

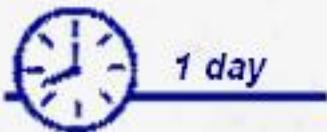
Course Presentation

Objectives of the course

In this course you will learn methods to create and modify sheet metal features and parts.

Targeted audience

New Catia V5 Users



Prerequisites

CATIA V5 Fundamentals
CATIA V5 Part Design
CATIA V5 Assembly
CATIA V5 Drafting

Table of Contents

1. Introduction to Sheet Metal Design

Workbench Presentation, User Interface, Terminology, General Process

2. Sheet Metal Parameters

Practice on the Control Bracket

3. Sheet Metal Walls

Practice on the Control Bracket

4. Sheet Metal Bends and Flanges

Practice on the Control Bracket

5. Sheet Metal Cut Outs, Flat Pattern Mode and Features

Practice on the Control Bracket

6. Sheet Metal 2D View Creation

Practice on the Control Bracket

7. Sheet Metal Modifications to a Part

Practice on the Control Bracket

Introduction to Sheet Metal Design

You will learn about the Sheet Metal Design Workbench by

- Accessing the workbench
- Exploring the User Interface
- Seeing Terminology
- Understanding the general process



Accessing the Workbench



Anywhere from 1- Start

2- File

3- Workbench Icon

1- 2- 3-

The screenshot shows the CATIA V5 interface. The menu bar includes Start, TeamPDM, File, Edit, View, Insert, Tools, Window, and Help. The tree view on the left shows a hierarchy: Part1 (gear icon), xy plane, yz plane, zx plane, and PartBody (gear icon). The PartBody element is highlighted with a red box. A 'Welcome to CATIA V5' dialog box is open in the center, displaying six workbench options: Assembly Design, Catalog Editor, Drafting, Part Design, Sheet Metal Design, and Sheet Metal Production. The Part Design option is selected. A red arrow points from the PartBody element to the Part Design option in the dialog. Another red arrow points from the Part Design option to the Workbench icon in the right-hand toolbar, which is also circled in red.

The first time you access the Sheet Metal Design Workbench, a Part Body is created, that will contain geometrical elements

User Interface: Sheet Metal Design General



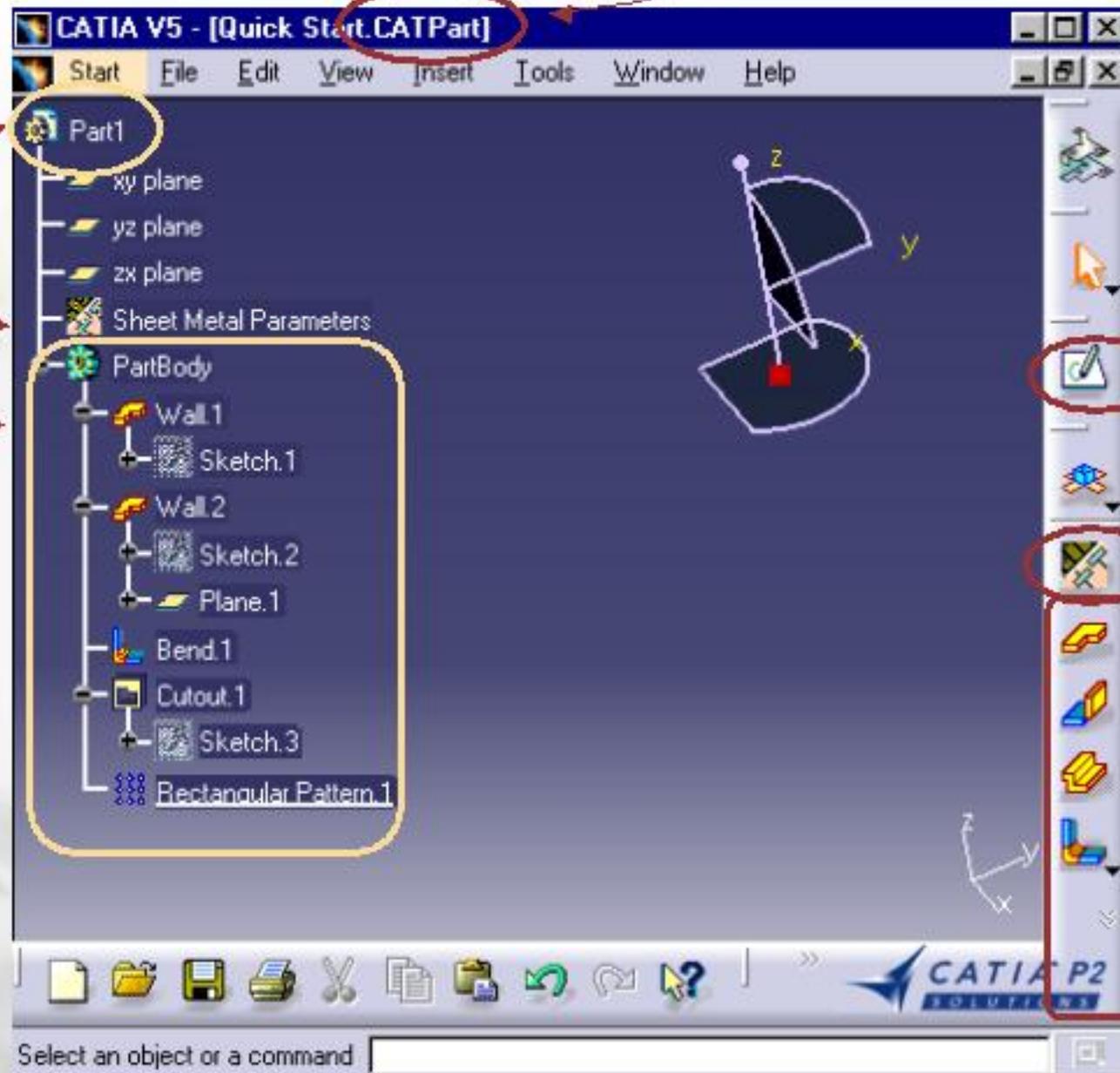
CATPart extension (THE SAME AS A REGULAR PART)

Part tree

Sheet Metal Parameters

Features...

Standard tools



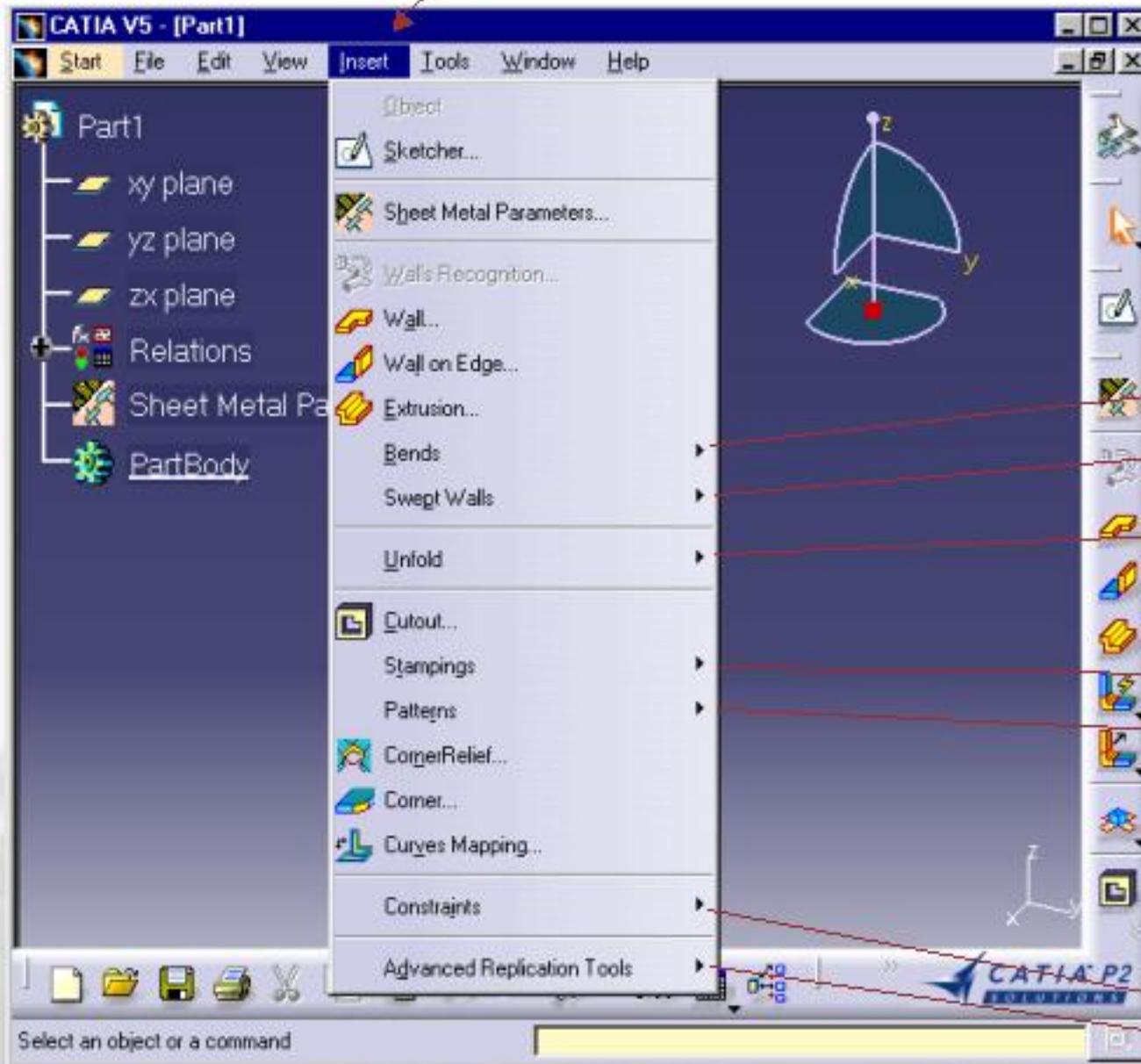
Sketcher access...

Sheet Metal Parameters

Sheet Metal Design tools...

User Interface: Sheet Metal Design Tools

Insert menu or toolbars



Sheet Metal Icons



Bend



Swept Walls



Unfold



Stampings



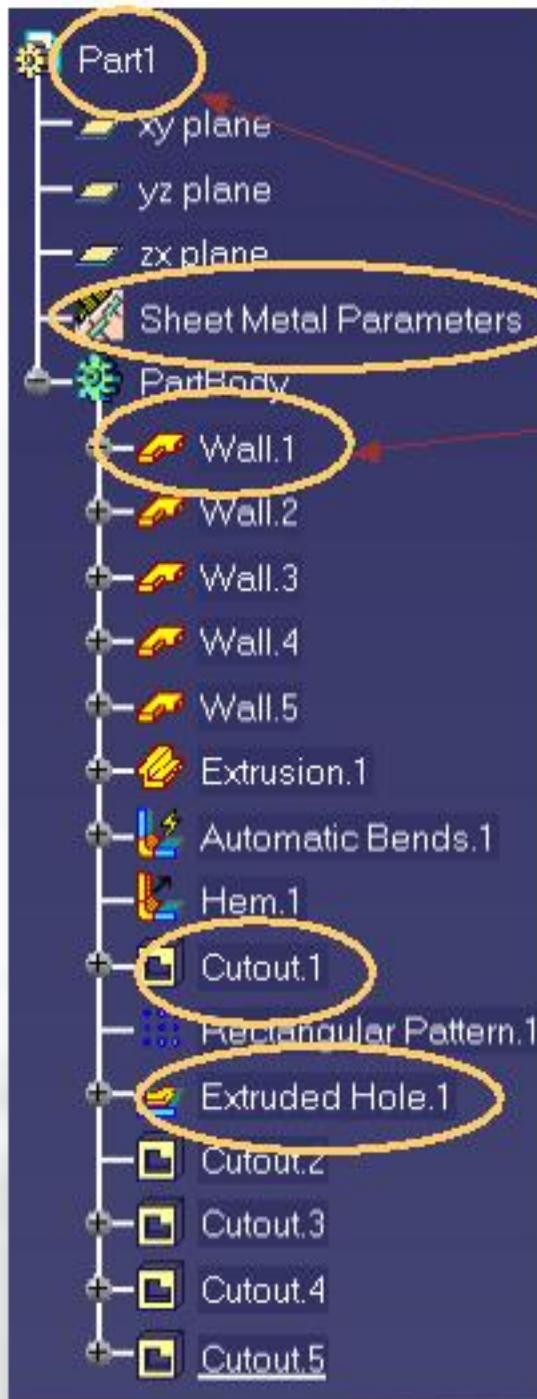
Patterns



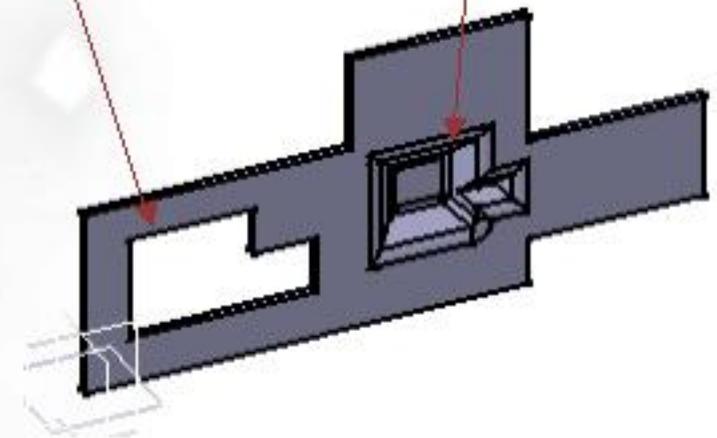
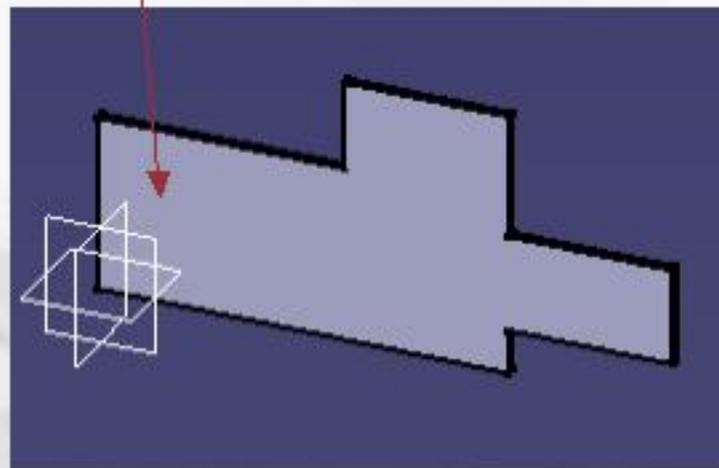
Power Copy



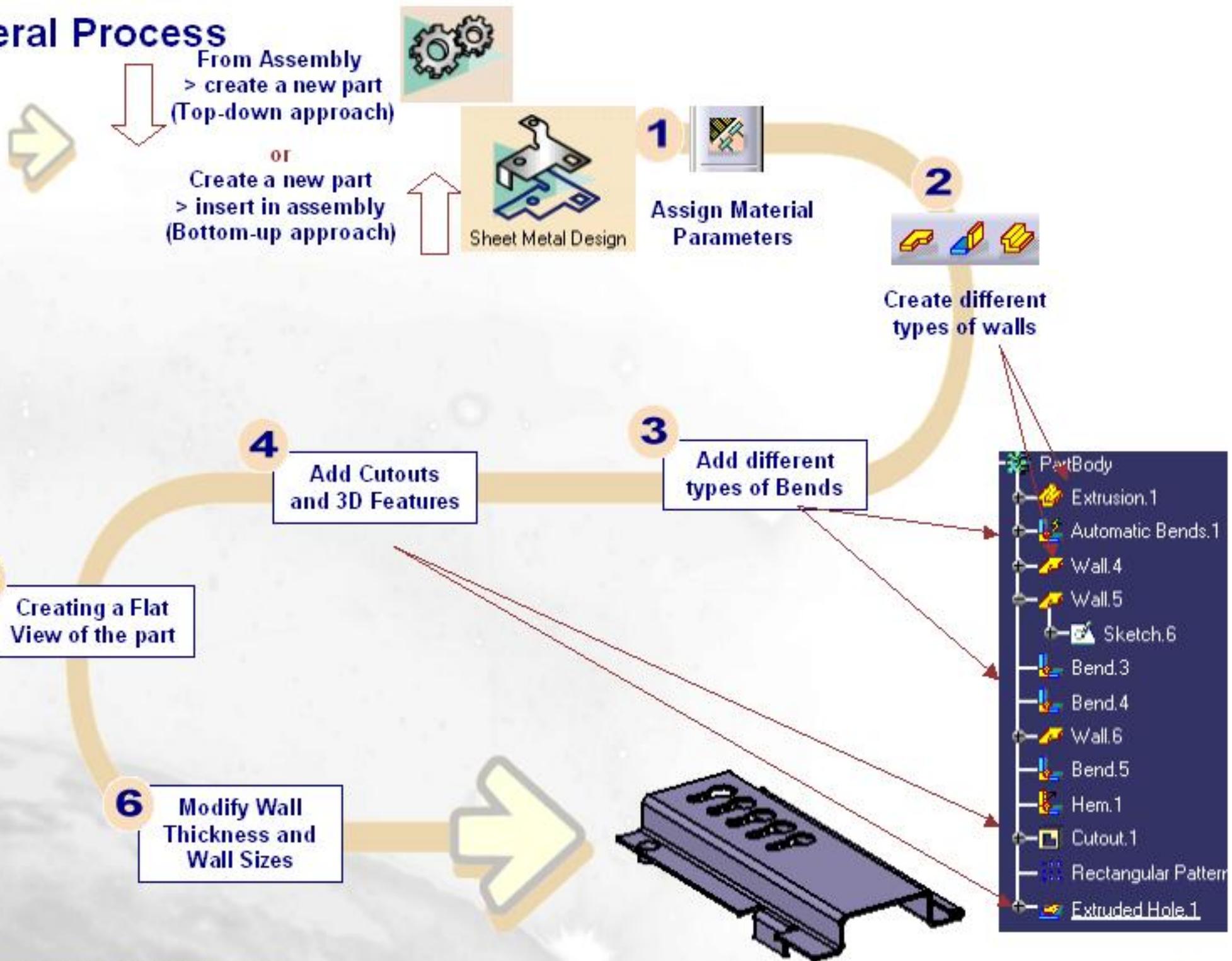
Terminology



- A Part is a combination of one or more walls, bends and features
- Parameters are the values of the general wall thickness, bend radii and type of relief used in the part.
- The first wall is generated from a sketch (profile), as either a single wall, a group of extruded walls or a rolled wall.
- Features are things such as cutouts, patterns and stampings added to the part.



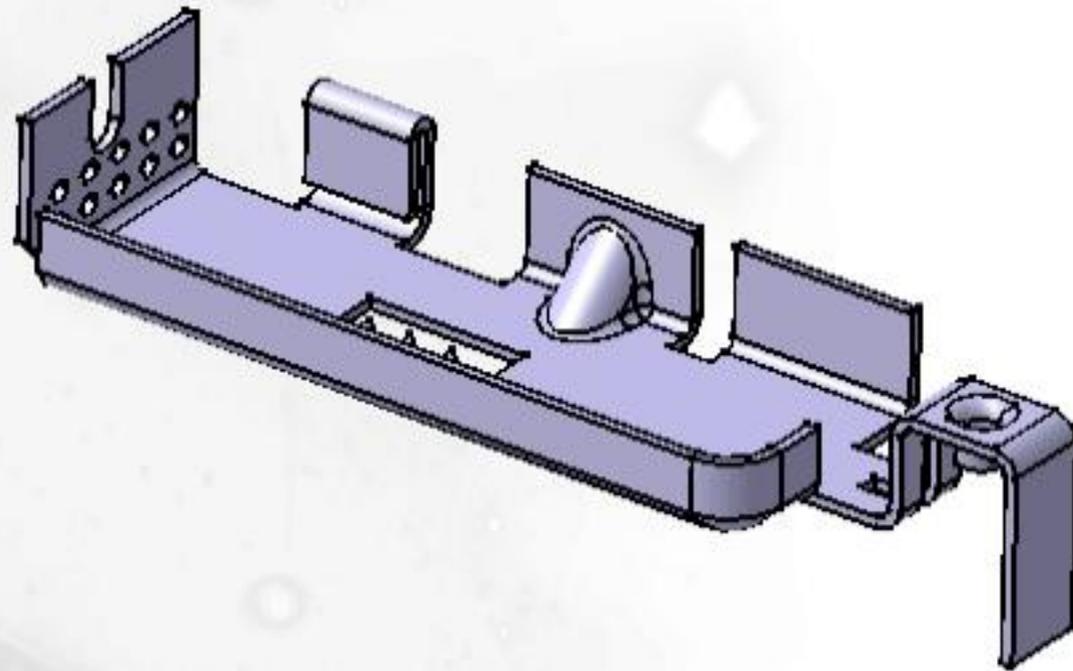
General Process



Sheet Metal Parameters

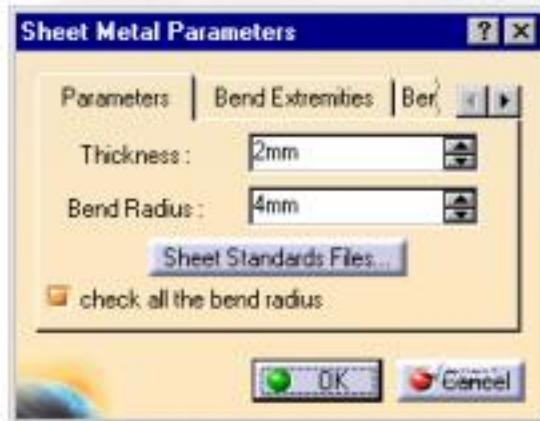
In this lesson, you will see how to start a Sheet Metal part by assigning the basic parameter needed to create the part.

■ Defining Sheet Metal Parameters

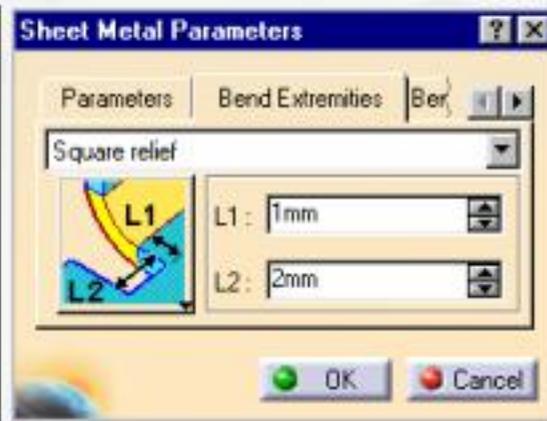


Defining Parameters

Before starting to create any Sheet Metal part, define the Sheet Metal Parameters of the part. These are common characteristics that when defined will make creation easier and thus limit your need to supply data during creation of the part.



Parameters Tab



Bend Extremities Tab



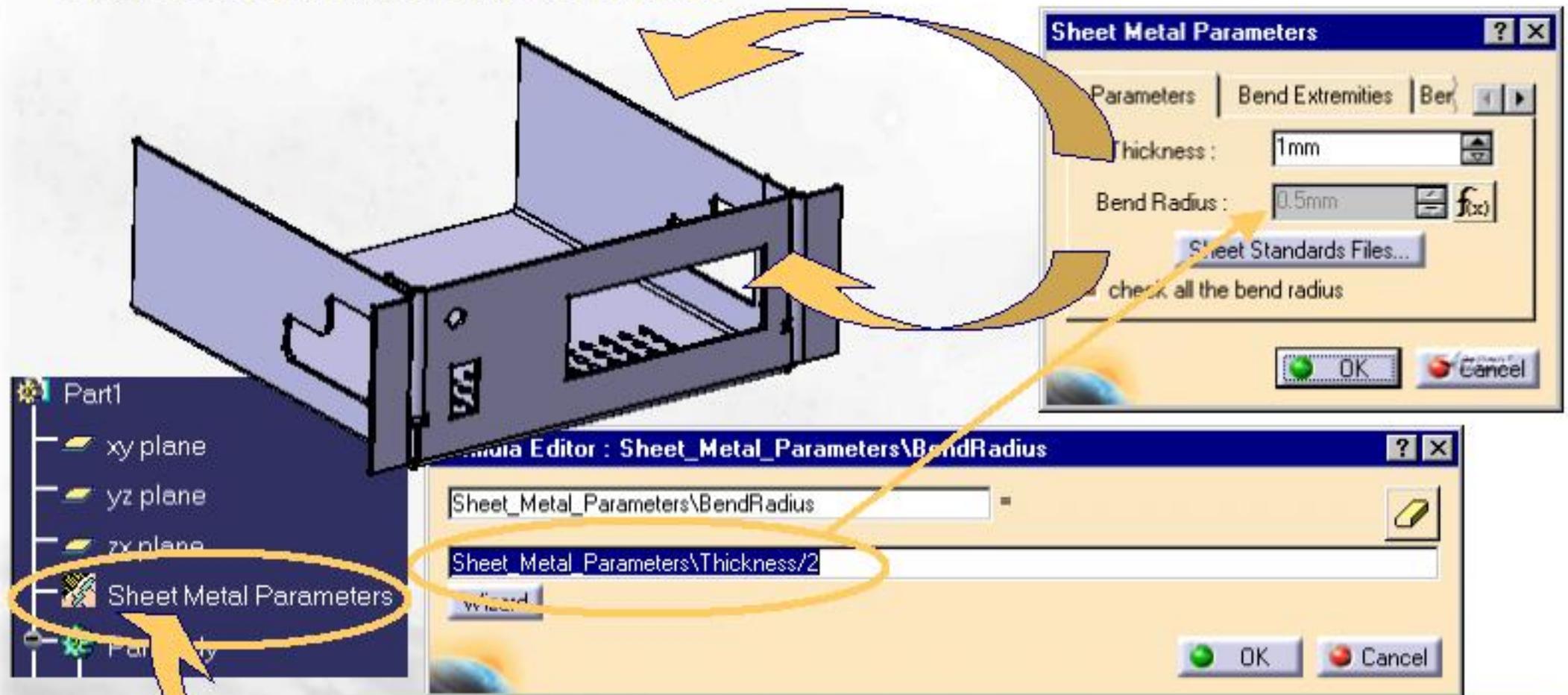
Bend Corner Relief Tab



Bend Allowance Tab

What are Parameters ?

Parameters are the common characteristic used in the creation of the part. They are as simple as stating the dimensional requirement of the material and bend radius



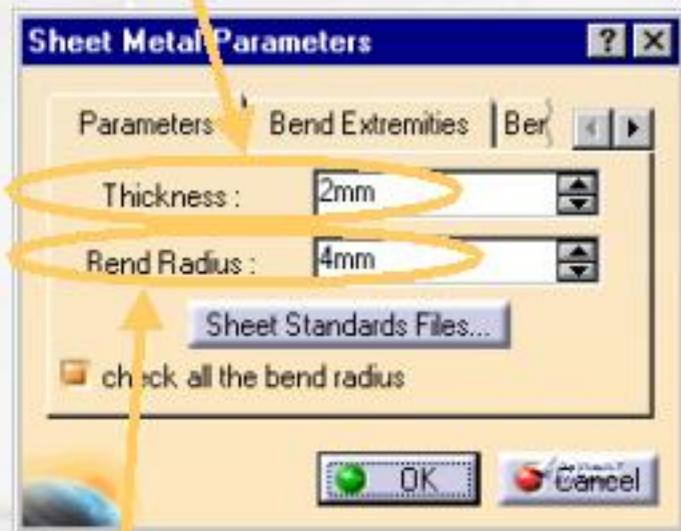
It is important to note here that only one set of parameters can be assigned in a CATPart, therefore you are limited to one part per document.

What Parameters need to be defined ?

In order to start there are three basic parameters that need to be assigned:

- Material Thickness
- Bend Radius
- Bend Relief Type

Material Thickness

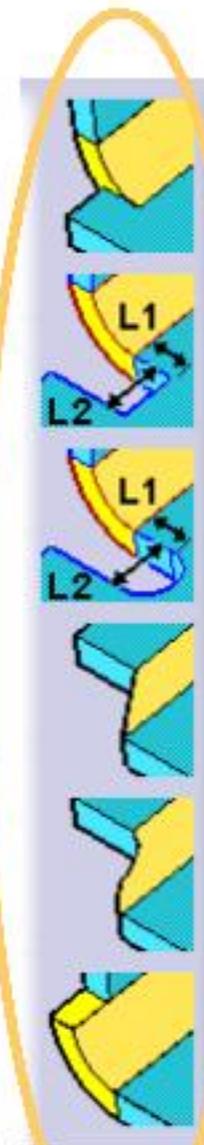
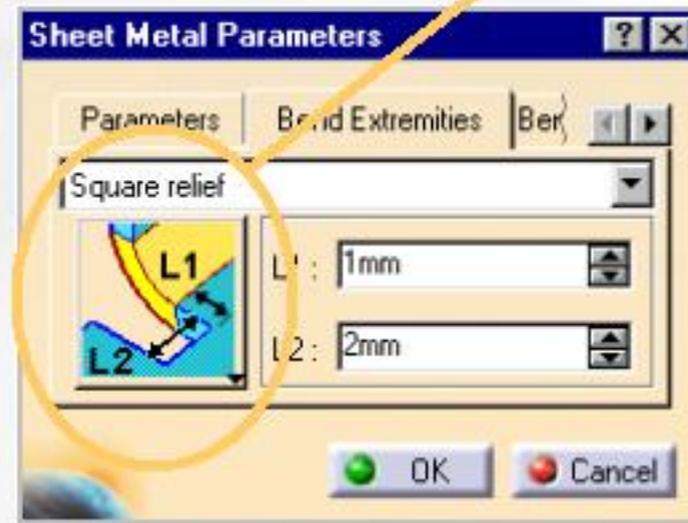


Bend Radius



Remember, the Bend Radius assigned here is the standard Bend Radius to be used on the part. This will be used to create all Bend Radii unless specified otherwise during the creation.

Types of different Bend Reliefs



Minimum with no relief

Square Relief

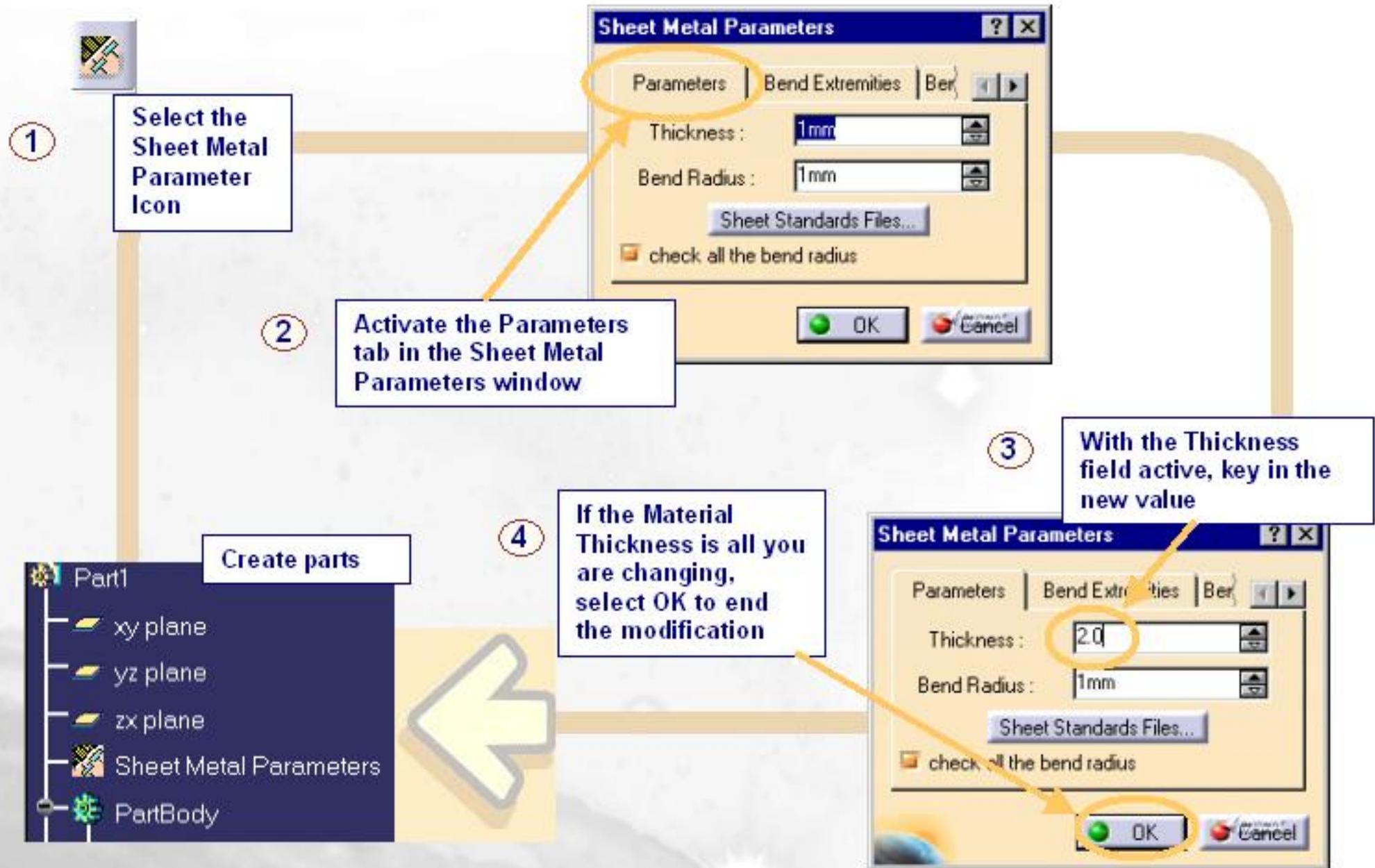
Round Relief

Linear

Tangent

Maximum

Defining Material Thickness



Defining Material Bend Radius

1 Select the Sheet Metal Parameter Icon

2 Activate the Parameters Tab in the Sheet Metal Parameters window

3 Key in the new Bend Radius

4 Select OK

Sheet Metal Parameters

Parameters | Bend Extremities | Ber

Thickness : 2mm

Bend Radius : 1mm

Sheet Standards Files...

check all the bend radius

OK Cancel

Sheet Metal Parameters

Parameters | Bend Extremities | Ber

Thickness : 2mm

Bend Radius : 3

Sheet Standards Files...

check all the bend radius

OK Cancel

Part1

- xy plane
- yz plane
- zx plane
- Sheet Metal Parameters
- PartBody

Create parts

←

Bend radii can be as small as 0.01 mm. 0 Radius is not available yet.

Selecting the Bend Relief Type

1 Select the Sheet Metal Parameter Icon

2 Select the Bend Extremities Tab in the Sheet Metal Parameters window

3 Select the type of Bend Relief displayed in the window

4 The relief with its dimensions are displayed

5 If the Bend Radius is all you are changing, select OK to end the modification

Create parts

- Part1
 - xy plane
 - yz plane
 - zx plane
 - Sheet Metal Parameters
 - PartBody

Where possible bend relief values, L1 and L2 can be modified to suit your special needs

Selecting the Bend Allowance

HINTS AND TIPS

1

Select the Sheet Metal Parameter Icon



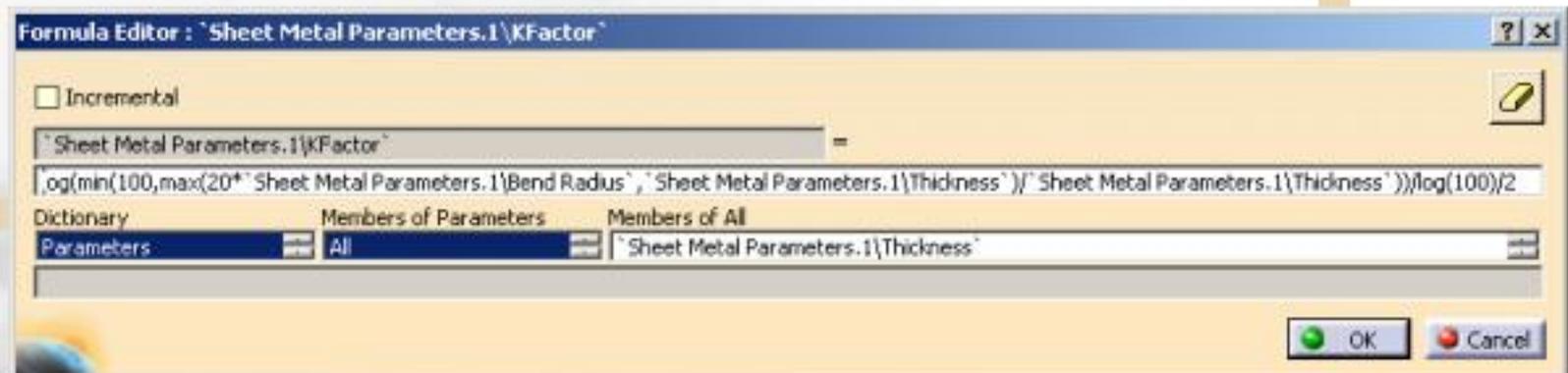
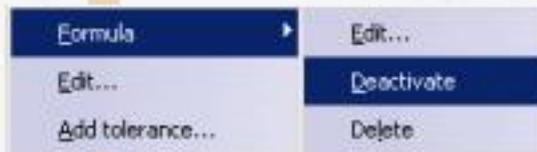
2

The k factor is managed by a formula including the material



3

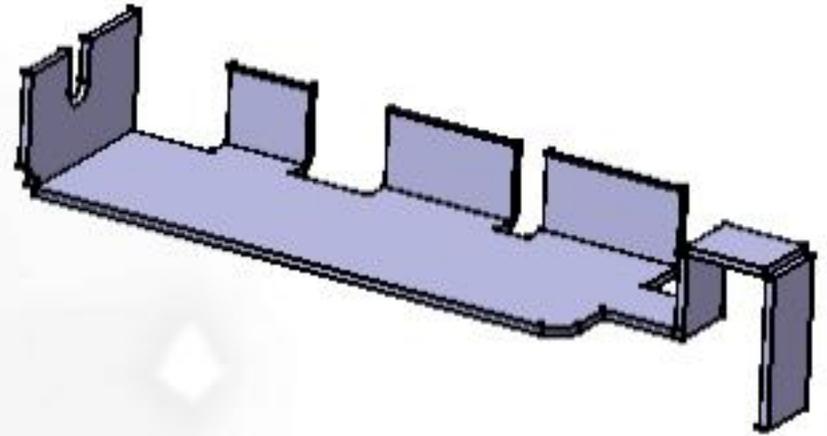
You can change it for a personal value or you can modify it for each bend deactivating the formula and typing the new one



Sheet Metal Walls

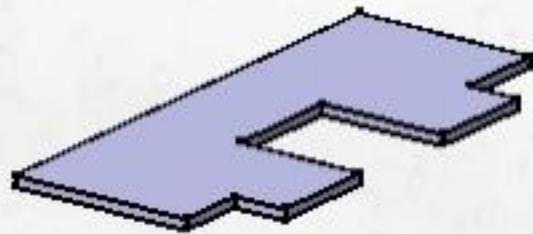
In this lesson, you will see how to start a sheet metal part and different ways to define walls

- Defining Walls from Scratch
- Defining Walls on Existing Parts
- Sheet Metal Walls recap exercise

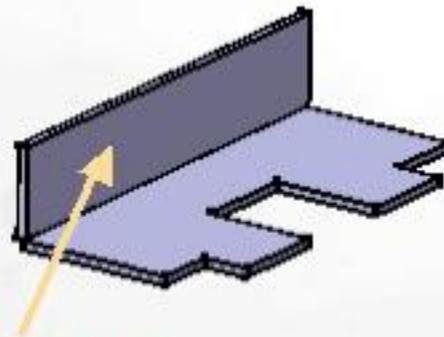


Defining Walls from Scratch

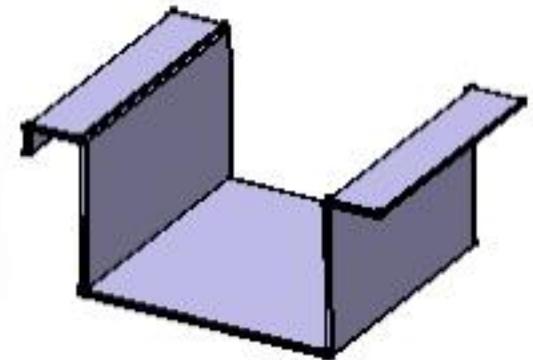
You will learn how to create the 5 basic types of walls offered in the Sheet Metal workbench



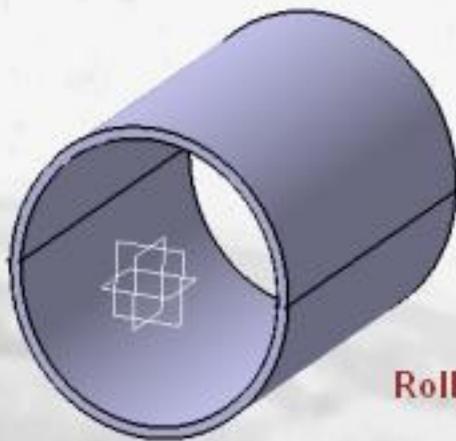
Profile wall



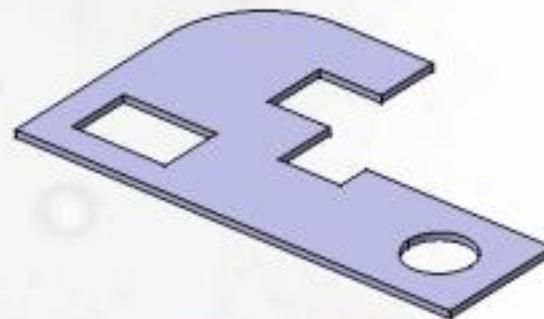
Wall on Edge



Extruded Wall



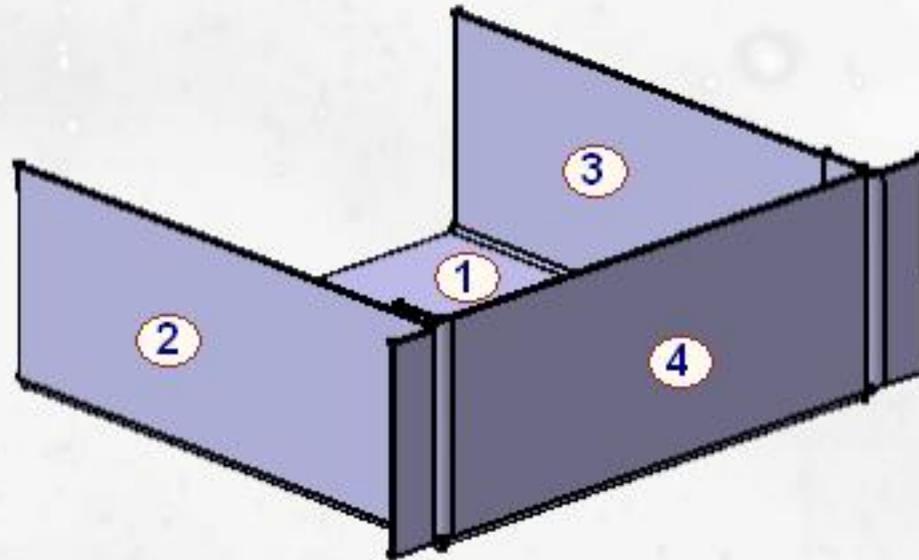
Rolled Wall



Pierced Wall

What are walls ?

To start a sheet metal part, create some type of wall first to start from. It can be as little as one line that you can extrude a wall from or a complex profile to create many walls from or a profile to make an individual wall.



An example of how different ways could be used to create this part is as follows:

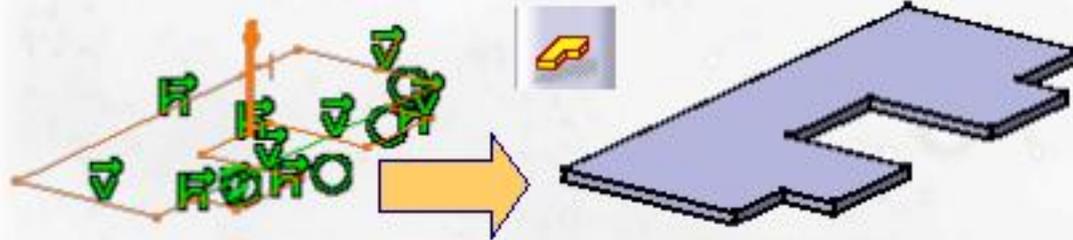
- Wall 1 could have been created first as a flat profile wall and then walls 2, 3, and 4 could have been done as walls on edge
- Wall 1, 2, and 3 could have been created with an extrude and then wall 4 could have been done as a wall on edge



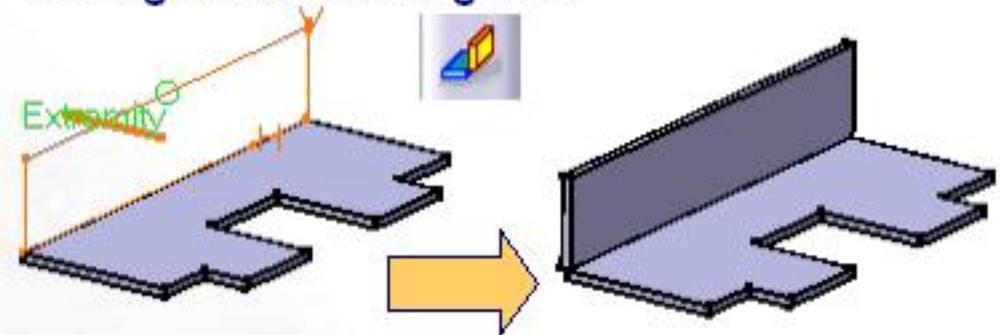
Be aware that where you select edges, top vs bottom will drive a different proposal for your wall. For example, a top selection may force your wall to go in an upward direction while a bottom selection may force your wall in a downward direction.

