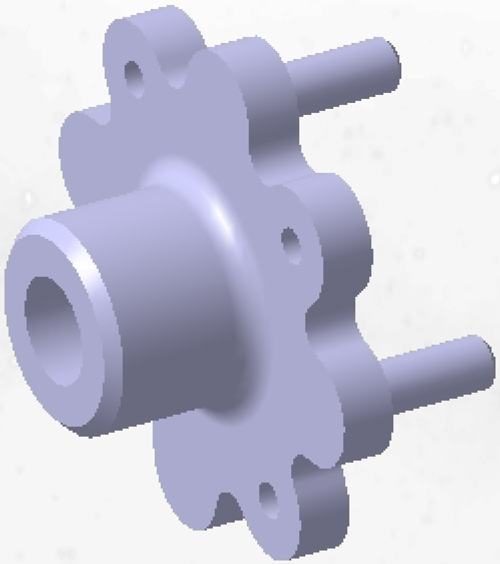
CATIA Training

Foils



CATIA
Part Design Advanced

Version 5 Release 8
January 2002

EDU-CAT-E-PDG-AF-V5R8

Course Presentation

Objectives of the course

In this course you will complete the knowledge acquired in the CATIA Part Design Fundamentals course

Targeted audience

New CATIA V5 users

0.5 day

Prerequisites

CATIA Part Design

Fundamentals course

Table of Contents (1/2)

Table of Contents (2/2)

Advanced Tools

You will learn how to create and use other tools of Sketch-Based, Surface-Based and Dress-up Features. You will also learn tools of Transformations, 3D Constraints, Local Axis and Annotation.

- **Holes/Pockets/Pads not Normal to Sketch Plane**
- **Creating Grooves**
- **Creating Ribs and Slots**
- **Creating Stiffeners**
- **Creating Lofts**
- **3D Wireframe Elements**
- **Surface-Based Features**
- **Thickness**
- **Using Transformation**
- **3D Constraints**
- **Local Axis**
- **Annotation**

Holes/Pockets/Pads not Normal to Sketch Plane

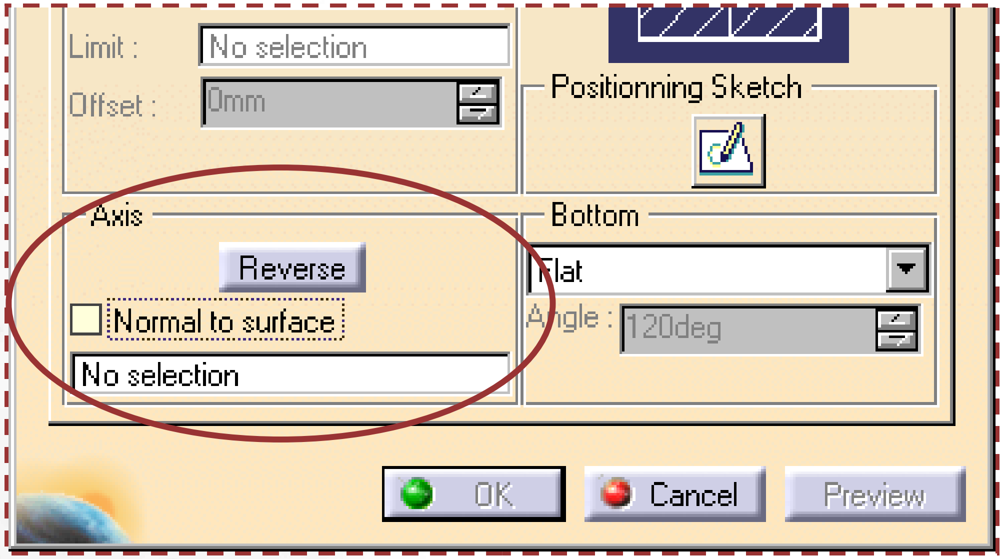
You will learn how to create Holes, Pockets or Pads with their direction not perpendicular to their sketch

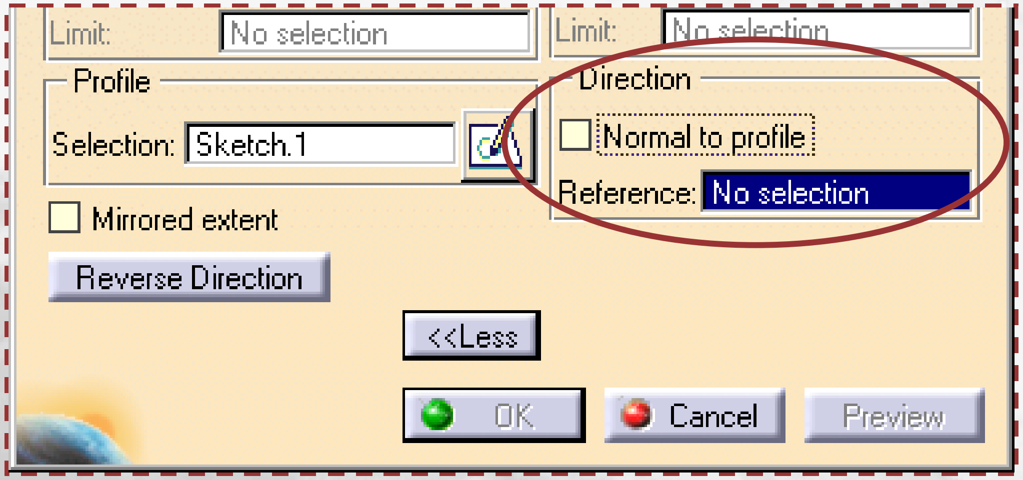
- **Defining a direction**

What are Holes/Pockets/Pads not Normal to Sketch Plane ?

Some Key Points:

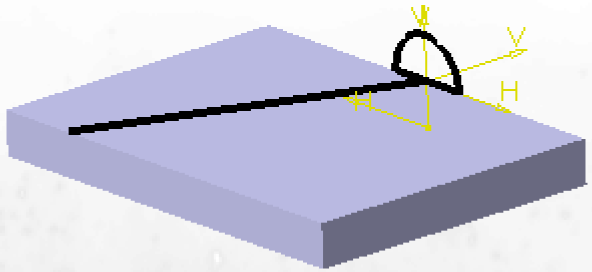
- **When creating a hole, a pocket or a pad, by default you get a result perpendicular to the sketch you have selected to get these features**
- **It is possible to define another direction by specifying a direction in the direction field**
- **The selected direction must not be in a plane parallel to the sketch plane nor in the same plane**



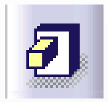


Holes/Pockets/Pads not Normal to Sketch Plane

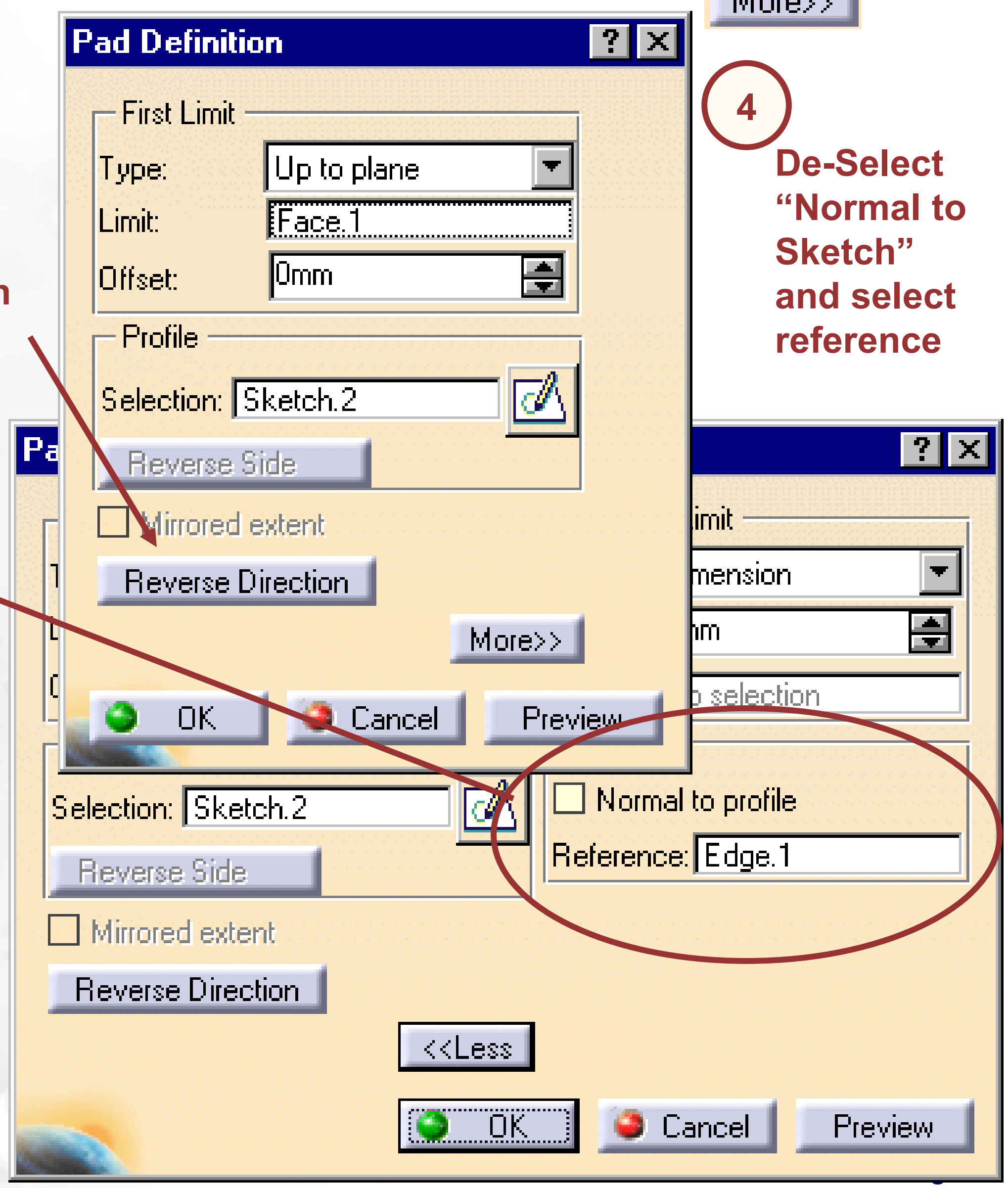
1 If a Pad or Pocket, select Profile sketch to be used



2 Select the appropriate icon

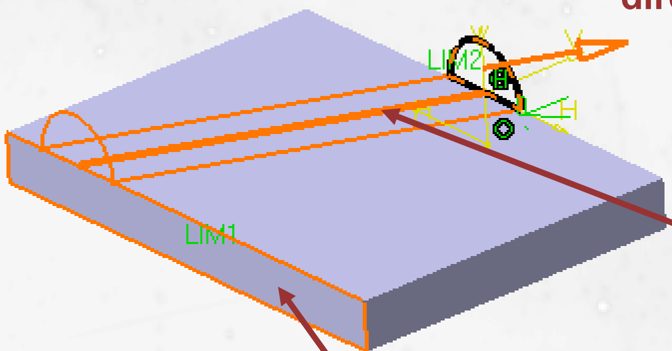


3 For this geometry, modify definition to include type “Up to Plane” and select More>>



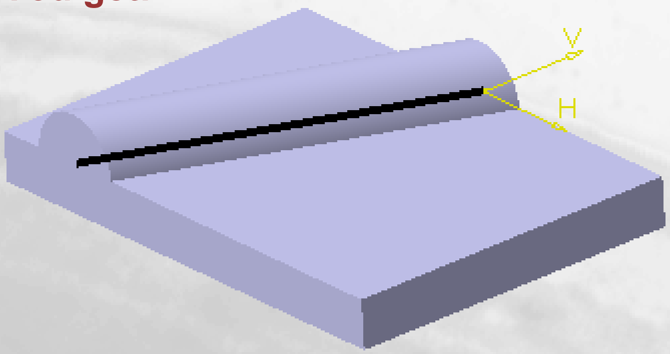
4 De-Select “Normal to Sketch” and select reference

Changes extrusion direction



5 Select limit surface on part

You get:



Creating Grooves

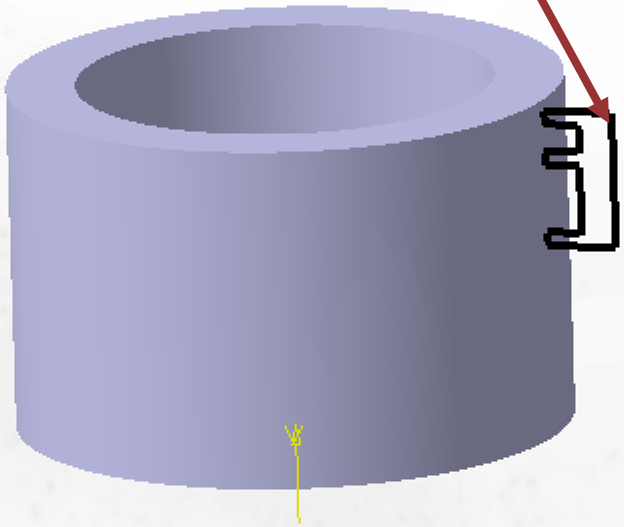
You will learn how to create grooves

•Material removing according to a revolution body

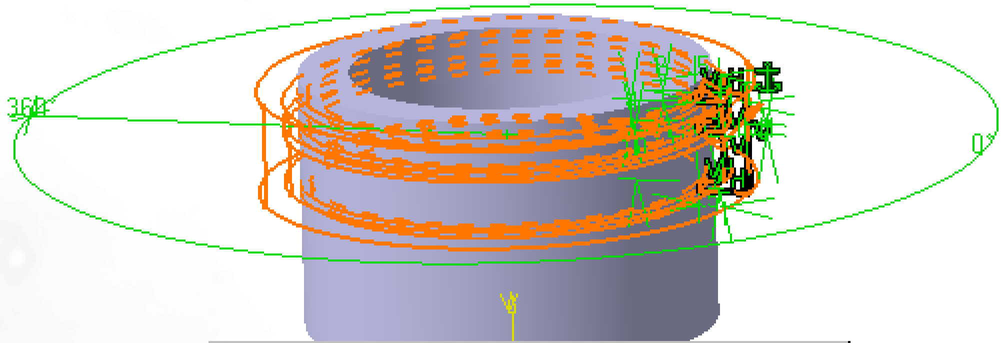
Creating Grooves

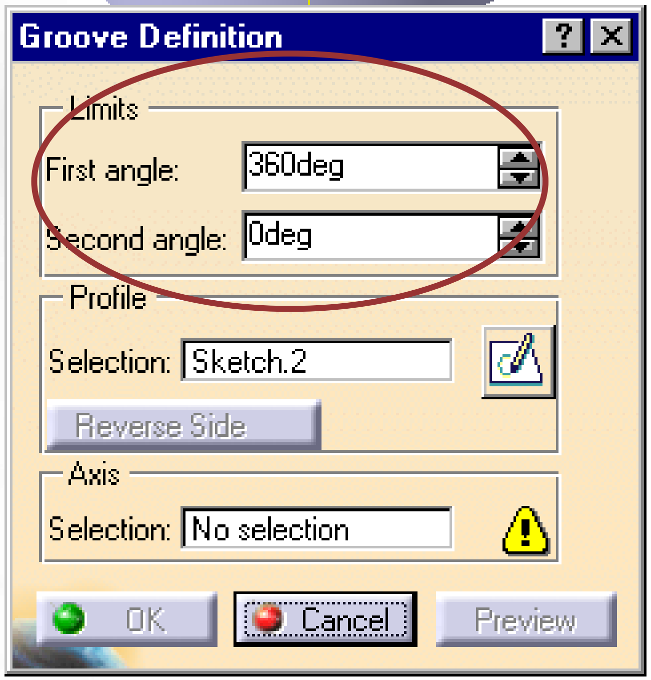
3 Select the Groove icon

2 Select the profile to be used for the groove (the sketch must contains an axis)



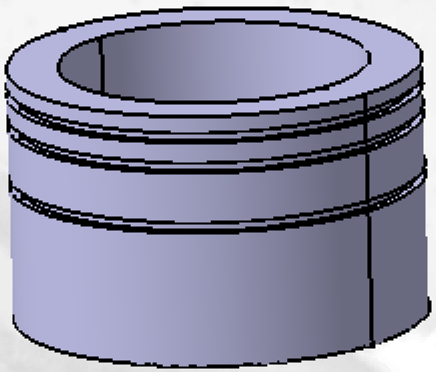
3 Define the limits of the groove





4 Select OK

You get:

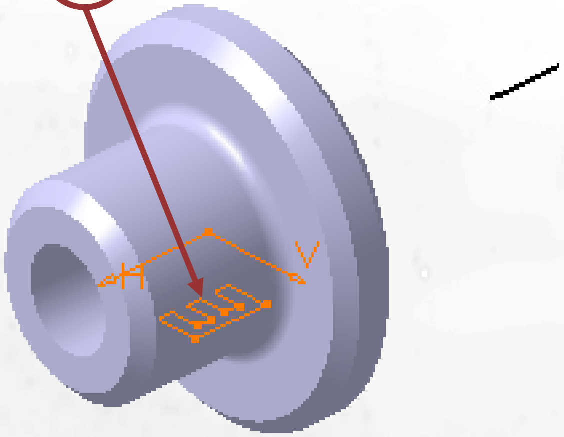


Groove : 3d Line axis

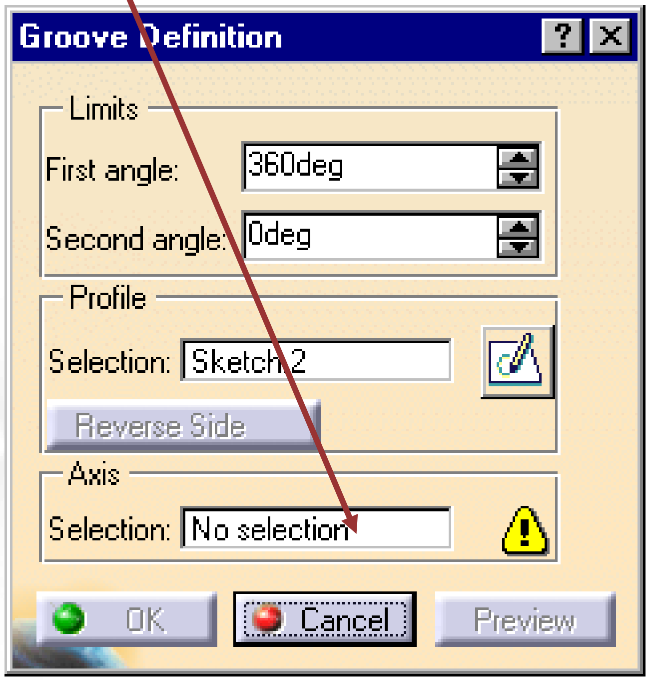
When creating a groove, it is possible to use a 3d line or a sketched line not included in the sketch of the profile as the rotation axis

1 Select the Groove icon

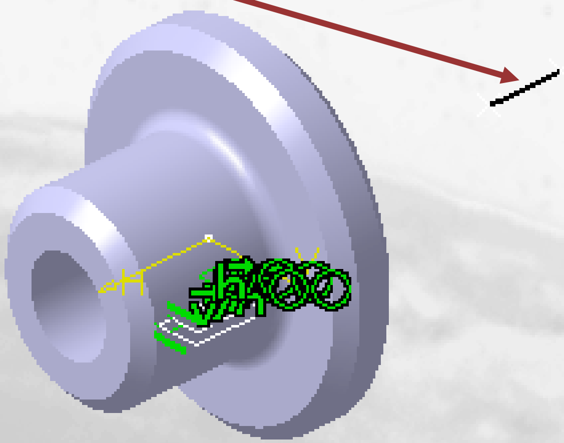
2 Select the profile



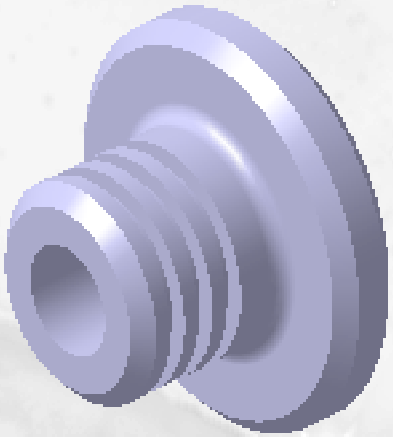
3 Select the Axis field in the dialog box



4 Select the 3d line as the rotation axis

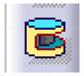


5 You can modify the Limits parameters then select OK, you get:

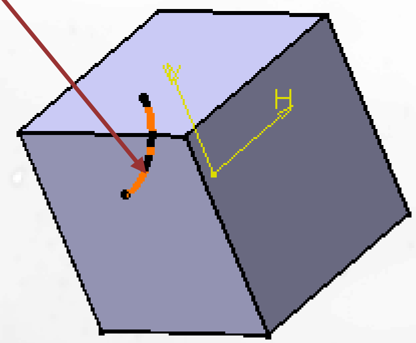


Groove : Reverse Side

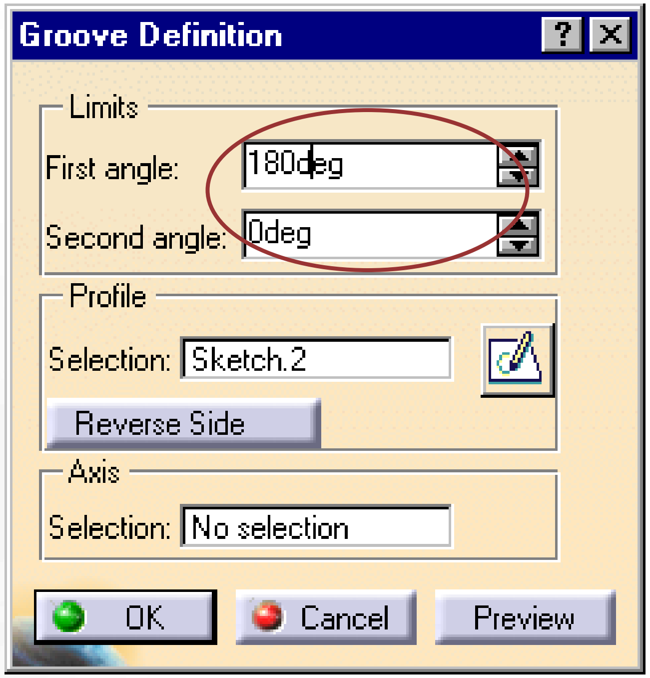
The Reverse Side button applies for open profiles only. This option lets you choose which side of the profile is to be extruded

1 Select the Groove icon 

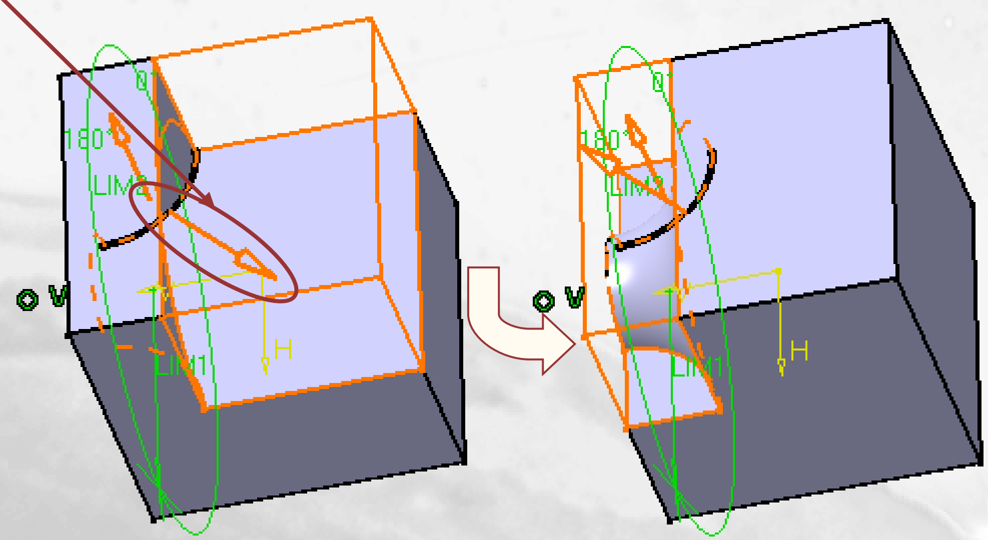
2 Select the open sketch



3 Modify the Groove Limit Angles

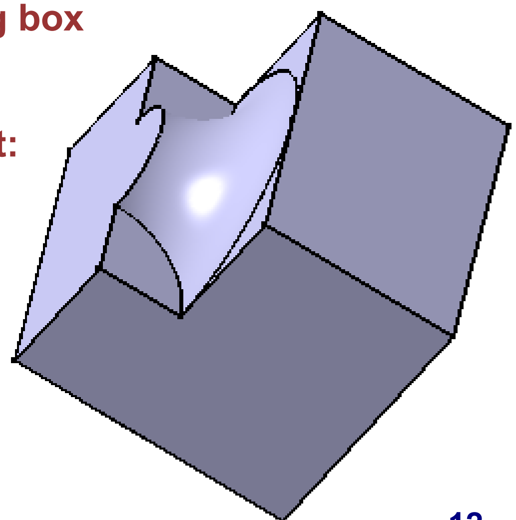


4 Select the arrow to reverse the groove side (or click the Reverse Side button in the dialog box)



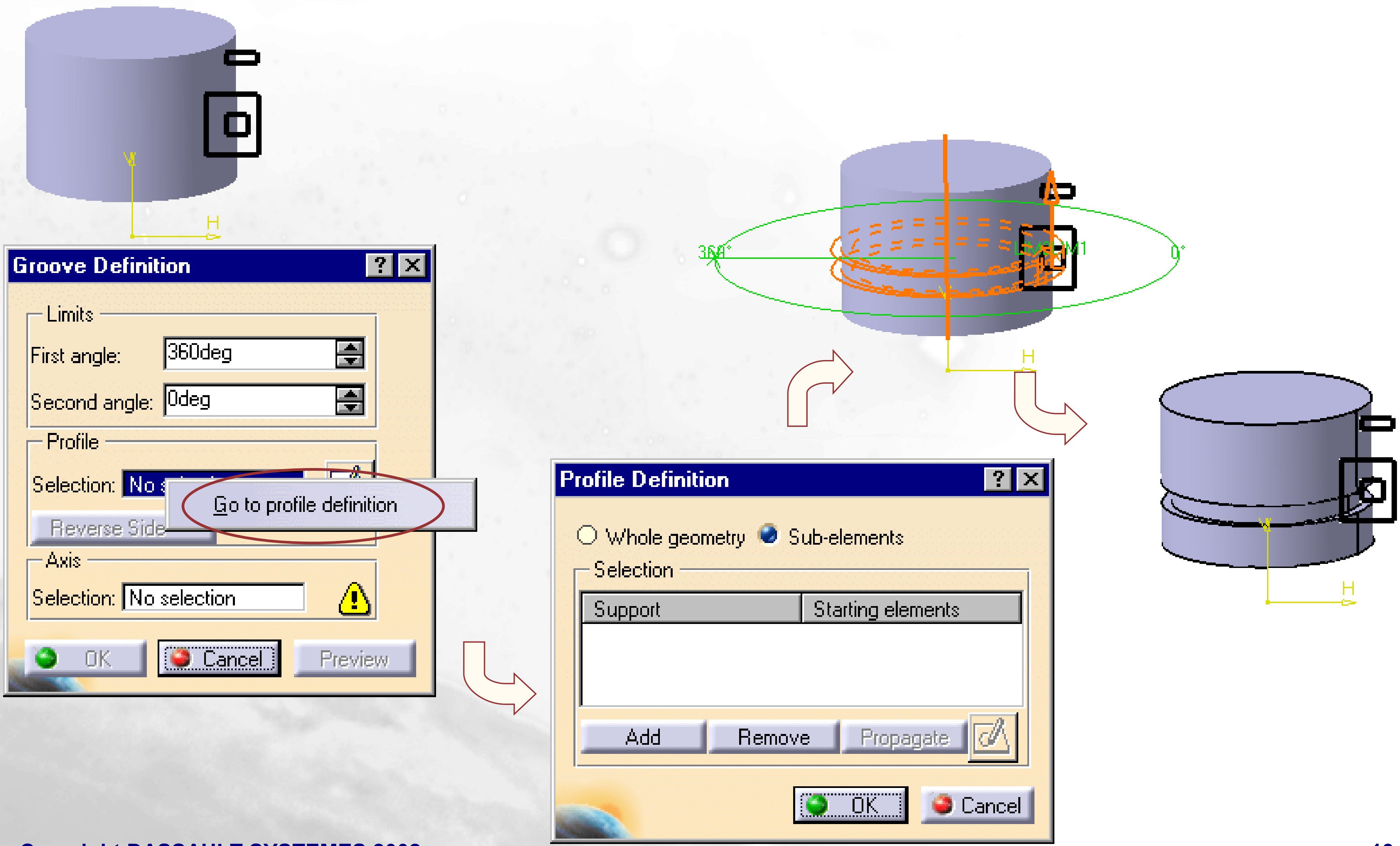
5 Select OK in the dialog box

You get:



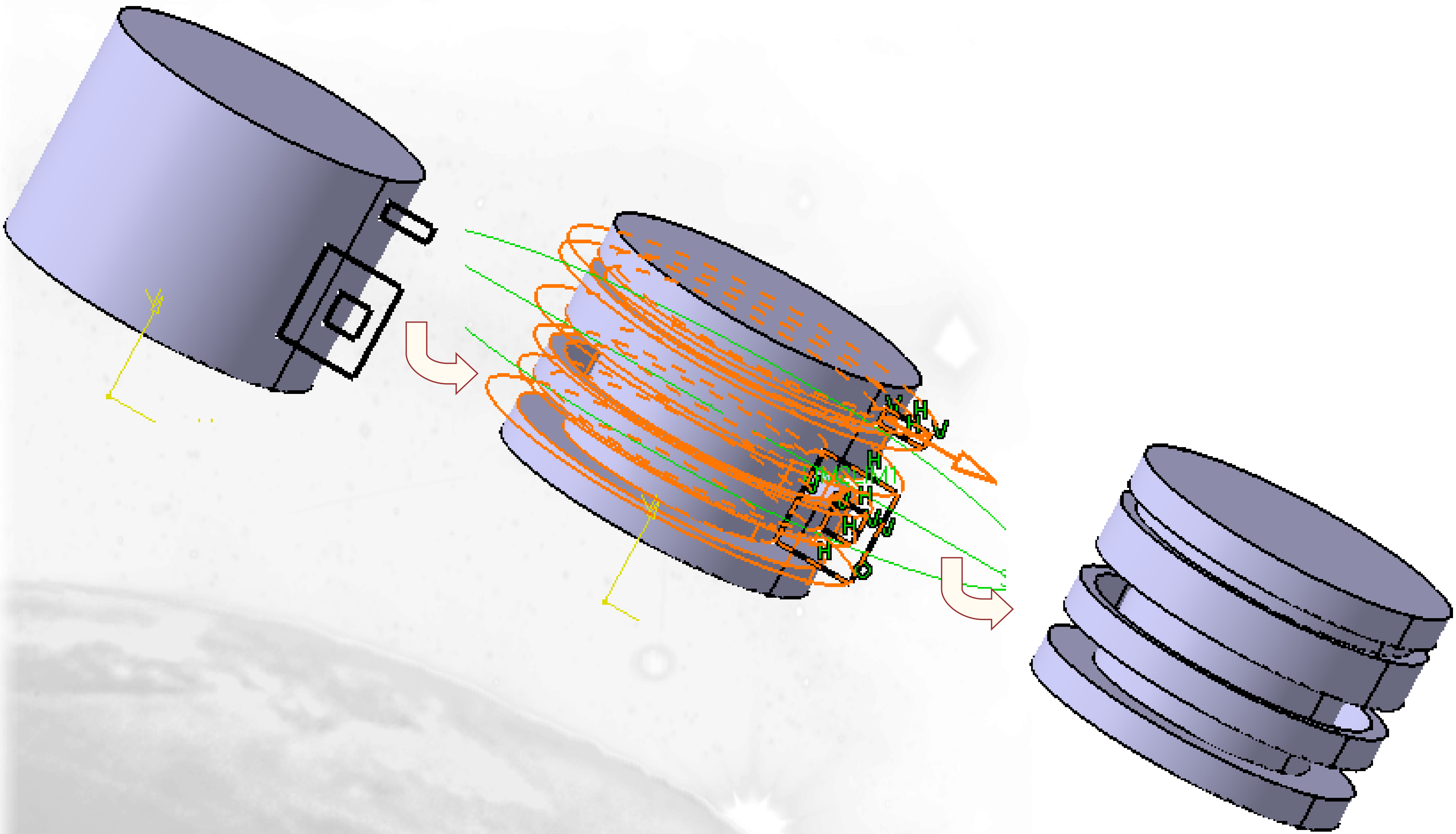
Additional Information (1/3)

You can use sub-elements of a sketch to create grooves, like for pads or pockets



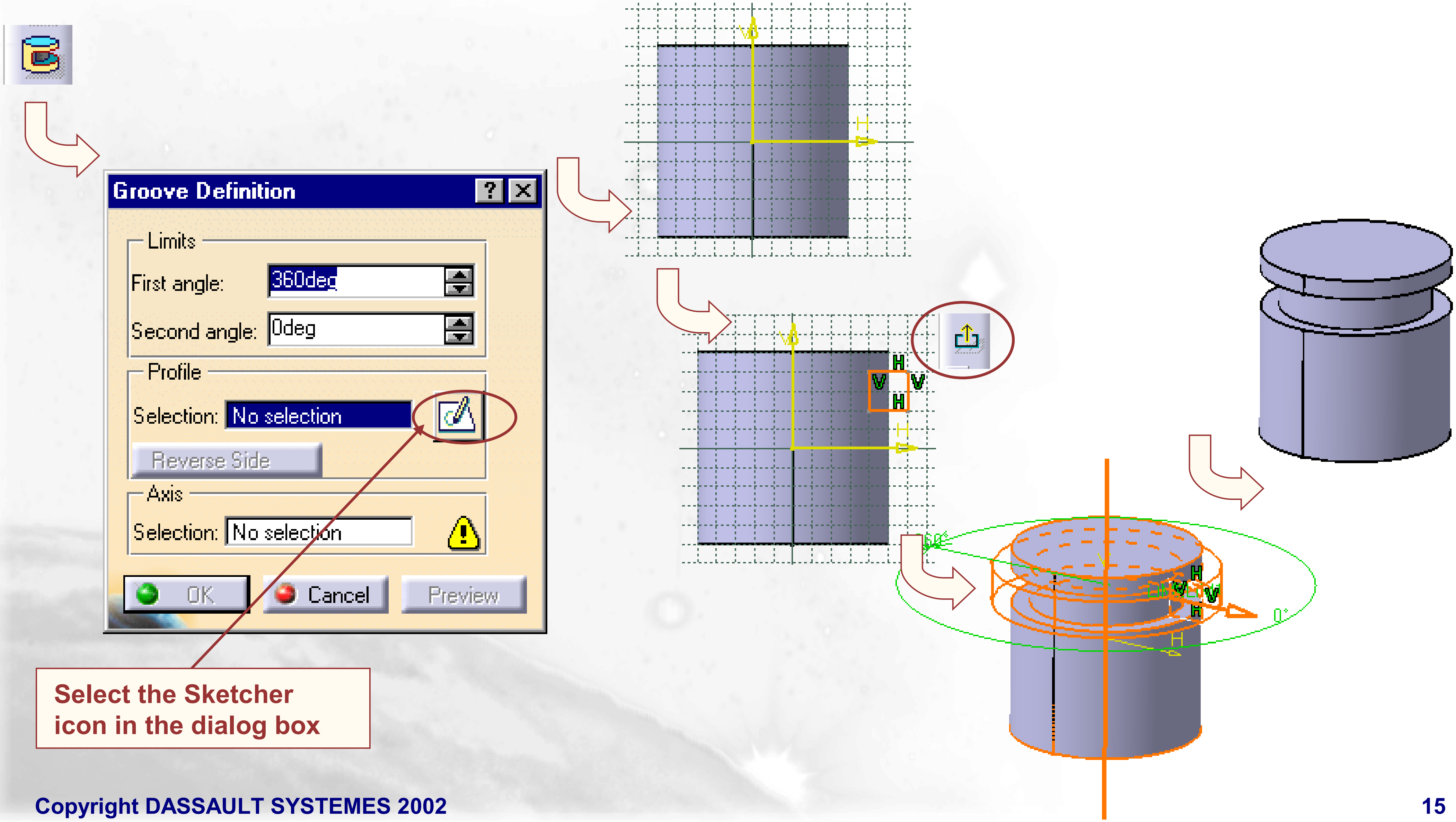
Additional Information (2/3)

You can create Grooves from sketches including several closed profiles. These profiles must not intersect



Additional Information (3/3)

If no sketches have been created when activating the Groove icon, you can access to the sketcher by selecting the Sketcher icon in the dialog box. When you have completed the sketch, you can leave the sketcher then you will return to the Groove creation



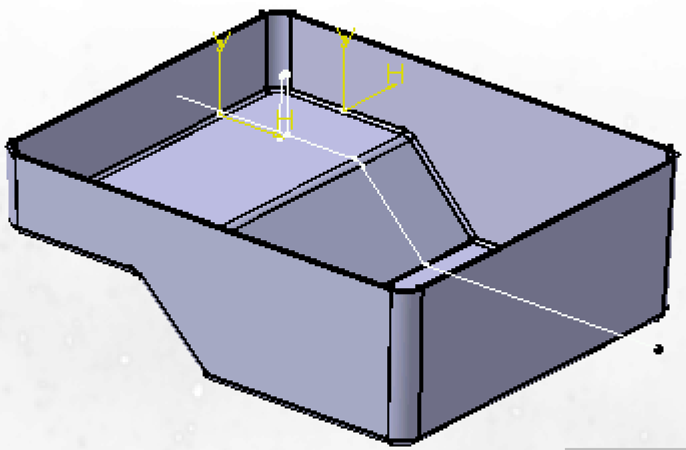
Creating Ribs and Slots

In this lesson we will learn how to create the sketch based features known as Ribs and Slots

- **Creating Ribs**
- **Creating Slots**

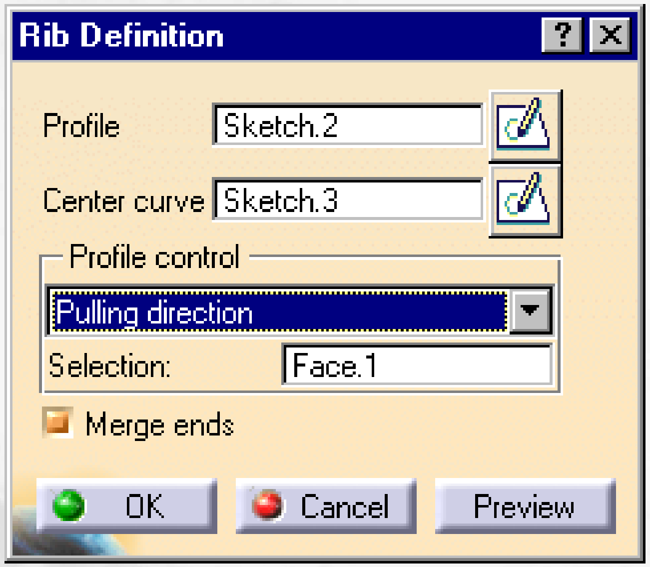
What is a Rib ?

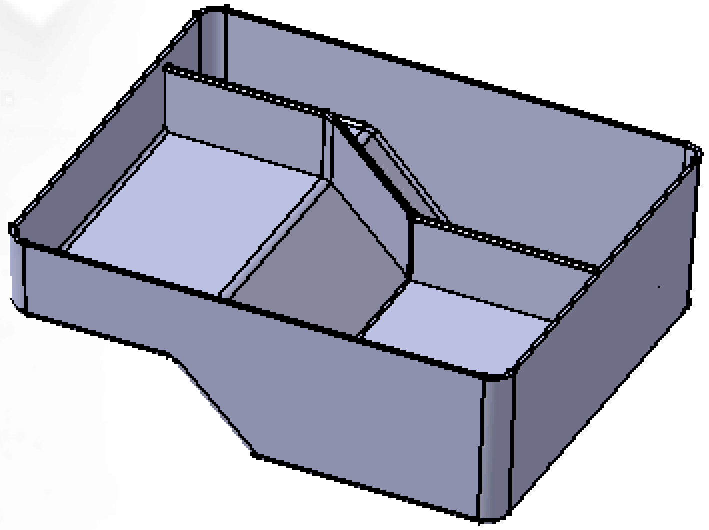
A Rib is a profile swept along an open or closed Center Curve to create a 3D feature



The profile can be swept along an open or a closed center curve to create the feature

The center curve does not have to extend to the end, Merge Ends can be used to extend or shorten the rib to its proper wall

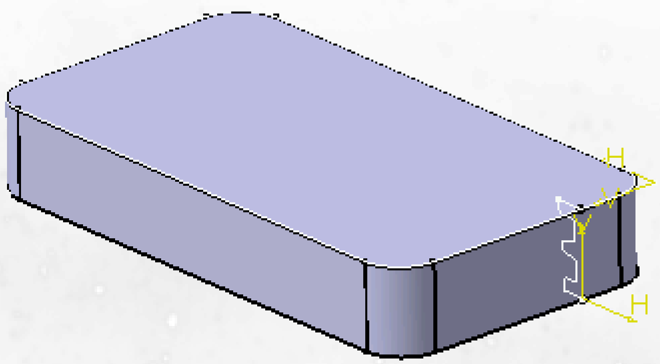




The Profile of the Rib can be controlled by simply using one of the 3 choices under the Profile control section of the window

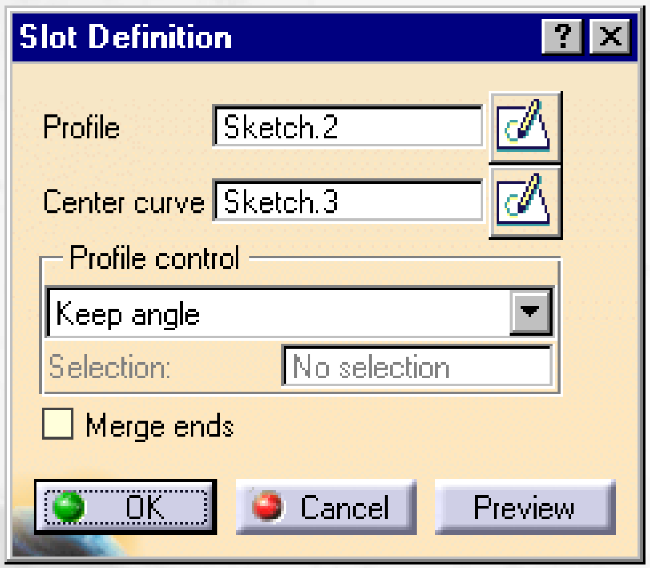
What is a Slot ?

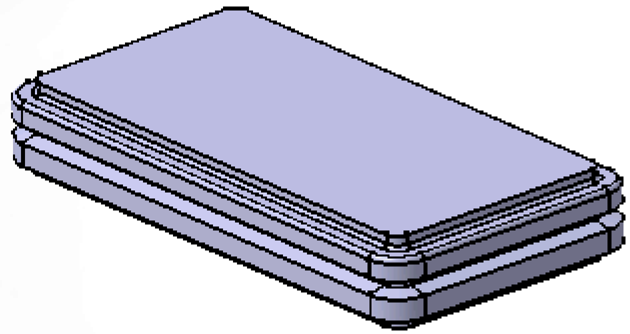
A slot is a profile that is swept along an open or closed Center Curve to remove material from a solid



The profile can be swept along an open or a closed center curve to remove the material

The center curve does not have to extend to the end, Merge Ends can be used to extend or shorten the slot to its proper wall





The Profile of the Slot can be controlled by simply using one of the 3 choices under the Profile control section of the window

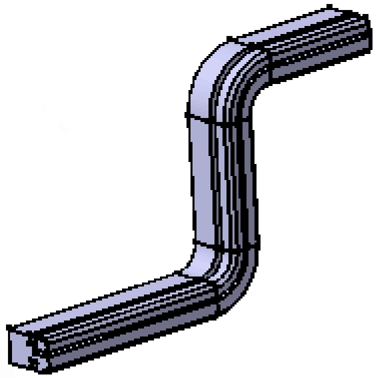
When Should we Use Ribs and Slots ?

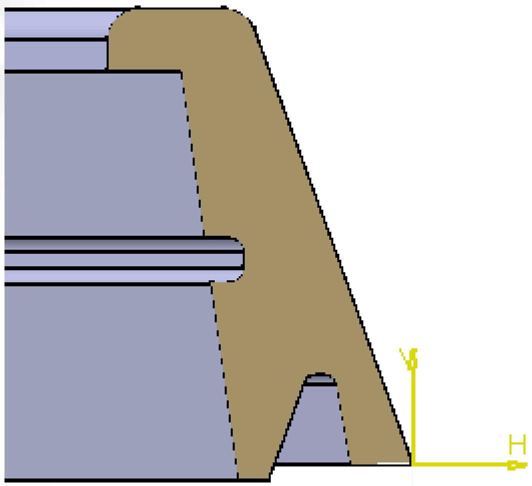
You will find Ribs useful when you need to sweep profiles from one surface to another

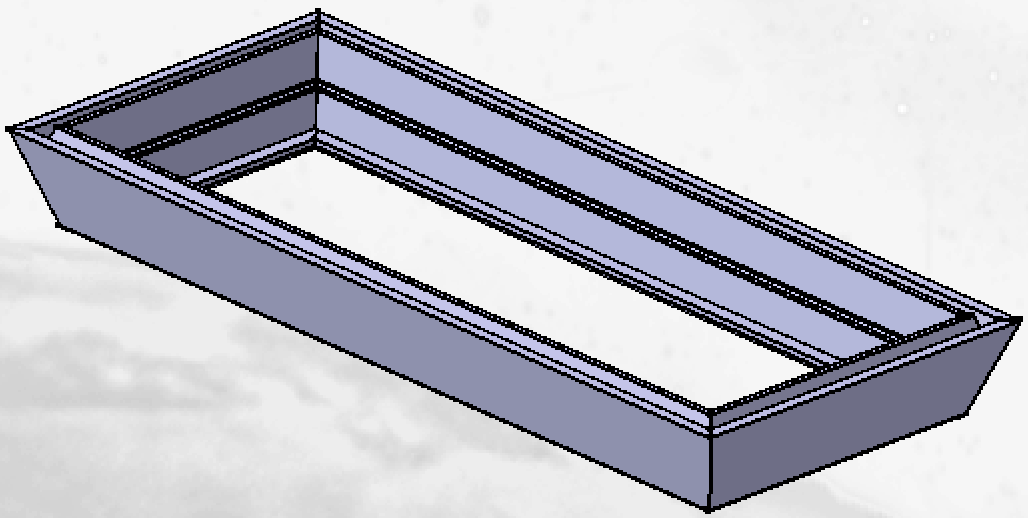
Ribs and Slots will also be useful to create complex walls of parts that have many details in them. Here you can control your complexity in one sketch and not have many small sketches or geometric features to work with

Slots and Ribs can be created on Planar as well as 3D Center Curves

Also a Rib can be used to create a pipe by sweeping a profile along a center curve



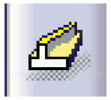


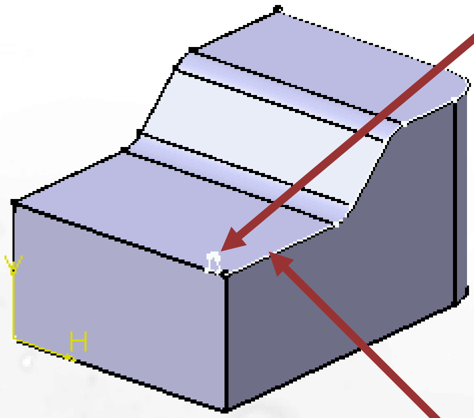


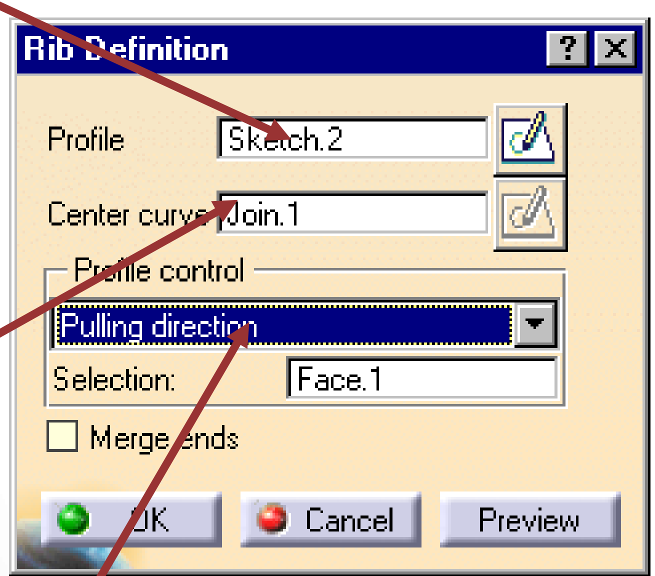
Creating a Simple Rib

2 Select the Profile to be swept

1 Select the Rib icon



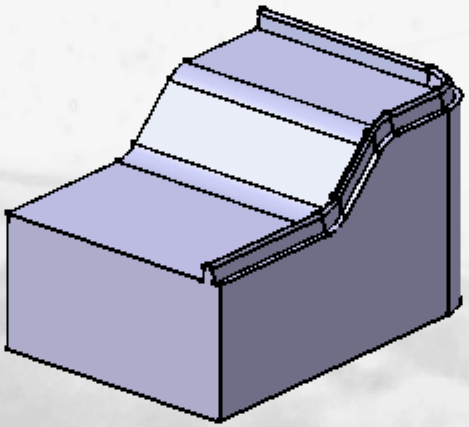


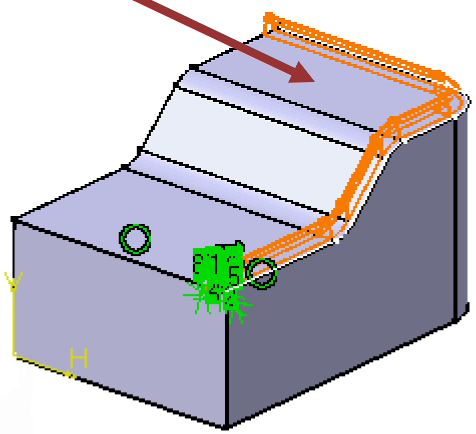


3 Select the 3D Center Curve

5 The Rib is displayed, select OK to create the Rib

4 Select Pulling direction and then select the indicated surface

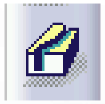




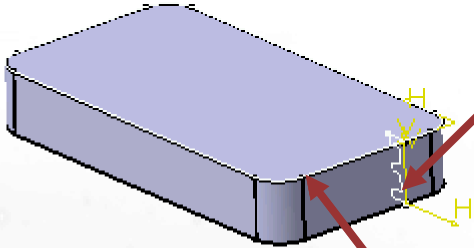
The 3 Dimensional curve was created in the Wire Frame workbench

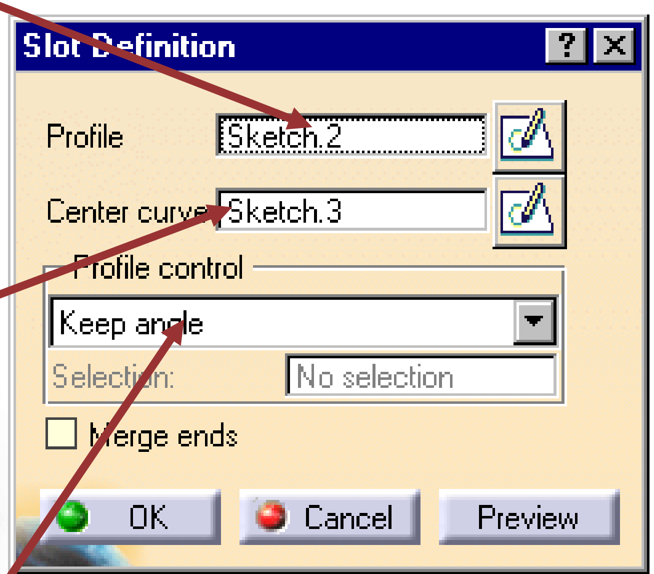
Creating a Slot

1 Select the Slot icon



2 Select the Profile to be swept

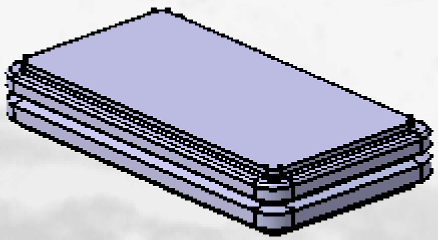


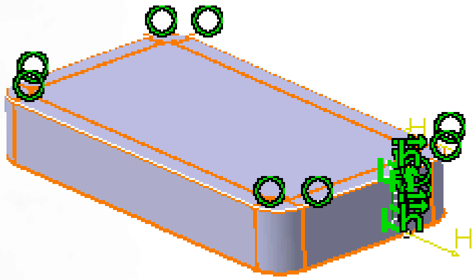


A Sketch for the Center Curve and the profile must exist prior to the icon being available for selection

3 Select the Center Curve to Sweep around

4 The Slot is previewed and in this case the Profile Control is left at Keep Angle. Select OK to create the Rib

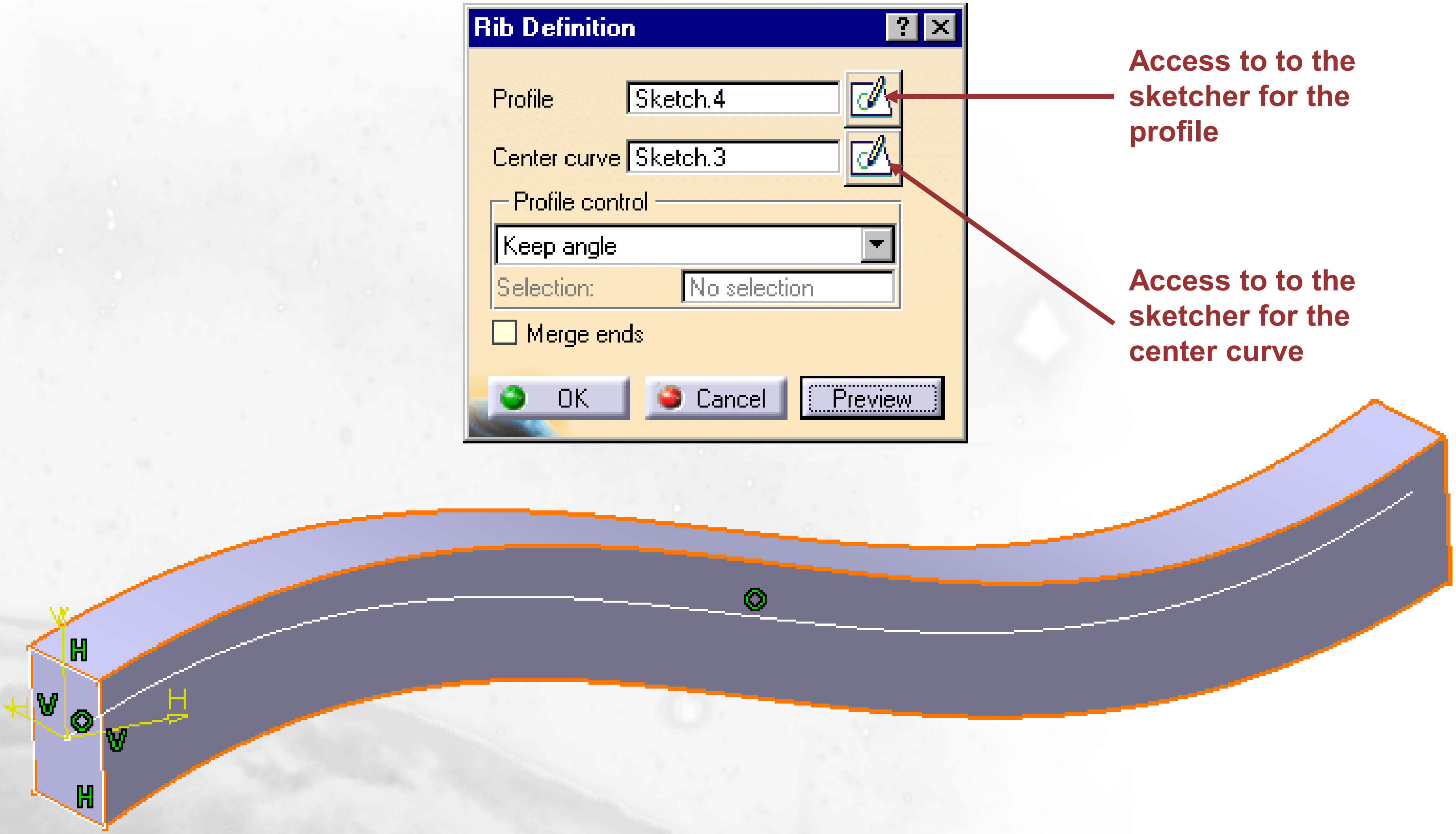




The depth of the profile must be equal to or less than the radius of the Center Curve

Additional Information (1/5)

Capability to edit the profile and center curve sketches during rib or slot creation or edition

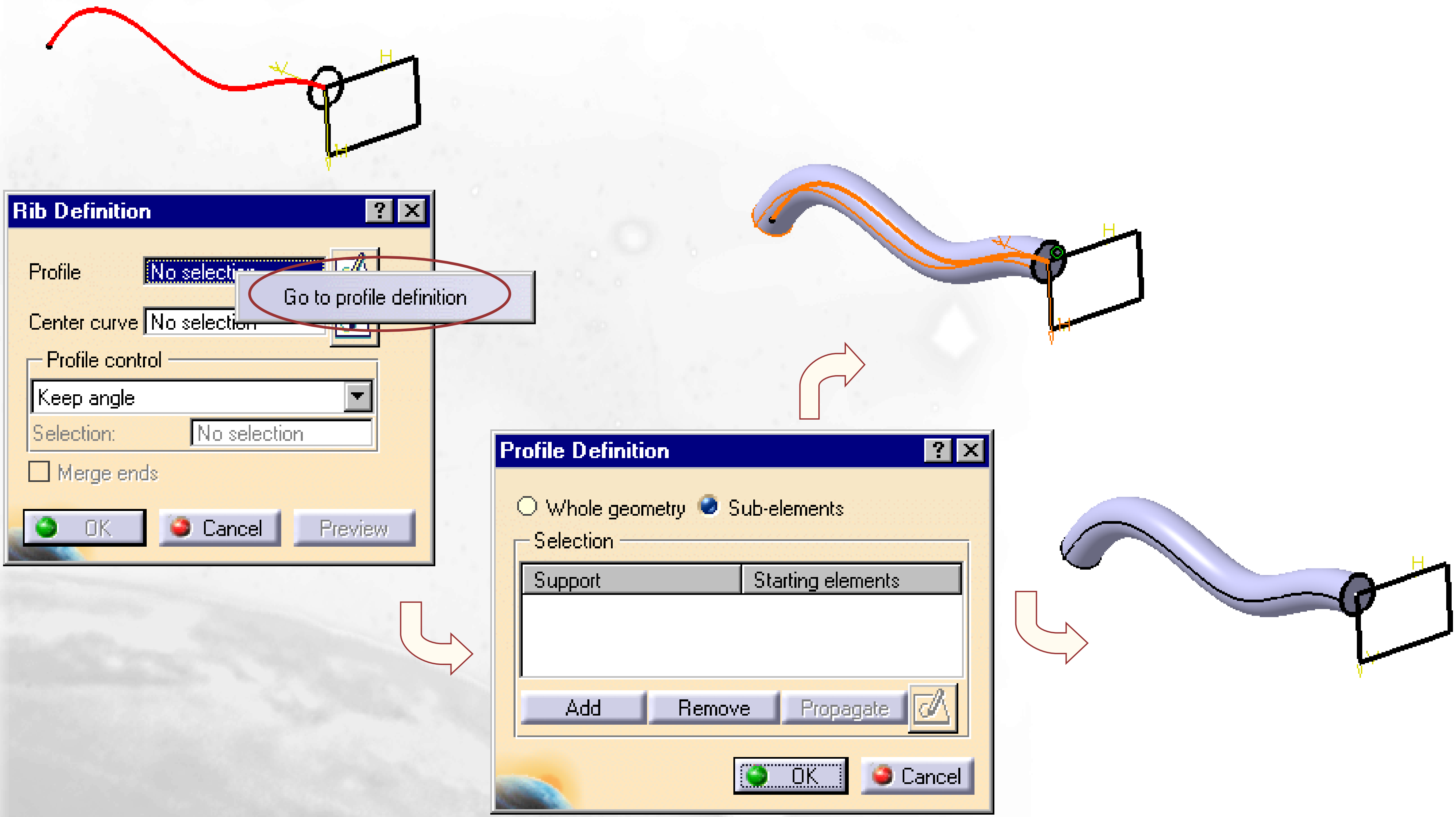


Access to to the sketcher for the profile

Access to to the sketcher for the center curve

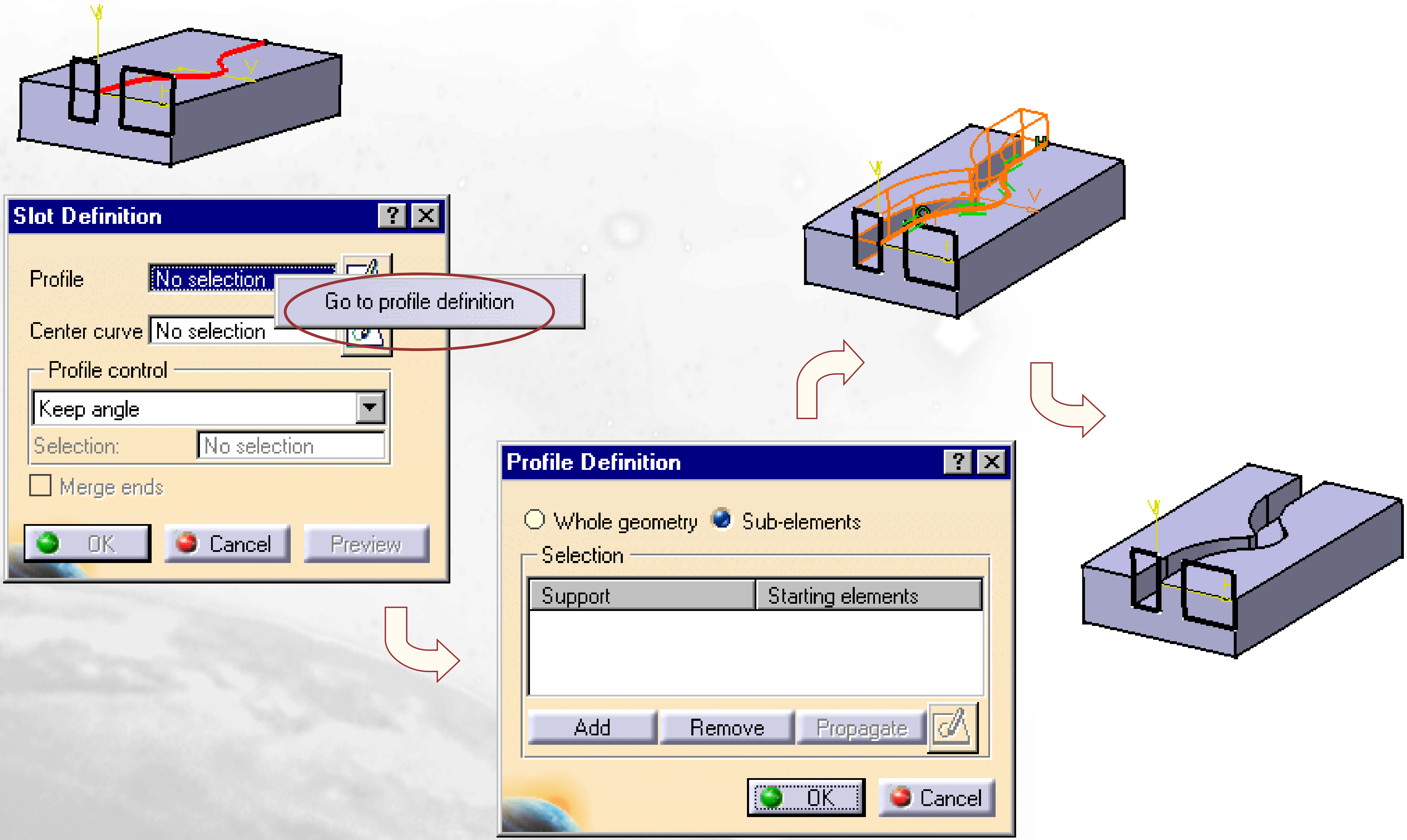
Additional Information (2/5)

You can use sub-elements of a sketch to create ribs, like for pads or pockets



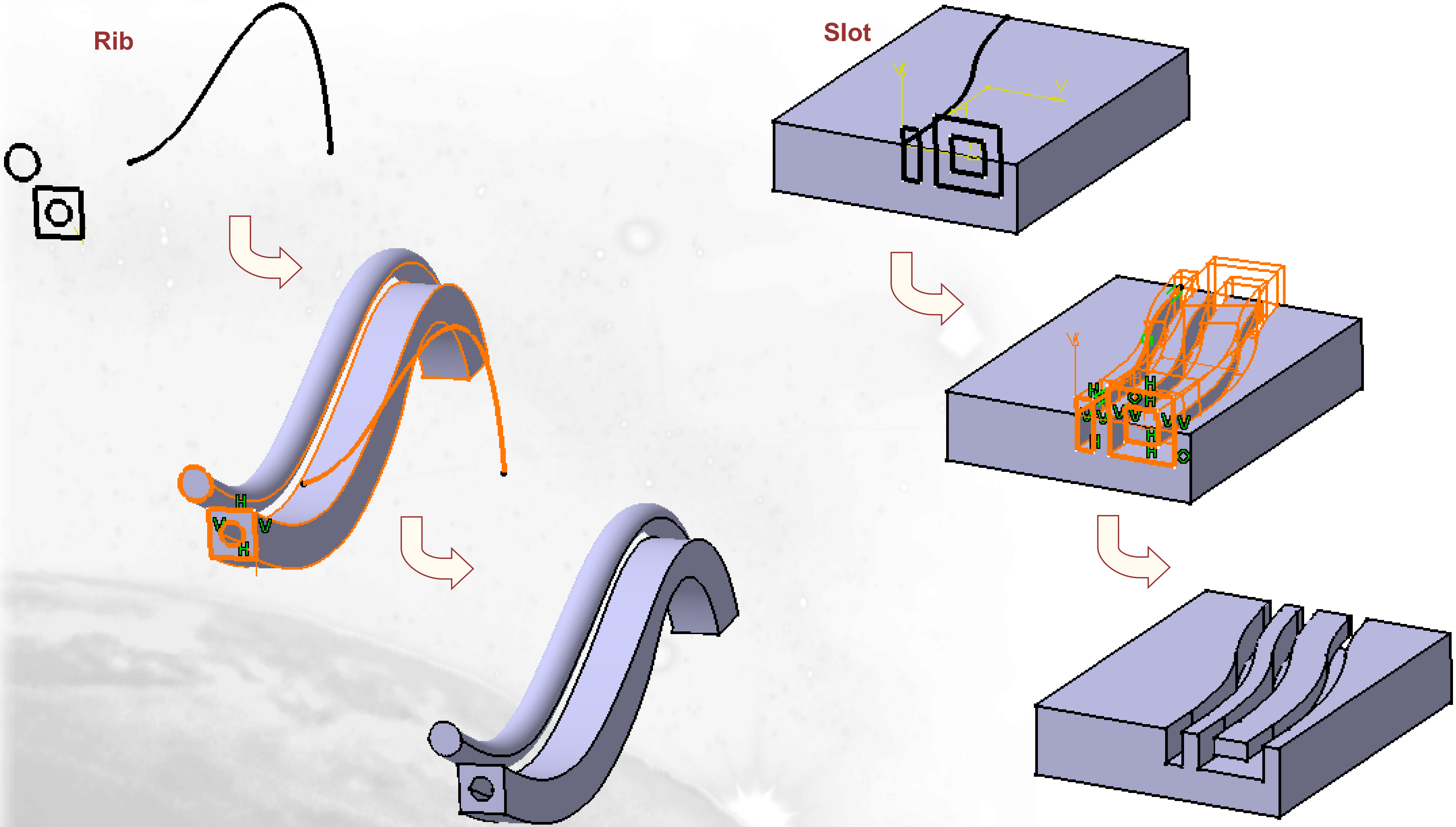
Additional Information (3/5)

You can use sub-elements of a sketch to create slots, like for pads or pockets



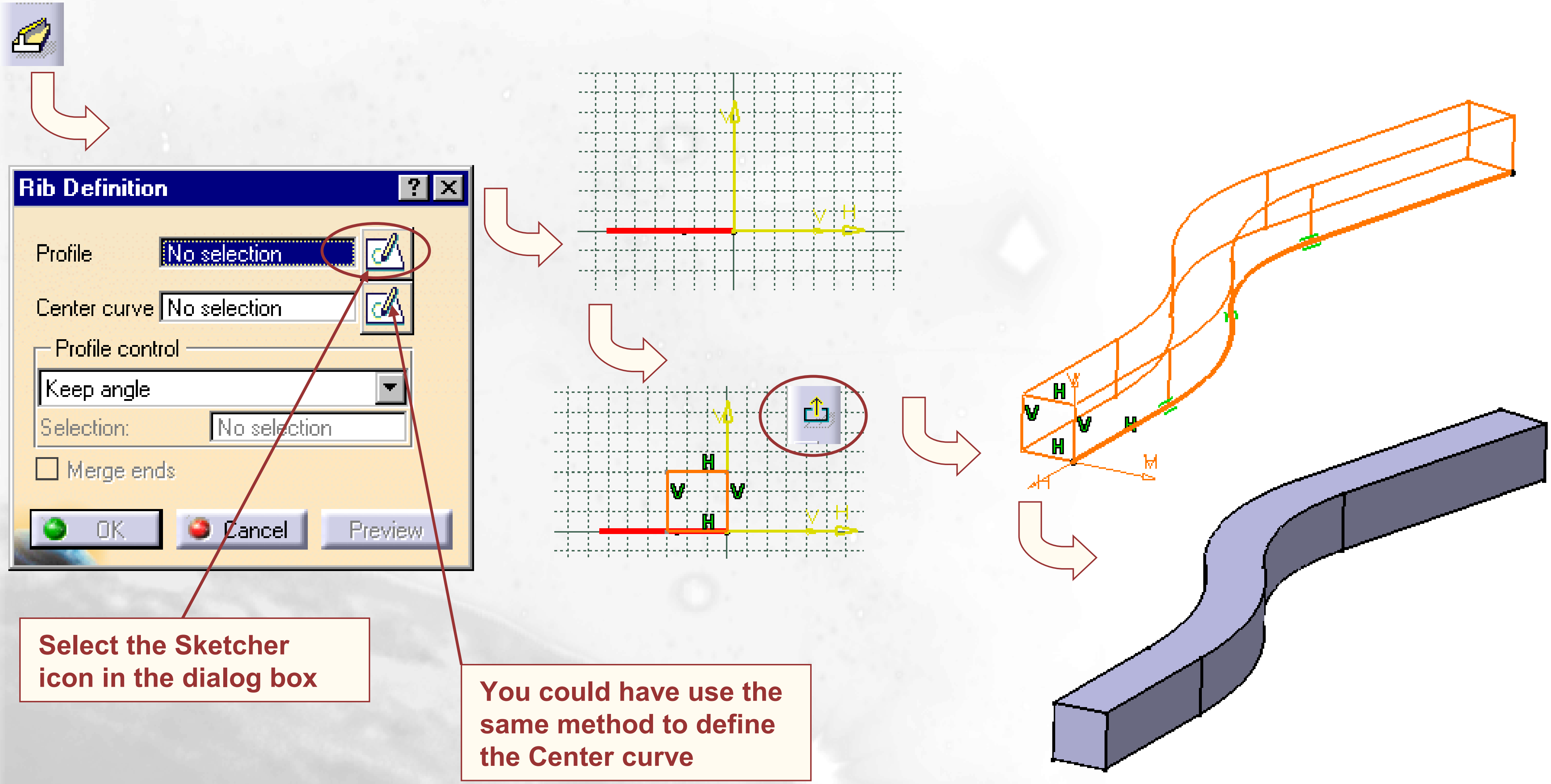
Additional Information (4/5)

You can create Ribs and Slots from sketches including several closed profiles. These profiles must not intersect



Additional Information (5/5)

If no sketches have been created when activating the Rib or Slot icon, you can access to the sketcher by selecting the Sketcher icon. When you have completed the sketch, you can leave the sketcher then you will return to the Rib or Slot creation



Creating Stiffeners

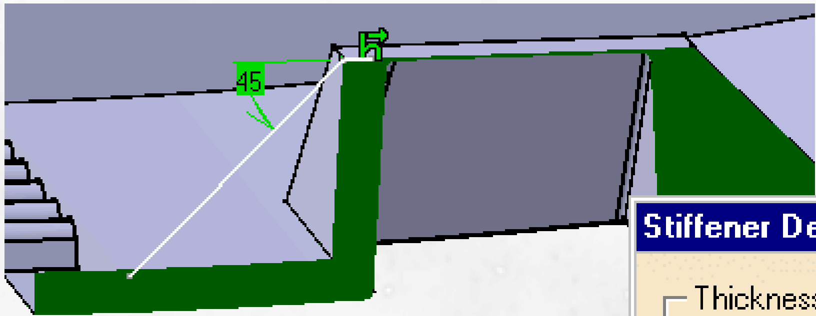
In this lesson we will learn how to create the sketch based features known as Stiffeners

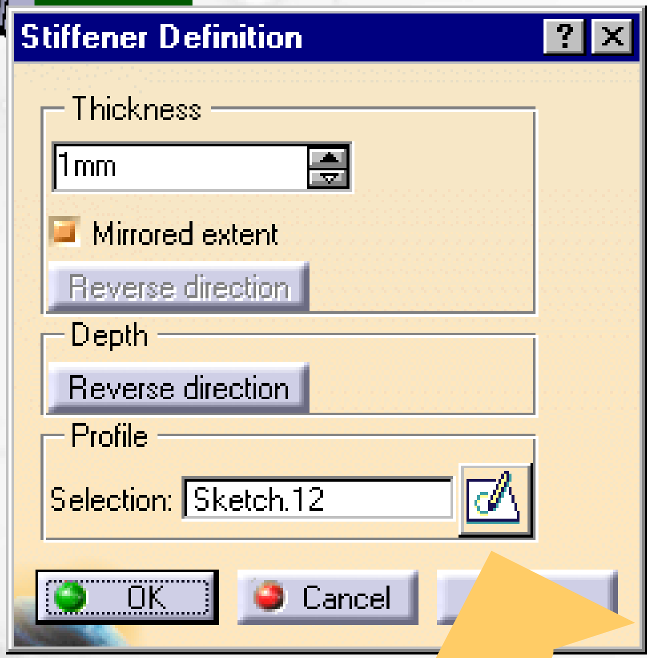
- **Creating a Stiffener**

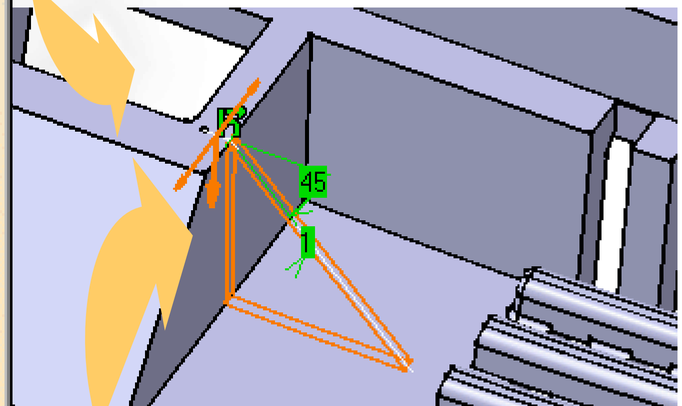
What is a Stiffener ?

A Stiffener is a brace or rib that is added to a wall or a stand-off to add strength to the wall or stand-off and thus prevent breakage. It is commonly found on molded plastic parts or castings

These two arrows are used to control the width of the part, it can be either symmetrical or all on one side or the other





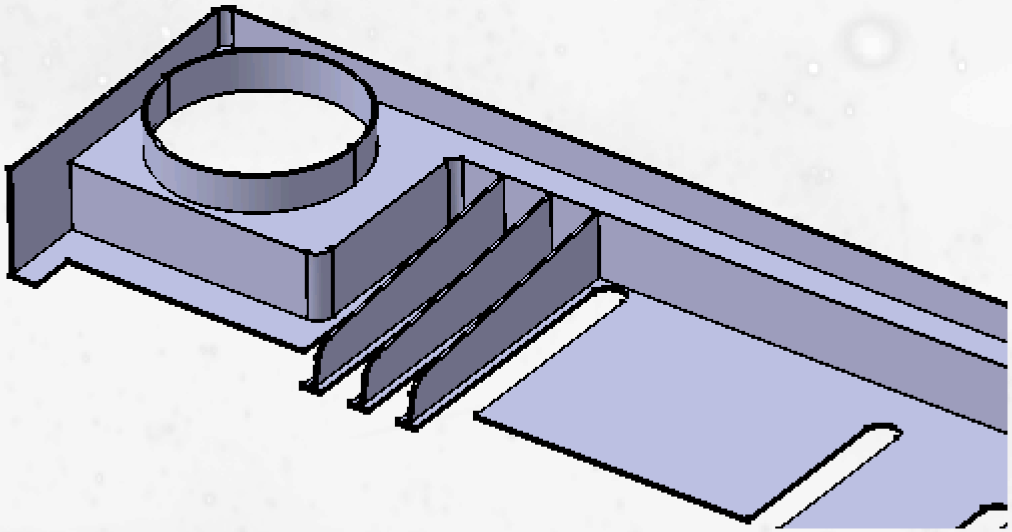


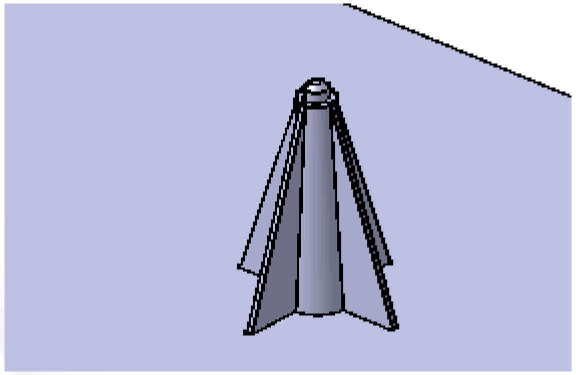
As with most features you can now access the sketch directly by selecting this button

The other arrow is used to control the direction of the rib

When Should we Use Stiffener ?

They can be used when you have a thin wall that you want to be more rigid without increasing the thickness of the wall

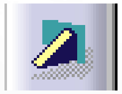




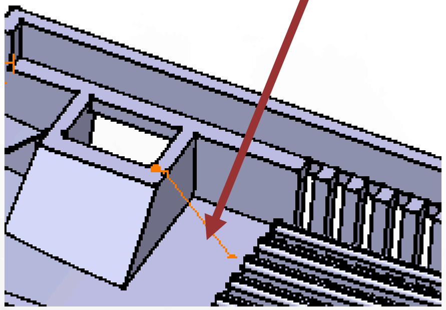
They can also be used when you have tall objects that are used to locate or support other objects and you want to prevent them from breaking off the surface they are attached to

Creating Stiffeners

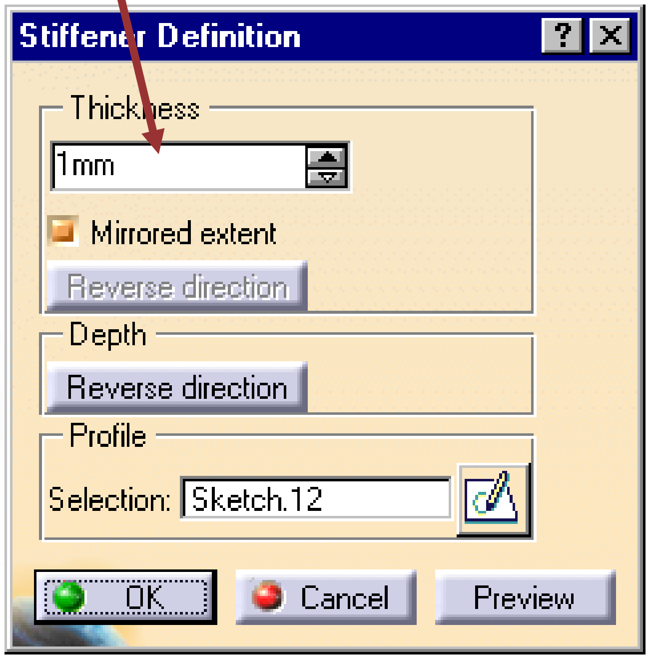
1 Select the Stiffener Icon



2 Make sure the sketch is highlighted

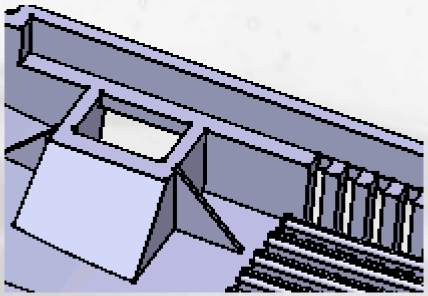


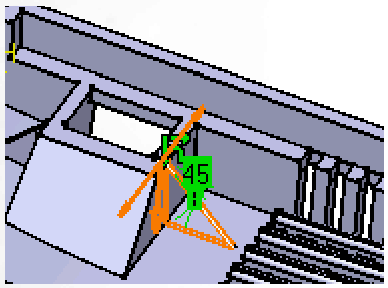
3 Key in the Thickness of the Stiffener



You will find that in many cases need to add a small line segment on to the top of the angled line used to create your stiffener. This allows for a coincidence constraint to be created between the rib and the part

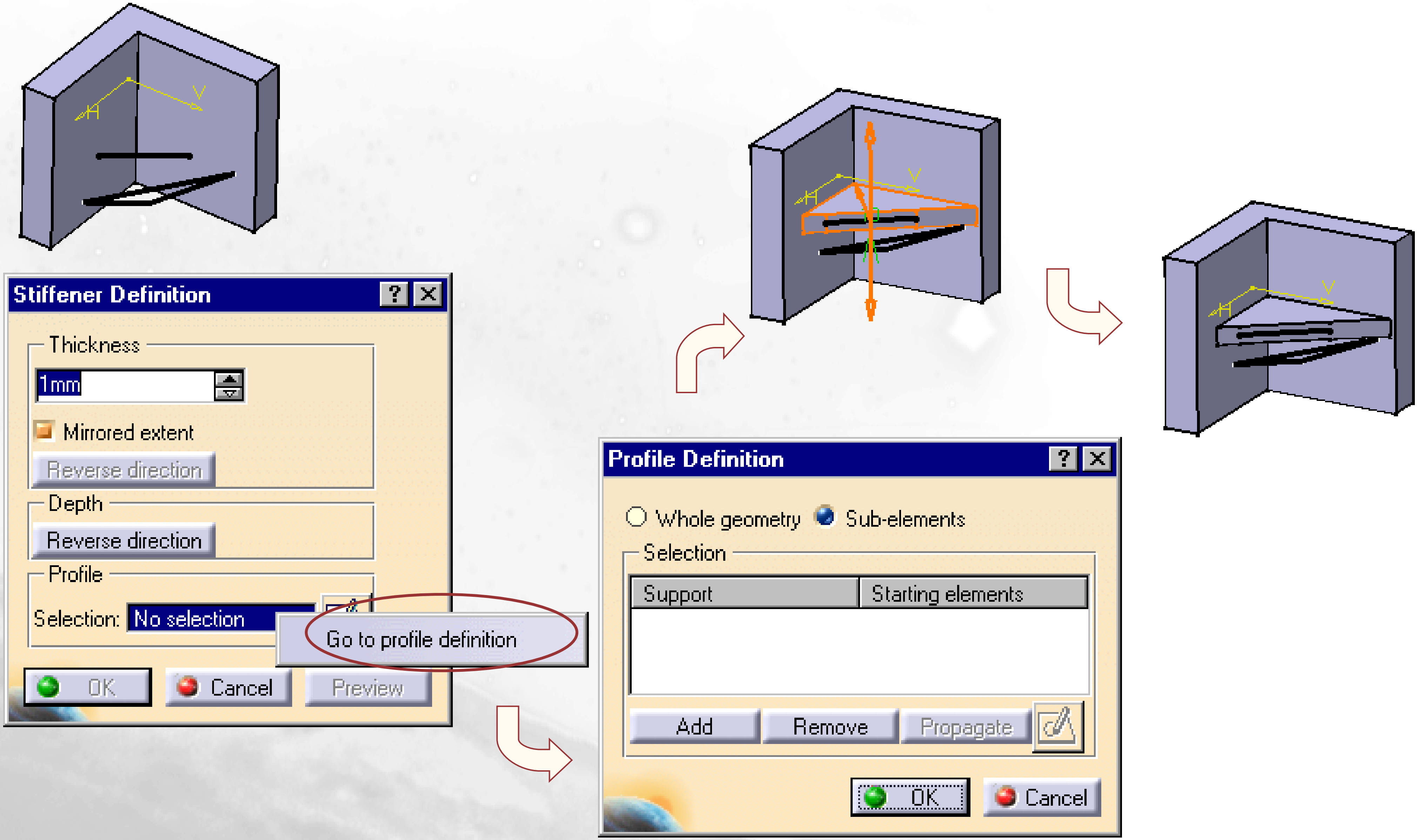
4 If the direction is correct select OK to create the Stiffener





Additional Information

You can use sub-elements of a sketch to create stiffeners, like for pads or pockets



Creating Lofts

You will learn how to create Lofts and Removed Lofts

- Creating a Simple Loft
- Remove Lofts
- Coupling
- Closing Points

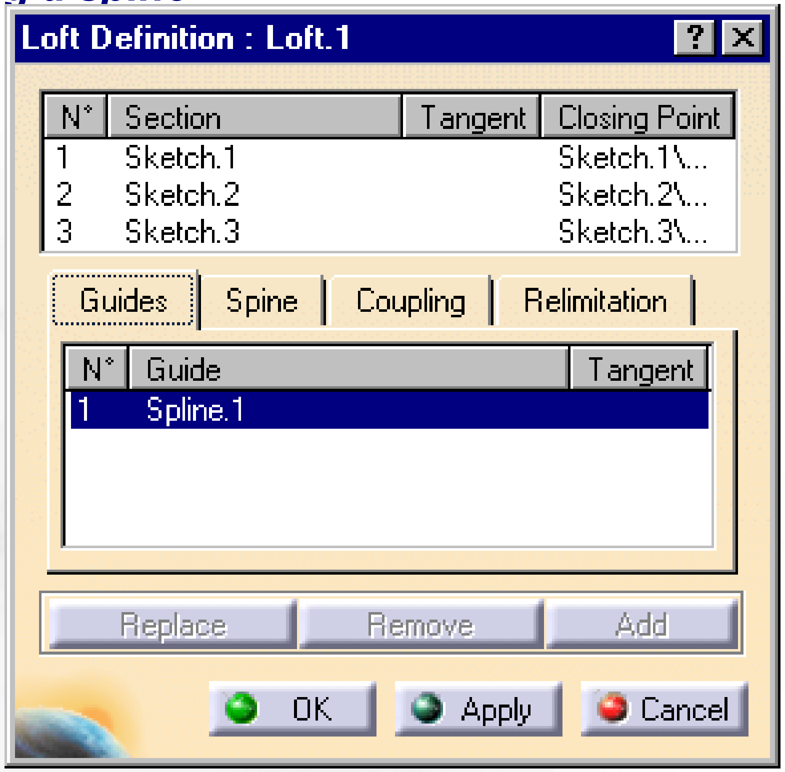
Creating Simple Lofts

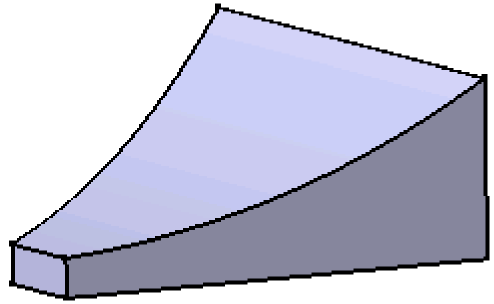
In this lesson we will learn how to create Lofts

- **Creating Simple Lofts**

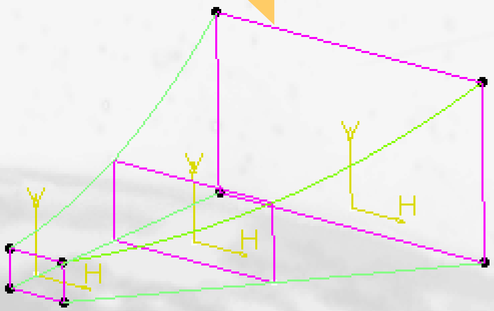
What is a Loft ?

