SheetMetal Design

Overview

Welcome to the *SheetMetal Design User's Guide*. This guide is intended for users who need to become quickly familiar with the SheetMetal Design Version 5 product.

This overview provides the following information:

- SheetMetal Design in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

SheetMetal Design in a Nutshell

The SheetMetal Design workbench is a new generation product offering an intuitive and flexible user interface. It provides an associative feature-based modeling, making it possible to design sheet metal parts in concurrent engineering between the unfolded or folded part representation.

SheetMetal Design offers the following main functions:

- Associative and dedicated sheet metal feature-based modeling
- Concurrent engineering between the unfolded or folded part representation
- Access to company-defined standards tables
- Dedicated drawing capability including unfolded view and specific settings.

All sheet metal specifications can be re-used by the Knowledge Advisor workbench to capture corporate knowledge and increase the quality of designs.

Natively integrated, SheetMetal Design offers the same ease of use and user interface consistency as all V5 applications.

As a scalable product, SheetMetal Design can be used in cooperation with other current or future companion products in CATIA V5 such as Assembly Design and Generative Drafting. The widest application portfolio in the industry is also accessible through interoperability with CATIA Solutions Version 4 to enable support of the full product development process from initial concept to product in operation.

The *SheetMetal Design User's Guide* has been designed to show you how to design sheet metal parts of varying levels of complexity.

Getting the Most out of This Guide

Before reading this guide, you should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. Therefore, we recommend that you read the *Infrastructure User's Guide* that describes generic capabilities common to all Version 5 products. It also describes the general layout of V5 and the interoperability between workbenches.

You may also like to read the following complementary product guides, for which the appropriate license is required:

- *Part Design User's Guide*: explains how to design precise 3D mechanical parts.
- *Assembly Design User's Guide*: explains how to design assemblies.
- *Generative Drafting User's Guide*: explains how to generate drawings from 3D parts and assembly definitions.

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading and performing the step-by-step Getting Started tutorial.

Once you have finished, you should move on to the next sections, which explain how to handle more detailed capabilities of the product.

The Workbench Description section, which describes the SheetMetal Design workbench will also certainly prove useful.

Accessing Sample Documents

To perform the scenarios, you will be using sample documents contained in the online\cfysa\samples\SheetMetal folder. For more information about this, refer to Accessing Sample Documents in the *Infrastructure User's Guide*.

Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications.

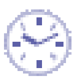
Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

This icon...	**Identifies...**
	estimated time to accomplish a task
	a target of a task
	the prerequisites
	the start of the scenario
	a tip
	a warning
	information
	basic concepts
	methodology
	reference information
	information regarding settings, customization, etc.
	the end of a task

	functionalities that are new or enhanced with this release
	allows you to switch back to the full-window viewing mode

Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

This icon...	Indicates functions that are...
P1	specific to the P1 configuration
P2	specific to the P2 configuration
P3	specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are denoted as follows:

This icon...	Gives access to...
	Site Map
	Split View mode
	What's New?
	Overview
	Getting Started
	Basic Tasks
	User Tasks or the Advanced Tasks
	Workbench Description
	Customizing
	Reference
	Methodology
	Glossary

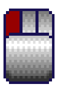
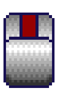
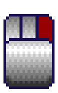
Text Conventions

The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

How to Use the Mouse

The use of the mouse differs according to the type of action you need to perform.

Use this mouse button...	Whenever you read...
	• Select (menus, commands, geometry in graphics area, ...) • Click (icons, dialog box buttons, tabs, selection of a location in the document window, ...) • Double-click • Shift-click • Ctrl-click • Check (check boxes) • Drag • Drag and drop (icons onto objects, objects onto objects)
	• Drag • Move
	• Right-click (to select contextual menu)

What's New?

There are no new or improved capabilities in Version 5 Release 14 of CATIA SheetMetal Design.

Getting Started

Before getting into the detailed instructions for using SheetMetal Design, the following tutorial provides a step-by-step scenario demonstrating how to use key functionalities.

The main tasks proposed in this section are:

Entering the Workbench
Defining the Sheet Metal Parameters
Creating the First Wall
Creating the Side Walls
Creating a Cutout
Creating Automatic Bends Automatically
Extracting Drawings from the Sheet Metal Part

All together, these tasks should take about 15 minutes to complete.

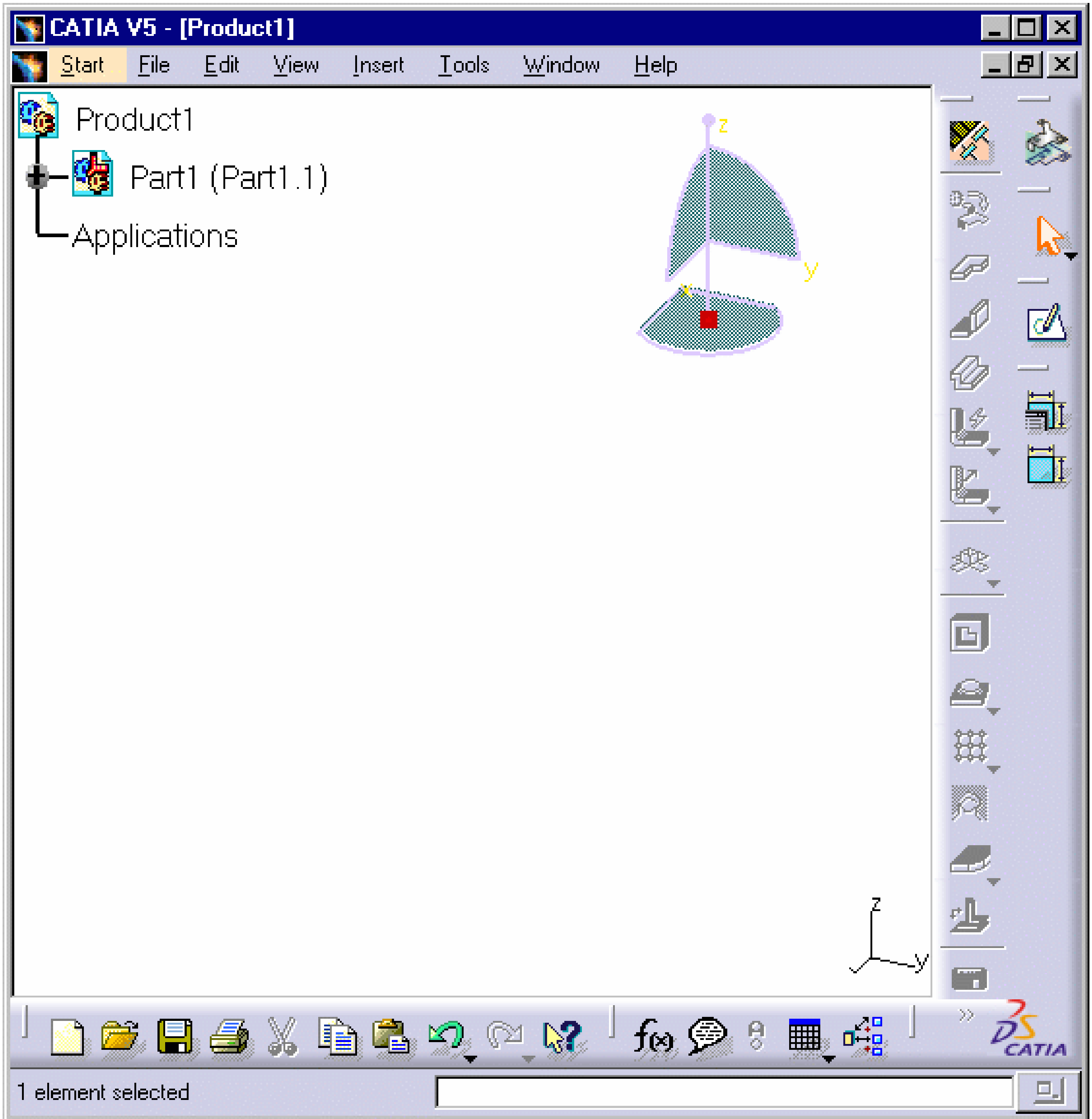
Entering the Workbench

The SheetMetal Design or Generative SheetMetal Design functions are available when you are in the Part environment. Several functions are integrated from the Part Design workbench.

This task shows how to enter the workbench.

For the SheetMetal Design workbench, choose the **Mechanical Design -> Sheet Metal Design** item from the **Start** menu.
For the Generative Sheetmetal Design workbench, choose the **Mechanical Design -> Generative Sheetmetal Design** item from the **Start** menu.

According to the chosen workbench, the corresponding Sheet Metal toolbar is displayed and ready to use.



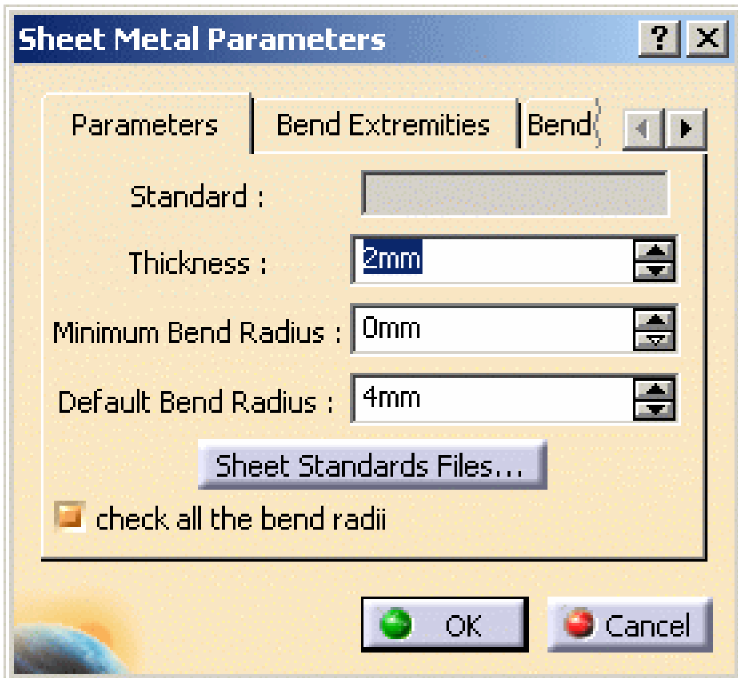
You may add the SheetMetal Design or the Generative SheetMetal Design workbench to your Favorites, using the **Tools -> Customize** item. For more information, refer to the *Infrastructure User's Guide*.

Defining the Sheet Metal Parameters

This task shows you how to configure the sheet metal parameters.

1. Click the **Sheet Metal Parameters** icon .

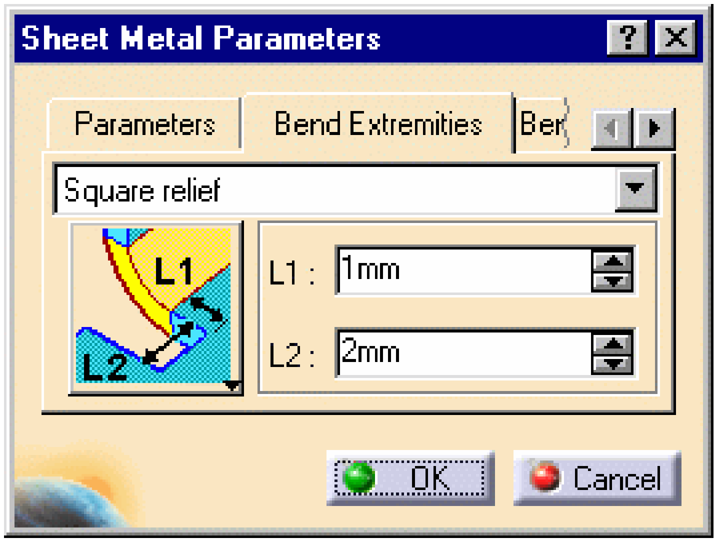
The Sheet Metal Parameters dialog box is displayed.



2. Enter 1mm in the **Thickness** field.

3. Enter 5mm in the **Default Bend Radius** field.

4. Select the **Bend Extremities** tab.

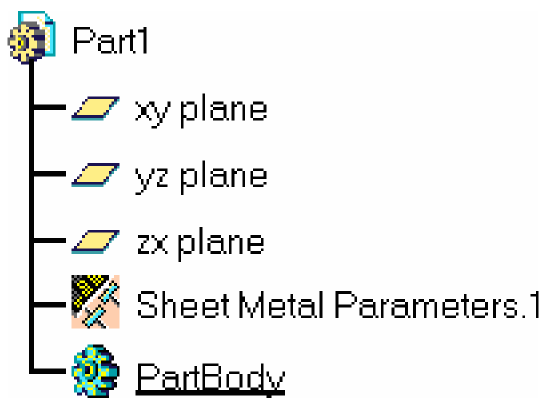


5. Select **Tangent** in the Bend Extremities combo list.

An alternative is to select the bend type in the graphical combo list.

6. Click **OK** to validate the parameters and close the dialog box.

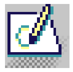
   The **Sheet Metal Parameters** feature is added in the specification tree.

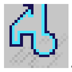


The other two tabs are not used in this scenario.

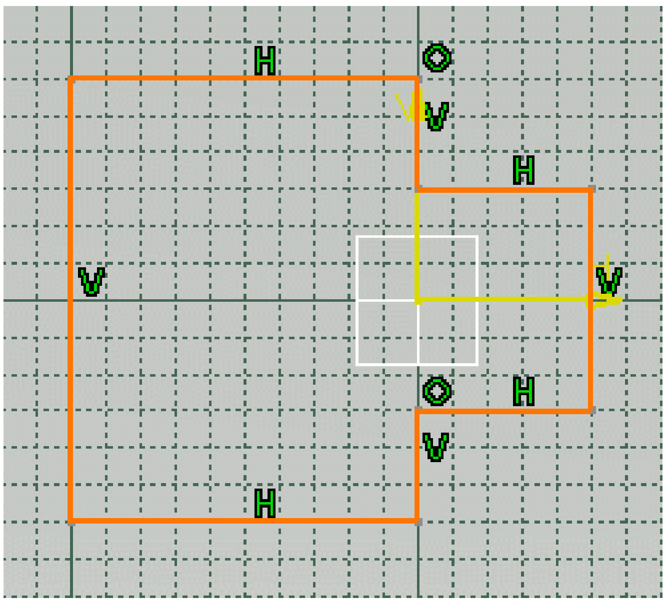
Creating the First Wall

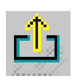
This task shows how to create the first wall of the sheet metal Part.

1. Click the **Sketcher** icon  then select the **xy plane**.

2. Select the **Profile** icon .

3. Sketch the profile as shown below:

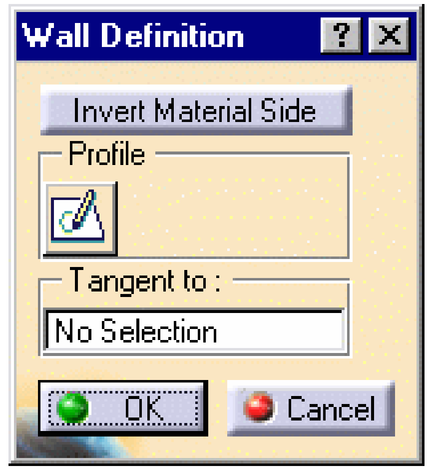


4. Click the **Exit workbench** icon  to return to the 3D world.

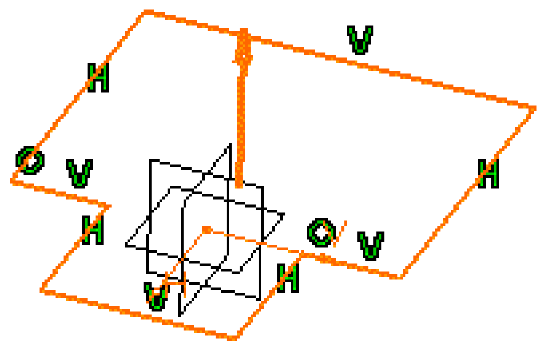
   The sketch remains selected.

5. Click the **Wall** icon .

The Wall Definition dialog box opens.

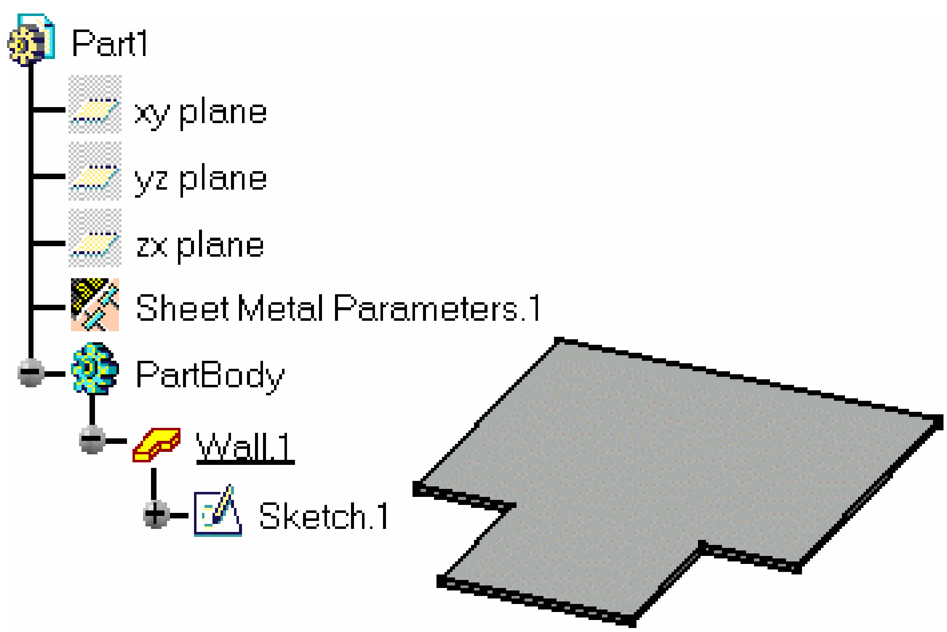


By default, the Material Side is set to the top.



6. Click OK.

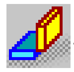
The **Wall.1** feature is added in the specification tree



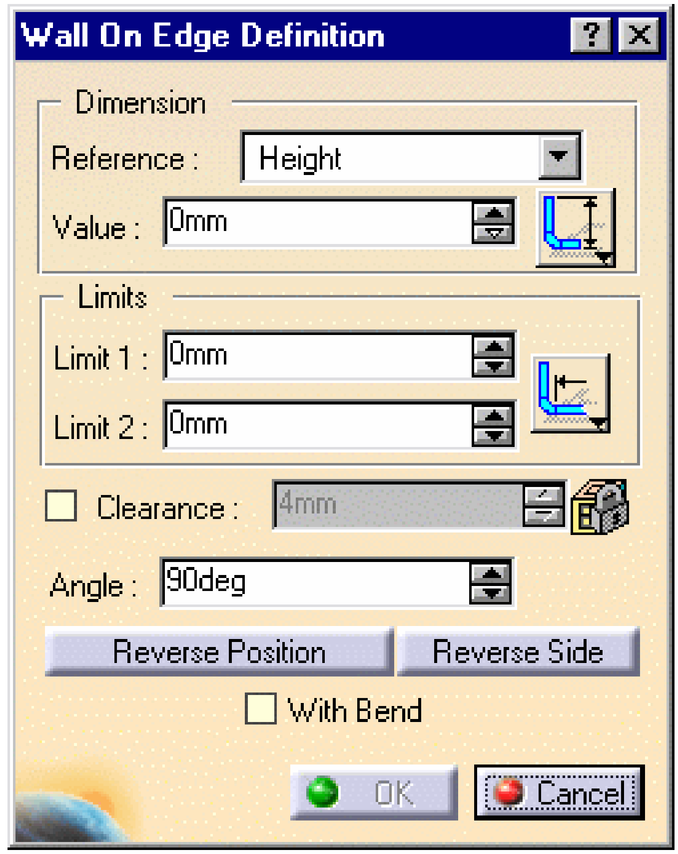
The first wall of the Sheet Metal Part is known as the reference wall.

Creating the Side Walls

This task shows you how to add other walls to the sheet metal part.

1. Click the **Wall on Edge** icon .

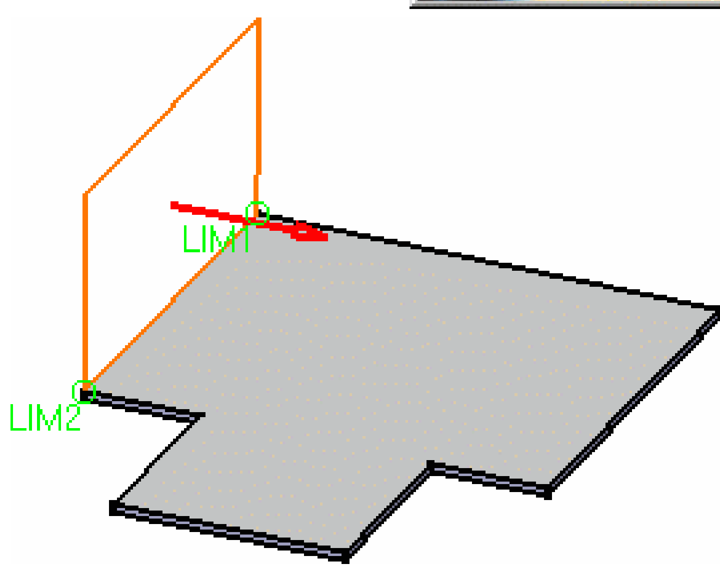
   The Wall On Edge Definition dialog box opens.



2. Select the left edge.

3. Enter 50mm in the **Value** field.

   The application previews the wall.

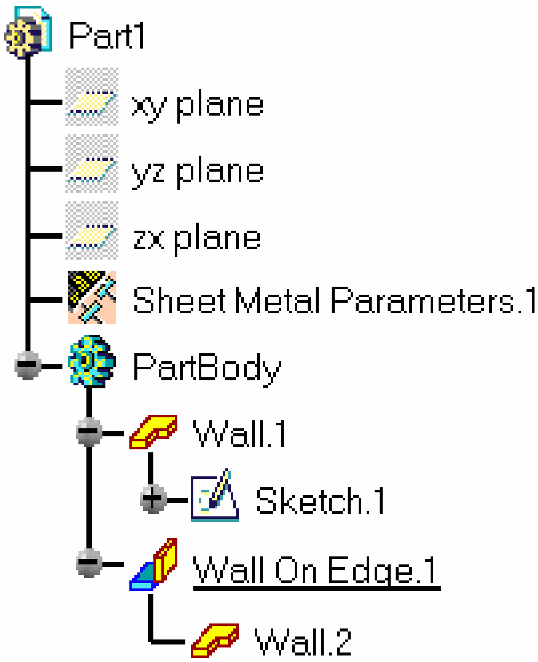


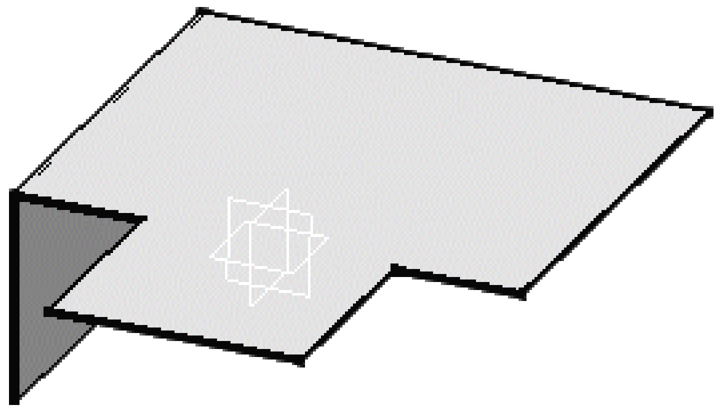
By default, the material side is such that it ensures a continuity with the reference profile. If needed, invert it using the **Reverse side** button, or clicking the arrow.

4. Click the **Reverse Position** button to Invert the sketch profile.

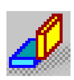
5. Click **OK**.

   The wall is created and the **Wall On Edge.1** feature is displayed in the specification tree:



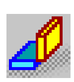


6. Select the right edge.

7. Click the **Wall on Edge** icon  again.

   The Wall On Edge Definition dialog box opens with the parameters previously selected.

8. Invert the sketch profile and click **OK** to validate.

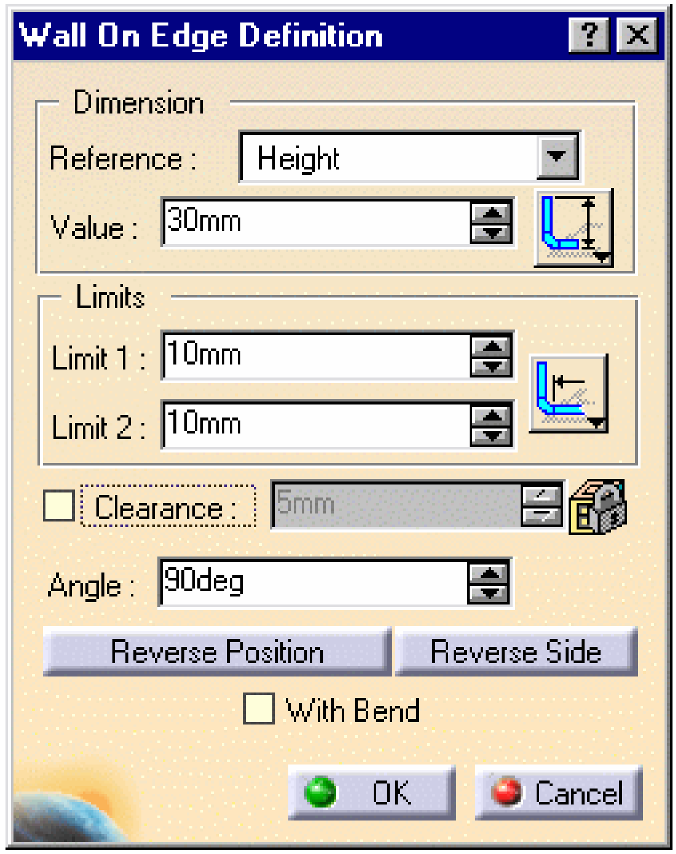
9. Click the **Wall on Edge** icon  again.

10. Select the front edge.

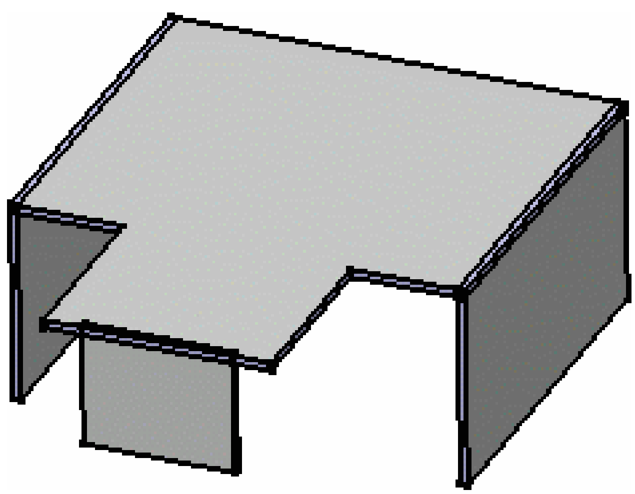
    The Wall Definition dialog box opens with the parameters previously selected.

11. Enter 30mm in the **Value** field and 10mm in the **Limit1** and **Limit2** fields, then invert the sketch profile.

12. Press **OK** to validate.



The final part looks like this:

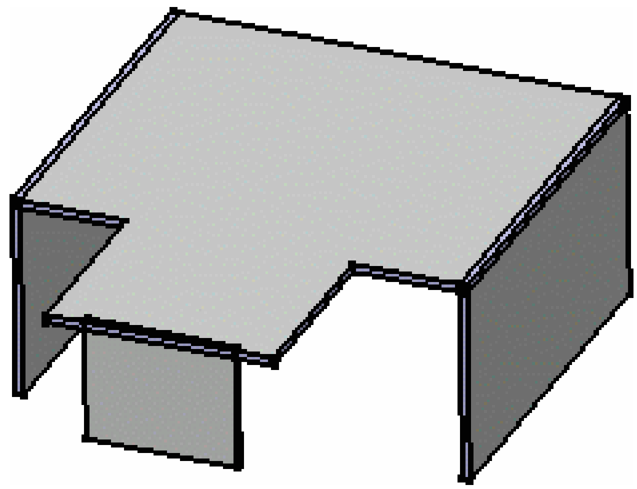


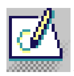
Creating a Cutout

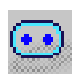
In this task, you will learn how to:

- open a sketch on an existing face
- define a profile in order to create a cutout.

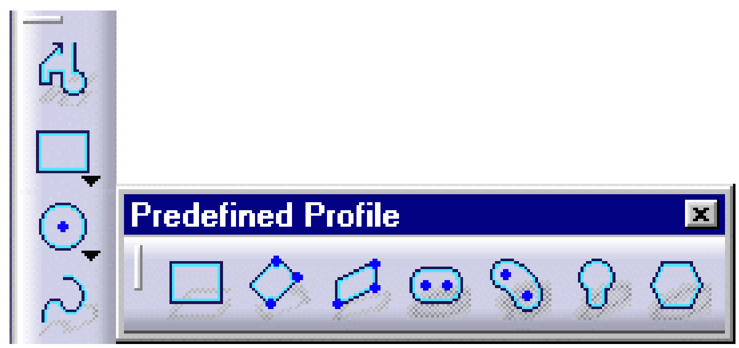
1. Select **Wall On Edge.2** from the geometry area to define the working plane.



2. Click the Sketcher icon .

3. Click the **Elongated Hole** icon  to create the profile.

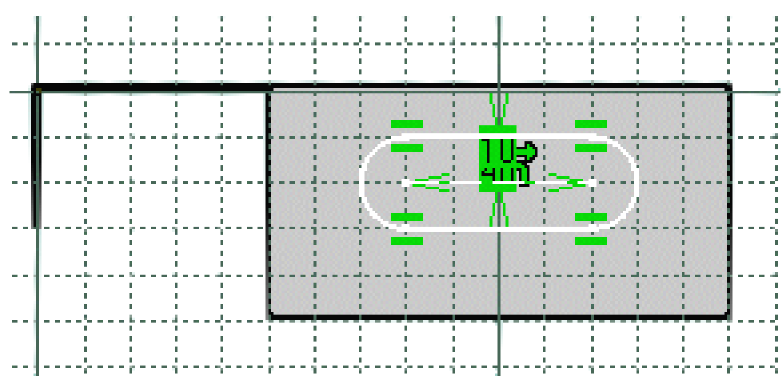
To access the oblong profile, click the black triangle on the **Rectangle** icon. It displays a secondary toolbar.



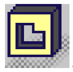
4. Click to create the first point and drag the cursor.

5. Click to create the second point. The first semi-axis of the profile is created.

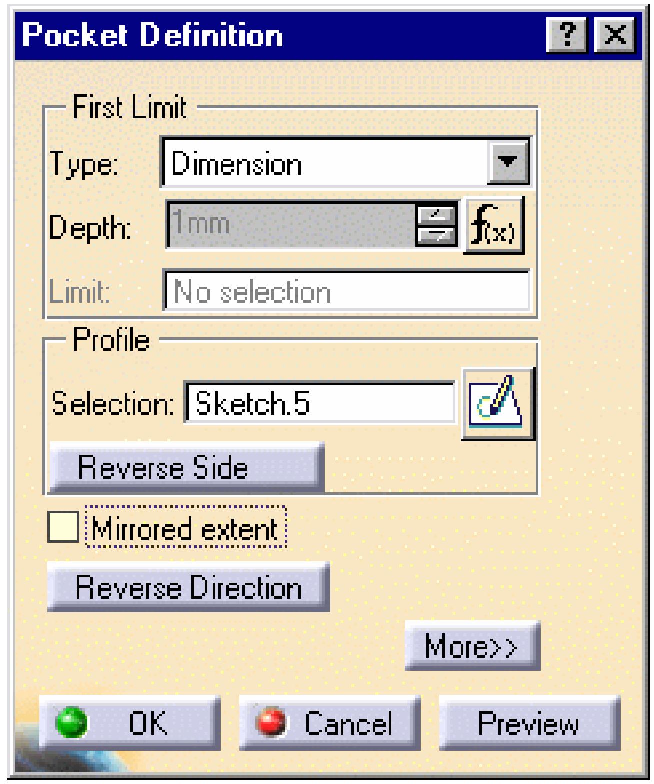
6. Drag the cursor and click to create the third point. The second semi-axis is created and the oblong profile is displayed.



7. Click the **Exit workbench** icon  to return to the 3D world.

8. Select the **Cutout** icon .

The **Pocket Definition** dialog box is displayed and a cutout is previewed with default parameters.

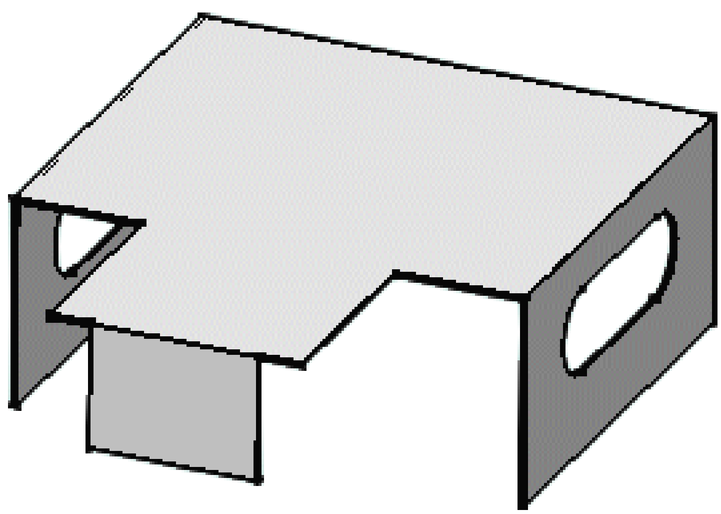


9. Set the Type to **Up to last** option to define the limit of your cutout.

This means that the application will limit the cutout onto the last possible face, that is the opposite wall.

10. Click **OK**.

This is your cutout:



Creating the Bends Automatically

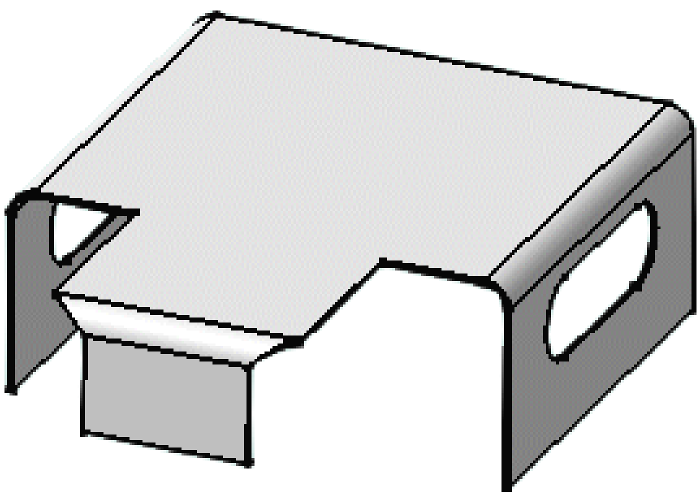
This task applies to the SheetMetal Design workbench only, NOT to the Generative Sheetmetal Design workbench.

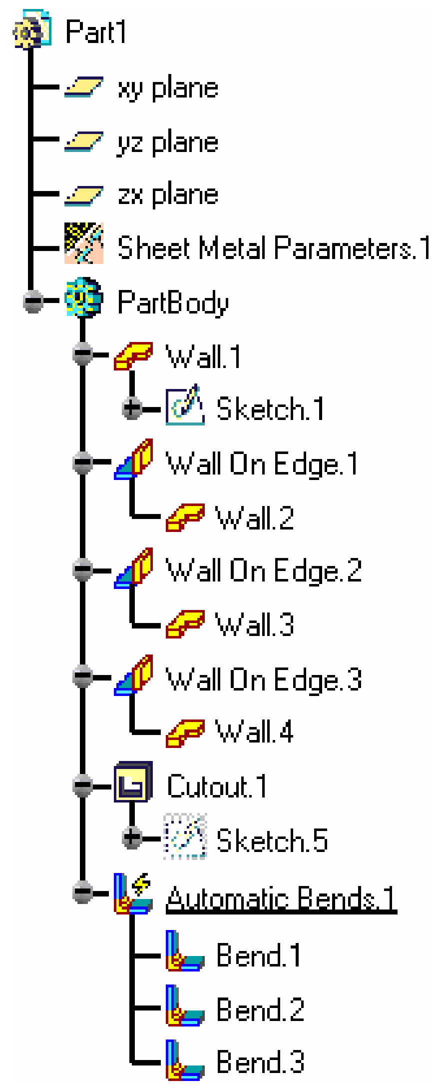
This task shows how to create the bends automatically.

1. Click the **Automatic Bends** icon . The bends are created.

The bends are displayed in the specification tree: **Automatic Bends.1**

The sheet metal part looks like this:





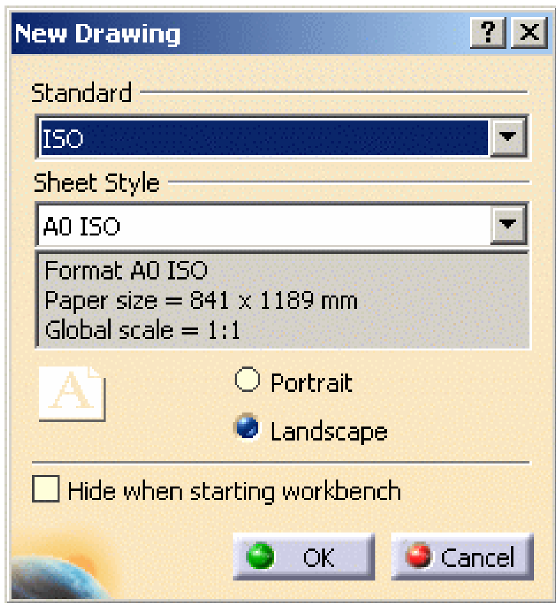
Extracting Drawings from the Sheet Metal Part

This task shows how to create the sheet metal part views in the Generative Drafting workbench.

The sheet metal part is displayed.

1. Click or select **File -> New...**
2. Select the **Drawing** type and click OK.

   The Generative Drafting workbench is launched. The New Drawing dialog box opens.



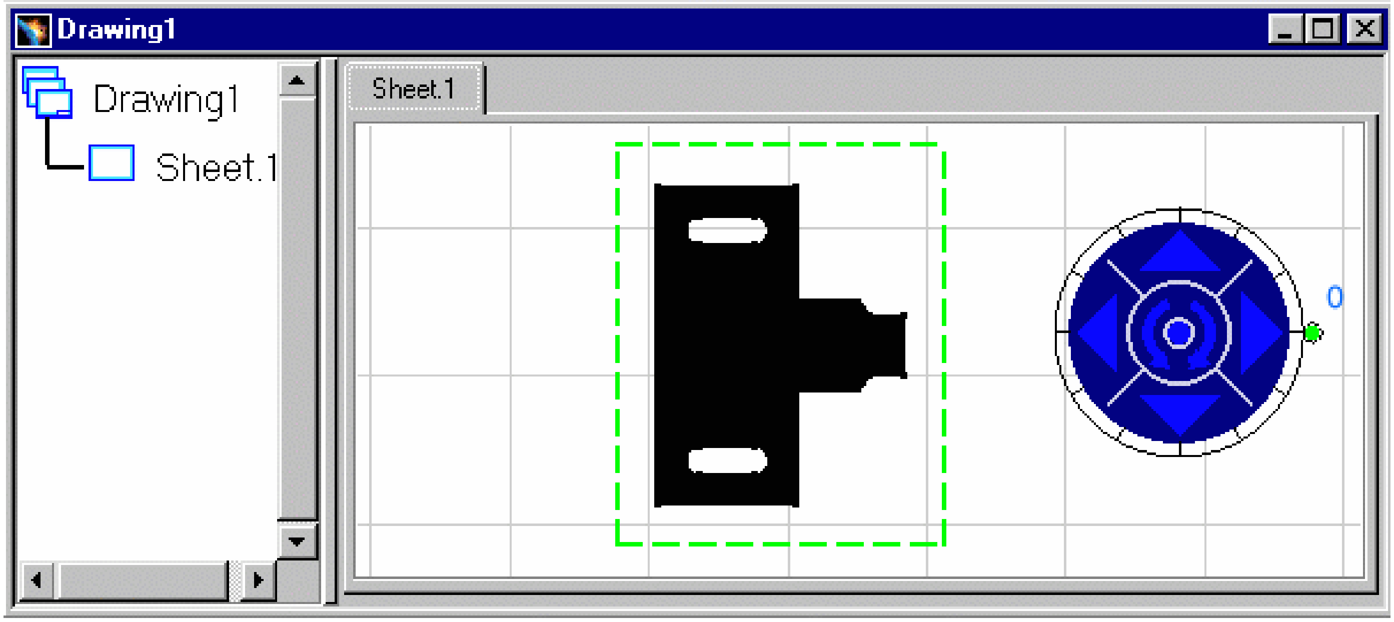
3. Click **OK**.

For more information about this workbench, refer to *Generative Drafting User's Guide*.

4. The drawing sheet appears.
5. Tile the windows horizontally using the **Window -> Tile Horizontally** menu item.
6. Select the Unfolded View icon in the Projections toolbar from Generative Drafting Workbench.

This icon is added to the Projections toolbar provided the Sheet Metal workbench is present.

7. Choose the **xy plane** in the Sheet Metal specification tree. The unfolded view is previewed.



8. Click in the drawing to validate and generate the view, with the bend axes and bend limits when applicable.

Basic Tasks

The Basic Tasks section explains how to create and modify various kinds of features.

Managing the Default Parameters

This section explains and illustrates how to use or modify various kinds of features. The information you will find is listed below.

Using SheetMetal Design assumes that you are in a CATPart document.

Edit the parameters: select the **Parameters** tab, the wall thickness and bend radius values.

Modify the bend extremities : select the **Bend Extremities** tab and choose a predefined bend type.

Define the bend corner relief: select the **Bend Corner Relief** tab and choose a predefined corner relief type.

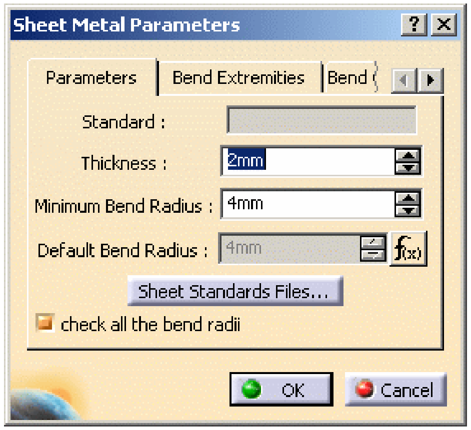
Define the bend allowance: select the **Bend Allowance** tab and define the allowance value (K factor).

Editing the Sheet and Tool Parameters

This section explains how to change the different sheet metal parameters needed to create your first feature.

1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box is displayed.



2. Change the **Thickness** if needed.

3. Change the **Minimum Bend Radius** if needed.

   The Minimum Bend radius defines the minimum internal radius allowing the creation of a bend.

You can set the value to 0 to create bend with no radius. If using the DIN standard, the KFactor automatically sets to 0 as well.

This option is only relevant with the Generative Sheetmetal Design workbench.

4. Change the **Default Bend Radius** if needed.

   The Default Bend Radius corresponds to the internal radius and is linked by default to the creation of the bends.

Convention dictates that the inner angle between the two elements is used to define the bend.
It can vary from 0deg to 180deg exclusive. This angle is constant and the bend axis is rectilinear.

This option is only relevant with the Generative Sheetmetal Design workbench.

5. Click **OK** to validate the Sheet Metal Parameters.

The Standard field displays the Standard to use with the part, if implemented. The name of this standard file is defined in a Design Table.
When the **Check all the bend radii** button is checked, and you click OK in the Sheet Metal Parameters dialog box, existing bend radii are checked and a list displays flanges or bends that do not use the minimum Bend Radius value as defined in step 3. Therefore, they will not be modified.

Parameters can be defined in a Design Table. To do so, press the **Sheet Standards Files...** button to access to the company-defined standards, if need be. For more information, refer to the Customizing Standard Files section.

All parameters hereafter, or only some of them, can be defined in this Design Table:

Sheet Metal Parameters	Column associated in the Design Table	Definition
Standard in Sheet Metal Parameters	SheetMetalStandard	sheet reference name
Thickness	Thickness	sheet thickness
Minimum Bend Radius	MinimumBendRadius	minimum bend radius
Default Bend Radius	DefaultBendRadius	default bend radius
K Factor	KFactor	neutral fiber position
Radius Table	RadiusTable	path to the file with all available radii

In all cases, the Thickness parameter must be defined in the Design Table in order for the other parameters to be taken into account.

Whenever both Radius Table and Default Bend Radius are defined in the Design Table, only the Radius Table will be taken into account for the bend creation.

Standard Names For Holes	Column associated in the Design Table	Definition
Clearance Hole	ClearanceHoleStd	path to the Clearance Hole Standard file
Index Hole	IndexHoleStd	path to the Index Hole Standard file
Manufacturing Hole	ManufacturingHoleStd	path to the Manufacturing Hole Standard file
Fastener Hole	FastenerHoleStd	path to the Fastener Hole Standard file
Standard Names For Stamps	**Column associated in the Design Table**	**Definition**
Flanged Hole	ExtrudedHoleStd	path to the Flanged Hole Standard file
Bead	BeadStd	path to the Bead Standard file
Circular Stamp	CircularStampStd	path to the Circular Stamp Standard file
Surface Stamp	SurfaceStampStd	path to the Surface Stamp Standard file
Flanged CutOut	FlangedCutoutStd	path to the Flanged CutOut Standard file
Curve Stamp	CurveStampStd	path to the Curve Stamp Standard file

When a parameter refers to a path, another sub-Design Table will be associated to the corresponding feature.

1. Here is an example for the use of a bend allowance table:

Main Sheet Metal Parameters Design Table

	A	B	C	D
1	SheetMetalStandard	Thickness (mm)	RadiusTable	
2	AG 3412	2	RadiusTableForThickness2.xls	
3	AG 3824	4	RadiusTableForThickness4.xls	
4				
5				

Radius Table For Thickness 2

This table defines available all bend radii for a thickness of 2 mm. A design table will be created on the Default Bend Radius of the Sheet Metal Parameters and on the Radius of each bend.

	A	B
1	BendRadius (mm)	BendTable
2	1	BendTableT2R1.xls
3	2	BendTableT2R2.xls
4	4	BendTableT2R4.xls
5		

Bend Table for Thickness 2 and Bend Radius 1

Whenever a bend is created, a radius table will be associated. If the configuration "Bend Radius = 1mm" is selected, a new design table (the Bend Table) will be created from BendTableT2R1.xls in order to compute the bend allowance.

According to the open angle, the bend deduction will be read in the Allowance column or interpolated if necessary.

	A	B	C
1	OpenAngle (deg)	Allowance (mm)	
2	25	-1.942	
3	90	-3.644	
4	160	-0.534	
5			

2. Here is an example for the use of a hole standard file:

Main Sheet Metal Parameters Design Table

	A	B	C	D	E	F	G	H	I
1	SheetMetalStandard	Thickness (mm)	MinimumBendRadius (mm)	DefaultBendRadius (mm)	KFactor	ClearanceHoleStd	FastenerHoleStd	IndexHoleStd	ManufacturingHoleStd
2	AG 3412	2	0	4	0.36	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls
3	ST 5123	3	1	5	0.27	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls
4									
5									

Hole Standard

Whenever a hole is created, a design table will associate its radius with a standard name.

	A	B
1	StandardName	Diameter (in)
2	M1	0.39
3	M2	0.65
4	M3	0.89
5	M4	0.25
6	M5	0.56
7		

2. Here is an example for the use of a stamp standard file:

Main Sheet Metal Parameters Design Table

	A	B	C	D	E	F	G	H	I
1	SheetMetalStandard	SurfaceStampStd	CurveStampStd	CircularStampStd	BeadStd	BridgeStd	FlangedCutoutStd	ExtrudedHoleStd	StiffeningRibStd
2	AG 3412	SurfaceStampAG3412.xls	CurveStampAG3412.xls	CircularStampAG3412.xls	BeadAG3412.xls	BridgeAG3412.xls	FlangedCutoutAG3412.xls	ExtrudedHoleAG3412.xls	StiffeningRibAG3412.xls
3	ST 5123	SurfaceStamp5123.xls	CurveStampST5123.xls	CircularStampST5123.xls	BeadST5123.xls	BridgeST5123.xls	FlangedCutoutST5123.xls	ExtrudedHoleST5123.xls	StiffeningRibST5123.xls

Whenever a stamp is created, a design table will associate its dimension with a standard name.

- **Surface Stamp**

	A	B	C	D	E
1	StandardName	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
2	S1	6	80	2	2
3	S2	8	75	1	1
4					

- **Curve Stamp**

	A	B	C	D	E	F
1	StandardName	Height (mm)	Length (mm)	Angle (deg)	Radius2 (mm)	Radius1 (mm)
2	C1	4	6	75	1	1
3	C2	5	7	80	1	1

- **Circular Stamp**

	A	B	C	D	E	F
1	StandardName	Diameter (mm)	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
2	C1	10	6	80	2	2
3	C2	20	5	85	1	1

- **Bead**

	A	B	C	D	E
1	StandardName	SectionRadius(mm)	EndRadius(mm)	Height(mm)	Radius1 (mm)
2	Bead04	4	6	4	2
3	Bead09	9	10	5	3
4					

- **Bridge**

	A	B	C	D	E	F	G	H
1	StandardName	Angle (deg)	PositioningAngle (deg)	Length (mm)	Radius1 (mm)	Radius2 (mm)	Height (mm)	Width (mm)
2	B1	80	5	10	2	2	6	5
3	B2	75	4	12	1	1	8	6

- **Flanged Cutout**

	A	B	C	D
1	StandardName	Height (mm)	Angle (deg)	Radius1 (mm)
2	F1	6	80	2
3	F2	8	75	1
4				

- **Extruded Hole**
  (or **Flanged Hole** in the Generative Sheetmetal Design workbench)

	A	B	C	D
1	StandardName	Diameter (mm)	Height (mm)	Angle (deg)
2	D20	20	6	90
3	D15	15	6	70
4				

- **Stiffening Rib**

	A	B	C	D	E
1	StandardName	Angle (deg)	Radius2 (mm)	Length (mm)	Radius1 (mm)
2	S1	80	2	30	2
3	S2	75	1	35	2
4					

Modifying the Bend Extremities

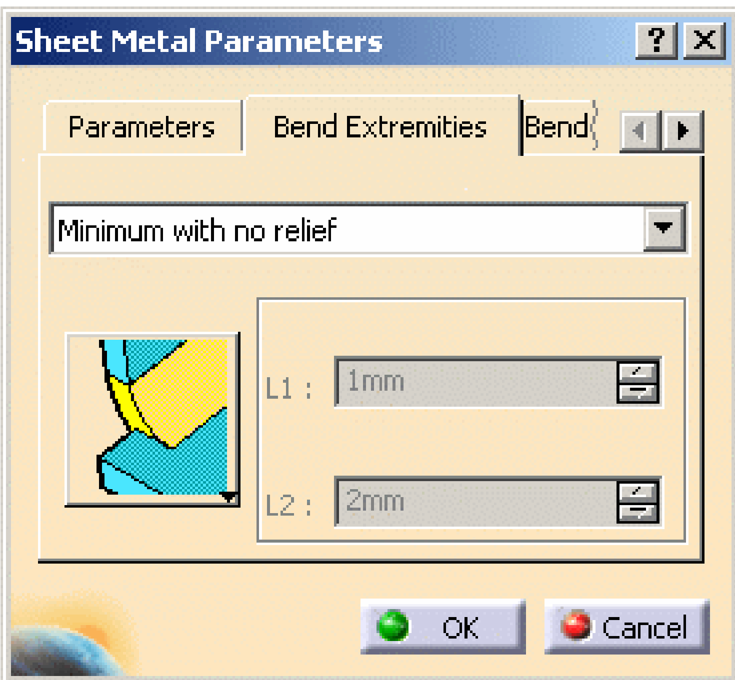
This section is only available with the SheetMetal Design products.

This section explains how to change the bend extremities, i.e. how to change axial relimitations for a straight bend.

1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box is displayed.

2. Click the **Bend Extremities** tab to access parameters defining bend extremities.



3. Choose a bend extremity, either from the drop-down list or using the graphical button underneath.

- **Minimum with no relief** (default option): the bend corresponds to the common area of the supporting walls along the bend axis, and shows no relief.
- **Square relief**: the bend corresponds to the common area of the supporting walls along the bend axis, and a square relief is added to the bend extremity. The L1 and L2 parameters can be modified if needed.
- **Round relief**: the bend corresponds to the common area of the supporting walls along the bend axis, and a round relief is added to the bend extremity. The L1 and L2 parameters can be modified if needed.
- **Linear**: the unfolded bend is split by two planes going through the corresponding limit points (obtained by projection of the bend axis onto the edges of the supporting walls).
- **Tangent**: the edges of the bend are tangent to the edges of the supporting walls.

- **Maximum**: the bend is calculated between the furthest opposite edges of the supporting walls.

4. Click **OK** to validate.

Defining the Bend Corner Relief

This section is only available for the Sheet Metal Design workbench, NOT for the Generative Sheetmetal Design workbench.

This section explains how to change the bend corner relief.

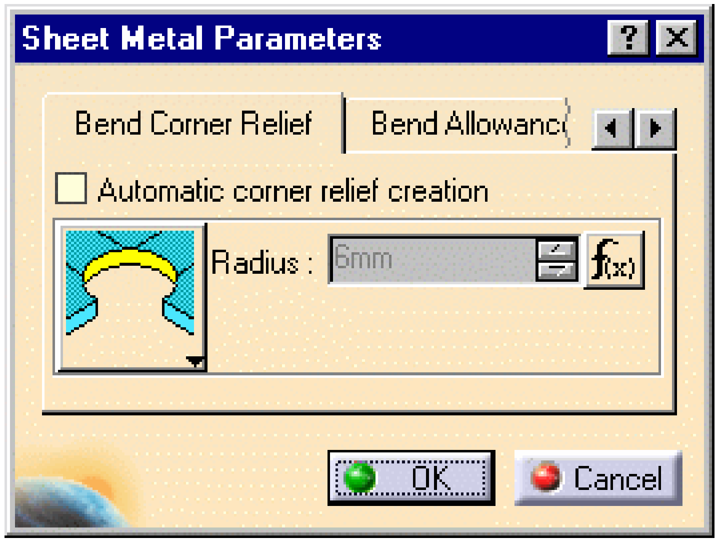
Open CornerRelief01.CATPart if you are using the Sheet Metal Design workbench.

Within the **Tools -> Options -> General -> Parameters -> Knowledge** tab, check the **Load Extended Language Libraries** option.

1. Click the **Sheet Metal Parameters** icon.

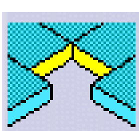
   The Sheet Metal Parameters dialog box is displayed.

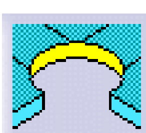
   The third tab concerns the bend corner relief.

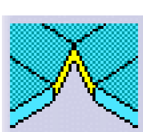


By default, no corner relief is created when a bend is created. Check the **Automatic corner relief creation** option to activate this creation every time a bend is created.

Three corner relief types are available. Select the icon corresponding to the requested type:

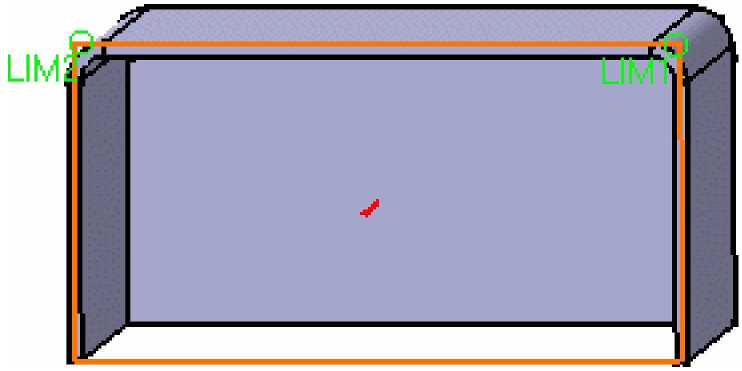
- : **square**: the square corner relief is created using the bend limits. Its dimensions are defined by the width of the unfolded bends.

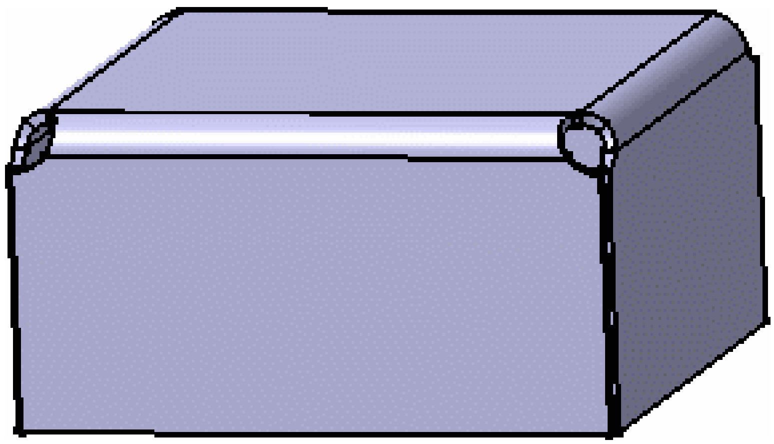
- : **circular**: its center is located at the intersection of the bend axes. For that option, a radius is proposed by default. It is equal to the bend radius + the thickness. To change it:
  - Select **Formula -> Deactivate** from the contextual menu of the input field and enter a new value,
  - click on the button and entering a new formula.

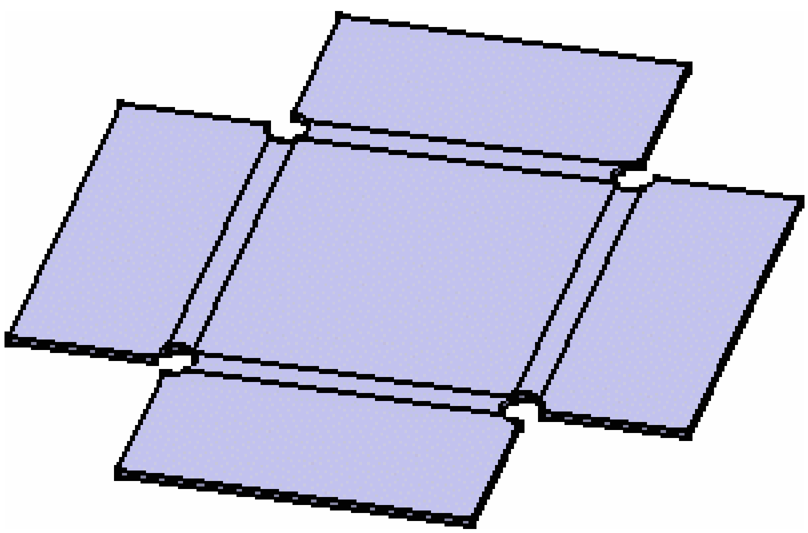
- : **triangular**: the triangular relief is created from the intersection point of the inner bend limits towards the intersection points of the outer bend limits with each wall.

The corner relief is not previewed during its creation.

The corner relief is taken into account in the unfolded view.





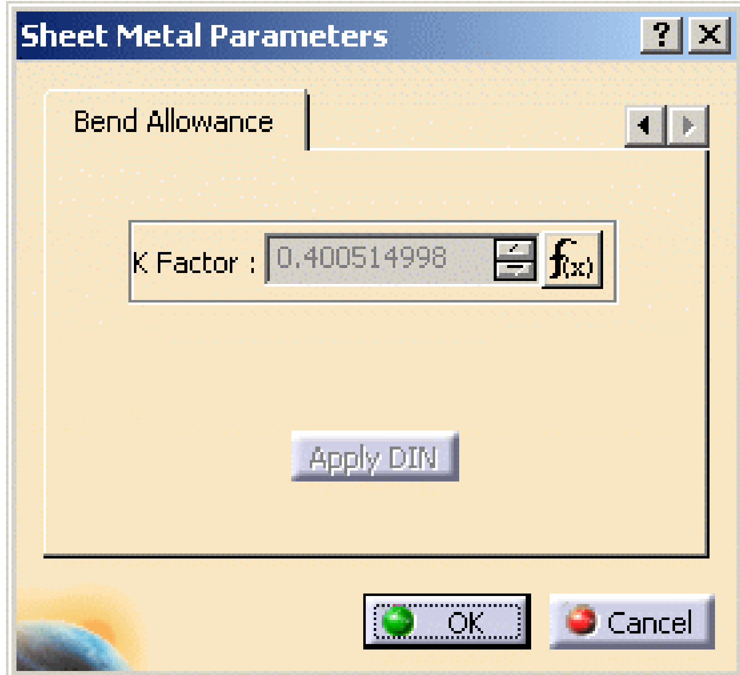


- For better result, you should select the Maximum Bend Extremities option when creating corner relief.
- These parameters are applied to each corner relief created or to be created, except to those with that have been redefined, or the locally defined corner relieves.

Computing the Bend Allowance

This section explains the calculations related to folding/unfolding operations.

1. Click the **SheetMetal Parameters** icon .

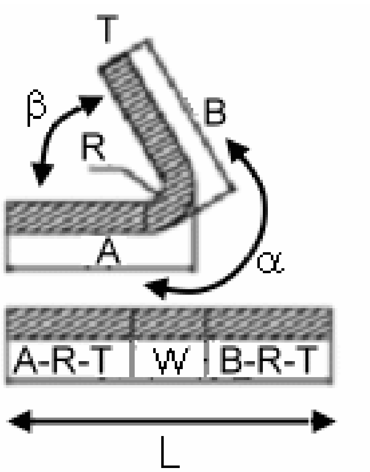


The Sheet Metal Parameters dialog box is displayed.

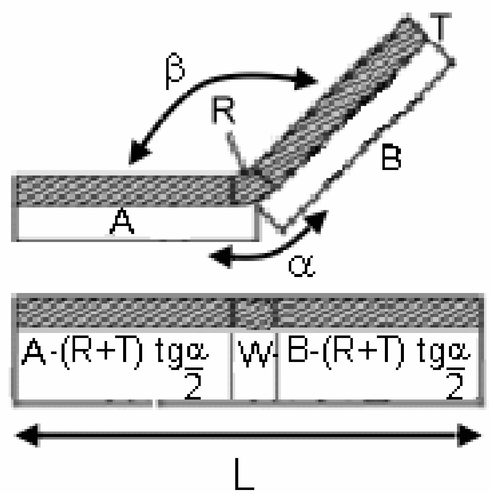
The fourth tab concerns the bend allowance.

- **Bend Allowance**

The bend allowance corresponds to the unfolded bend width.



bend < 90deg



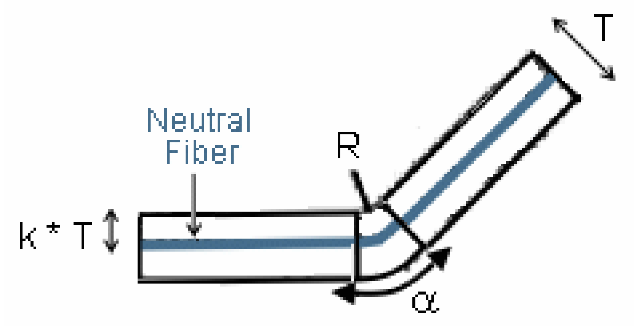
bend > 90deg

L is the total unfolded length
A and **B** the dimensioning lengths as defined on the above figure. They are similar to the DIN definition.

- **K Factor**

Physically, the neutral fiber represents the limit between the material compressed area inside the bend and the extended area outside the bend. Ideally, it is represented by an arc located inside the thickness and centered on the bend axis.

The K factor defines the neutral fiber position:



$$\mathbf{W = \alpha * (R + k * T)}$$

where:

W is the bend allowance
R the inner bend radius
T the sheet metal thickness
α the inner bend angle in radians.

If β is the opening bend angle in degrees:

$$\alpha = \pi * (180 - \beta) / 180$$

When you define the sheet metal parameters, a literal feature defines the default K Factor and a formula is applied to implement the **DIN** standard. This standard is defined for thin steel parts. Therefore the K Factor value ranges between 0 and 0.5.

The DIN definition for the K factor slightly differs.

$$\mathbf{W = \alpha * (R + k' * T/2)}$$

Therefore **k' = 2 * k** and ranges from 0 to 1.

This formula can be deactivated or modified by right-clicking in the K factor field and choosing an option from the contextual menu. It can be re-activated by clicking the Apply DIN button. Moreover, the limit values can also be modified.

When a bend is created, its own K Factor literal is created.
Two cases may then occur:

a. If the Sheet Metal K Factor has an activated formula using the default bend radius as input parameter, the same formula is activated on the bend K Factor replacing the default bend radius by the local bend radius as input.
b. In all other cases, a formula "equal to the Sheet Metal K Factor" is activated on the local bend K Factor.
   This formula can also be deactivated or modified.

- **Bend Deduction**

When the bend is unfolded, the sheet metal deformation is thus represented by the bend deduction **V,** defined by the formula:

L = A + B + V

(refer to the previous definitions).

Therefore the bend deduction is related to the K factor using the following formula:

$$V = \alpha * (R + k * T) - 2 * (R + T) * \tan(\min(\pi/2, \alpha) / 2)$$

This formula is used by default. However, it is possible to define bend tables on the sheet metal parameters. These tables define samples: thickness, bend radius, open angle, and bend deduction. In this case, the bend deduction is located in the appropriate bend table, matching thickness, bend radius, and open angle. If no accurate open angle is found, an interpolation will be performed.

When updating the bend, the bend deduction is first computed using the previously defined rules. Then the bend allowance is deduced using the following formula:

$$W = V + 2 * (R + T) * \tan(\min(\pi/2, \alpha) / 2)$$

When the bend deduction is read in the bend table, the K factor is not used.

Creating Walls

This section explains and illustrates different methods to create walls.

Create a wall from a sketch: use the sketcher to define the profile, and set the material side.

Create a wall tangent to another one: select a profile coincident with an existing wall, and select the wall to which it should be tangent

Create a wall from an edge: select a wall edge, set the height, limits, angle, then the sketch and material sides

Creating Walls from a Sketch

This task shows how to create a wall from a sketch.

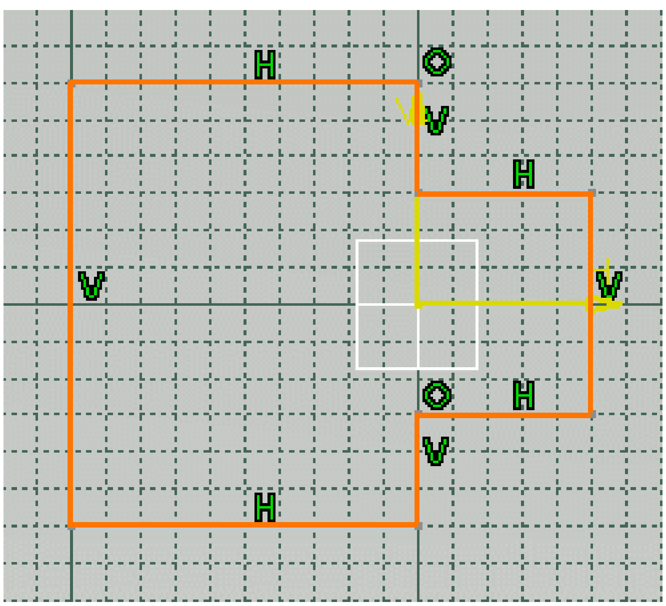
You must be in the Sheet Metal Workbench, with a .CATPart document open, and you must have defined the sheet metal parameters.

Set the sketcher grid to **H = 100mm** and **V = 100mm**, using the **Tools -> Options**, **Mechanical Design -> Sketcher**, **Sketcher** tab.

1. Click the **Sketcher** icon then select the **xy plane**.

2. Select the **Profile** icon .

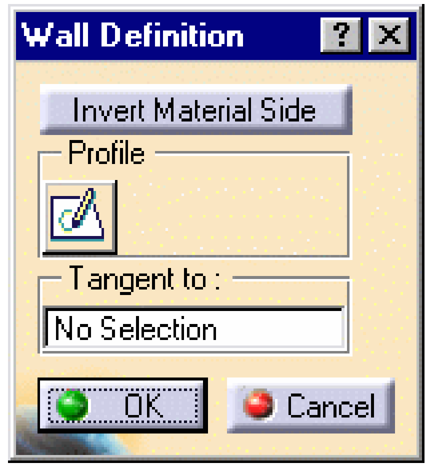
3. Sketch the profile as shown below:



4. Click the **Exit workbench** icon to return to the 3D world.

5. Click the **Wall** icon .

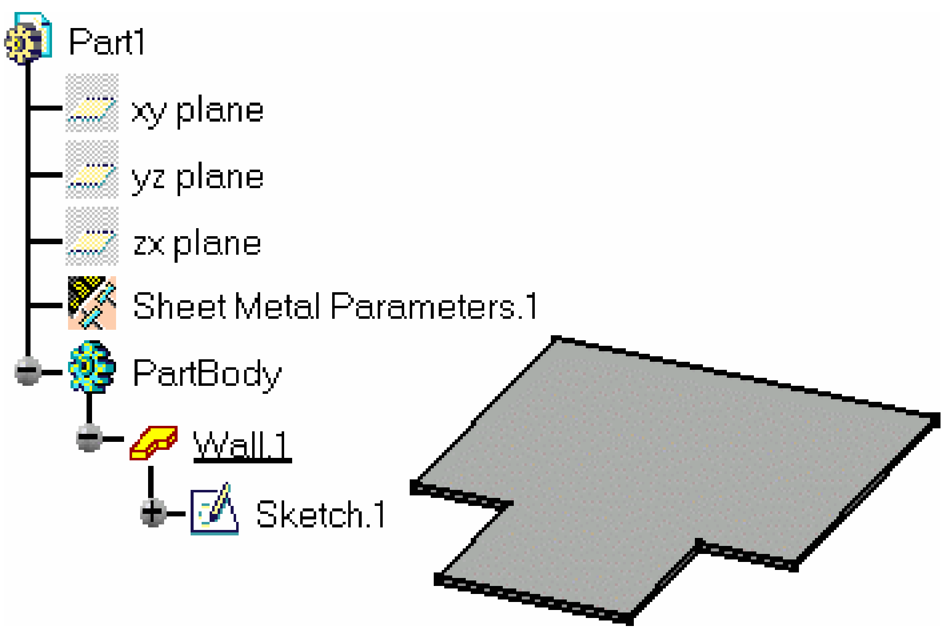
The Wall Definition dialog box opens.



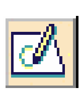
By default, the Material Side is set to the top.

6. Click OK.

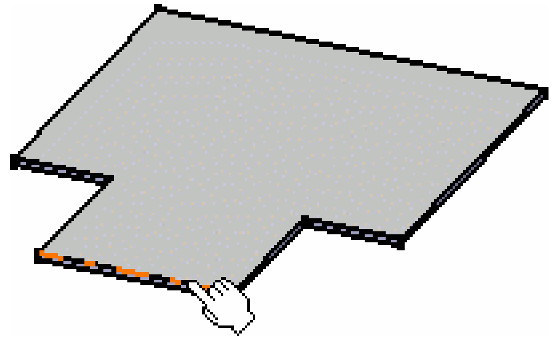
The **Wall.1** feature is added in the specification tree.



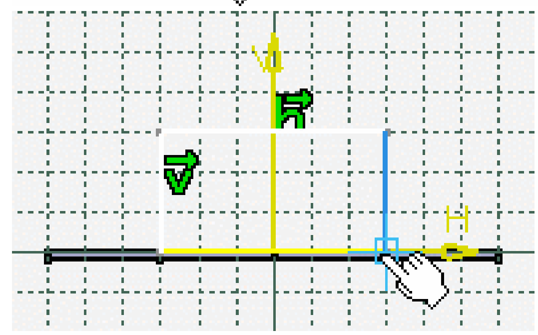
The first wall of the Sheet Metal Part is known as the Reference wall.

- Click the sketcher icon  from the Wall Definition dialog box, if you wish to directly edit the selected sketch. When exiting the sketcher, you then go back to the wall creation step, without having to reactivate the Wall icon.

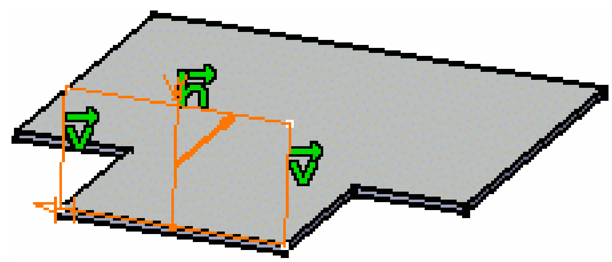
This is also very useful if you have selected an edge from a wall and clicked the **Wall** icon .



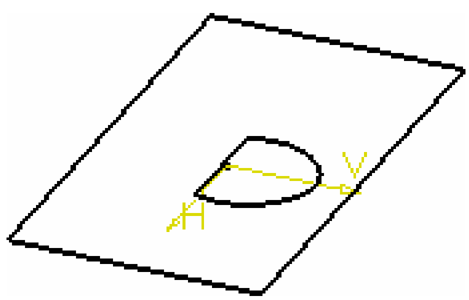
In this case, the sketcher is automatically activated and the plane defined as being the selected edge's plane.



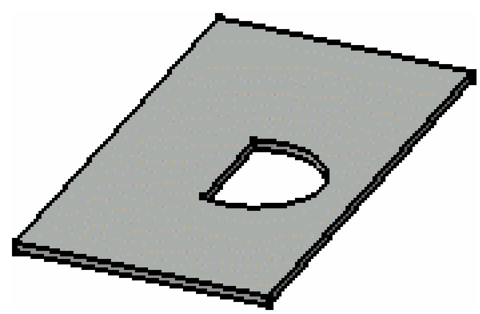
You can then directly draw a sketch, then exit the sketcher and return to the wall creation step.



- You can directly create a wall with a hole, by selecting a sketch with an inner profile (the profiles must not intersect):



Sketch with inner profile



Resulting wall

Note however, that the emptied area is part of the wall and is not a separate cutout that can be edited.

Creating Tangent Walls

This task shows how to create a wall tangent to a planar part of another wall or of a flange. This capability will then allow this tangent wall to be seen when unfolding the part, even though there is no bend linking it to its tangent support, provided this support is unfoldable too.

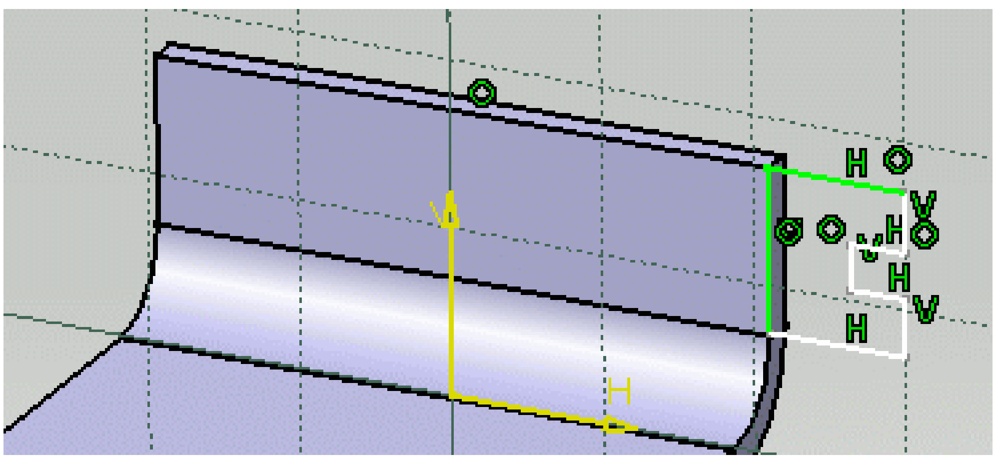
Open the TangentWall1.CATPart document from the samples directory.

1. Select a face of an existing wall and click the Sketcher icon .
   Here we selected the flange's planar face.

2. Select the **Profile** icon and sketch the profile as shown below.

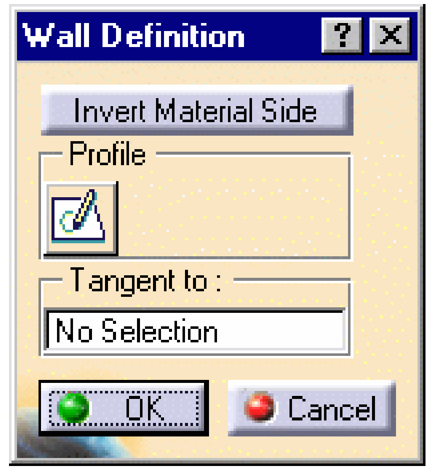
You can also select Sketch.2 from the specification tree.

3. Using the Constraint Defined in Dialog box icon, set coincidence constraints between the edges where the support and sketch are to coincide.



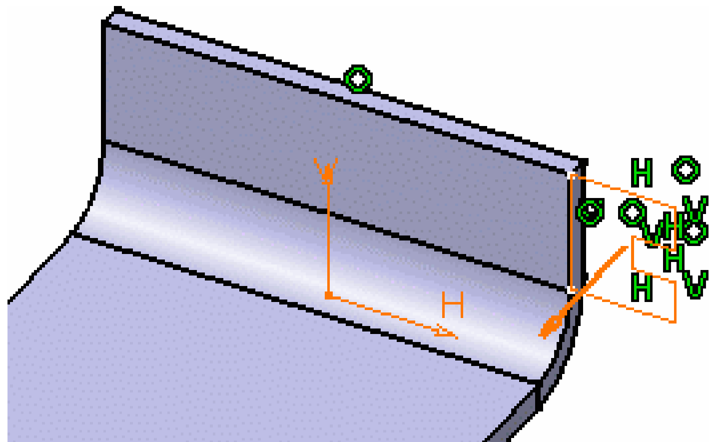
4. Click the **Exit workbench** icon to return to the 3D world.

5. Make sure the sketch is still active, then click the **Wall** icon .



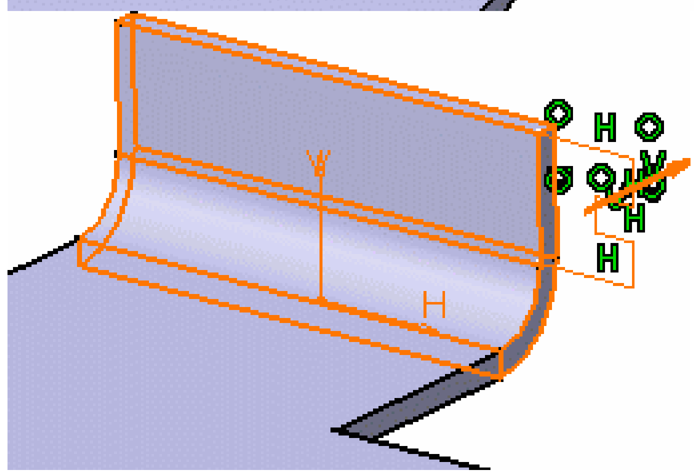
The Wall Definition dialog box opens.

Note the orientation of the wall to be created.