When to use the various types of walls ?

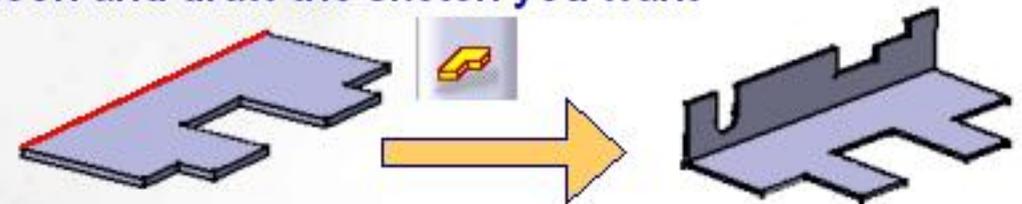
- The Profile wall is used to create a wall of a specific shape. Often used as a first wall of a part



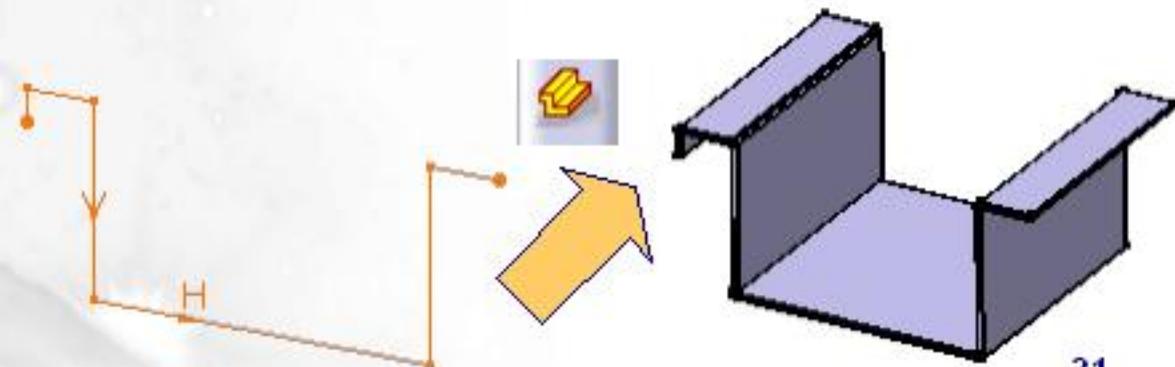
- The Wall on Edge is used to create a wall on an edge of an existing wall.



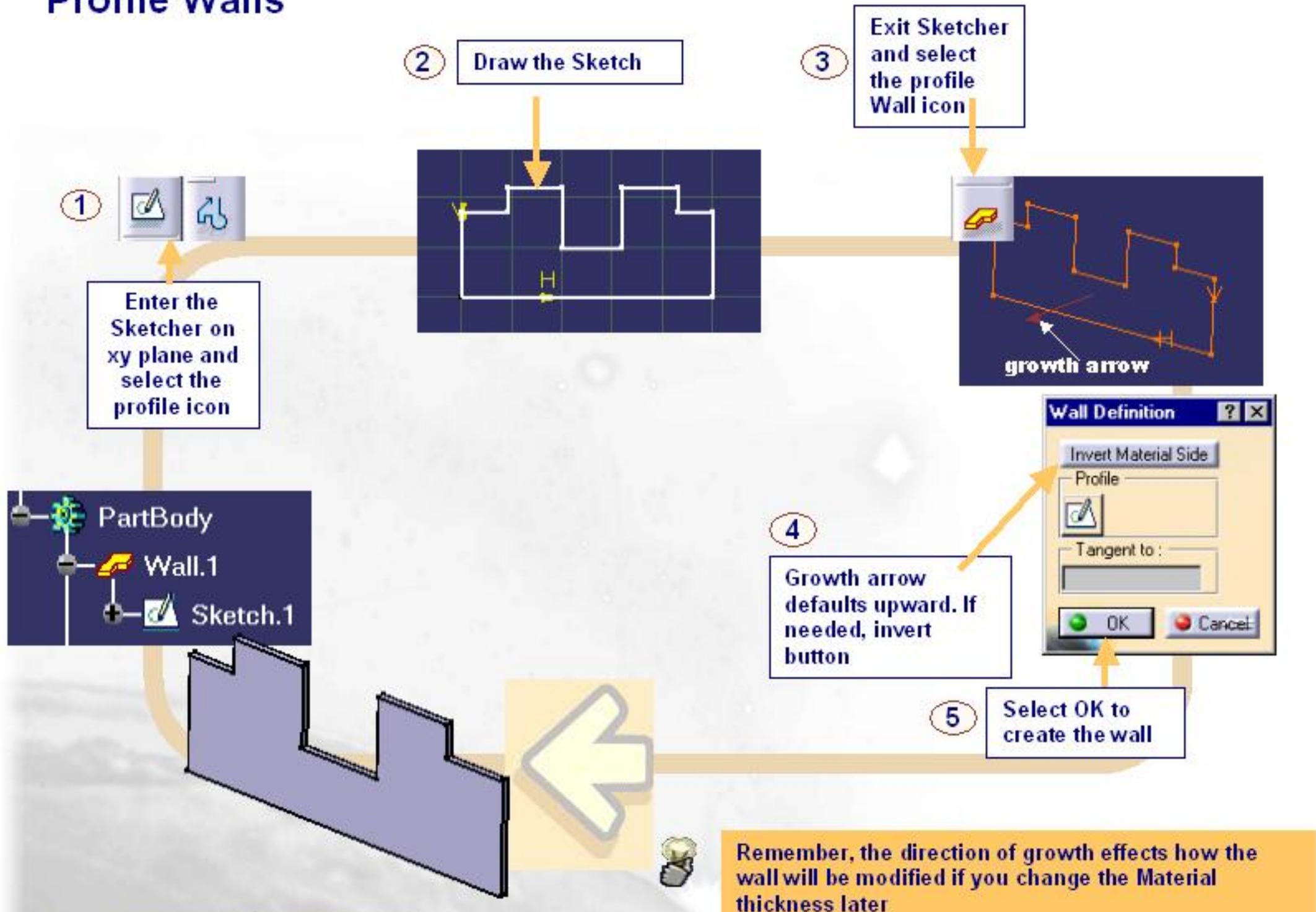
You can also select an edge, click on the wall icon and draw the sketch you want



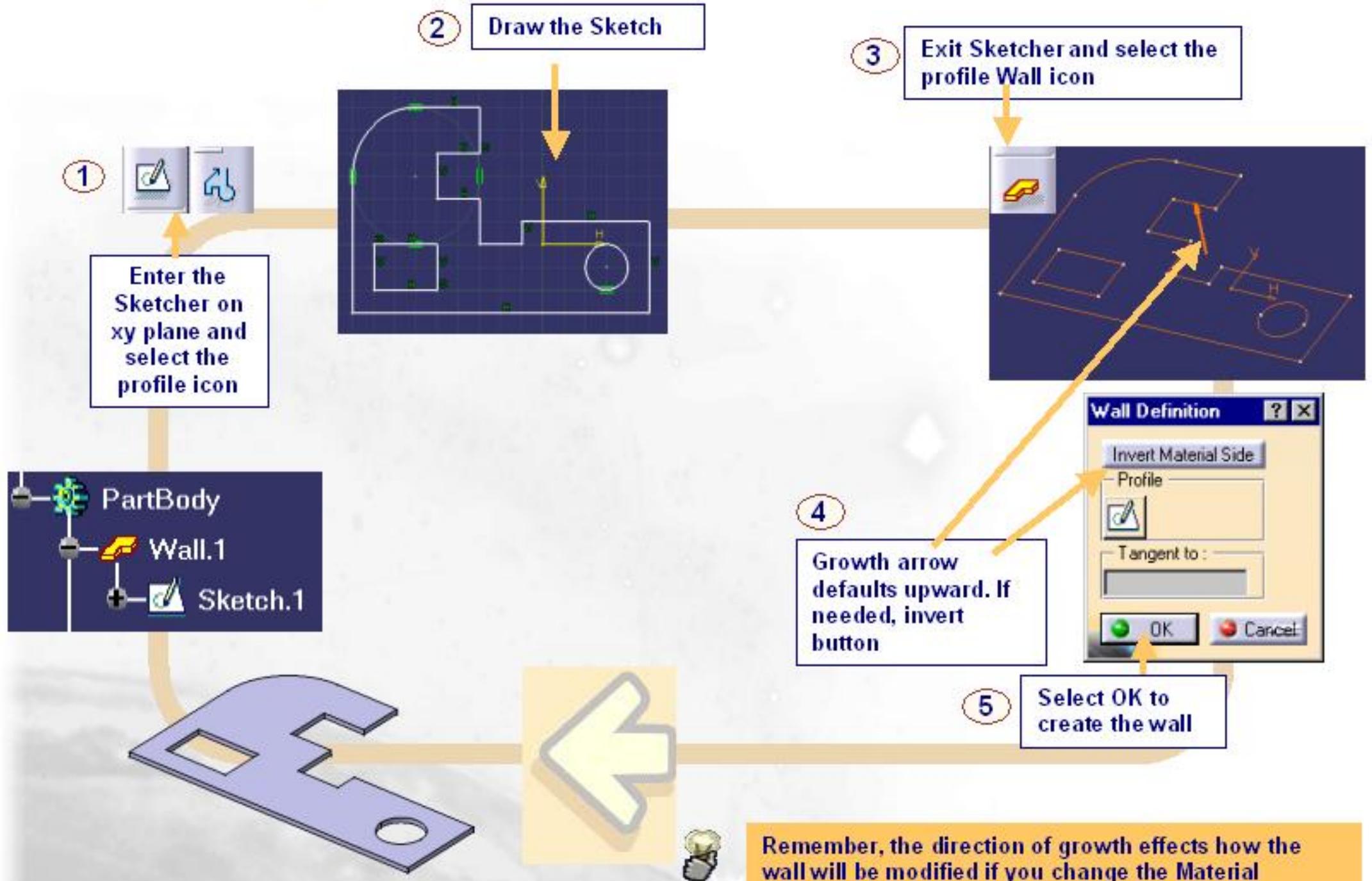
- The Extruded walls are used to create a group of walls using a specific profile. Here it is used to build a simple “U” shaped chassis or a hat section used to strengthen other parts.



Profile Walls



Profile Walls (with cutouts)

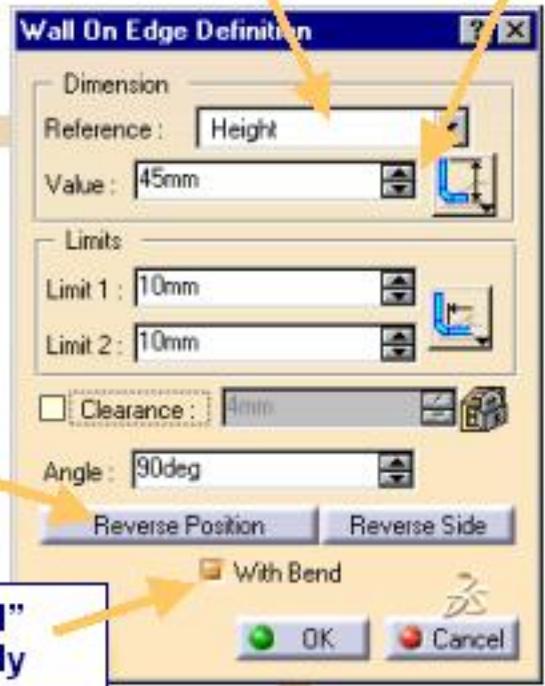
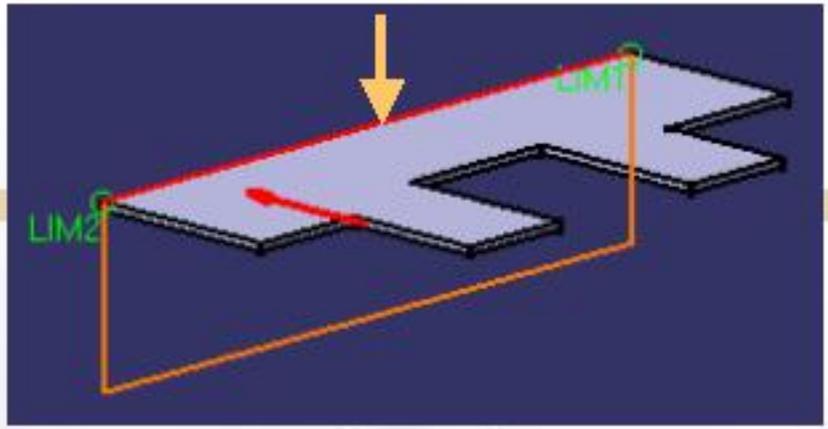


Walls on an Edge

2 Select the edge to attach the wall too

3 Key in the new values and select a dimension options

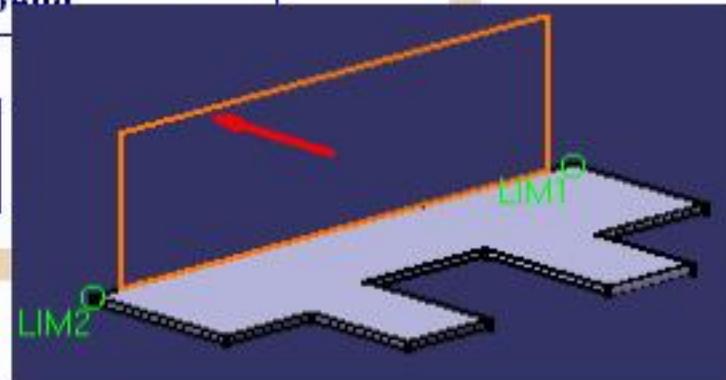
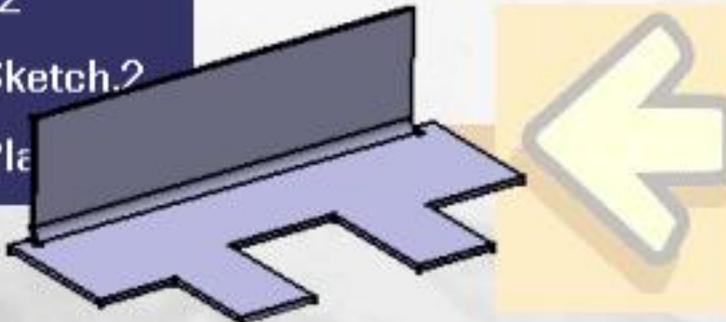
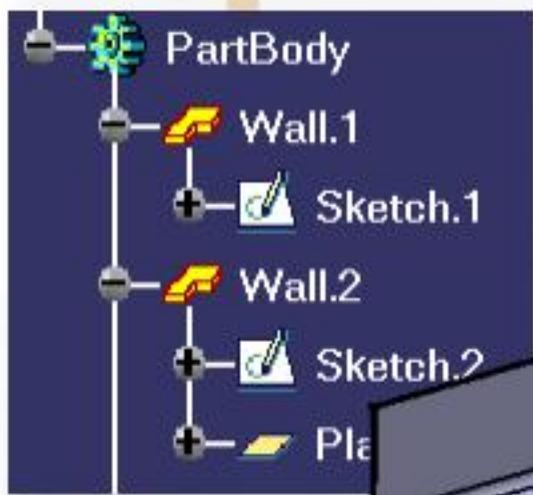
1 Select the wall on edge icon



4 To reverse sketch profile direction, select this button

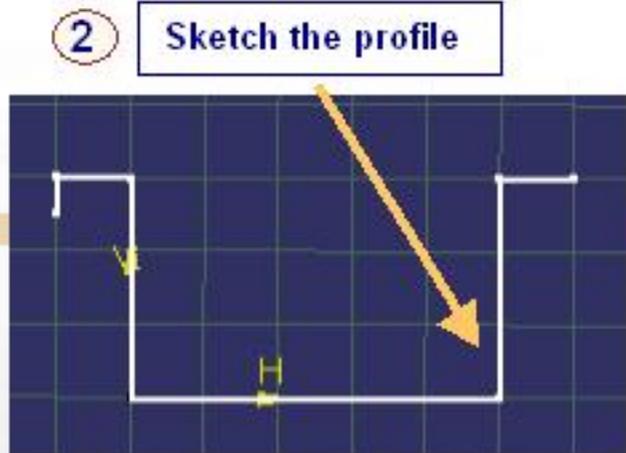
5 Select the "With Bend" option to automatically create the bend

6 Select OK to create the wall

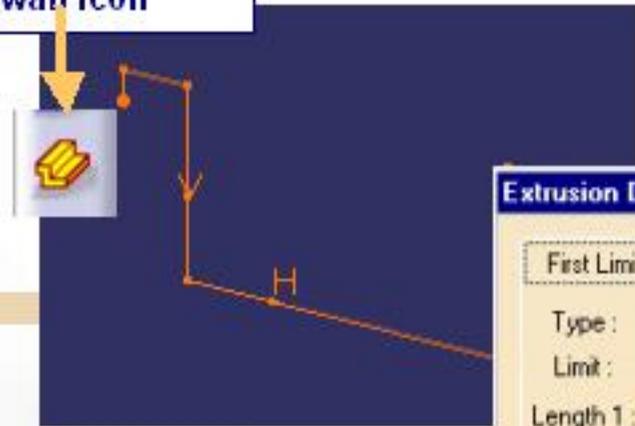


Extruded Walls (1/2)

1
Enter the Sketcher on xy plane and select the profile icon



3 Exit Sketcher and select the Extrusion wall icon

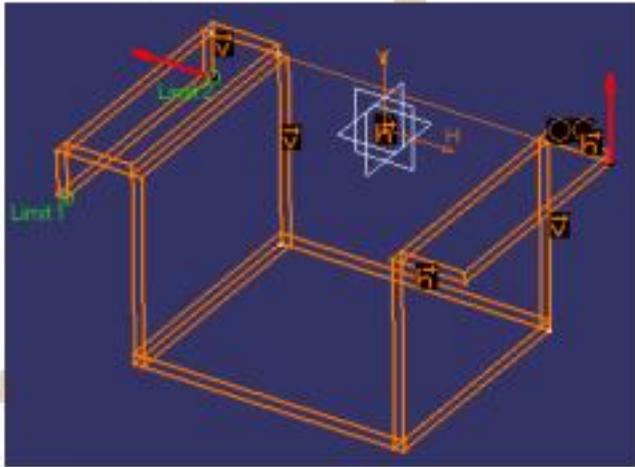
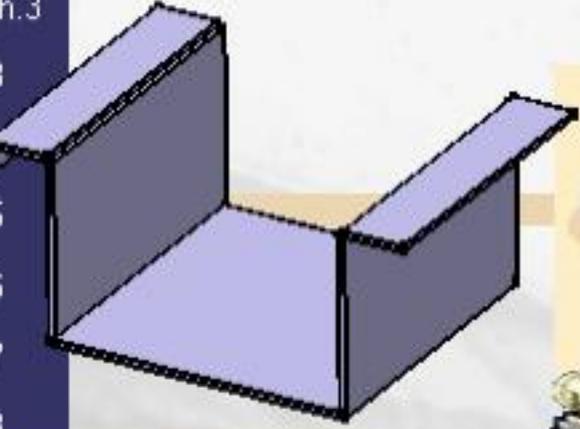


4 Key or select the extrusion parameters.



5 Select OK

The walls can be created on one or both sides of the sketch (Second Limit). Also, the material thickness can be in either direction or symmetrically around the profile (symmetrical thickness checkbox)



Note: in the tree, each wall is created separately under the extrusion. No sketches for the walls are created; the sketch for the original profile is kept.

Extruded Walls (2/2)

HINTS AND TIPS

1

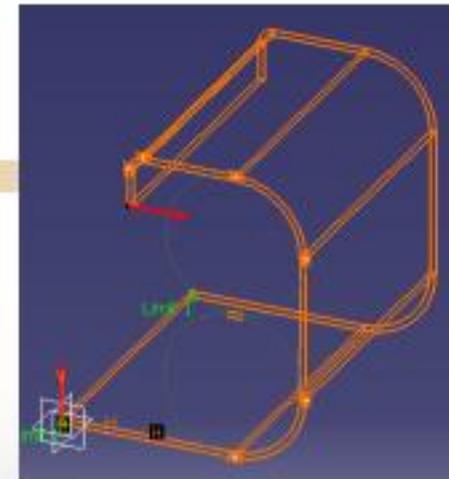


Other possibilities of the extrusion



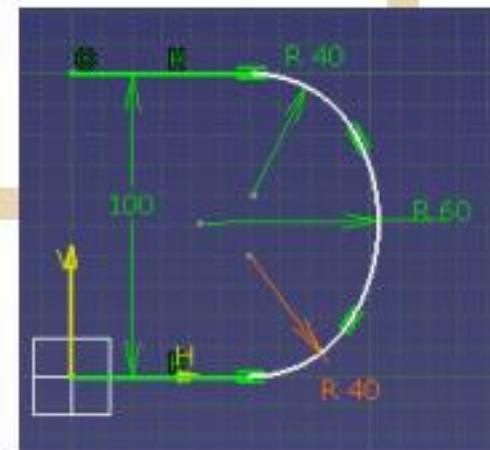
2

All the arcs created in the sketch will be replaced by bends



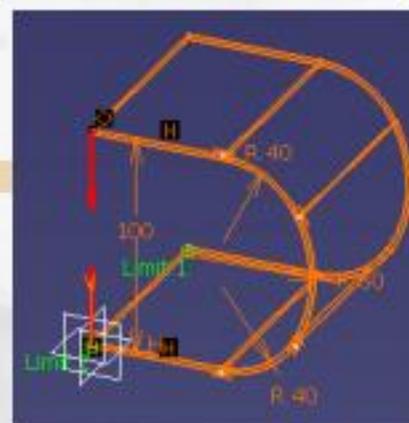
3

You can do the same creating 3 consecutive arcs



4

You must have a tangency constraint between each arc



5

The tree will be defined this way

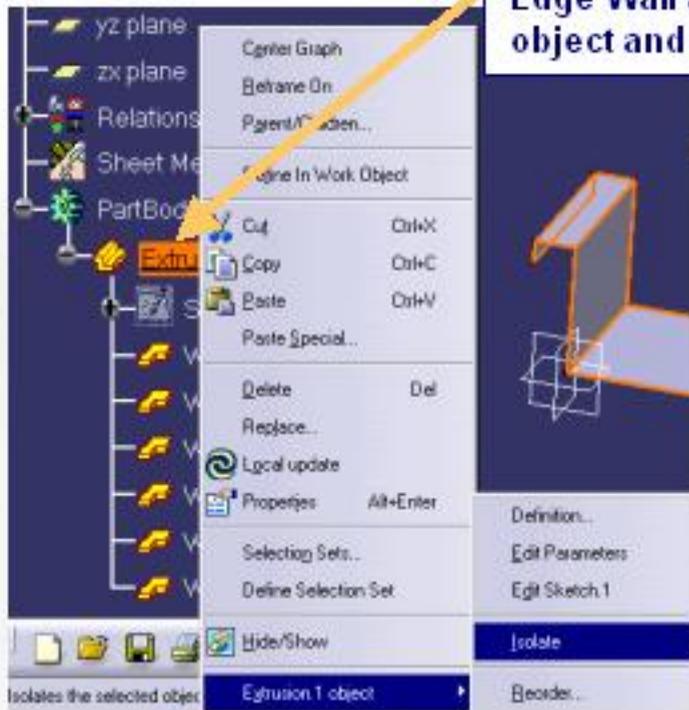


Isolate Walls

Can be performed on:

- extrusion walls
- wall on edge

1 MB3 on Extrusion or Edge Wall and select item object and Isolate



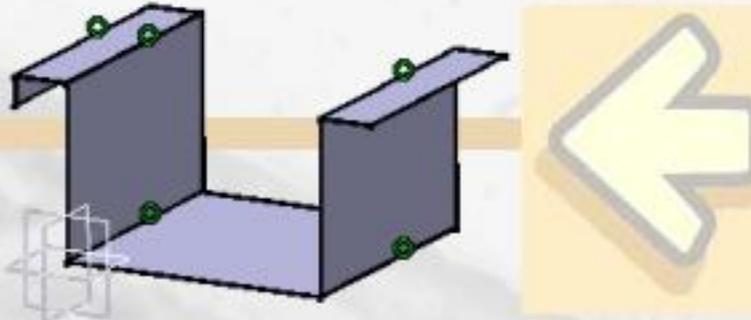
2 If an Extrusion, Select a wall



5 local axis, 6 sketches and some coincidence constraints are automatically created for this extrusion.



You cannot undo an Isolate action after having modified the wall. Isolating a Wall on Edge erases all updating data.



Walls on an Edge (with clearance)

1 Select the wall on edge icon

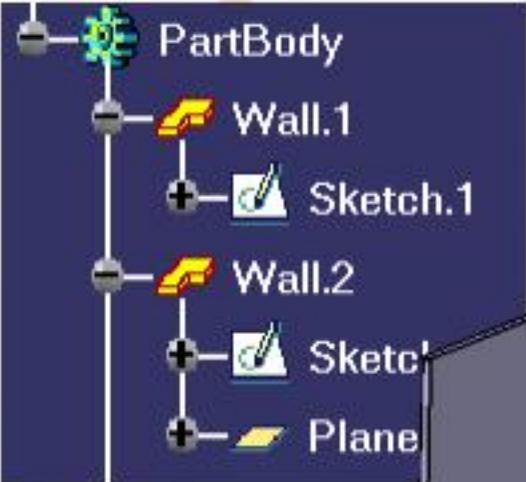
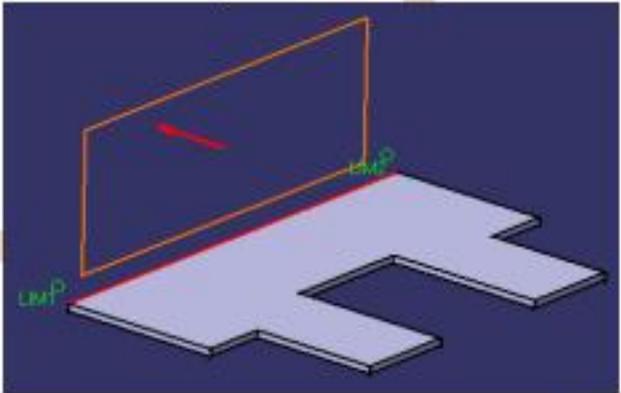
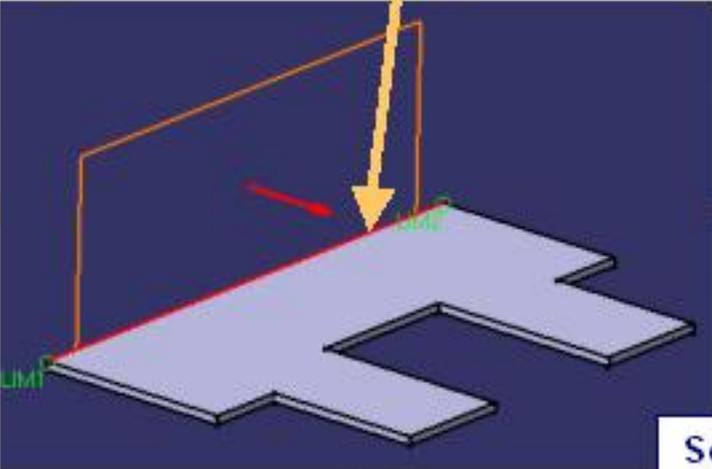
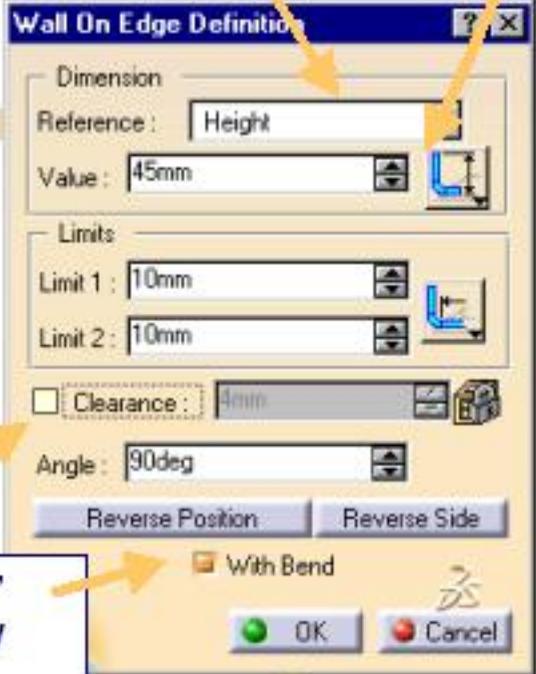
2 Select the edge to attach the wall to

3 Key in the new values and select a dimension options

4 Select the clearance option. The value is by default driven by bend radius

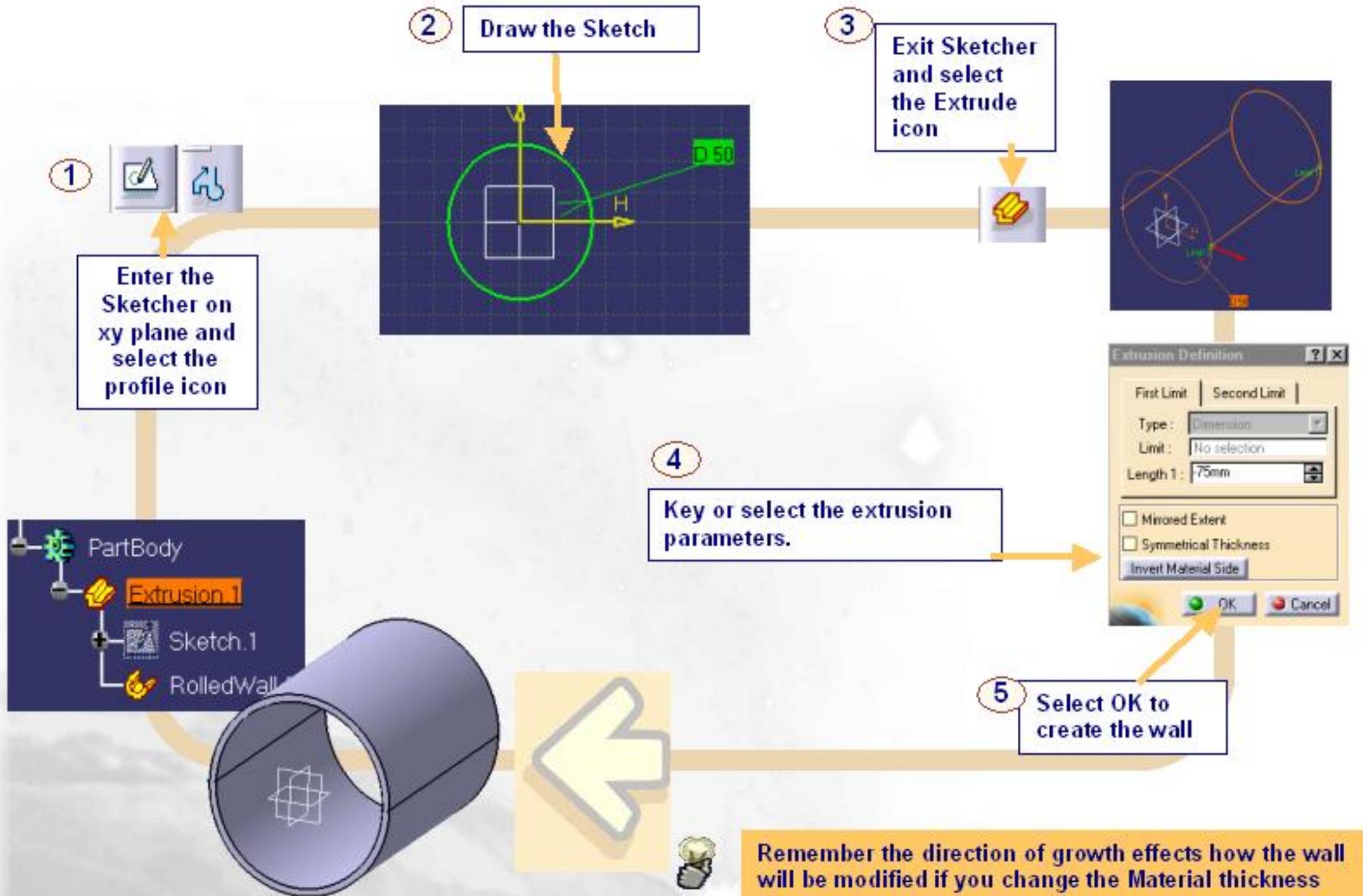
5 Select the "With Bend" option to automatically create the bend

6 Select OK to create the wall



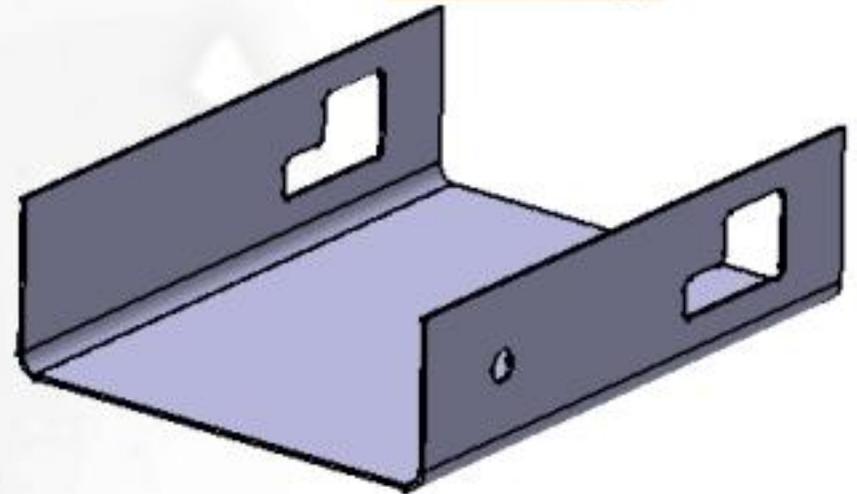
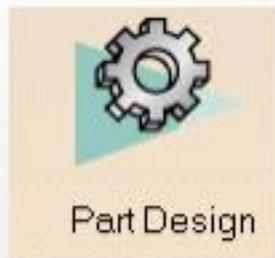
Note: the reference wall is not impacted by the new wall

Rolled Walls



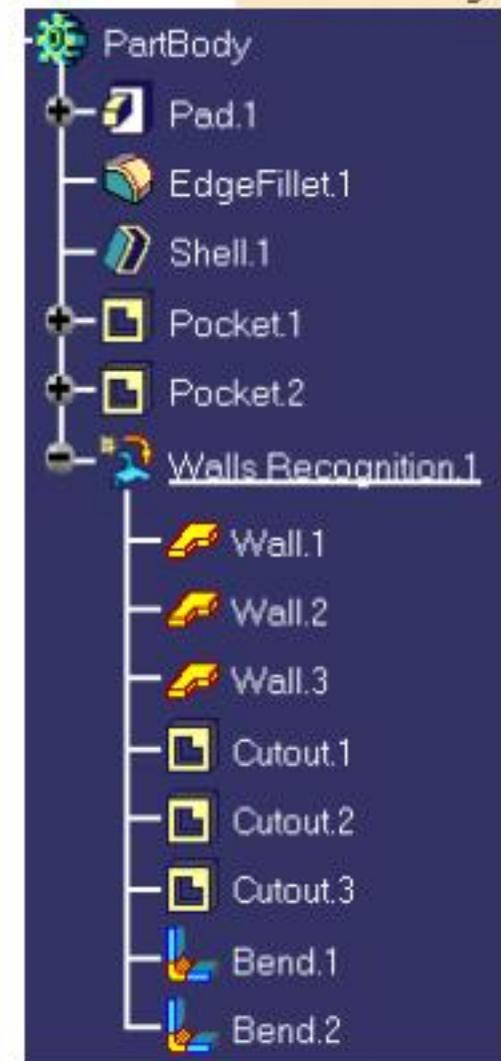
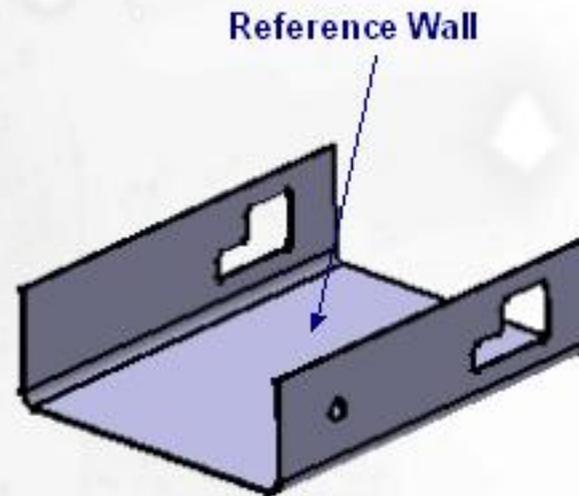
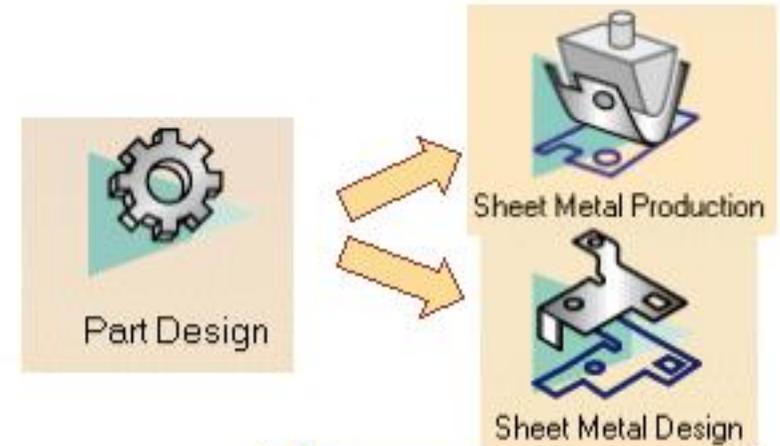
Defining Walls on Existing Parts

- You will learn how to define walls on existing solid parts



What is Wall Recognition?

Another way to start creating sheet metal walls is to take an existing solid and recognize (= change) the solid walls into sheet metal walls.



Be aware that any cut outs in the part, when it is recognized, can only be modified in the Part Design workbench.

Walls Recognition Definition dialog box

2 wall creation modes:

Part body recognition: the whole solid is processed and walls are created wherever possible.

Only selected faces: only explicitly selected faces of the solid are processed and the corresponding walls are created.

Reference wall – this wall represents the face you select on the body to generate the wall recognition. You do not have to select it again.

Compulsory walls - are faces from which the walls are to be generated when there might be an ambiguity. For example, if the initial part is a box, you will need to select two opposite inner faces and outer faces on the other two sides of the box, in order to avoid overlapping when generating the walls.

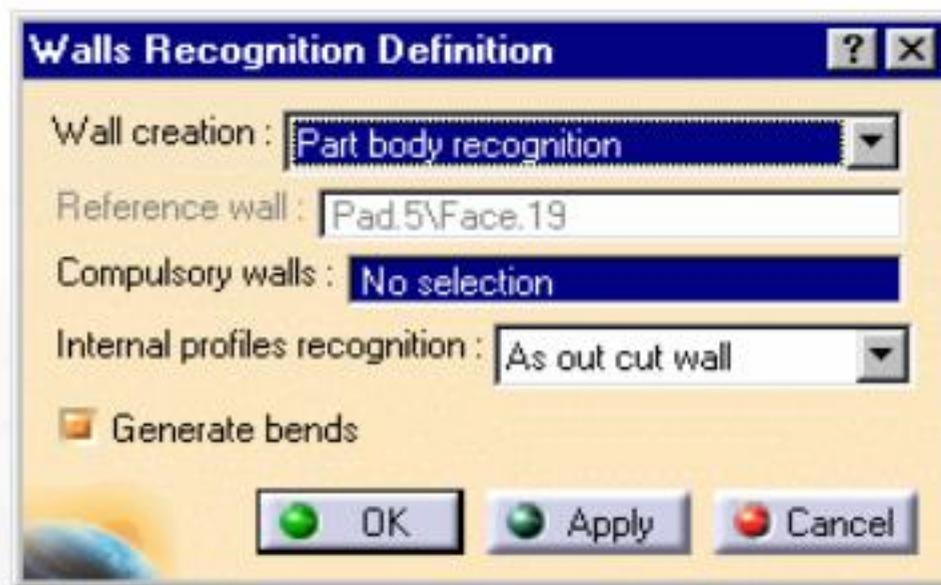
Internal profiles recognition mode:

As cut out wall: generates walls with inner contours (no cutout feature is generated)

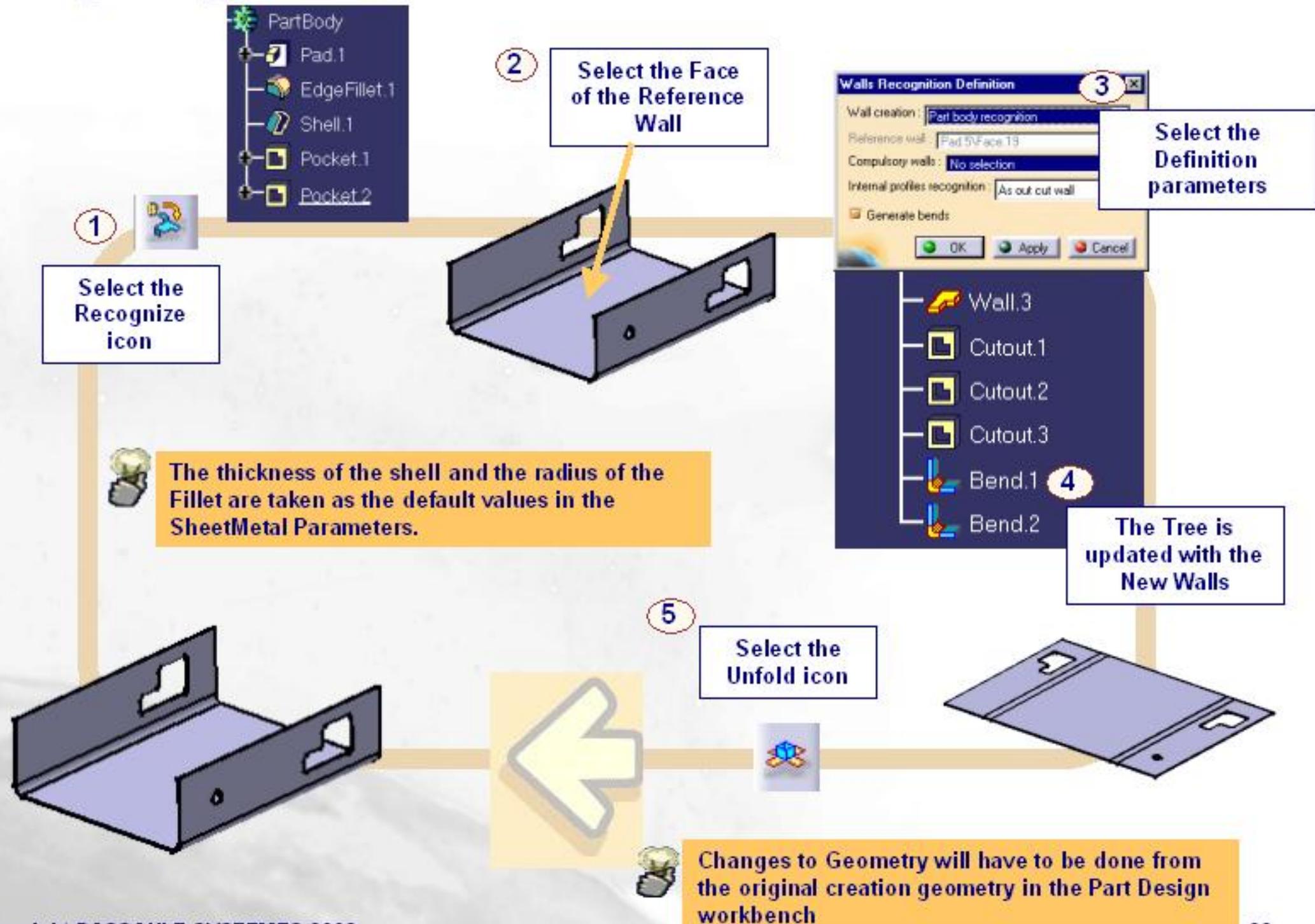
One cut out by wall: regardless of how many pockets there are on a face of the solid, only one cutout feature is generated per wall

One cut out by profile: for each inner contour on the sketch-based solid, a cutout feature is generated

None: whether there are pockets on the solid faces, or not, no cutout feature is created in the resulting Sheet Metal features.