A Loft can be a Positive (add material) or Negative (substract material) solid that is generated by two or more planar sections swept along a spine





Guide Line



Closing Point

Directional arrows are provided to get the proper orientation of the Loft

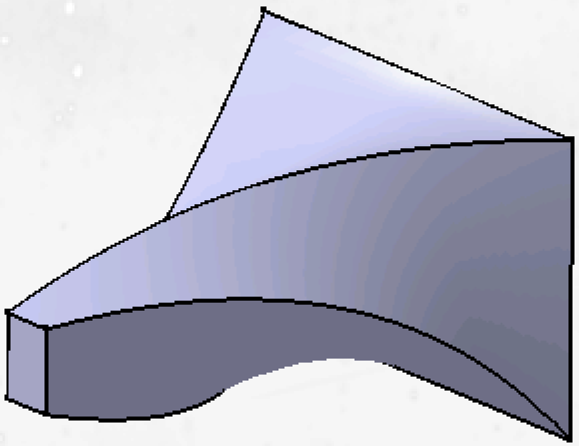
The Planar sections can be connected with Guide Lines

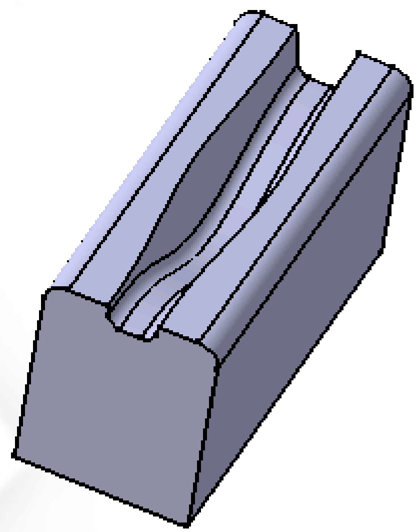
Be aware Closing Points on the sketch must be aligned to get the proper orientation of the sections otherwise the loft would be twisted

When Should we Use Lofts and Removed Lofts ?

Lofts can be used for several reasons.

- ***To create complex solids.***
- ***Or to create some transition geometry between two existing solids in a part***





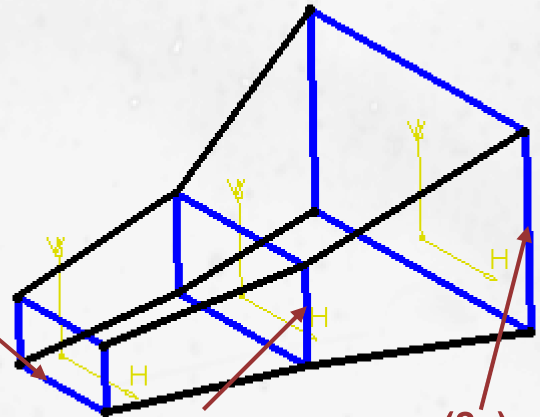
Removed Lofts are used the same way when you wish to subtract a transitioned surface from another solid

Loft Creation : Guide Lines

1 Select the Loft icon

(1)

2 Select the sections the loft is going to pass through. The order in which you select the sections is important, it will define the order of connection between the sections



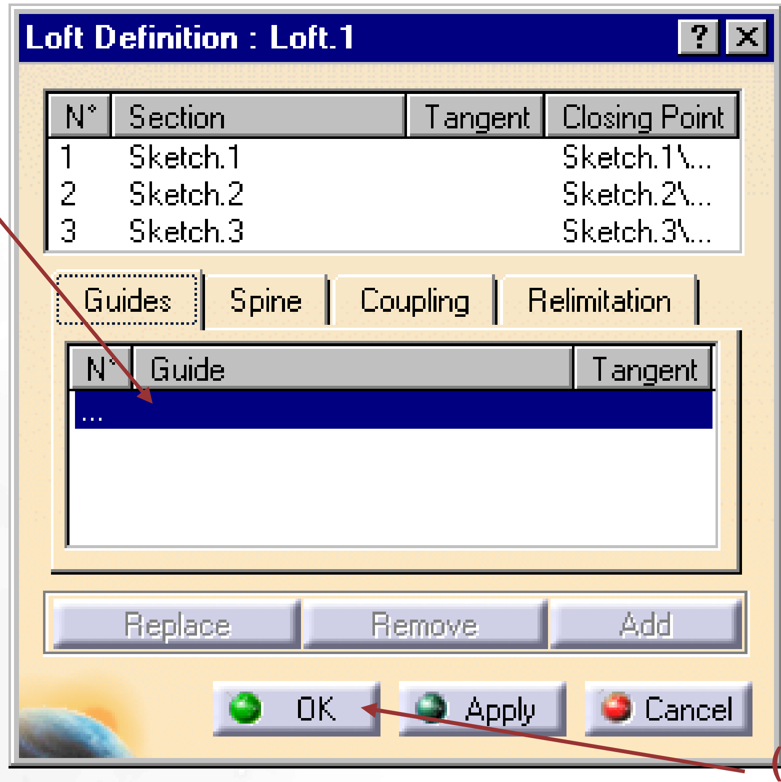
(2a)

(2b)

(2c)

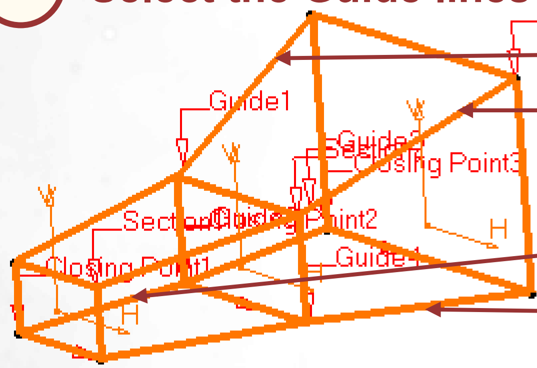
3 Select the Guide option from the dialog box

(3)



(5)

4 Select the Guide lines



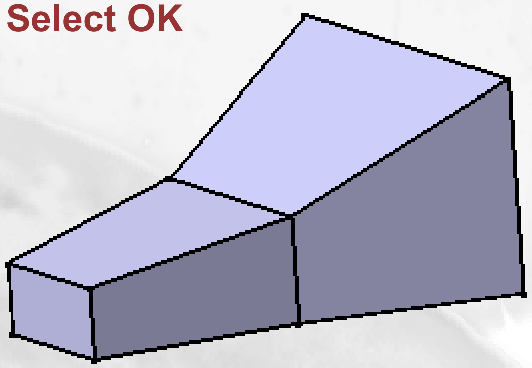
(4a)

(4b)

(4c)

(4d)

5 Select OK



The loft is passing through the sections and it is limited by the guide lines

Loft Creation : Spine

1 Select the Loft icon

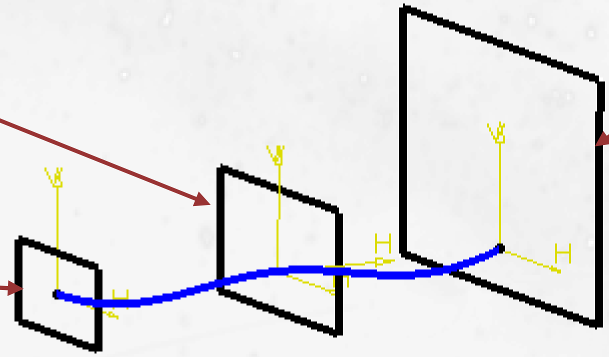
(1)

2 Select the sections the loft is going to pass through. The order in which you select the sections is important, it will define the order of connection between the sections

(2a)

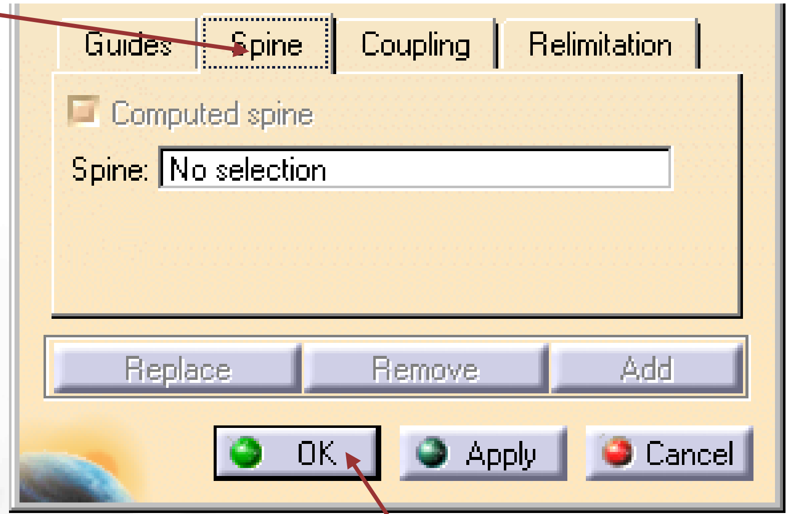
(2b)

(2c)



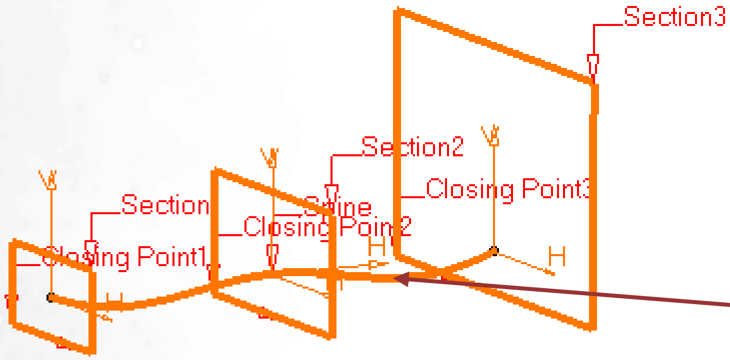
3 Select the Spine tab from the dialog box

(3)



(5)

4 Select the Spine



(4)

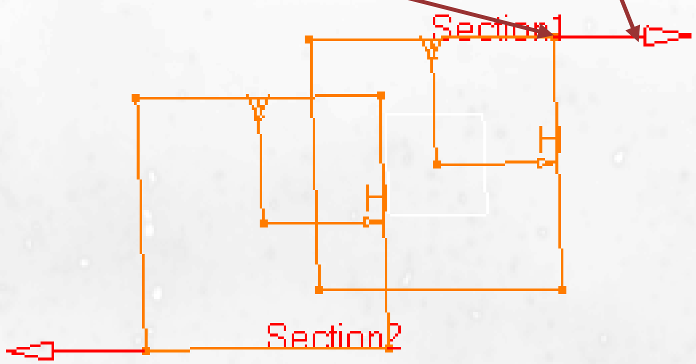
5 Select OK

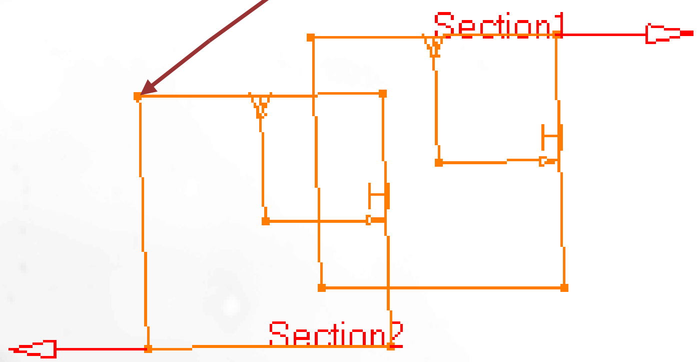
From the first to the last section, the solid is generated by doing a sweep along the spine. The sections always stay fix in space

Loft Creation : Closing Point and Orientation

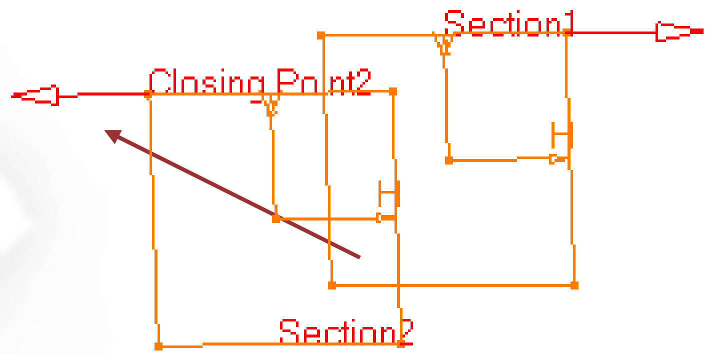
Orientation of the section

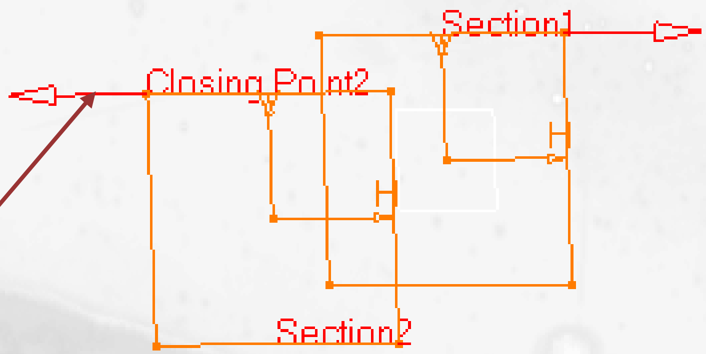
Closing point of the section



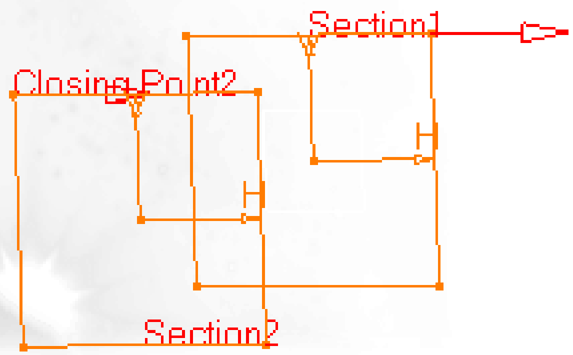


To change the closing point of a section, select another point on this section





When selecting another section, it might happen that the section is orientated in the other direction than the previous one, so, to reverse the section orientation select the arrow which indicates the section orientation



Loft Creation : Tangent Surfaces

1 Select the first section

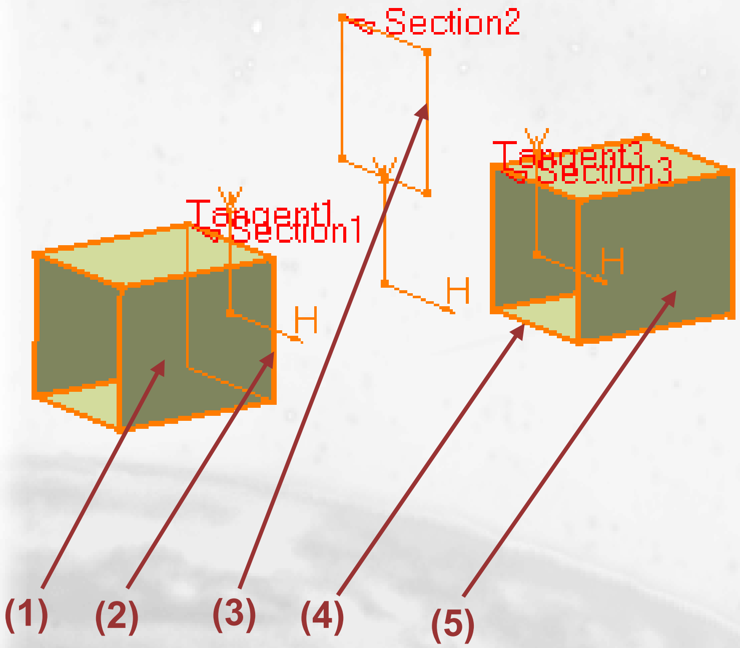
2 Select the surface (corresponding to the first section) the loft will be tangent to

3 Select the intermediary sections

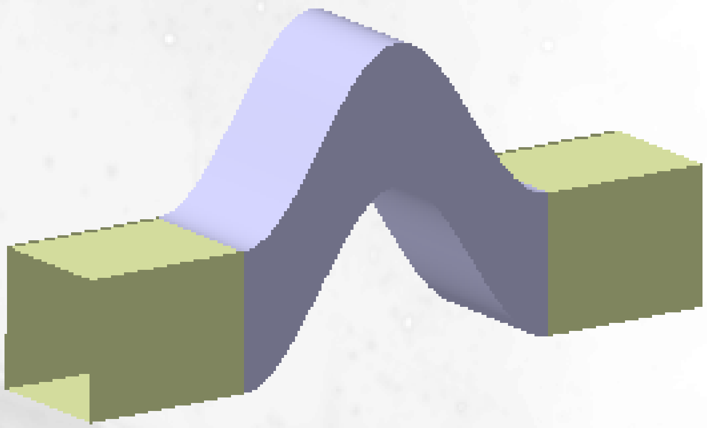
4 Select the last section

5 Select the surface (corresponding to the last section) the loft will be tangent to

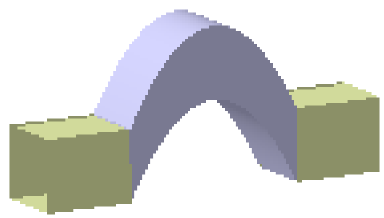
6 Validate



You get :



Result with the same sections but without any tangent surfaces



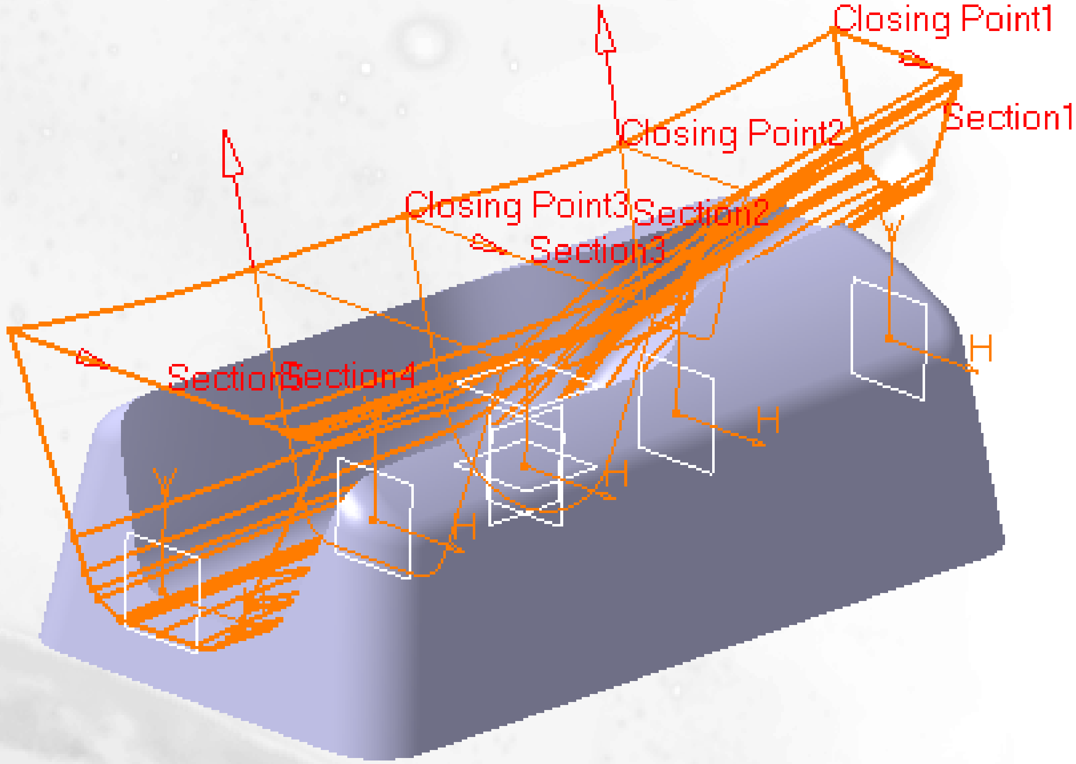
Remove Lofts

In this lesson we will learn how to create Remove Lofts

•Creating Simple Remove Lofts

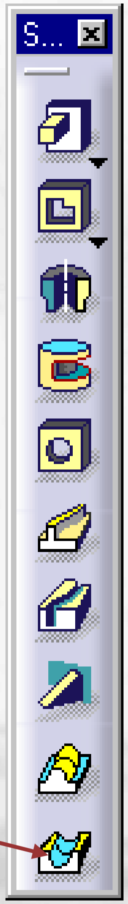
What is Remove Loft Material ?

The Remove Loft capability generates lofted material, by sweeping one or more planar section curves along a computed or user-defined spine, and then removes this material. The material can be made to respect one or more guide curves

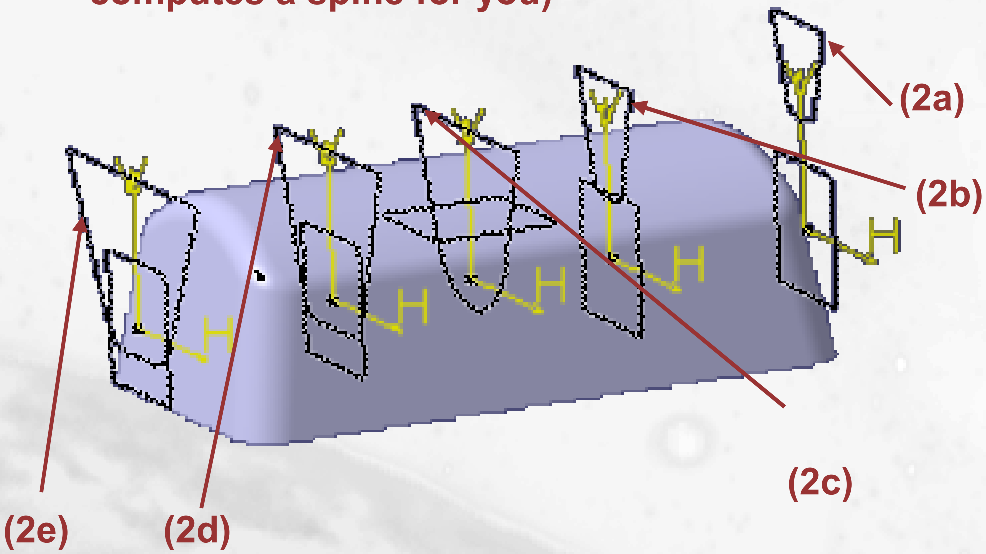


Remove Loft Material

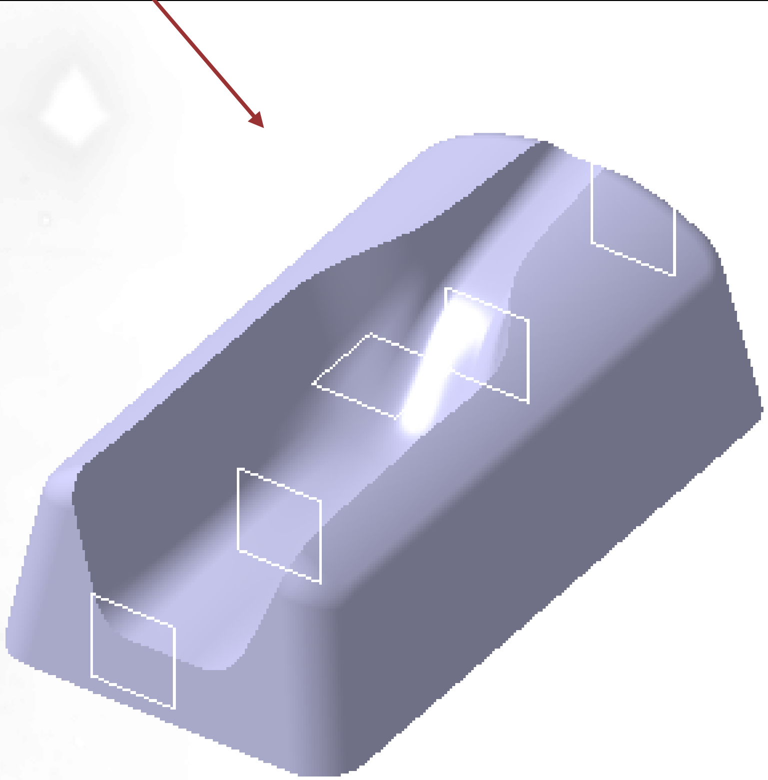
1 Select the Remove Lofted Material icon



2 Select the sections the loft is going to pass through. The order in which you select the sections is important, it will define the order of connection between the sections (You could have defined a spine or several guide lines, if no spine is selected, the system computes a spine for you)



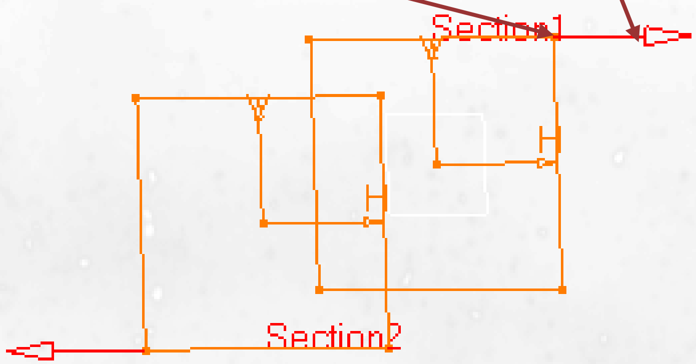
3 Select OK

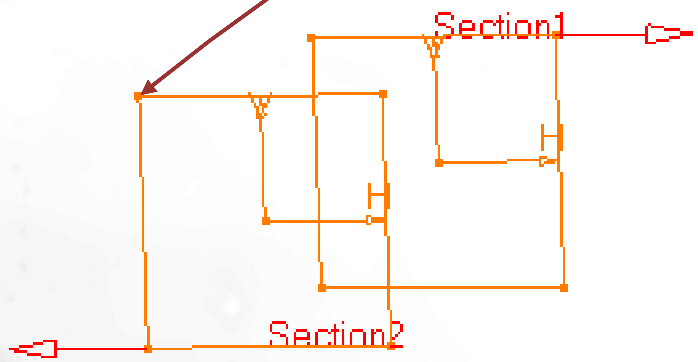


Remove Loft Material : Closing Point and Orientation

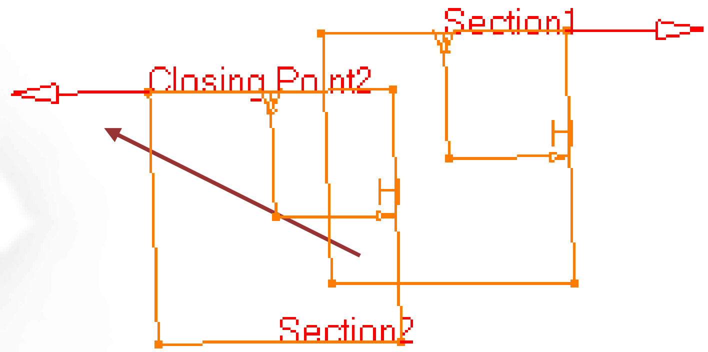
Closing point of the section

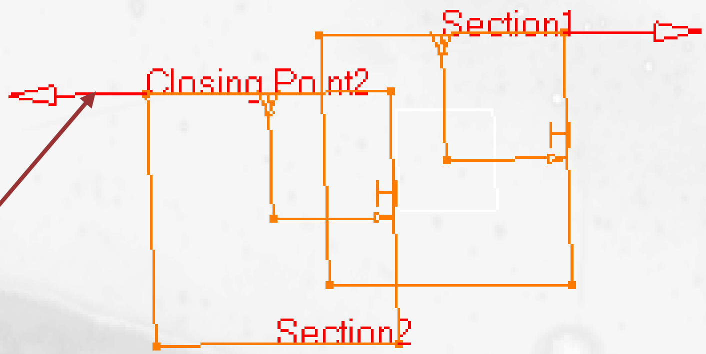
Orientation of the section



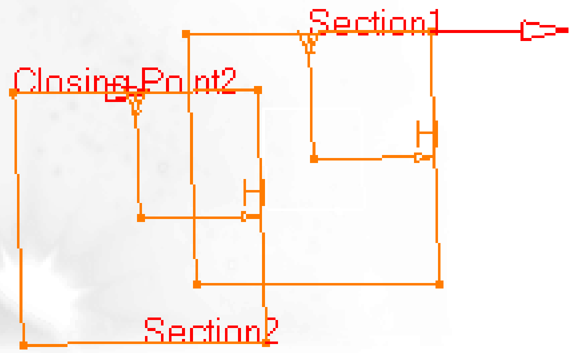


To change the closing point of a section, select another point on this section





When selecting another section, it might happen that the section is orientated in the other direction than the previous one, so, to reverse the section orientation select the arrow which indicates the section orientation



Remove Loft Material : Tangent Surfaces

1 Select the first section

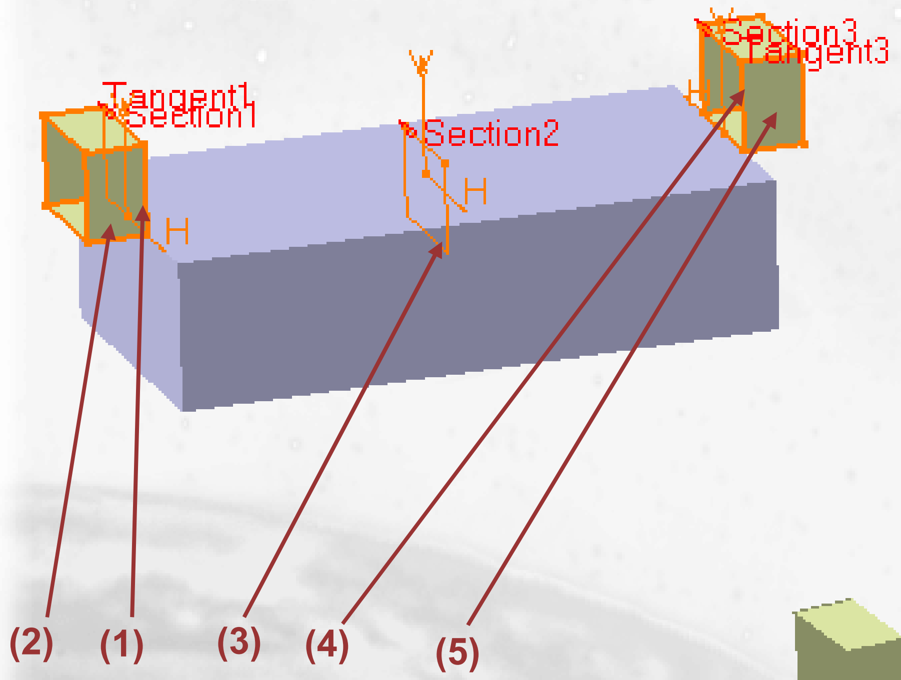
2 Select the surface (corresponding to the first section) the removed loft will be tangent to

3 Select the intermediary sections

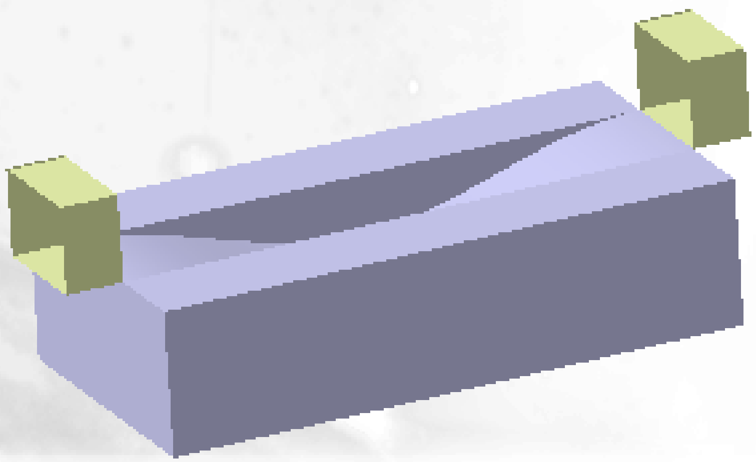
4 Select the last section

5 Select the surface (corresponding to the last section) the removed loft will be tangent to

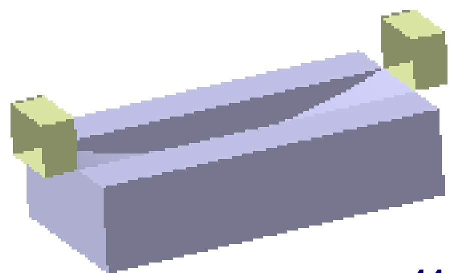
6 Validate



You get :



Result with the same sections but without any tangent surfaces



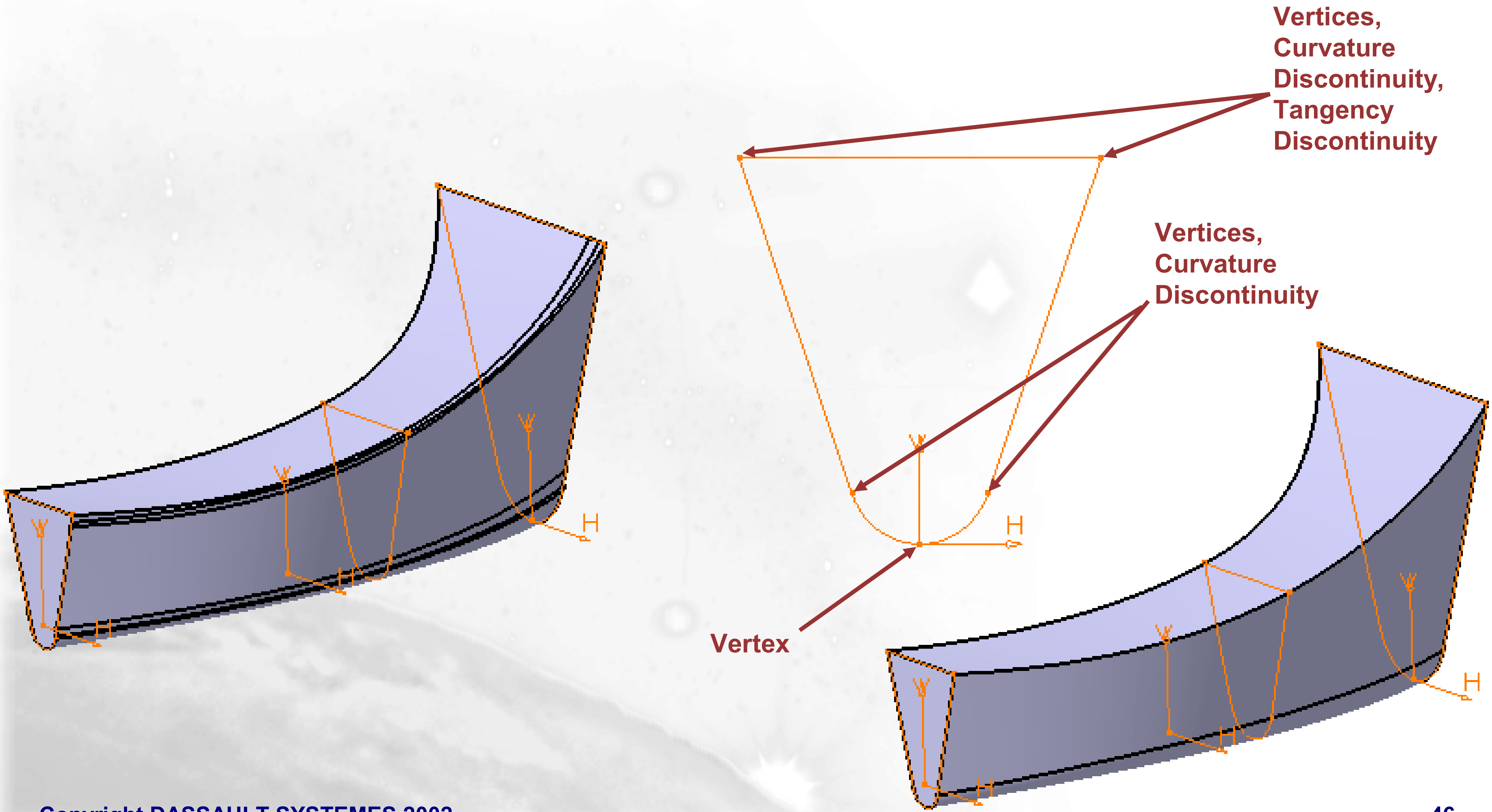
Coupling

In this lesson we will learn how to use Coupling when creating Lofts

•**Coupling when Creating Loft**

What is Coupling when Creating Loft ?

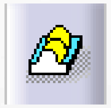
A Coupling tab in the loft and remove loft functions to compute the loft using the total length of the sections (ratio) or between the vertices of the sections or between the curvature discontinuity points of the sections or between the tangency discontinuity points of the sections



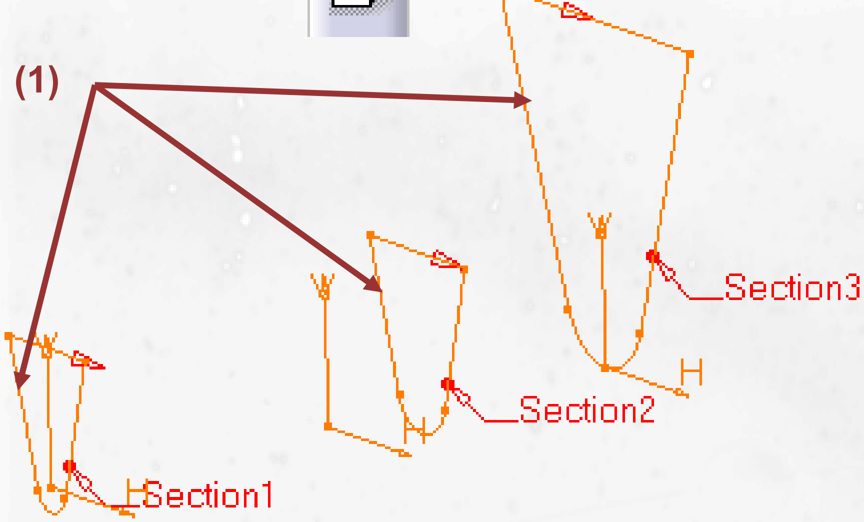
Coupling when Creating Loft

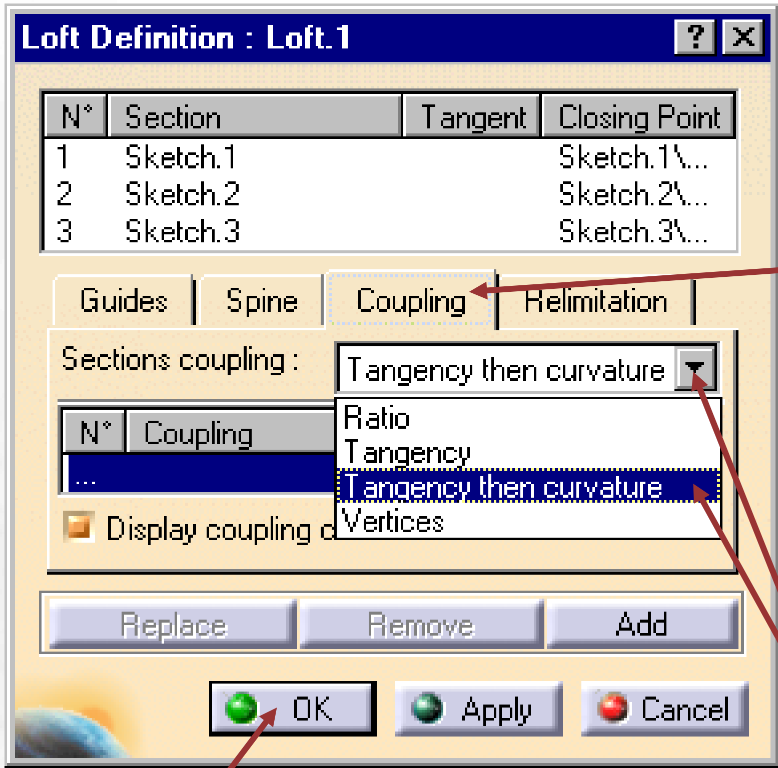
A coupling tab in the loft and remove loft functions to compute the loft on the total length of the sections (ratio) or between the vertices of the sections or between the curvature discontinuity points of the sections or between the tangency discontinuity points of the sections

1. Activate the Loft icon and select and orient the sections.



(1)





(2)

(3)

(4)

2. Select the Coupling tab from the dialog box

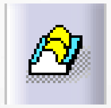
3. Select the desired kind of coupling from the combo

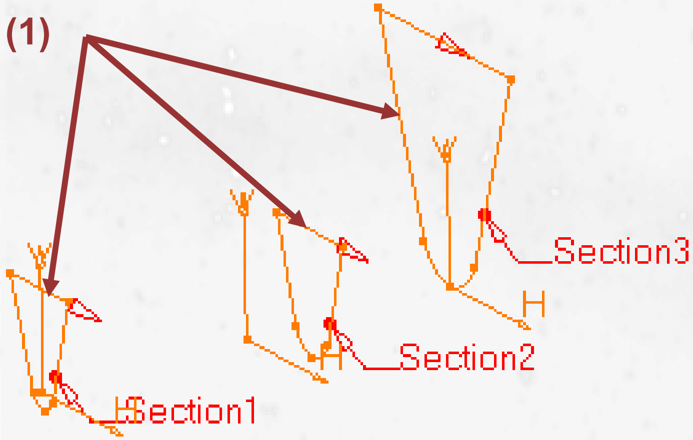
4. Select OK

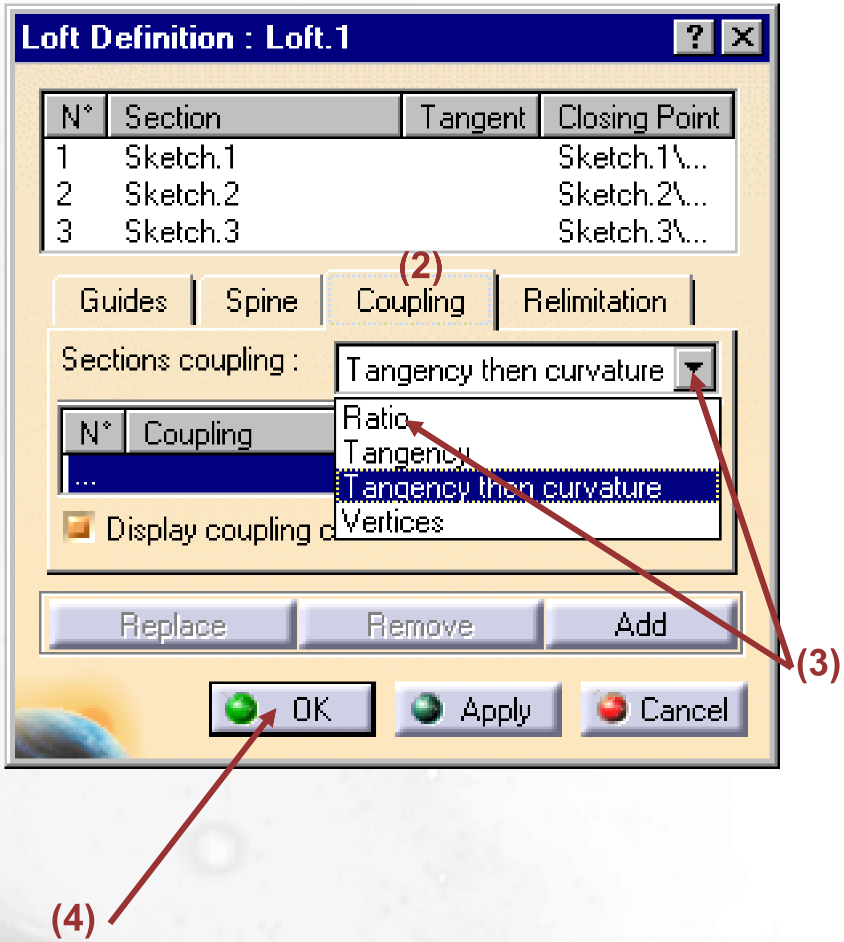
Coupling when Creating Loft : Ratio

A coupling tab in the "loft" and "remove loft" functions to compute the loft using the total length of the sections (ratio)

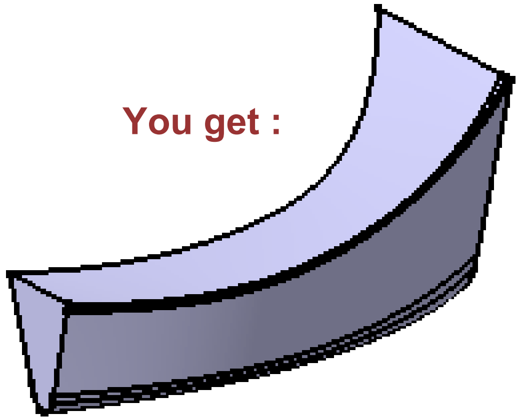
1 Activate the Loft icon and select and orient the sections.







You get :



2 Select the Coupling tab from the dialog box

3 Select Ratio from the combo

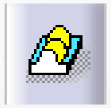
4 Select OK

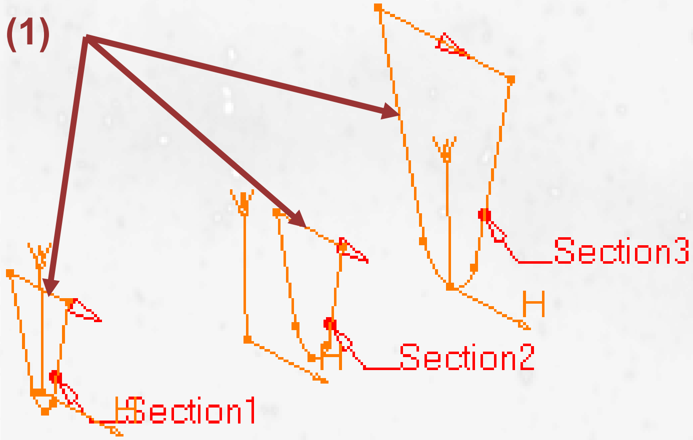
The solid is passing through the sections and the variation between the sections is computed by a ratio corresponding to the length of each section

Coupling when Creating Loft : Tangency

A coupling tab in the loft and remove loft functions to compute the loft between the tangency discontinuity points of the sections

1 Activate the Loft icon and select and orient the sections.

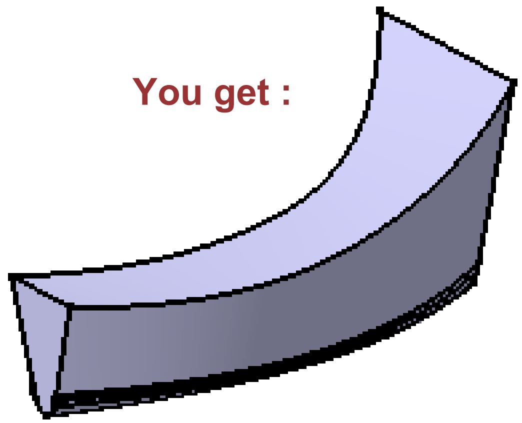




(3)

(4)

You get :



2 Select the Coupling tab from the dialog box

3 Select Tangency Discontinuities from the combo

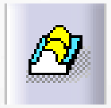
4 Select OK

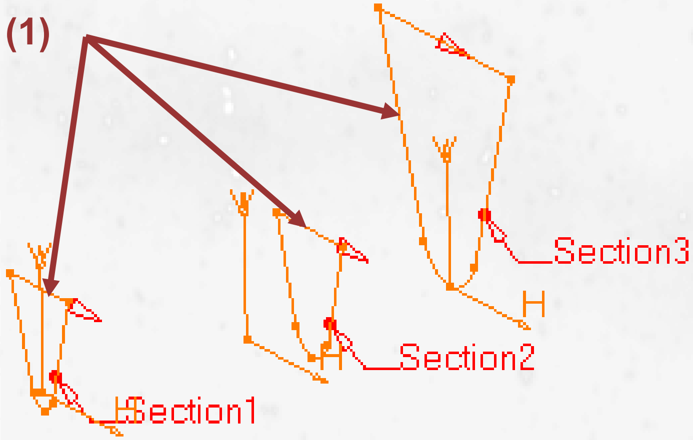
The solid is passing through the sections and each section is split at each tangency discontinuity point. The solid is computed between each split section

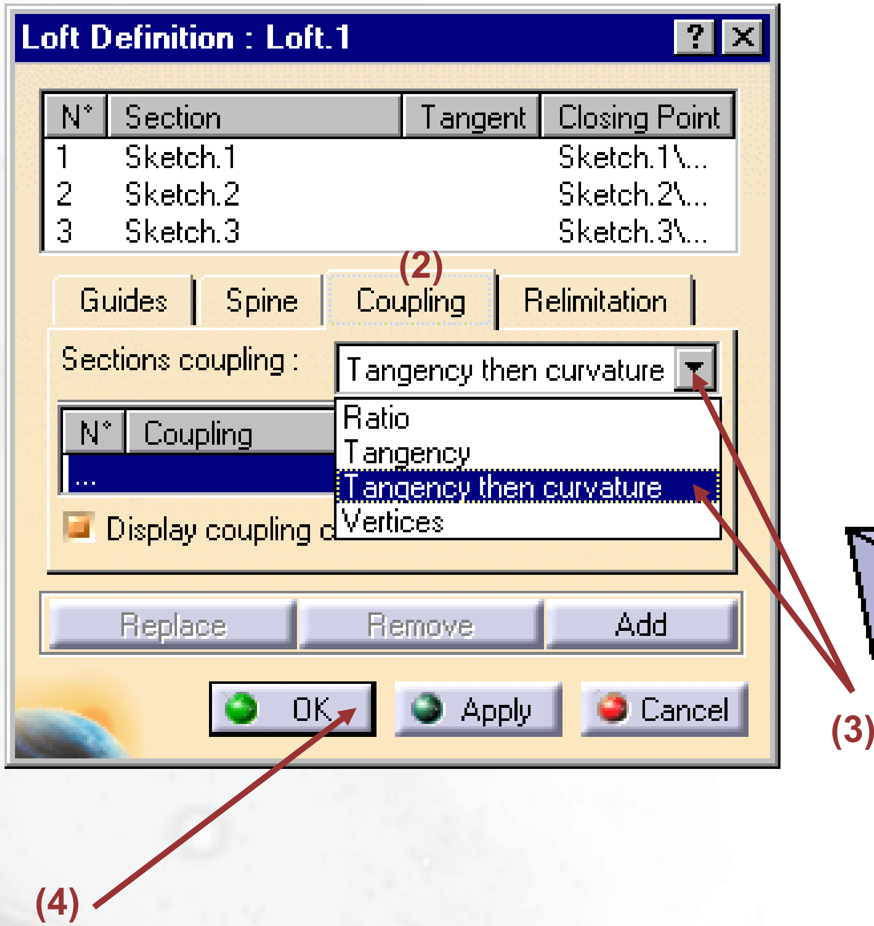
Coupling when Creating Loft : Tangency then Curvature

A coupling tab in the loft and remove loft functions to compute the loft between the curvature discontinuity points of the sections

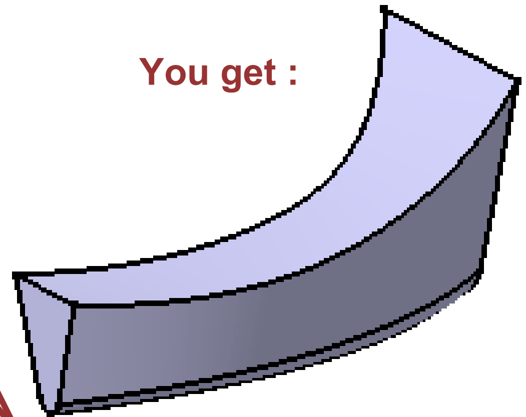
1 Activate the Loft icon and select and orient the sections.







You get :



2 Select the Coupling tab from the dialog box

3 Select Curvature Discontinuities from the combo

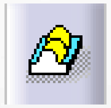
4 Select OK

The solid is passing through the sections and each section is split at each curvature discontinuity point. The solid is computed between each split section

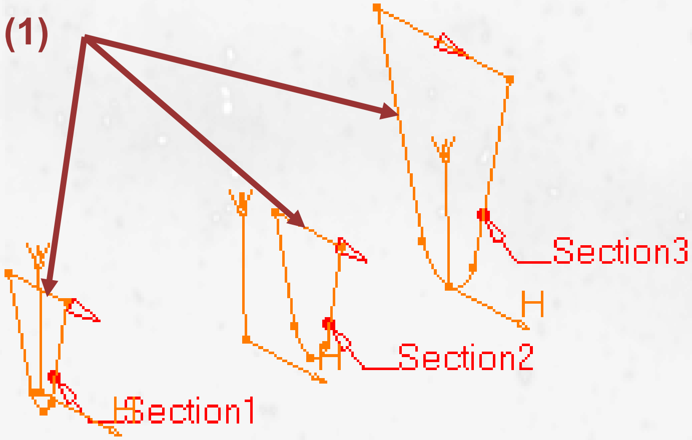
Coupling when Creating Loft : Vertices

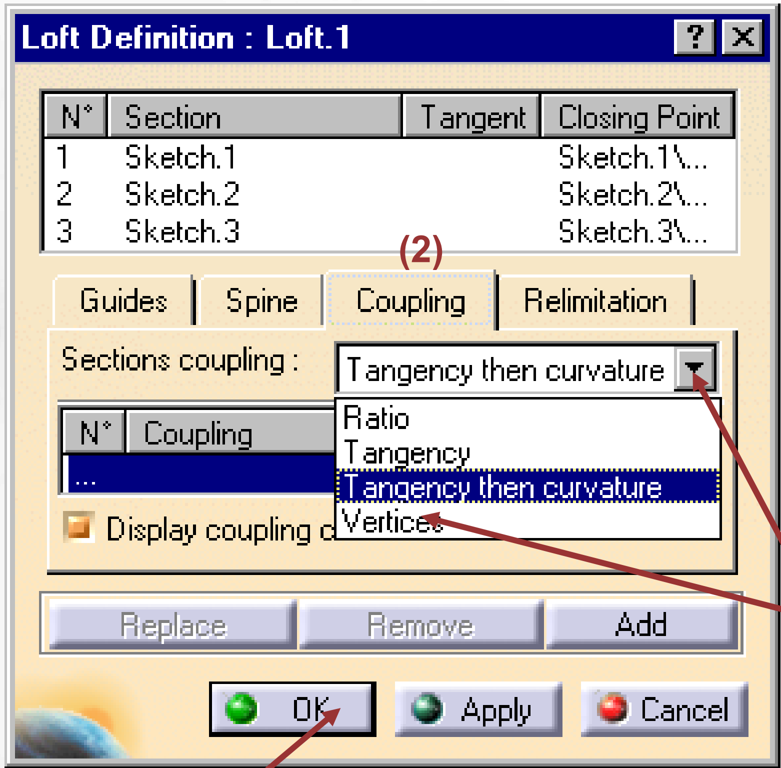
A coupling tab in the loft and remove loft functions to compute the loft between the vertices of the sections

1 Activate the Loft icon and select and orient the sections.



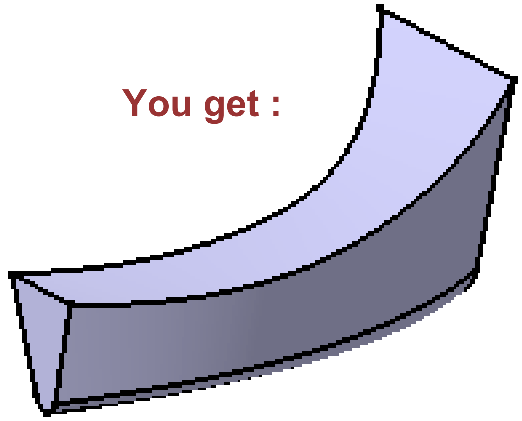
(1)





(3)

You get :



2 Select the Coupling tab from the dialog box

3 Select Vertices from the combo

(4)

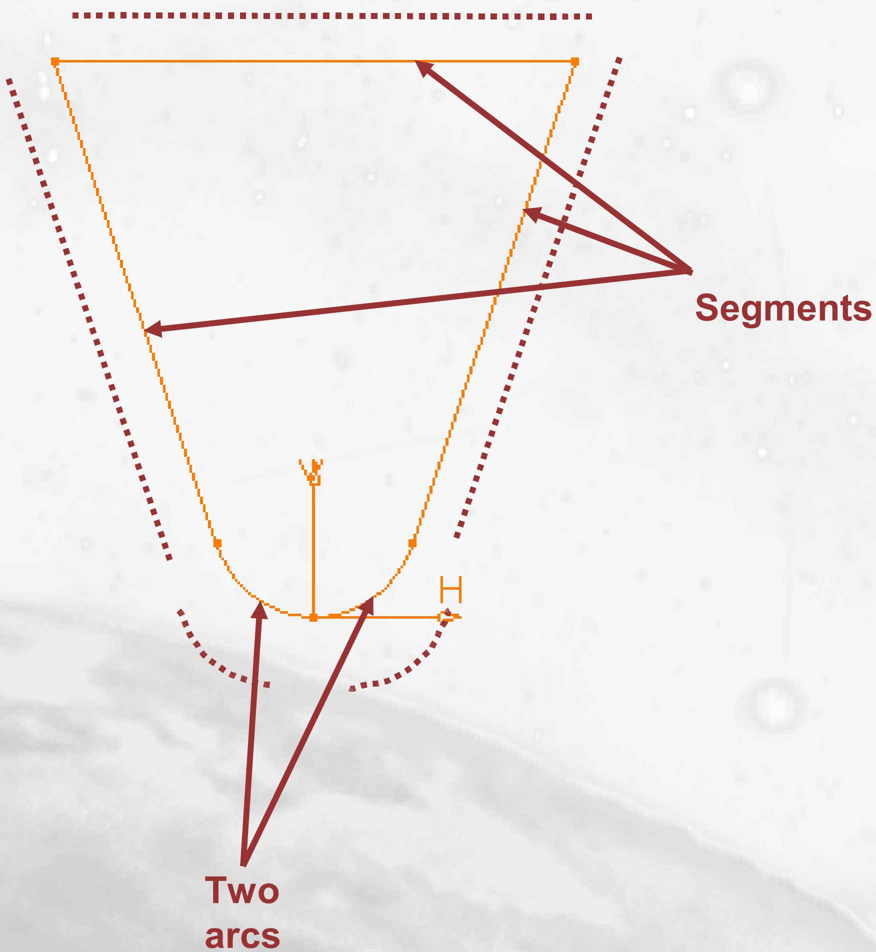
4 Select OK

The solid is passing through the sections and each section is split at each vertex. The solid is calculated between each split section

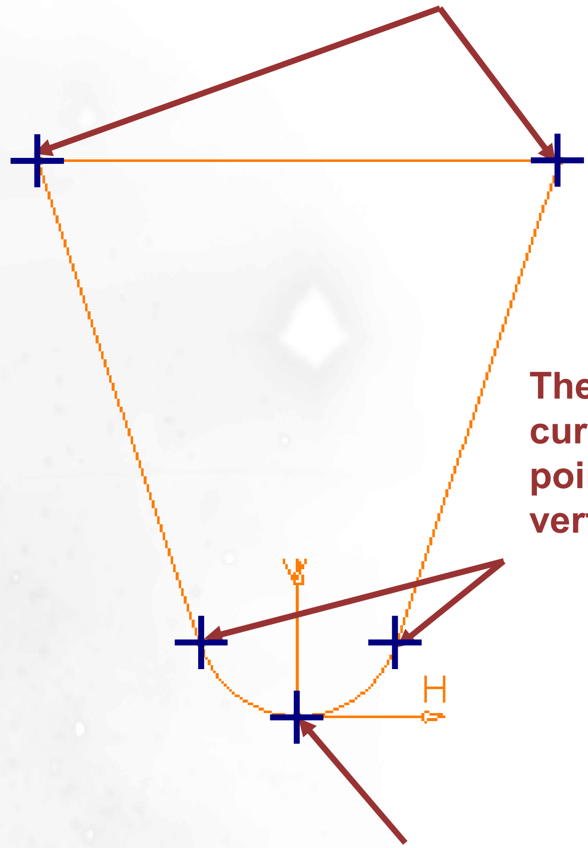
Coupling when Creating Loft : Points of Discontinuity

These are the different kinds of points that CATIA can use to split the sections when creating lofts using coupling

To have a look at the different discontinuity, we have sketched a profile as shown below :



These two points are tangency and curvature discontinuity points. They are also vertices



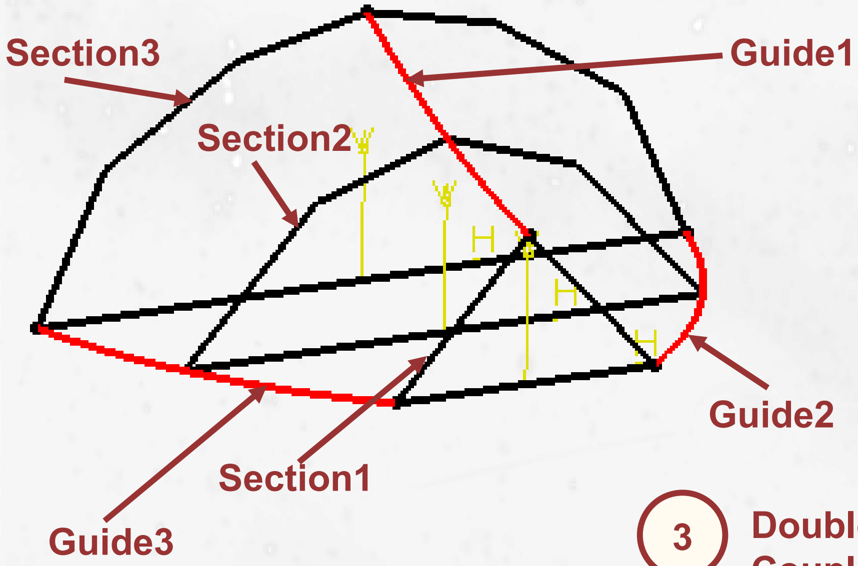
These two points are curvature discontinuity points. They are also vertices

This point is a tangency and curvature continuity point. This point is a pure vertex

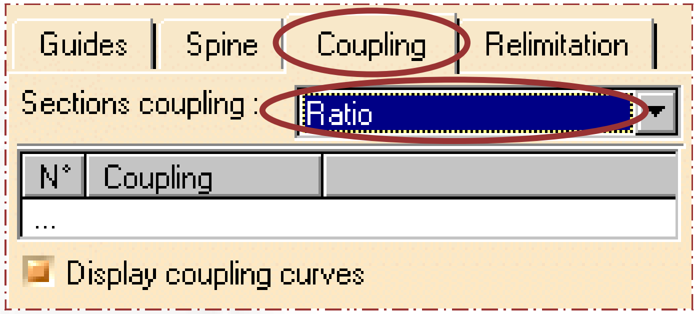
Loft: Manual Coupling (1/2)

When the sections of the lofted solid do not have the same number of vertices you may define a manual coupling instead of changing or creating closing points

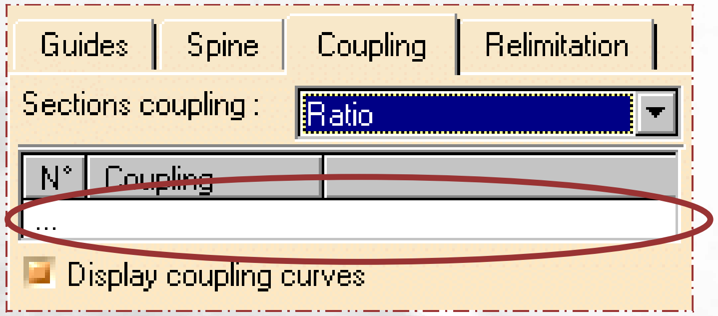
1 Activate the Loft icon select the sections and the guide curves (If necessary, change the section orientation)



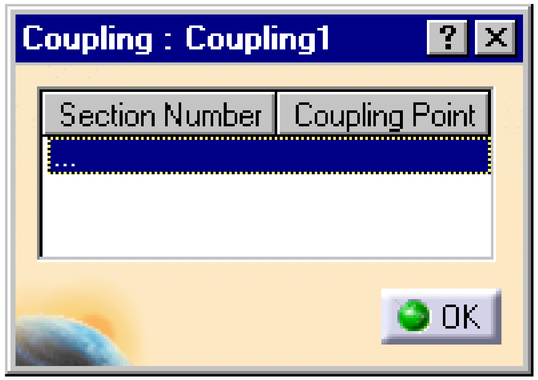
2 Select the Coupling tab then set the Sections coupling to Ratio



3 Double click in the Coupling field to display the Coupling window



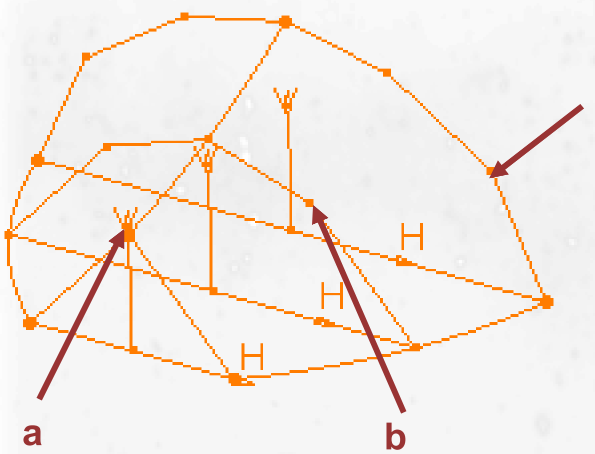
You get:

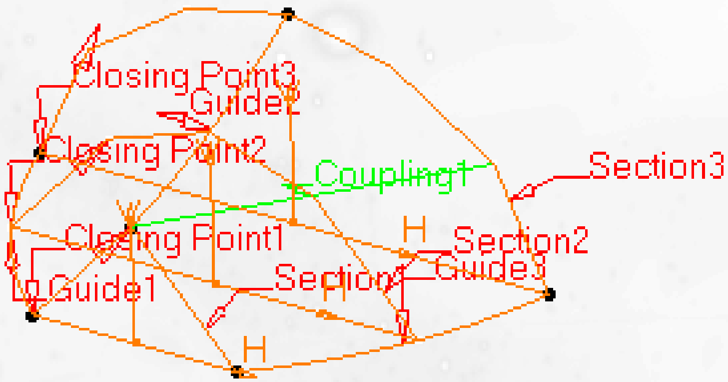


Loft: Manual Coupling (2/2)

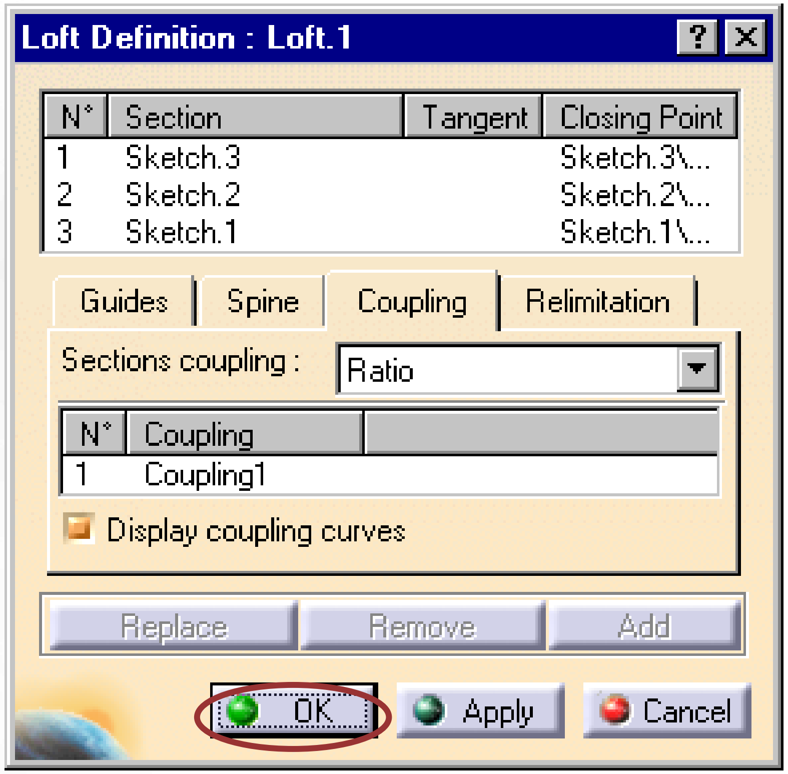
When the sections of the lofted solid do not have the same number of vertices you may define a manual coupling instead of changing or creating closing points

4 For each section select the vertex to be taken into account in the coupling. You can visualize the coupling curve if the corresponding option is checked. The Vertices selection must be done in the same order than the sections selection

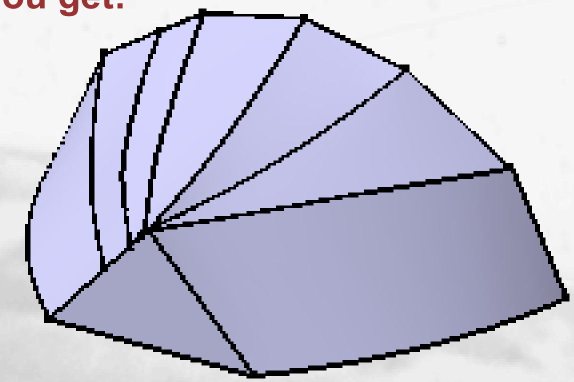




5 Click OK to end lofted surface definition



You get:

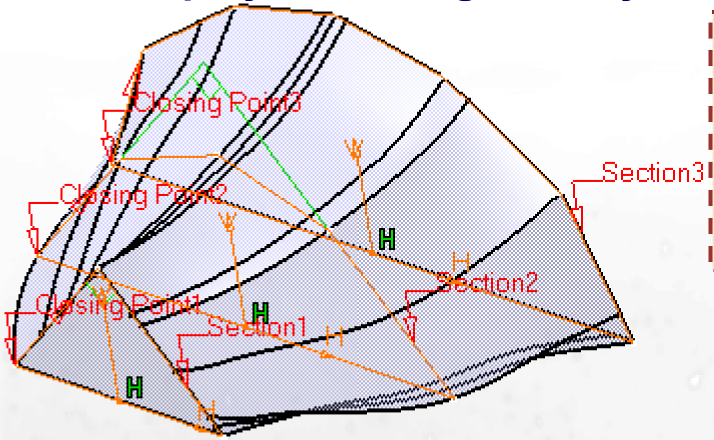


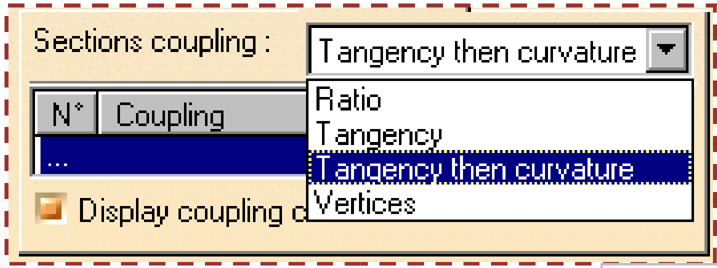
Note: This is also possible with the Remove Loft command

Note:To refine the shape of the lofted surface you can define another coupling curve : select the first coupling and click on the Add button, then define the new coupling curve as explained above.