6. Click inside the **Tangent to** field, then select the wall to which the new wall has to be tangent.

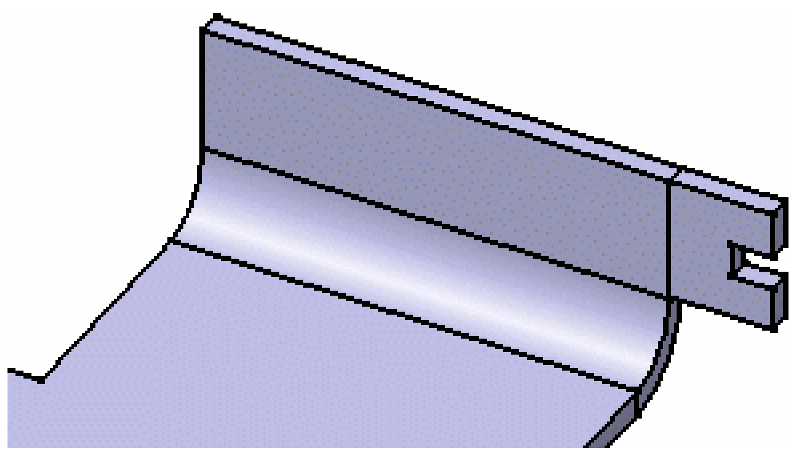
Here, you need to select the planar face of the flange.

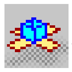


Note that the orientation automatically changes to conform to the material orientation already defined on the support wall.

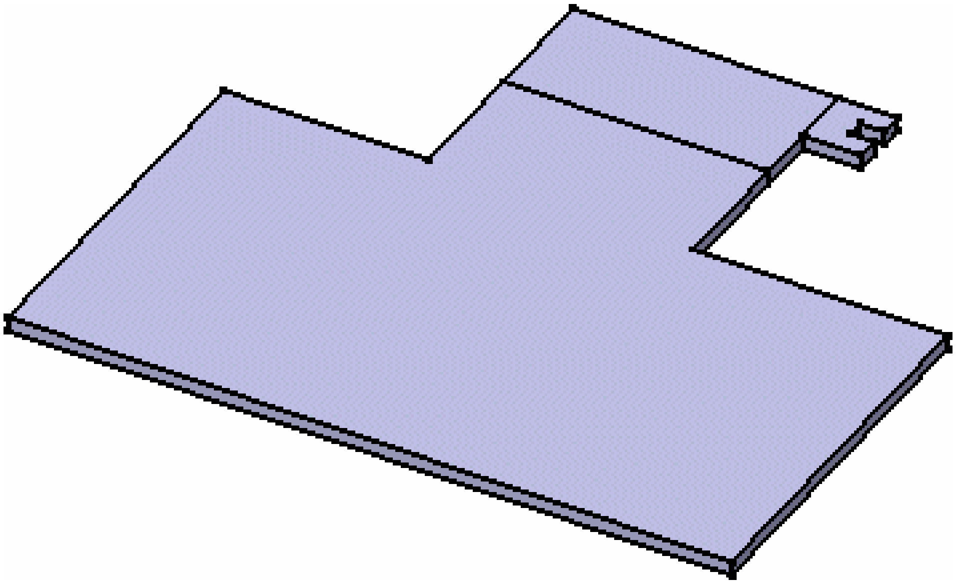
7. Click **OK**.

The wall is created and a **Wall.xxx** feature is added in the specification tree.



8. Click the **Unfold** icon .

The tangent wall is unfolded as a wall linked by a bend to another wall, would be.



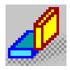
If you want to create the wall on a wall that cannot be unfolded, the system issues a warning and prevents you from creating the tangent wall.

Creating Walls From An Edge

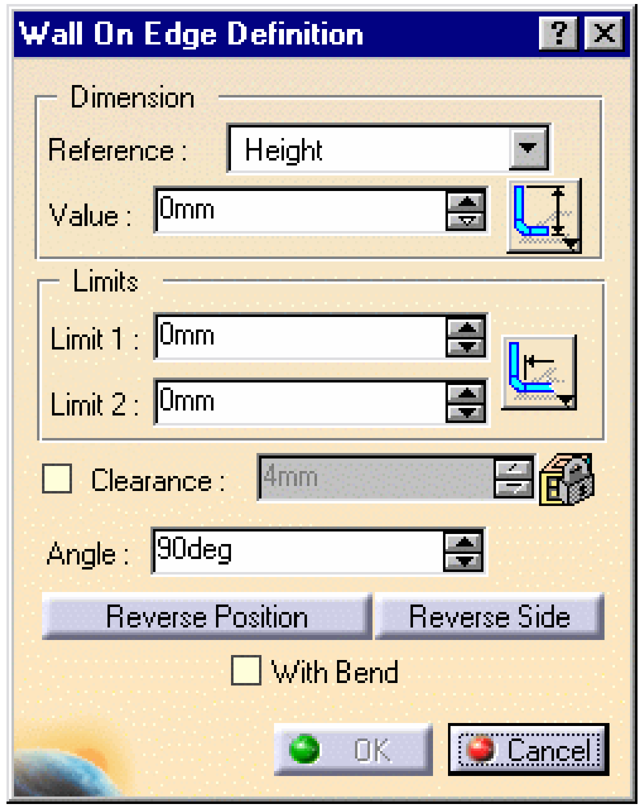
This task shows how to create walls from edges of an existing wall. This function is used to create a box in an easy and quick way from an existing reference wall.

At least one wall must already exist.

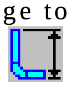
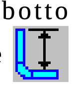
Open the Wall1.CATPart document from the samples directory.

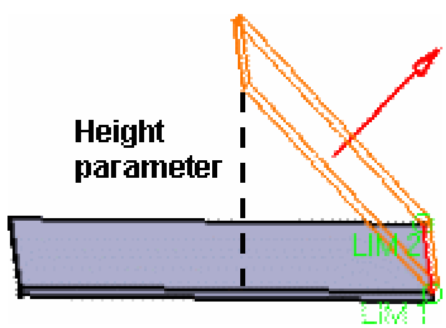
1. Click the **Wall on Edge** icon  and select an edge of an existing wall.

The Wall On Edge Definition dialog box is displayed together with a preview of the wall.

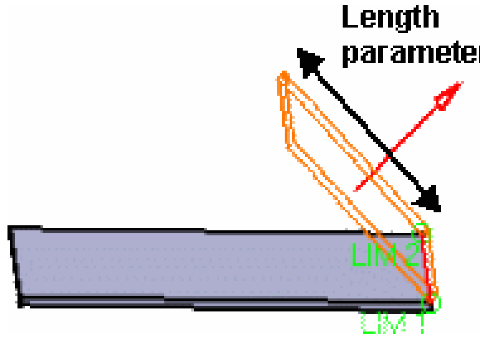


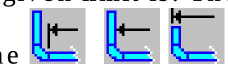
2. Define the type of wall to be created by specifying the Reference, that is:

- the **Height** of the wall: that is the orthogonal projection from the top of the wall on edge to the reference wall. Select the  icon to define the height of the wall from the bottom of the reference wall or the  icon to define the height of the wall from the top of the reference wall.



- the **Length** of the wall: that is the absolute value of the wall on edge without bend.

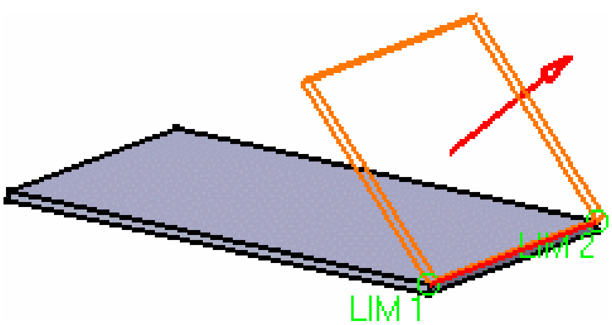


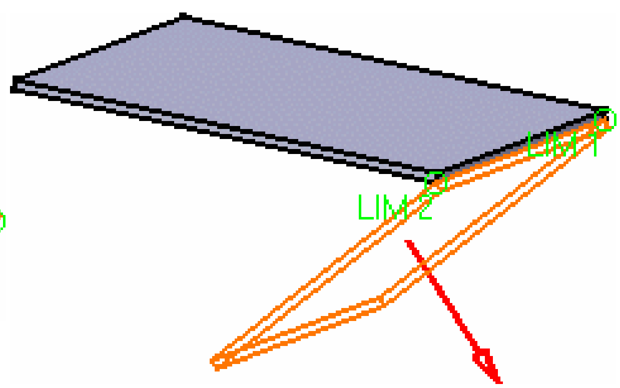
- the limits of the wall: Limit 1 and Limit 2. These texts only indicate on which side a given limit is. They are not precisely on the limit spots. The actual locations of the limits are defined with the  icons and an input distance that is taken into account respectively from the inner side of an existing bend, the inner side of an existing wall or the outer side of an existing wall.

- the angle of the wall: by default it is perpendicular to the plane containing the edge. You may modify it according to your needs. it is updated dynamically on the screen.

This preview gives information about:

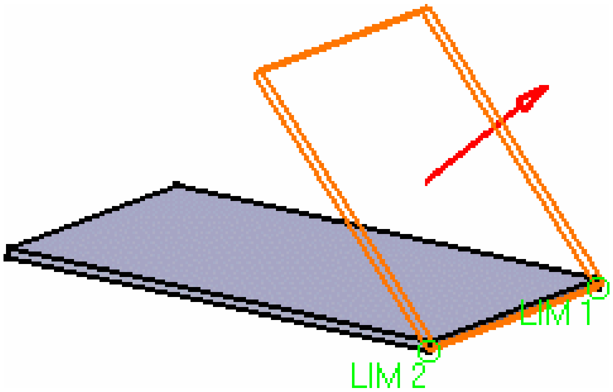
- the Sketch Profile: by default, if you pick an edge on the top of the reference wall, the direction of the wall is upwards, if you pick an edge on the bottom of the reference wall the direction of the wall is downwards.





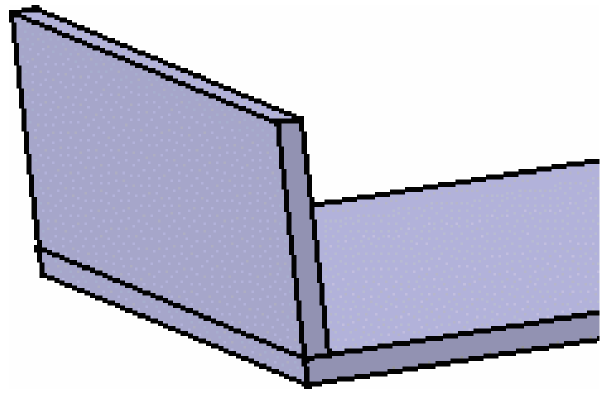
Preview with top edge selected *Preview with bottom edge selected*

You can invert the sketch's position, and therefore the wall's, using the **Reverse Position** button.

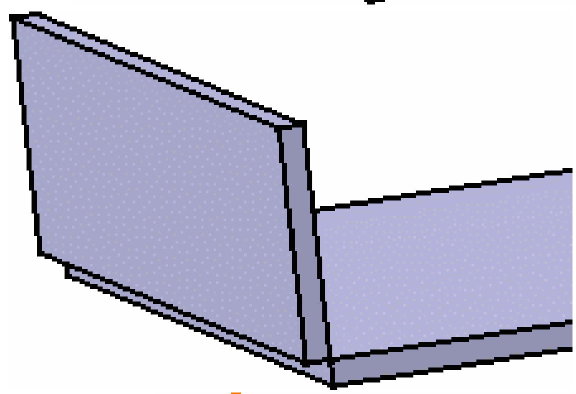


Preview with bottom edge selected and sketch profile inverted

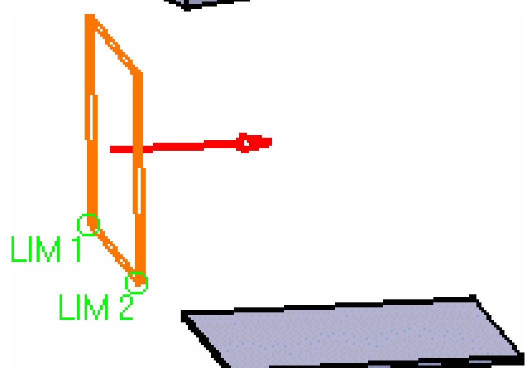
- the material side: by default the material side is proposed so as to keep a continuity with the reference profile.



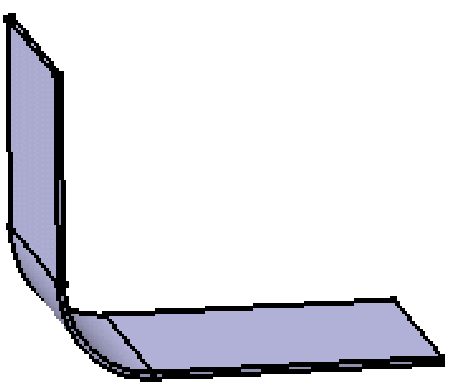
However, you can change it by clicking the red arrow or the **Reverse Side** button.



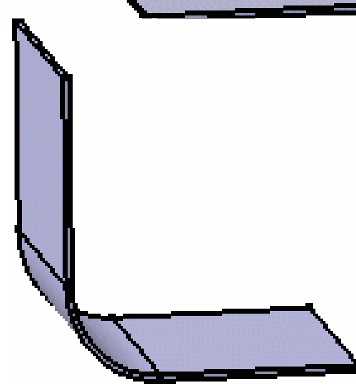
3. Check the **Clearance** option to offset the wall on edge from the selected edge.



The entered value is the radius of the bend on this edge.



Wall on edge with clearance



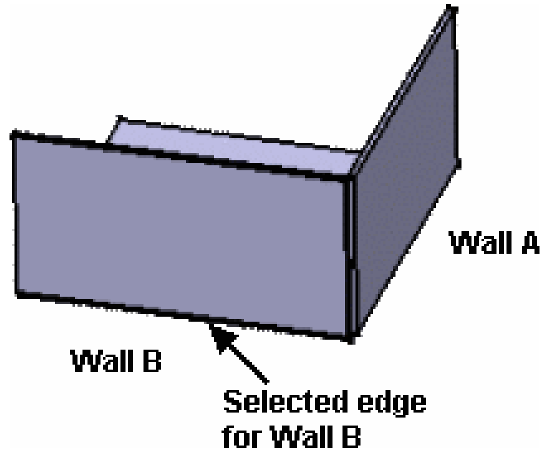
Wall on edge without clearance

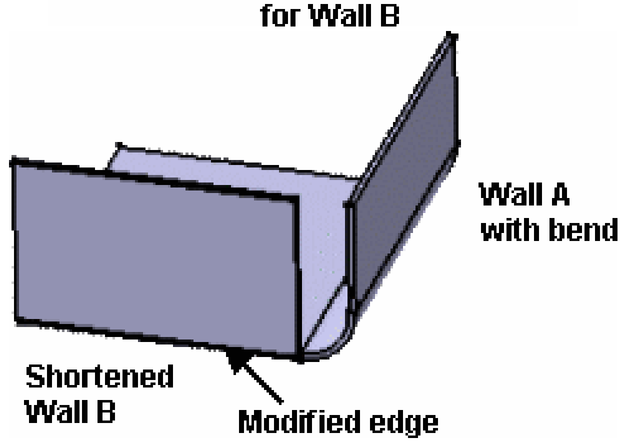
Note that the reference wall remains unchanged when changing the bend radius value. It is the Wall on Edge's length that is affected.

4. You can also choose to create the wall with or without a bend by checking the **With Bend** option.

- If there is no wall from which a limit can be computed, the reference element is the edge of the reference wall.

- A wall on edge is defined by the selected edge (reference edge). When the reference edge is modified, by adding any feature that shortens the edge (a bend to an adjacent wall on edge or a cutout for example) the wall on edge based on this reference edge is recomputed.
  To avoid this you may:
  - create the bend manually on the wall modifying the edge used as the reference to create the other wall
  - reorder the creation of walls to postpone the creation of the modifying feature

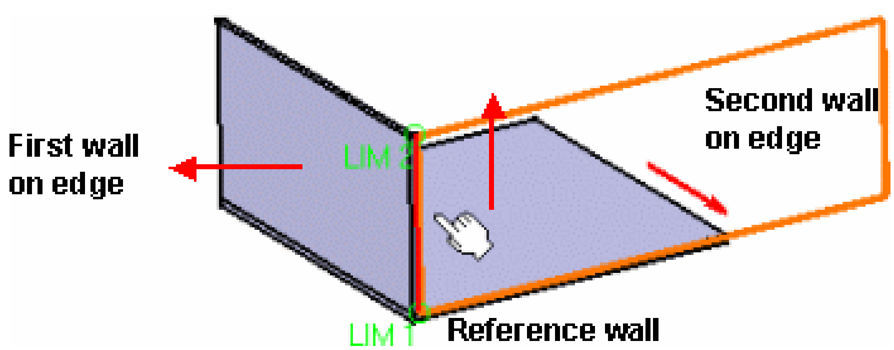




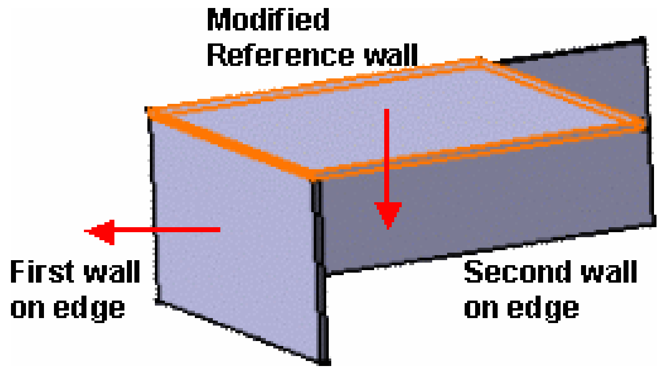
- Both limits are computed with the same reference icon.
- The bend is not previewed, even if the option **With Bend** is checked. However it will be created.
- The selected options are modal and will be proposed to create the next wall.

- Walls on edge being contextual features, if you break the profiles continuity by inverting the material side of a wall, you may have to manually re-specify all features based upon the modified one, even if they are not directly connected to the modified wall, in order to update the part afterwards.

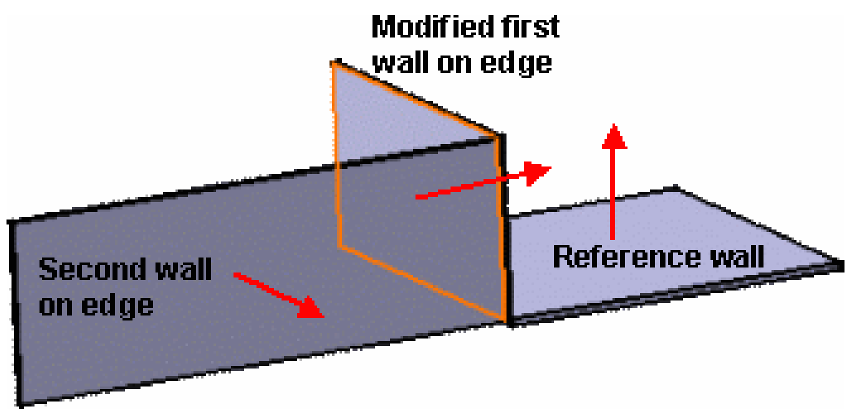
This is the case when creating a wall on edge from another wall on edge, for example:



- If you invert the material side of the reference wall on which the first wall on edge has been created, both walls on edge are relocated (as if you were flipping the geometry):



- If you invert the material side of the first wall on edge (and not the material side of the reference wall), the second wall on edge is relocated. Indeed, its specification being relative to the first wall on edge, when its input data (i.e. the edge selected on the first wall on edge) is modified, the second wall on edge is rebuilt at a new location.



5. Click **OK** in the **Wall On Edge Definition** dialog box.

A WallOnEdge.x element is created in the specification tree.

- A WallOnEdge.x element can be edited: double-click it in the graphic area or in the specification tree to display its creation dialog box and modify the parameters described above, including the edge from which it is created. However, the sketch of a wall on edge cannot be edited directly.
- Would you need to edit the sketch of a wall on edge, you have to isolate it first. See Isolating Walls.
- You can cut or copy and paste a wall on edge.

- If you cut and paste a wall on edge with children elements, these children elements are lost. This may result in update errors.
- You cannot undo an **Isolate** action after having modified the wall.
- Isolating a wall on edge erases all updating data.

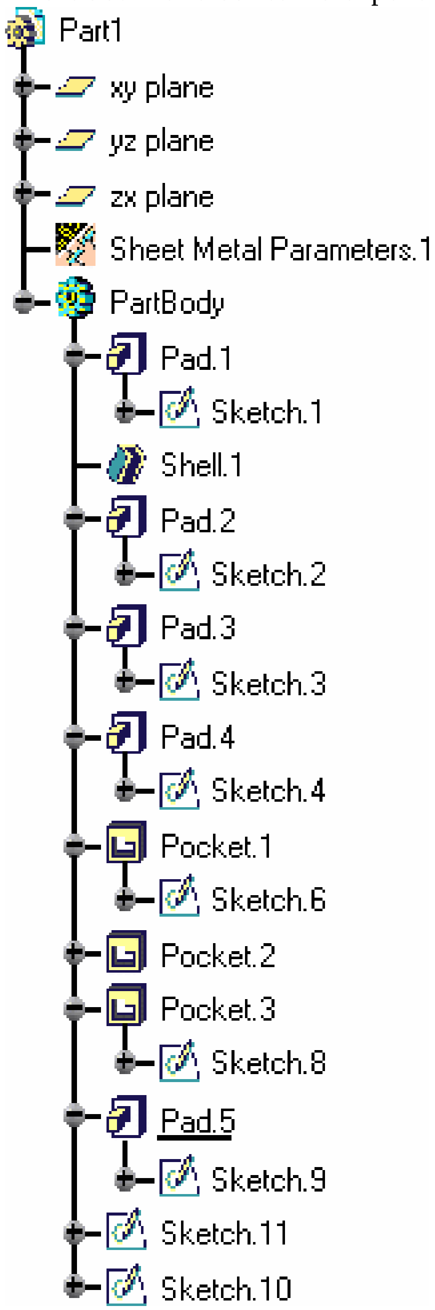
Recognizing Walls From an Existing Part

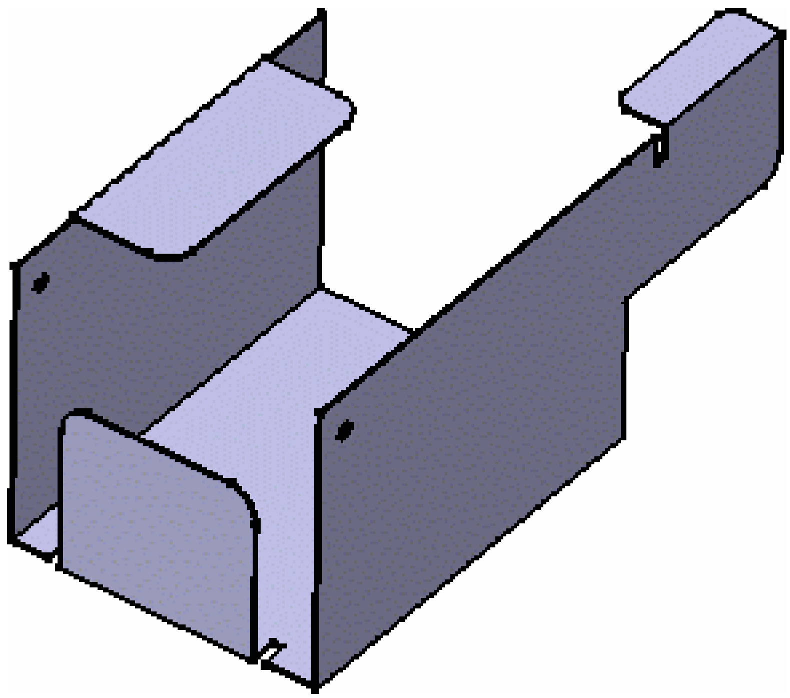
This functionality is only available with SheetMetal Design.

This task illustrates how to create a sheet metal part using an existing Part, that is recognizing the thin part shapes of the part as created using the Part Design workbench or from a CATIA Version 4 Solid for example.

Open the WallScenario1.CATPart document from the samples directory.

The document contains a part created in the Part Design workbench and it looks like this:

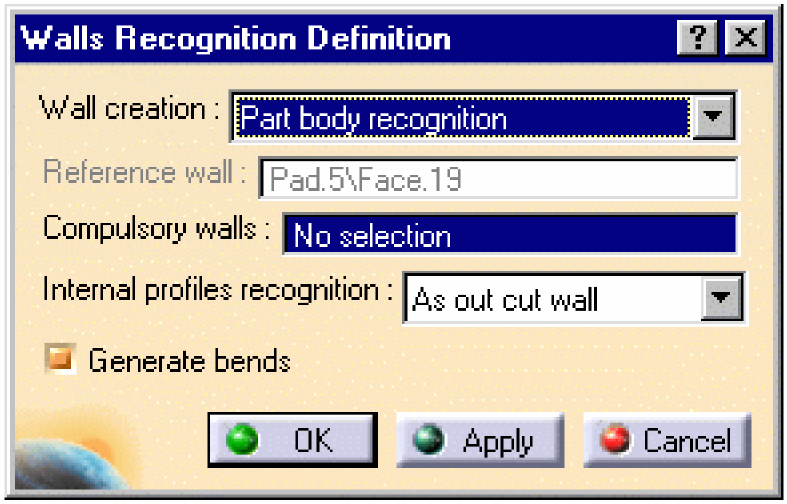




1. Click the **Walls Recognition** icon .

2. Click any face to be the reference wall.

The Walls Recognition Definition dialog box is displayed.



3. Choose the **Wall creation** mode:

- **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.

The **Reference wall** is indicated in the Walls Recognition Definition dialog box for information only (it is grayed out).

4. Select faces as the **Compulsory walls**.

   These 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.

5. Set the Internal profiles recognition mode:

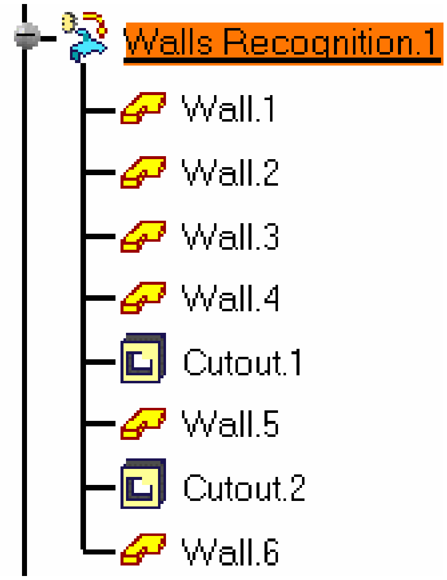
- **As cut out wall**: generates walls with inner profiles (no cutout feature is generated)
- **One cut out by wall**: regardless of how many pockets there are on a face of the solid, only one cutout feature is generated per wall
- **One cut out by profile**: for each inner profile on the sketch-based 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 SheetMetal features.

The **Generate Bends** check button allows the automatic creation of bends as the walls are being created, wherever applicable.

6. Click Apply.

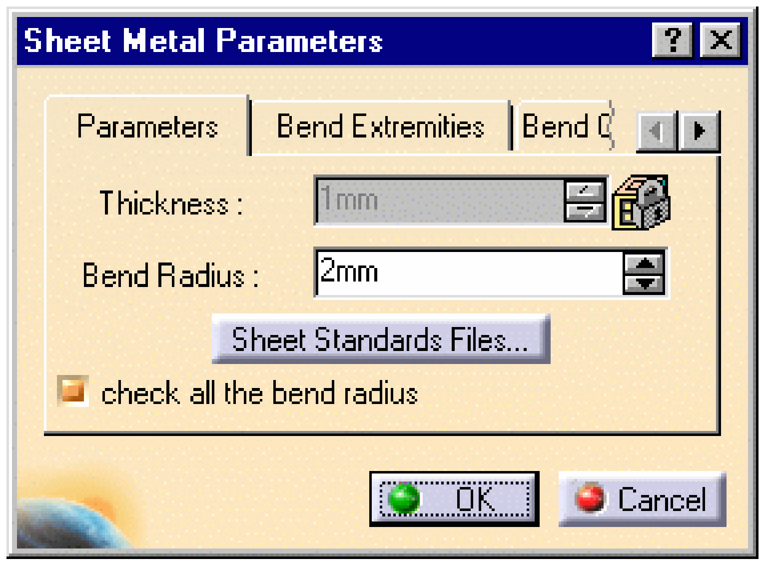
The walls are generated from the Part Design geometry.

The **Walls Recognition.1** feature is added to the tree view.

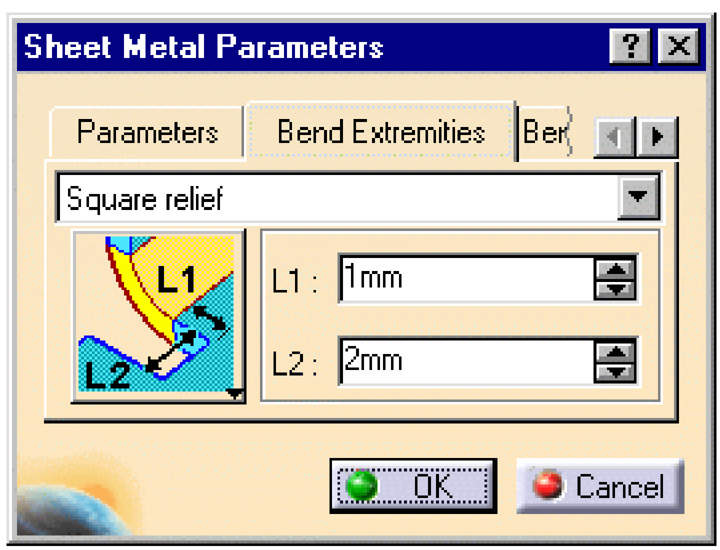


At the same time, the sheet metal parameters are created, deduced from the Part geometry.

7. Select the icon  to display the sheet metal parameters:



- the **Thickness** is equal to 1mm
- the **Bend radius** is twice the thickness value
- the **Bend Extremities** field is set to Square relief.



You can modify a few of these parameters.
The **Thickness** parameter cannot be modified because it is based, like the bend extremities and radius, on the initial solid geometry . However you can modify these parameters (bend radius and bend extremities) to be taken into account for sheet metal features other than the "recognized" ones.
The bend allowance, being used to unfold the part, and the corner relief affect all features, and therefore can be edited even for "recognized" features.

8. Click OK in the Sheet Metal Parameters when all parameters have been redefined where needed.

The solid is now a Sheet Metal part. All the features are displayed in the specification tree.

- Once the solid has been converted to a Sheet Metal part, you can create bends as with any other Sheet Metal part, or modify/add Sheet Metal features to complete the design.
- Uncheck the **Generate Bends** button, if you do not wish bends to be created automatically.

Extruding

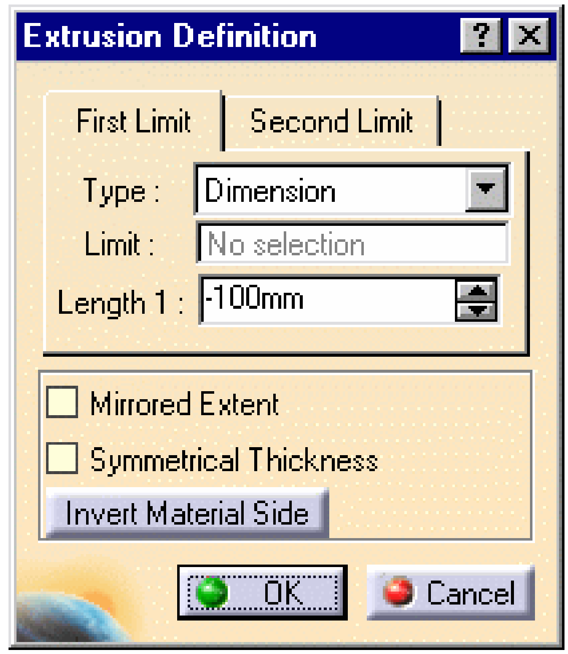
This task explains how to create a wall by extrusion.
You can extrude sketches composed of lines, circles, projection of lines, and projection of circles.

You must have defined the Sheet Metal parameters.
A model is available in the Extrude1.CATPart from the samples directory.

1. Click the **Extrusion** icon .

2. Select a sketch.

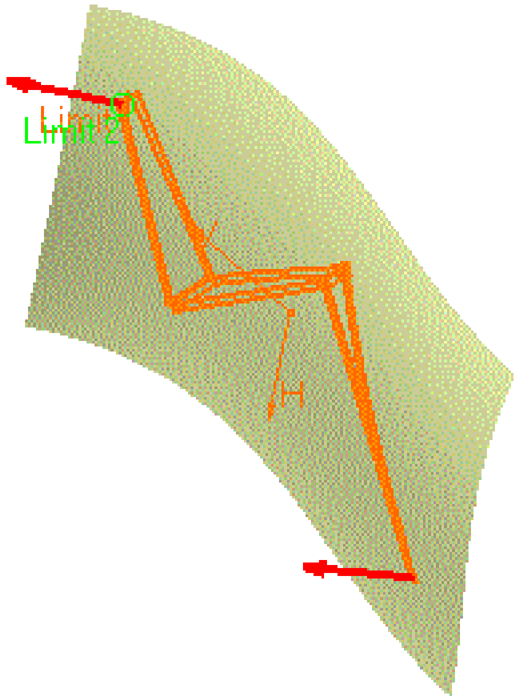
The Extrusion Definition dialog box is displayed.

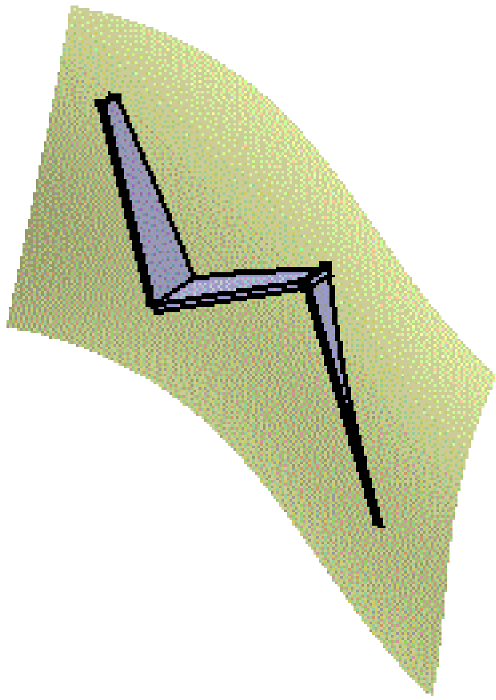


Several types of extrusion are available:

- **Dimension** : the requested input data are a sketch and a dimension,
- **Up to plane** or **Up to surface**: a plane or a surface are input as limit to the extrusion. These functions are used to create walls that are not rectangular.

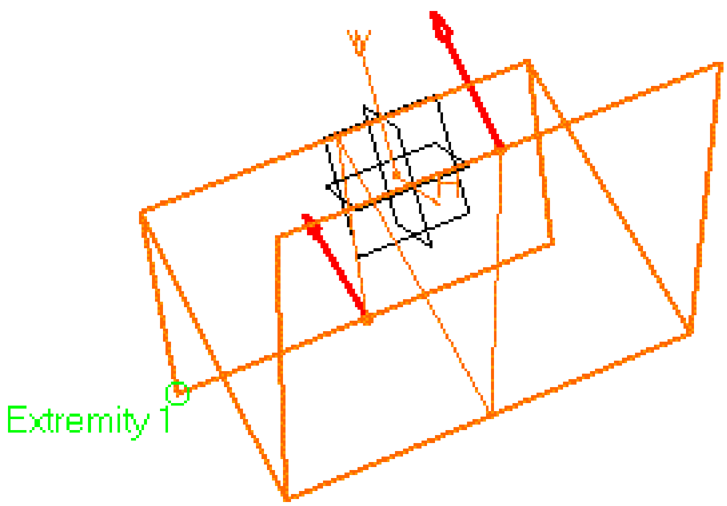
3. Edit the **Length1** and **Length2** to set both extremities, for option **Dimension**.





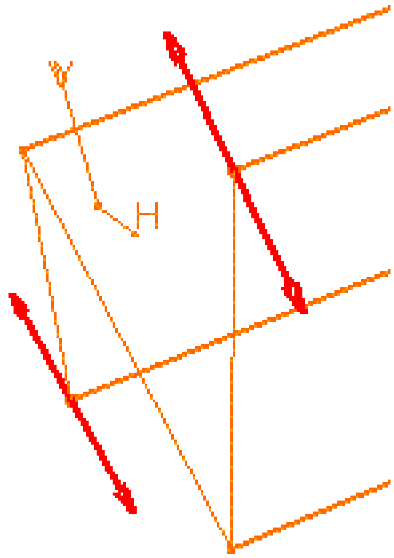
4. Define the options as needed:

- Check the option **Mirrored extent** to extent the material on both sides of the sketch. In that case, only **Length1** can be edited.

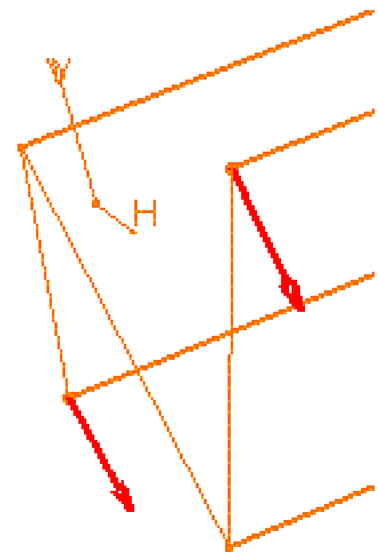


This option is only available if the type is set to **Dimension**.

- Check **Symmetrical thickness** to create thickness inside and outside the sketch.



- Push **Invert Material Side** to invert the direction of the creation of the material.

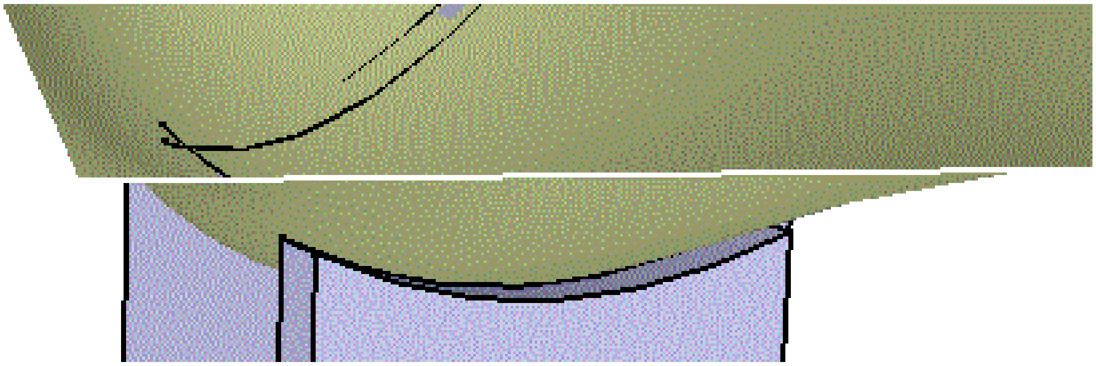


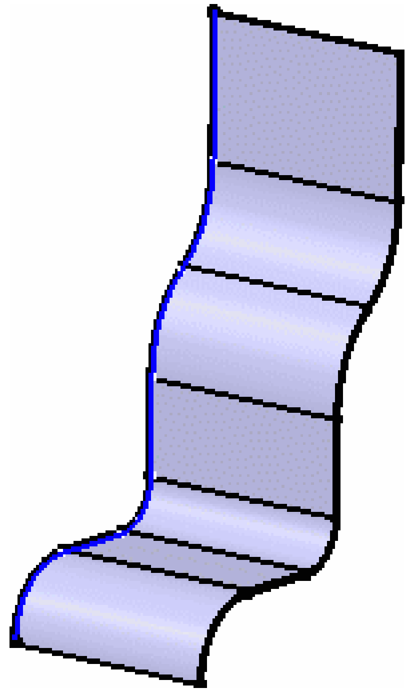
5. Click **OK**.

The walls corresponding to the selected sketch are created according to the specified options, and added to the specification tree.

- When the extrusion is the first Sheet Metal feature of the Part, the reference wall is the first wall created based on the first segment of the sketch.

- For option **Up to Surface**, while the wall end that is limited by the surface has the shape of the surface, its thickness does not fit the surface. It is a "rectangular" polygon defined by the first edge that comes into contact with the surface.





- Such an extrusion can also be performed on a sketch made of lines and arcs of circle, provided there are no tangency discontinuities between the different elements.
  However, in this case:
  - the **Up to plane** or **Up to surface** capabilities are not available,
  - you cannot isolate such an extrusion,
  - if the element of the extruded sketch connected to the part is an arc of circle, the extrusion will not display in the unfolded view. To avoid this, prefer to create a User Defined Flange or remove the arc of circle of the extruded sketch and create a bend to connect the extrusion to the part.

- Extrusion walls can be edited and/or isolated.

The sketch may not be closed.

Isolating Walls

This task explains how to isolate a wall. This is possible in two cases:

- after having created walls by extrusion (see Extruding)
- after having created a wall on edge (see Creating Walls from an Edge).

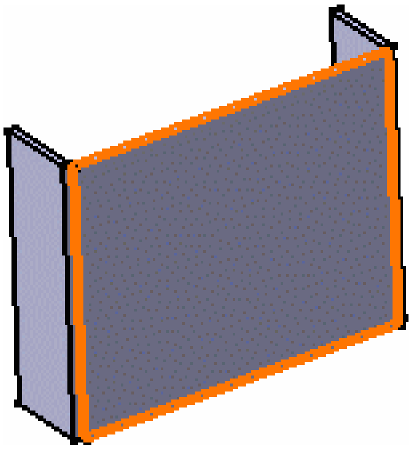
You must have defined the Sheet Metal parameters.
A model is available in the Extrude2.CATPart from the samples directory.

Isolating Extruded Walls

1. Right-click the Extrusion.1 feature and choose the **Extrusion.1 object -> Isolate** contextual menu item.

The Extrusion Isolate dialog box is displayed.

2. Select one of the wall of the extrusion to be isolated.

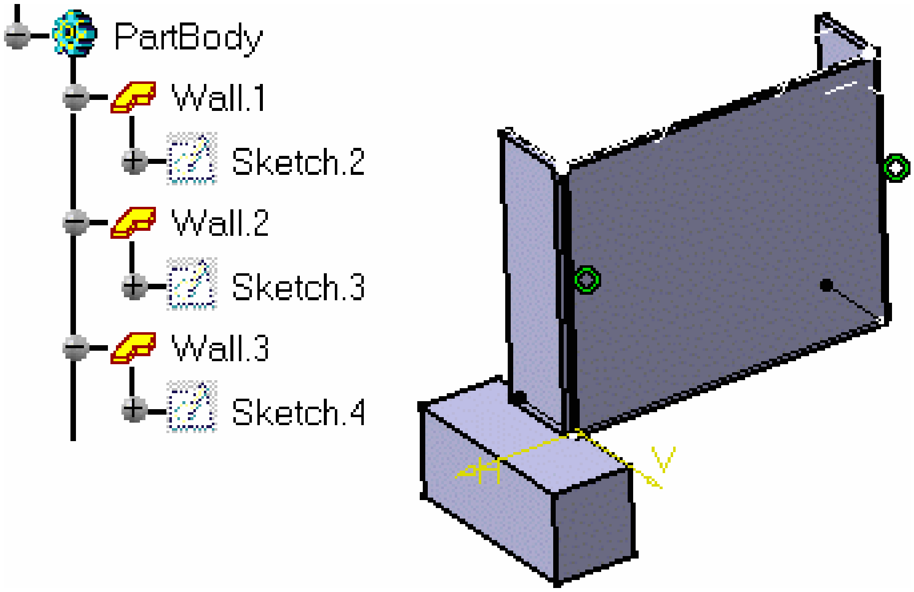


The selected wall is highlighted in the geometry. This wall is the reference wall, meaning that it can be modified and the other walls will take the modification into account. On the other hand if the other walls are modified the reference wall is an anchoring wall, and modifications will be made around it.

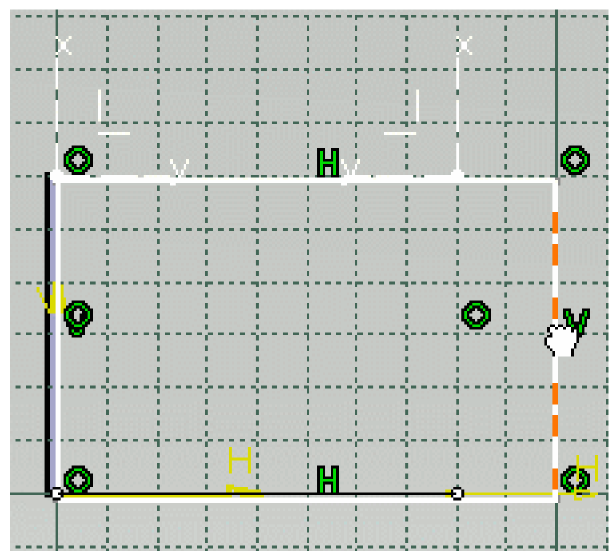
The Extrusion Isolate dialog box is updated.

3. Click **OK** in the dialog box.

The walls of the extrusion have been isolated, each with its own sketch. Yet coincidence constraints are automatically generated between the isolated walls.

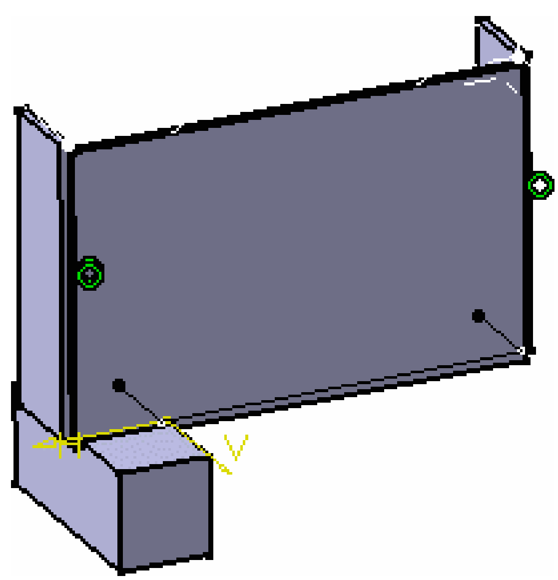


The extrusion's initial sketch is retained (Sketch.1 in the example above).



4. Double-click the sketch of the reference wall (here Sketch.3) and modify it by increasing its length.

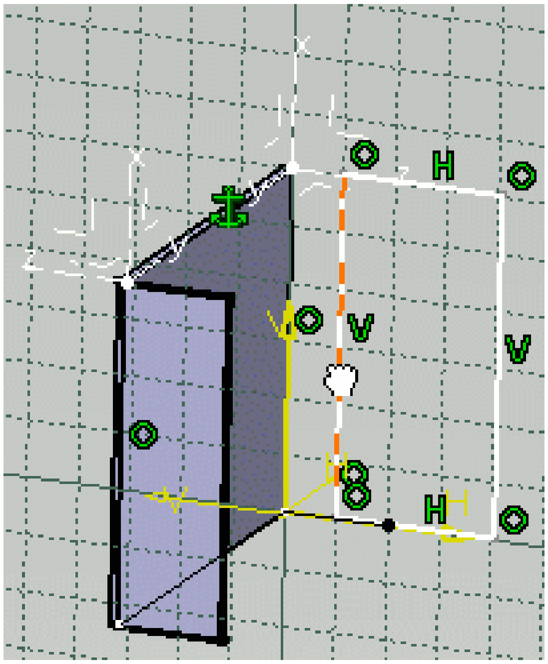
5. Exit the Sketcher using the **Exit** icon .



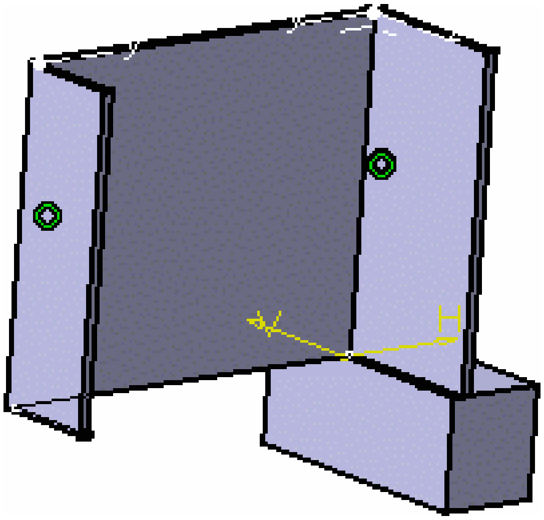
The Part is updated.

You can note that the wall that was adjacent to the pad, now lies on it, as it is the reference wall that was modified.

However, had you modified the sketch of the wall lying on the pad (Sketch.4), moving it further away from Wall.2 as shown here to the right, the updated pad would not take the gap between the walls into account.



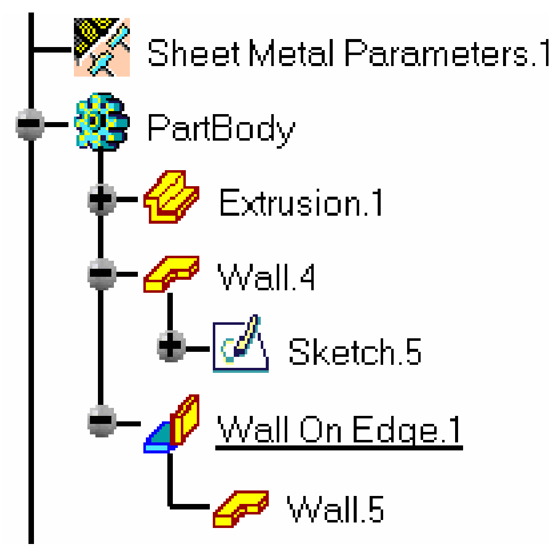
The resulting part looks like this (Wall.3 has been modified but still coincides with Wall.2):

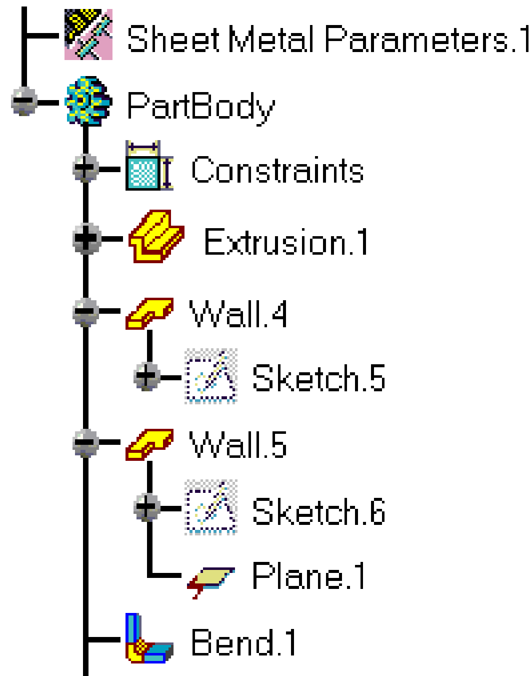


Isolating Walls on Edge

1. Right-click the wall on edge and choose the **Wall On Edge** contextual menu item.

The wall on edge is then changed to a standard wall, as you can see from the specification tree.

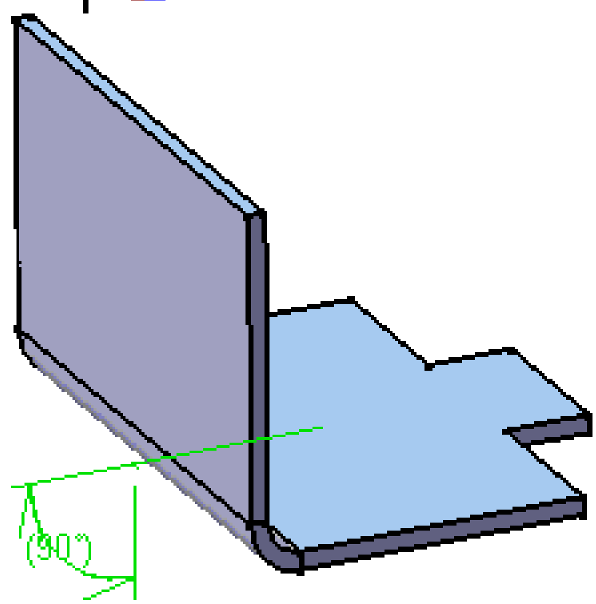




You can then edit its sketch if needed.

In the present case, the wall on edge had been created with a bend. Therefore when isolating this wall from the reference wall, the bend is created as a separate feature that can be edited as well.

The angle value between the two walls is displayed for edition.

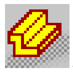


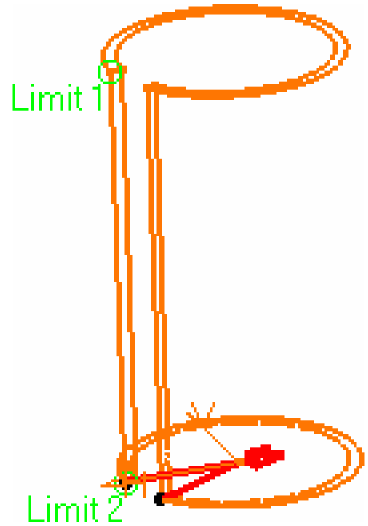
- You cannot undo an **Isolate** action after having modified the wall.
- Isolating a wall on edge erases all updating data.

Creating Rolled Walls

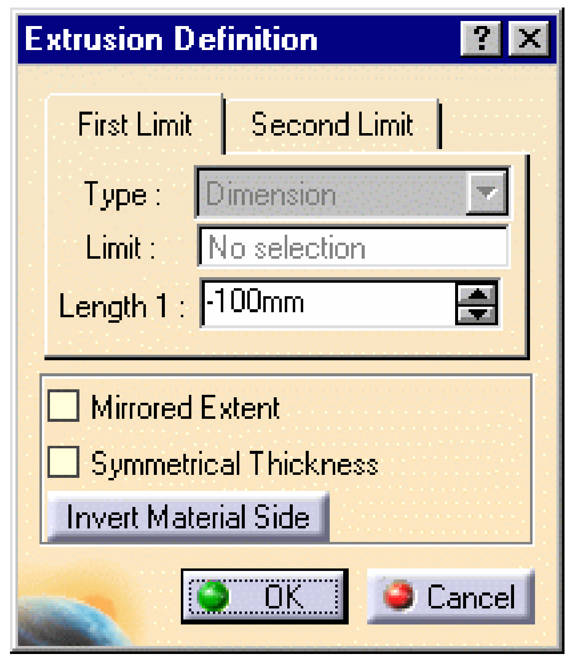
This task shows how to create rolled walls (such as pipes, open pipes with flange, etc.).

You must have defined the Sheet Metal parameters, and have a sketch available, in the form of an circular arc.
The part is available in the Rolledwall1.CATPart from the samples directory.

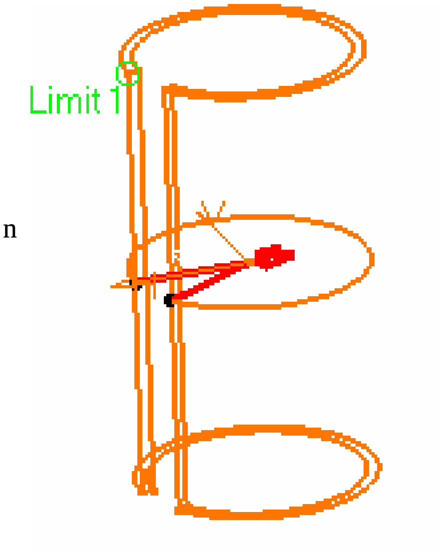
1. Click the **Extrusion** icon .
2. Select the circular sketch.



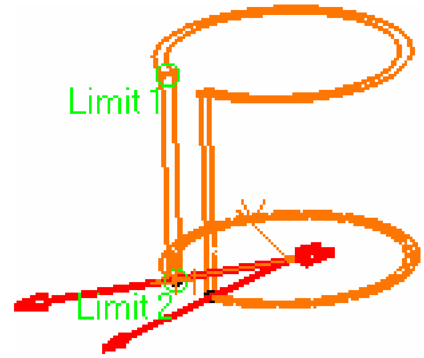
3. Make sure the type is set to **Dimension**.
4. **Length 1** and **Length 2** indicate the location of **Limit 1** and **Limit 2**.



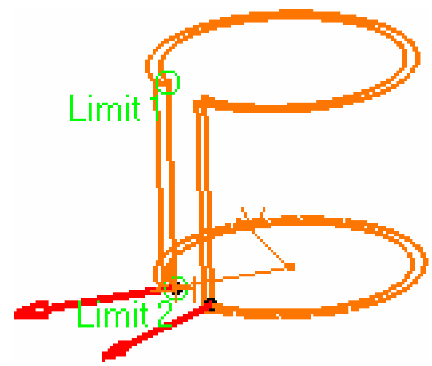
5. Define the options as needed (the length being down to -50 mm):



- Check the option **Mirrored extent** to extend the material on both sides of the sketch. In that case, only **Length1** can be edited.

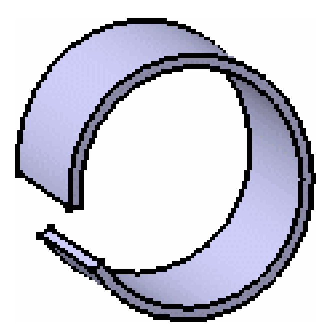


- Check **Symmetrical thickness** to create thickness inside and outside the sketch.



- Click **Invert Material Side** to invert the direction of the creation of the material.

6. Click OK in the dialog box to create the rolled wall:



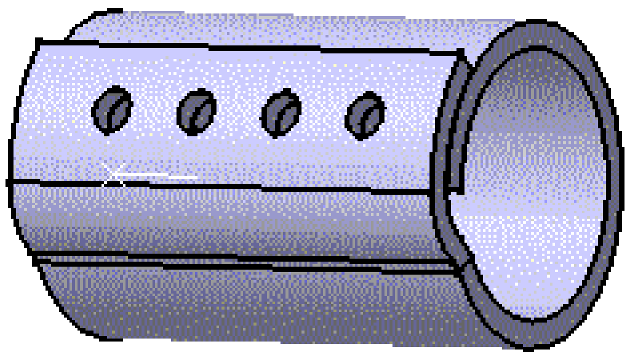
The rolled wall is a particular extrusion:

- the input sketch is either a circular arc or a closed circle,
- the creation type is always **Dimension**.

The sketch may be open. In that case, you can define where the opening should be.

The sketch may be closed. In that case, you have no control on the opening location.

The only operations you can combine with a rolled wall in a Sheet Metal model are flanges and cutouts.



No other elements (standard wall, bend,...) are allowed.

Creating Bends on Walls

This section explains and illustrates different methods to create bends on walls. Bends can only be created between walls, whether manually or automatically, and not between any other Sheet Metal features, such as stamps for example.

Manually create bends from walls: select the two walls, set the bend radius value, the bend extremities, and specify the use of corner relief

Generate bends automatically: select the part, then a reference wall

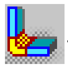
Create conical bends: select the part, and choose a reference wall

Create bends from a line: select a sketch, set the creation mode and limiting option, set the radius and angle in relation to the selected sketch.

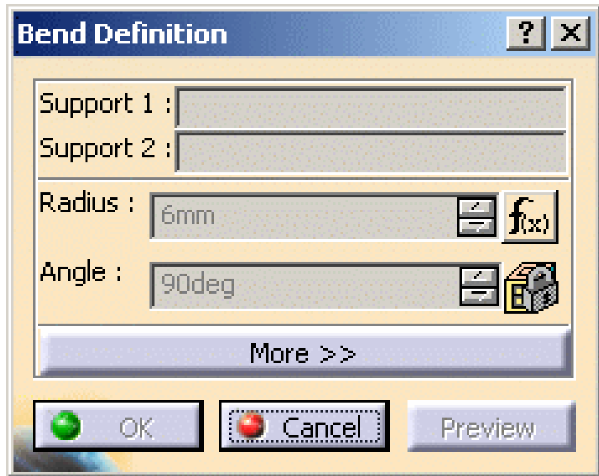
Manually Creating Bends from Walls

This task explains two ways to create bends between walls in the Sheet Metal part.
These bends can be created on non-connex walls, and with a constant radius value.

Open the BendExtremities01.CATPart document from the samples directory.

1. Select the **Bend** icon .

   The Bend Definition dialog box opens.

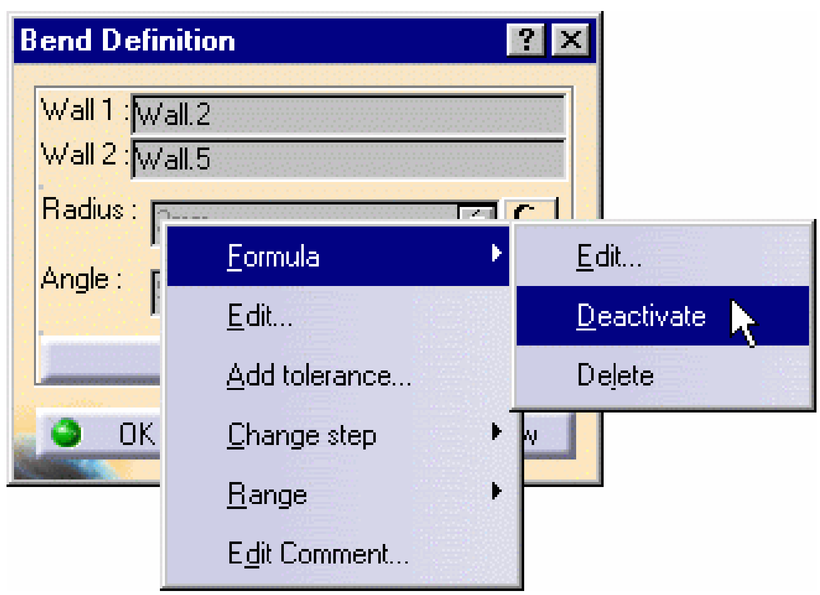


Note that the **Radius** field is in gray because it is driven by a formula when editing the sheetmetal parameters: at that time, you cannot modify the value.

2. Select **Wall.2** and **Wall.5** in the specification tree.

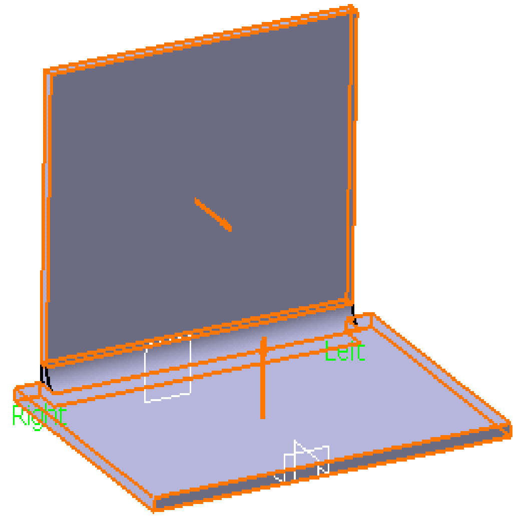
   The Bend Definition dialog box is updated.

3. Right-click the **Radius** field and select **Formula -> Deactivate** from the contextual menu to change the value.

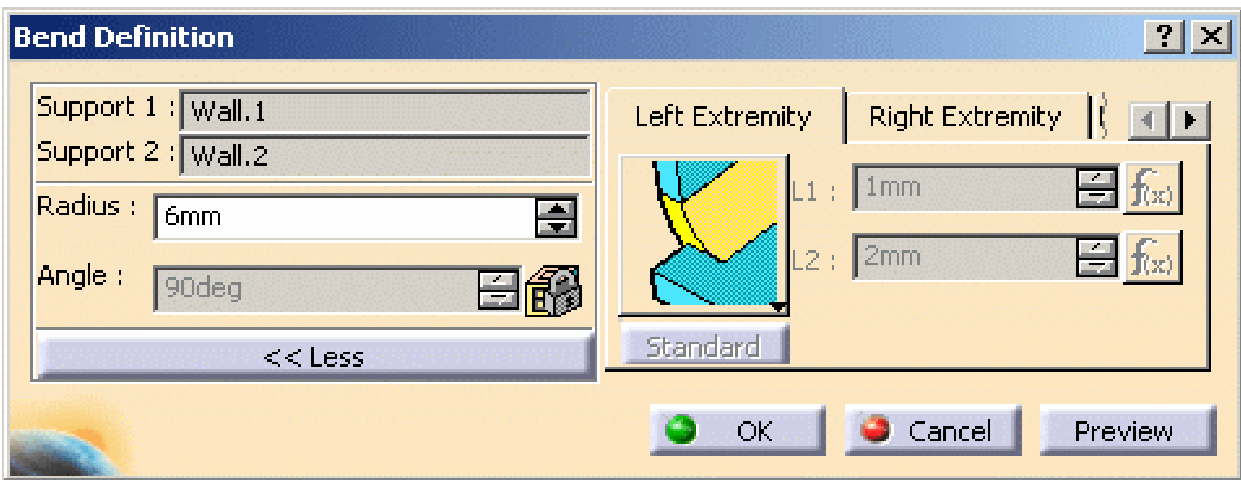


4. Enter 4mm for the Radius and click Preview.

The bend is previewed, along with its orientation symbolized by arrows. The Left and Right texts further indicate this orientation and are useful to define different bend extremities.



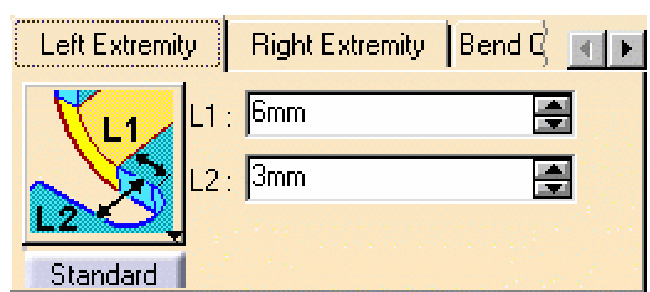
5. Click the **More>>** button to display further options:



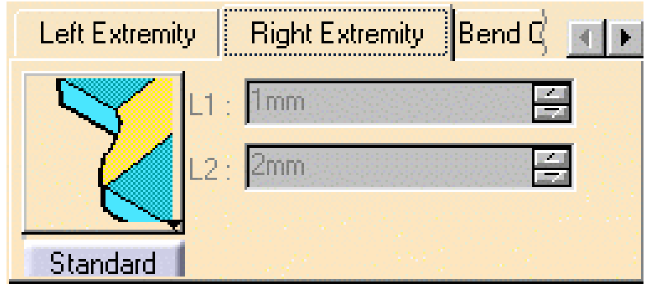
You can define:

- the left and right extremity settings (see also extremities definition settings)
- the corner relief definition settings
- and the bend allowance settings.

6. Within the Left Extremity tab, choose the Mini with round relief bend extremity type, deactivate the L1 and L2 length formulas, and set them to 6mm and 3mm respectively.



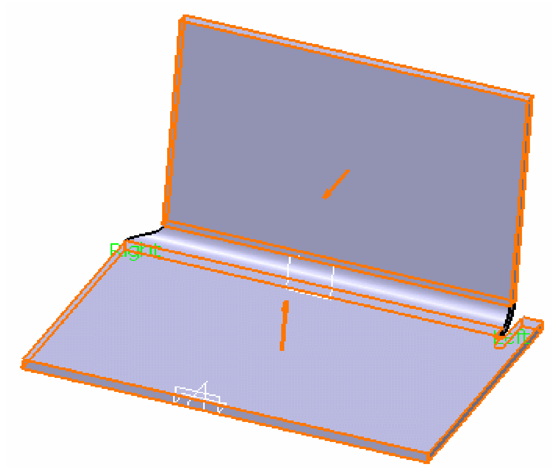
7. Click the Right Extremity tab, and choose the Curved shape type.

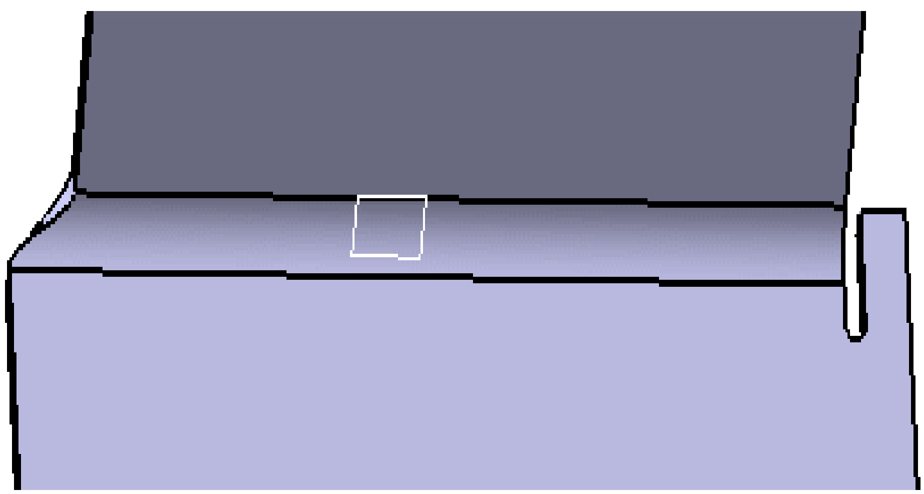


8. Click Preview to visualize the left and right extremities.

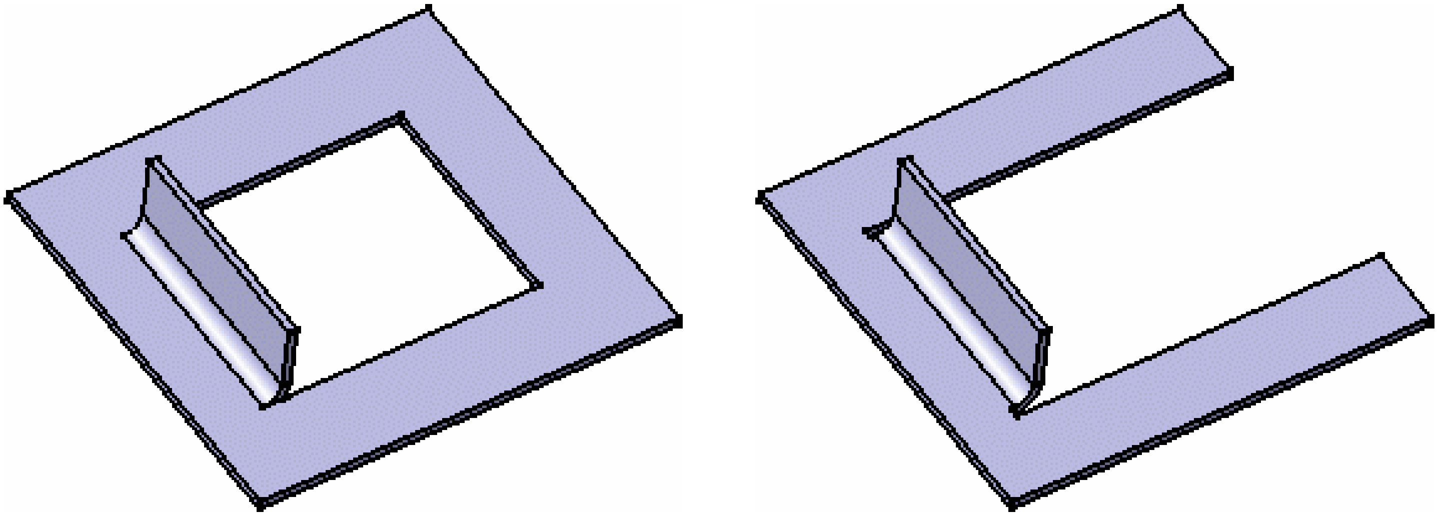
9. Click OK in the Bend Definition dialog box.

The Bend (identified as Cylindrical Bend) is created with the specified extremity types and is added to the specification tree.



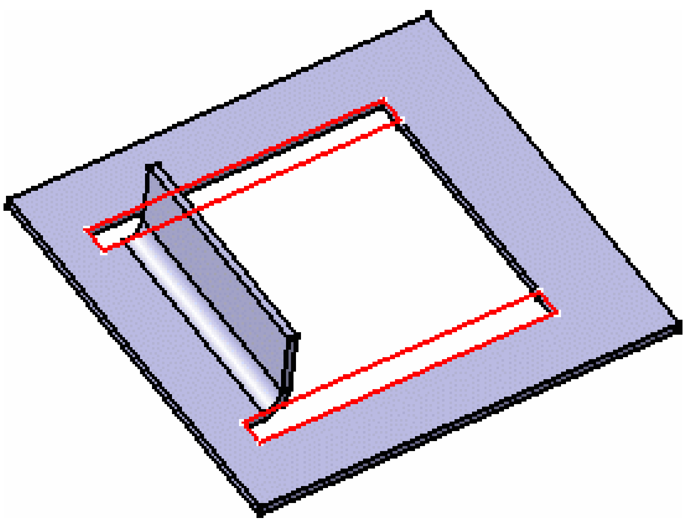


- The extremities and the corner relief defined with the current dialog box will apply locally and prevail over any other global definition.
- Be careful when creating bends with **square** or **round** relief. Depending on the geometry configuration, this can lead to removing more matter than you would expect. Indeed, a corner relief being computed on the whole intersection of the elements involved (bends or bend/wall), in the following configuration, the matter is removed till the end of the wall.



Bend with no relief *Bend with square relief*

In this case, it is best to replace the bend corner relieves with cutouts (identified in red in the figure below) that will act as corner relieves removing matter to the part.



Generating Bends Automatically

This functionality is only available with SheetMetal Design.

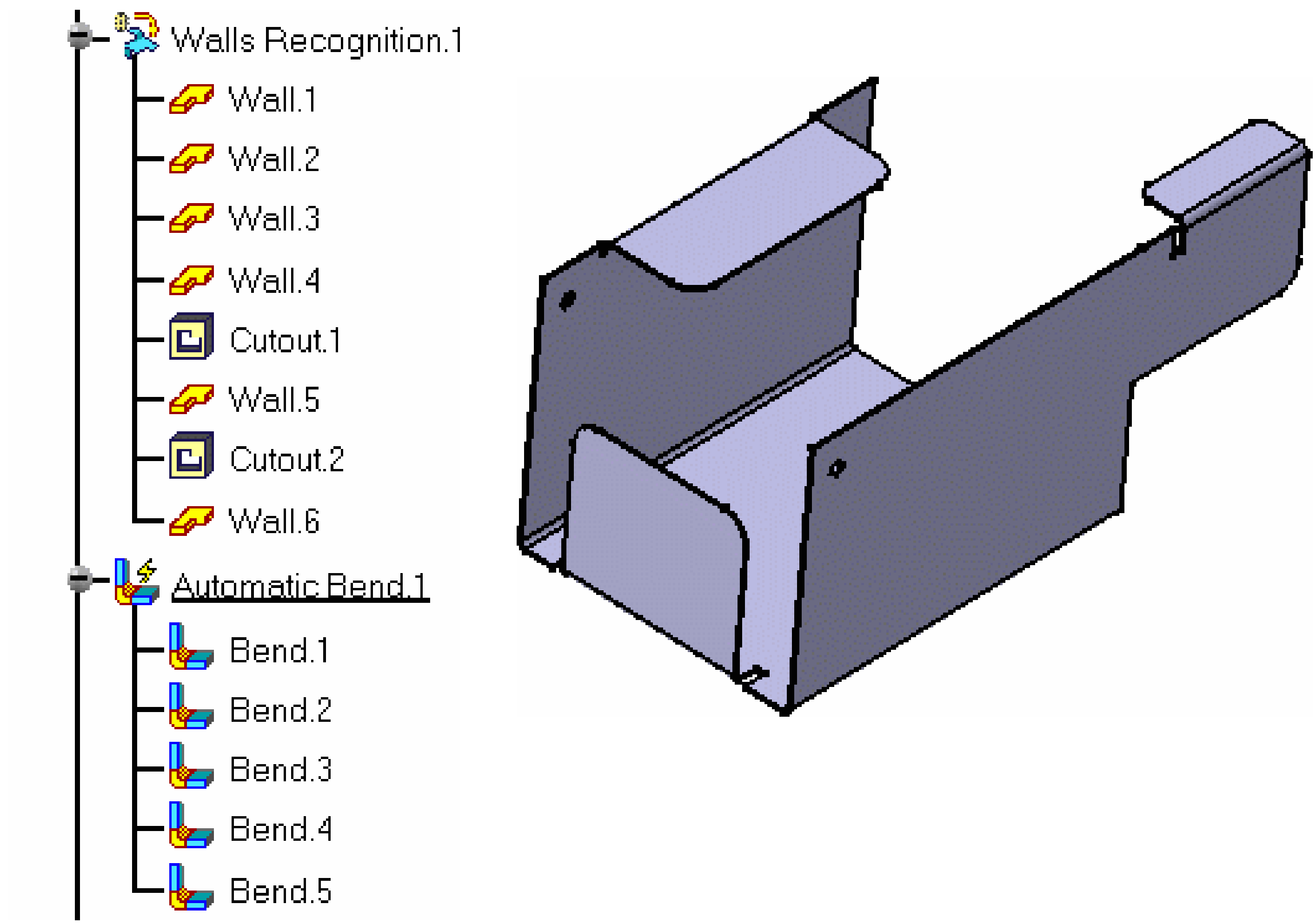
This task explains how to automatically generate bends between walls in the Sheet Metal part. You can first create all the bends, then modify the parameters for any of the generated bends.

However, when an ambiguity arises, that is when more than two bends end on the same vertex, the bends are not automatically generated. You then need to create them manually, so as to explicitly select the walls between which the bends are to be created.

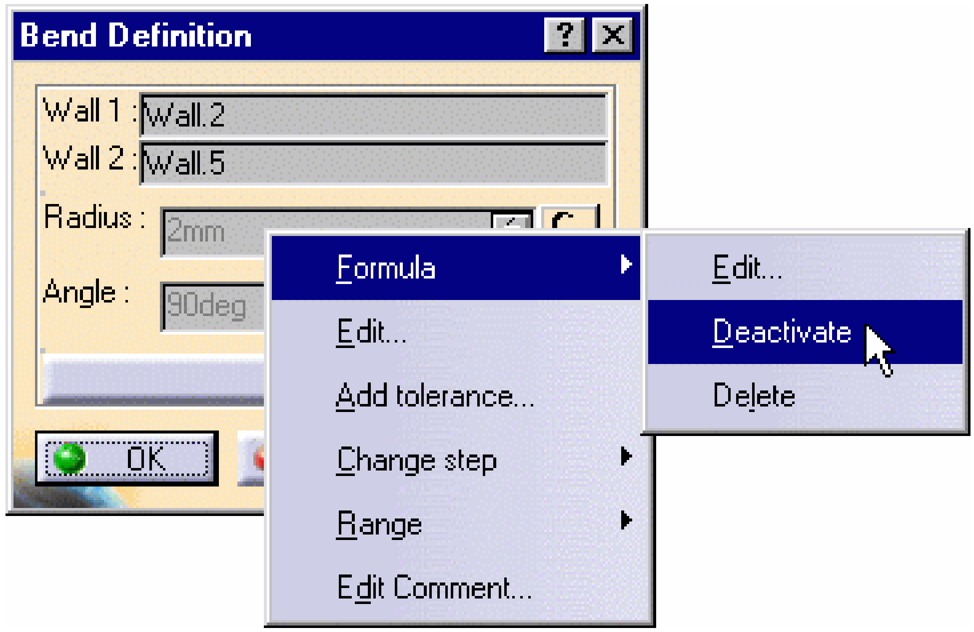
Open the BendExtremities01.CATPart document from the samples directory.

