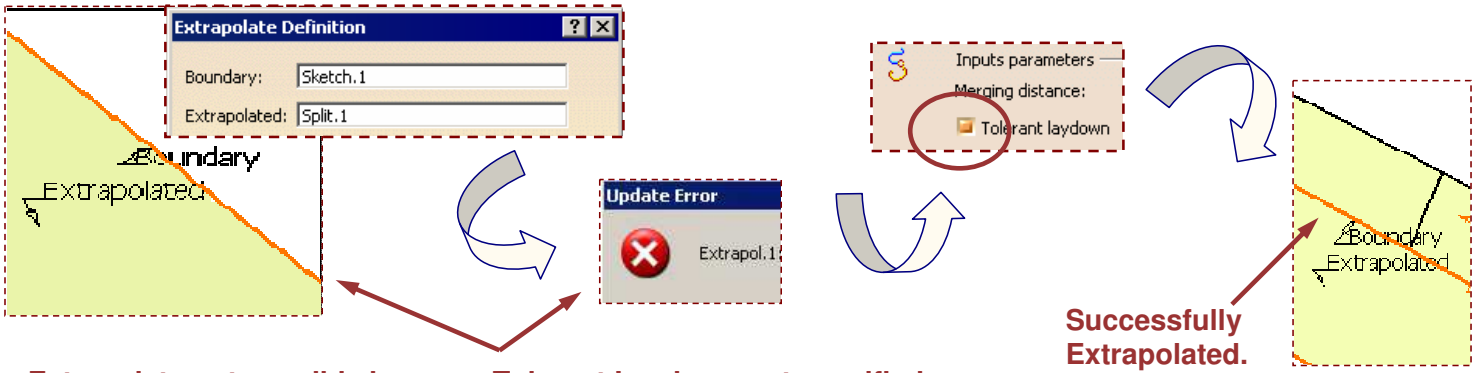
**Tolerant Projection for Fill and Extrapolate lays down the input wire on the surface within the 0.1mm limit specified for Tolerance Laydown and makes possible the creation of Extrapolate and Fill which which otherwise would not have been possible.**




**The surface shown here has been created by sweep operation using the curve as shown.**

**Tolerant smoothening parameters were used and thus the surface is smoothened but there is a distance between the surface and the input curve.**



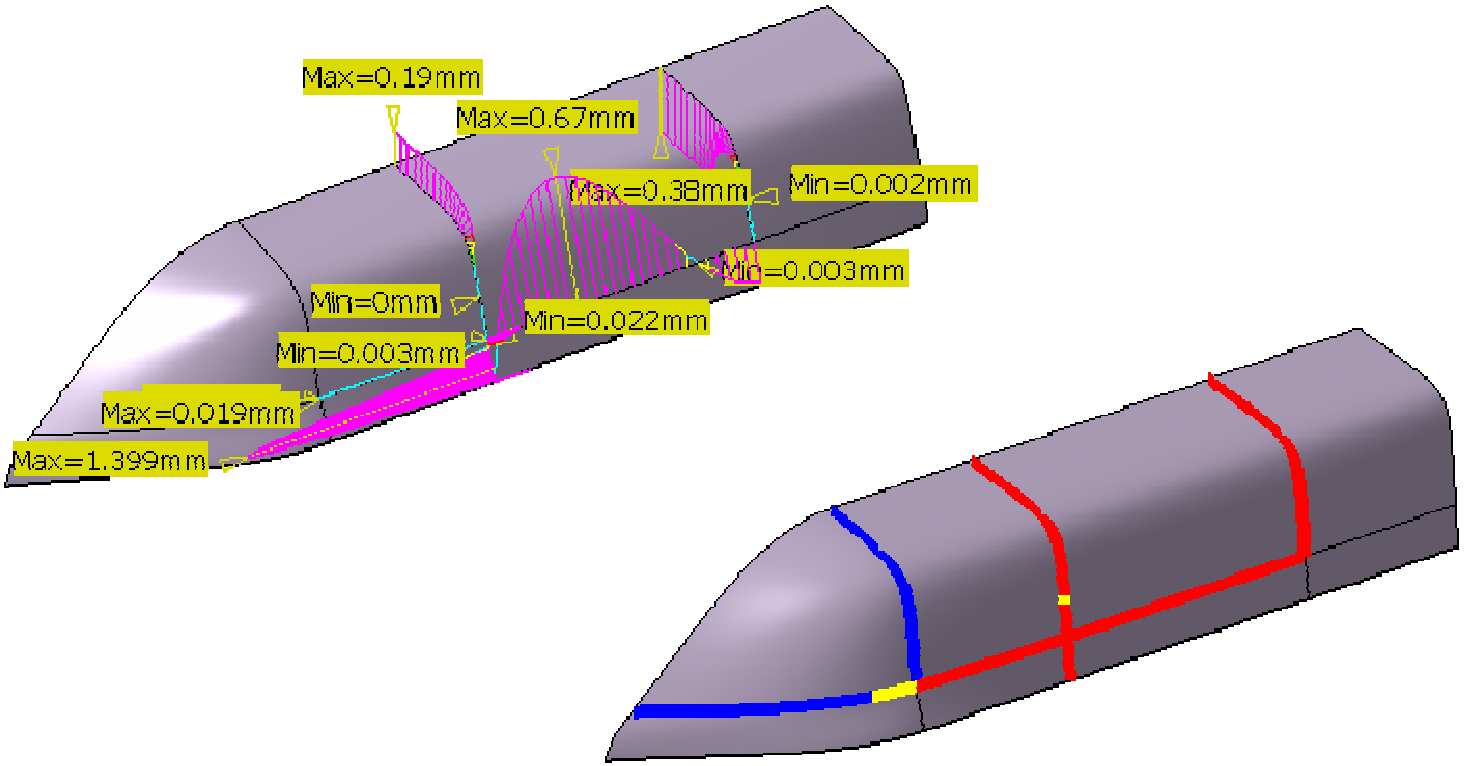

**Extrapolate not possible because Tolerant Laydown not specified.**

**Successfully Extrapolated.**

Student Notes:

# Connect Checker

*You will learn how to use the Connect Checker tool to analyze the connection between surfaces.*

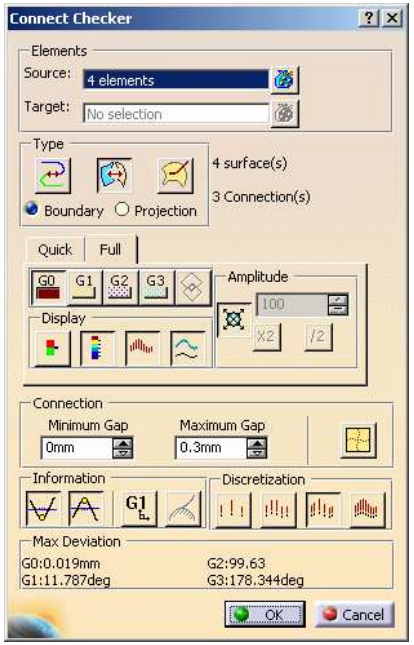
Student Notes:

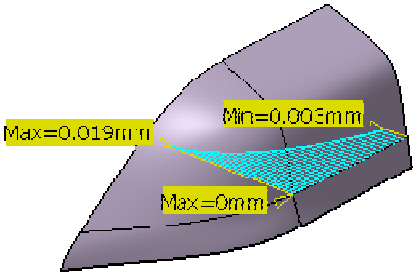
# Why the Connect Checker?