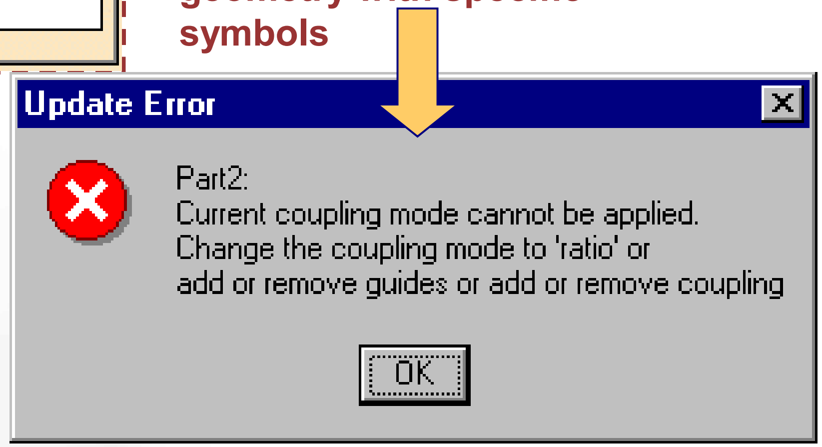
Manual Coupling: Displaying Uncoupled Points

For each coupling mode, the points that could not be coupled are displayed in the geometry with specific symbols

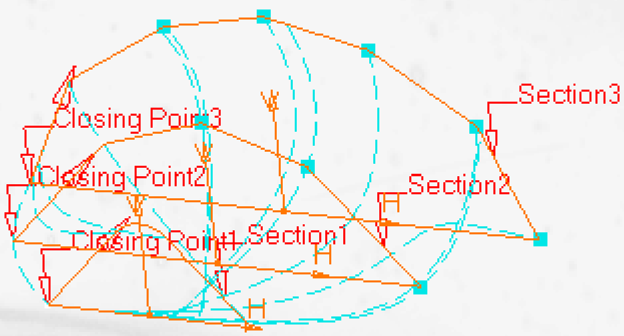




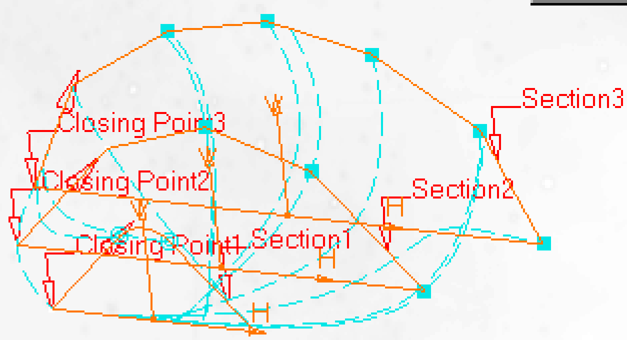
For each coupling mode, the points that could not be coupled are displayed in the geometry with specific symbols



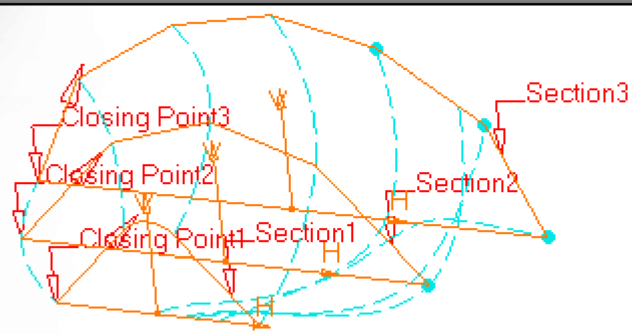
On this loft, the sections have not the same number of vertices and have some discontinuity in curvature and tangency



Tangency mode : uncoupled tangency discontinuity points are represented by a square



Tangency the Curvature mode : uncoupled tangency discontinuity points are represented by a square. Uncoupled curvature discontinuity points are represented by an empty circle

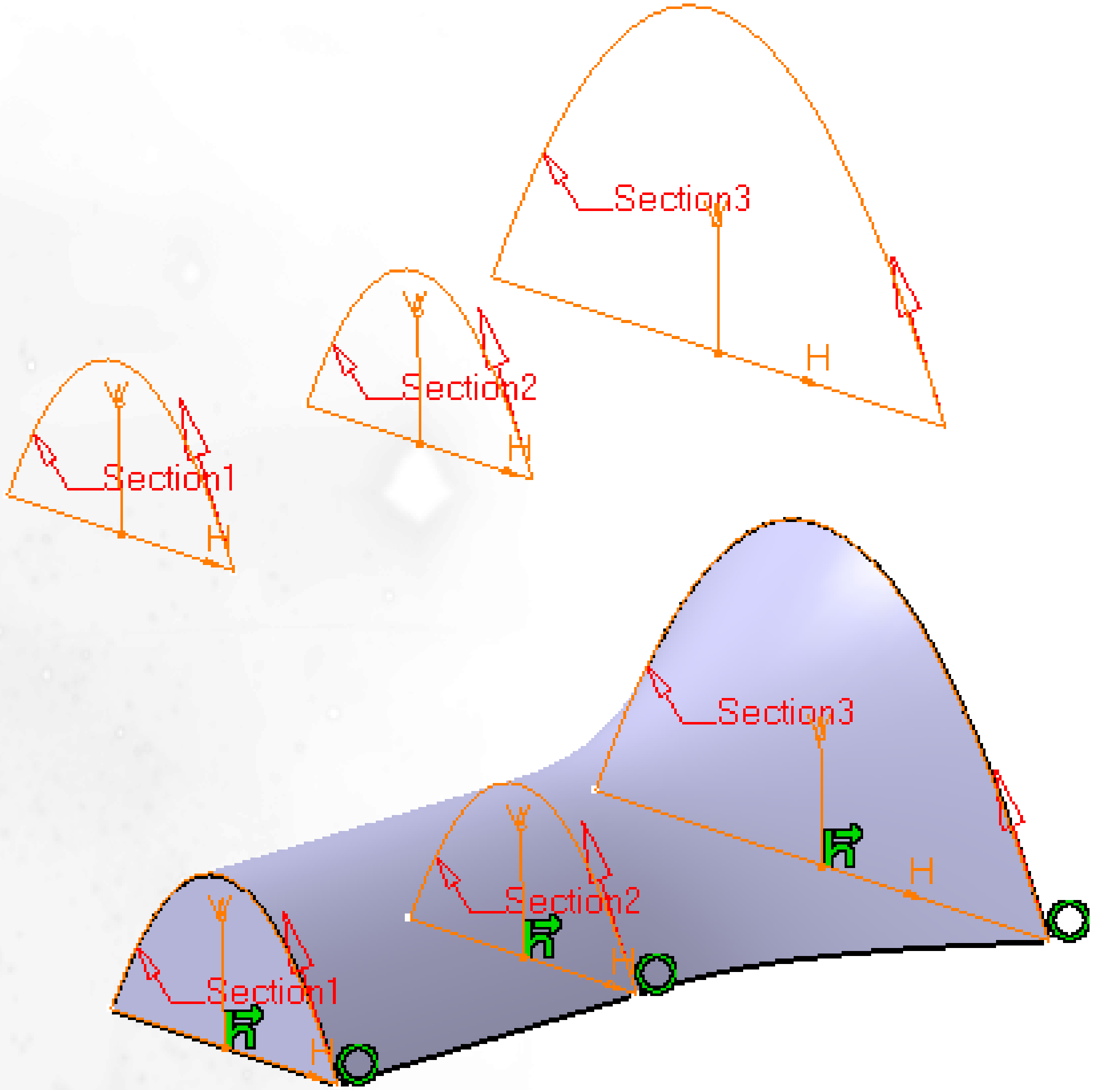


Vertices mode : uncoupled vertices are represented by a full circle

Loft: Relimitation (1/3)

By default the lofted surface is limited by the start and end sections. However you can choose to limit it on the spine or on the guide lines extremities

When the limitation option is checked, the loft is limited to the start or (and) end sections even is a larger spine or guide curves have been used

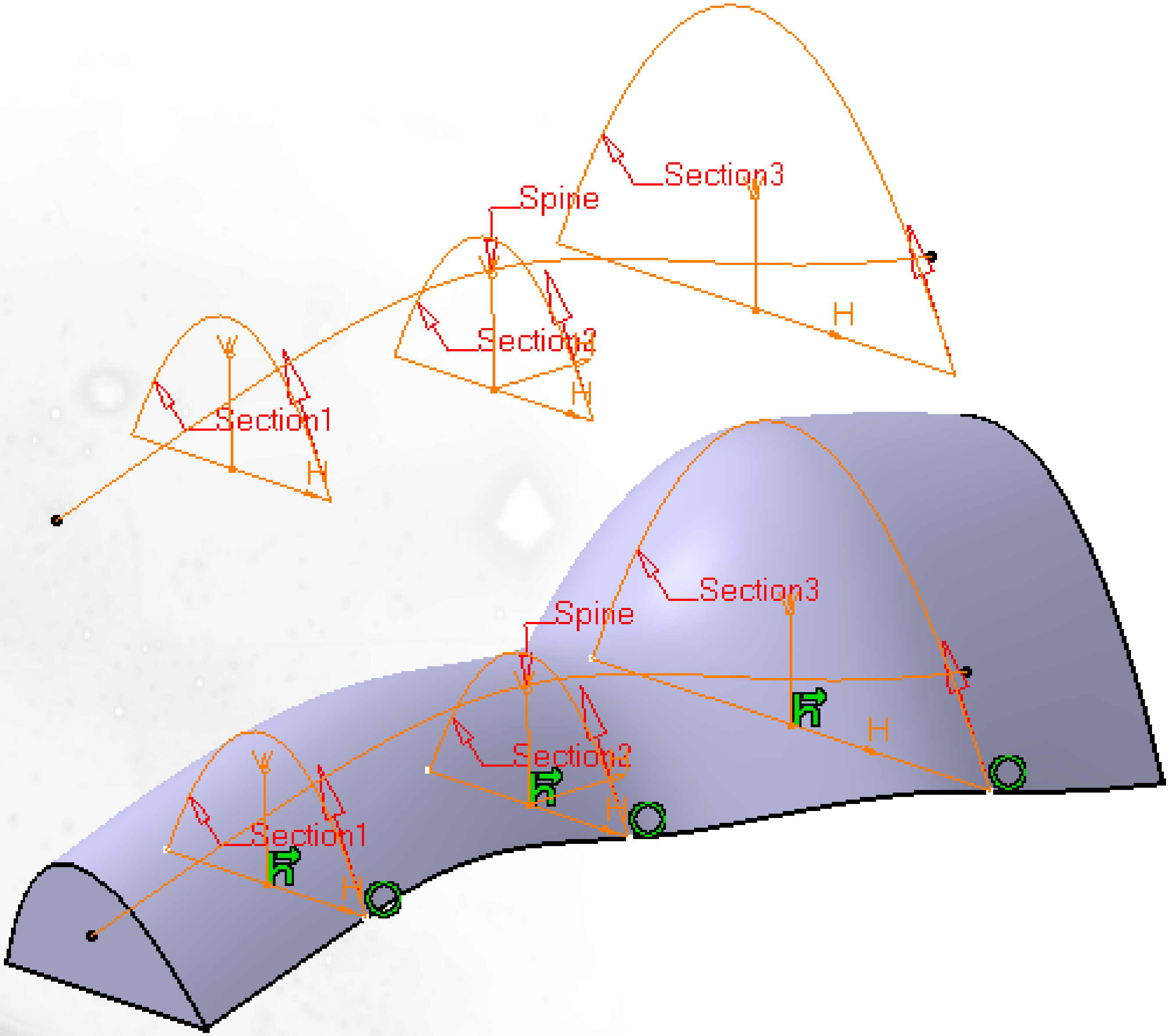


Note: **This is also possible with the Remove Loft command**

Loft: Relimitation (2/3)

By default the lofted surface is limited by the start and end sections. However you can choose to limit it on the spine or on the guide lines extremities

When the limitation option is unchecked, and when a spine has been used, the loft is limited by the spine extremities

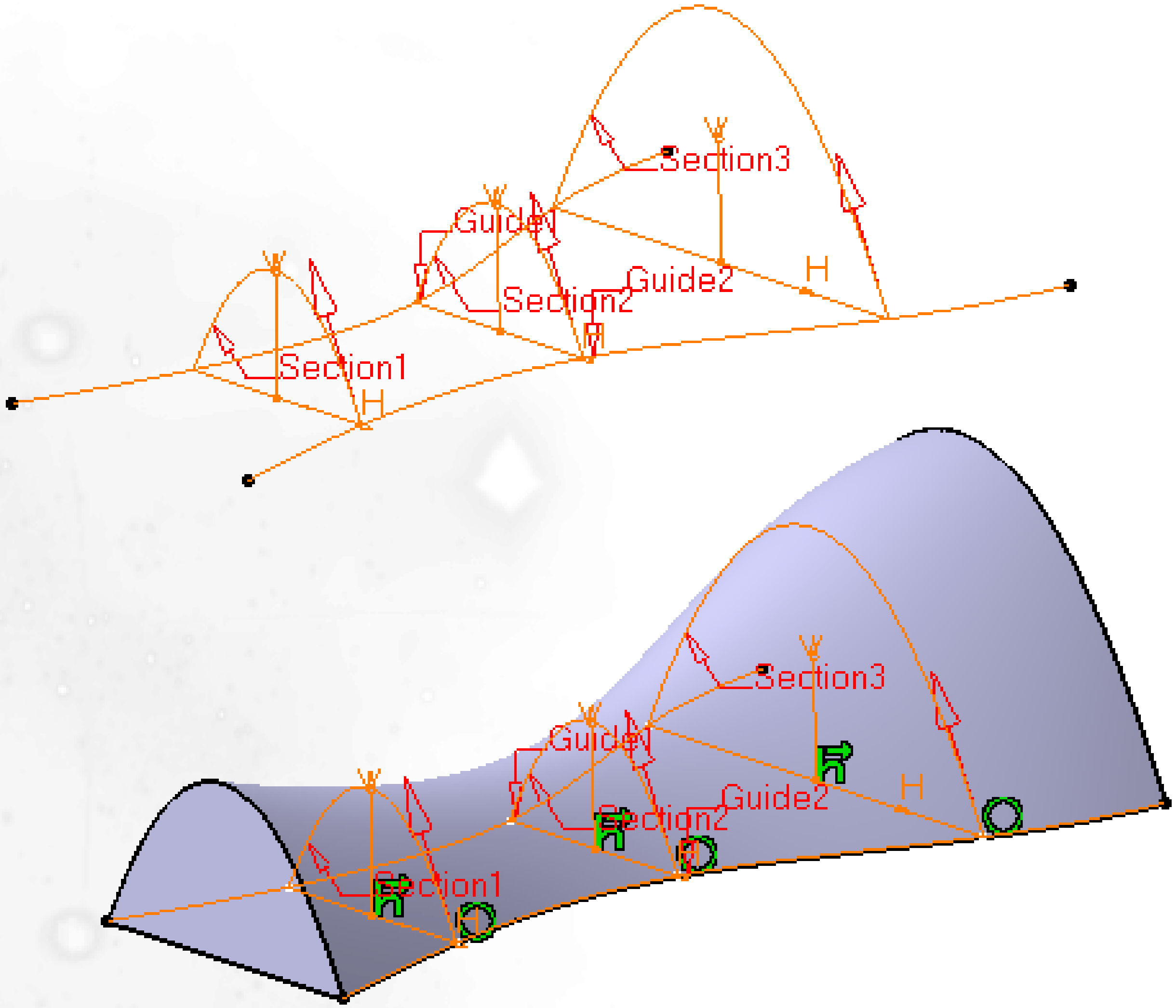


Note: **This is also possible with the Remove Loft command**

Loft: Relimitation (3/3)

By default the lofted surface is limited by the start and end sections. However you can choose to limit it on the spine or on the guide lines extremities

When the limitation option is unchecked, and when guide lines have been used, the loft is limited by the guide lines extremities



Note: **This is also possible with the Remove Loft command**

Note: **If a spine an guide lines have been used the loft will be limited on the shorter line**

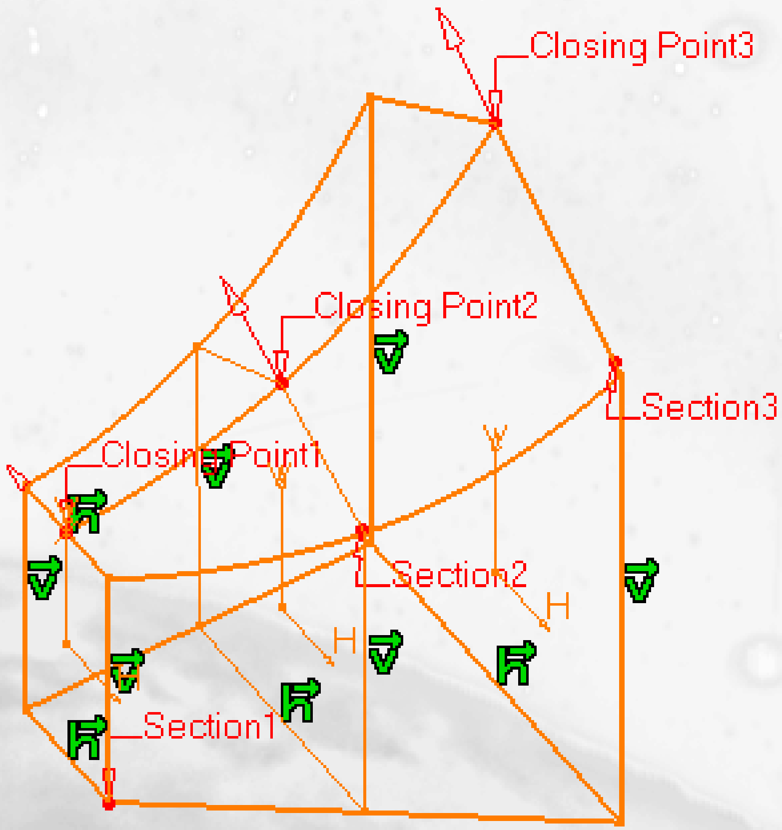
Changing the Closing Point

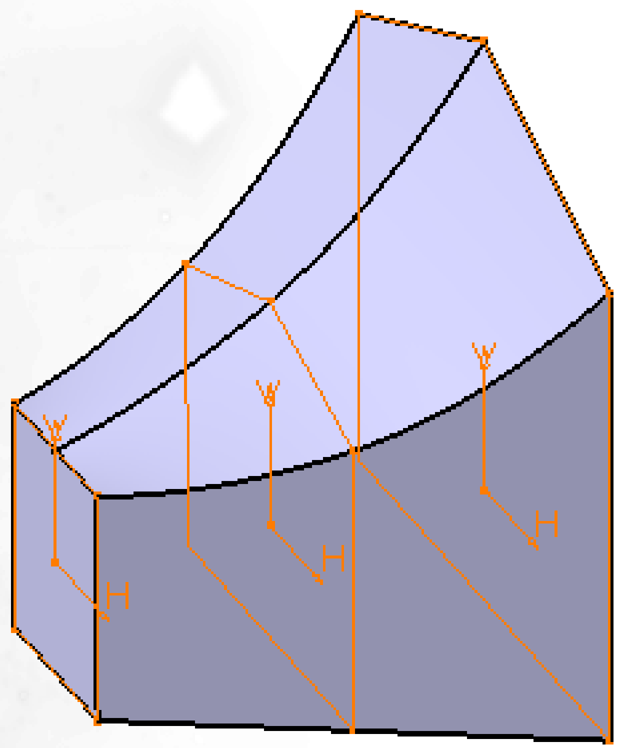
In this lesson we will learn how to change the closing point when creating a Loft

- Changing the Closing Point

What is Changing the Closing Point when Creating Loft ?

When selecting the sections to create a loft (or remove loft), you can change the closing point after the selection of the sections and you can create a closing point anywhere on a section profile



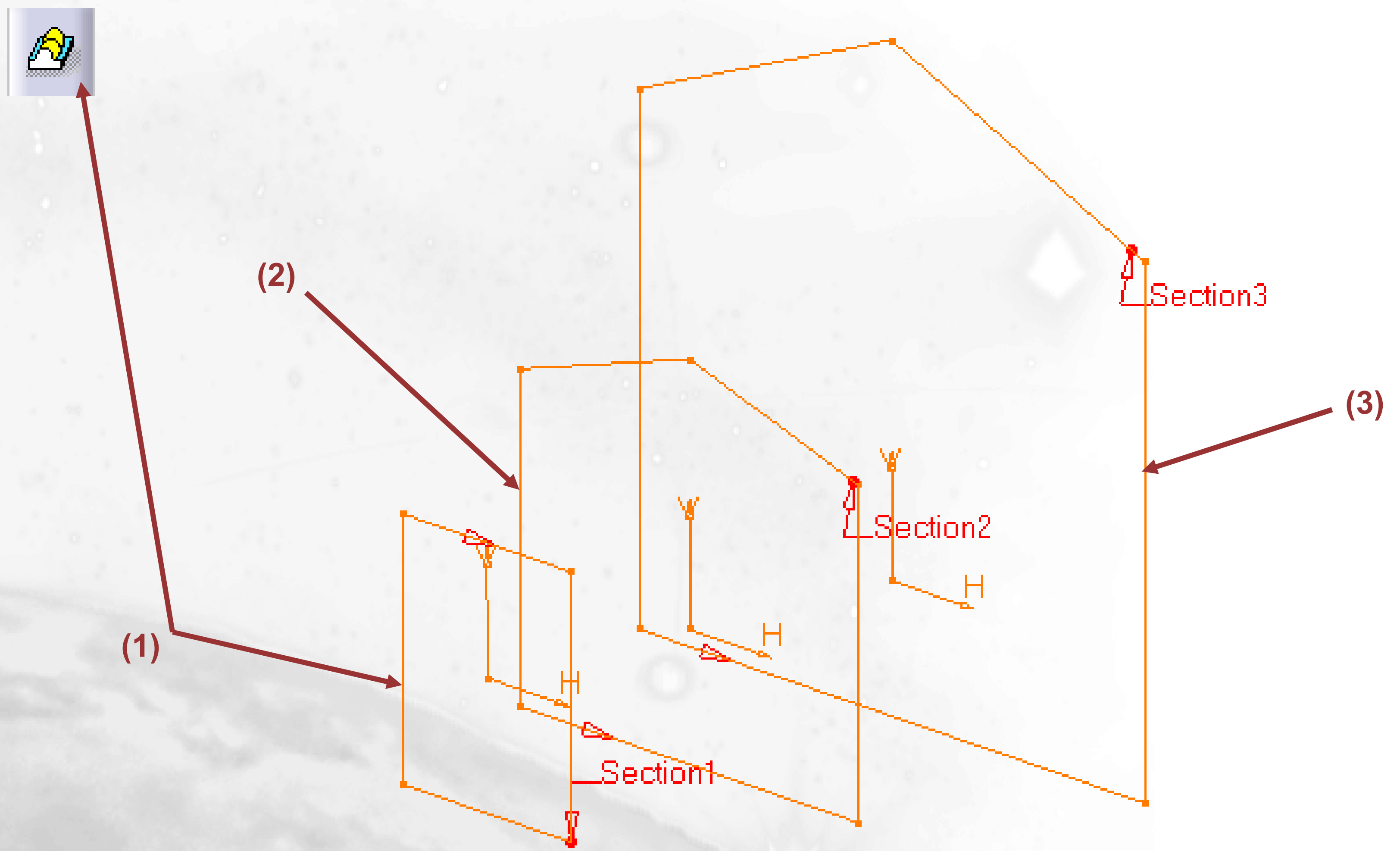


Changing the Closing Point when Creating Loft (1/6)

1 Activate the loft icon and select the first section

2 Select the second section

3 Select the third section



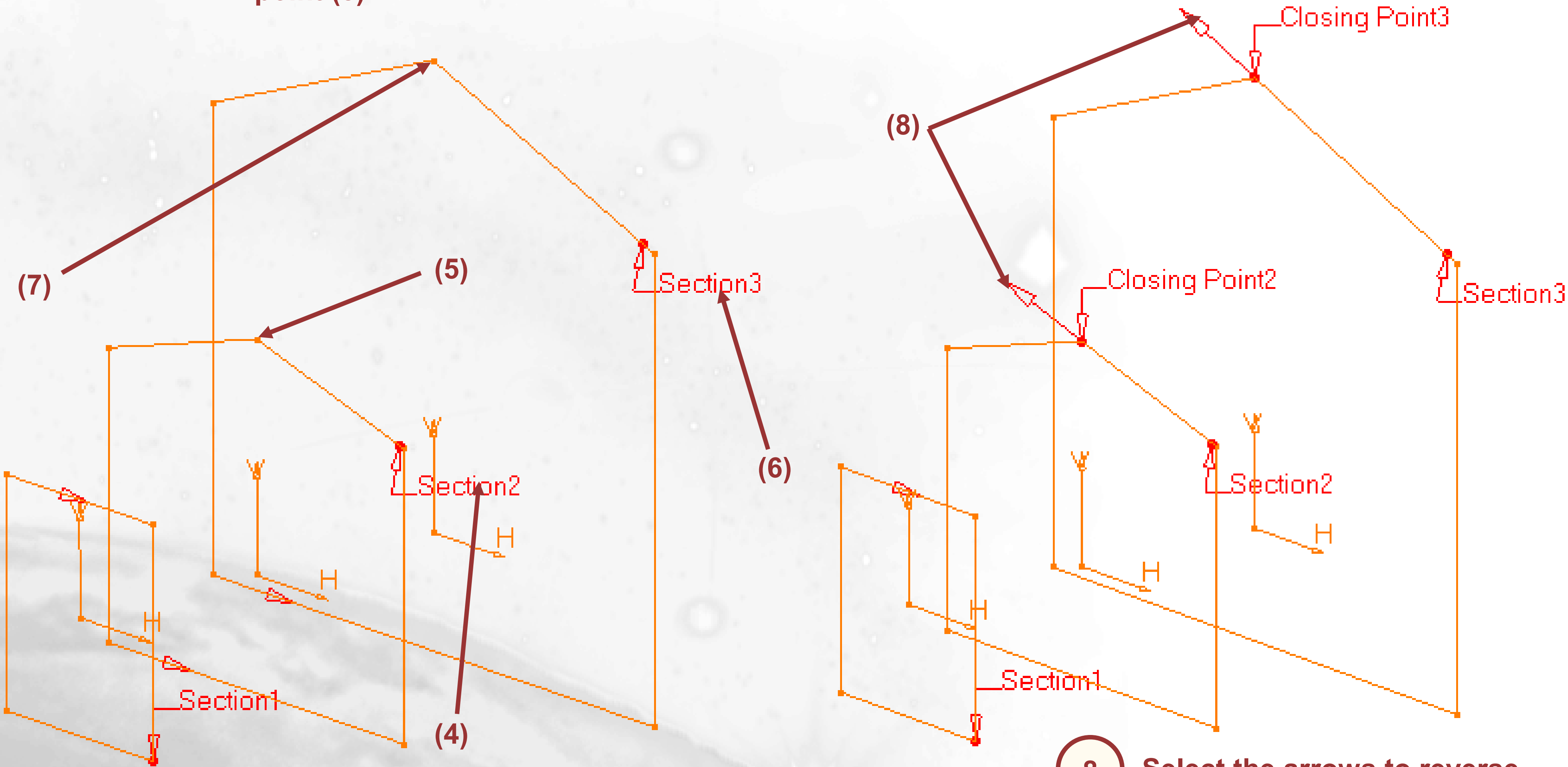
Changing the Closing Point when Creating Loft (2/6)

4 Click on Section2 (Label)

5 Select Replace Closing Point from the contextual menu, then select a new closing point (5)

6 Click on Section3 (Label)

7 Select Replace Closing Point from the contextual menu, then select a new closing point (7)

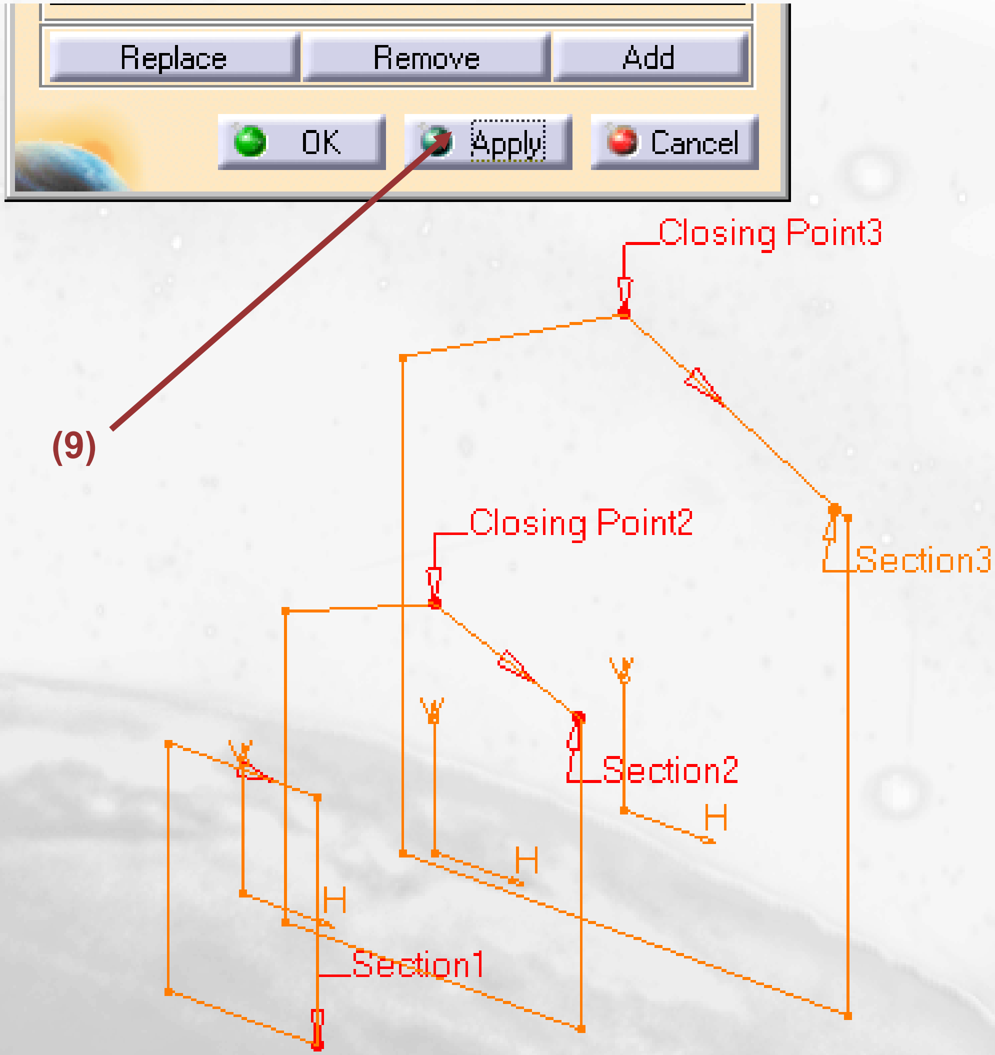


8 Select the arrows to reverse Section2 and Section3

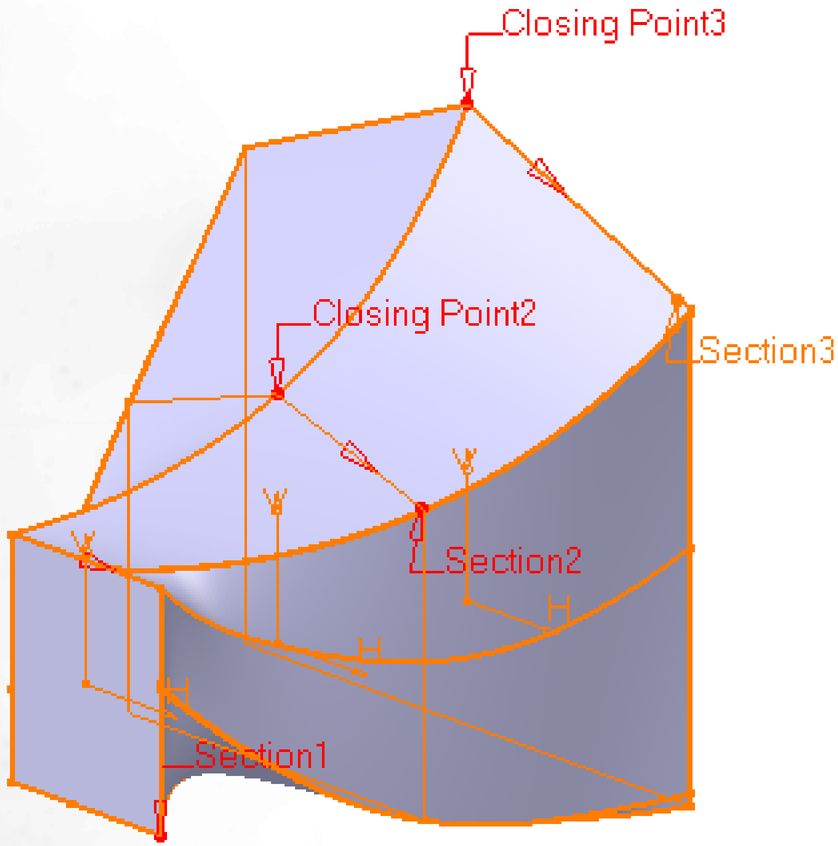
Changing the Closing Point when Creating Loft (3/6)

9 Check that the coupling is at Ratio then Select Apply in the dialog box

You can see that the solid is twisted because the default closing point of Section1 is not aligned with the closing points of the other sections



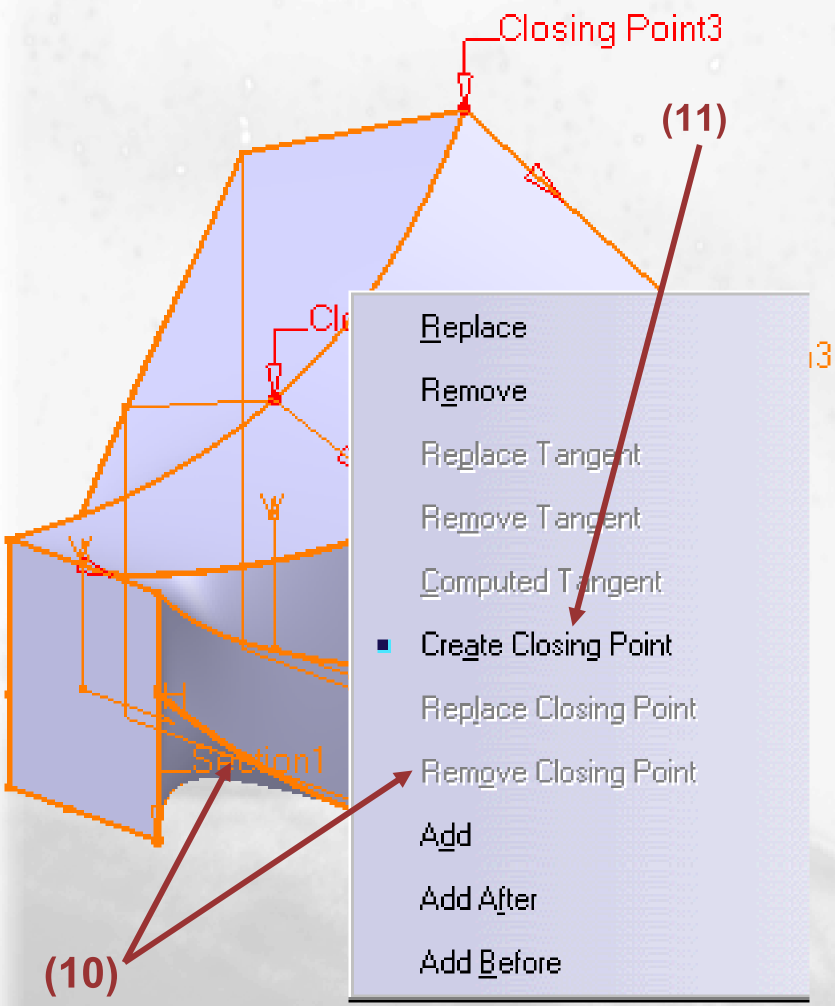
(9)



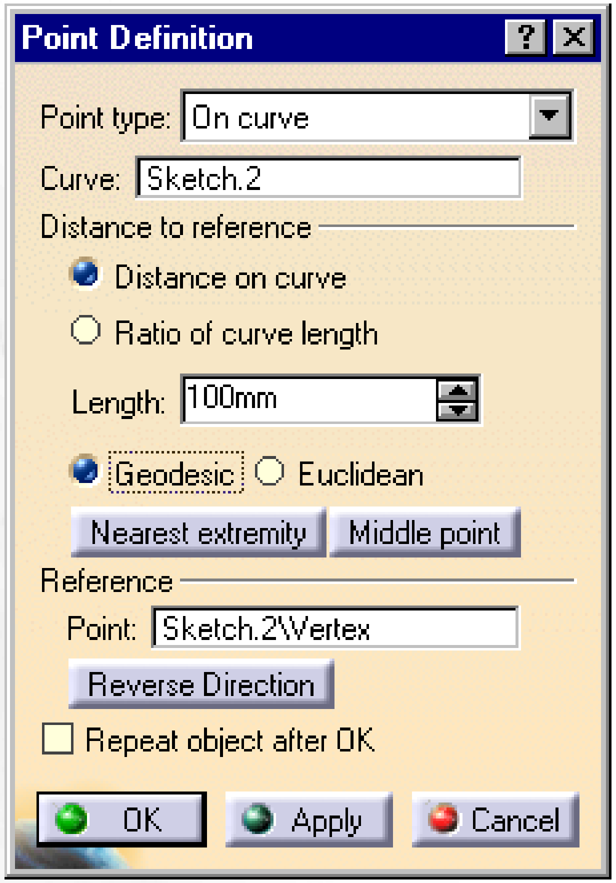
Changing the Closing Point when Creating Loft (4/6)

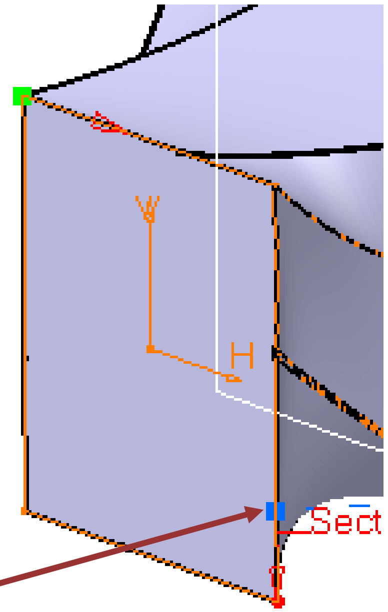
10 In order to create a closing point on Section1, select the Section1 label with MB3, then select Remove Closing Point

11 Then again, select Create Closing Point from the contextual menu



A new dialog box corresponding to a point creation on a curve appears





The point appears in blue before validation

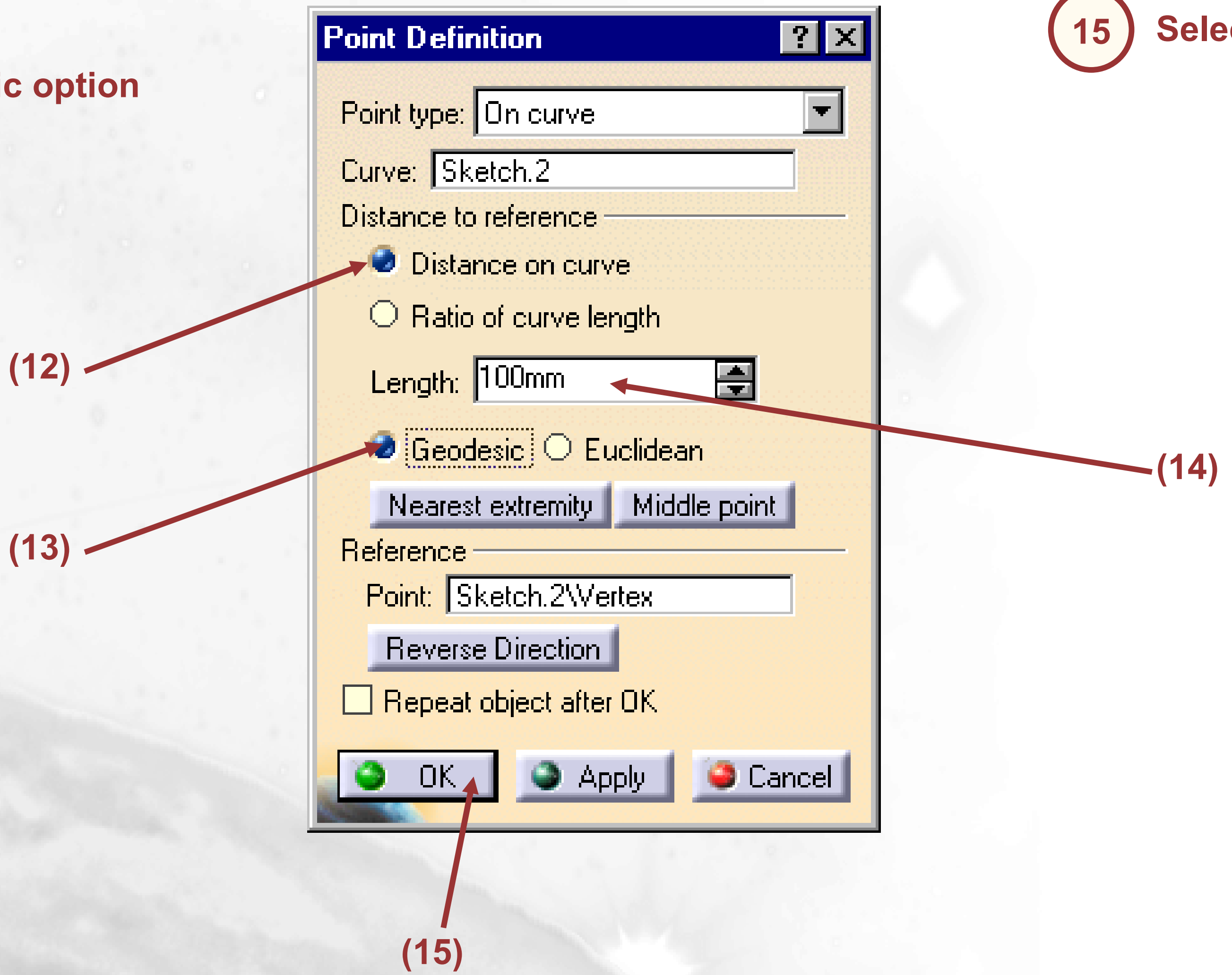
Changing the Closing Point when Creating Loft (5/6)

12 Select the Distance on curve option

13 Select the Geodesic option

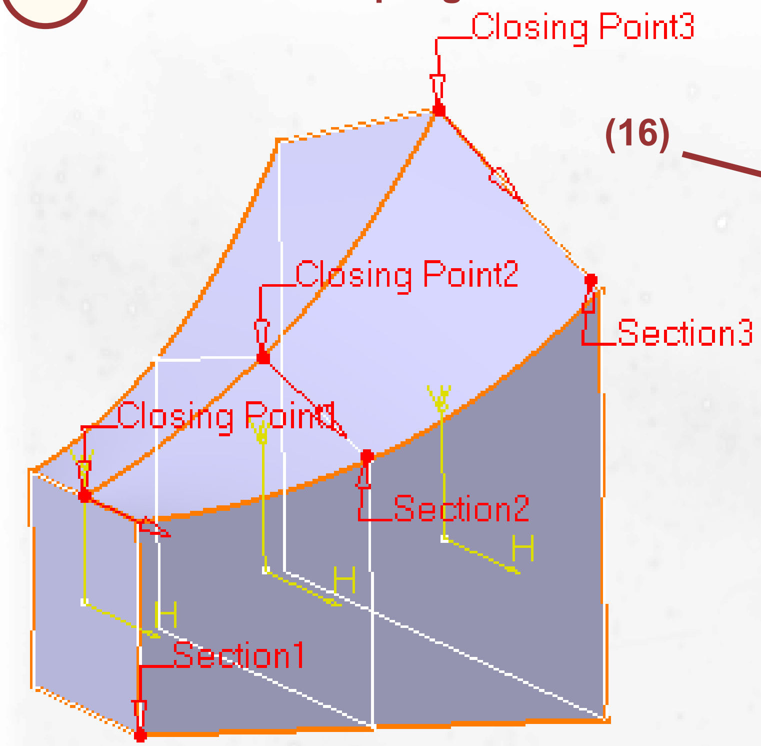
14 Enter 100 as the Length

15 Select OK

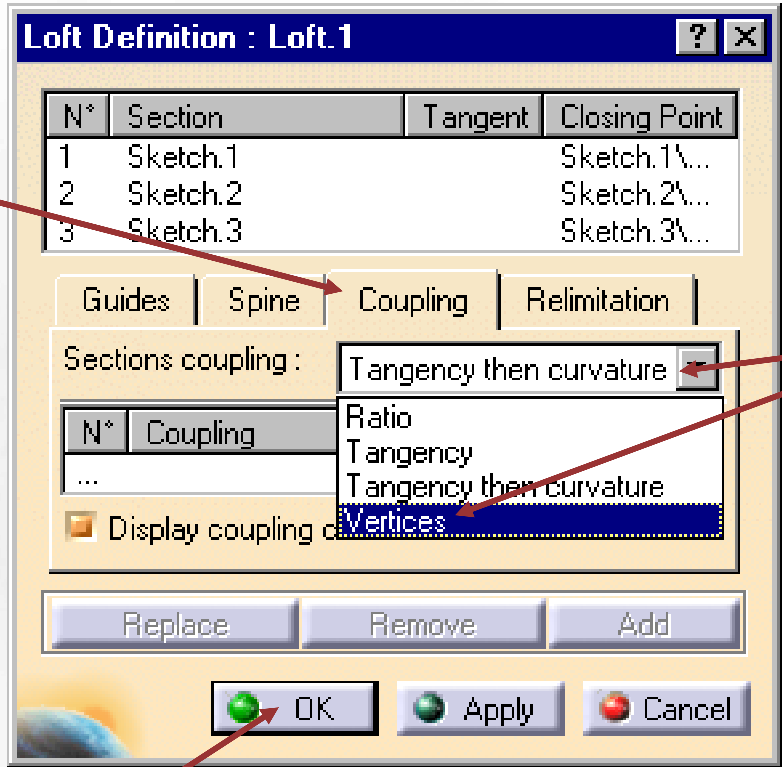


Changing the Closing Point when Creating Loft (6/6)

16 Select the Coupling tab

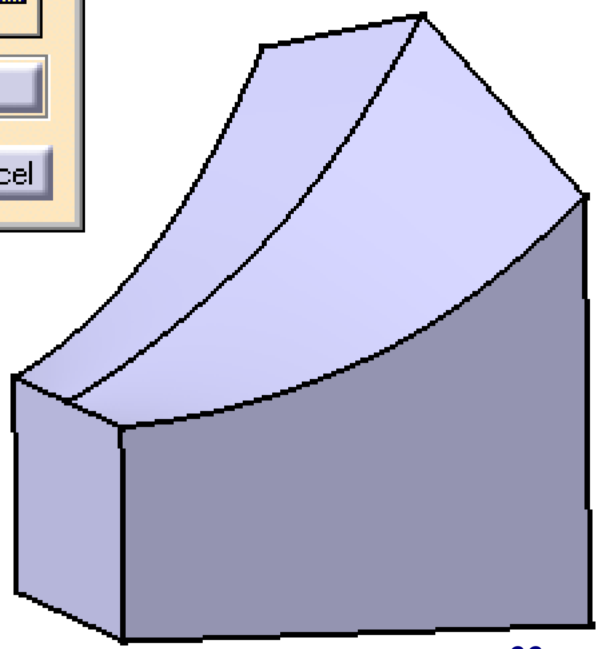


(16)



(17)

You get :



17 Select Vertices option from the combo

(18)

18 Select OK

3D Wireframe Elements

You will learn more about 3D Wireframe Elements and how we use them to help construct our Part

- 3D Wireframe Elements
- Use of Wireframe Elements in Part Design

What are 3D Wireframe Elements ?

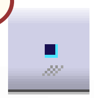
In the Part Design Workbench, we can create points, lines and planes without using the Sketcher.
These elements belong to the "*Reference Element*" toolbar.

In the Specification Tree, they are inserted under "Open_Body" which contain all 3D Wireframe elements.

Even if these elements are some Wireframe Elements, we can use them with the Part Design Tools.

Creating 3D Wireframe Point

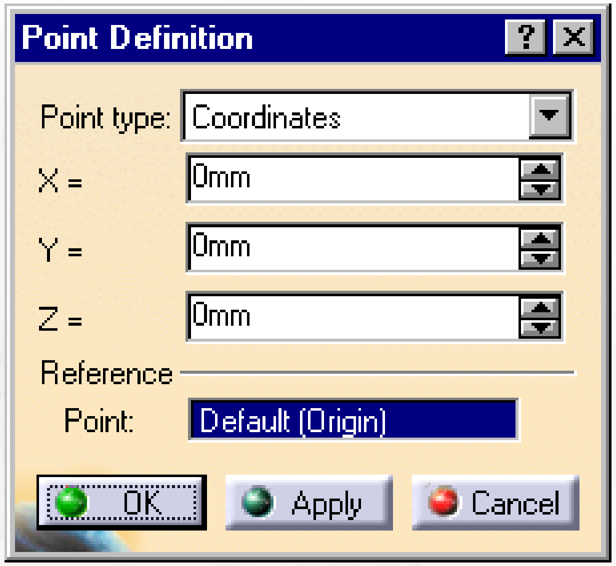
1



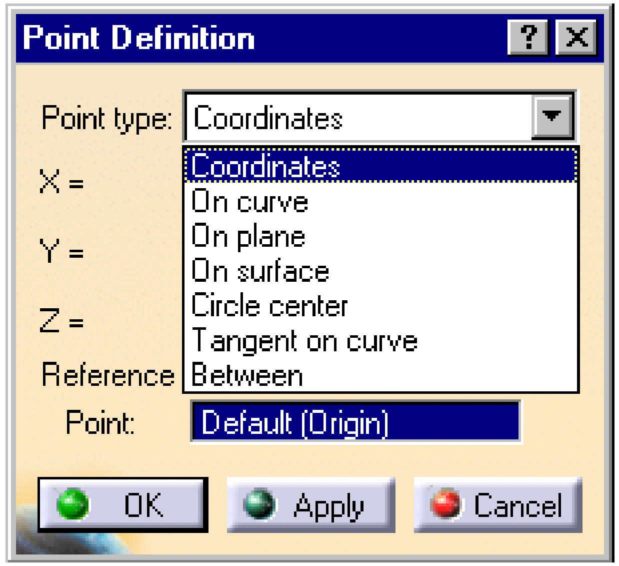
In Reference Toolbar, select Point by clicking on icon

2

A dialog Box is displayed



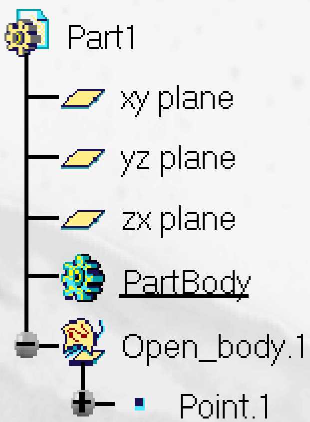
Notice that we can choose between several types of points



We create the desired point

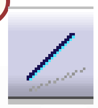
3

The created point appears under Open_Body



Creating 3D Wireframe Line

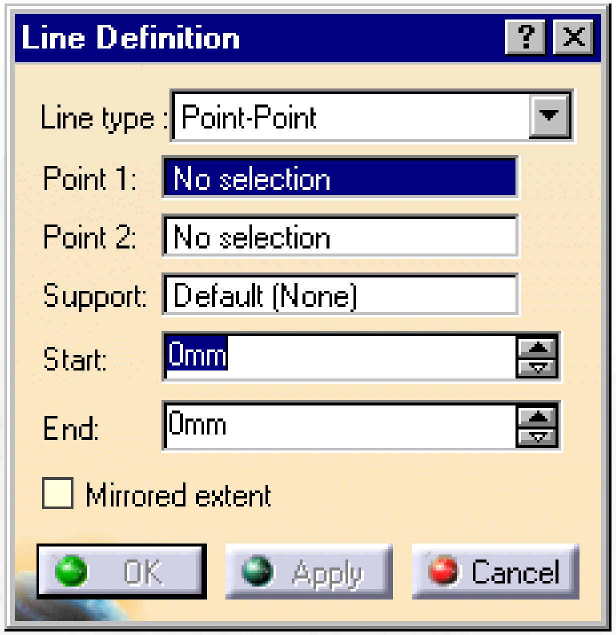
1



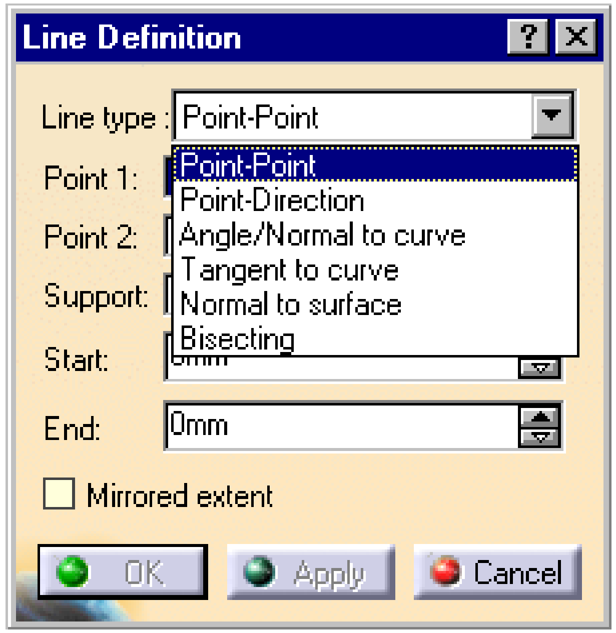
In Reference Toolbar, select Line by clicking on icon

2

A dialog Box is displayed



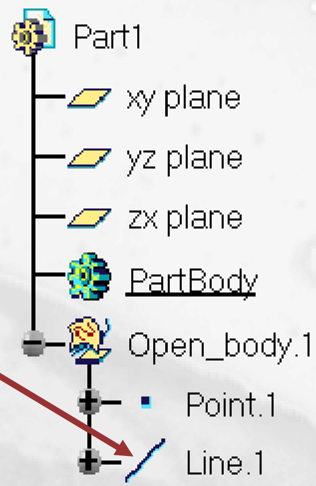
Notice that we can choose between several types of lines



We create the desired line

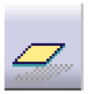
3

The created line appears under Open_Body



Creating 3D Wireframe Plane

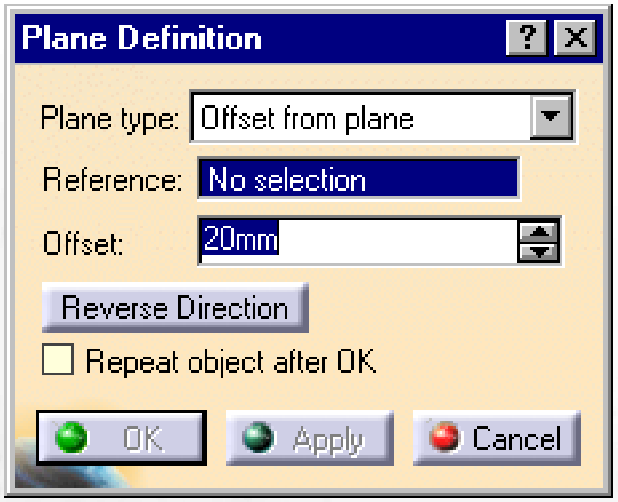
1



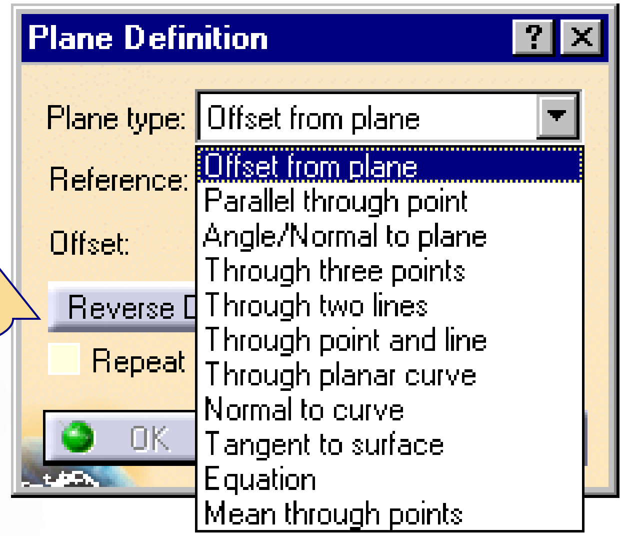
In Reference Toolbar, select Plane by clicking on icon

2

A dialog Box is displayed



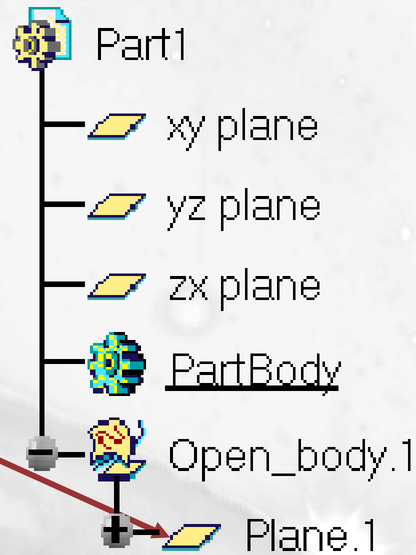
Notice that we can choose between several types of planes



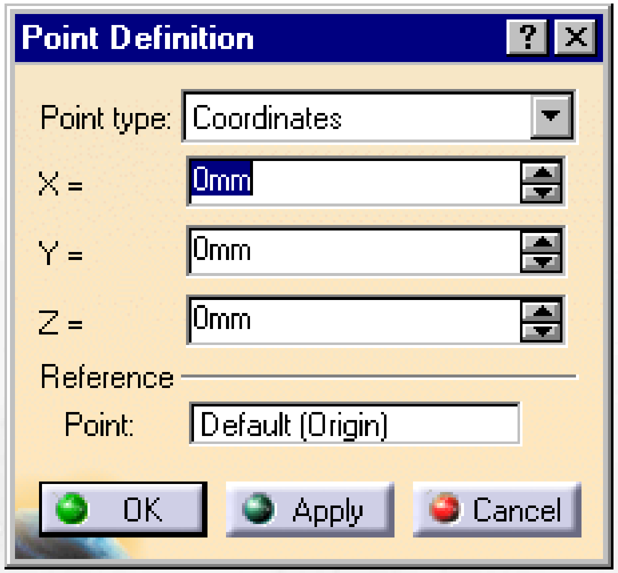
We create the desired plane

3

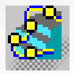
The created plane appears under Open_Body



Using 3D Wireframe Elements to Create a 3D Curve



1 You can create points in space according to their coordinates by using the Points tool from the Reference Element tool bar

2 

Create the 3D curve by using the Curve in Space tool from the Free-Style Workbench

This curve can now be used to extrude a rib or create a slot

Surface Based Features

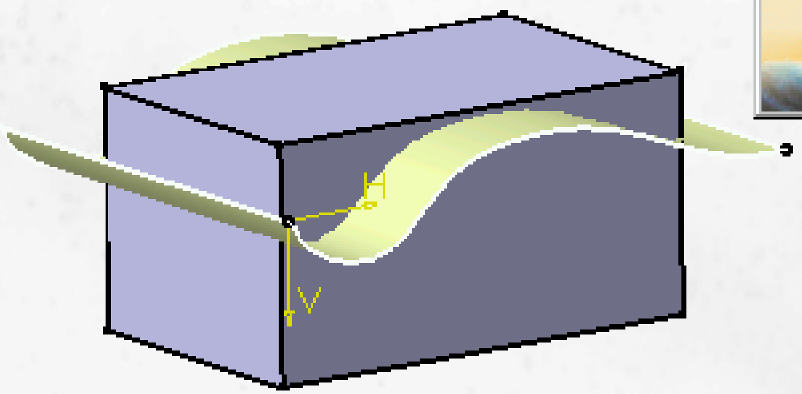
We will learn how to use all of the various types of Surfaced Based Features Split, Thick Surface, Close Surface and Sew Surfaces

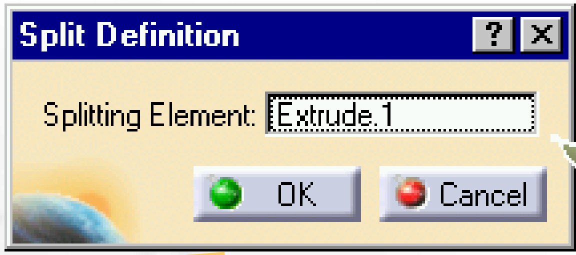
- **Split**
- **Thick Surface**
- **Close Surface**
- **Sew Surface**

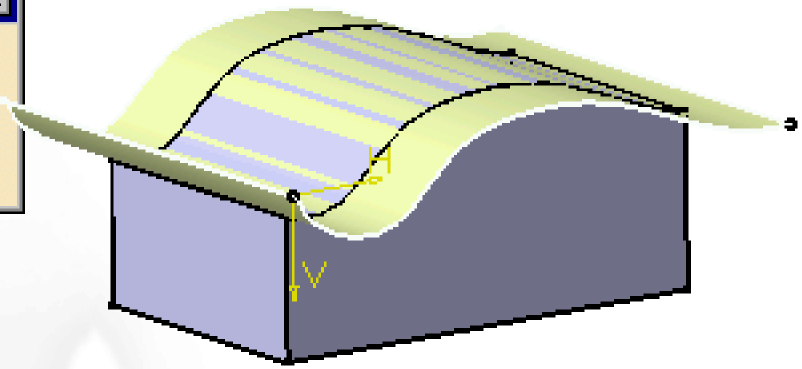
What is a Surface Based Feature and when Do You Use It (1/2) ?

There are four Surface Based Features

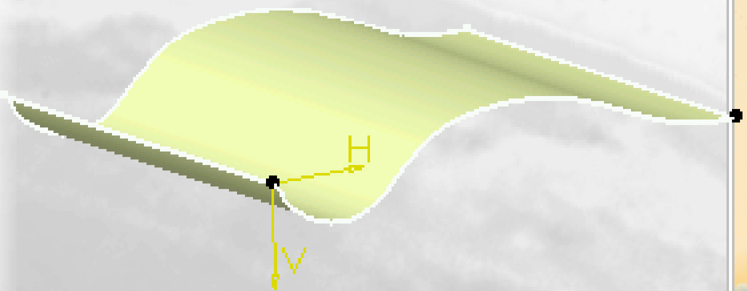
Split: Used to split a solid with either a plane or a surface.

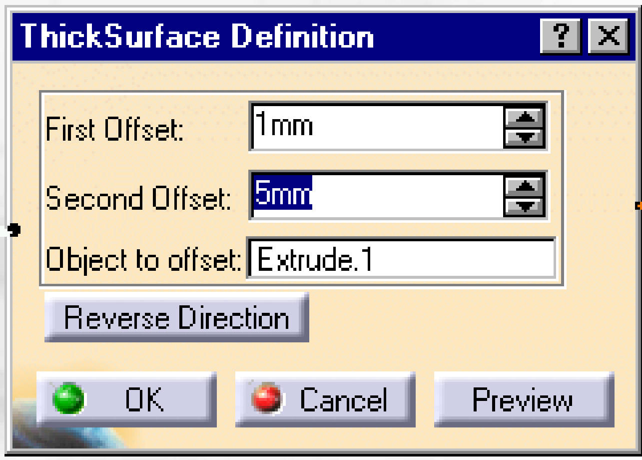


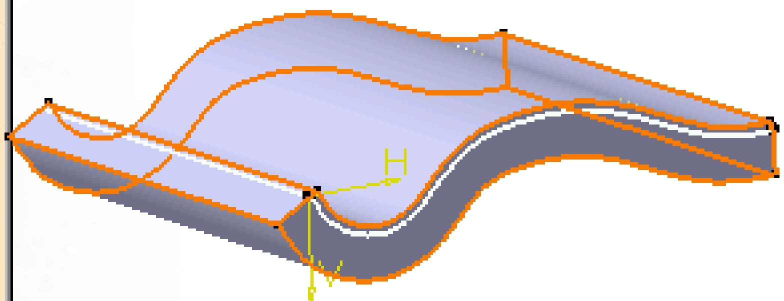




Thick Surface: Used to create solids from surfaces. Material can be added from either or both sides of the surface



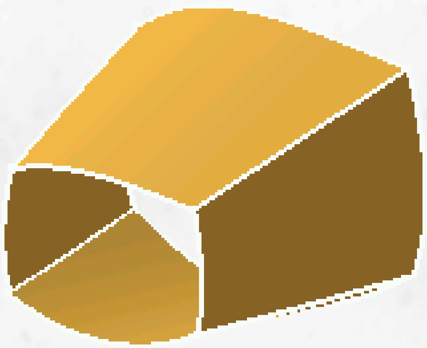


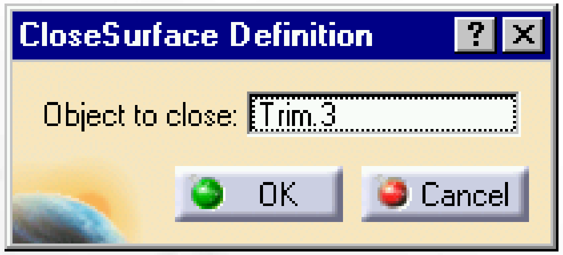


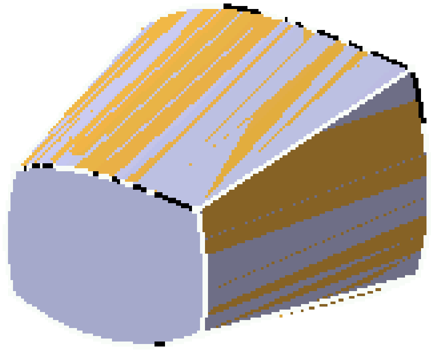
What is a Surface Based Feature and when Do You Use It (2/2) ?

There are four Surface Based Features

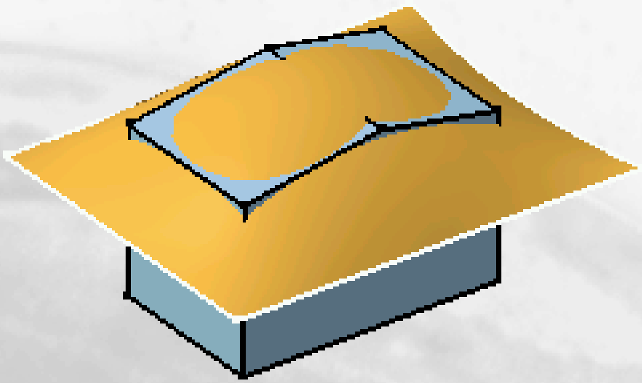
Close Surface: Used to take a closed surface and turn it into a solid.

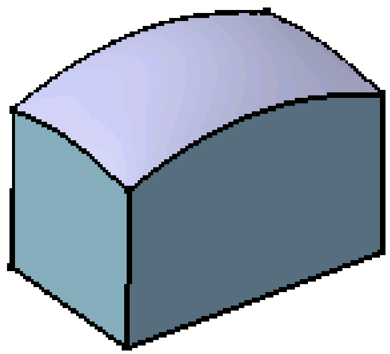






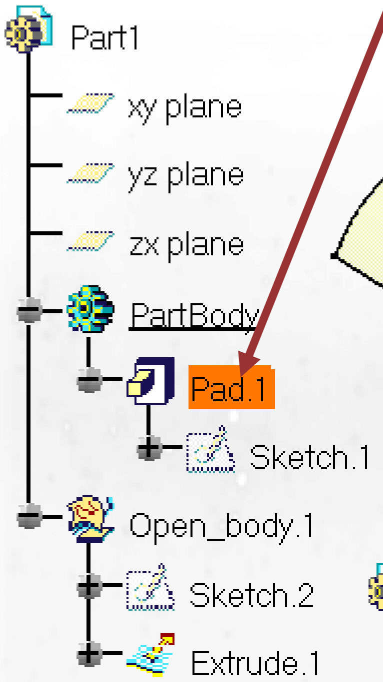
Sew Surface: Used to glue a surface feature to an existing 3D solid.

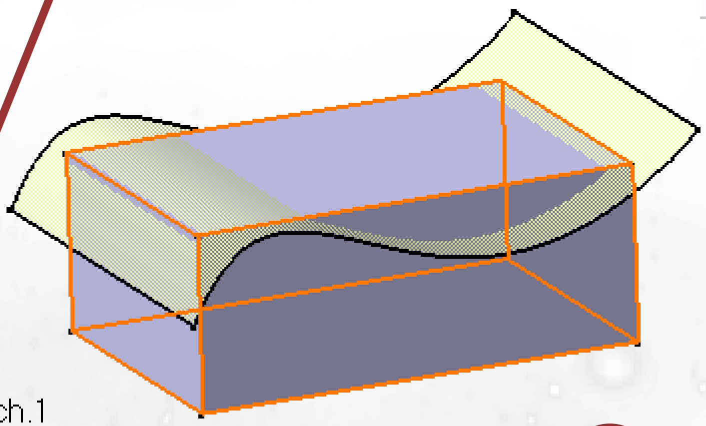




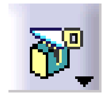
Split

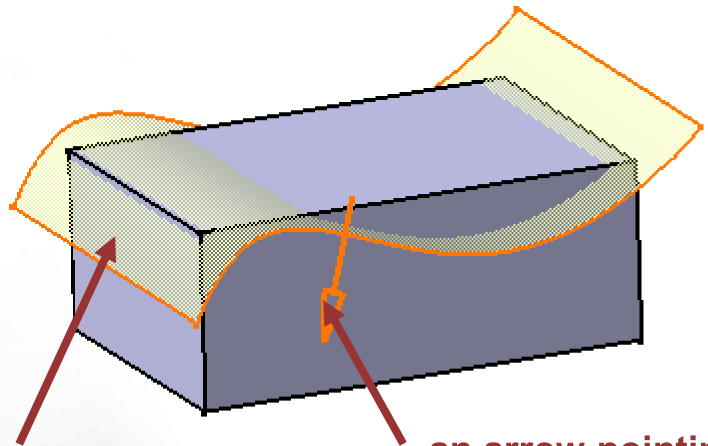
1 Select body to be split





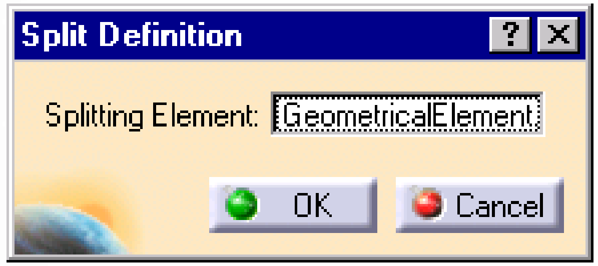
2 Select Split icon





3 Select splitting element

an arrow pointing to material to keep appears. Click to change direction if needed

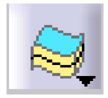


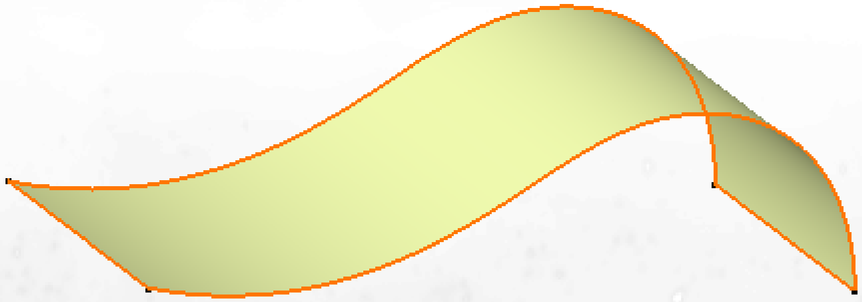
You can split a body with a plane, face or surface. A typical use is where internal structure must be trimmed and associated to an outer aerodynamic shape to allow rapid future change

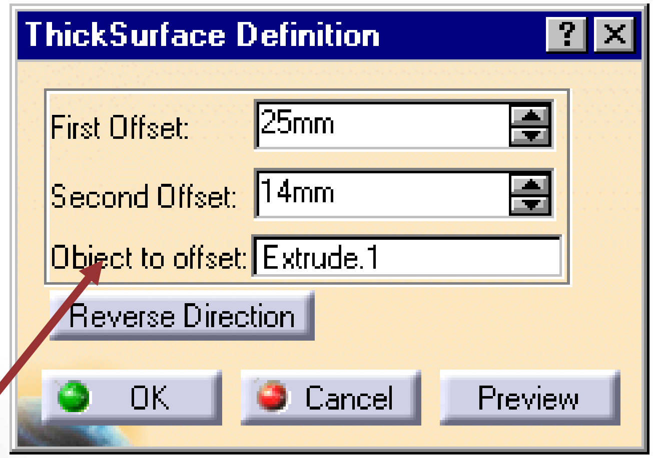
Thick Surface

1 Select surface to be thickened

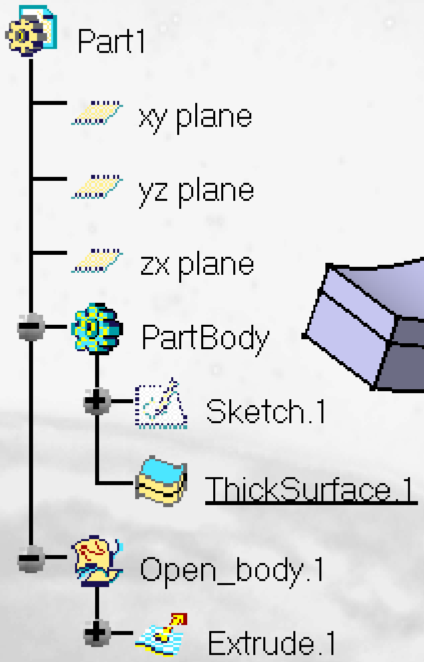
2 Select Thick Surface icon

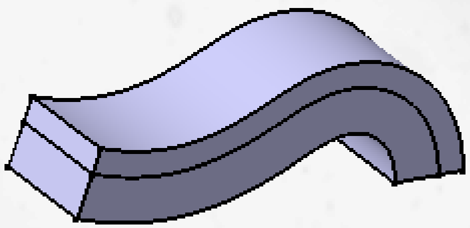


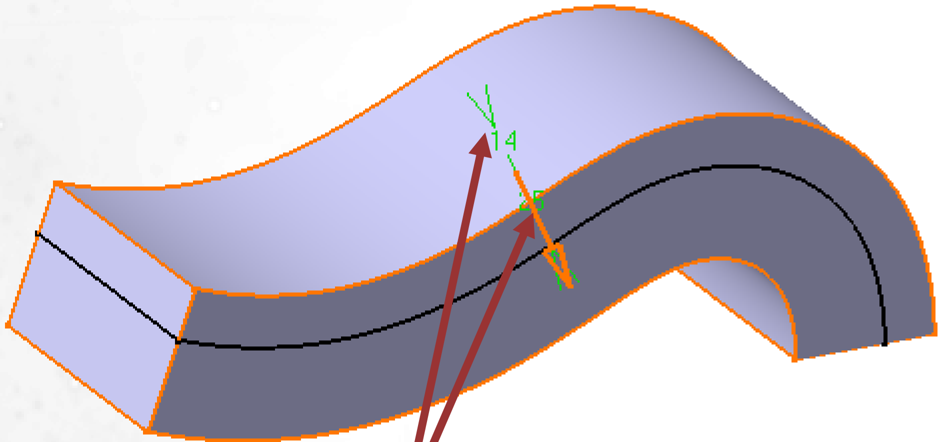




3 Modify thickness offsets





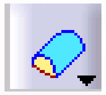


and the preview shows the corresponding upper and lower thickness

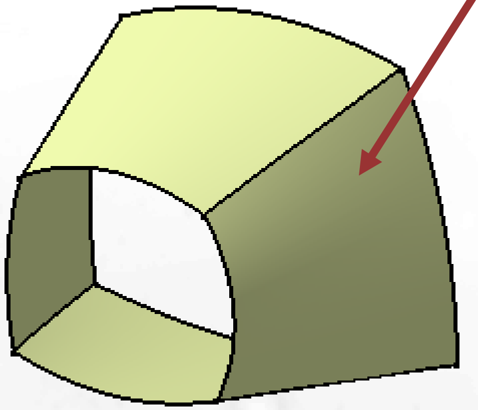
The resulting feature does not keep the color of the original surface

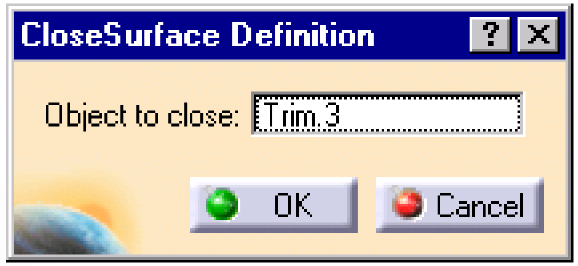
Close Surface

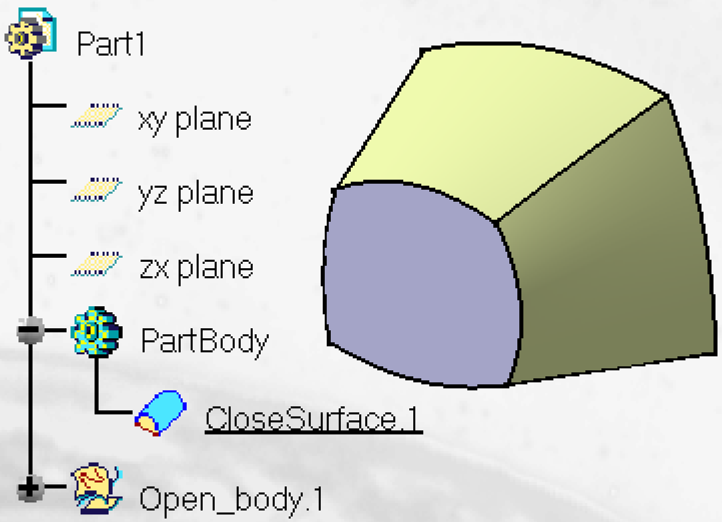
1 Select Close Surface icon

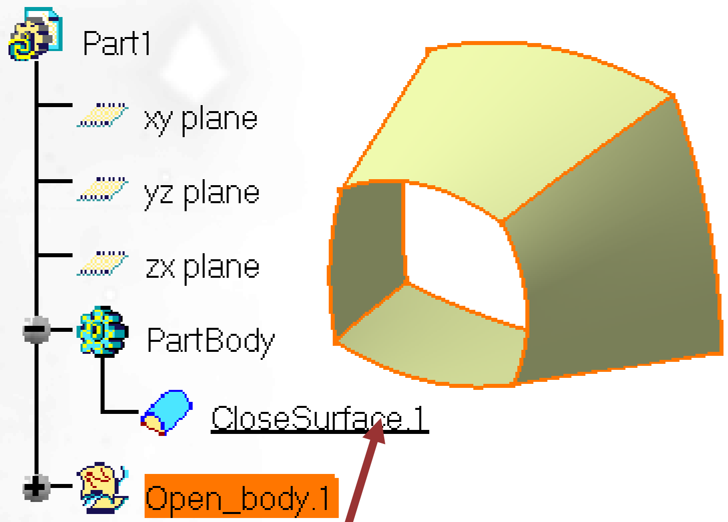


2 Select surface to be closed





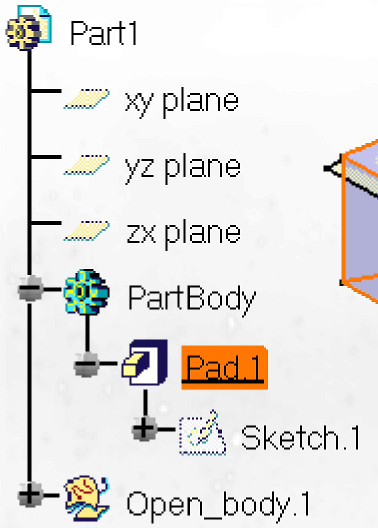


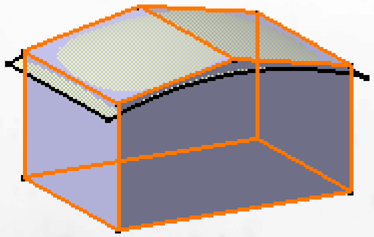


3 Closed Surface appears in specification tree

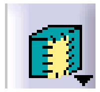
Sew Surface

1 Select the object to sew surface to

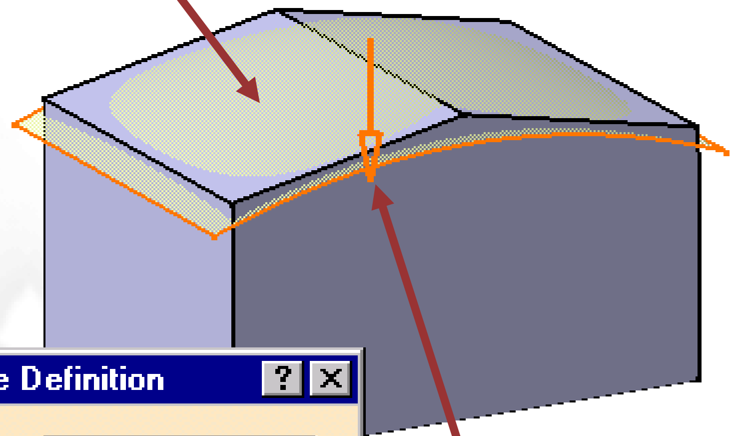


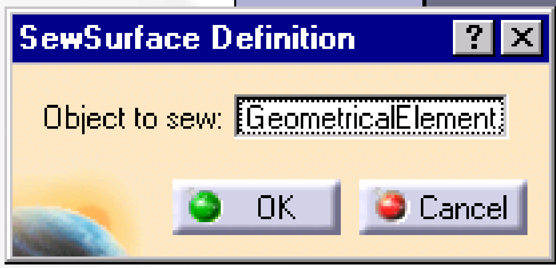


2 Select Sew Surface icon

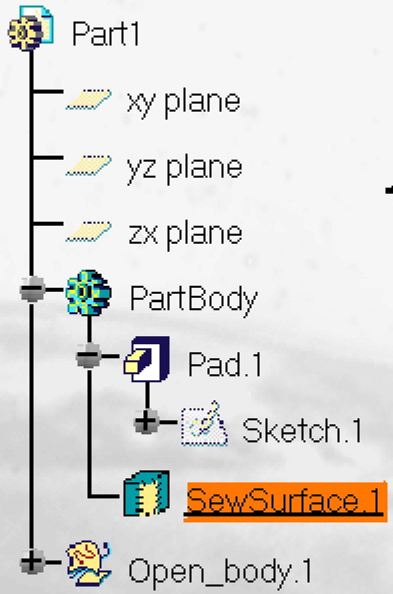


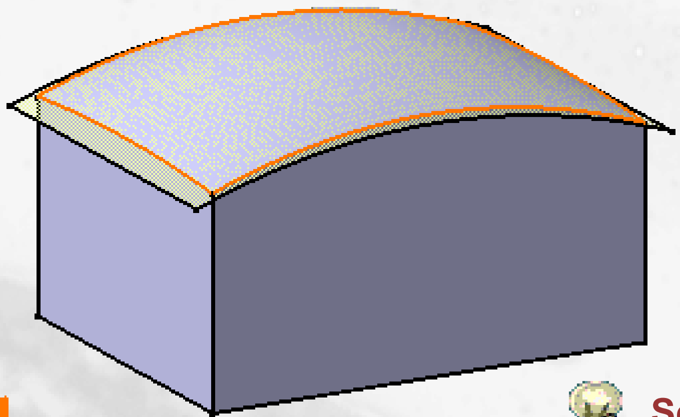
3 Select surface to sew





an arrow pointing to material to keep appears. Click to change direction if needed





Sewing means joining together a surface and a body. This capability consists in computing the intersection between a given surface and a body while removing useless material

Advance Draft

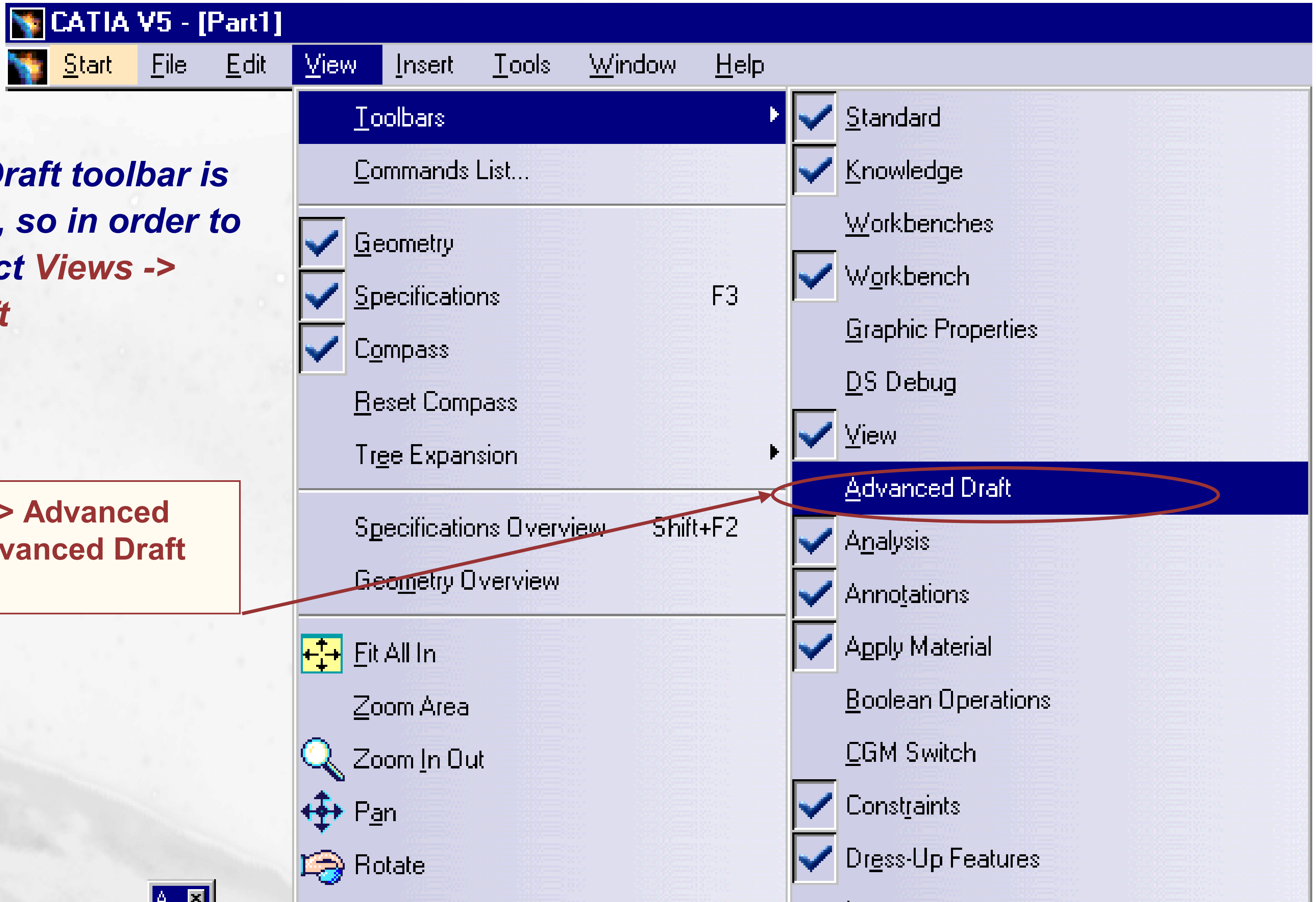
In this lesson we will see the Advanced Draft command

- **Advanced Draft**

What is the Advance Draft Command ? (1/5)

The Advanced Draft command lets you draft basic parts or parts with reflect lines but it also lets you specify two different angle values for drafting complex parts. This task shows you how to draft two faces with reflect lines, and this by specifying two different angle values and by using both modes available.