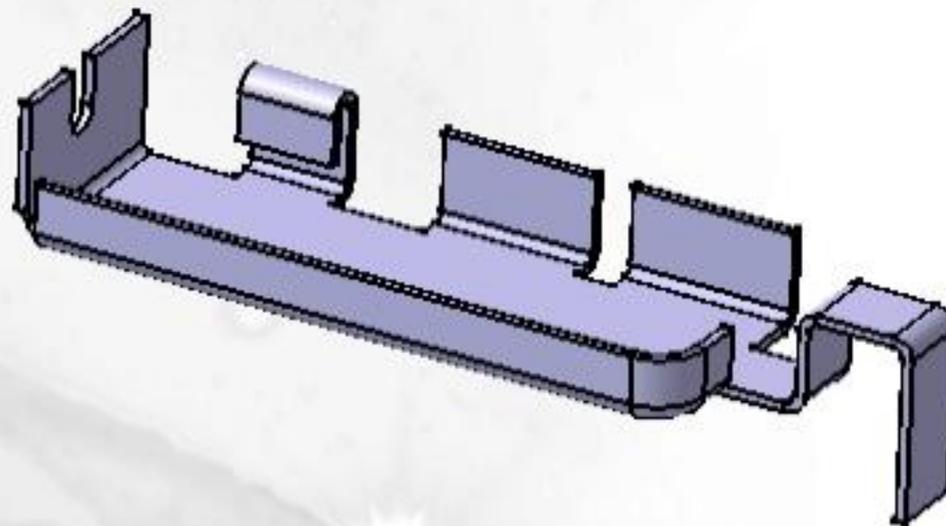
Recognizing Walls on Parts



Sheet Metal Bends and Flanges

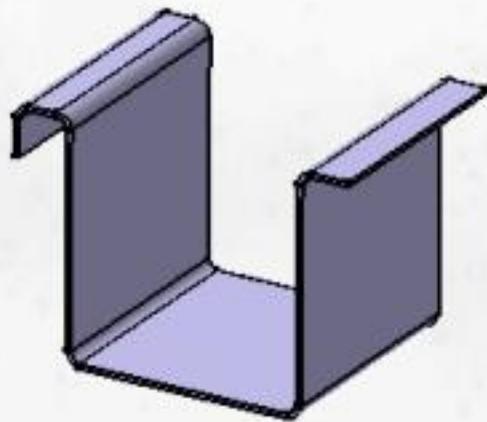
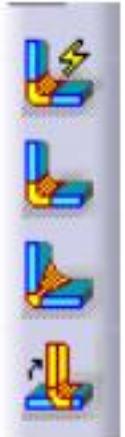
In this lesson, you will see how to add bends to a part and also see some different type of flanges that can be used when creating parts

- Defining Bends between Walls
- Defining Different types of Flanges available
- Sheet Metal Bends and Flanges recap exercise

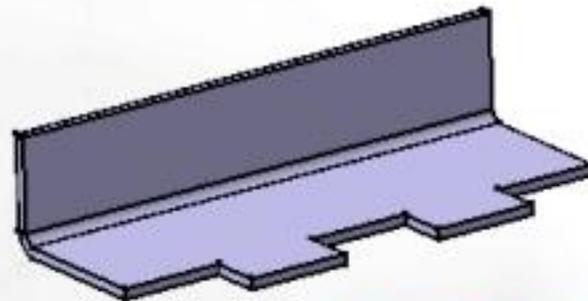


Defining Bends for Walls

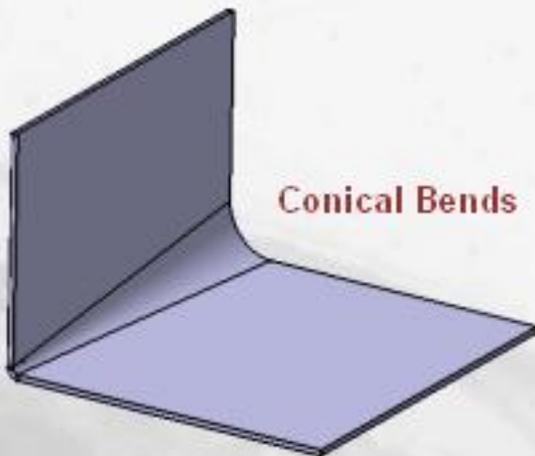
Bends can be created by four ways. If you choose, you can create your bend each time you create a wall on another wall or you can create a bunch of walls that show how your part will look and then use Automatic Bends to create all of them at one time.



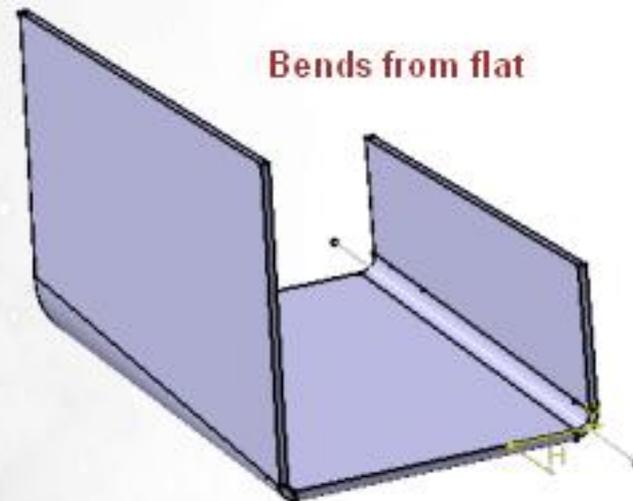
Automatic Bends



Singular Bend



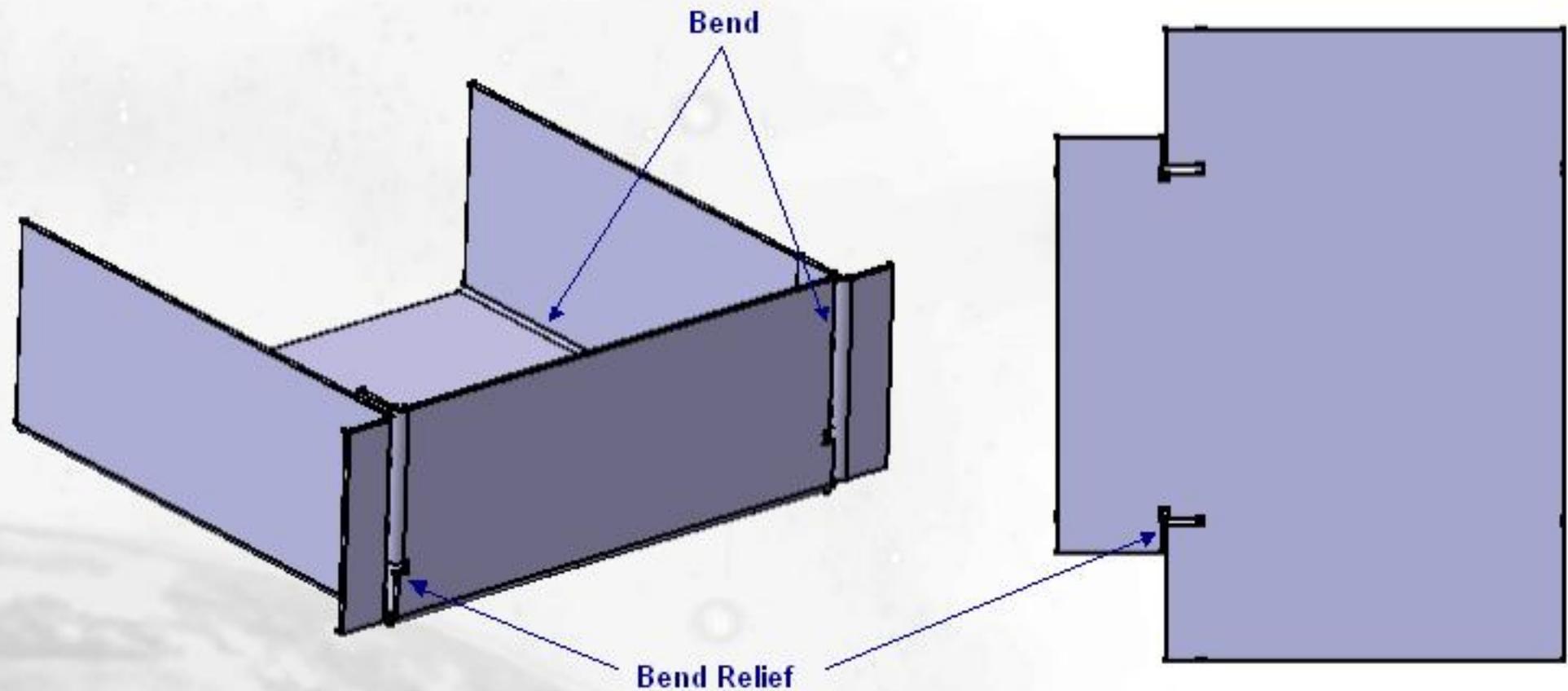
Conical Bends



Bends from flat

What are Bends ?

A Sheet Metal part is created from one piece of metal that is cut to the shape needed and folded to create the part. These folds are what is call bends.

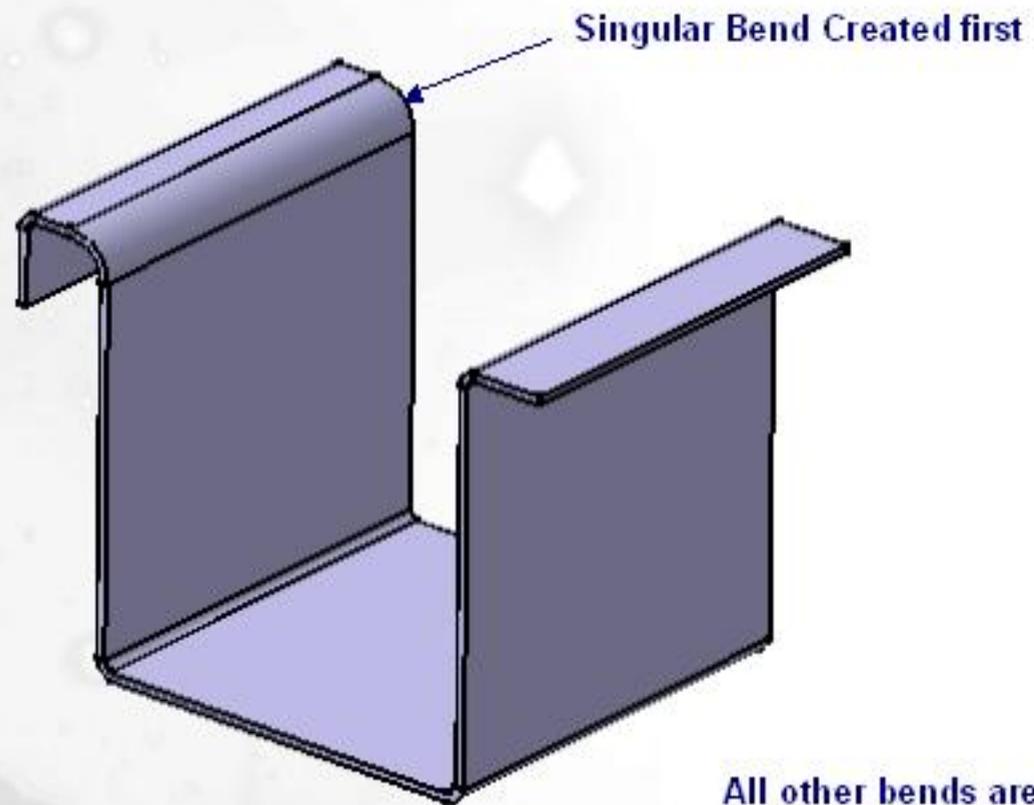
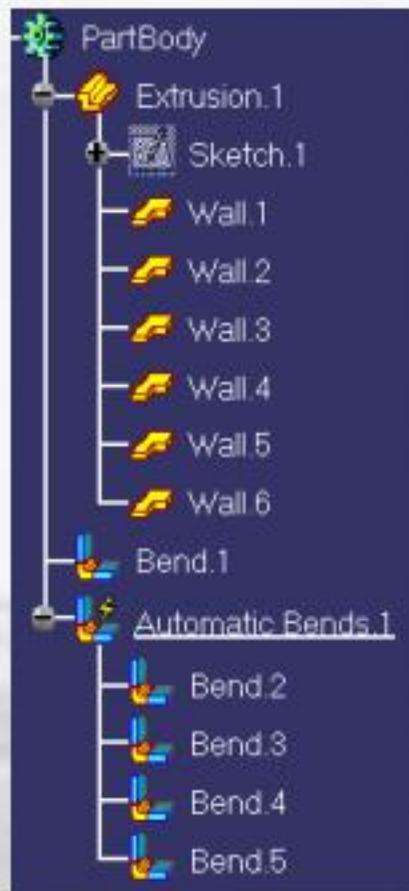


Bend Relief is the material cut out (removed) to allow the folding of the part without tearing the material or causing it deform

When to use Automatic Bends ?

In V5, the new methodology is to create groups of walls which have a common bend size. Allowing a group of walls to be created all at one time and save the steps of having to confirm the direction each time.

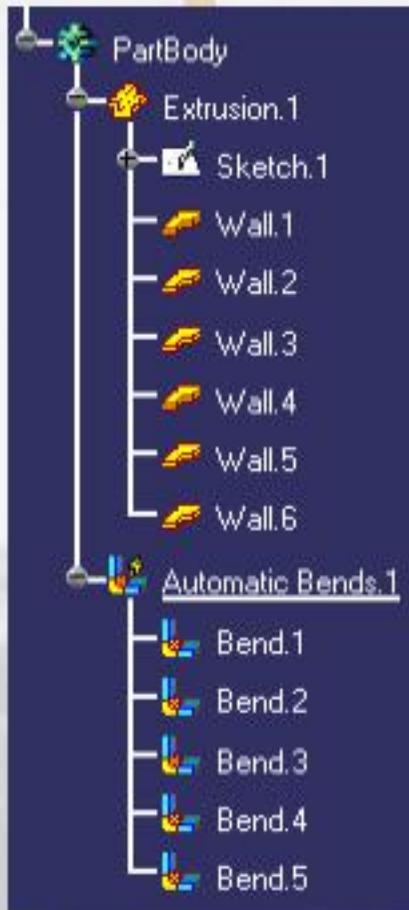
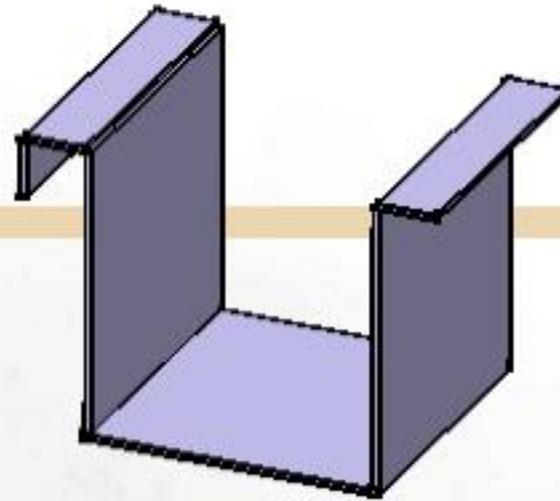
In cases where you have non-standard bends, you will have to use the singular bend capability and define direction. Therefore, it is important to make sure your standard bend is the most common bend size on your part.



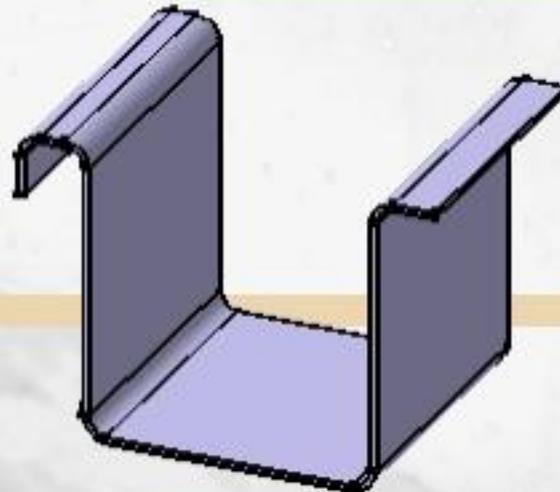
All other bends are created with Automatic Bends

Using the Auto Bend Function

1 Display the part you wish to have the bends added too.



All of the bend radii are created at the default value.

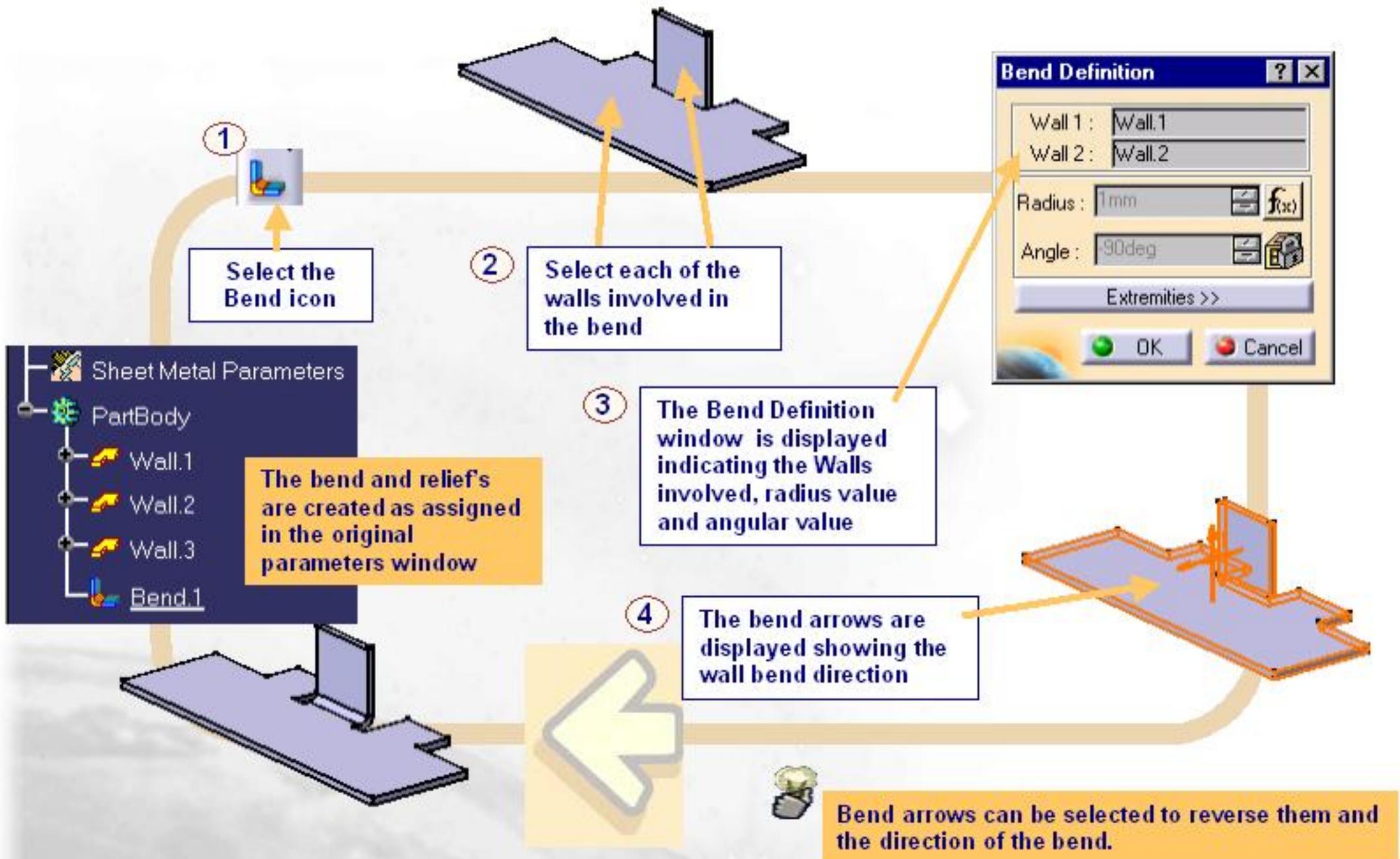


2 Select the Automatic Bend icon

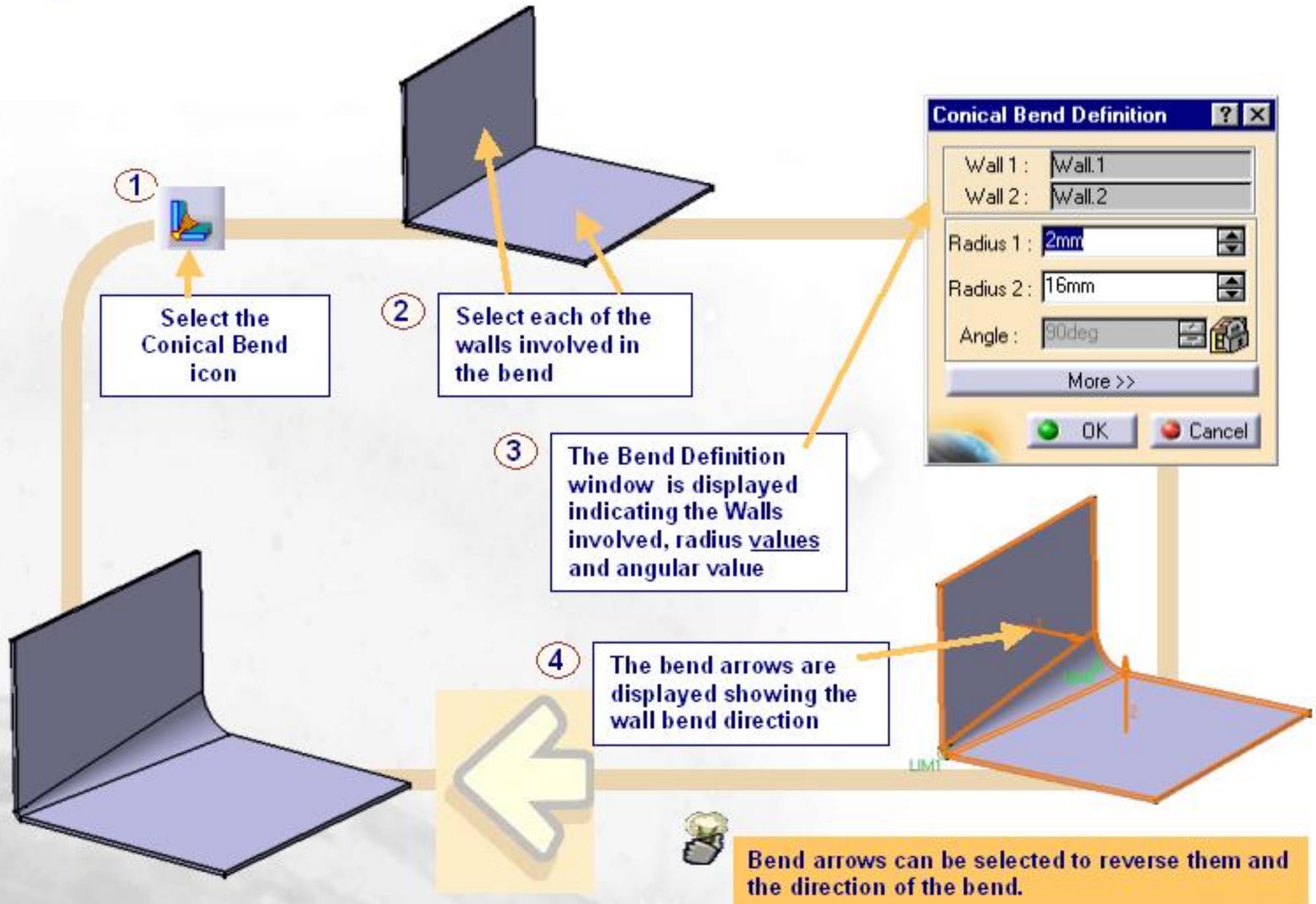


Today, the direction of the bend is chosen for you.

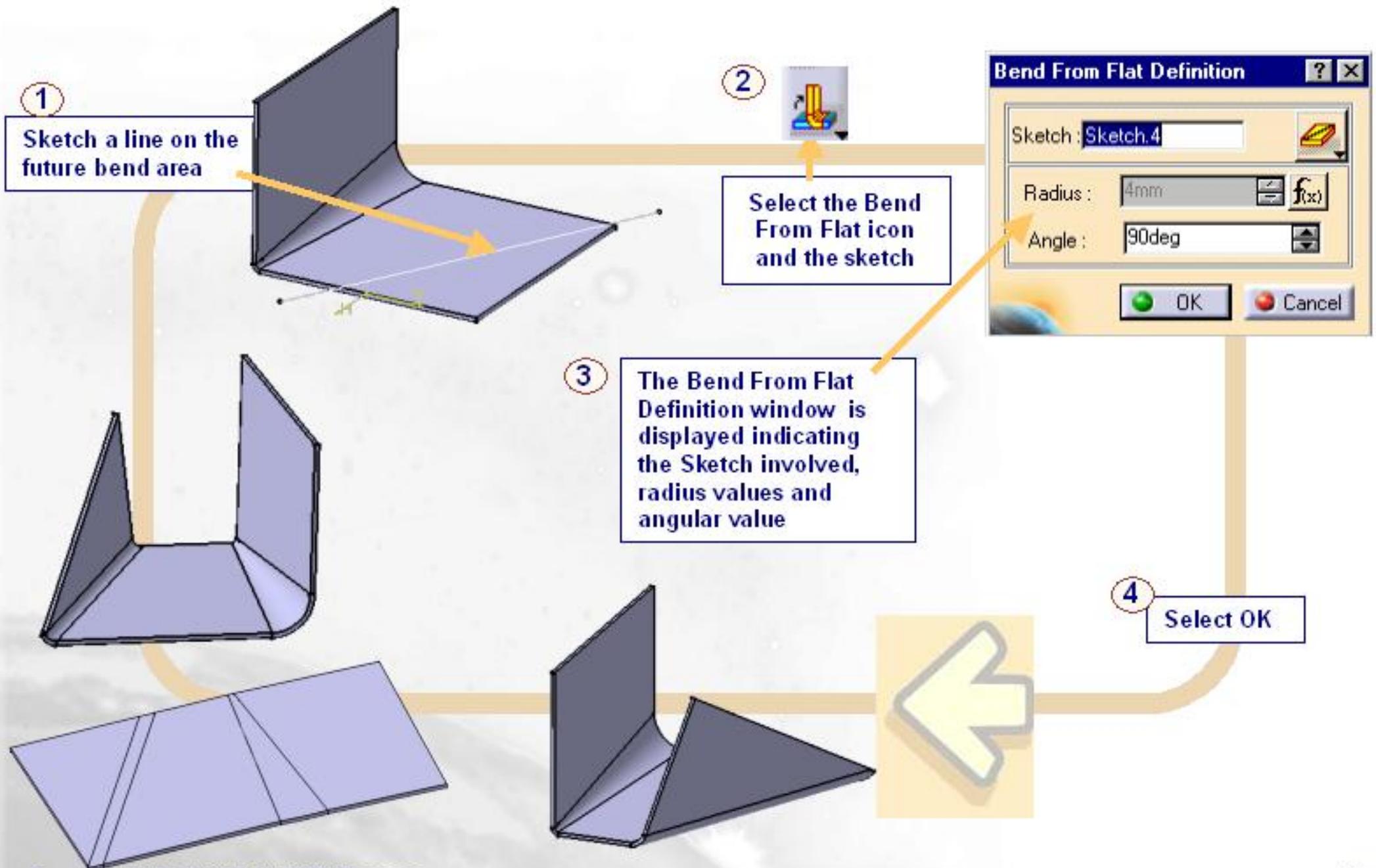
Using the Singular Bend function



Using the Conical Bend function



Using the Bend From Flat function



Modifying a Standard Bend Radius

1 Select the Bend icon

2 Select each of the walls involved in the bend

3 The Bend Definition window is displayed indicating the Walls involved, radius value and angular value

4 Right-click in the Radius field and select Formula/Deactivate

5 Key in the new value in the Bend definition window. Make sure the bend arrows are pointing in the proper direction

The bend and reliefs are created as assigned in the original parameters window

This process can also be used to create a bend by selecting the equation icon

Bend Definition

Wall 1 :	Wall.1
Wall 2 :	Wall.2
Radius :	1mm
Angle :	90deg

Extremities >>

OK Cancel

Formula Edit...
Edit... Deactivate
Add tolerance... Delete

Sheet Metal Parameters

- PartBody
- Wall.1
- Wall.2
- Wall.3
- Bend.1

Using the Corner Relief function

1 Double-click on the bend to edit it

2 Select the *More* button and switch to *Bend Corner Relief* Tab and activate the *Corner Relief* option

Bend Definition

Wall 1: |Wall 1|
Wall 2: |Wall 1|

Radius: |3mm| [f10]
Angle: |30deg| [f10]

<< Less

Bend Corner Relief | Bend Allowance

Corner relief

Radius: |5mm| [f10]

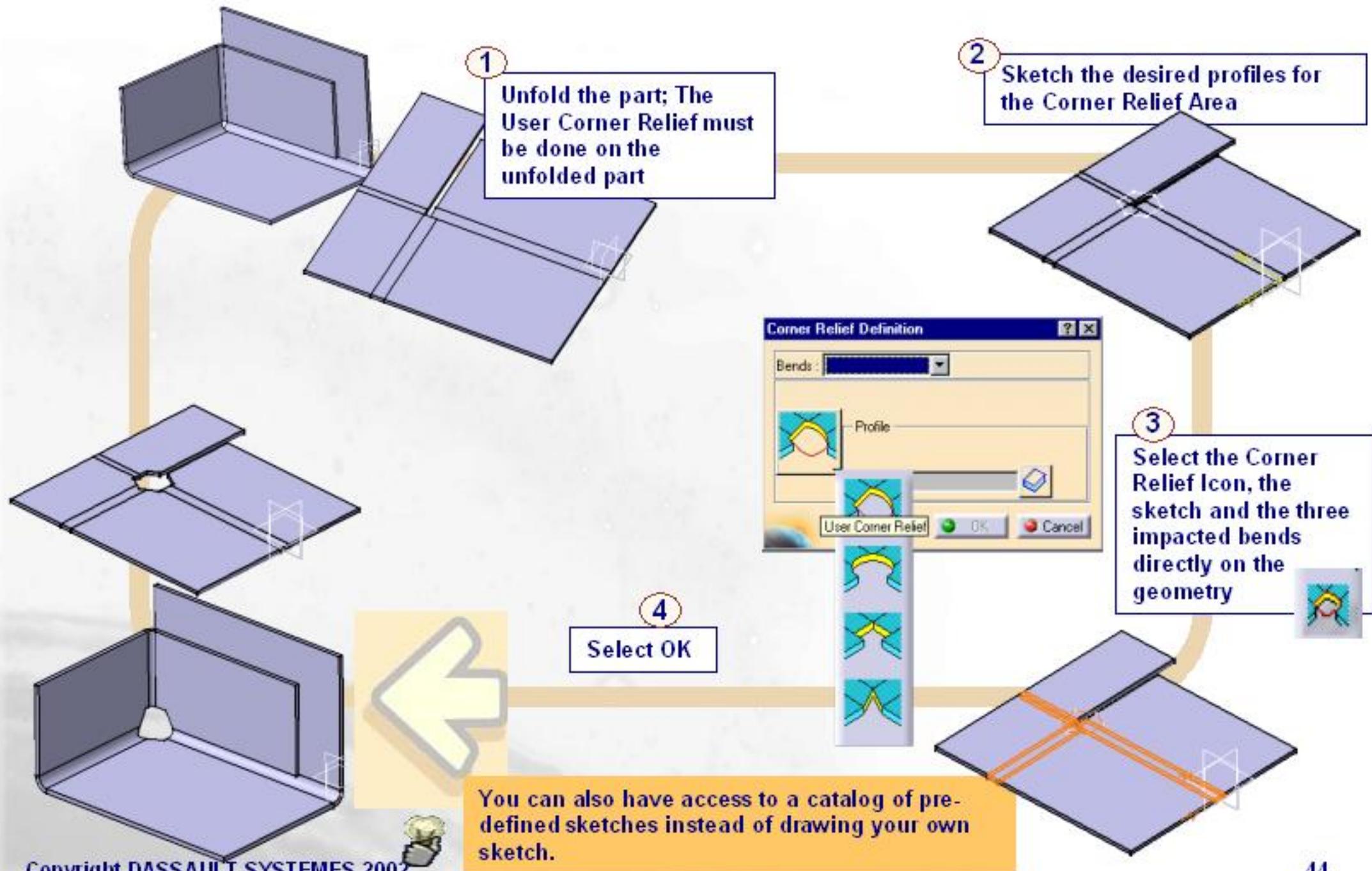
OK Cancel

3 Click OK to validate

You have several shapes for relieves: round, square, triangular.

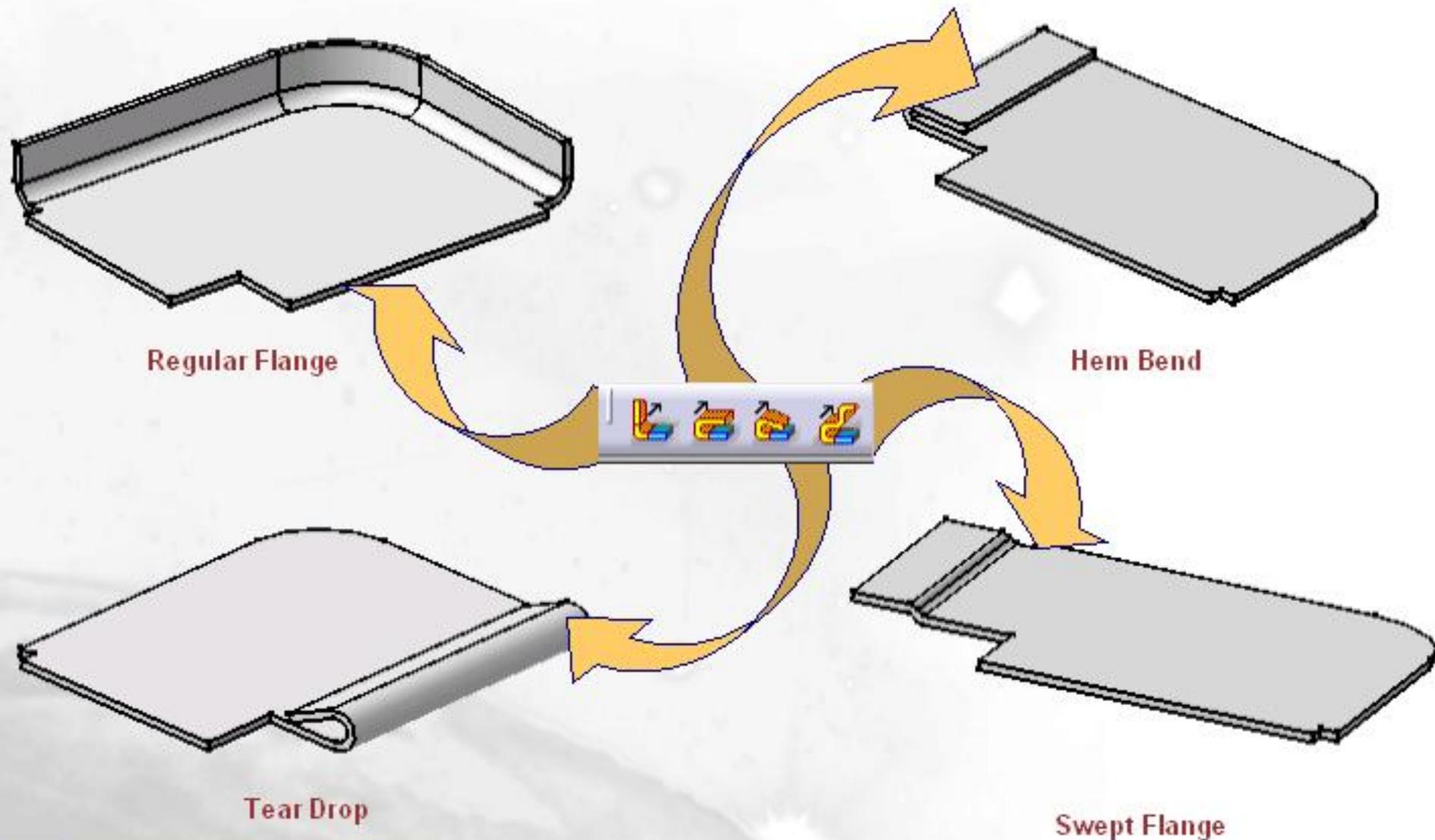
You can also define the Corner Relief in the SheetMetal Parameters

Using the User Corner Relief function



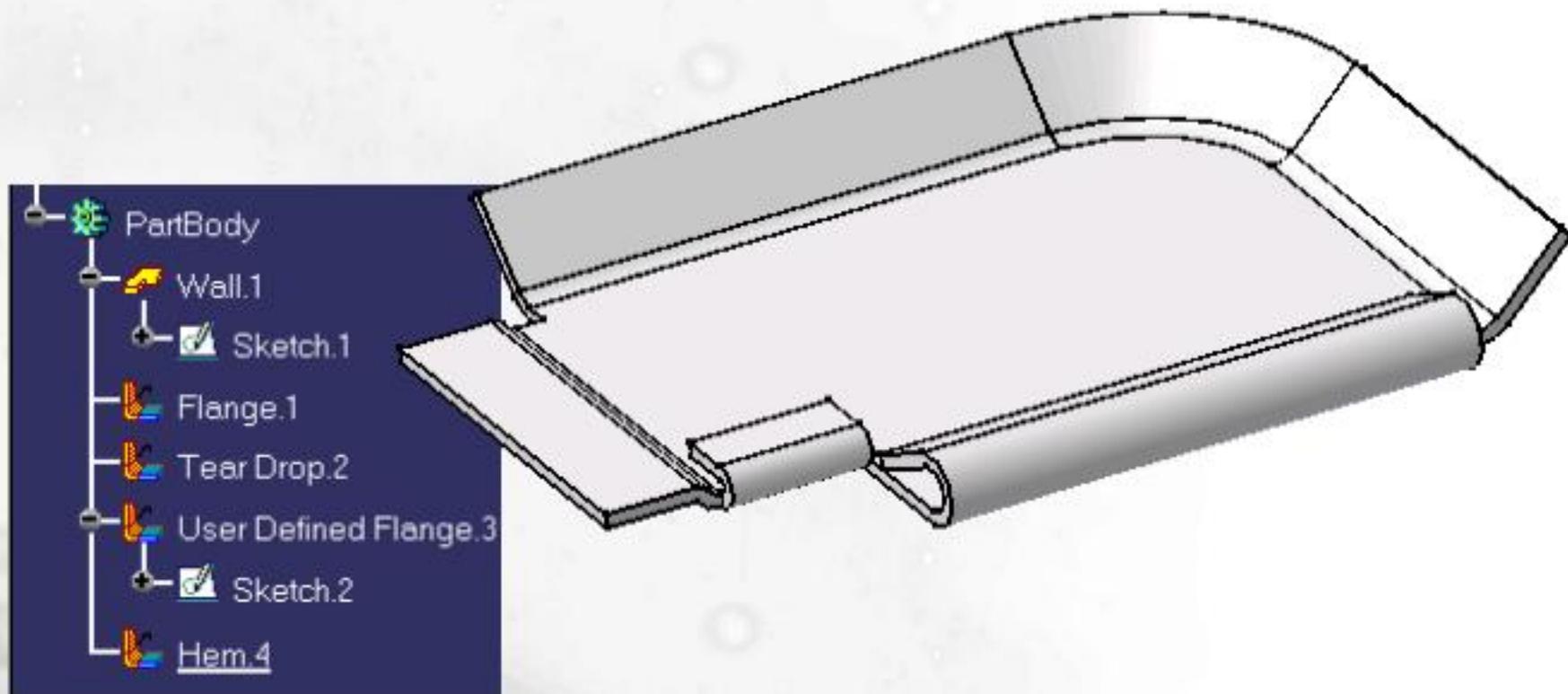
Defining Flanges

Flanges come in four different types: Regular Flange, Hem Bend, Teardrop and Swept Flange.



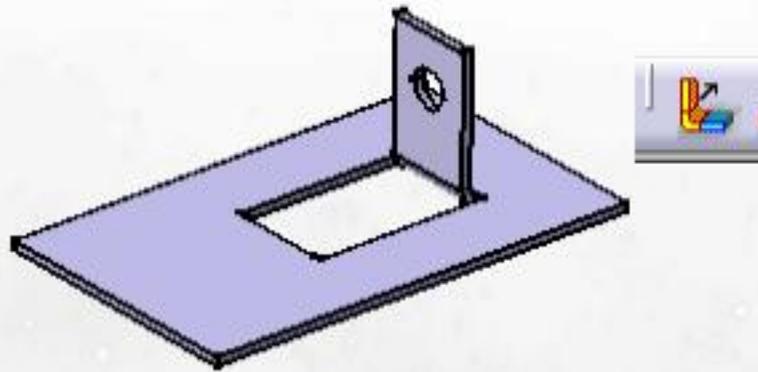
What are Flanges ?

Flanges are different ways to form sheet metal into tabs, walls and edges that give functionality, beauty, strength and safety to the part.

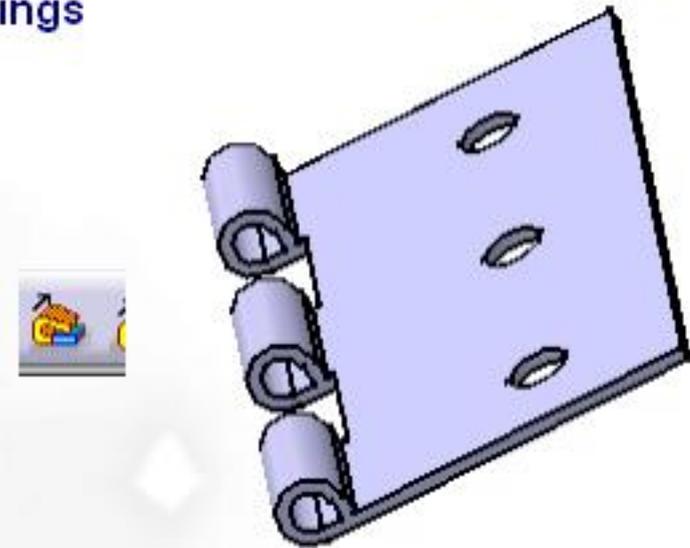


When to use the various types of Flanges ?