**For surface modeling, to ensure good transition from one surface to another, the Connect Checker allows the user to examine :**

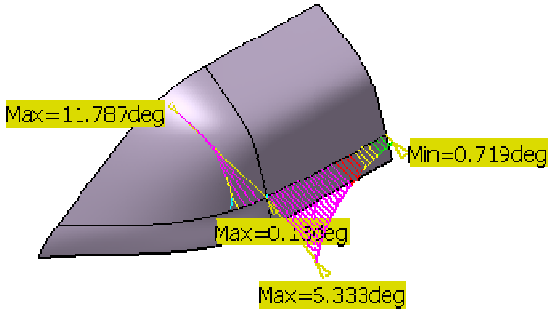
- **G0 (mm)**
- **G1 (deg)**
- **G2 (%)**
- **G3 (deg)**

**along an edge joining two surfaces.**

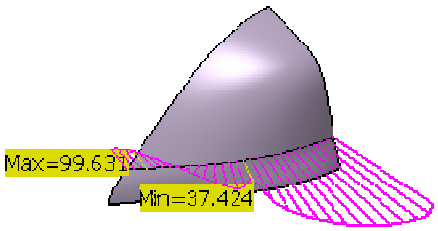



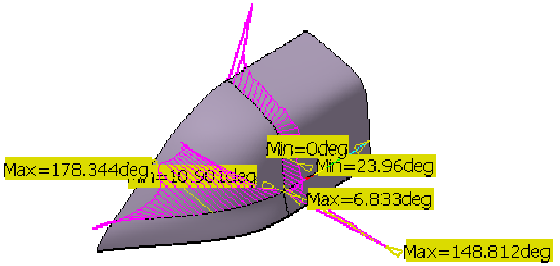

**G0 analysis**




**G1 analysis**




**G2 analysis**




**G3 analysis**

Student Notes:

# How to use the Connect Checker (1/2)

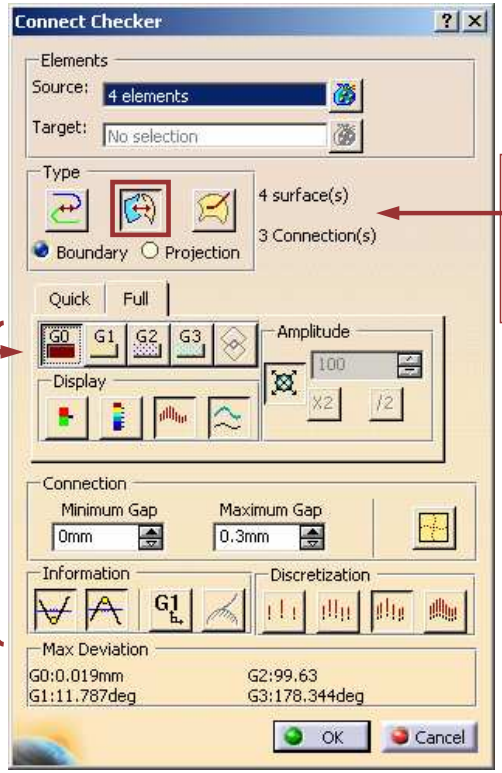
1. Select the Connect Checker Icon 

2. Click Surface-Surface Connection type.

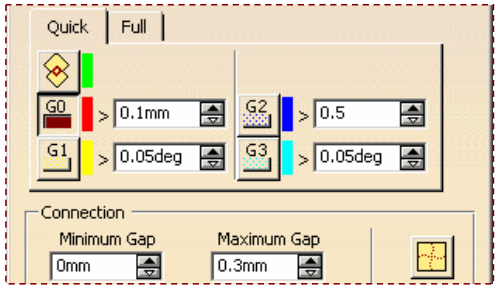
3. Multi-Select the two surfaces between which you would like to check the connection. (In case no connections is found between the selected surfaces CATIA displays an information).

4. Choose the Analysis Type : G0, G1,G2 or G3

5. Choose the type of Display you require.




The number of selected elements and the number of detected connections are displayed.



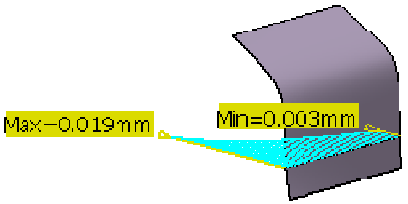

Select the Quick tab to obtain a simplified analysis taking into account tolerances (G0,G1,G2 and G3.)

Student Notes:

# How to use the Connect Checker (2/2)

6 Adjust the color ranges taking account your Minimum and Maximum values

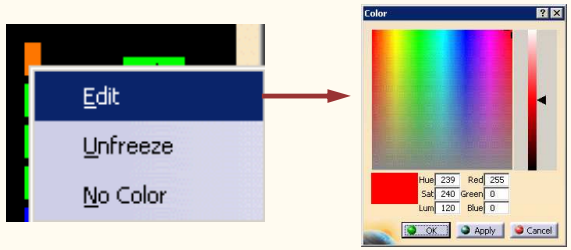
7 Check the analysis result on the geometry.




Note the Minimum and Maximum values between the two surfaces.


You can modify the thresholds values and the corresponding colours by right-clicking on it.

This way, you can adjust the colour range fields.

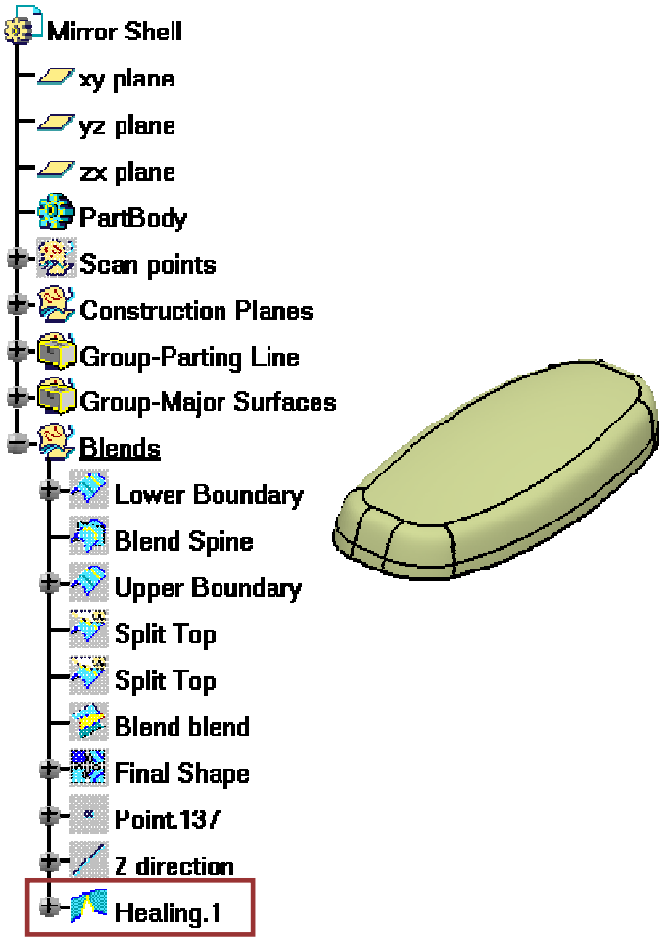



8 Click OK to confirm.The Connection Analysis is added to the specification tree

Student Notes:

# Healing Surfaces

*You will learn about the Healing operation*

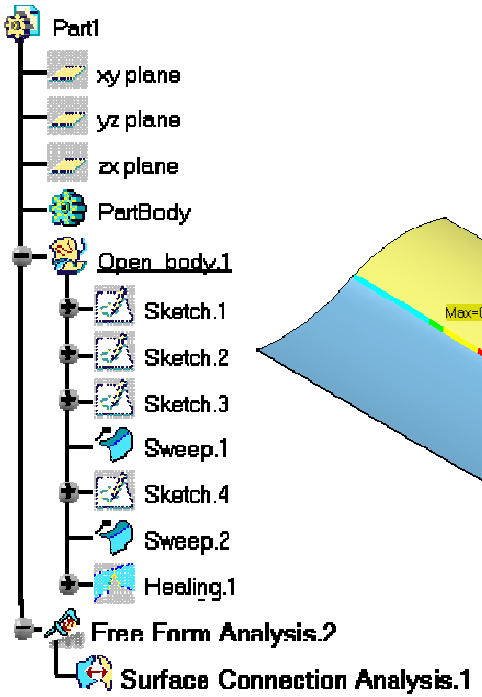


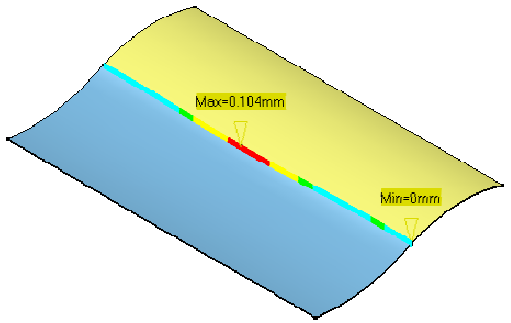
Student Notes:

# Why Healing?

While Join is a topological integration of surfaces into one logical surface, HEALING will mathematically deform the shape of surfaces at boundary areas so they smoothly blend into one another.

