Generative Sheetmetal Design

Overview

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

This overview provides the following information:

Generative Sheetmetal Design in a Nutshell

The Generative 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.

Generative 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, Generative Sheetmetal Design offers the same ease of use and user interface consistency as all V5 applications.

As a scalable product, Generative 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 *Generative Sheetmetal Design User's Guide* has been designed to show you how to design sheet metal parts of varying levels of complexity.

Before Reading 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 Generative Sheetmetal Design workbench, and the Customizing section, which explains how to customize the Generative 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.

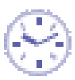
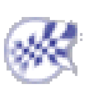
Graphic Conventions

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

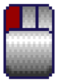
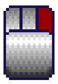
Text Conventions

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?

New Functionalities

Local fold and unfold of bends

You can now fold planar faces, as well as unfold cylindrical or conical faces in folded views, without changing the flattened view.

Overlap detection

A new command now lets you check overlapping areas on flat views.

Mirror

You can now mirror a given sheet metal feature. This duplicates a sheet metal feature symmetrically with respect to a plane.

Hybrid Design

You can now create wireframe and surfacic features within the same solid body which impacts the behavior of overlapping and local fold/unfold of bends.

Enhanced Functionalities

Cutouts

Additional possibilities are now available when creating a cutout: you can choose a direction for the cutout that is different from, or equal to, the normal direction. Additionally, the extrusion can now be of lesser length than the thickness. You can also now specify several supports for the cutout, instead of just one previously.

Stamp recognition

Stamps can now be recognized as Generative SheetMetal Design stamps.

Half-pierce for stamps

The new half pierce feature is available for circular stamps, curve stamps and surface stamps.

New options for surface stamps

Additional possibilities are now available when creating surface stamps. You can now define a stamp based on profile containing a punch and die sketch, a 3D curve sketch or a sketch with several inner contours.

Flange pattern

You can now create a pattern from a flange on rectangular, circular or user-defined patterns.

Hopper

Selecting a ruled surface is now possible when creating a hopper.

Document chooser integration

You can now customize the document environment (Tools > Options > General > Document tab) in order to select documents or paths using various interfaces (folder, Enovia, and so on). The interface can be customized for a folder or DLName path selection interface.

Getting Started

Before getting into the detailed instructions for using Generative 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
Extracting Drawings from the Sheet Metal Part

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

This tutorial, which is common to the *Sheet Metal Design User's Guide* and to the *Generative Sheet Metal Design User's Guide*, is illustrated using screen captures from the Sheet Metal Design workbench. At times, the user interface or results will slightly differ in the Generative Sheetmetal Design workbench. For more specific information concerning the Generative Sheetmetal Design workbench, refer to the User Tasks in this guide.