1. Select the **Automatic Bends** icon .

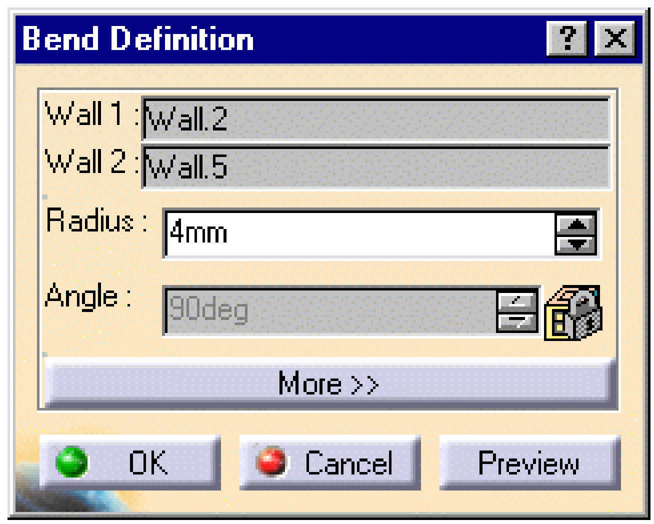
The bends are created.



2. Double-click the bend of interest: **Bend.4**
   The Bend Definition dialog box opens.

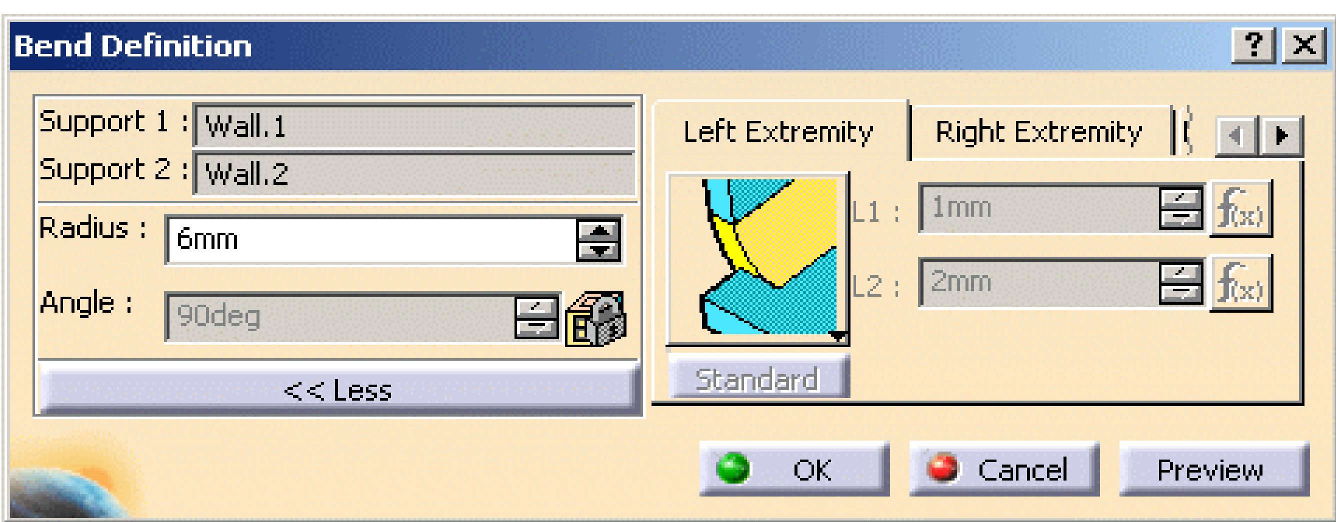


3. Right-click the **Radius** field: the contextual menu appears.
4. Deactivate the formula: you can now change the value.
5. Enter 4mm for the Radius and click Preview.



Bend.4 is modified.

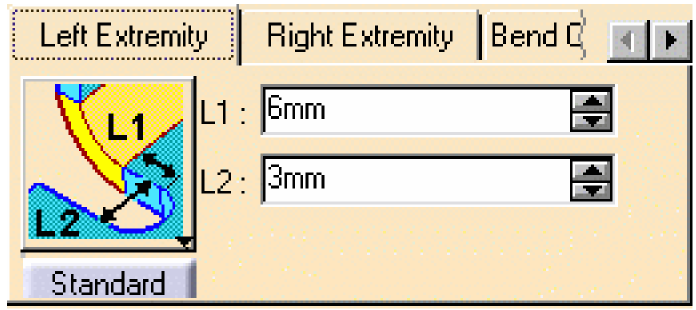
6. Click the More button to display further options:



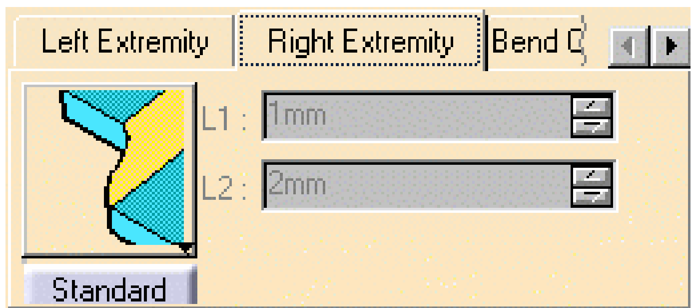
You can re-define:

- the left and right extremity settings (see also extremities definition settings)
- the corner relief definition settings
- and the bend allowance settings.

7. Within the Left Extremity tab, choose the Mini with round relief bend extremity type, deactivate the L1 and L2 length formulas, and set them to 6mm and 3mm respectively.

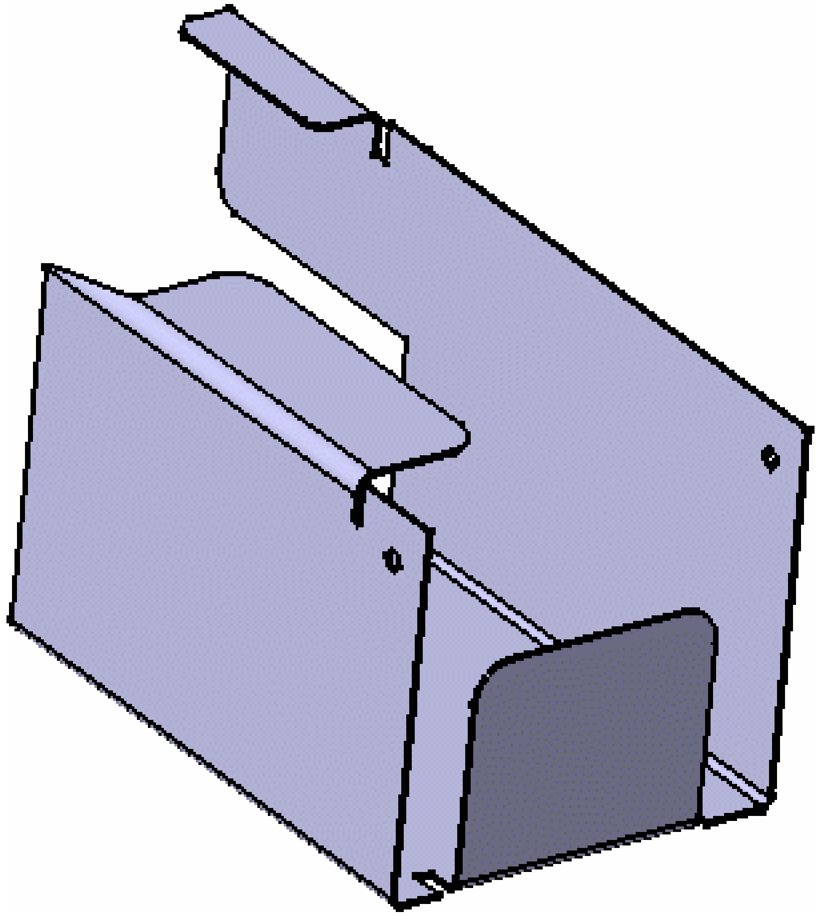


8. Click the Right Extremity tab, and choose the Curved shape type.



9. Click OK in the Bend Definition dialog box.

The bend is modified with the specified options



The extremities and the corner relief defined with the current dialog box will apply locally and prevail over any other global definition.

Push the more button to display:

- the extremities definition settings,
- the corner relief definition settings,
- and the bend allowance settings.

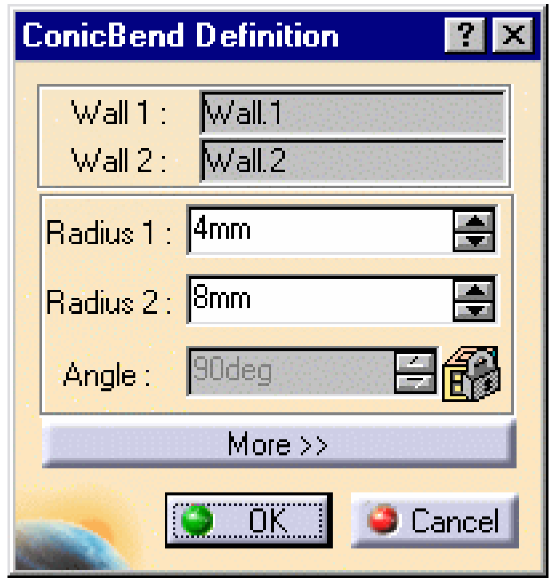
Creating Conical Bends

This task explains how to generate conical bends between two walls in the Sheet Metal part.
These bends are different from the standard bend in that they allow different radius values at each end of the bend.

Open the ConicalBend1.CATPart document from the samples directory.

1. Select the **Conic Bend** icon .

   The Conic Bend Definition dialog box opens.

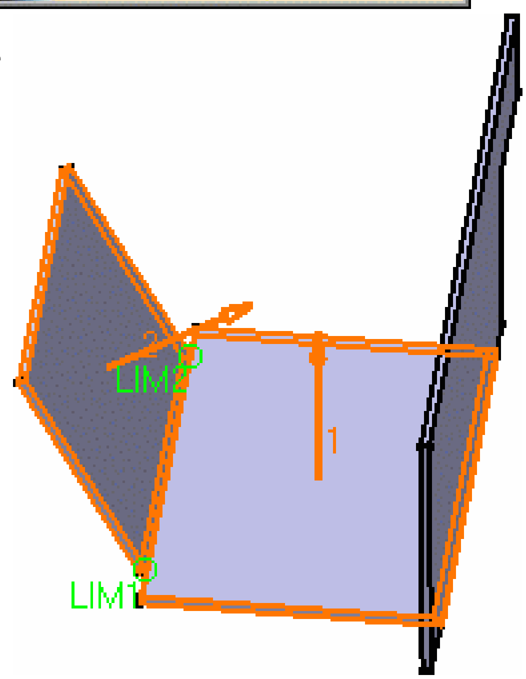


2. Select **Wall.1** and **Wall.2** in the specification tree or in the geometry.

   The Bend Definition dialog box is updated, and arrows are displayed indicating the walls orientation.

You can click on the arrow to invert them if needed.

The LIM1 and LIM2 texts indicate the endpoints for the bend.



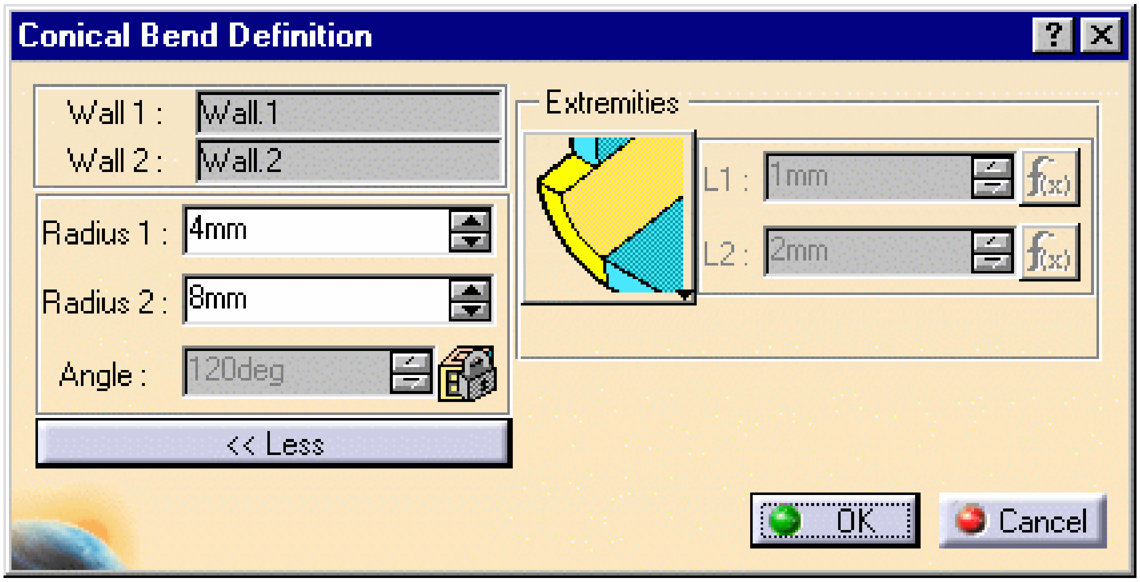
3. Enter the radius values for each end of the conical bend.

   If the difference between the specified radius values does not allow the generation of a cone with an angle greater than 1 degree, a warning is issued prompting you to increase one of the radii.

   Click OK in the Warning dialog box, and increase/decrease the radius values.

By default, **Radius 2** is twice **Radius 1**.

4. Click **More...** to display further options.

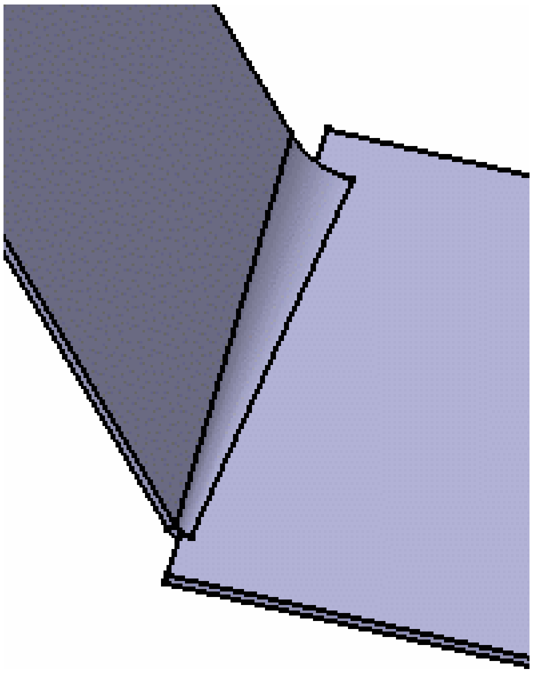


5. Choose the bend extremities:

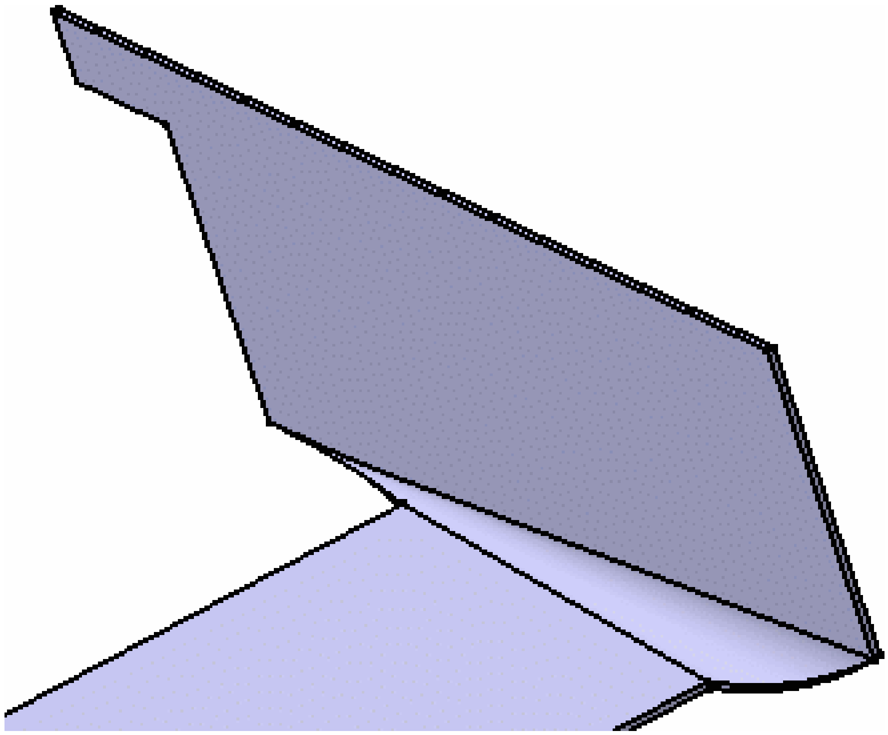
- **Mini with no relief**: the shortest possible bend is created, and presents no relief
- **Curve shaped**: the bend is created keeping the tangency continuity with the support walls.
- **Maximum**: the bend is calculated between the furthest opposite edges of the supporting walls.

6. Click OK.

The Bend is created.



- The two walls must be connected by the edge of their internal faces.
- The **Angle** field is locked. It indicates the angle value between the two walls between which the bend is computed.
- Should you choose the **Curve shaped** extremity option, the bend would look like this:



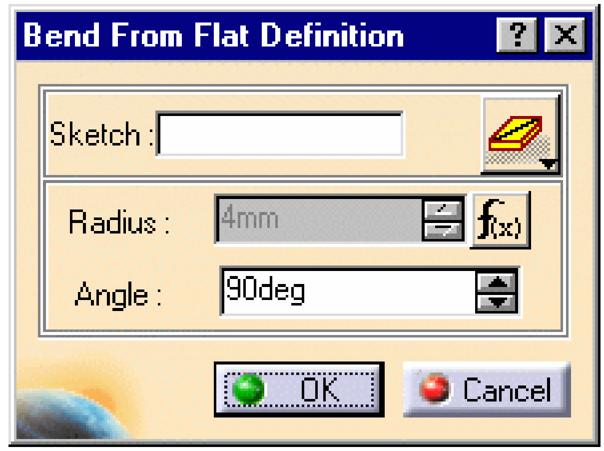
Creating Bends From a Line

This task explains how to generate bends based on a line (also called flat bends). A wall and a bend are created.

Open the FlatBend1.CATPart document from the samples directory.

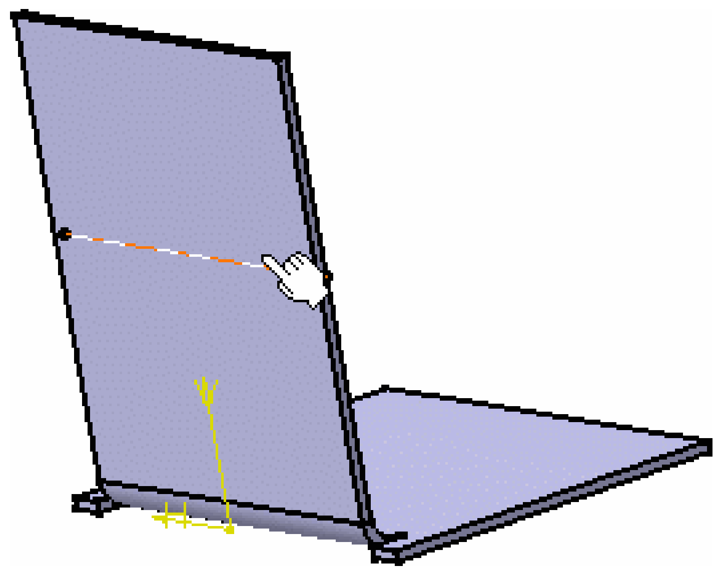
1. Select the reference wall then the **Bend From Flat** icon.

   The Bend From Flat Definition dialog box opens.



2. Select a sketch (Sketch.3 here).

   This sketch must necessarily be a line.



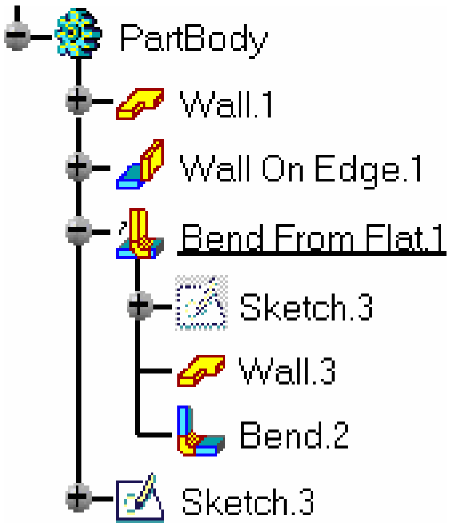
3. You can choose the line extrapolation option:

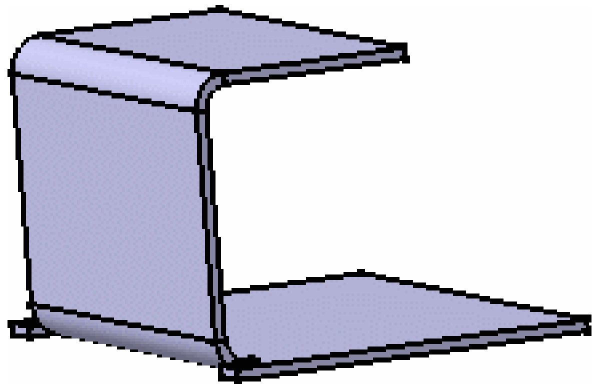
- the line is extrapolated up to the wall edge (**Bend From Flat Until**)
- the line is not extrapolated, and the bend is limited to the line's length (**Bend From Flat Length**)

The Radius value is the one defined when editing the sheetmetal parameters:
Right-click the **Radius** field and select **Formula -> Deactivate** from the contextual menu to change the value.

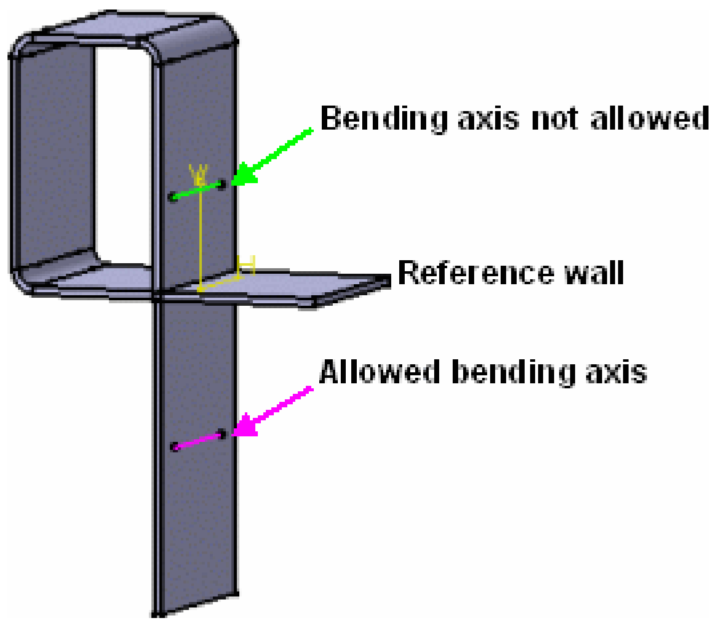
4. Set the angle value between the generated wall and the reference wall.

5. Click OK to create the bend.





- When creating such a bend on a reference wall (first wall), an arrow indicates which part is to be folded. Click this arrow to invert the side that will be bent.



- Such bends cannot be created, if the section to be folded already intersects the part.

- Bends from line should be performed on end walls, or prior to creating further walls on the bent one.
- Perform the bend before creating the stamping features, as stamps are not retained when the part is folded with the bend.

Creating Swept Walls

This section explains and illustrates how to create and use various kinds of swept walls, i.e. walls based on a given profile that is swept along a spine.

 Create a flange: select a spine, and set the radius, length, and angle values.

 Create a hem: select a spine, and set the radius, and length values.

 Create a tear drop: select a spine, and set the radius, and length values.

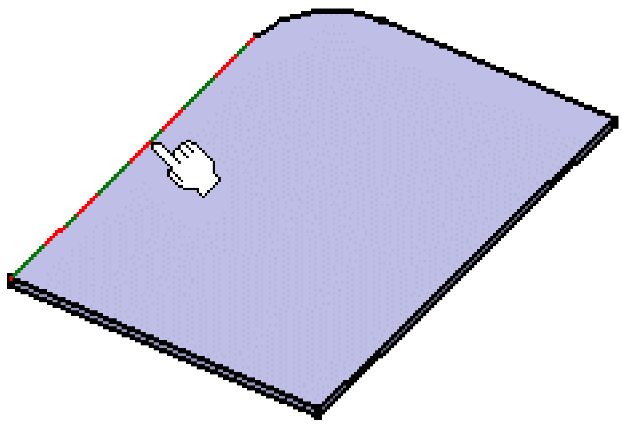
 Create a swept flange: select a spine, and a user-defined profile

Redefine swept walls limits: choose the **Relimited** type, and select a point lying on the spine or a plane normal to the spine and intersecting it as limits

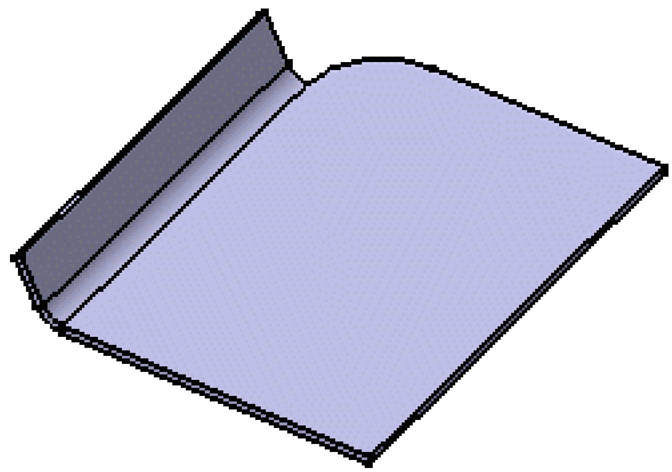
Selecting the Spine

Whatever the type of the swept wall you wish to create, you first need to select one or more contiguous edges to make up the spine along which the profile, either pre- or user-defined, is to be swept. You can:

- manually select one, or more, edge(s)

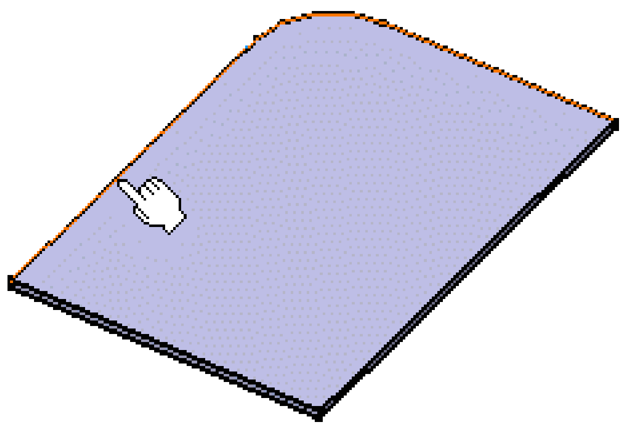


Selection without propagation

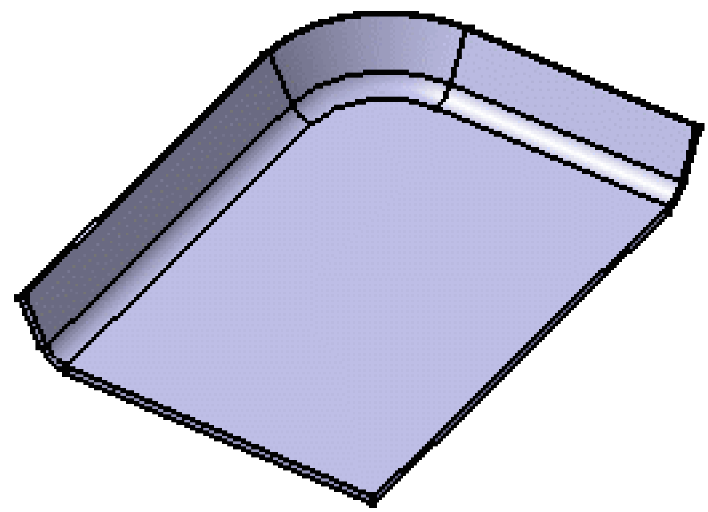


Resulting flange without propagation

- select one edge and click the **Tangency Propagation** button: all contiguous and tangent edges are selected. In this case, would you need to remove one edge, you need to manually select it. Remember that only extremity edges can be removed without breaking the continuity between edges.



Selection with propagation

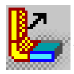


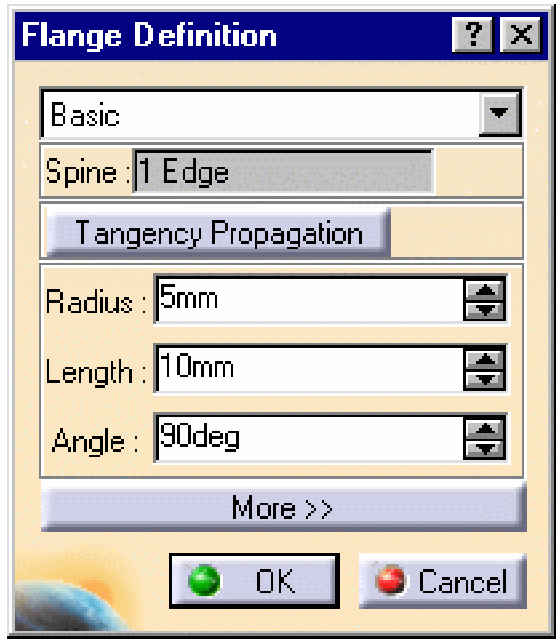
Resulting flange with propagation

Creating a Flange

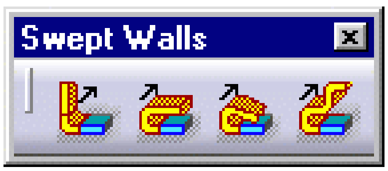
This task explains how to generate a flange from a spine and a profile.

Open the SweptWall01.CATPart document from the samples directory.

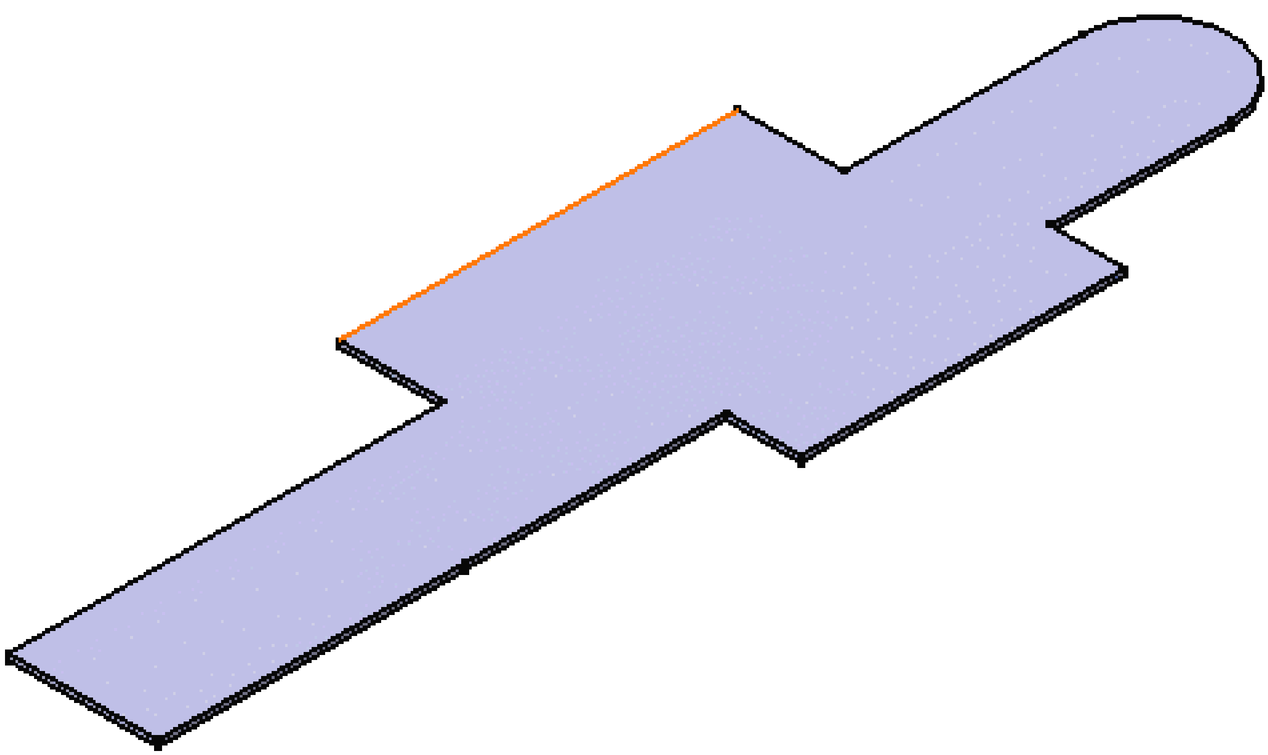
1. Select the **Flange** icon .



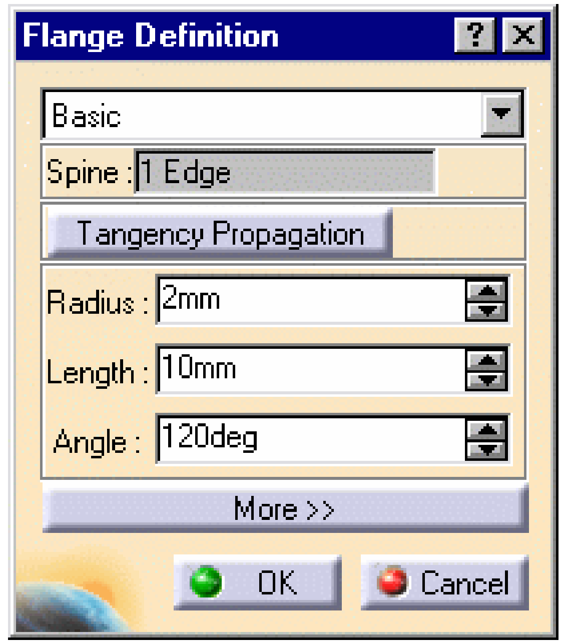
The Flange Definition dialog box opens.



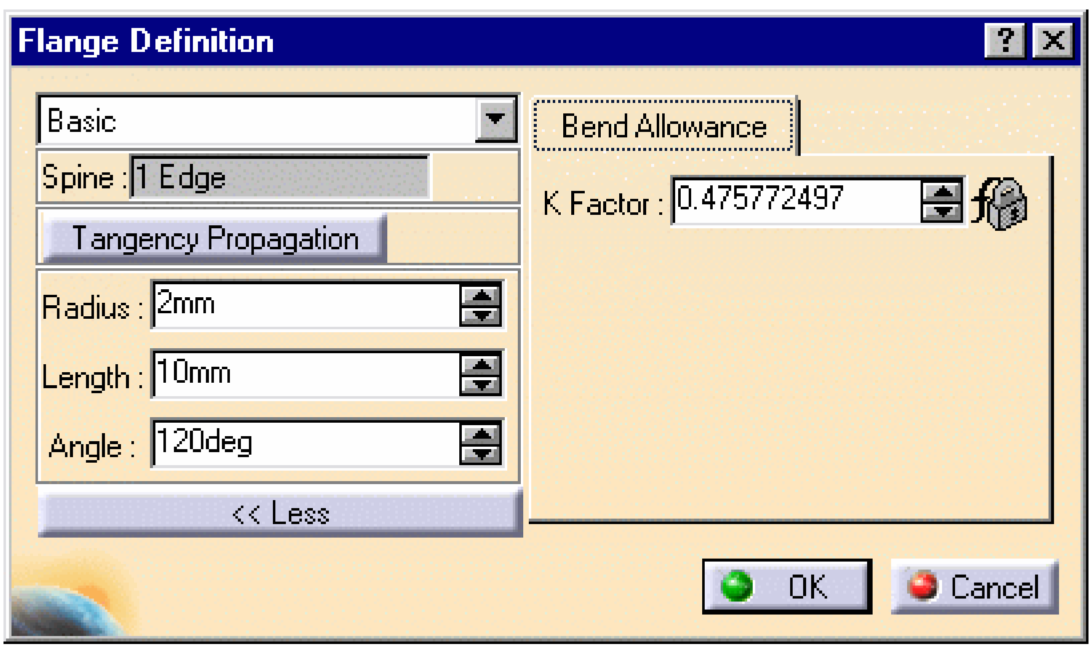
2. Select the edge as shown in red.



3. Enter 2mm in the Radius field,
10mm in the Length field and 120deg for the Angle.

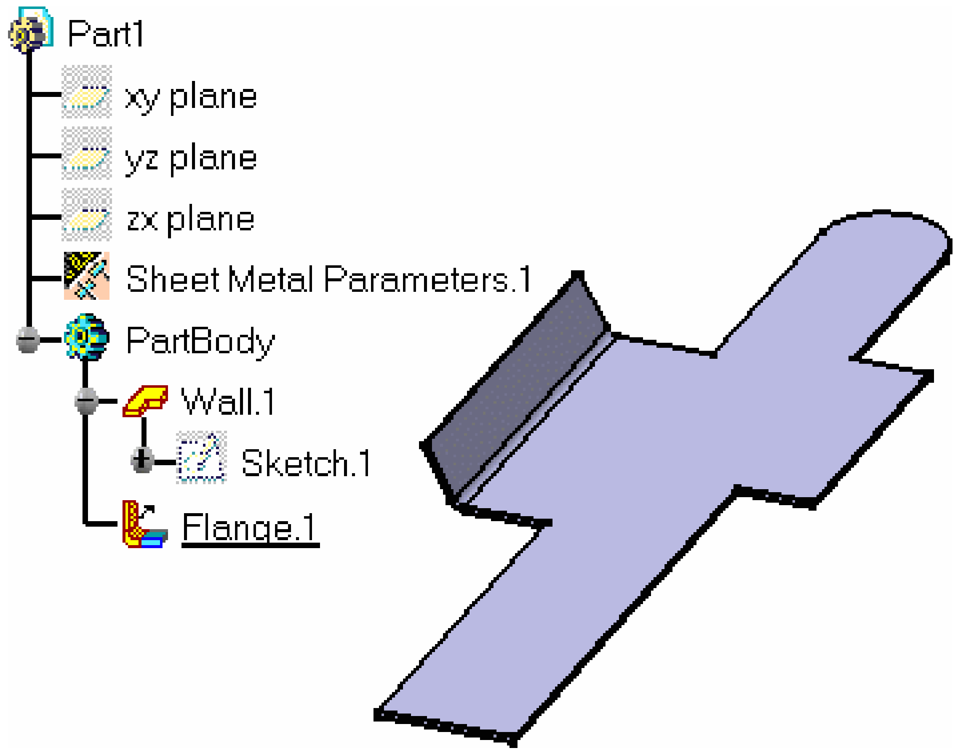


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click OK to create the flange.



The feature is added to the specification tree.

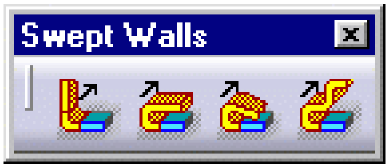
- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the flange limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

Creating a Hem

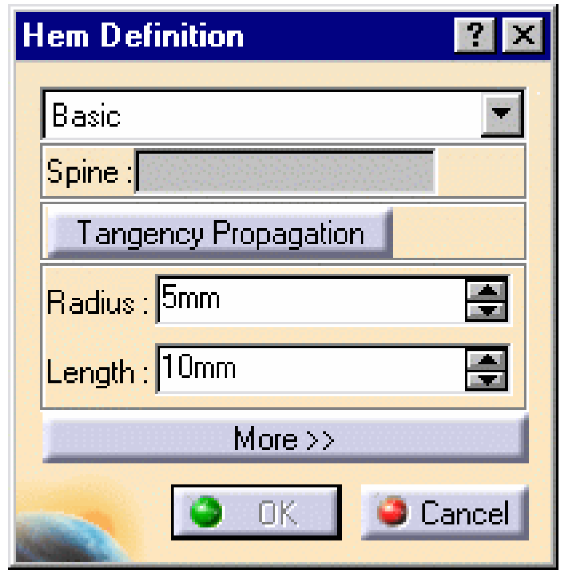
This task explains how to generate a hem from a spine and a profile.

The SweptWall01.CATPart document is still open from the previous task.
If not, open the SweptWall02.CATPart document from the samples directory.

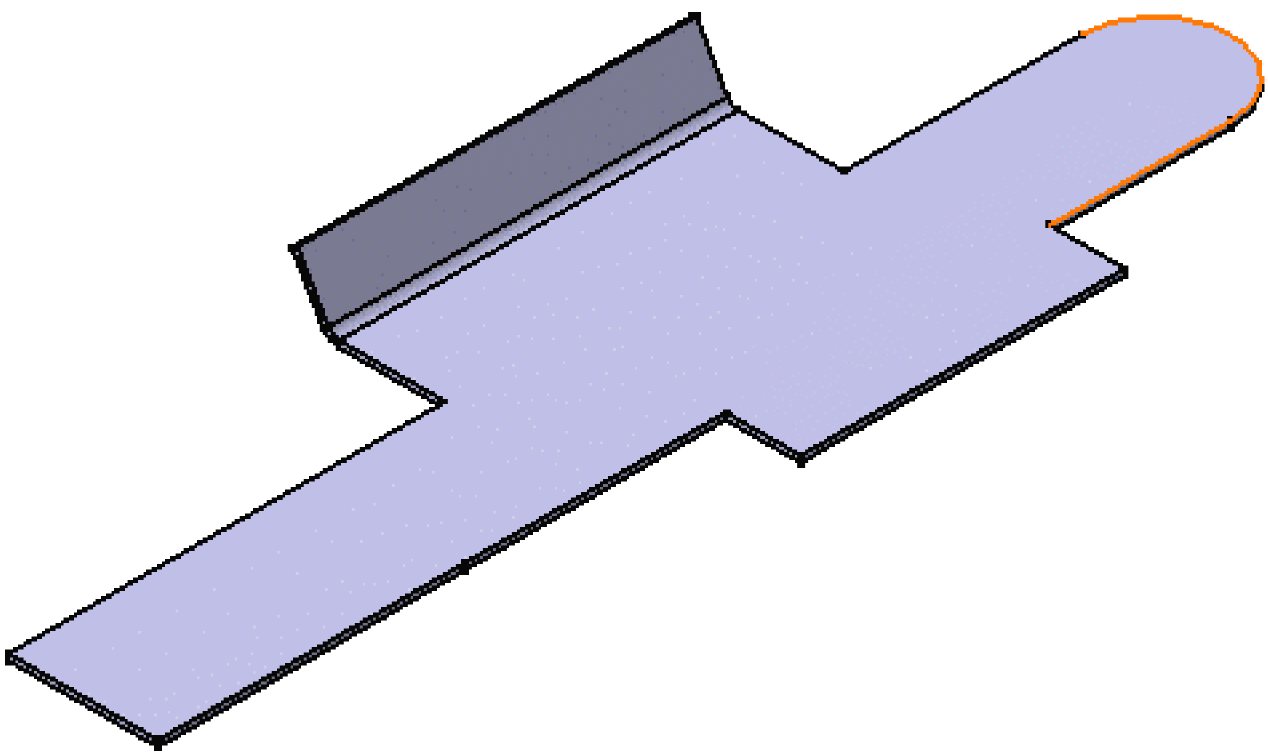
1. Select the **Hem** icon in the **Swept Walls** sub-toolbar.



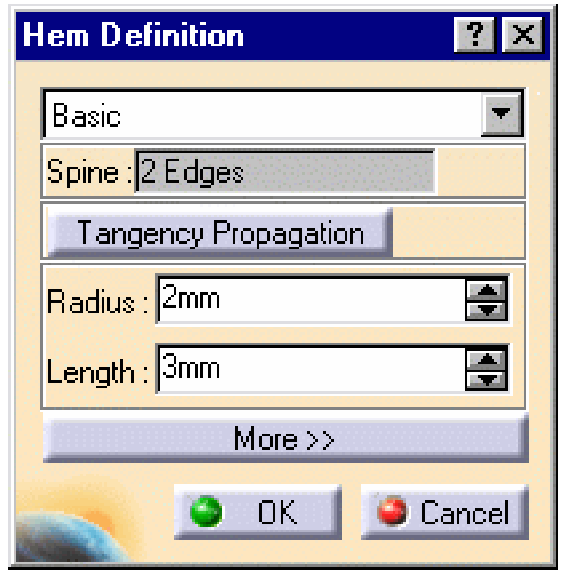
The Hem Definition dialog box opens.



2. Select the edges as shown in red.

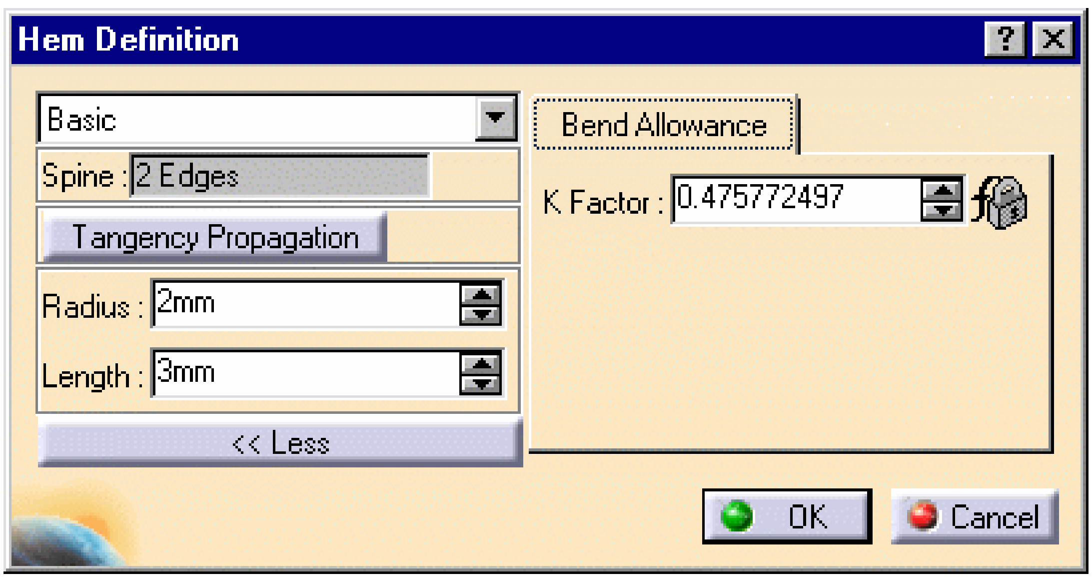


3. Enter 2mm in the **Radius** field, and 3mm in the **Length** field.



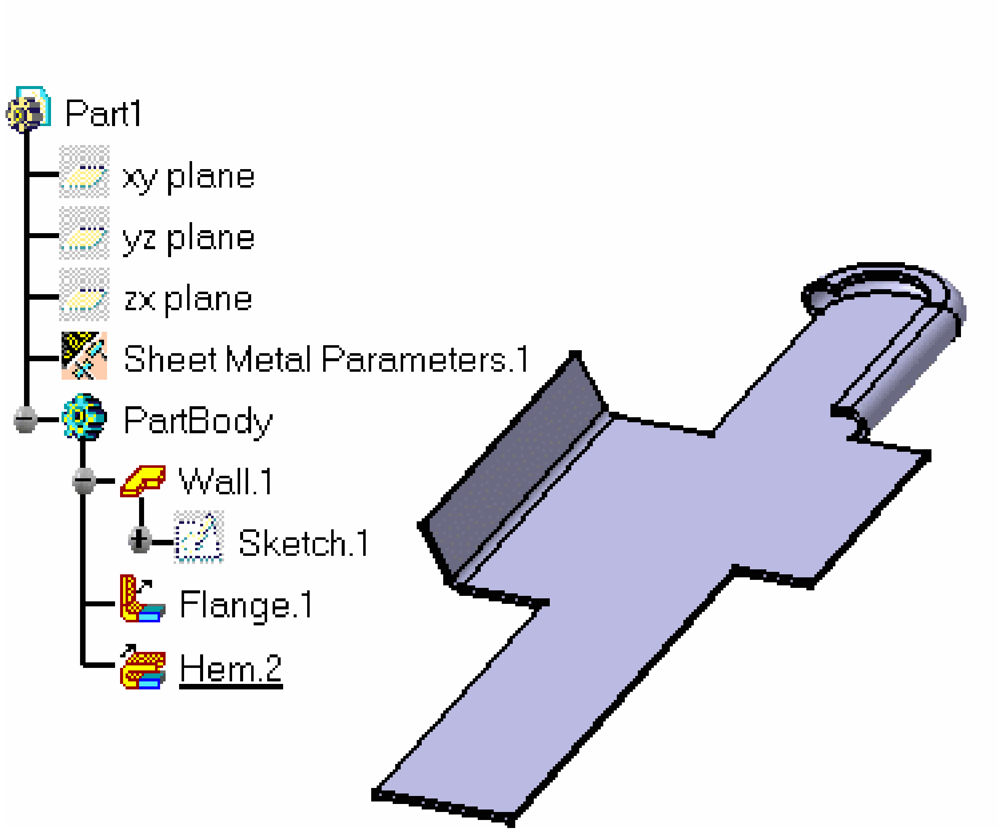
4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the hem.



The feature is added to the specification tree.

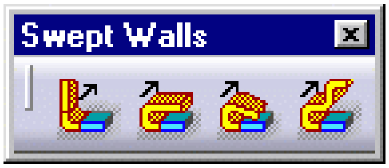
- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the hem limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

Creating a Tear Drop

This task explains how to generate a tear drop from a spine and a profile.

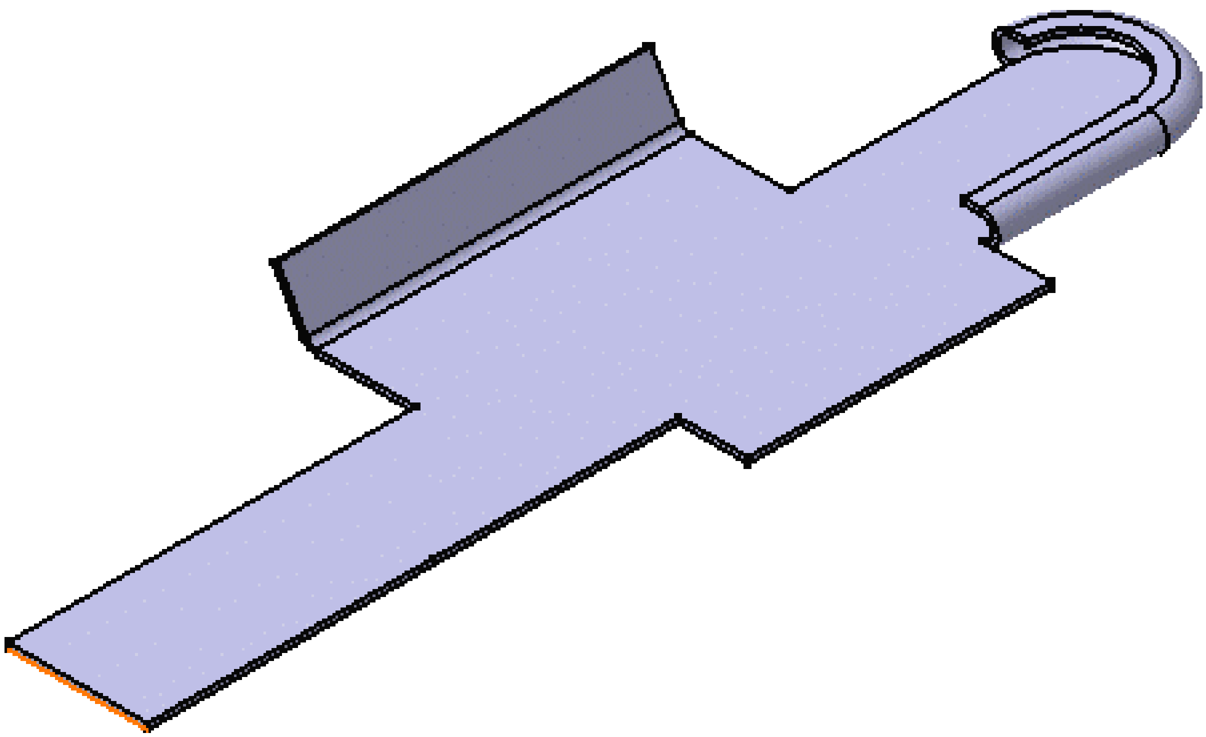
The SweptWall01.CATPart document is still open from the previous task.
If not, open the SweptWall03.CATPart document from the samples directory.

1. Select the **Tear Drop** icon  in the **Swept Walls** sub-toolbar.

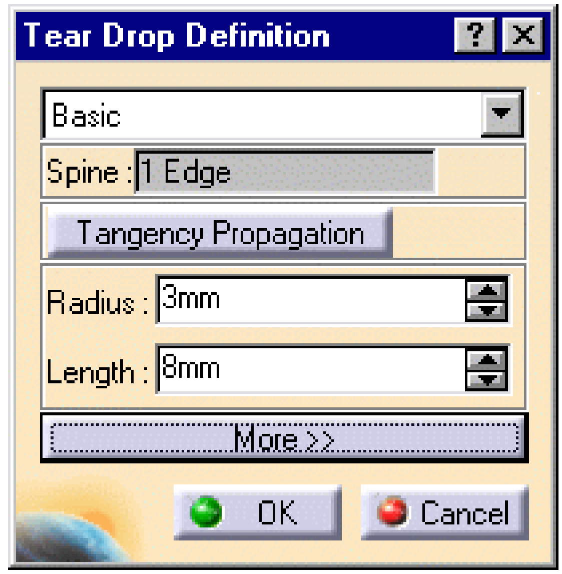


The Tear Drop Definition dialog box opens.

2. Select the edge as shown in red.

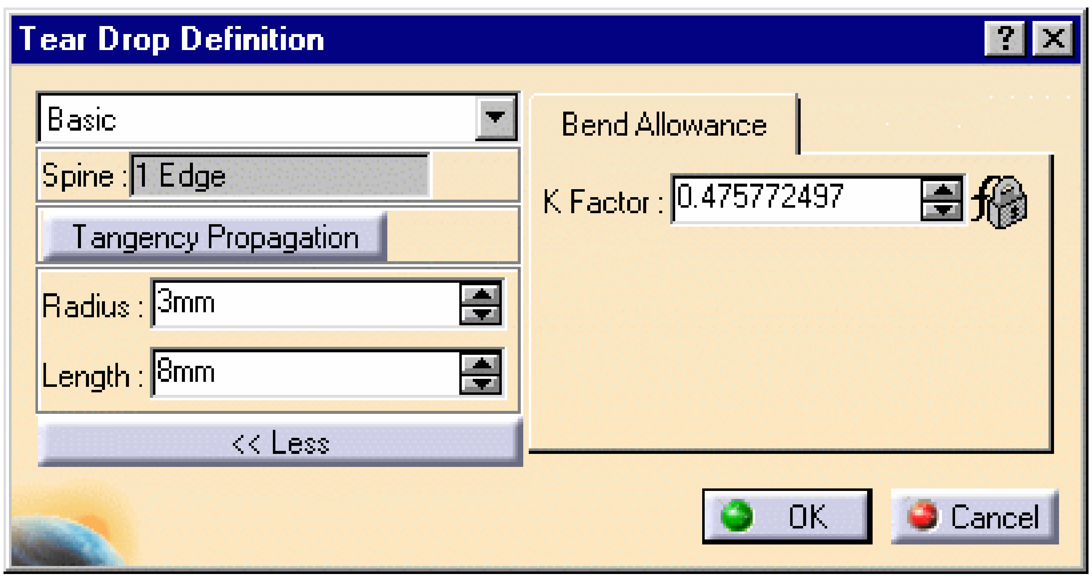


3. Enter 3mm in the **Radius** field, and 8mm in the **Length** field.



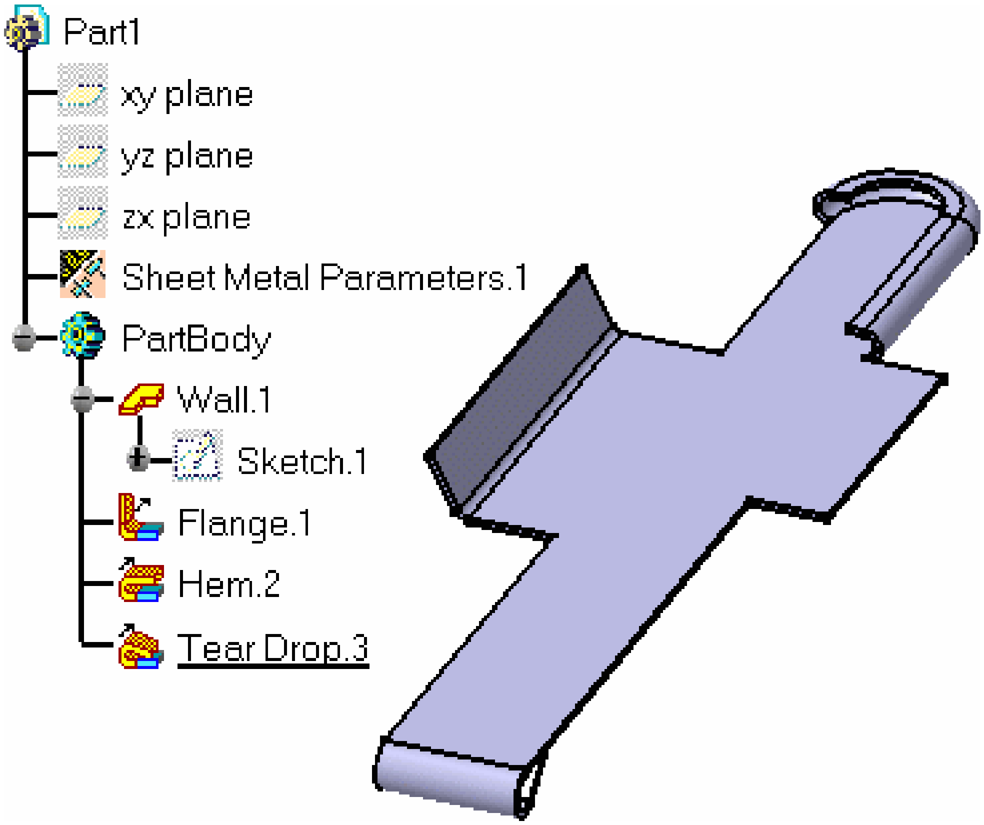
4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the tear drop.



The feature is added to the specification tree.

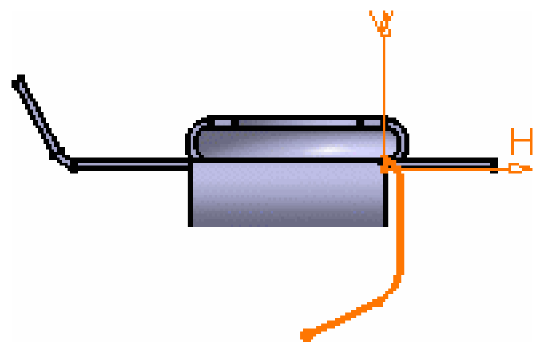
- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the hem limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

Creating a Swept Flange

This task explains how to generate a swept flange from a spine and a user-defined profile.

The SweptWall01.CATPart document is still open from the previous task.
If not, open the SweptWall04.CATPart document from the samples directory. As a profile is already defined on the part, you will be able to skip step 1 of the scenario.

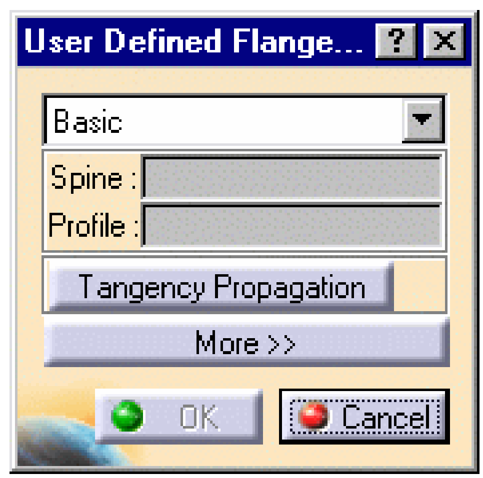
1. If you are using the SweptWall01.CATPart document, click the Sketcher icon , and define a profile in the yz plane as shown below:



Then quit the Sketcher, using the **Exit** icon .

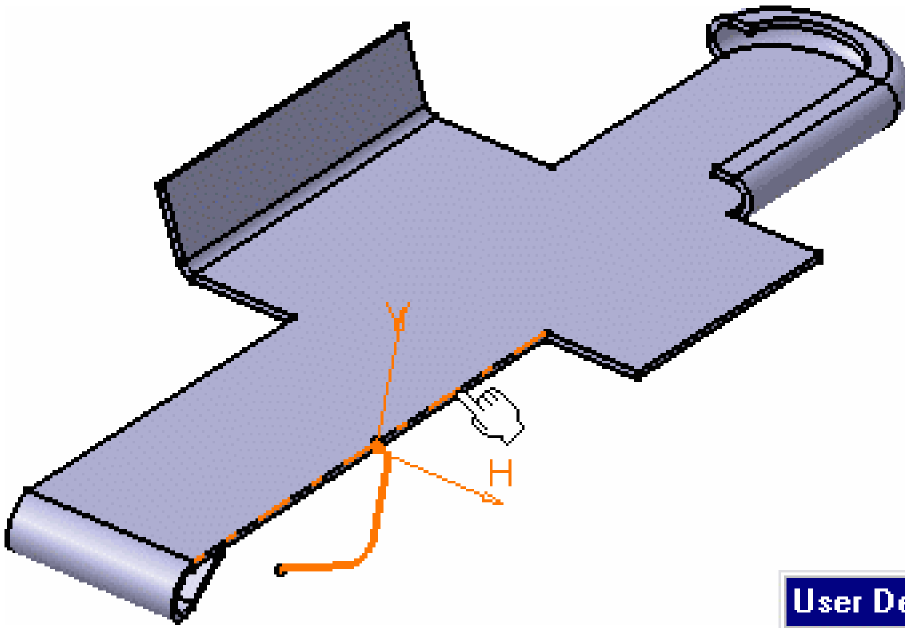
If you are using the SweptWall04.CATPart, go directly to step 2 as the profile is already defined.

2. Select the **Swept Flange** icon in the **Swept Walls** sub-toolbar.

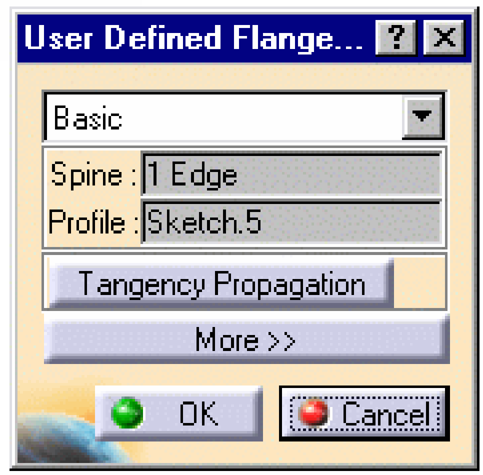


The **User Defined Flange** Definition dialog box opens.

3. Select the edge and the profile, as shown in red.

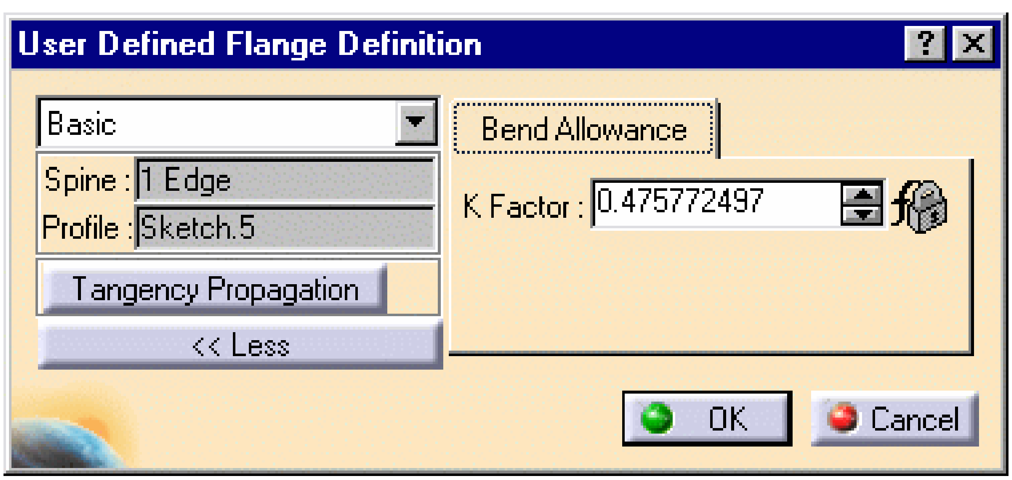


The dialog box looks like this:



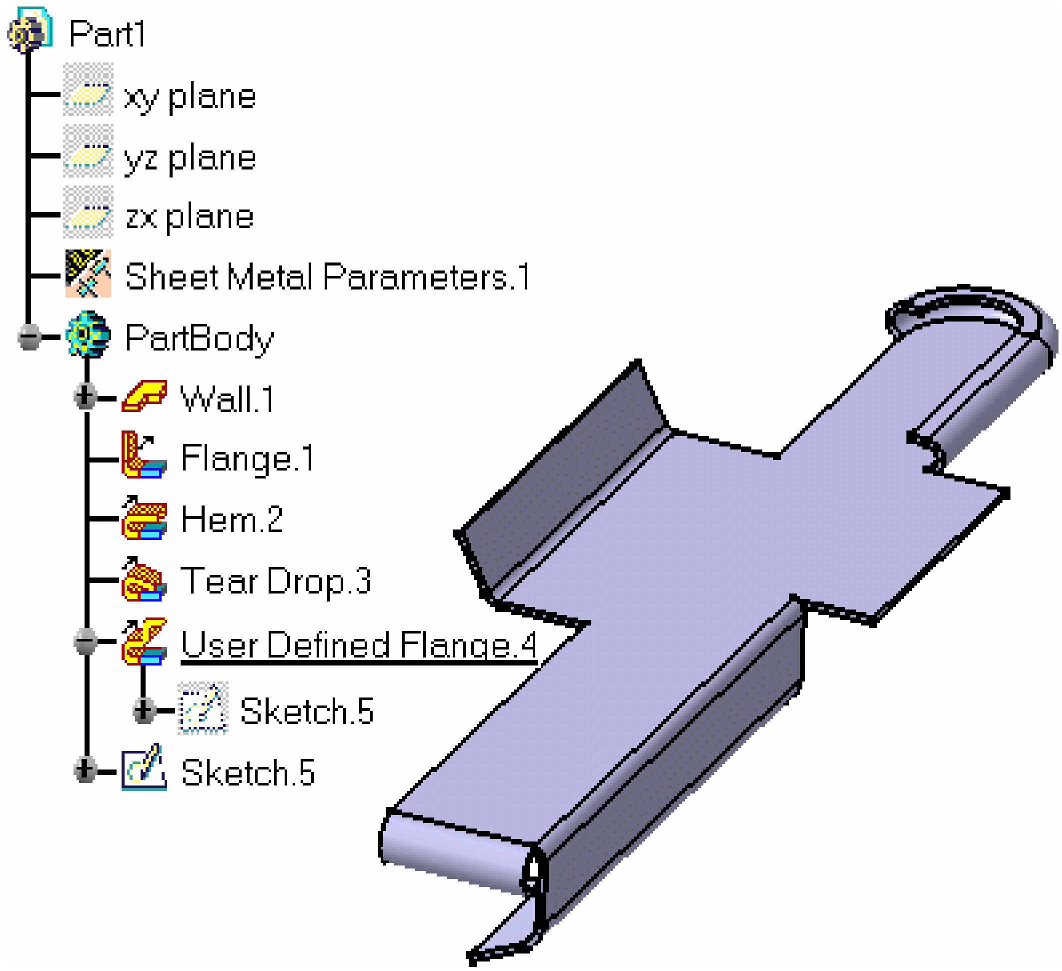
4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the swept flange.



The feature is added in the specification tree.

- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the hem limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

Redefining Swept Wall Limits

This task explains how to redefine the spine's limits when creating any type of swept walls, using existing geometric elements: points lying on the spine or intersecting planes.

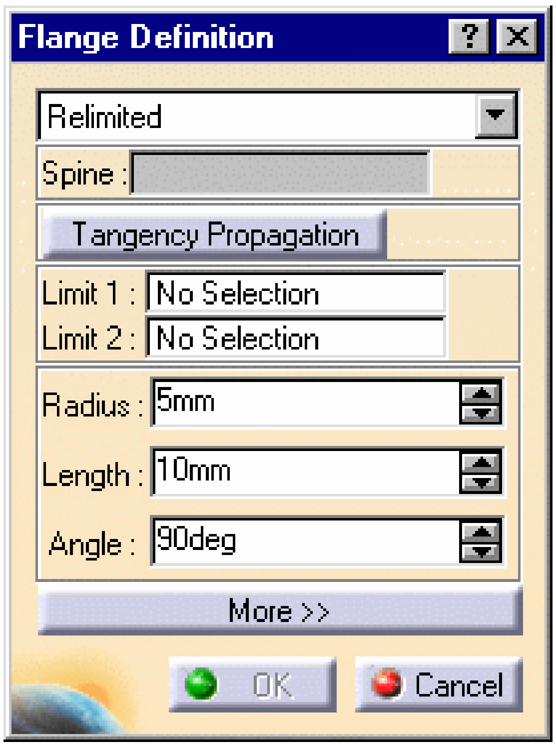
Open the SweptWall01.CATPart document.

1. Select the **Flange** icon .

The Flange Definition dialog box opens.

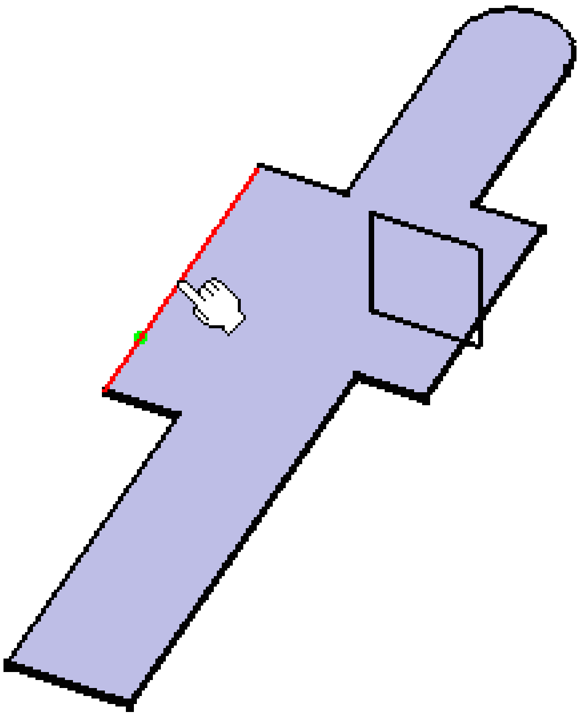
2. Using the combo list, choose the Relimited type.

The Flange Definition dialog box is updated and now displays two **Limit** fields.



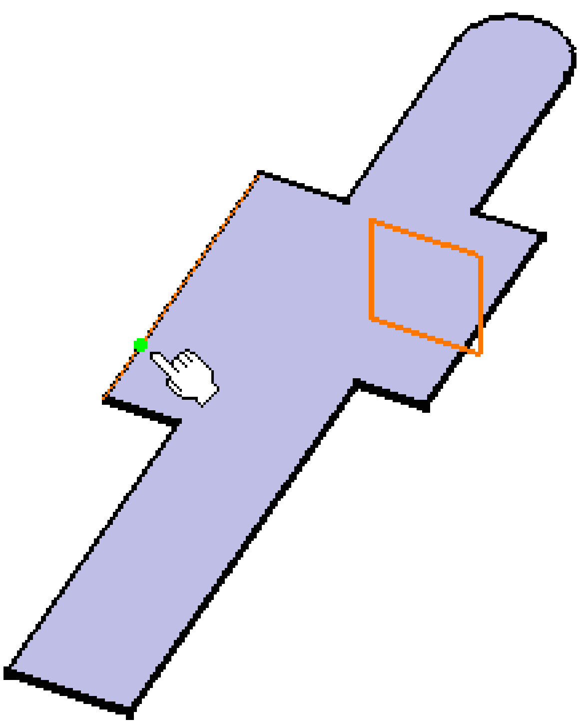
3. Select the spine.

   Here we select a single edge. See also Selecting the Spine.



4. Successively select the two limiting elements.

Here we select a point lying on the spine as the first selecting element, and a plane intersecting the spine as the second limiting element.



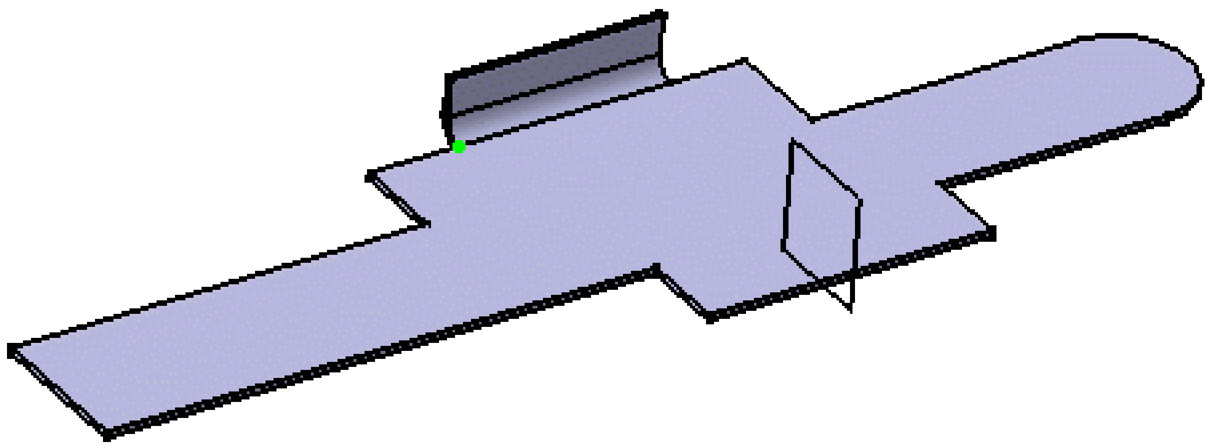
Make sure intersecting elements are normal to the spine, and they intersect it only once.

5. Specify the swept wall values.

   In the example of the flange you set the Radius, Length and Angle values. You may also click **More >>** to display further options. See Creating a Flange.

6. Click OK.

The swept wall is created within the limits on the spine.



Unfolding

Unfolded Sheet Metal parts can be displayed in two ways:

Folded/Unfolded View Access
Concurrent Access

Each Sheet Metal feature is created in a given view: folded, or unfolded. Editing a feature must be done in its definition view. If not, a message is automatically issued, prompting you to change views, before editing the feature.

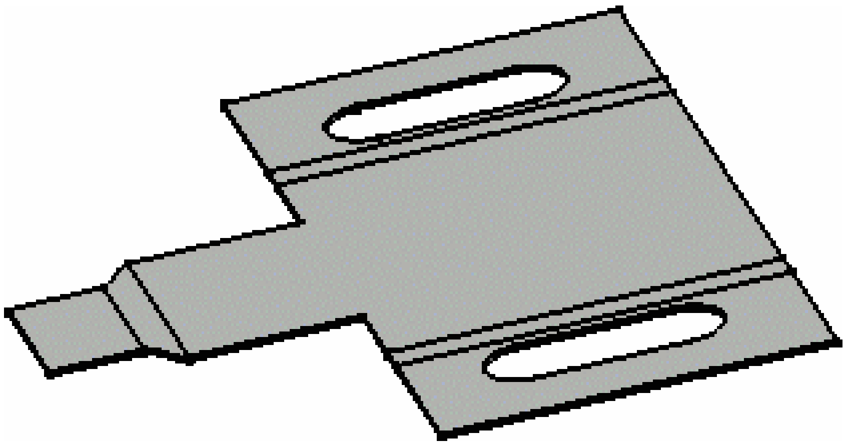
Folded/Unfolded View Access

This task shows how to unfold the part.

To perform this scenario, you can open any sheet metal sample provided in this user's guide.

1. Click the **Unfold** icon .

The part is unfolded according to the reference wall plane or web, as shown below.



2. Click this icon again to refold the part for the next task.

- In SheetMetal Design, bend limits and stamping are now displayed in the unfolded view. However, cutouts created on stamps are not.

- When designing in context, if a CATProduct document contains several sheet metal parts, only one part can be visualized in the unfolded view at a time.

Concurrent Access

This functionality is P2 for SheetMetal Design.

To perform this scenario, you can open any sheet metal sample provided in this user's guide.

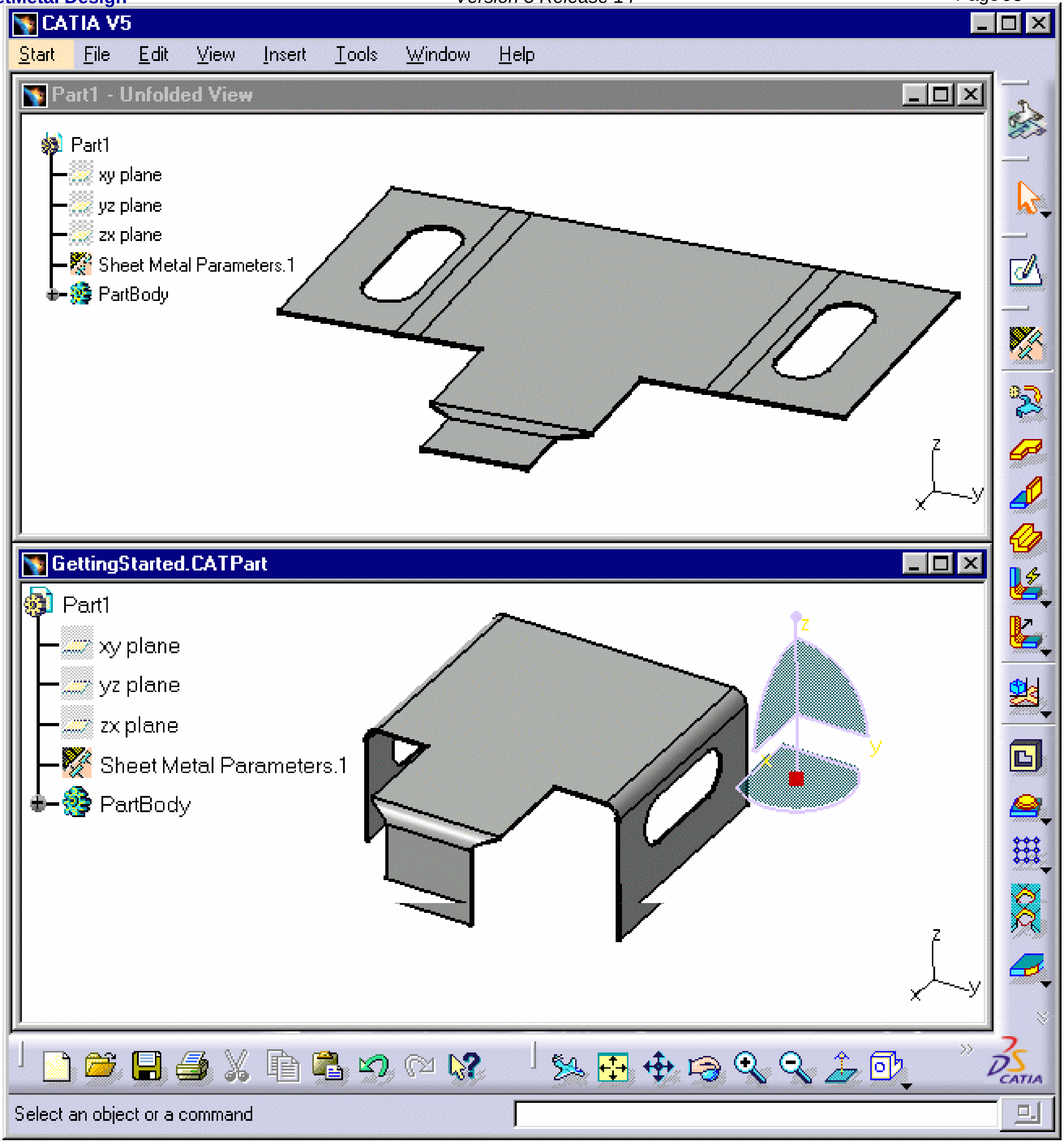
This task explains how to display the sheet metal part in two windows: one with the folded view, one with the unfolded view. Any modification in one window is displayed in the other window.