When physical parts are manufactured from CAD models, the machining is guided by the exact representation of the individual surfaces. Hence, Healing is important to ensure that each one of these surfaces transitions smoothly between one another.

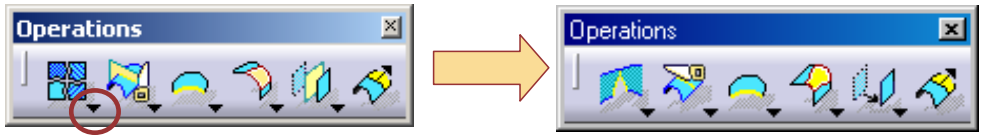


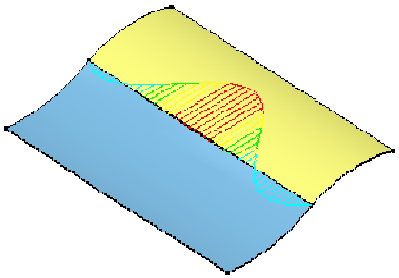
Student Notes:

# Healing Surfaces (1/3)

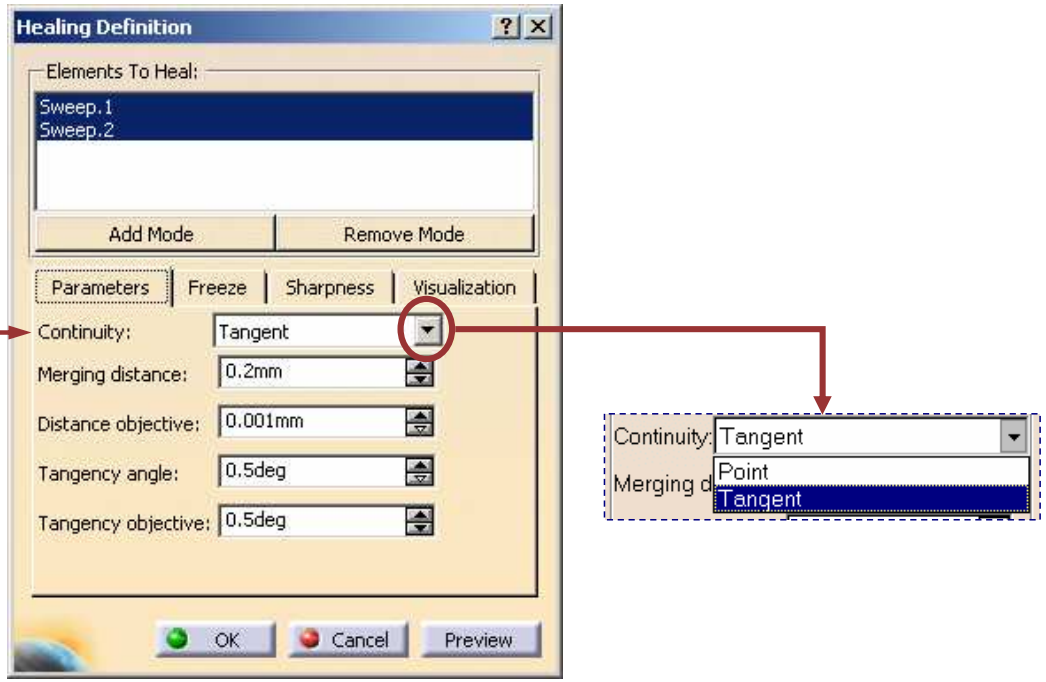
1




2 Select directly the surfaces to heal.



3 Choose if you want to heal the point discontinuities or the tangency discontinuities.



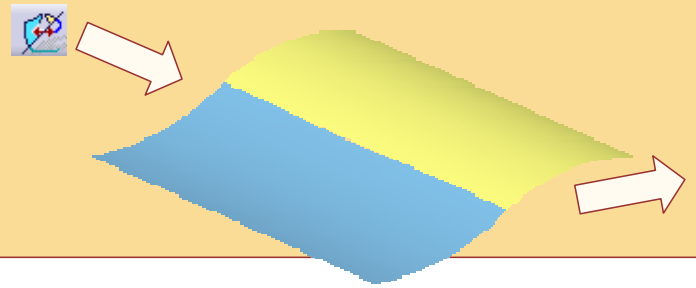
Student Notes:

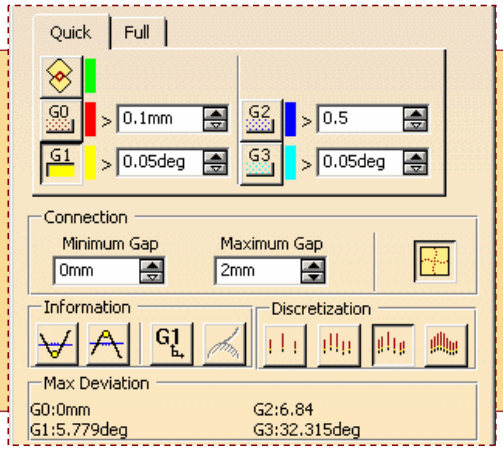
# Healing Surfaces (2/3) : Parameters

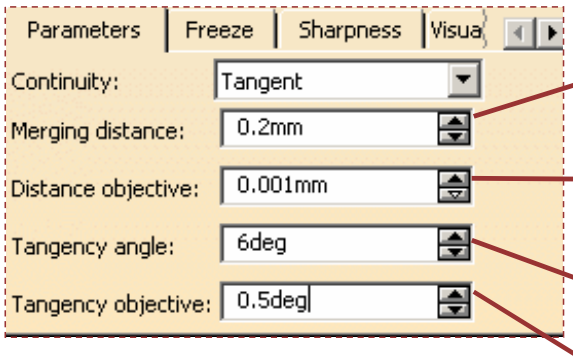
**The objective of the parameters is to choose the discontinuities you want to heal or not :**

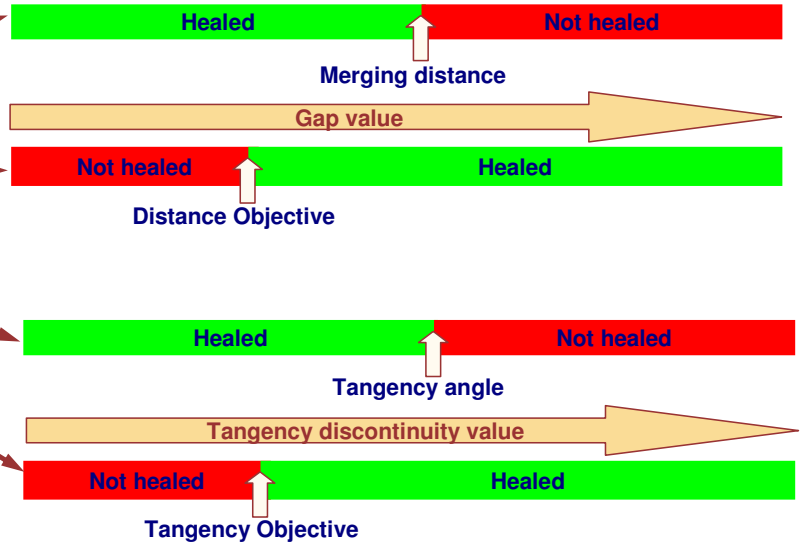
**4** **Key in parameters :**

**Note : a Connect Checker analysis can help to choose these parameters :**





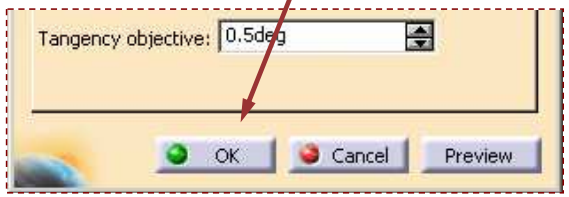



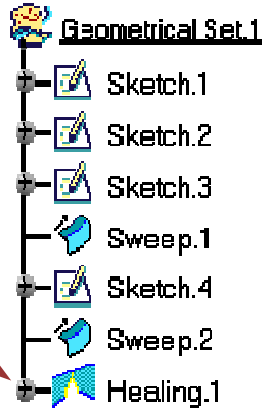

**These parameters are thresholds that allows you to:**
**- define the discontinuities to be healed (Merging distance and Tangency angle).**
**- define the discontinuities you consider it is not necessary to heal (Distance Objective and Tangency Objective).**