By default, the Advanced Draft toolbar is not accessible from CATIA, so in order to get it, you will have to select Views -> Toolbars -> Advanced Draft

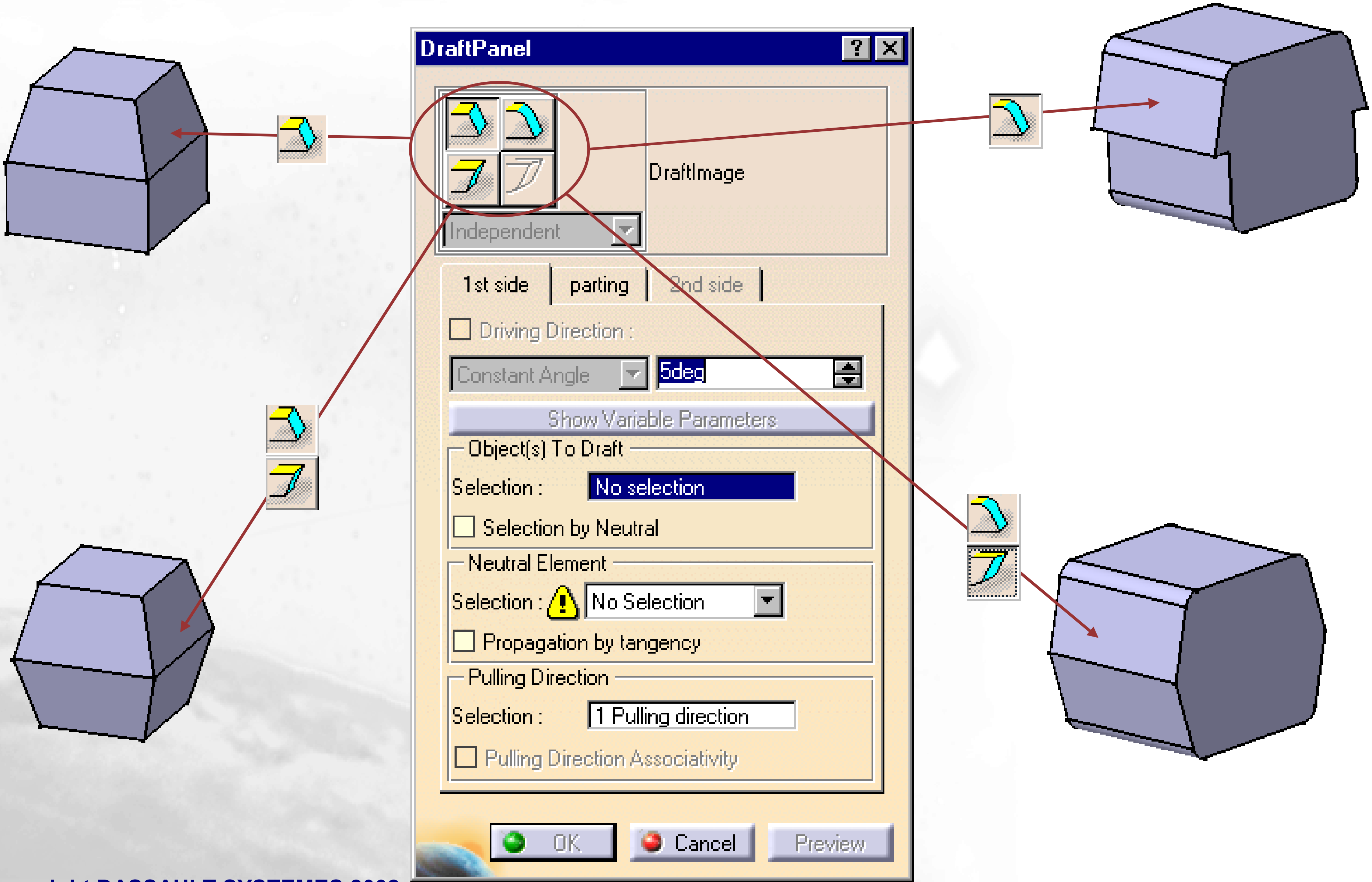


Select the View -> Toolbars -> Advanced Draft command to get the Advanced Draft toolbar

You will see the following toolbar:

What is the Advance Draft Command ? (2/5)

With the Advanced Draft command, you can define if you want to draft both sides or not and if you want to draft with reflect lines or not. To do so, you will have to activate one or two buttons as described hereafter



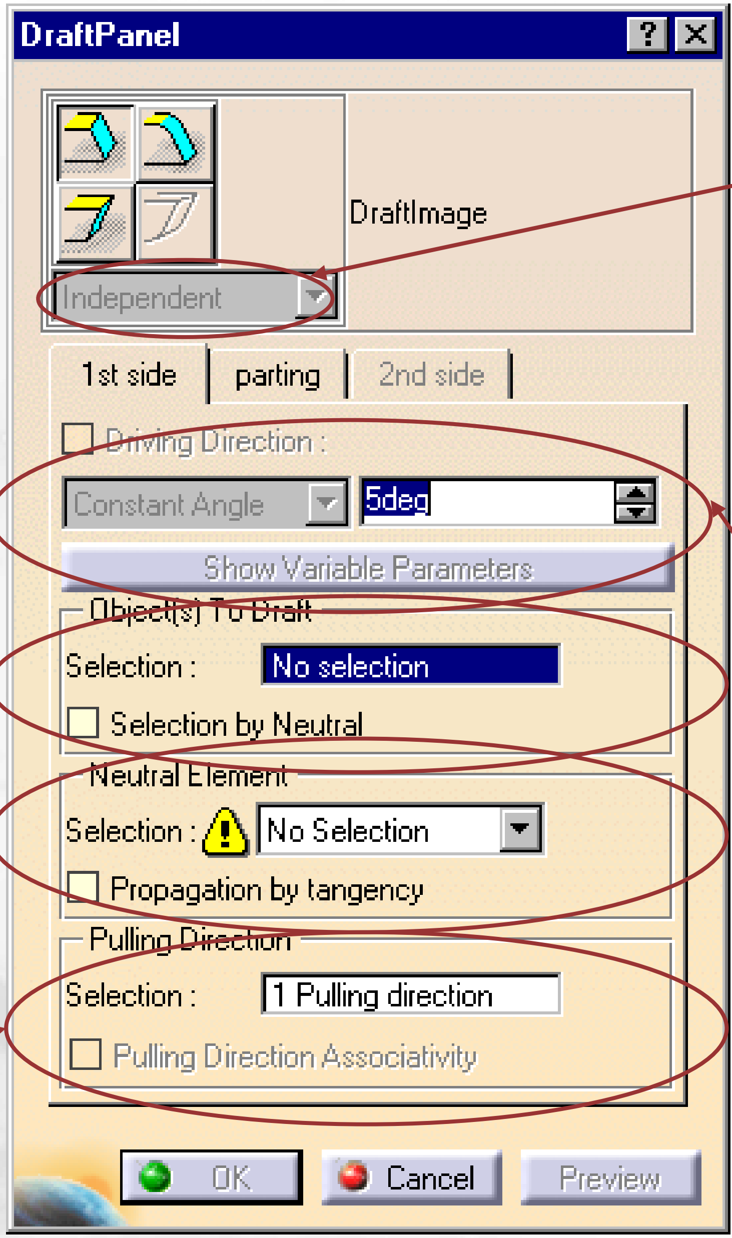
What is the Advance Draft Command ? (3/5)

The 1st side tab is used to define the characteristics of the draft angle for the selected faces. If you have decided to draft both sides, you will have to define the draft angle characteristics for the second side using the 2nd side tab. When drafting both sides with reflect lines, you can decide to get the draft angles independent or not

To define the Faces to be drafted

To define the neutral element

To define the pulling direction

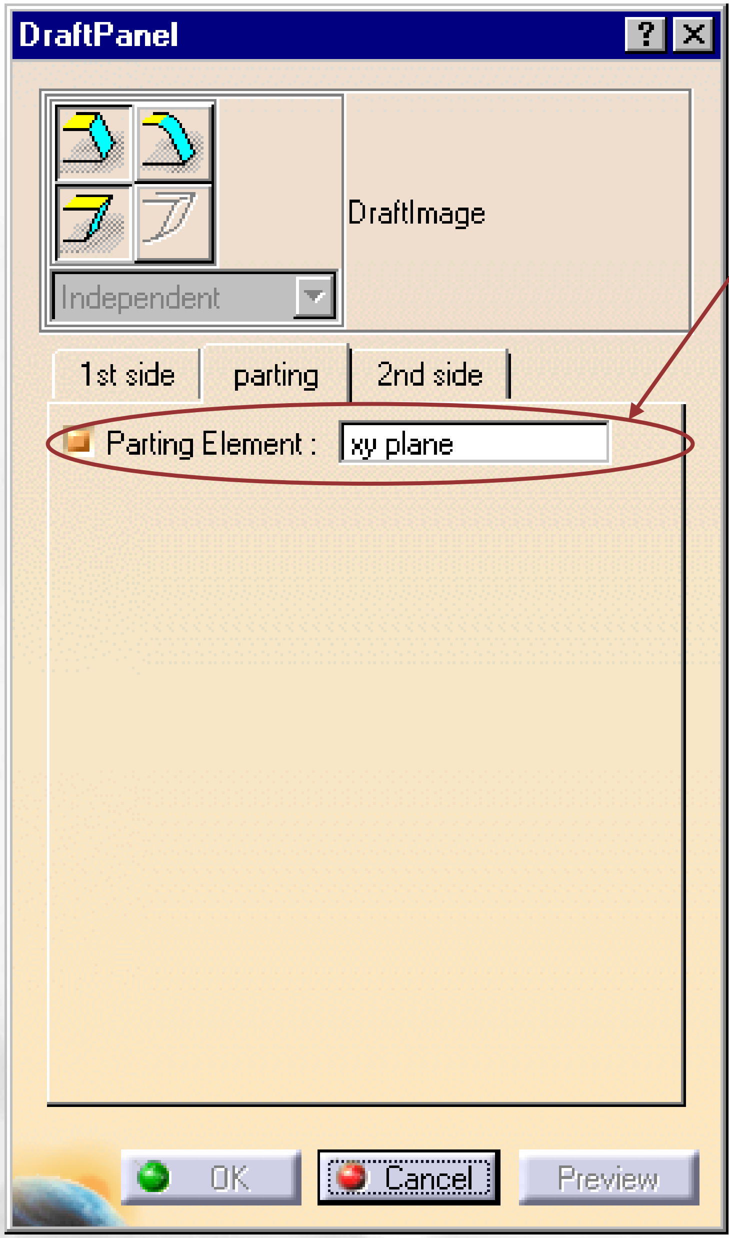


To define if the angle are the same or not when drafting both sides

To define the draft angle value

What is the Advance Draft Command ? (4/5)

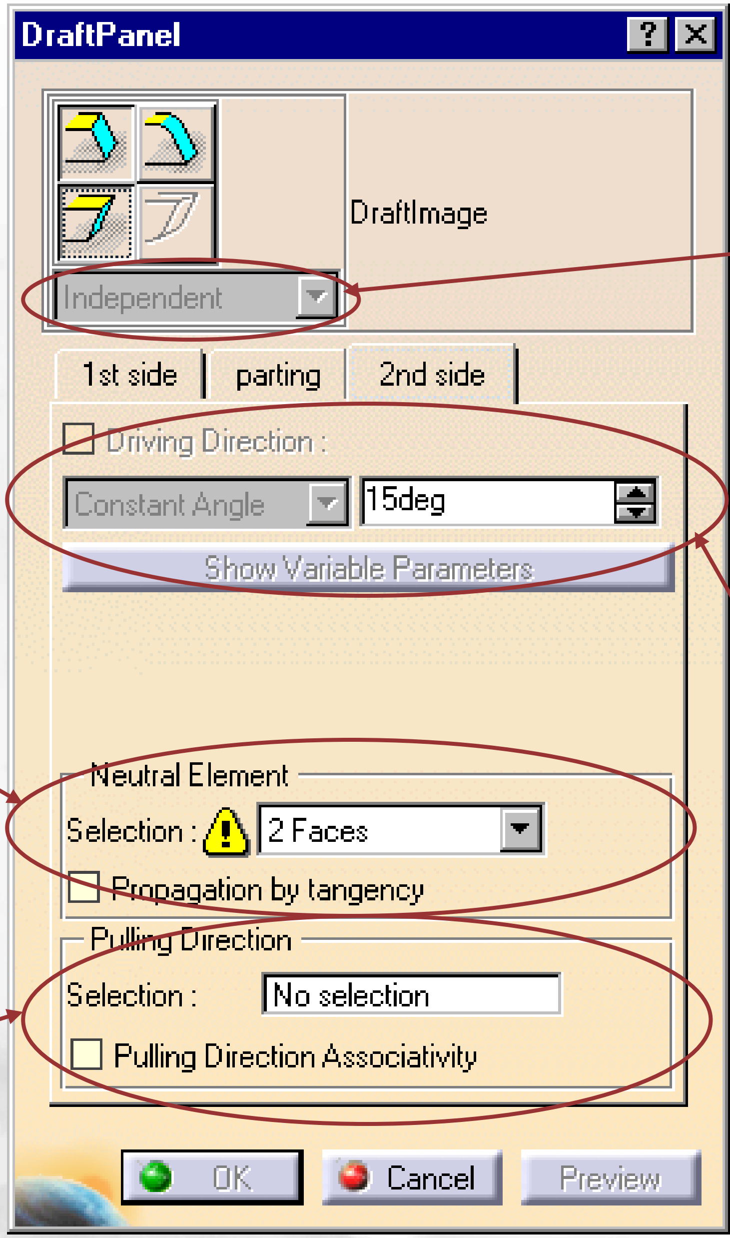
To define the Parting Element, you will have to used the parting tab. The parting Element can be a plane, a surface or a face



To define the parting element

What is the Advance Draft Command ? (5/5)

When you have decided to draft both sides with independent angle, you have to define the second side characteristics



To define if the angle are the same or not when drafting both sides

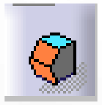
Neutral element

To define the draft angle value

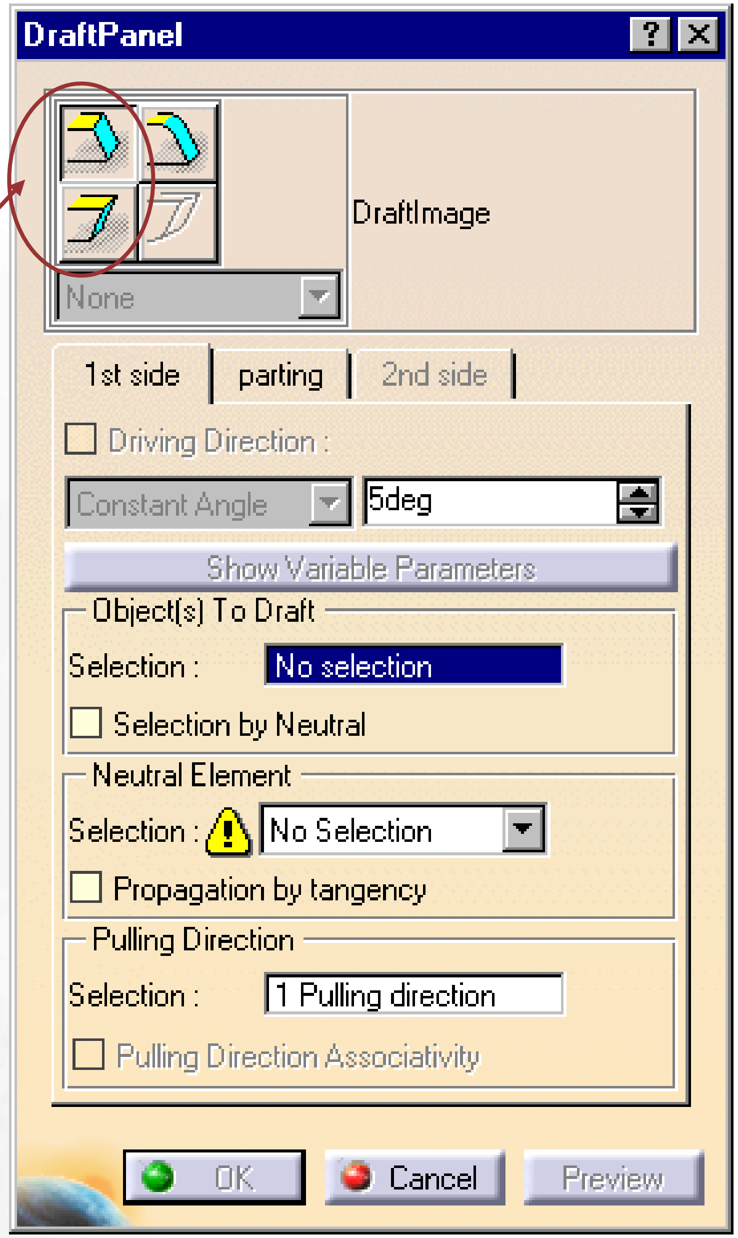
To define the pulling direction

Advanced Draft Angle: Draft Both Sides (1/9)

You are going to see how to draft both sides using the Advanced Draft icon

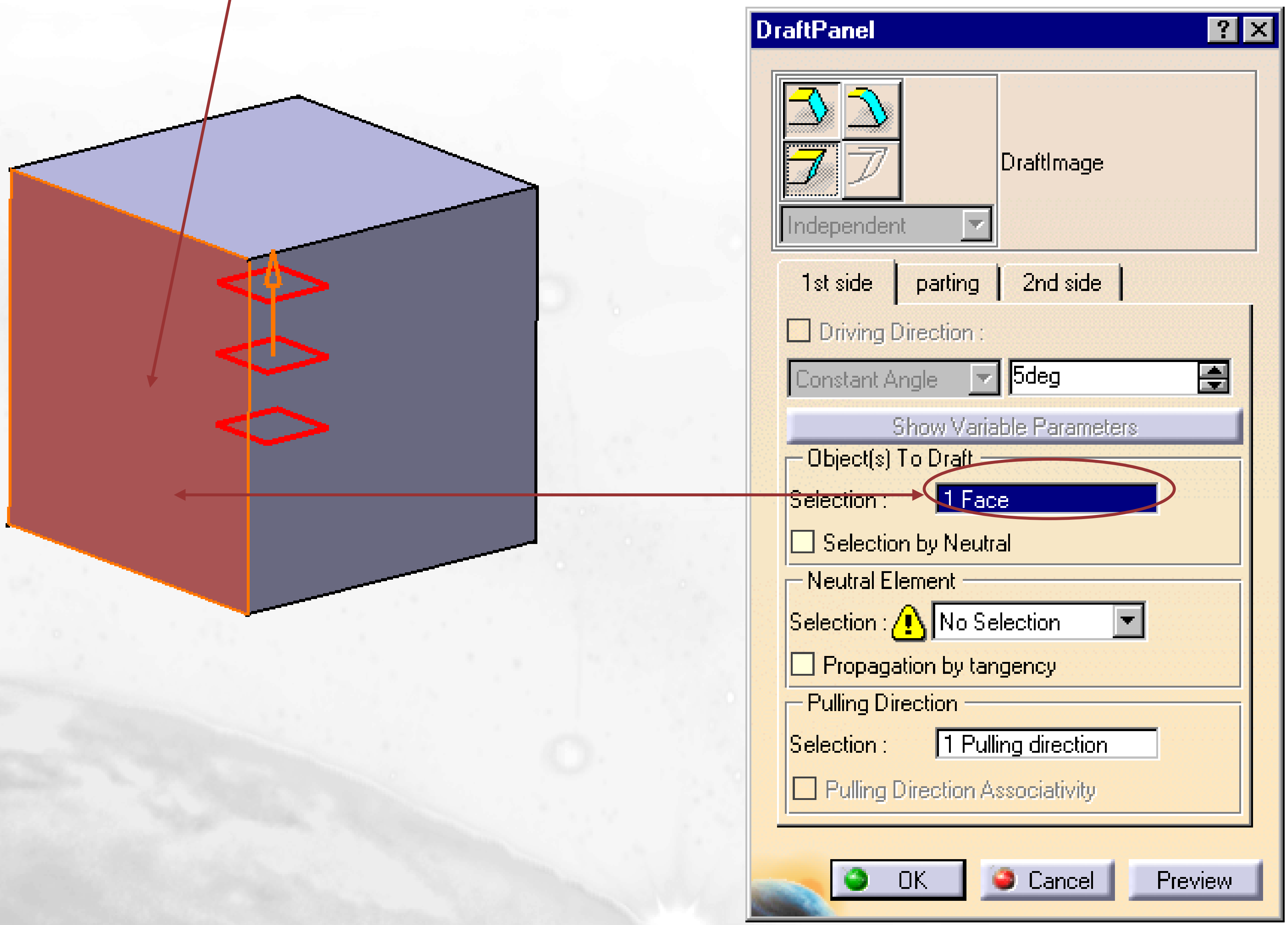
1. Select the Advanced Draft icon 

2. Activate these two buttons



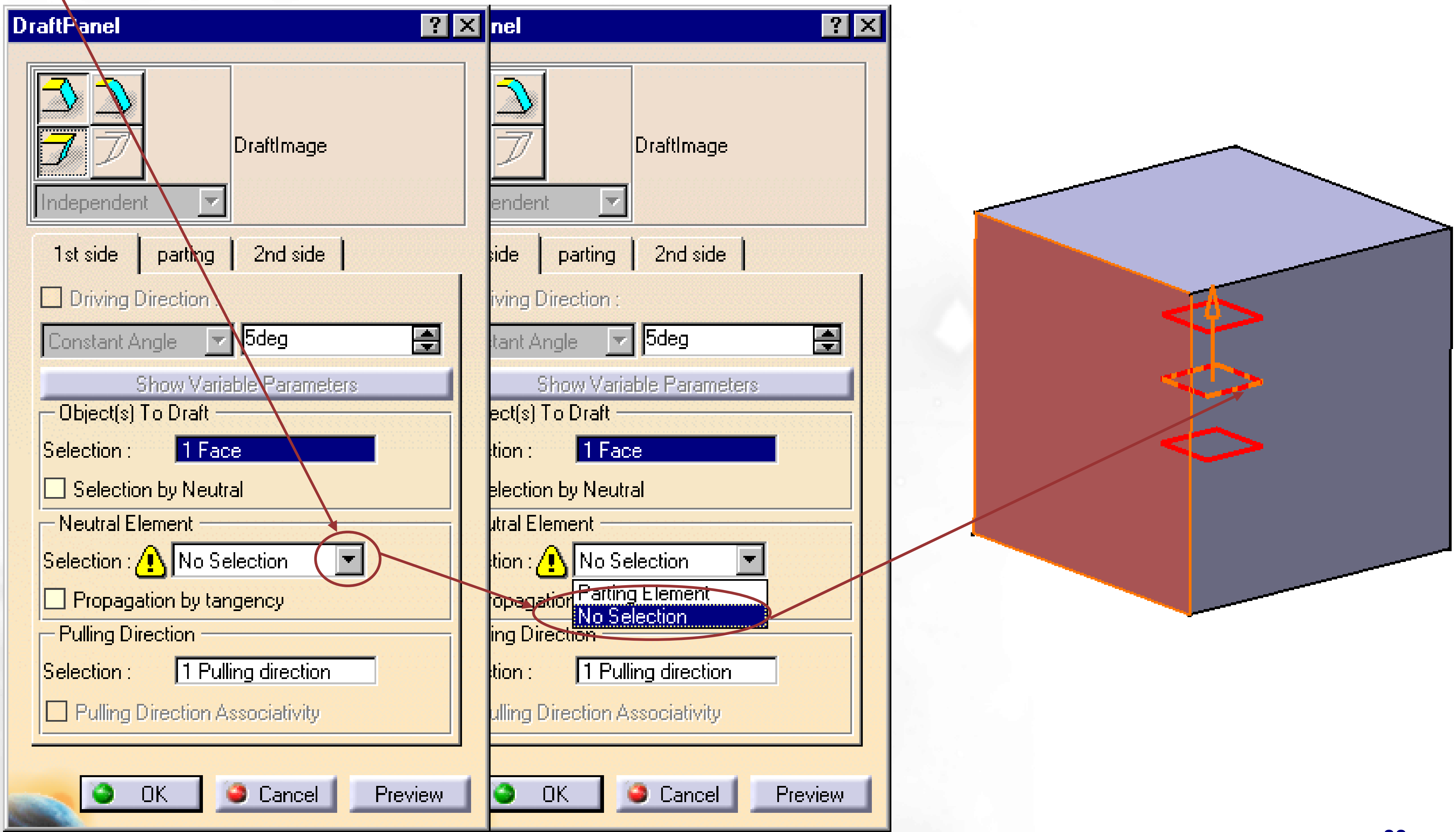
Advanced Draft Angle: Draft Both Sides (2/9)

3 As the object to be drafted, select this face



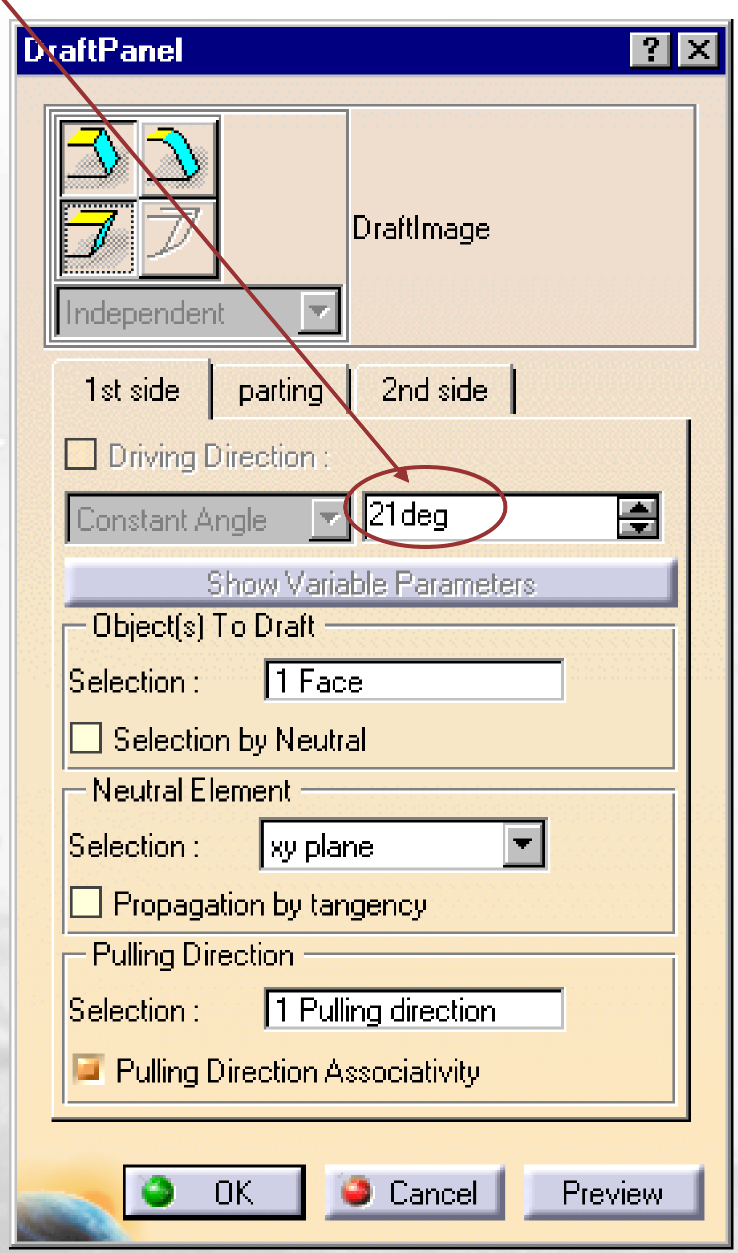
Advanced Draft Angle: Draft Both Sides (3/9)

4 Select the No Selection option from the Neutral Element combo, then select the indicated plane

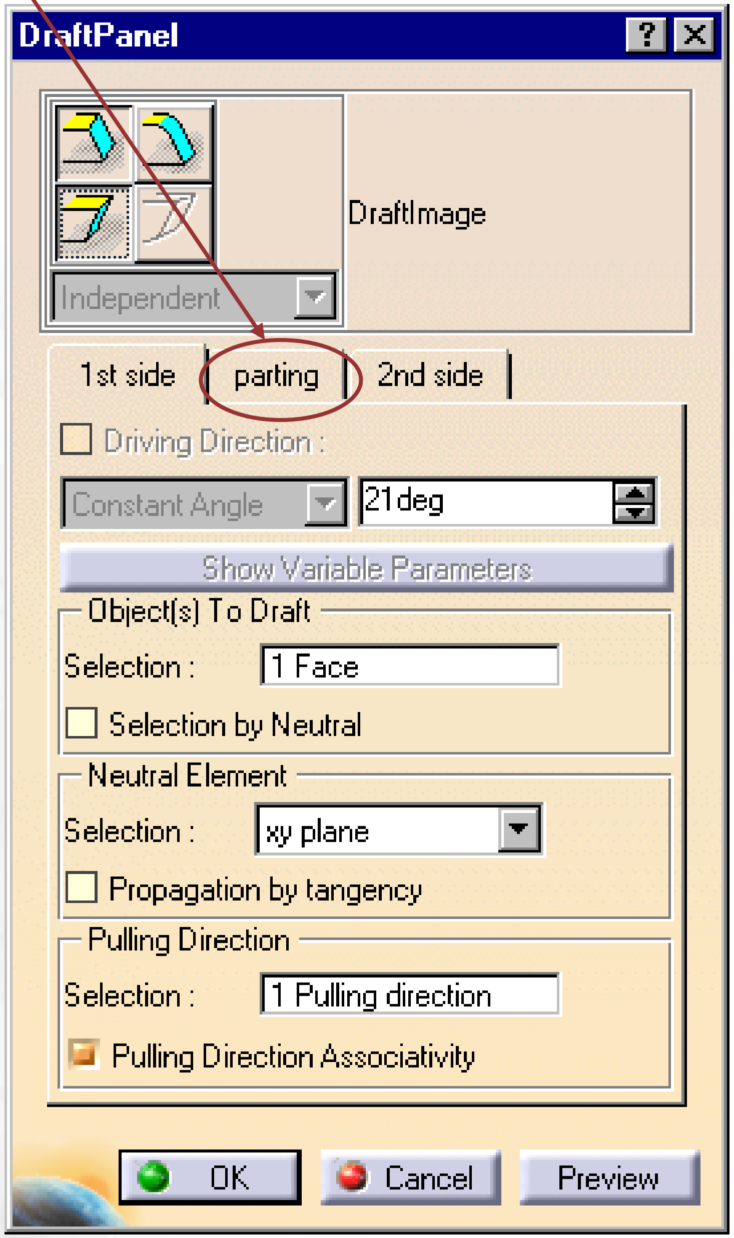


Advanced Draft Angle: Draft Both Sides (4/9)

5 Enter 21 in the angle field

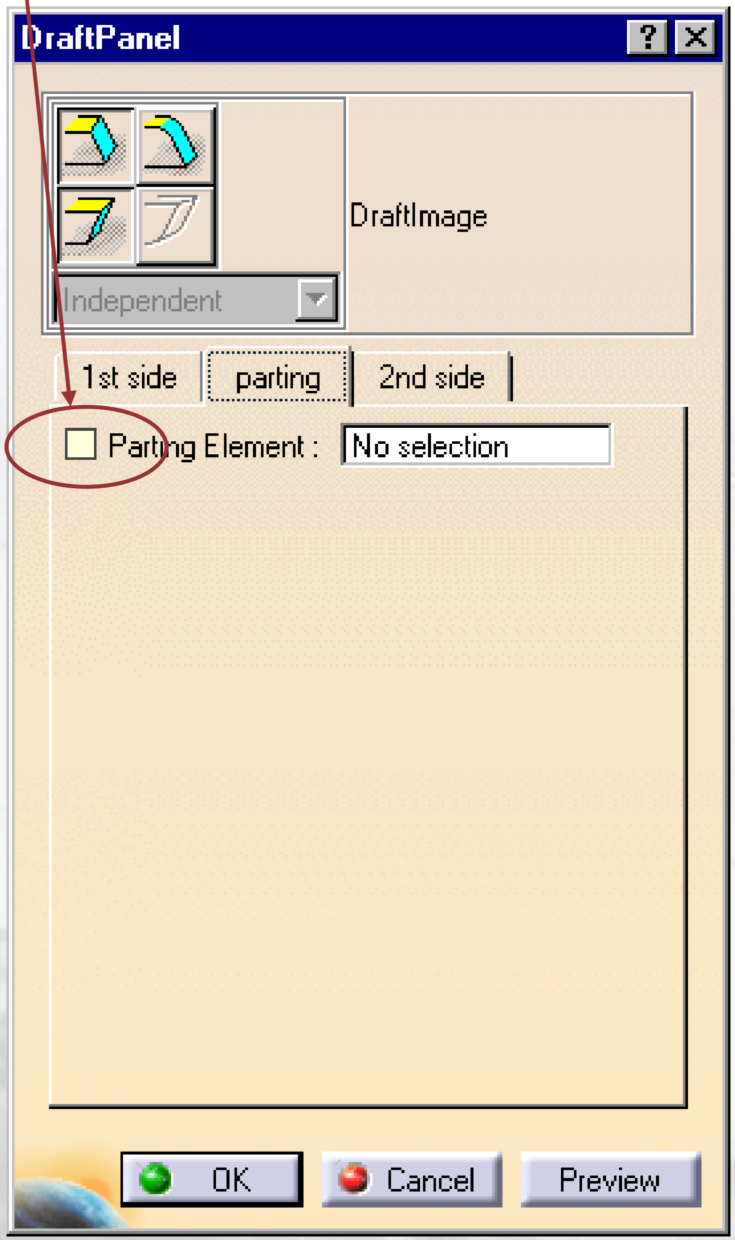


6 Select the parting tab

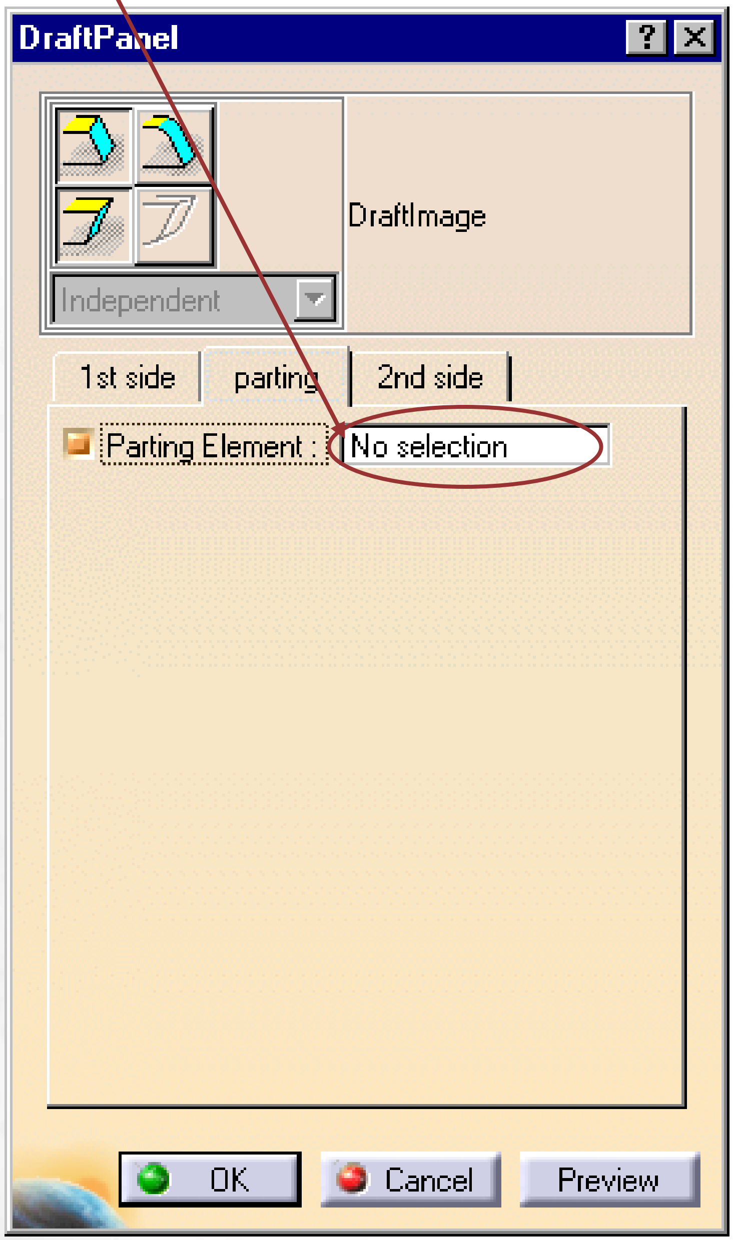


Advanced Draft Angle: Draft Both Sides (5/9)

7 Select the Parting Element button

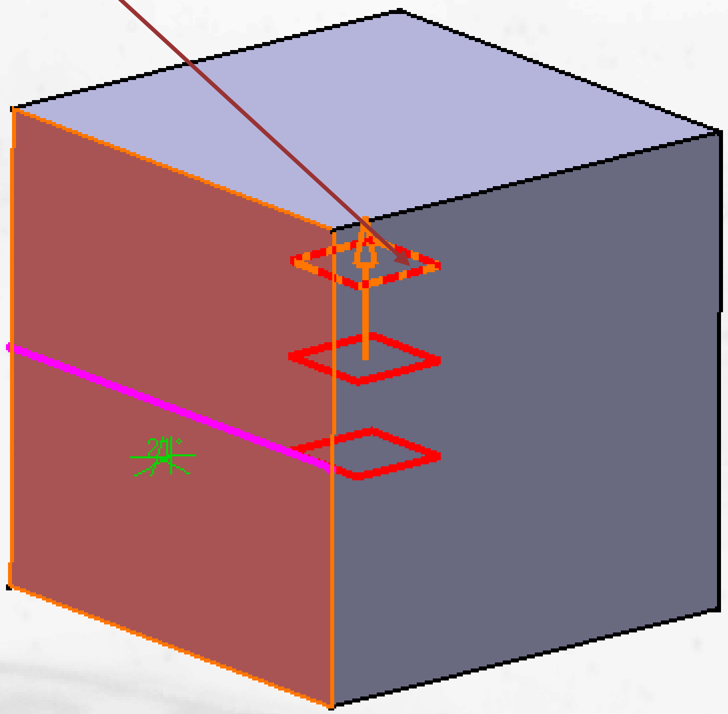


8 Select the Parting Element field

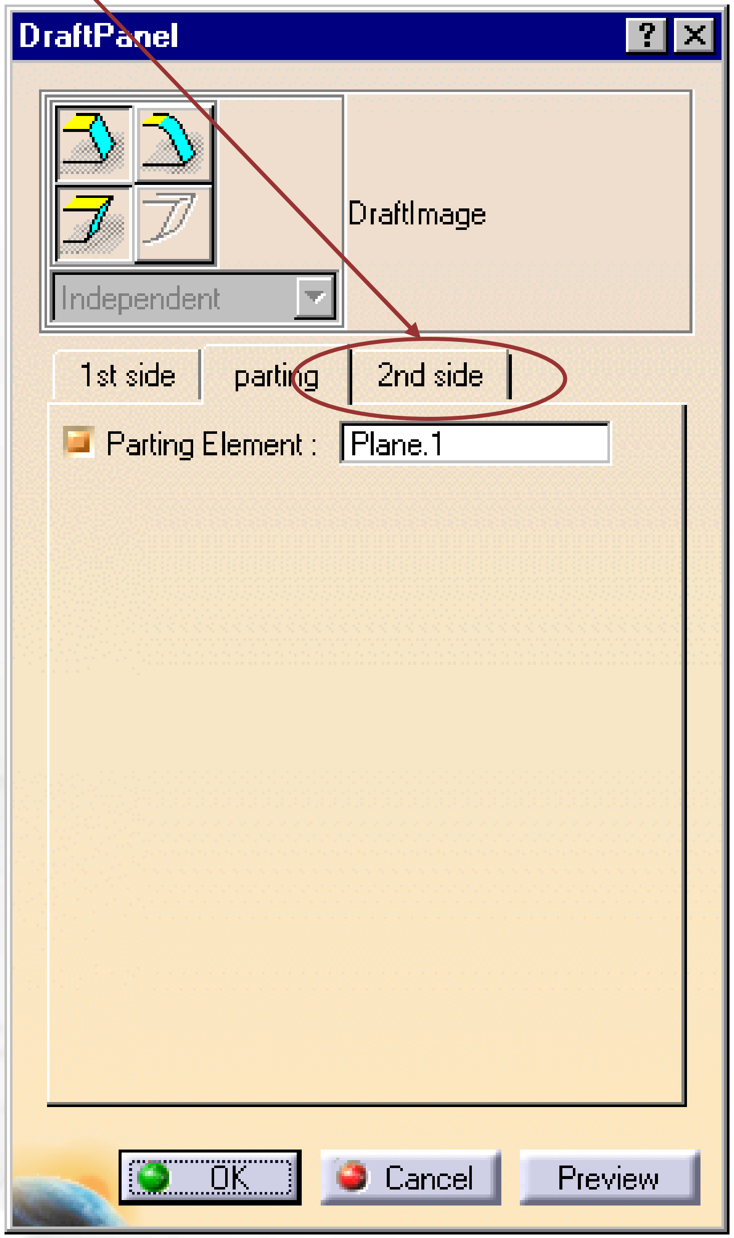


Advanced Draft Angle: Draft Both Sides (6/9)

9 Select the indicated plane

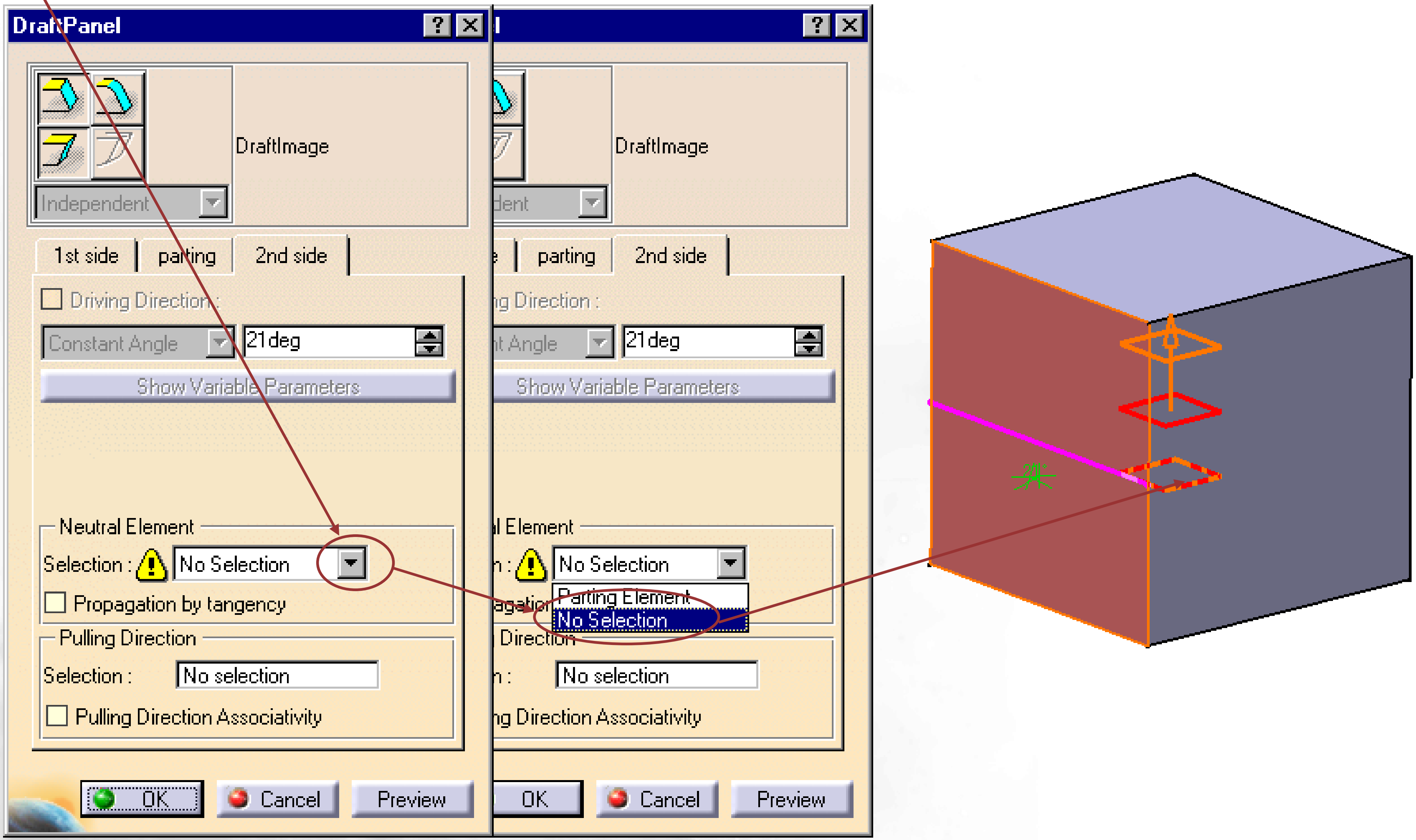


10 Select the 2nd side tab



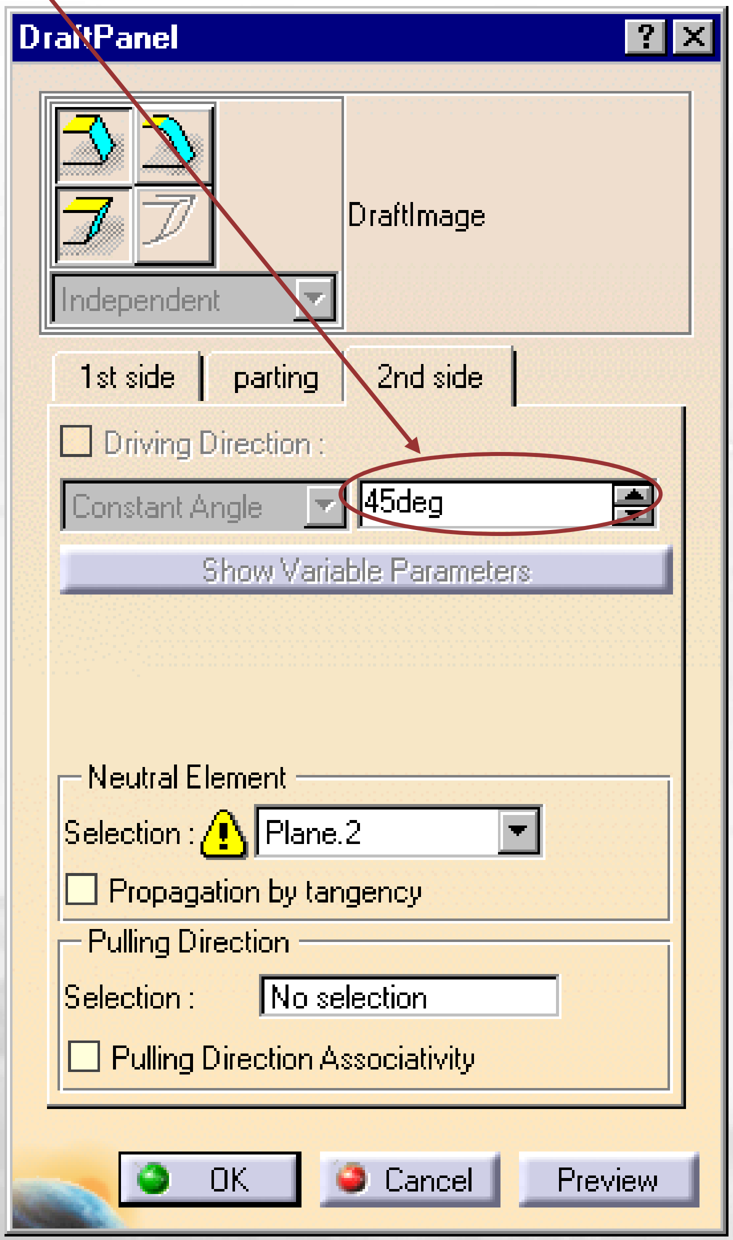
Advanced Draft Angle: Draft Both Sides (7/9)

11 Select the No Selection option from the Neutral Element combo, then select the indicated plane

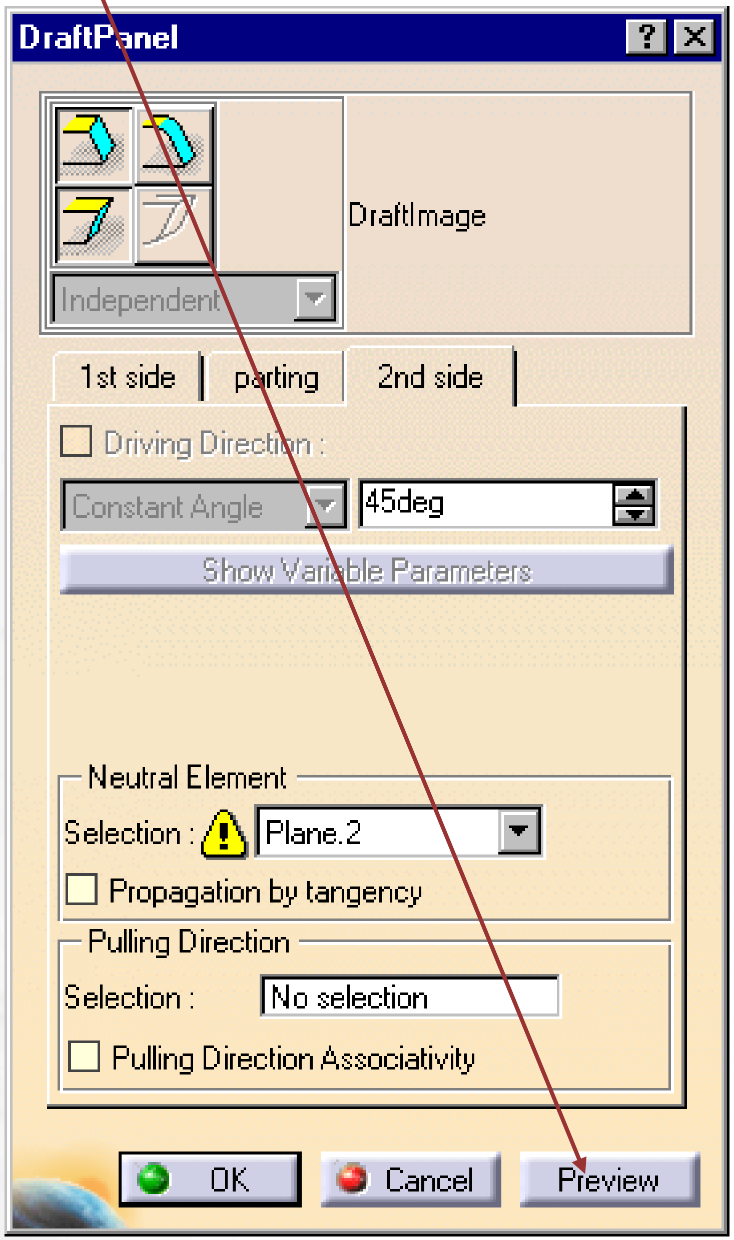


Advanced Draft Angle: Draft Both Sides (8/9)

12 Enter 45 in the angle field

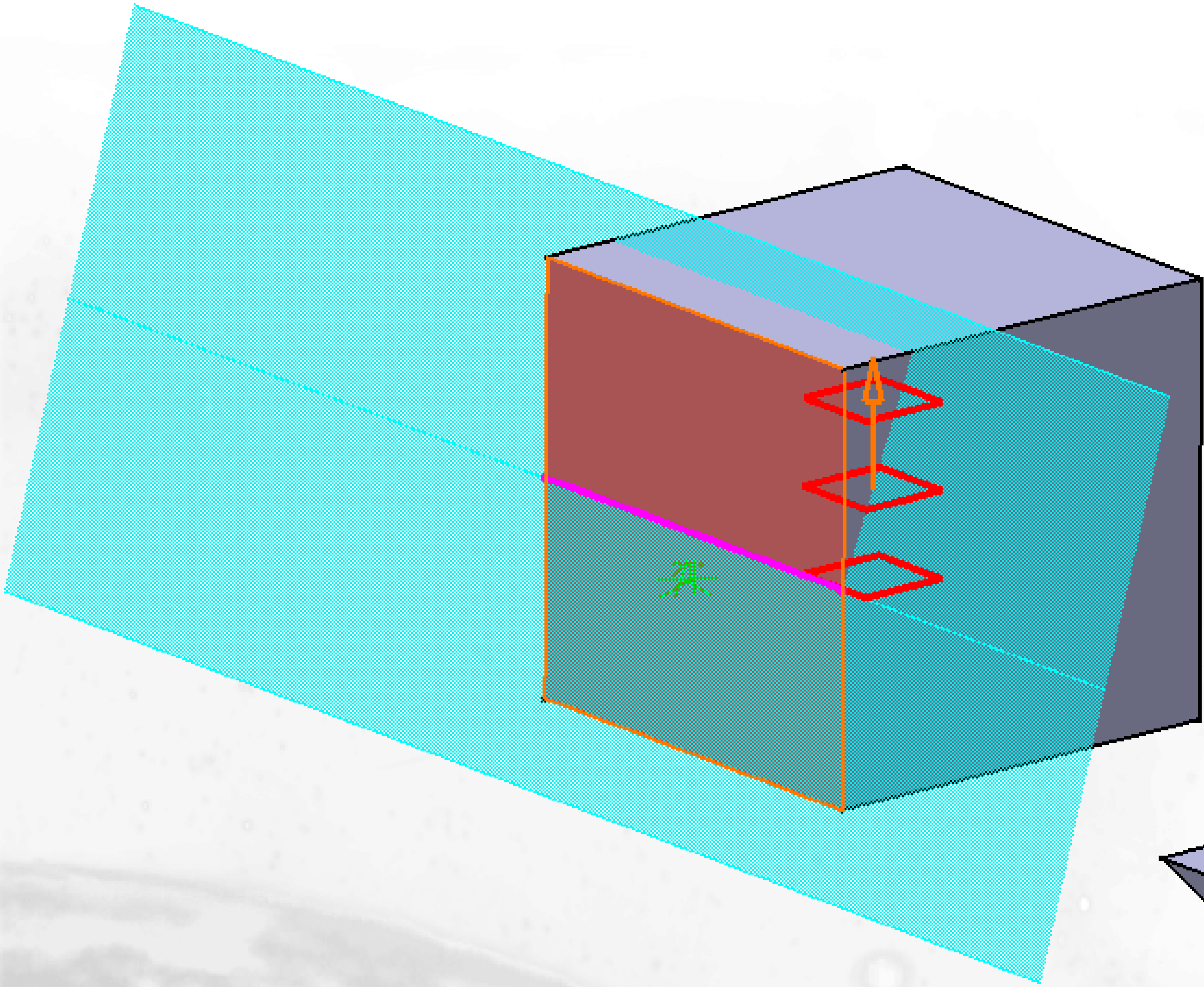


13 Select Preview



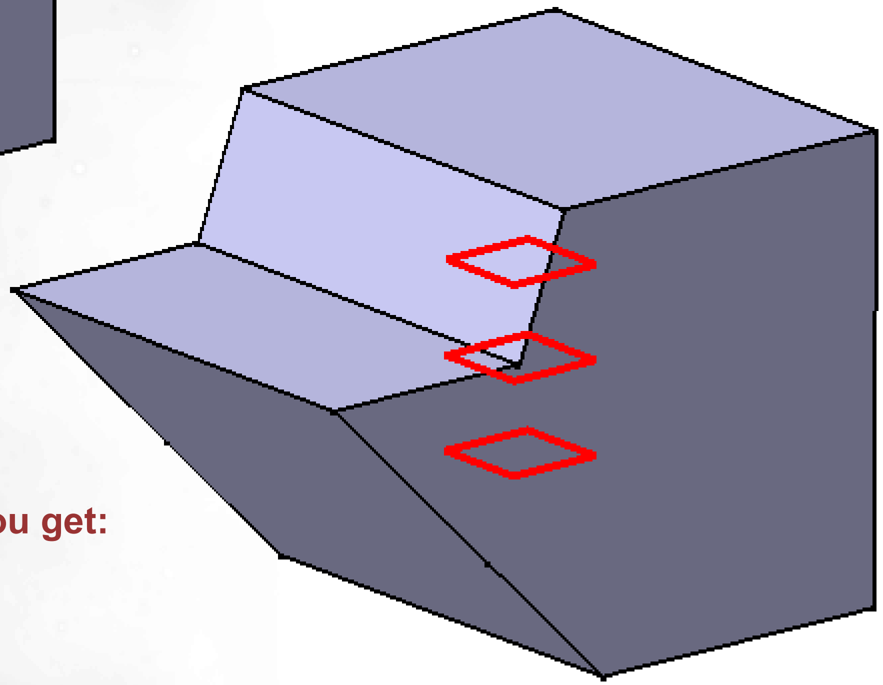
Advanced Draft Angle: Draft Both Sides (9/9)

You will see



14 Select OK in the dialog box

You get:



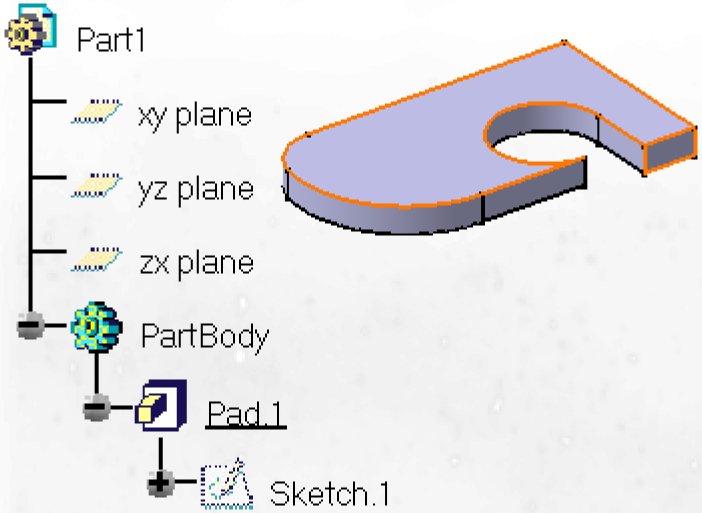
Thickness

We will see how to add material on a selected face by defining a thickness

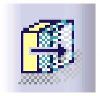
- Thickness Creation

Thickness

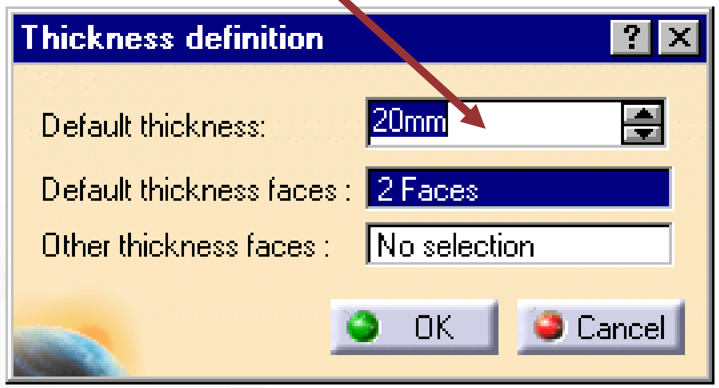
1 Multi-select faces to be thickened



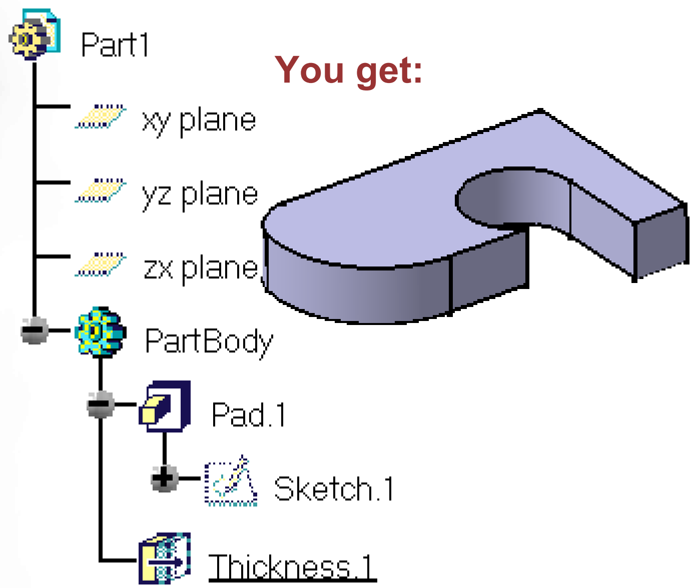
2 Select Thickness icon



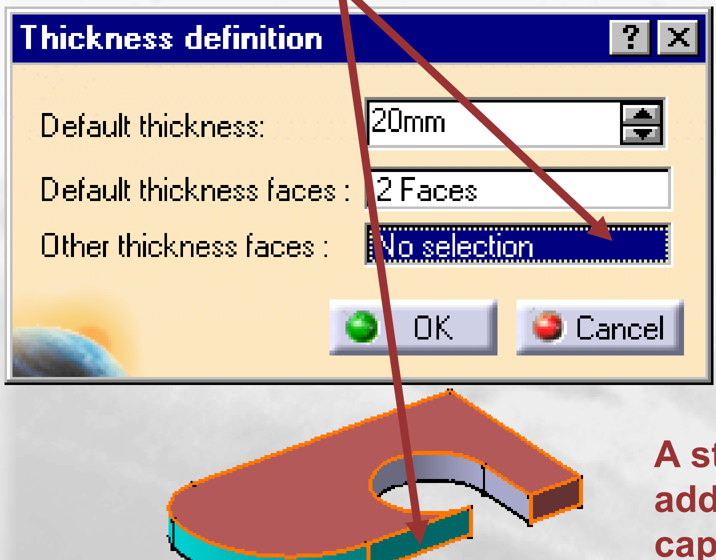
3 Modify thickness definition of material to add in the dialog box



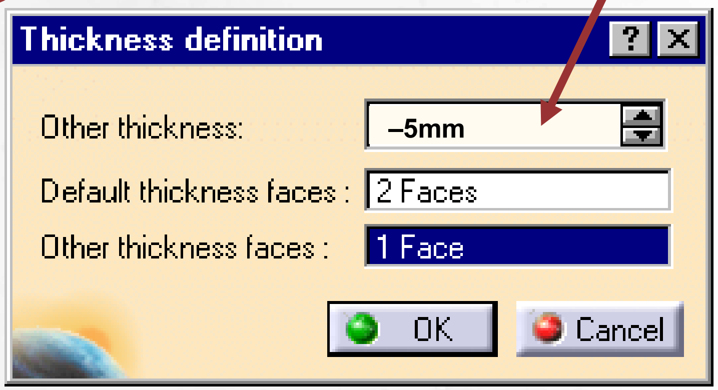
You get:



4 Select the Other thickness faces field then select another face



5 Enter –5 in the Other thickness field then select OK



A standard use of thickness is when material has to be added or removed before machining a part. Thickness captures the design intent and allows rapid change

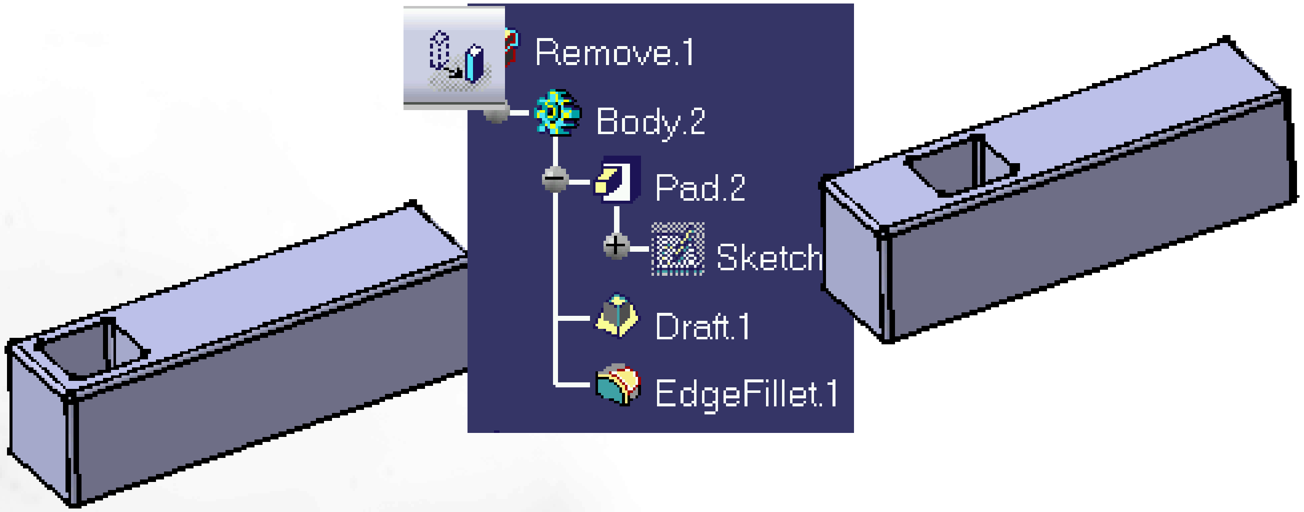
Using Transformations

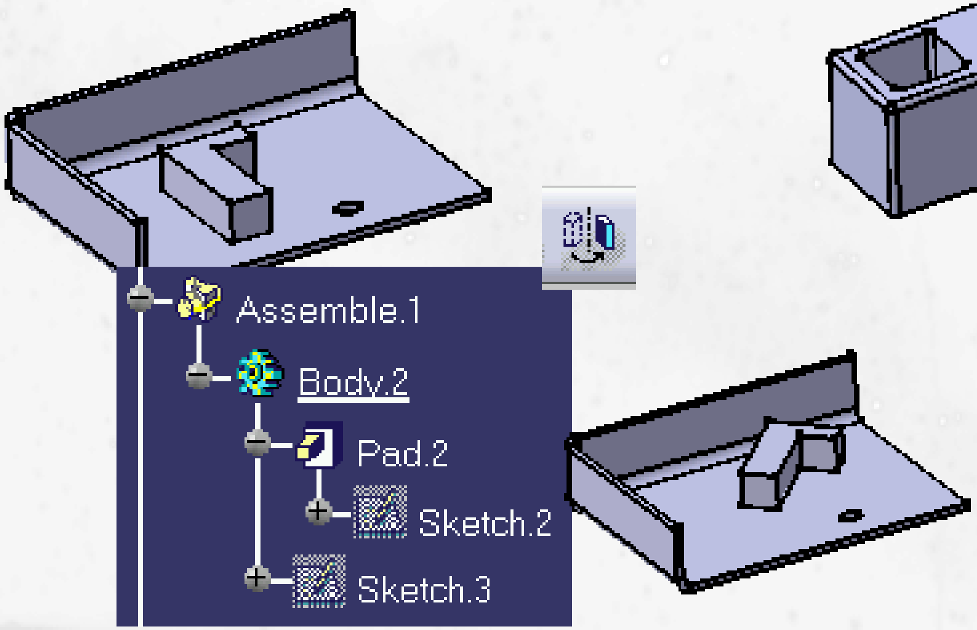
In this lesson we will learn how to use all of the various types of Transformations

- Using Transformations

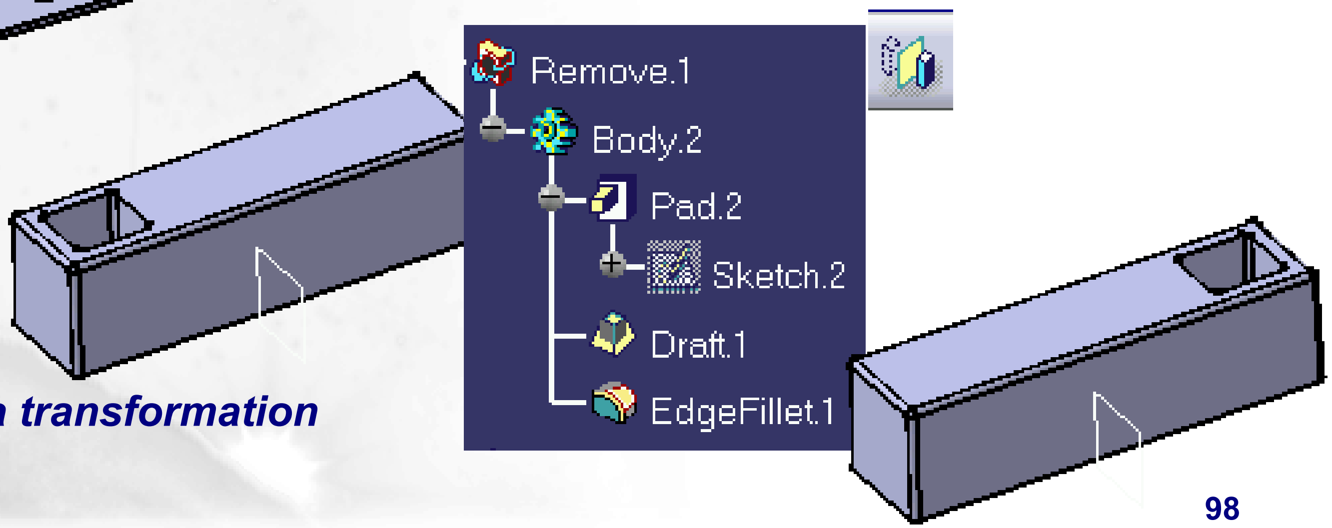
What is Transformation ?

Transformation is the ability to move a body either by translating it along an axis, rotating it around an axis or moving it symmetrically around a plane





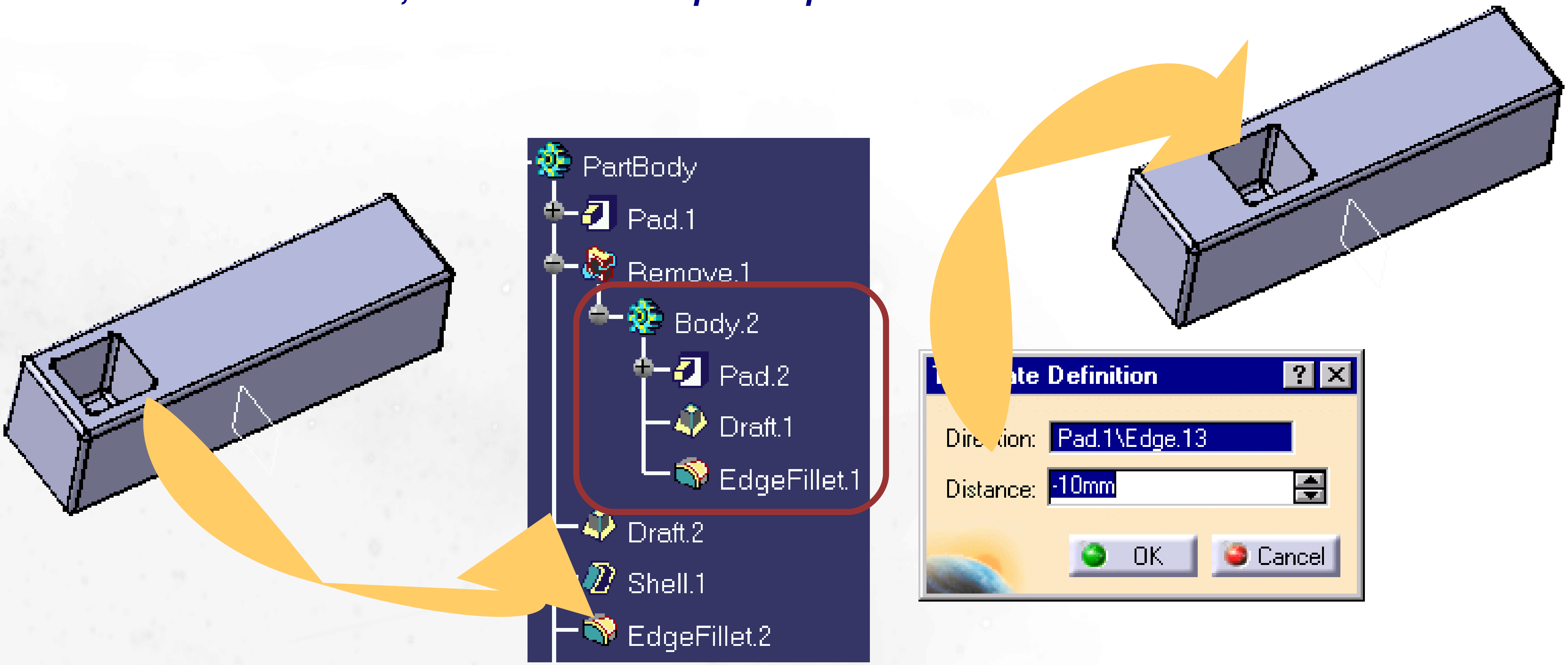
Transformations come in 3 types :
- Translation along an edge
- Rotation about an axis
-Symmetry around a plane



Geometry cannot be duplicated using a transformation

When Should we Use a Transformation ?

Transformations are useful when you have created some geometry and decide that it needs to be moved, or rotated into a specific position

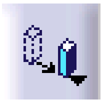


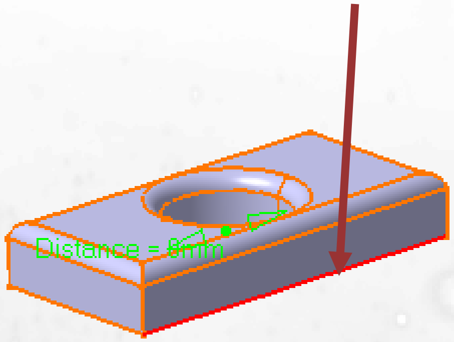
There are some cases where it would not be easy to create the geometry in the plane that it is needed in because it requires the use of geometry not in the plane. You can create the geometry in the wrong plane and then rotate or translate to its proper position

Transformations can only be used on either the whole Part Body or an individual Body within the Part

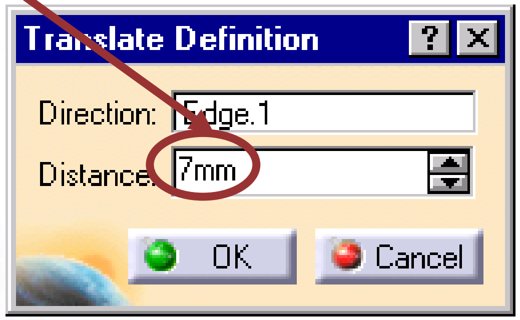
Translation

1 Select the Translation icon

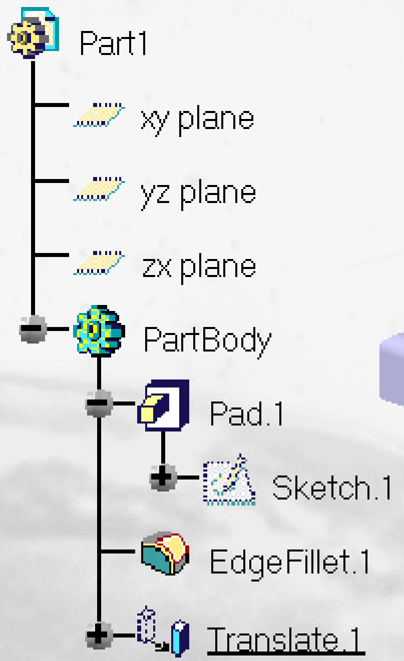


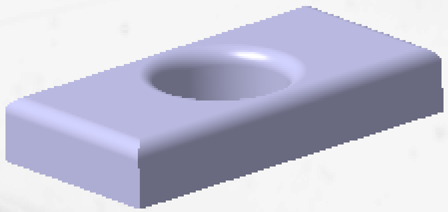


3 Enter the amount of translation or drag the icon, then select OK

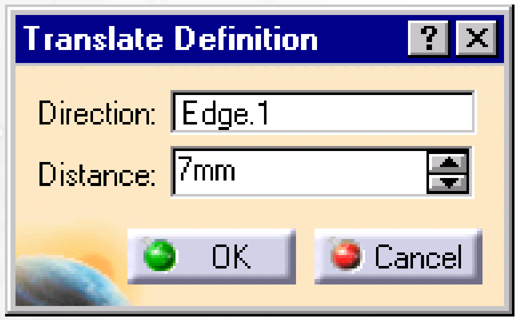


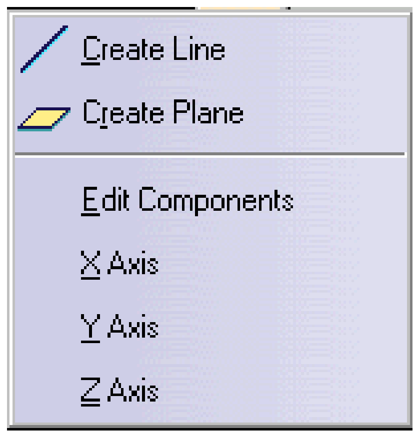
4 Select OK





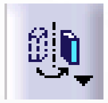
You can also define the direction using the contextual menu on the Direction field



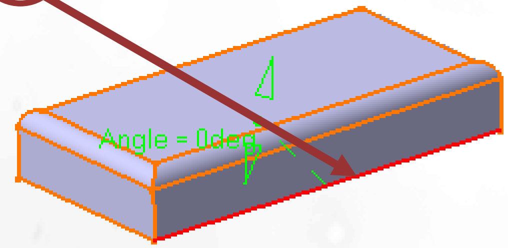


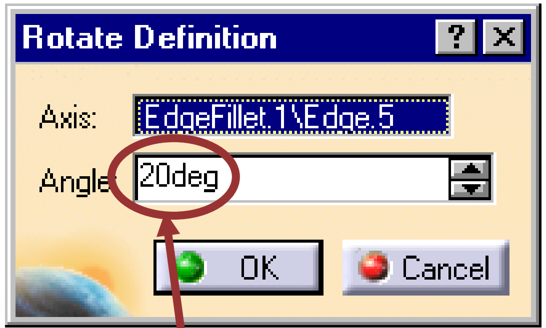
Rotation

1 Select the Rotation icon

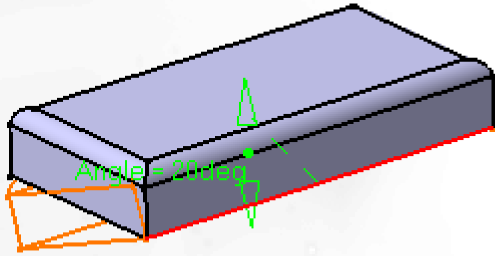


2 Select an edge for rotation axis…

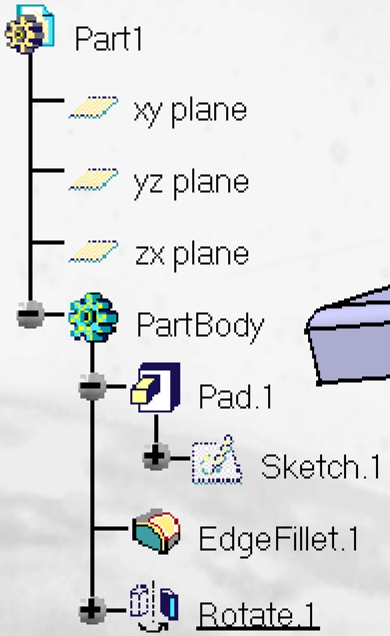


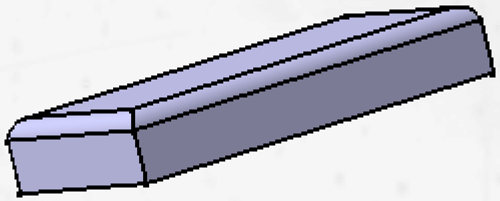


…and enter the amount of rotation or drag the icon

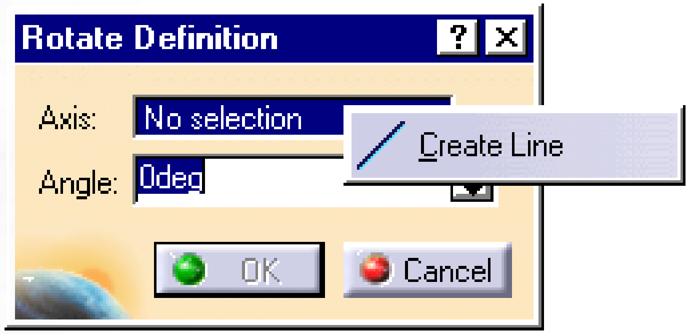


3 Select OK





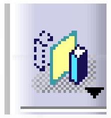
You can also define the axis using the contextual menu on the Axis field

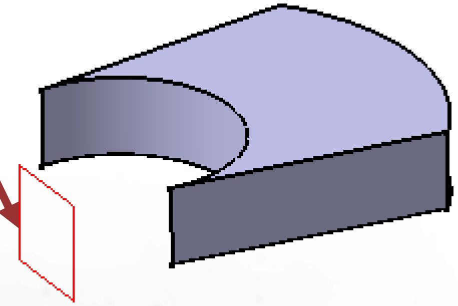


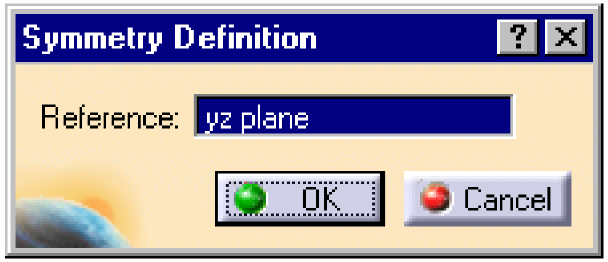
Symmetry

2 Select plane defining the symmetry

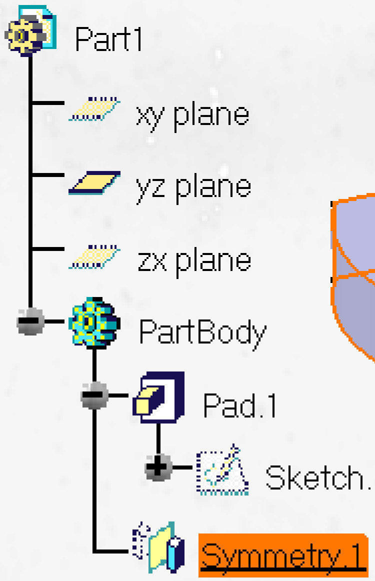
1 Select the Symmetry icon

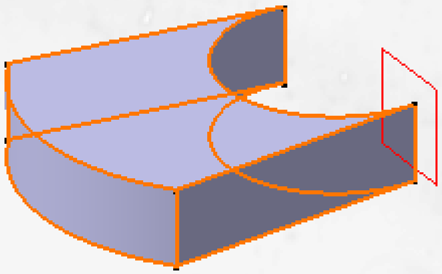


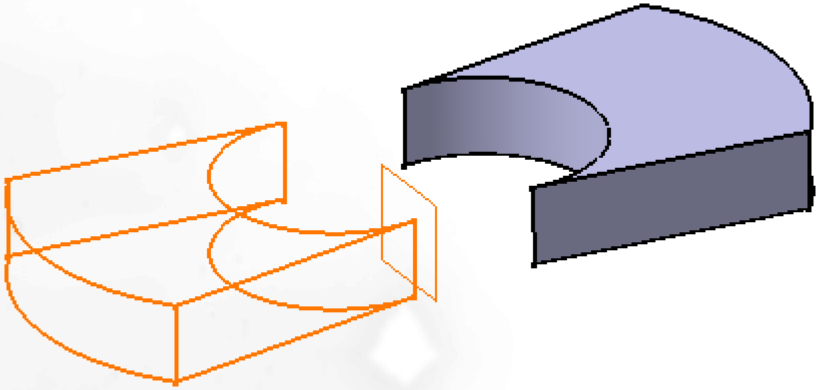




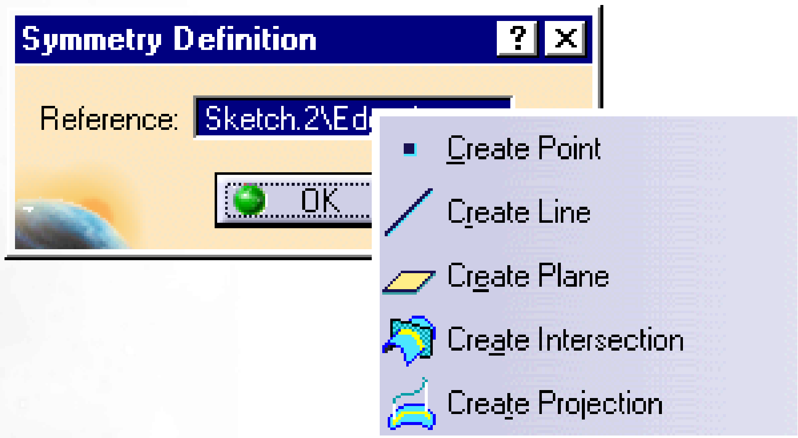
4 Select OK



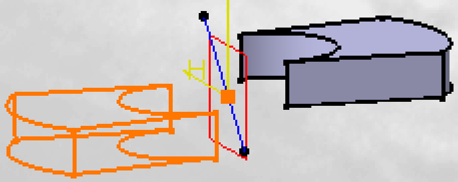




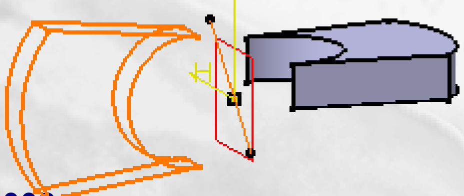
You can also define the Reference using the contextual menu on the Reference field



Instead of a plane, you can use a Point as the Reference element



Instead of a plane, you can use a Segment as the Reference element



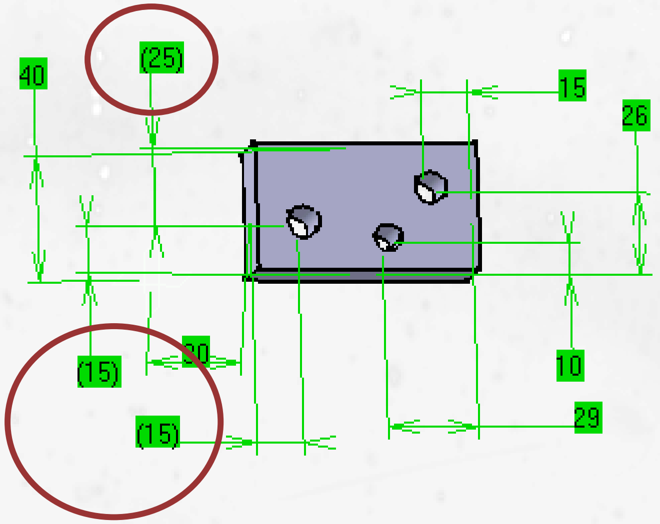
3D Constraints

In this lesson we will learn how to use 3D constraints

- **Creating and Using 3D Constraints**

What is a 3D Constraint ?

A 3D Constraint is the same as any other constraint only it is applied in the 3D model itself. Basically you will note that some are reference type constraints and others are regular constraints. Creation is the same as in the sketcher, so we will concentrate on their usage here



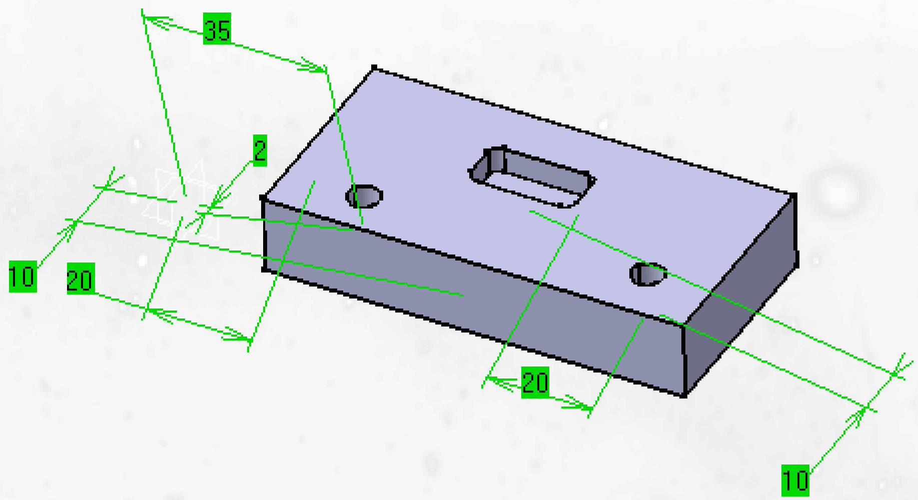
Reference constraints are shown in parenthesis and cannot be modified

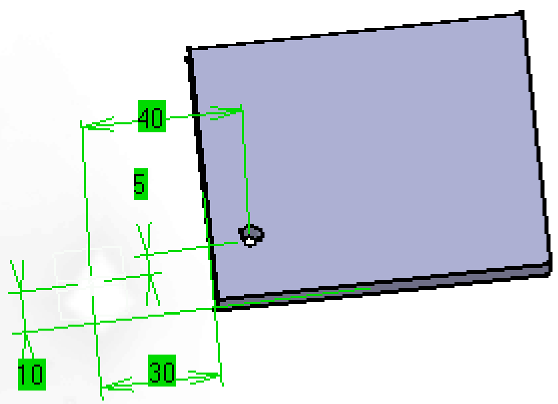
They are reference because there are general other constraints in the sketcher or implicit to the geometry that are constraining the geometry

Normally, 3D constraints are modifiable and can be linked and driven as others are in the sketcher

When Do we Use 3D Constraints ?