1. Click the **Multi-view** icon .

The part is unfolded in a second window.

2. Choose the **Window -> Tile Horizontally** menu item.

Both windows are tiled. Activate the window in which you want to work.



- Any modification in one view is taken into account in the other view enabling the user to make modifications in the best possible context.
- In the multi-view mode as in the standard unfolded view, all constraints are displayed in the geometrical views.

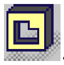
- Once in the Multi-view mode, the standard icon **Unfold** is not longer available.
- The Multi-view function is not available from a standard unfolded view.
- Only parts with bends can be unfolded.
- Cutting faces and open faces are not displayed in Multi-view mode (SheetMetal Design)

Pockets

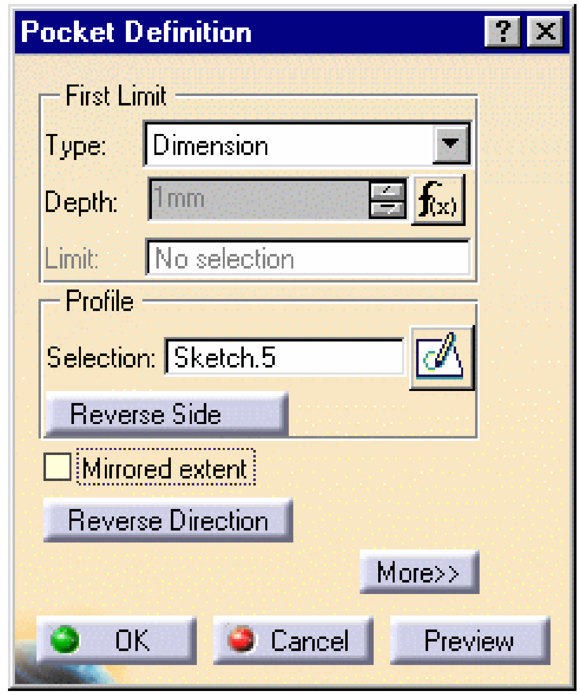
Creating a Cutout

This task explains how to create a cutout in a wall.
Creating a cutout consists in extruding a profile and removing the material resulting from the extrusion.

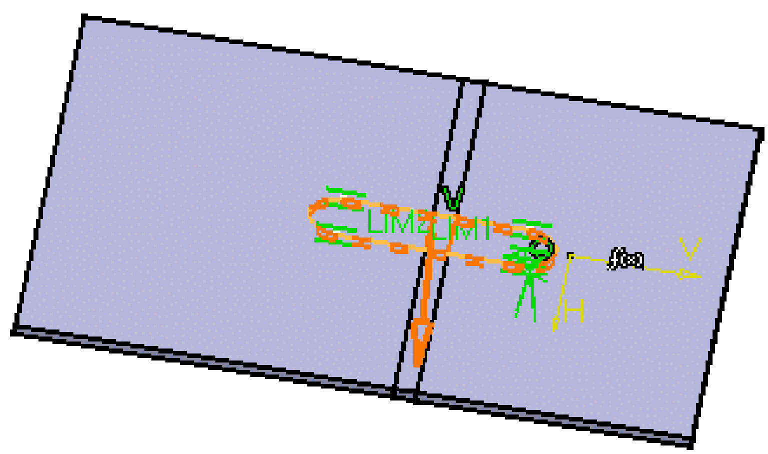
Open the Cutout1.CATPart document.

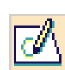
1. Click the **Cutout** icon .

2. Select a profile.



The Pocket Definition dialog box is displayed and CATIA previews a cutout with default parameters.



Once the sketch is selected, you can modify it by clicking the Sketcher icon .

3. Select the type.

Several limit types are available:

- **Dimension**: the cutout depth is defined by the specified value
- **Up to next**: the limit is the first face the application detects while extruding the profile. This face must stops the whole extrusion, not only a portion of it, and the hole goes through material.
- **Up to last**: the last face encountered by the extrusion is going to limit the cutout
- **Up to plane**: the cutout is limited by the selected plane
- **Up to surface**: the cutout is limited by the selected surface

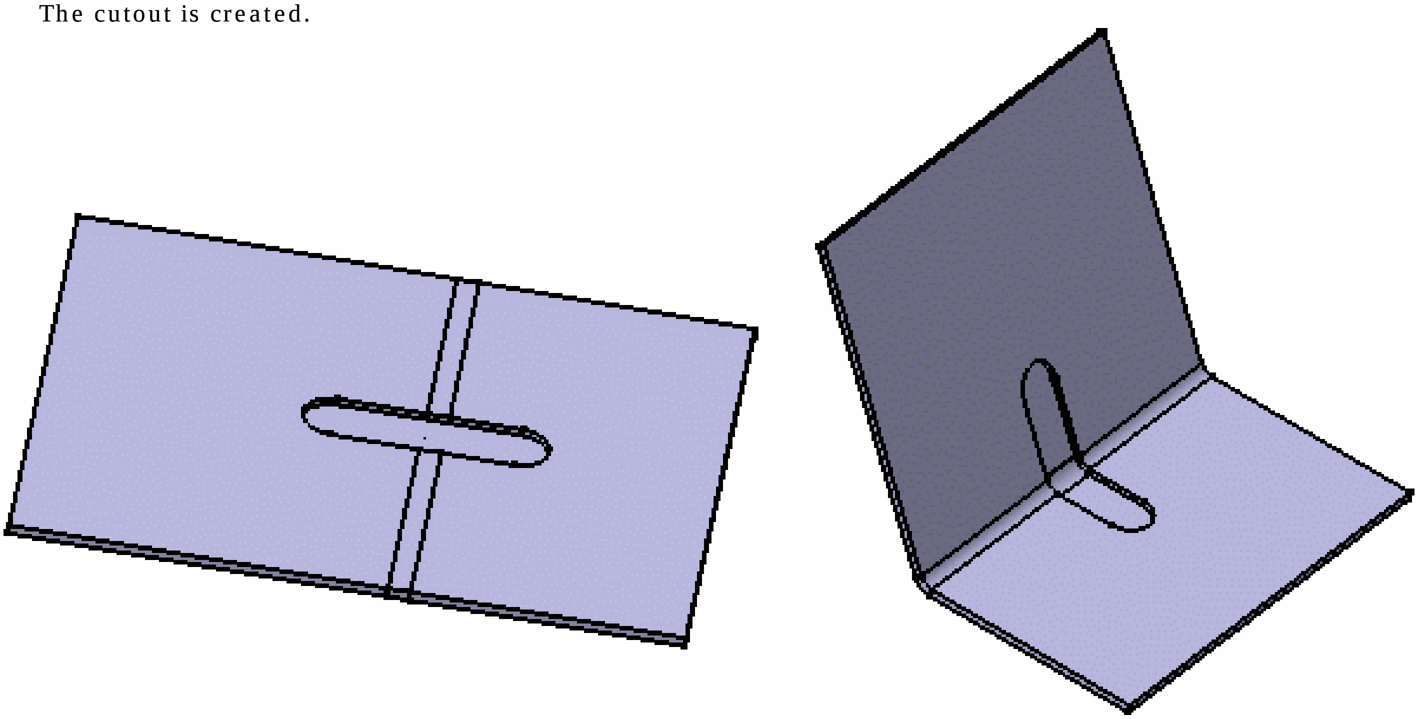
When an **Offset** value is required, it is the distance between the limiting element and the top face of the cutout, if the latter does not result in a complete hole through the material.

The LIM1 and LIM2 texts in the geometry area indicate the top and bottom limits of the cutout.

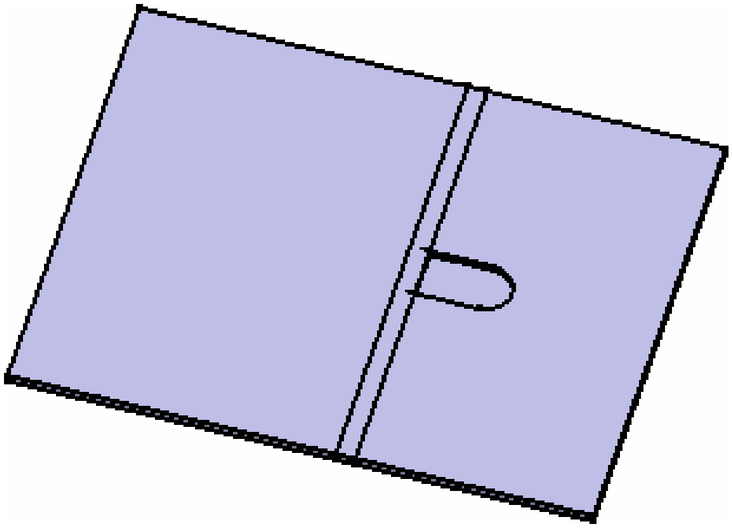
- The **Reverse Side** option lets you choose between removing the material defined within the profile, which is the application's default behavior, or the material surrounding the profile.
- The **Reverse Direction** option allows you to invert the direction of the extrusion pointed by the arrow.

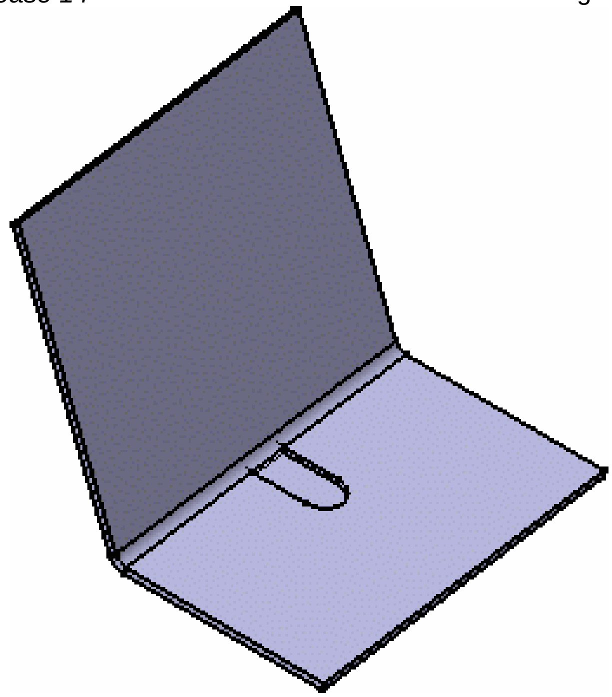
4. Click OK in the Pocket Definition dialog box.

The cutout is created.

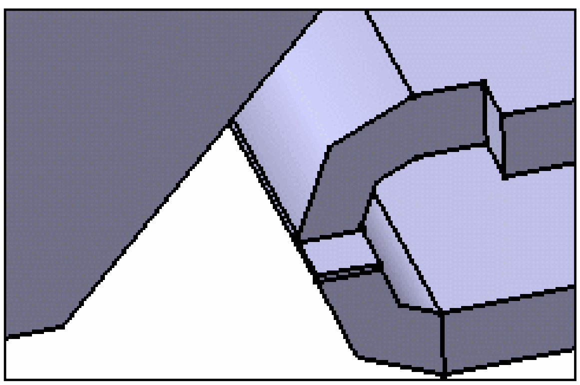


You may want to fold or unfold the part prior to creating the cutout, depending on the selected profile and the expected results. Selecting the same profile, the resulting cutout when created in the unfolded view is seen above, while below, the cutout was created in folded view.

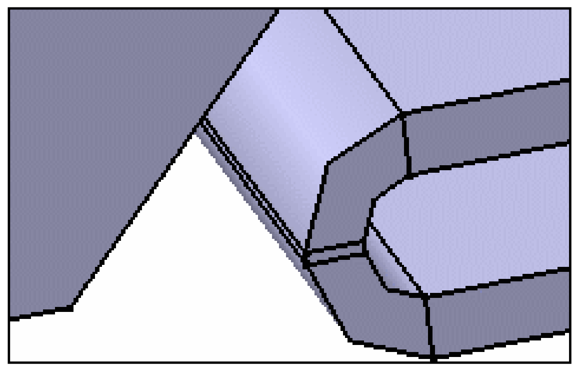




However, you also have to be careful when designing the Part, not to create the cutout in the unfolded view as this may lead, when folding the part, to completely or partially removing another section of the Part. It is best, whenever possible, to create the wall based on a sketch integrating the shape of the cutout.



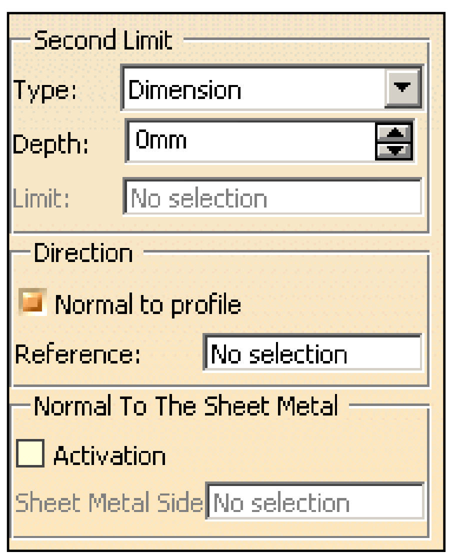
Cutout removing material



Cutout not removing material based on wall's sketch modification

4. Click **More>>** to display the maximum information.

- You can define 'Limit2' as the second limit by using the same options as for Limit 1 (Dimension, Up to last, up to plane, up to surface).
- You can choose between a direction normal to the profile or define a new direction by selecting geometry in the Reference field.
- You can choose the cutout to be normal to the sheet metal part: check the **Activation** button and define the sheet metal side.



5. Select the **Support** (here we chose the wall)

May you want to create a cutout on an overlapping element or a bend with radius=0, either choose the top skin of the element, or unfold the part to create the cutout.

- Refer to the *Component Catalog Editor* documentation to have further information on how to use catalogs.
- Refer to the Pocket task in the *Part Design User's Guide* for further details on how to create cutouts.

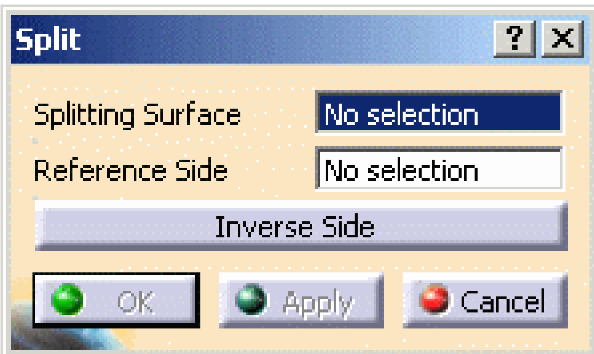
Splitting Geometry

This functionality is only available with SheetMetal Design.

This task shows how to create a split normal to a sheet metal part by means of a cutting surface.

Open the Split1.CATPart document.

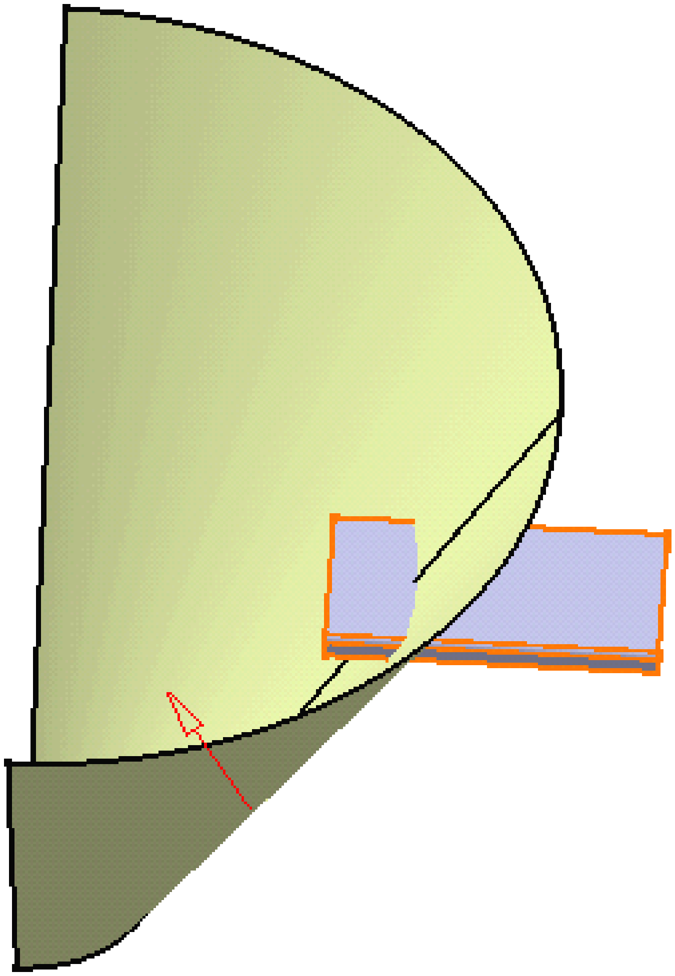
1. Click the **Split** icon . The Split dialog box appears.



2. Select the **Splitting Surface**.

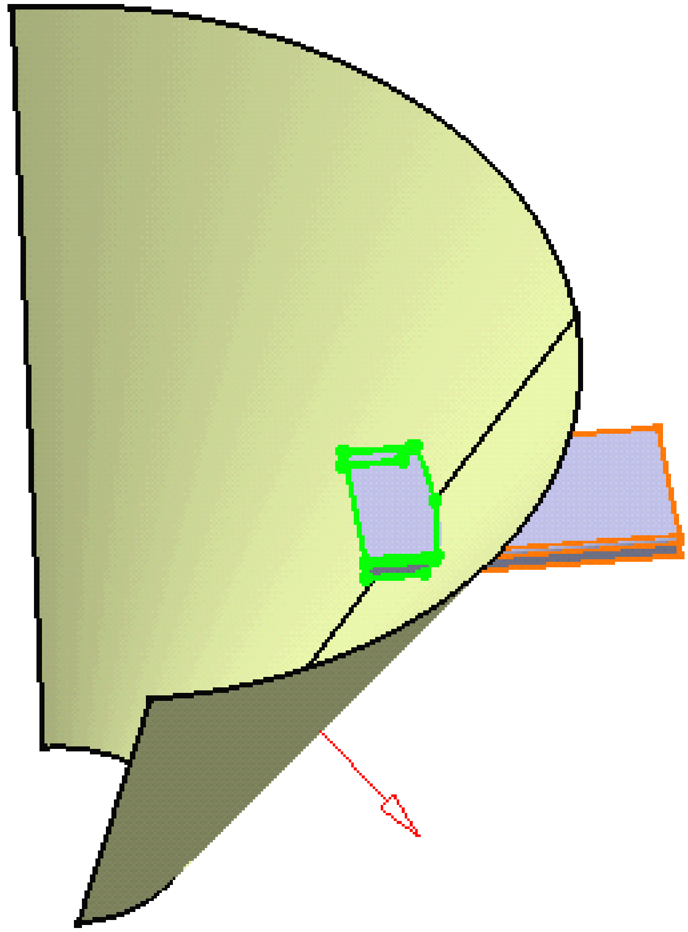
3. Select the **Reference Side** of the element to split.

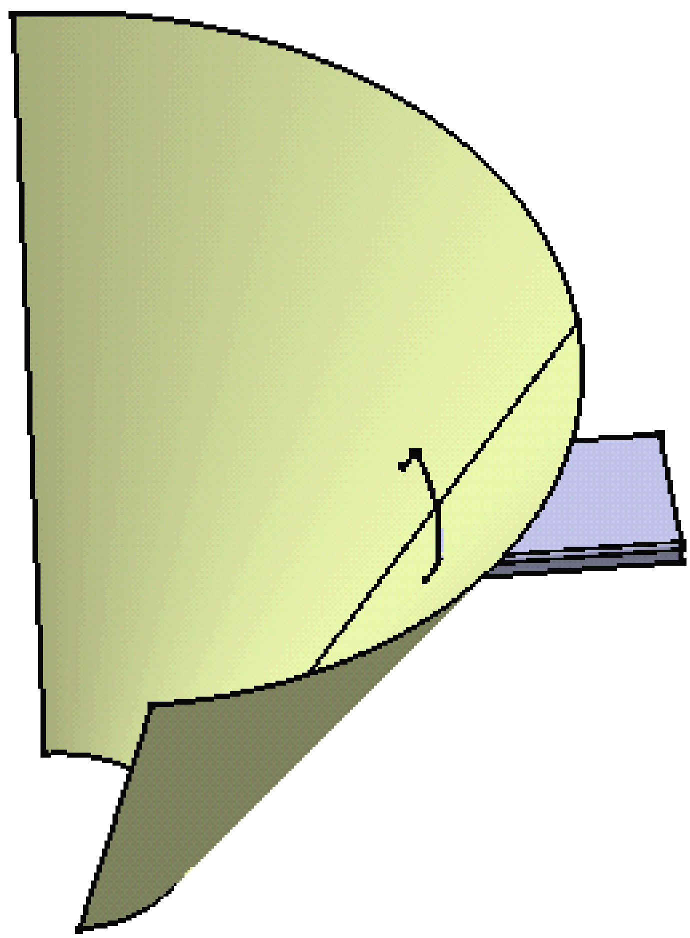
The red arrow shows the side to keep



4. Click the **Inverse Side** button to keep the other side.

5. Click **Apply** to preview the split: the side to be removed is highlighted in green.





6. Click **OK** to split the element.

The created element (identified as Split.xxx) is added to the specification tree.

You can create a split using the unfolded view of the part.

Stamping

This section explains and illustrates how to create and use various kinds of stamps.

- Stamps must be created on walls, or walls on edge, except for the stiffness rib that is to be created on a bend.
- If a stamp is created over the limit between several supports, such as walls, bends, and so forth, this stamp is not visible on the unfolded view.
- When unfolding a part, only the largest imprint of the stamp is retained on the stamped wall.
- Stamps cannot be created on an unfolded part.

Creating Standard Stamping Features
Creating User-Defined Stamping Features

Creating Standard Stamping Features

This section explains and illustrates how to create and use various kinds of stamps.
The table below lists the information you will find.

Create a point stamp: select a point on a face, and set the stamping parameters.

Create an extruded hole: select a point on a face, and set the stamping parameters.

Create a curve stamp: select a sketch, and set the stamping parameters.

Create a surface stamp: select a sketch, and set the stamping parameters.

Create a bridge: select a point on a face, set the stamping parameters, and select an edge to give the bridge orientation.

Create a louver: select a sketch, and set the stamping parameters.

Create a stiffening rib: select the external surface of a bend, and set the stamping parameters.

Creating a Point Stamp

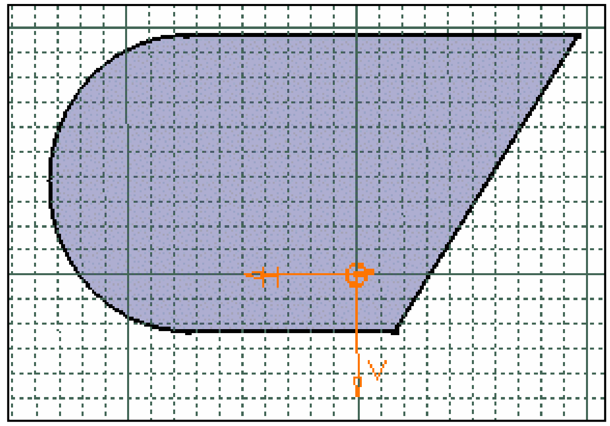
This task shows you how to create a point stamp by specifying the punch geometrical parameters.

Open the Stamping.CATPart document from the samples directory.

1. Click the **Point Stamp** icon .

2. Select a point on the top face.

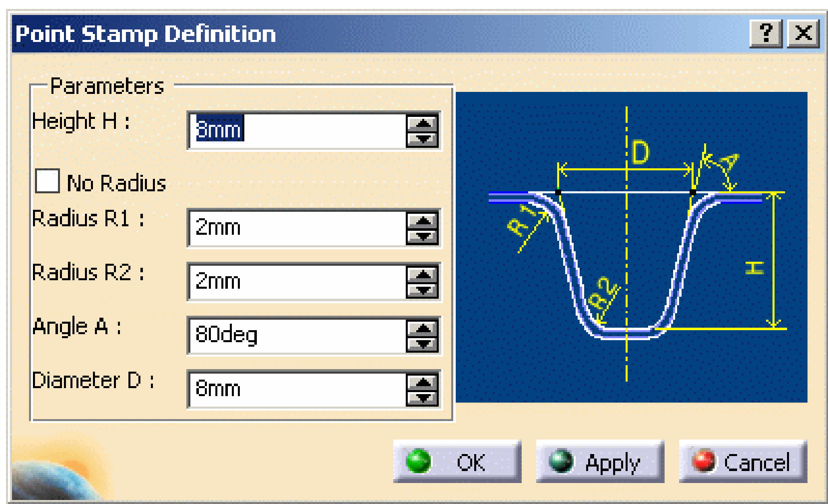
A grid is displayed to help you position the point stamp.



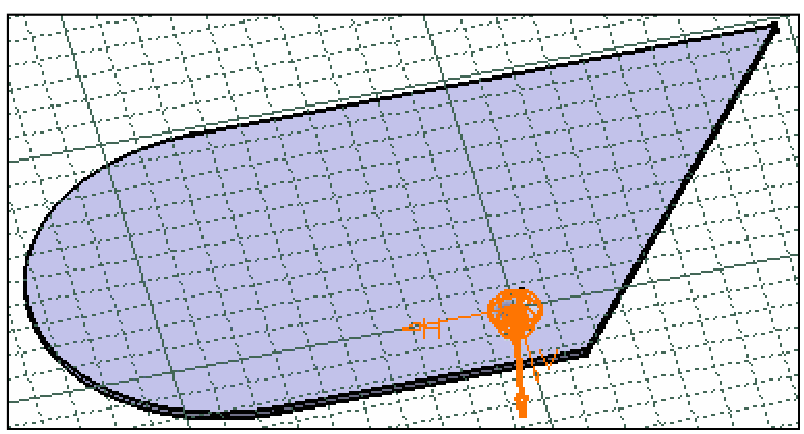
The Point Stamp Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

- Height H
- Radius R1
- Radius R2
- Angle A
- Diameter D

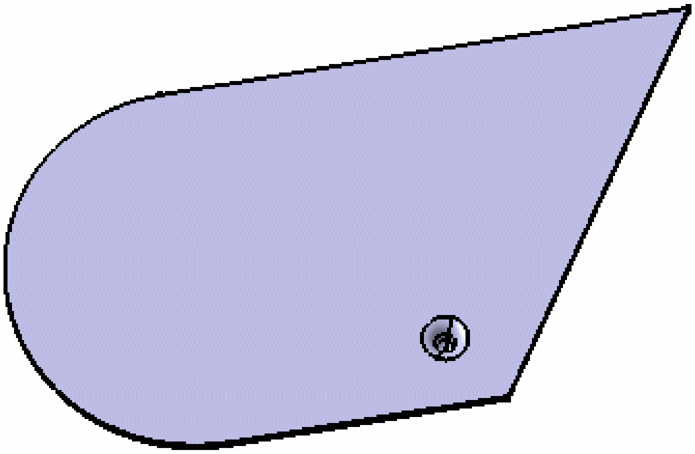


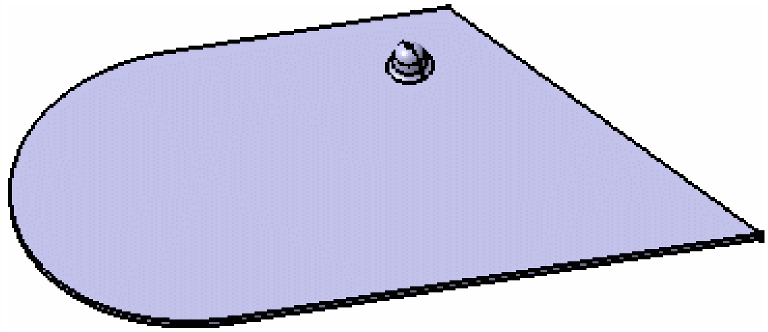
4. Click **Apply** to preview the point stamp.



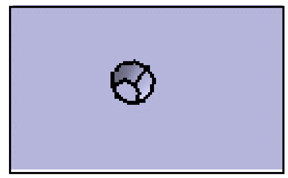
5. Click **OK** to validate.

The point stamp (identified as Point Stamp.xxx) is created and the specification tree is updated accordingly.





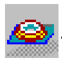
Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the bridge stamp without a fillet.



Creating an Extruded Hole

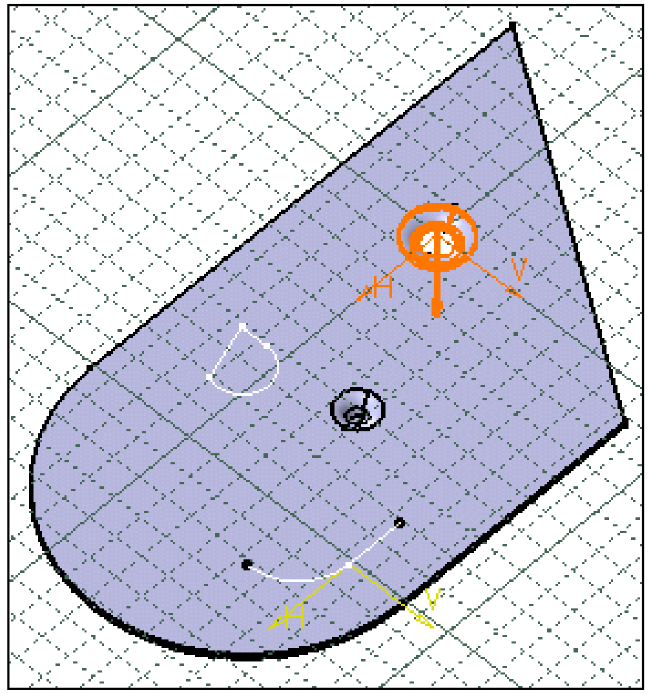
This task shows you how to create an extruded hole by specifying the punch geometrical parameters.

The Stamping.CATPart document is still open from the previous task.
If not, open the Stamping8.CATPart document.

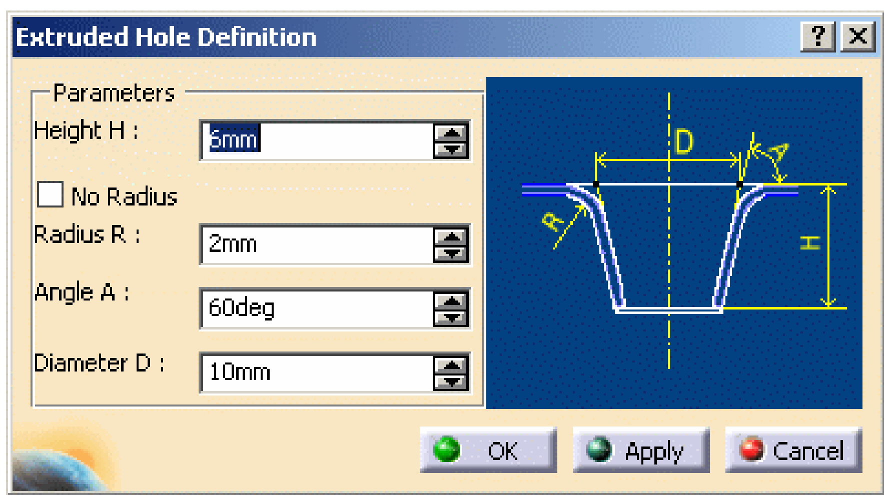
1. Click the **Extruded Hole** icon .

2. Select the surface where you want to place the hole.

   A grid is displayed to help you position the flanged hole and the Flanged Hole Definition dialog box opens, providing default values.



3. Change the value in the different fields, if needed:

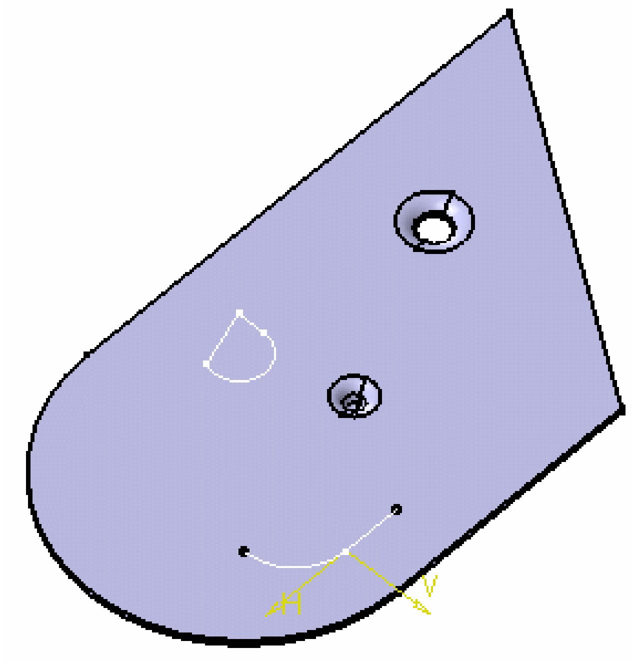


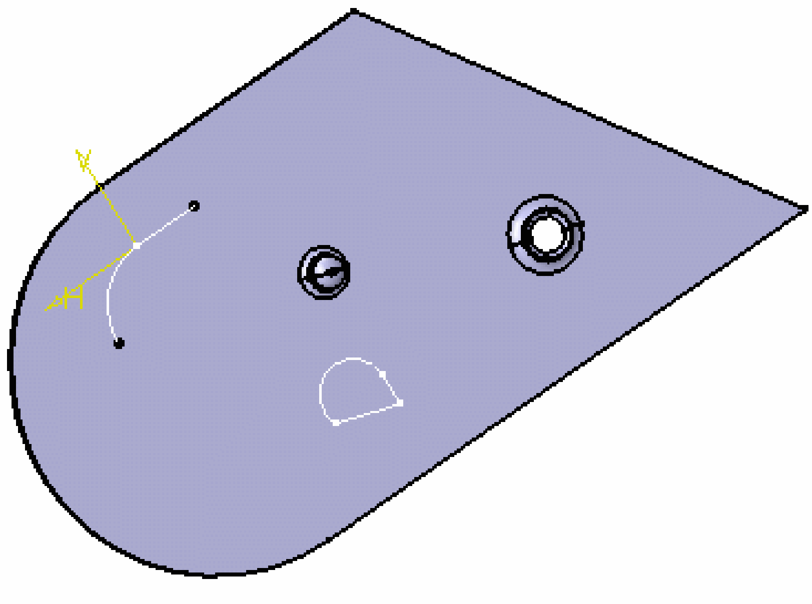
- Height H
- Radius R
- Angle A
- Diameter D

4. Click **Apply** to preview the flanged hole.

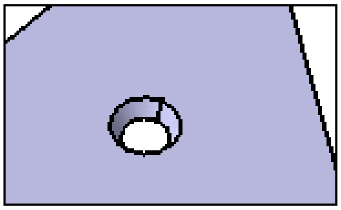
5. Click **OK** to validate.

The extruded hole (identified as Extruded Hole.xxx) is created and the specification tree is updated accordingly.





Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the point stamp without a fillet.



Creating a Curve Stamp

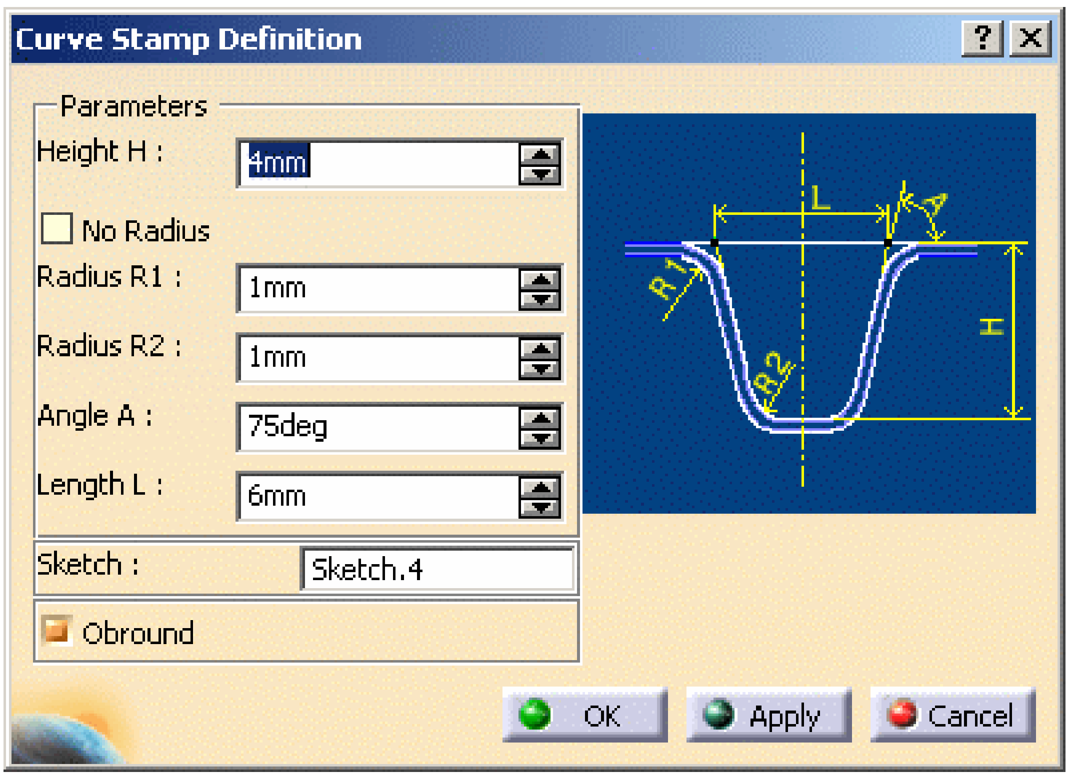
This task shows you how to create a curve stamp by specifying the punch geometrical parameters.

The Stamping.CATPart document is still open from the previous task.
If not, open the Stamping3.CATPart document from the samples directory.

1. Click the **Curve Stamp** icon .

2. Select **Sketch-for-Curve-Stamp**, the curve previously defined.

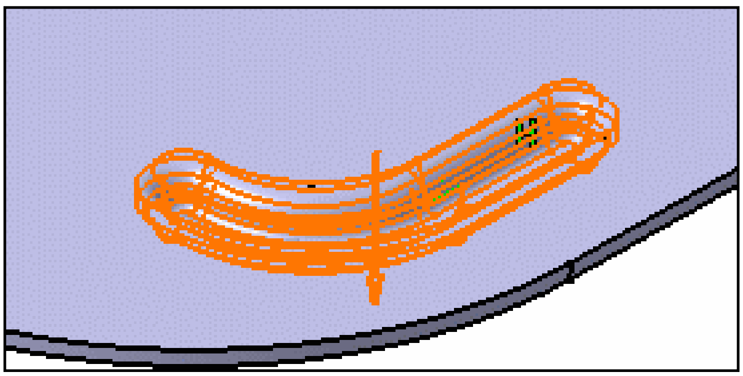
The Curve Stamp Definition dialog box opens, providing default values.



3. Change the value in the different fields, if needed:

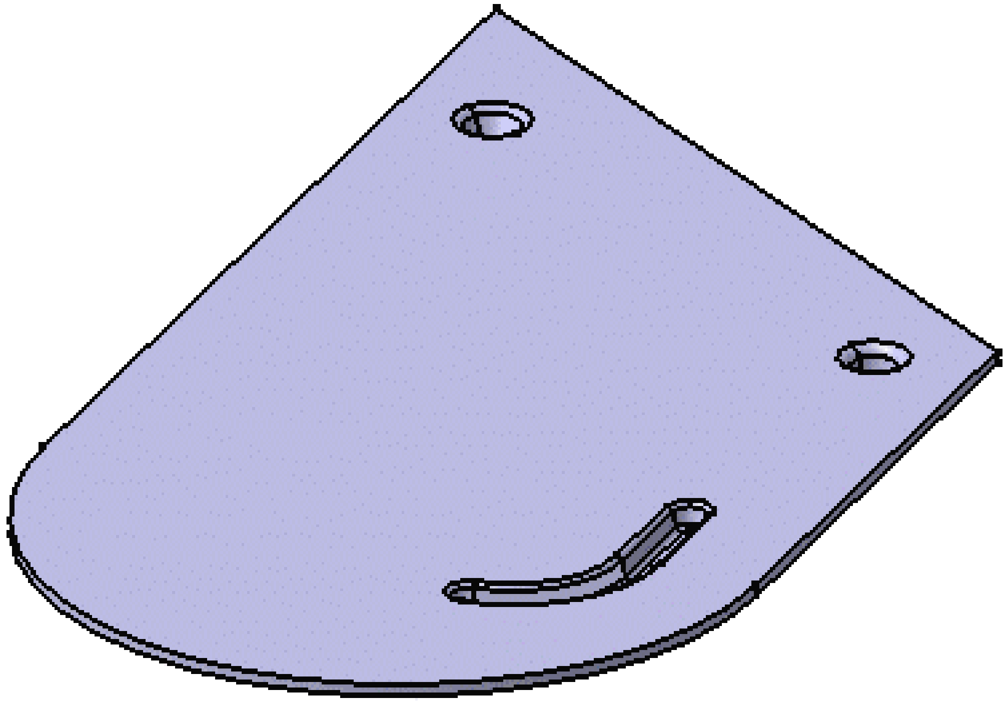
- Height H: the total height
- Radius R1: the outer bend radius
- Radius R2: the outer bend radius
- Angle A: the stamping draft angle
- Length L: the stamps' maximum width

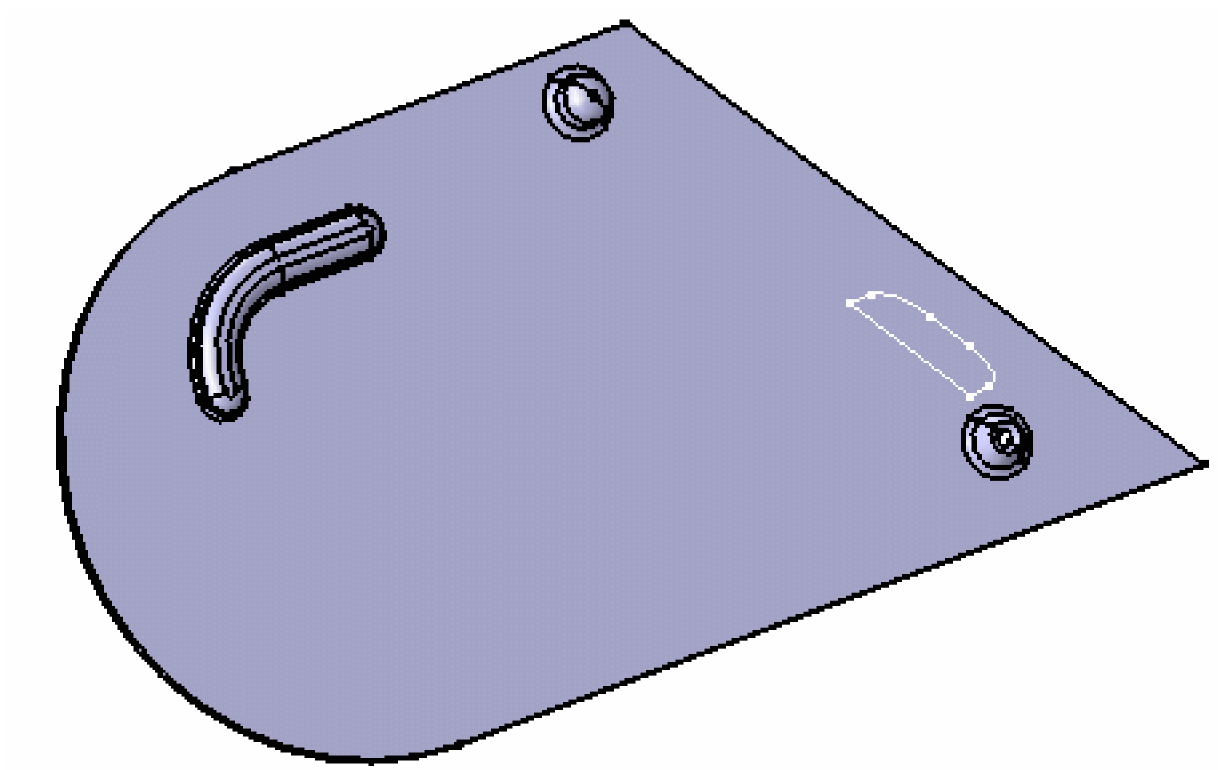
4. Click Apply to preview the curve stamp.



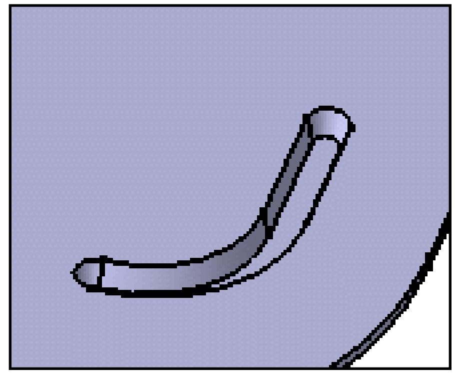
5. Click **OK** to validate.

The curve stamp (identified as Curve Stamp.xxx) is created and the specification tree is updated accordingly.

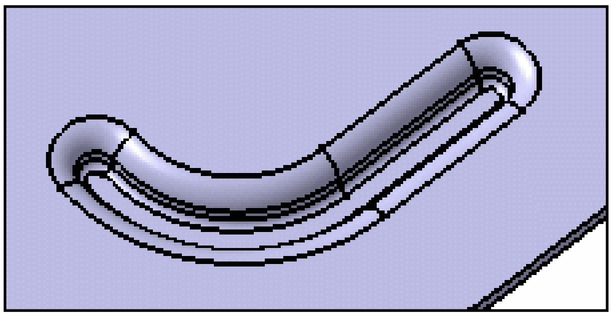




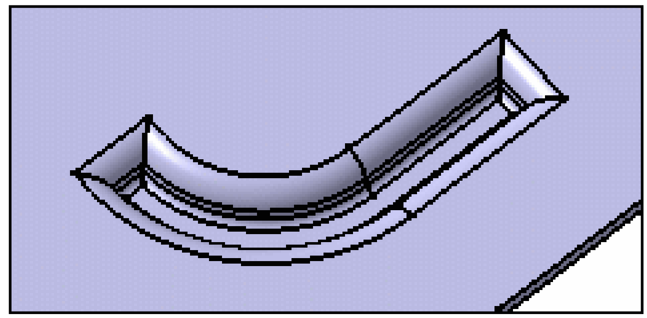
- Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the curve stamp without a fillet.



- Check the **Obround** option to round off the edges of the curve stamp.



Obround option checked



Obround option unchecked

Creating a Surface Stamp

This task shows you how to create a surface stamp by specifying the punch geometrical parameters.

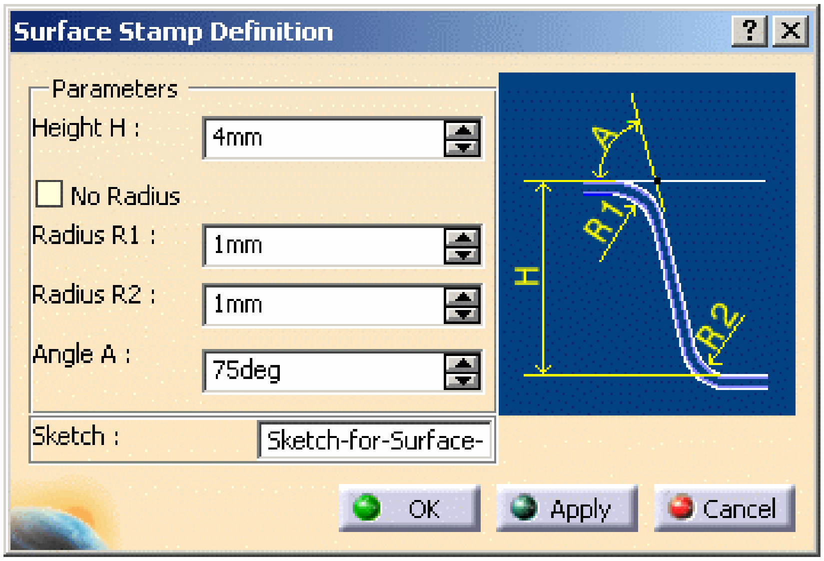
The Stamping.CATPart document is still open from the previous task.

If not, open the Stamping4.CATPart document from the samples directory.

1. Click the **Surface Stamp** icon .

2. Select **Sketch-for-Surface-Stamp**, the profile previously defined.

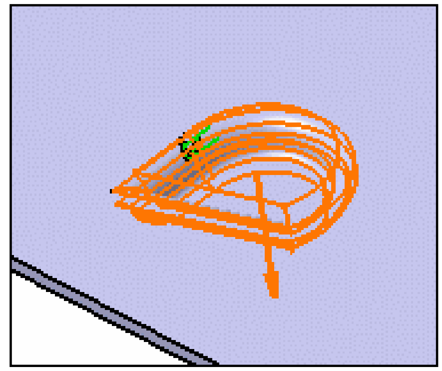
The Surface Stamp Definition dialog box opens, providing default values.



3. Change the value in the different fields, if needed:

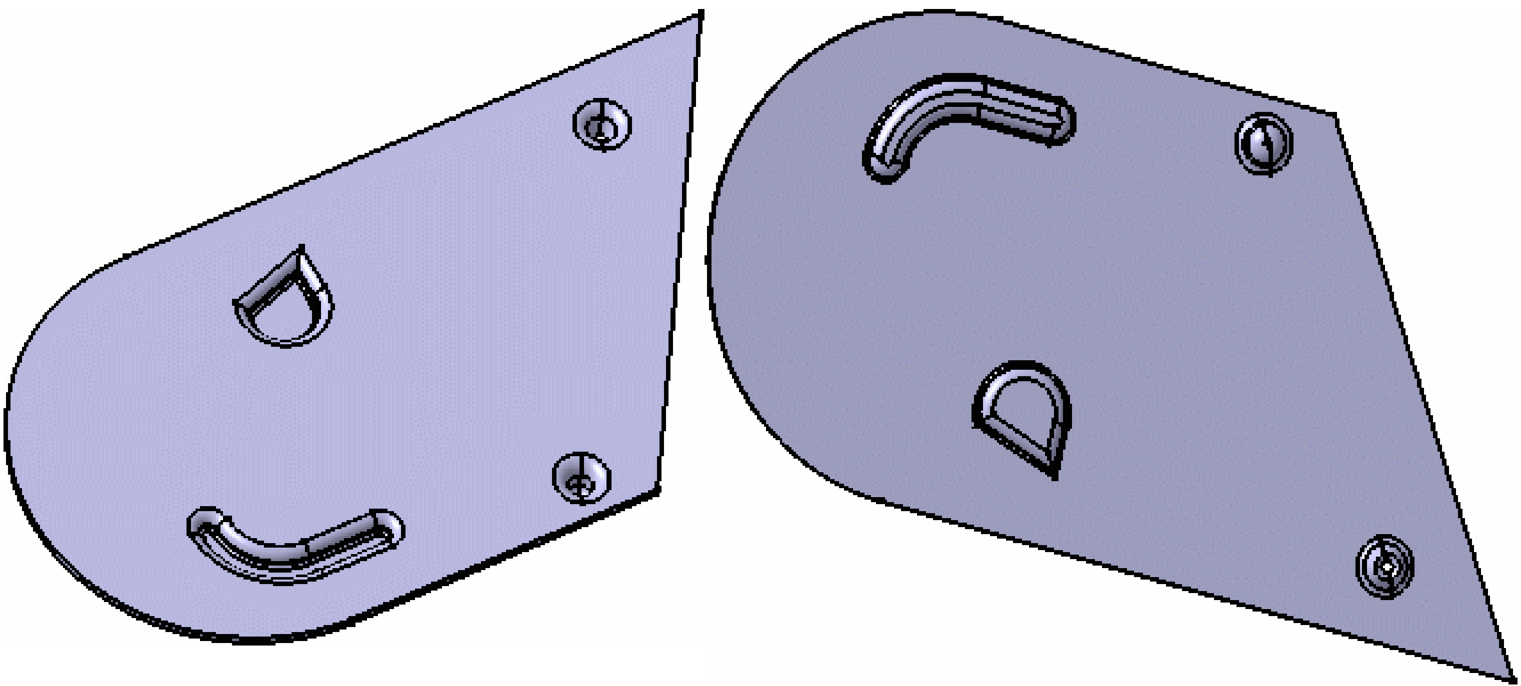
- Height H
- Radius R1
- Radius R2
- Angle A

4. Click **Apply** to preview the surface stamp.

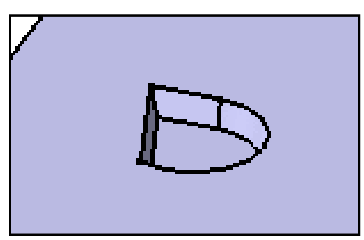


5. Click **OK** to validate.

The surface stamp (identified as Surface Stamp.xxx) is created and the specification tree is updated accordingly.



Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the surface stamp without a fillet.



Creating a Bridge

This task shows you how to create a bridge by specifying the punch geometrical parameters.

The Stamping.CATPart document is still open from the previous task.
If not, open the Stamping5.CATPart document from the samples directory.

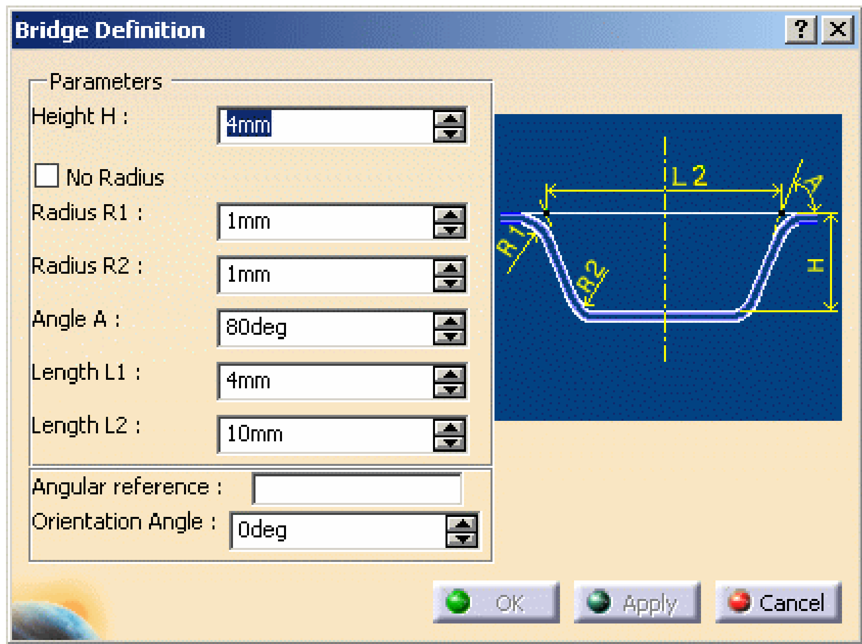
1. Click the **Bridge** icon .

2. Select a point on the top face where you want to place the bridge.

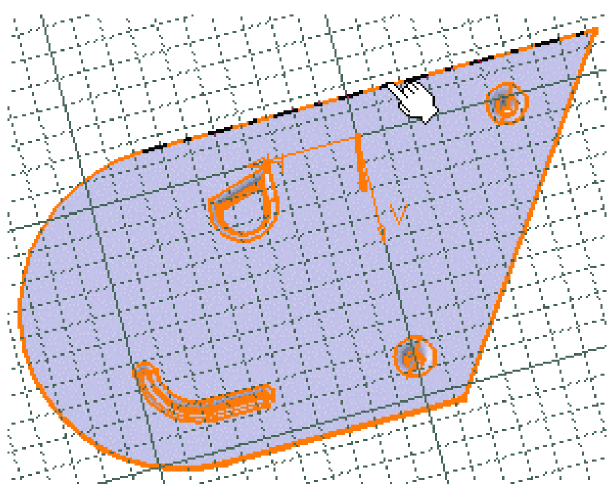
The Bridge Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

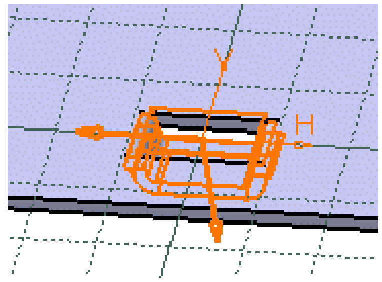
- Height H
- Radius R1
- Radius R2
- Angle A
- Length L1
- Length L2



4. Select an edge to give the direction of the bridge.

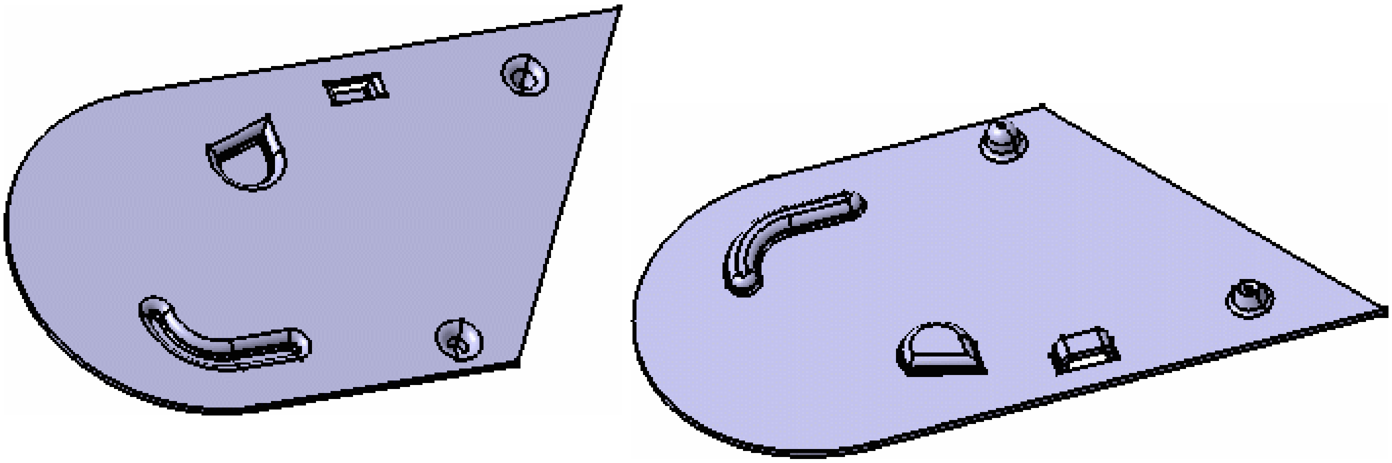


5. Click Apply to preview the bridge.

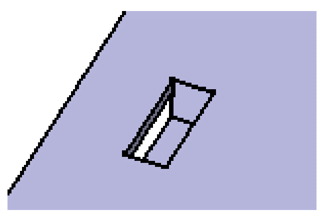


6. Click **OK** to validate.

The bridge (identified as Bridge.xxx) is created and the specification tree is updated accordingly.



Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the bridge stamp without a fillet.



Creating a Louver

This task shows you how to create a louver by specifying the punch geometrical parameters.

Open the Stamping6.CATPart document from the samples directory.

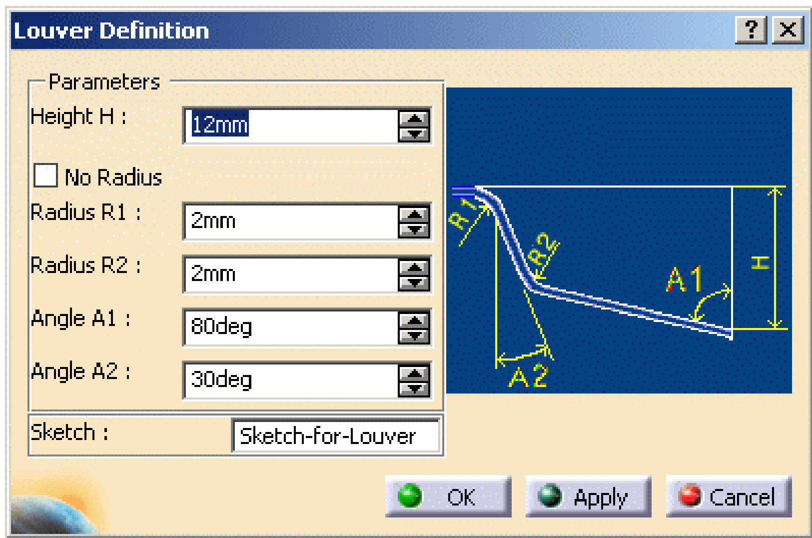
1. Click the **Louver** icon .

2. Select **Sketch-for-Louver**, a profile previously defined on **Wall.2**. The **Louver Definition** dialog box opens, providing default values.

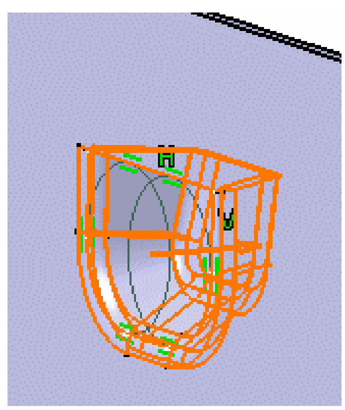
The louver opening face is represented in the sketch by the element that does not present any tangency continuity with the other lines/curve segments of the sketch. In case there are several non-continuous elements, the first one is used as the opening face.

3. Change the value in the different fields, if needed:

- Height H
- Radius R1
- Radius R2
- Angle A1
- Angle A2

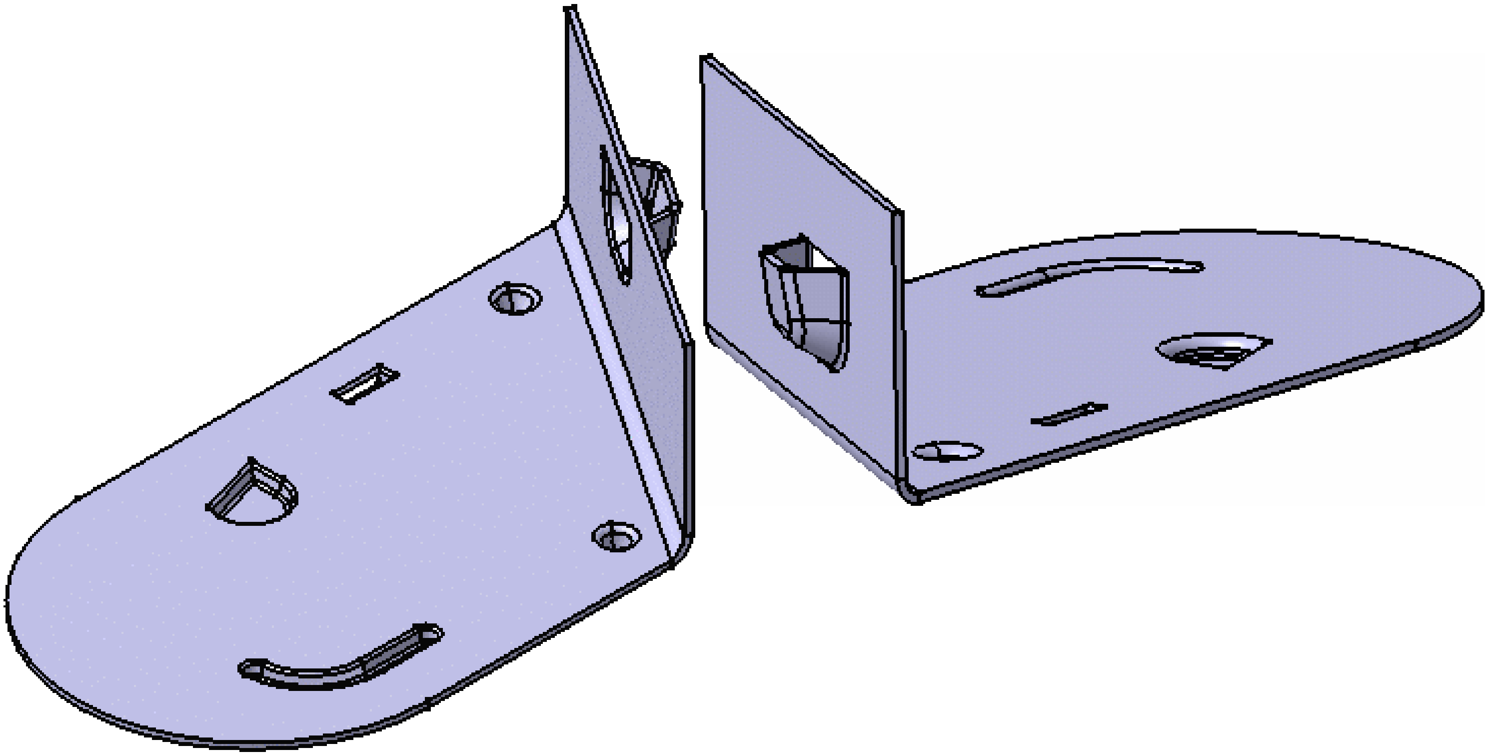


4. Click **Apply** to preview the louver.

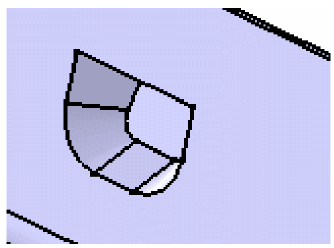


5. Click **OK** to validate.

The louver (identified as Louver.xxx) is created and the specification tree is updated accordingly.



Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the louver stamp without a fillet.



Creating a Stiffening Rib

This task shows you how to create a stiffness rib by specifying the punch geometrical parameters.

Open the Stamping7.CATPart document from the samples directory.

1. Click the **Stiffness Rib** icon  .

2. Select the external surface of Bend.1, where you want to place a stiffener.

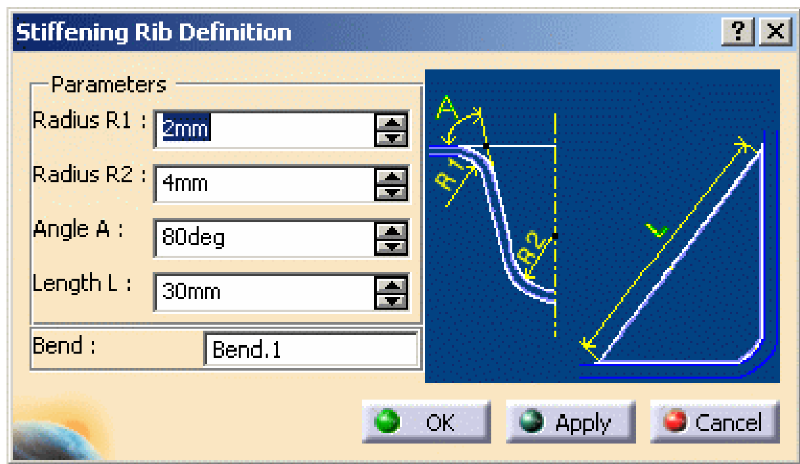
   Note that the stiffener will always be centered on the bend radius, wherever the point may be along the curve.

A grid is displayed.

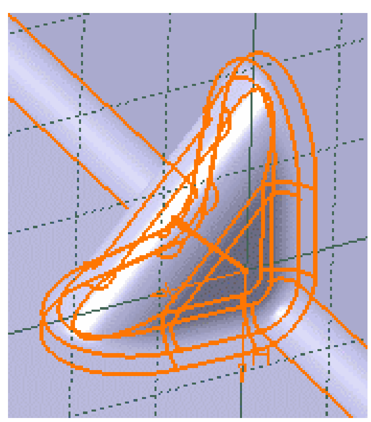
The Stiffening Rib Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

- Radius R1
- Radius R2
- Angle A
- Length L

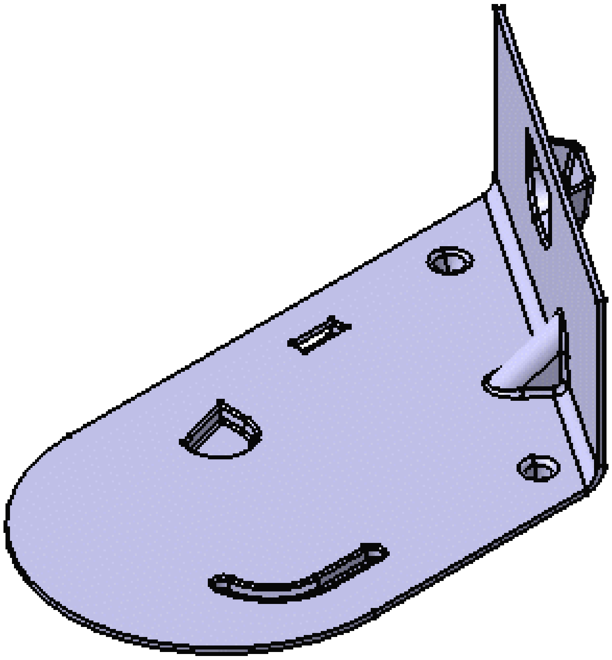
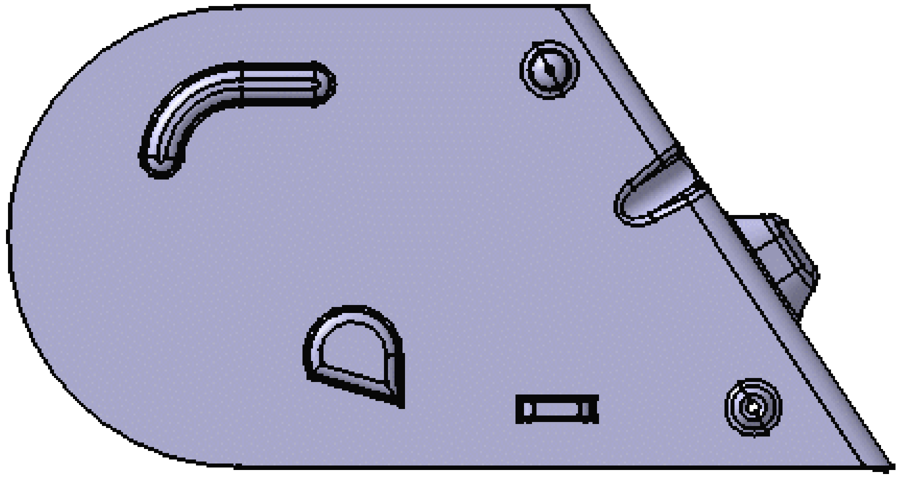


4. Click **Apply** to preview the stiffness rib.



5. Click **OK** to validate.

   The stiffening rib (identified as Stiffnening Rib.xxx) is created and the specification tree is updated accordingly.

Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the stiffening rib without a fillet.

Creating User-Defined Stamping Features

Two user-defined stamping features are available:

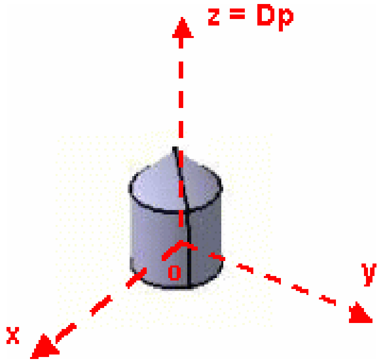
- Create a punch and a die: define the punch and die features, select a wall, choose the punch and die as stamping elements, select an edge on the wall and give an angle for orientation purposes.
- Open and cut faces: define the punch, select a wall, define the cutting faces and opening faces of the punch, select an edge on the wall and give an angle for orientation purposes.
- Edit a user-defined stamp: double-click the existing stamp and change its type, or select, or remove cutting and opening faces

What You Should Know

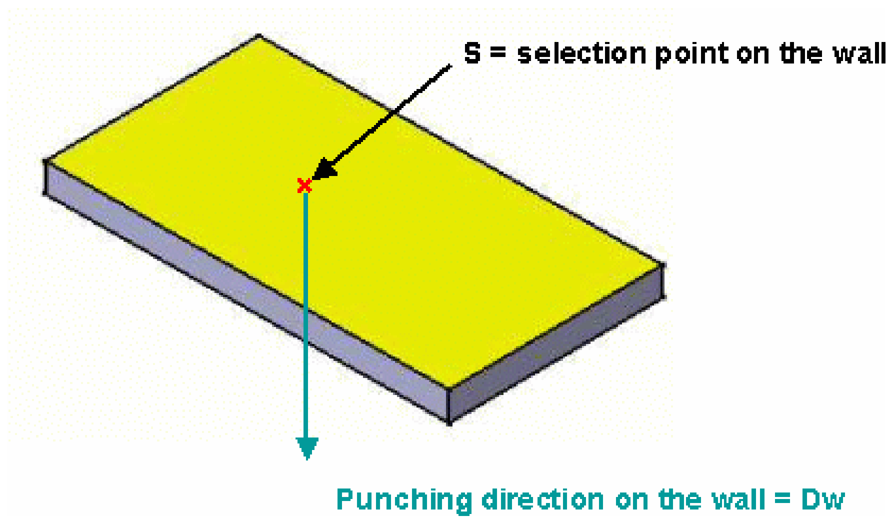
In both tasks illustrating either a stamp based on a punch and a die, or a punch with cutting and opening faces, the punch positioning is defined as below:

Defining the Punch in Relation to the Wall to be Stamped

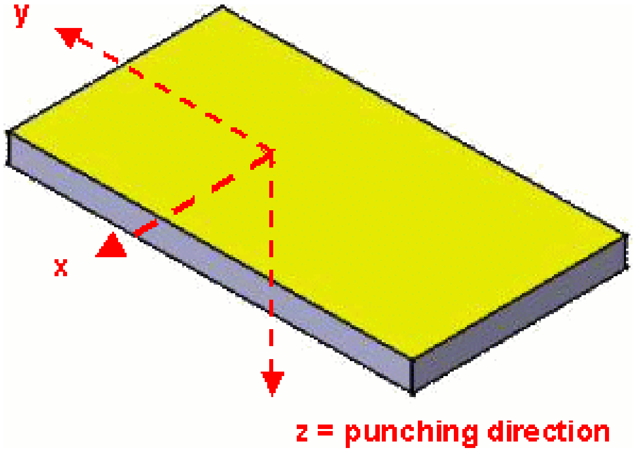
The punch is defined within the absolute (default) axis-system of the .CATPart document. (o, x, y, z) is the axis associated with the punch. The punching direction on the punch (*Dp*) must be equal to z.

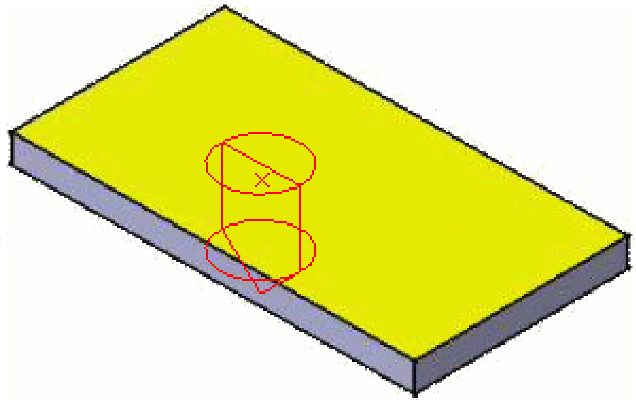


The punching direction on the wall (*Dw*) is normal to the selected wall face, and is oriented from the selected wall face towards the opposite face.



The punch is applied matching *Dp* on *Dw* and matching the punch's (x, y) plane onto the selected wall face:



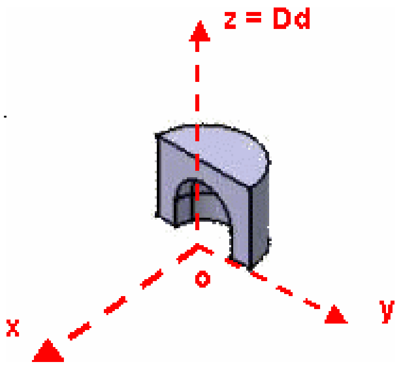


Defining the Die in Relation to the Wall to be Stamped

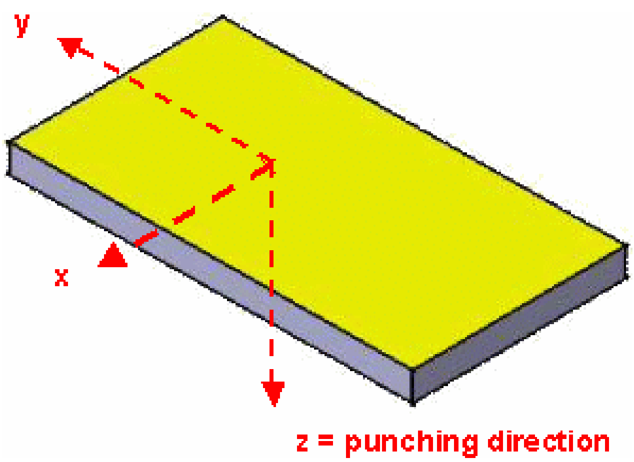
This is useful only when defining a punch a die, and does not apply to punches with cutting and opening faces.

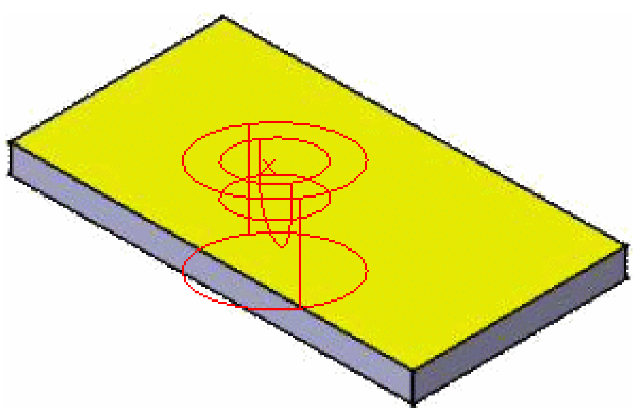
The die is also defined within the absolute (default) axis-system of the .CATPart document. (o, x, y, z) is the axis associated with the punch. The punching direction on the die (*Dd*) must be equal to z.

The illustration is a section view of the die.



The die is applied matching *Dd* on *Dw* and matching the die's (x, y) plane onto the selected wall face:





Creating a Punch with a Die

This task explains how to create a stamp from punch and die features.