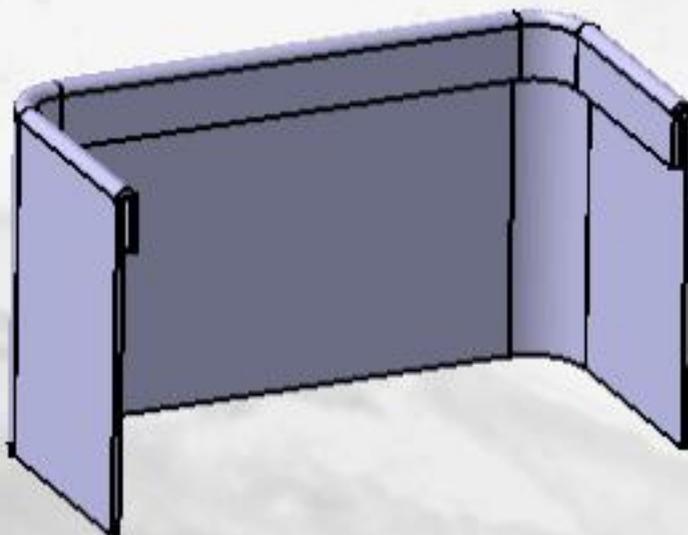
Regular Flanges are used to create tabs in cut outs and simple flanges around the edges of parts



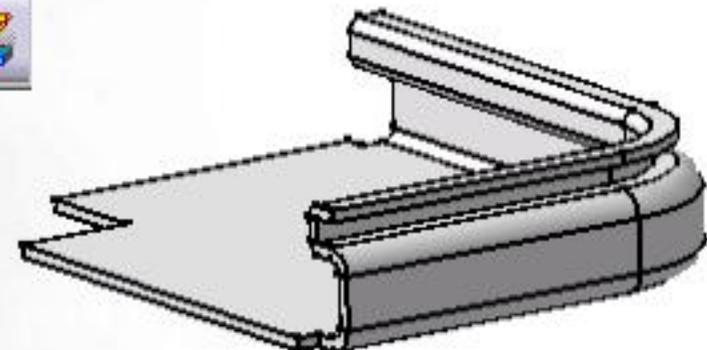
Tear Drops can also serve to create decorative edges as well as create things like hinge pins openings



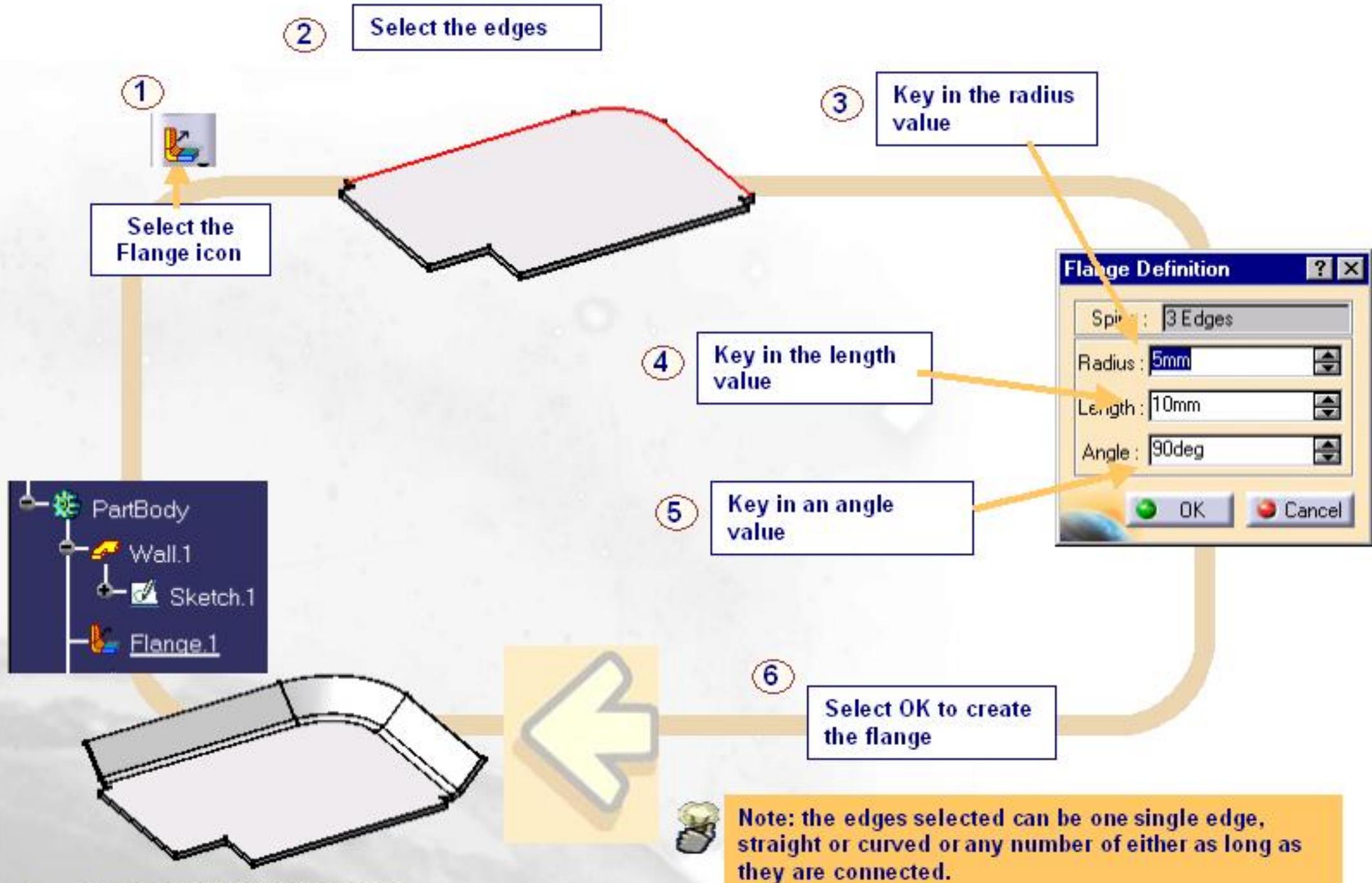
Hem bends are used to strengthen long thin sheet metal parts and to also supply a smooth safe edge



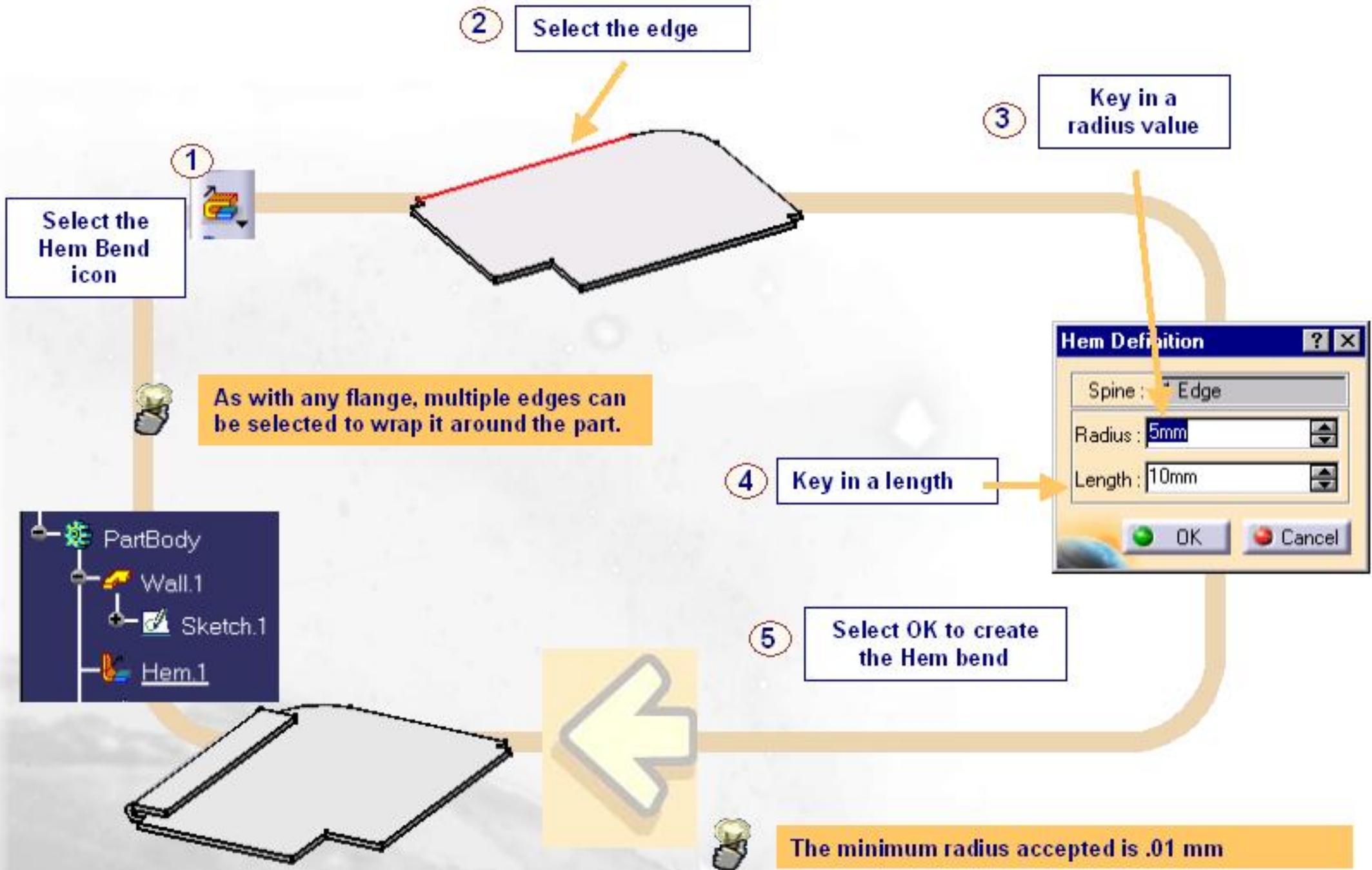
Swept flanges are used to sweep profiles around exterior edges of parts.



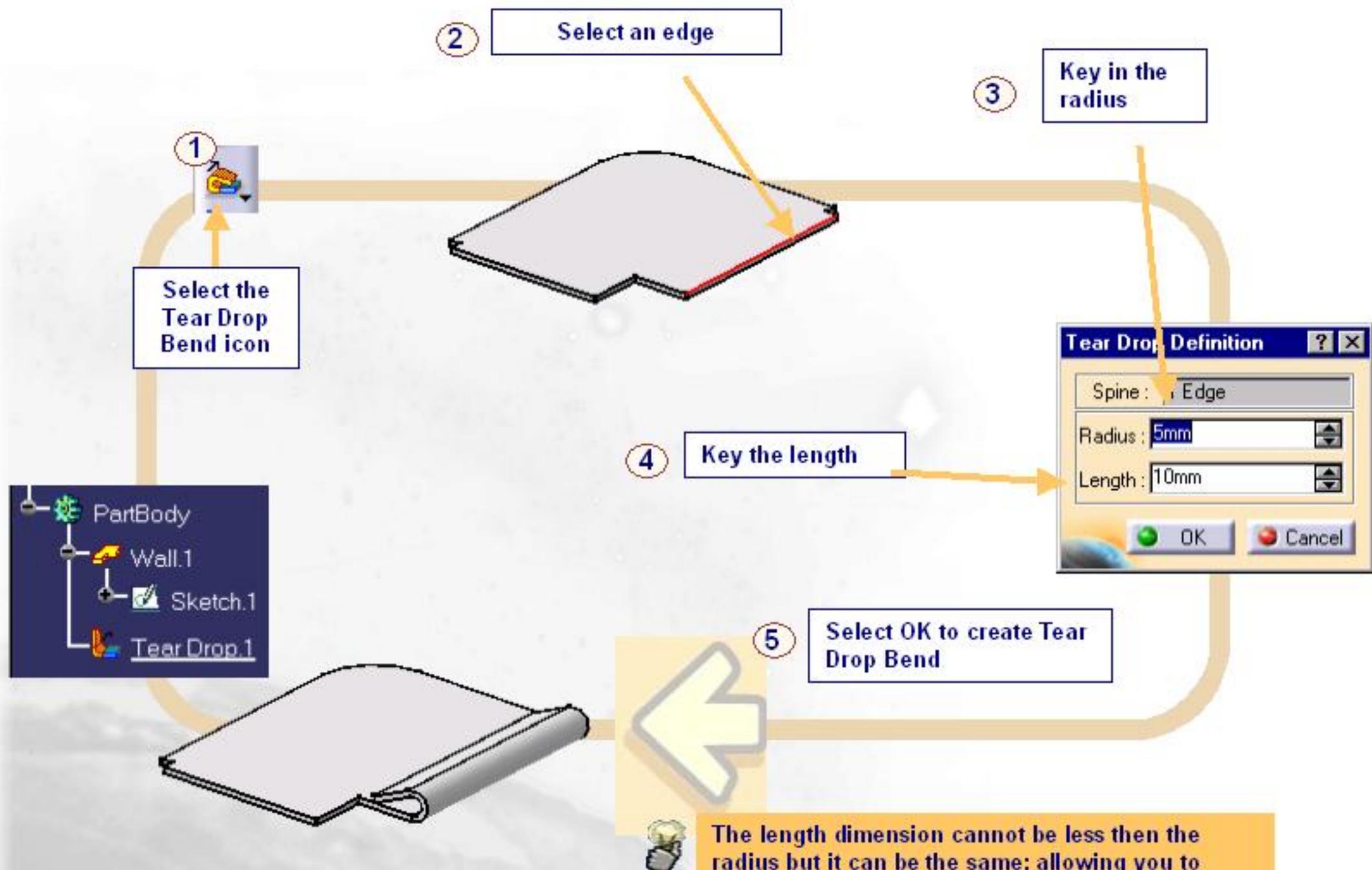
Simple Flanges



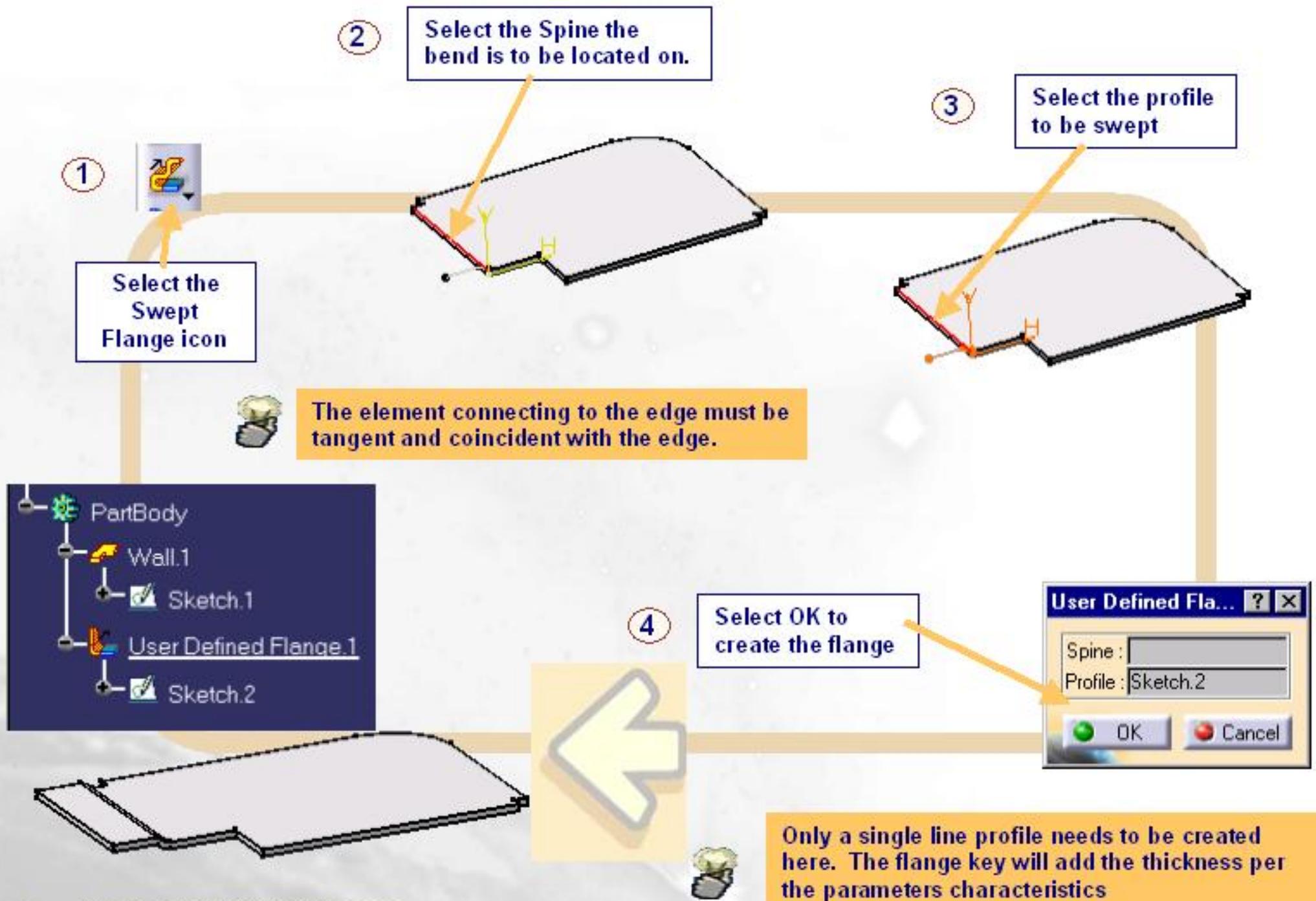
Hem Bends



Tear Drop Flanges



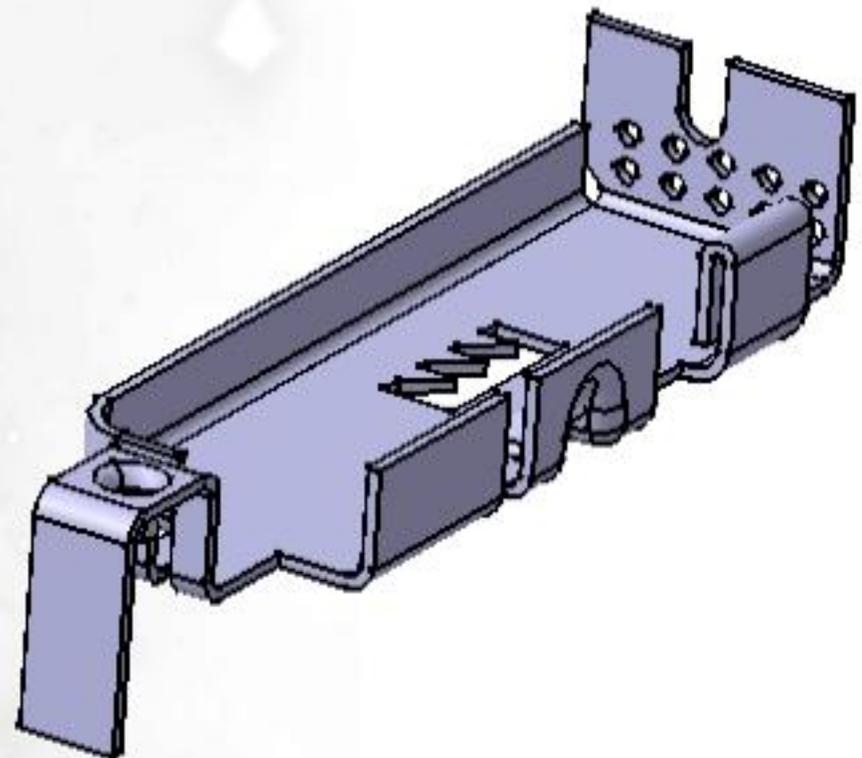
Swept Flanges



Sheet Metal Cut Outs, Flat Pattern Mode and Features

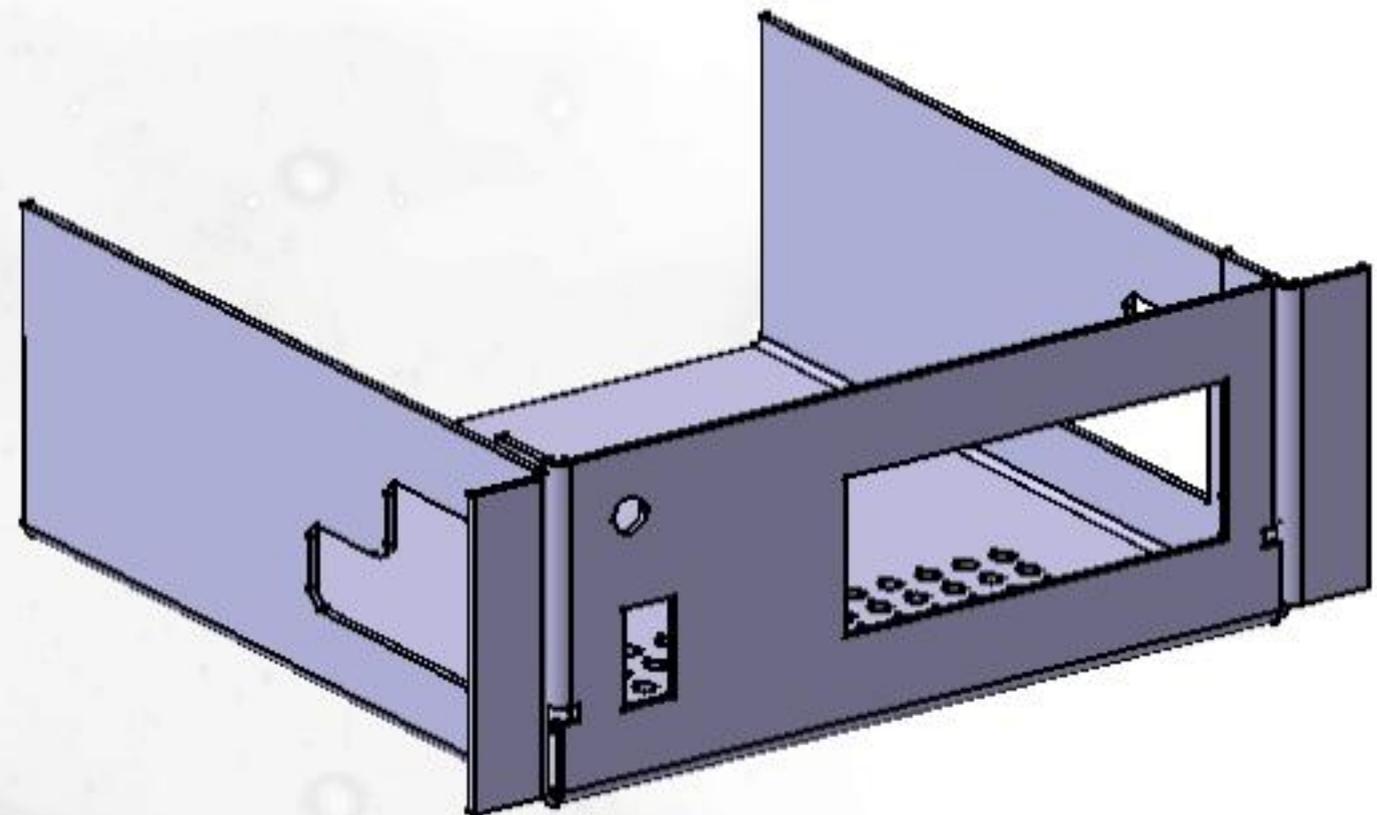
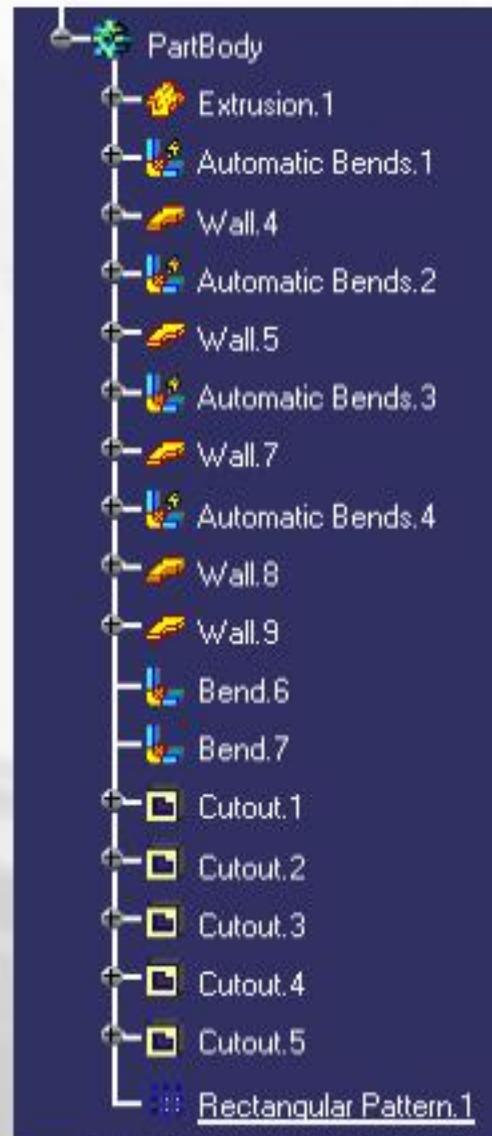
In this lesson, you will see how to add Cut Outs to parts, pattern them, as well as how to make use of the 3D Features available.

- Defining Cut Outs in Walls
- Going to Flat Pattern Mode
- Defining a 3D Feature
- Sheet Metal Cut Outs, Patterns and Features recap exercise



Defining Cut Outs in Walls

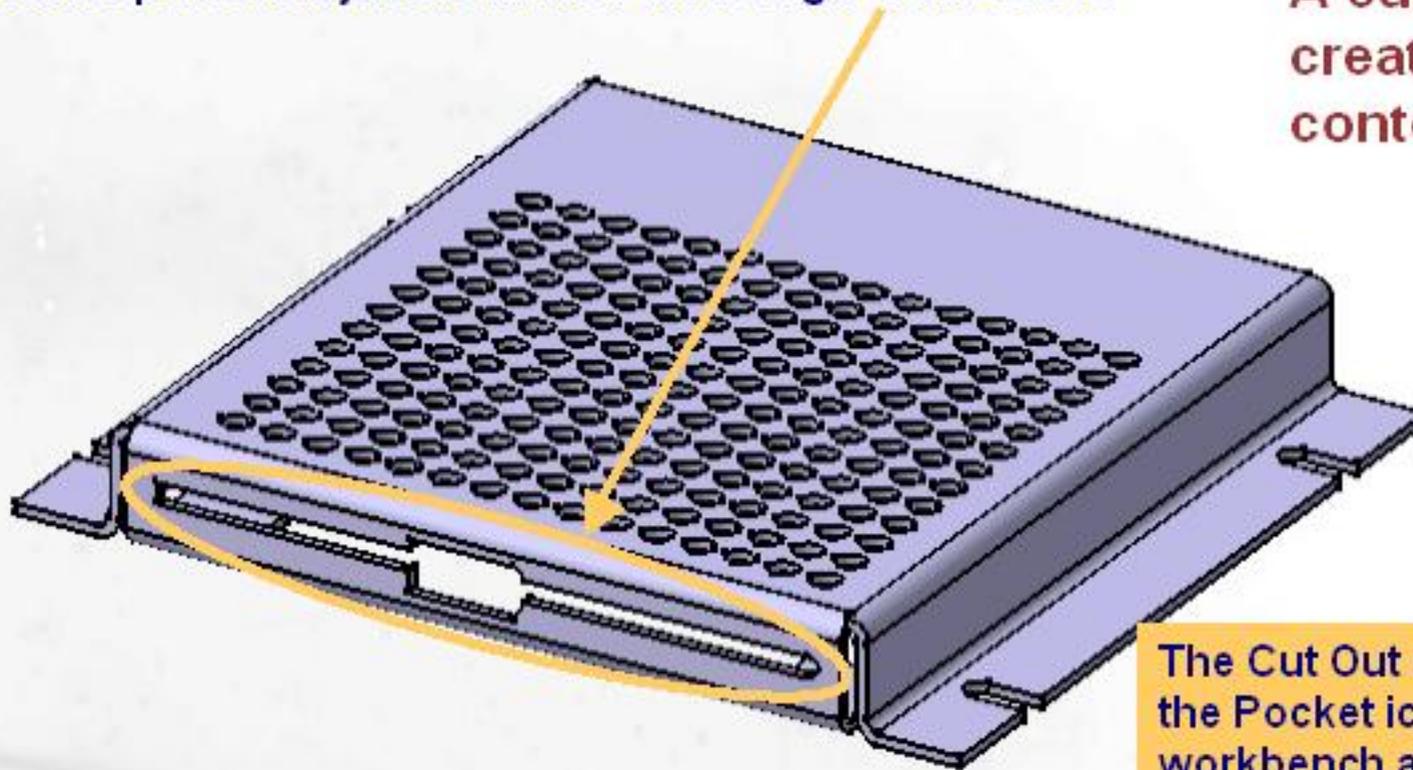
In the Sheet Metal workbench there are no Holes or Pocket icons as in the Part Design workbench. Openings in a Sheet Metal part are all done with a Cut Out icon.



What are Cut Outs ?

A Cut Out is any opening in a Sheet Metal wall. It can be anything, from as simple as circular hole to a more complex cut out. Cut Outs can be patterned just as in the Part Design workbench.

A cut-out can be created with an open contour sketch.

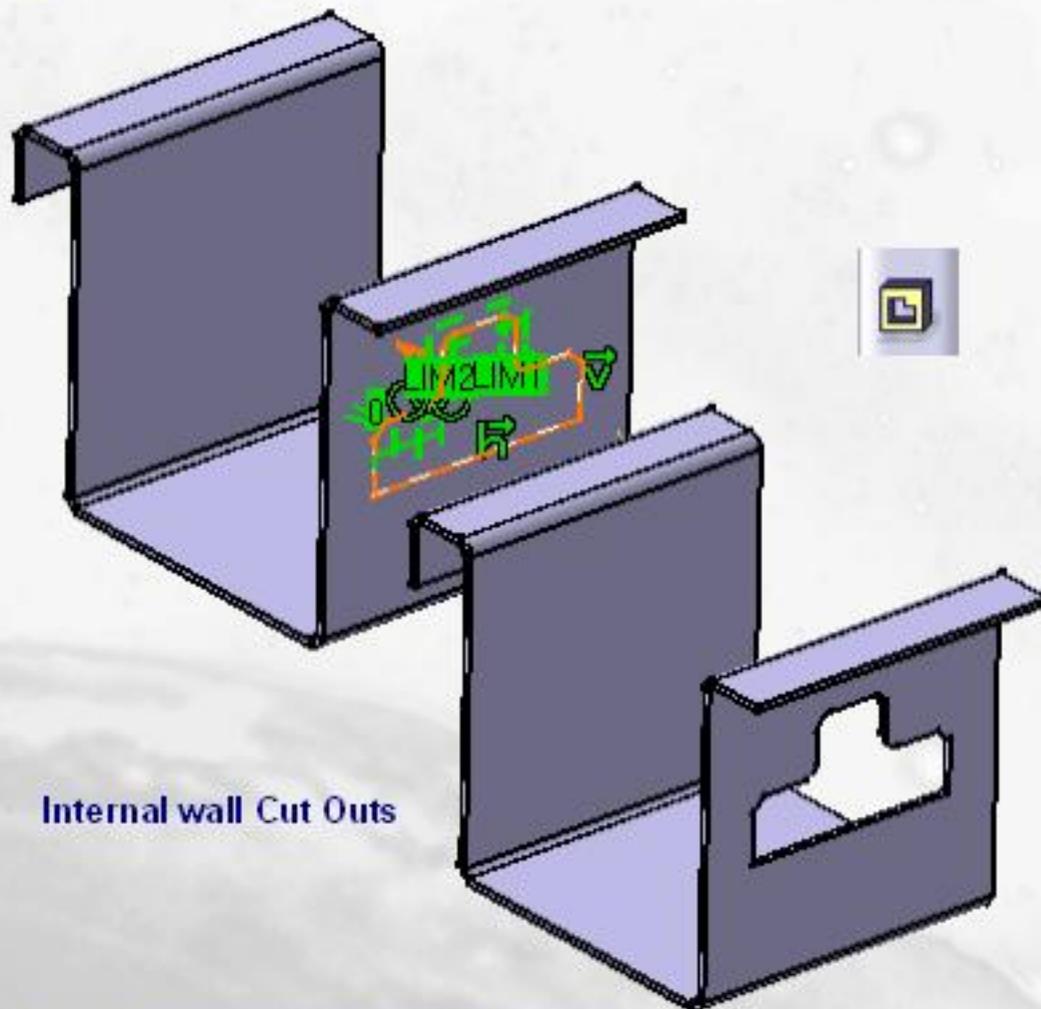


The Cut Out icon is very similar to the Pocket icon in the Part Design workbench and has all of the depth capabilities such as “Up to Next” and “Up to Last”

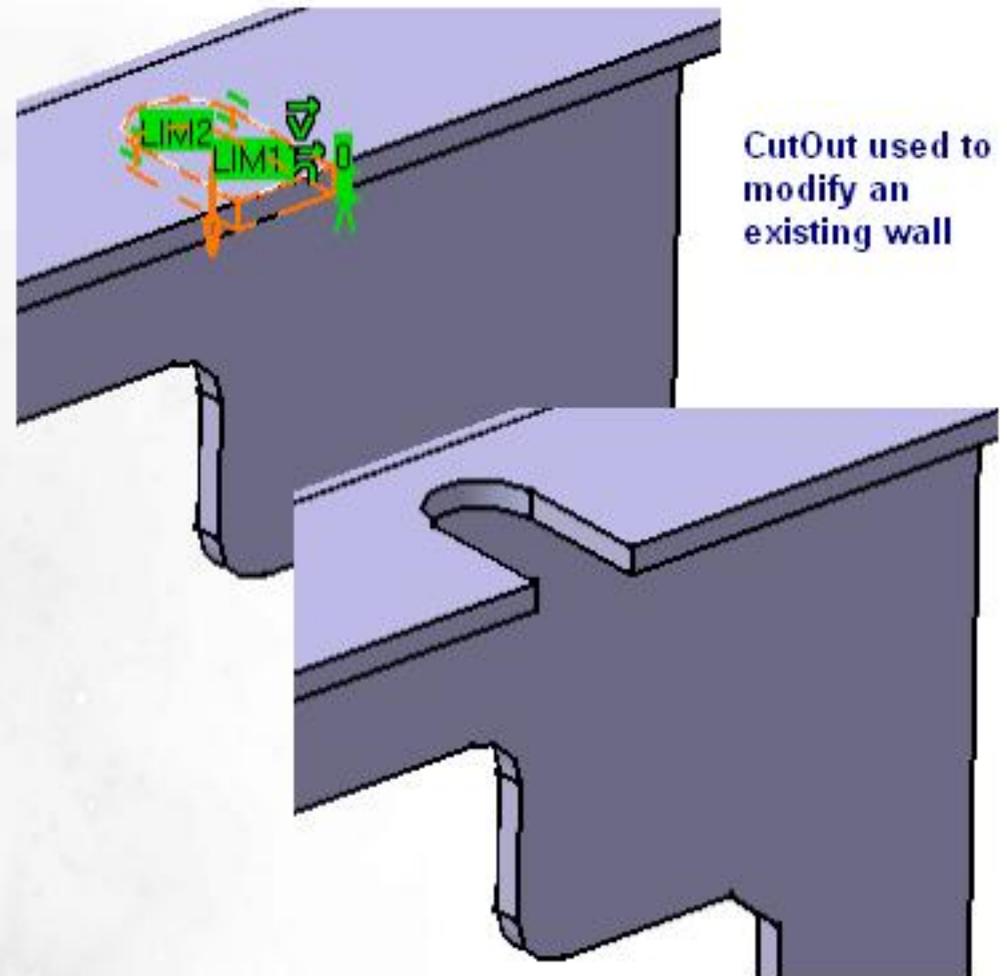
When to use Cut Outs ?

Cut Outs are used to subtract material to create a hole, pocket, notch, slot, or any other type opening. They can be as simple as a circle or more complex by having many line, arc, spline segments.

After creating a wall, it is possible to add additional slots on the exterior of the wall, add notches, or chamfers with cut outs without modifying the wall sketch.

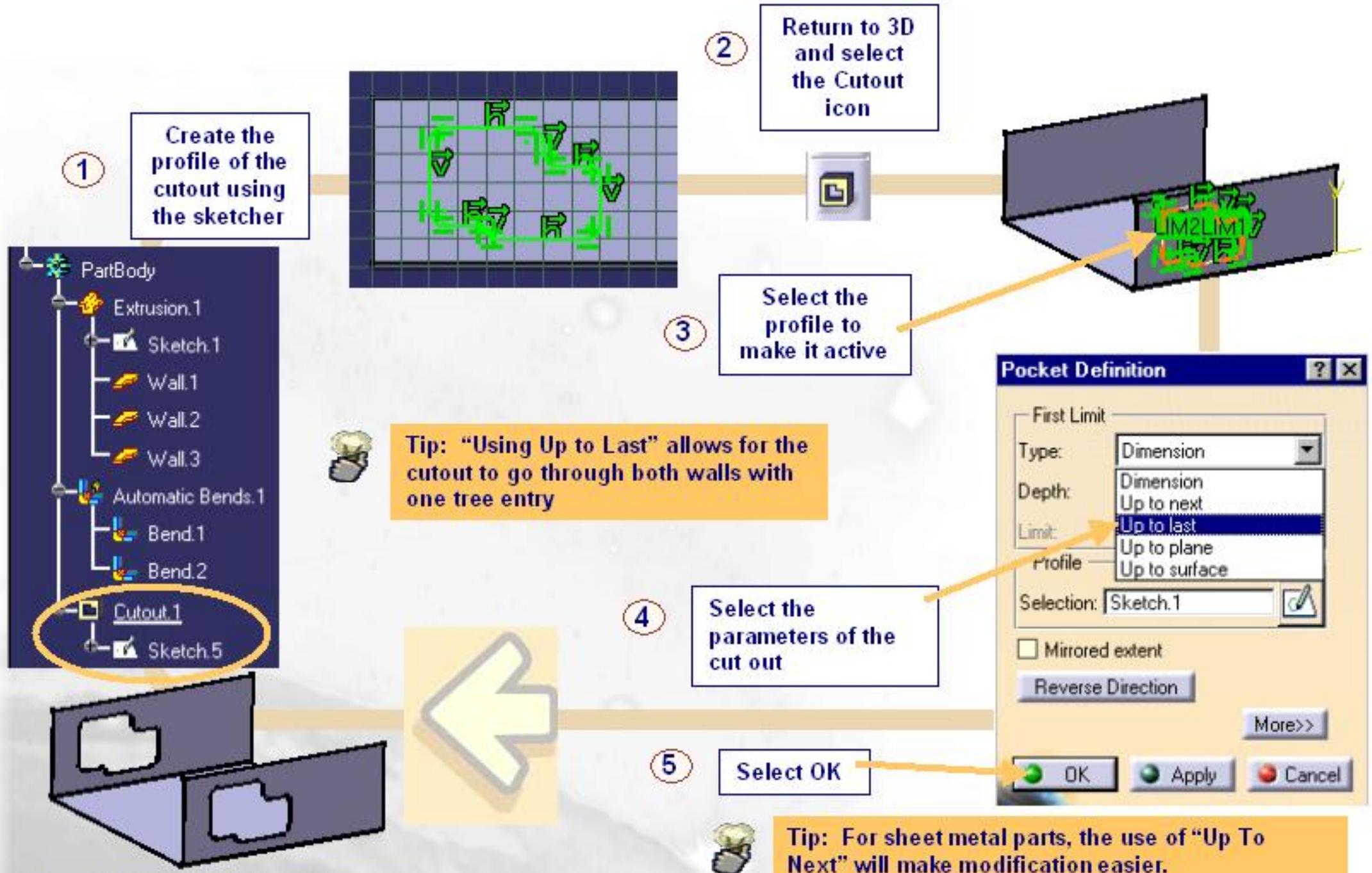


Internal wall Cut Outs

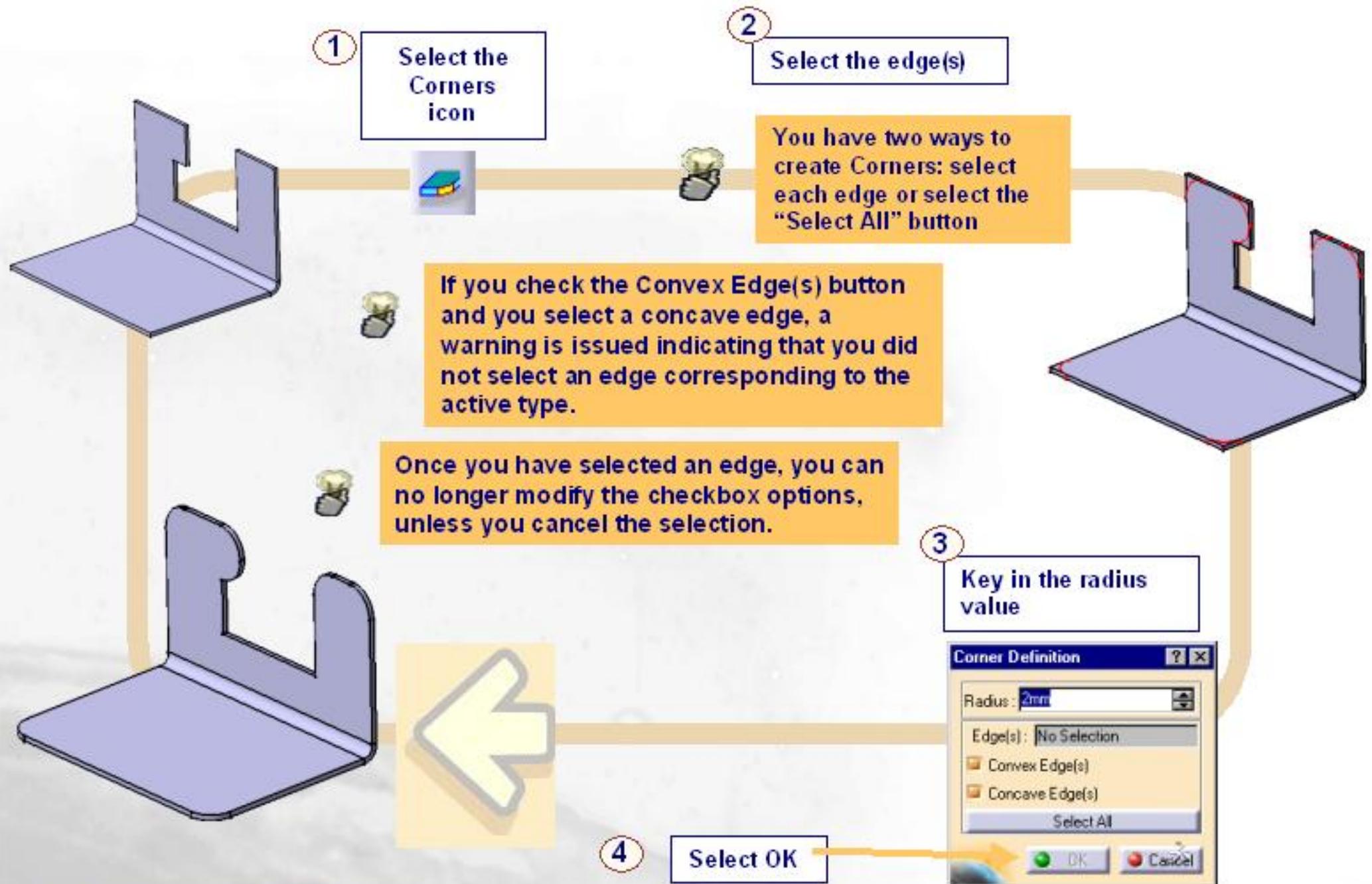


CutOut used to modify an existing wall

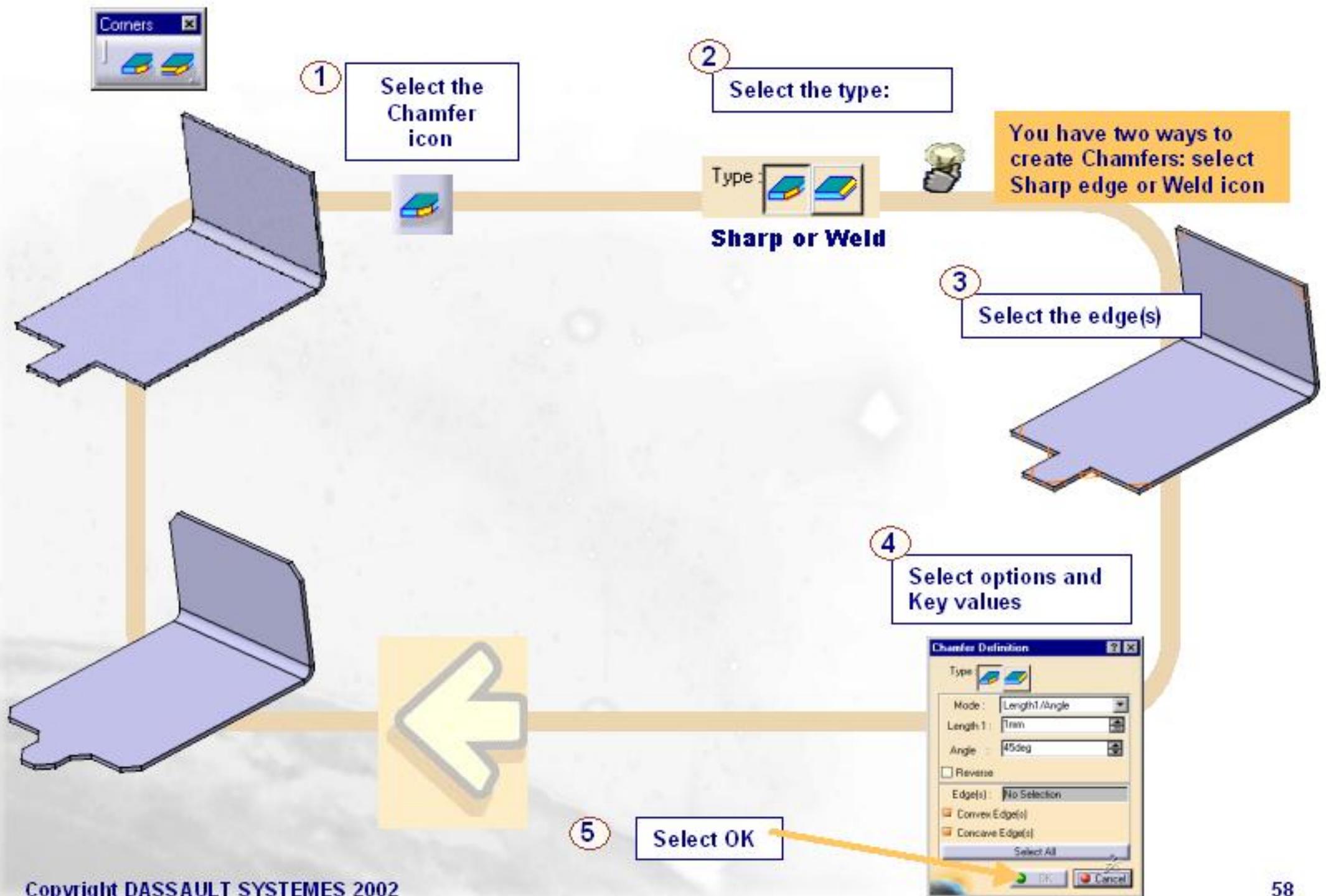
Creating a Cut Out



Creating Corners



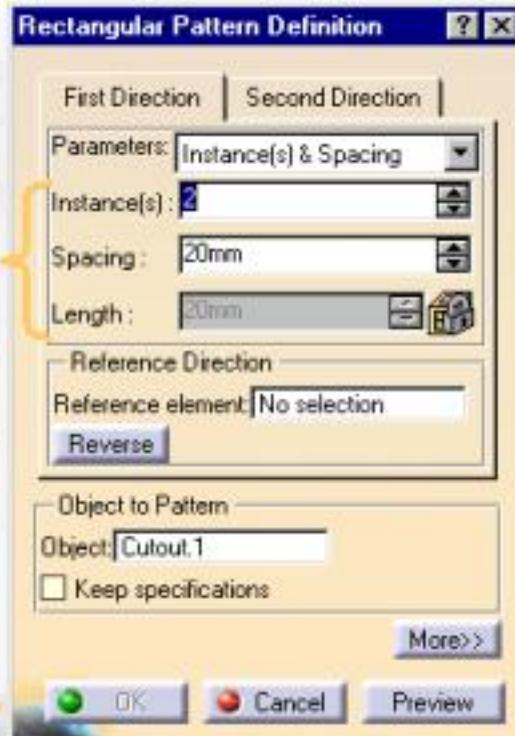
Creating Chamfers



The different type of Patterns

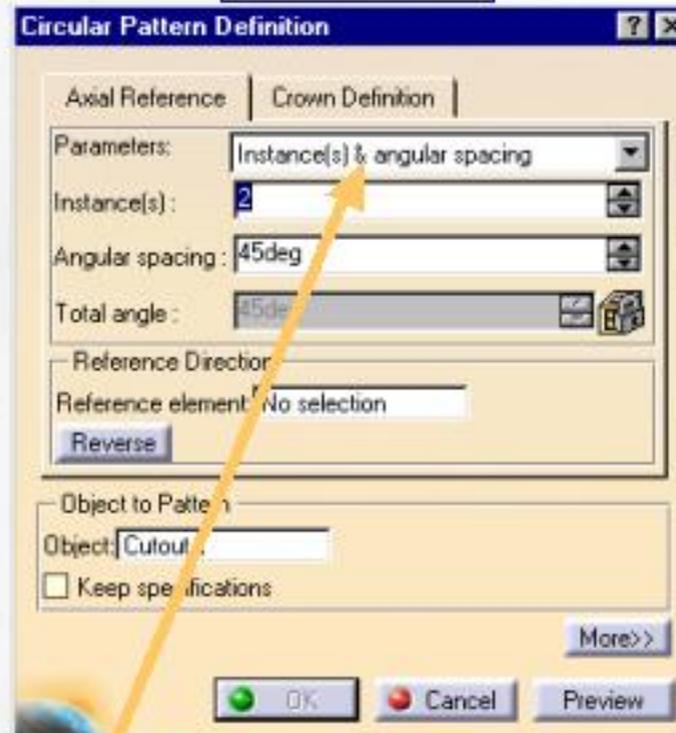


Rectangular Pattern



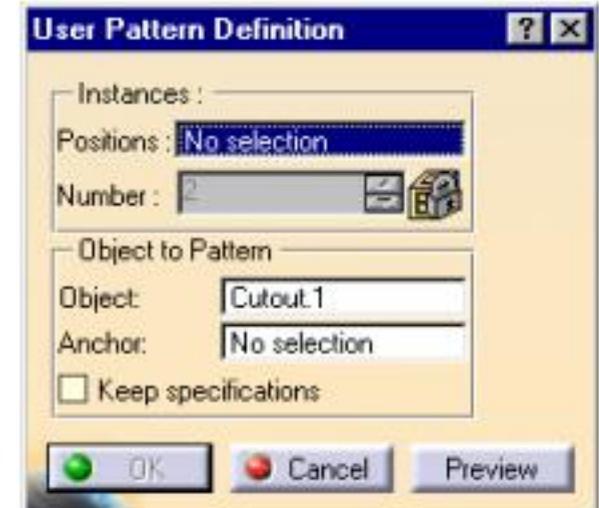
Several Parameter configurations. On each case, fill two of the three parameters. The reference direction element can be selected directly on the geometry (edges).