They can be used whenever you have 3D geometry that you wish to link to some type of 3D datum plane or surface



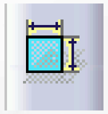


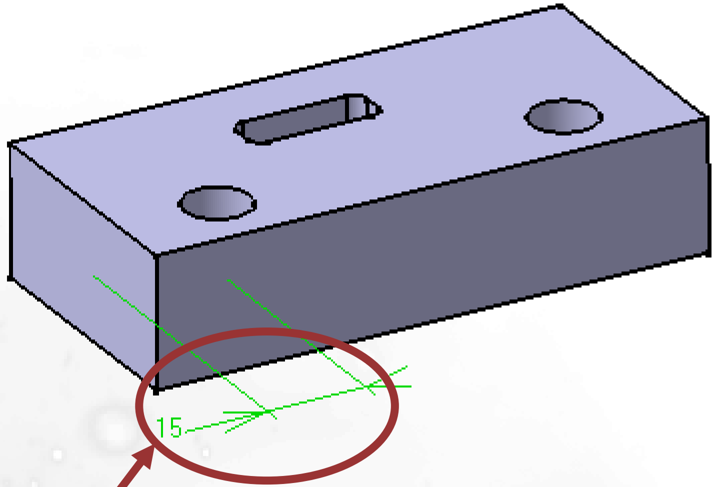
They are also useful when you need to drive the location of a piece of geometry created earlier in the design from a piece of geometry created later in the model. Thus this will limit some of the need to re-ordering of the part

You may also find it useful when you are using Copy and Paste to locate the pasted piece of Geometry from where you wish

Creating 3D Constraints

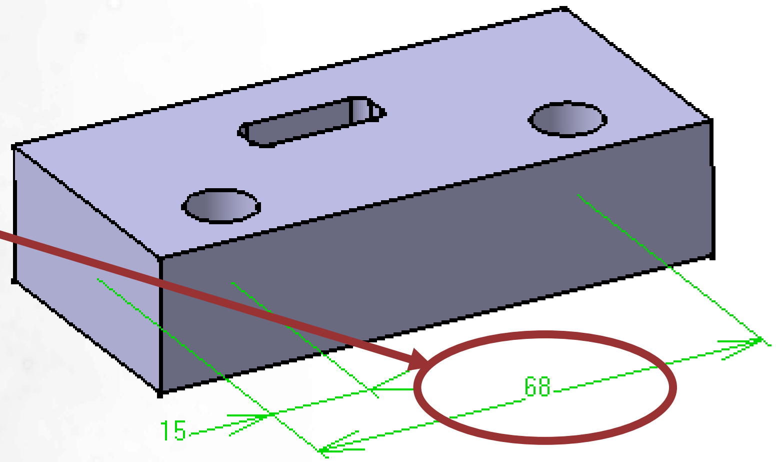
1 Select the Constraint icon and create a constraint between the left side surface and the hole on the left side of the part





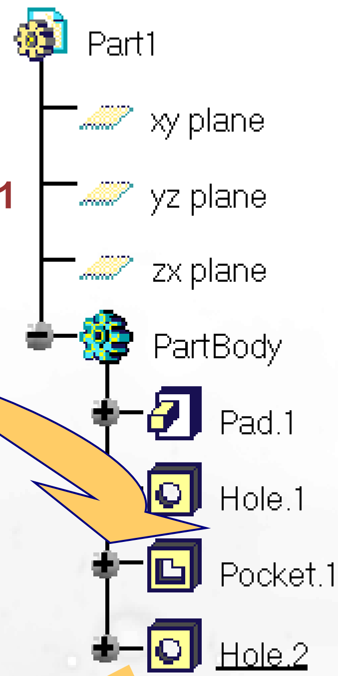
2 Now repeat the process from the same side to the hole on the left side of the part

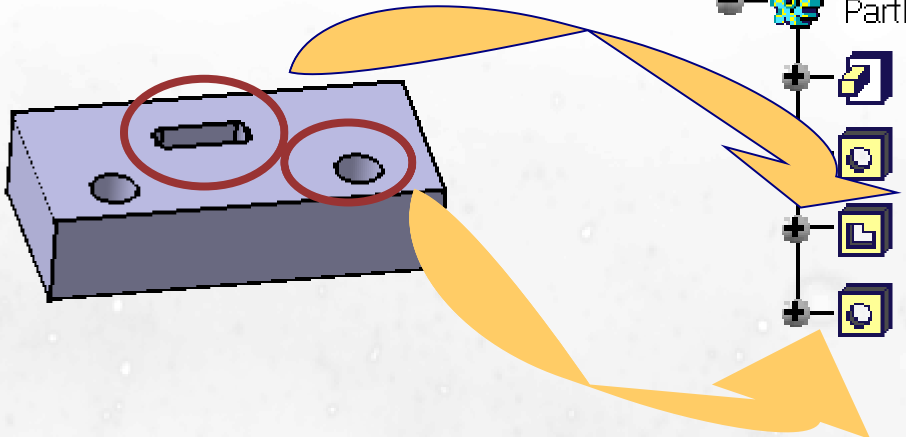
Note: The first dimension created was not a reference dimension. No Parenthesis were on the value. The second dimension was a reference dimension because the hole is located with the sketch for the hole from the same or the right side edge



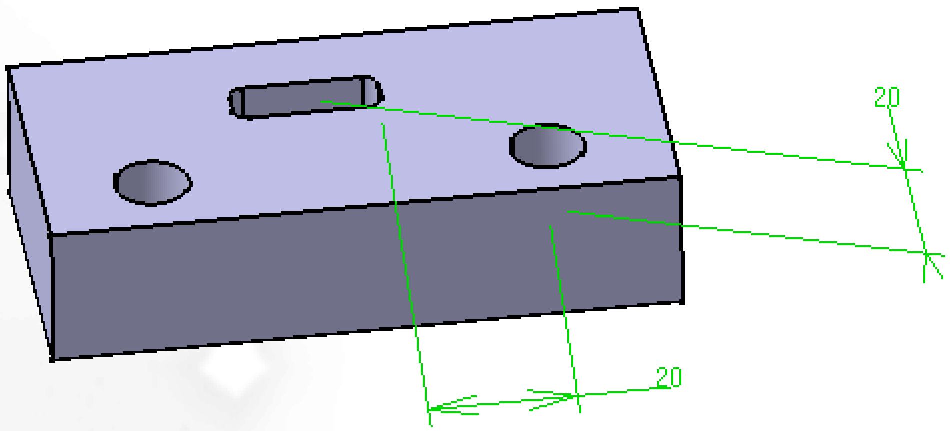
Using 3D Constraints

1 We want to drive the location of Pocket.1 from Hole.2 created after it in the tree

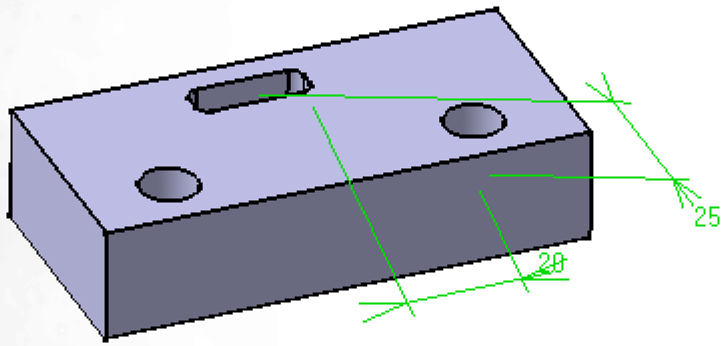




2 Create the two constraints shown below from the center line of Hole.2 to the edges of the Pocket.1



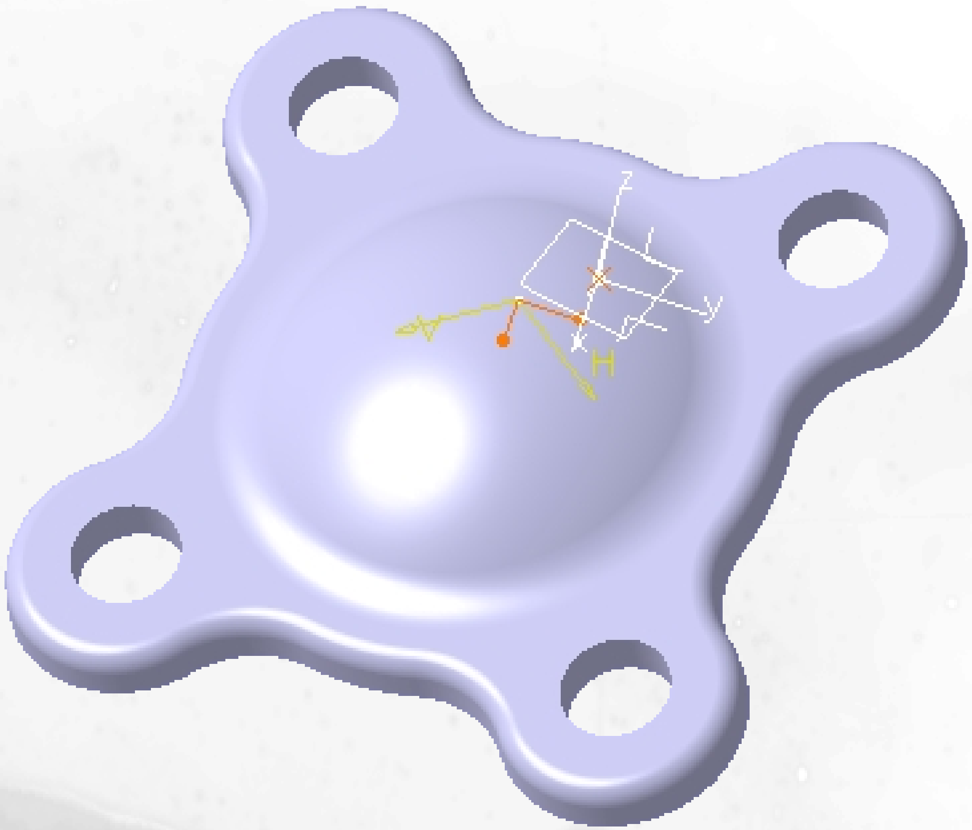
3 Modify the constraint indicated in red to 25mm and the Pocket.1 is now driven from the Hole.2 location

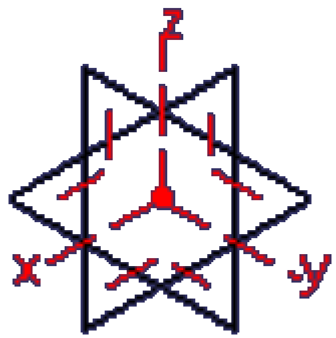


Note: This capability will allow you to drive location of features in the tree from features created after them without having to do re-location of features in the tree.

Local Axis

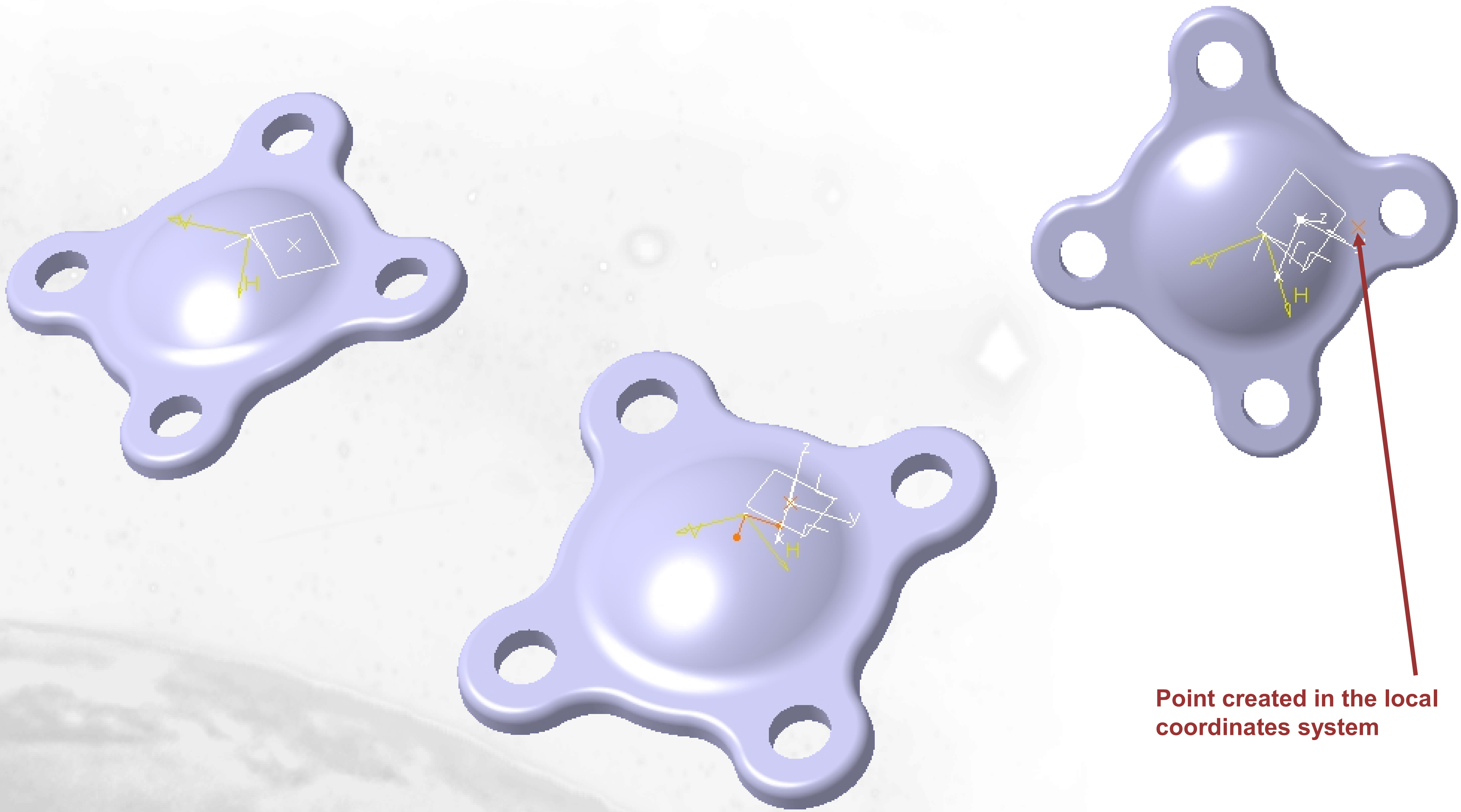
You will learn how to create a local axis in order to define local coordinates





What is a Local Axis ?

It is possible to create a local axis in order to define local coordinates. For example, it is, sometime, easier to build a point by coordinates in a local axis rather than creating it in the absolute coordinates system

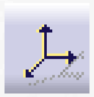


Point created in the local coordinates system

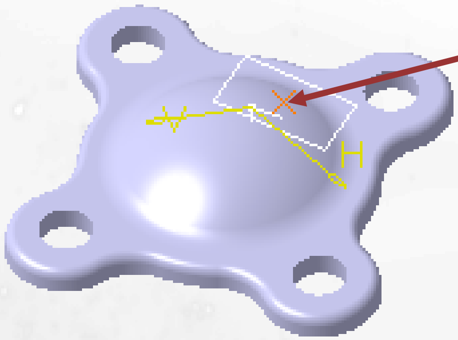
Local Axis : Creation

It is possible to create a local axis in order to define local coordinates. For example, it is, sometime, easier to build a point by coordinates in a local axis rather than creating it in the absolute coordinates system

1 Select the Axis System icon

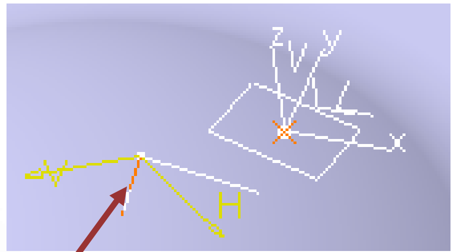


2 Select the local axis origin point



(2)

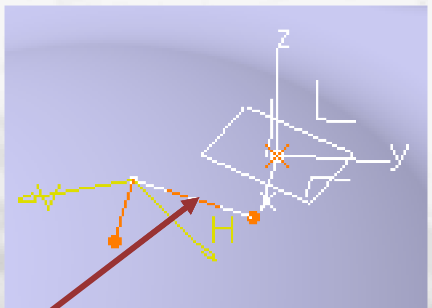
3 Select the OX direction



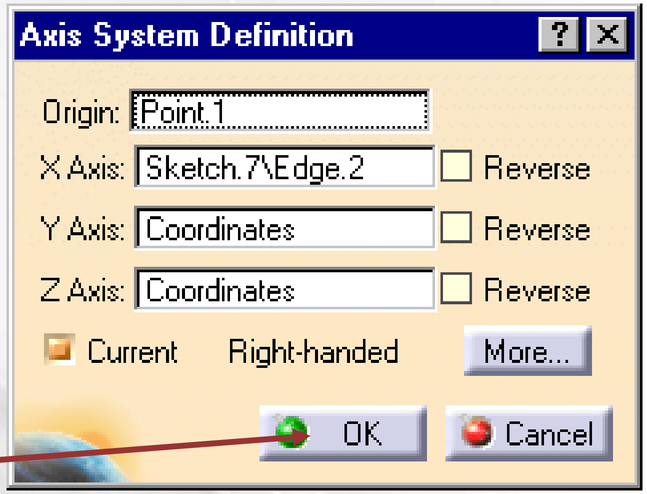
(3)

5 Select OK in the dialog box

4 Select the OY direction

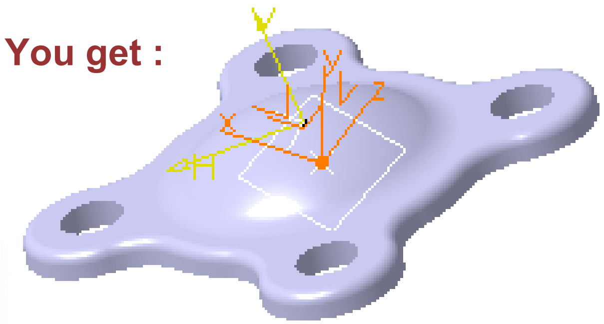


(4)



(5)

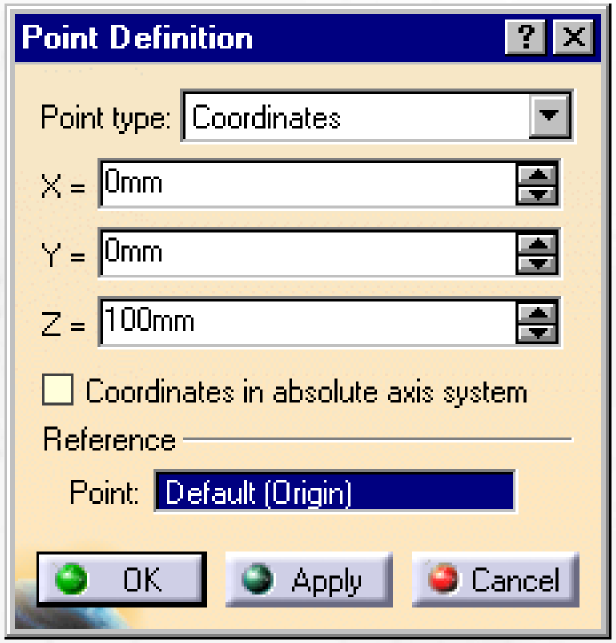
You get :



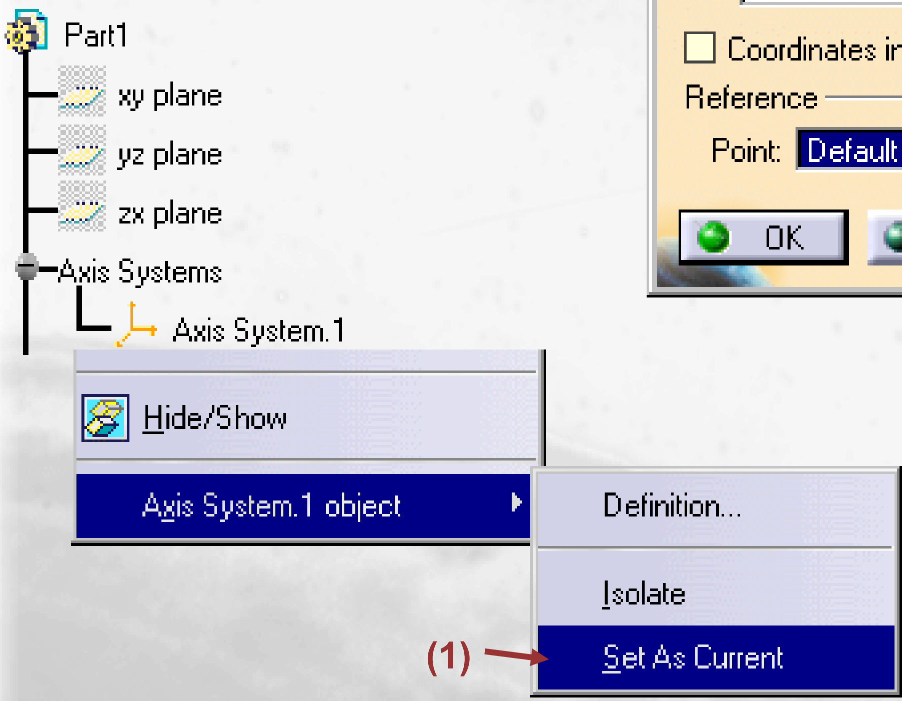
Local Axis : Use

It is possible to create a local axis in order to define local coordinates. For example, it is, sometime, easier to build a point by coordinates in a local axis rather than creating it in the absolute coordinates system

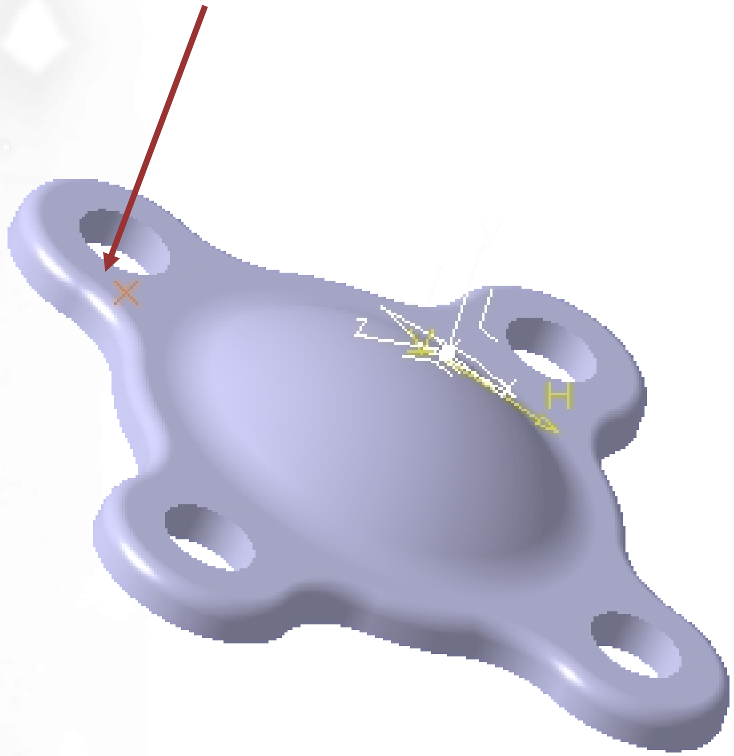
2 Using the Point function (Coordinates options), create a point with X=0, Y=0 and Z=100



1 Set the axis system As the Current one with the contextual menu



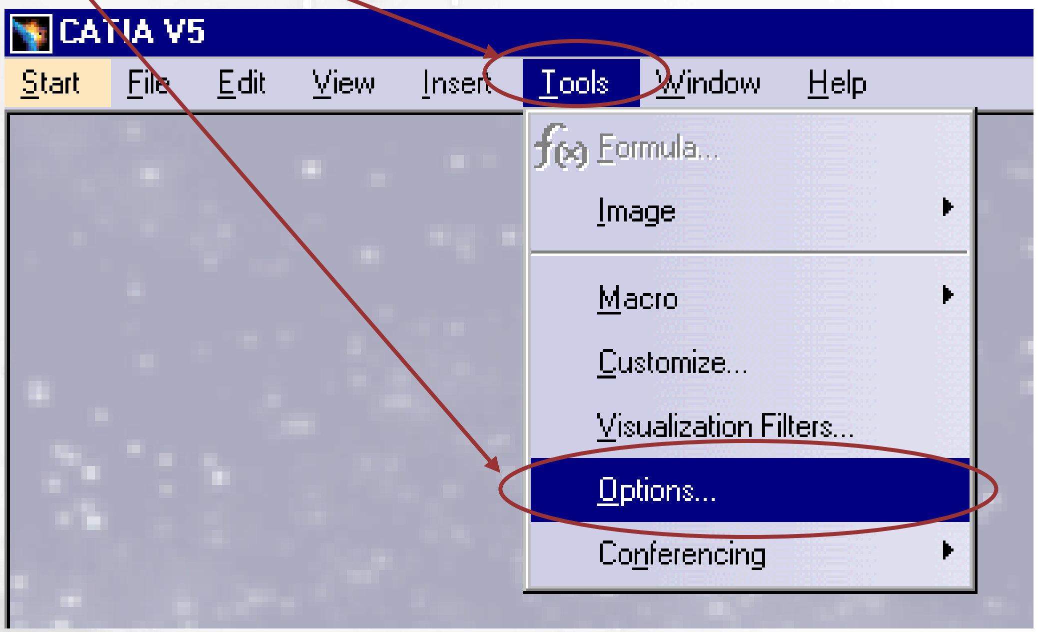
You get :



Customizing Local Axis (1/3)

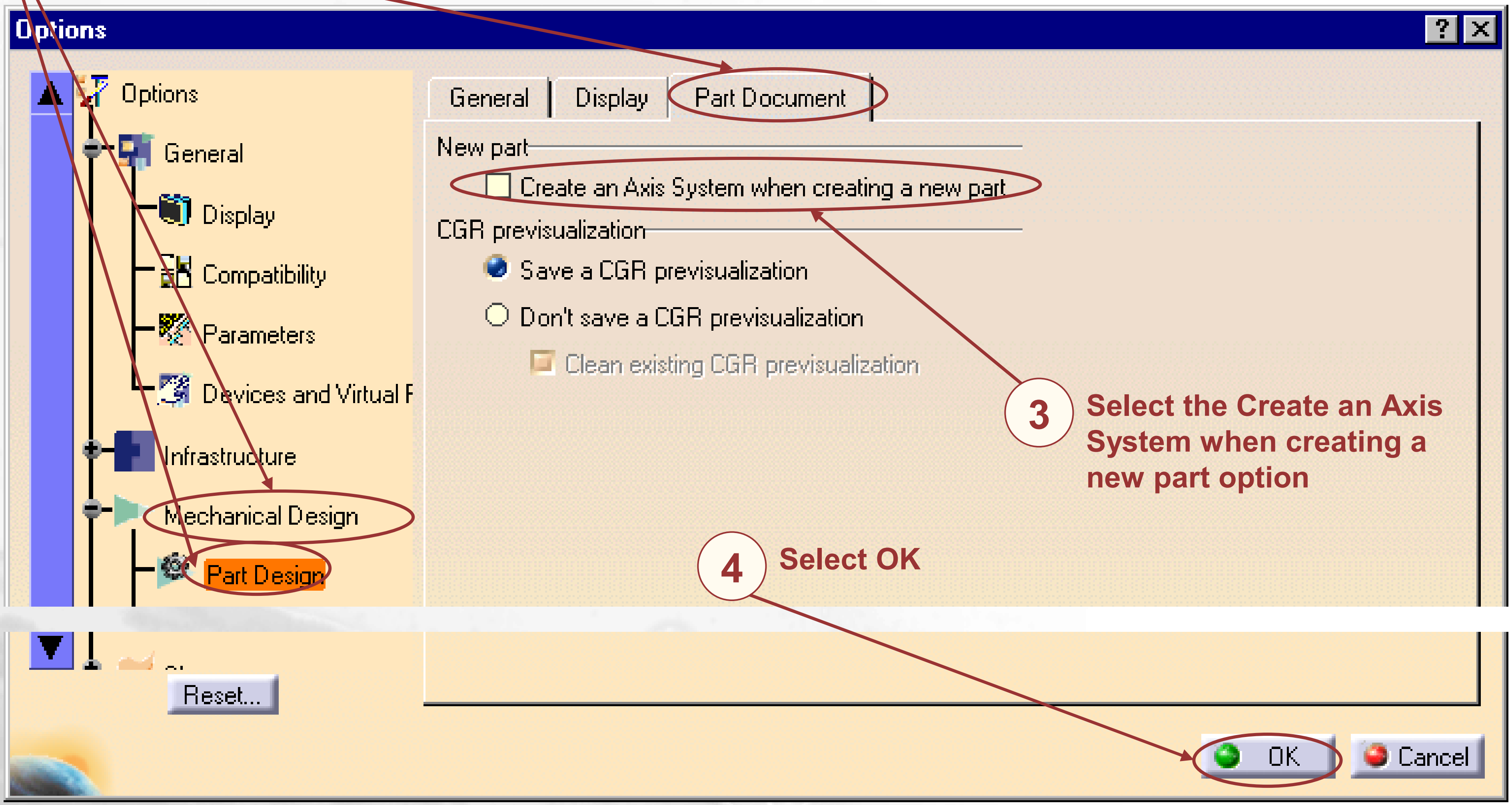
Check Create an Axis System when creating a new part if you wish to create a three axis system which origin point is defined by the intersection of the default planes that is plane XY, plane YZ and plane ZX

1 Select Tools -> Options



Customizing Local Axis (2/3)

2 In the Options dialog box, select Mechanical Design -> Part Design then the Part Document tab

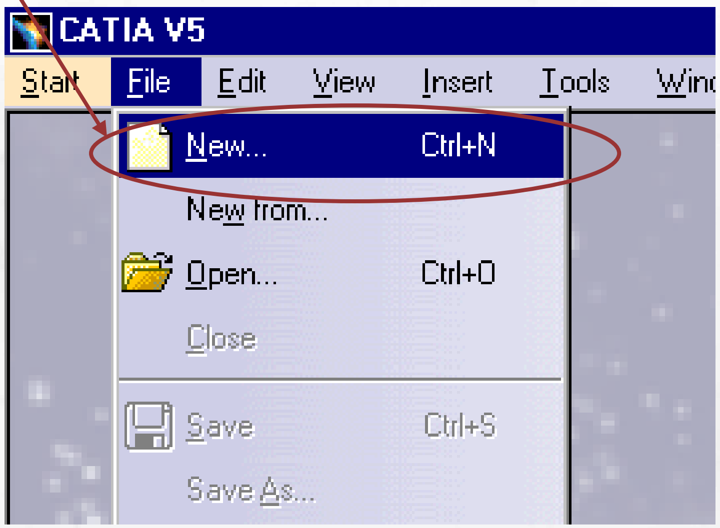


3 Select the Create an Axis System when creating a new part option

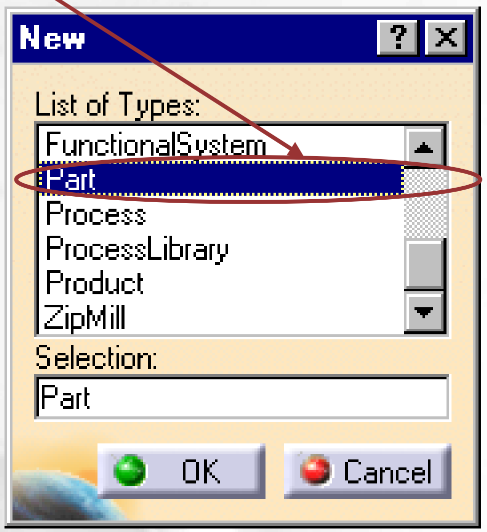
4 Select OK

Customizing Local Axis (3/3)

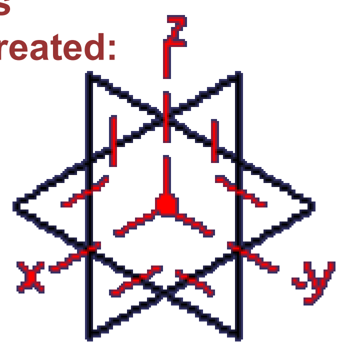
6 Select the File -> New command



5 Double click on Part in the dialog box

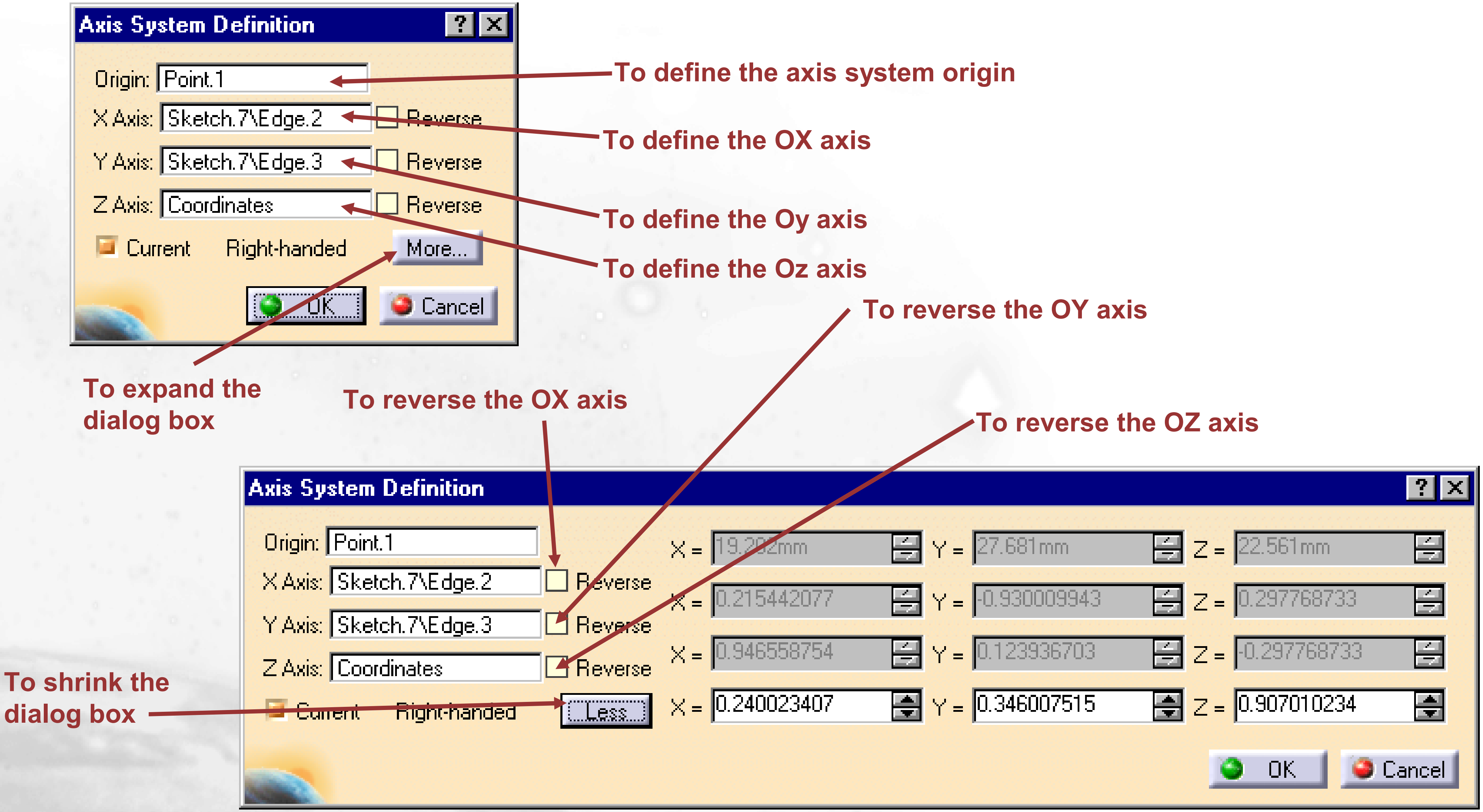


The local axis is automatically created:



Additional Information

Local Axis dialog box



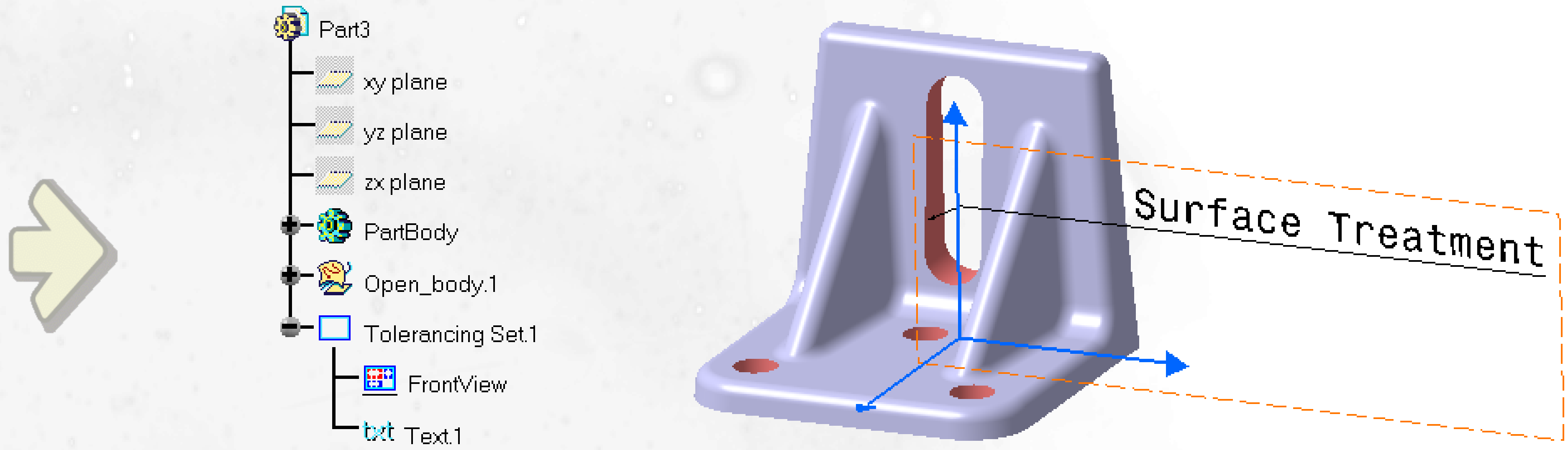
Annotation

You will learn how to attach a text to a part and how to add hyperlinks to your document and then use them to jump to a variety of locations

- **Text with Leader**
- **Flag Note with Leader**

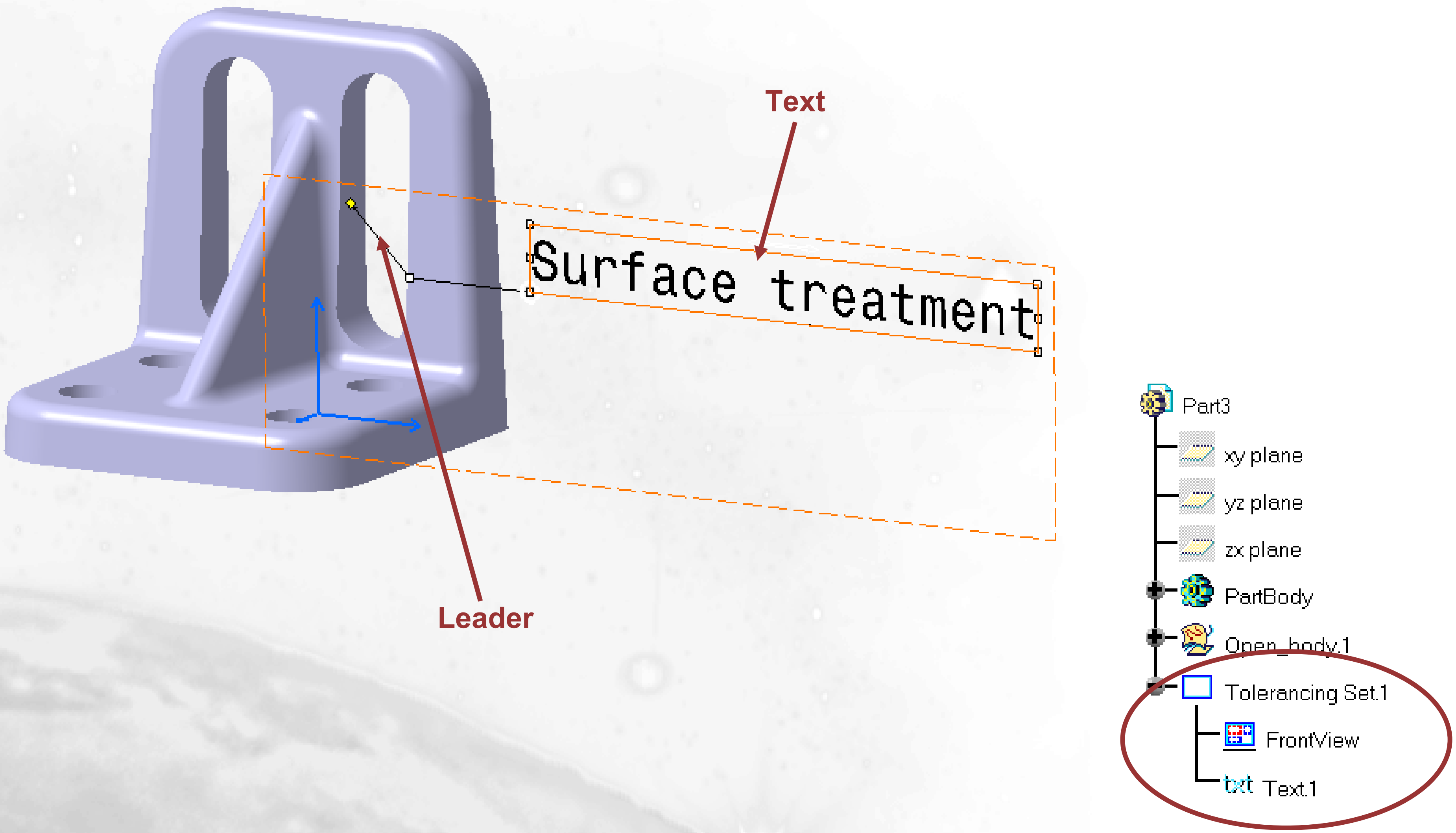
Text with Leader

You will learn how to attach a text to a part

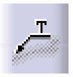


What are Texts with Leader?

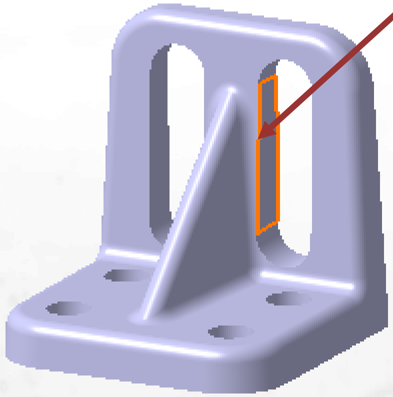
A text with leader can be attached to a part in order to give information for example on surface treatment. This text can appears on the drawing



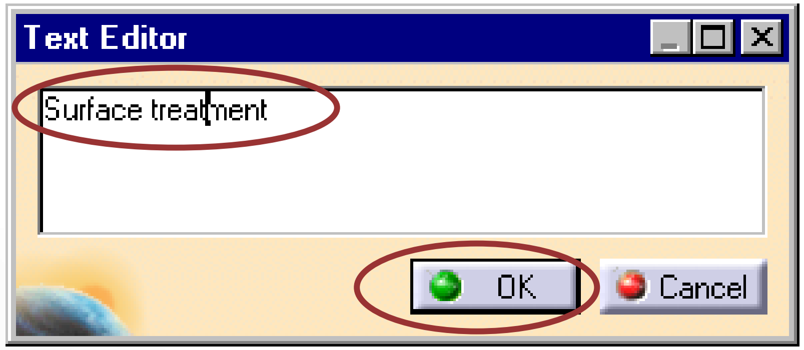
Texts with Leader

1 Select the Text with Leader icon 

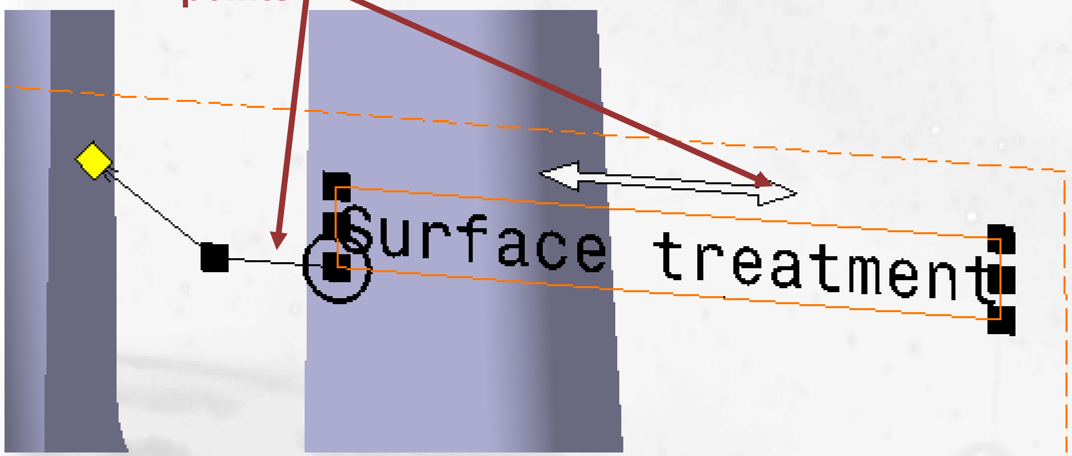
2 Select the position of the leader on the part



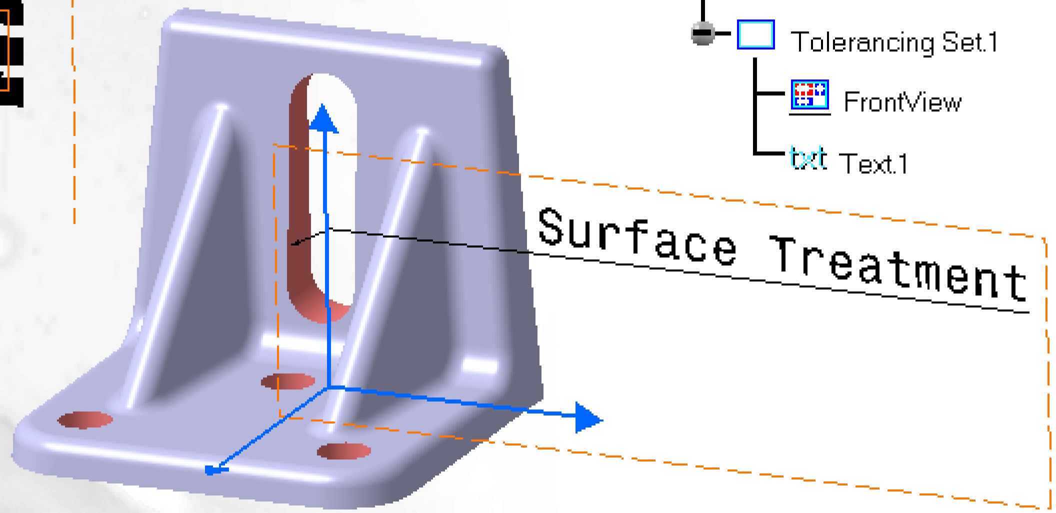
3 Enter the text in the dialog box then select OK

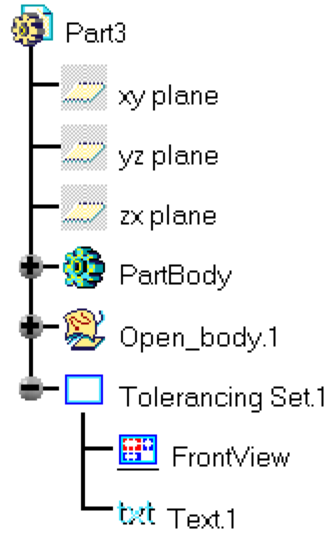


4 Place the text and the leader by dragging the arrow or the square points



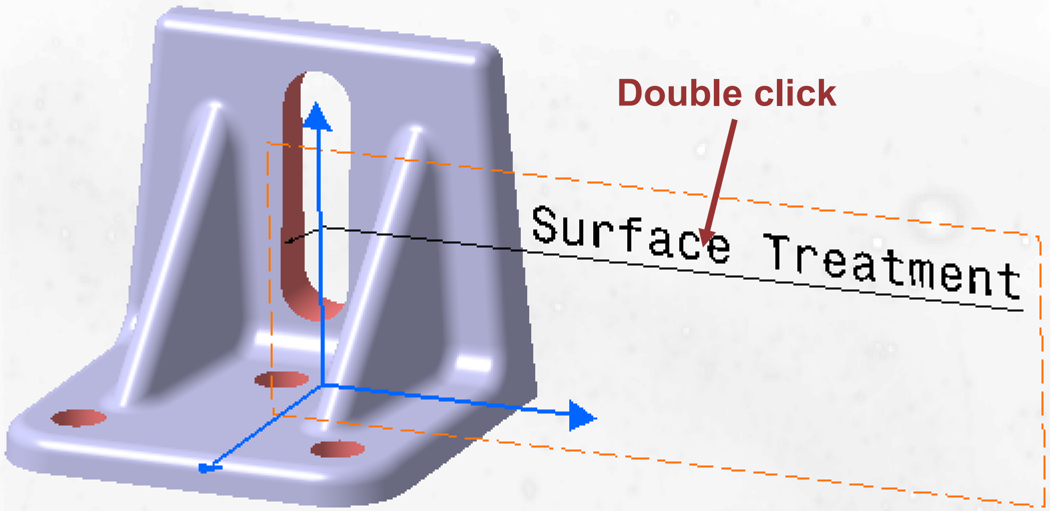
You get:

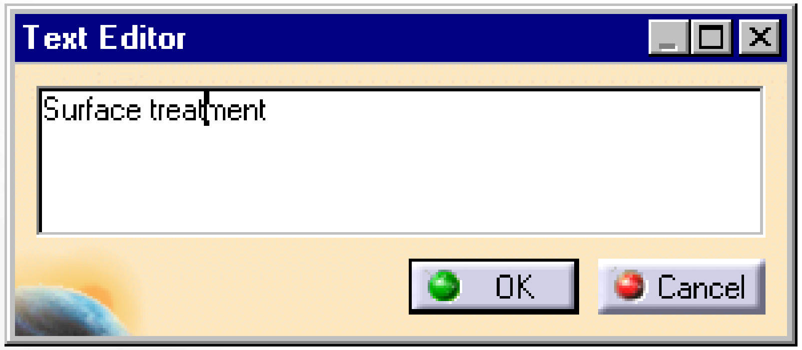




Additional Information

To Modify the text of a text with leader, double click on the text, you will recover the dialog box where you can change the text

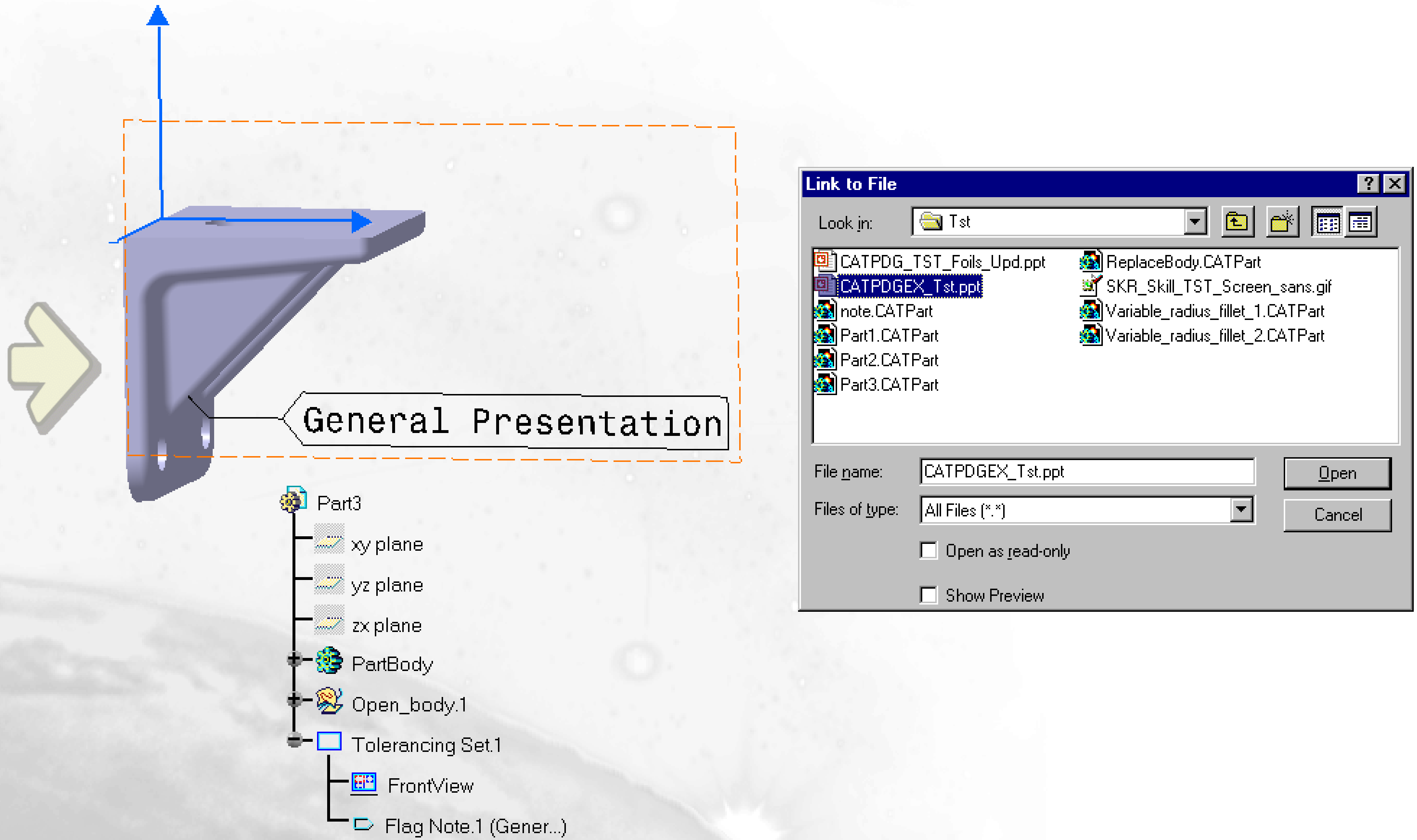




Using the Properties command from the contextual menu will give you access to text, font and graphic modifications

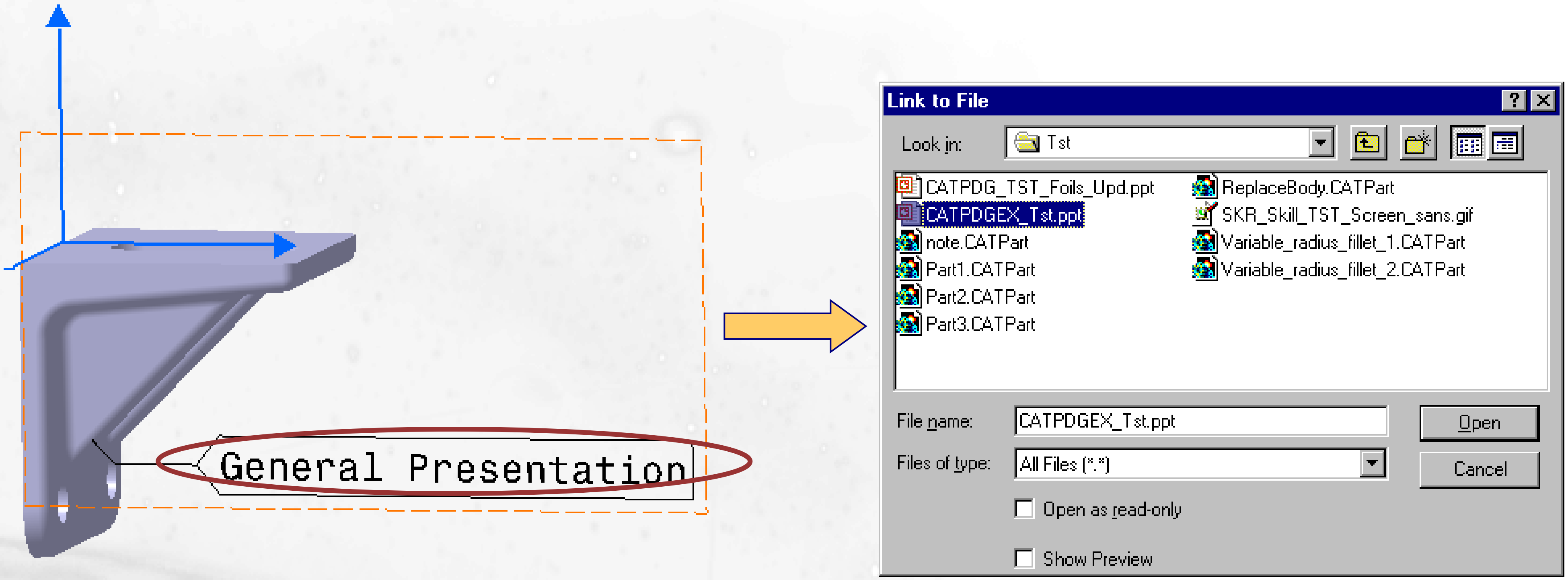
Flag Note with Leader

You will learn how to add hyperlinks to your document and then use them to jump to a variety of locations

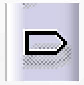


What are Flag Notes with Leader?

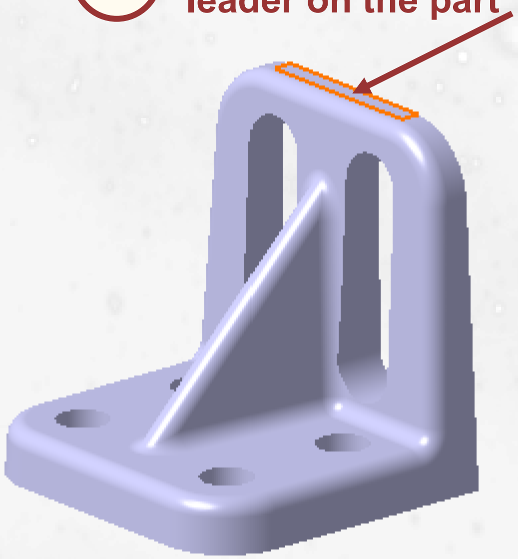
A flag note with leader can be attached to a part in order to give information for example on surface treatment. This flag is an hyperlink that can start any documents such as a presentation, a Microsoft Excel spreadsheet or a HTML page on the intranet



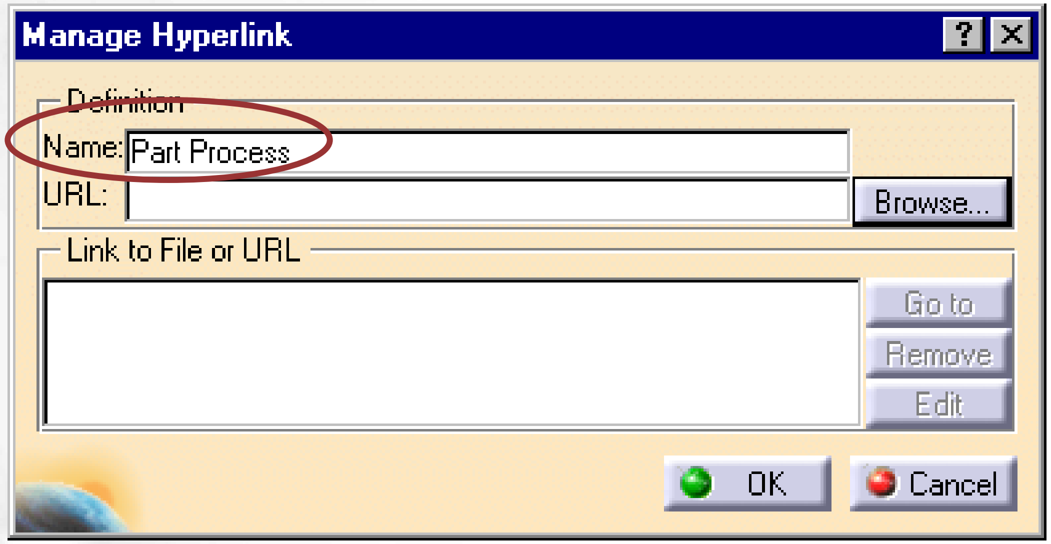
Flag Notes with Leader (1/2)

1 Select the Flag Note with Leader icon 

2 Select the position of the leader on the part

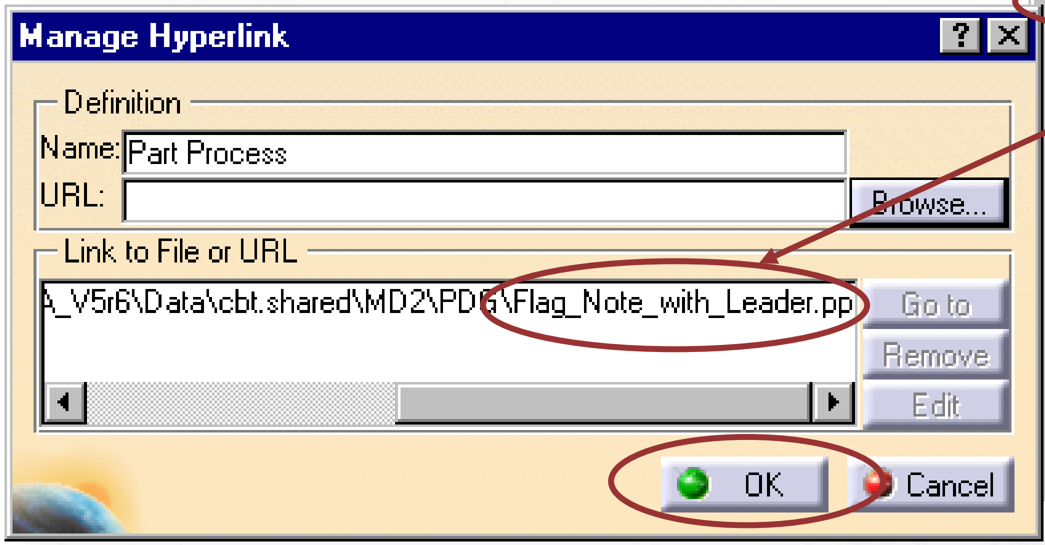


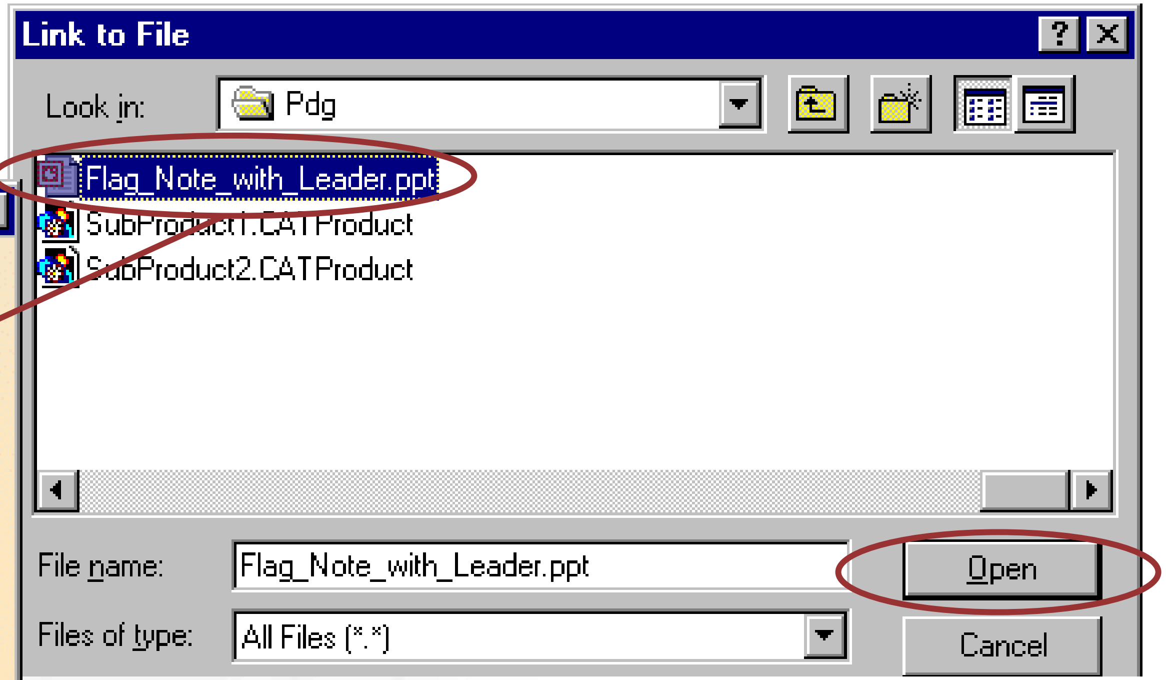
3 Enter Part Process in the Name field



Flag Notes with Leader (2/2)

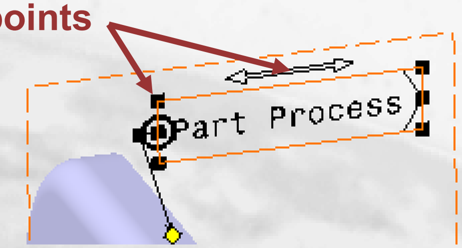
4 Select the Browse button then select the file to which you want to be linked then select OK

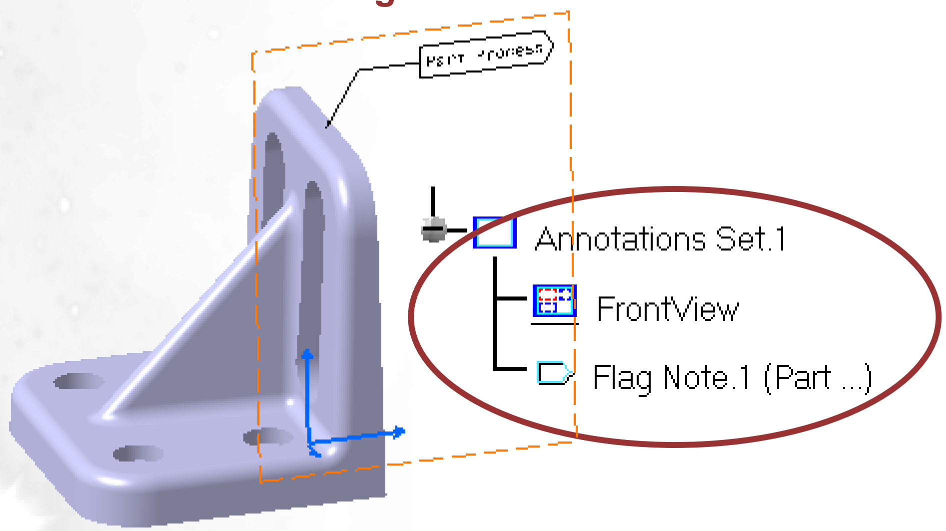




You get:

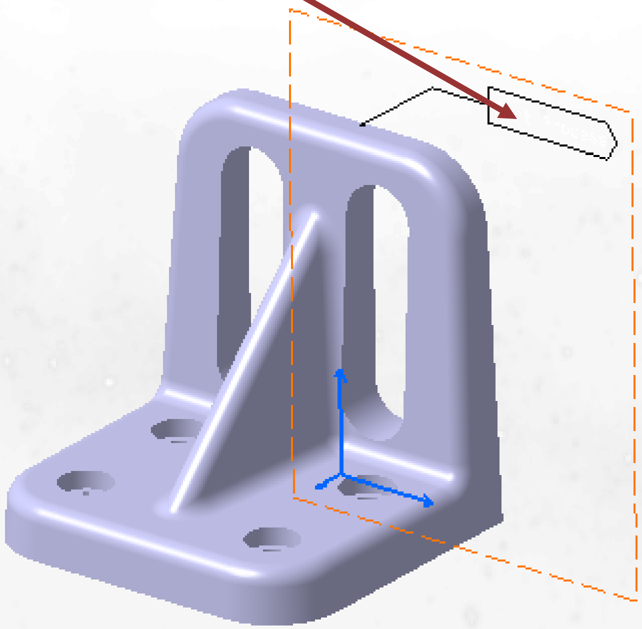
5 Place the text and the leader by dragging the arrow or the square points



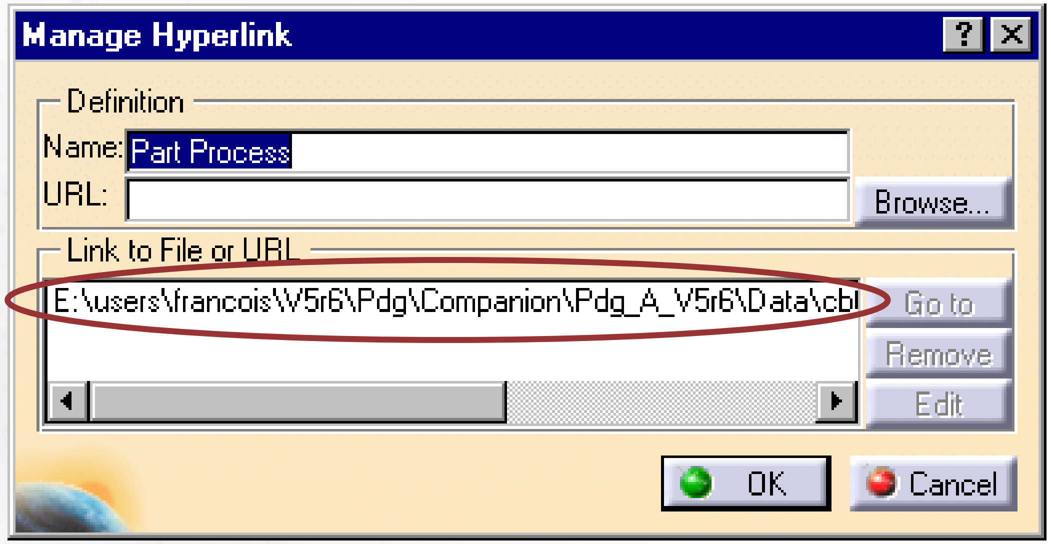


Using Flag Notes with Leader

1 Double click on the flag



2 Select the Link in the dialog box

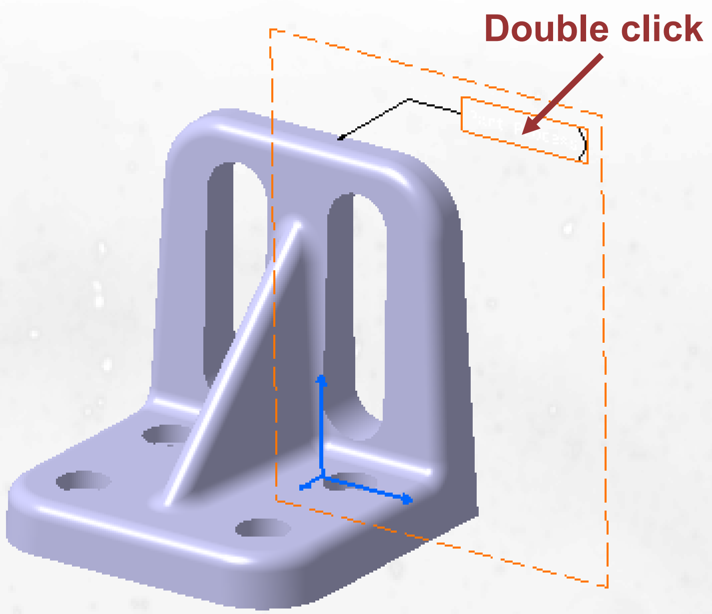


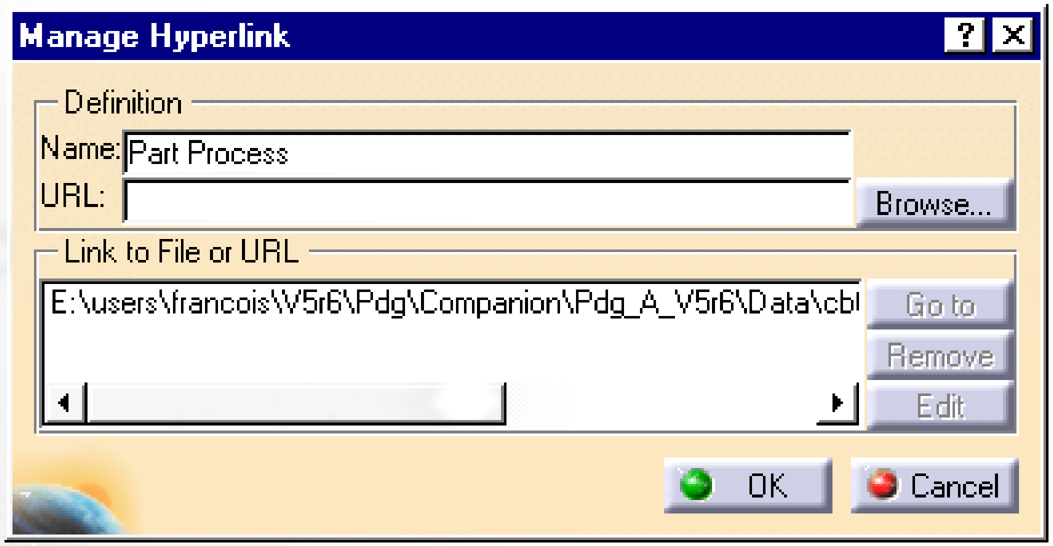
3 Select the Go to button in the dialog box

The linked file is now started

Additional Information

To Modify the text of a flag note with leader, double click on the text, you will recover the dialog box where you can change the text





Using the Properties command from the contextual menu will give you access to text, font and graphic modifications

You can have several files linked to a flag note

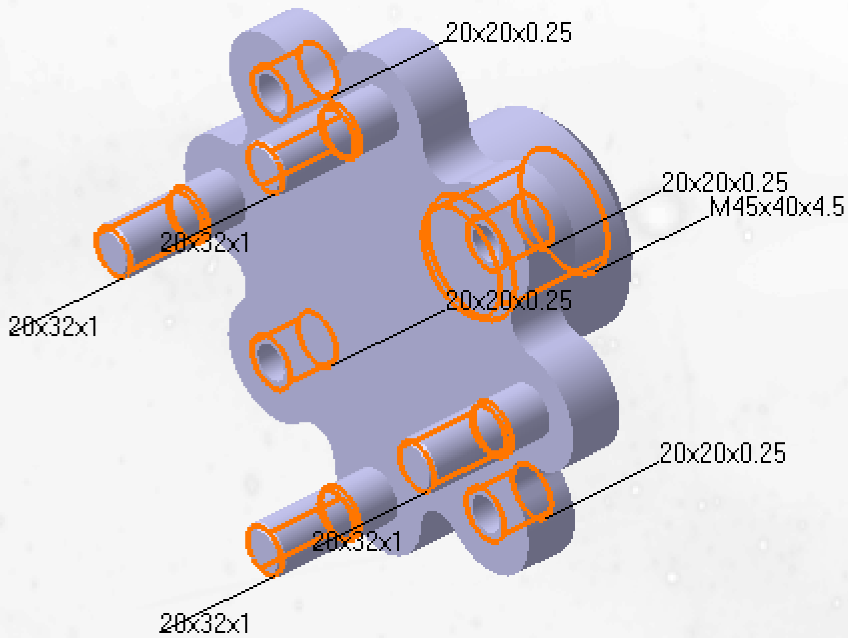
Analysis

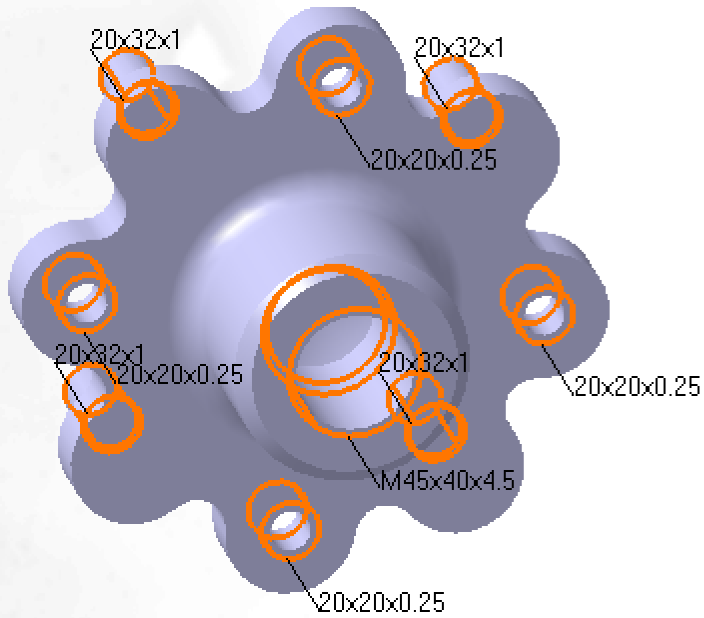
You will learn how to analyze part in order to display the threads and tap, and to check if a part can be removed from mold in accordance with its draft angles

- Analysing Threads and Taps
- Draft Analysis

Analysing Threads and Taps

You will learn how to display and filter out information about threads and taps contained in a part

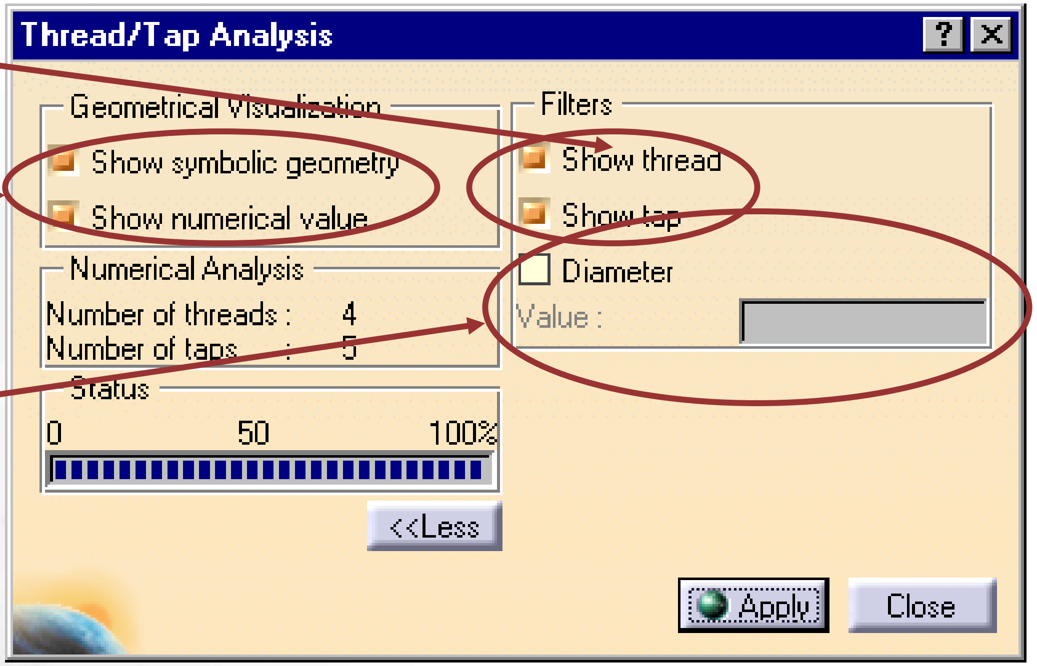


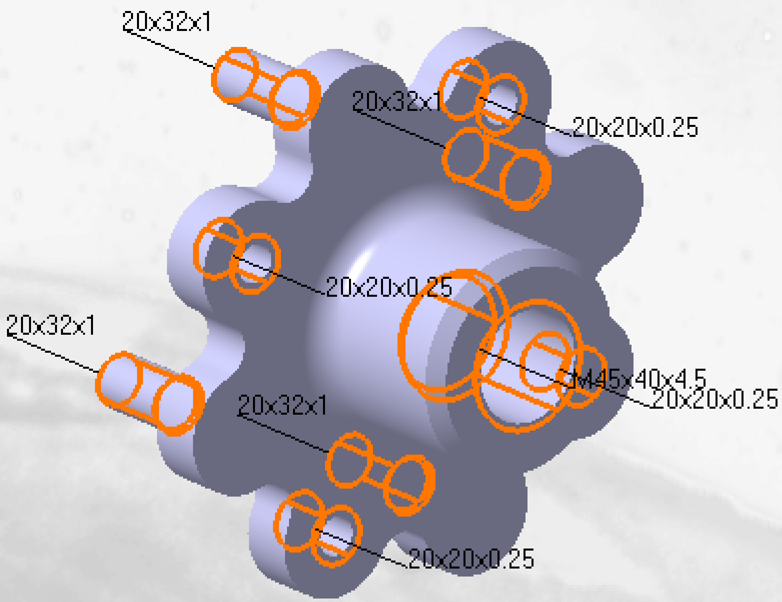


What is the Threads and Tap Analysis ?