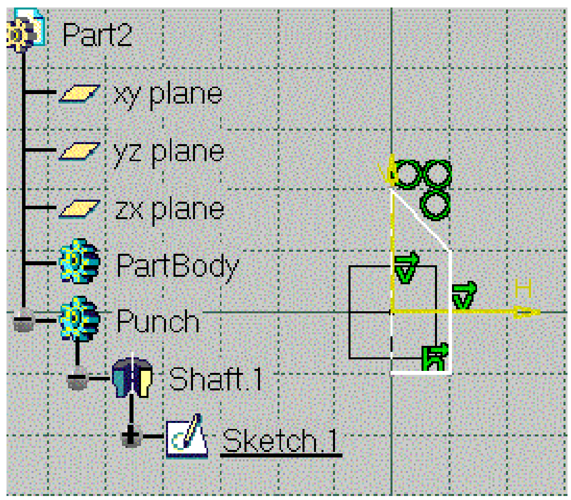
First, you will define a punch and a die in Part Design, in the absolute axis-system.
Then, in a Sheet Metal part, you will bring the punch and the die features (and their axis system) to a point you have selected. If necessary, you will define a rotation of the axis system from a reference line.

This user-defined stamping cannot be combined with the Opening and Cutting Faces approach.

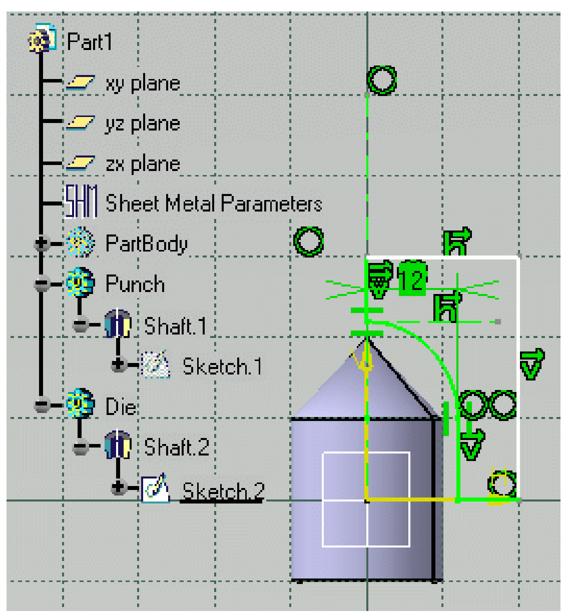
All .CATParts are available from the samples directory (PunchDie1.CATPart, Punch1.CATPart and Die1.CATPart or NEWPunchDie1.CATPart, NEWPunch1.CATPart and NEWDie1.CATPart for Generative Sheetmetal Design or Aero_PunchDie1.CATPart, Aero_Punch1.CATPart and Aero_Die1.CATPart for Aerospace Sheetmetal Design).

1. Start the Part Design application.
2. Insert a PartBody (menu **Insert -> Body**) to define the punch.
3. Enter the sketcher select the yz plane, and draw the profile of the punch, and a rotation shaft.

The punch must be oriented as described in Defining the Punch in Relation to the base feature to be Stamped.



4. Return to the 3D space and create the punch using the Shaft icon.

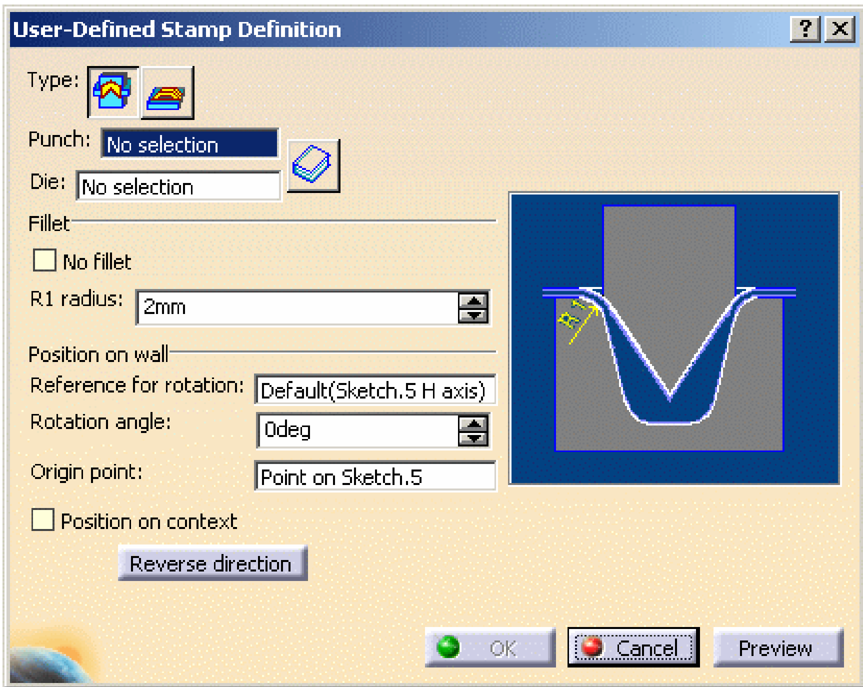


5. Repeat from step 2 to step 4 to define the die, making sure that it is oriented as described in Defining the Die in Relation to the base feature to be Stamped.

6. Return to the Sheet Metal application, and if needed, use the **Define In Work Object** on the PartBody containing the wall or the base feature to be stamped.

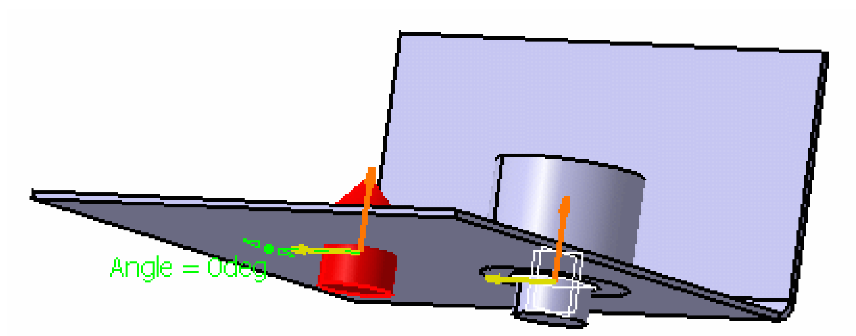
7. Click the **User Stamping** icon from the Stamping tool bar and select a base feature, or a face where the stamping is to be created. This base feature or face is used to define the stamping location and direction, by matching the punch's origin to the selected point on the base feature.

The User Defined Stamp Definition dialog box is displayed:



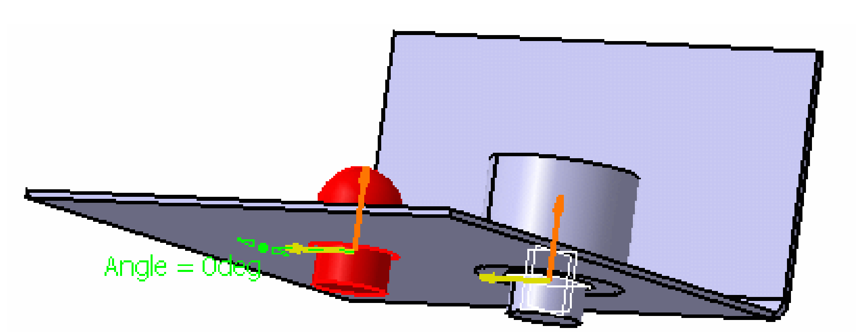
8. Make sure the **With die** icon is pressed down and select the Punch feature from the specification tree.

The punch's positioning is previewed in the geometry.

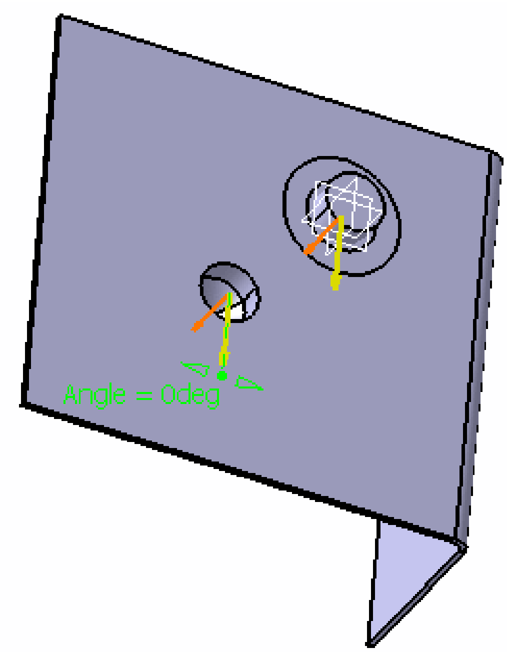


9. Select the Die feature.

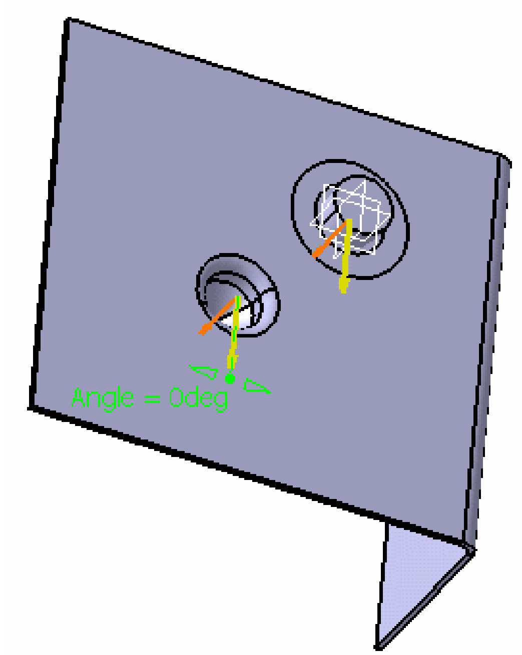
The die's positioning is previewed in the geometry as well.



10. Check the No Fillet button is you do not wish the stamp to be filleted, or set the radius value if you wish the stamp to be filleted.



Stamp without fillet



Stamp with fillet

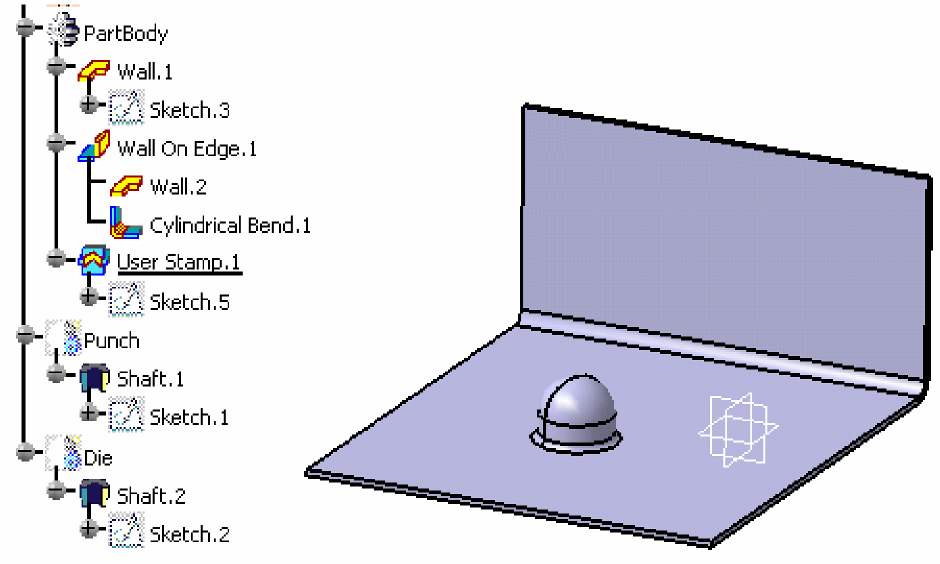
11. If needed, define the stamp's positioning on the selected base feature by choosing:

- a **Reference for rotation**: by default, it is the sketch axis, but you can also select any line or edge on base feature.
- a **Rotation angle** value: you can either enter a value in the dialog box, or use the manipulator in the geometry to define this value.
- a new **Origin point** on the base feature to coincide with the punch's point of origin.

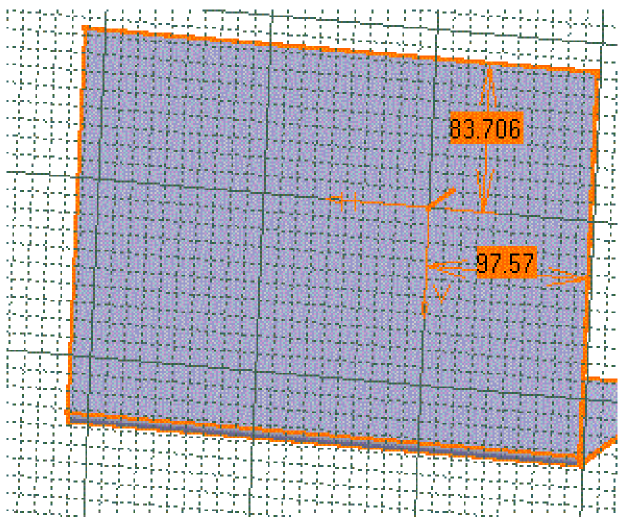
This is especially useful for non-circular stamps, but you can very well create the stamp as is, without further positioning.

13. Click OK to validate and create the stamping.

By default the Punch and Die parts are set in No Show mode when clicking OK to create the stamp on the base feature.

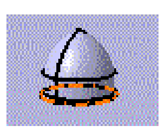


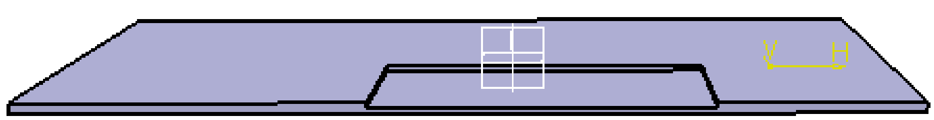
- **Radius** is the radius of the bend between the stamping and the base feature.
- **Punch** and **Die** are the bodies you have defined previously. If the punch and the die are in another CATPart document, activate this document before clicking the punch or the die.
- If you select two reference lines in addition to the plane, this will create two editable constraints to position the stamping. These constraints are editable.



- A user-defined stamping can be edited (punch, die, position, constraints).
- As the punch and die are not symmetrical, you cannot create such features as a cutout, a hole, a corner, etc., on this kind of stamping.

- If you enter a punch and a die, the stamping is the difference of the shape of both features.





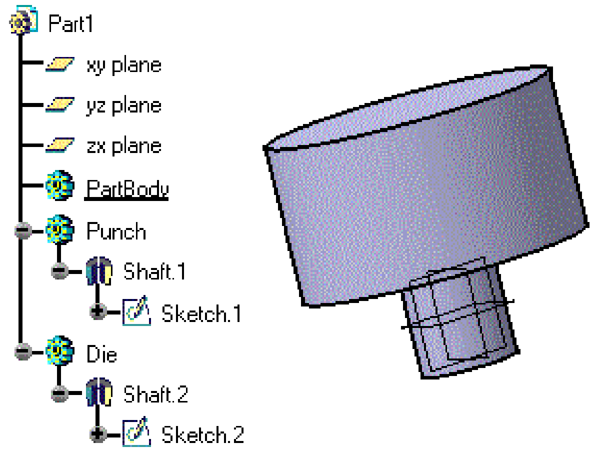
- The punch height cannot be superior to the base feature height, otherwise it is considered as a cutout.

- You may create a user-defined stamping from a punch only but you cannot create a fillet.

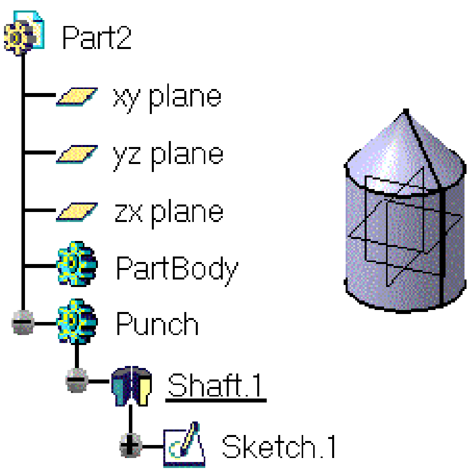
- Only the stamping sketch is displayed in unfolded views.

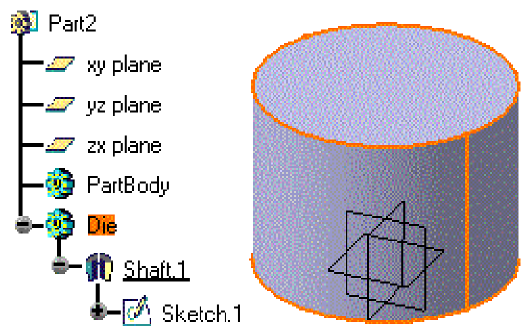
- The punch and die bodies can be defined in the Sheet Metal part where the stamping is to be created (see PunchDie1.CATPart or NEWPunchDie1.CATPart in the samples directory).

In this case, make sure you select the **Define In Work Object** on the PartBody containing the base feature to be stamped, prior to actually creating the stamp.



or as two separate Part Design parts (Punch1.CATPart and Die1.CATPart from the samples directory)





In this case, when selecting the punch or die feature, the system automatically copies this feature into the .CATPart document into which the base feature to be stamped is located.
A link is retained between the initial punch or die feature and its copy.

Opening and Cutting Faces

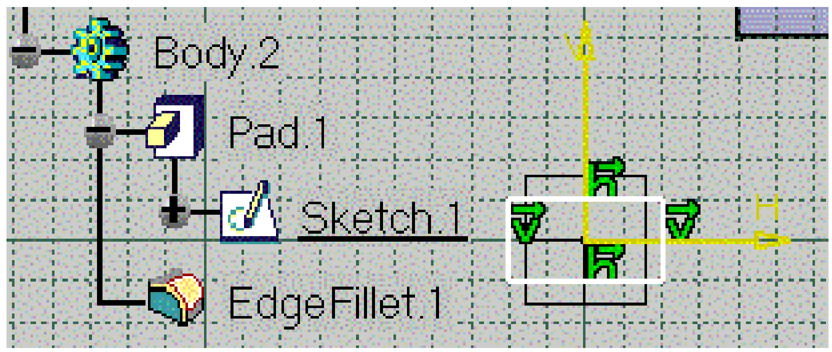
This task explains how to create a stamp from a punch feature with cutting and opening faces.

First, you will define a punch in Part Design, in the absolute axis system.
Then, in a Sheet Metal part, you will bring the punch feature (and its axis system) to a point you have selected. If necessary, you will define a rotation of the axis system from a reference line.

This user-defined stamping cannot be combined with the Punch and Die approach.

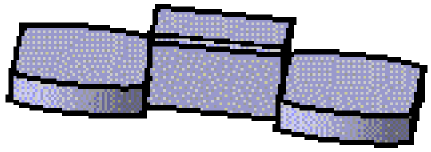
All CATParts, OpenFaces1.CATPart and CuttingFaces1.CATPart, are available from the samples directory.

1. Start the Part Design application.
2. Insert a PartBody (menu **Insert -> Body**) to define the punch.
3. Enter the sketcher select the yz plane, and draw the profile of the punch.
4. Return to the 3D space and create the punch using the pad icon and the fillet icon .

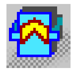


The punch must be oriented as described in Defining the Punch in Relation to the Wall to be Stamped.

- The punch can be defined in the Sheet Metal part where the stamping is to be created or in another part.
  In this case, when selecting the punch feature, the system automatically copies it into the .CATPart document into which the wall to be stamped is located.
  A link is retained between the initial punch feature and its copy.
- If you define a punch with cutting faces, they should come below the sheet.

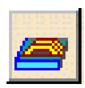


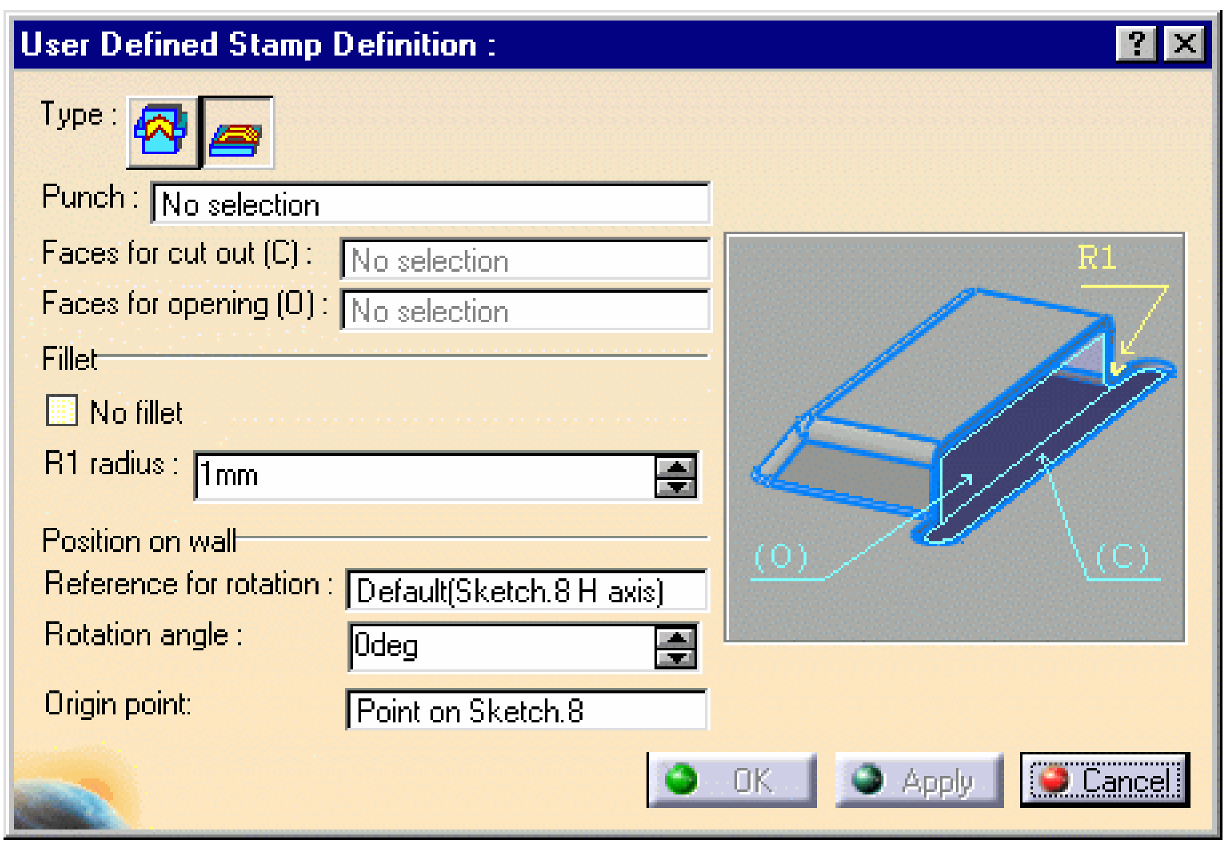
5. Return to the Sheet Metal application, and if needed, use the **Define In Work Object** on the PartBody containing the wall to be stamped.

6. Click the **User Stamping** icon  from the Stamping tool bar and select a wall or a face where the stamping is to be created.

   This wall or face is used to define the stamping location and direction, by matching the punch's origin to the selected point on the wall.

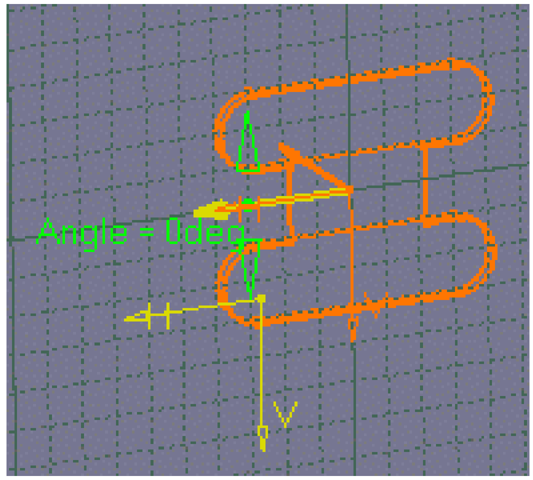
The User Defined Stamp Definition dialog box is displayed, along with a grid that will help you position the punch.

7. Click the **With cut-out and opening**  icon.



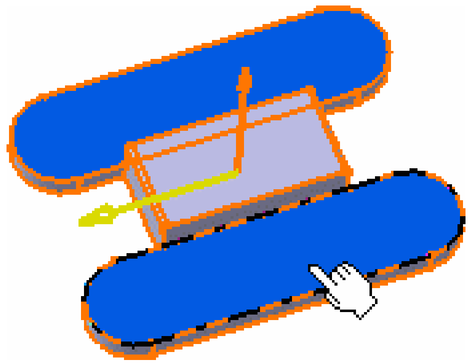
8. Select the punch (Body.2).

The punch is previewed on the wall.

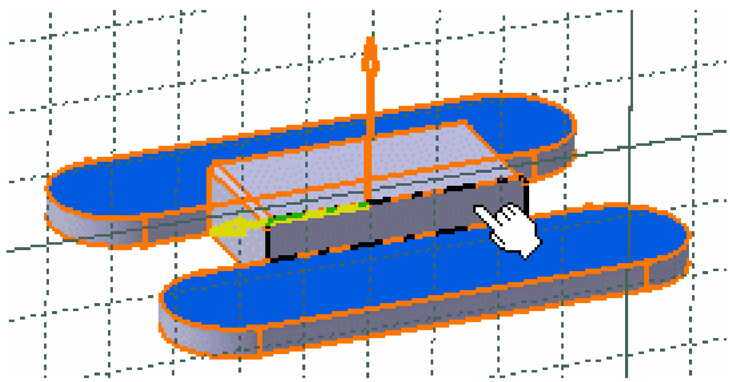


9. Select both top faces of the oblong features of the part (Pad.2 and Pad.3).

The **Faces for cut-out** field is updated in the dialog box, and now reads: **2 Faces**.

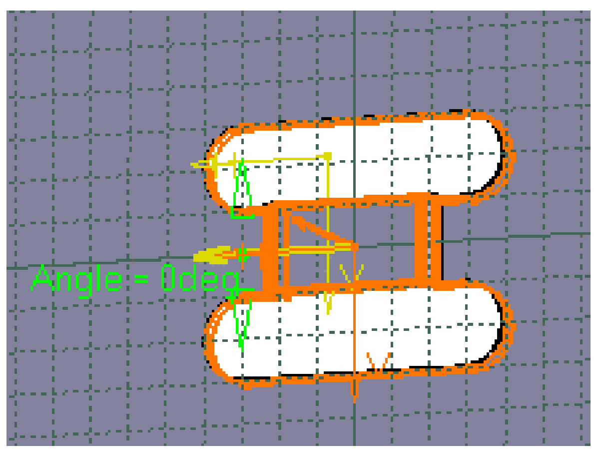


10. Click the **Faces for opening** field and select the lateral faces of the punch (Pad.1).



11. Click Apply.

The stamp is previewed with the opening faces:



12. Check the No Fillet button is you do not wish the stamp to be filleted, or set the radius value if you wish the stamp to be filleted.

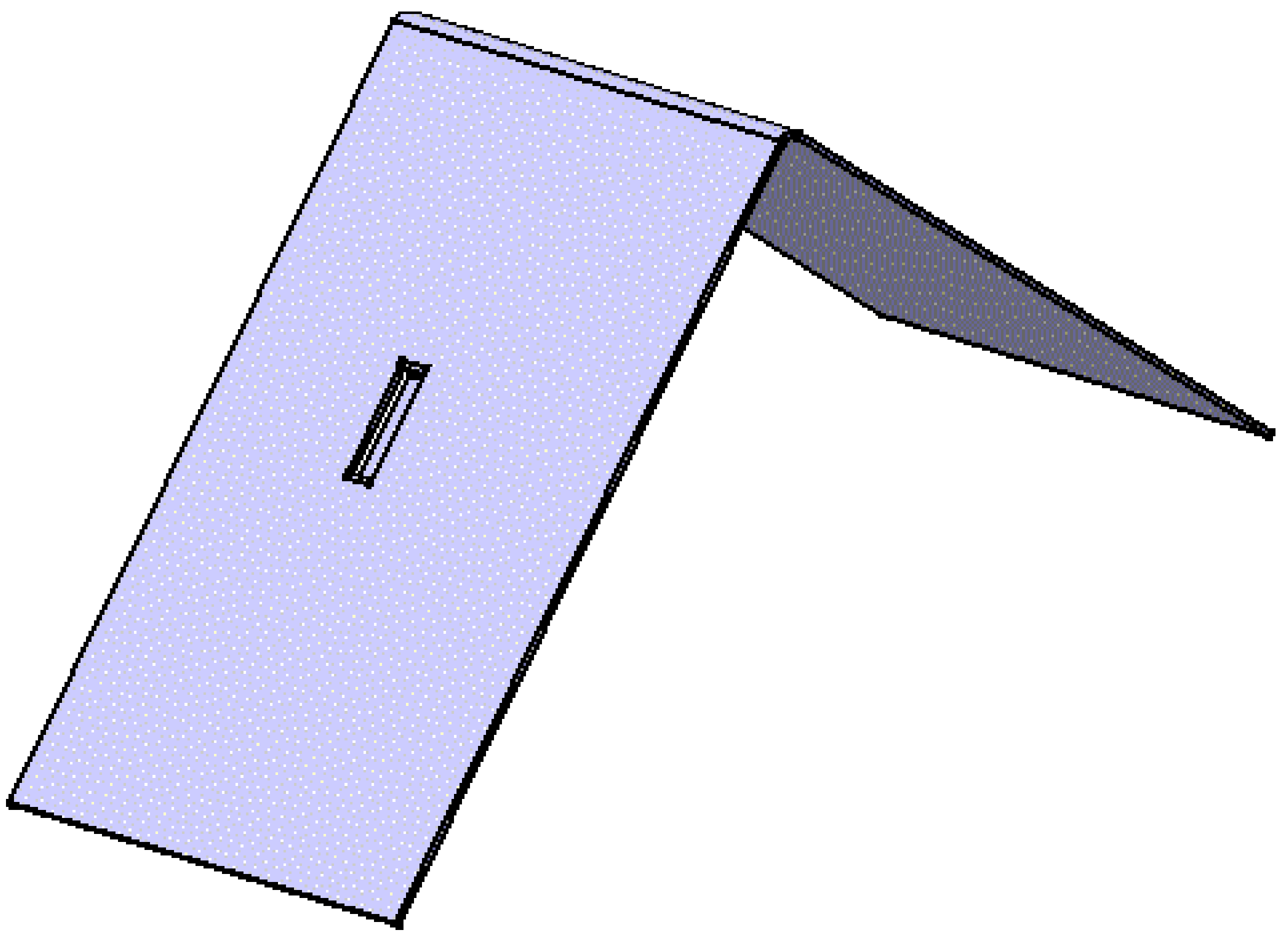
13. If needed, define the stamp's positioning on the selected wall by choosing:

- a **Reference for rotation**: by default, it is the sketch axis, but you can also select any line or edge on the wall.
- a **Rotation angle** value: you can either enter a value in the dialog box, or use the manipulator in the geometry to define this value.
- a new **Origin point** on the wall to coincide with the punch's point of origin.

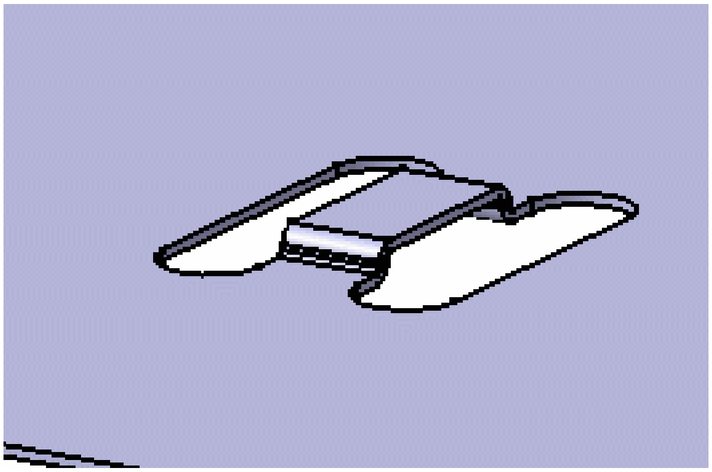
This is especially useful for non-circular stamps, but you can very well create the stamp as is, without further positioning.

14. Click OK to validate and create the stamping.

The stamp is automatically set in No Show mode.



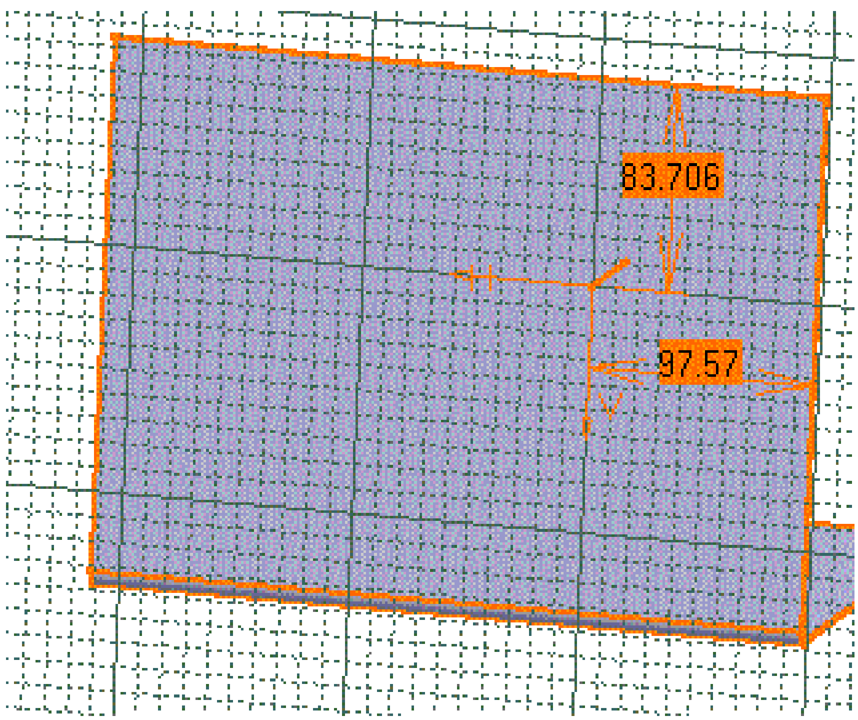
Stamping with opening faces



Stamping with opening and cutting faces

- **Radius** is the radius of the bend between the stamping and the wall.
- **Punch** is the body you have defined previously. If the punch is in another CATPart document, activate this document before clicking the punch.
- The **Faces for cut-out** and **Faces for opening faces** must be picked on the punch, not on the wall.
  If the punch is located into another .CATPart document, these faces must be picked on the copy of the punch where the wall to be stamped is located.

If you select two reference lines in addition to the plane, this will create two editable constraints to position the stamping. These constraints are editable.



- A user-defined stamping can be edited (punch, die, position, constraints)
- Check the **No fillet** option to deactivate the Radius R1 value, and to create the stamp without a fillet.

Editing User-Defined Stamps

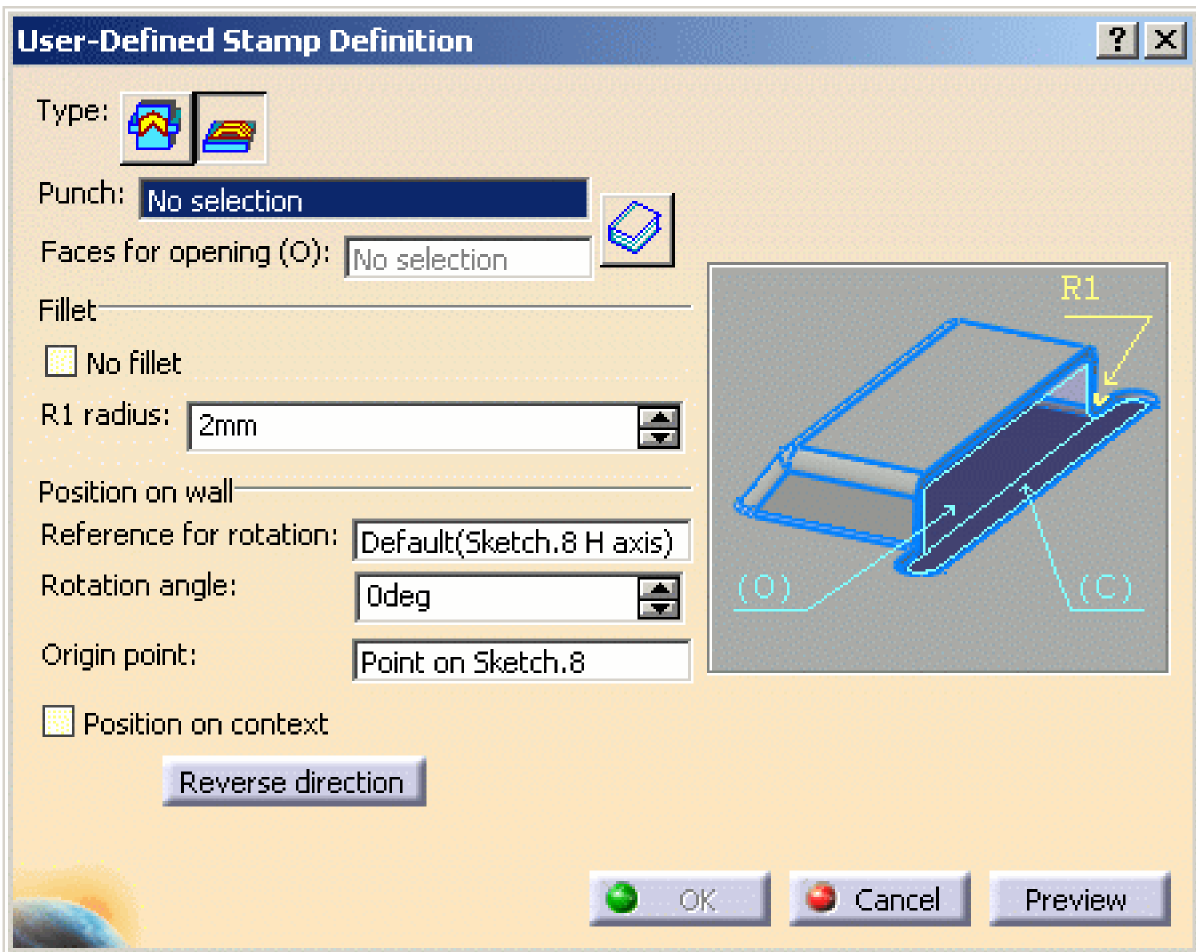
This task explains how to edit a user-defined stamp, that is:

- to change its type
- add or remove cutting and opening faces

To perform this scenario, you can open any .CATPart document containing a user-defined stamp.

1. Double-click the existing user-defined stamp from the specification tree.

The User Defined Stamp Definition dialog box is displayed.



2. Change stamp type using the icons:

- If you change from **With die** to **With opening**, the Die feature no longer is selected, and you need to select **Faces for opening**.
- If you change from **With opening** to **With die**, the punch faces no longer are selected and you may select a die feature if you wish (it is not compulsory).

Basically, only the punch remains selected.
If you are working with a punch with opening faces (**With opening** option) you may want to add or remove some opening faces:

3. Click in the **Faces for opening** field then:

- select a face in the geometry to add it to the already selected opening faces
- select an already selected face to remove it from the opening faces
- use the **Clear selection** contextual menu to remove all opening faces that have been previously selected.

4. Modify any other parameter as needed.

5. Click OK in the User Defined Stamp Definition dialog box to take these modifications into account.

The stamp is updated accordingly.

Patterning

This section explains and illustrates how to create various kinds of patterns on Sheet Metal parts.

Create rectangular patterns: select the element to be duplicated, set the patterning type, and its parameters, and the reference direction

Create circular patterns: select the element to be duplicated, set the axial reference parameters, the reference direction, and possibly the crown definition

Create user-defined patterns: select the element to be duplicated, and the positioning sketch and anchor point

To know more about patterns, refer to the *Part Design User's Guide*.

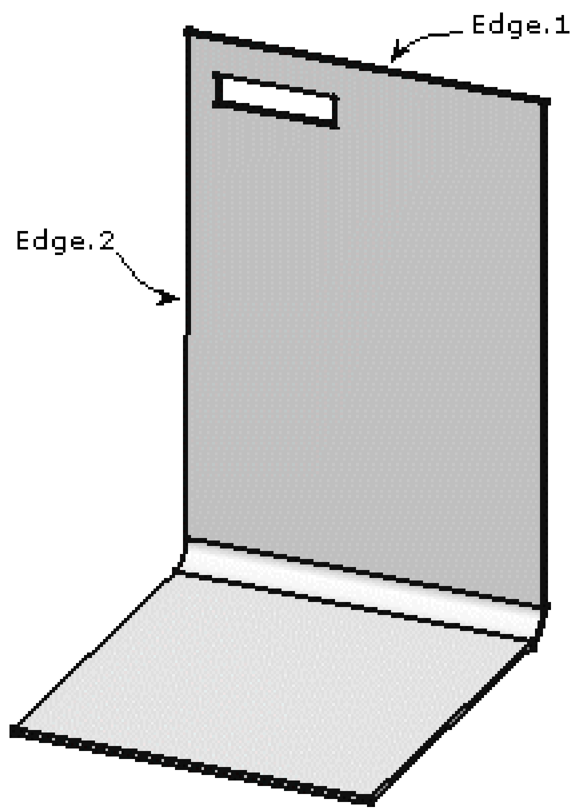
Creating Rectangular Patterns

In this task, you are going to create rectangular cutouts according to a pattern.
These features make the creation process easier.

In the Generative Sheetmetal Design workbench, you can only duplicate flanges, cutouts, holes, stamps (except stiffening ribs), stamps without fillet (radius=0) and Generative Sheetmetal Design patterns.
These features must lie on a unique and planar wall.

For the SheetMetal Design workbench, open the RectangularPattern1.CATPart document.
For the Generative Sheetmetal Design workbench, open the NEWRectangularPattern1.CATPart document.

The Sheet Metal part looks like this:



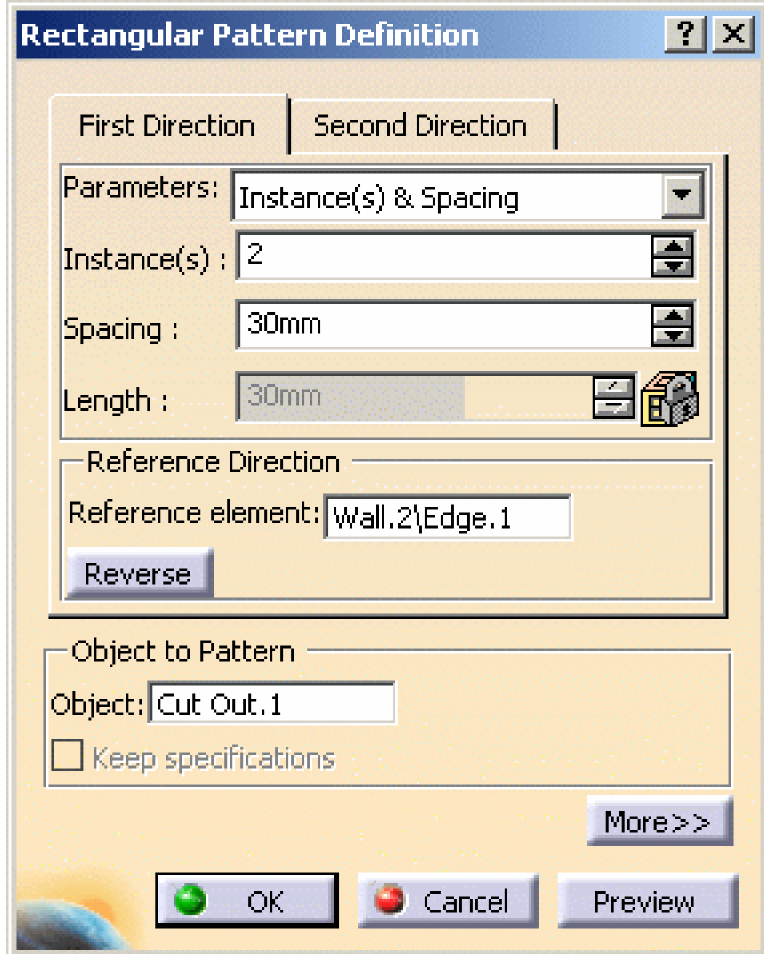
1. Select the rectangular cutout you want to duplicate.

2. Click the **Rectangular Pattern** icon .

The **Rectangular Pattern Definition** dialog box is displayed. Each tab is dedicated to a direction to define the location of the duplicated feature.

3. Set the specification for the First Direction by selecting the first edge (**Edge.1**) as shown, to specify the first direction of creation.

   An arrow is displayed on the wall.



The **Reverse** button enables to modify the direction.

You can also click the arrow in the 3D geometry.

4. Keep the **Instances & Spacing** options to define the parameters.

   Choosing these parameters types dims the **Length** field because the application no longer needs this specification to space the instances.

You can set the duplication parameters by choosing the number of instances, the spacing between instances, or the total length of the zone filled with instances.
Three options are available:

- **Instances & Length**: the spacing between instances is automatically computed based on the number of instances and the specified total length
- **Instances & Spacing**: the total length is automatically computed based on the number of instances and the specified spacing value
- **Spacing & Length**: the number of instances is automatically computed to fit the other two parameters.

For each of these cases only two fields are active, allowing you to define the correct value.

If you set **Instances & Length** or **Spacing & Length** parameters, note that you cannot define the length by using formulas.

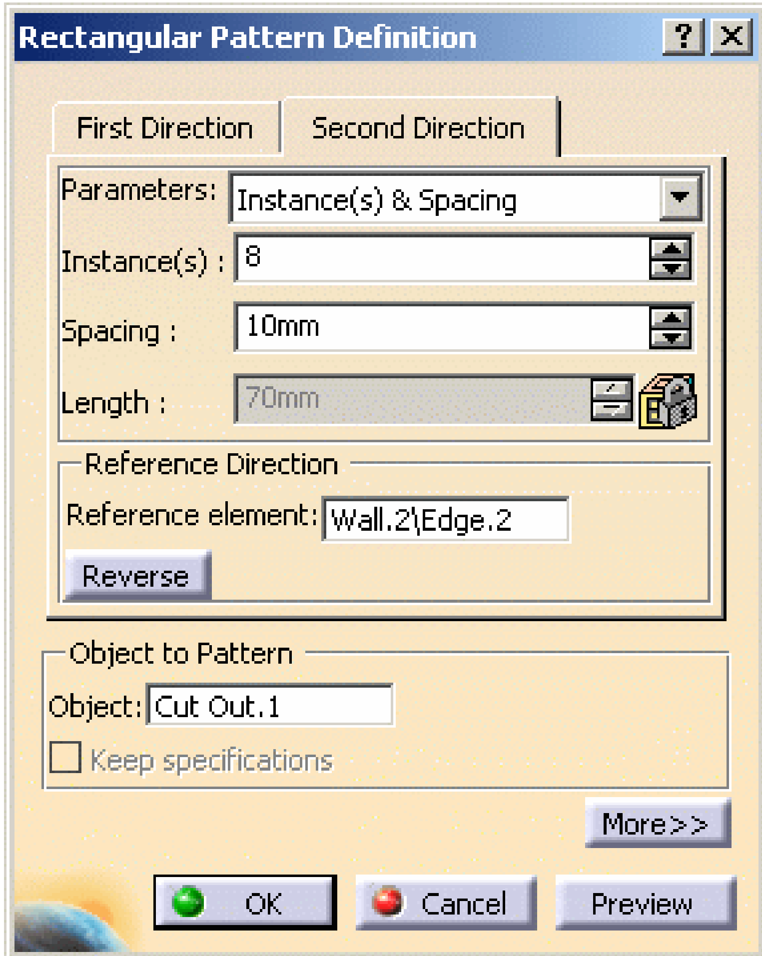
Patterns should not go beyond the model.

5. Enter 2 as the number of instances you wish to obtain in the first direction.

6. Define the spacing along the grid: enter 30mm.

Defining the spacing along the grid and the length of your choice, would make the application compute the number of possible instances and space them at equal distances.

7. Now, click the **Second Direction** tab to define the other parameters.



Note that defining a second direction is not compulsory. Creating a rectangular pattern defining only one direction is possible.

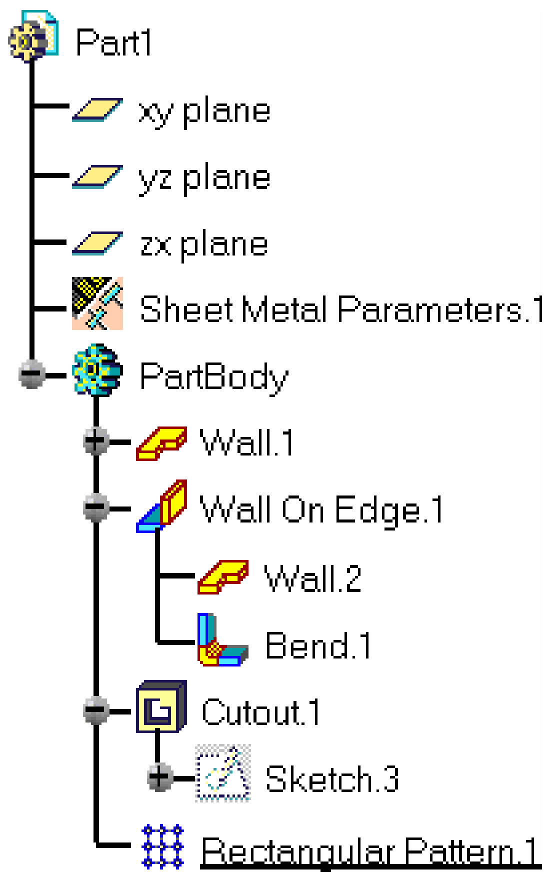
8. Select the second edge (**Edge.2**), as shown, to define the second direction.

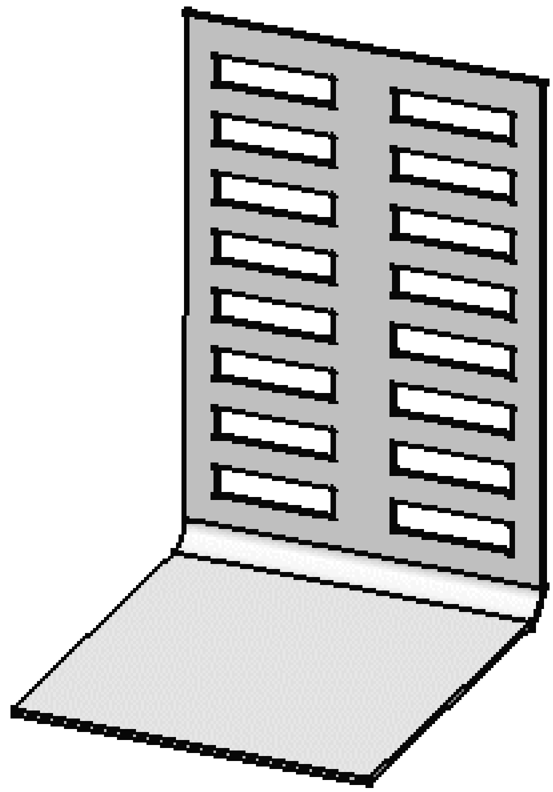
9. Keep the **Instances & Spacing** option: enter 8 and 10 mm in the appropriate fields.

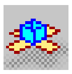
Additional cutouts have been aligned along this second direction.

10. Click **OK** to repeat the cutouts.

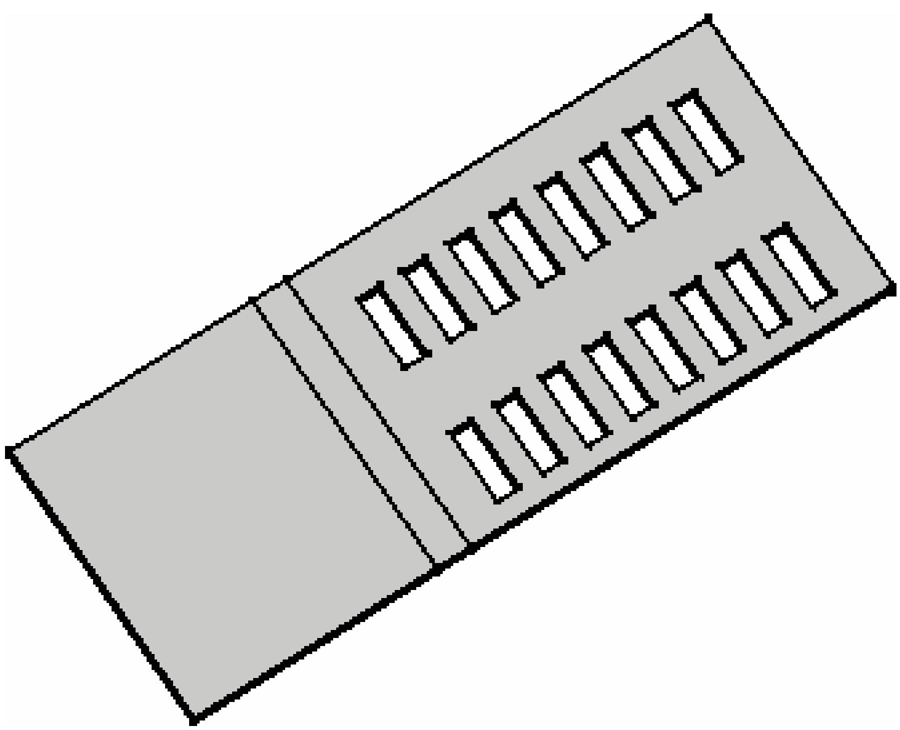
After the update, the Sheet Metal part looks like this:





11. Select this icon  to unfold the part:

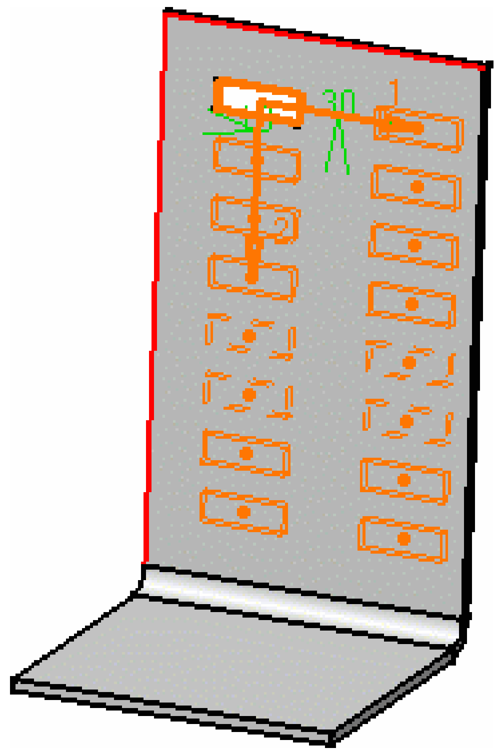
The pattern is updated on the unfolded view.



12. Click the **More>>** button to display further options.

The **Simplified representation** option lets you lighten the pattern geometry, when more than 10 instances are generated in one direction. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:

You can also specify the instances you do not want to see by double-clicking the dots. These instances are then represented in dashed lines during the pattern definition and then are no longer visible after validating the pattern creation. The specifications remain unchanged, whatever the number of instances you view. This option is particularly useful for patterns including a large number of instances.



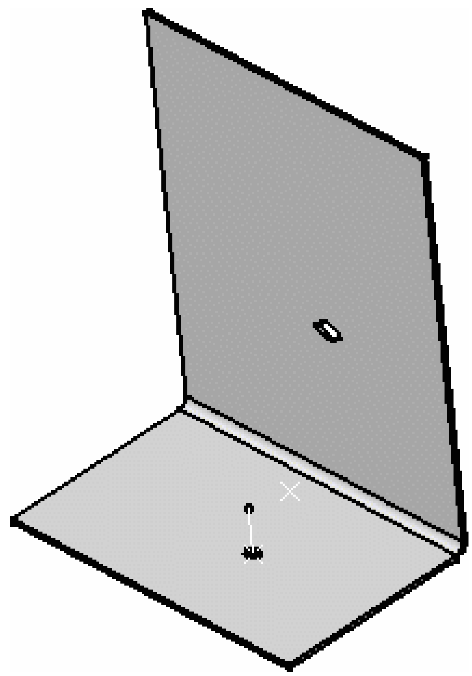
Creating Circular Patterns

In this task, you are going to create circular cutouts according to a pattern.
These features make the creation process easier.

In the Generative Sheetmetal Design workbench, you can only duplicate flanges, cutouts, holes, stamps (except stiffening ribs), stamps without fillet (radius=0) and Generative Sheetmetal Design patterns.
These features must lie on a unique and planar wall.

For the SheetMetal Design workbench, open the CircularPattern1.CATPart document.
For the Generative Sheetmetal Design workbench, open the NEWCircularPattern1.CATPart document.

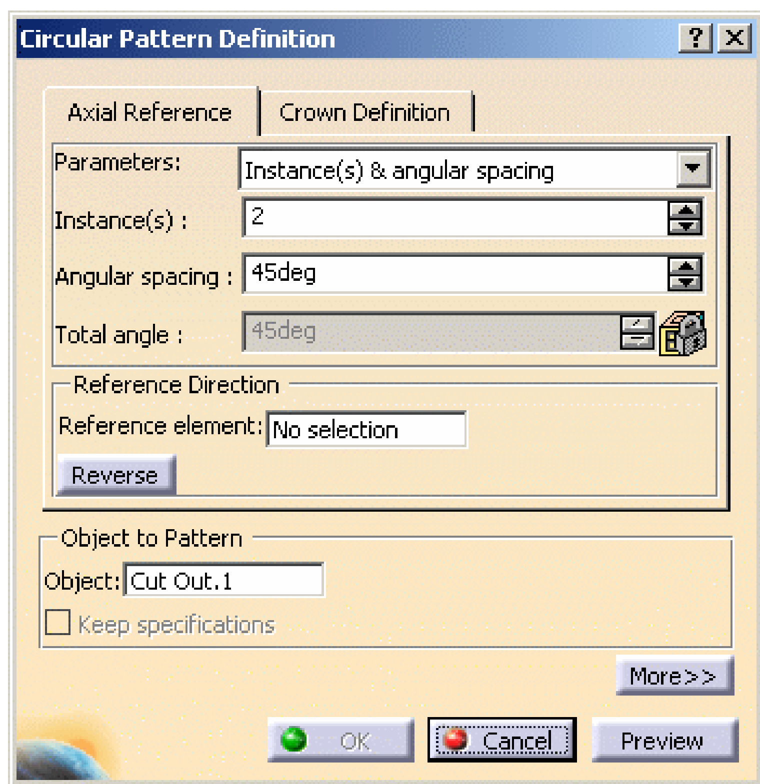
The Sheet Metal part looks like this:



1. Select the circular cutout you want to duplicate.

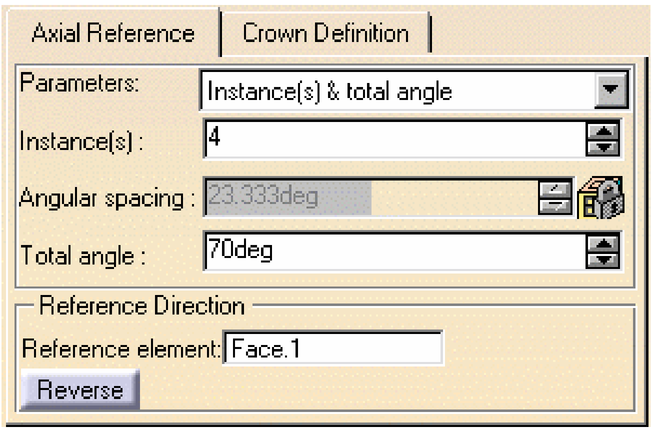
2. Click the **Circular Pattern** icon .

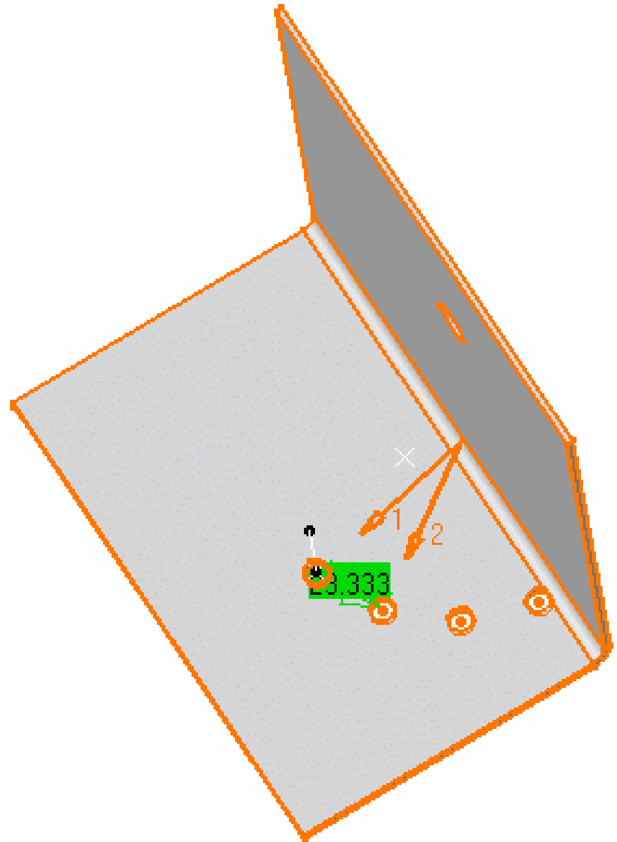
The **Circular Pattern Definition** dialog box is displayed.



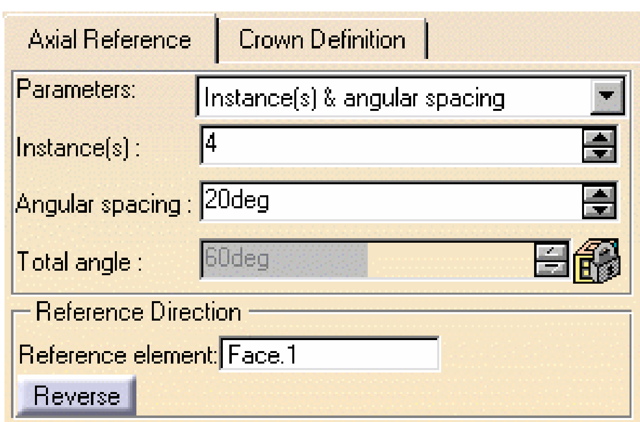
3. Define the **Axial Reference** by choosing the **Parameters** type, and reference direction.

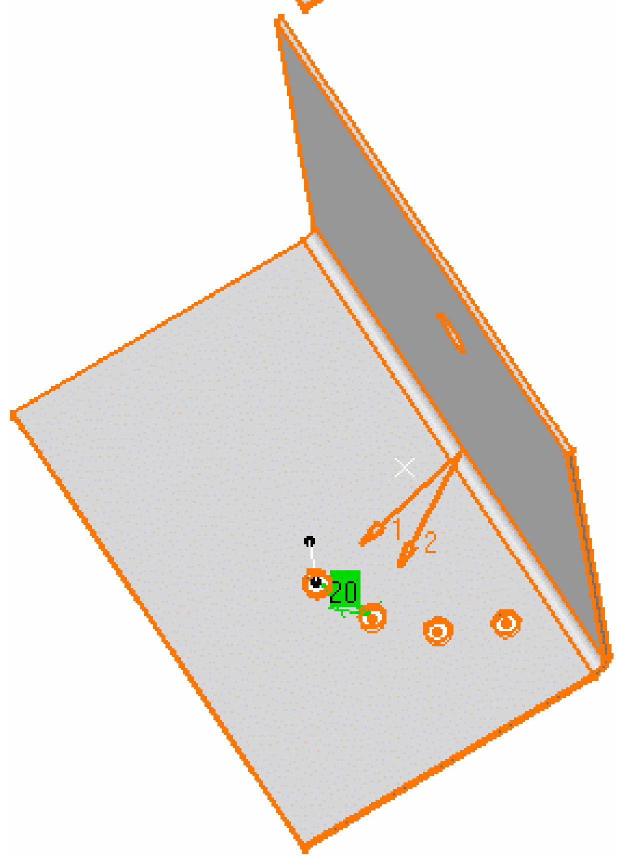
- **Instance(s) & total angle**: the number of patterns as specified in the instances field are created, in the specified direction, and evenly spread out over the total angle.

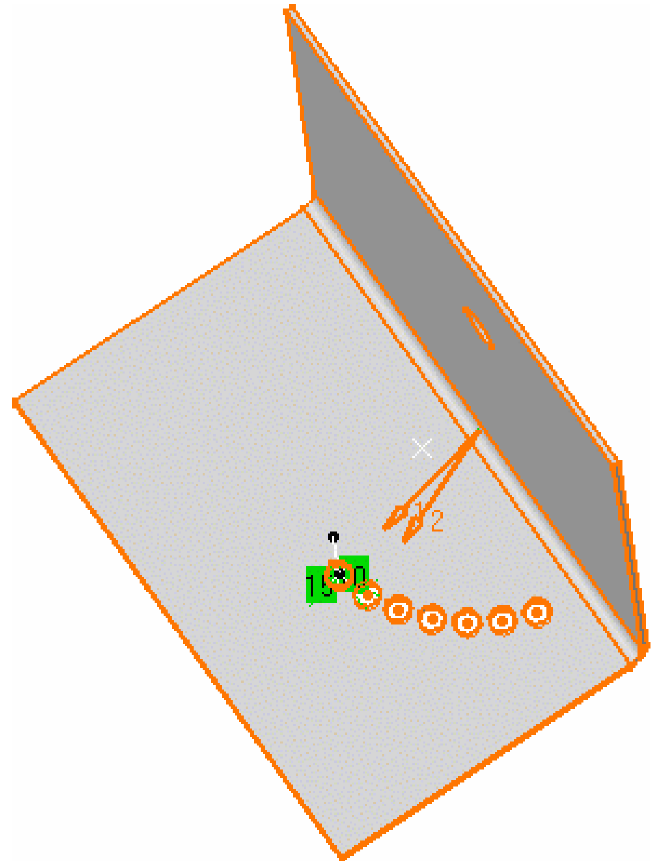




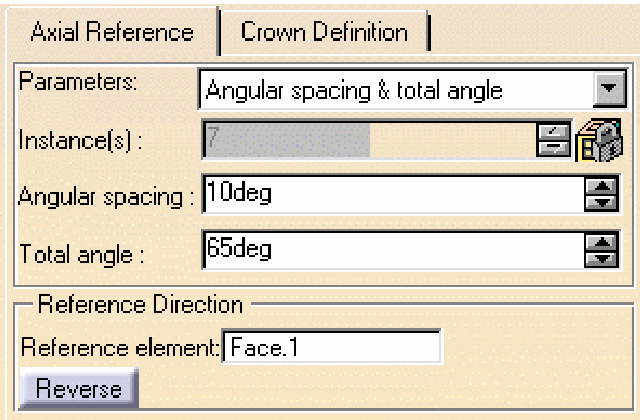
- **Instance(s) & angular spacing**: the number of patterns as specified in the instances field are created in the specified direction, each separated from the previous/next one of the angular angle value.

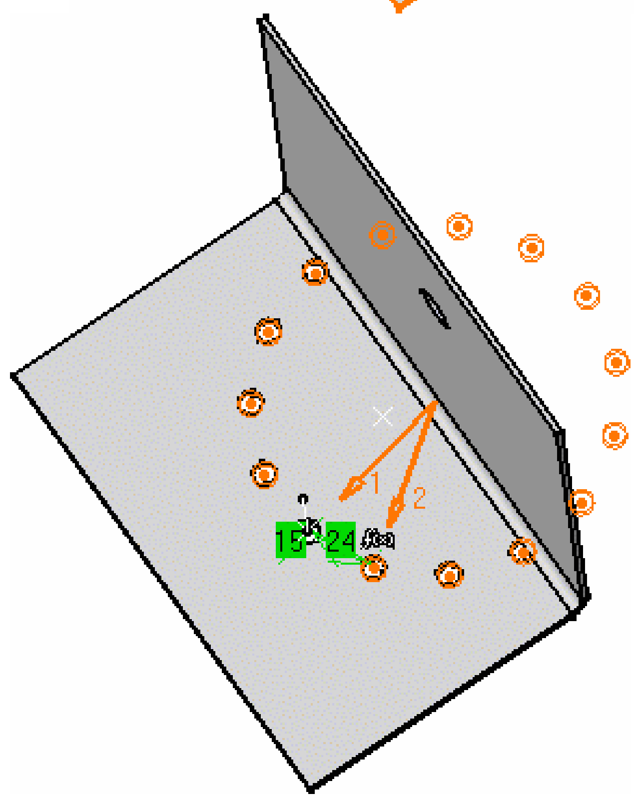




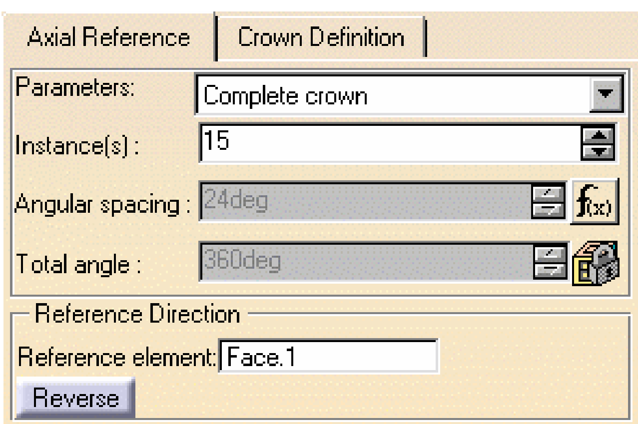


- **Angular spacing & total angle**: as many patterns as possible are created over the total angle, each separated from the previous/next one of the angular angle value.





- **Complete crown**: the number of patterns as specified in the instances field are created over the complete circle (360deg).



If you set **Instance(s) & total angle** or **Angular spacing & total angle** parameters, note that you cannot define the length by using formulas.

4. Click the Reference element and select the element defining the rotation axis.

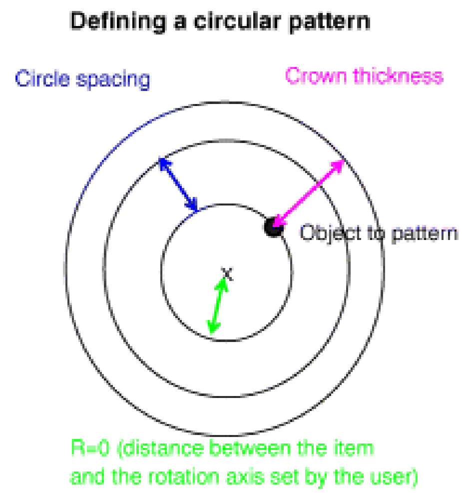
   Here select the face on which lies the circular cutout.

- To define a direction, you can select an edge or a planar face.
  Should you select the face of a wall, the rotation axis would be normal to that face.
- Click the **Reverse** button to inverse the rotation direction.

Now you are going to add a crown to this pattern.

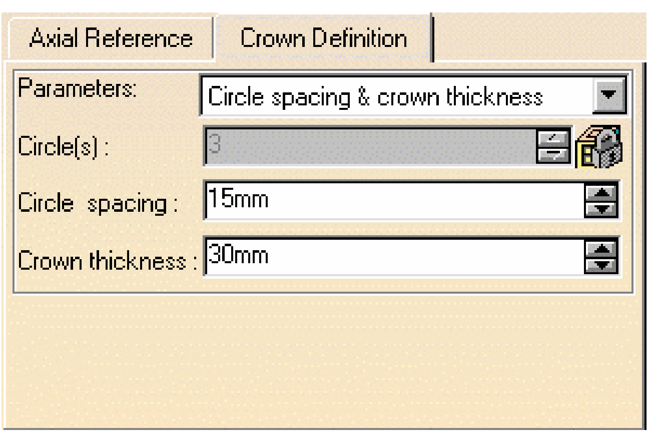
5. Click the **Crown Definition** tab, and choose which parameters you wish to define the crown.

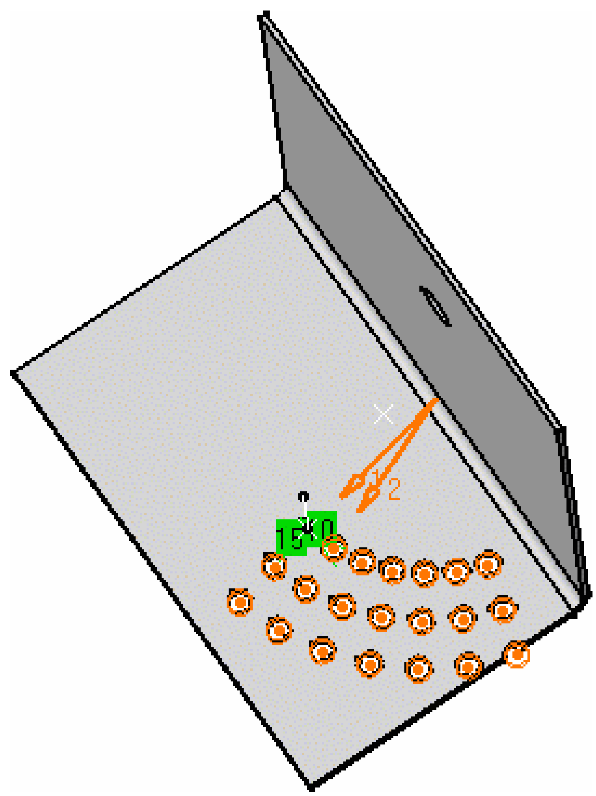
This figure may help you define these parameters:



- Circle(s) and crown thickness: you define the number of circles and they are spaced out evenly over the specified crown thickness
- Circle(s) and circle spacing: you define the number of circles and the distance between each circle, the crown thickness being computed automatically
- Circle(s) spacing and crown thickness: you define the distance between each circle and the crown thickness, and the number of circles is automatically computed.

For example, using the values described above for the **Angular spacing & total angle** option, you could define the crown as:

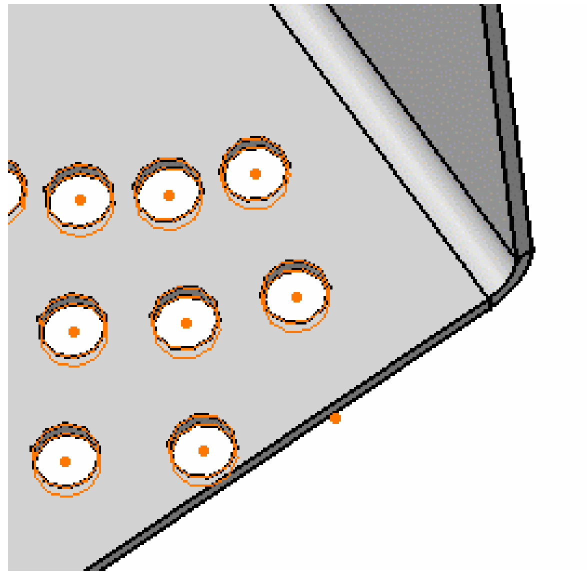




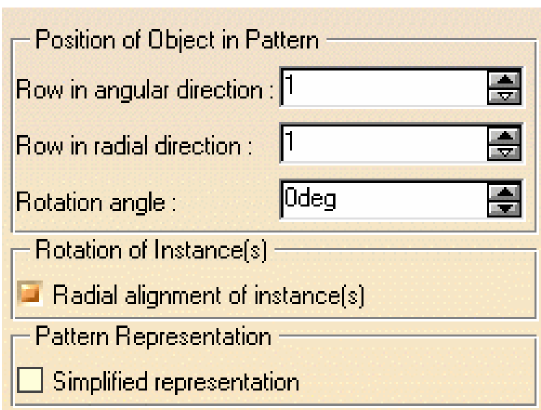
Note that one of the pattern is created beyond the wall.

You can delete instances of your choice when creating or editing a pattern. To do so, just select the points materializing instances in the pattern preview.

The instance is deleted, but the point remains, as you may wish to click it again to add the instance to the pattern definition again.

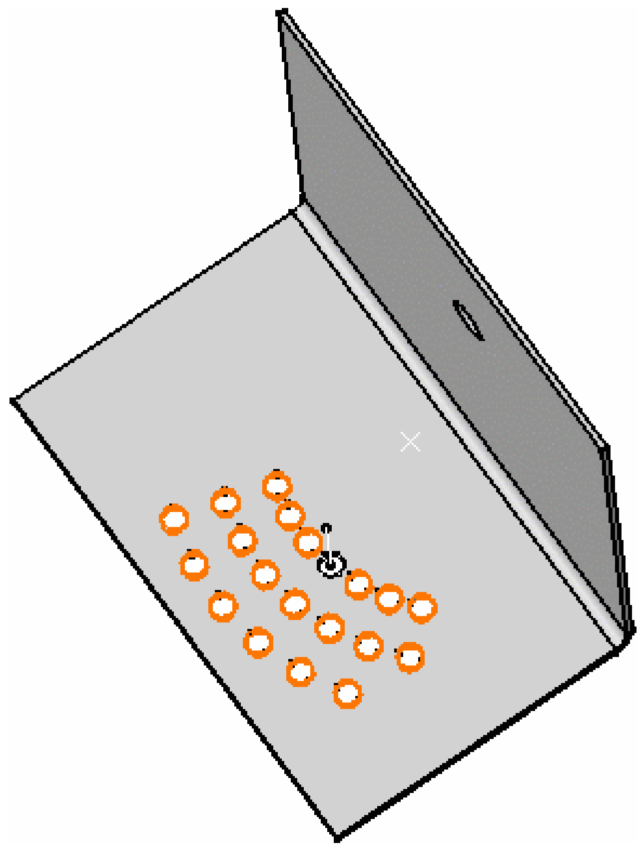


6. Click the **More>>** button to display further options:



Using these options, you can change the position of the selected cutout within the crown. For example, if you set the **Row in angular direction** parameter to 4, this is what you obtain: the initially selected cutout is the fourth instance, based on the rotation direction, of the pattern.

Typically, in this case, you might want to edit the pattern and click again the instance that you removed above, to get a full pattern.



- The **Simplified representation** option lets you lighten the pattern geometry, when more than 10 instances are generated in one direction. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:
  You can also specify the instances you do not want to see by double-clicking them . These instances are then represented in dashed lines during the pattern definition and then are no longer visible after validating the pattern creation. The specifications remain unchanged, whatever the number of instances you view. This option is particularly useful for patterns including a large number of instances.
- When checking the **Radial alignment of instances**, all instances have the same orientation as the original feature. When unchecked, all instances are normal to the lines tangent to the circle.

In case you use the circular cutout as a reference element, it means the axial reference of the pattern will be the same as the rotation axis of the circular cutout. As a result, when the cutout is duplicated, the first crown instances will be superimposed on one another.

To avoid this, uncheck **Radial alignment of instances**, so that the instances are properly positioned around the cutout.

7. Click **OK** to create the pattern.

Creating User-Defined Patterns

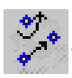
The User Pattern command lets you duplicate a cutout, a stamp, or any other feature as many times as you wish at the locations of your choice.
Locating instances consists in specifying anchor points. These points are sketches.

In the Generative Sheetmetal Design workbench, you can only duplicate flanges, cutouts, holes, stamps (except stiffening ribs), stamps without fillet (radius=0) and Generative Sheetmetal Design patterns.
These features must lie on a unique and planar wall.