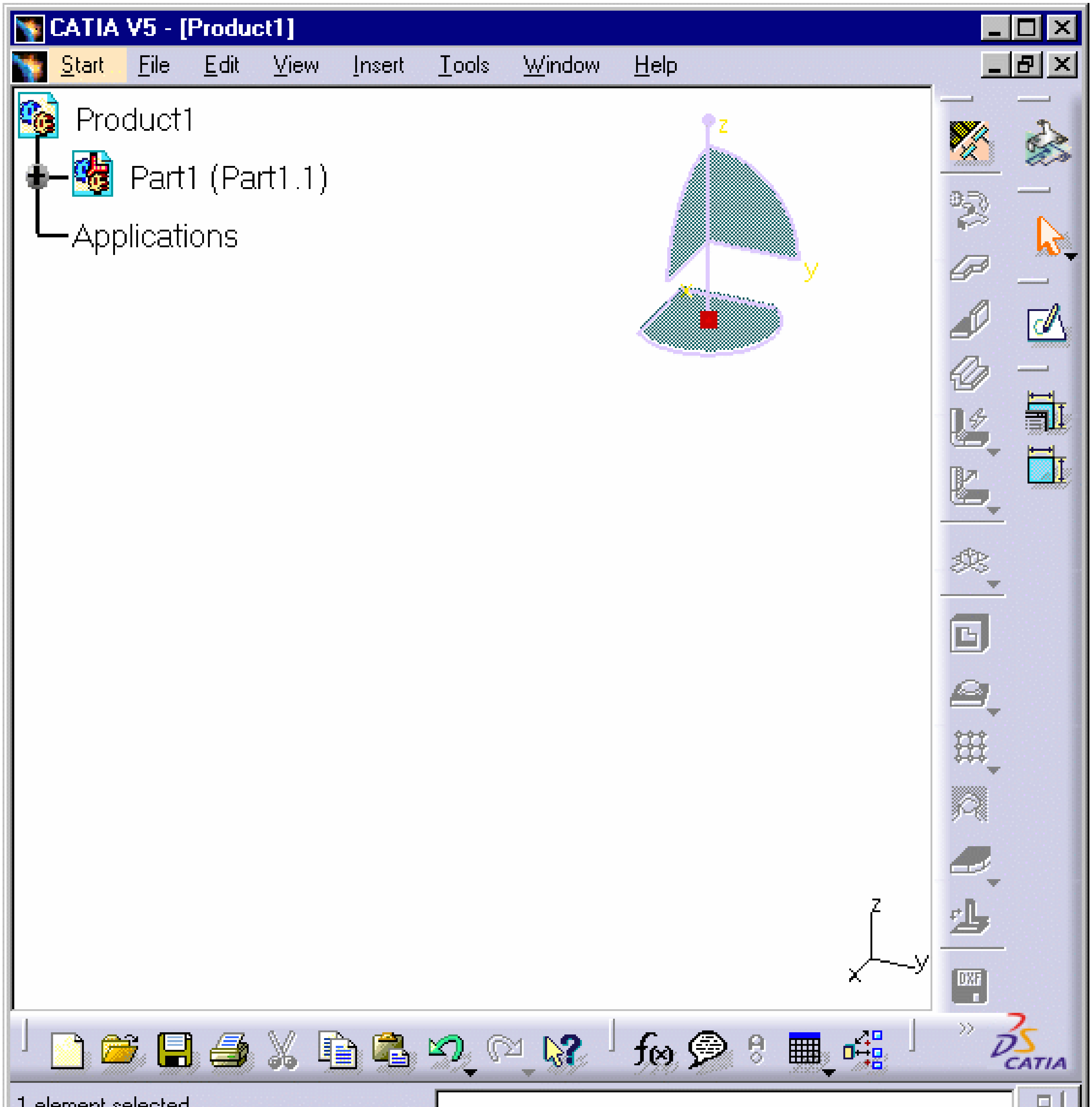
Entering the Workbench

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.



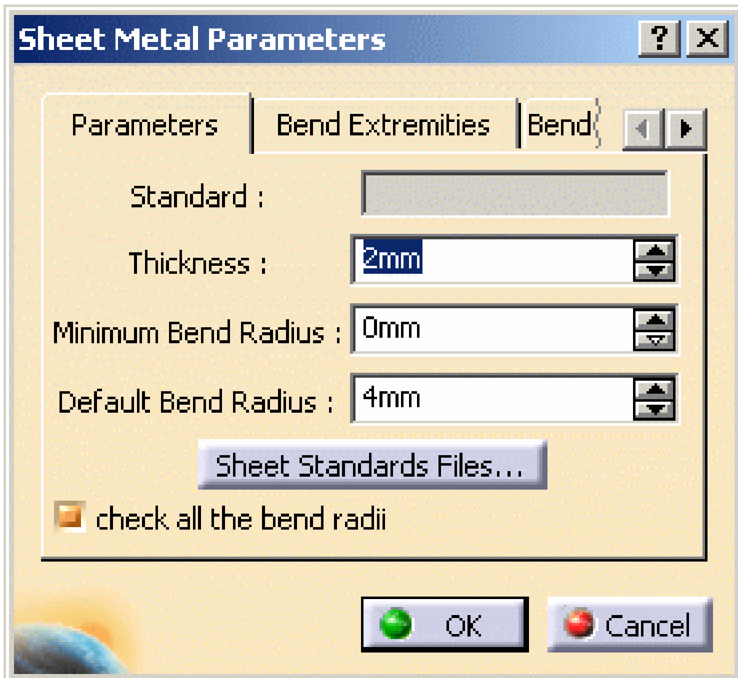
You may add the SheetMetal Design or the Generative SheetMetal Design workbench to your Favorites, using the **Tools -> Customize** item. For more information, refer to the *Infrastructure User's Guide*.

Defining the Sheet Metal Parameters

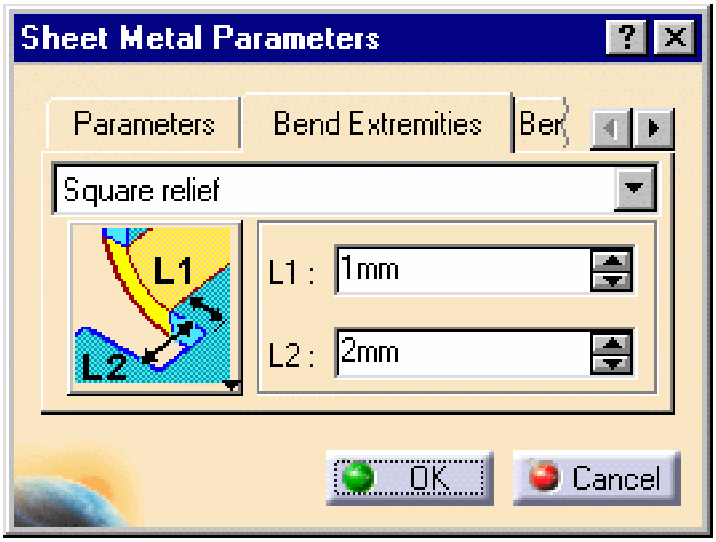
This task shows you how to configure the sheet metal parameters.