When a part has been created with threads and taps, CATIA does not physically displays these features. There is a way to quickly know all the information about threads and taps by using the Thread and Taps Analysis icon

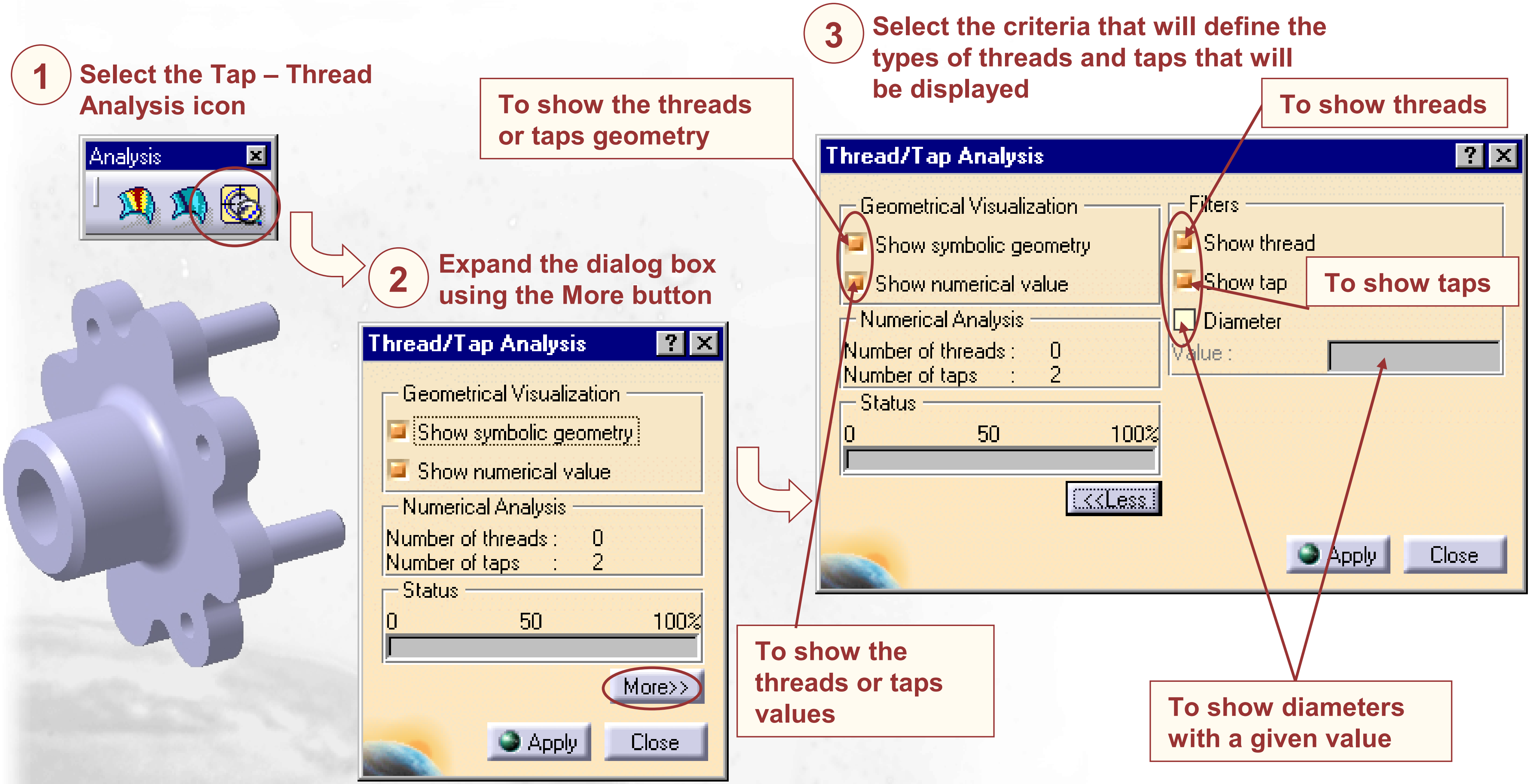
- **You can display the threads or the taps or both of them**
- **You can display the threads and taps numerical values**
- **You can display threads or/and taps of a given diameter value**





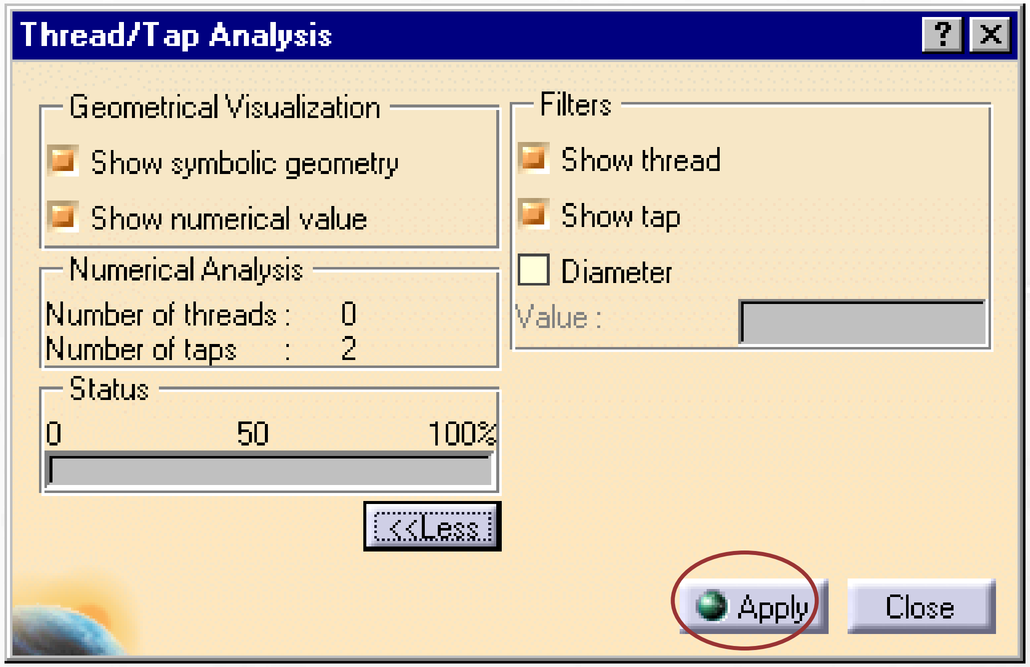
Analysing Threads and Taps (1/2)

You can display and filter out information about threads and taps contained in a Part

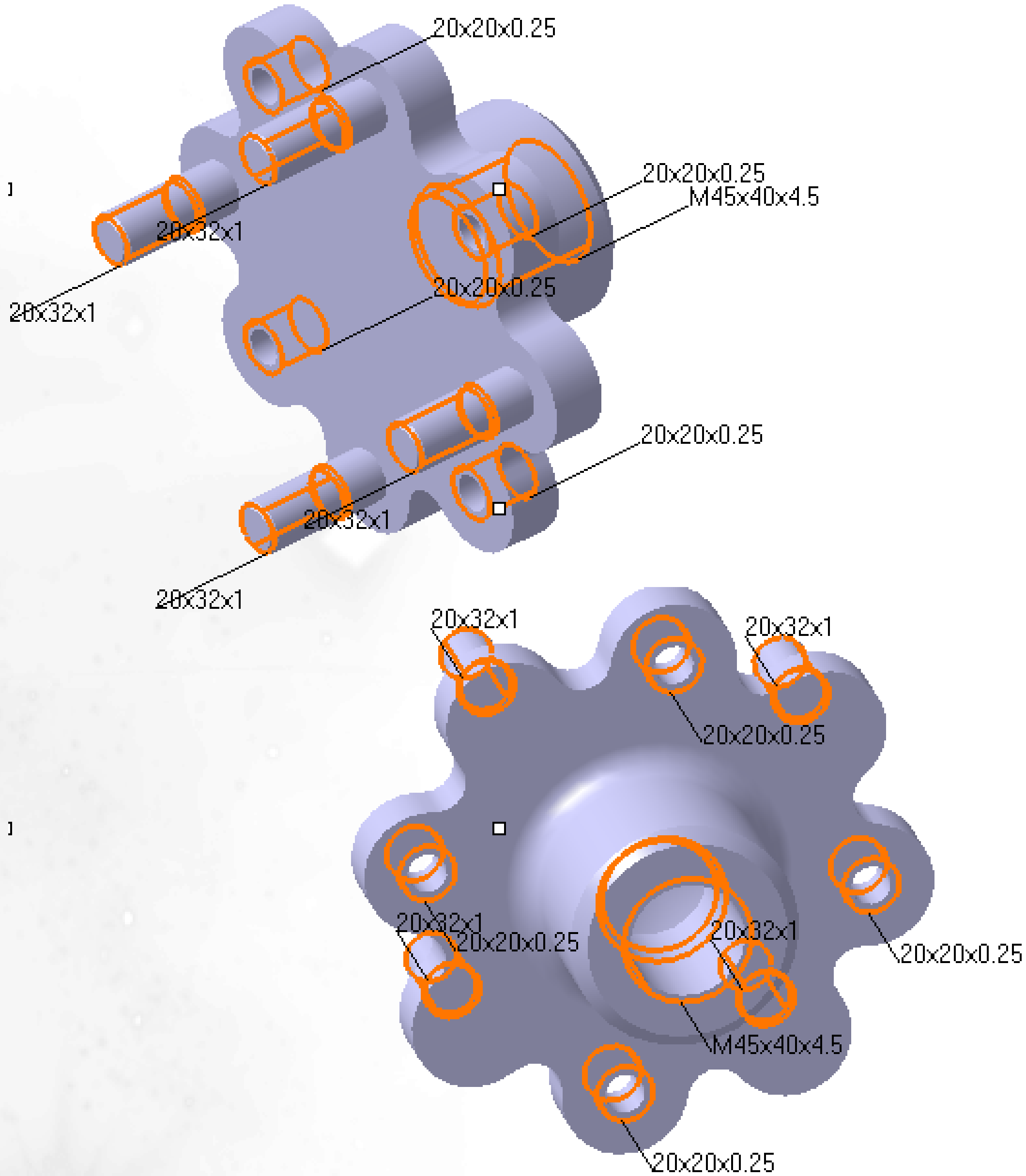


Analysing Threads and Taps (2/2)

4 **Select Apply in the dialog box**

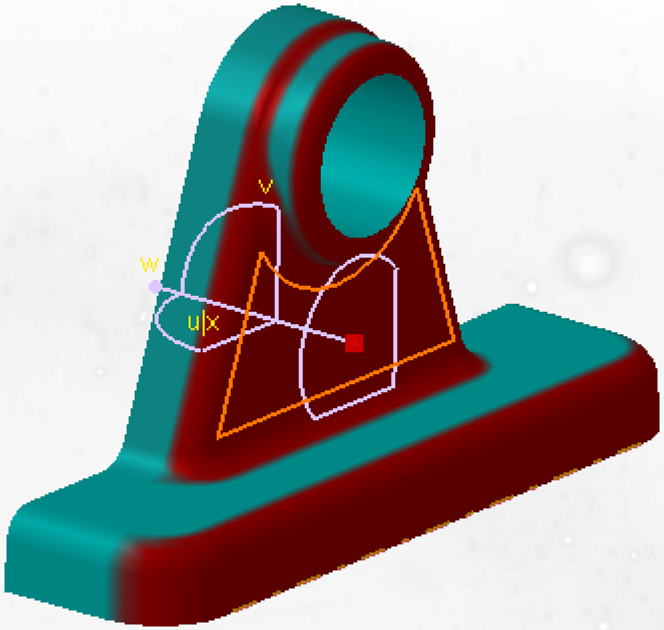


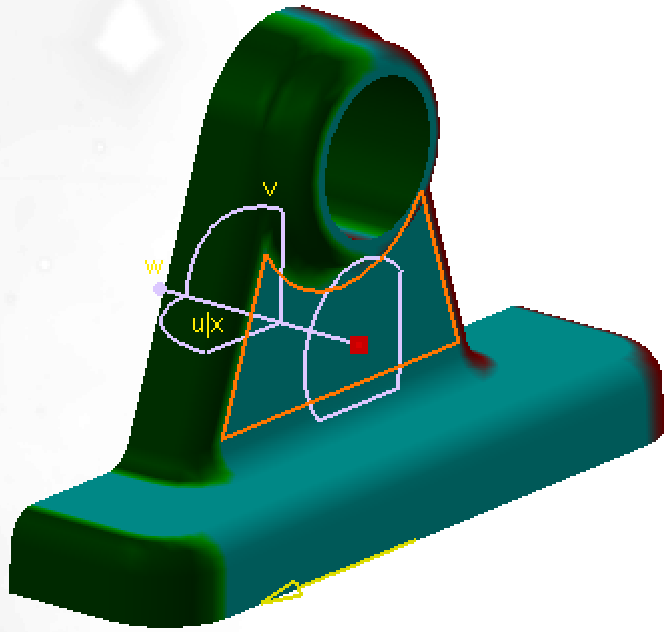
You get:



Draft Analysis

You will learn how to analyze the draft angle on the surface of a part

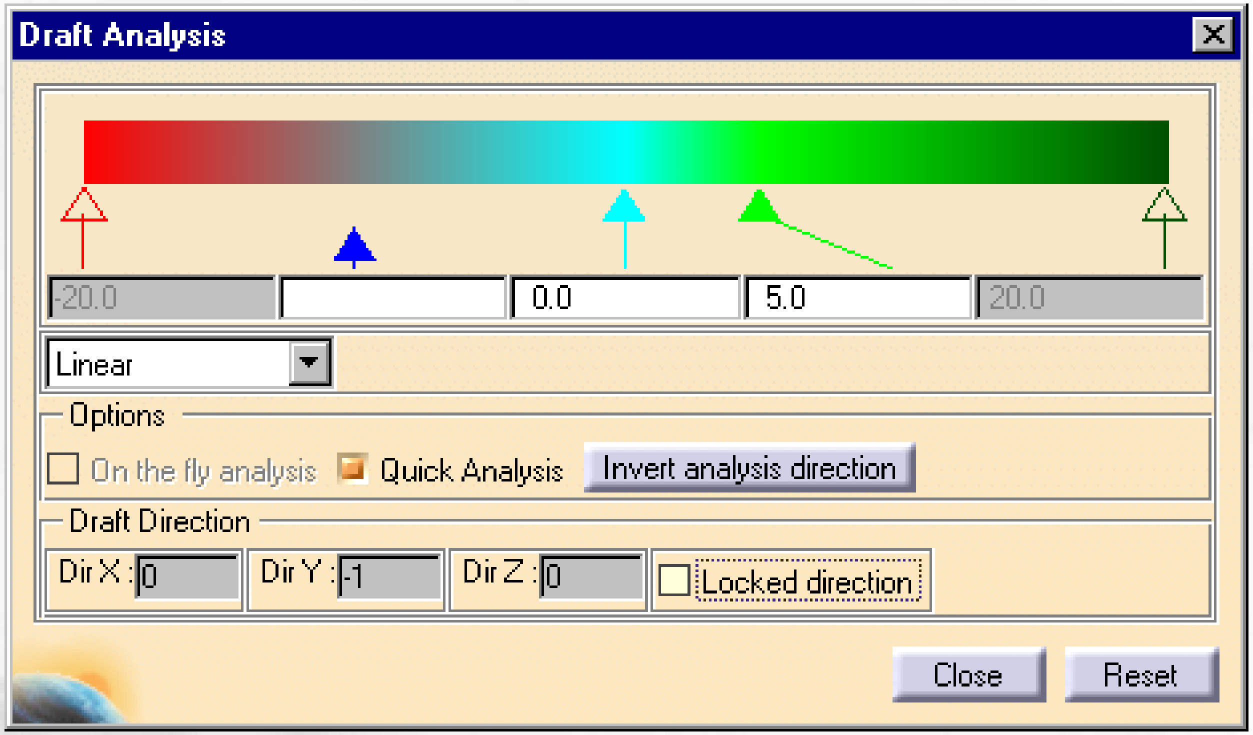


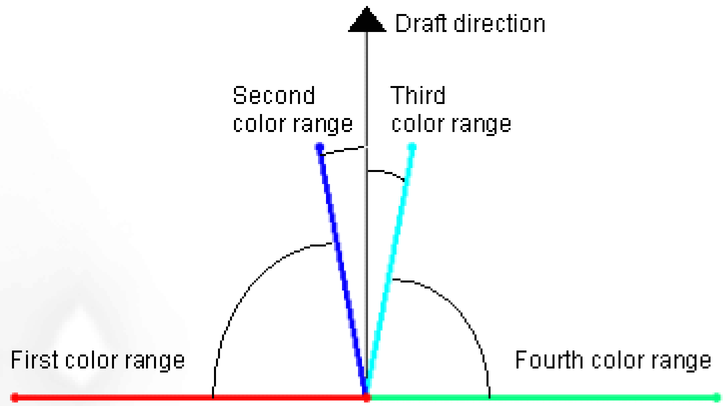


What is the Draft Analysis ?

The Draft Analysis command lets you analyze the draft angle on the surface of a part. You will be able to detect if the part you drafted will be easily removed from the associated mold

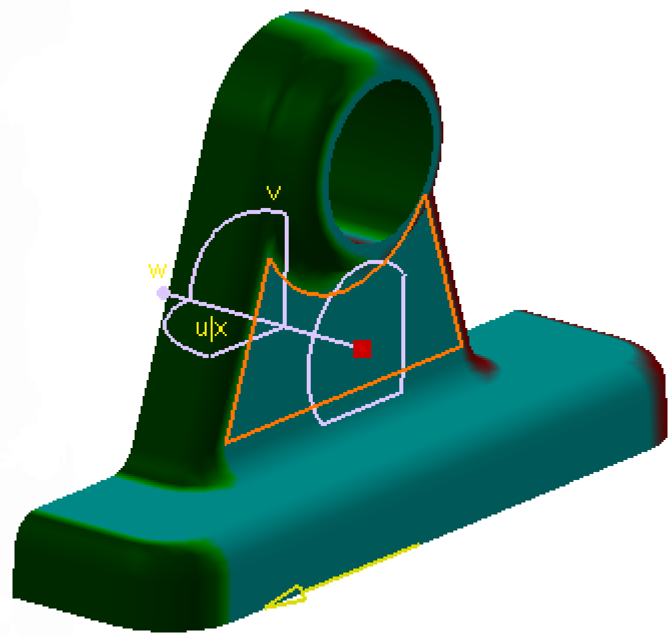
This type of analysis is performed based on color ranges identifying zones on the analyzed element where the deviation from the draft direction, represented by the normal to the surface at a given point, corresponds to specified values





The cursor manipulation for colors is limited to -20 and 20 but the analysis is performed between -90 and 90 degrees.

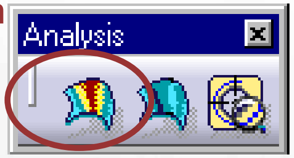
To get a result, the view mode must be turned to Material display



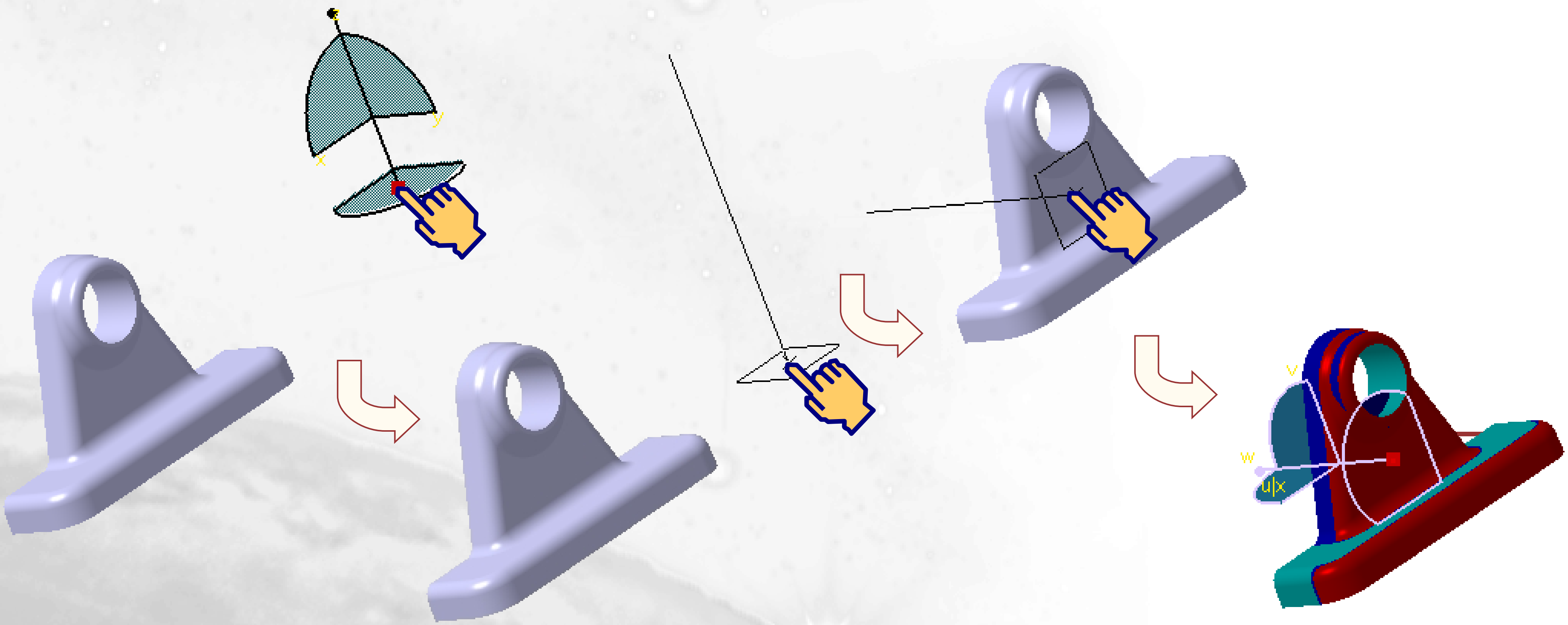
Draft Analysis (1/2)

The Draft Analysis command lets you analyze the draft angle on the surface of a part. You will be able to detect if the part you drafted will be easily removed from the associated mold

1 Select the Draft Analysis icon

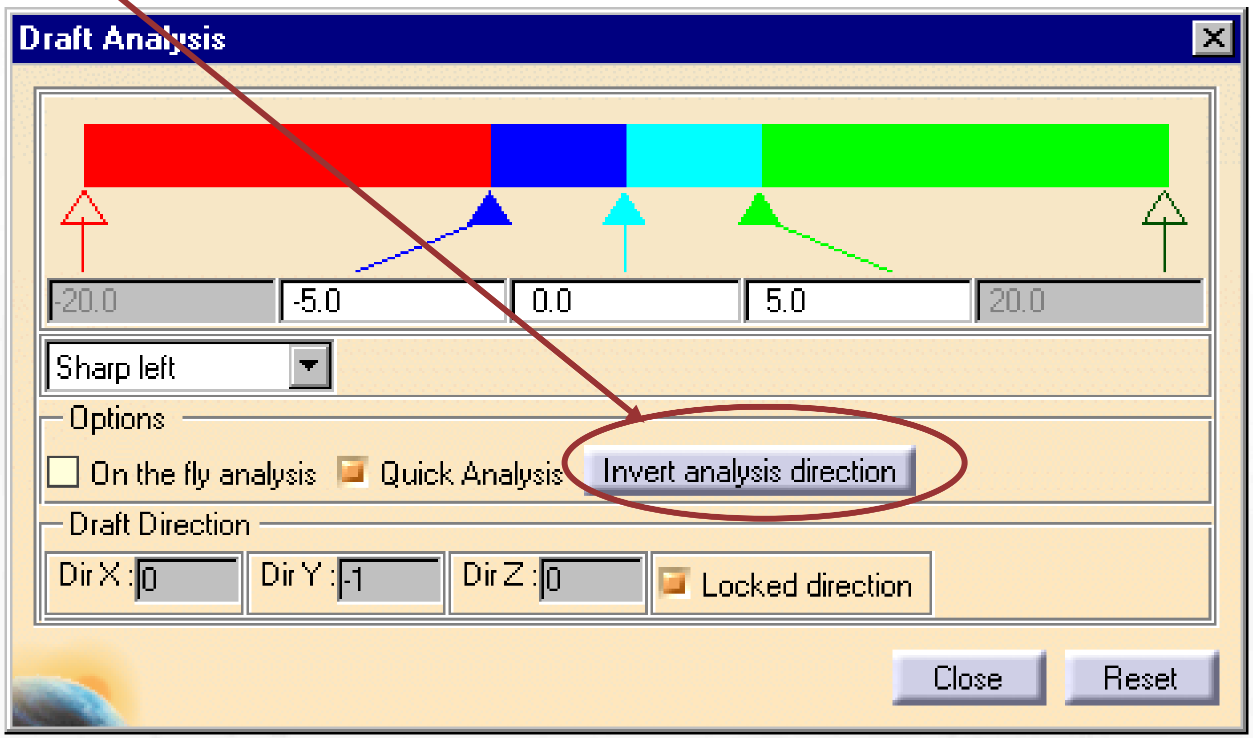


2 Drag the red point of the Compass and drag it onto a face perpendicular to the direction of extraction

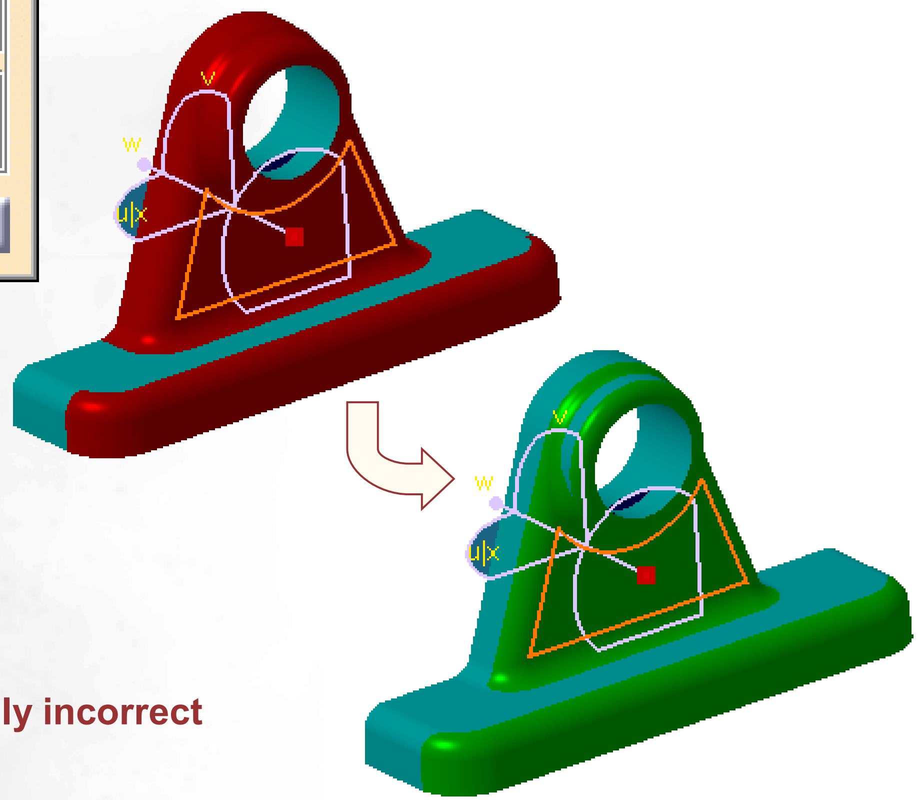


Draft Analysis (2/2)

2 Select the Invert analysis direction in the dialog box



You get:



Everything in green is correct

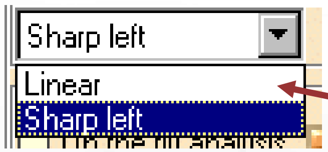
Everything in red is incorrect

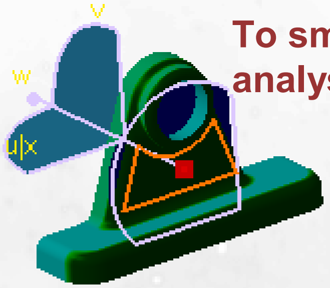
You have to take care of the light blue faces but it might correct

You have to take care of the dark blue faces because it is certainly incorrect

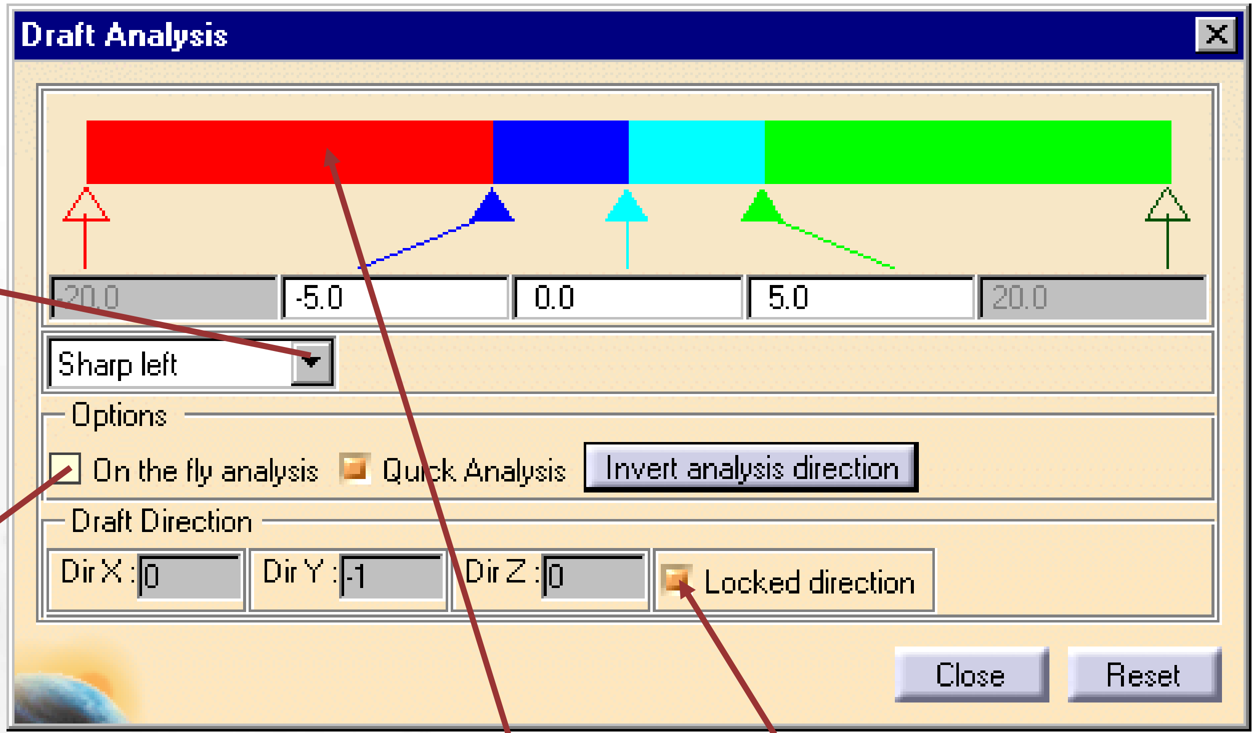
Additional Information

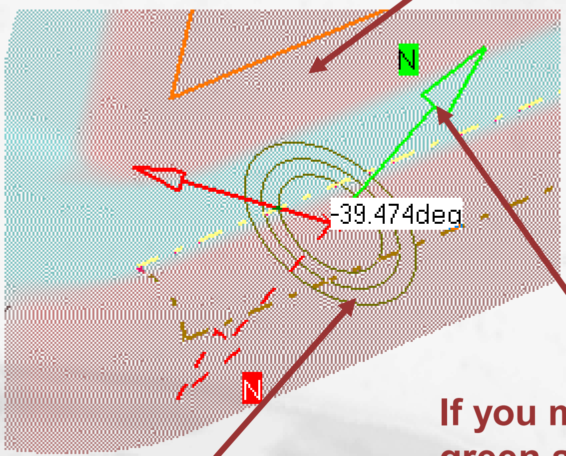
Draft Analysis dialog box





To smooth the analysis





You can customize these colors by double clicking

If you move the pointer over the green arrow (Normal), the inverted normal is displayed in dotted line

To fix a direction of extraction, uncheck the Locked direction option, and select a direction (a line, or a plane which normal is used), or use the compass manipulators, when available

Circles are displayed indicating the plane tangent to the surface at this point

Part Management

You will learn Part Management tools that you will need to design complex parts and integrate these parts into a Multi-model Environment

- Measure, Mean Dimensions, Scan, Parents-Children
- Cut, Paste, Isolate, Break
- Inserting and Managing Bodies
- Multi-Model Links
- Sketch Selection with Multi-Documents Links
- Scaling

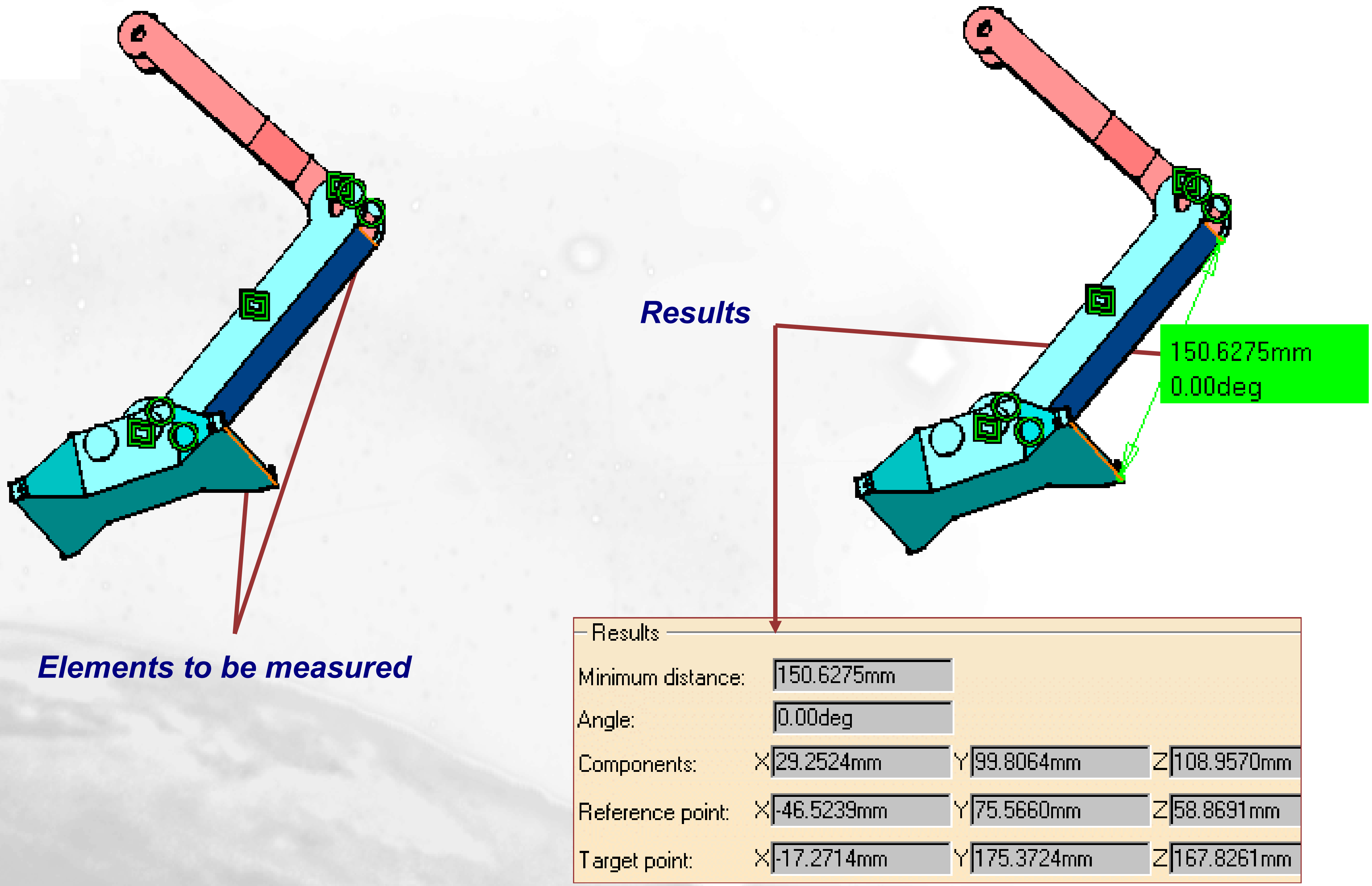
Measure, Mean Dimensions, Scan, Parents-Children

In this lesson, you will see how to measure angle and distance between geometrical entities, then how to replay the construction history of a part and isolate temporarily any feature to work locally, then to provide an accurate view of genealogical links between elements. We recommend you to use it before deleting elements

- **Measuring Elements**
- **Mean Dimensions**
- **Scanning a Part**
- **Parents-Children Relationship**

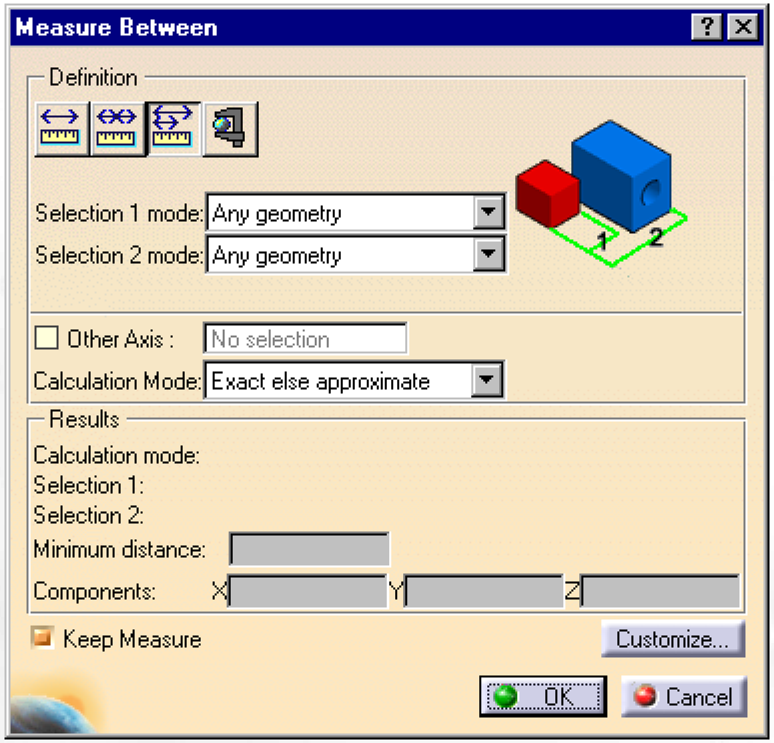
What is Measuring Elements ?

Measuring Elements means to get the angle and the distance between two geometric entities

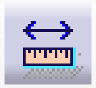


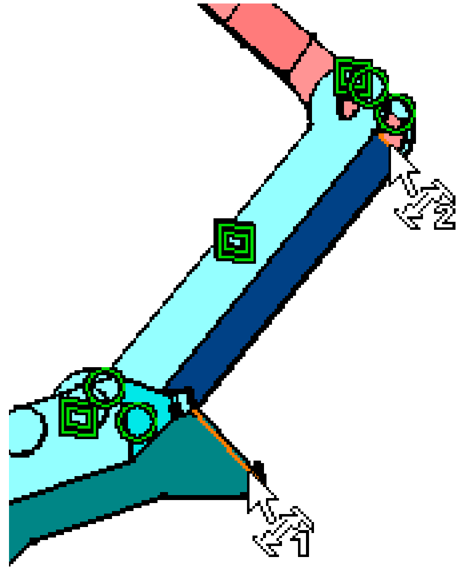
Measuring Elements

2 Set the desired type of Measurement

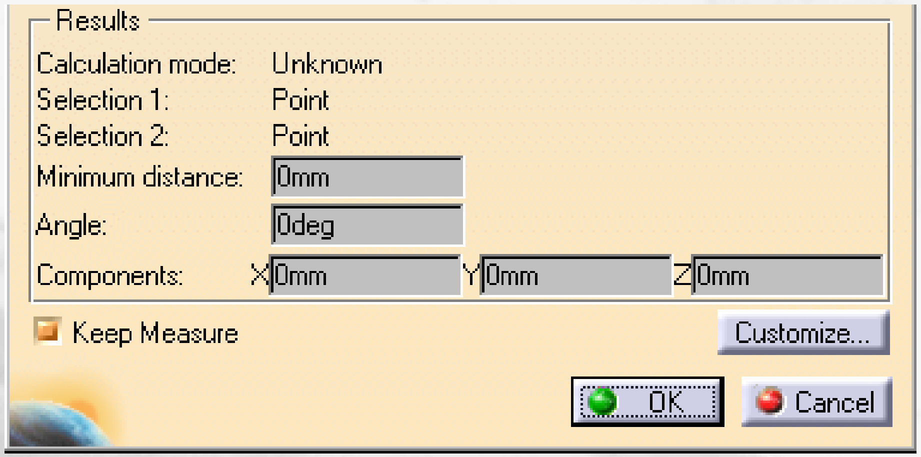


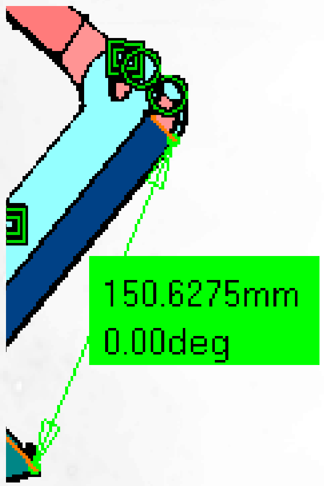
1 Select the Measure Between icon





3 Select your reference (1) and target (2) elements

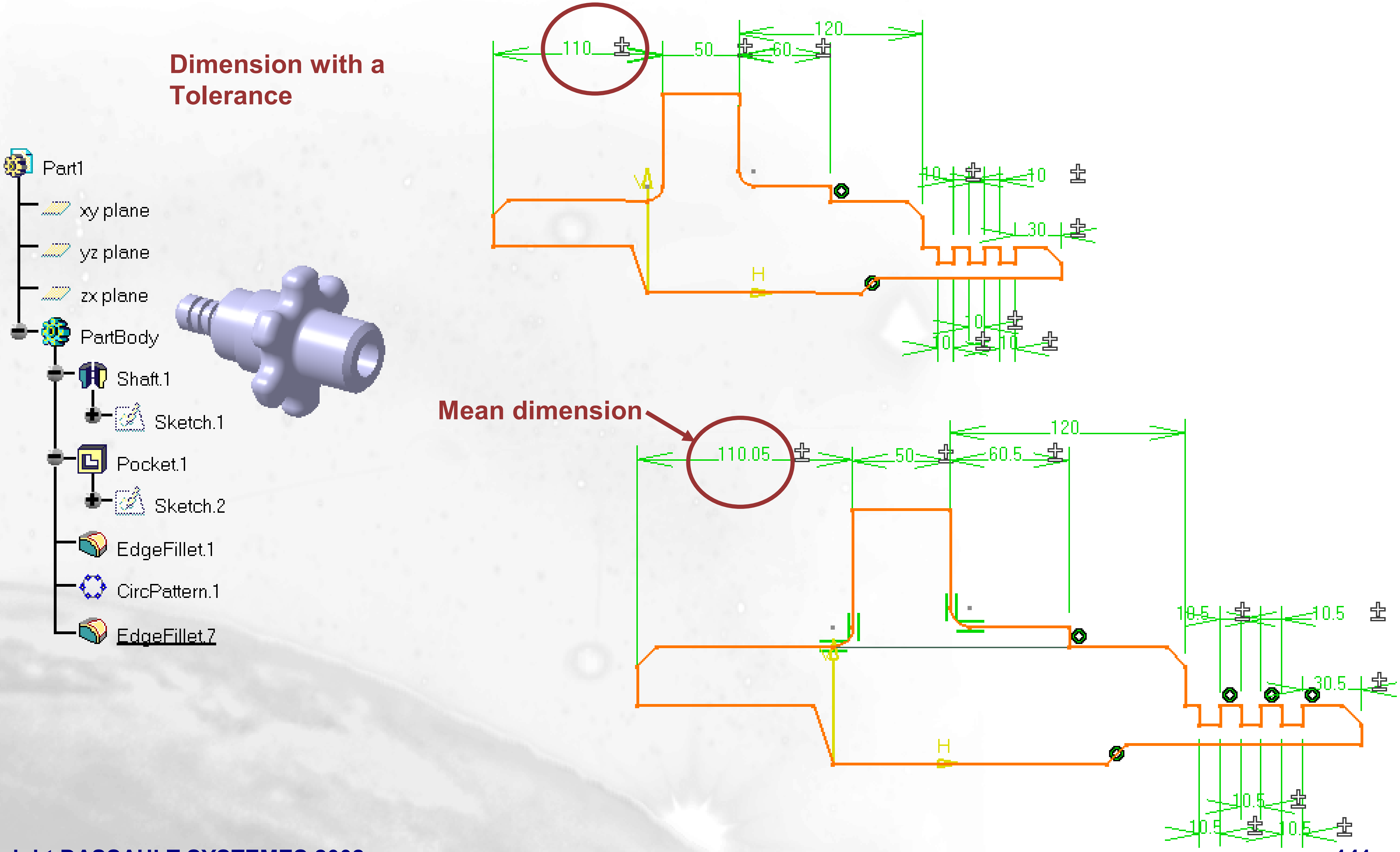




4 Minimum distance and angle (if you customize your dialog box) are displayed on the geometry and in the results Window

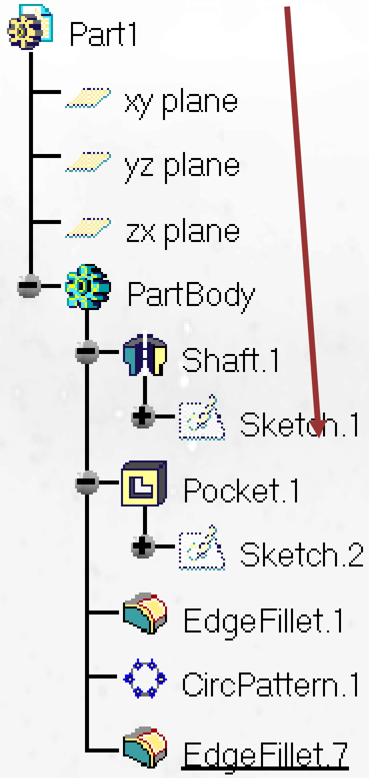
What are Mean Dimensions?

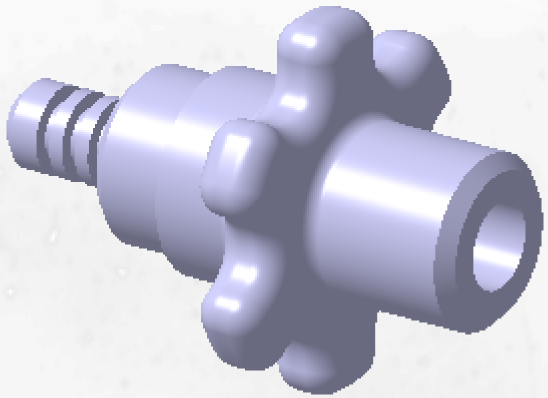
When creating dimensional constraints, you can define a tolerance. Using the Mean Dimensions icon you can compute the mean dimensions and the part will be updated. This can be useful for a part to be machined



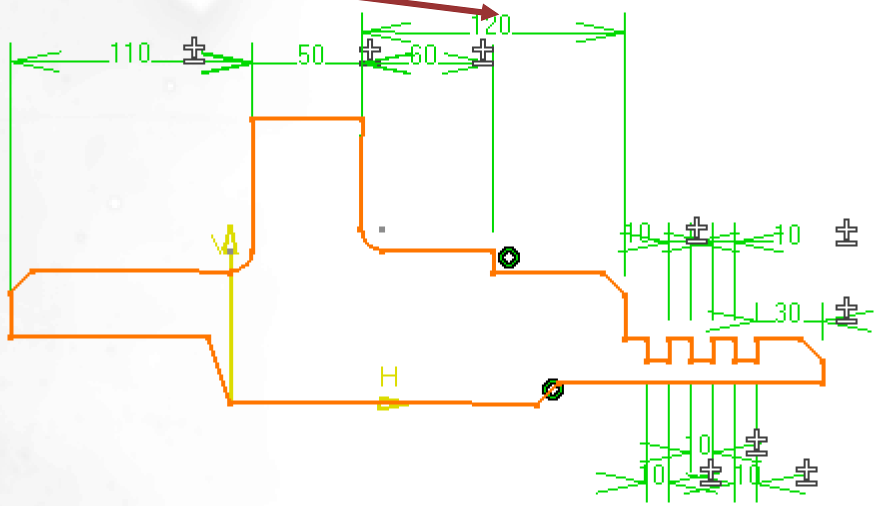
Mean Dimensions (1/4)

1 We are going to add tolerances on dimensions which have been created in the sketch of the shaft. Double click on Sketch.1



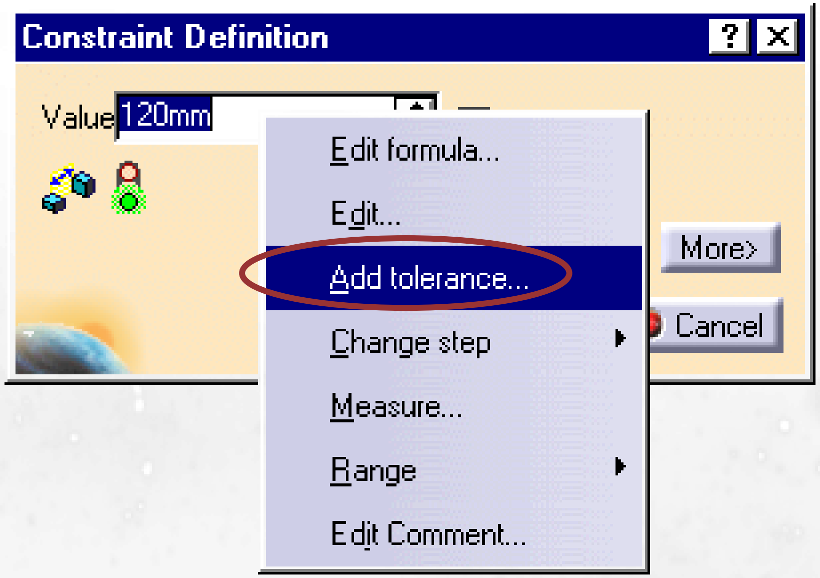


2 Double click on the indicated dimension

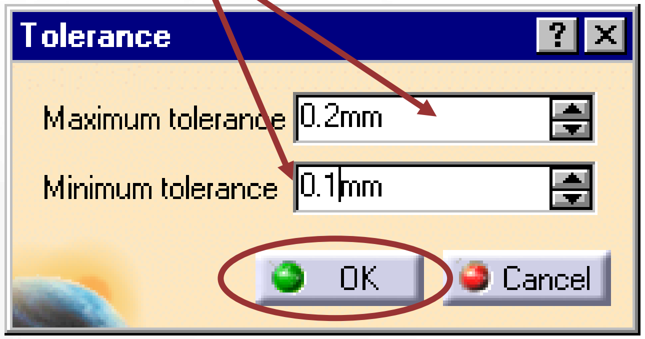


Mean Dimensions (2/4)

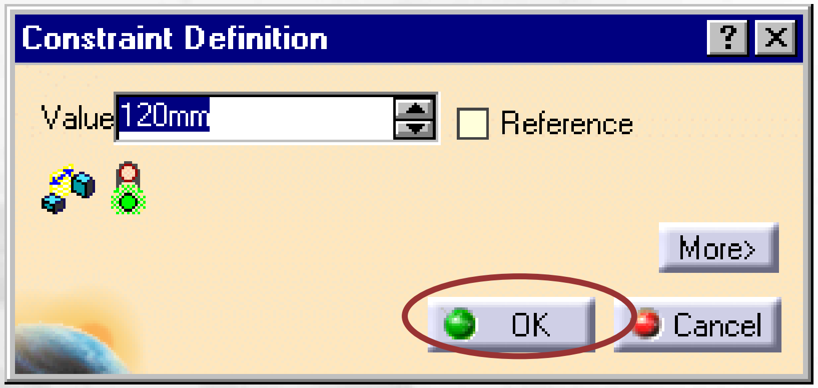
3 Using the contextual menu on the Value field, select the Add Tolerance command



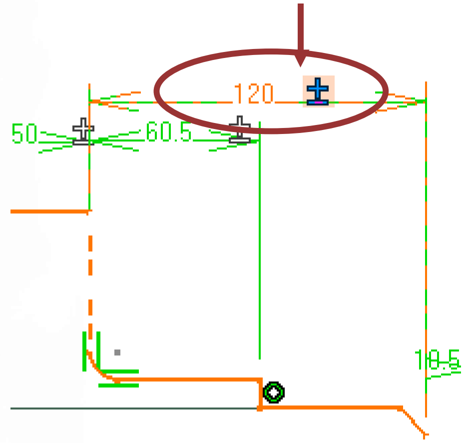
4 In the appearing dialog box, enter 0.2 in the Maximum tolerance filed and enter 0.1 in the Minimum tolerance field then select OK



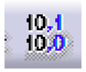
5 The tolerance is created. Select Ok in the Constraint Definition dialog box then Exit the sketcher



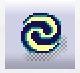
Dimension with a tolerance

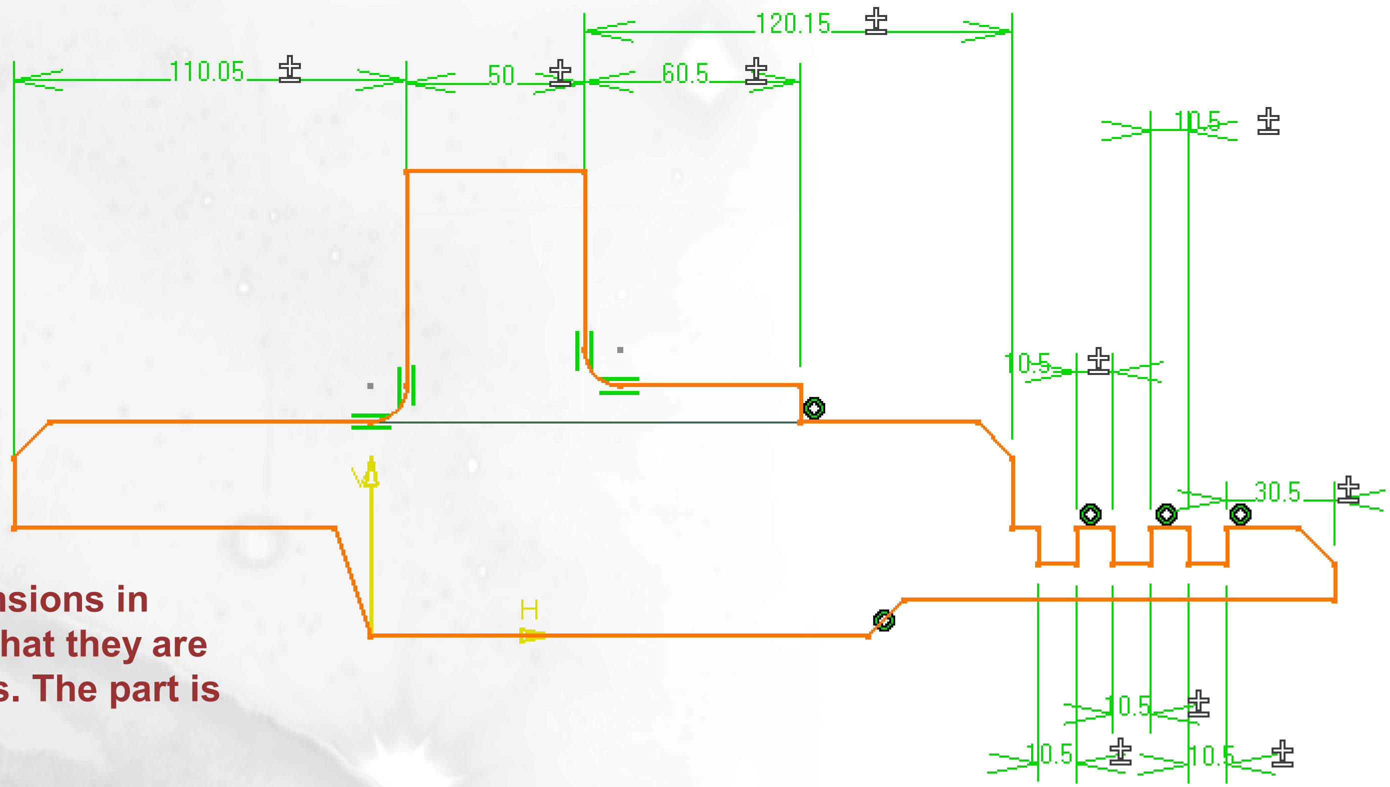


Mean Dimensions (3/4)

6 To compute the mean dimensions, select the Mean Dimension icon 

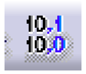
7 Select OK

8 Select the Update All icon 

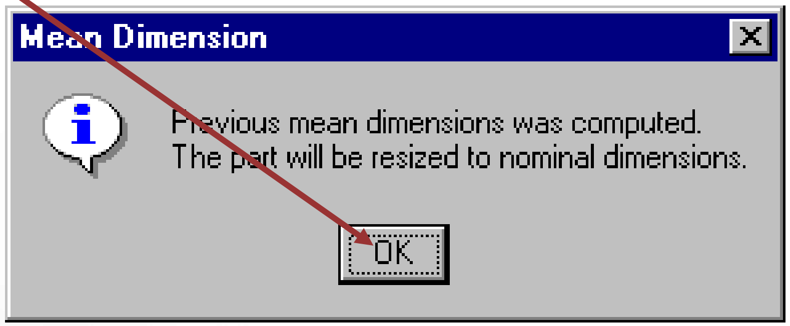


If you look at the dimensions in Sketch.1, you will see that they are changed to their means. The part is also updated

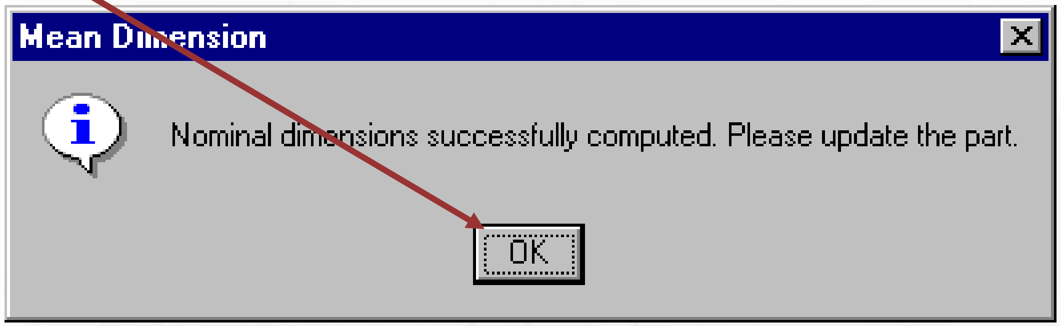
Mean Dimensions (4/4)

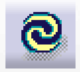
9 To come back to nominal dimensions, select the Mean Dimension icon 

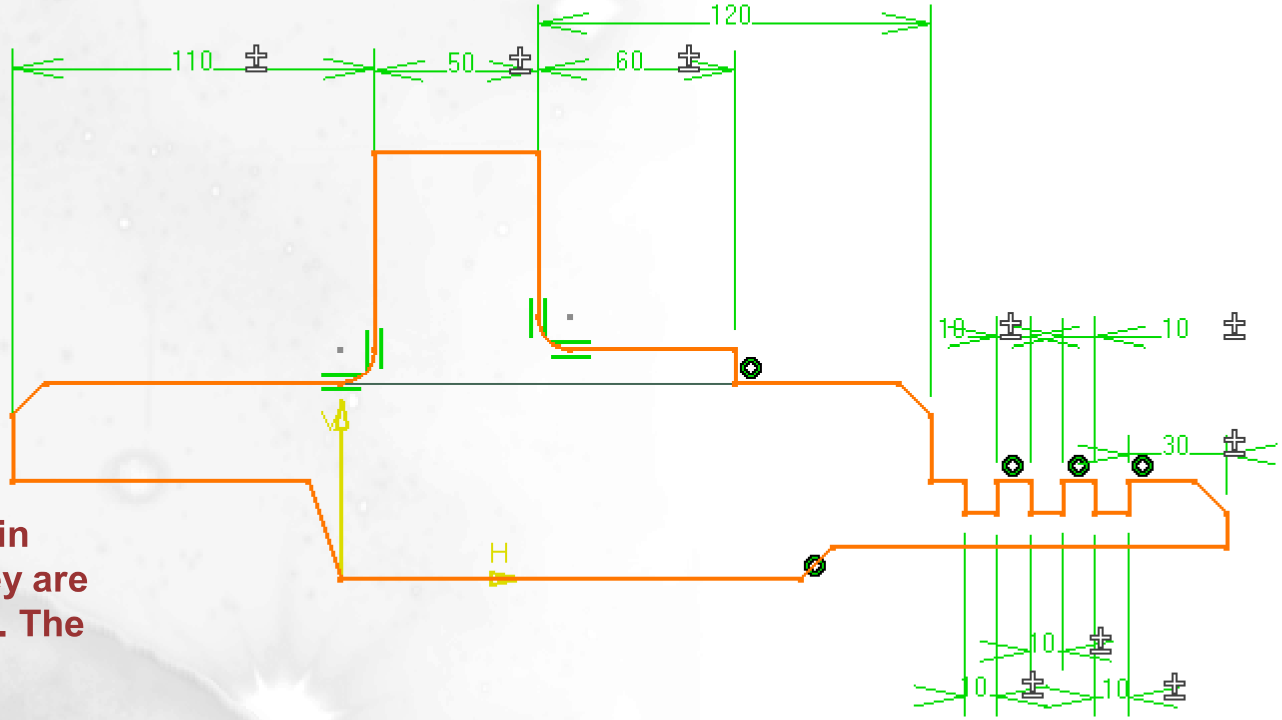
10 Select OK



11 Select OK



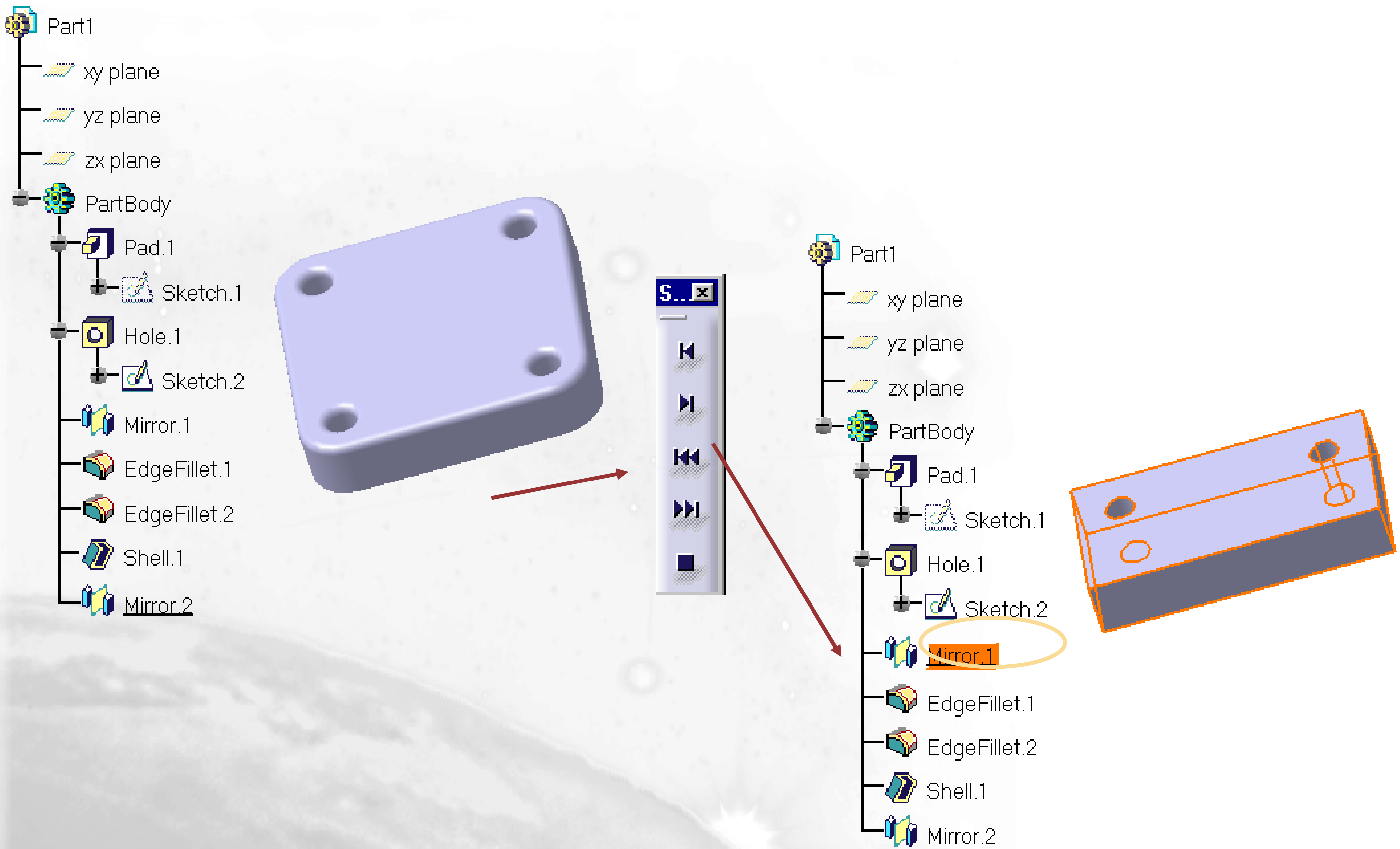
12 Select the Update All icon 



If you look at the dimensions in Sketch.1, you will see that they are changed to their nominal size. The part is also updated

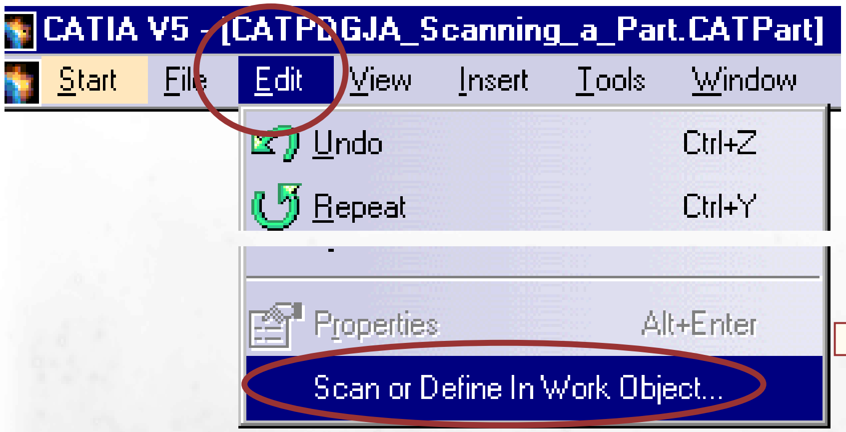
What is Scanning a Part ?

Scanning a part means to replay the construction history of a part and isolate temporarily any feature to work locally

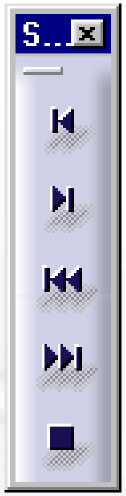


Scanning a Part

1 Select Edit > Scan ... Menu option



2



Use the Scan tools to navigate through the part structure

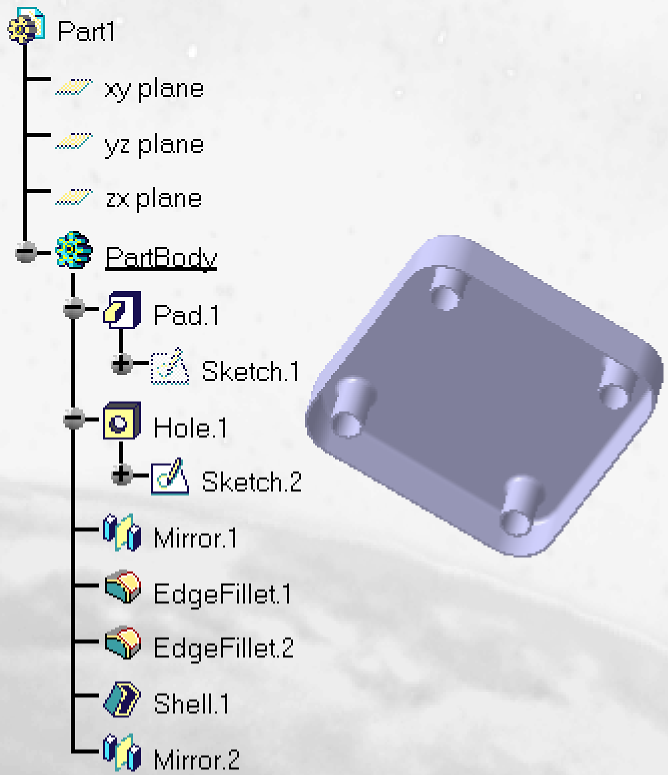
Backward: goes to the previous feature in the tree

Forward: goes to the next feature in the tree

Starting feature: feature active when starting scanning

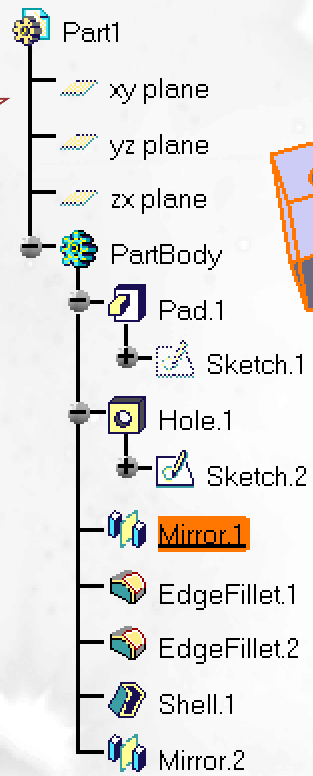
Last feature: last feature in the tree

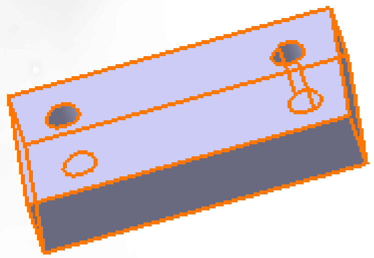
Exit: when you exit the active feature becomes in work (it is underlined in the tree)



Initial part

3





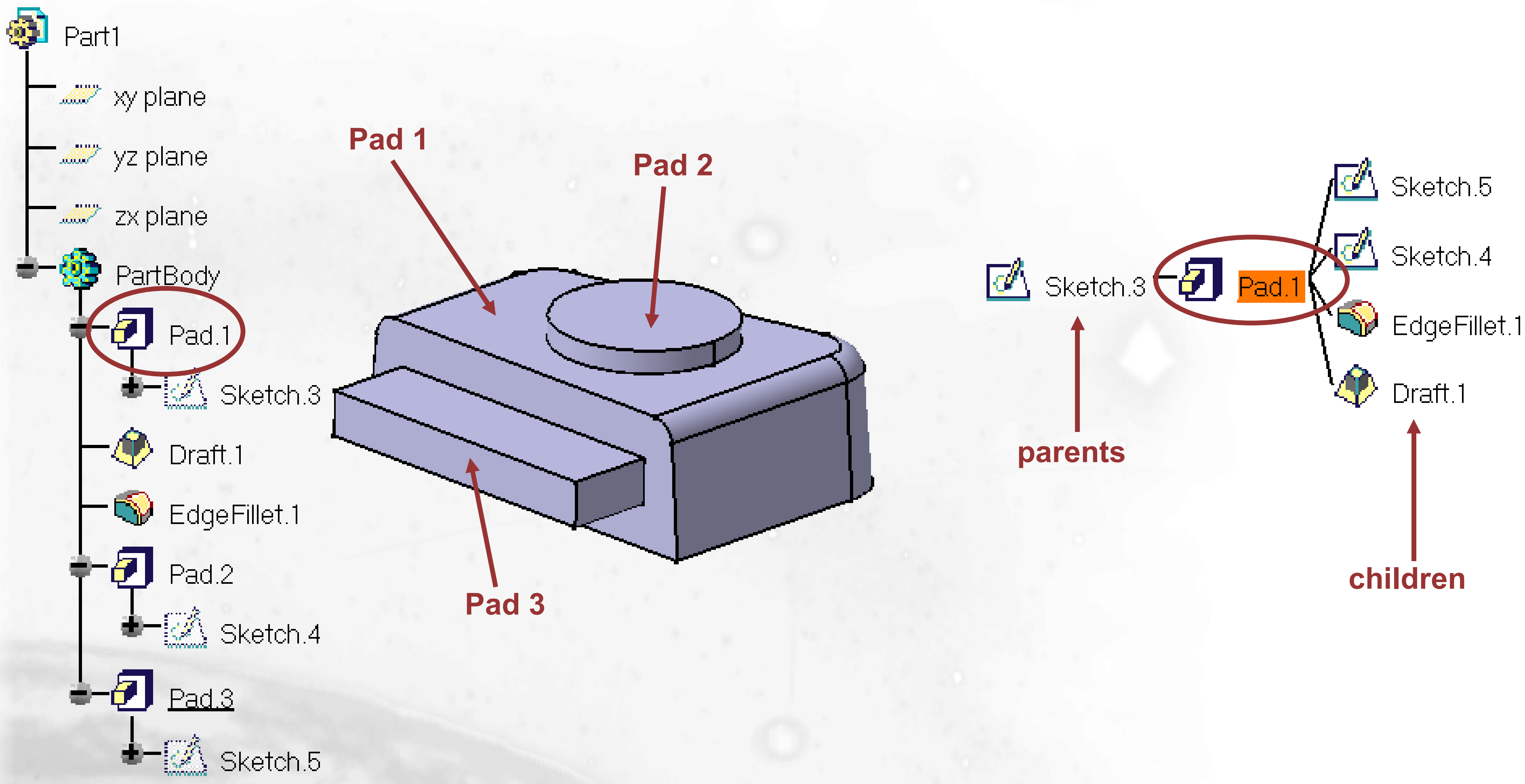
4

To work again on the whole part, click the last feature in the tree and select the Define in work option in the contextual menu (MB3)

The Mirror.1 feature is in work: you can make local changes

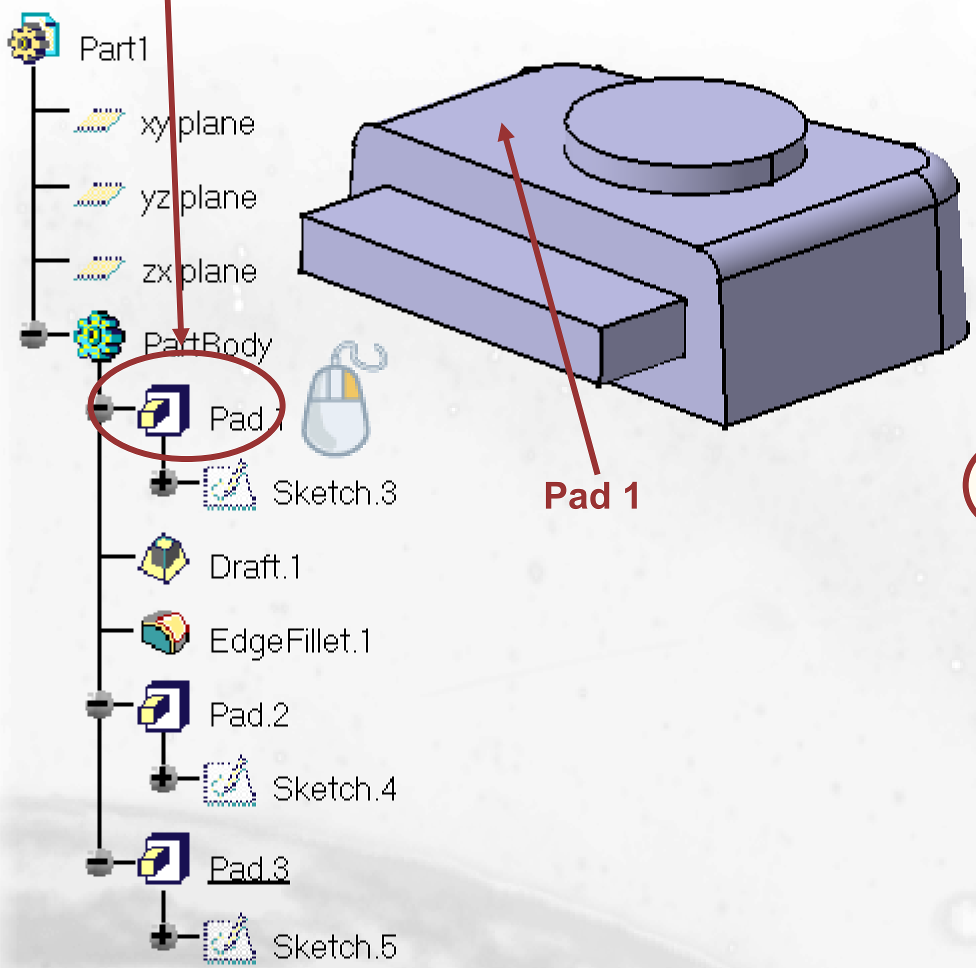
What is Parents-Children Relationship ?

The parents-children relationship provides an accurate view of genealogical links between elements. We recommend you to use it before deleting elements

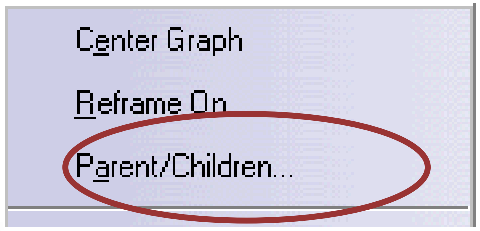


Parents-Children Relationship

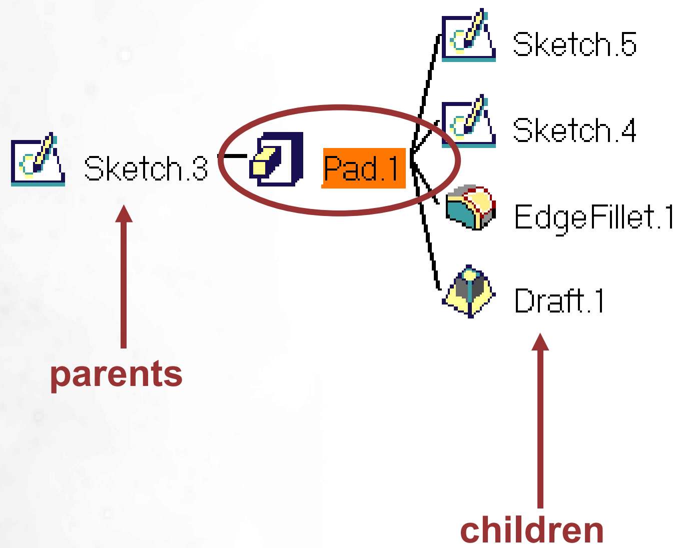
1 Activate the contextual menu on the desired feature (here, Pad.1)



2 Select the Parent-Children option

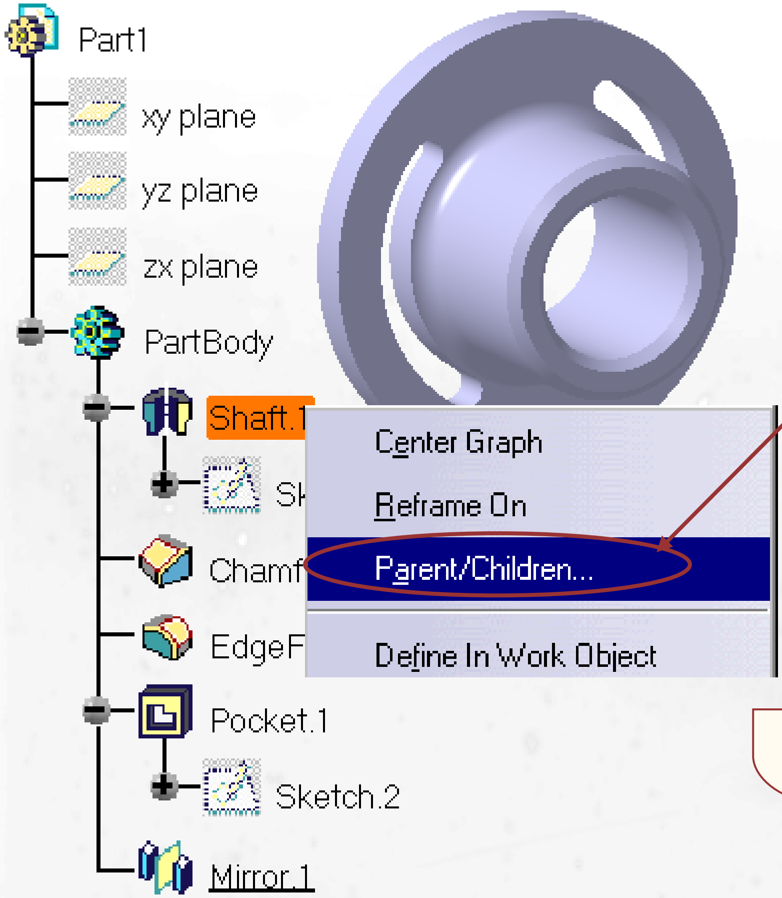


3 The graph allows you to show all parents / children (MB3)
Double-click a component to show/hide parents or children



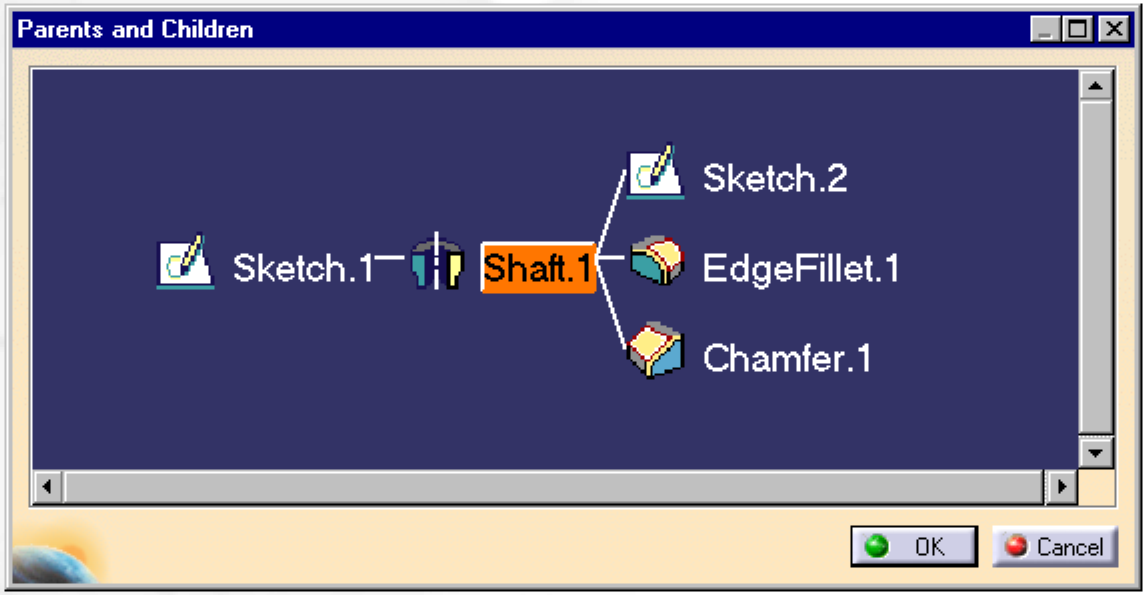
Parents-Children (Edition) (1/2)

Parent Children command lets you edit features

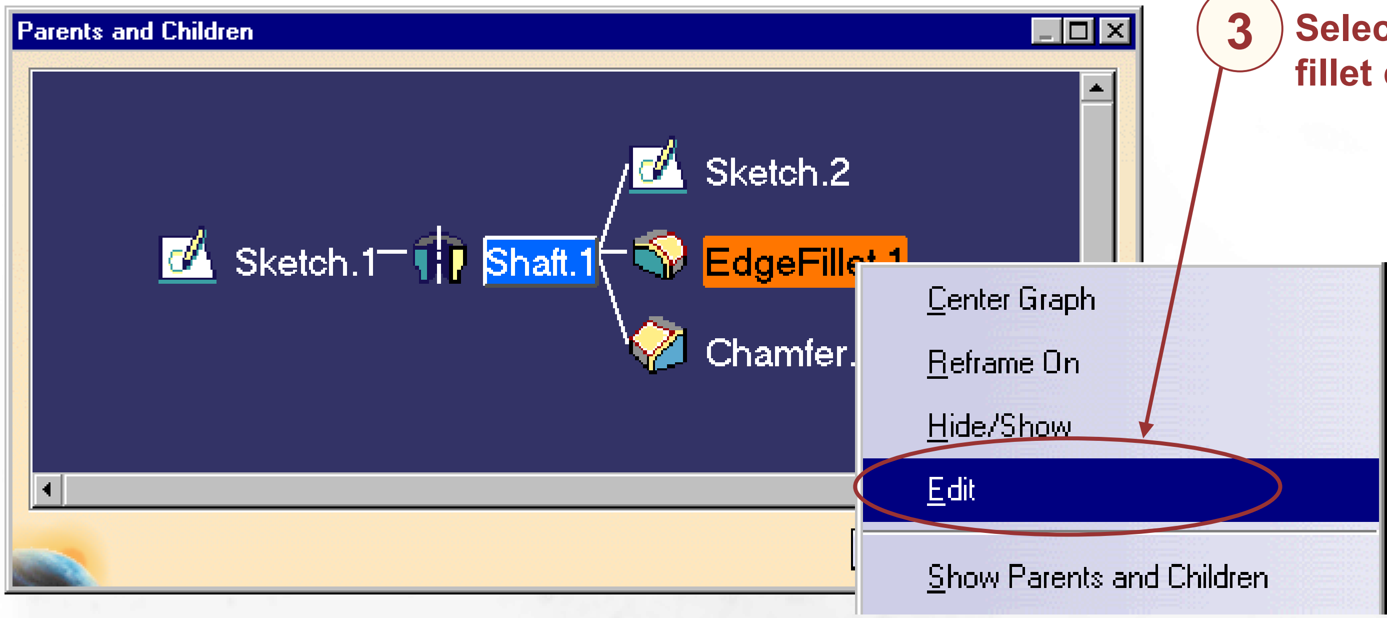


1 Select the Parent/Children command from the contextual menu

2 You get:

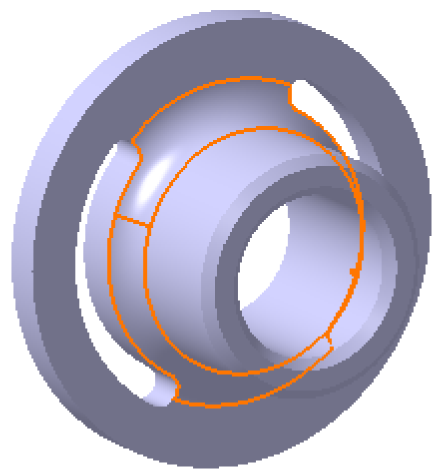


Parents-Children (Edition) (2/2)

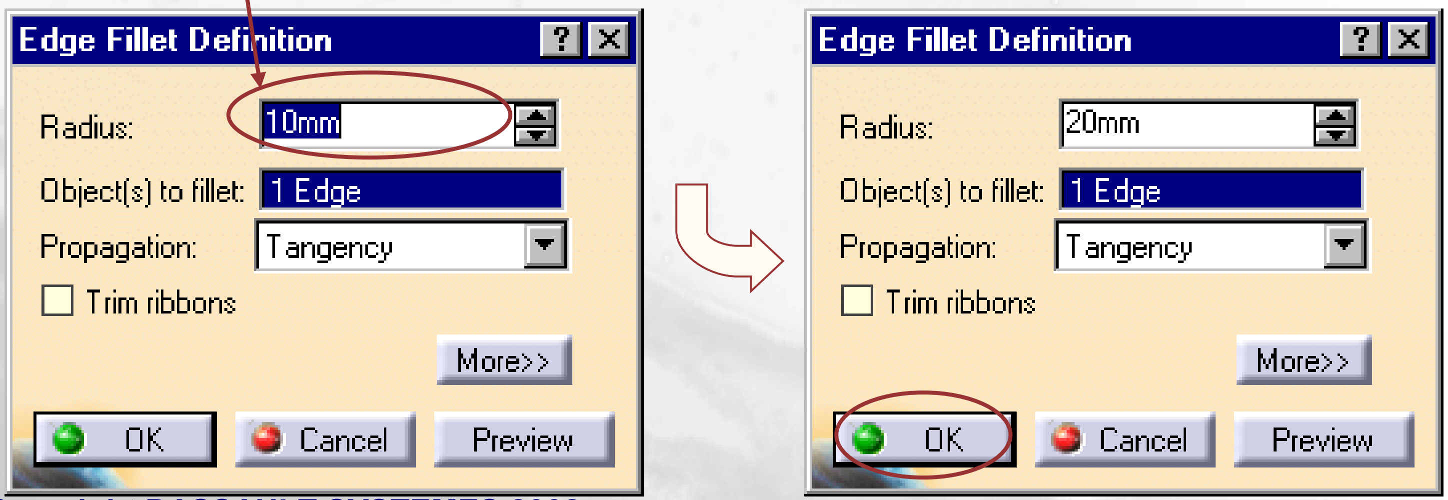


3 Select the Edit command from the fillet contextual menu

5 You get:



4 Modify the fillet radius then select OK



Cut, Paste, Isolate, Break

In this lesson, you will see how to cut or copy a feature and paste it onto a body and you will also see how to isolate or break 3D geometry from their parents

- **Cut/Copy and Paste (Drag and Drop)**
- **Isolate/Break**

What is Cut/Copy and Paste (Drag and Drop) ?