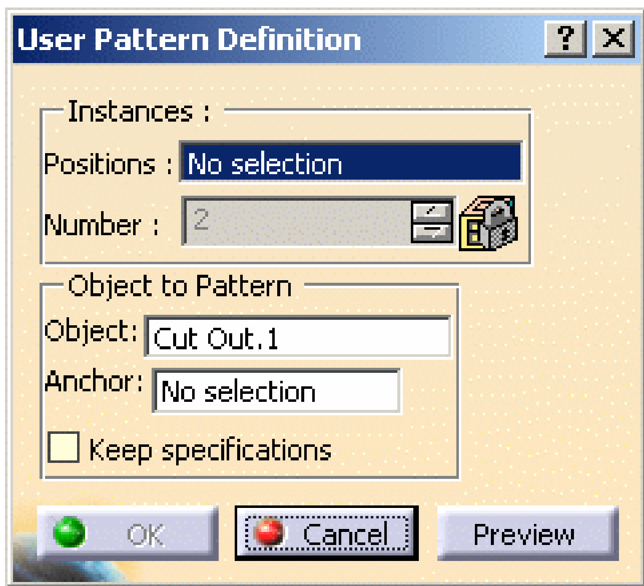
For the SheetMetal Design workbench, open the UserPatterns1.CATPart document.
For the Generative Sheetmetal Design workbench, open the NEWUserPatterns1.CATPart document.

1. Select the feature to be duplicated.

   Here we selected the cutout.

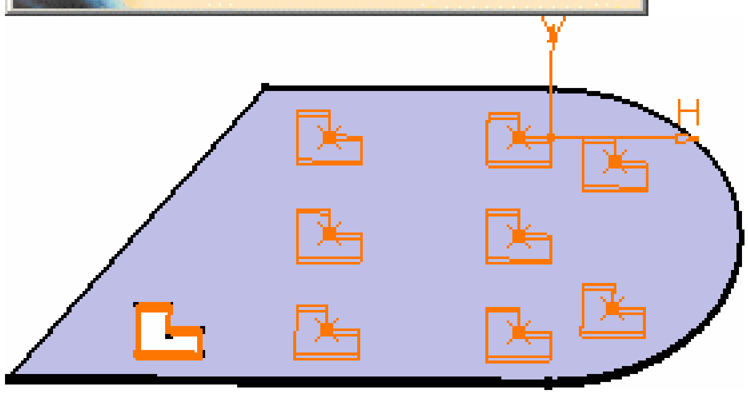
2. Click the **User Pattern** icon .

   The User Pattern Definition dialog box is displayed.



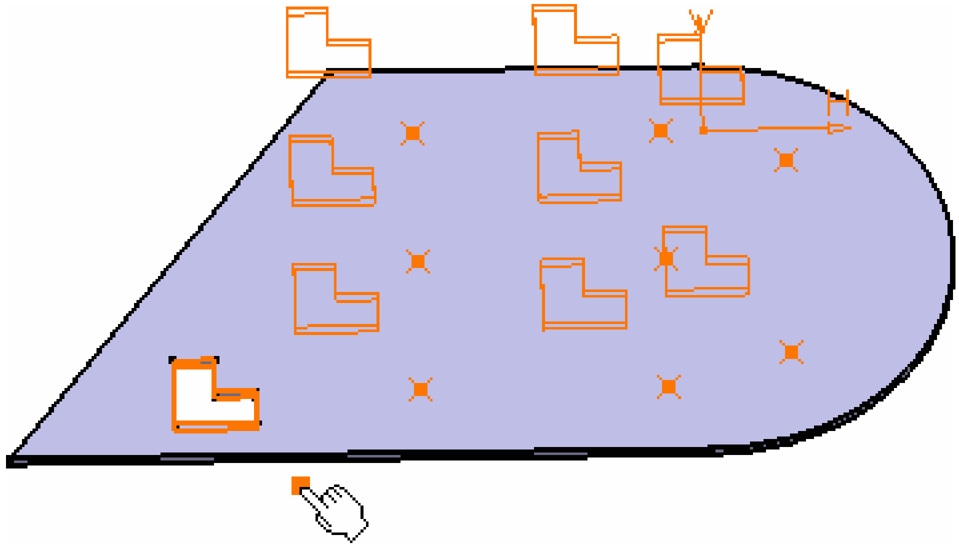
3. Select '**Sketch 3**' in the specification tree and click **Preview**.

   The sketch contains the points you need to locate the duplicated cutouts.

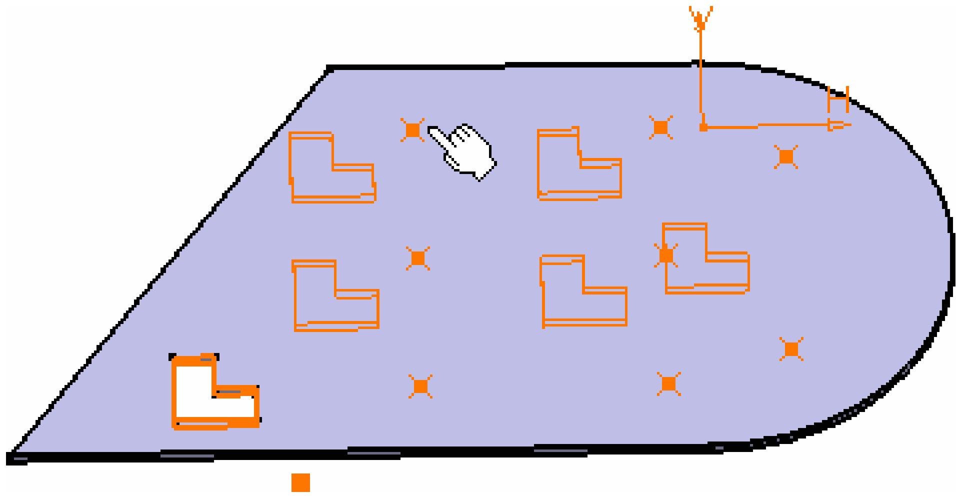


By default, the application positions each instance with respect to the center of gravity of the element to be duplicated. To change this position, use the anchor field: click the anchor field and select a vertex or a point.

4. Click inside the Anchor field and select the point (Point.1) to indicate a new reference location.

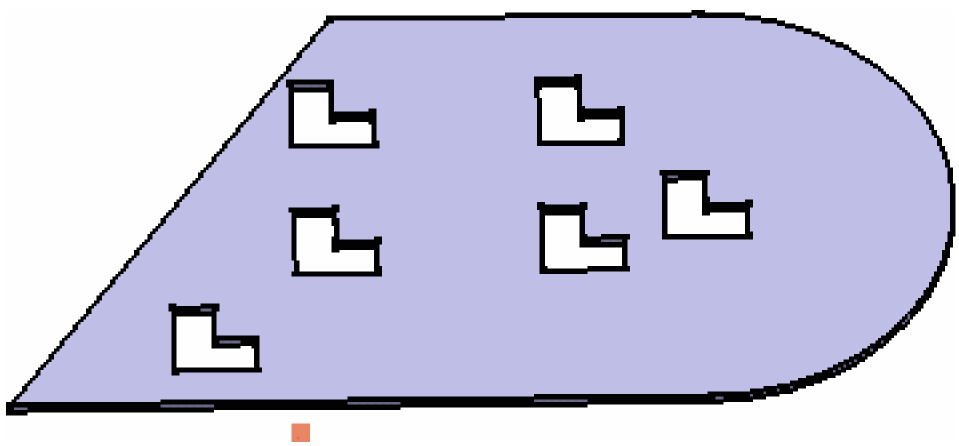


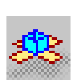
5. You can then click the points corresponding to the pattern instances to be removed.



6. Click **OK** in the User Pattern Definition dialog box.

Cutouts are created at the points of the sketch.



Would you need to unfold the part using the  icon, you would notice that the pattern is updated.

Corner Relief

This section explains and illustrates different methods to create corner relieves on bends.

Redefine an automatic corner relief: double-click an automatic corner relief, edit its parameters in the dialog box


Create a local corner relief: select two or more bends, the corner relief type and parameters

See also Bend Corner Relief parameters settings.

Redefining an Automatic Corner Relief

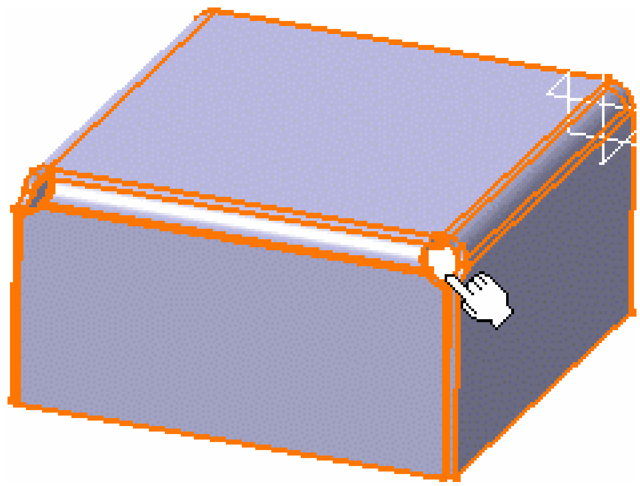
This task explains how to redefine automatic corner relieves on a Sheet metal part.

Open the CornerRelief01.CAPTPart model from the samples directory.

1. Double-click the bend on which the corner relief you want to redefine is located.

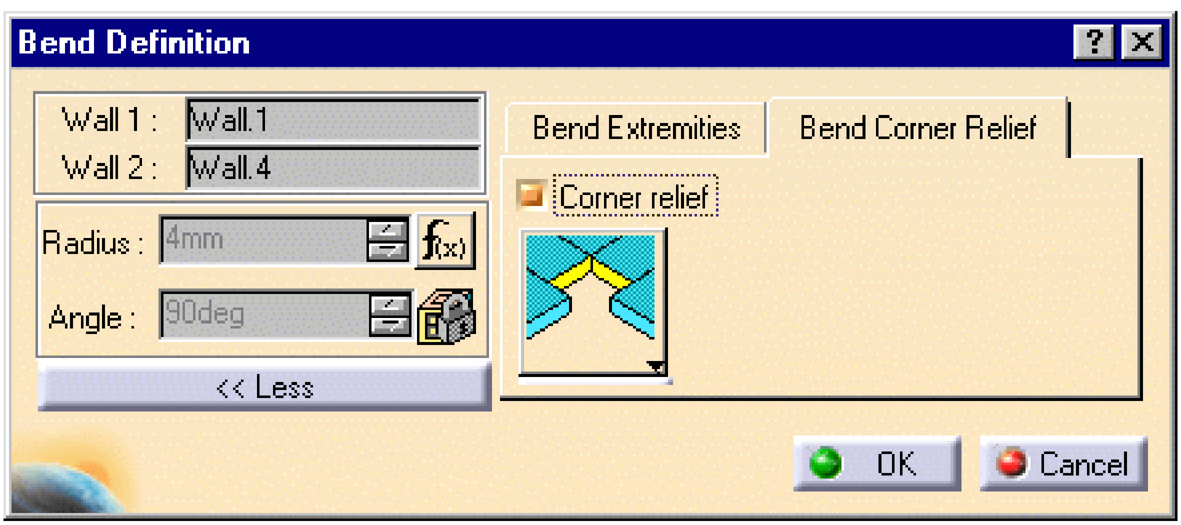
   It may be easier to double-click it from the specification tree.

   The Bend Definition dialog box is displayed.

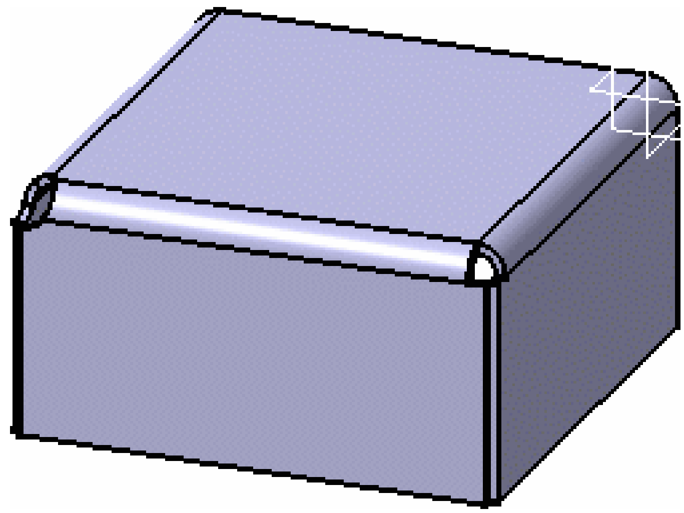


2. Click the More button and select the Bend Corner Relief tab.

   This tab is similar to the Corner Relief tab of Sheet Metal Parameters dialog box.



3. Choose a new corner relief type, and click **OK** to validate.



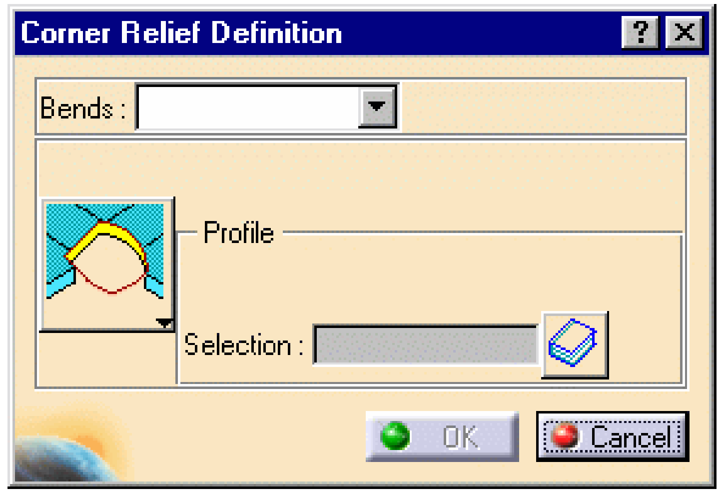
This definition will apply to the current corner relief, and will prevail over any other corner relief definition you may enter through the Sheet Metal Parameter dialog box.

Creating A Local Corner Relief Relief

This task explains how to define a corner relief locally on a set of bends.
Depending on the number of bends involved, not all types of corner relief are available.

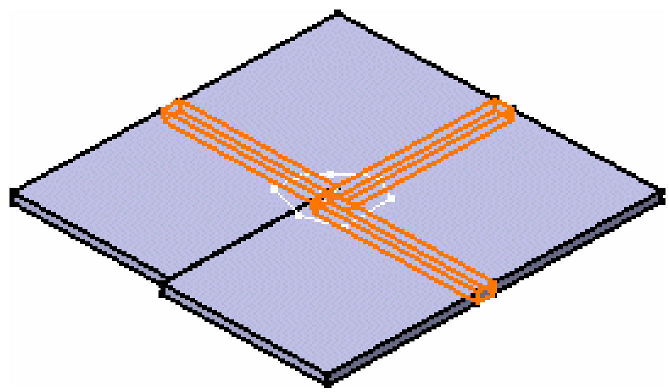
Open the CornerRelief02.CATPart document from the samples directory.
The part needs to be unfolded prior to creating the corner relief.

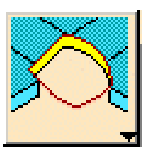
1. Click the Corner Relief icon .



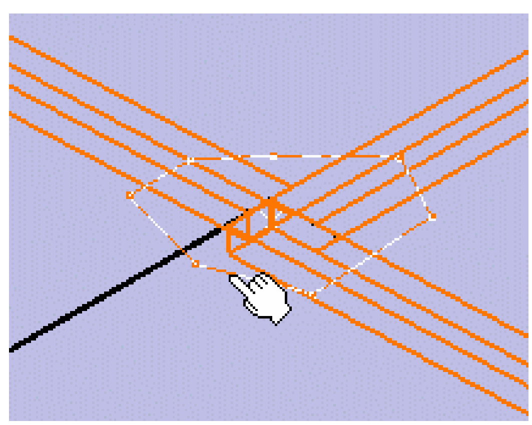
The Corner Relief Definition dialog box is displayed.

2. Select the supports on which a corner relief should be created.



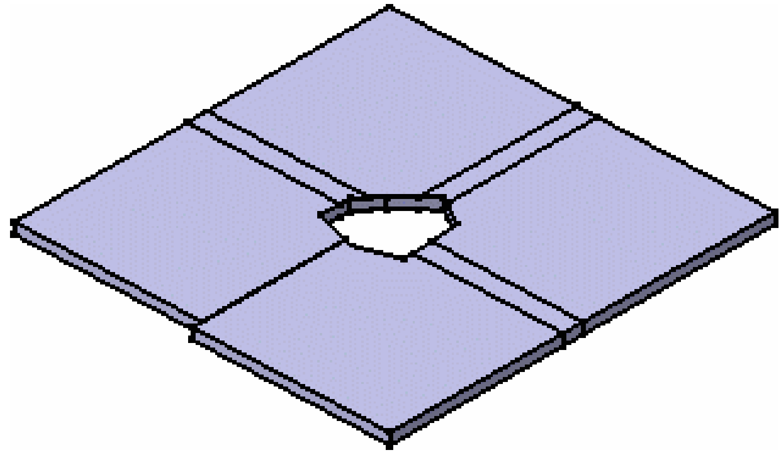
By default the User Corner Relief  is active in the Corner Relief Definition dialog box.

3. Select the sketch, directly in the document.

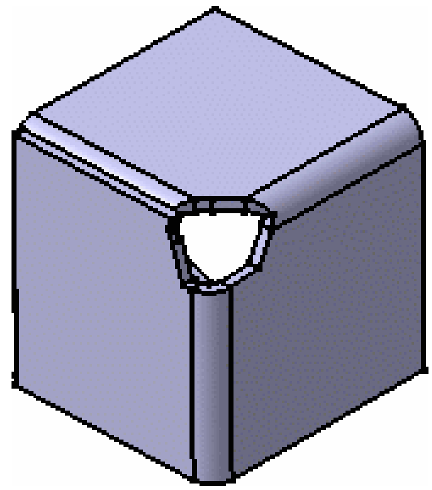


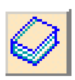
As soon as the sketch has been selected the sketcher icon is displayed in the dialog box allowing you to edit the selected sketch, if needed.

4. Click OK in the Corner Relief Definition dialog box.

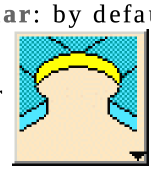


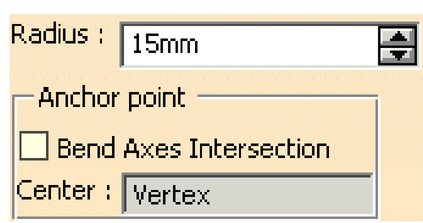
5. Fold the part to check the corner relief in 3D.



You can use the **Catalog** icon  to open the Catalog Browser.
For more information on catalogs, please refer to the *Component Catalog Editor* documentation.

If you choose another corner relief type, the scenario maybe slightly different:

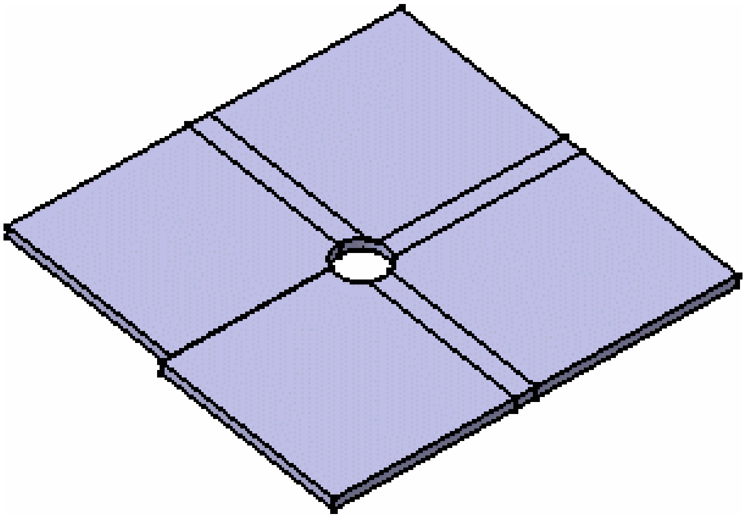
- **circular**: by default the corner relief center  is located at the intersection of the bend axes.



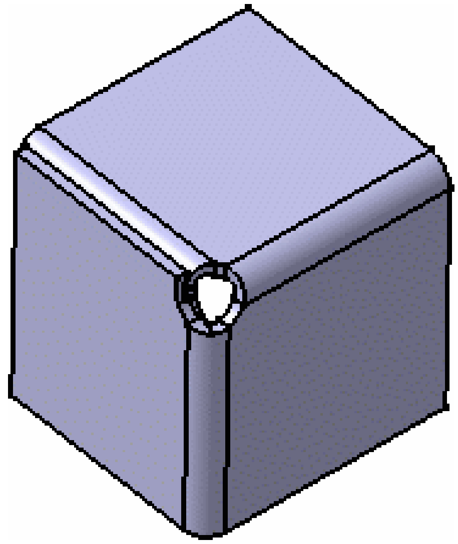
You can select a point as the circle's center.

A radius is proposed by default. It is equal to the bend radius + the thickness. You can change it by:

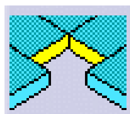
- **Selecting Formula -> Deactivate** from the contextual menu of the input field and enter a new value,
- or clicking on the  button and entering a new formula.



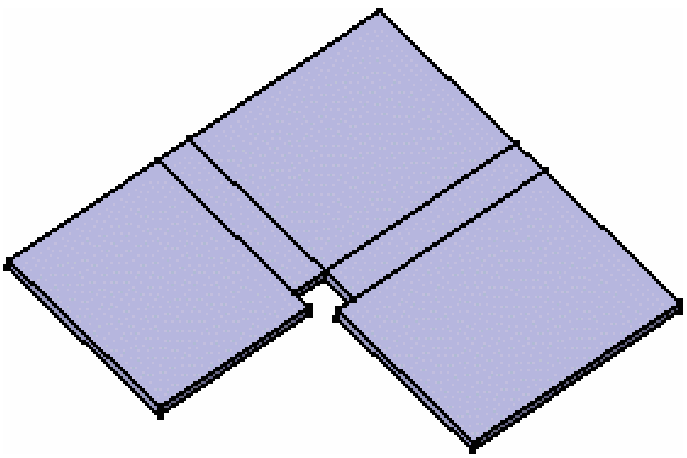
Unfolded circular corner relief



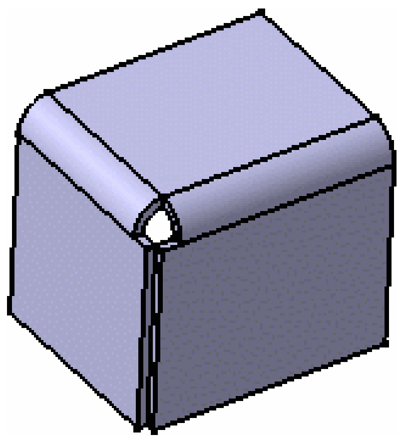
Folded circular corner relief

- **square**: the square corner relief  is created using the bend limits. Its dimensions are defined by the width of the unfolded bends.

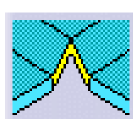
Available between two bends only.



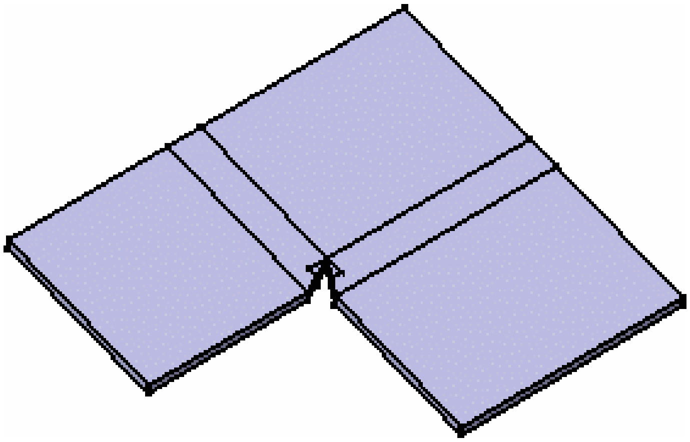
Unfolded square corner relief



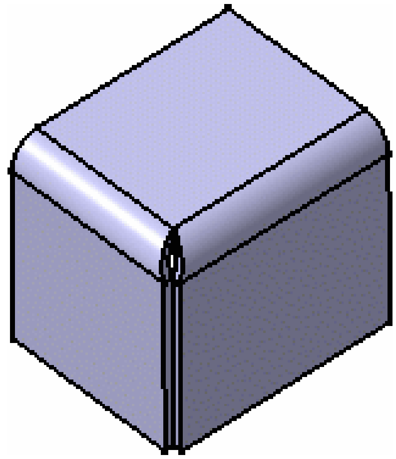
Folded square corner relief

- **triangular**: the triangular corner relief  is created from the intersection point of the inner bend limits towards the intersection points of the outer bend limits with each wall.

Available between two bends only.



Unfolded triangular corner relief



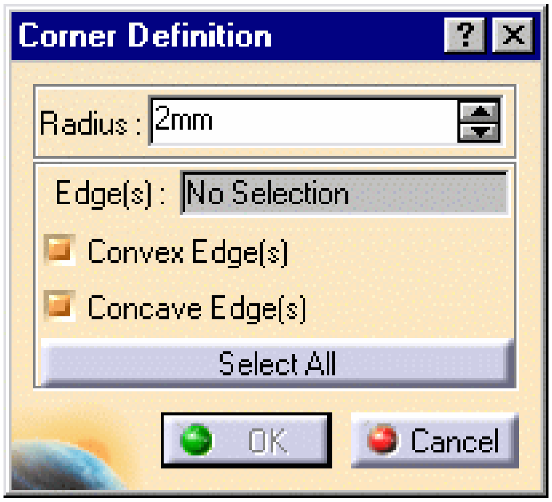
Folded triangular corner relief

Only the User-defined and Circular corner relief type allow the selection of more than two bends.

Creating Corners

This task shows how to create one or more corner(s) on a Sheet Metal part, that is to round off sharp edges, much like a fillet between two faces of a Part Design Body.
This corner creation operation can be performed indifferently on the folded or unfolded view, and only one support (i.e. the corner when previewed should not lie over two supports).

Open the Corners1.CATPart document.



1. Click the **Corner** icon .

The Corner Definition dialog box is displayed.

2. Set the radius value.

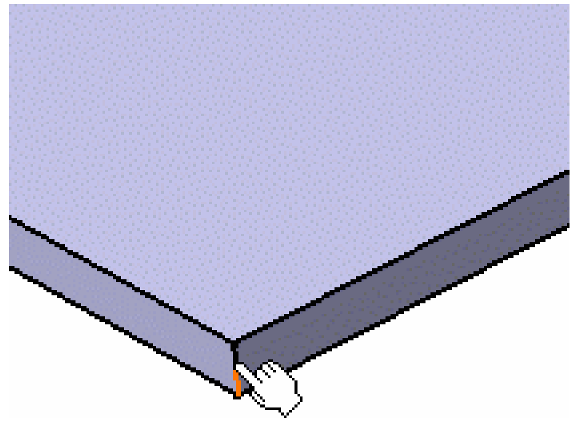
3. Choose the type of edge you wish to round off:

- using the **Select All** button: all convex or concave edges, or all edges of both types
- any edge manually selected

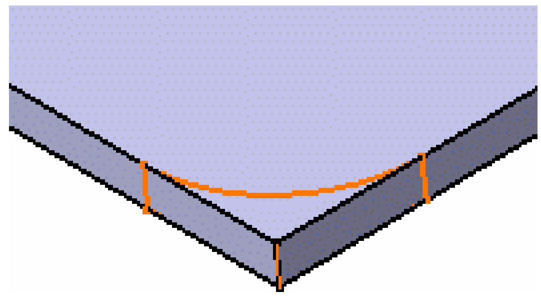
By default both buttons are checked, to allow the selection of any edge type whether manually or automatically.

- Once you have selected an edge, you can no longer modify this option, unless you cancel the selection.
- If you check the **Convex Edge(s)** button and you select a concave edge, a warning is issued indicating that you did not select an edge corresponding to the active type.

4. With only the **Convex Edge(s)** button checked, select a sharp edge on a part.



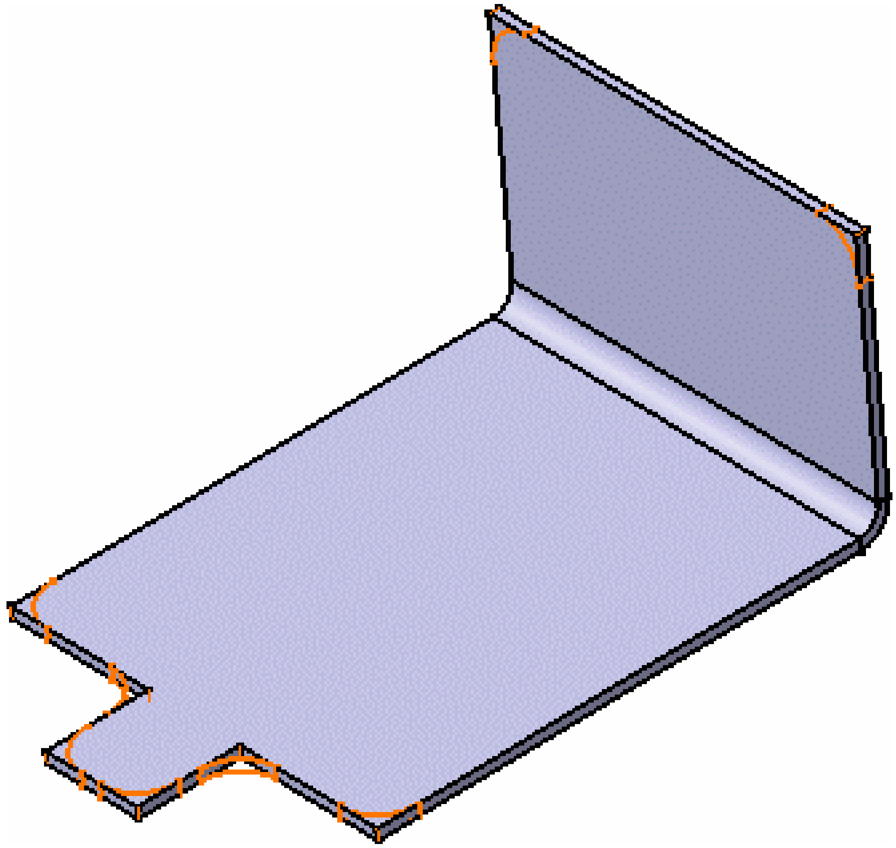
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.



The corner is previewed on the edge, with the current radius value.

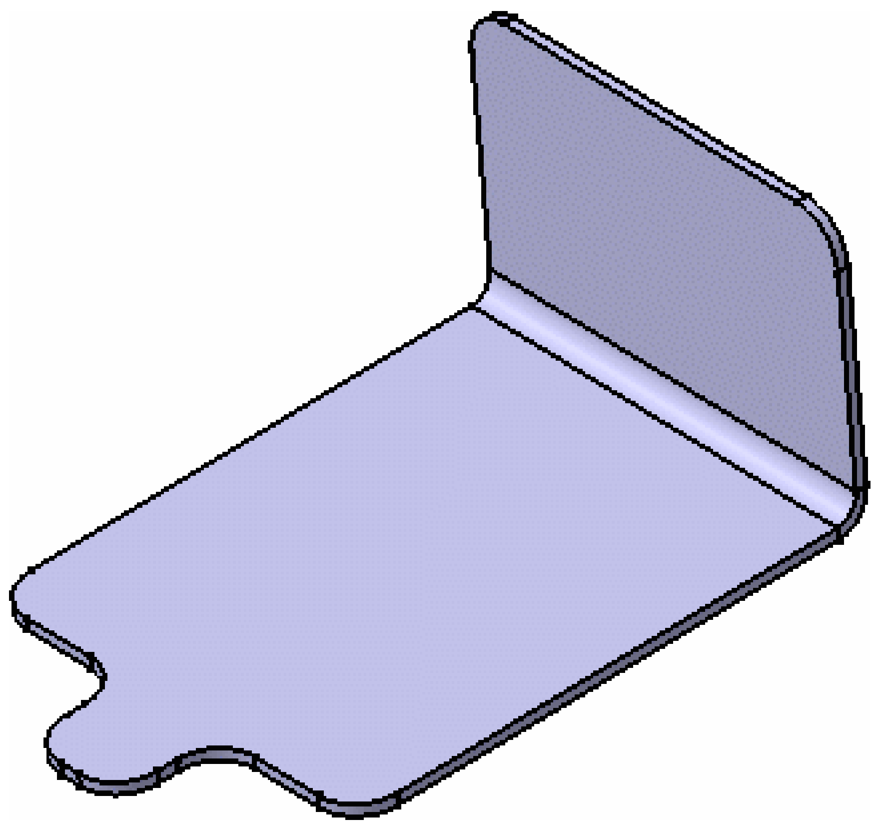
5. Click **Cancel Selection**, make sure that both **Convex Edge(s)** and **Concave Edge(s)** buttons are checked, then click the **Select All** button.

All sharp edges of the part are selected, the **Select All** button taking into account the chosen type (convex, concave, or both) and the corners previewed.



6. Click OK in the dialog box.

All sharp edges of the part are rounded off to create smooth corners.



To deselect an edge, simply click it again. For quick selection in a complex part, you can select all edges with the **Select All** check button, then deselect one or two edges.

- When you select an edge that is not sharp, such as the edge between a wall and a bend for example, a warning is issued.
- As you select more edges, the **Edge(s)** field of the dialog box is updated.
- When using the **Select All** button, you select all edges (whether concave, convex, or both) present at the time. If when modifying the Sheet Metal part, new edges are created, these will not be automatically rounded off.

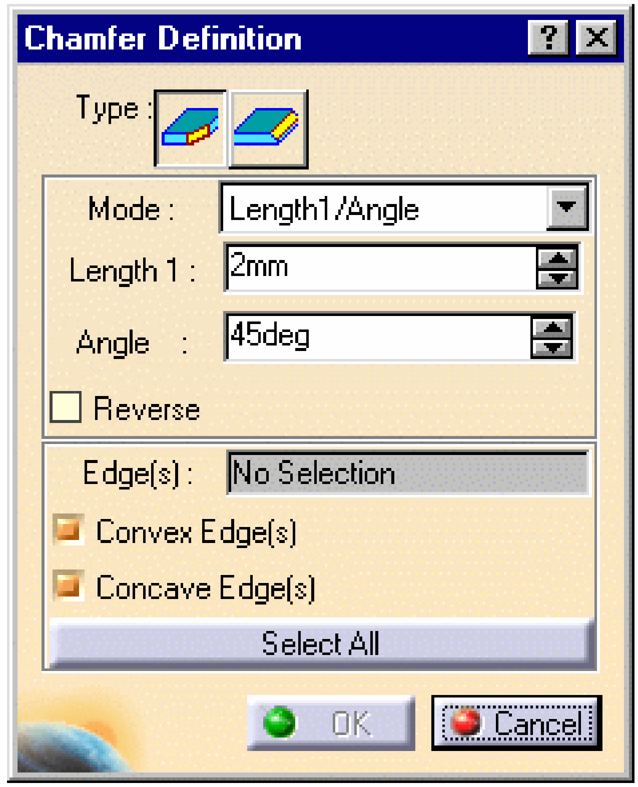
Creating Chamfers

This task shows how to create one or more chamfer(s) on a Sheet Metal part, that is to cut off, or fill in sharp edges of Sheet Metal parts.
This chamfer creation operation can be performed indifferently on the folded or unfolded view, and only one support (i.e. the chamfer when previewed should not lie over two supports).

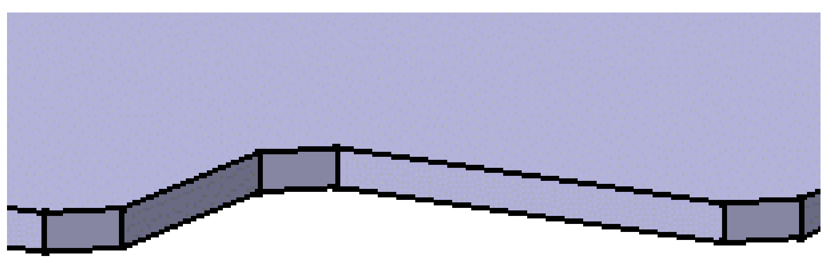
Open the Corners1.CATPart document.

1. Click the **Chamfer** icon .

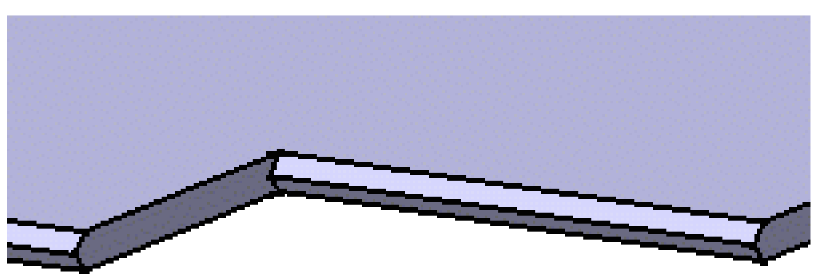
The Chamfer Definition dialog box is displayed.



2. Choose the chamfer **Type**:



- **Thickness chamfer** : to be able to select edges that represent the thickness of the part



- **Welding chamfer** : to be able to select edges that represent the area of the part where it can be welded to another part.

With the Thickness chamfer type only, you can choose the type of edge you wish to chamfer:

- using the **Select All** button: all convex or concave edges, or all edges of both types
- any edge manually selected

By default both buttons are checked, to allow the selection of any edge type whether manually or automatically.

- Once you have selected an edge, you can no longer modify this option, unless you cancel the selection.
- If you check the **Convex Edge(s)** button and you select a concave edge, a warning is issued indicating that you did not select an edge corresponding to the active type.

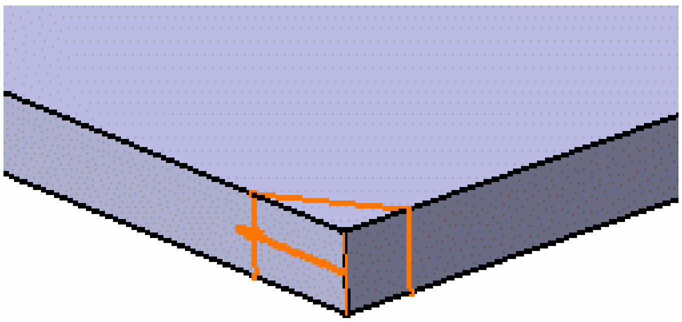
3. Select a sharp edge on a part.

The chamfer is previewed on the edge.

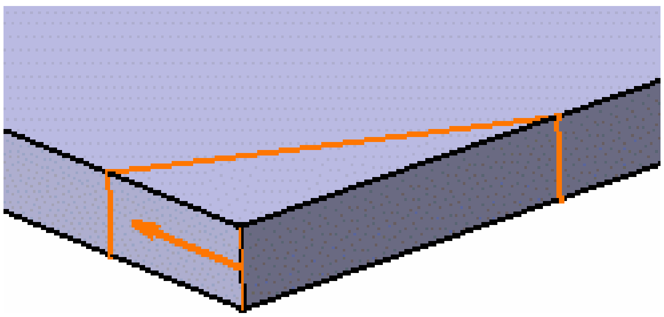
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.

4. Choose a chamfer **Mode**. You can either enter:

- a length value and an angle: the length is computed on one side of the edge and the angle from the chamfer's limit on the same side



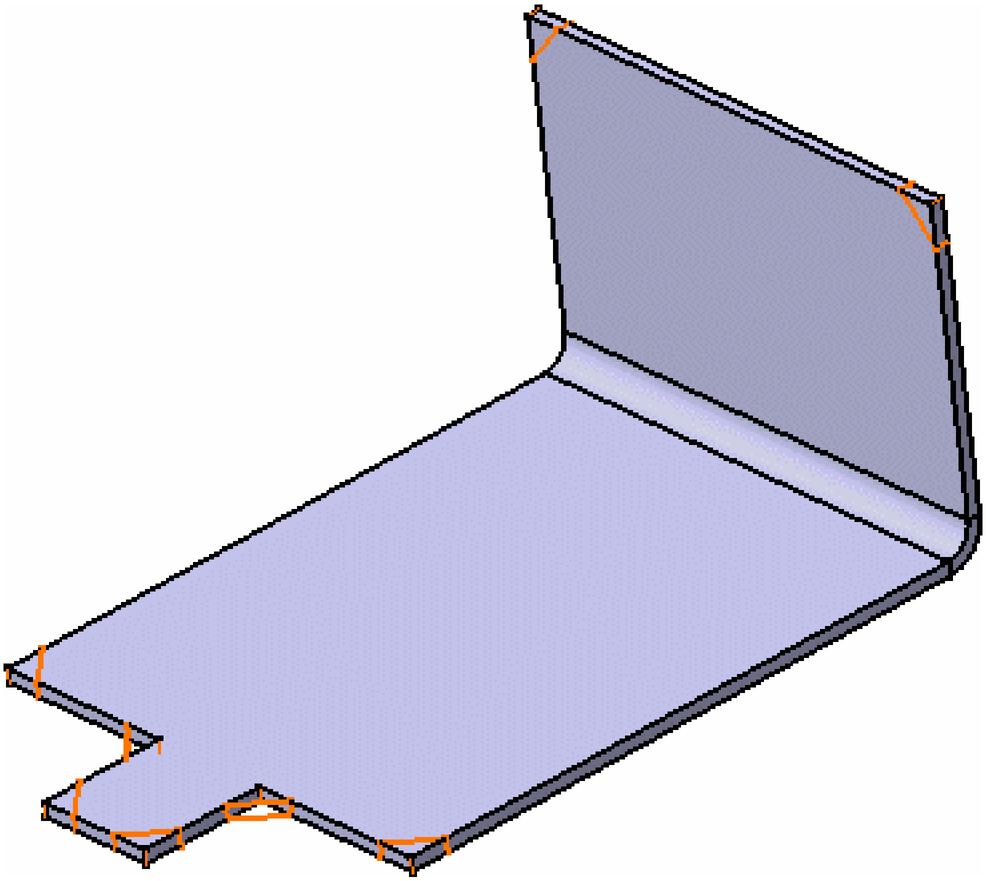
- two lengths: these lengths are computed from the selected edge on both sides.



You can use the Reverse button to inverse all edges' side, on which the values are taken into account; Use the arrow displayed on each edge to locally invert only one edge.

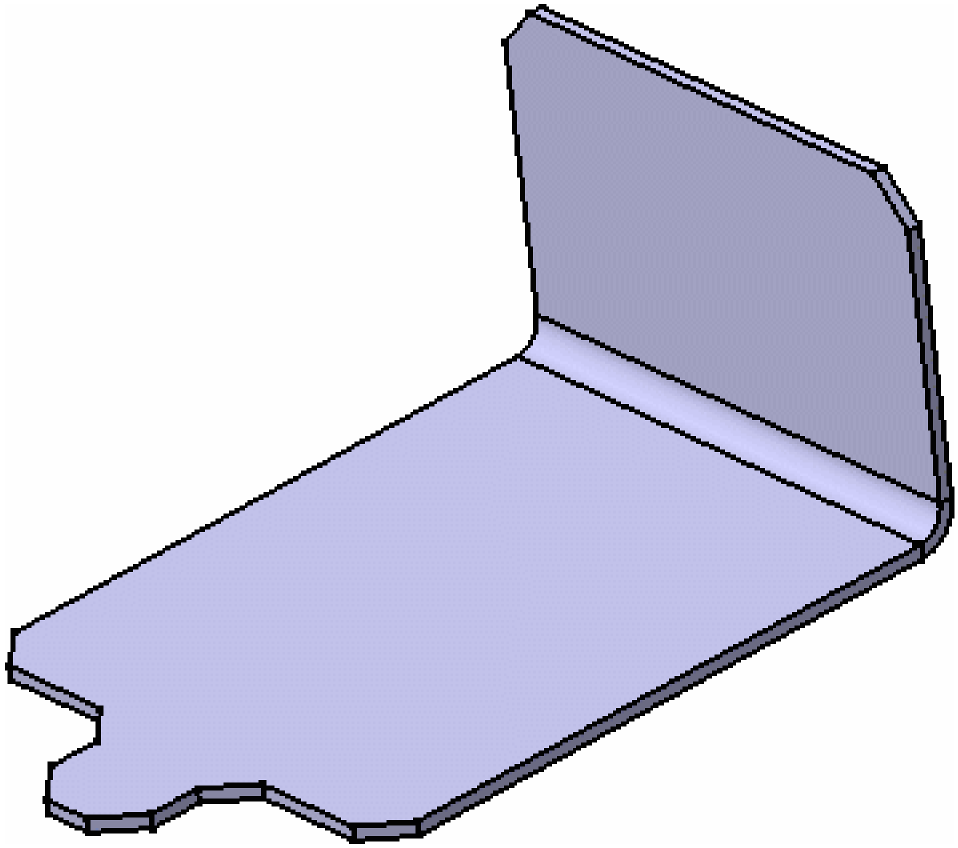
5. Click **Cancel Selection** then, make sure that both **Convex Edge(s)** and **Concave Edge(s)** buttons are checked, and click the **Select All** button.

All sharp edges of the part are selected, the **Select All** button taking into account the chosen type (convex, concave, or both) and the chamfers previewed.



6. Click OK in the dialog box.

All sharp edges of the part are cut off or filled in.



To deselect an edge, simply click it again. For quick selection in a complex part, you can select all edges with the **Select All** button, then deselect one or two edges.

- When you select an edge that is not sharp, such as the edge between a wall and a bend for example, a warning is issued.
- As you select more edges, the **Edge(s)** field of the dialog box is updated.
- When using the **Select All** button, you select all edges (whether concave, convex, or both) present at the time. If when modifying the Sheet Metal part, new edges are created, these will not be automatically chamfered.

Mapping Elements

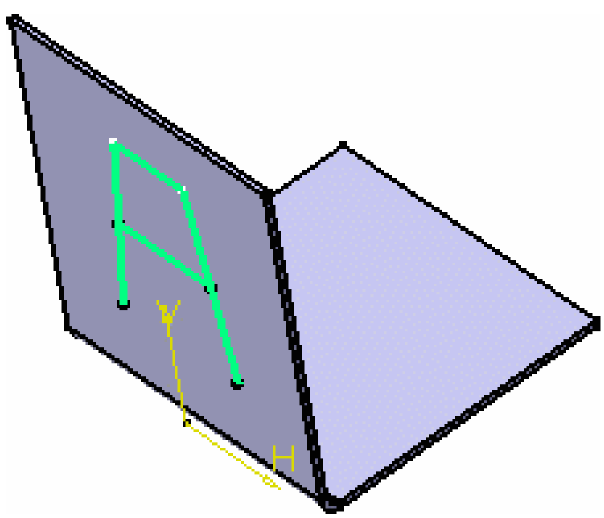
This task shows how to create curves or points from a sketch (as designed using the Sketcher) or from existing curves or points, onto a Sheet Metal part; and to fold/unfold it, just as other Sheet Metal elements.
This is especially useful when:

- you want to generate a logotype
- you want to define an area for chemical milling
- you want to create a cutout (pocket) to solve the overlapping of walls for example (the overlapping can be checked with the Sheet Metal Production product).

If you use SheetMetal Design, open the Mapping1.CATPart document.
If you use Generative Sheetmetal Design, open the NEWMapping1.CATPart document.
If you use Aerospace SheetMetal Design, open the Aero_Mapping.CATPart document.

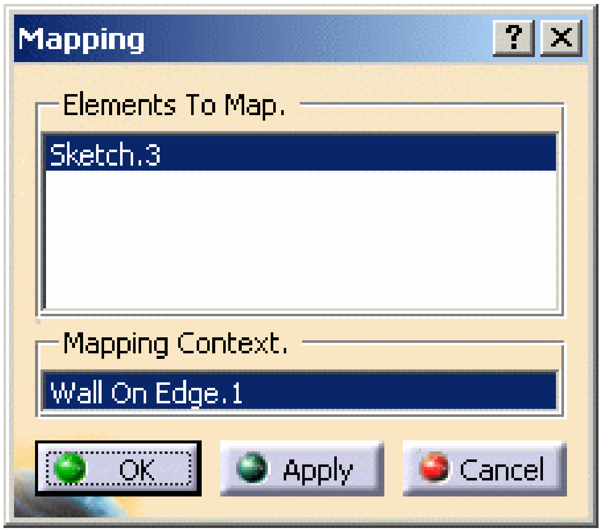
These samples already contain a pre-defined sketch that will be mapped onto the part.

Otherwise, you would need to defined a sketch by entering the Sketcher workbench , selecting the wall onto which the curve should lie, and drawing the sketch you wish.



1. Make sure the sketch is selected, and click the **Point or Curve Mapping** icon .

The Elements To Map definition dialog box is displayed, indicating which elements have been selected for mapping.



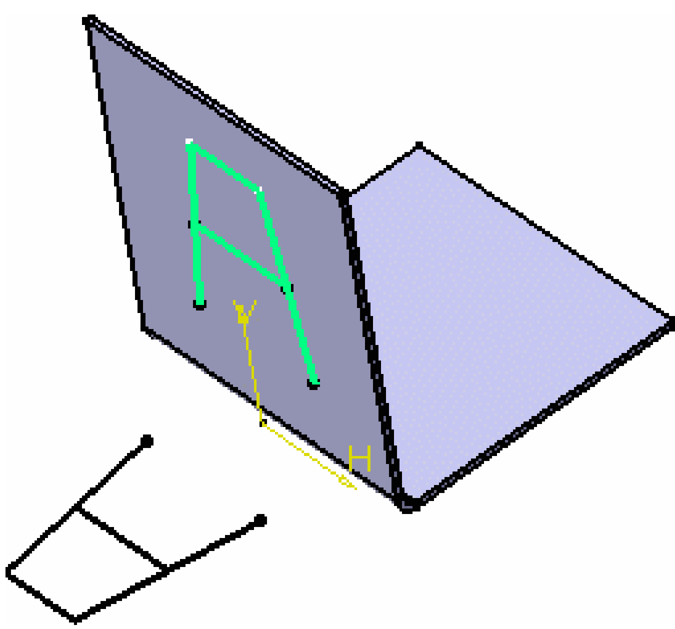
You can manage the list of elements:

- to remove an element, select it from the list and use the **Clear selection** contextual menu
- to add an element, select it directly in the geometry.
  Order in the list does not matter.

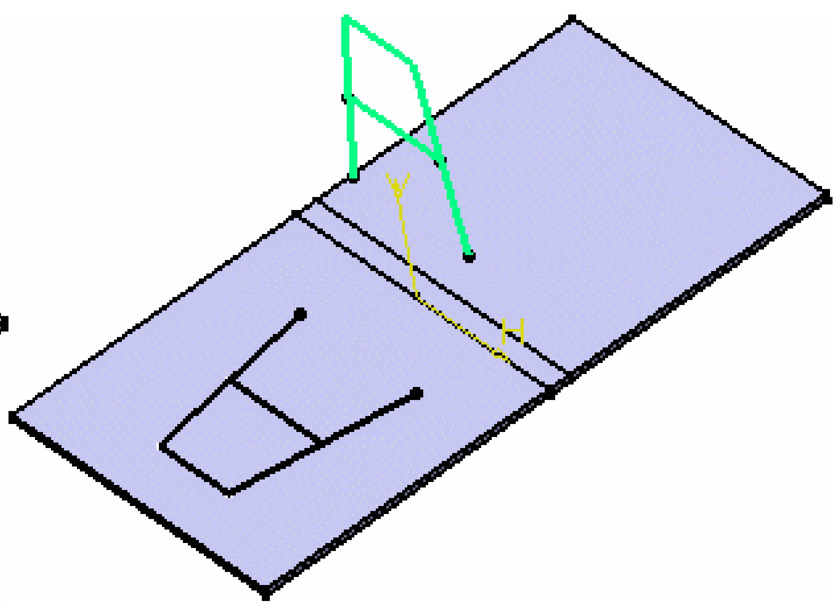
2. Select the **Mapping Context**, that is the element of the part on which the curve should be generated when folding or unfolding.

The **Mapping Context** is not necessarily the support element on which the element to be mapped has been drawn. Indeed, by default, the **Mapping Context** is the last Sheet Metal feature that has been created or modified, that is the current feature in the specification tree.

3. Click **OK**. The curve mapping is created and added in the specification tree.



Folded view of the curve mapping



Unfolded view of the curve mapping

- You can select several sketches/curves/points to be mapped at a time.
- Mapped curves can be created across several walls and bends.

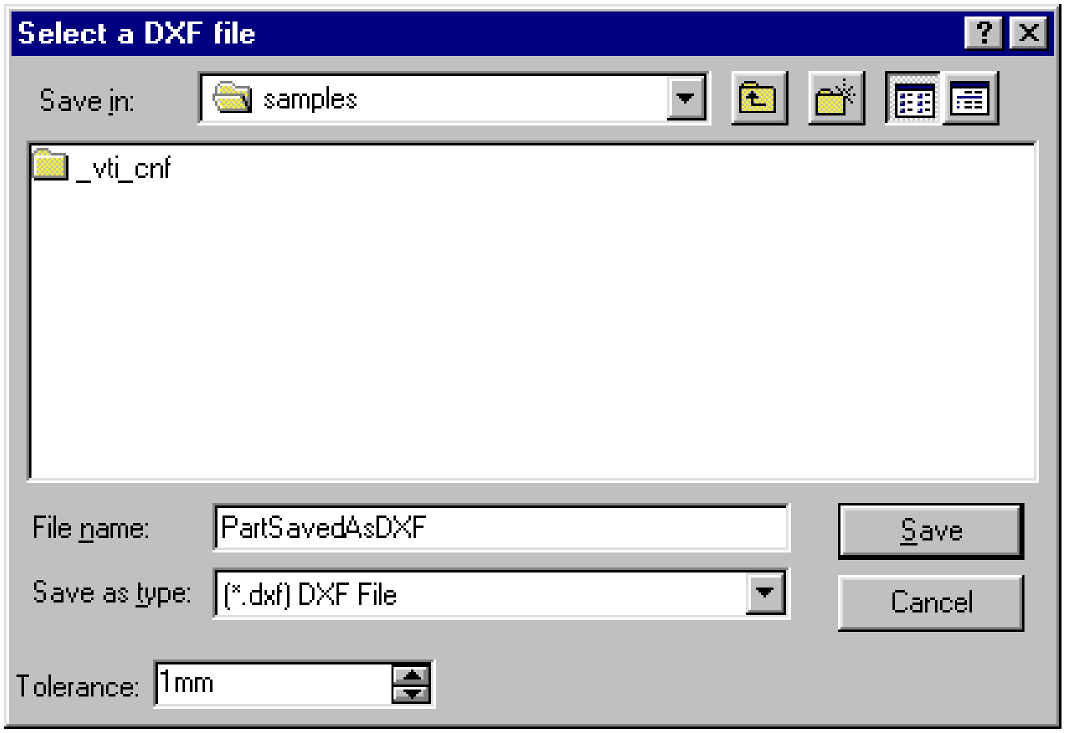
Saving As DXF

This task shows how save the generated geometry as a DXF document.

To perform this scenario, you can open any sheet metal sample provided in this user's guide.

1. Click the **Save As DXF** icon .

   The **Select a DXF file** dialog box is displayed allowing you to navigate to the correct location.



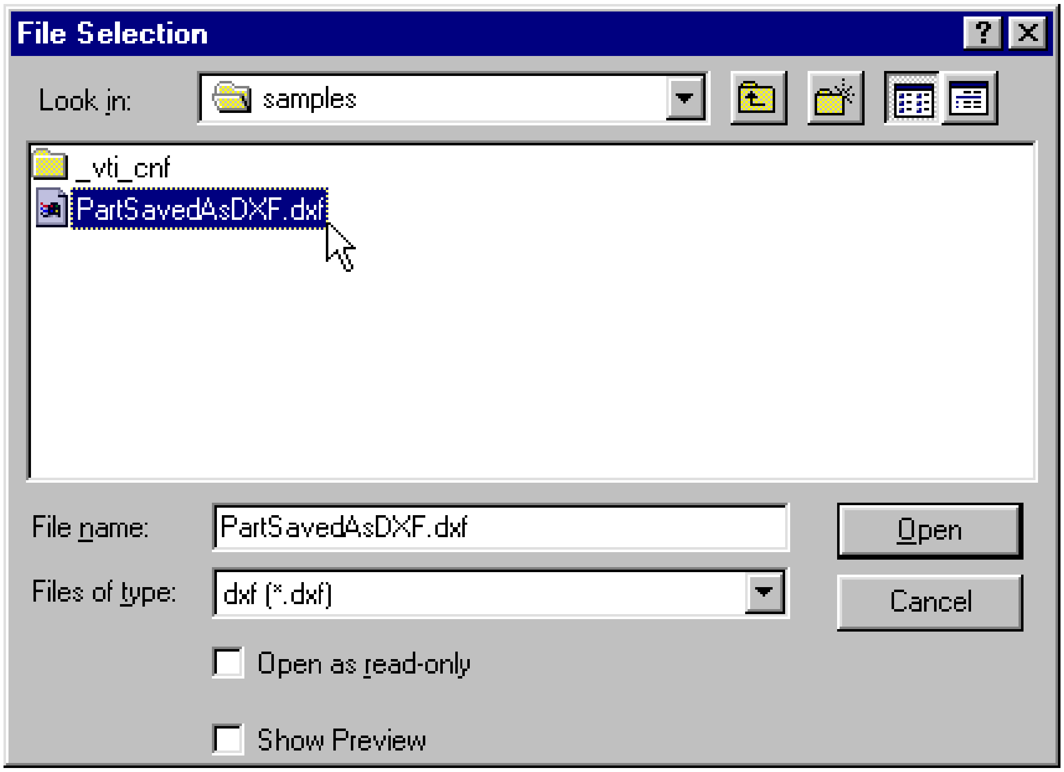
2. Indicate the correct path and file name (we saved it as PartSavedAsDXF.dxf).
3. Click **Save**.

The geometry has been saved, and can be imported as a DXF file in any system supporting this type.

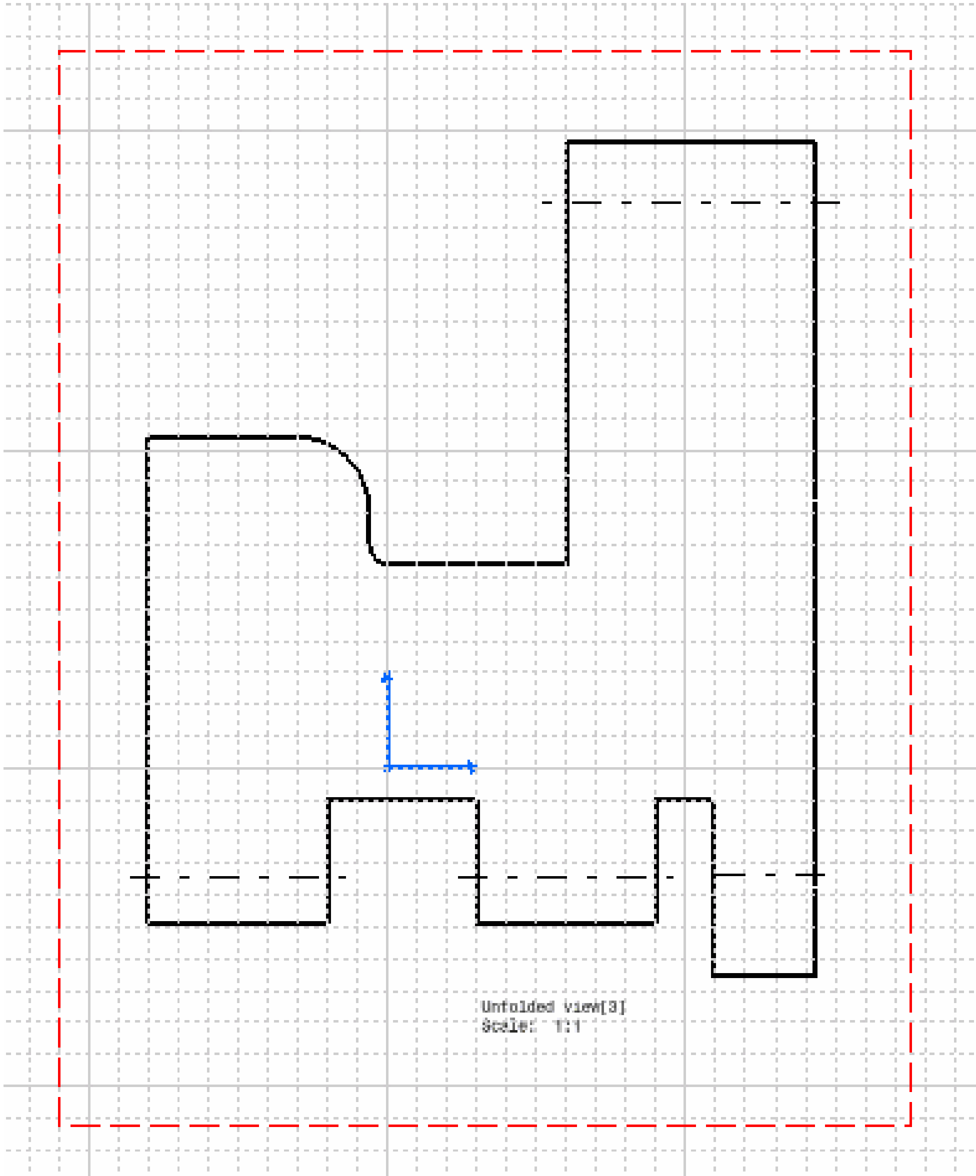
4. Close the CATPart Document.
5. Click **File -> Open**.

6. From the File Selection dialog box, choose the .dxf file type, then select the saved part (PartSavedAsDXF.dxf).

7. Click **Open**.



The unfolded view of the part is opened within the Drafting workbench, because the .dxf type is recognized as being a drafting type of document.



Note that the axes of bends and planar hems, tear drops, or flanges are automatically displayed on the drawing.

Refer also to DXF/DWG Settings from the *Infrastructure User's Guide*.

Interoperability With Wireframe

Creating Points

This task shows the various methods for creating points:

- by coordinates
- on a curve
- on a plane
- on a surface
- at a circle/sphere center
- tangent point on a curve
- between

Open the Points3D1.CATPart document.

1. Click the **Point** icon .

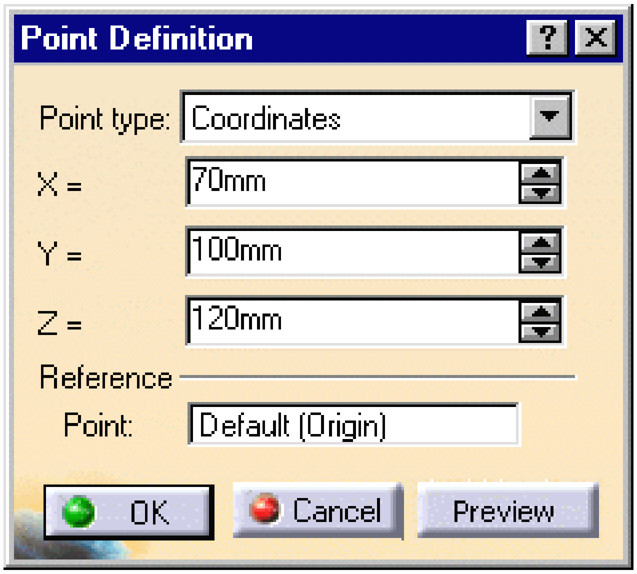
The Point Definition dialog box appears.

2. Use the combo to choose the desired point type.

Coordinates

- Enter the X, Y, Z coordinates in the current axis-system.
- Optionally, select a reference point.

The corresponding point is displayed.



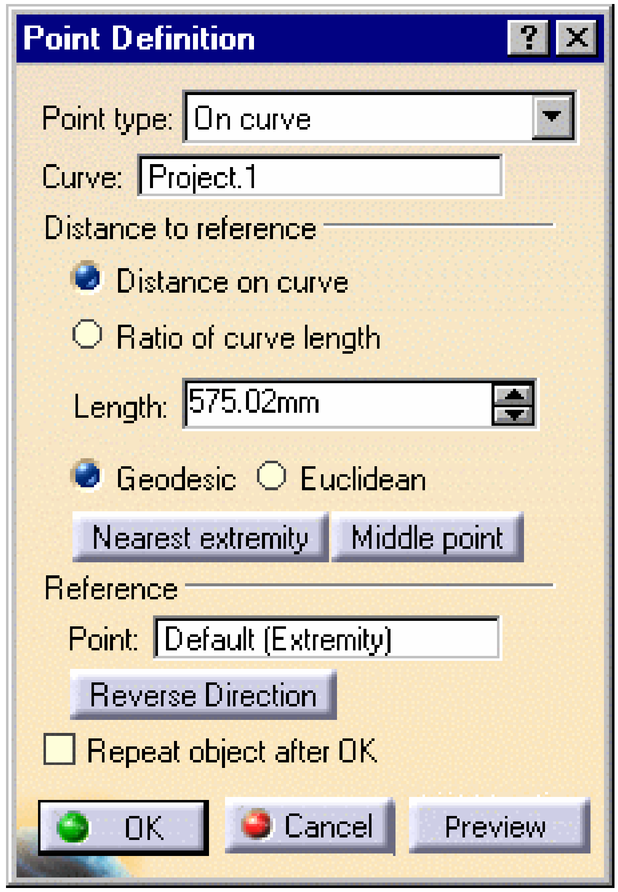
When creating a point within a user-defined axis-system, note that the **Coordinates in absolute axis-system** check button is added to the dialog box, allowing you to be define, or simply find out, the point's coordinates within the document's default axis-system.
If you create a point using the coordinates method and an axis system is already defined and set as current, the point's coordinates are defined according to current the axis system. As a consequence, the point's coordinates are not displayed in the specification tree.

The axis system must be different from the absolute axis.

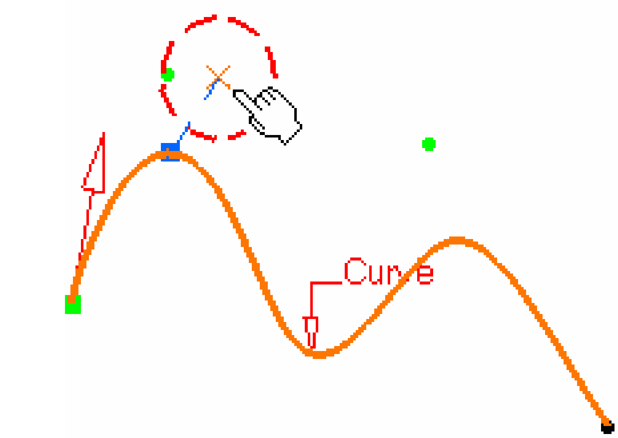
On curve

- Select a curve
- Optionally, select a reference point.

  If this point is not on the curve, it is projected onto the curve.
  If no point is selected, the curve's extremity is used as reference.

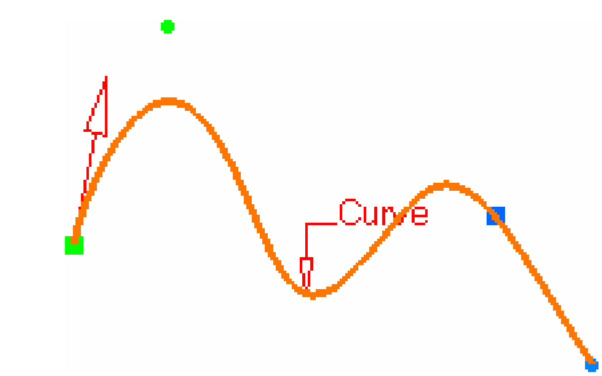


- Select an option point to determine whether the new point is to be created:
  - at a given distance along the curve from the reference point
  - a given ratio between the reference point and the curve's extremity.



- Enter the distance or ratio value.
  If a distance is specified, it can be:
  - a geodesic distance: the distance is measured along the curve
  - an Euclidean distance: the distance is measured in relation to the reference point (absolute value).

  The corresponding point is displayed.



If the reference point is located at the curve's extremity, even if a ratio value is defined, the created point is always located at the end point of the curve.

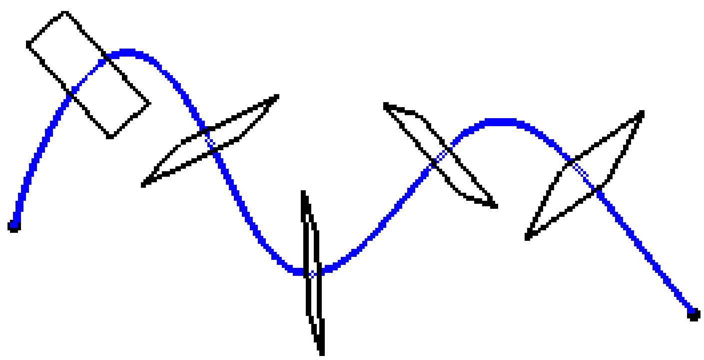
You can also:

- click the **Nearest extremity** button to display the point at the nearest extremity of the curve.
- click the **Middle Point** button to display the mid-point of the curve.

Be careful that the arrow is orientated towards the inside of the curve (providing the curve is not closed) when using the **Middle Point** option.

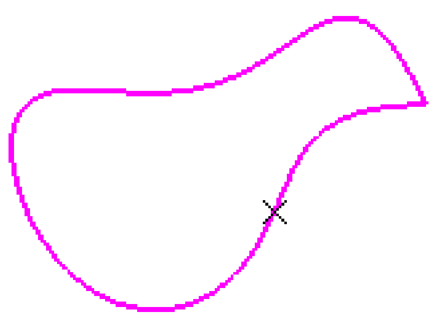
- use the **Reverse Direction** button to display:
  - the point on the other side of the reference point (if a point was selected originally)
  - the point from the other extremity (if no point was selected originally).
- click the **Repeat object after OK** if you wish to create equidistant points on the curve, using the currently created point as the reference, as described in Creating Multiple Points in the Wireframe and Surface User's Guide.

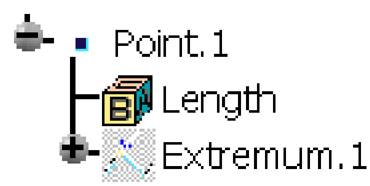
You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new geometrical set by checking the **Create in a new geometrical set** button.
If the button is not checked the instances are created in the current geometrical set .



- If the curve is infinite and no reference point is explicitly given, by default, the reference point is the projection of the model's origin
- If the curve is a closed curve, either the system detects a vertex on the curve that can be used as a reference point, or it creates an extremum point, and highlights it (you can then select another one if you wish) or the system prompts you to manually select a reference point.

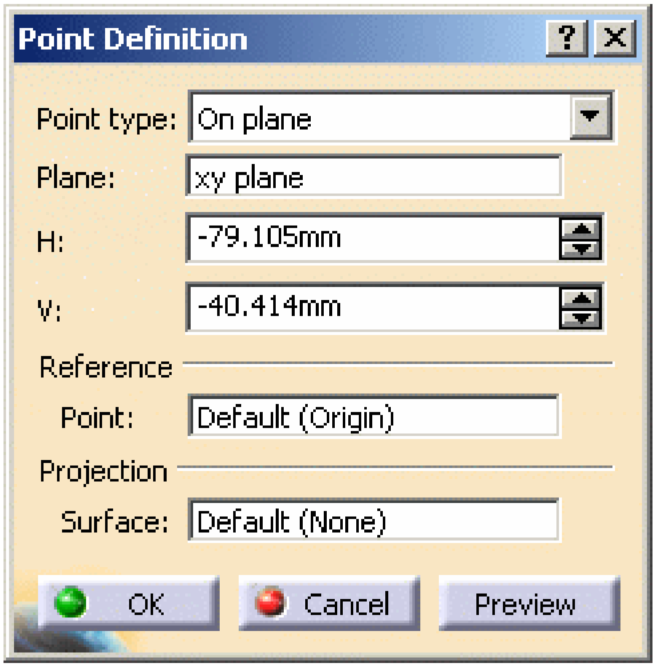
Extremum points created on a closed curve are now aggregated under their parent command and put in no show in the specification tree.





On plane

- Select a plane.
- Optionally, select a point to define a reference for computing coordinates in the plane.

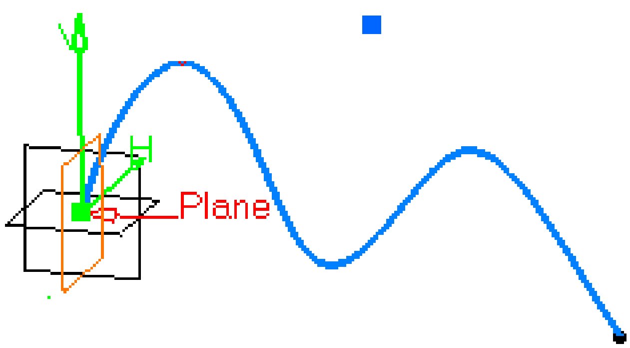


If no point is selected, the projection of the model's origin on the plane is taken as reference.

- Optionally, select a surface on which the point is projected normally to the plane.

If no surface is selected, the behavior is the same.

Furthermore, the reference direction (H and V vectors) is computed as follows:
With N the normal to the selected plane (reference plane), H results from the vectorial product of Z and N (H = Z^N).
If the norm of H is strictly positive then V results from the vectorial product of N and H (V = N^H).
Otherwise, V = N^X and H = V^N.

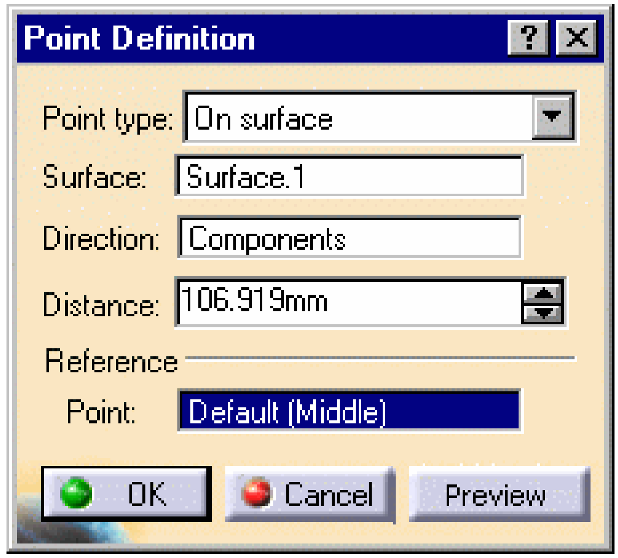


Would the plane move, during an update for example, the reference direction would then be projected on the plane.

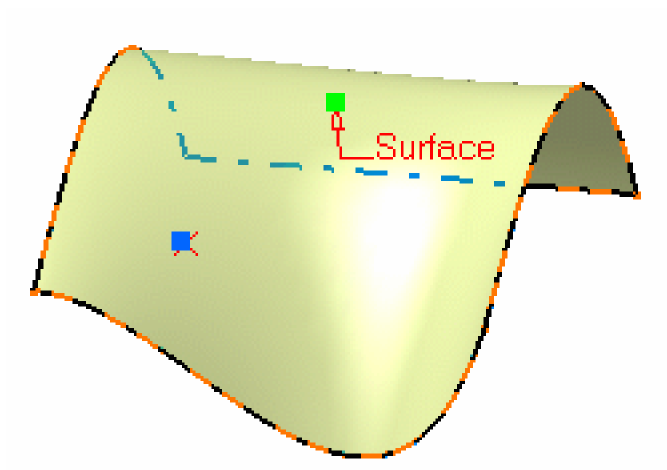
- Click in the plane to display a point.

On surface

- Select the surface where the point is to be created.



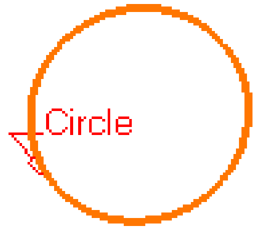
- Optionally, select a reference point. By default, the surface's middle point is taken as reference.
- You can select an element to take its orientation as reference direction or a plane to take its normal as reference direction.
  You can also use the contextual menu to specify the X, Y, Z components of the reference direction.
- Enter a distance along the reference direction to display a point.

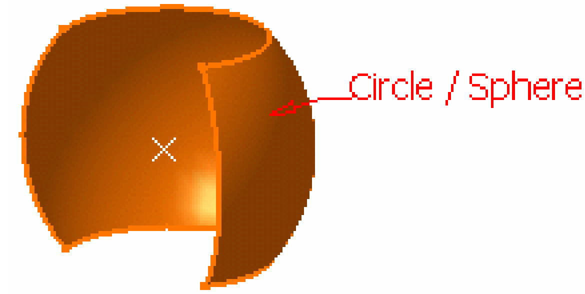


Circle/Sphere center

- Select a circle, circular arc, or ellipse, or
- Select a sphere or a portion of sphere.

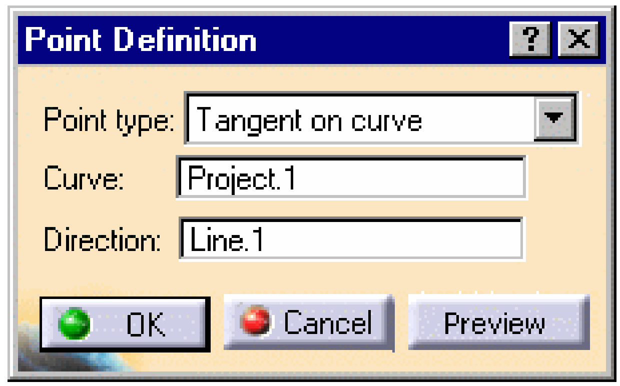
A point is displayed at the center of the selected element.





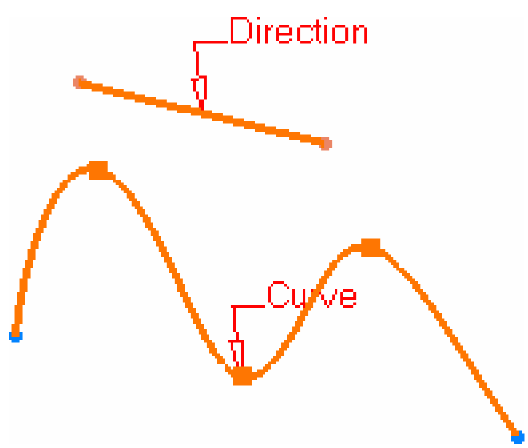
Tangent on curve

- Select a planar curve and a direction line.



A point is displayed at each tangent.

The Multi-Result Management dialog box is displayed because several points are generated.

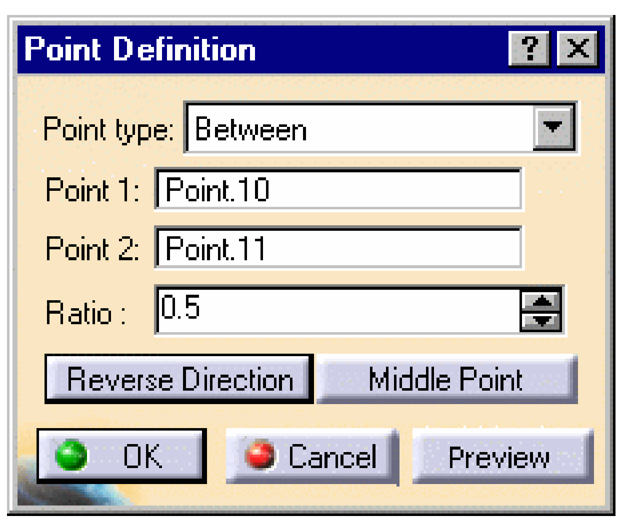


- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.