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.

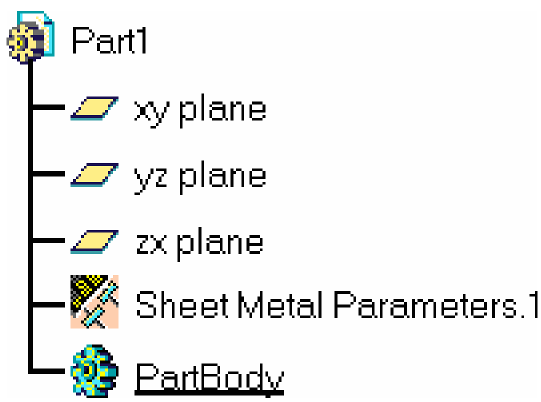


5. Select **Tangent** in the Bend Extremities combo list.

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.

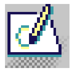
   The **Sheet Metal Parameters** feature is added in the specification tree.

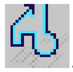


The other two tabs are not used in this scenario.

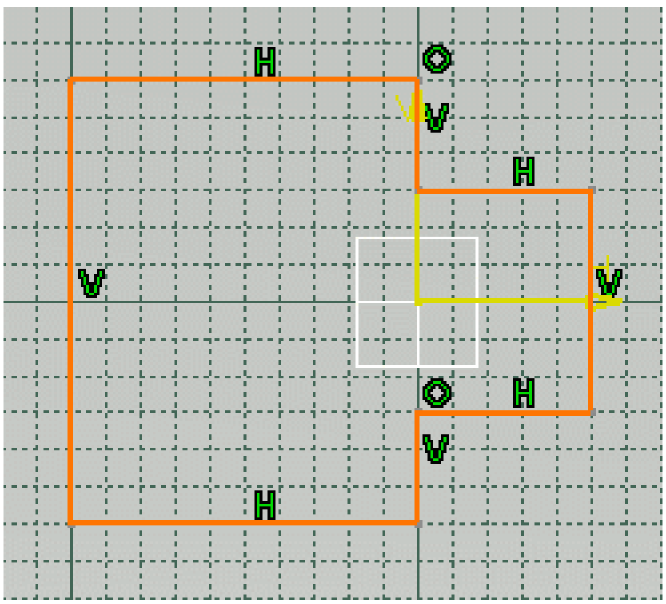
Creating the First Wall

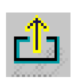
This task shows how to create the first wall of the sheet metal Part.

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

2. Select the **Profile** icon .

3. Sketch the profile as shown below:

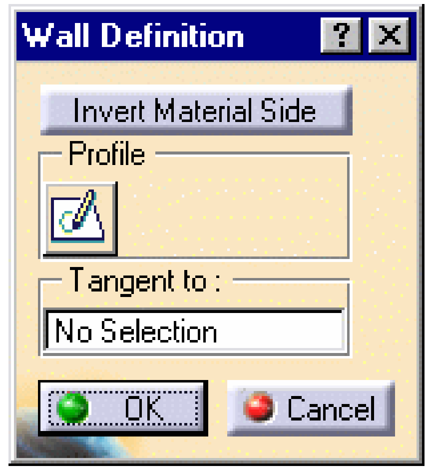


4. Click the **Exit workbench** icon  to return to the 3D world.

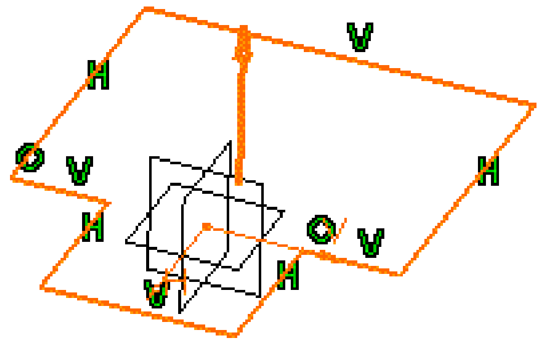
   The sketch remains selected.

5. Click the **Wall** icon .

The Wall Definition dialog box opens.