Circular Pattern



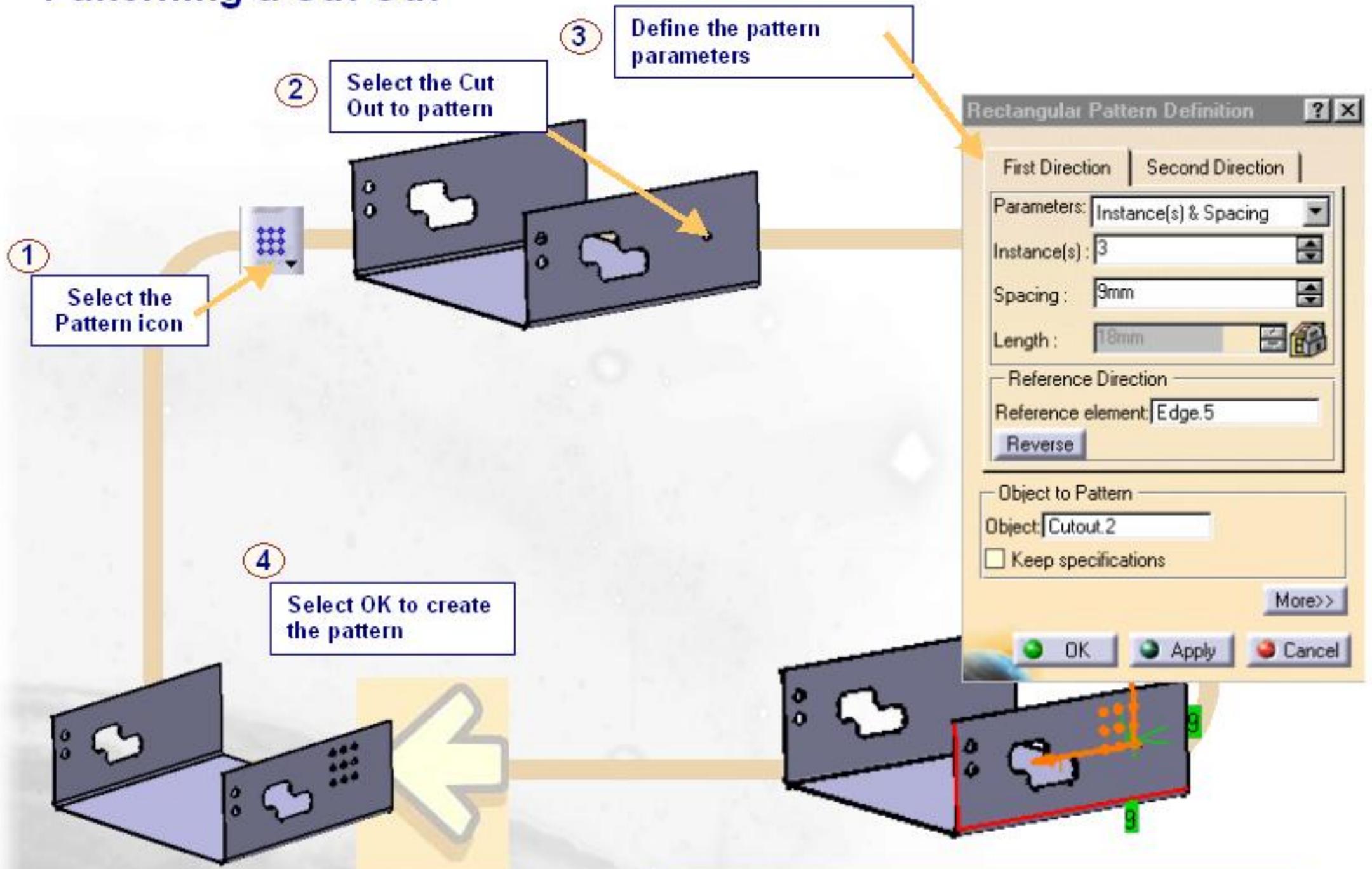
The "Complete Crown" Parameter is very useful to instantiate a feature all around a crown.

User Pattern



For the position, select a sketch containing points where the new features will be instantiated.

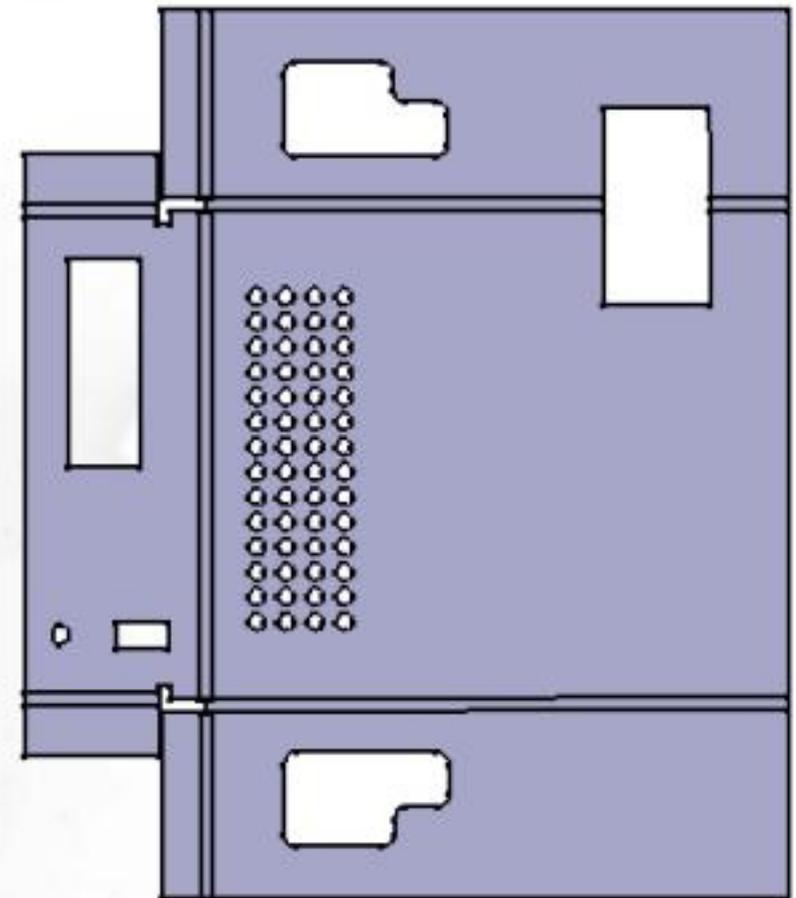
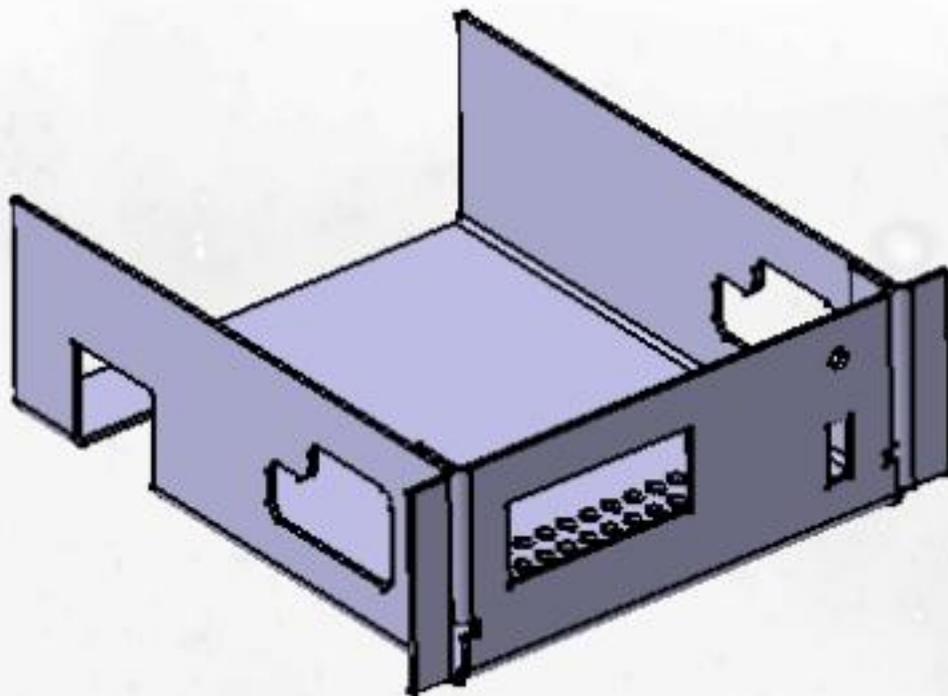
Patterning a Cut Out



Note: instances of the pattern can be turned off achieve a staggered pattern

Going to Flat Pattern Mode

When designing a Sheet Metal part, you may find times when you need to see the part in its Flattened state for checking or adding physical cut outs.

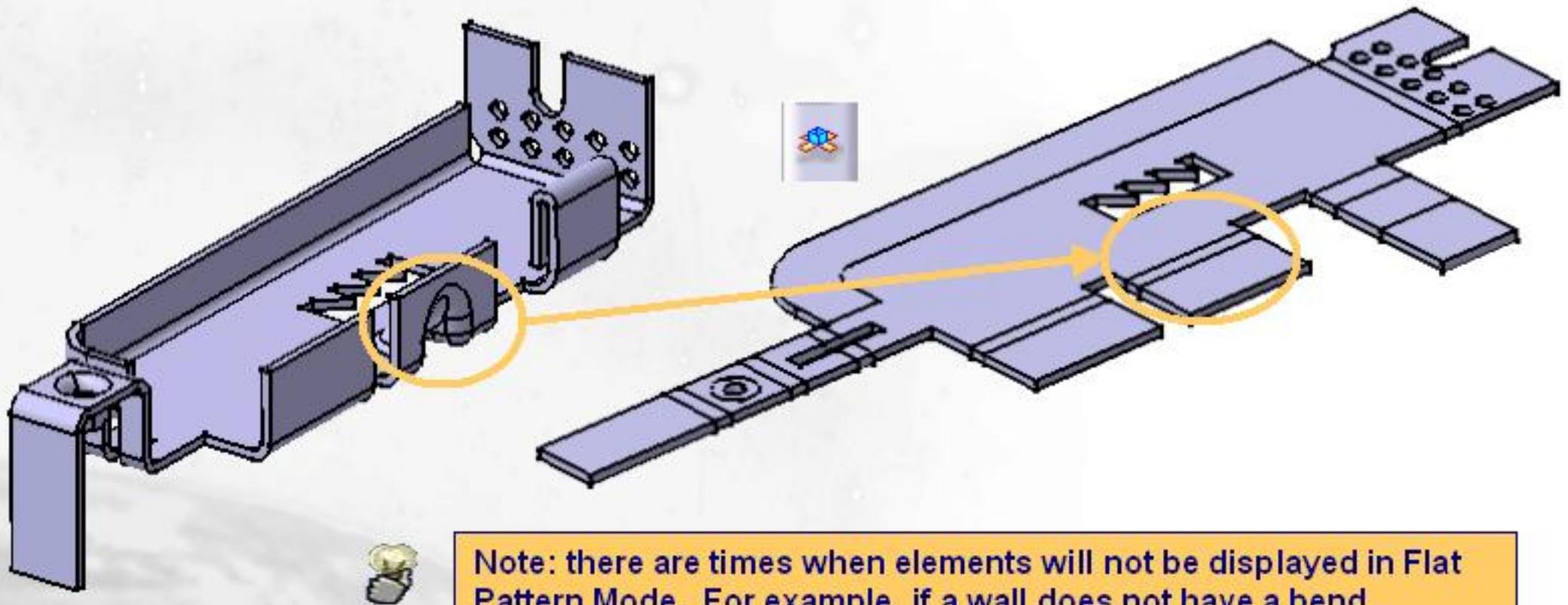


The part can also be display in folded and the unfolded states in two different windows at the same time



What is Flat Pattern Mode ?

Flat Pattern Mode is when you are viewing the sheet metal part in it's unfolding condition. Allowing all of the cut outs, edges, bend limits and foot print stamp to be displayed

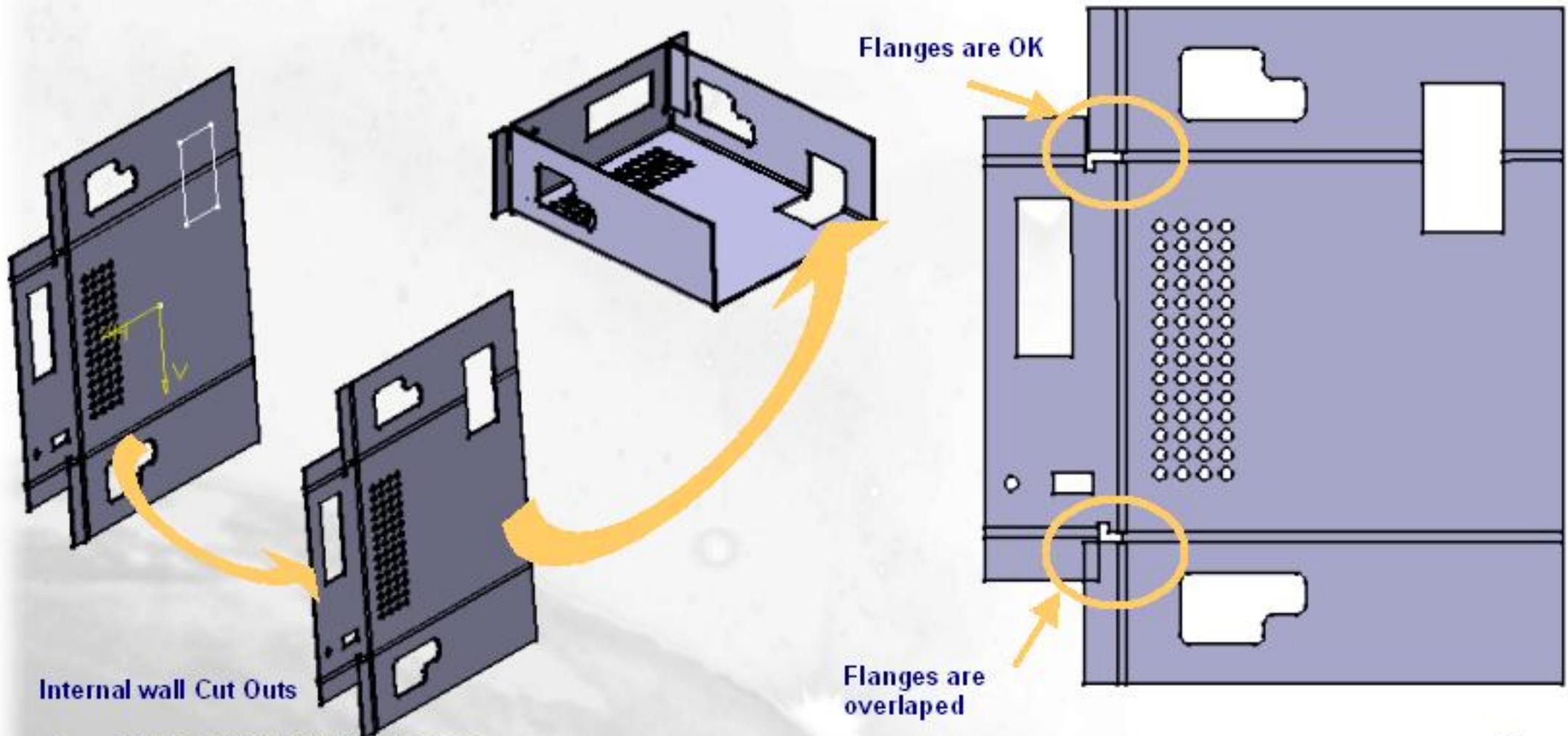


Note: there are times when elements will not be displayed in Flat Pattern Mode. For example, if a wall does not have a bend assigned to it. Also, if a stamp feature is created over the limit between several supports, such as walls, bends, and so forth, this stamp is also not visible.

When to use Flat Pattern Mode ?

Flat Pattern Mode should be used when a Cut Out will be part of more than one wall and go through a bend. This allows it to be located from the edges on the flattened pattern of the part.

Another use of Flat Pattern Mode is to be able to locate any overlap of walls. Typically, this occurs where flanges overlap each other.

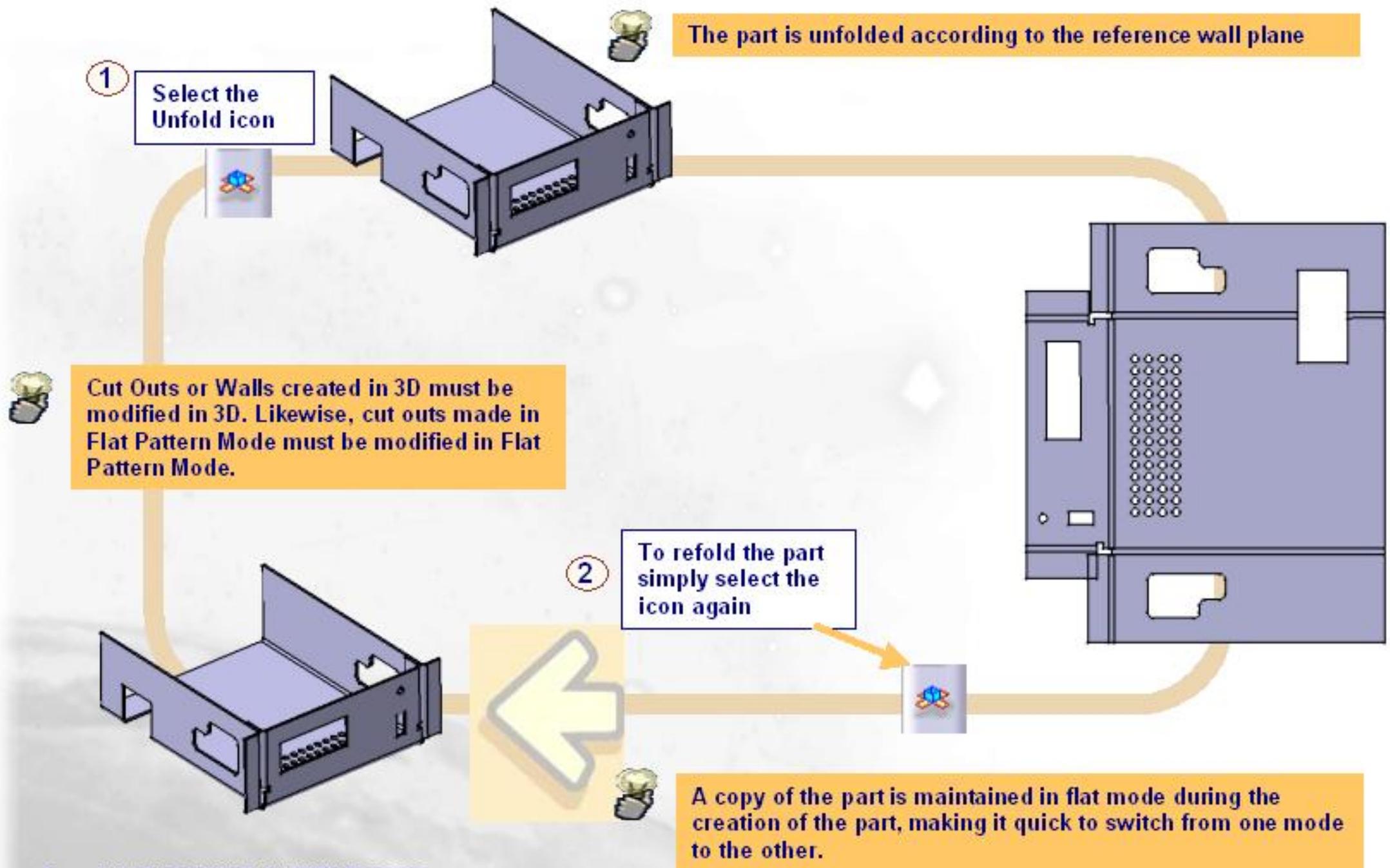


Internal wall Cut Outs

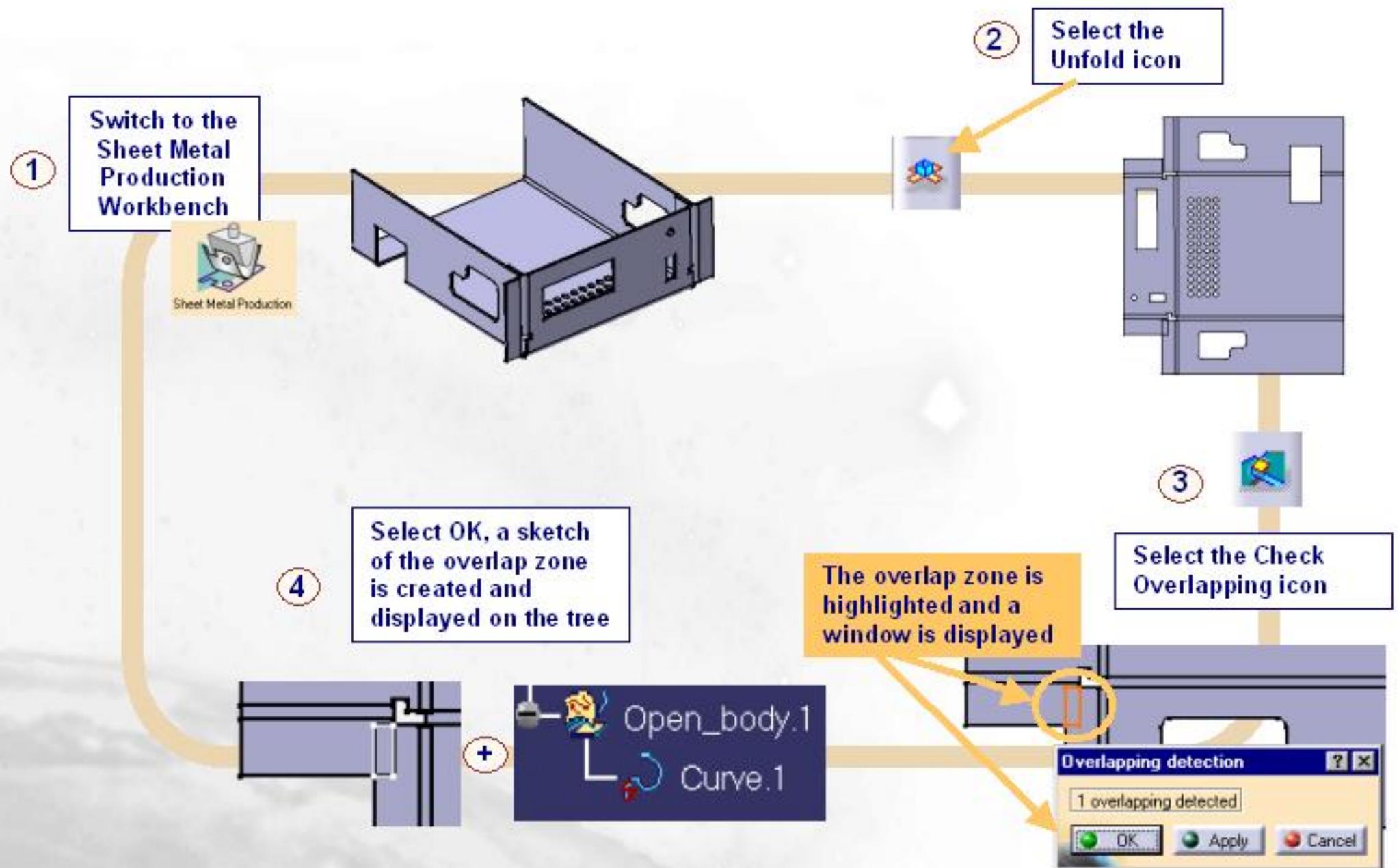
Flanges are OK

Flanges are overlaped

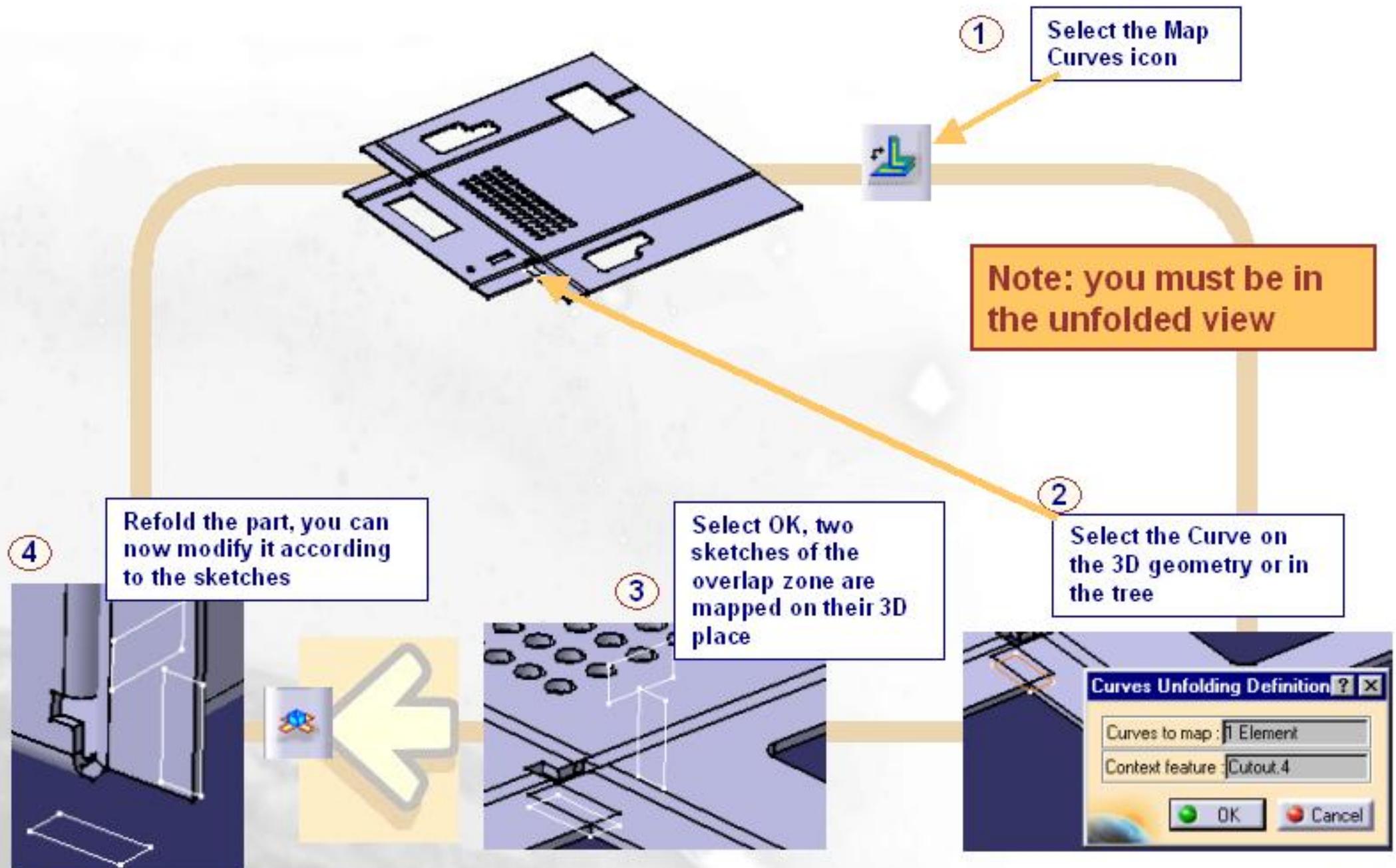
Going to Flat Pattern/Part Mode



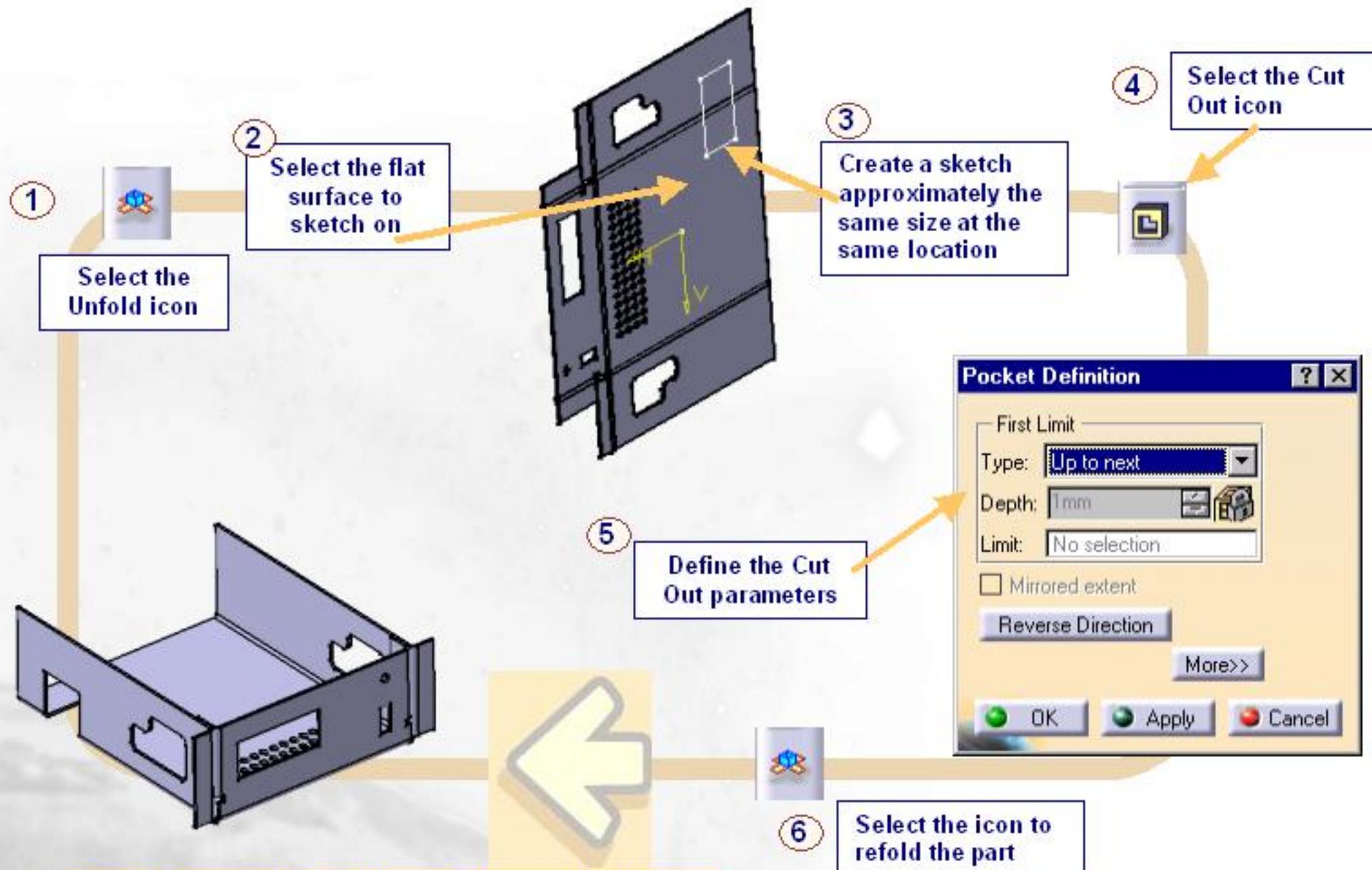
Checking the overlaps



Checking the overlaps



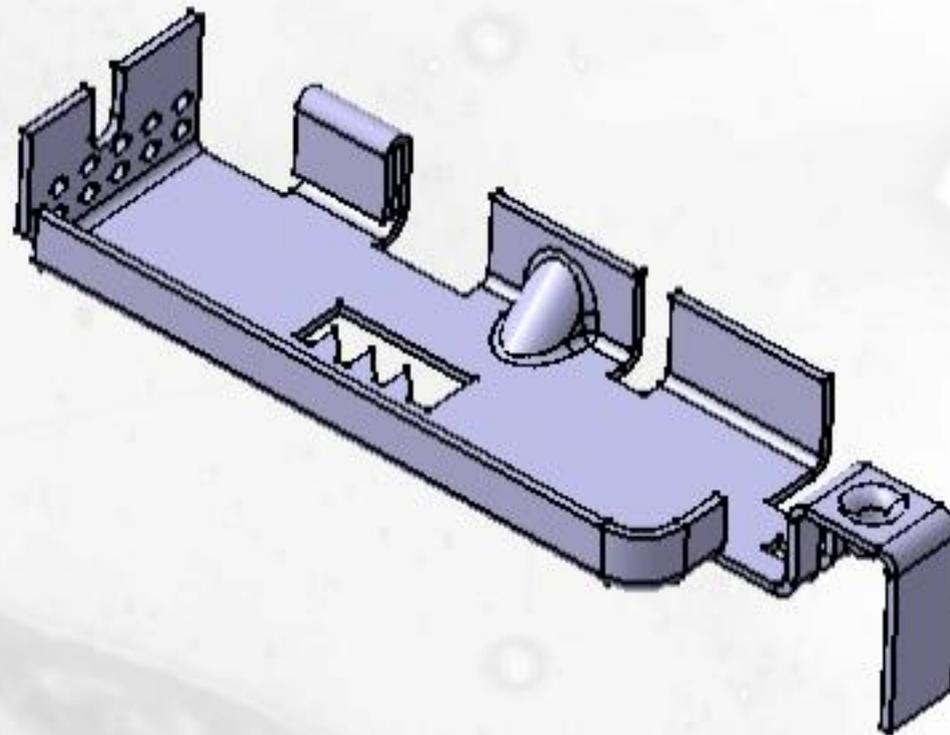
Creating a Cut Out in Flat Pattern Mode



Cut Out's created in Flat Pattern Mode can also be patterned

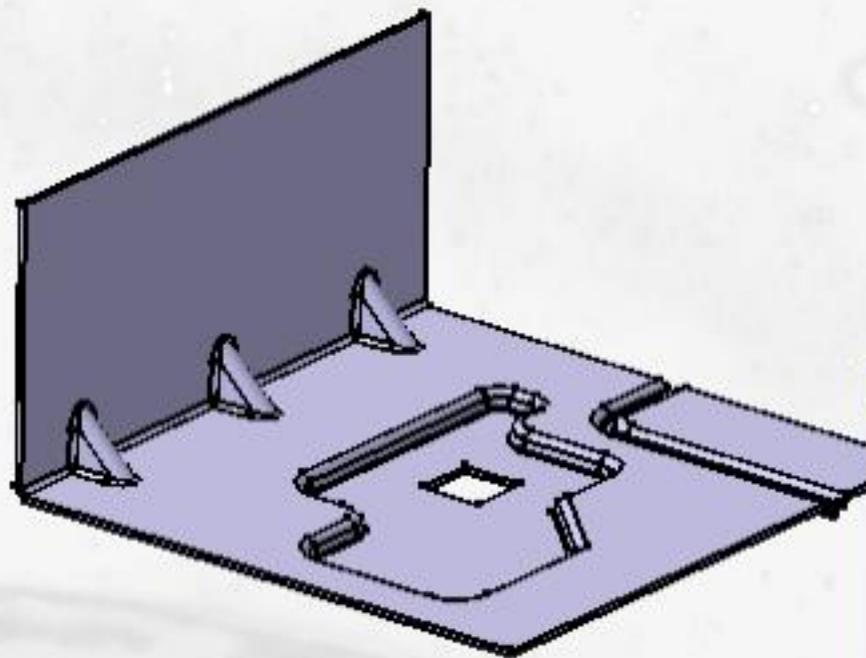
Defining a Stamp Feature

Designing a Sheet Metal part, you will find a need to use 3D Features (Stamps) such as an extruded hole or a bridge. The CATIA Sheet Metal workbench provides some of the common Stamps Features or allows user-defined ones.

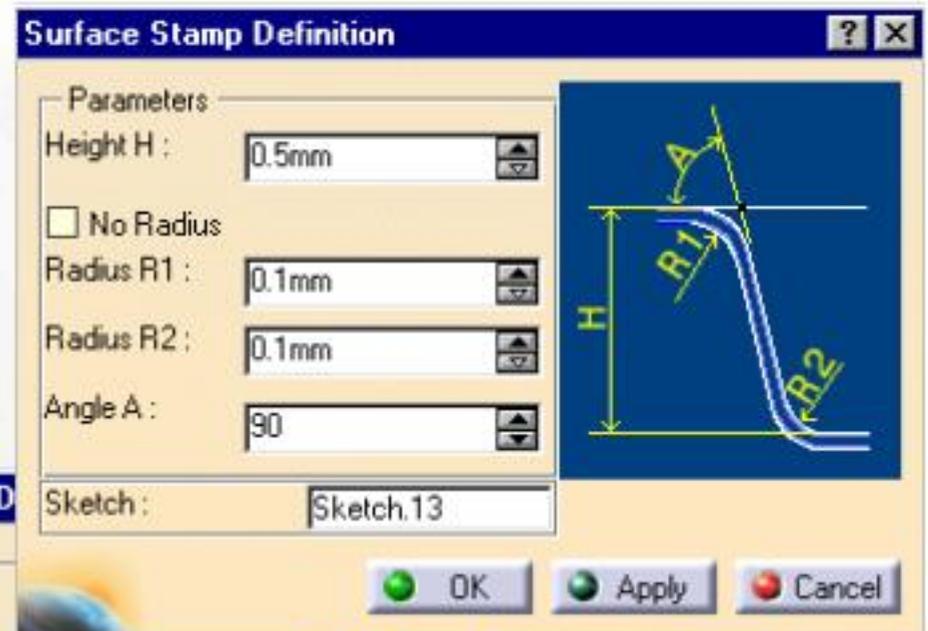


What is a Stamp Feature ?

A Stamp Feature is a common piece of geometry used in creating sheet metal parts through a stamping and/or die process. It comes with parameters that can be modified to change its shape or size and sometimes requires a sketch to complete its creation

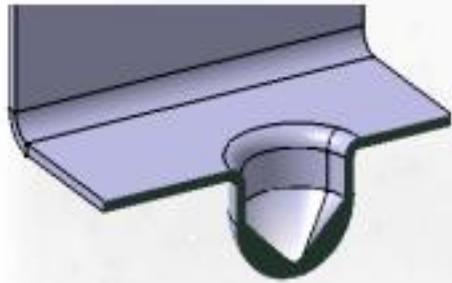


Stamp features can have cut outs in them.

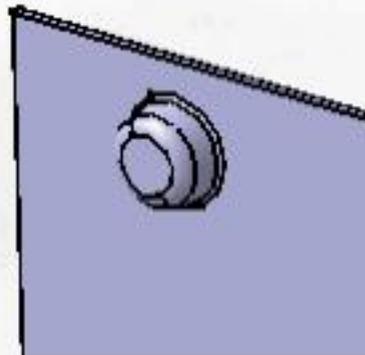


What types of Stamp Features are available ?

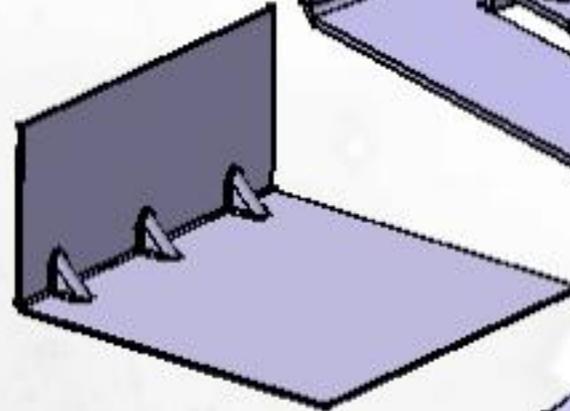
Shown below are the various types of Features available.