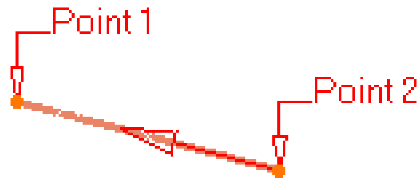
For further information, refer to the Managing Multi-Result Operations chapter.

Between

- Select any two points.

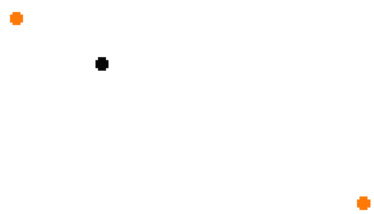


- Enter the ratio, that is the percentage of the distance from the first selected point, at which the new point is to be.
  You can also click **Middle Point** button to create a point at the exact midpoint (ratio = 0.5).



Be careful that the arrow is orientated towards the inside of the curve (providing the curve is not closed) when using the **Middle Point** option.

- Use the **Reverse direction** button to measure the ratio from the second selected point.



If the ratio value is greater than 1, the point is located on the virtual line beyond the selected points.

3. Click OK to create the point.

The point (identified as Point.xxx) is added to the specification tree.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a point in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Creating Lines

This task shows the various methods for creating lines:

- point to point
- point and direction
- angle or normal to curve
- tangent to curve
- normal to surface
- bisecting

It also shows you how to create a line up to an element, define the length type and automatically reselect the second point.

Open the Lines1.CATPart document.

1. Click the **Line** icon .

   The Line Definition dialog box is displayed.

2. Use the drop-down list to choose the desired line type.

A line type will be proposed automatically in some cases depending on your first element selection.

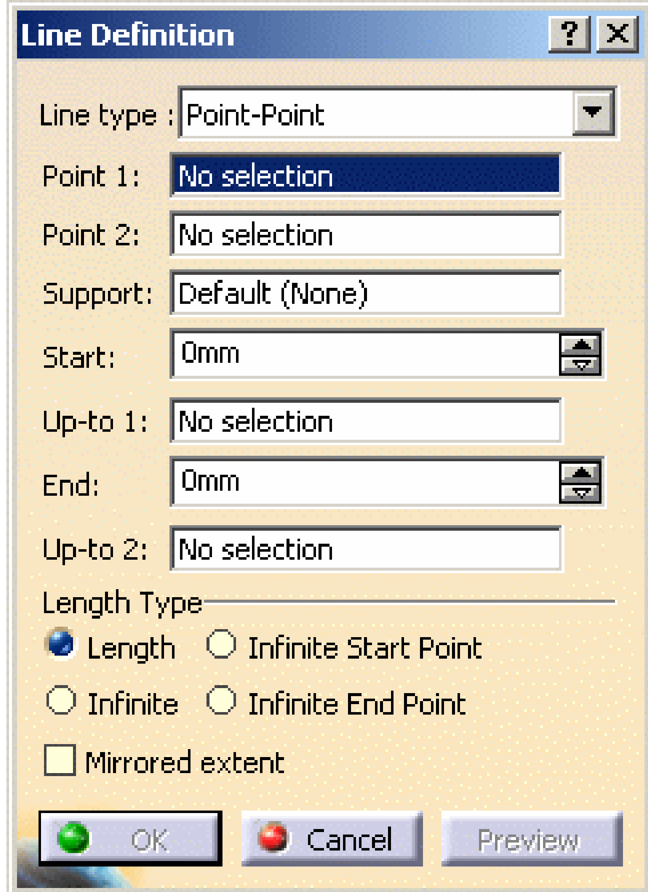
Defining the line type

Point - Point

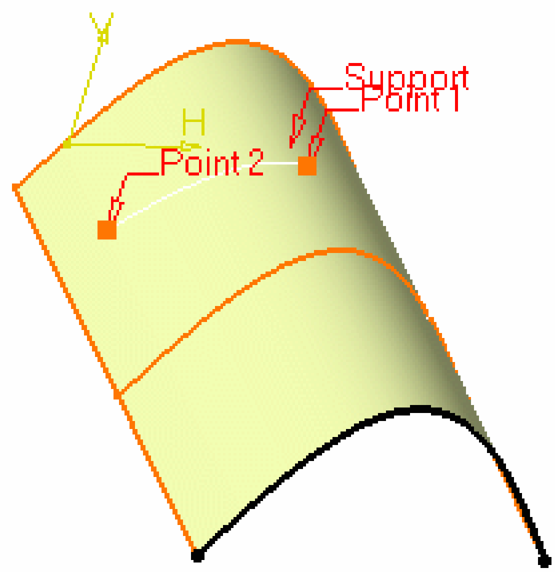
This command is only available with the Generative Shape Design 2 product.

- Select two points.

A line is displayed between the two points.
Proposed **Start** and **End** points of the new line are shown.

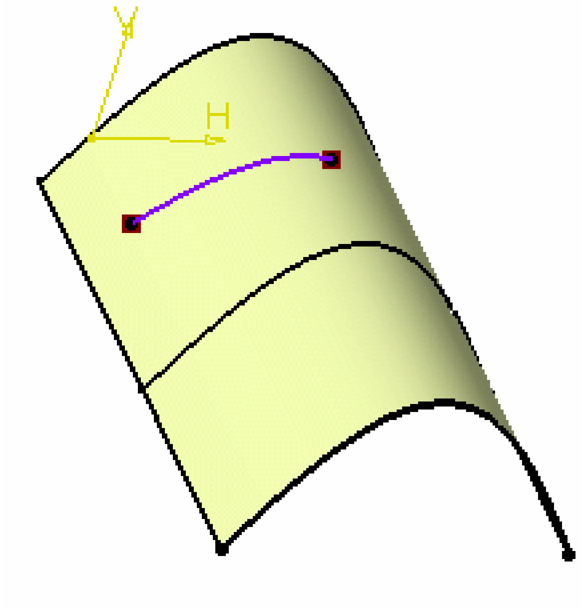


- If needed, select a support surface.
  In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below).
  If no surface is selected, the line is created between the two points based on the shortest distance.



If you select two points on closed surface (a cylinder for example), the result may be unstable. Therefore, it is advised to split the surface and only keep the part on which the geodesic line will lie.

The geodesic line is not available with the Wireframe and Surface workbench.

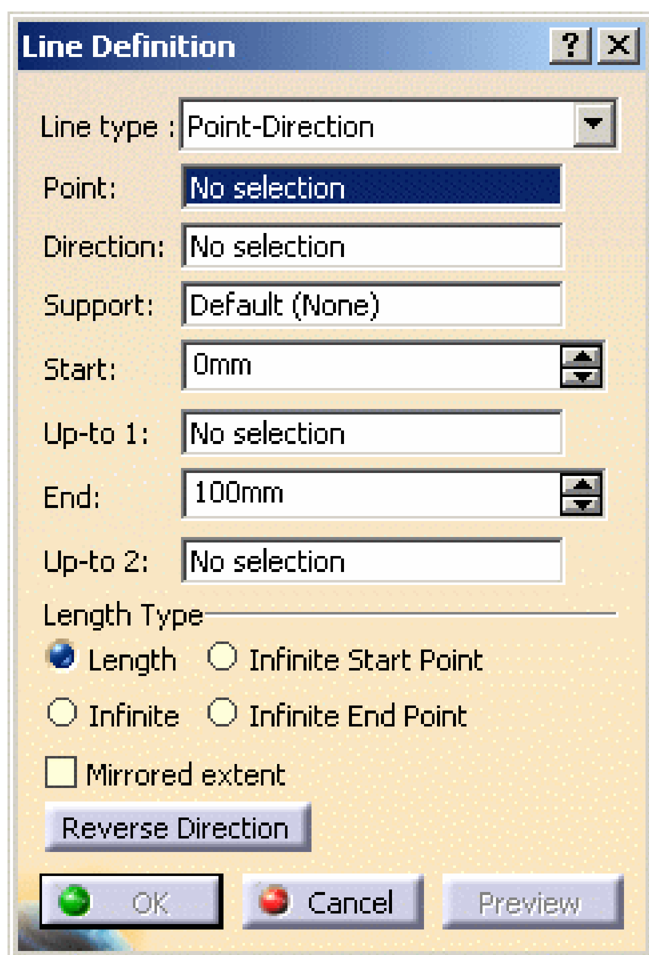


- Specify the **Start** and **End** points of the new line, that is the line endpoint location in relation to the points initially selected. These **Start** and **End** points are necessarily beyond the selected points, meaning the line cannot be shorter than the distance between the initial points.
- Check the **Mirrored extent** option to create a line symmetrically in relation to the selected **Start** and **End** points.

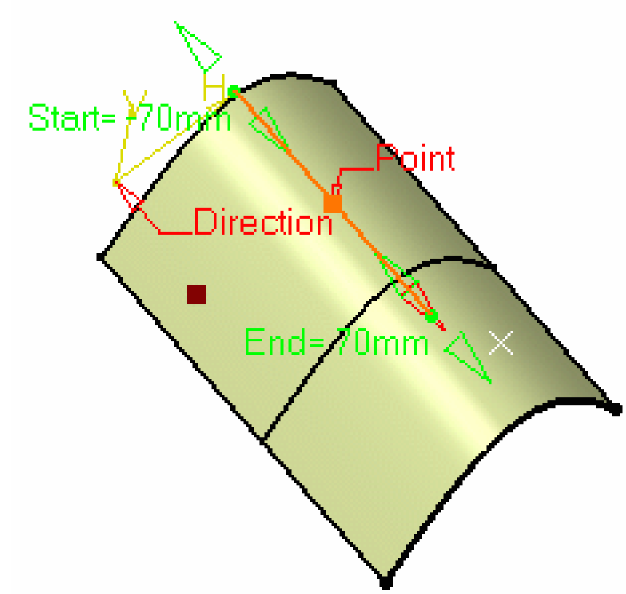
The projections of the 3D point(s) must already exist on the selected support.

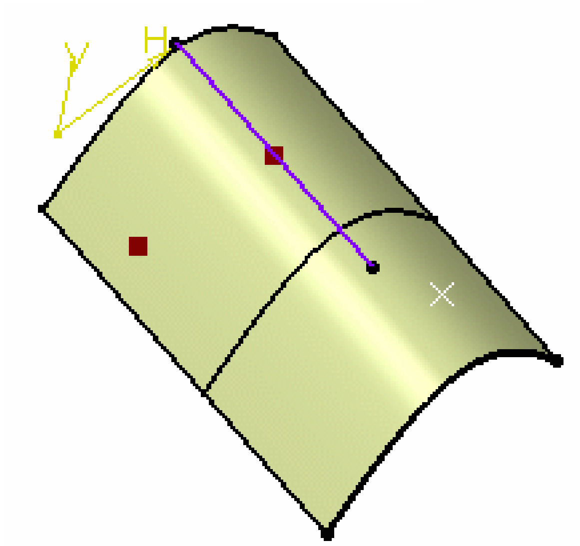
Point - Direction

- Select a reference **Point** and a **Direction** line.
  A vector parallel to the direction line is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.



- Specify the **Start** and **End** points of the new line.
  The corresponding line is displayed.





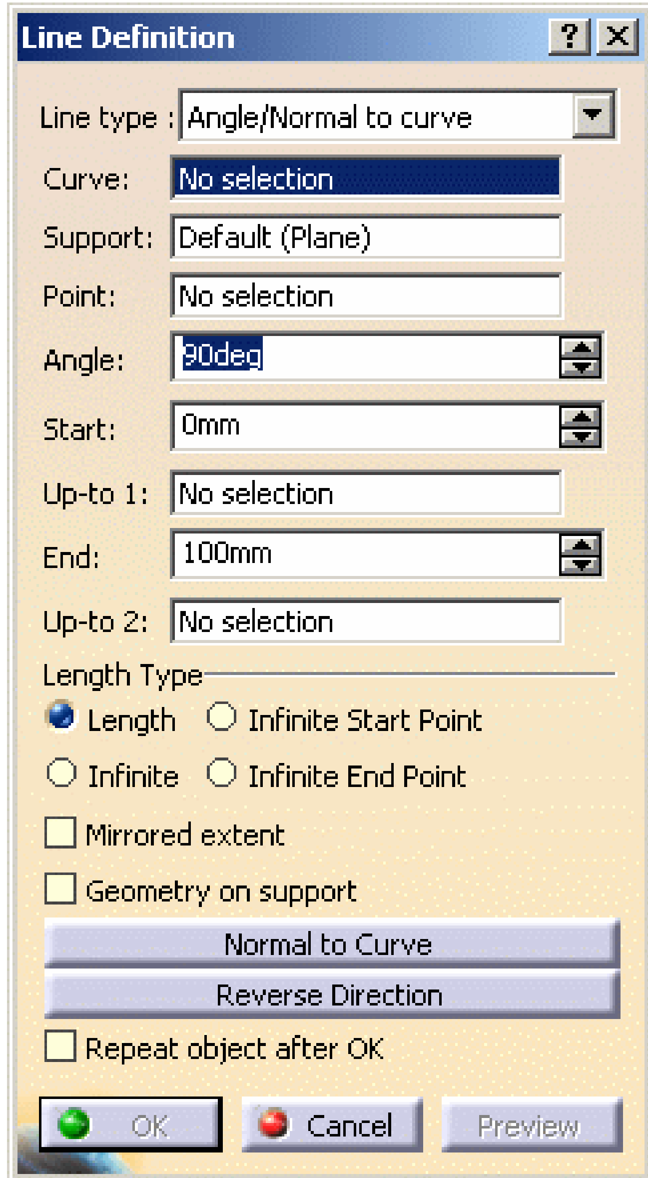
The projections of the 3D point(s) must already exist on the selected support.

Angle or Normal to curve

- Select a reference **Curve** and a **Support** surface containing that curve.

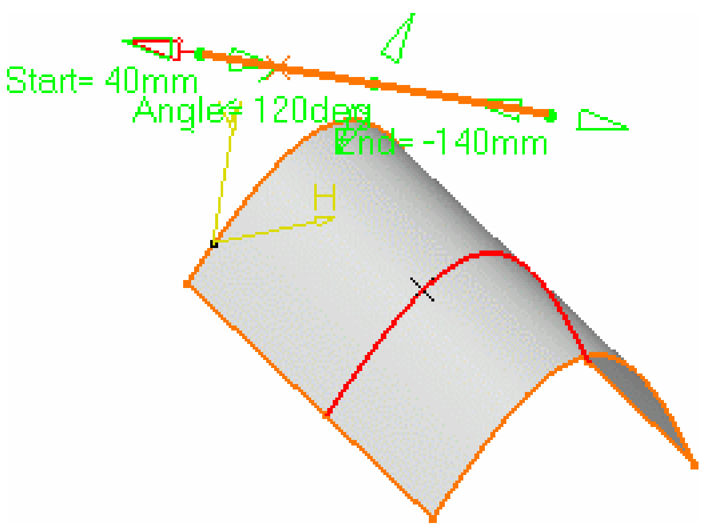
- If the selected curve is planar, then the **Support** is set to Default (Plane).

- If an explicit **Support** has been defined, a contextual menu is available to clear the selection.



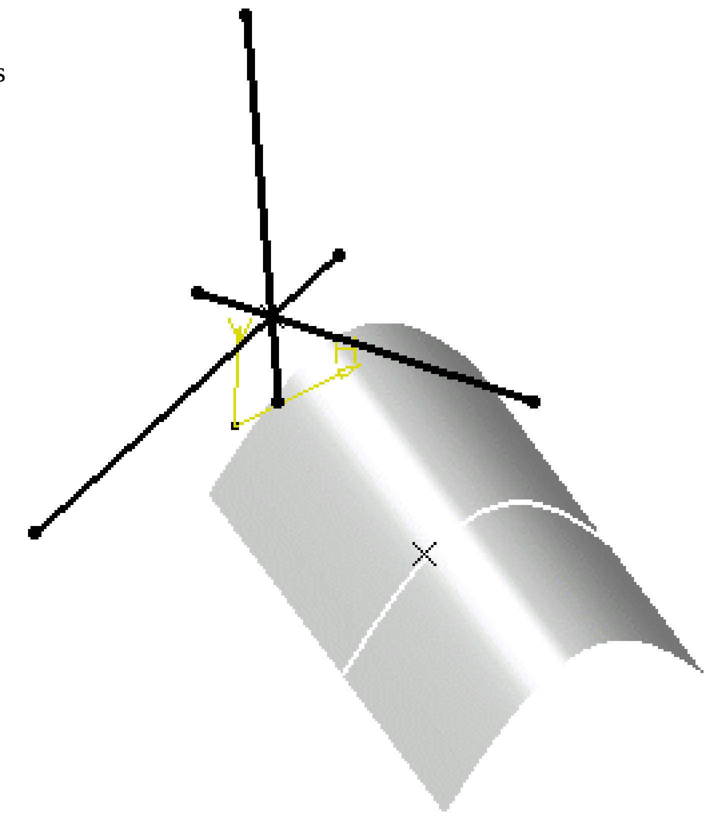
- Select a **Point** on the curve.
- Enter an **Angle** value.

A line is displayed at the given angle with respect to the tangent to the reference curve at the selected point. These elements are displayed in the plane tangent to the surface at the selected point.
You can click on the **Normal to Curve** button to specify an angle of 90 degrees.
Proposed **Start** and **End** points of the line are shown.



- Specify the **Start** and **End** points of the new line. The corresponding line is displayed.

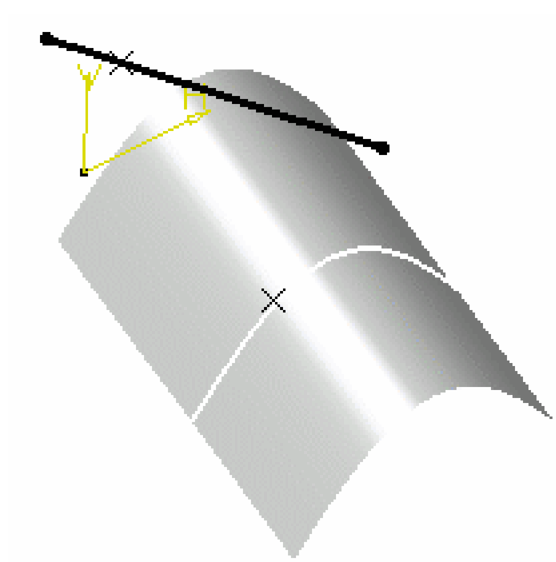
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.

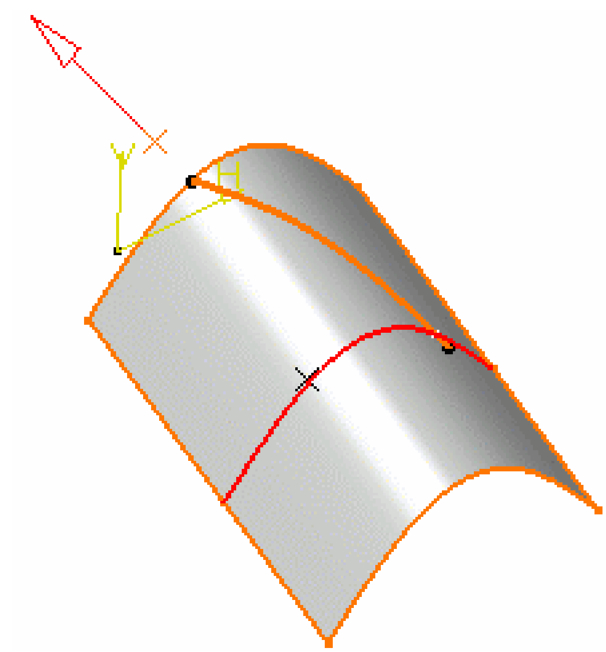


As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.

You can select the **Geometry on Support** check box if you want to create a geodesic line onto a support surface.

The figure below illustrates this case.

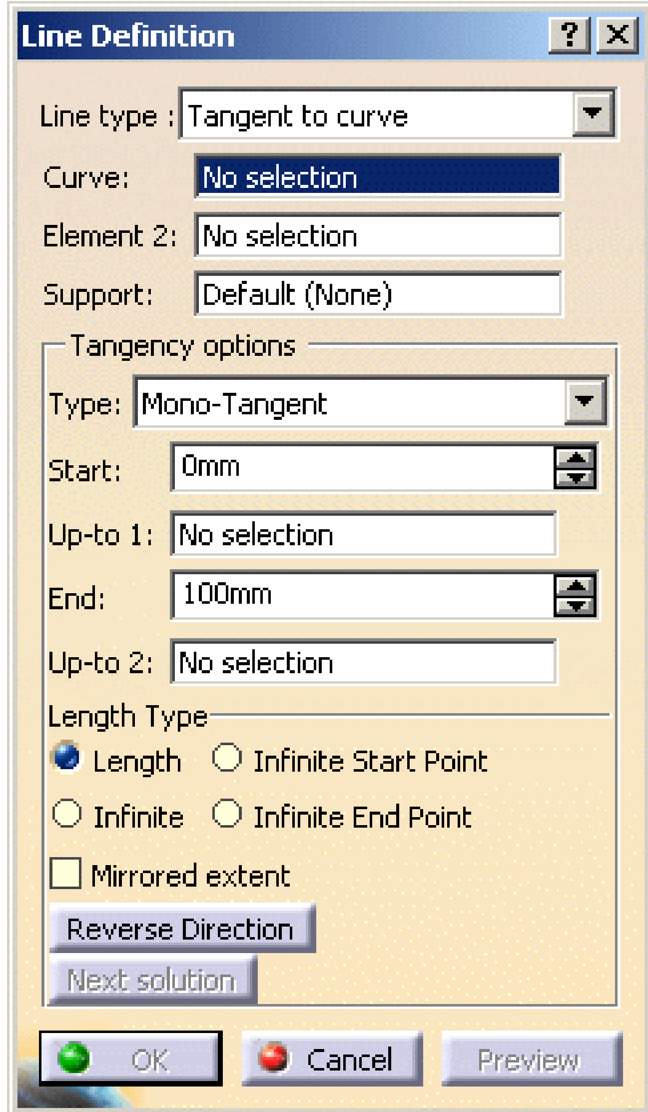




Geometry on support option not checked *Geometry on support option checked*

This line type enables to edit the line's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

Tangent to curve

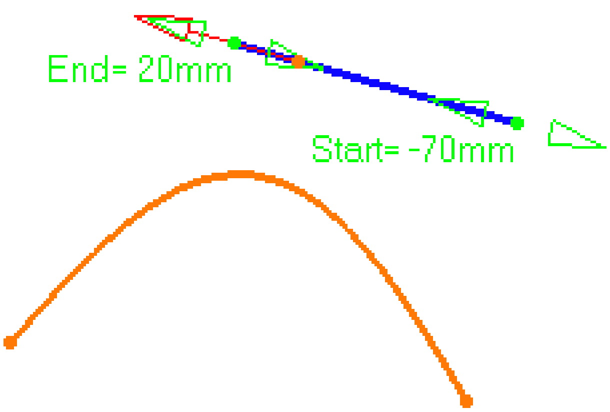


- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
  - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
  - If a second curve is selected (or a point in bi-tangent mode), you need to select a support plane. The line will be tangent to both curves.

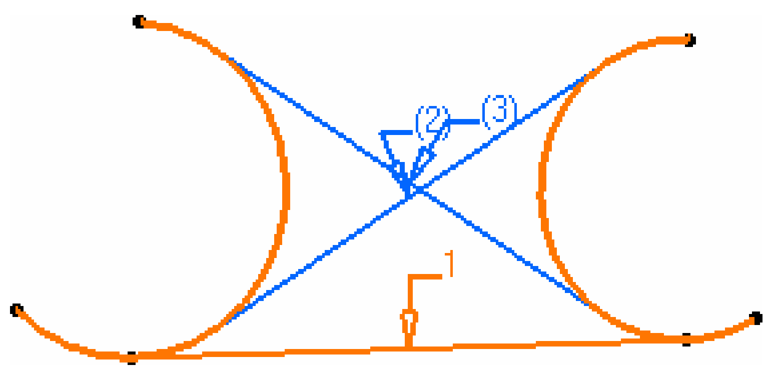
- If the selected curve is a line, then the **Support** is set to Default (Plane).

- If an explicit **Support** has been defined, a contextual menu is available to clear the selection.

When several solutions are possible, you can choose one (displayed in red) directly in the geometry, or using the **Next Solution** button.

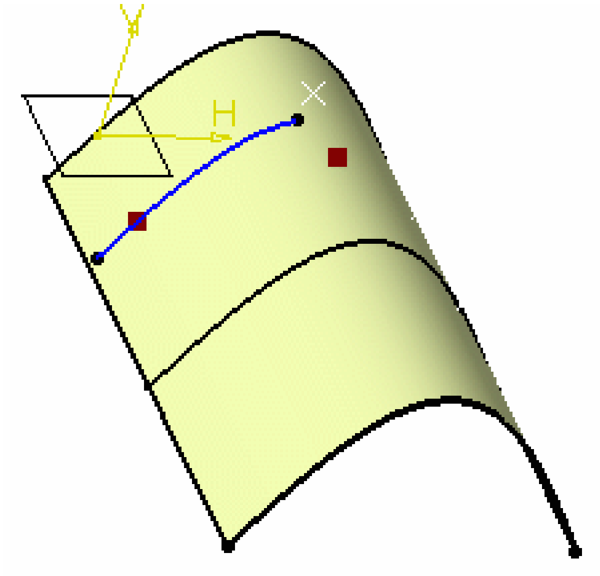


Line tangent to curve at a given point



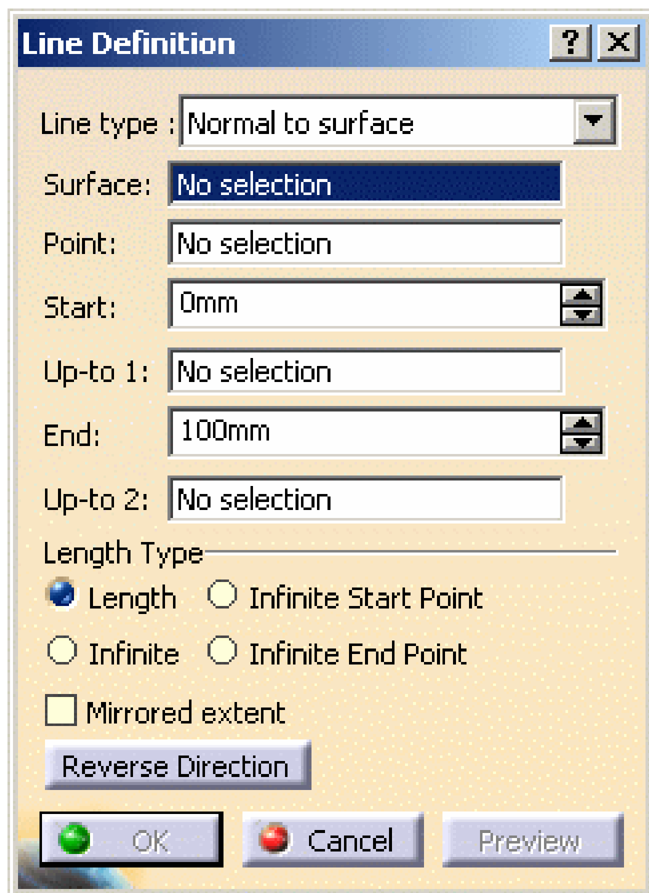
Line tangent to two curves

- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.



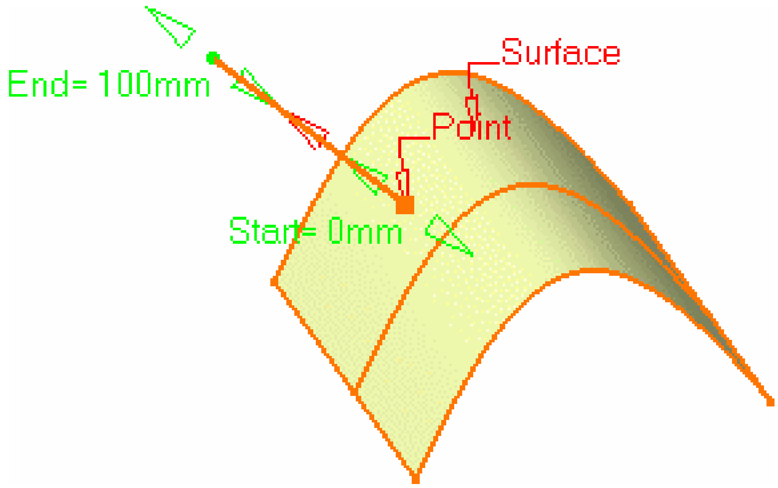
Normal to surface

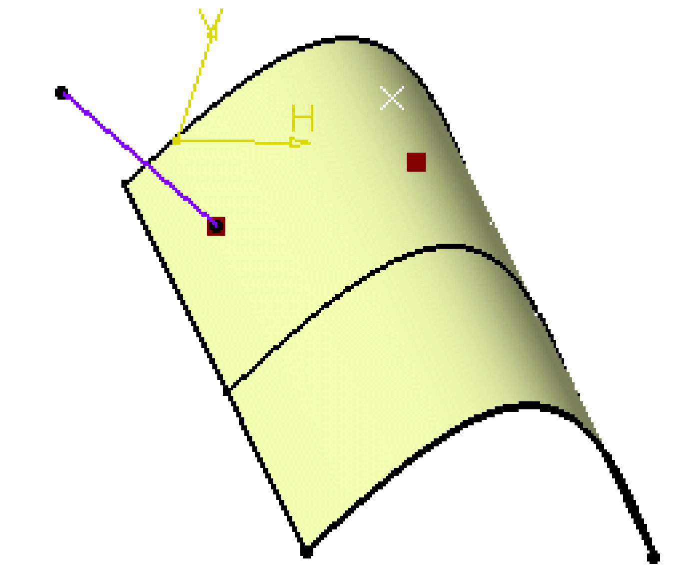
- Select a reference **Surface** and a **Point**.
  A vector normal to the surface is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.



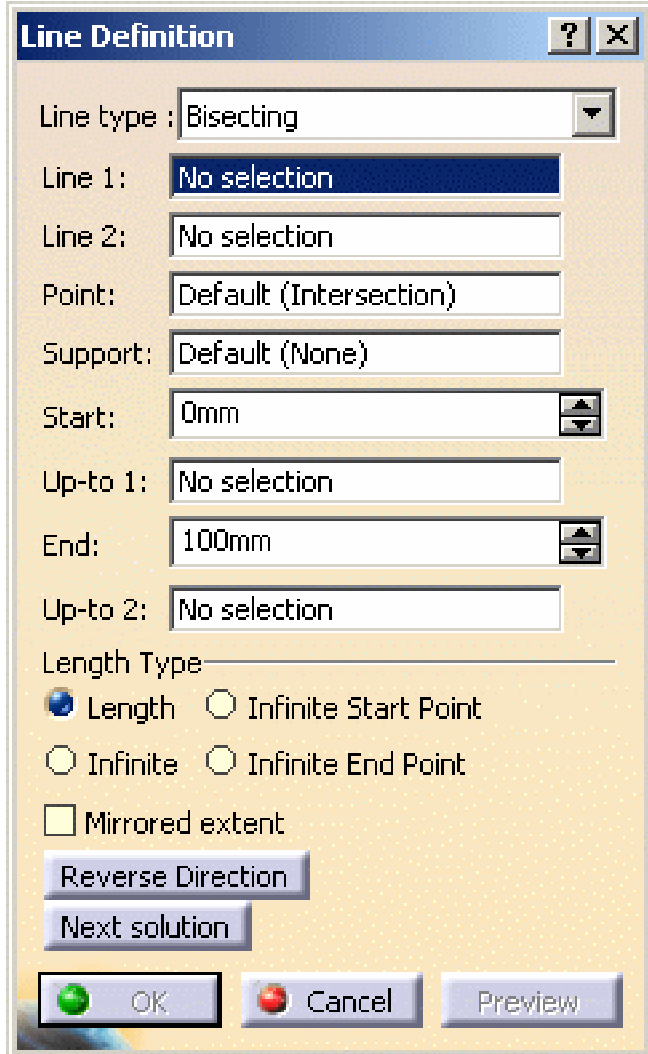
If the point does not lie on the support surface, the minimum distance between the point and the surface is computed, and the vector normal to the surface is displayed at the resulted reference point.

- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.

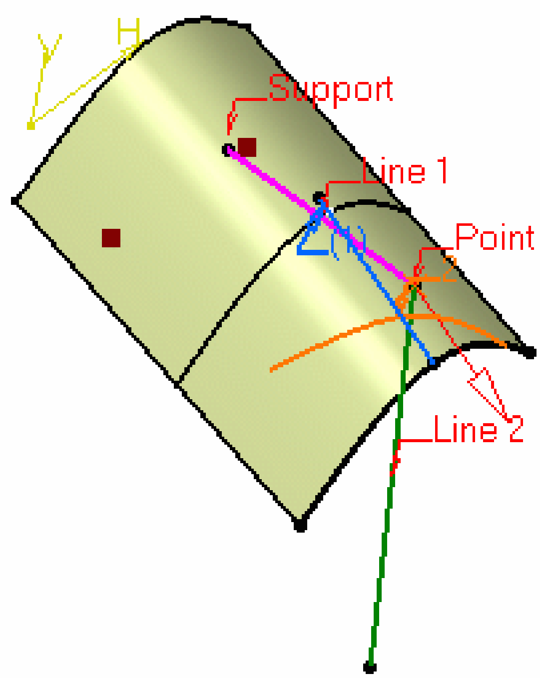




Bisecting



- Select two lines. Their bisecting line is the line splitting in two equals parts the angle between these two lines.
- Select a point as the starting point for the line. By default it is the intersection of the bisecting line and the first selected line.



- Select the support surface onto which the bisecting line is to be projected, if needed.
- Specify the line's length in relation to its starting point (**Start** and **End** values for each side of the line in relation to the default end points).
  The corresponding bisecting line, is displayed.
- You can choose between two solutions, using the **Next Solution** button, or directly clicking the numbered arrows in the geometry.

3. Click **OK** to create the line.

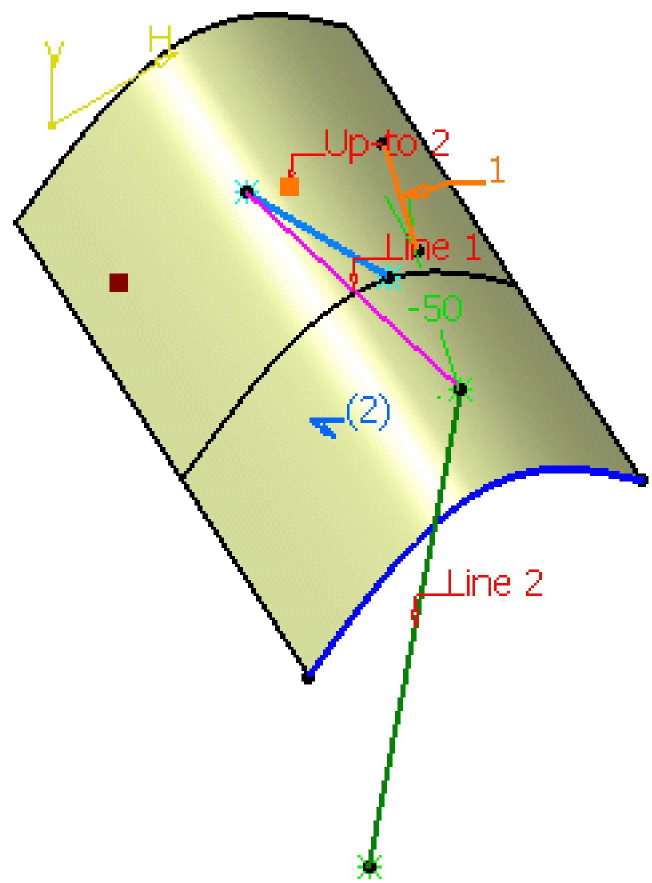
   The line (identified as Line.xxx) is added to the specification tree.

- Regardless of the line type, **Start** and **End** values are specified by entering distance values or by using the graphic manipulators.
- **Start** and **End** values should not be the same.
- Check the **Mirrored extent** option to create a line symmetrically in relation to the selected **Start** point.
  It is only available with the **Length** Length type.
- In most cases, you can select a support on which the line is to be created. In this case, the selected point(s) is projected onto this support.
- You can reverse the direction of the line by either clicking the displayed vector or selecting the **Reverse Direction** button (not available with the point-point line type).

Creating a line up to an element

This capability allows you to create a line up to a point, a curve, or a surface.

- It is available with all line types, but the Tangent to curve type.



Up to a point

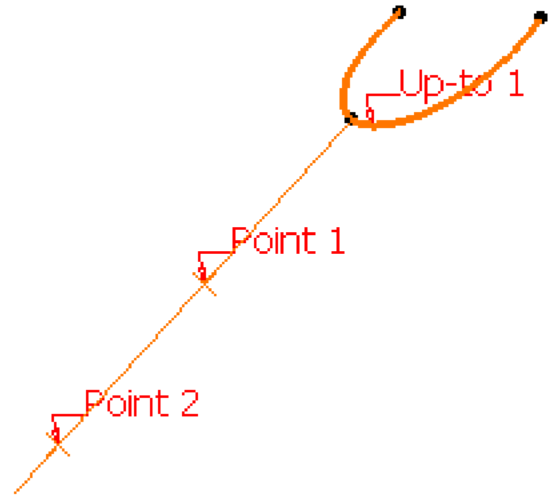
- Select a point in the **Up-to 1** and/or **Up-to 2** fields.

  Here is an example with the Bisecting line type, the **Length** Length type, and a point as **Up-to 2** element.

Up to a curve

- Select a curve in the **Up-to 1** and/or **Up-to 2** fields.

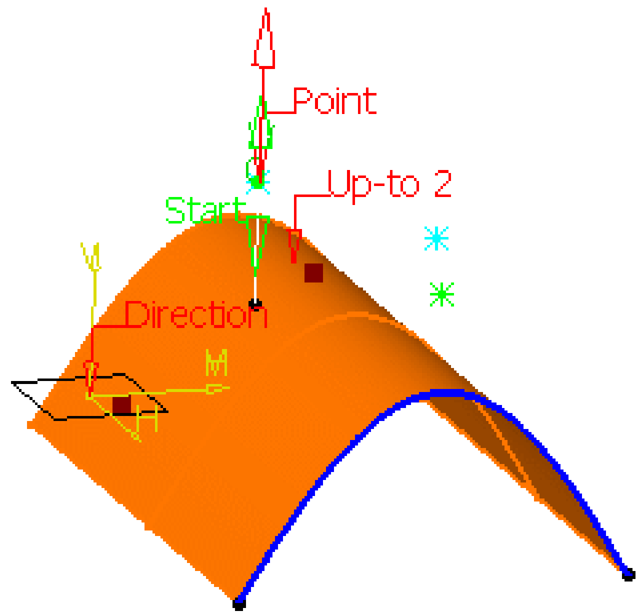
  Here is an example with the Point-Point line type, the **Infinite End** Length type, and a curve as the **Up-to 1** element.



Up to a surface

- Select a surface in the **Up-to 1** and/or **Up-to 2** fields.

  Here is an example with the Point-Direction line type, the **Length** Length type, and the surface as the **Up-to 2** element.



- If the selected Up-to element does not intersect with the line being created, then an extrapolation is performed. It is only possible if the element is linear and lies on the same plane as the line being created.
  However, no extrapolation is performed if the Up-to element is a curve or a surface.
- The **Up-to 1** and **Up-to 2** fields are grayed out with the **Infinite** Length type, the **Up-to 1** field is grayed out with the **Infinite Start** Length type, the Up-to 2 field is grayed out with the **Infinite End** Length type.
- The **Up-to 1** field is grayed out if the **Mirrored extent** option is checked.
- In the case of the Point-Point line type, **Start** and **End** values cannot be negative.

Defining the length type

- Select the Length Type:
  - **Length**: the line will be defined according to the **Start** and **End** points values
  - **Infinite**: the line will be infinite
  - **Infinite Start Point**: the line will be infinite from the **Start** point
  - **Infinite End Point**: the line will be infinite from the **End** point

By default, the Length type is selected.
The **Start** and/or the **End** points values will be greyed out when one of the **Infinite** options is chosen.

Reselecting automatically a second point

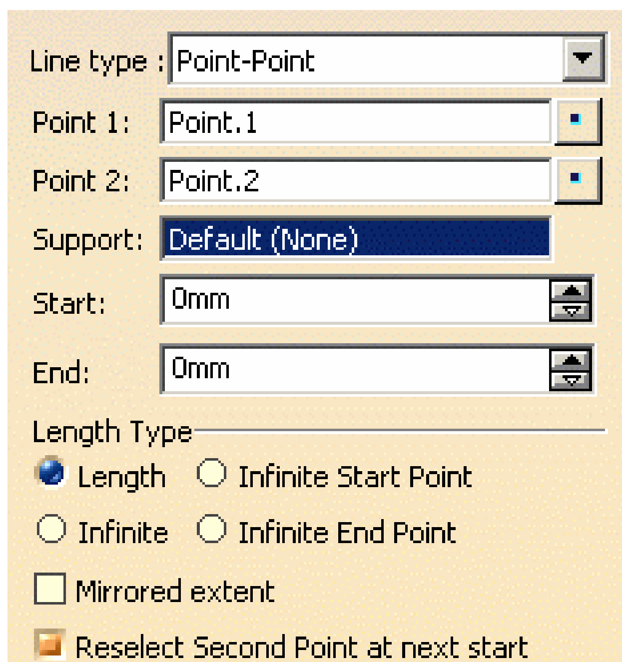
This capability is only available with the Point-Point line method.

1. Double-click the **Line** icon.

   The Line dialog box is displayed.

2. Create the first point.

The **Reselect Second Point at next start** option appears in the Line dialog box.

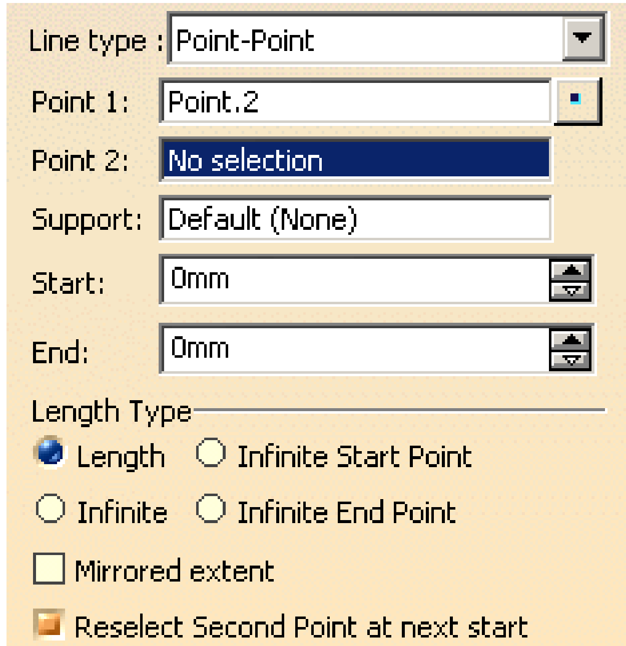


3. Check it to be able to later reuse the second point.

4. Create the second point.

5. Click OK to create the first line.

The Line dialog box opens again with the first point initialized with the second point of the first line.



6. Click OK to create the second line.

To stop the repeat action, simply uncheck the option or click Cancel in the Line dialog box.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a line in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Creating Planes

This task shows the various methods for creating planes:

- offset from a plane
- parallel through point
- angle/normal to a plane
- through three points
- through two lines
- through a point and a line
- through a planar curve
- normal to a curve
- tangent to a surface
- from its equation
- mean through points

Open the Planes1.CATPart document.

1. Click the **Plane** icon .

The Plane Definition dialog box appears.

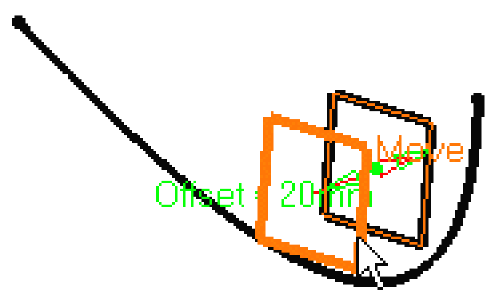
2. Use the combo to choose the desired **Plane type**.

Once you have defined the plane, it is represented by a red square symbol, which you can move using the graphic manipulator.

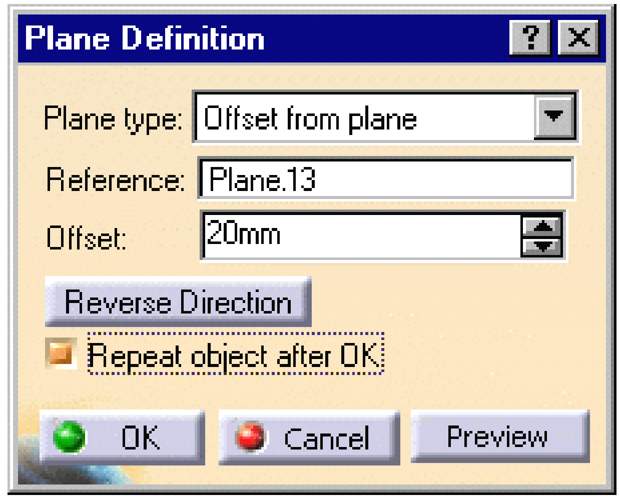
Offset from plane

- Select a reference **Plane** then enter an **Offset** value.

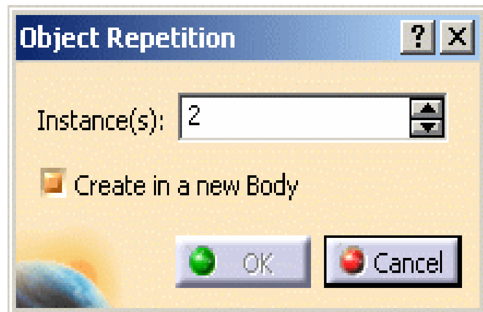
A plane is displayed offset from the reference plane.



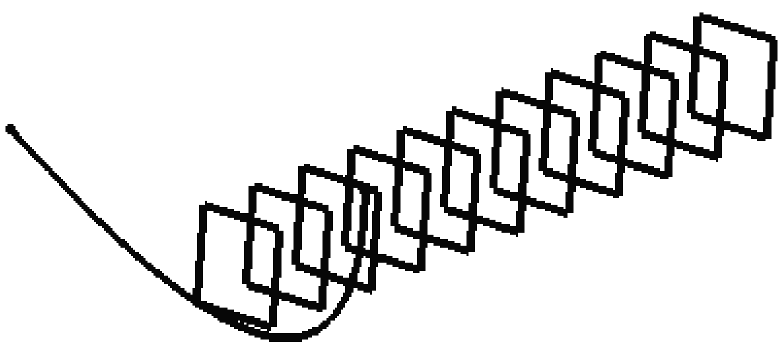
Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.



- Click the **Repeat object after OK** if you wish to create more offset planes .
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.

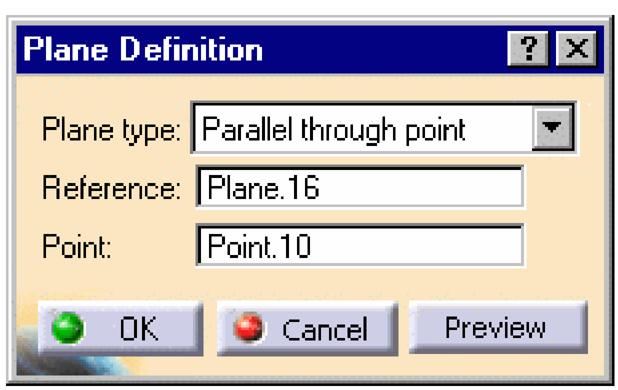


As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Offset** value.

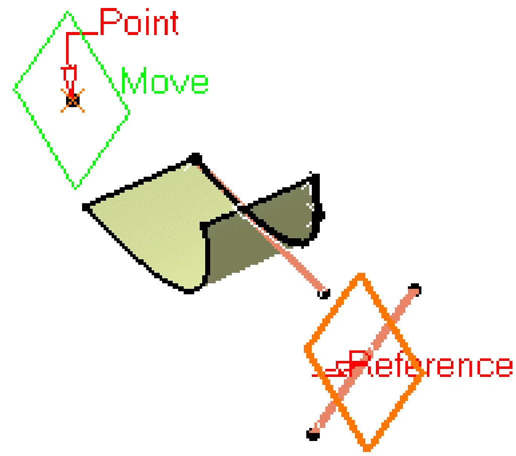


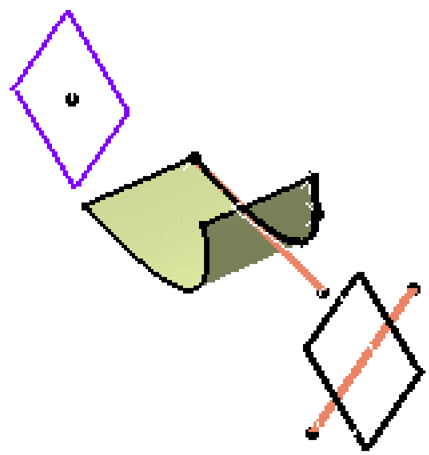
Parallel through point

- Select a reference **Plane** and a **Point**.



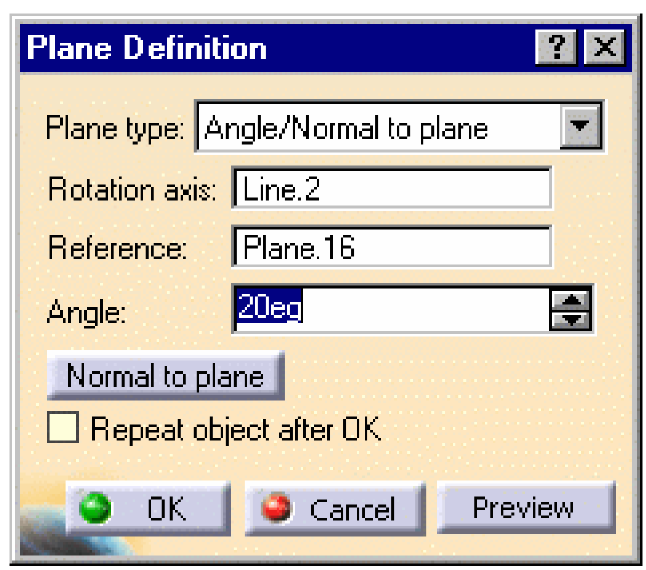
A plane is displayed parallel to the reference plane and passing through the selected point.



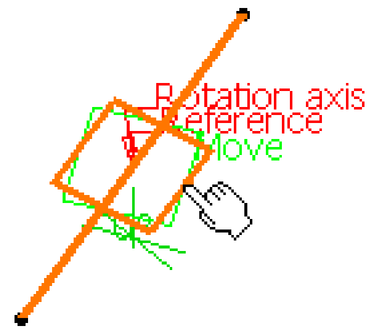


Angle or normal to plane

- Select a reference **Plane** and a **Rotation axis**.
  This axis can be any line or an implicit element, such as a cylinder axis for example. To select the latter press and hold the Shift key while moving the pointer over the element, then click it.
- Enter an **Angle** value.



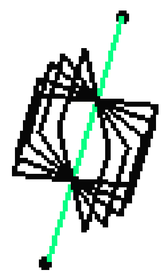
A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



- Click the **Repeat object after OK** if you wish to create more planes at an angle from the initial plane.
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.

As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Angle** value.

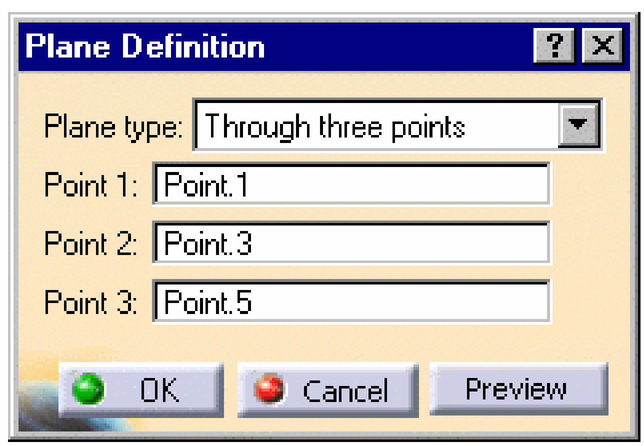
Here we created five planes at an angle of 20 degrees.



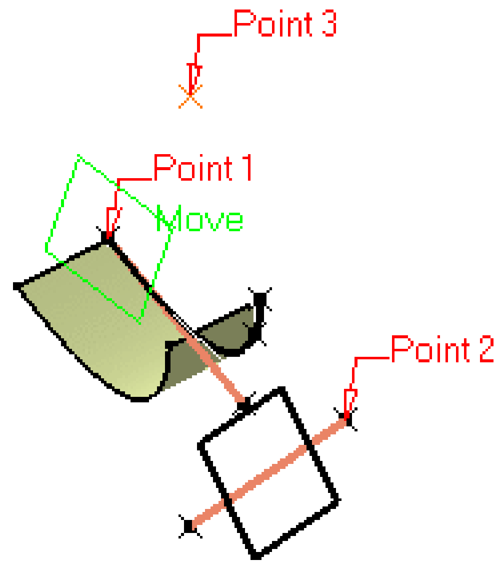
This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

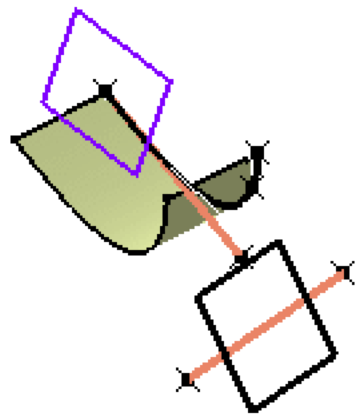
Through three points

- Select three points.



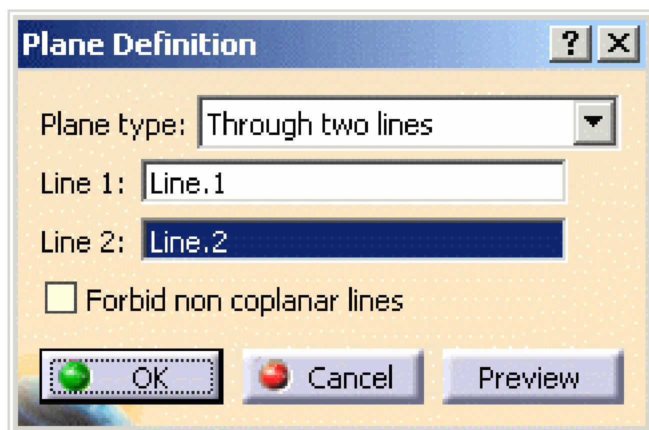
The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.



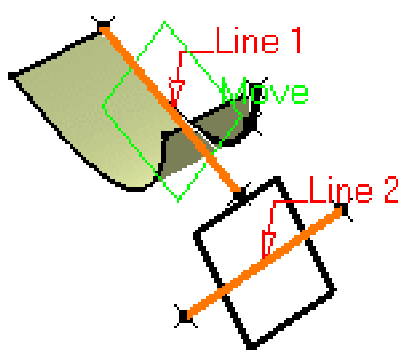


Through two lines

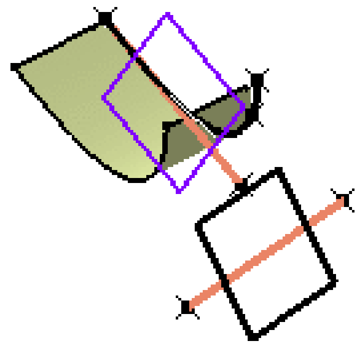
- Select two lines.



The plane passing through the two line directions is displayed.
When these two lines are not coplanar, the vector of the second line is moved to the first line location to define the plane's second direction.

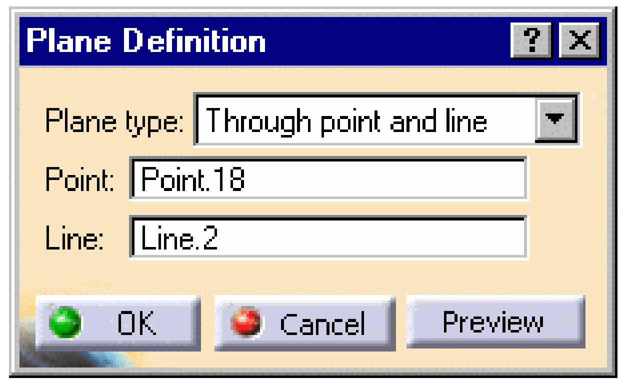


Check the **Forbid non coplanar lines button** to specify that both lines be in the same plane.

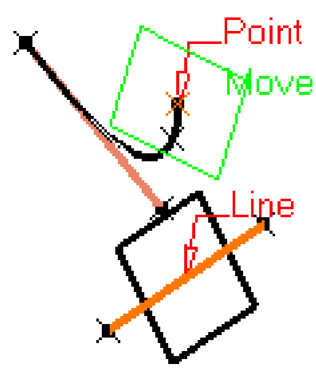


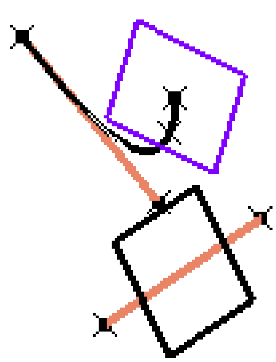
Through point and line

- Select a **Point** and a **Line**.

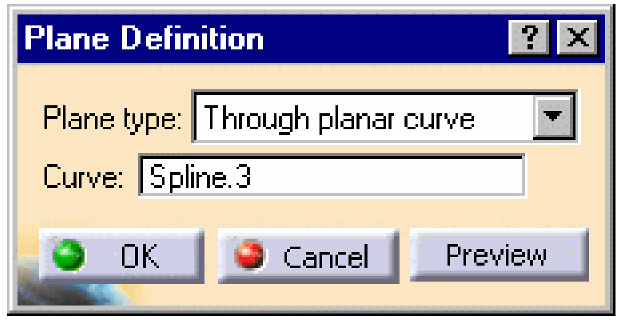


The plane passing through the point and the line is displayed.



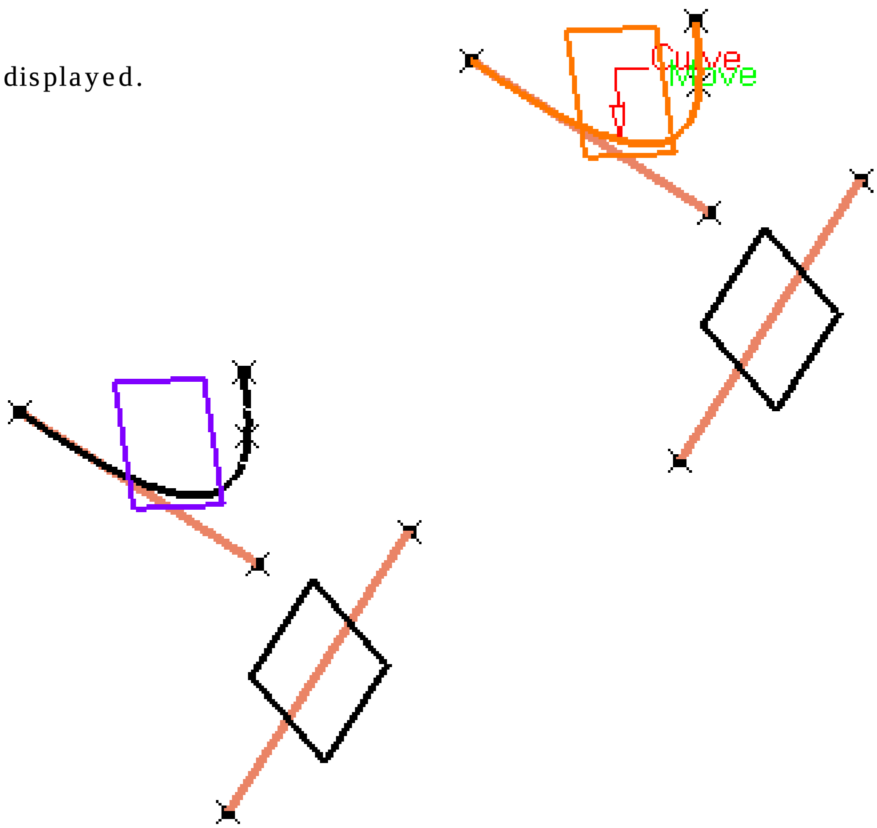


Through planar curve

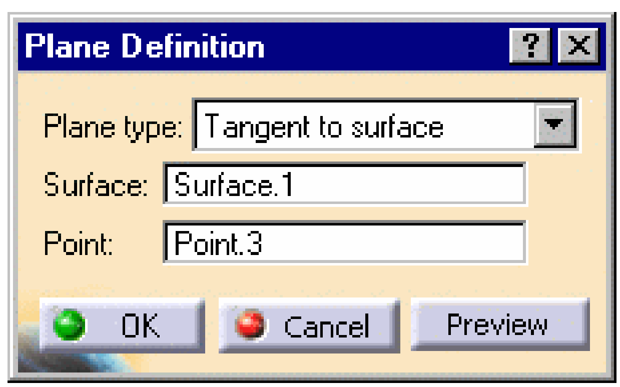


- Select a planar **Curve**.

The plane containing the curve is displayed.

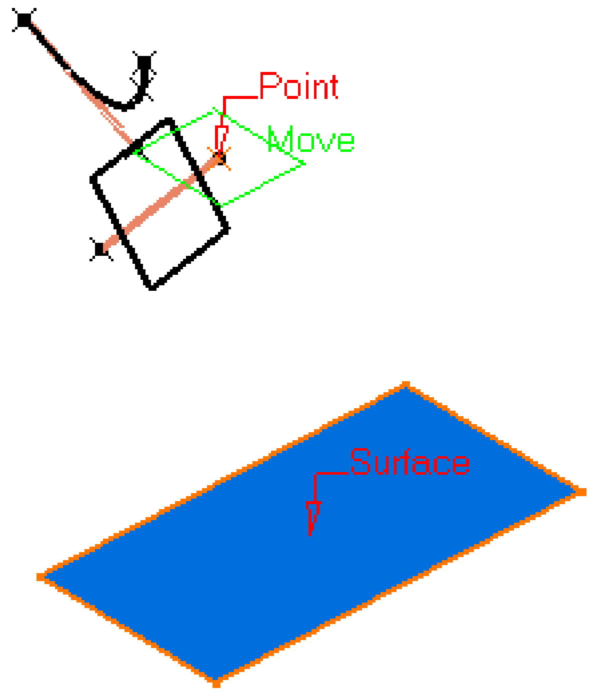


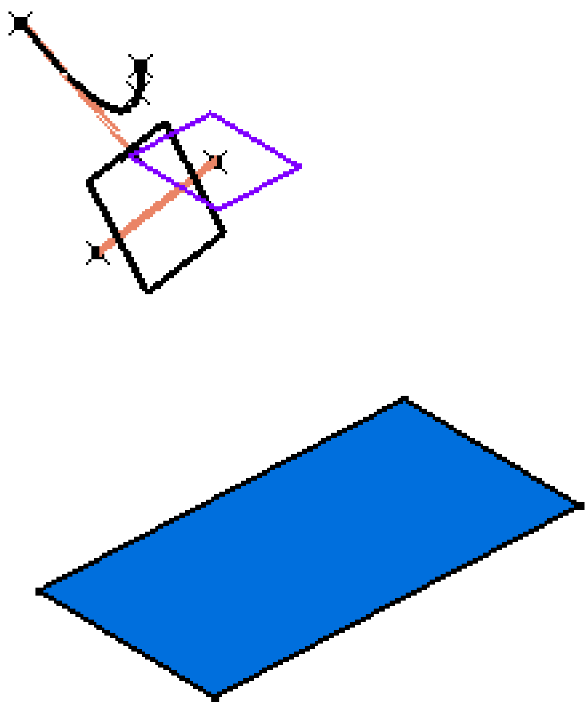
Tangent to surface



- Select a reference **Surface** and a **Point**.

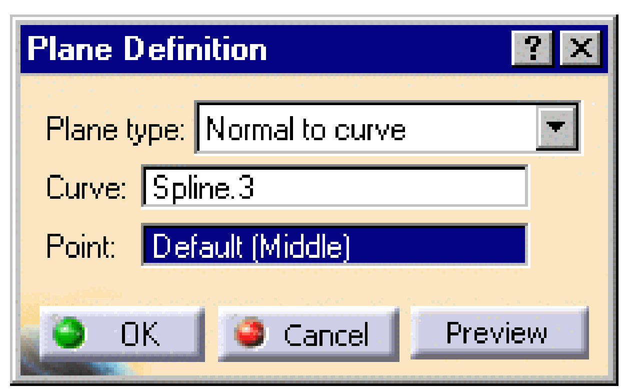
A plane is displayed tangent to the surface at the specified point.

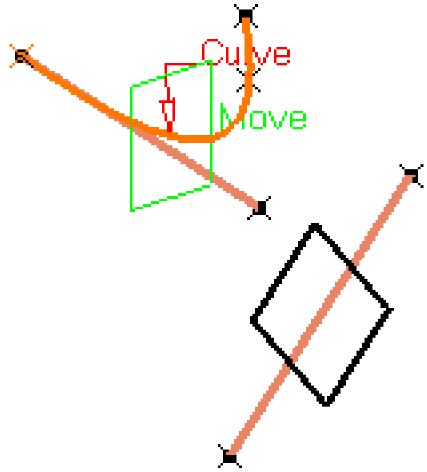




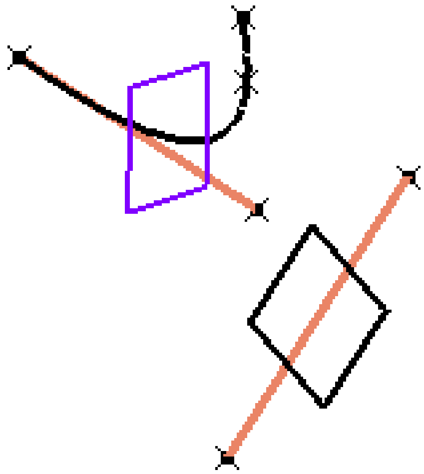
Normal to curve

- Select a reference **Curve.**
- You can select a **Point**. By default, the curve's middle point is selecte.



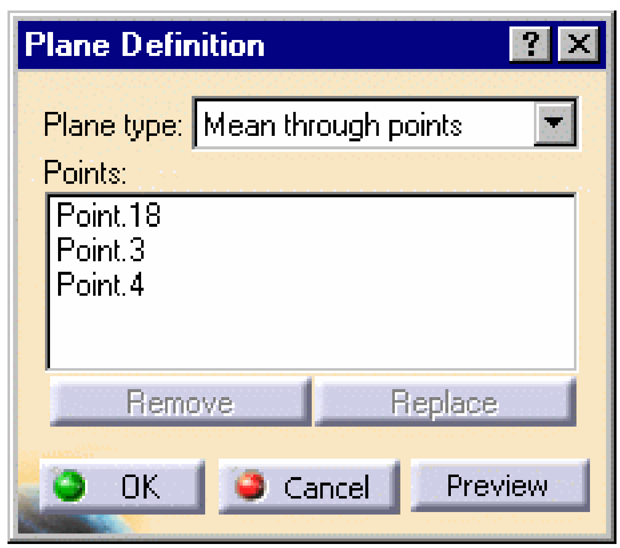


A plane is displayed normal to the curve at the specified point.



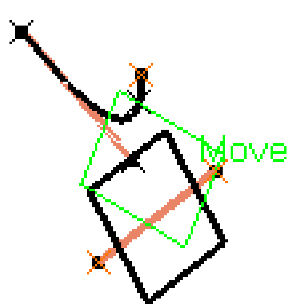
Mean through points

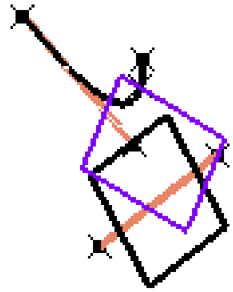
- Select three or more points to display the mean plane through these points.



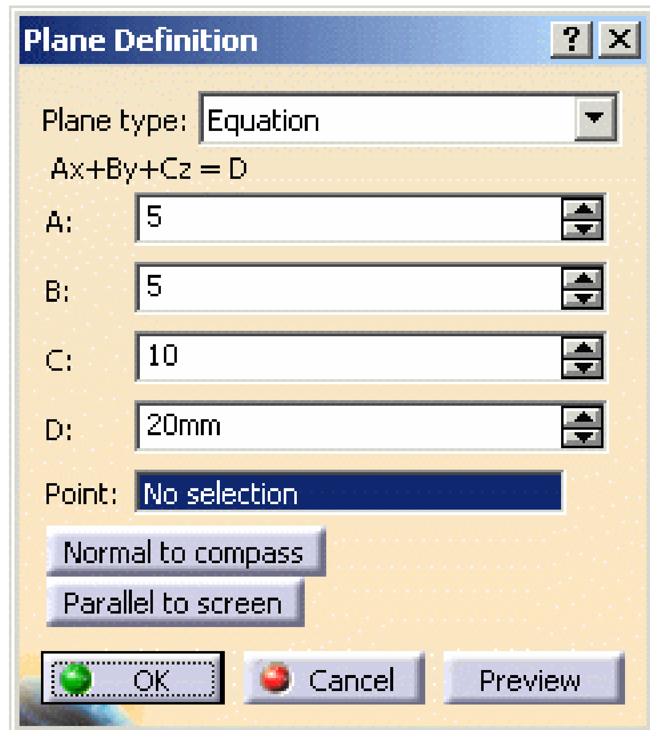
It is possible to edit the plane by first selecting a point in the dialog box list then choosing an option to either:

- **Remove** the selected point
- **Replace** the selected point by another point.



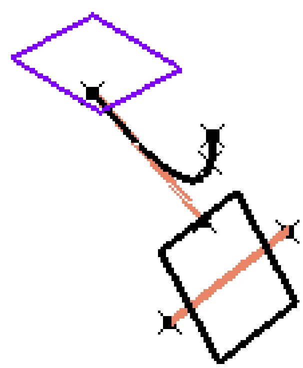


Equation



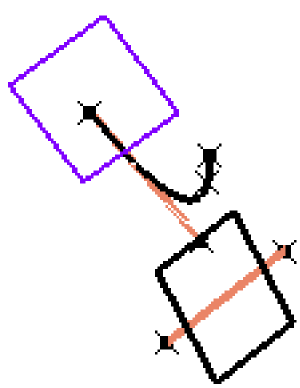
- Enter the **A**, **B**, **C**, **D** components of the **Ax + By + Cz = D** plane equation.

Select a point to position the plane through this point, you are able to modify **A**, **B**, and **C** components, the **D** component becomes grayed.



Use the **Normal to compass** button to position the plane perpendicular to the compass direction.

Use the **Parallel to screen** button to parallel to the screen current view.



3. Click **OK** to create the plane.

The plane (identified as Plane.xxx) is added to the specification tree.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a plane in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Advanced Tasks

The Advanced Tasks section explains how to use further functions that may not be as common as the ones described in the Basic Tasks section, as well as the integration of the SheetMetal Design workbench and elements with other workbenches.

Integration With Part Design

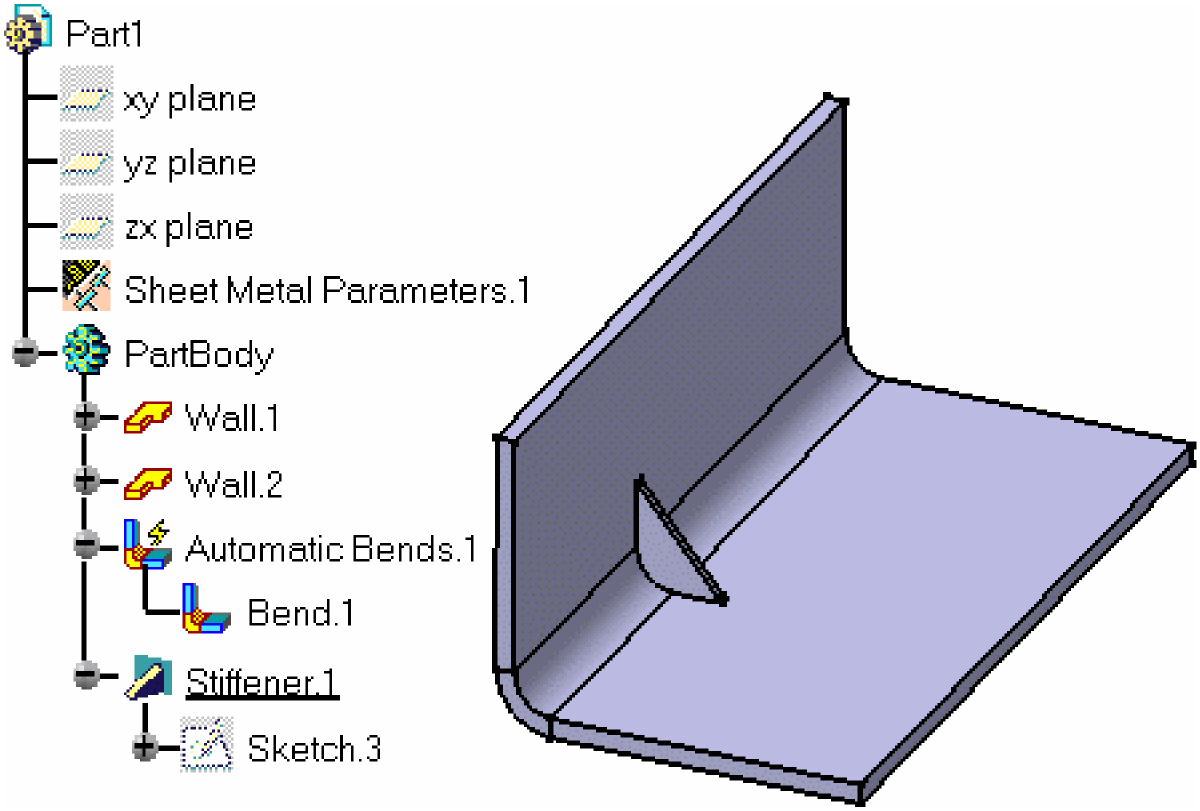
For the Sheet Metal Design workbench, open the Stiffener1.CATPart document.
For the Generative Sheetmetal Design workbench, open the NEWStiffener1.CATPart document.

This scenario, which is valid for both the Sheet Metal Design workbench and the Generative Sheet Metal Design workbench, is illustrated using screen captures from the Sheet Metal Design workbench. Results will slightly differ in the Generative Sheetmetal Design workbench, for which Automatic bends are not available.

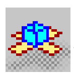
In a CATPart document, you may have Part Design features and Sheet Metal features according to the following rules:

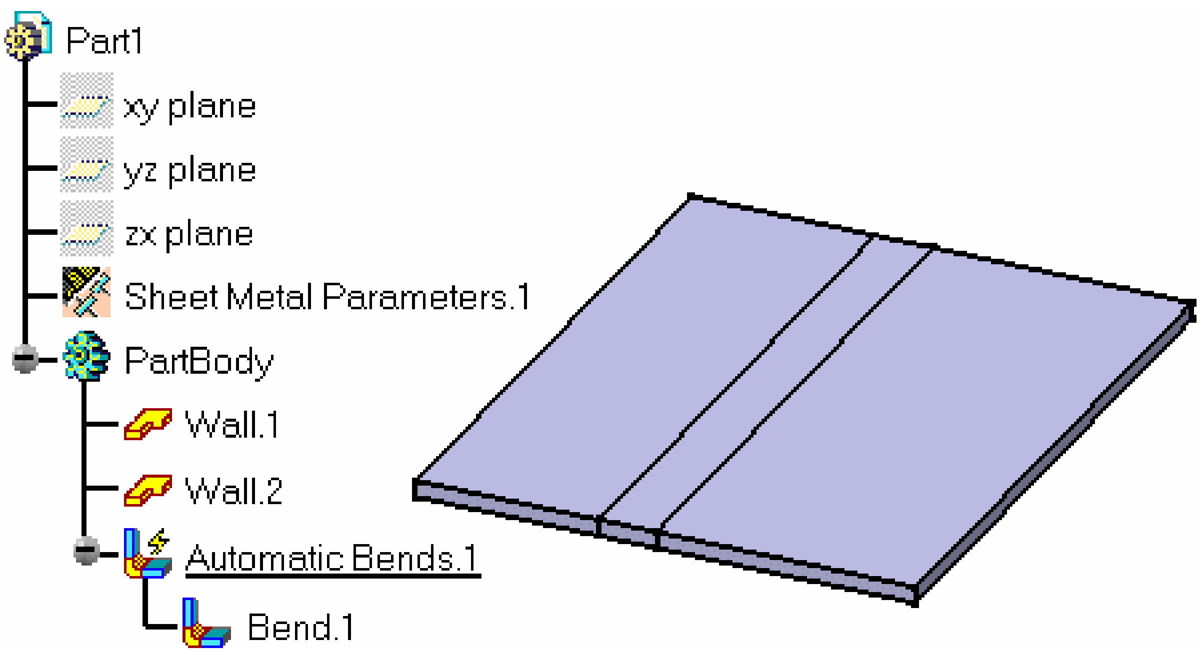
- Part Design features can be created before Sheet Metal features.
- a Part Design feature can also be created after Sheet Metal features as long as the part is in folded view.
- in the unfolded view, the Part Design feature will not be displayed.
- it is no longer possible to create Sheet Metal features after this last Part Design feature in folded view.

1. Create two walls with an Automatic or Conical Bend for the Sheet Metal Design workbench or two walls with a Cylindrical or Conical Bend for the Generative Sheetmetal Design workbench.
2. Switch to the Part Design workbench.
3. Launch the Sketcher and draw an oblique line in the yz plane.
4. Click the icon to create a **Stiffener**.



5. Switch to the Sheet Metal Design workbench or to the Generative Sheet Metal Design workbench.

6. Click the **Unfold** icon .



The stiffener is not displayed on the unfolded view.

To add a Generative Sheetmetal Design feature and switch to the Sheet Metal Design workbench or to the Generative Sheetmetal Design Design workbench, select the Bend for example and right-click the **Define In Work Object** item. The Generative Sheetmetal Design feature will be added after the Bend but before the Stiffener.

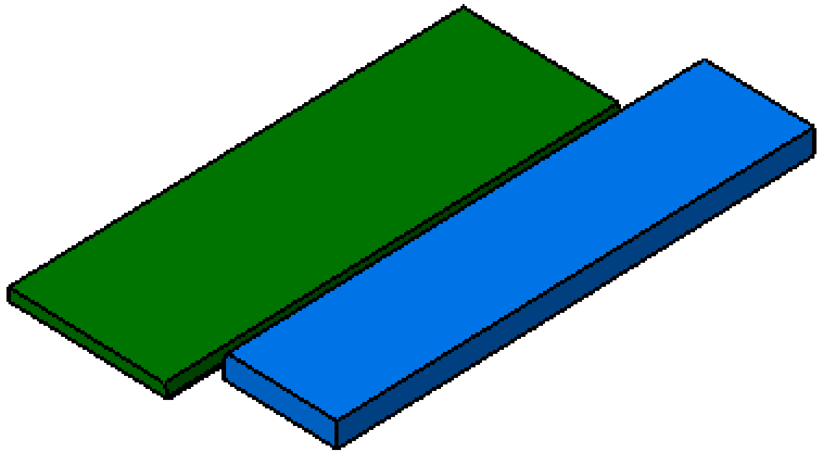
Integration With Weld Design

You can open the Integration.CATProduct document from the samples directory to replay the scenario.
If you use the Generative Sheetmetal Design workbench, open the NEWIntegration.CATProduct document.