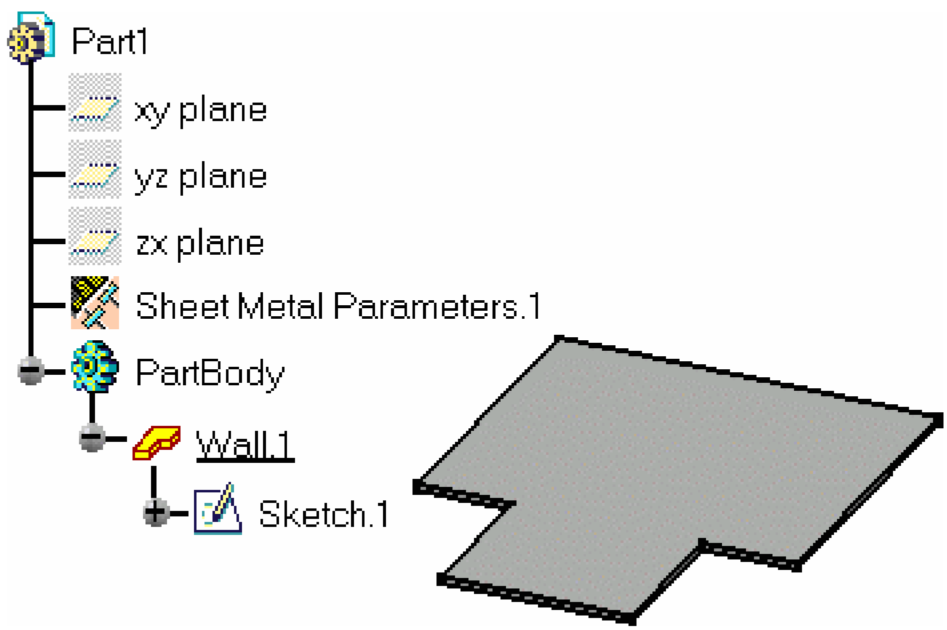


By default, the Material Side is set to the top.



6. Click OK.

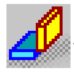
The **Wall.1** feature is added in the specification tree



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.

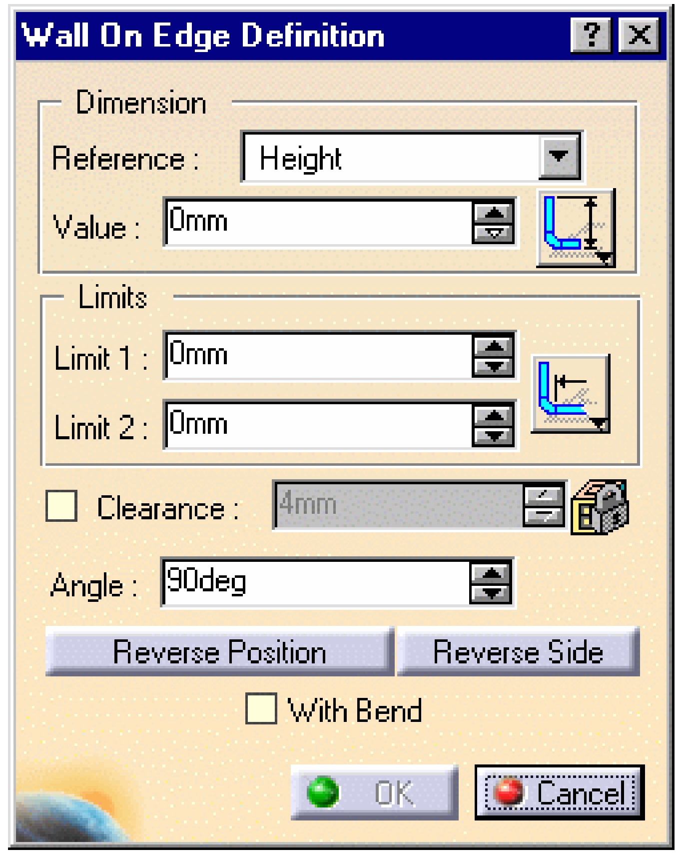
1. Click the **Wall on Edge** icon .

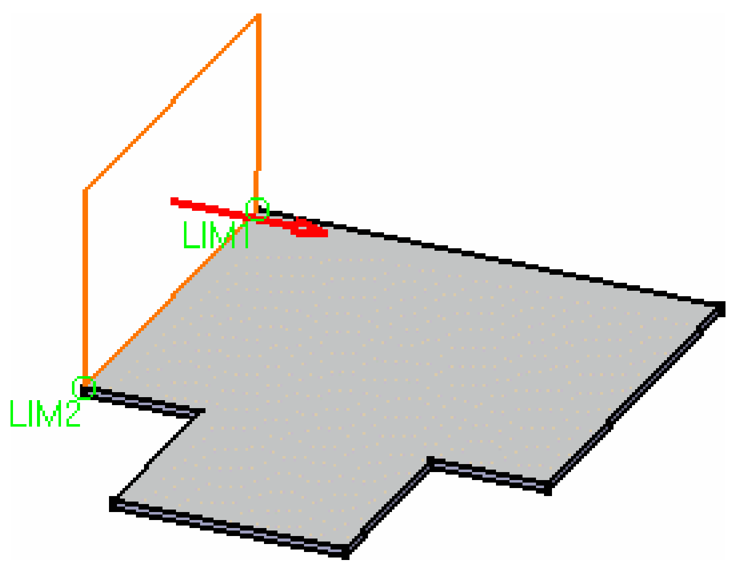
   The Wall On Edge Definition dialog box opens.