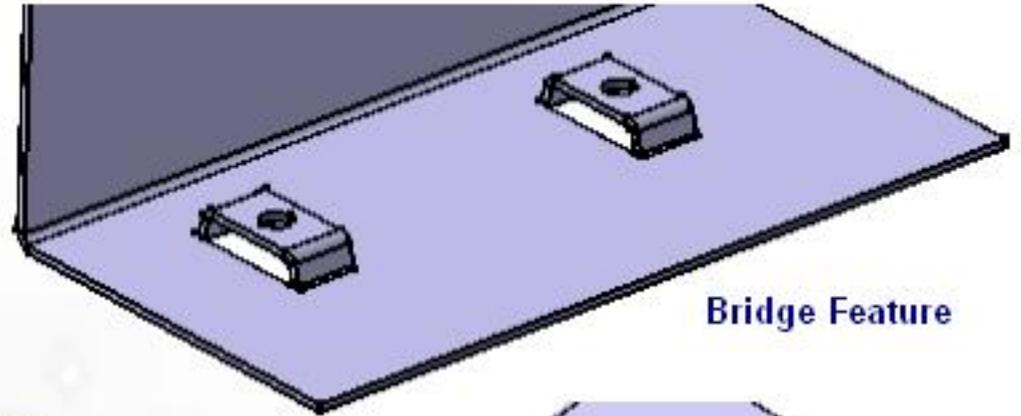
User defined Feature



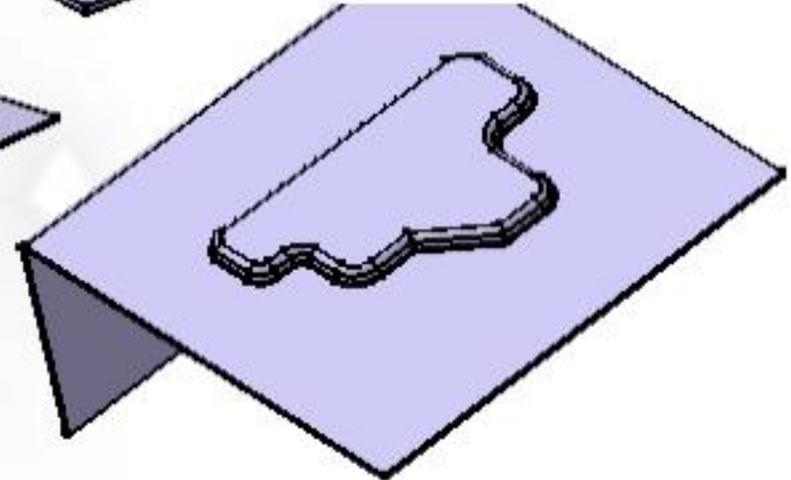
Point Stamp Feature



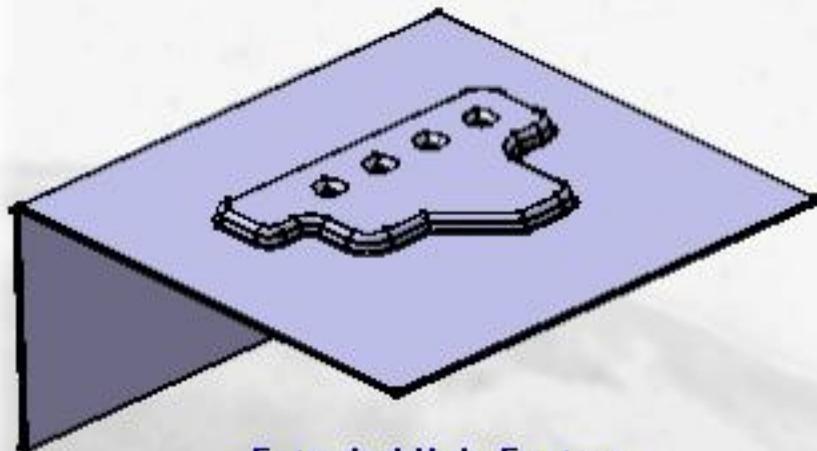
Stiffening Rib Feature



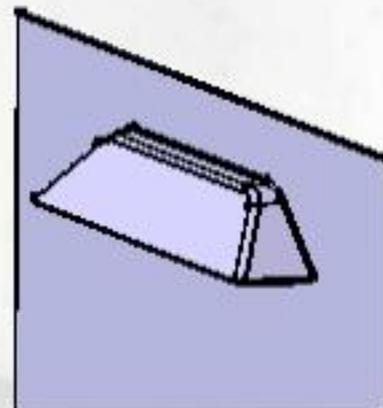
Bridge Feature



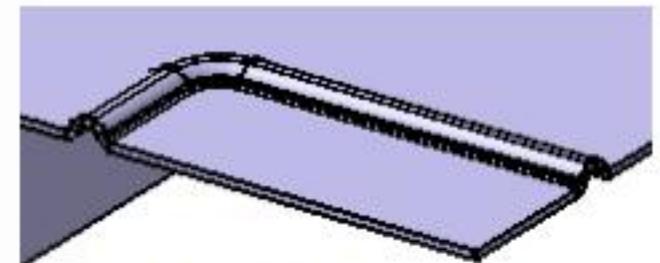
Surface Stamp Feature



Extruded Hole Feature

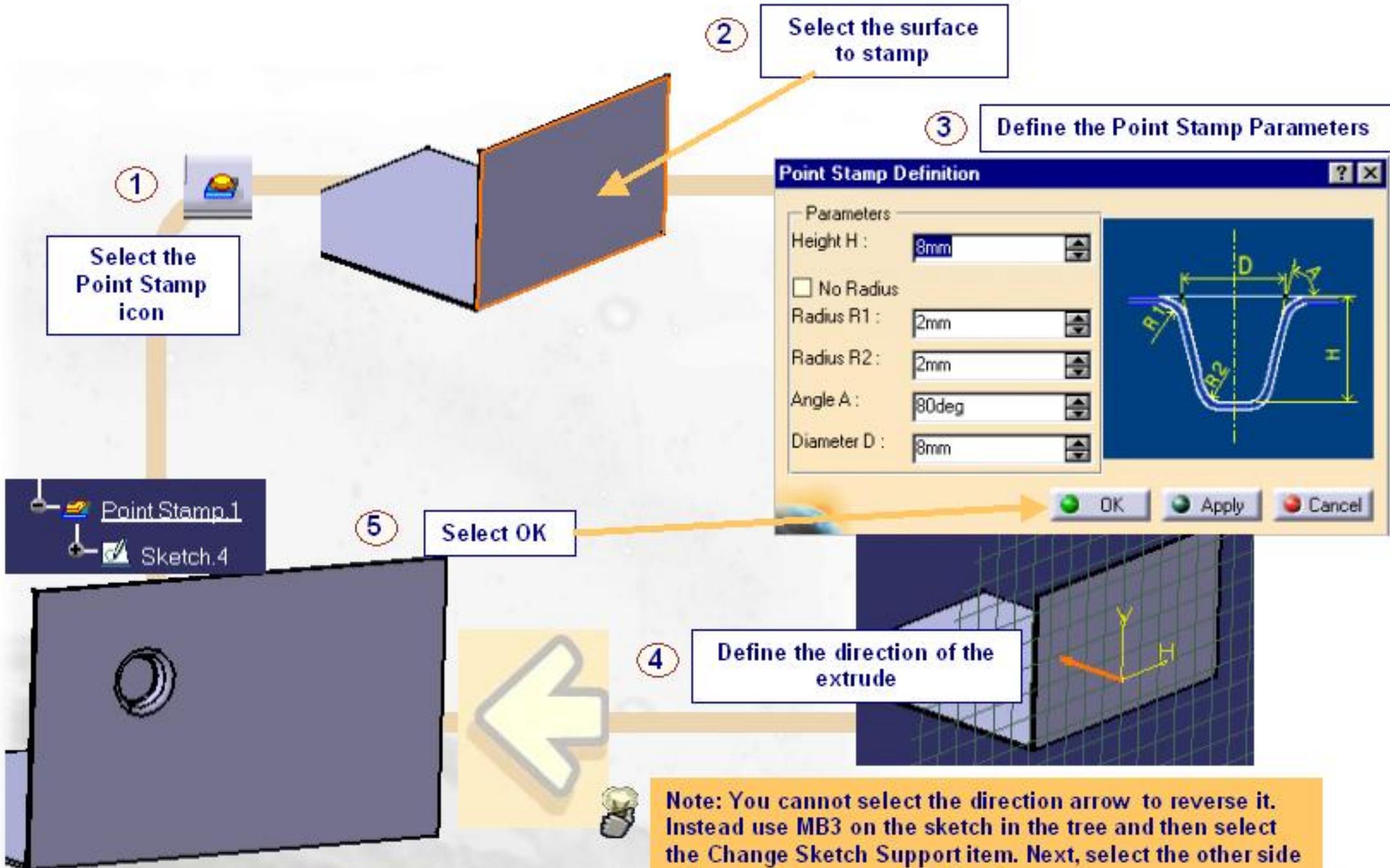


Louver Feature



Curve Stamp Feature

Point Stamp Feature



Note: You cannot select the direction arrow to reverse it. Instead use MB3 on the sketch in the tree and then select the Change Sketch Support item. Next, select the other side of the wall to reverse the direction.

Extruded Hole Stamp Feature

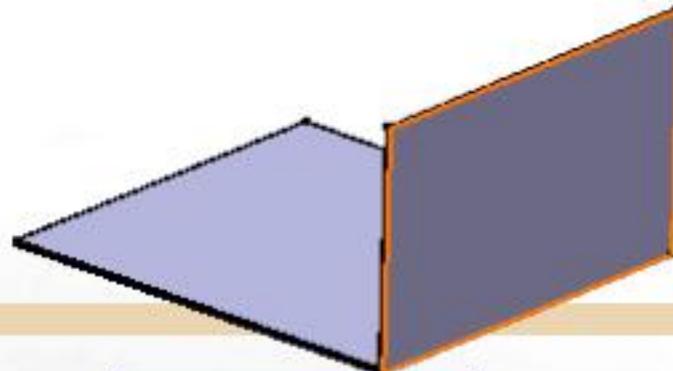
1



Select the Extruded Hole stamp icon

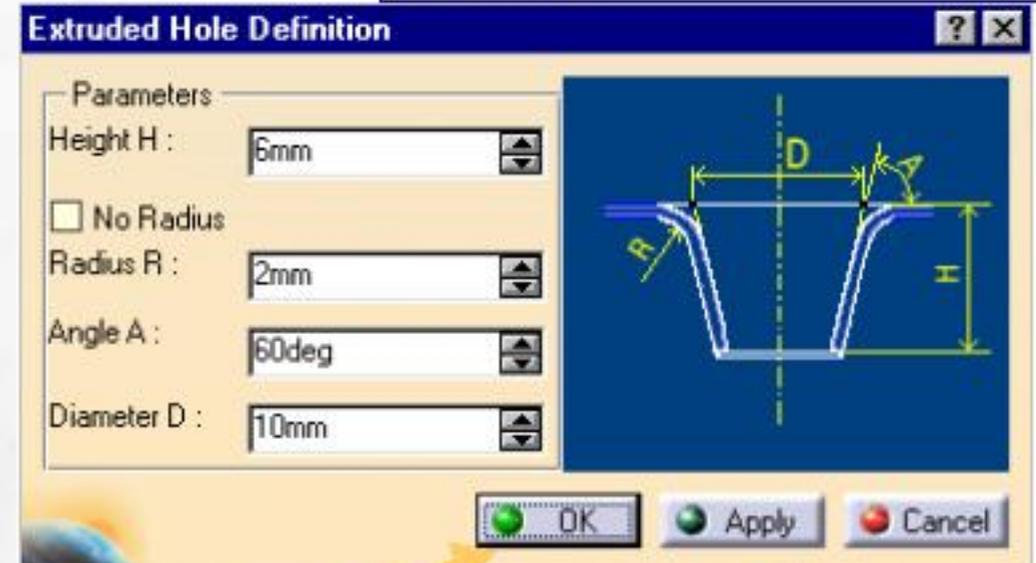
2

Select the surface to stamp



3

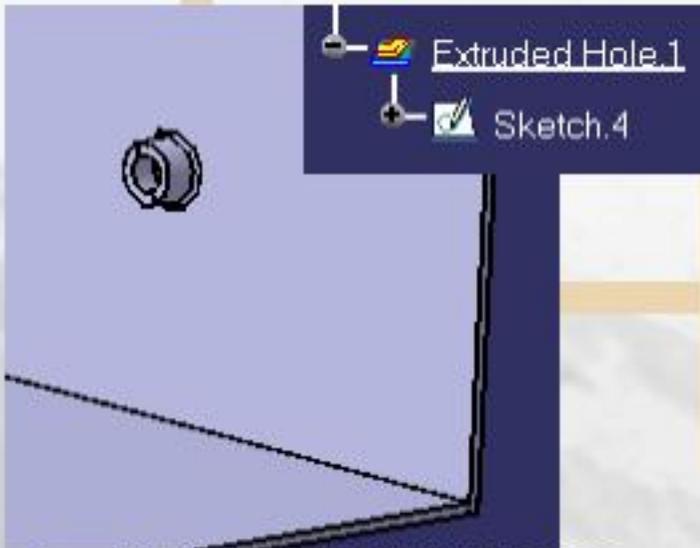
Define the Extruded Hole Stamp Parameters



4

Select OK

Each feature has a Sketch attached to it where the locational data is contained. Which can be used for modification.



As with the Hole icon in Part Design, Pre-Selecting two edges allows location constraints for the feature to be created.

Curve Stamp Feature

1 Create/Select a Sketch element

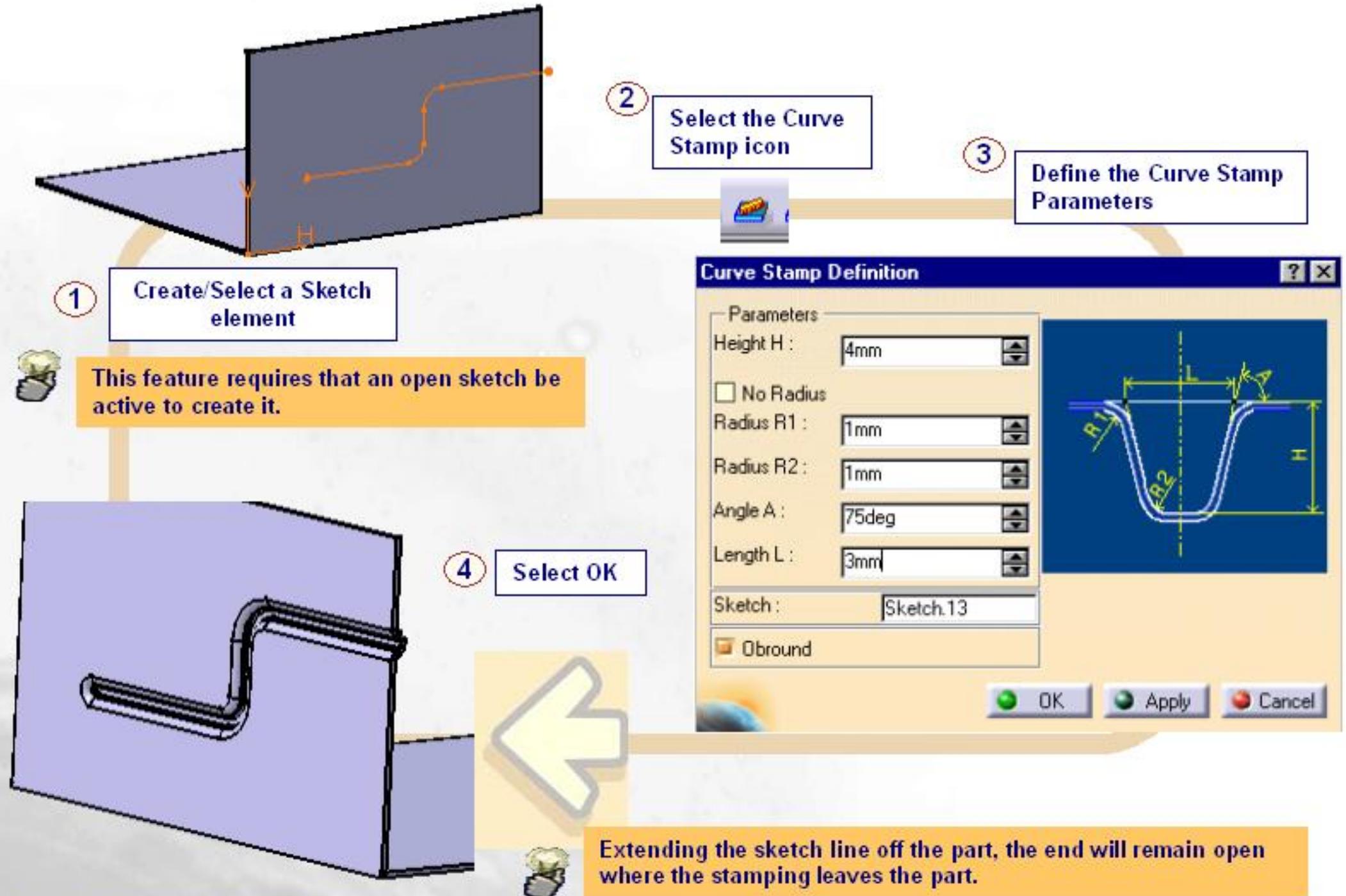
This feature requires that an open sketch be active to create it.

2 Select the Curve Stamp icon

3 Define the Curve Stamp Parameters

4 Select OK

Extending the sketch line off the part, the end will remain open where the stamping leaves the part.



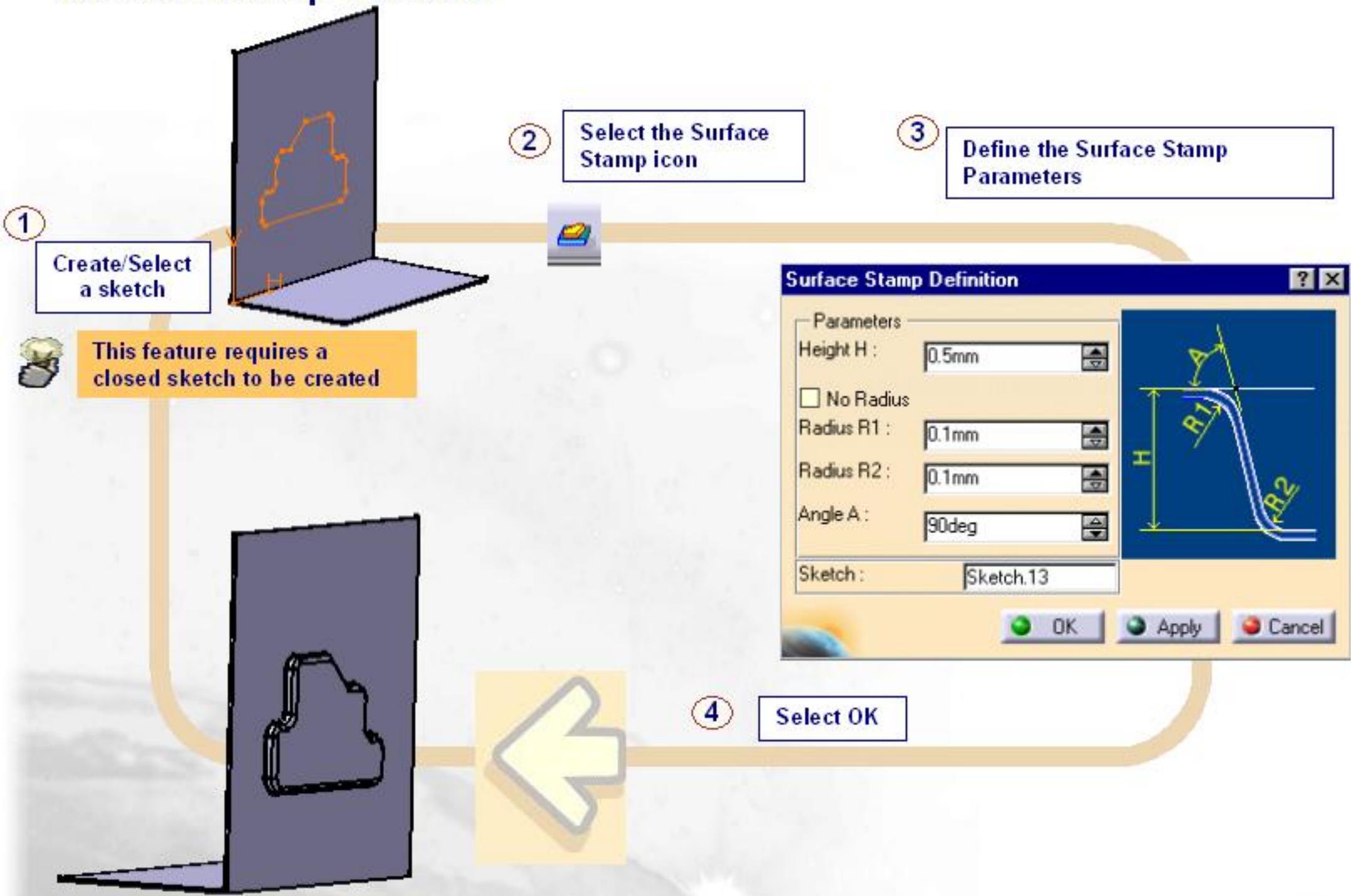
Curve Stamp Definition

Parameters:

- Height H : 4mm
- No Radius
- Radius R1 : 1mm
- Radius R2 : 1mm
- Angle A : 75deg
- Length L : 3mm
- Sketch : Sketch.13
- Obround

OK Apply Cancel

Surface Stamp Feature



1 Create/Select a sketch

2 Select the Surface Stamp icon

3 Define the Surface Stamp Parameters

This feature requires a closed sketch to be created

Surface Stamp Definition

Parameters

Height H : 0.5mm

No Radius

Radius R1 : 0.1mm

Radius R2 : 0.1mm

Angle A : 90deg

Sketch : Sketch.13

OK Apply Cancel

4 Select OK

Bridge Stamp Feature

1 Select the Bridge icon

2 Select the surface to stamp

3 Define the Bridge Parameters

4 Select OK

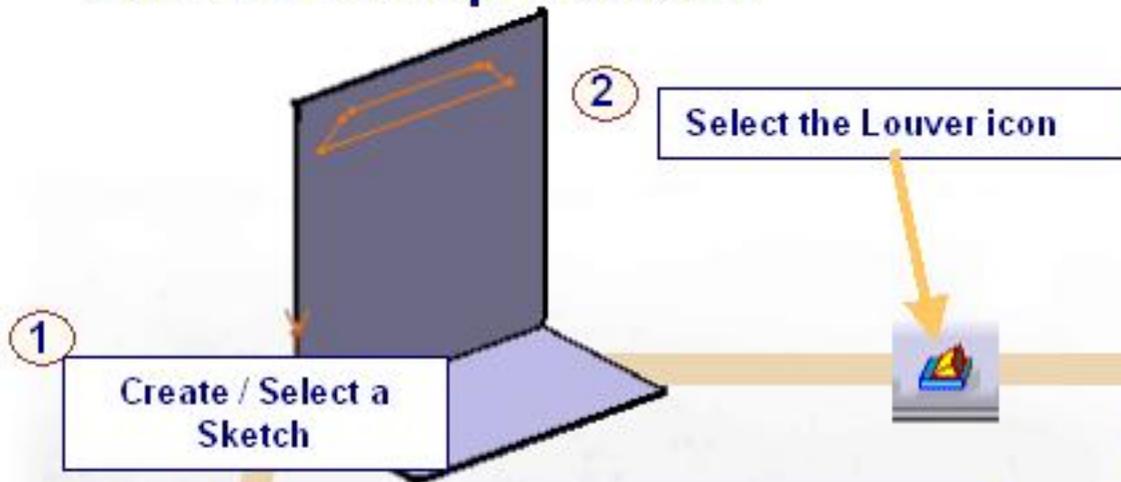
The Bridge Definition dialog box contains the following parameters:

- Parameters:
- Height H: 4mm
- No Radius
- Radius R1: 1mm
- Radius R2: 1mm
- Angle A: 80deg
- Length L1: 4mm
- Length L2: 10mm
- Angular reference: Edge
- Orientation Angle: 0deg

The diagram shows a bridge profile with dimensions: R1, R2, L1, L2, and H.

Bridge.1
Sketch.4

Louver Stamp Feature



The sketch must be the profile of the cutout, but cannot have tangency on the bottom/open edge.

Louver Definition

Parameters

Height H : 12mm

No Radius

Radius R1 : 2mm

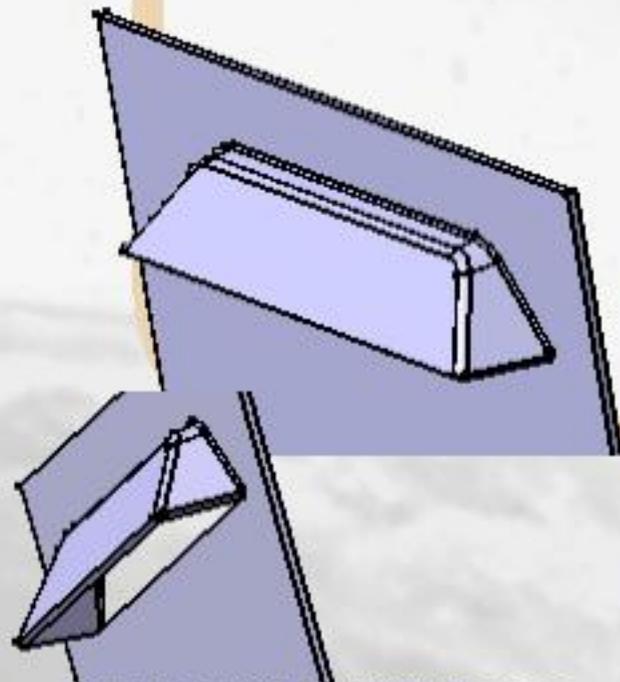
Radius R2 : 2mm

Angle A1 : 80deg

Angle A2 : 30deg

Sketch : Sketch.13

OK Apply Cancel



Stiffening Rib Stamp Feature

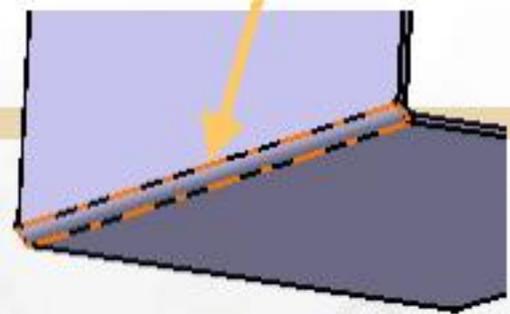
1



Select the Stiffening Rib icon

2

Select the exterior surface of the bend.



3

Define the Stiffening Rib Parameters

Stiffening Rib Definition [?] [X]

Parameters

No Radius

Radius R1 : 2mm

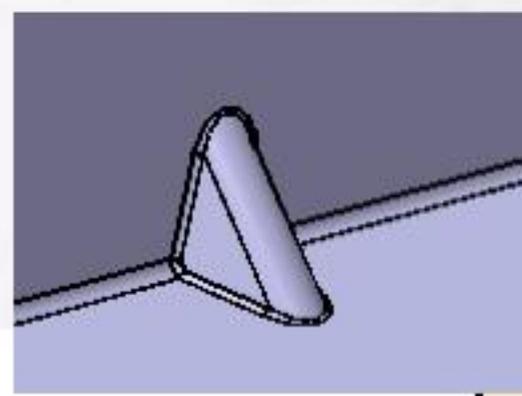
Radius R2 : 4mm

Angle A : 80deg

Length L : 10mm

Bend : Bend.1

OK Apply Cancel



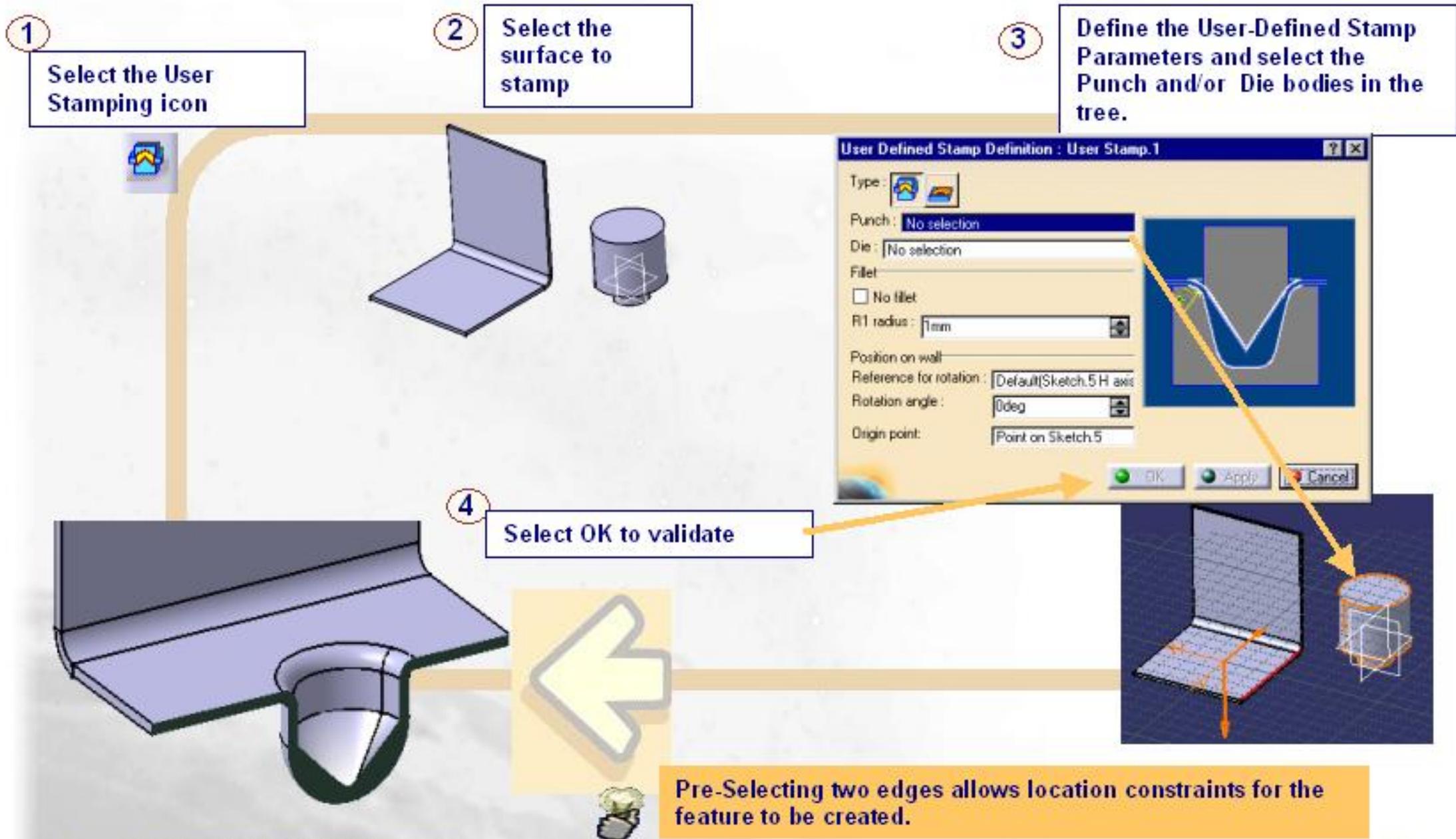
4

Select OK



Pre-Selecting two edges allows location constraints for the feature to be created.

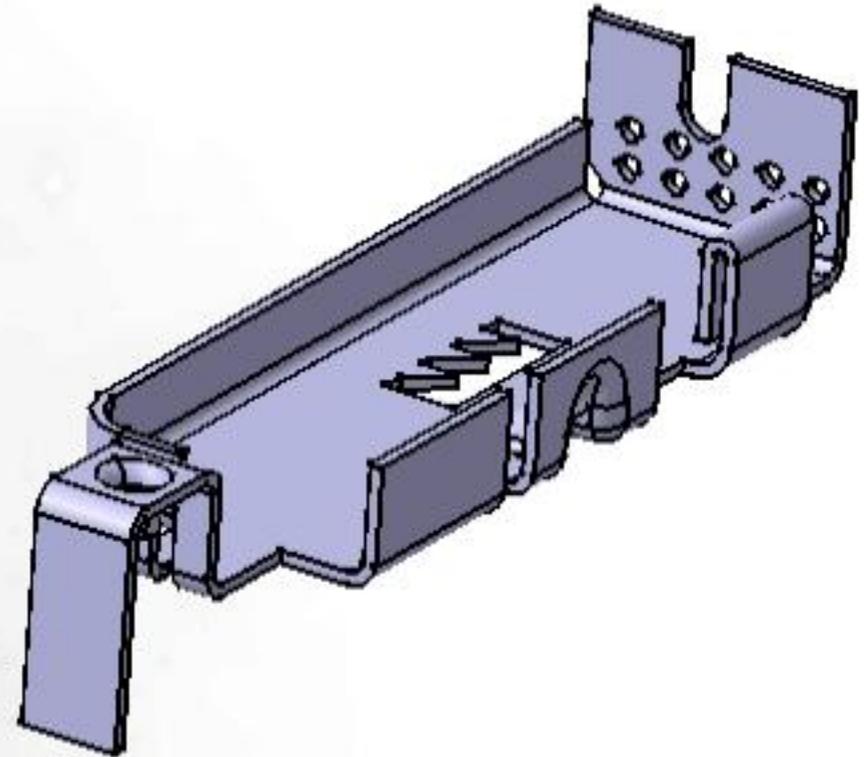
User-Defined Stamping Feature



Sheet Metal 2D View Creation

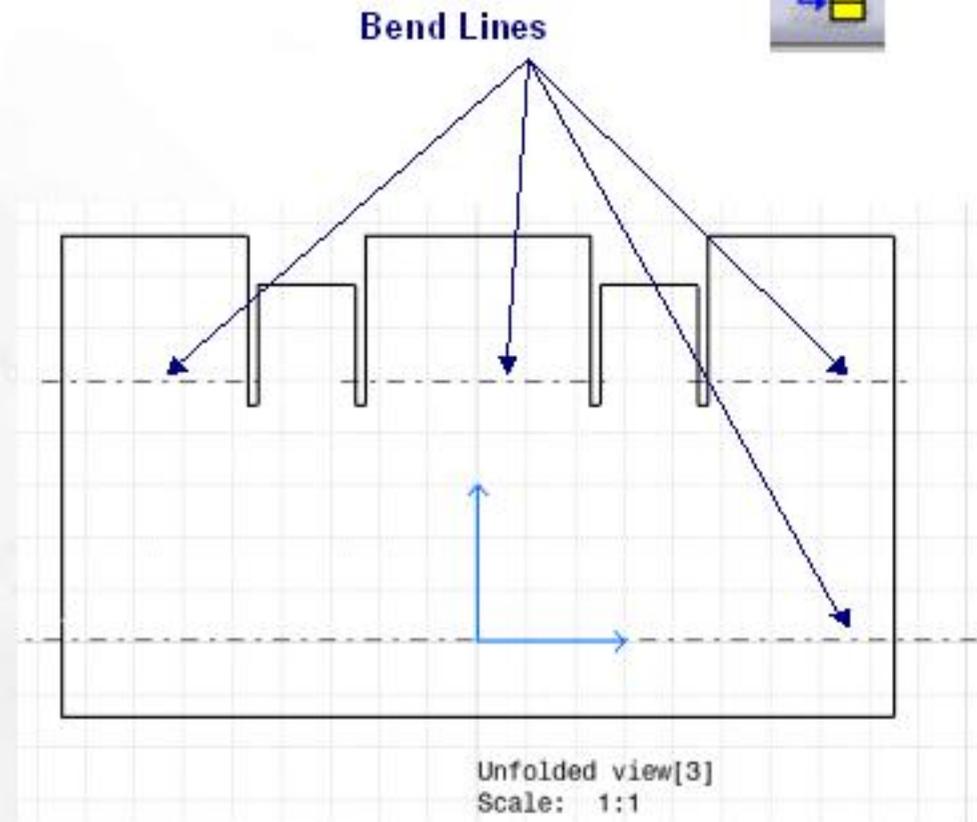
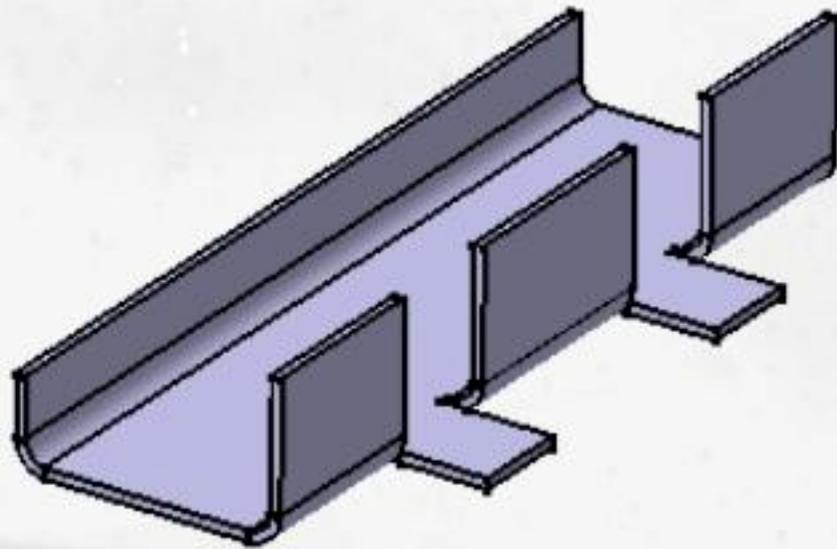
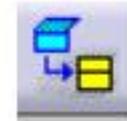
In this lesson, you will see how to create a 2D (Flatten) View of a sheet metal part

- Creating a 2D (Flatten) View



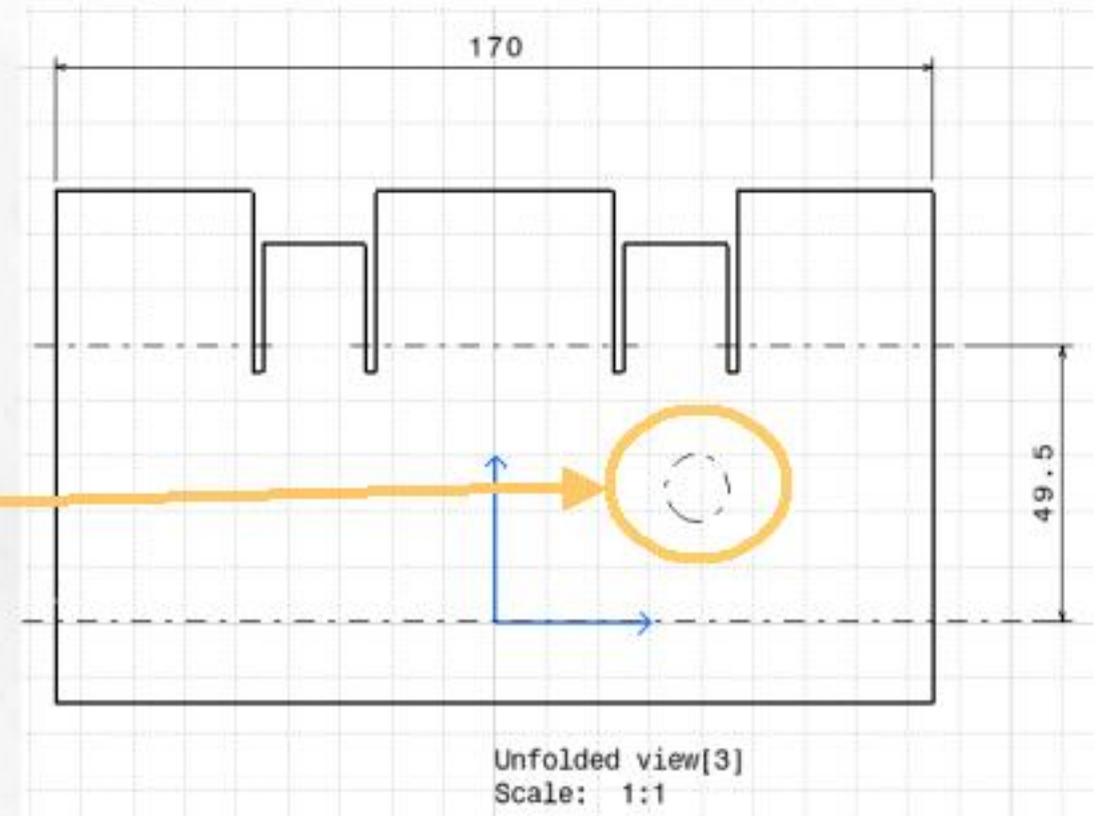
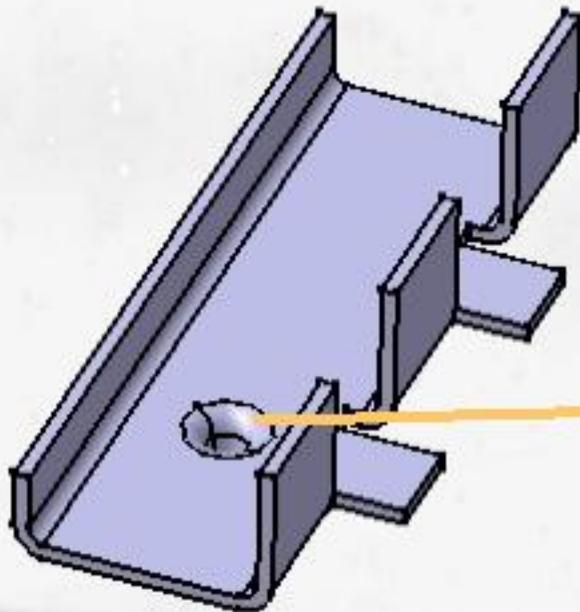
Creating a 2D (Flatten) View

When creating a Sheet Metal part, you may need to get a flatten view of the part unfolded on a 2D drawing.



What are 2D (Flatten) Views ?

A 2D (Flatten) View is a 2D drawing view of a Sheet Metal part which has been flatten for production. Shown on the view are the bend lines where the folds will occur on the part.

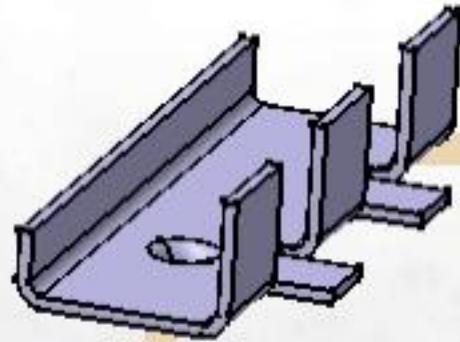


Dimensions can be added to the 2D drawing as needed.