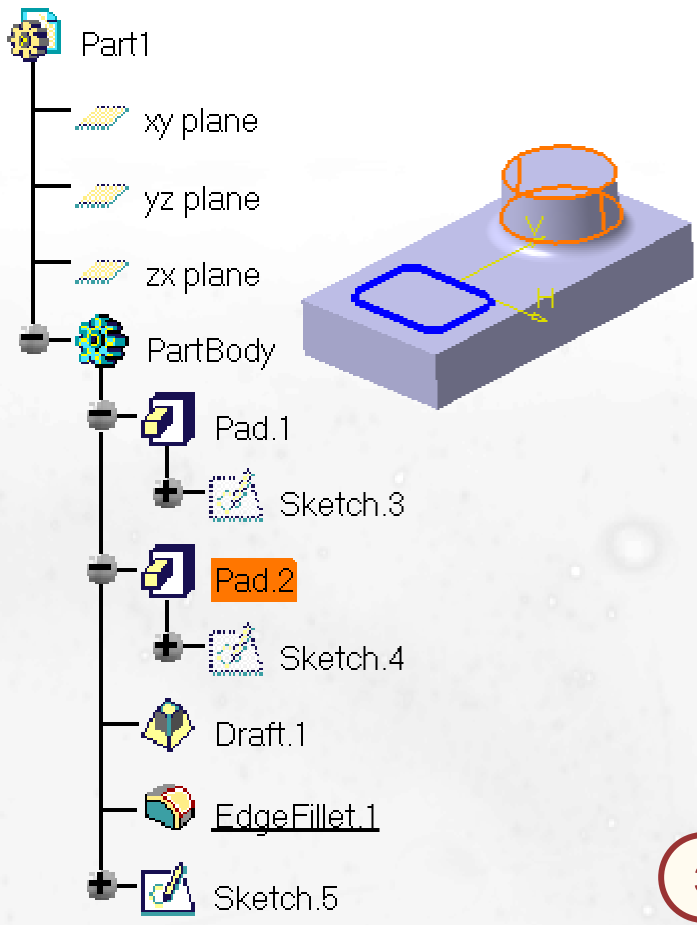
Cut/Copy then Paste captures the node specified into the clipboard and either replaces (Cut) or copies (Copy) the content into a different selected point in the part structure. The action is interpreted by the system in a context sensitive manner. For example, if a pad is copied onto a different sketch, the new sketch is used for the profile and information on extrusion limits will be those of the pad. However, if pad1 is copied onto pad2, since this action has no real meaning, it is interpreted as generically copying the clipboard's content into the part. The effect is to create another copy of pad1 (with its original sketch) in the part structure. This copy will be placed after whatever node is currently the "In Work" node

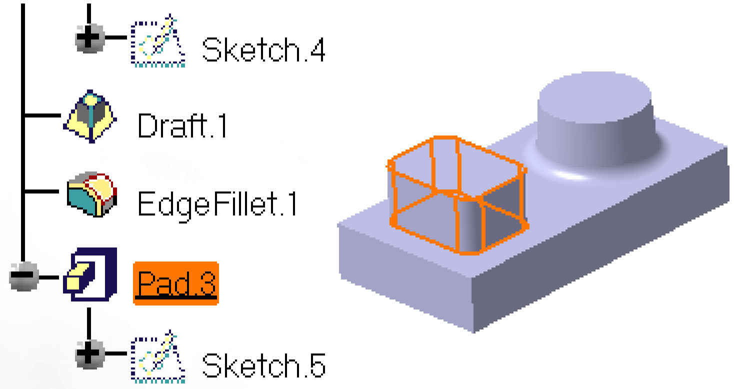
Cut/Copy then Paste an be achieved by drag and drop. If the CTRL key is pressed during the drag and drop, the action is interpreted as a copy

Cut/Copy and Paste (Drag and Drop)

1

One way we can copy the limits of the circular pad to apply to the rectangular pad is to work within the Part tree and use the 3rd. Mouse button to Copy Pad.2 and Paste onto Sketch.3



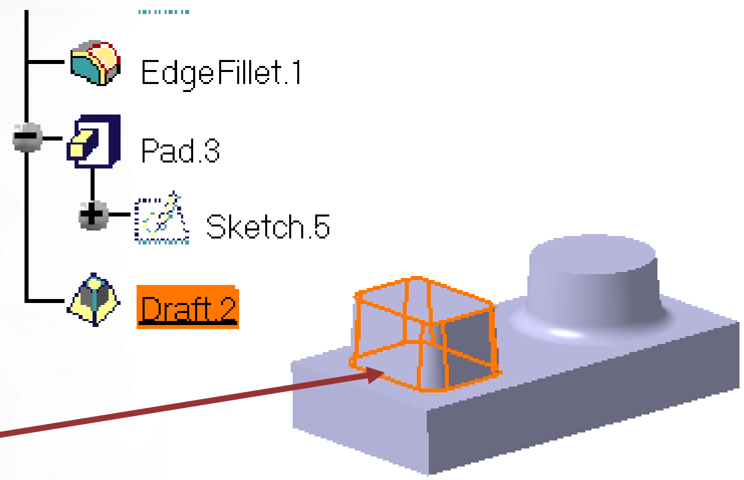


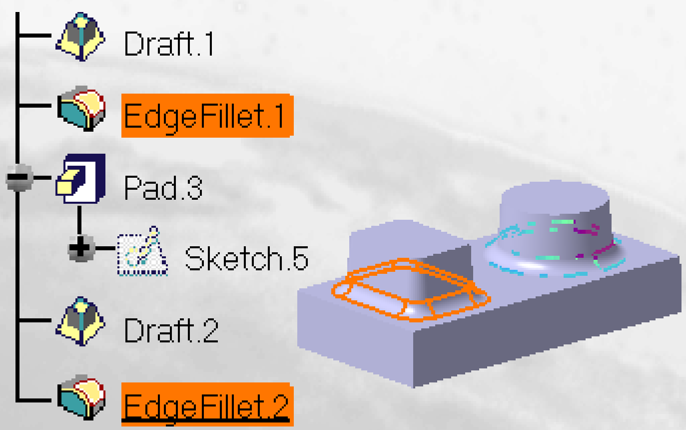
2

We can copy the draft by using another variation - 3rd. Mouse button to copy Draft.1 from the tree then select a vertical face on the rectangular pad and 3rd. Mouse button to Paste

3

Another variation for the fillet - Keeping the CTRL key pressed, Drag with the 1st. Mouse button to one of the base edges of the rectangular pad





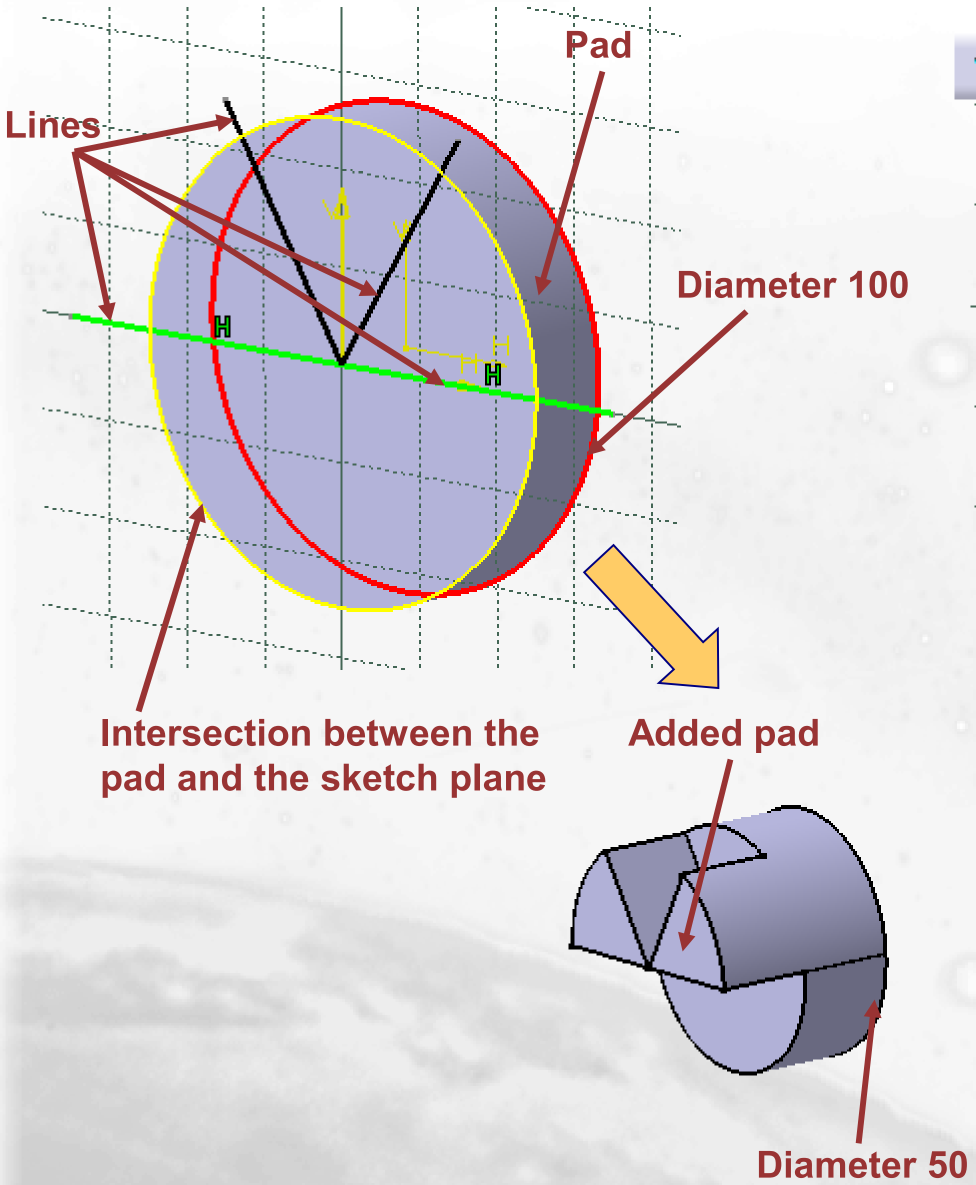
What are Isolate and Break ?

Isolate is used when 3D geometry is projected into a sketch in order to be modified and used as part of the sketch's profile. Isolate duplicates the element since the original element cannot be changed since other geometry depend on it

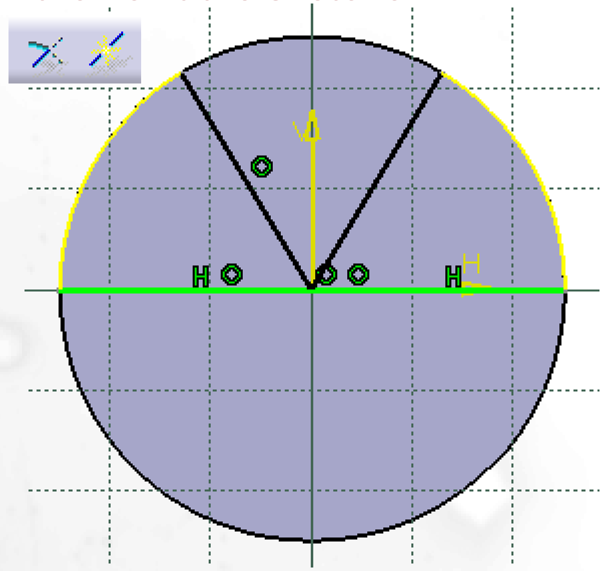
Break is used to divide an isolated element into two parts at a specified point (usually to use one side of this element in the sketch)

Isolate, Break (1/3)

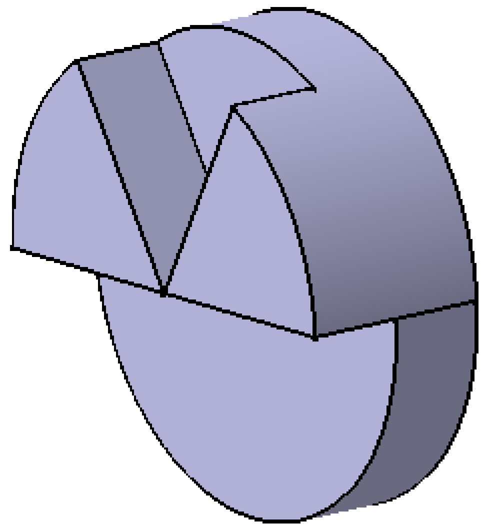
1 Starting with the geometry below, we want to add a pad



2 Using the Trim and Break icon in the sketcher, modify the sketch as follows, then exit the sketcher

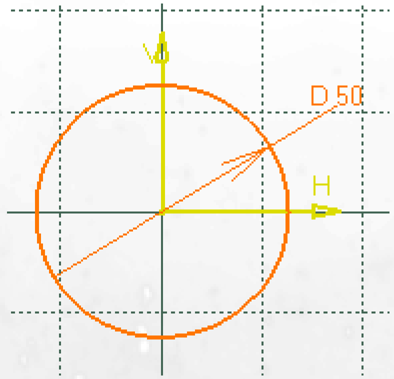


3 Create a pad with an length of 20

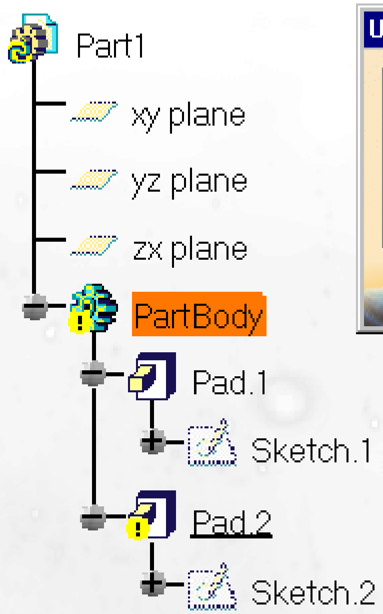


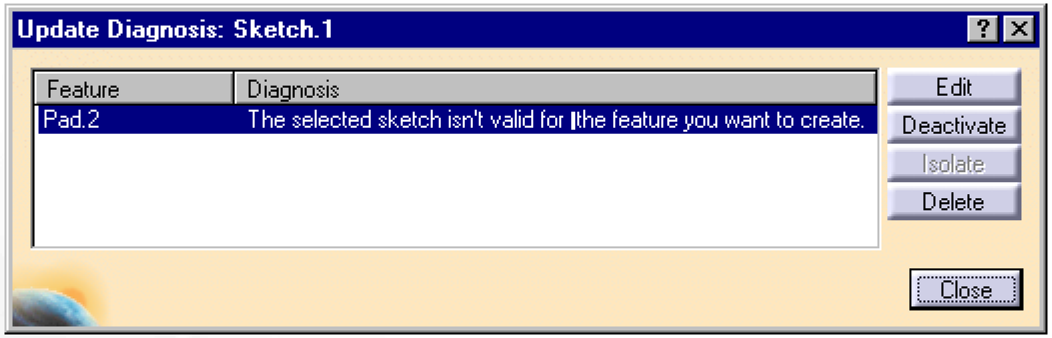
Isolate, Break (2/3)

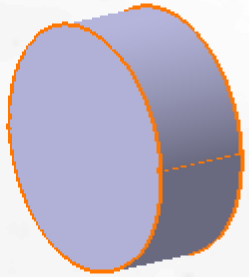
4 Edit the Sketch of the first pad then change the circle diameter to 50



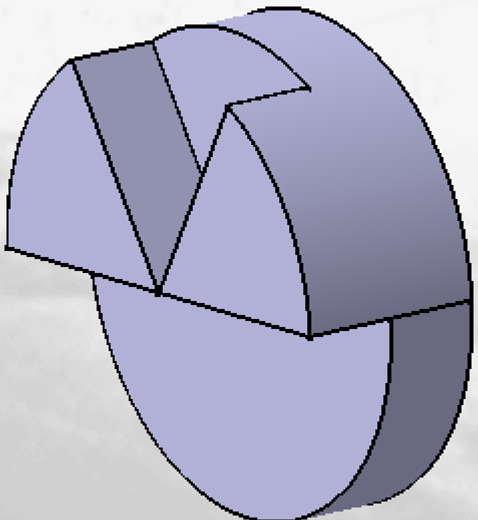
5 Exit the sketcher (Sketch.1) then, if necessary, Update the part. You will get:



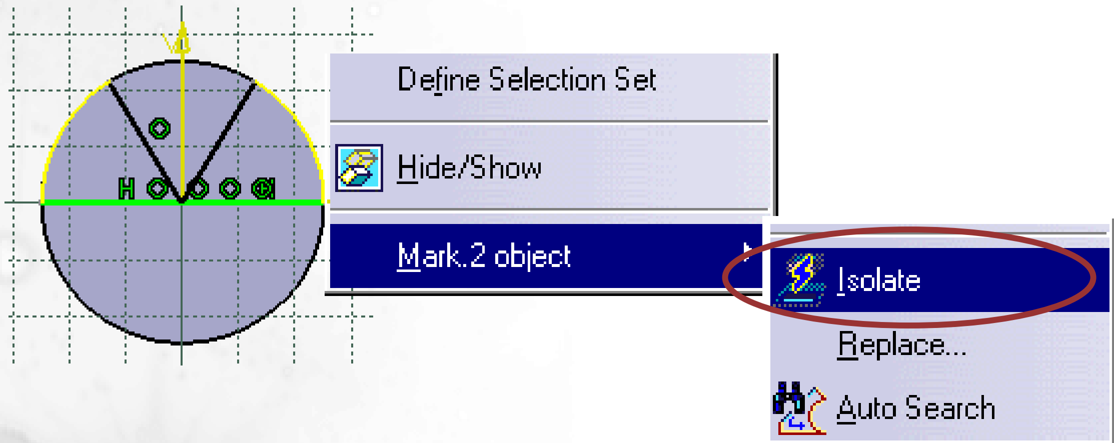




6 Select the Undo icon (may be several times) in order to come back to diameter 100

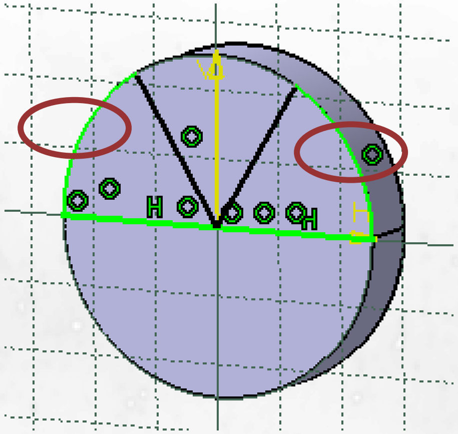


7 Edit Sketch.2, then place the cursor on the yellow line then select Isolate from the contextual menu

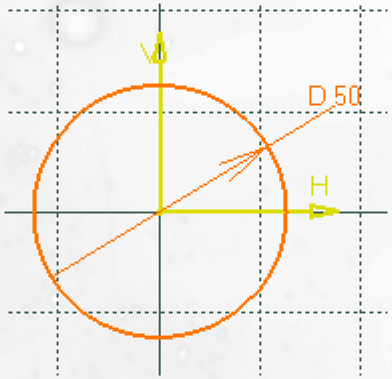


Isolate, Break (3/3)

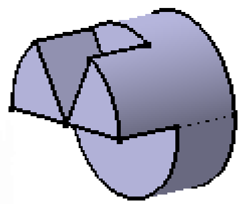
8 Create two Coincidence between the isolated arcs and the cylinder then exit the sketcher



9 Edit the Sketch (Sketch.1) of the first pad then change the circle diameter to 50



10 Exit the sketcher then, if necessary, Update the part. You will get:



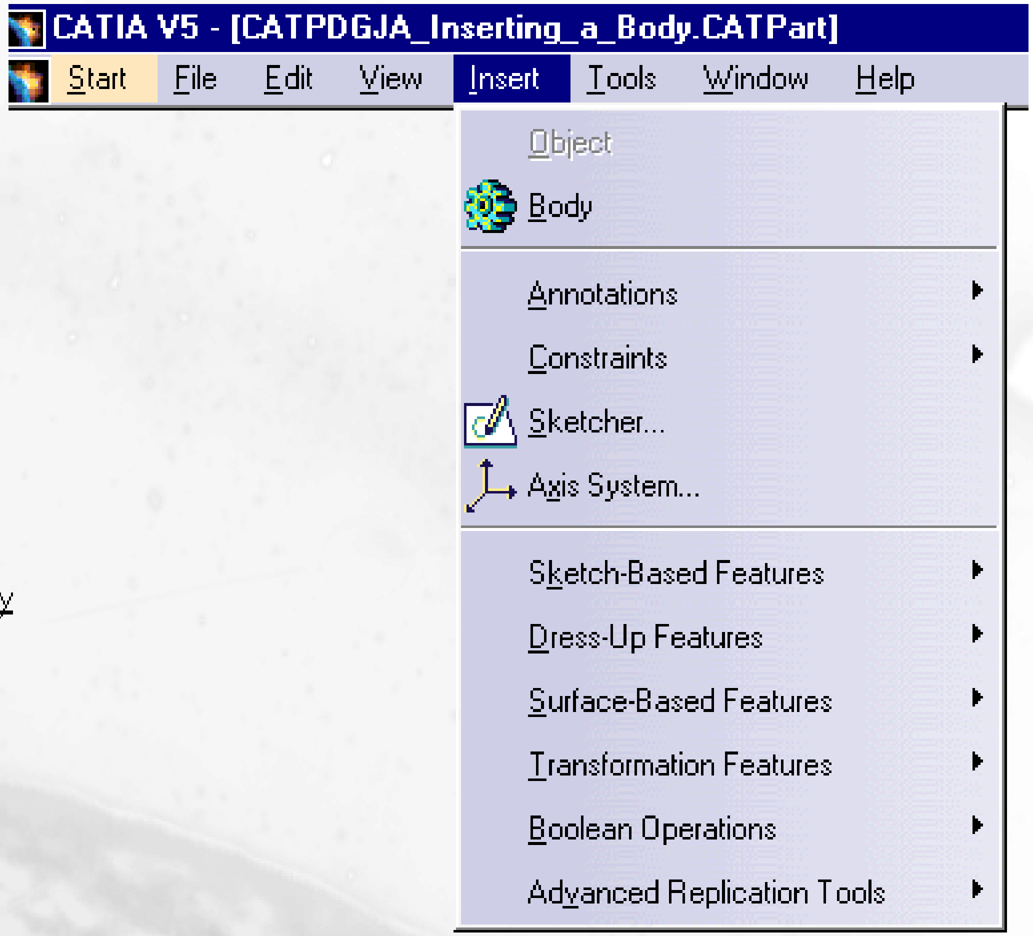
Inserting and Managing Bodies

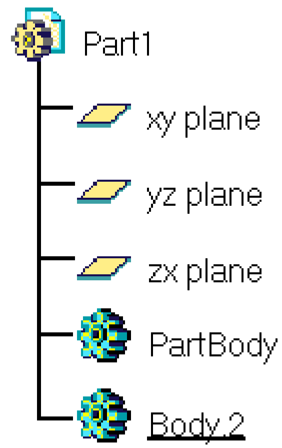
You will learn ways to manage Bodies using tools such as Assembling, Intersecting, Adding, Removing, and Trimming bodies

- Inserting a Body
- Assembling/Intersecting/Adding/Removing Bodies
- Union Trimming Bodies
- Removing Lumps
- Replacing a Body
- Change Boolean Type

What is Inserting a Body?

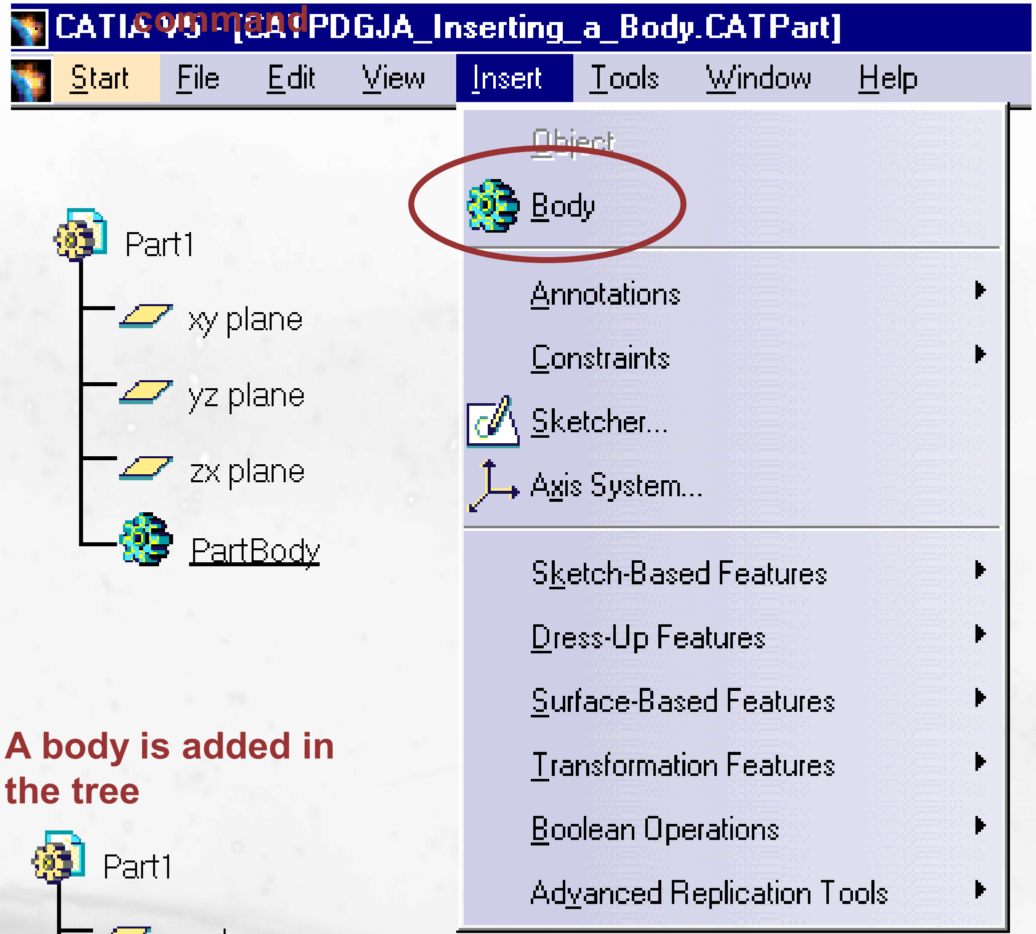
Using several bodies in a part allows you to design different step of a part without any operations between bodies. You will be able to perform operations (add, assemble, remove, …) later. This method can be use when, for example, you create a mold part. You can create the outside of the part in a body and the core in another one then you can remove the core from the main part. Later it will be easy for you to separate the part and it core



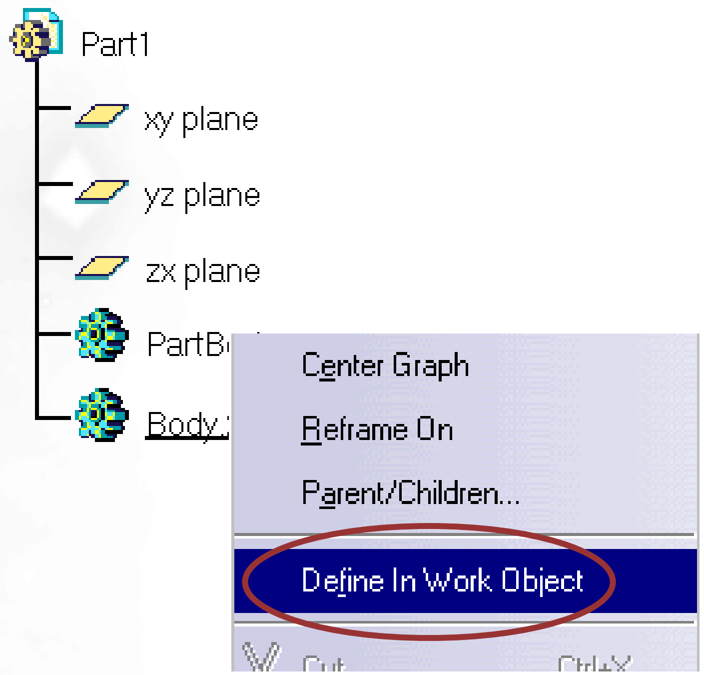


Inserting a Body

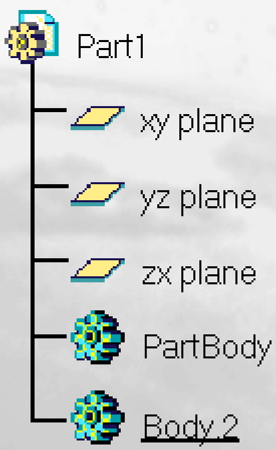
1 In order to insert a Body, select the Insert -> Body command



2 You can work in the PartBody or in Body.2. Top Switch from one Body to another, select the Define in workobject command from the contextual menu of the desired body



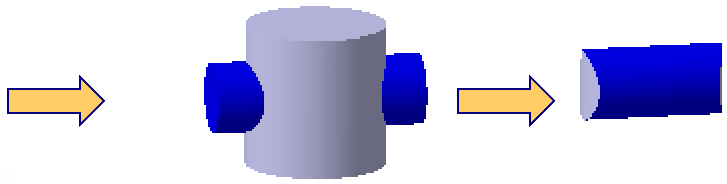
A body is added in the tree



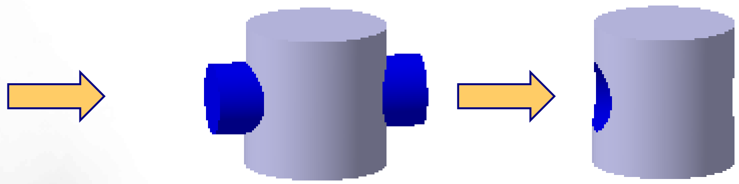
What is Inserting and Managing Bodies ?

***Assembling/Adding :** If Body2 is Assembled or Added to Body1, the operation between the bodies is a Union. The only difference between the two is that Assemble will respect the "nature" of features. If Body2 contains as its first node a Pocket feature (permissible), Assemble will see it as a Pocket and remove material from Body1. In this case, if Add is used, the Pocket will be seen by Body1 as a Pad*

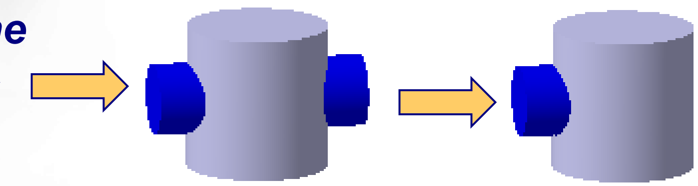
***Intersecting :** The resulting material is the intersection between the two bodies*



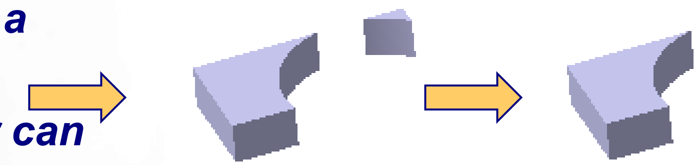
***Removing :** If Body2 is Removed from Body1, the operation is Body1 minus Body2*



***Union Trim :** The Union Trim is basically a Union with an option to remove or keep one side or the other. In the picture on the right, the purple face is selected to remove the right side and the blue face is selected to keep only the top side. For the Union Trim to work, the geometry must have sides that are clearly defined*

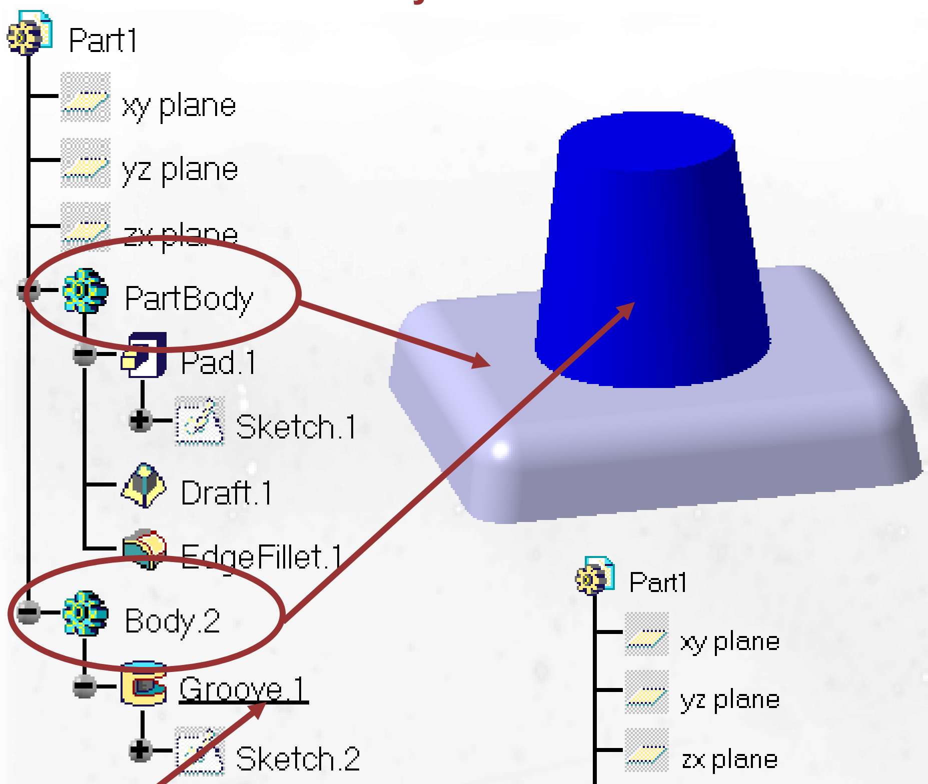


***Remove Lump :** All the above options work between two bodies. The Remove Lump works on geometry within a specific Body. If a single Body has material that is completely disconnected, each piece of disconnected material is defined as a "Lump". The user can delete any Lump as a single entity even if the Lump is a combination of numerous features*



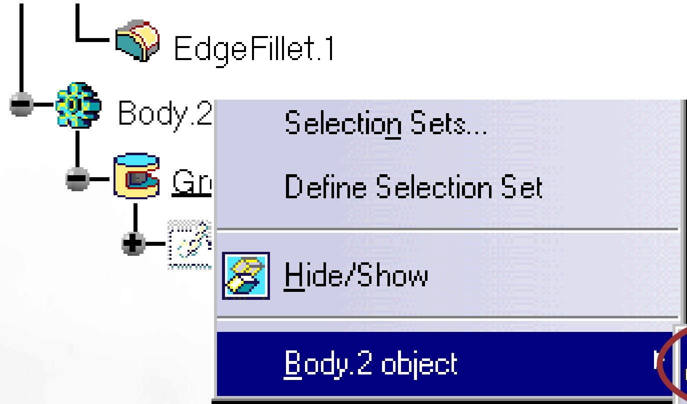
Assemble

1 We want to assemble Body.2 with PartBody

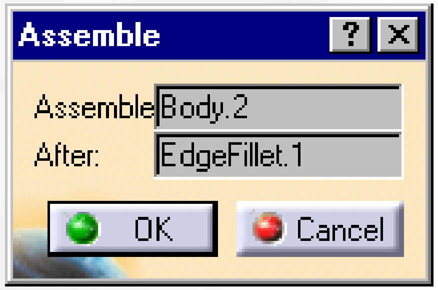


Body.2 contains a groove

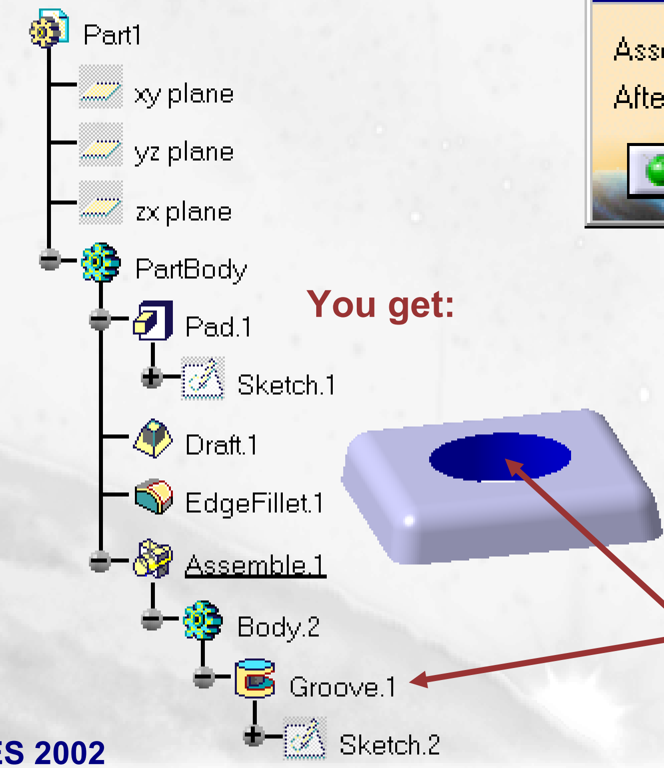
2 With the cursor on Body.2, select Assemble from the contextual menu (MB3)



3 Select OK in the Dialog box



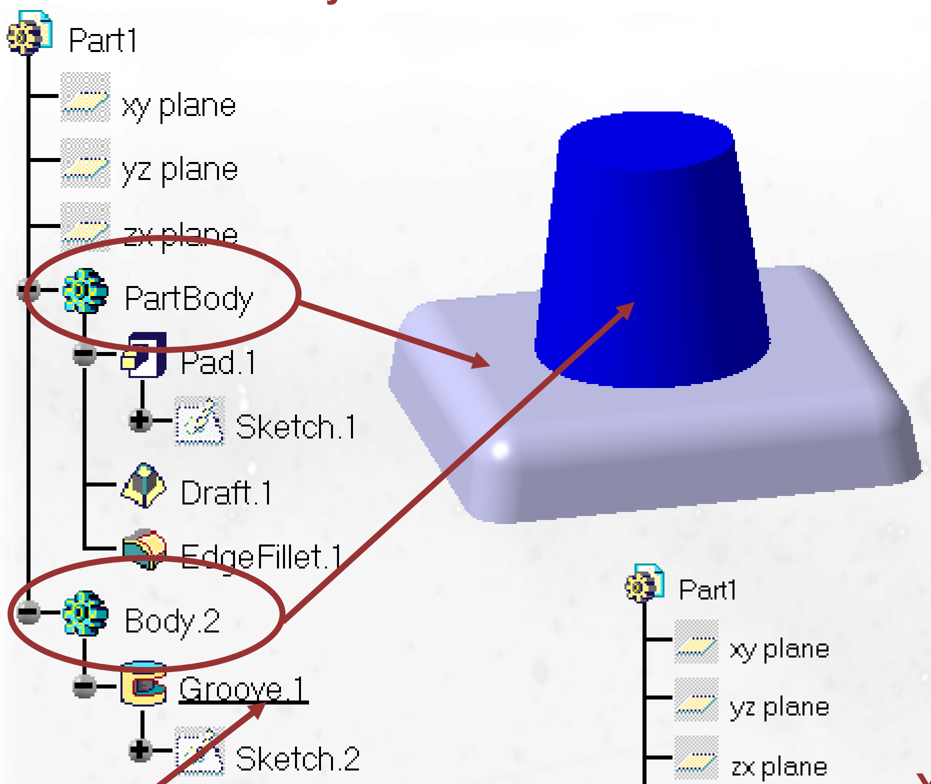
You get:



Because Body.2 contains a groove which is a feature that removes material, the result of the assemble operation is also removing material

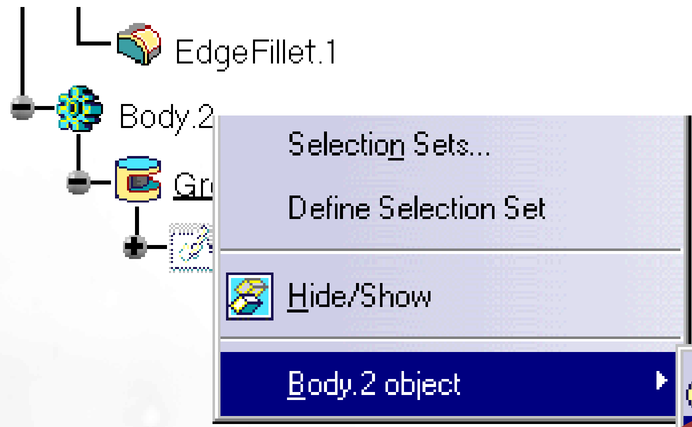
Add

1 We want to add Body.2 with PartBody

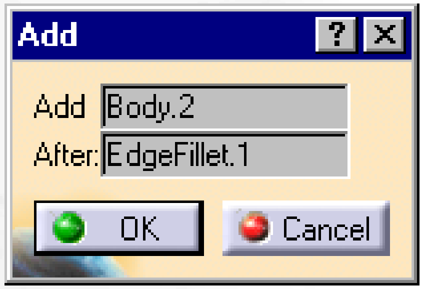


Body.2 contains a groove

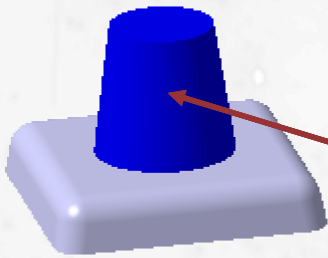
2 With the cursor on Body.2, select Add from the contextual menu (MB3)



3 Select OK in the Dialog box



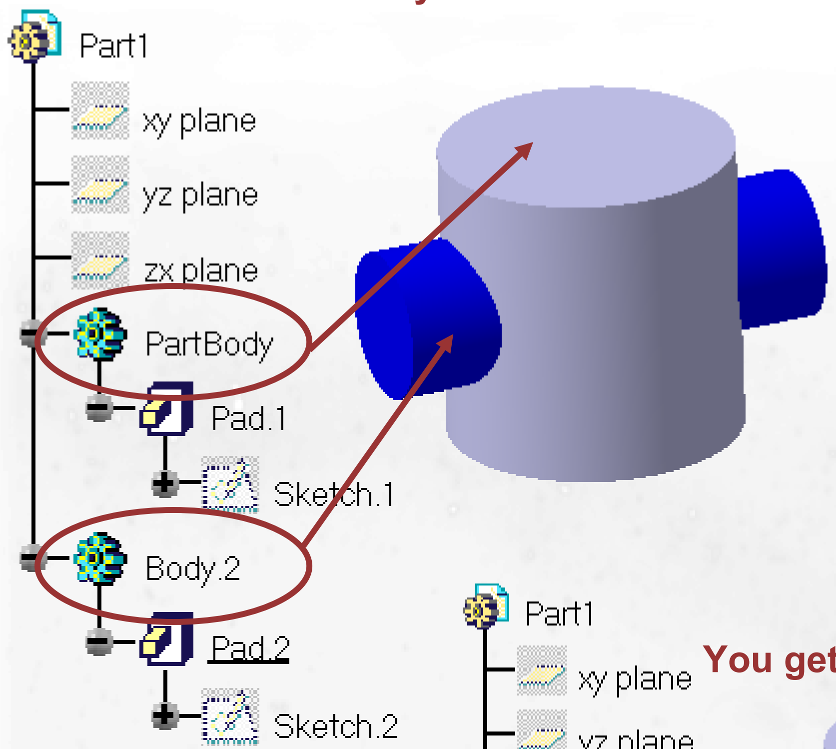
You get:



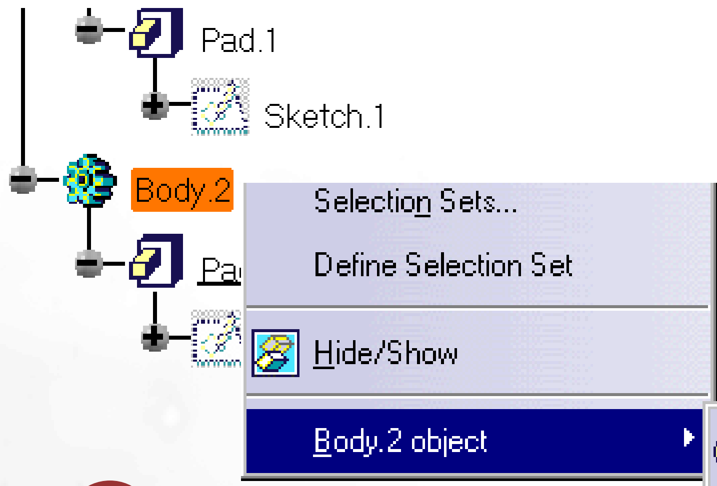
Body.2 contains a single groove, so it is appears as a solid (even if it normally removes material). When you Add a Body, CATIA keeps the feature like it appears before the addition.

Remove

1 We want to remove Body.2 from PartBody

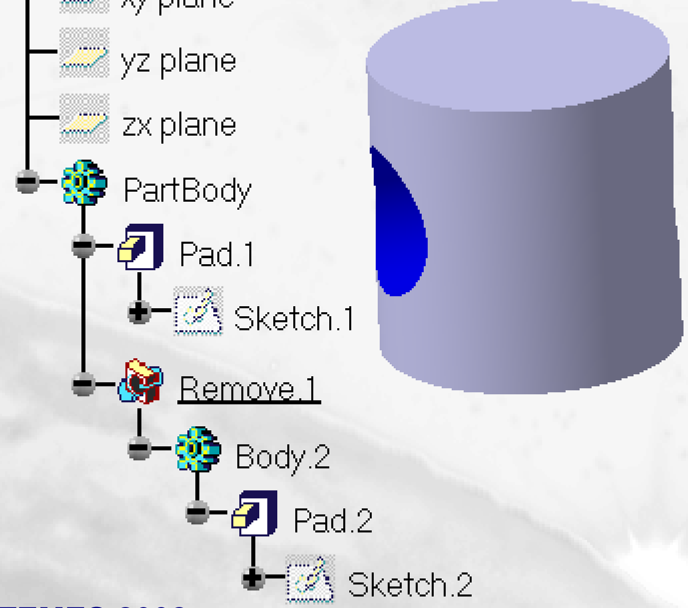


2 With the cursor on Body.2, select Remove from the contextual menu (MB3)



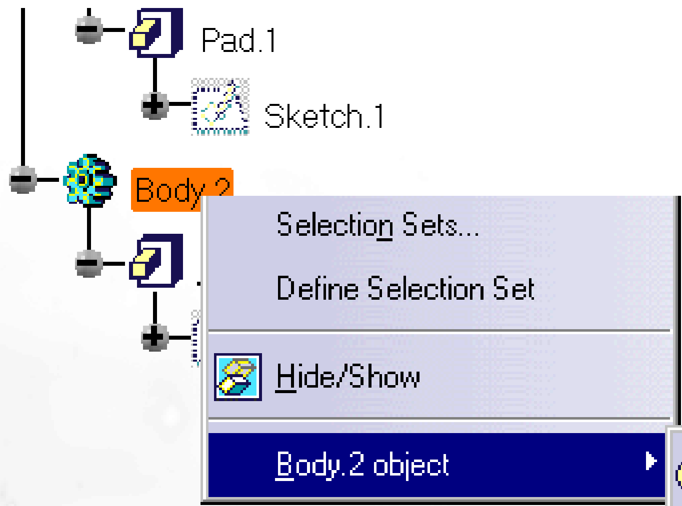
3 Select OK in the Dialog box

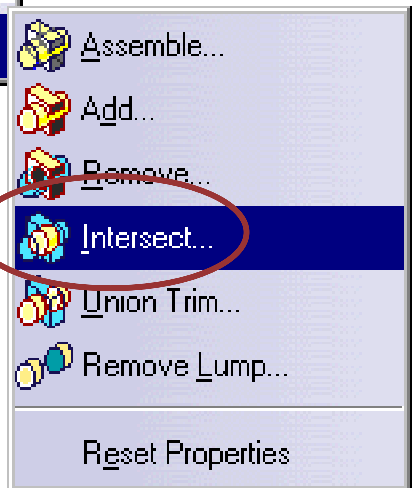
You get:



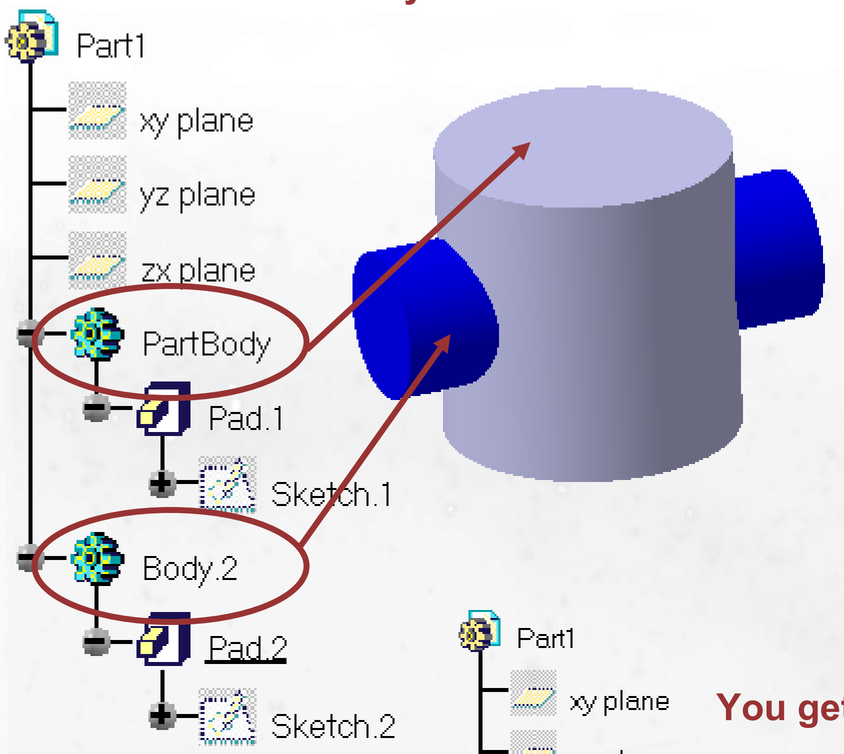
Intersect

2 **With the cursor on Body.2, select Intersect from the contextual menu (MB3)**



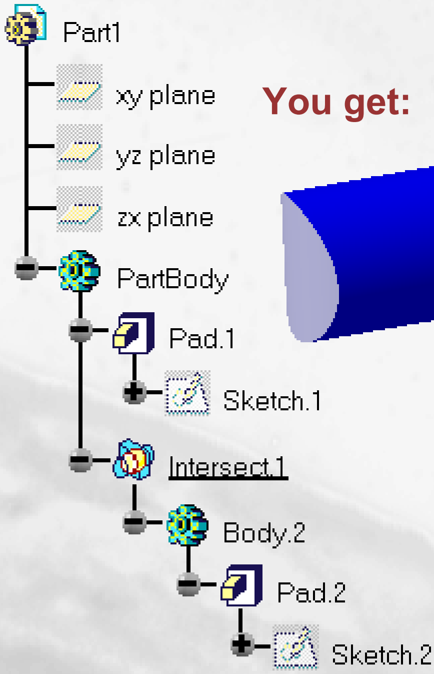


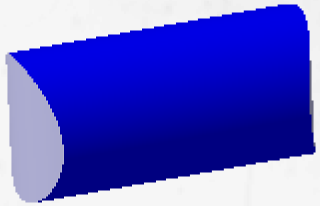
1 **We want to intersect Body.2 with PartBody**



3 **Select OK in the Dialog box**

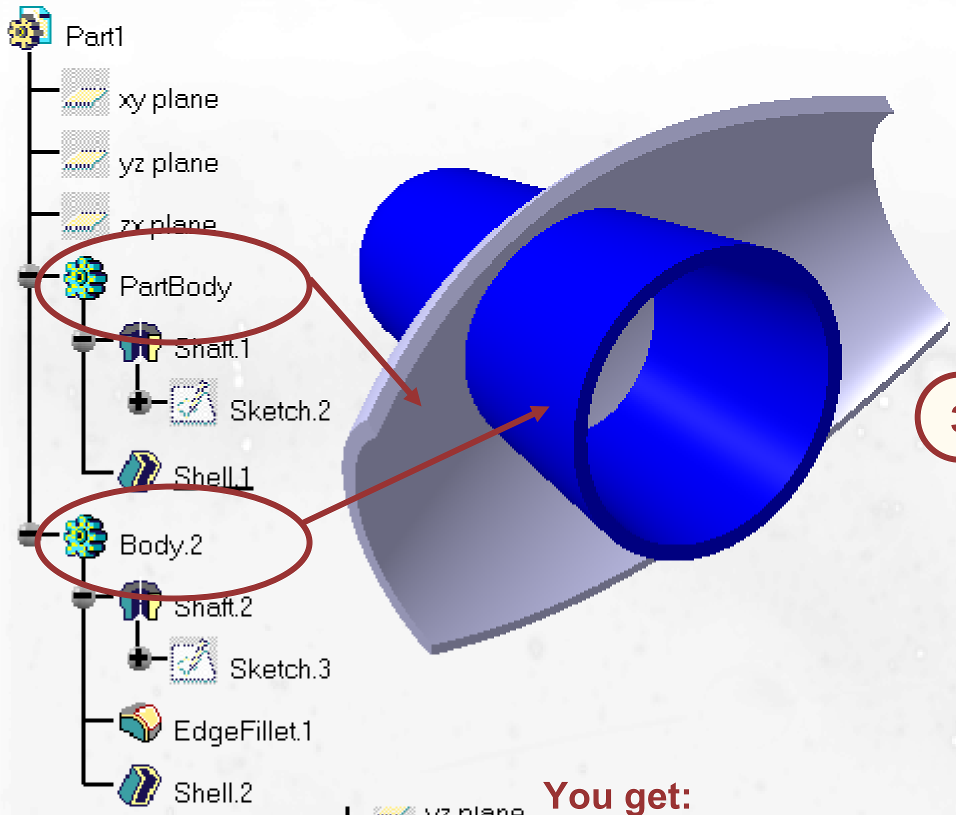
You get:



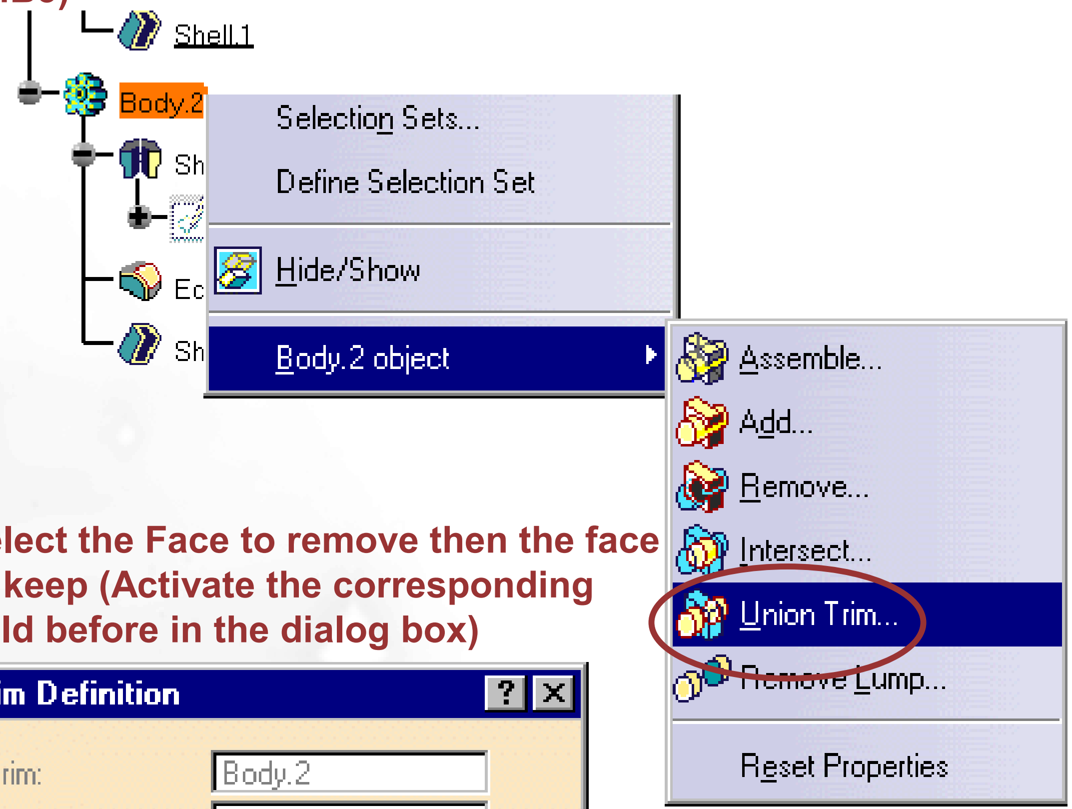


Union Trimming Bodies

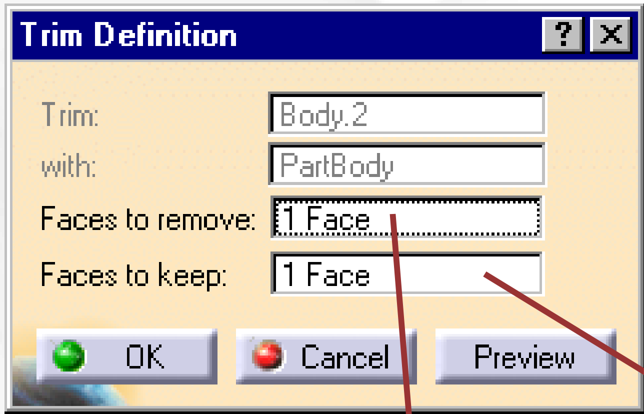
1 We want to do an Union Trim of Body.2 with PartBody

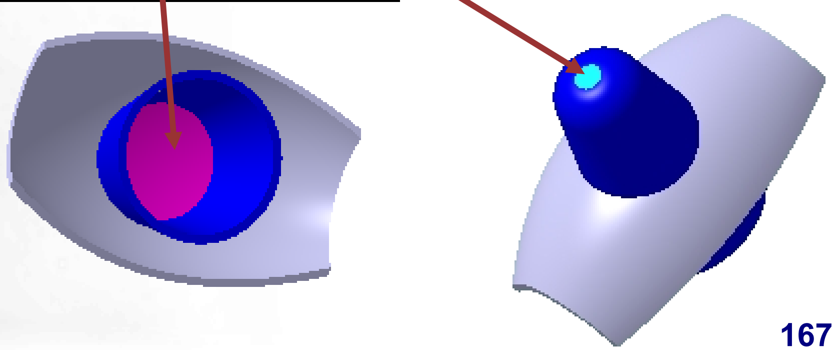


2 With the cursor on Body.2, select Union Trim from the contextual menu (MB3)

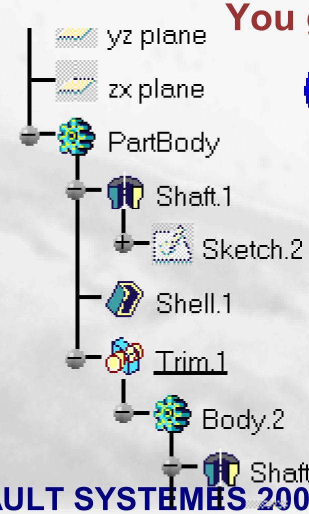


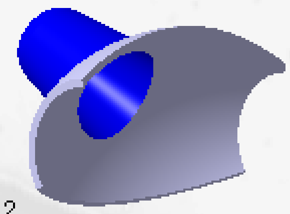
3 Select the Face to remove then the face to keep (Activate the corresponding field before in the dialog box)





You get:

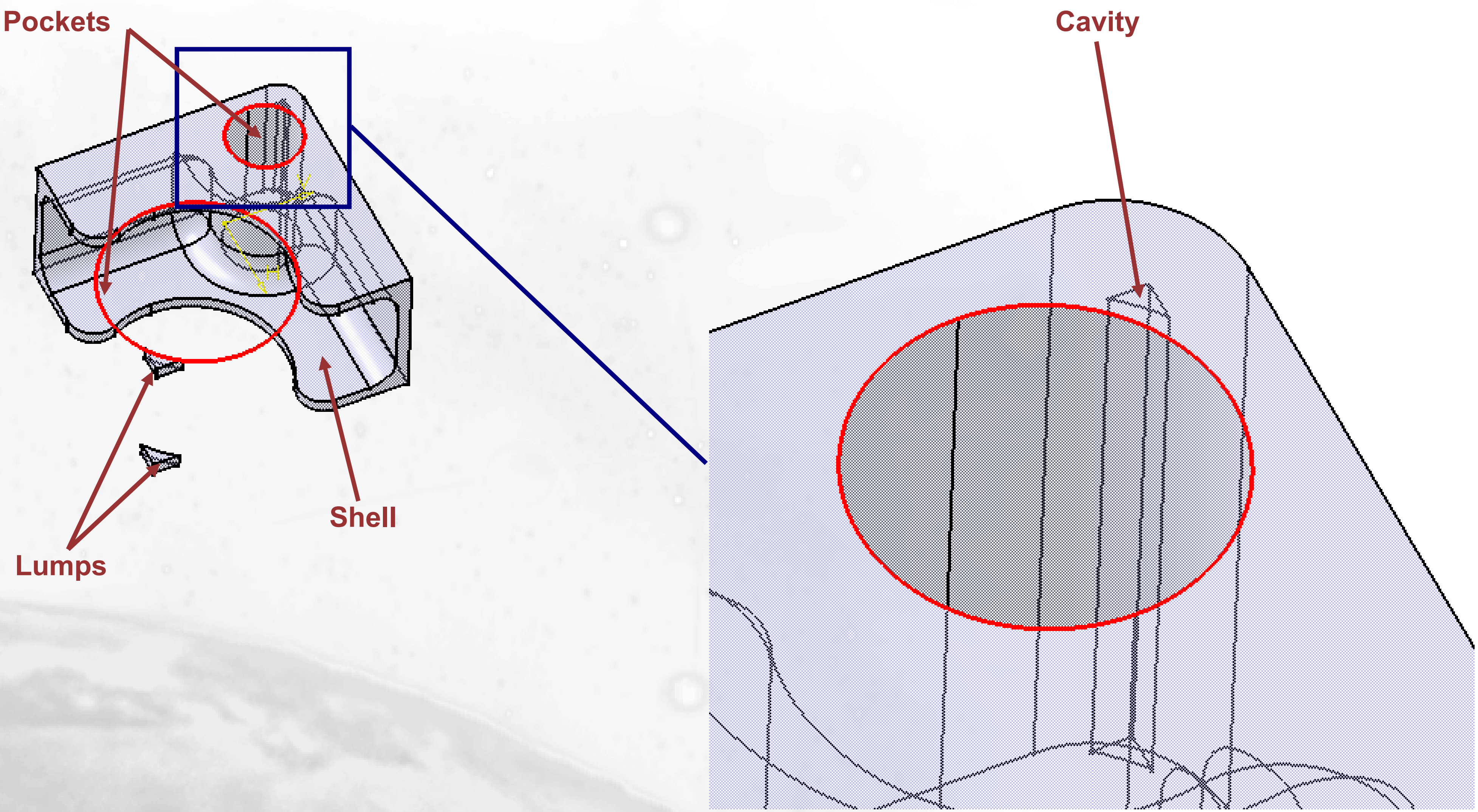




4 Select OK

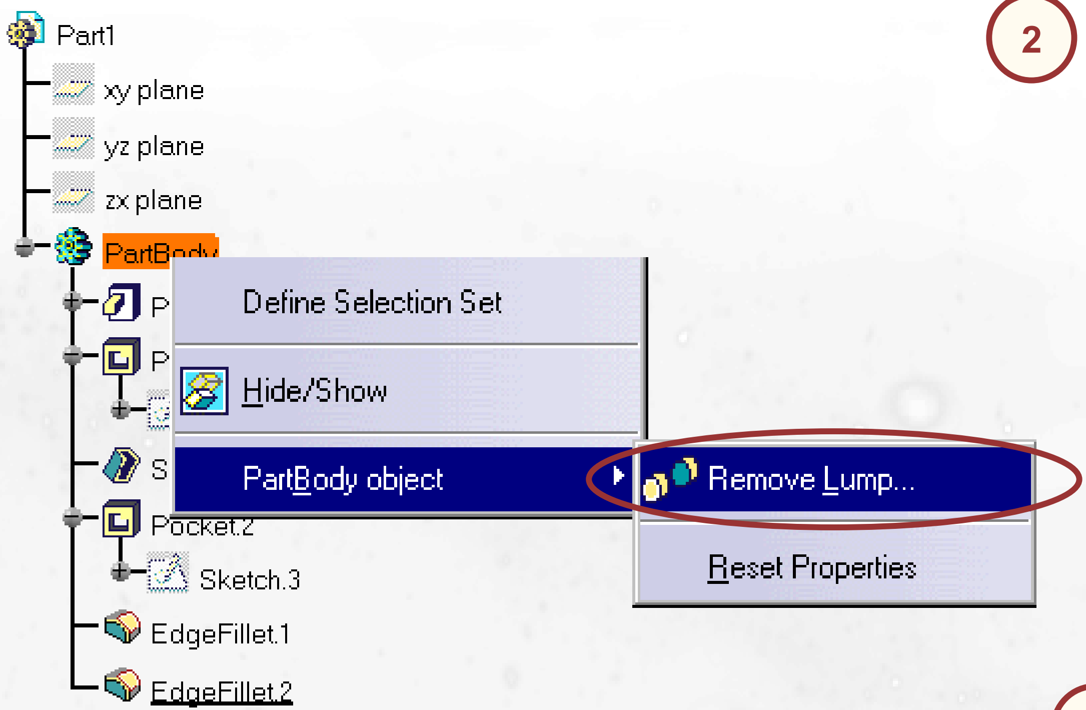
Removing Lumps (1/3)

After certain operations, it may happen that some Lumps or Cavities appear in the part. We need to remove them. The Remove Lump command allows you to remove Lumps and Cavities

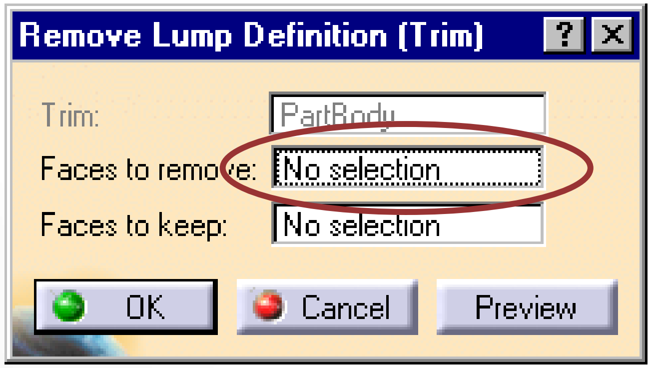


Removing Lumps (2/3)

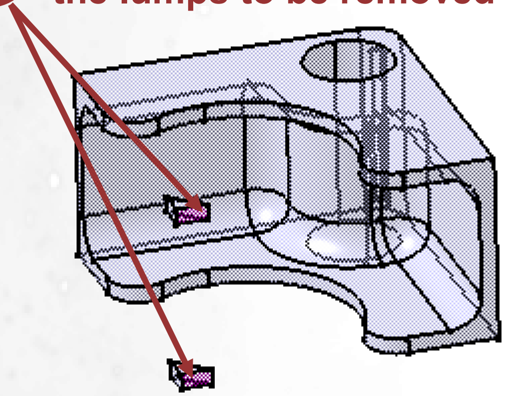
1 With the cursor on PartBody, select Remove Lump from the contextual menu (MB3)



2 Select the Faces to remove field in the dialog box

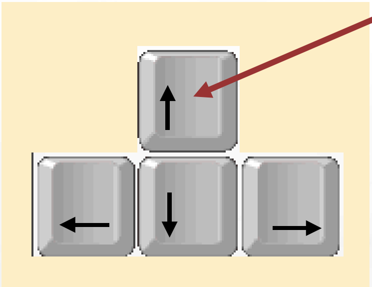


3 Select the two following faces belonging to the lumps to be removed

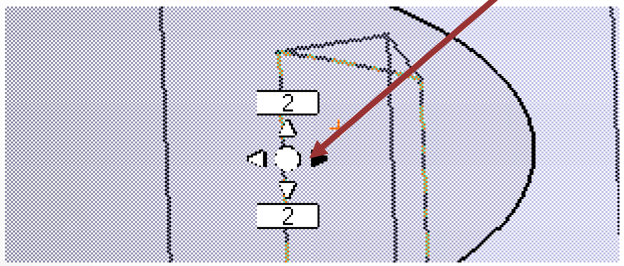


Removing Lumps (3/3)

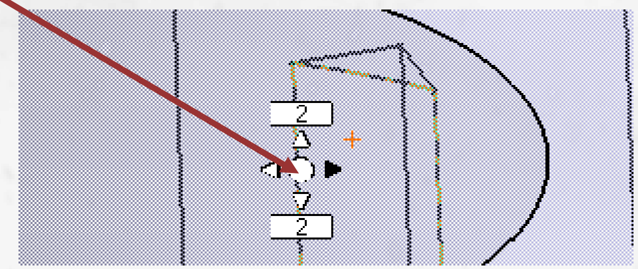
4 In order to select a face of the cavity, place the cursor on the cavity to be remove then press the Up arrow key on the keyboard



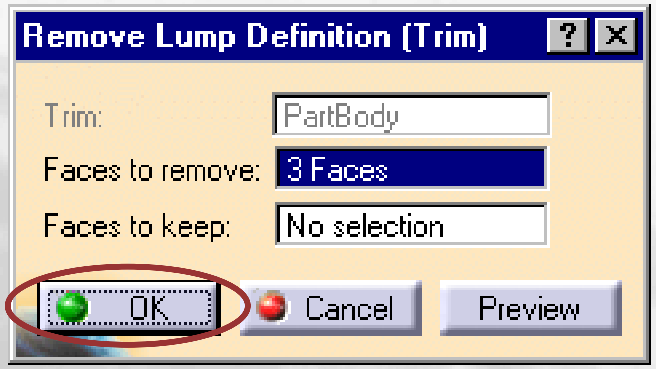
5 Using the small arrows, highlight one of the cavity face



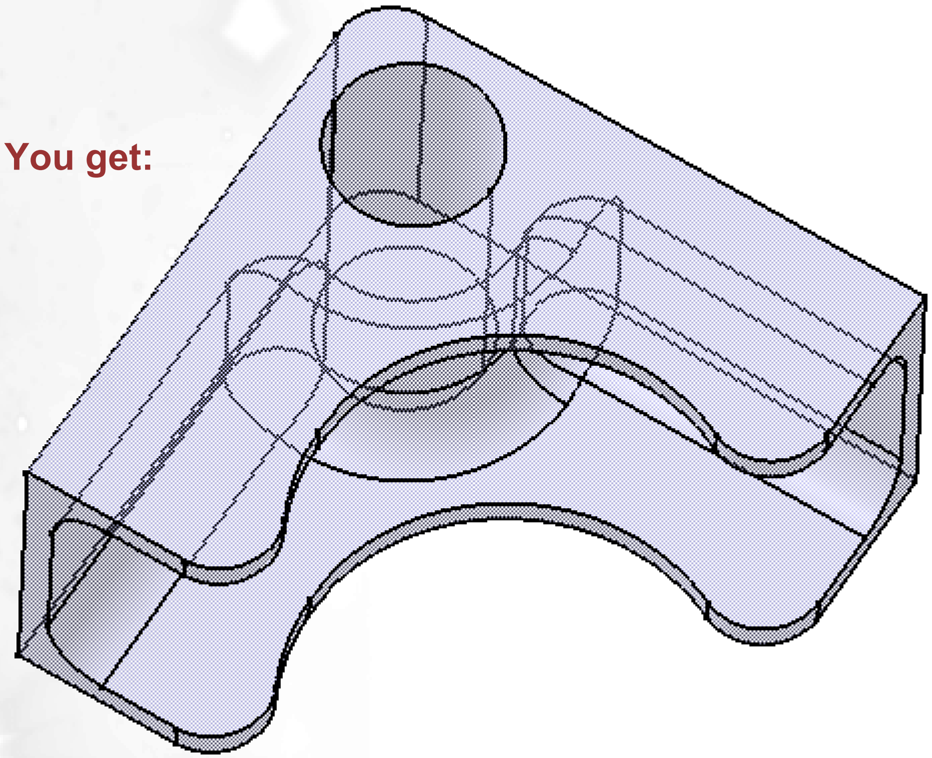
6 To confirm the face selection select the circle



7 Select OK

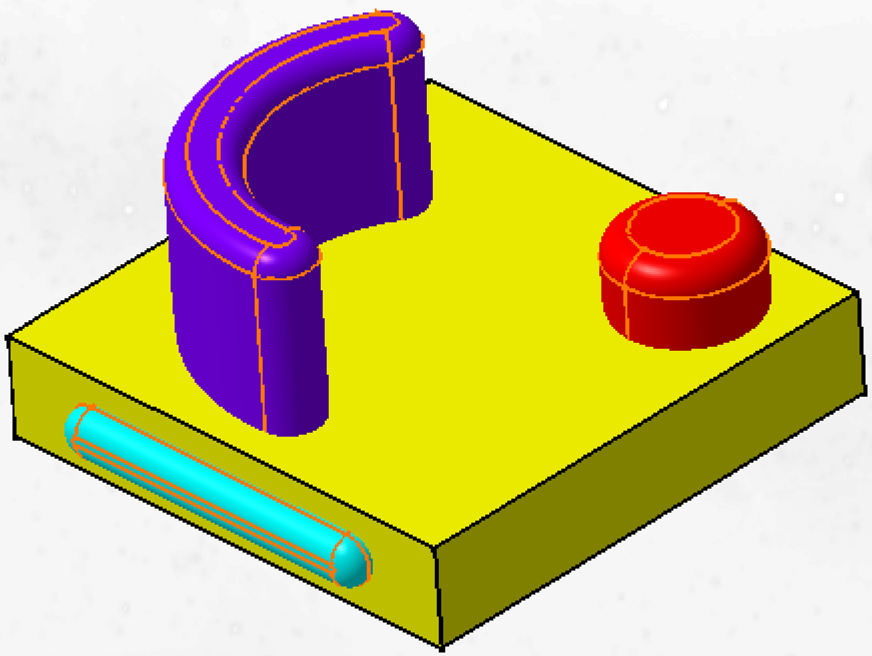


You get:

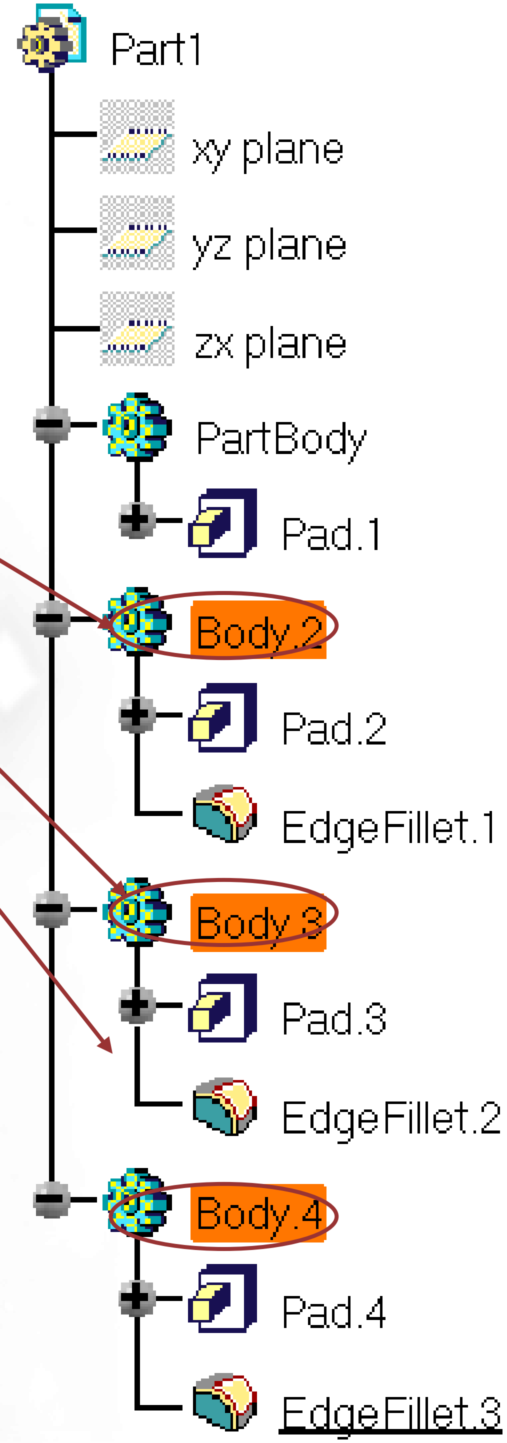


Assembling a Set of Bodies (1/3)

Assembling a set of bodies (Multi selected via the Ctrl key) is possible

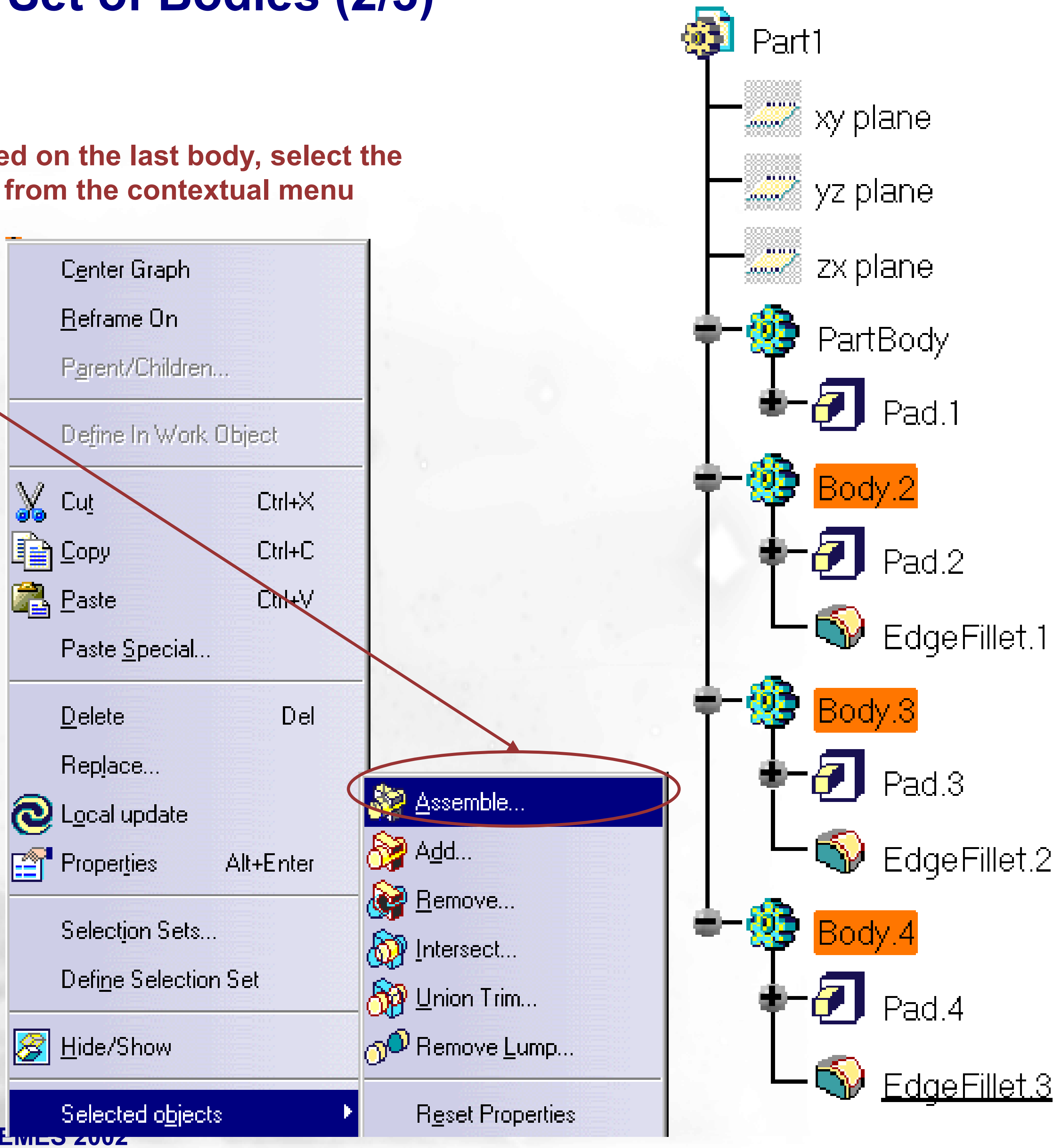


1 Using the Ctrl key, select the three following bodies to be assembled



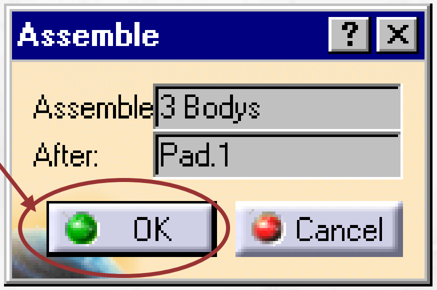
Assembling a Set of Bodies (2/3)

2 With the cursor placed on the last body, select the Assemble command from the contextual menu

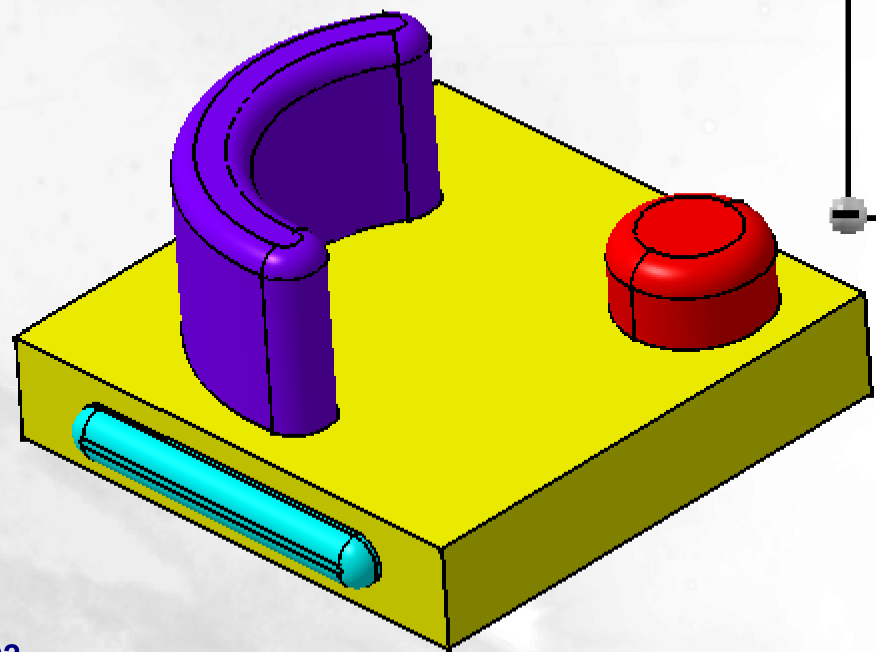


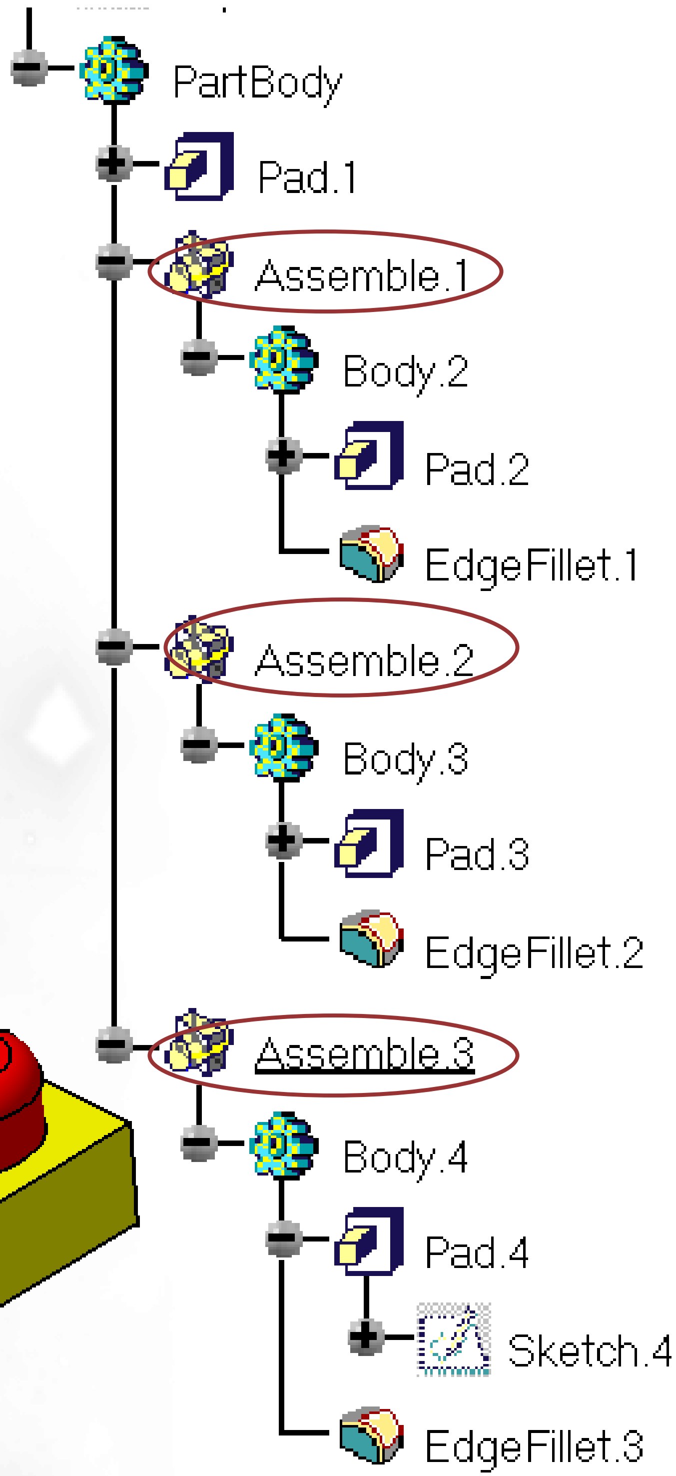
Assembling a Set of Bodies (3/3)

3 **Select OK in the dialog box**



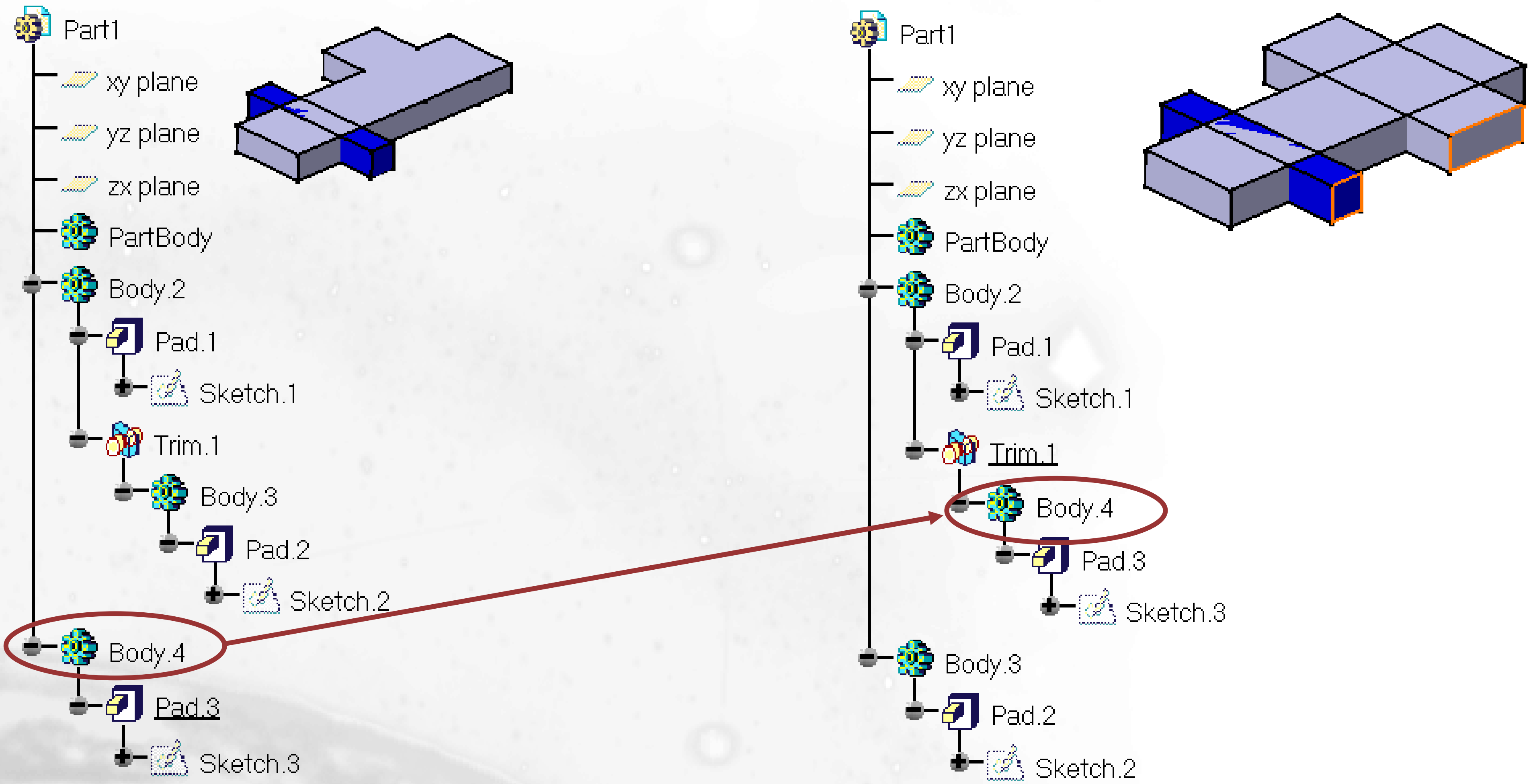
You get:



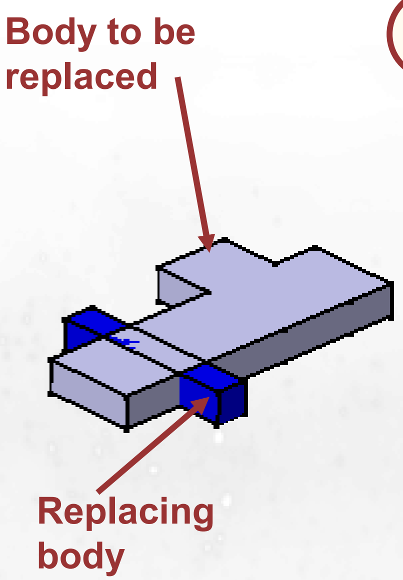


What is Replacing a Body?