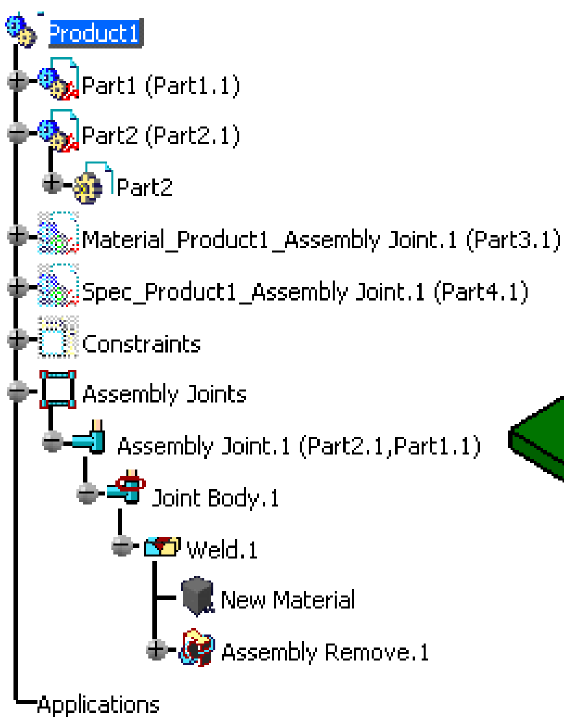
In a CATProduct document, you may have Weld Design features and Sheet Metal features according to the following rules:

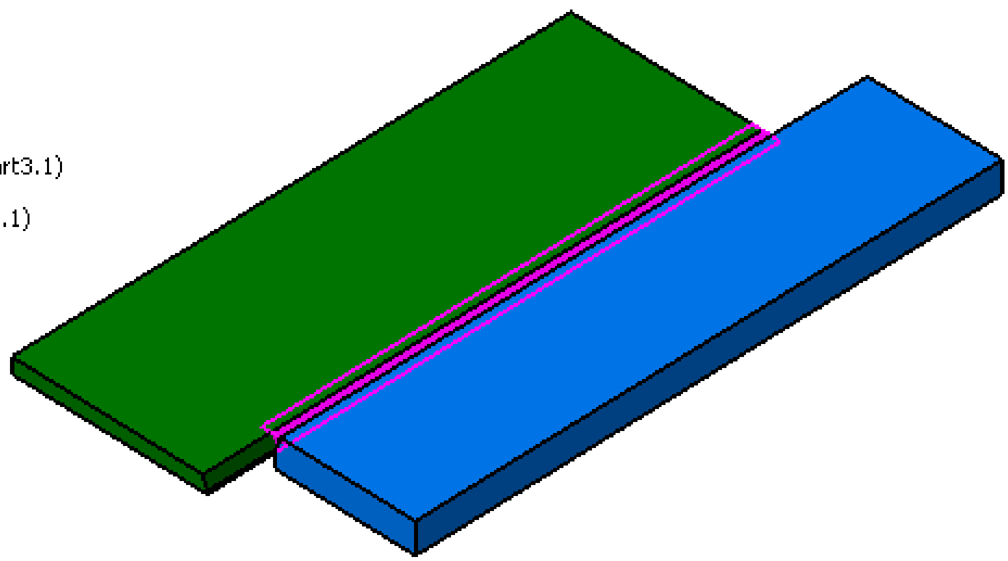
- Weld Design features are created after Sheet Metal features.
- in the unfolded view, the Weld Design feature will not be displayed.
- it is no longer possible to create Sheet Metal features after this last Weld Design feature.

1. Create two walls.
2. Switch to Weld Design workbench.



3. Create a weld (in our scenario, we create a Single-V Butt weld)





To add a Generative Sheetmetal Design feature and switch to the Sheet Metal Design workbench, select any of the two walls and make it current.

The Generative Sheetmetal Design feature will be added after the Bend but before the Stiffener.

Integration with Generative Drafting

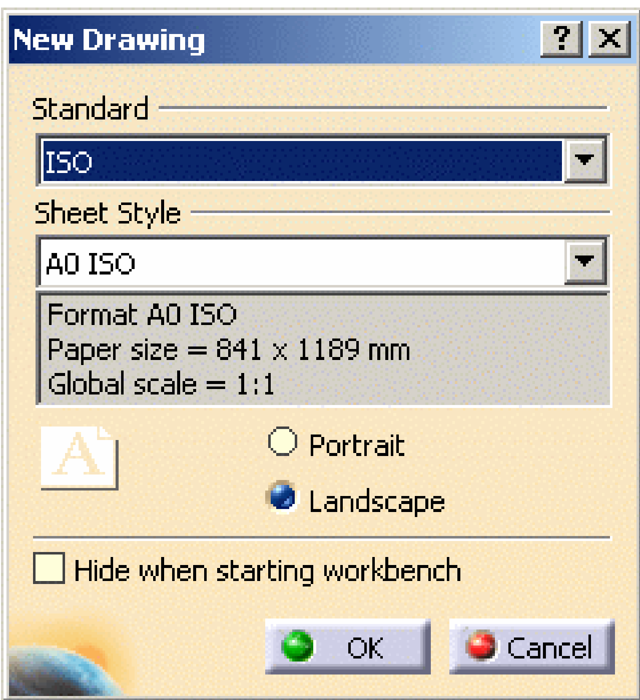
This task explains how to generate an unfolded view of a part with swept wall in a drawing sheet. The axes of planar hems, tear drops, and flanges are displayed within the drawing.

For the SheetMetal Design workbench, open the SweptWall01.CATPart document.
For the Generative Sheetmetal Design workbench, open the NEWSweptWall1.CATPart document.

Make sure you have an appropriate Generative Drafting license.

Make sure that the **Generate axis** option is checked in the **Tools -> Options -> Mechanical Design -> Drafting -> View** tab, prior to generating a view in a .CATDrawing document.

1. Click or select **File -> New...**
2. Select the Drawing type and click **OK.**

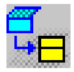


3. Click **OK**.

For more information about this workbench, refer to *Generative Drafting User's Guide*.

4. The drawing sheet appears.

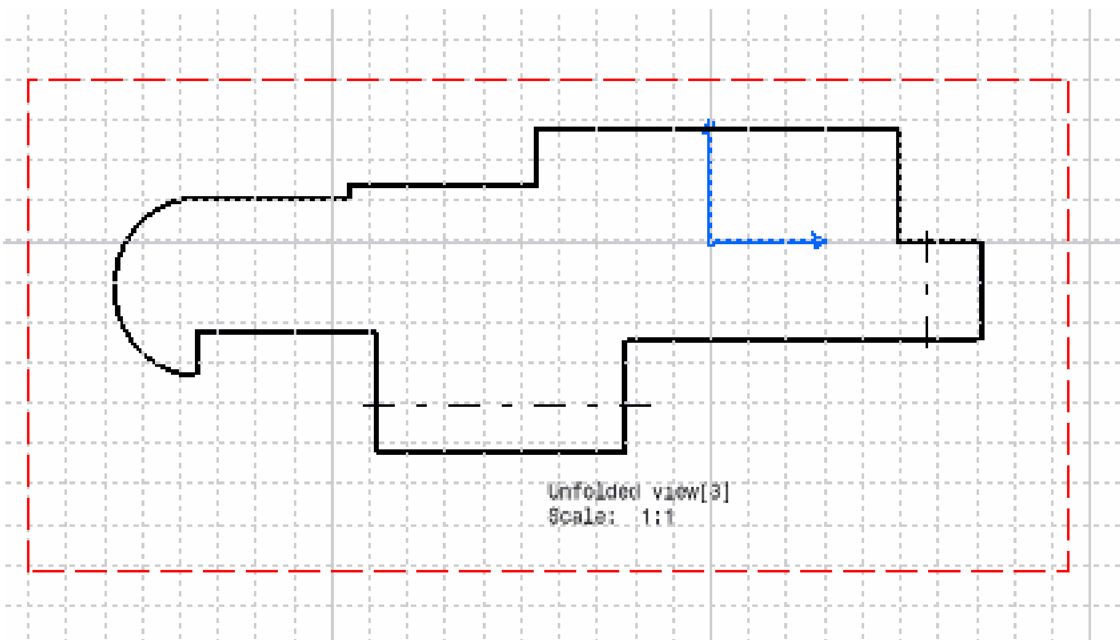
5. Tile the windows horizontally using the **Window -> Tile Horizontally** menu item.

6. Select the **Unfolded View** icon  in the Projections toolbar from the Generative Drafting workbench.

This icon is active in the Projections toolbar provided the Generative Sheetmetal Design workbench is present.

7. Choose the **xy plane** in the Sheet Metal specification tree. A preview of the view is displayed in the drawing.

8. Click in the drawing to validate the view creation. The unfolded view is displayed with the planar swept wall axes.



Designing in Context

This section explains and illustrates how to create and use various kinds of features.
The table below lists the information you will find.

Design in context: from a CATProduct document, create a new CATPart document, and create a complete SheetMetal Part.

Modify the design: modify another part from another document and update the SheetMetal Part.

Designing in Context

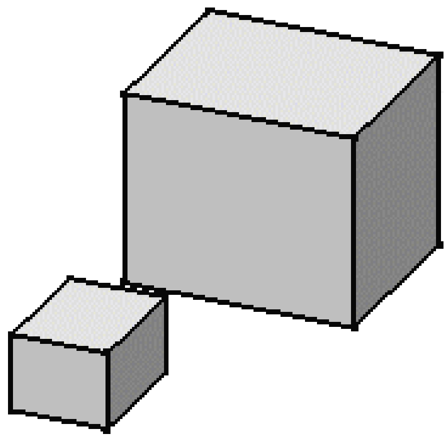
This task explains how to create a Sheet Metal part in an Assembly context.

For the Sheet Metal Design workbench, open the Scenario2.CATProduct document.
For the Generative Sheetmetal Design workbench, open the NEWScenario2.CATProduct document.

This scenario, which is valid for both the Sheet Metal Design workbench and the Generative Sheet Metal Design workbench, is illustrated using screen captures from the Sheet Metal Design workbench. Results will slightly differ in the Generative Sheetmetal Design workbench, for which Automatic bends are not available.

You are in Assembly Design workbench.

The document contains two parts.



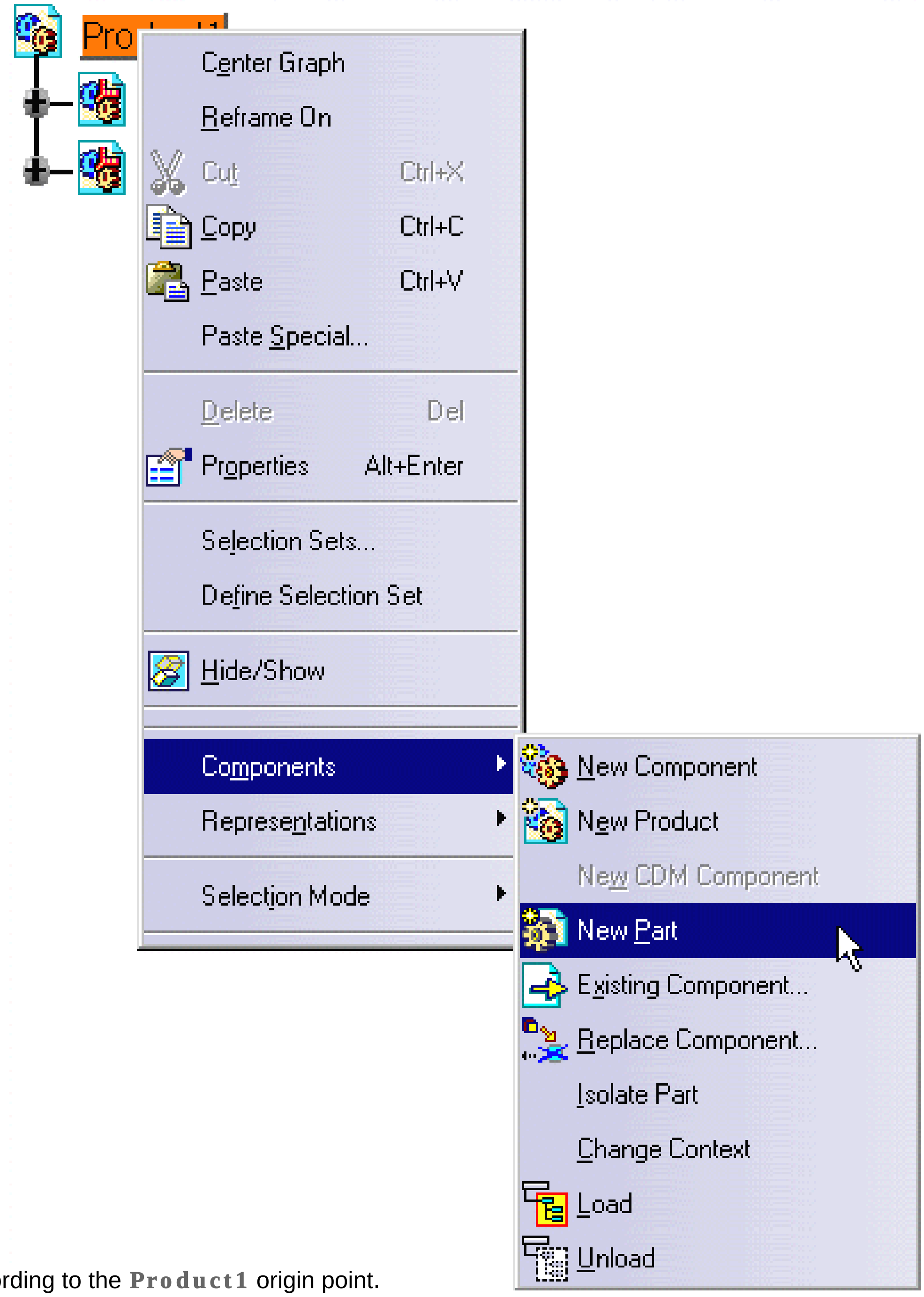
1. Right-click **Product1** in the specification tree and select **Components -> New Part...**

Provided the **Manual Input** option is checked in **Tools -> Options -> Infrastructure -> Product Structure**, **Product Structure** tab, the Part Number dialog box is displayed:

2. Enter **Part3** in the New part Number field and click **OK.**

   A New Part dialog box proposes two locations to define the origin point.

   For more information, refer to Inserting a New Part, in the *Product Structure User's Guide*.



3. Click **No** to locate the part origin according to the **Product1** origin point.

Make sure you are in **Design Mode:**

- Select **Product1**
- Choose **Edit -> Representations ->Design Mode**

4. Expand the tree and activate the **Part3** Part body by double-clicking.

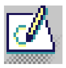
5. Switch to the Sheet Metal Design workbench or to the Generative Sheetmetal Design workbench.

6. Select the **Sheet Metal parameters** icon  to create the Sheet Metal characteristics for the part:

- 1mm for the Thickness
- 3mm for the Bend radius
- Linear for the Bend extremities

and click OK.

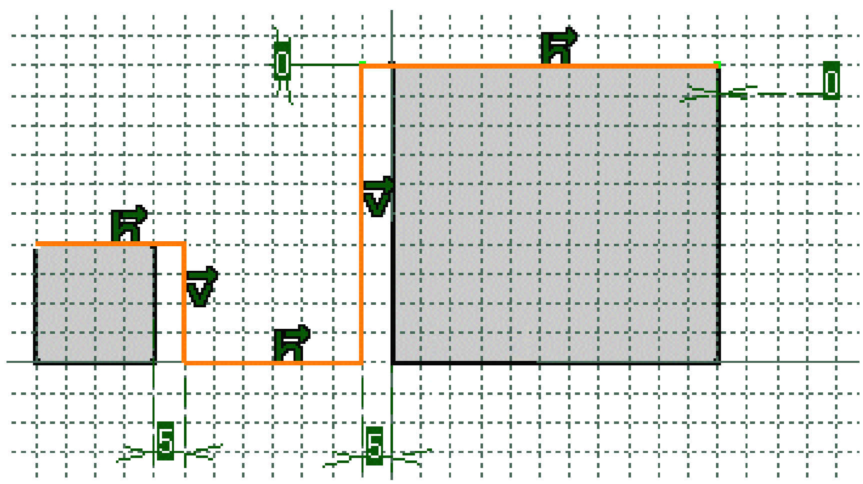
7. Choose the **Tools -> Options -> Infrastructure -> Part Infrastructure, General** tab and check the **Keep link with selected object** option, then click OK.

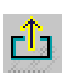
8. Click the **Sketcher** icon  and select the **zx plane**.

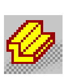
9. Select the **Profile** icon .

10. Sketch the profile and set the constraints as shown below:

- 5mm between the Sheet Metal vertical walls and each pad
- 0mm between the Sheet Metal horizontal walls and each pad top
- 0mm between the last point of the Sheet Metal sketch and the right pad side.

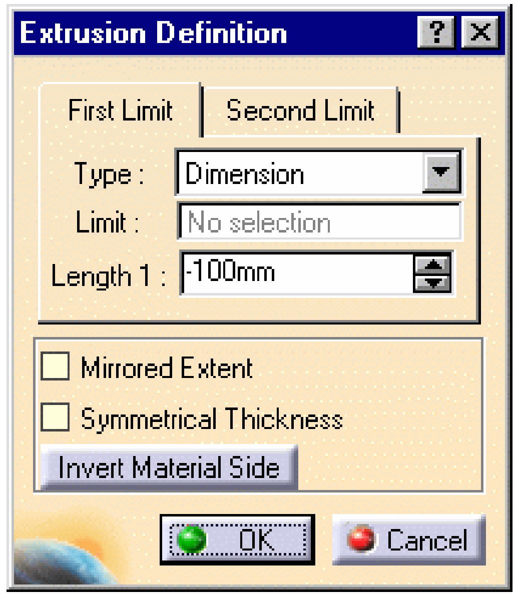


11. Click the **Exit** icon  to return to the 3D world.

12. Select the **Extrusion** icon 

13. Select the Sheet Metal profile.

The Extrusion Definition dialog box appears.



14. Enter 70mm for **Length1** then click **OK**.

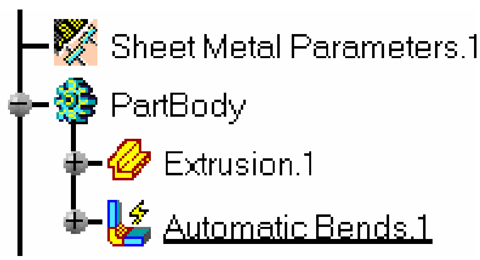
The Material Side should be set to the outside.

15. Perform this step only if you are using the Sheet Metal Design workbench: select the **Automatic Bends** icon .

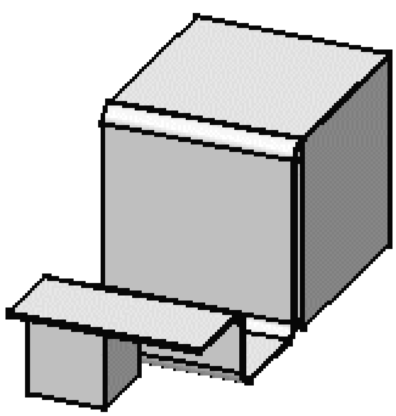
The bends are created.

The new features are shown in the specification tree:

- **Extrusion.1** with five walls
- **Automatic Bends.1** with four bends (for the Sheet Metal Design workbench only).



The Sheet Metal part looks like this:



Modifying the Design

In this task, you are going to modify the height and the sketch of Pad.1.

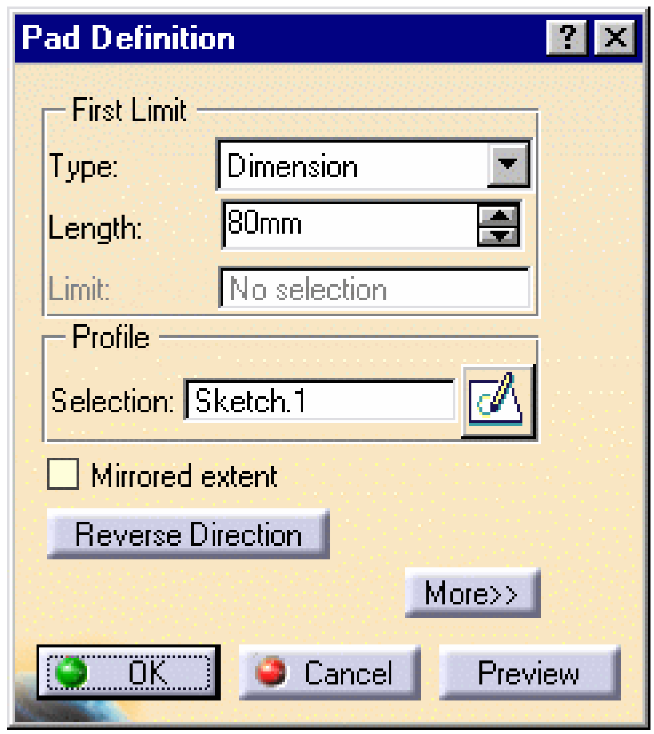
For the Sheet Metal Design workbench, the Scenario2.CATProduct document is open from the previous task. If not, open the Scenario2_2.CATProduct document from the samples directory.

For the Generative Sheetmetal Design workbench, open the NEWScenario2.CATProduct document.

This scenario, which is valid for both the Sheet Metal Design workbench and the Generative Sheet Metal Design workbench, is illustrated using screen captures from the Sheet Metal Design workbench. Results will slightly differ in the Generative Sheetmetal Design workbench, for which Automatic bends are not available.

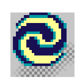
1. Double-click **Part1\PartBody\Pad.1** in the specification tree.

   The dialog box is displayed.



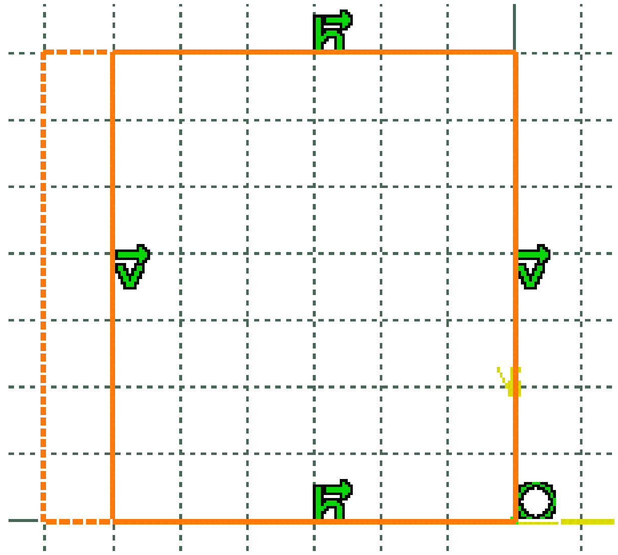
2. Enter 40mm for the **Length** and click **OK**.

   The pad is updated.

3. Double-click Part3 and **Update** the Sheet Metal part using the **Update**  icon.

4. Double-click **Part1\Pad.1\Sketch.1**.

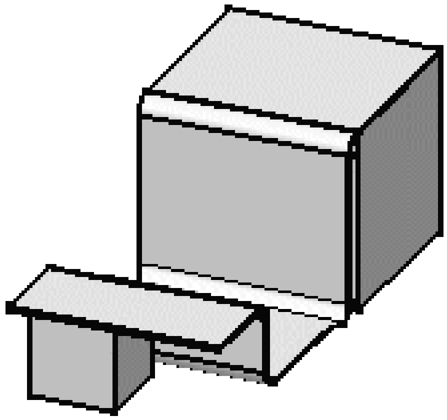
5. Modify the sketch:



6. Click the **Exit** icon  to return to the 3D world.

The constraints are respected.

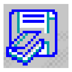
After the Part3 update, the document looks like this:



Managing PowerCopies

 Create PowerCopies: Select the **Insert ->Advanced Replication Tools -> PowerCopy Creation** command, select the elements making up the PowerCopy from the specification tree, define a name for the PowerCopy and its reference elements then choose an icon for identifying it.

 Instantiate PowerCopies: Select the **Insert -> Instantiate From Document** command, select the document or catalog containing the powercopy, complete the **Inputs** within the dialog box selecting adequate elements in the geometric area.

 Save PowerCopies into a Catalog: Select the PowerCopy from the specification tree, select the **Insert -> Advanced Replication Tools -> PowerCopy Save In Catalog...** command, enter the catalog name and click Open.

Creating PowerCopy Features

This task shows how to use create PowerCopy elements, to be reused later.
A PowerCopy is a set of features (geometric elements, formulas, constraints and so forth) that are grouped in order to be used in a different context, and presenting the ability to be re-specified according to the context when pasted.
This PowerCopy captures the design intent and know-how of the designer thus enabling greater reusability and efficiency.

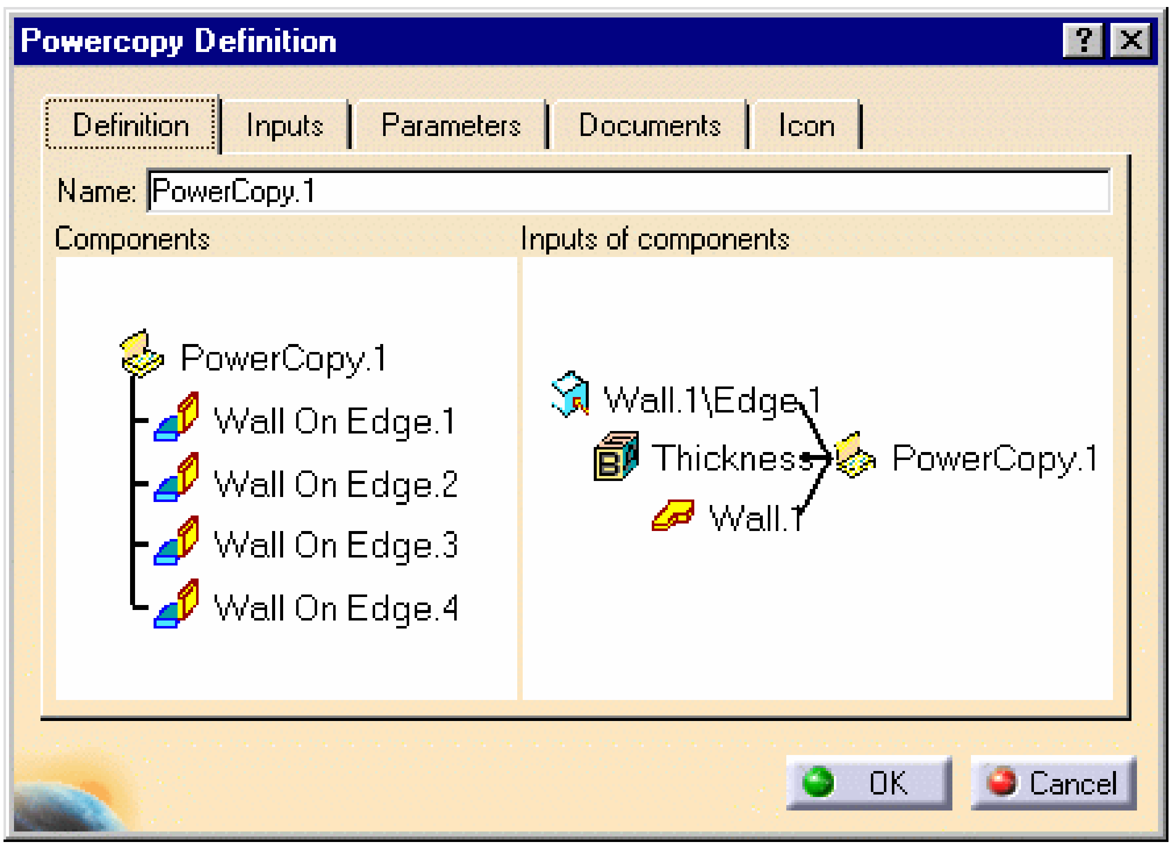
Open the PowerCopyStart.CATPart document.

1. 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.

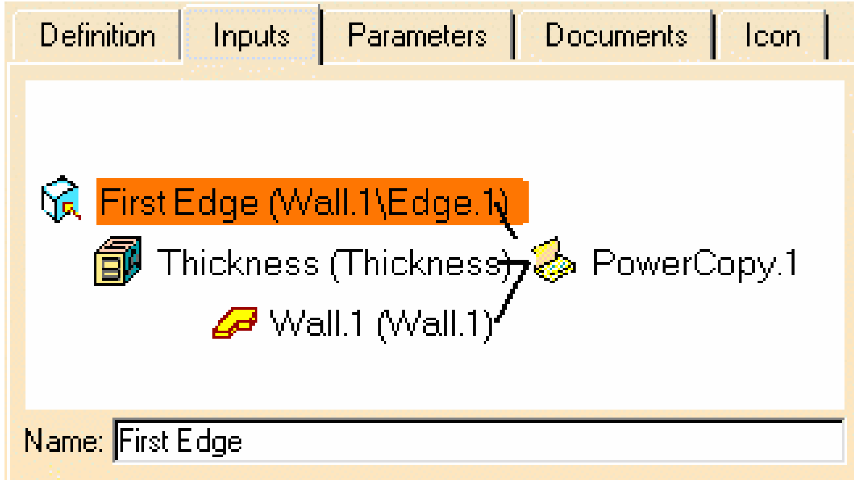
The **PowerCopy Definition** dialog box is automatically filled with information about the selected elements.



3. Define the PowerCopy as you wish to create it:

The **Definition** tab lets you assign a name to the PowerCopy and presents its components in the 3D viewer.

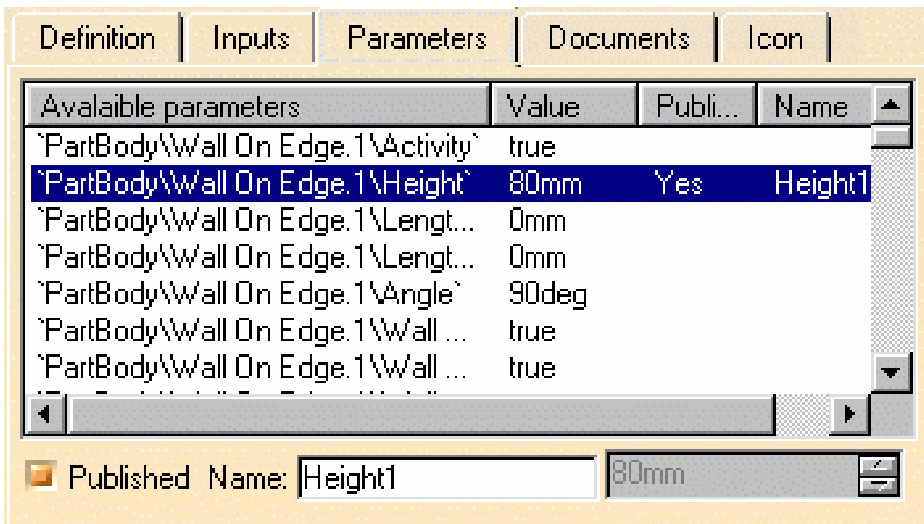
The **Inputs** tab lets you rename the reference elements making up the PowerCopy.



You can do that for clarification purposes as to their roles, by selecting the elements in the viewer and entering a new name in the **Name** field.

In this example, we renamed all three elements and in brackets you still can read the elements' default name based on their type.

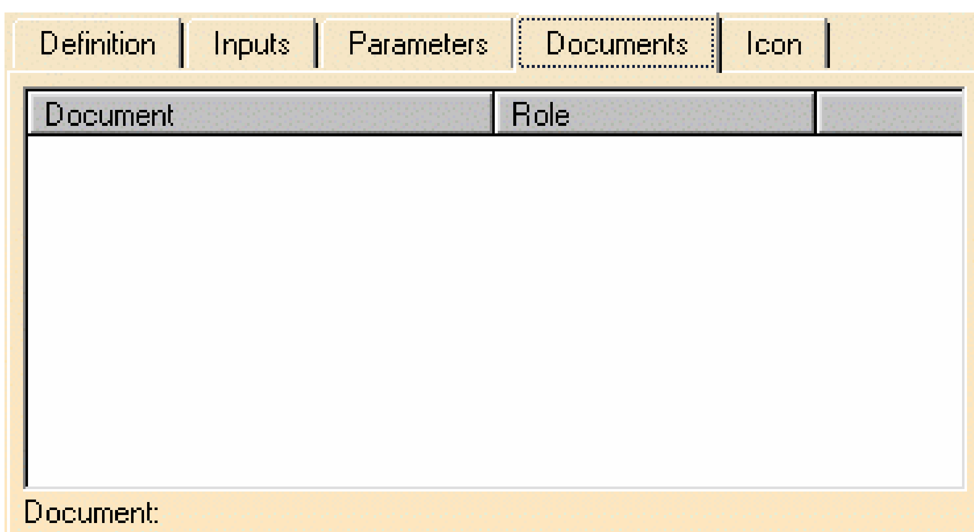
The **Parameters** tab lets you define which of the parameter values used in the PowerCopy you will be able to modify at instantiation time.



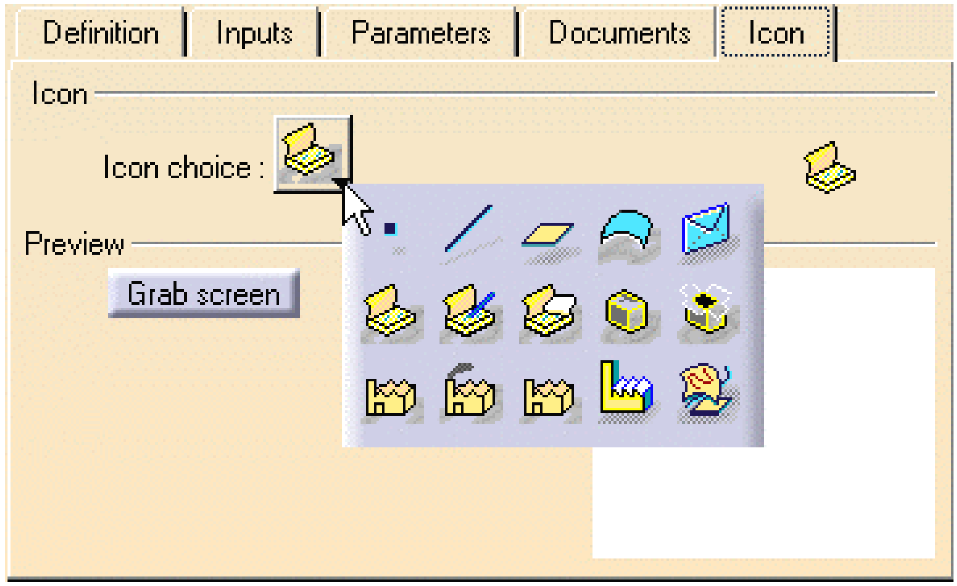
Simply check the Published button.

Use the **Name** field to give a more explicit name to the element.

The Documents tab shows the complete path and role of Design tables that are referenced by an element included in the Power Copy.

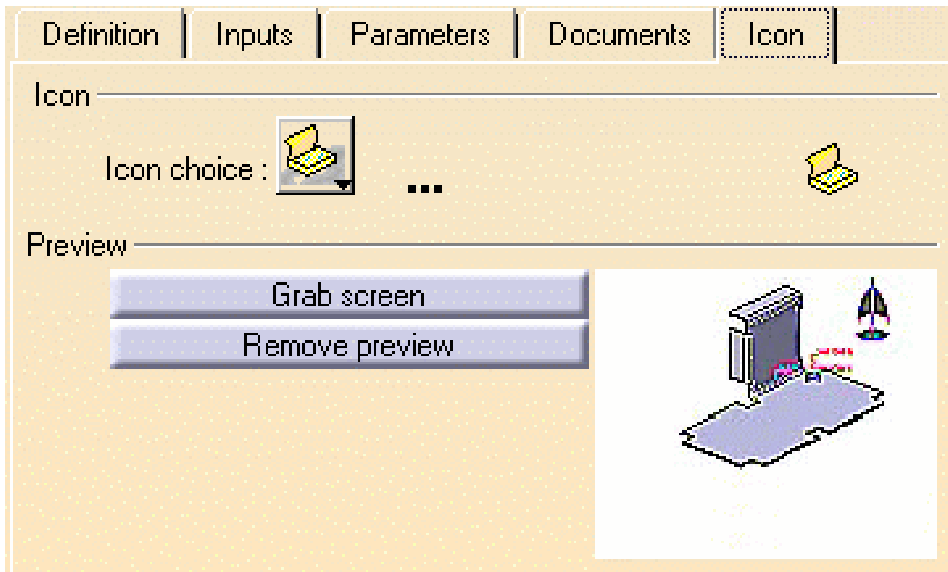


The **Icon** tab lets you modify the icon identifying the PowerCopy in the specifications tree.



A subset of icons is available from the Icon choice button.
If you click ... the Icon Browser opens, giving you access to all the graphic icons installed with the CATIA software.

Use the **Grab screen** button to capture an image of the PowerCopy to be stored with its definition in the catalog (see Saving PowerCopy Features).



Use the **Remove preview** button to delete the image captured with the **Grab screen** button.

4. Click OK to create the PowerCopy.

The PowerCopy is displayed close to the top of the specification tree.

- Double-click the PowerCopy in the specification tree to display the **PowerCopy Definition** dialog box and edit its contents.
- A formula is automatically included in a Power Copy definition when all its parameters are included. Otherwise, i.e. if at least one parameter is not selected as part of the Power Copy, you have to manually select the formula to make it part of the definition. If you do so, all the formula's parameters that have not been explicitly selected, are considered as inputs of the Power Copy.
- Once your PowerCopy is created, do not delete the referenced elements used to make up the PowerCopy.

Instantiating PowerCopy Features

This task shows how to instantiate PowerCopies once they have been created as described in Creating PowerCopy Features.
There are three ways to do this:

1. using the contextual menu
2. using the **Insert -> Instantiate From Document** menu item
3. using a catalog

The PowerCopyStart.CATPart document is still open from the previous task, otherwise, Open the PowerCopyStartResults1.CATPart document.

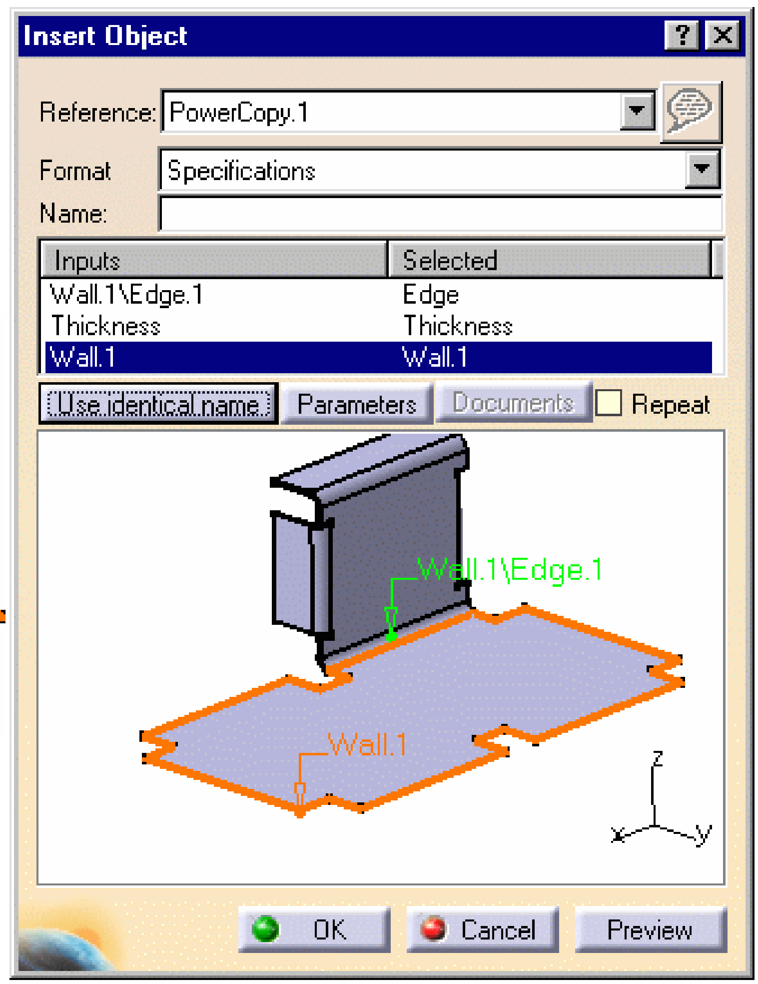
Using the Contextual Menu Item:

1. Select the PowerCopy feature from the specification tree.

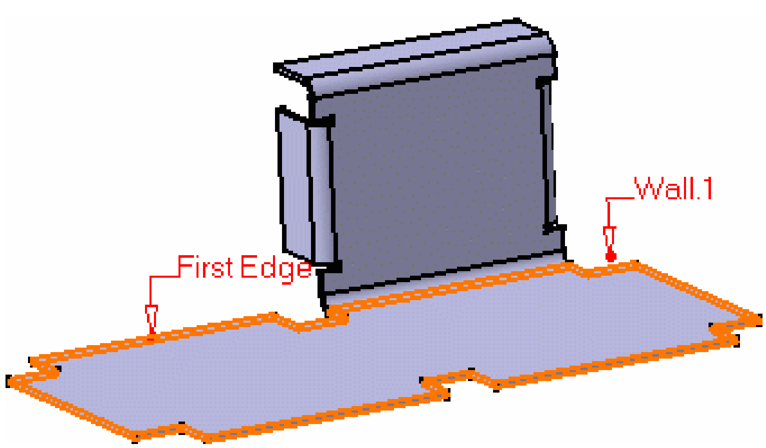
2. Right-click to display the contextual menu, and choose the **PowerCopy.1 object -> instantiate** menu item.

The **Insert Object** dialog box is displayed.

3. Complete the **Inputs** within the dialog box by selecting the adequate element in the geometric area.

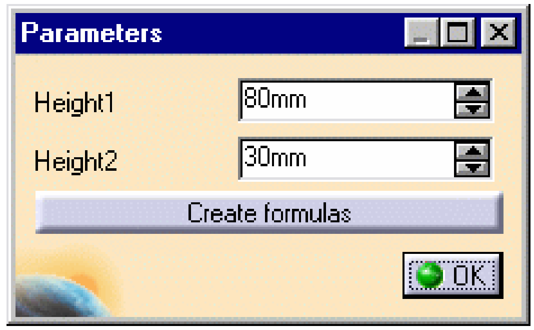


You need to click the arrow to invert the First Edge orientation.



Check the **Repeat** option to be able to repeat the instantiation.
In this case, once you have clicked OK in the Insert Object dialog box, the latter remains open, the PowerCopy's **Inputs** are listed and ready to be replaced by new inputs, as described above.
Modified parameters using **Parameters** button are retained as well for the next instantiation.
To exit the command, you then need to uncheck the **Repeat** button or click Cancel.

5. You can also click on the **Parameters** button to display the **Parameters** dialog box and modify values, if needed.



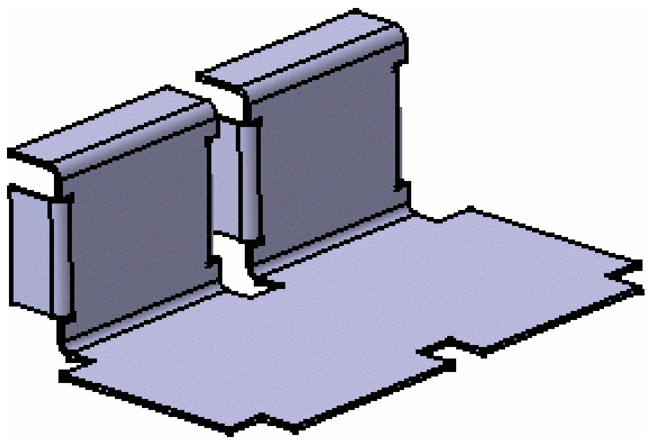
6. Use the Create formulas button to automatically create a formula on every parameters with the same name provided there are any.

7. Click OK in the **Parameters** dialog box.

The Documents button lets you access the list of documents (such as design tables) pointed by one of the elements making up the Power copy.
If there are documents, the Documents dialog box opens and you can click the Replace button to display the File Selection dialog box and navigate to a new design table to replace the initial one.
When no document is referenced, the Documents button is grayed within the Insert Object dialog box.

8. Click OK to create the PowerCopy instance.

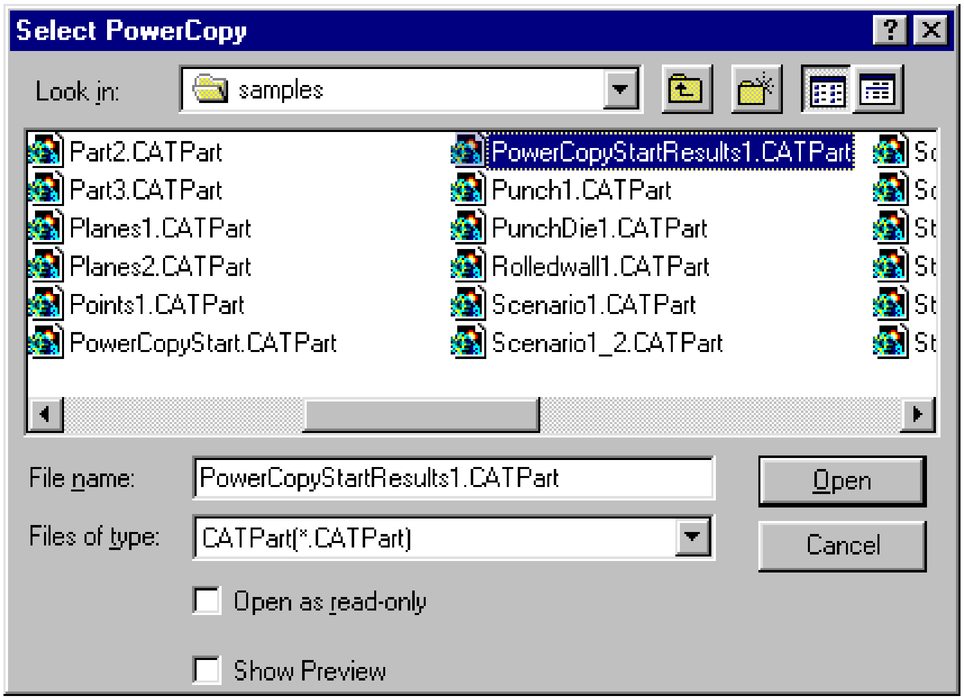
The PowerCopy is instantiated in context, meaning its limits are automatically re-defined taking into account the elements on which it is instantiated.



Using the Insert Menu:

1. Select the **Insert -> Instantiate From Document** menu item.

   The **Select PowerCopy** dialog box is displayed allowing you to navigate to the document or catalog where the power copy is stored.



2. Select the document containing the Powercopy.

The **Insert Object** dialog box is displayed.

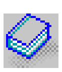
Use the **Reference** list to choose the correct PowerCopy when several have been defined in the document.

3. Continue the instantiation as described in Using the Contextual Menu item, step 3.

Using the catalog:

You need to have a catalog available, created either:

- using the Catalog capability, see the *Infrastructure User's Guide*.
- using the **Insert -> Advanced Replication Tools -> PowerCopy Save In Catalog...** menu item.

1. Click the  icon.
   If accessing a catalog for the first time, you need to navigate to the catalog location. This location is stored in the settings for faster access later on.
2. Select the catalog containing the PowerCopy you wish to instantiate.

3. Select the PowerCopy to be instantiated, then you can:

- drag and drop it onto the reference element
- double-click the PowerCopy
- or right-click on the PowerCopy in the dialog box and use the **Instantiate** contextual menu.

From then on, you instantiate the PowerCopy as described Using the Contextual Menu item, step 3.

- You can only instantiate a PowerCopy if the PowerCopy itself and the features making it up have been created in the current view mode: i.e. you will be able to instantiate a PowerCopy created in 3D mode, only on a feature in 3D mode, not on an unfolded feature.
- The icon is always grayed when instantiating Power Copies. It is available with User Features and allows you to create and modify URLs.

Saving PowerCopy Features

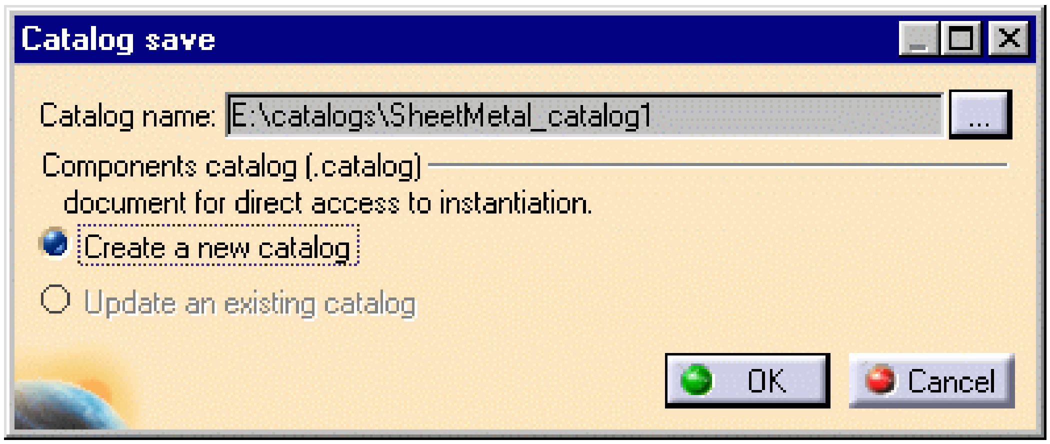
This task shows how to use store Power Copy elements into a catalog, for later use as described in Instantiating PowerCopy Features.

Open the PowerCopyStartResults1.CATPart document.

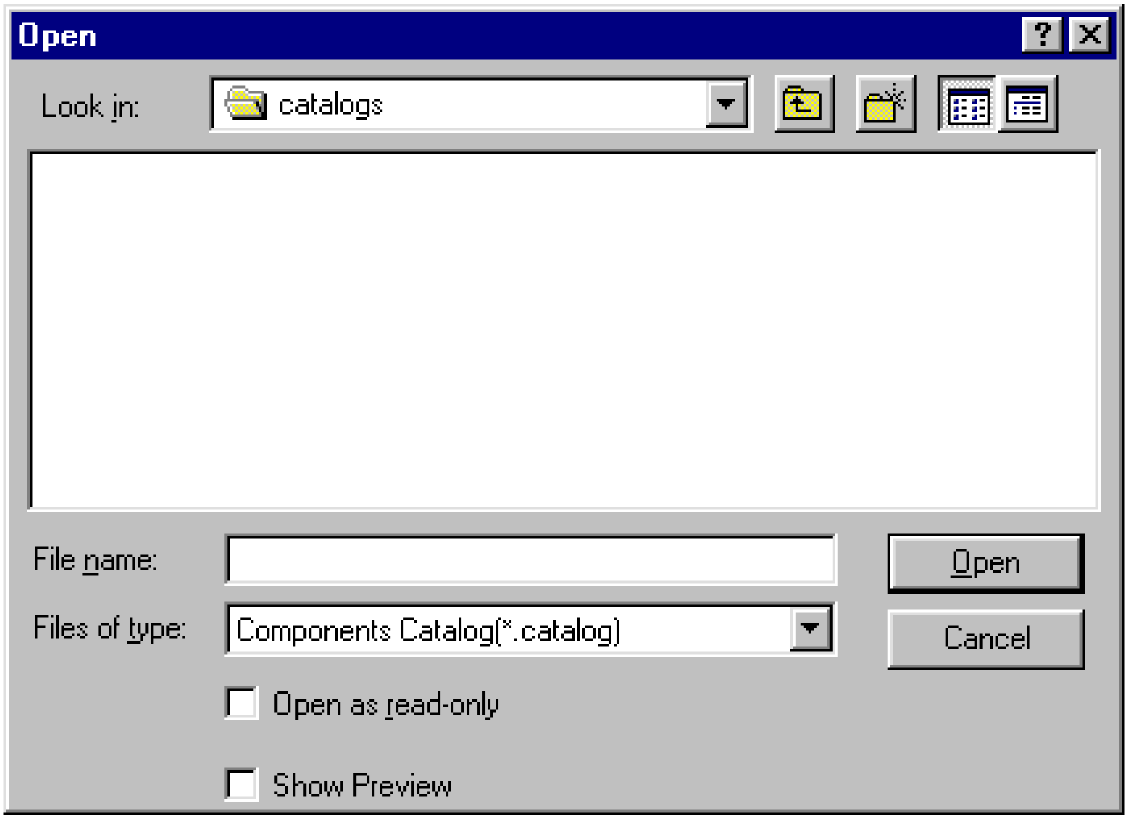
1. Select the PowerCopy from the specification tree for example.

2. Choose the **Insert -> Advanced Replication Tools -> (PowerCopy) Save In Catalog...** menu item.

   The **Catalog Save** dialog box is displayed:



- When creating a catalog for the first time, click the ... button to display the **Open** dialog box, and navigate to the location where you wish to create a catalog.
  Then simply key in the catalog name and click Open.



- If you wish to add a PowerCopy to an existing catalog, simply activate the **Update an existing catalog** option in the **Catalog Save** dialog box

By default, the **Catalog Save** dialog box recalls the catalog accessed last.

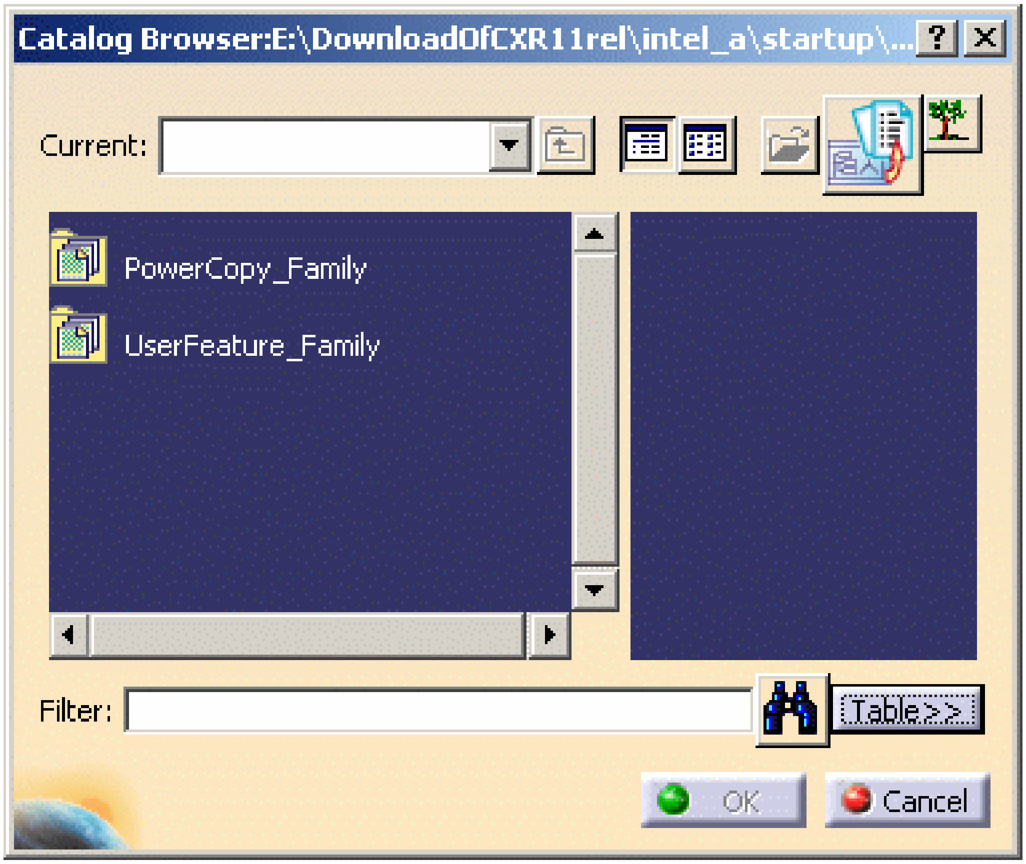
3. Click **OK**.

The PowerCopy has been stored in the catalog.

Browsing the Sheet Metal Catalog

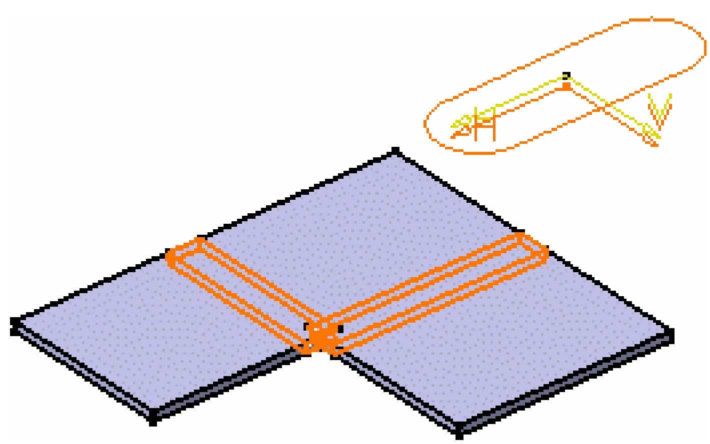
Open the CornerRelief03.CATPart document from the samples directory.

In this case, the Catalog Browser opens, and lets you choose from a number of sketches.

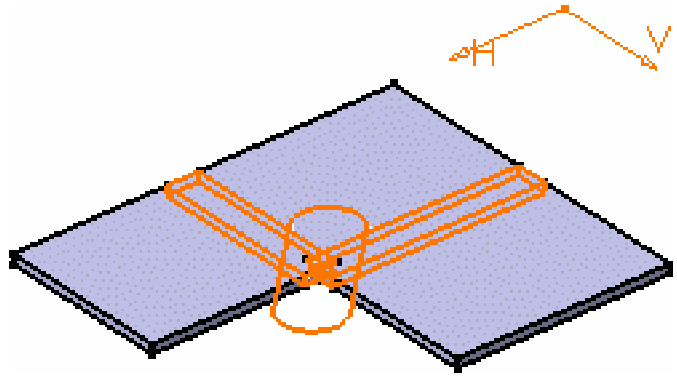


1. Select a sketch and click OK in the Catalog Browser.

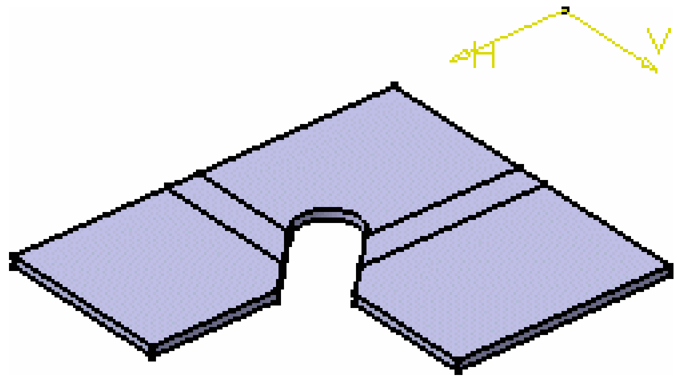
You return to the Corner Relief Definition dialog box, that has been updated with the Sketcher icon.

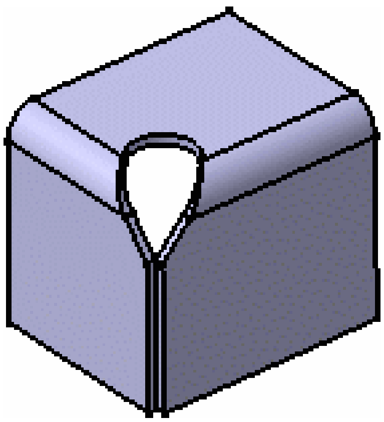


2. Click the sketcher icon, and move the sketch to position it properly in relation to the part.



3. Click OK, and check the part in a folded view.





Looking For Sheet Metal Features

This task shows how to use the Search capabilities on Sheet Metal Features, in order to detect any specific kind of feature.

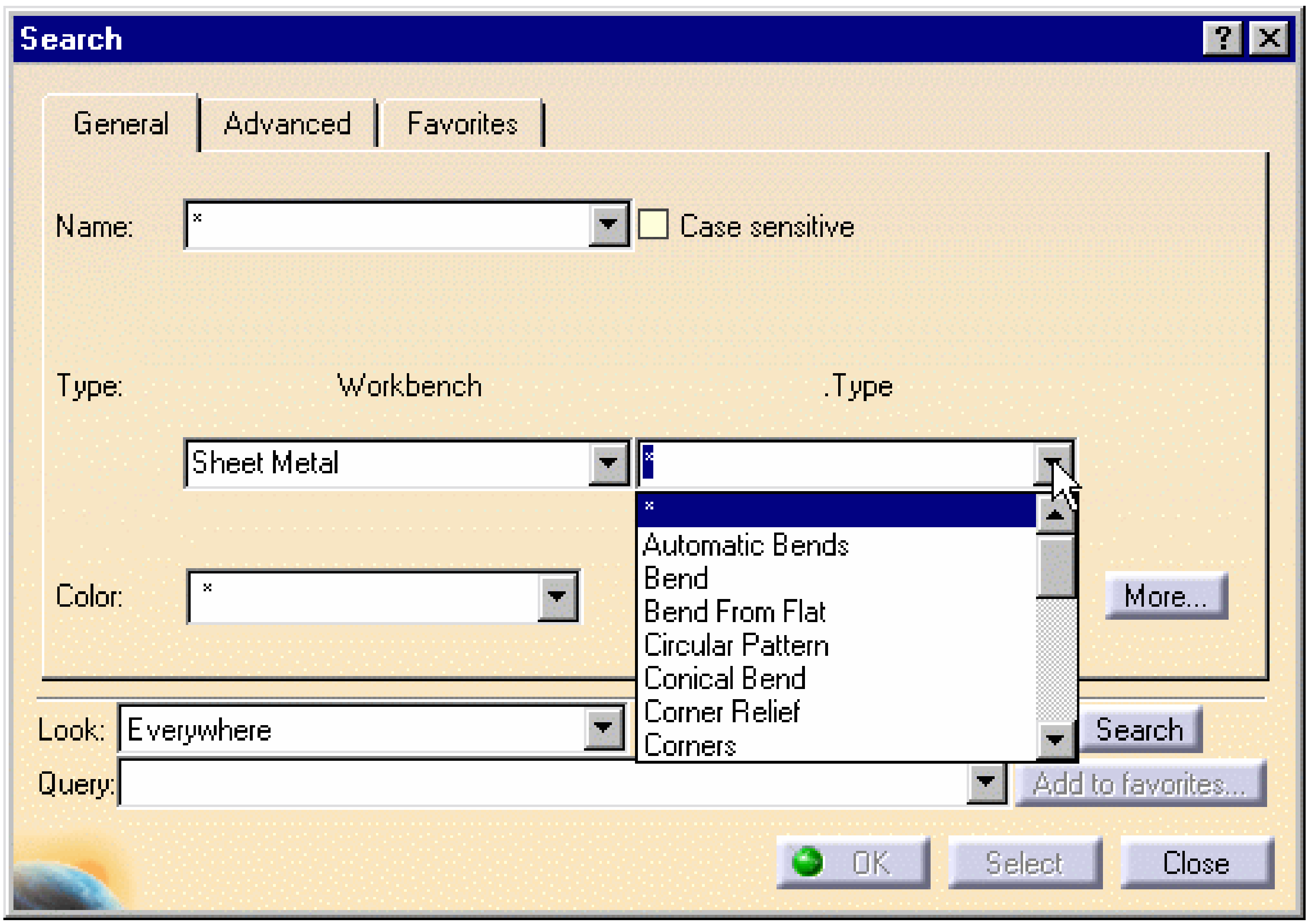
Open the PowerCopyStart.CATPart document.

1. Select the **Edit** -> **Search** menu item.

The Search dialog box is displayed.

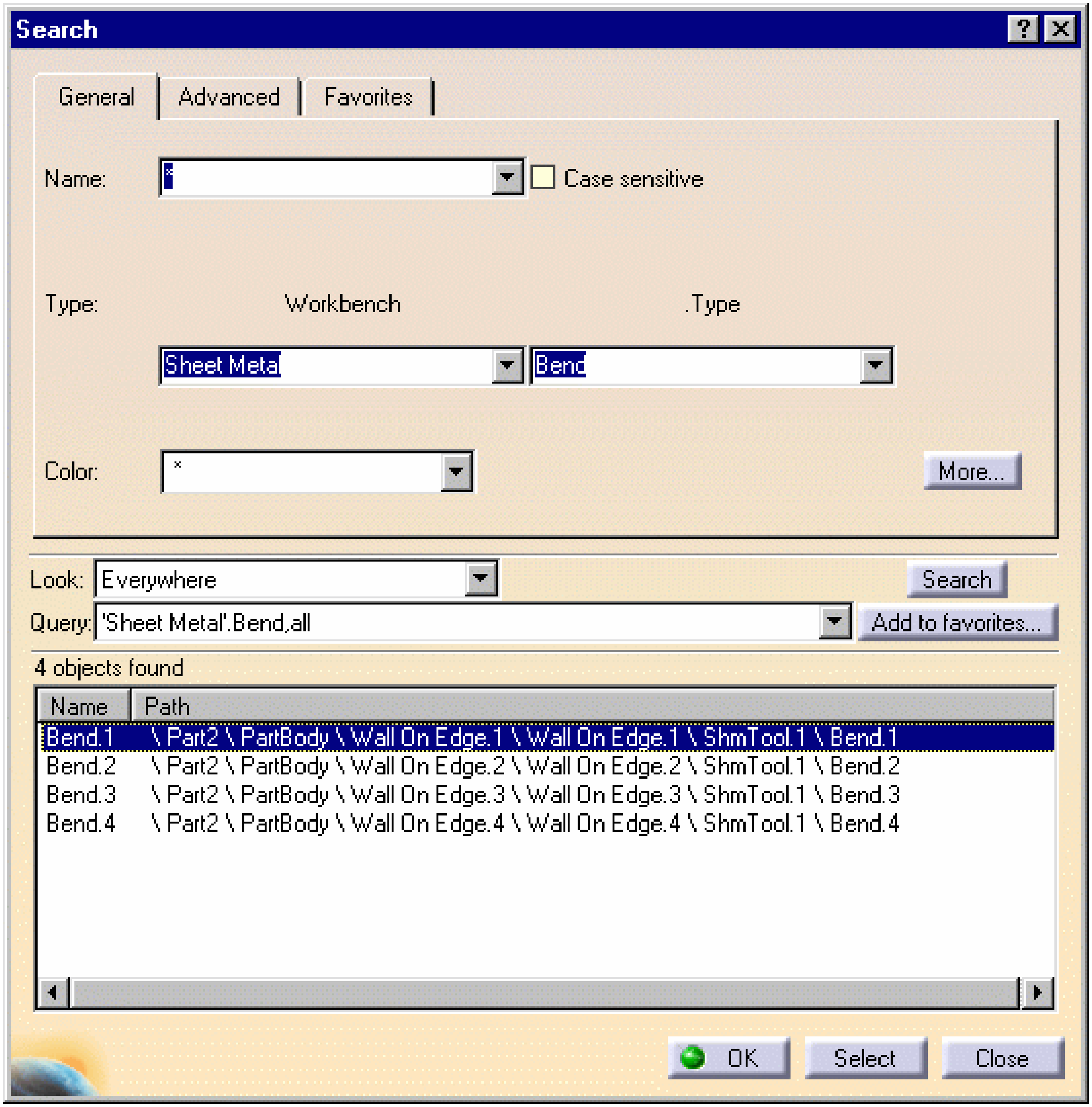
2. From the Type Workbench list choose Sheet Metal.

You can then display the list of Sheet Metal features from the **Type** list:



3. Select the type of feature you wish to find within the open .CATPart document, and click Search.

The list of all elements of the selected type is displayed in the Objects found field:

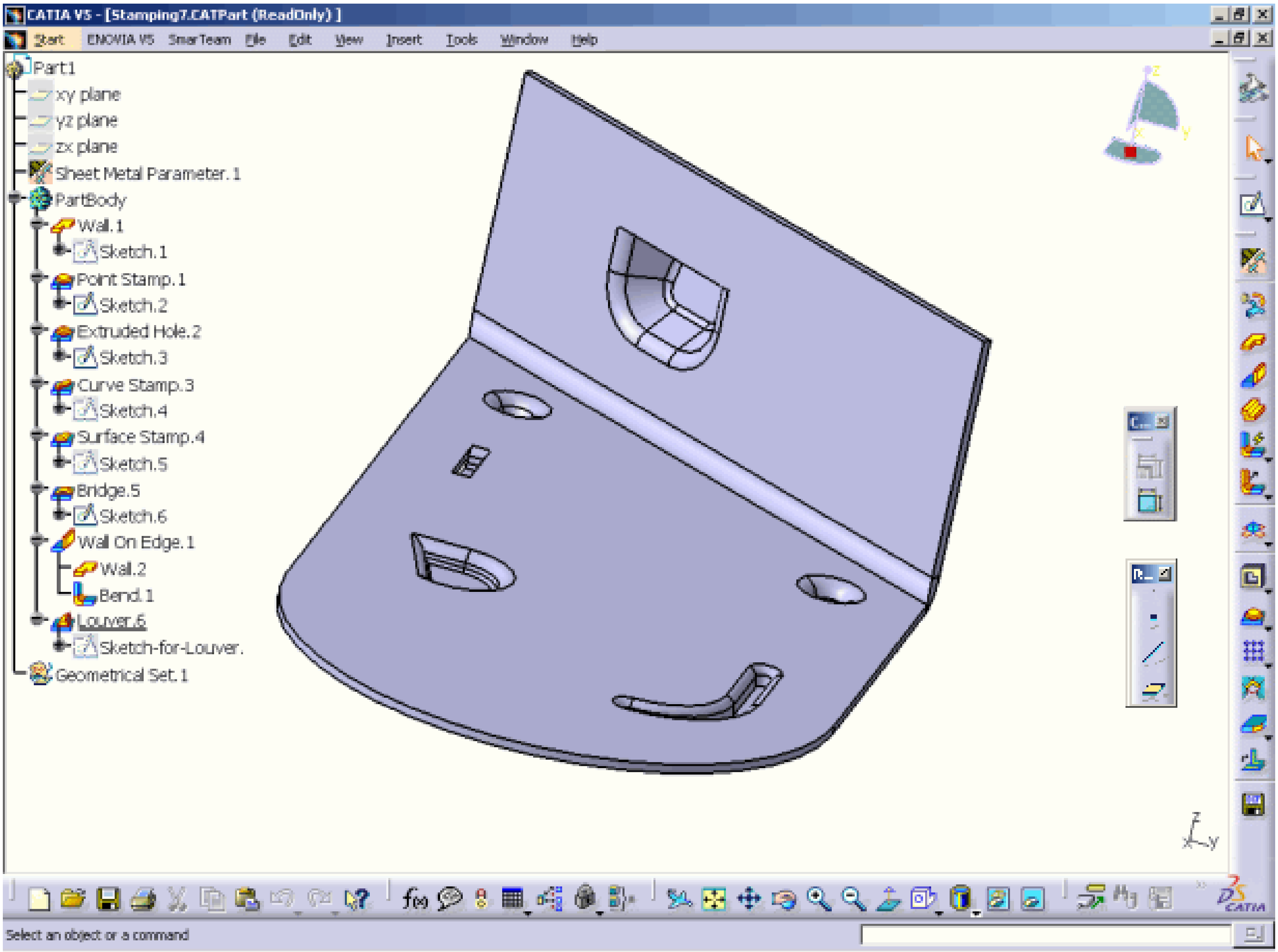


You can select an element from the list, it will be highlighted in the geometry area.

To find out more on the search capabilities, refer to Selecting Using the Search... Command (General Mode) and Selecting Using the Search... Command (Favorites and Advanced Modes) from the *Infrastructure User's Guide*.

Workbench Description

The SheetMetal Design application window looks like this:
Click the hotspots to display the related documentation.



Menu Bar
Sheet Metal Toolbar
Constraints Toolbar
Reference Elements Toolbar
Specification Tree

Menu Bar

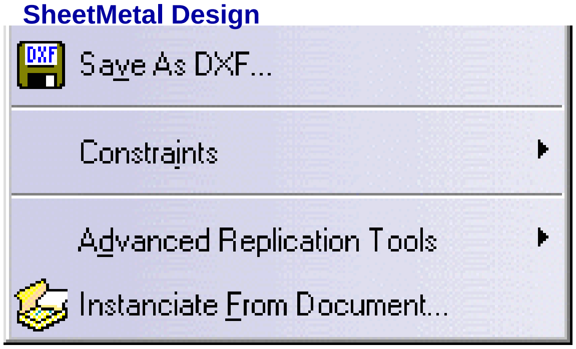
The various menus and menu commands that are specific to Sheet Metal Design are described below.

Start	File	Edit	View	Insert	Tools	Windows	Help

Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*. Refer to the Menu Bar section.

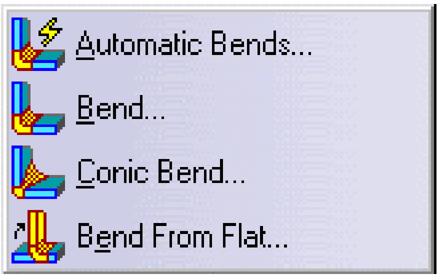
Insert

For...	See...
Sketcher...	Refer to Sketching in the *Sketcher User's Guide*.
Sheet Metal Parameters...	Managing the Default Parameters
Walls Recognition...	Creating Walls from an Existing Part
Wall...	Creating Walls from a Sketch
Wall on Edge...	Creating Walls from an Edge
Extrusion...	Extruding
Bends	Insert -> Bends
Swept Walls	Insert -> Swept Walls
Unfold	Insert -> Unfold
ShePocket	Insert -> ShePocket
Stampings	Insert -> Stampings
Patterns	Insert -> Patterns
CornerRelief...	Creating a Local Corner Relief
Corners	Insert -> Corners
Mapping...	Mapping Curves



Save As DXF...	Saving As DXF
Constraints	Setting Constraints in the *Part Design User's Guide*
Advanced Replication Tools	Insert -> Replication Tools
Instantiate From Document...	Instantiating PowerCopies

Insert -> Bends



For...	See...
Automatic Bends	Generating Bends Automatically
Bend	Creating Bends From Walls
Conic Bend	Creating Conical Bends
Bend	Generating a Bend from a Line

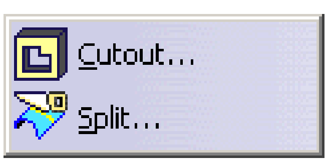
Insert -> Swept Walls

For...	See...
Flange	Creating a Flange
Hem	Creating a Hem
Tear Drop	Creating a Tear Drop
Swept Flange	Creating a Swept Flange

Insert -> Unfold

For...	See...
Unfold	3D View
MultiView	Concurrent Access

Insert -> ShePocket



For...	See...
Cutout...	Creating a Cutout
Split...	Splitting Geometry

Insert -> Stampings

For...	See...
Point Stamp	Creating a Point Stamp

Extruded Hole	Creating an Extruded Hole
Curve Stamp	Creating a Curve Stamp
Surface Stamp	Creating a Surface Stamp
Bridge	Creating a Bridge
Louver	Creating a Louver
Stiffening Rib	Creating a Stiffening Rib
User Stamping	Creating User-Defined Stamping Features

Insert -> Patterns

For...	See...
Rectangular Pattern	Creating Rectangular Patterns
Circular Pattern	Creating Circular Patterns
User-Defined Pattern	Creating User-Defined Patterns

Insert -> Corners

For...	See...
Corner	Creating Corners
Chamfer	Creating Chamfers

Insert -> Replication Tools

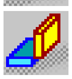
For...	See...
PowerCopy Creation	Creating PowerCopies
PowerCopy Save in Catalog	Saving PowerCopies

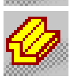
Sheet Metal Toolbar

 See Managing the Default Parameters

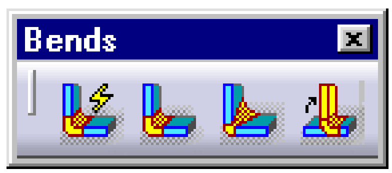
 See Creating Walls from an Existing Part

 See Creating Walls from a Sketch

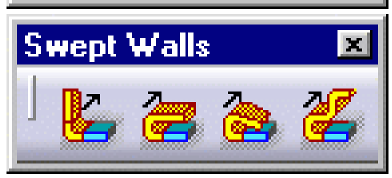
 See Creating Walls from an Edge

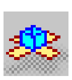
 See Extruding

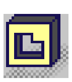
 See Creating Bends

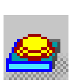


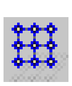
 See Creating Swept Walls

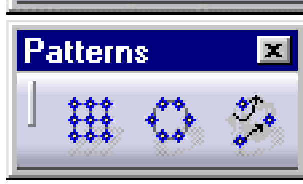


 See Unfolding the Part

 See Pockets

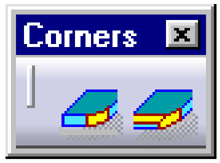
 See Stamping

 See Patterning



 See Creating a Local Corner Relief

See Creating Corners

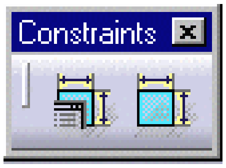


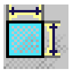
See Creating Chamfers

See Mapping Elements

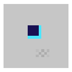
See Saving As DXF

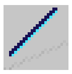
Constraints Toolbar

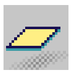




See Setting Constraints from the *Part Design User's Guide*

Reference Elements Toolbar

 See Creating Points

 See Creating Lines

 See Creating Planes

Specification Tree

Within the SheetMetal Design workbench, you can generate a number of features that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

- Sheet Metal Parameters
- Wall
- Wall On Edge
- Extrude
- Automatic Bends
- Bend
- Conical Bend
- Flat Bend
- Flange
- Hem
- Tear Drop
- User-defined Flange
- Cutout
- Split
- Point Stamp
- Extruded Hole
- Curve Stamp
- Surface Stamp
- Bridge
- Rectangular Pattern
- Circular Pattern
- User-Defined Pattern
- Corner Relief
- Corner
- Chamfer
- Mapping
- Point
- Line
- Plane

Louver

Stiffening Rib

User-Defined Stamp

Glossary

B C E F P R S W

B

bend A feature joining two walls

bend extremity Axial relimitation for a straight bend

C

corner relief A feature created at the intersection of several bends or of a bend and a wall allowing to remove matter so that the part can fold/unfold without overlapping.

cutout A feature corresponding to an opening through a feature.
The shape of the opening corresponds to the extrusion of a profile.

E

extrusion A feature created by extruding a profile and adding thickness

F

flange A feature created by sweeping a profile along a spine.
The different flanges or swept walls available are: simple and swept flange, hem and tear drop.

P

pattern A set of similar features repeated in the same feature or part

profile An open or closed profile including arcs and lines created by the **Profile** command in the Sketcher workbench

R

reference wall The first created wall; when unfolding the part, it is the fixed wall.

S

stamping A feature created by embossing sheet metal.
The different stampings available are:
point/circular, extruded/flanged hole, flanged cutout, curve, surface, bridge, louver and stiffening rib.

W

wall A feature created by adding thickness to a profile

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workbench

Generative Drafting

Sheet Metal Design