Creating a 2D (Flatten) View

1

Go to Drafting workbench and create a sheet

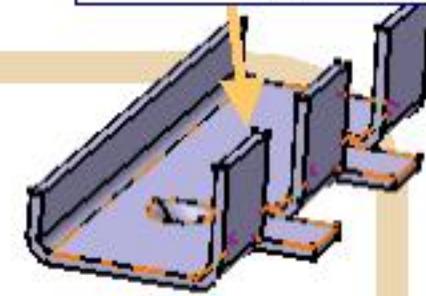


2

Select the Unfolded View icon

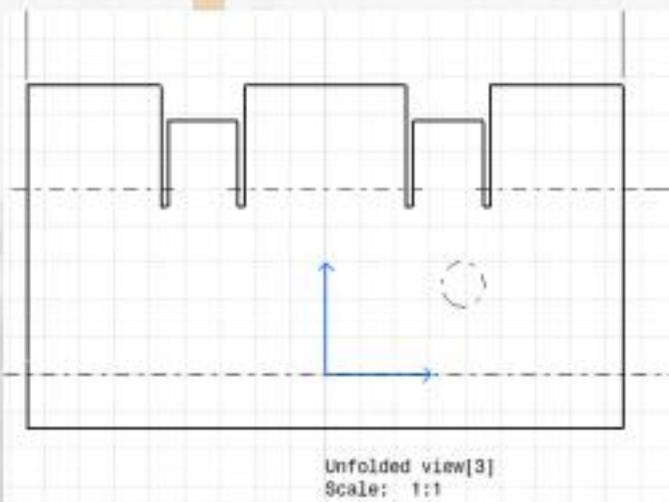
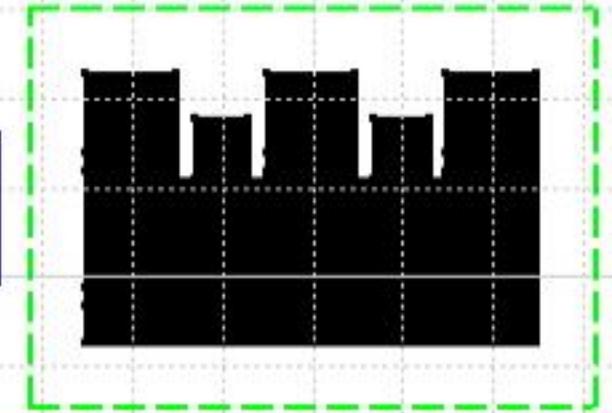
3

Next, select a flatten reference plane on the 3D geometry



4

Click anywhere on the drawing to create the view

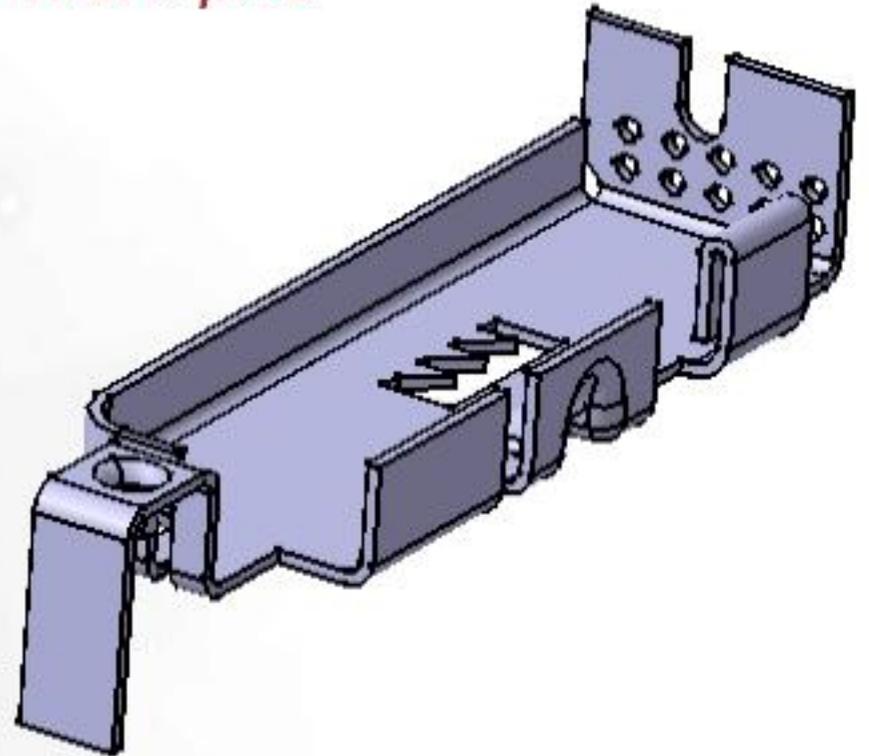


Note: if a stamp feature is created over the limit between several supports, such as walls, bends, and so forth, the stamp is not visible in the view.

Sheet Metal Modifications to a Part

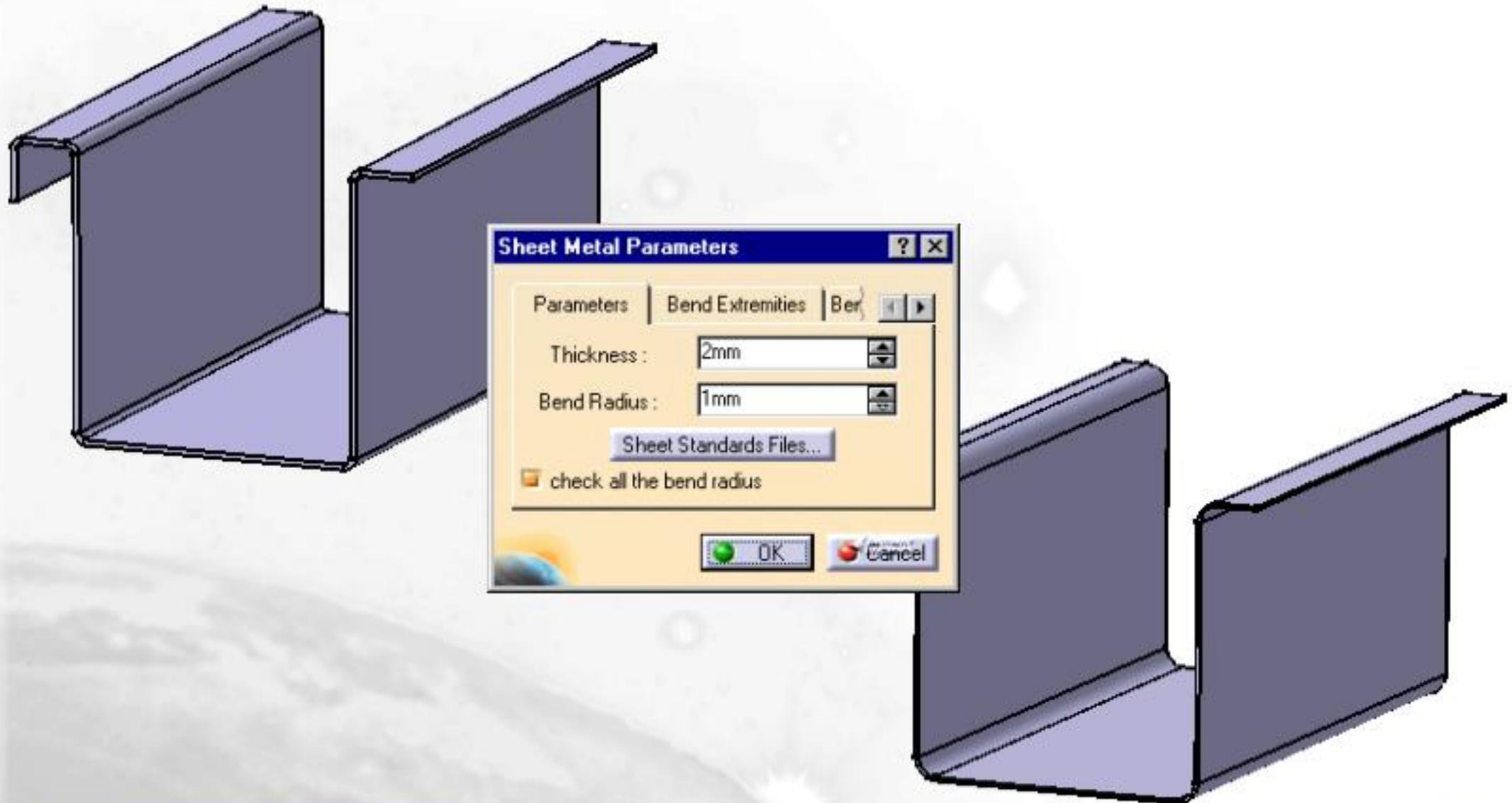
In this lesson, you will see how we can do various modifications to different parameters, walls, bends, cut outs and features of a part.

- **Modifying Part Parameters**
- **Modifying Walls**
- **Modifying Bends**
- **Modifying Flanges, Cut Outs and Stamp Features**



Modifying Part Parameters

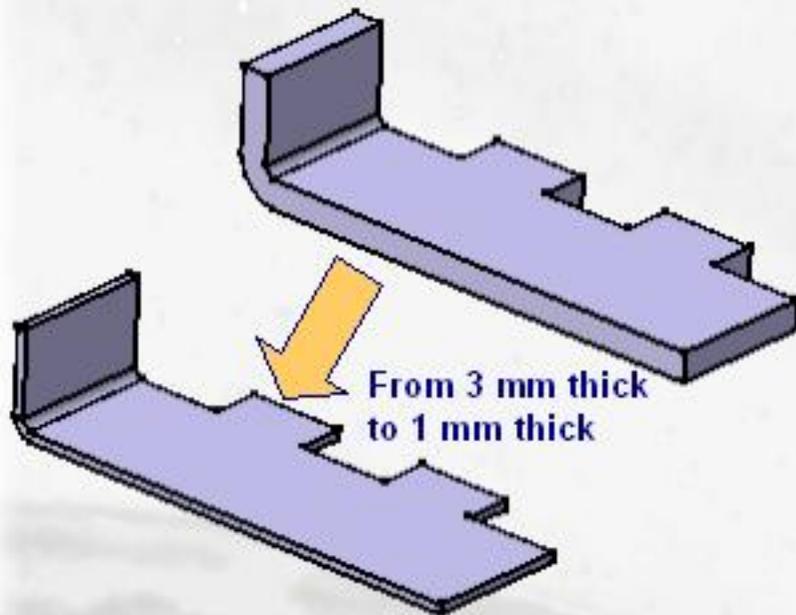
A Sheet Metal part's parameters can be modified for material thickness, bend radius, and relief changes. When modified, all features using the default values will be update.



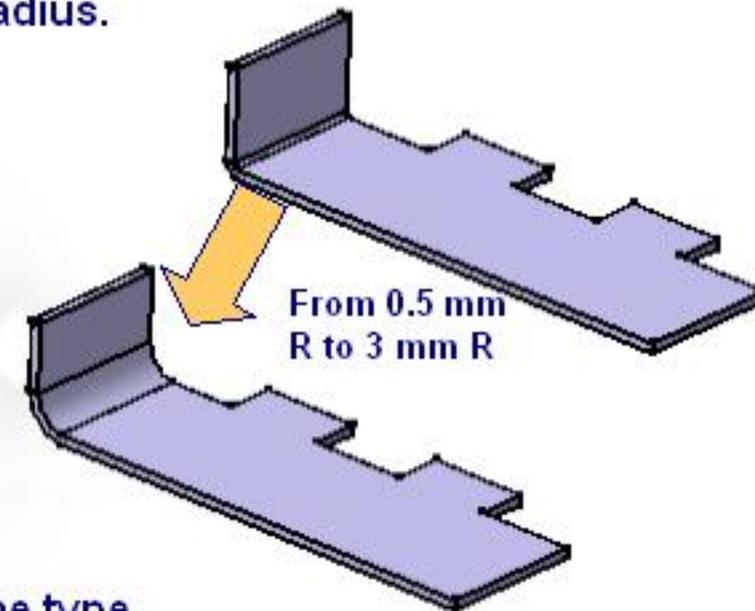
What are the Modifications to perform ?

There are three main parameters to modify: standard bends, wall thickness or bend relief's in the part.

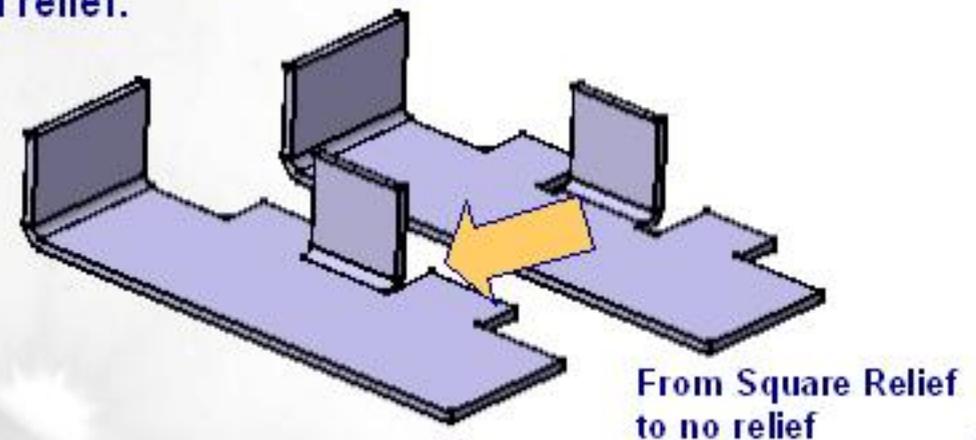
The first is the material thickness of the part. This allows changes to the material thickness for reason such as rigidity, weight, etc...



The second is the default bend radius.



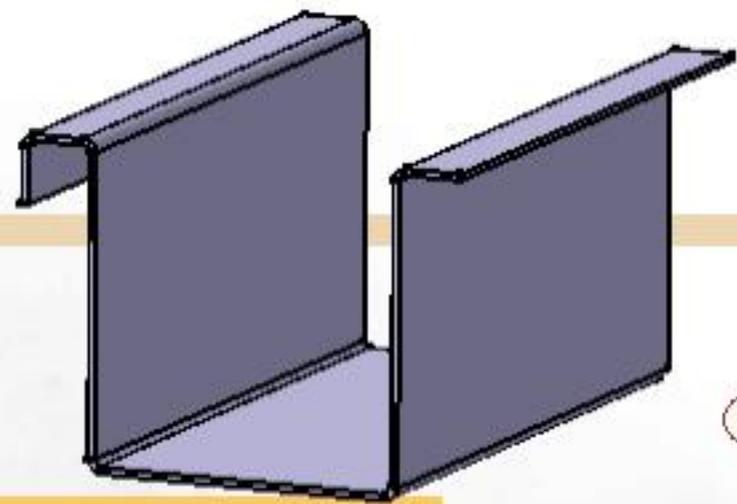
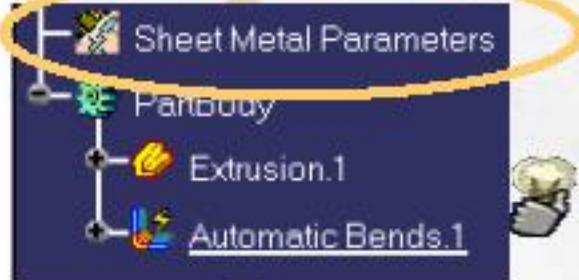
The third to modify is the type of bend relief.



Modifying the Wall Thickness and Bend radii

1

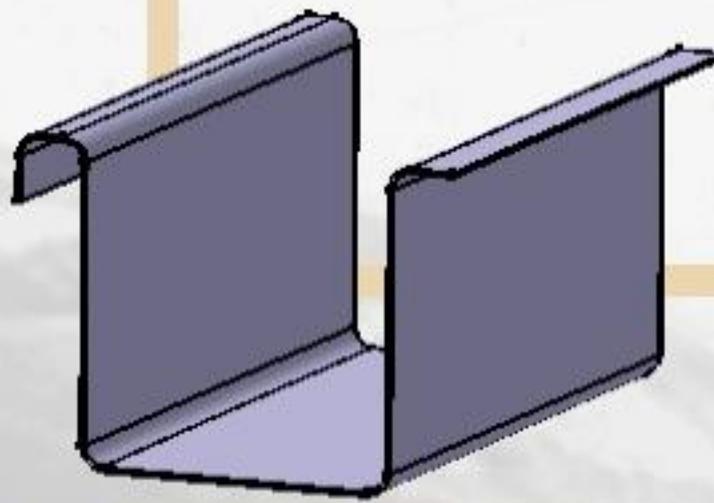
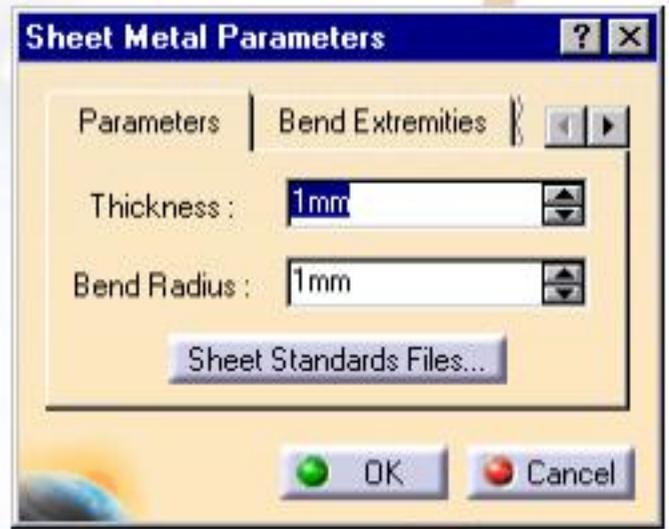
Select the Sheet Metal Parameters icon



2

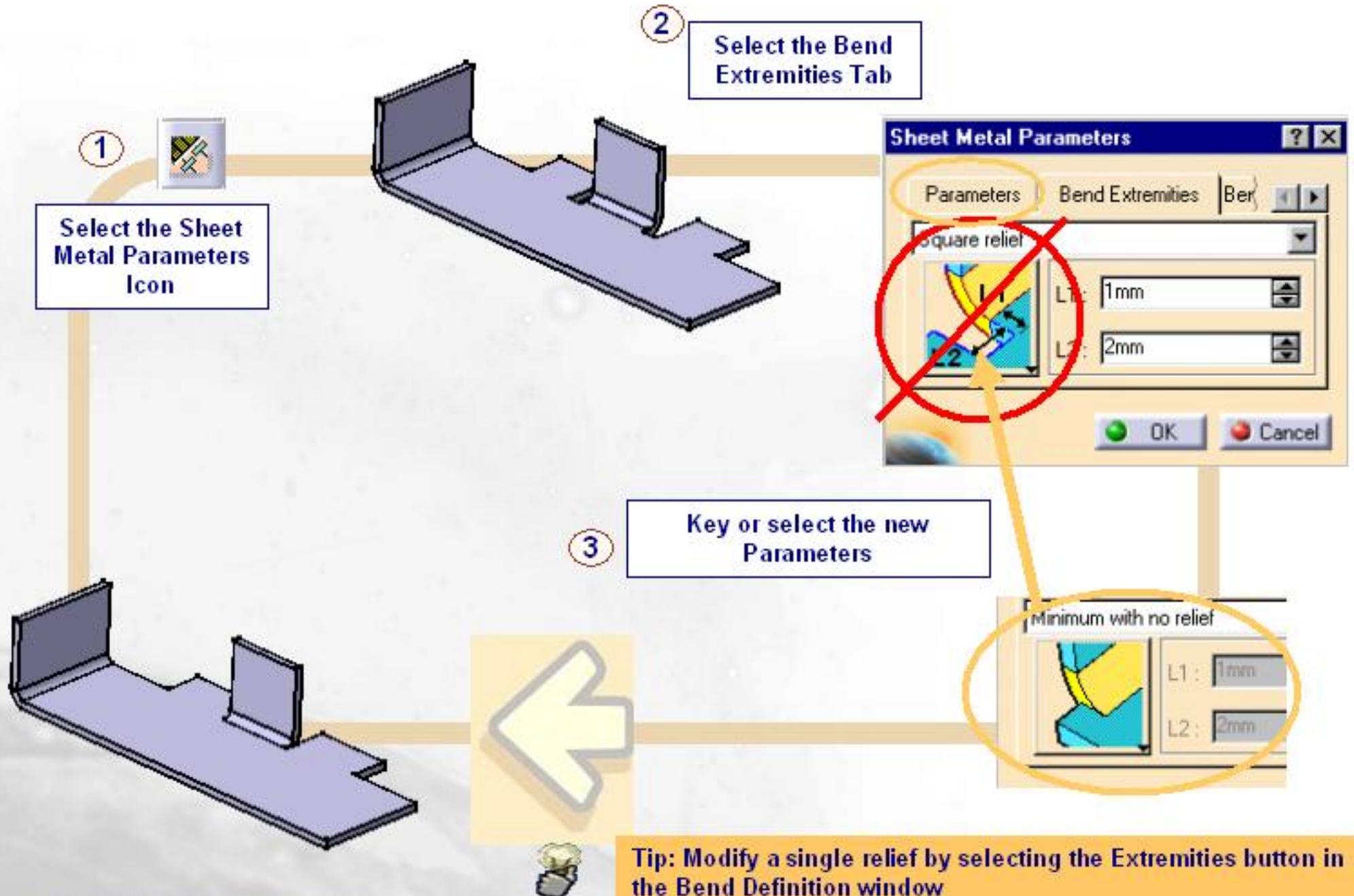
Key the new values in Sheet Metal Parameters Tab

Tip: You could select the Sheet Metal Parameters item in the tree also



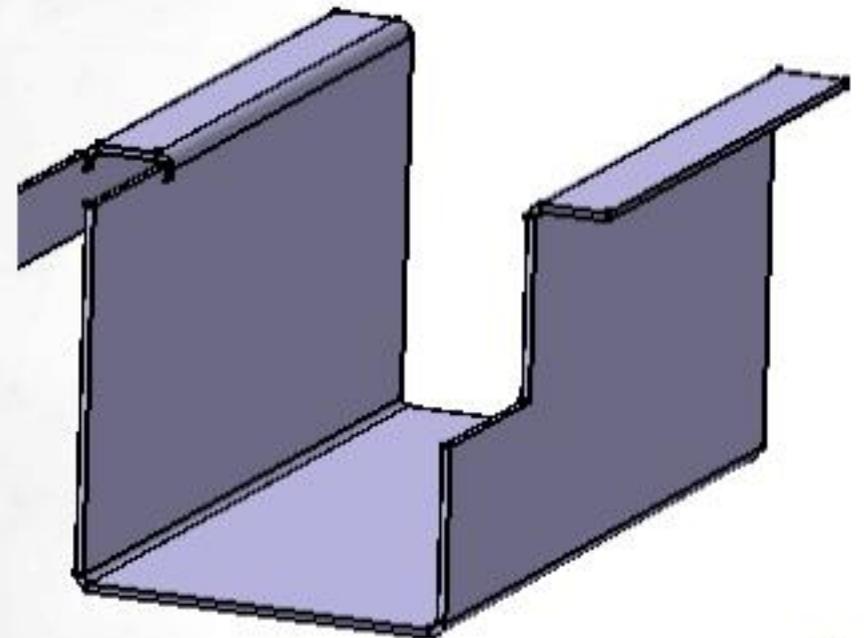
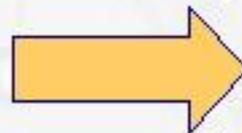
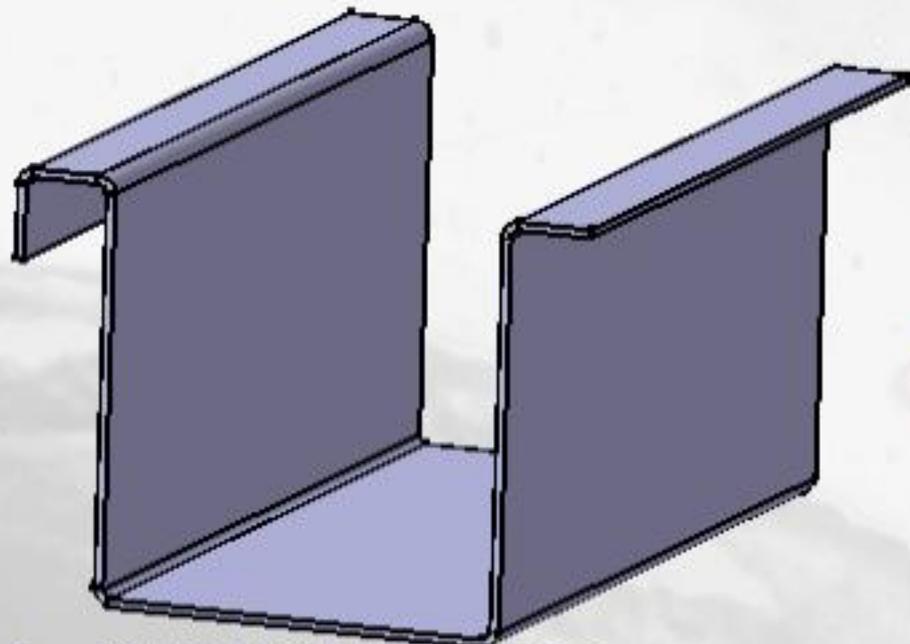
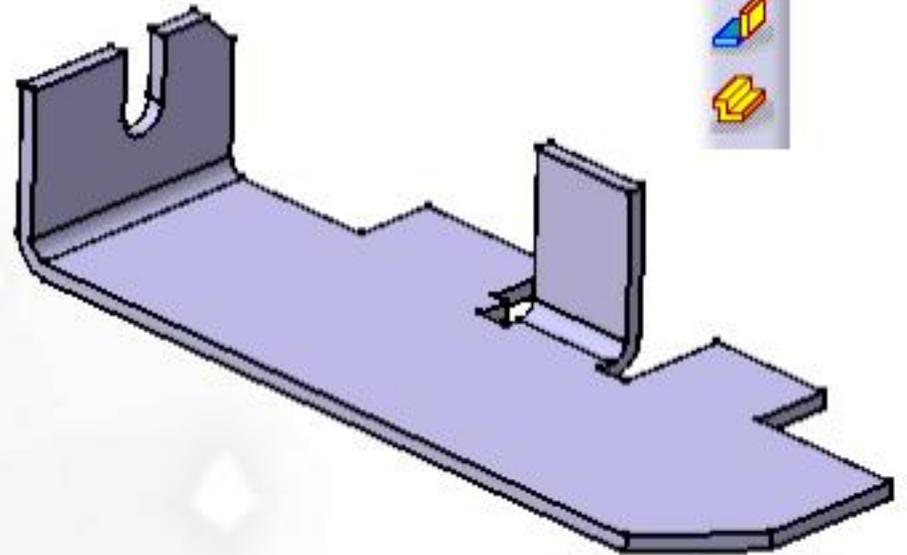
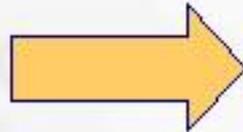
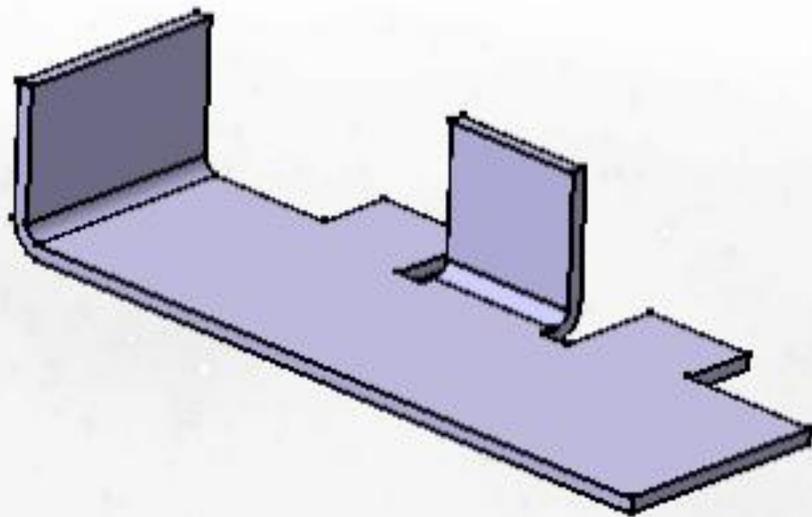
Tip: the wall thickness will modify only in the direction of the original wall thickness. To change this, you must edit the original wall and modify it there.

Modifying Bend Relief's



Modifying Walls

Modifying a Sheet Metal part's walls (shape, length, width etc...)



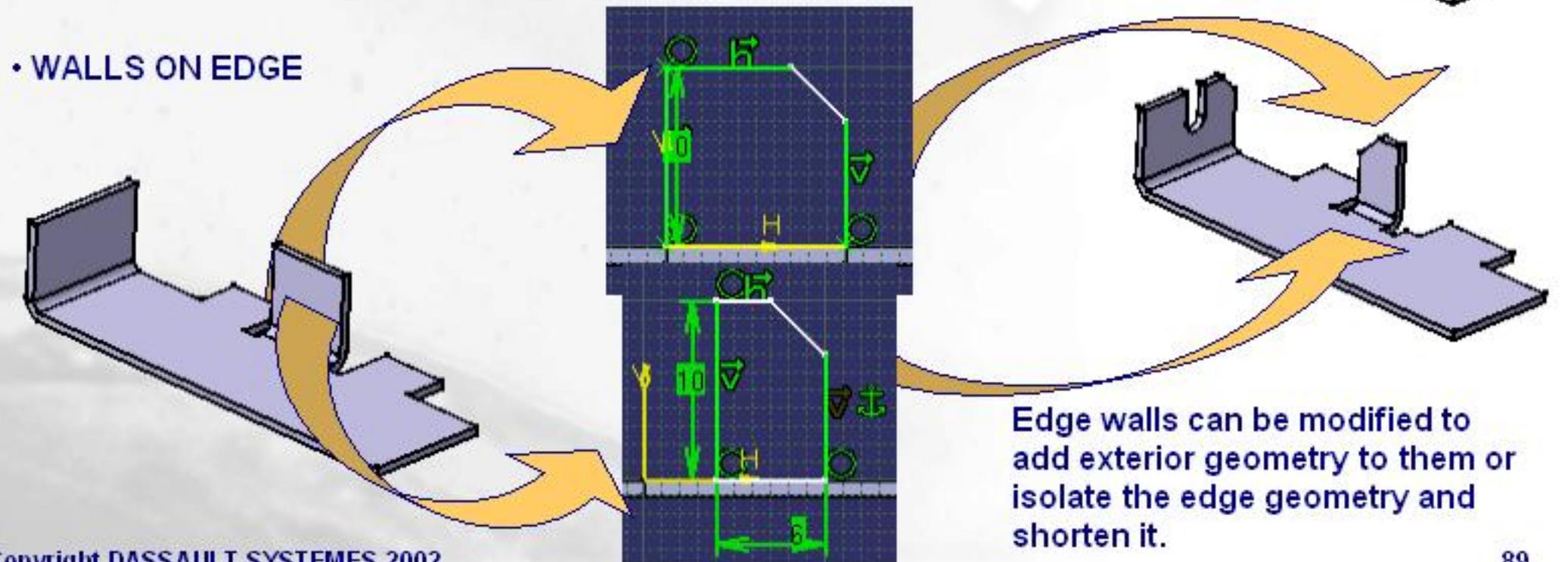
What types of modifications can be done to different walls ? (1/2)

• PROFILE WALLS

Profile walls can be modified by the original sketch; eliminating extra items in the tree. It also simplifies locating them for future changes.



• WALLS ON EDGE

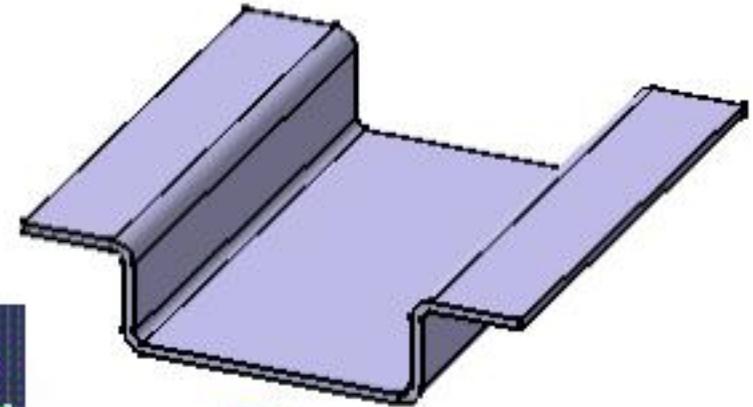
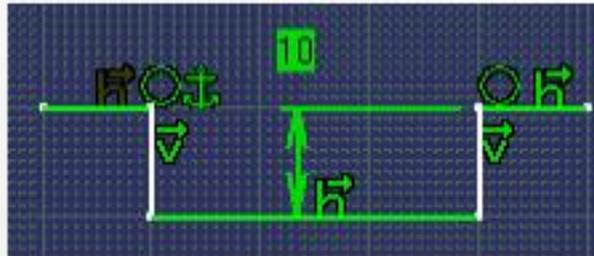
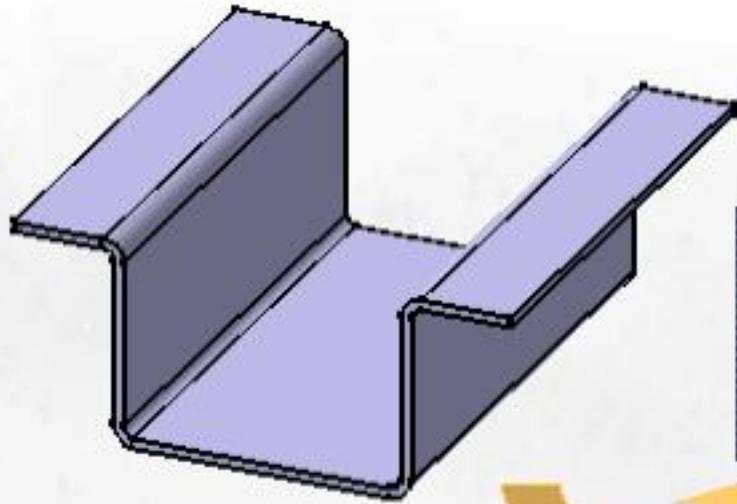


Edge walls can be modified to add exterior geometry to them or isolate the edge geometry and shorten it.

What types of modifications can be done to different walls ? (2/2)

• EXTRUDED WALLS

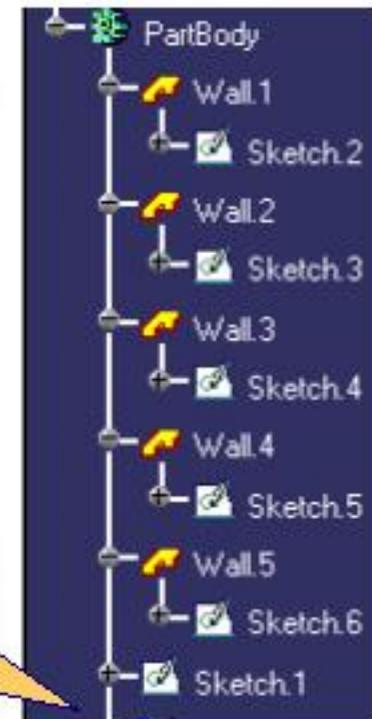
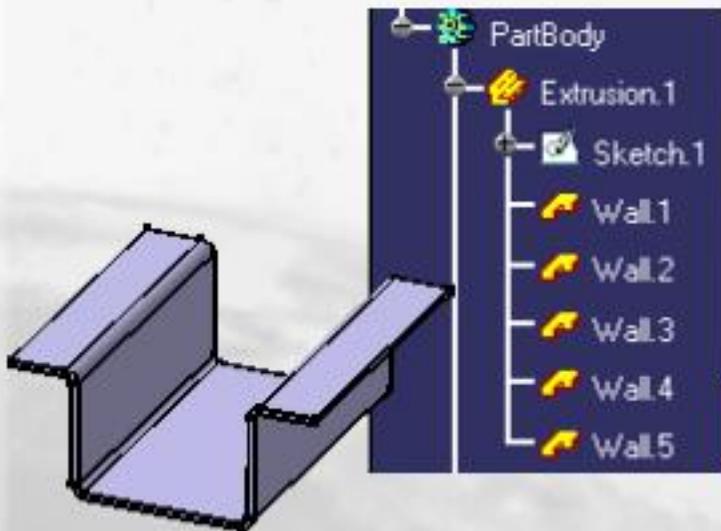
Modify the original sketch to change the walls.



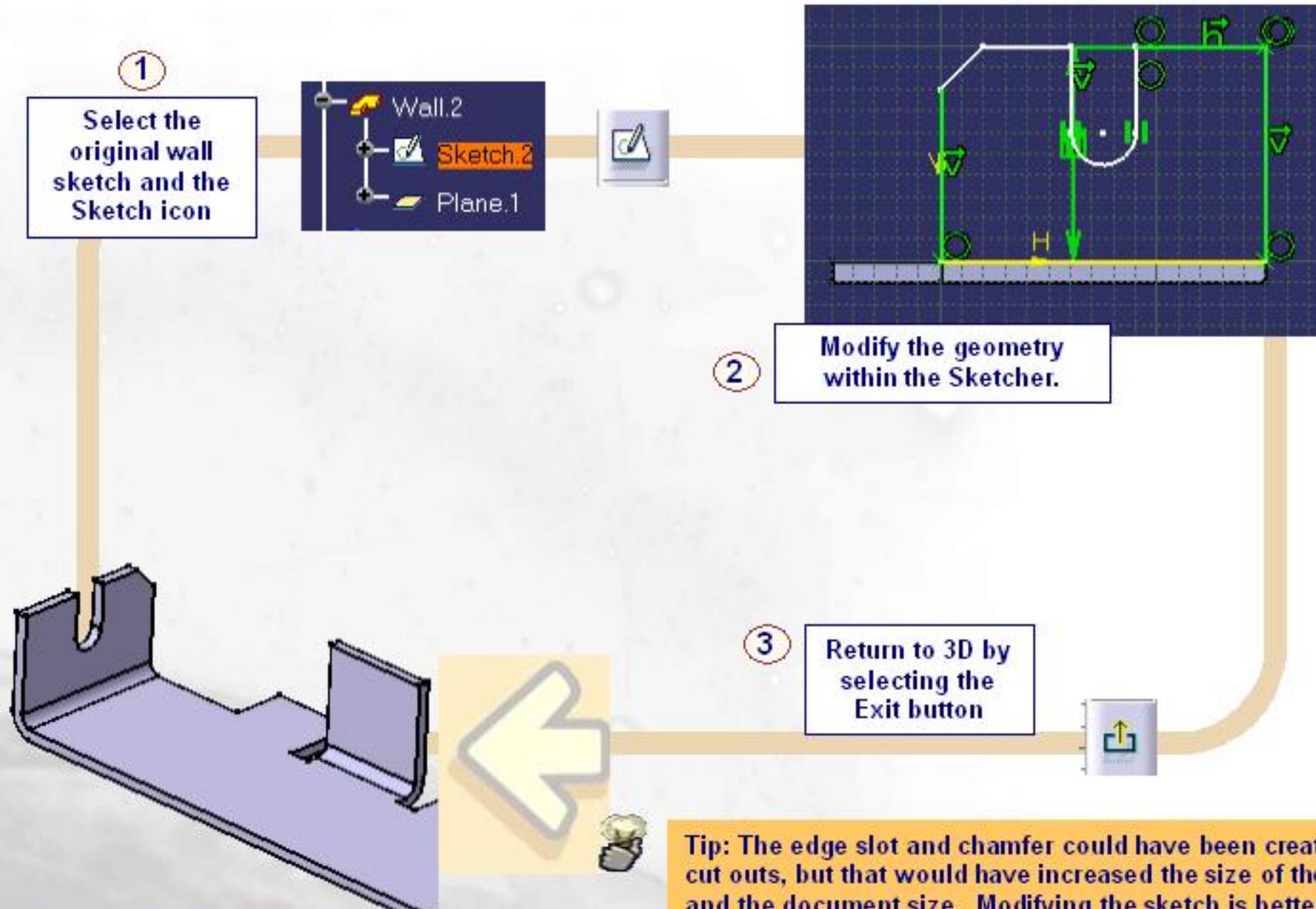
The original Extrusion used to create the extruded walls can be exploded to create separate walls so each can be modified individually

MB3 on Extrusion.1, Extrusion.1 object and isolate

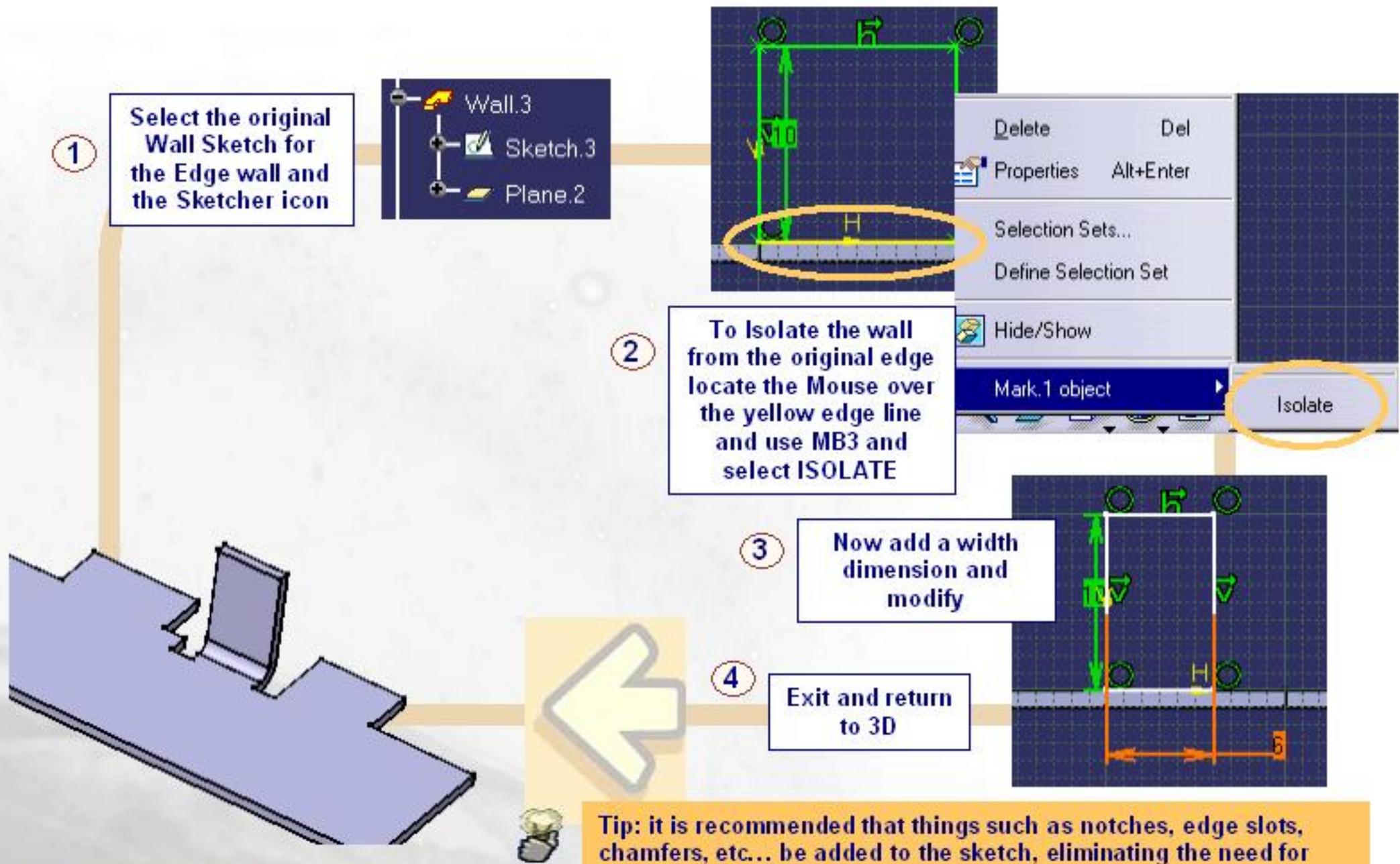
Isolate



Modifying a Profile Wall



Modifying a Wall on Edge

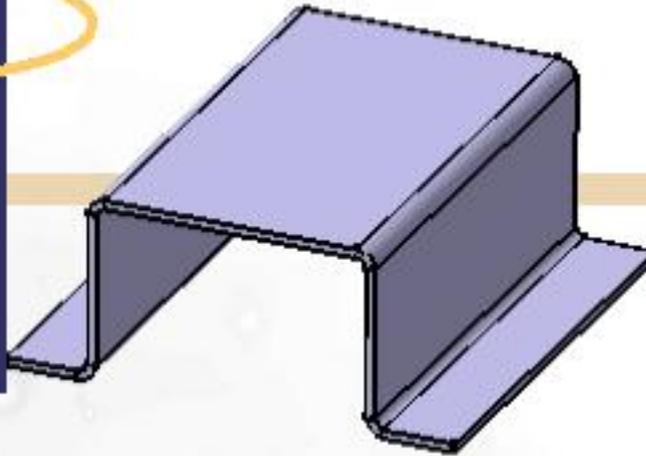


Tip: it is recommended that things such as notches, edge slots, chamfers, etc... be added to the sketch, eliminating the need for extra items in the tree.

Modifying a Group of Extruded Walls

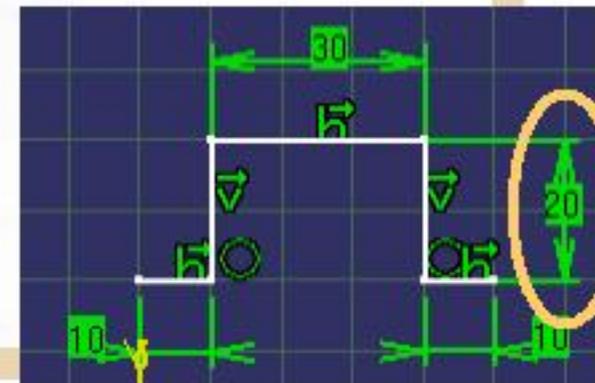
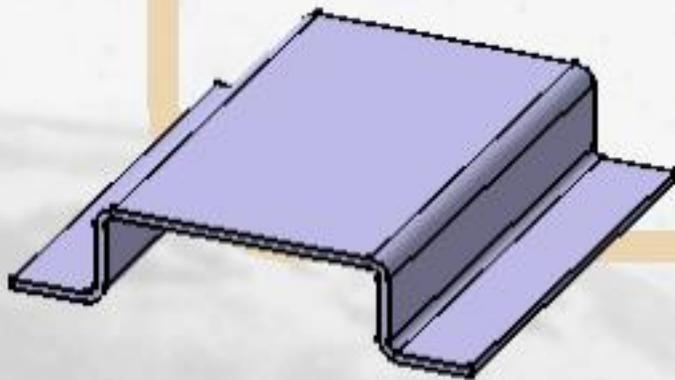
1

Select the Sketch of the Extruded walls



2

Modify the dimensions

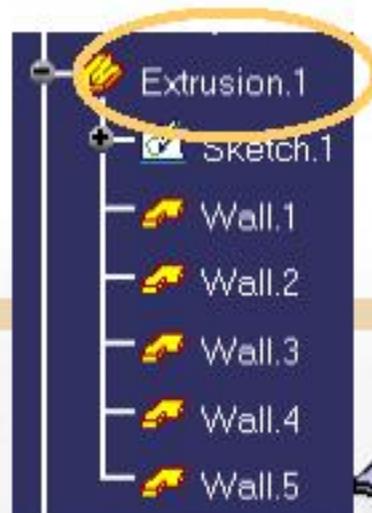


Tip: to add notches, edge slots, chamfers, etc... use cut outs to perform the modification.