Student Notes:

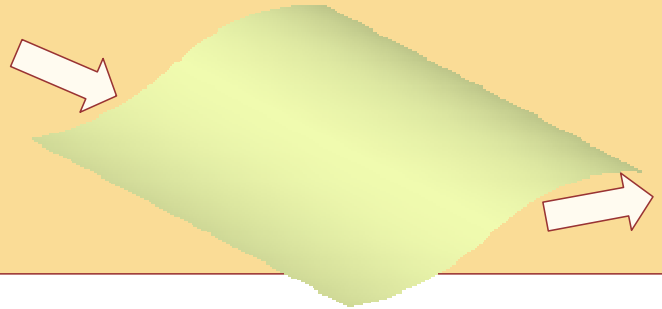
# Healing Surfaces (3/3)

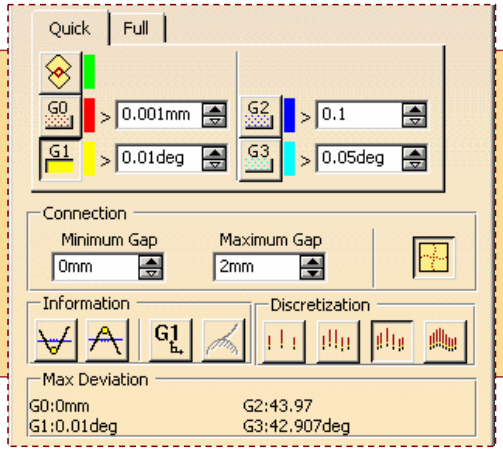
5 Click on OK to confirm the healed surface creation.




Note : a Connect Checker analysis now shows :



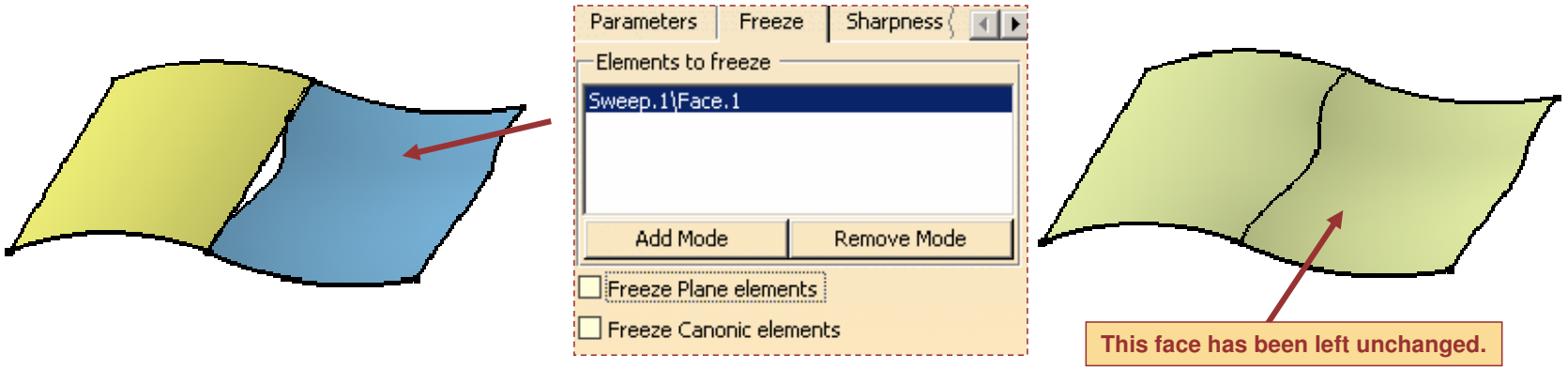


Student Notes:

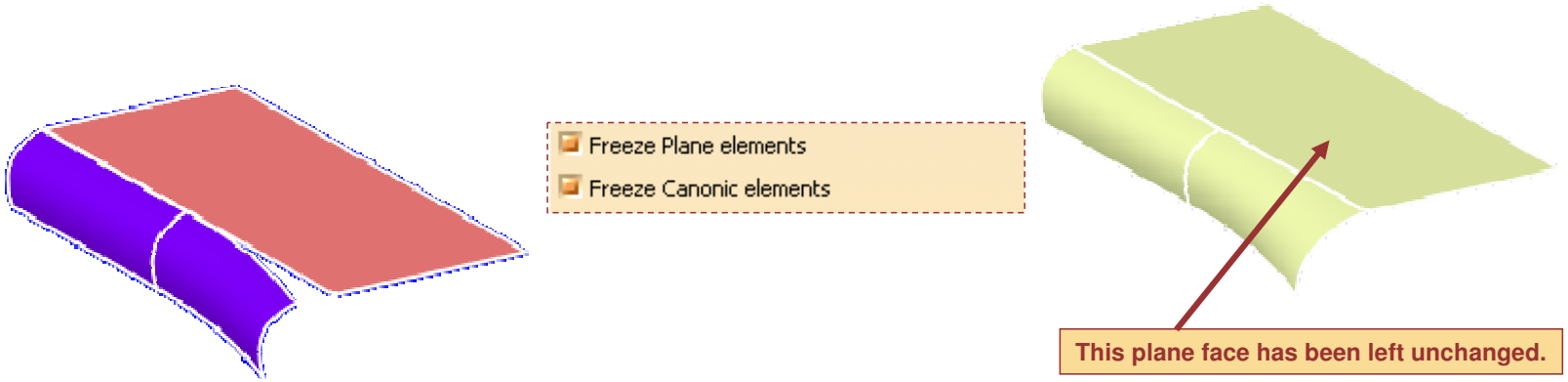
# Additional Information on healing (1/2)