2. Select the left edge.

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

   The application previews the wall.



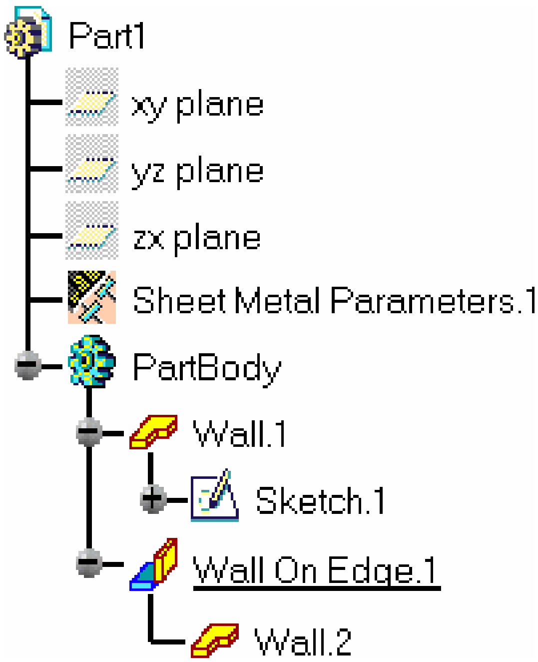


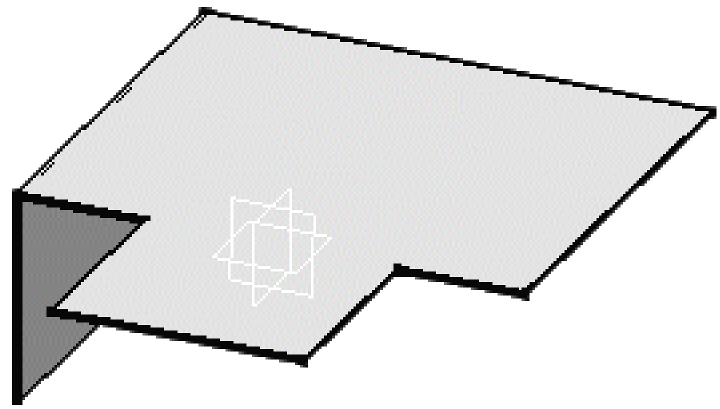
By default, the material side is such that it ensures a continuity with the reference profile. If needed, invert it using the **Reverse side** button, or clicking the arrow.

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

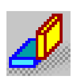
5. Click **OK**.

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



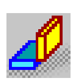


6. Select the right edge.

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.

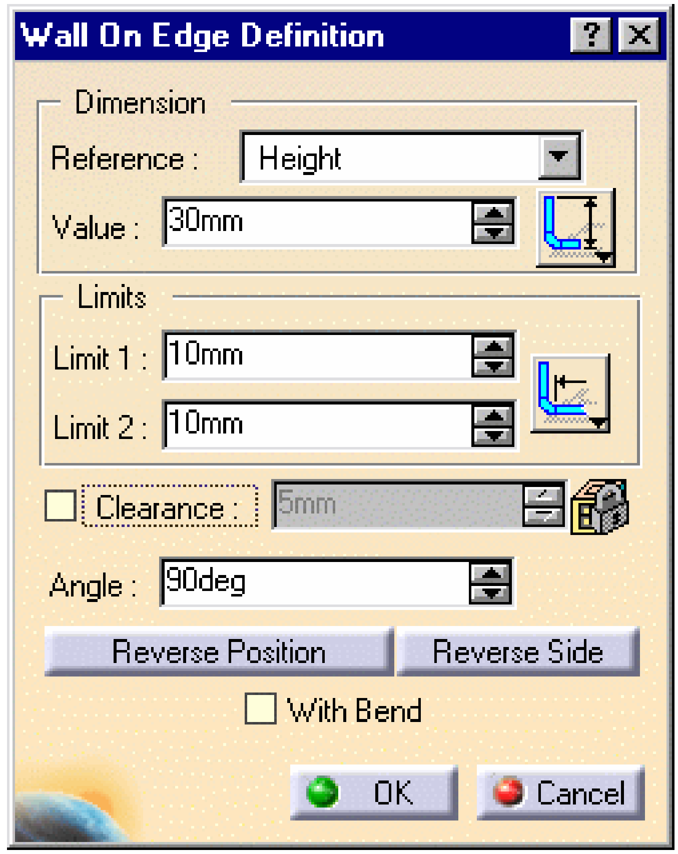
9. Click the **Wall on Edge** icon  again.