Exploding Extruded Walls into Single Walls

1

Select the Extrusion of the walls with MB3 from the Tree



2

Move over the object item in the tree, select ISOLATE



Note: This procedure can not be undone to return to the Extrusion type wall.

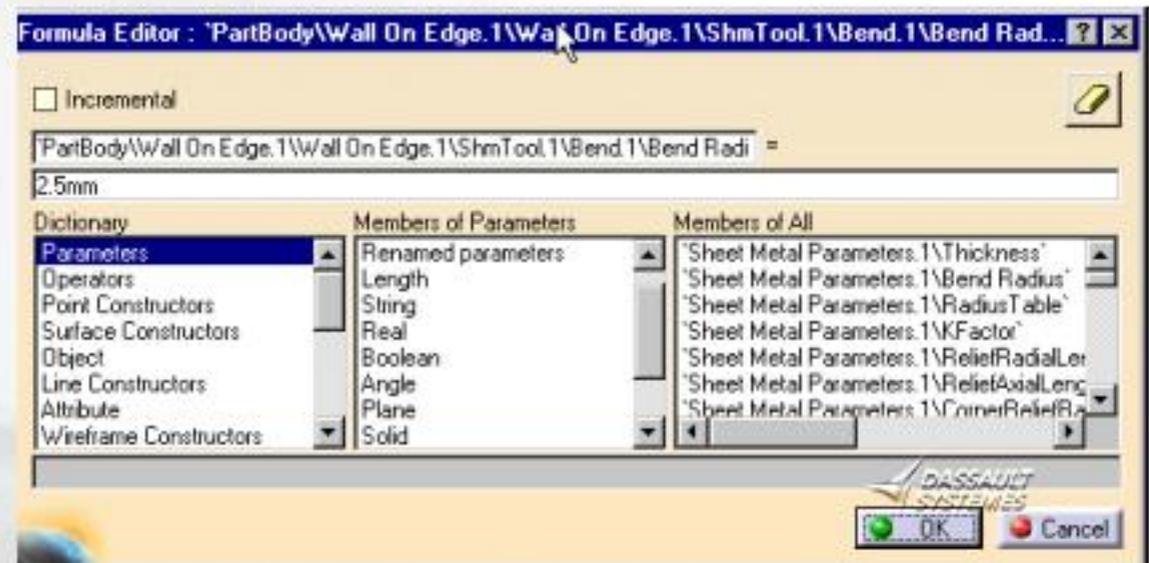
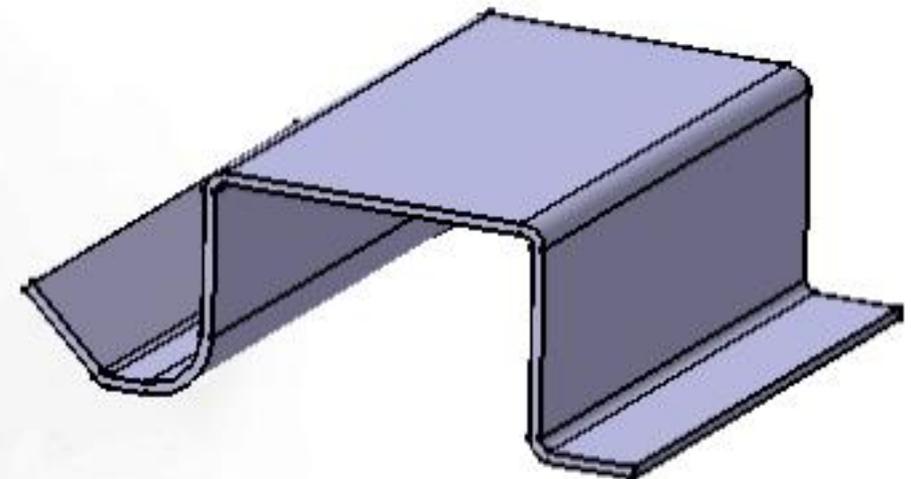
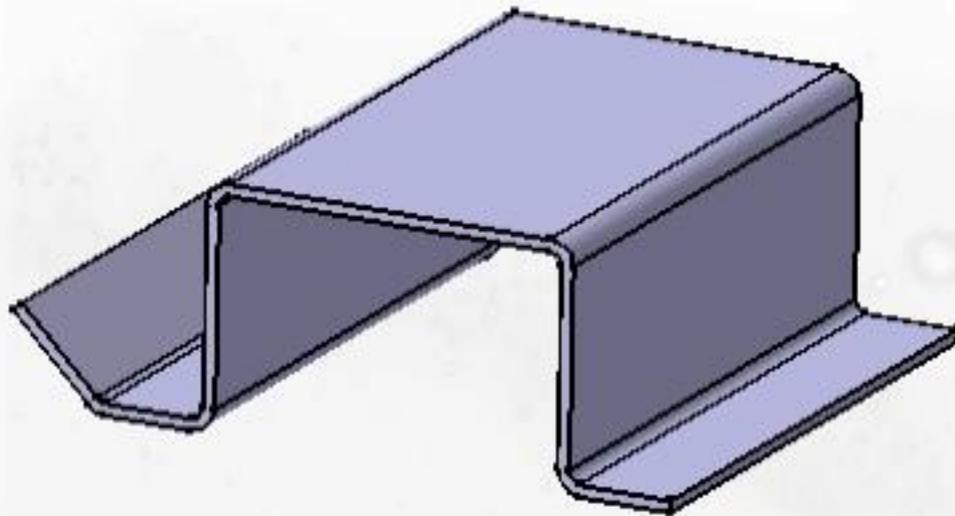
The Extrusion is gone and all walls contain a defining sketch



Tip: to modify the length or width of the wall and/or add notches, edge slots, chamfers, etc... modify the sketches of the walls

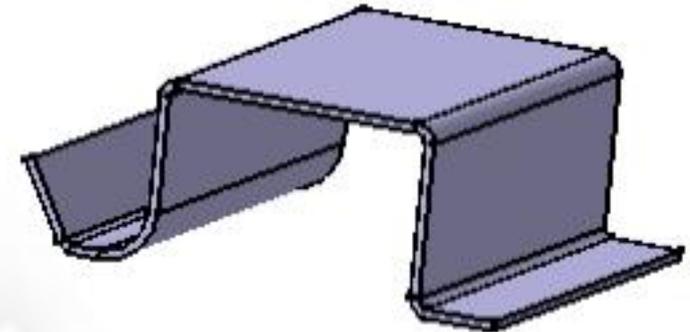
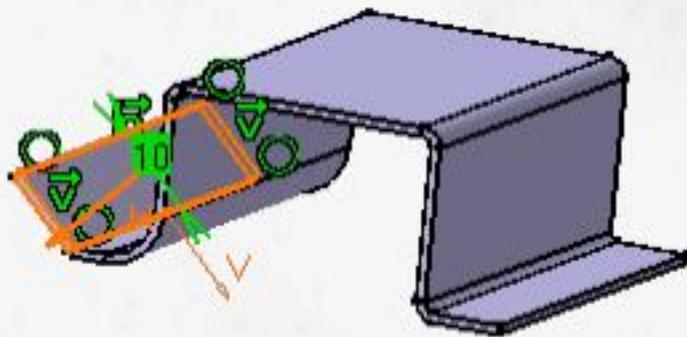
Modifying Bends

To modify one of the bend radii or angle parameters in a part



What types of Bend modifications can be done ?

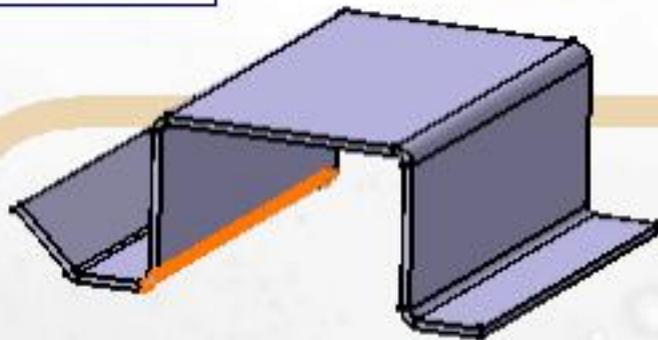
Bend Modifications are considered to be either the Radius value or angle of the wall. However they are accomplished using different windows.



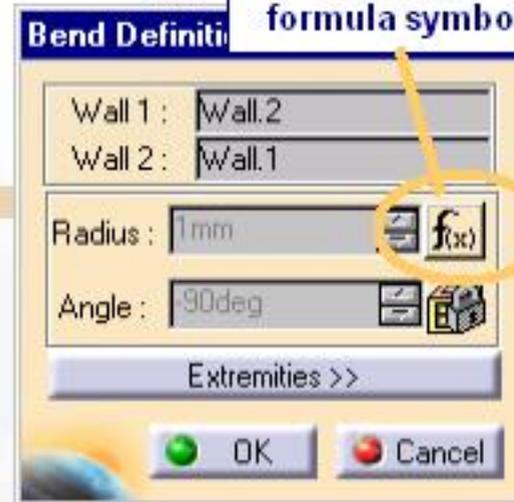
Once a Bend Radius has been unlocked from the formula that drives it, it always remains unlocked and thus will not change even if you change it back. The link must be restored using the Wizard before it will again be driven by the standard radius.

Modifying a Bend

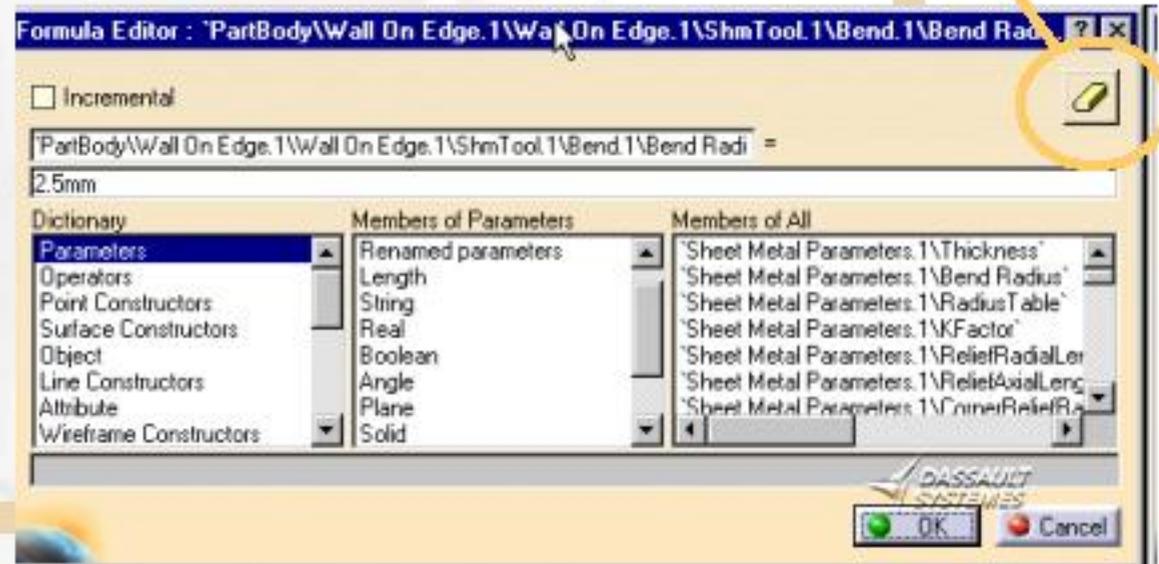
1 Select the Bend



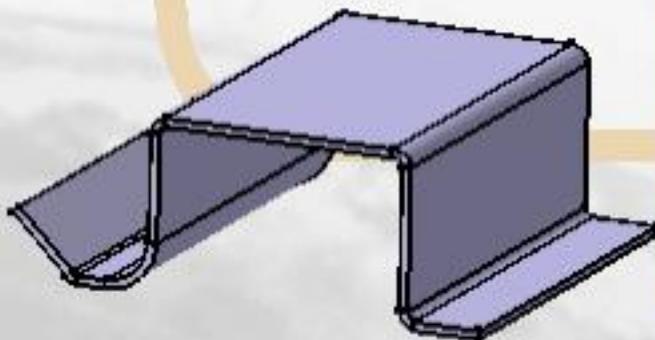
2 Select the formula symbol



3 Select the erase icon to clear the formula and key in a new value



Tip: during the modification the direction arrows are shown allowing for modification (reverse)



Only the bend radius is modifiable in the operation; angles must be managed from the wall creation window.

Modifying the Angle of a Bend

1 Select the Wall

2 Key in the new angle

Wall On Edge Definition

Height : 45mm

Limit 1 : 10mm

Limit 2 : 10mm

Clearance : 4mm

Angle : 90deg

Invert Sketch Profile

Invert Material Side

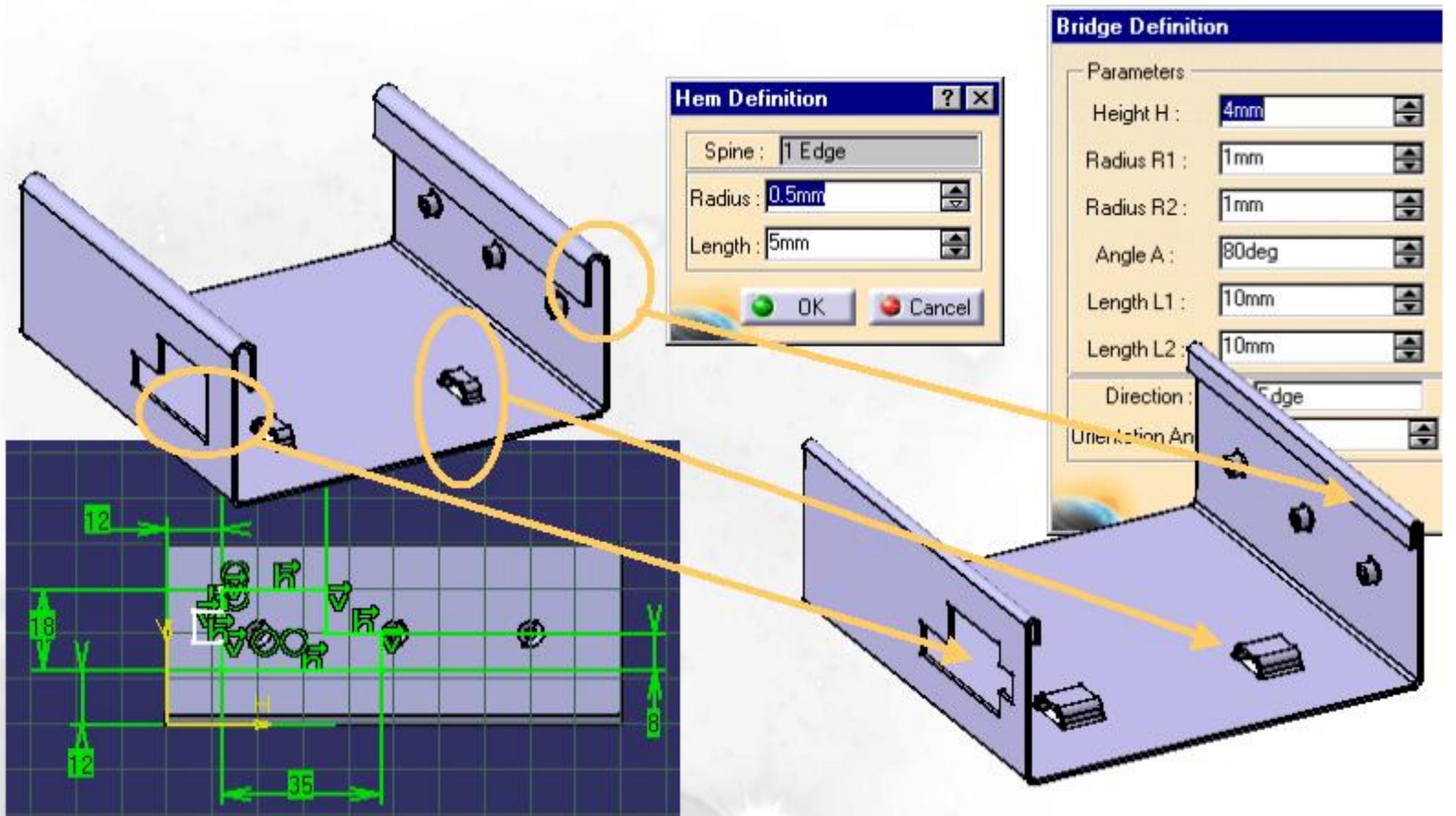
With Bend

OK Cancel

Note: walls created from an extrusion are not able to have the angle modified using this operation; modify the sketch.

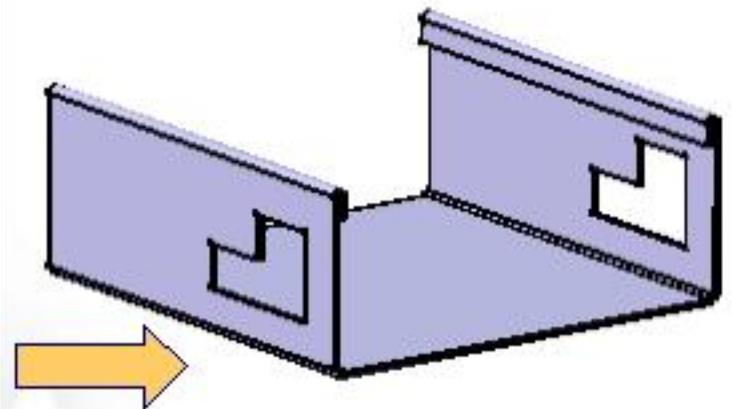
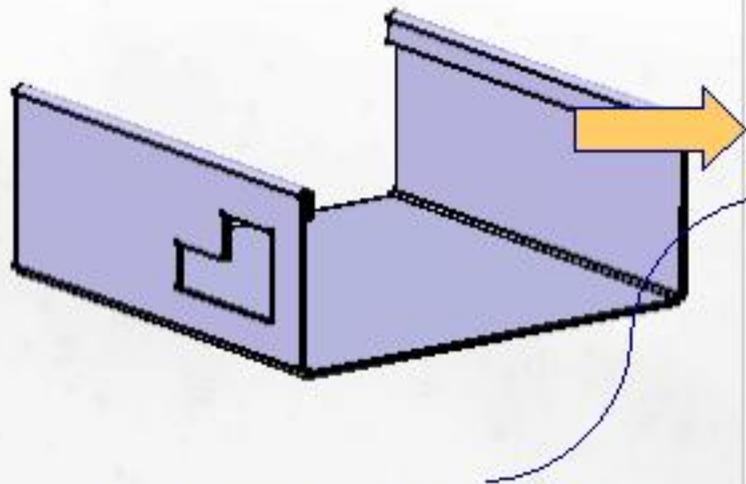
Modifying Flanges, Cut Outs and Stamp Features

Each of these items can be modified using their parameters

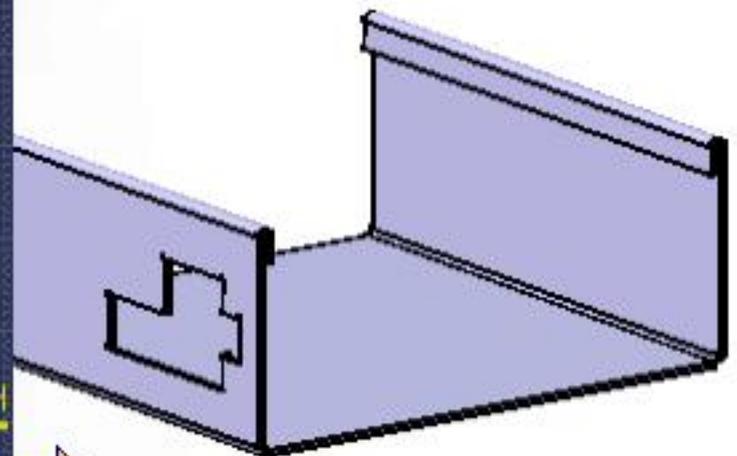
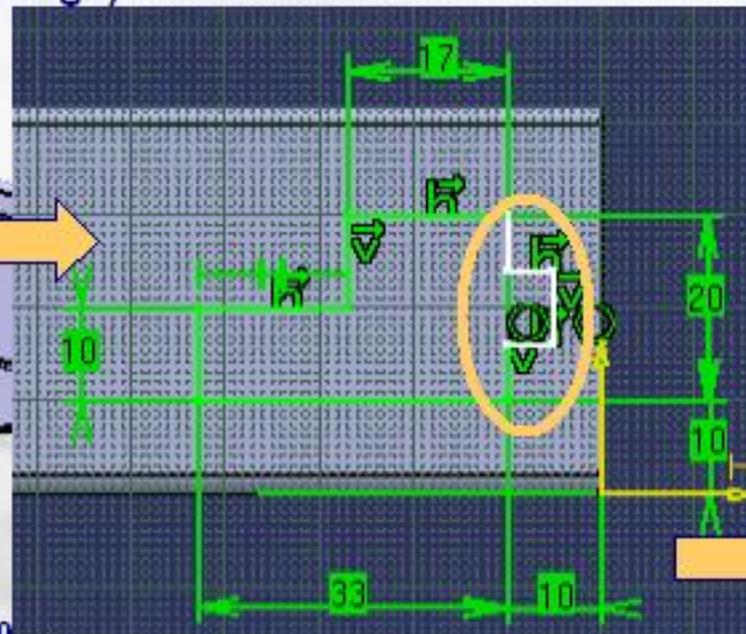
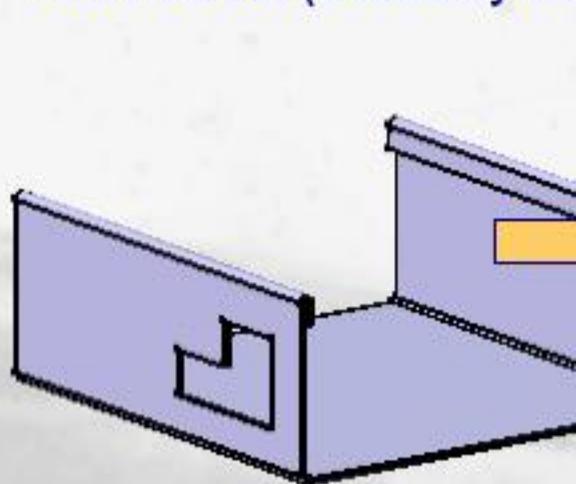


What types of modifications can be done ? (1/2)

• CUT OUTS: (Depth change)

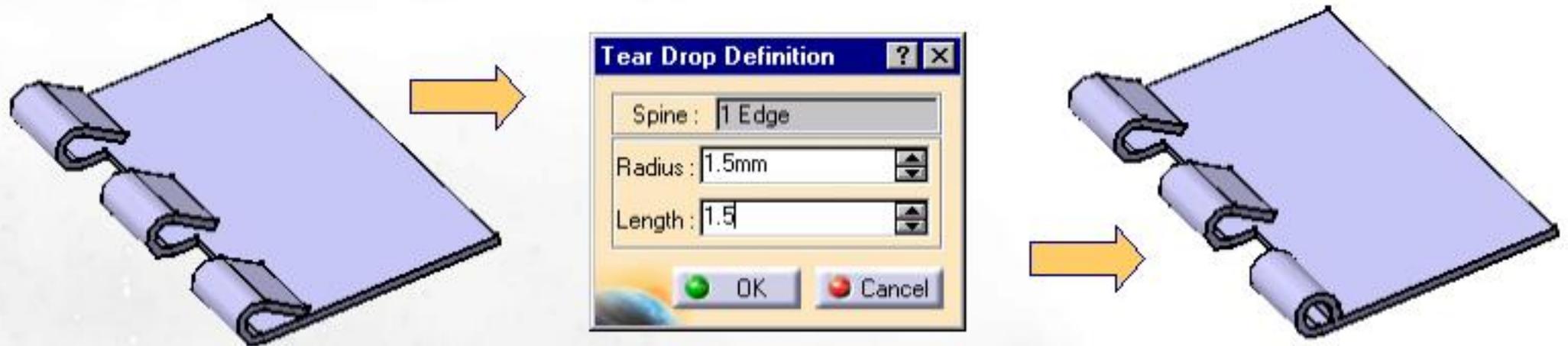


• CUT OUTS: (Geometry change)

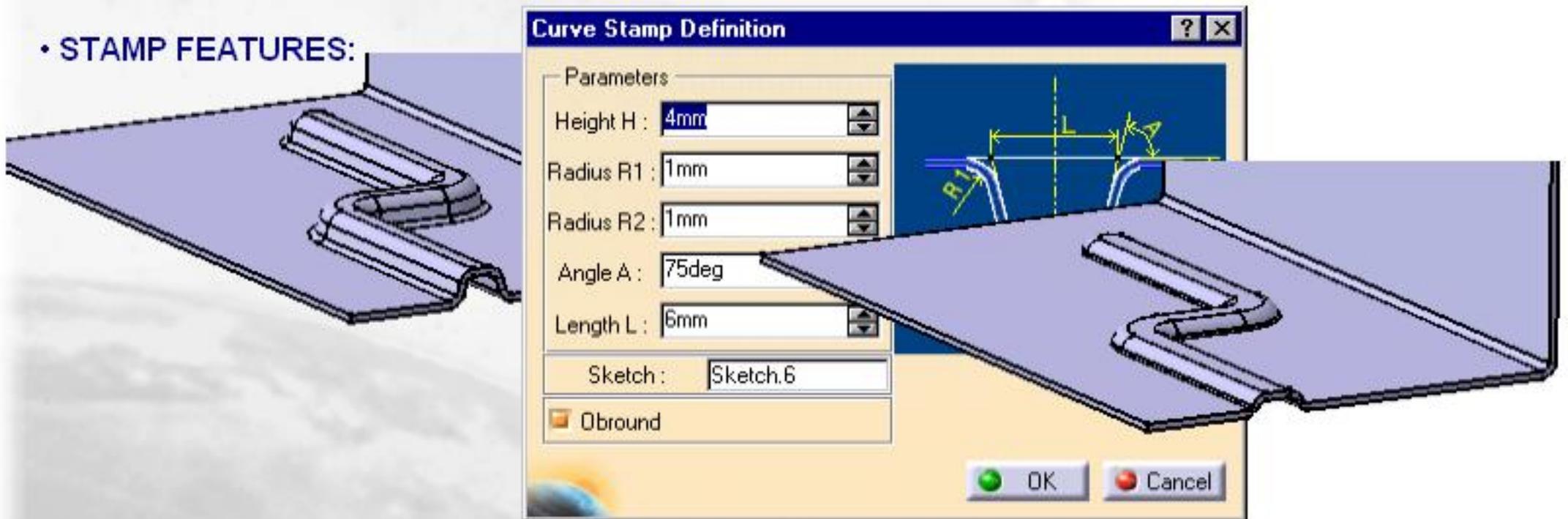


What types of modifications can be done ? (2/2)

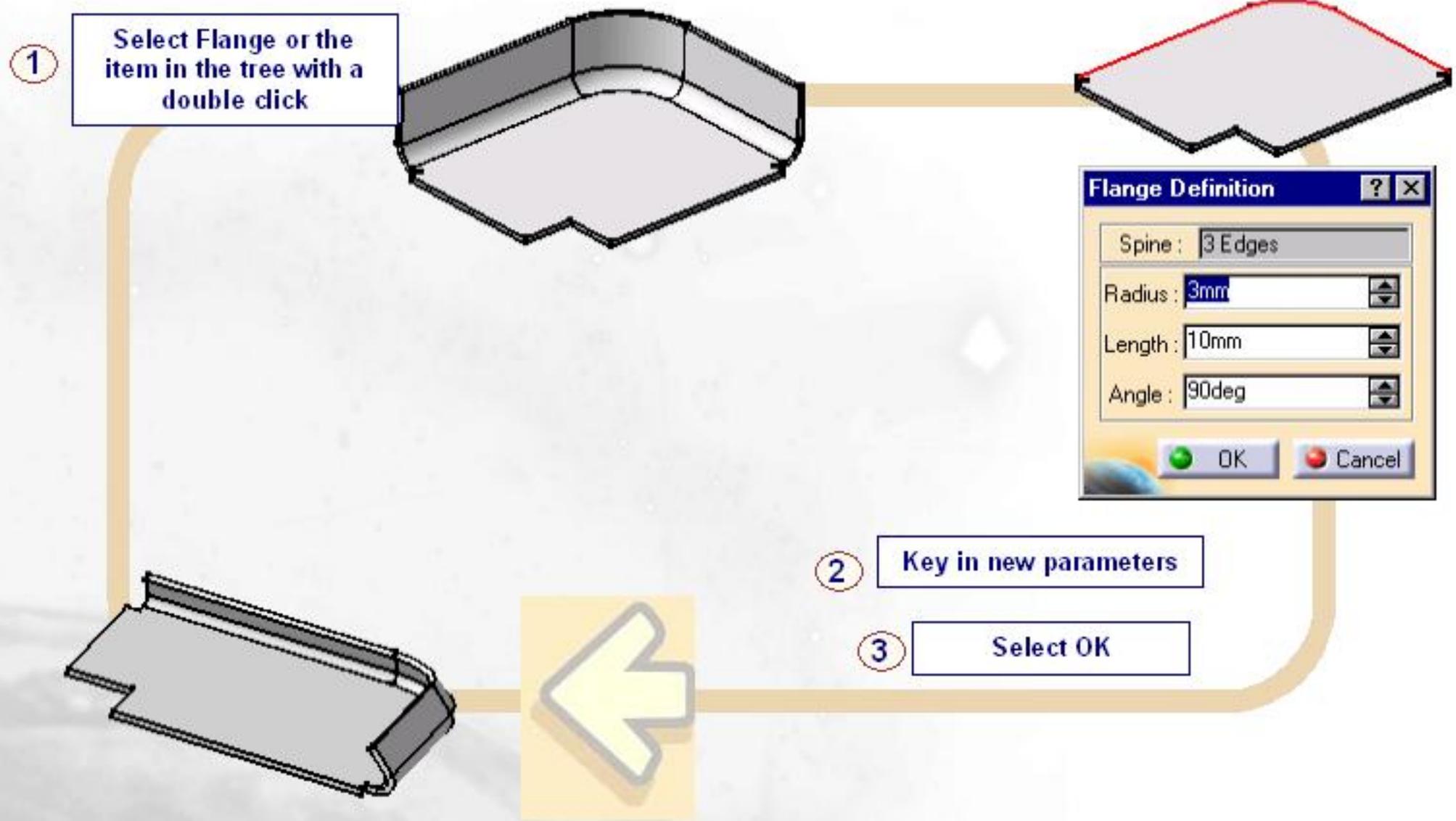
• FLANGES:



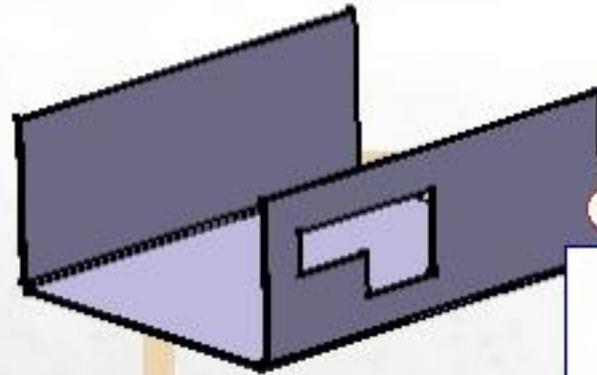
• STAMP FEATURES:



Modifying a Flange



Modifying a Cut Out



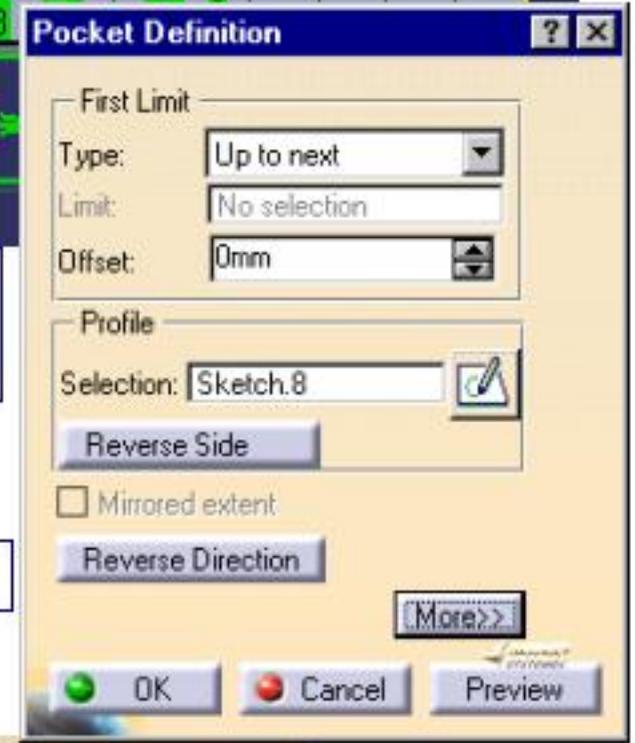
1

Select Cut Out or the item in the tree in the tree with a double click



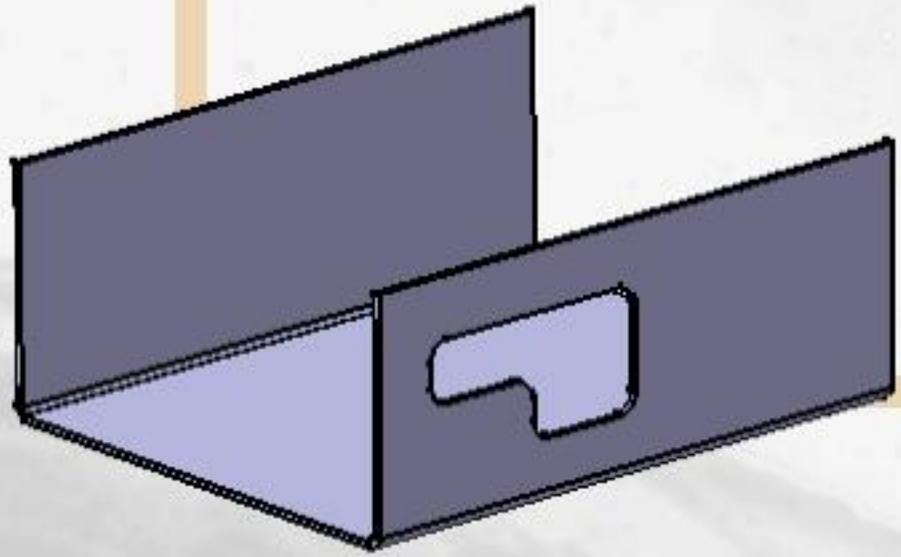
2

Modify the parameters or sketch.

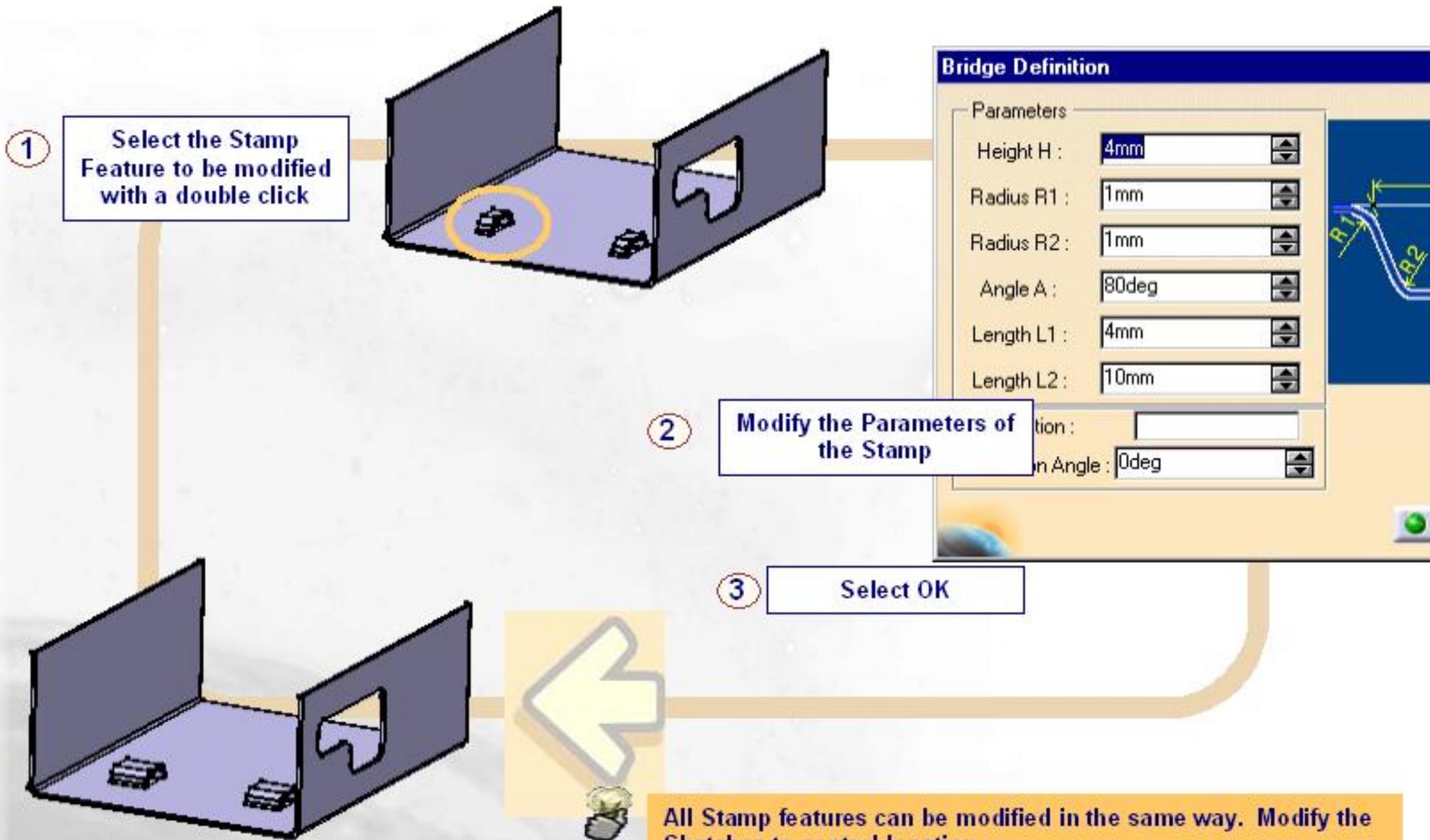


3

Select OK



Modifying a Stamp Feature

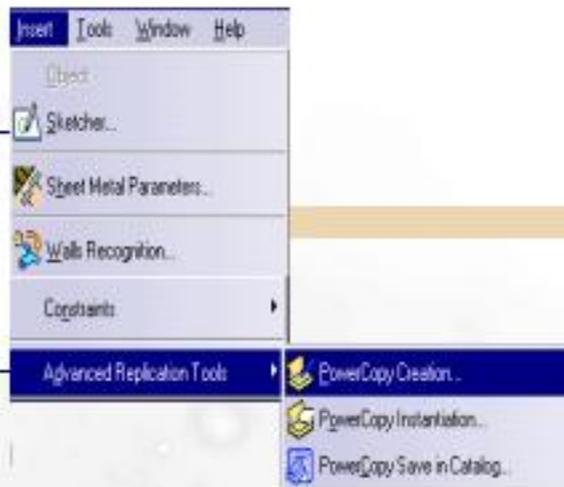


All Stamp features can be modified in the same way. Modify the Sketches to control location.

Create Power Copy

1

In an existing SheetMetal part, select the Insert/Advanced Replication Tools / PowerCopy Creation menu item. The PowerCopy Definition dialog box is displayed.



2

Select, from the specification tree, the elements to be included in the PowerCopy.



The Power Copy is created in the specification tree

The PowerCopy Definition dialog box is automatically filled with information about the selected elements.



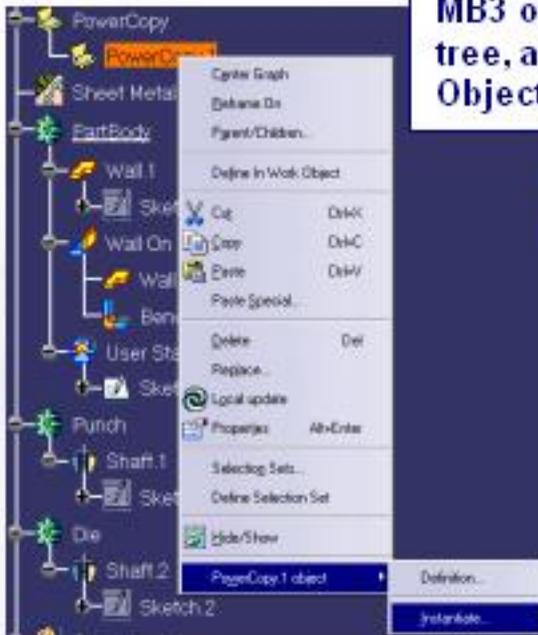
3

Click OK to validate

Instantiate Power Copy

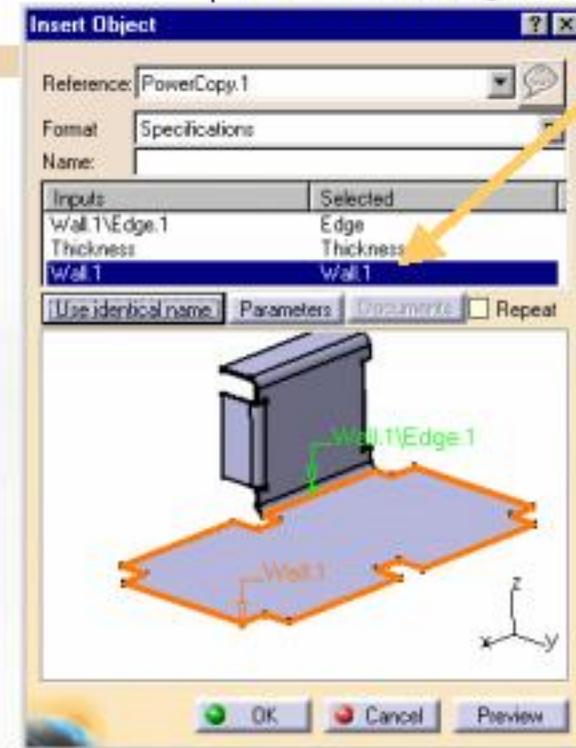
1

MB3 on Power Copy in the tree, and select Power Copy Object and Instantiate...



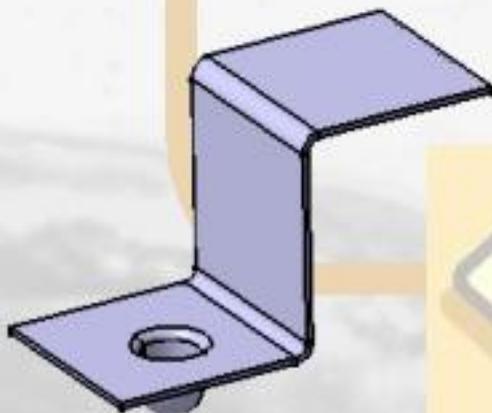
2

The Insert Object window is displayed. Answer it with the new inputs by clicking on it and on the 3D geometry.



3

Click OK to validate. The new feature is displayed.

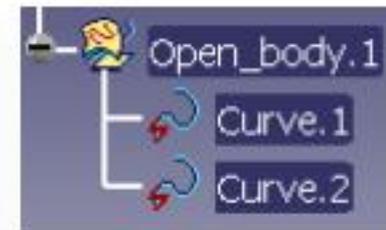
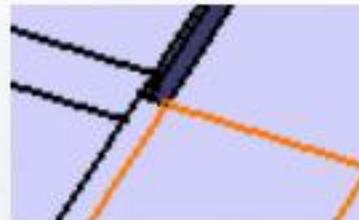


Using the SheetMetal Production Workbench (1/2)



Check Overlapping:

This tool will verify if there is no overlapping in the SheetMetal Part.
All the overlapped elements will be highlighted in the 3D view



Clicking OK, a curve will be created, in an open body, representing the overlapped area.

Check Bend Radius:

This tool will verify the value of the bends inside a SheetMetal part. If this value is different from the standard value, the non-standard bends will be highlighted and their names will be inserted inside an information panel.