**While healing surfaces, you can freeze some faces for them not to be modified by the healing operation.**

**You can choose to freeze any face you want …**



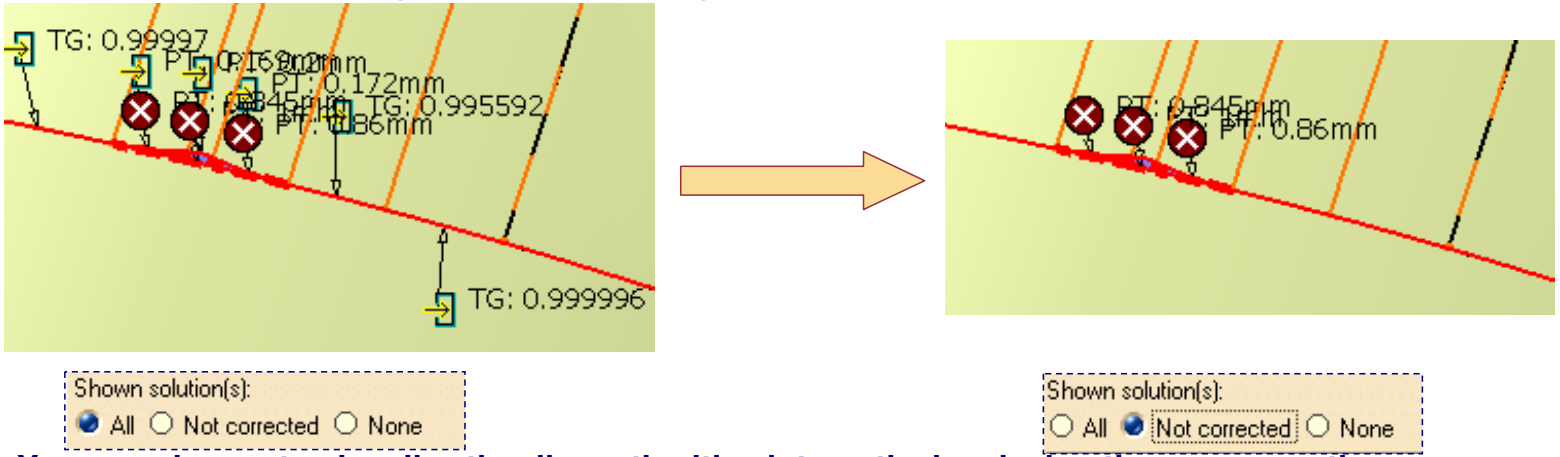

**… or choose to freeze plane elements or canonic elements.**



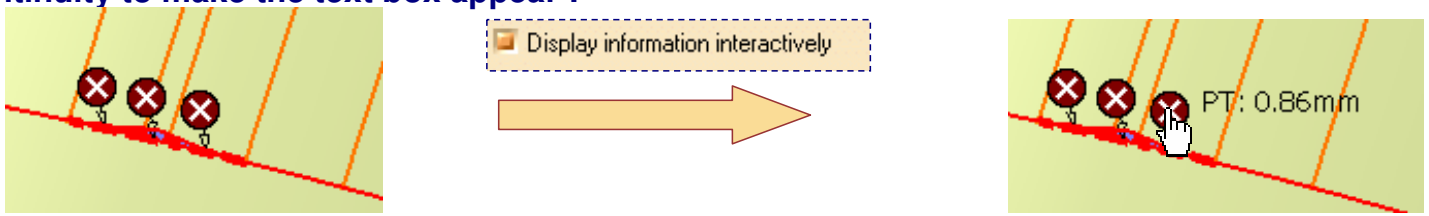
Student Notes:

# Additional Information on healing (2/2)

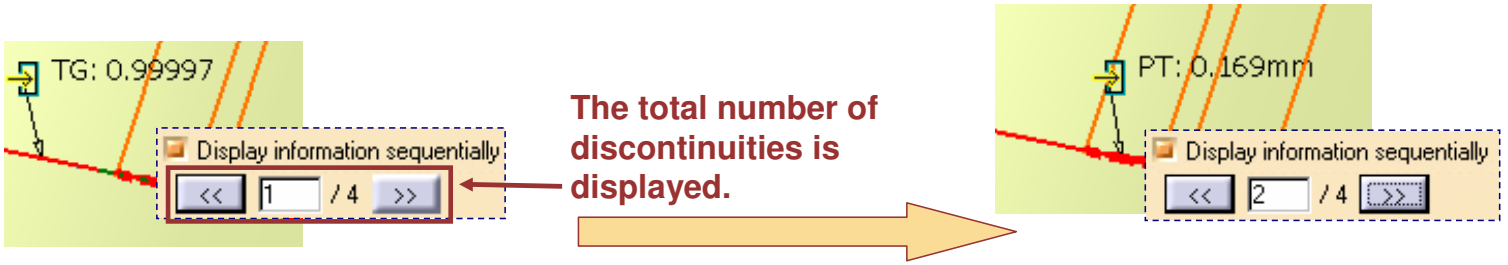
**You can visualize a diagnosis while healing surfaces**




**You can choose to visualize the discontinuities interactively, placing the mouse on the discontinuity to make the text box appear :**



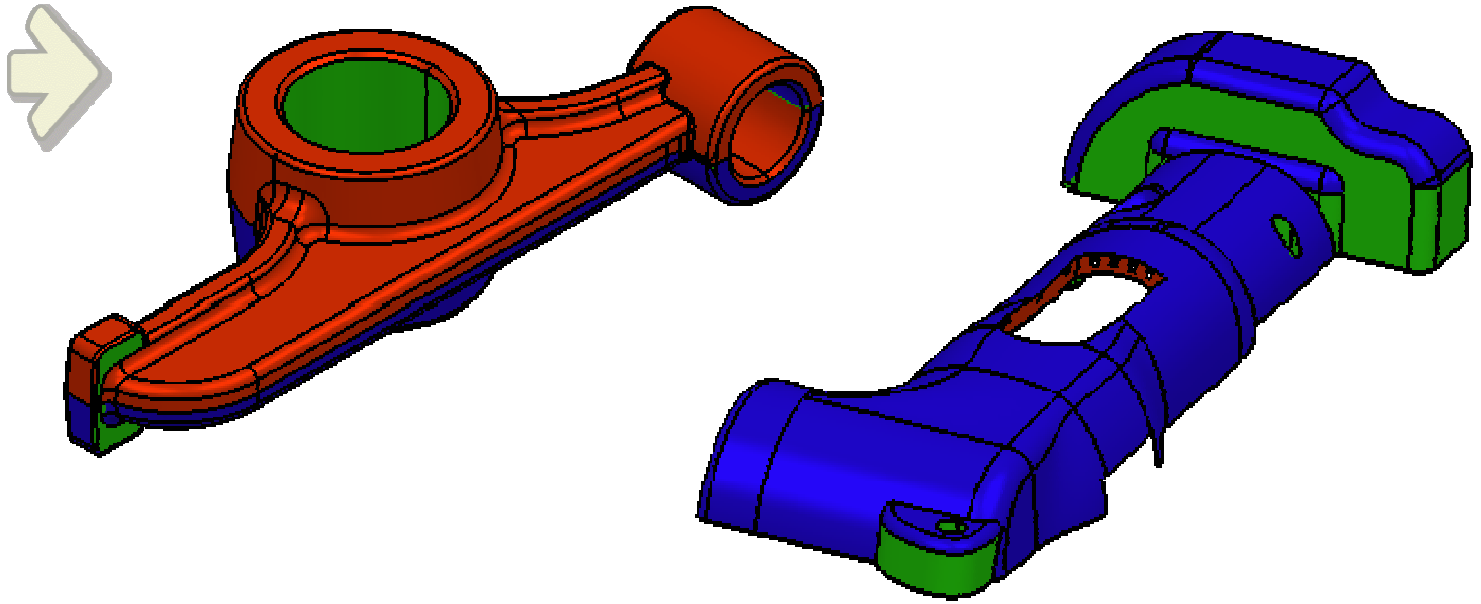

**You can also display the information sequentially :**



Student Notes:

# Checking Molded Parts

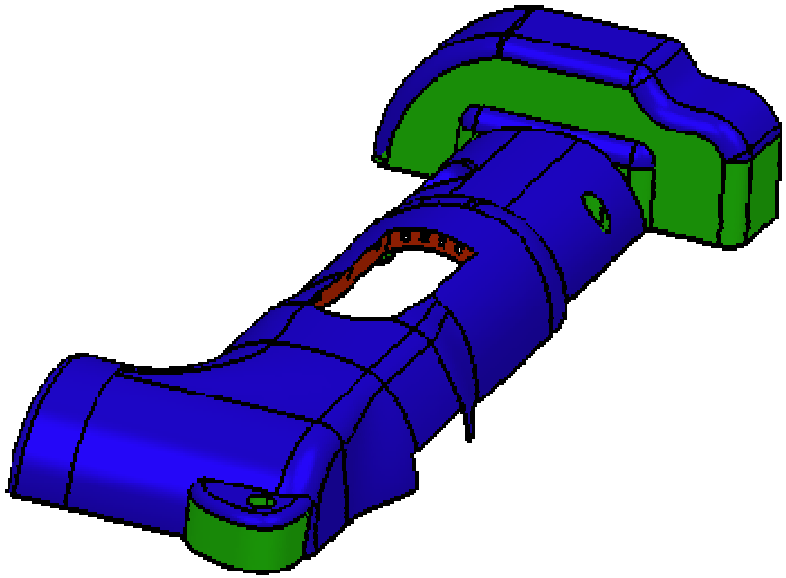
*You will learn about tools, commonly used while creating a Molded Parts.*



Student Notes:

# Draft Analysis

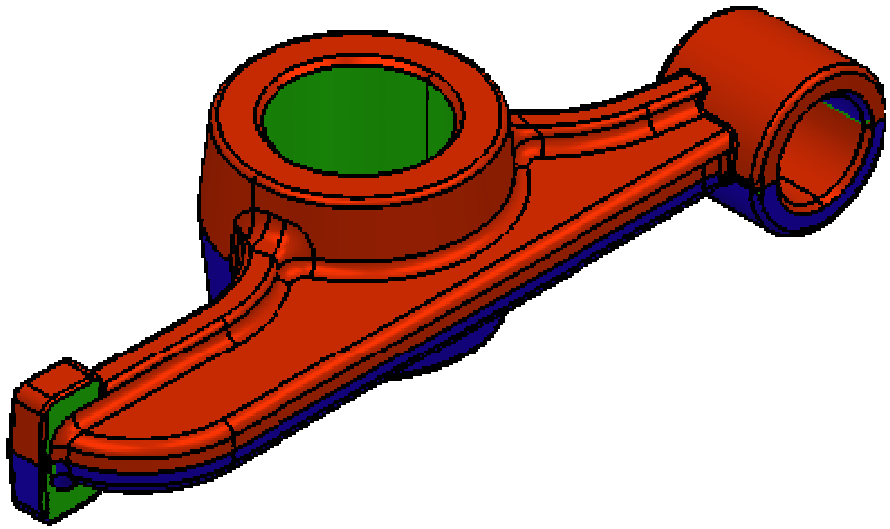
*You will learn how to use the Draft Analysis tool to analyze the draft values of surfaces or solids*



# Why Analyze Draft?

**Cast and Forged parts need dies and molds for manufacturing. While doing the die/mold design, draft allowances are required to be given so that the parts can be extracted. Drafts need to be analyzed to determine extractability of the part.**

**For NC Machining, a part is analyzed to look for negative Draft angles in order to determine if a 5-Axis NC machine will be required to cut the part.**



Student Notes:

Student Notes:

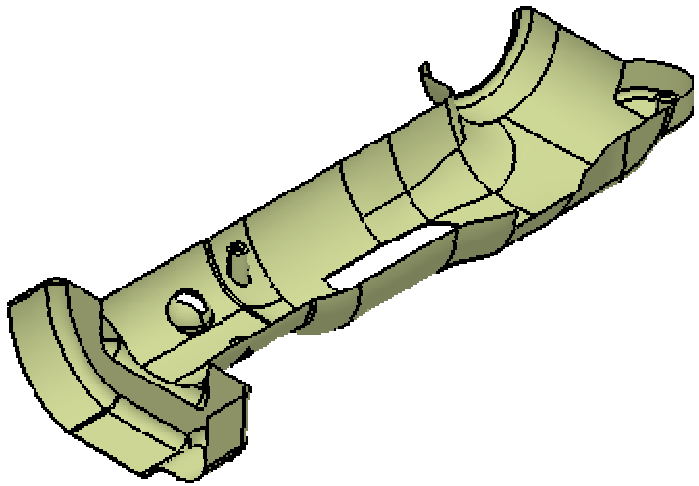
# Using the Draft Analysis Tool (1/5)

1 Select the customized view render style : 

2 


Select the Draft Analysis tool

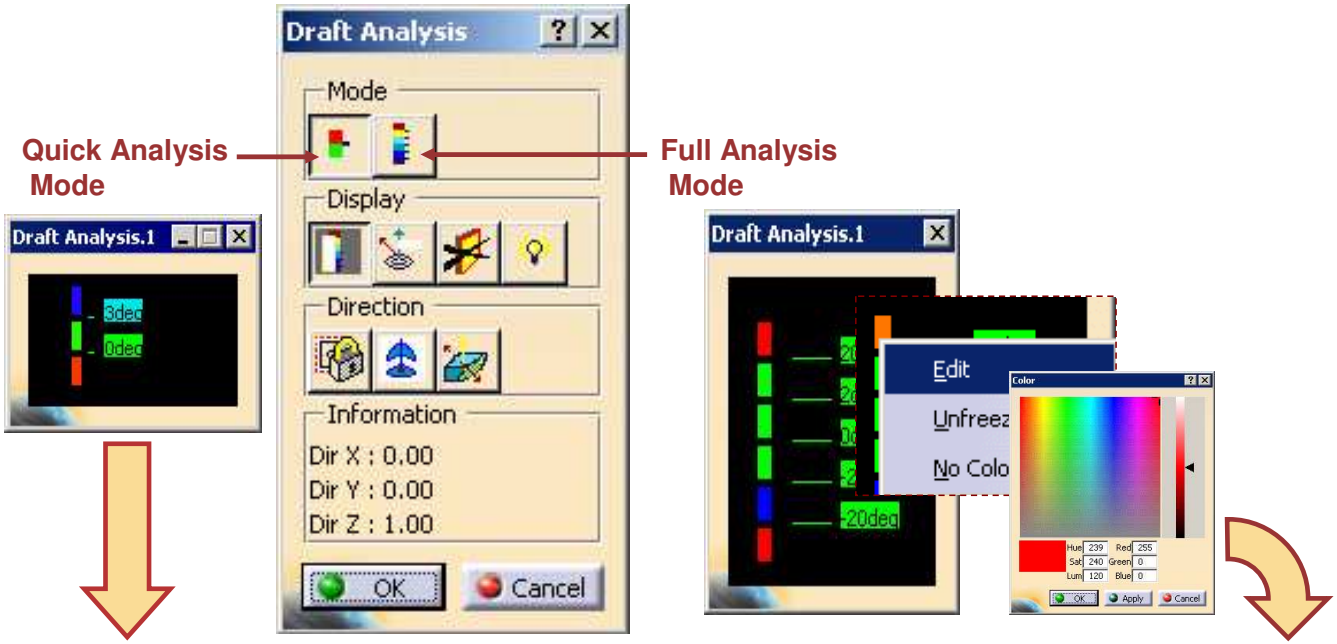

3 Select the surface to be analysed



Student Notes:

# Using the Draft Analysis Tool (2/5)

4 Choose the quick analysis mode or the full analysis mode :

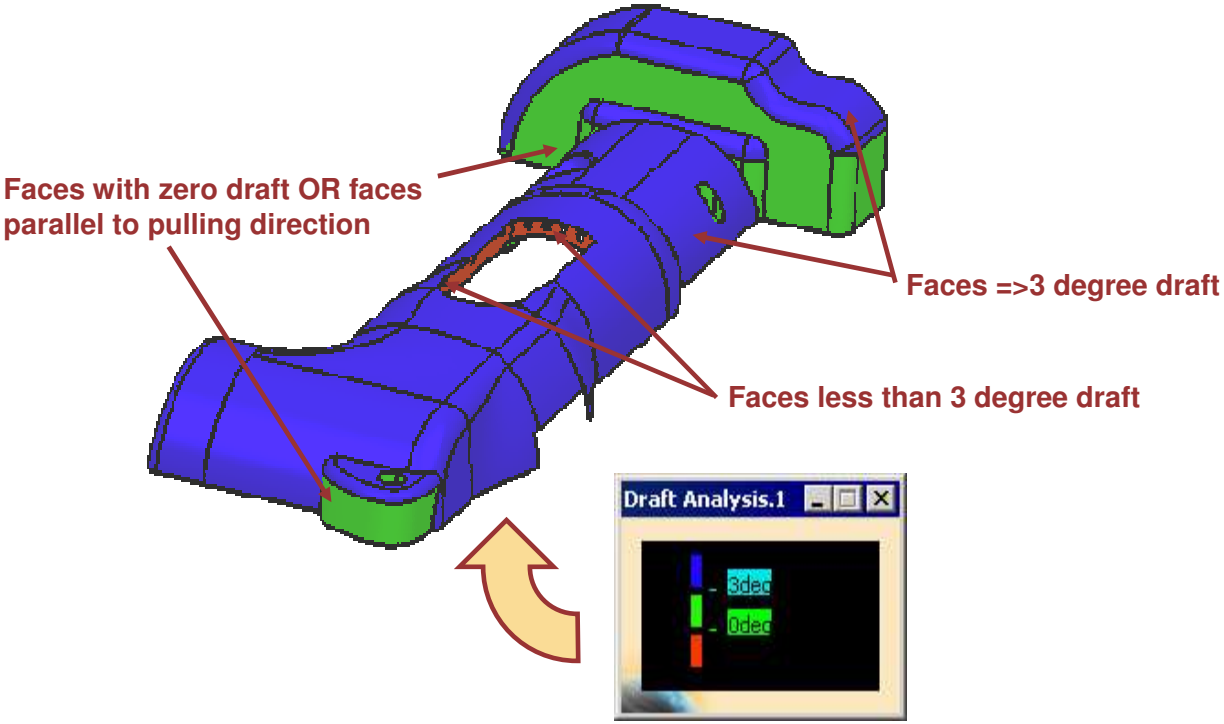



In quick analysis, you can set three parameters showing positive, Negative and zero (neutral) angle and assign the colors to these parameters for a quick view of a draft angle in a part

You can modify the thresholds values and the corresponding colours by right-clicking on them. This way, you can adjust the colour range fields.