10. Select the front edge.

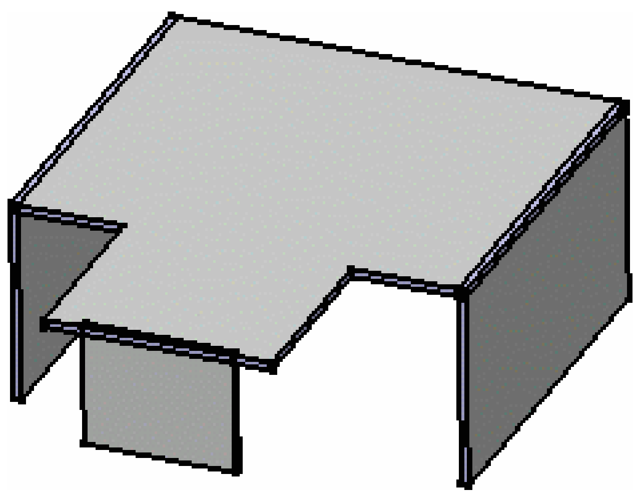
    The Wall Definition dialog box opens with the parameters previously selected.

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:

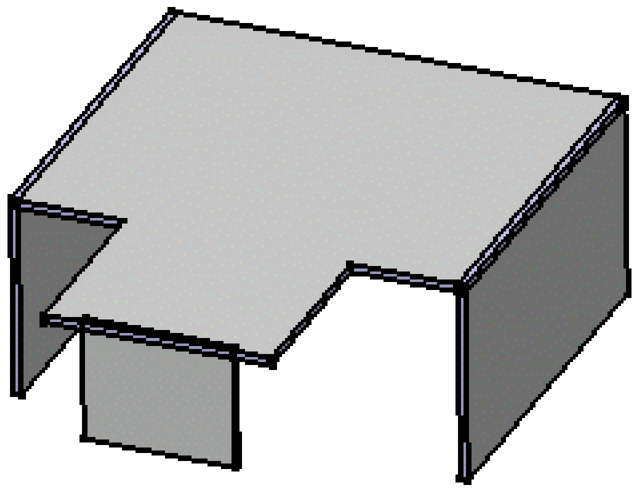


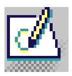
Creating a Cutout

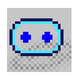
In this task, you will learn how to:

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

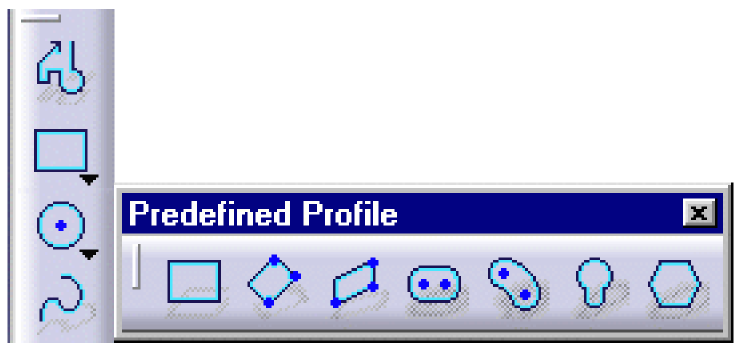
1. Select **Wall On Edge.2** from the geometry area to define the working plane.



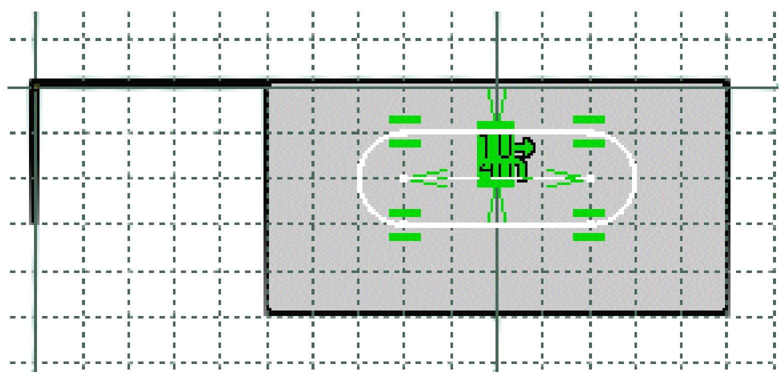
2. Click the Sketcher icon .

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.



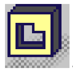
4. Click to create the first point and drag the cursor.