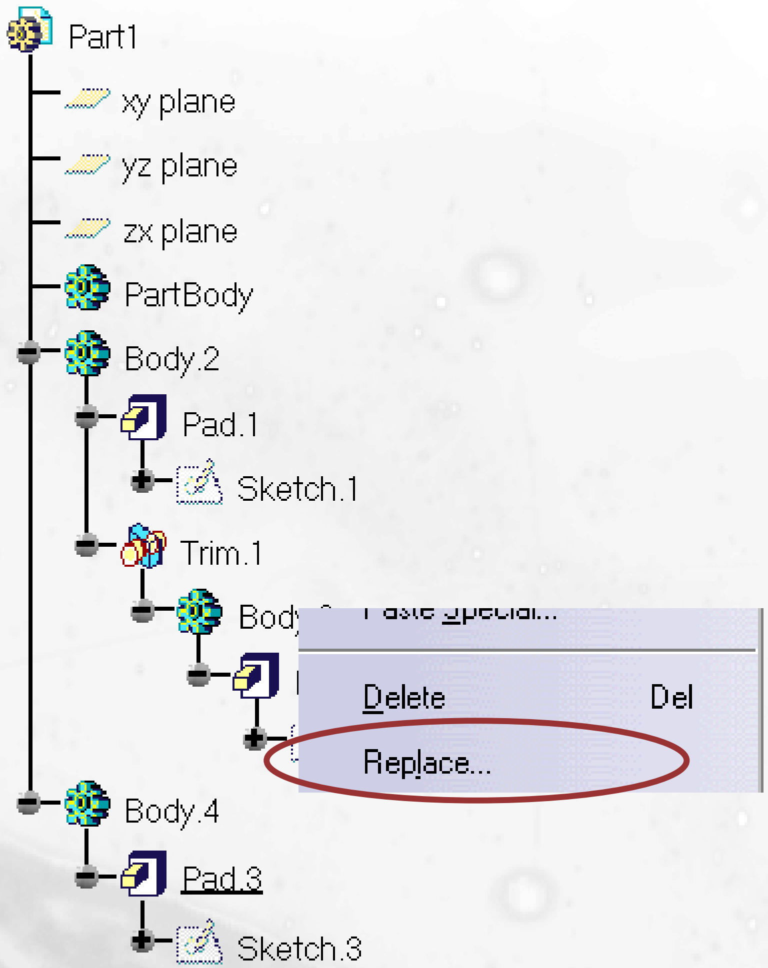
You can replace a body use in an operation by another one



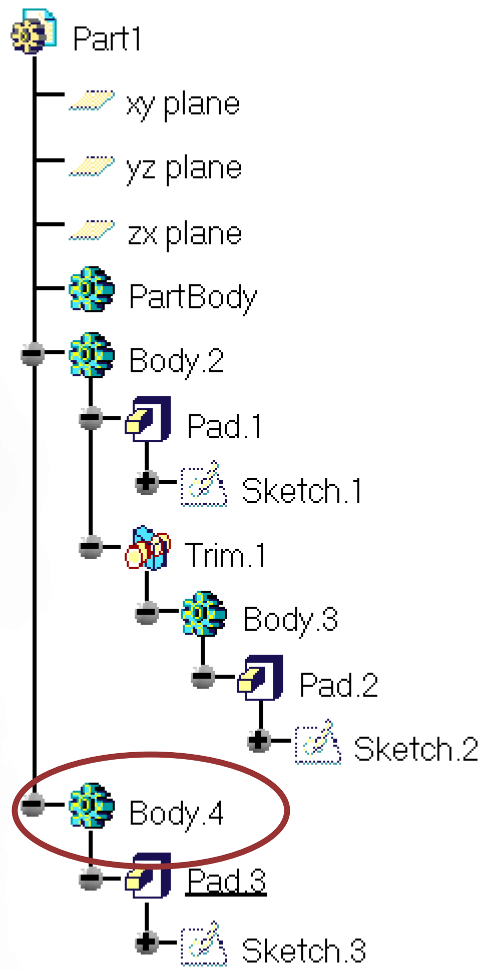
Replacing a Body (1/3)



1 Select the Replace command from Body.3 contextual menu

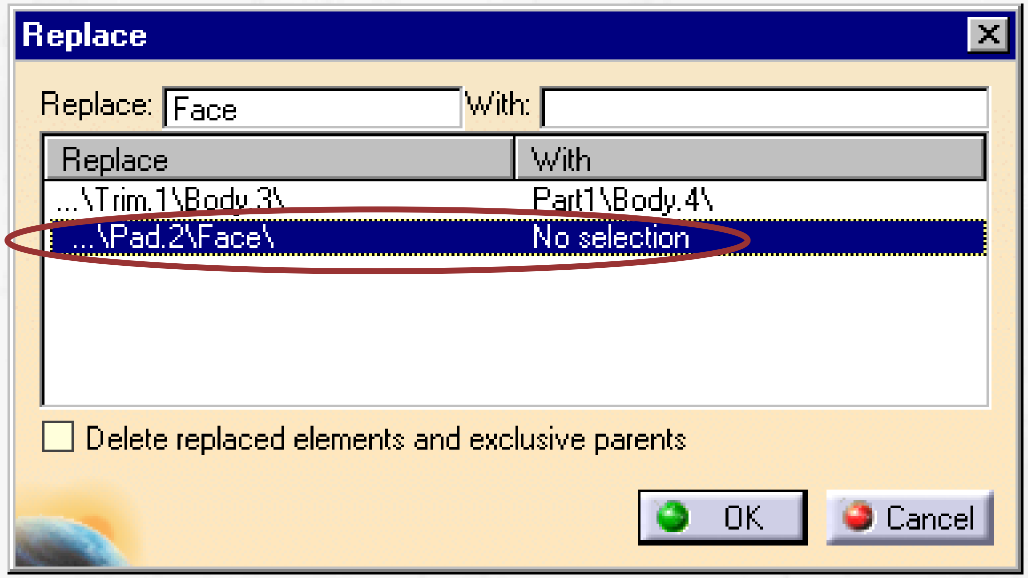


2 Select Body.4

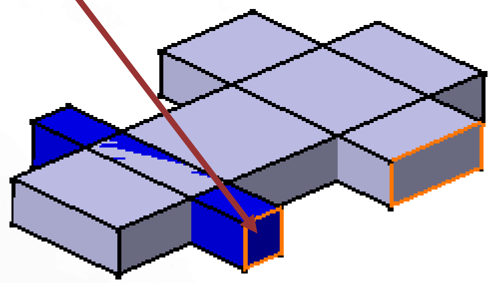


Replacing a Body (2/3)

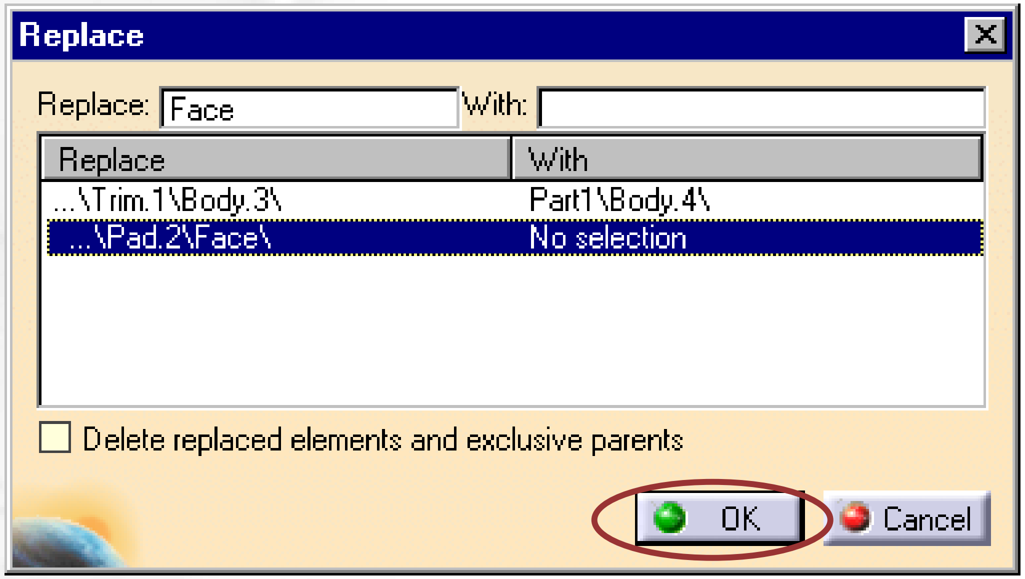
3 Select the following line in the dialog box



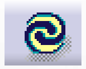
4 Select the following face. This face is the face that will be removed during the Union Trim operation

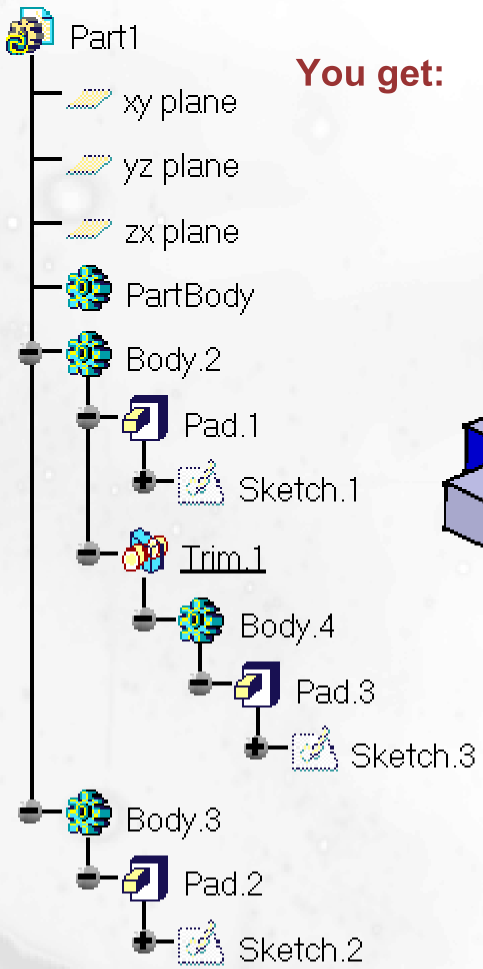


5 Select OK

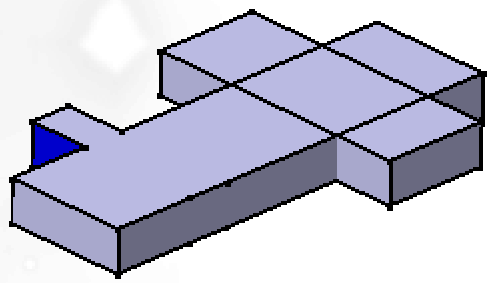


Replacing a Body (3/3)

6 If necessary, update the part by selecting the Update All icon 

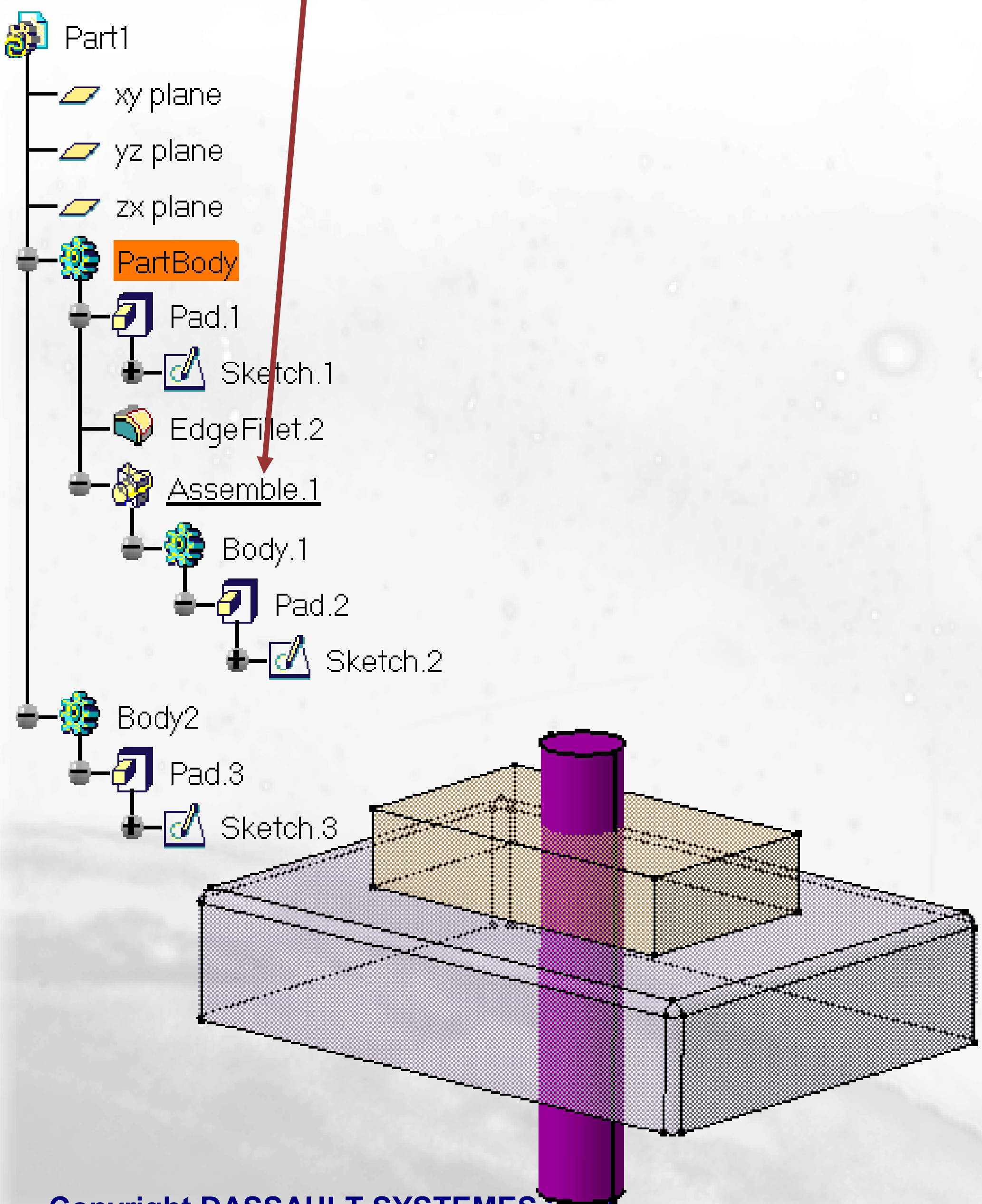


You get:

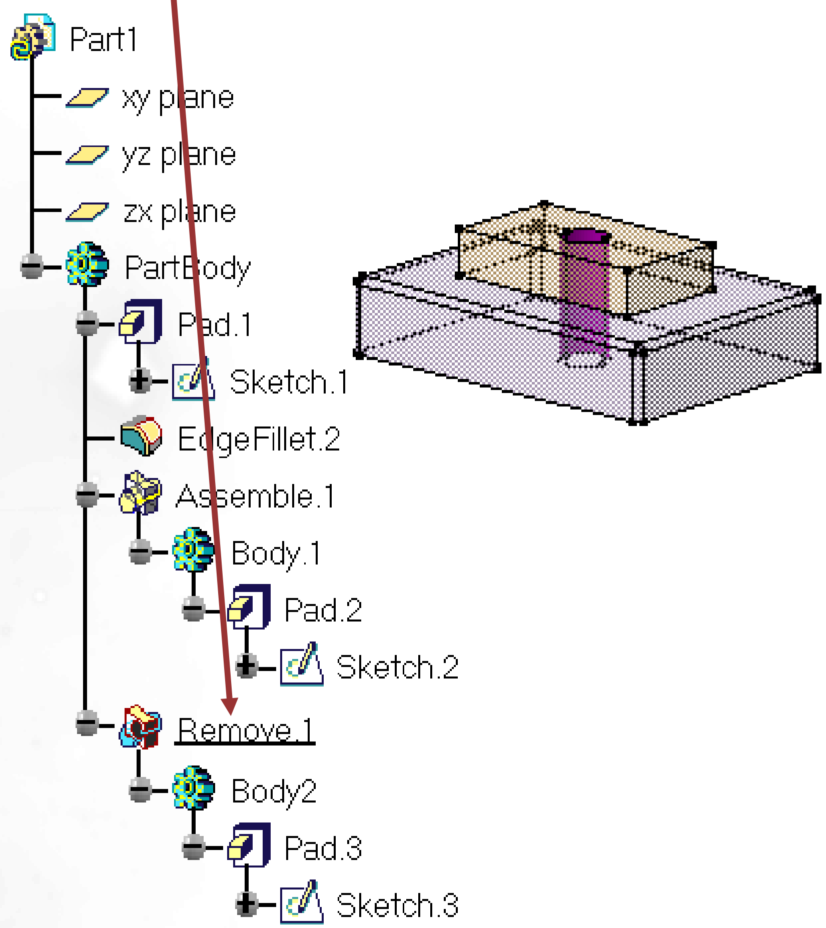


Change Boolean Type (1/4)

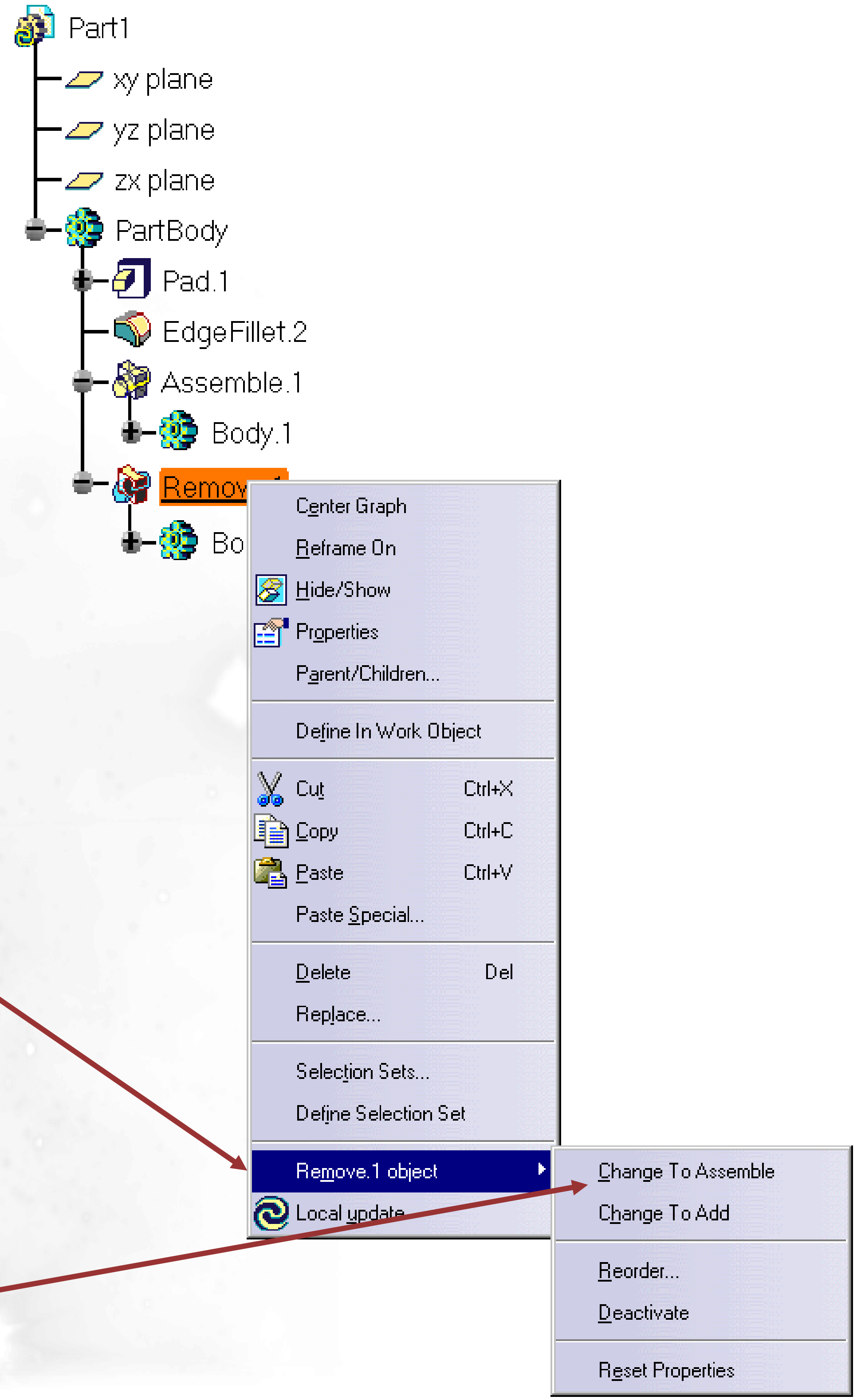
1 **The initial part is composed of three bodies. *Assemble* Body.1 to Part Body.**



2 ***Remove* Body.2 from Assemble.1.You obtain Remove.1.**



Change Boolean Type (2/4)

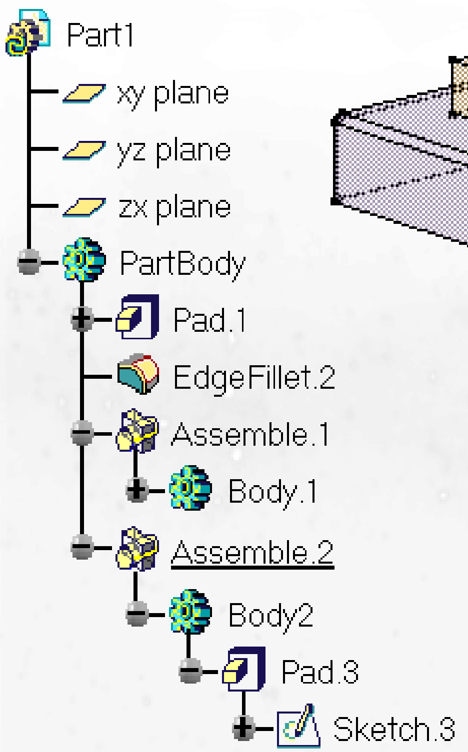


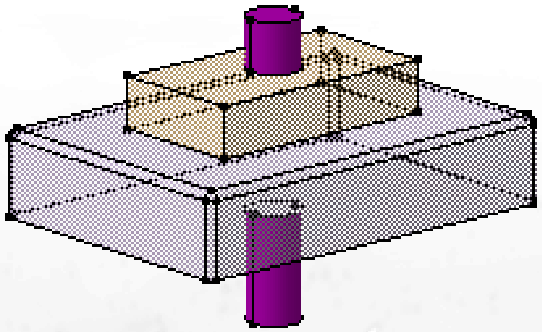
3 Click with the right button mouse on Remove.1. In the contextual menu, select Remove.1 object

4 Choose now the new operation. For example, click on *Change To Assemble.*

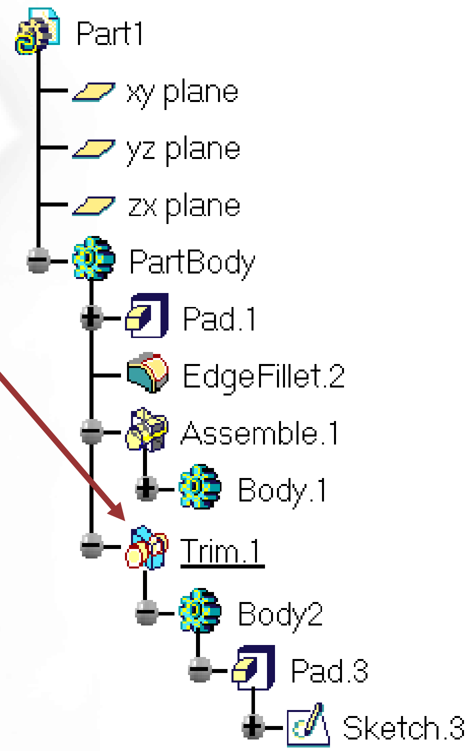
Change Boolean Type (3/4)

5 **You obtain :**



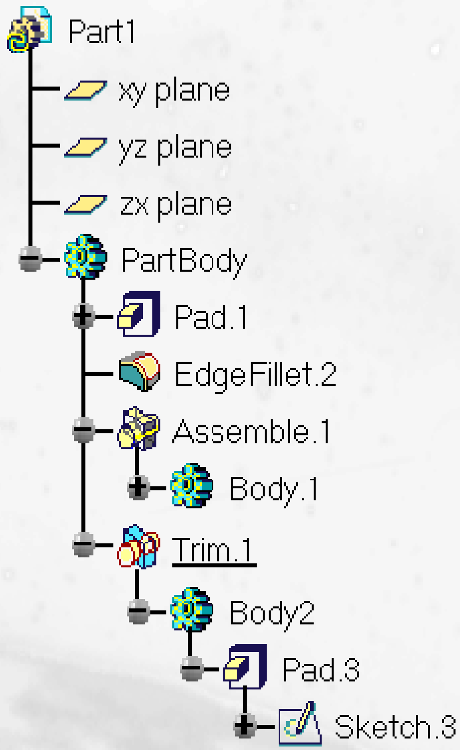


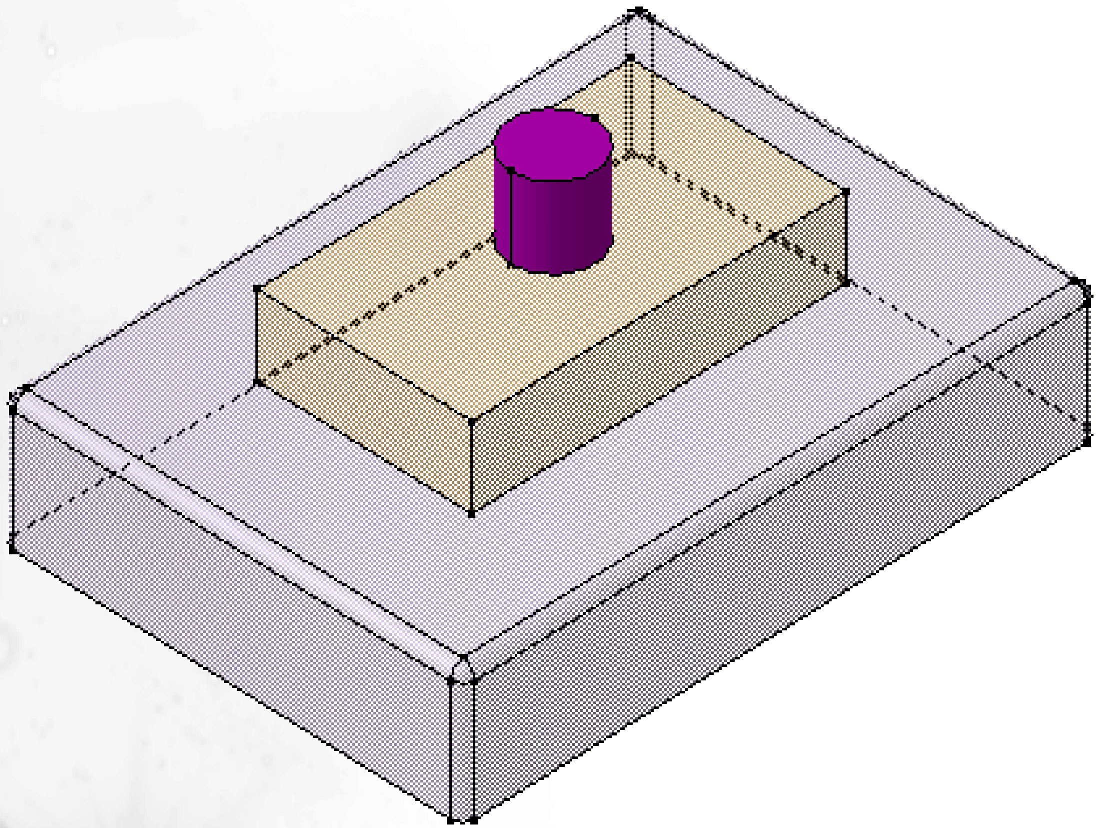
6 **Change now *Assemble.2* to *Union Trim.* You obtain :**



Change Boolean Type (4/4)

7 You can edit *Trim.1*. For instance, select the cylinder's top face as the face to keep. You obtain :





Multi-Model Links

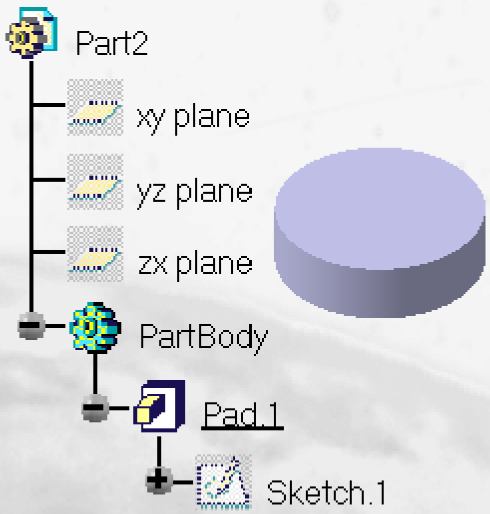
You will learn ways to use Multi-model links to help propagate design changes

- Establishing Multi-Model Links

What are Multi-Model Links ?

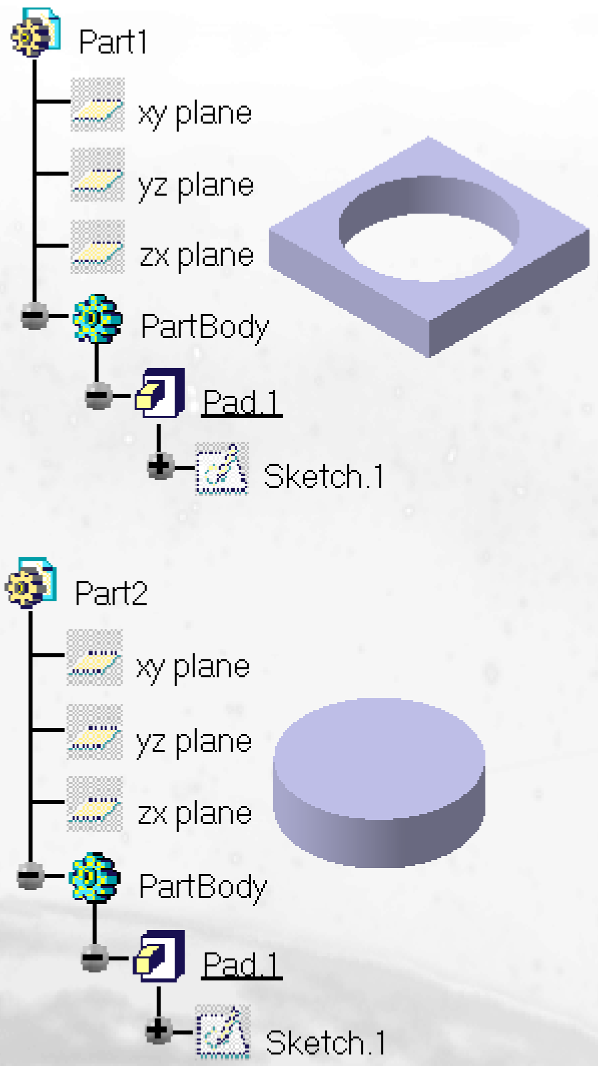
The concept of working within an independent "Body" and then having the ability to Add, Remove, or Intersect this Body with our "Master" PartBody gives us this added modeling flexibility

There are different ways that the independently modeled Body can be assimilated into the PartBody

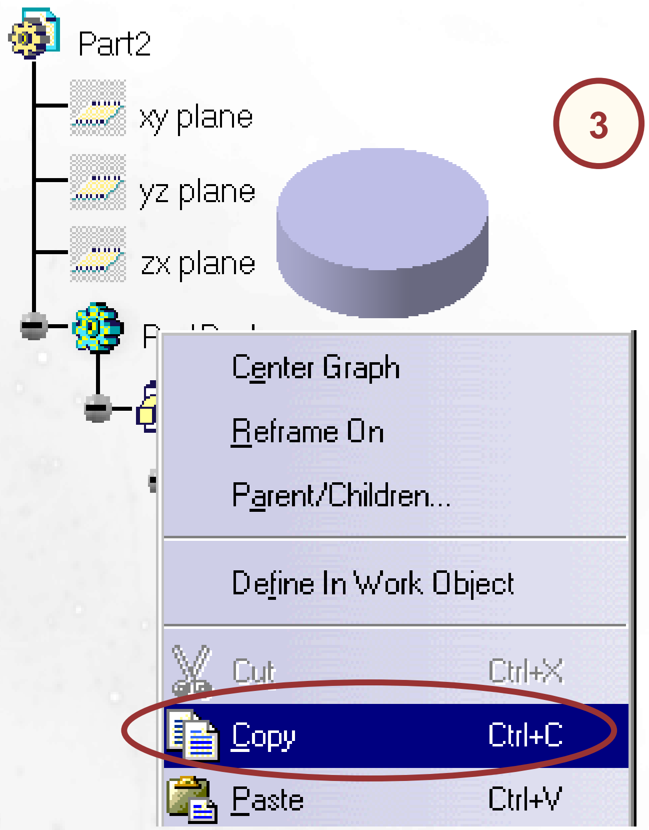


Establishing Multi-Model Links (1/3)

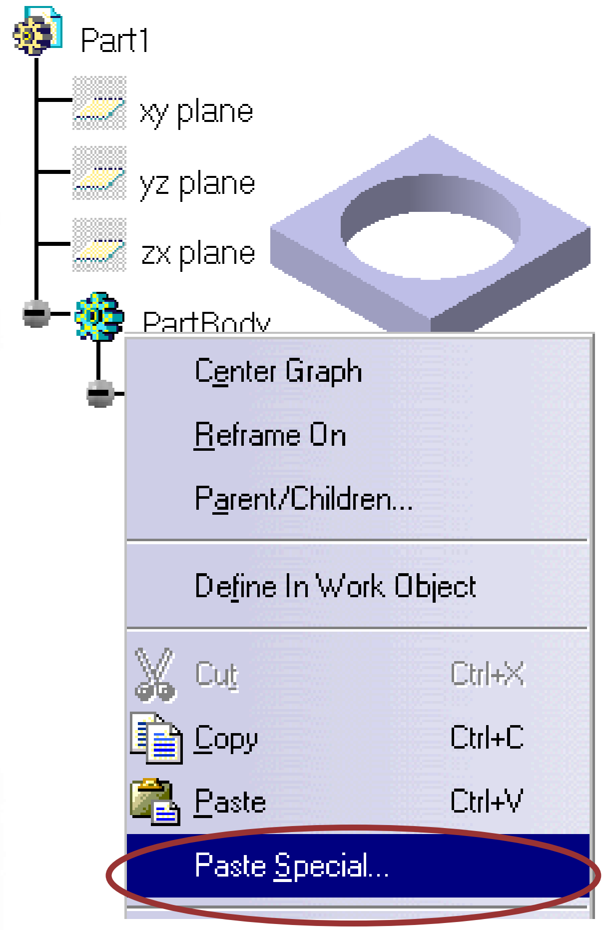
1 In a CATIA session you have two separate parts



2 Using the Contextual Menu, copy the PartBody of Part2

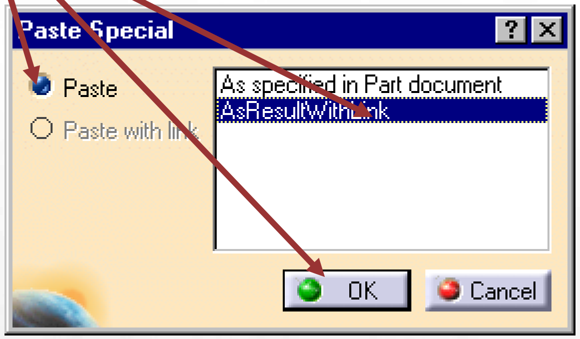


3 Place the cursor on the PartBody of Part1 then Select Paste Special from the contextual menu

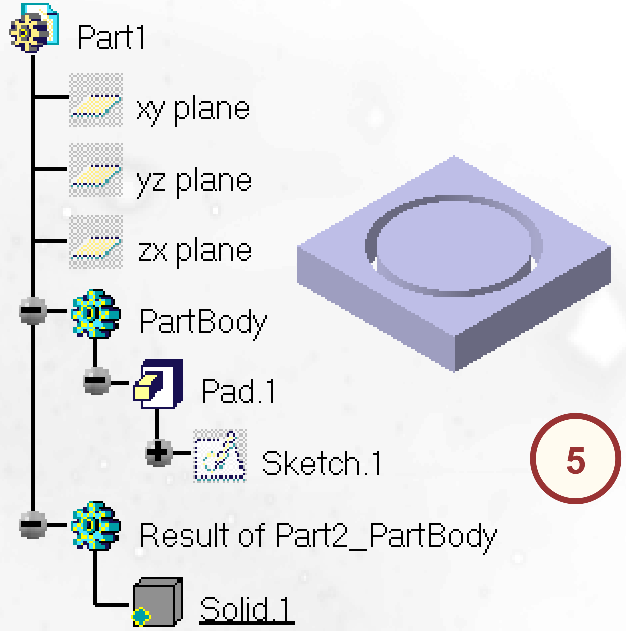


Establishing Multi-Model Links (2/3)

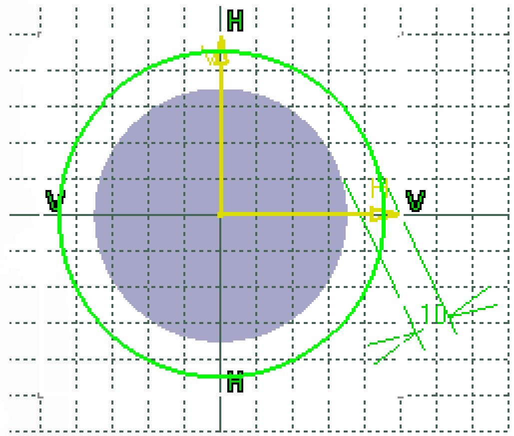
3 In the dialog box, select AsResultWithLink and the Paste button, then select OK



4 Part1 becomes:

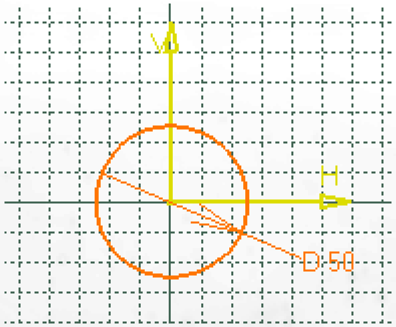


5 In Sketch.1 of part1, create a distance (10mm) between the circle and the copied cylinder then exit the sketcher

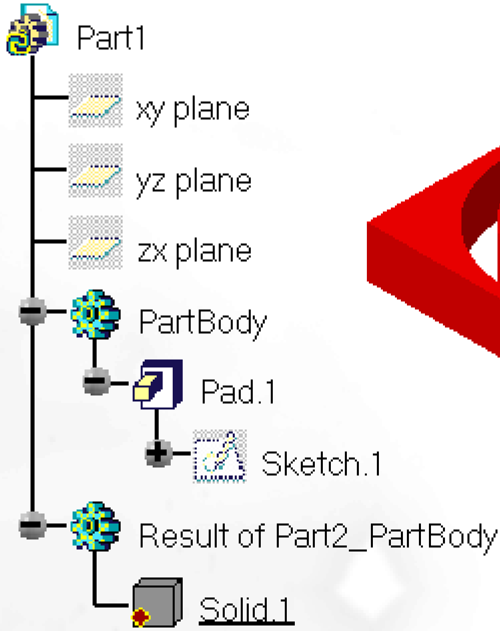


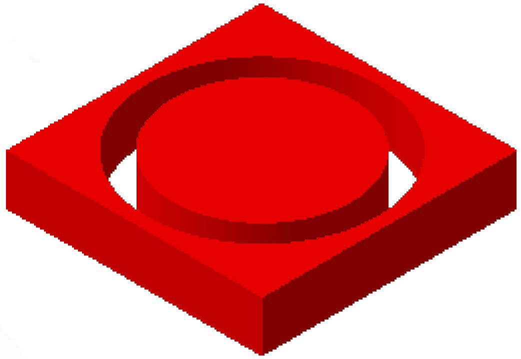
Establishing Multi-Model Links (3/3)

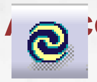
6 Now, in Sketch.1 of part2, create a diameter constraint of 50 then exit the sketcher



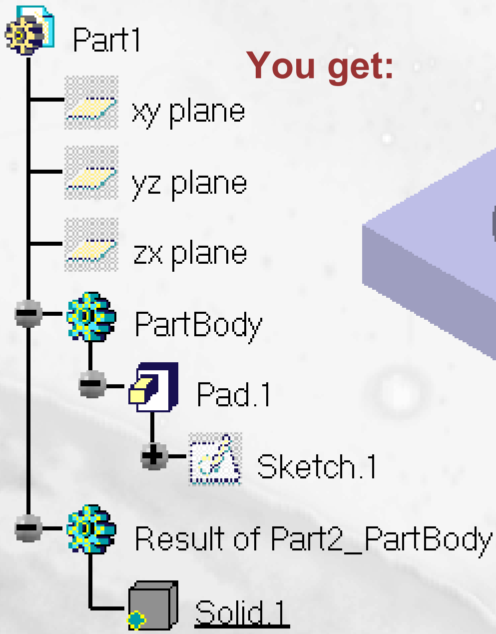
7 Part1 becomes:

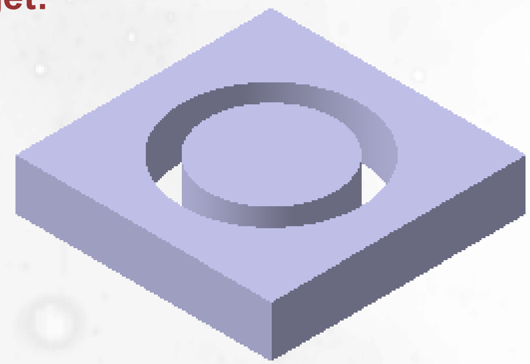




8 With Part1 active, select the Update icon 

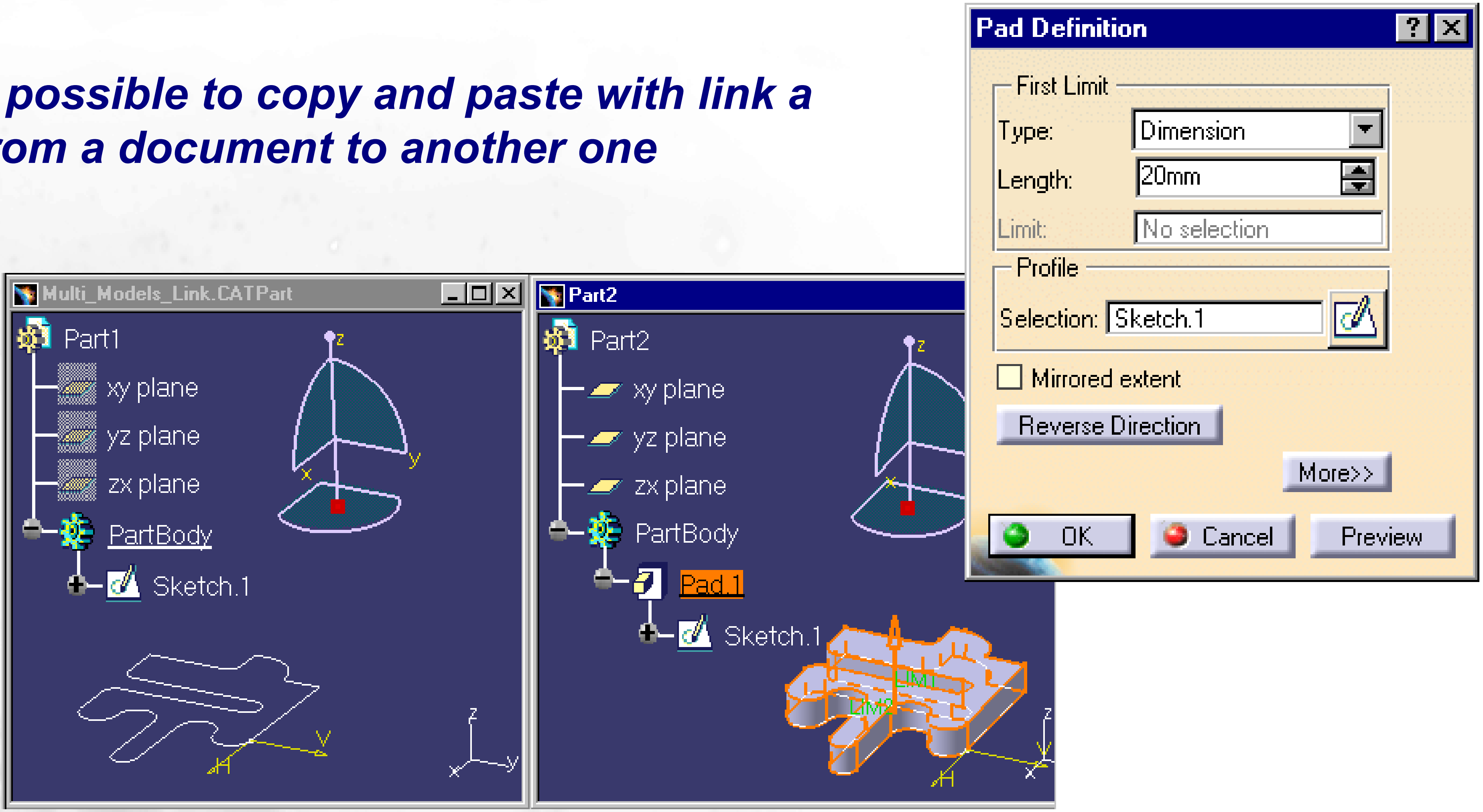
You get:





Sketch Selection with Multi-Documents Links

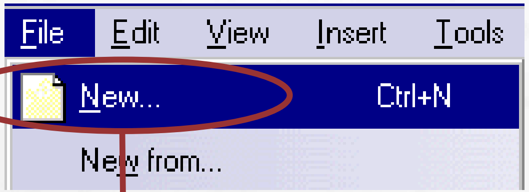
It is now possible to copy and paste with link a sketch from a document to another one

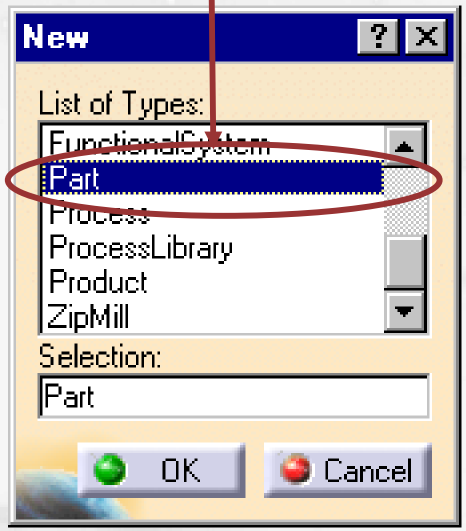


Sketch Selection with Multi-Documents Links (1/5)

You can copy a sketch in a document then paste it into another document keeping the link with the first one. You can use this copied sketch and in case of modification of the original sketch the document in which the copy is used will be also modified

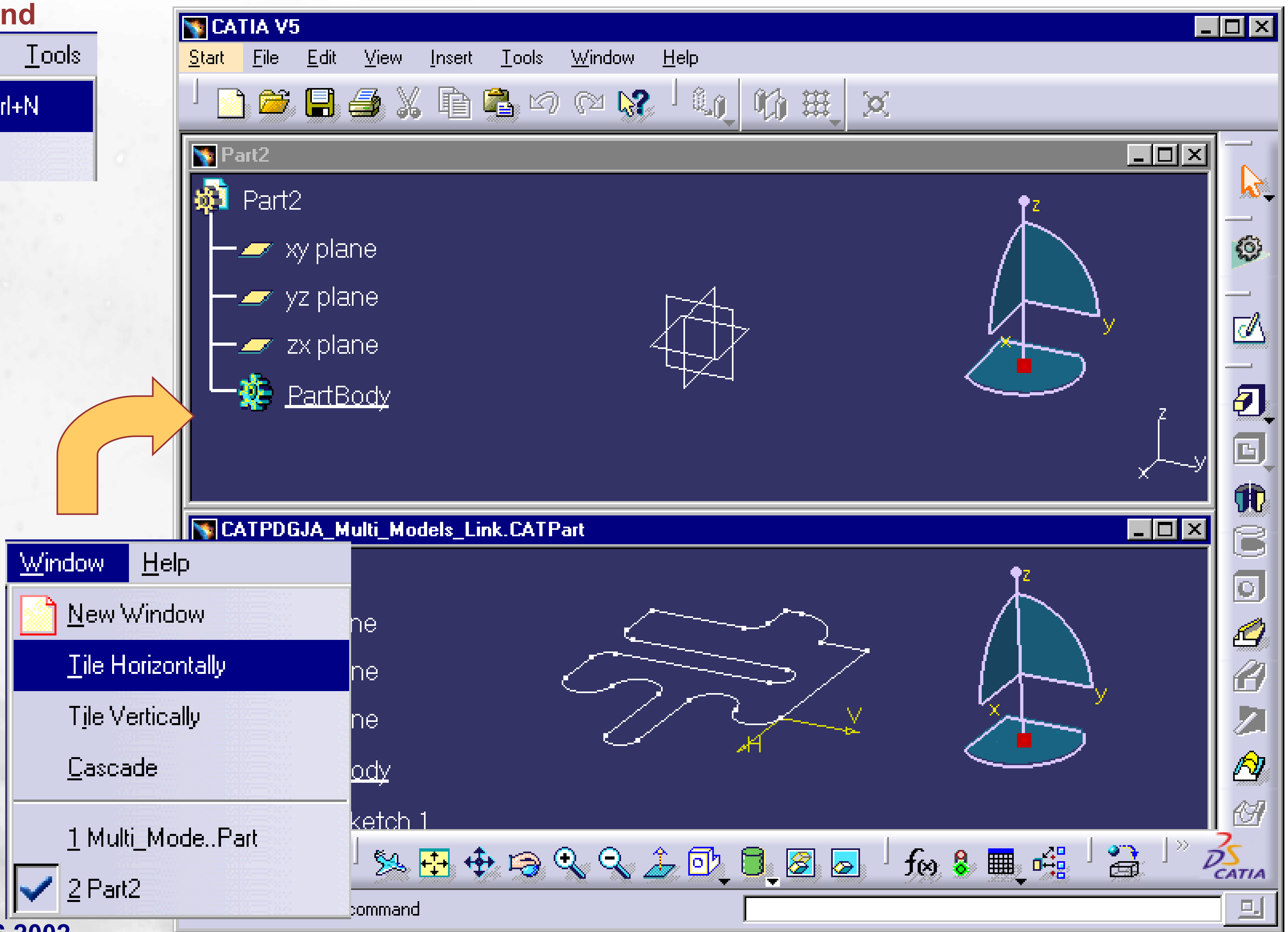
1. After loading a part containing a sketch, start a new part using the File + New command





2. Display the two parts using the Window + Tile Horizontally command

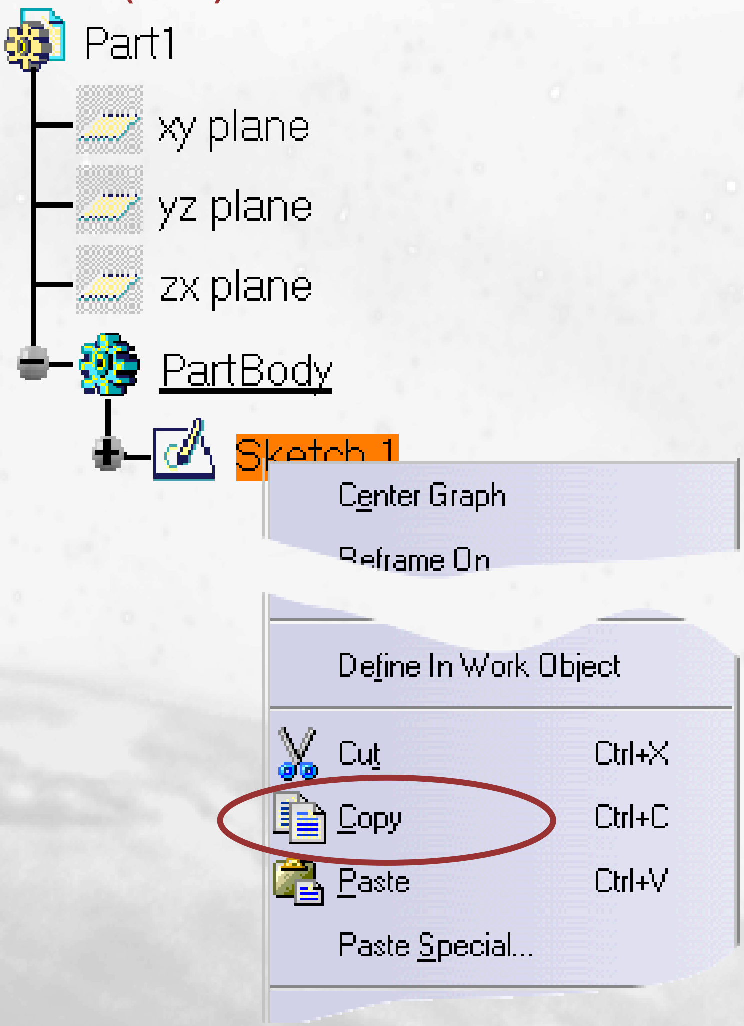
You get:



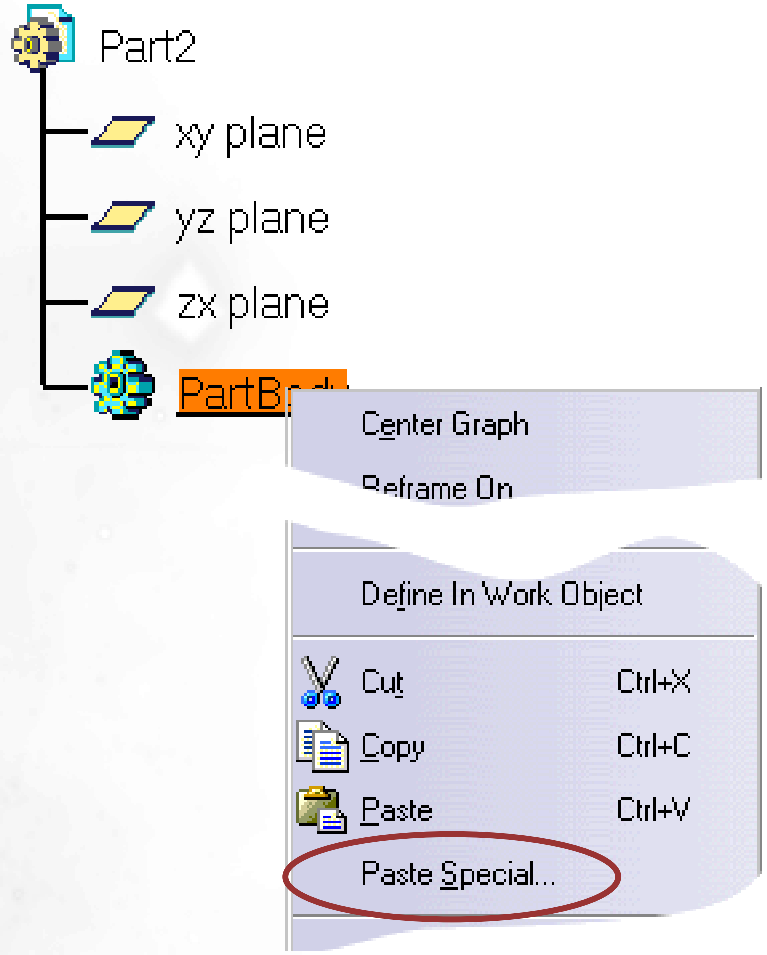
Sketch Selection with Multi-Documents Links (2/5)

You can copy a sketch in a document then paste it into another document keeping the link with the first one. You can use this copied sketch and in case of modification of the original sketch the document in which the copy is used will be also modified

3 **With the cursor on Sketch.1 in the tree, select the Copy command from the contextual menu (MB3)**



4 **In the second part, place the cursor on PartBody, then select Paste Special from the contextual menu (MB3)**

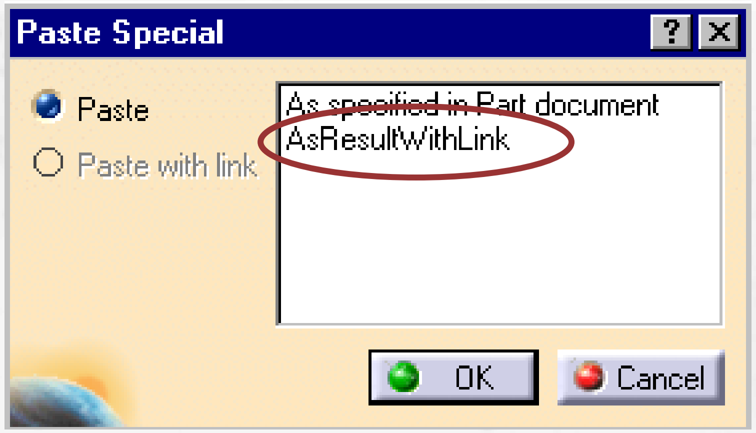


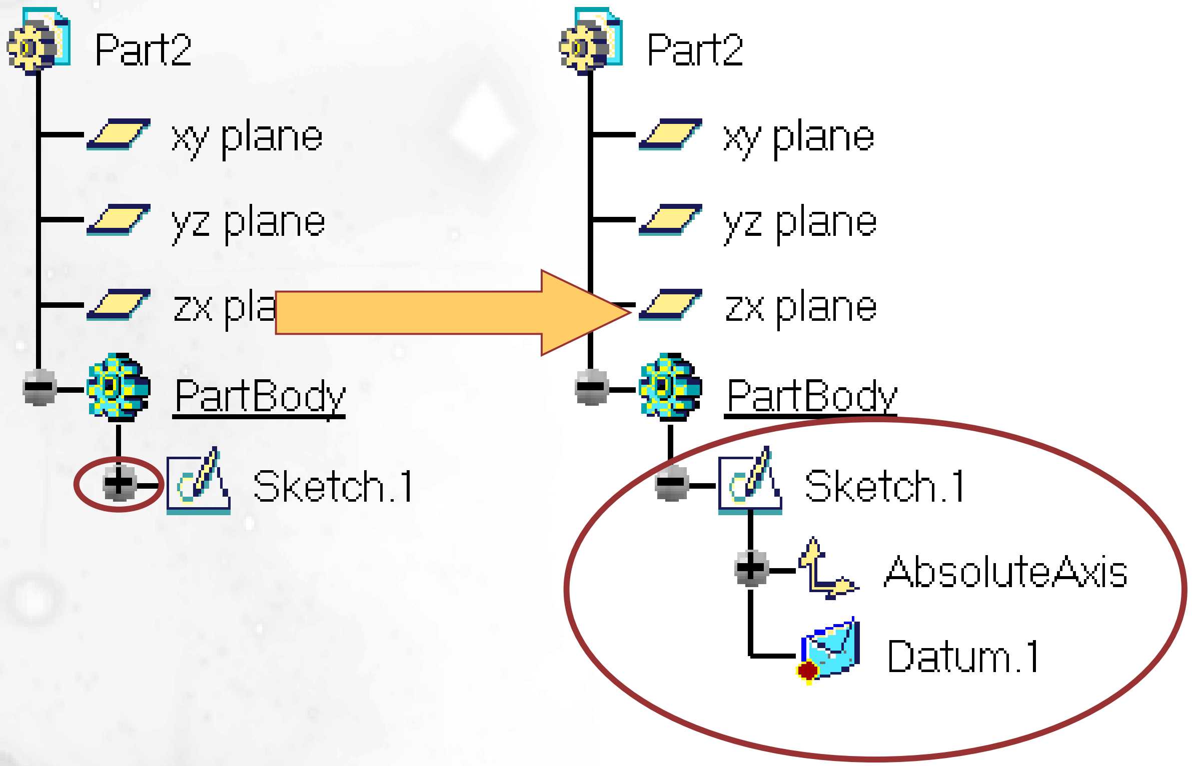
Sketch Selection with Multi-Documents Links (3/5)

You can copy a sketch in a document then paste it into another document keeping the link with the first one. You can use this copied sketch and in case of modification of the original sketch the document in which the copy is used will be also modified

5 **Select AsResultWithLink in the dialog box**

6 **Expand Sketch.1 in order to see what has been copied (by selecting +)**

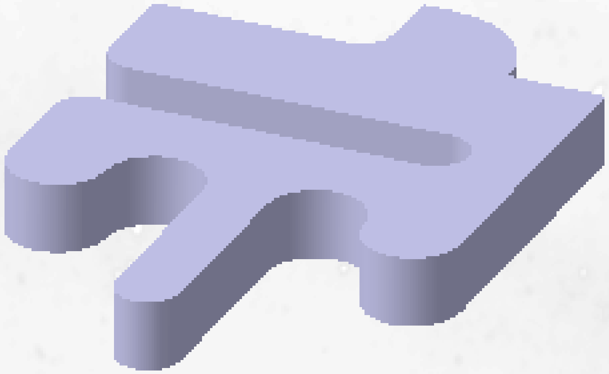




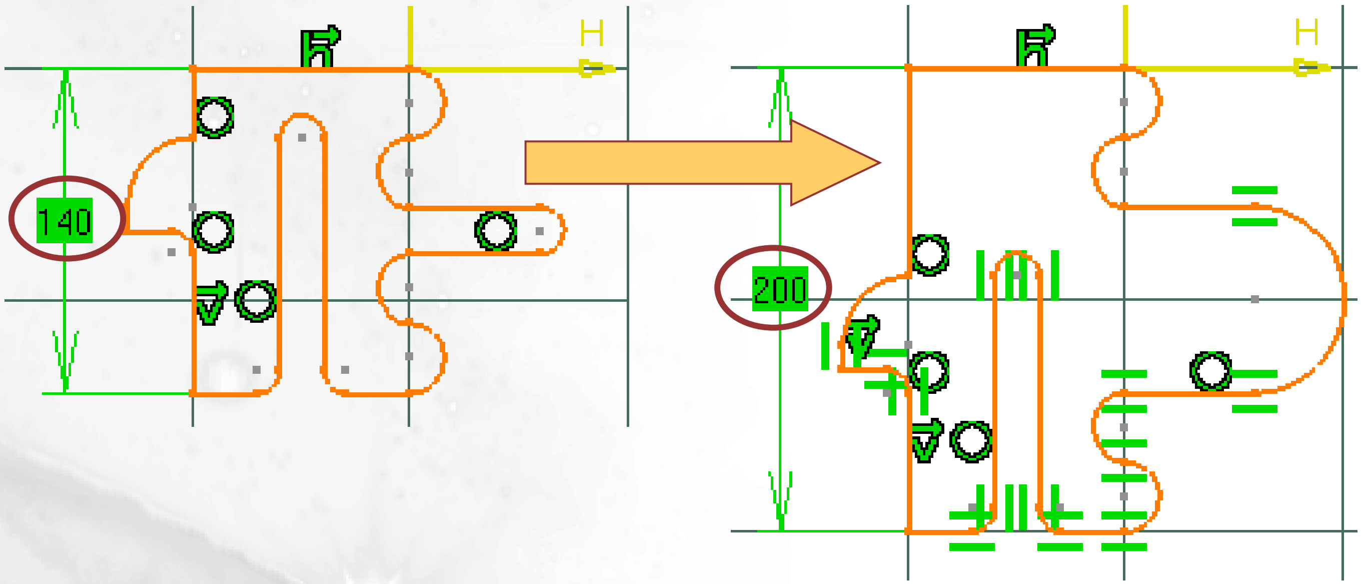
Sketch Selection with Multi-Documents Links (4/5)

You can copy a sketch in a document then paste it into another document keeping the link with the first one. You can use this copied sketch and in case of modification of the original sketch the document in which the copy is used will be also modified

7 **Create a 20mm height pad using the copied sketch**



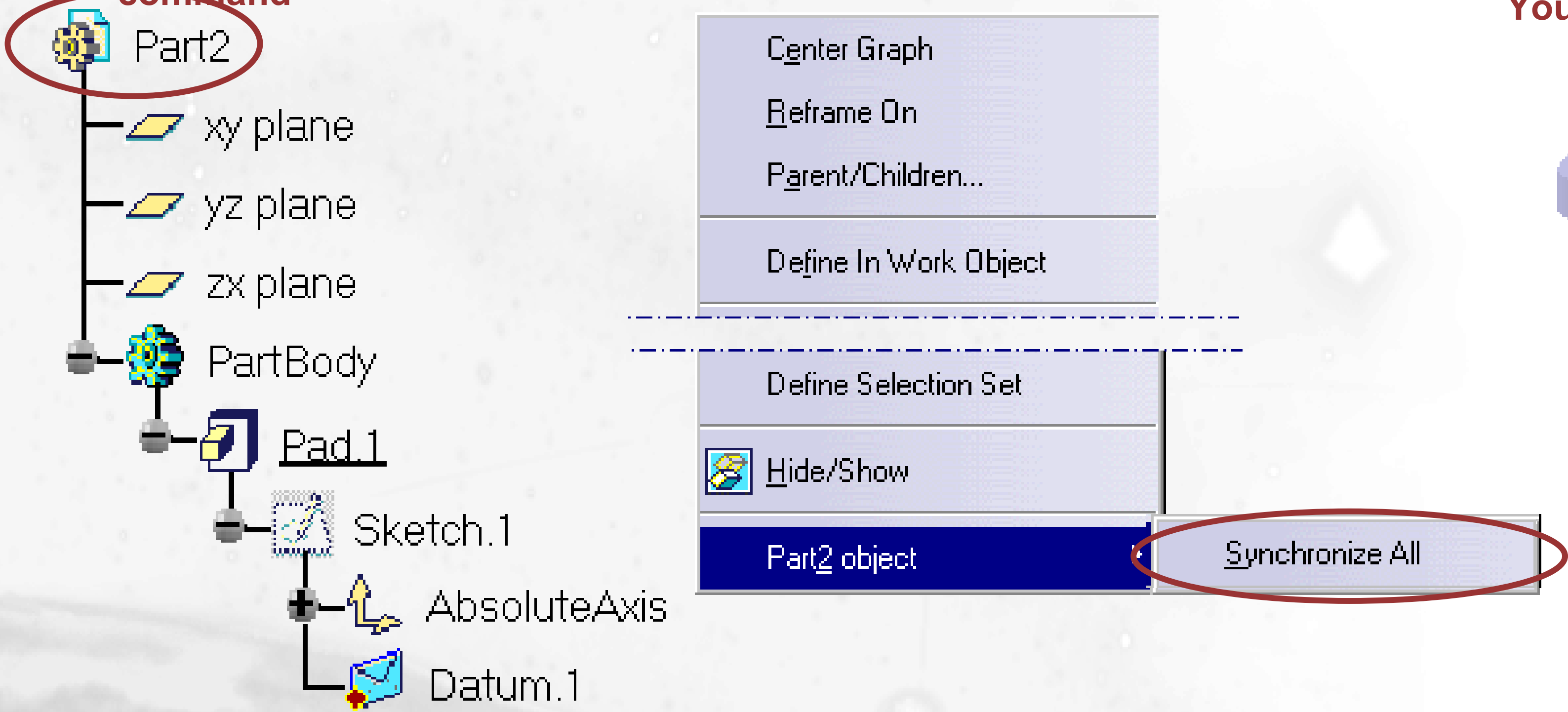
8 **In the first part, modify the sketch as follows**



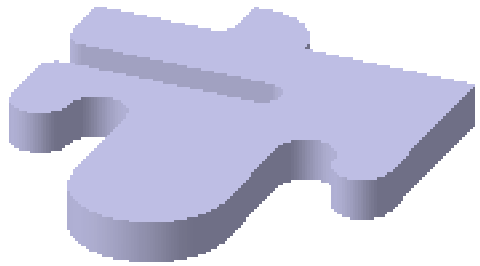
Sketch Selection with Multi-Documents Links (5/5)

You can copy a sketch in a document then paste it into another document keeping the link with the first one. You can use this copied sketch and in case of modification of the original sketch the document in which the copy is used will be also modified

9 To take the modification into account in the second part (the one which contains the copied sketch), place the cursor on Part2 then select the Part2object + Update All Links command

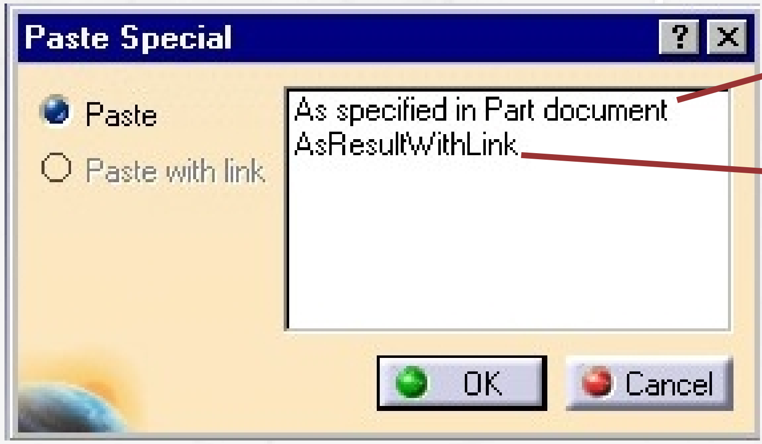


You get:



Additional Information

The different Paste Special options:



As specified in Part document: The copied element can be modified and has no link with the original one. The original element is duplicated

AsResultWithLink: The copied element cannot be modify (it is a datum)but in case of modification of the original element, the copied one is updated

Scaling

You will learn how to apply an affinity to a part with reference to a point

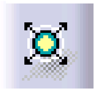
- Scaling/Affinity

What is Scaling ?

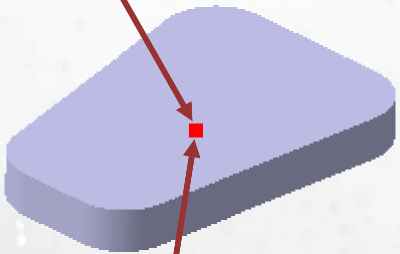
A scaling is a part transformation which is calculated by selecting a reference point and by entering a ratio

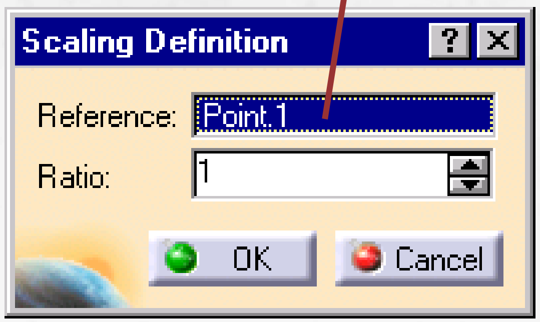
The system computes the distance between all the points of the outer skin of the part and the reference point, then these distances are multiplied by the ratio to get the new distances between the reference point and all the point of the new outer skin

Scaling/Affinity

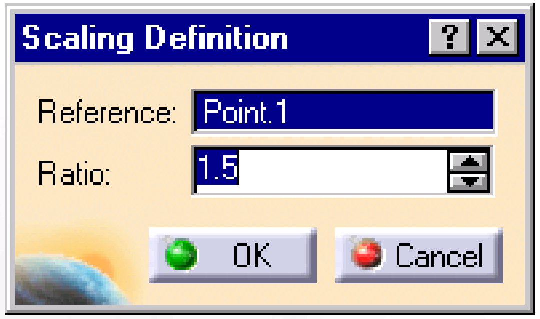
1 Select the Scaling icon 

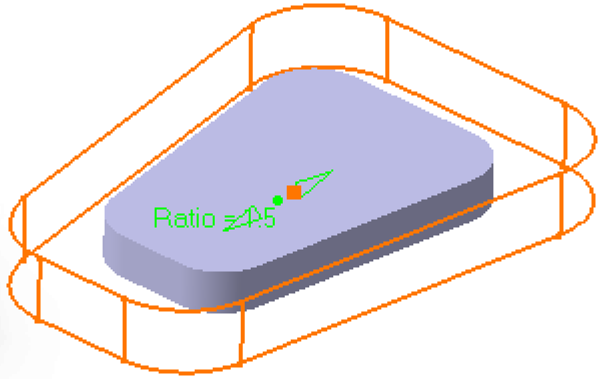
2 Select the reference point

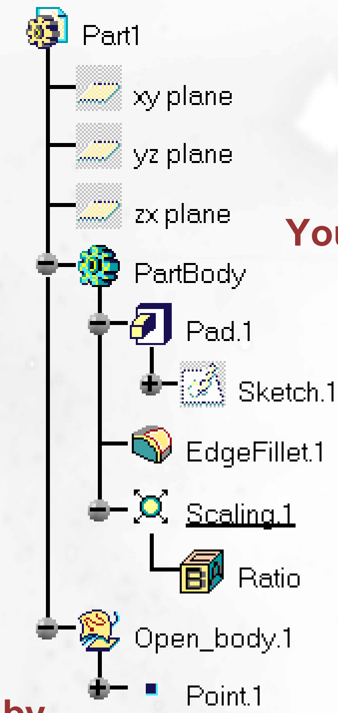




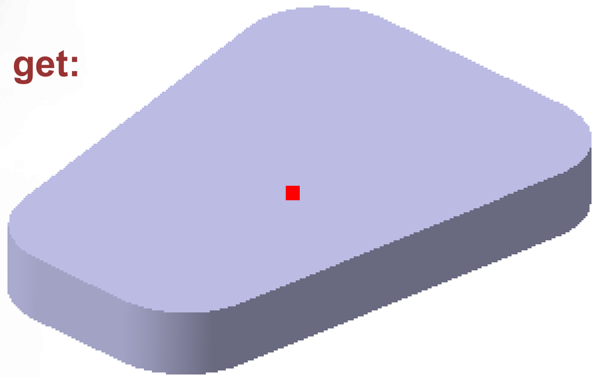
3 Modify scaling ratio then select OK







You get:



You can also resize a body in relation to a face or plane by selecting it instead of a reference point. The body will scale with it. You will obtain an affinity