Student Notes:

# Using the Draft Analysis Tool (3/5)



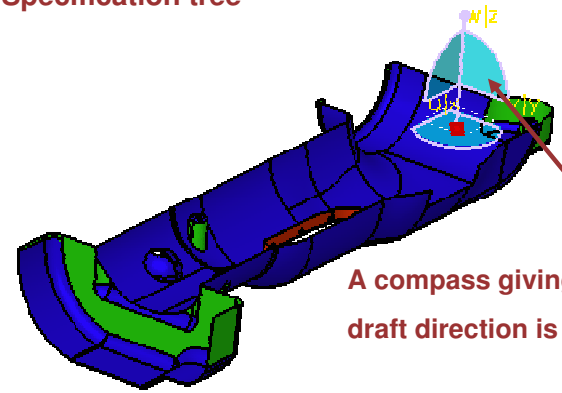

You can Visually analyze the part based on the color assigned with an angle value.A range of colors can be assigned to visualize the varying draft angle.

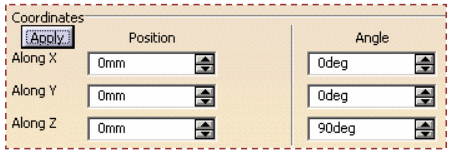
Student Notes:

# Using the Draft Analysis Tool (4/5)

5 You have now to adjust the draft direction. In the dialog box,select 

6 Select OK to keep the history of the analysis in to the Specification tree



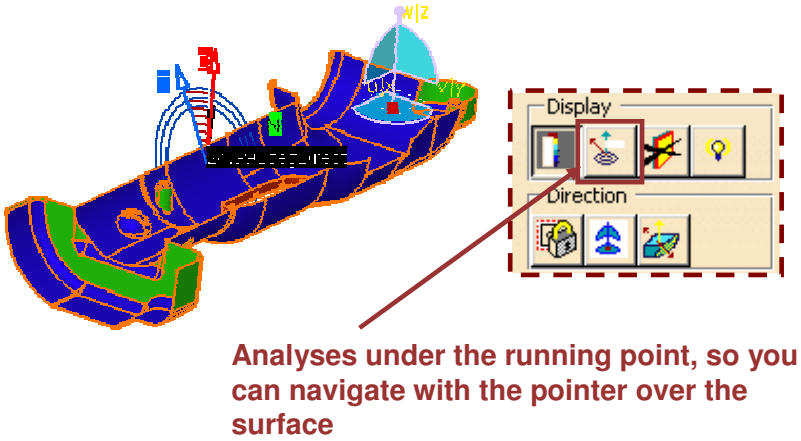

You can edit the compass proprieties to precisely define the draft direction.


The Blue plane is the plane tangent to the analyzed surface at this point..

Arrows are displayed under the pointer.

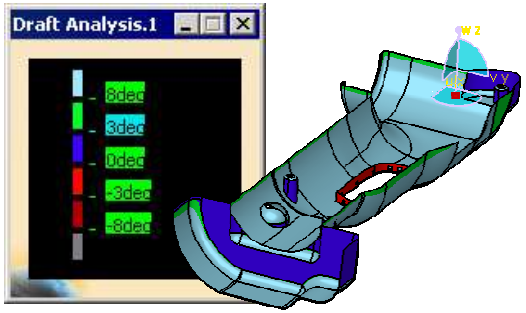
Green arrow is the direction normal to the surface, the Red arrow represent draft direction.

The displayed value indicates the angle between the draft direction and the normal vector to the surface at the current point.

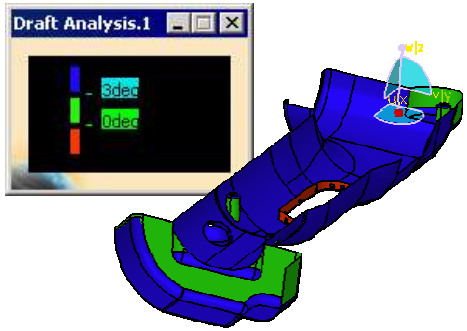


# Using the Draft Analysis Tool (5/5)

Student Notes:




Results of full Draft analysis

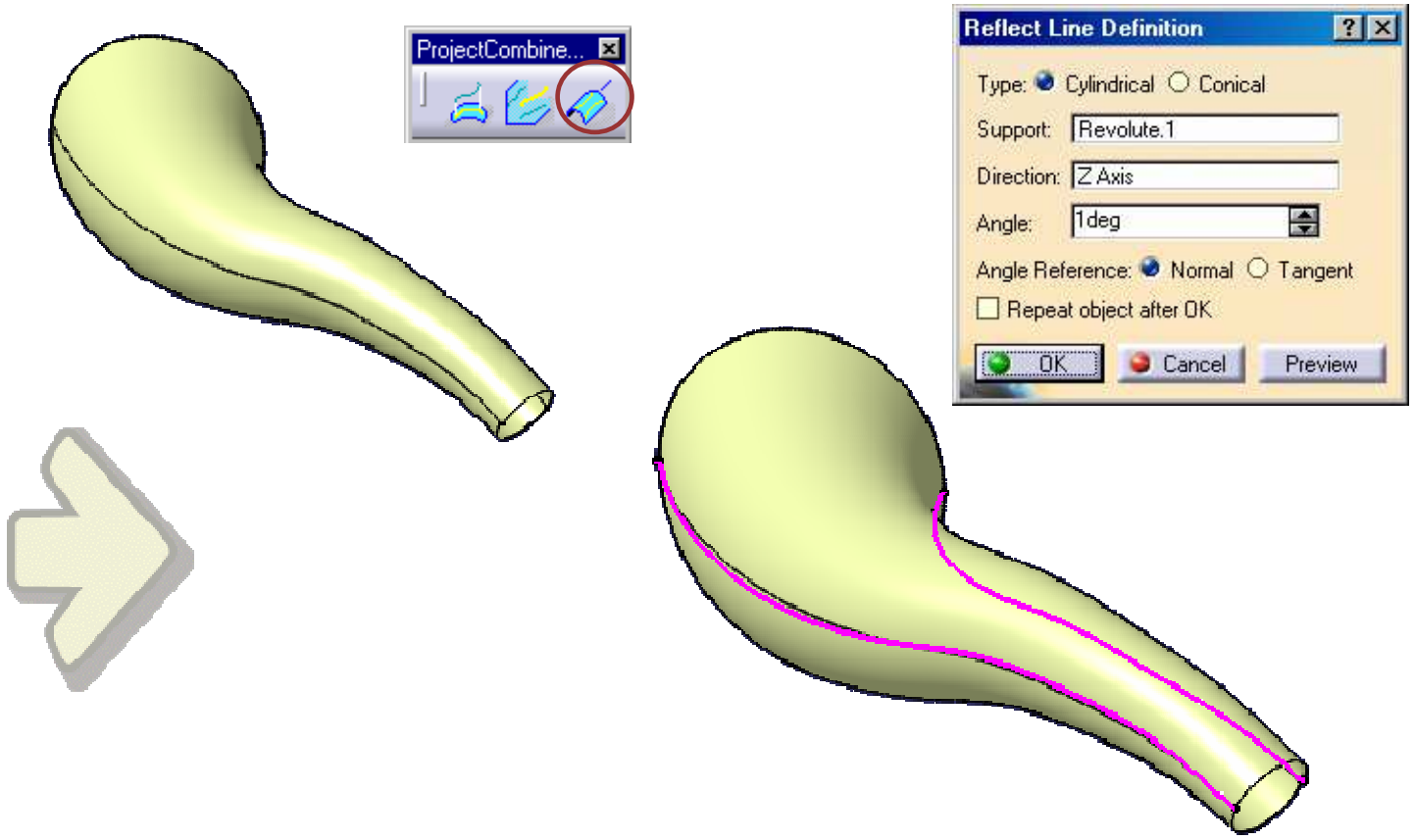



Results of Quick Draft analysis

Student Notes:

# Reflect Line

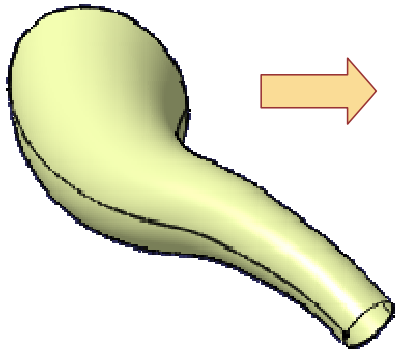
*You will learn what a Reflect Line is and how to create it.*



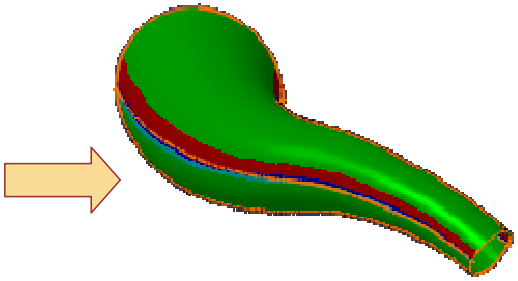
Student Notes:

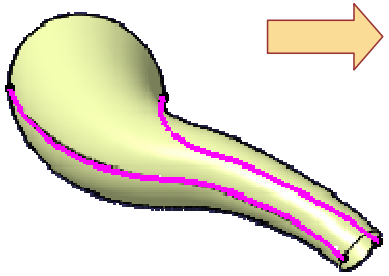
# What is a Reflect Line

**Reflect lines are curves for which the normal to the support surface in each point presents the same angle with a specified direction. It is very useful to find the parting plane of a complex surface.**

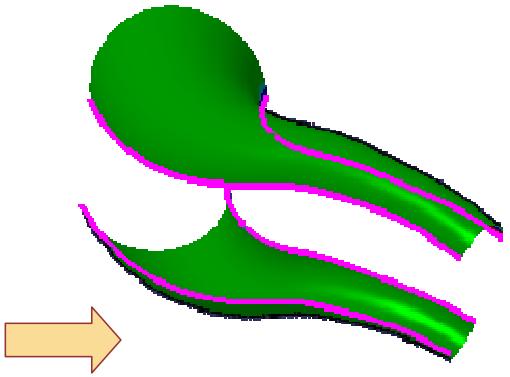


**If we perform a Draft analysis on this part, we can see, thanks to the red areas that the part is non extractible.**





**Thanks to the Reflect Line curve, we can cut the part in two extractible parts.**



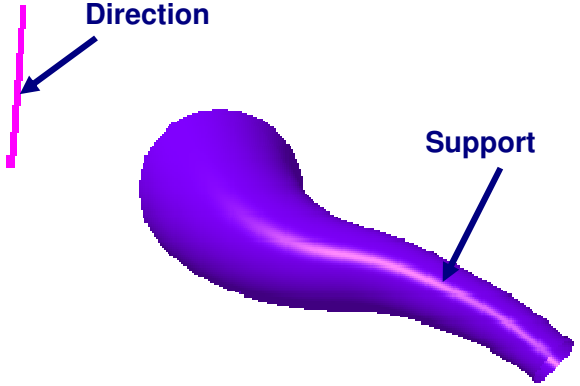
Student Notes:

# Creating a Reflect Line

1

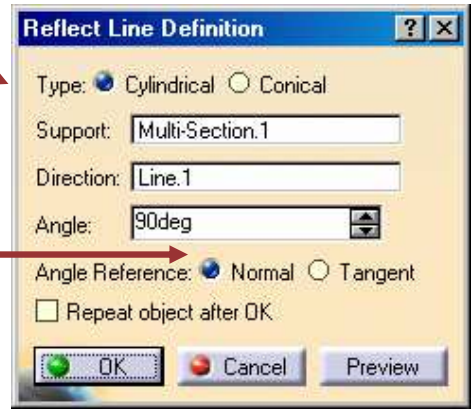


2 Select a support surface and a direction.

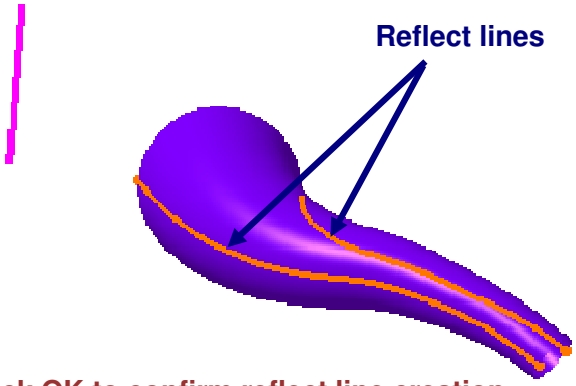



You can define one of the X,Y or Z axis by opening a contextual menu in the Direction field.

3 Key in an angle representing the value between the selected direction and the normal to the surface.




You can select the Normal option for the angle to be computed between the normal to the support and the direction.




4 Click OK to confirm reflect line creation

Student Notes:

# Summary (1/3)

## Congratulations

You have become familiar with Advanced tools of Generative Shape Design.
You have learnt different tools from different lessons as follows:

Lesson 1: Introduction to Surface Design

- Revised the Surface Design basic tools which are covered under Fundamental course.
- Overviewed the Generative Shape Design Workbench.
- Learnt more about Managing Features and Geometrical Sets.
- Finally performed the a recap exercise ‘Managing features’ and ‘Shampoo Bottle’.

Lesson 2: Creating Advance Wireframe Geometry

- Learnt GSD Similarities for MD2/HD2 configurations.
- Learnt creating an Extremum, Connect Curve.
- Performed recap exercise on ‘Advance Wireframe Geometry’.

Student Notes:

# Summary (2/3)

**Lesson 3: Wireframe Analysis and Repair**

- Learnt the need to analyze the wireframe.
- Learnt about Connect Checker and Smoothing Curves.
- Performed the a recap exercise on 'Wireframe Analysis and Repair'.

**Lesson 4: Creating Advance Swept Surface**

- Learnt about 'Swept Surface', 'Inputs for Swept Surface' , 'Different methods of creating Swept Surface','Adaptive Swept Surface'.
- Performed two recap exercise 'Knob' and 'Housing'.

**Lesson 5: Creating Advance Blend Surface**

- Learnt about 'Blend Surface', 'Types of Surface' , 'Creating fillets using Hold curve and Spine','Creating Fillet using Law'.
- Performed recap exercise 'Advance Blending Surface'.

Student Notes:

# Summary (3/3)

**Lesson 6: Surface Analysis and Repair**

- Learnt the need to analyze the Surface.
- Learnt about Curvature Analysis.
- Performed the a recap exercise on 'Surface Analysis'.

**Lesson 7: Additional Surface Design Tools**

- Learnt about other surface Design tools which are commonly used to improve the quality and stability of the surface and wireframe geometry.
- Also learnt about tools used to measure the molded parts in virtual mode such as Draft analysis and reflect line which helps the tool designer to decide the parting line and manufacturability.

**Master Exercise: You have performed the Master Exercise at the end of the each lessons to practice the tools learnt in each lesson.**