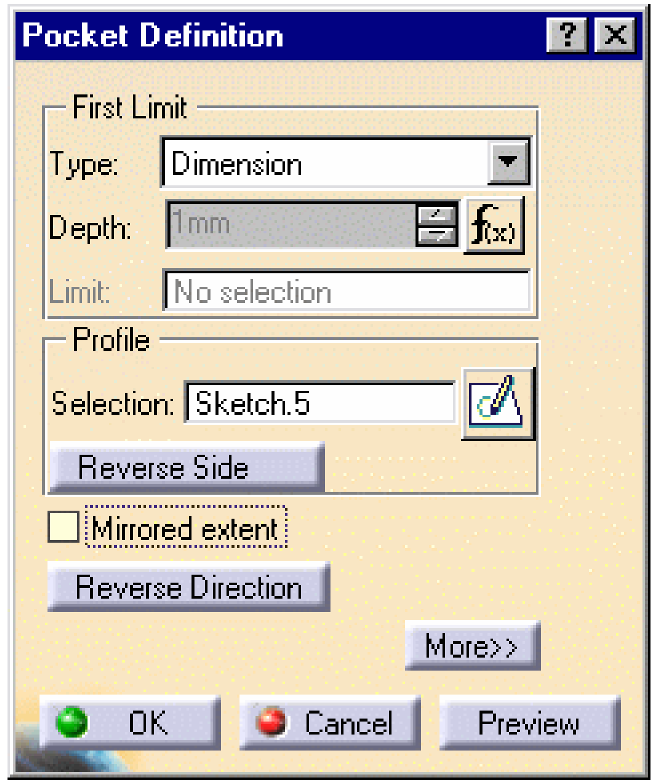
5. Click to create the second point.
   The first semi-axis of the profile is created.

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.

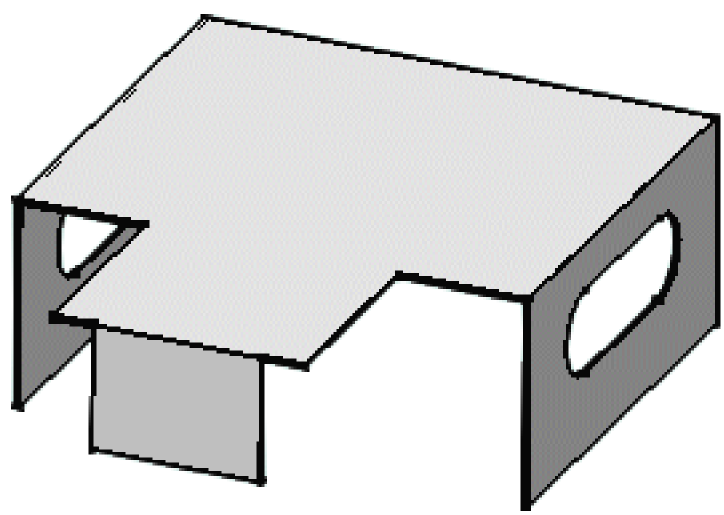


9. Set the Type to **Up to last** option to define the limit of your cutout.

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:



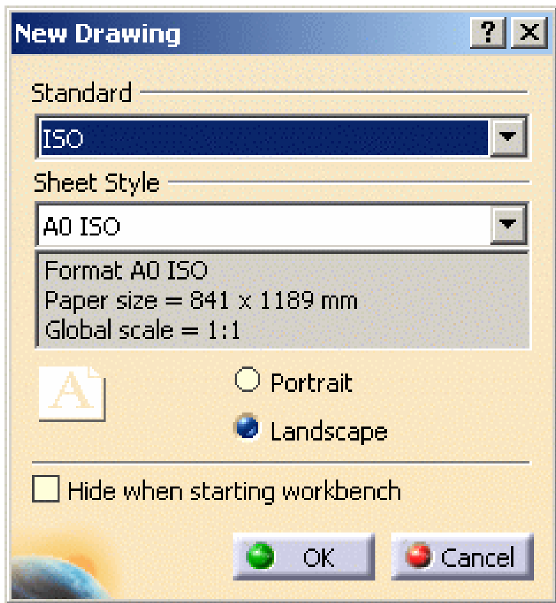
Extracting Drawings from the Sheet Metal Part

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.



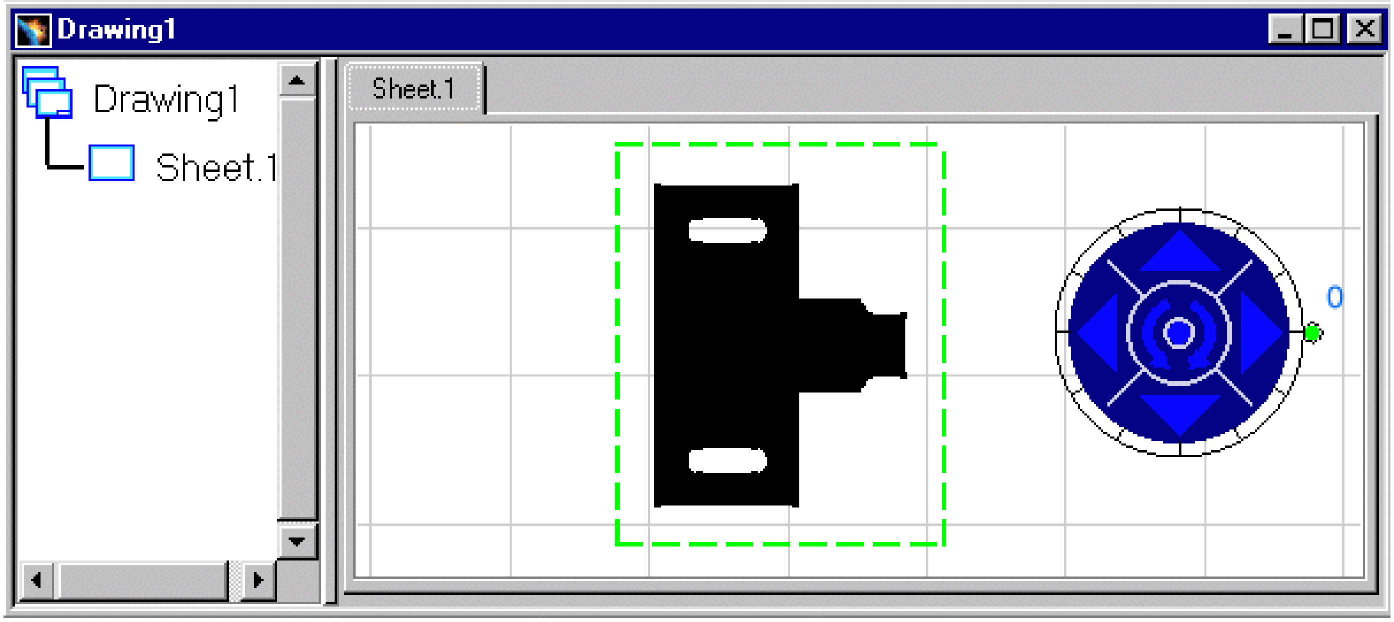
3. Click **OK**.

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.

User Tasks

The tasks you can perform in the Generative Sheetmetal Design workbench are described in the following chapters and tasks.

Managing the Default Parameters

This section explains and illustrates how to use or modify various kinds of features.

Using Generative 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.

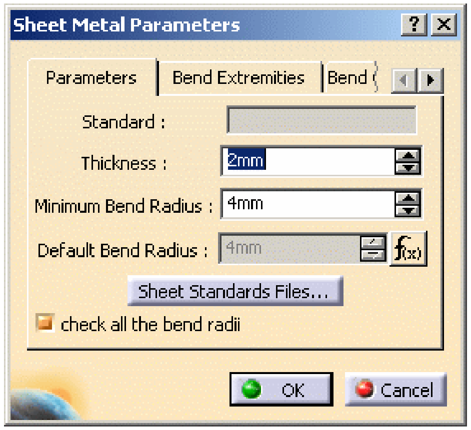
Define the bend allowance: select the **Bend Allowance** tab and define the allowance value (K factor).

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.

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.

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.

Example for a bend allowance table:

A bend table is associated to the default bend radius. Each bend radius is linked by a formula to the default bend radius.

If a bend table is defined, the bend allowance (for a given angle) is taken into account when the bend is created. Otherwise, the default K factor is used.

You should know that bend tables are used only for:

- the default bend radius; in the case of a non-standard bend, the K factor is used by default. In this case, you will need to modify manually the local K factor if you want the required allowance to be taken into account.
- the Bend and Bend from Flat features; other features use the K factor for flat length computation.

Main Sheet Metal Parameters Design Table

	A	B	C	D
1	SheetMetalStandard	Thickness(mm)	DefaultBendRadius(mm)	BendTable
2	AG 3412	2	3	BendTableT2R3
3	AG 3824	4	5	BendTableT4R5

Bend Table for Thickness 2 and Default Bend Radius

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

	A	B
1	OpenAngle(deg)	Deduction(mm)
2	20	-0.3
3	30	-0.531
4	40	-0.762

Example for 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		

Example for a stamp standard file:

Main Sheet Metal Parameters Design Table

SheetMetalStandard	SurfaceStampStd	CurveStampStd	CircularStampStd	BeadStd	BridgeStd	FlangedCutoutStd	ExtrudedHoleStd	StiffeningRibStd
AG 3412	SurfaceStampAG3412.xls	CurveStampAG3412.xls	CircularStampAG3412.xls	BeadAG3412.xls	BridgeAG3412.xls	FlangedCutoutAG3412.xls	ExtrudedHoleAG3412.xls	StiffeningRibAG3412.xls
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**

StandardName	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
S1	6	80	2	2
S2	8	75	1	1

- **Curve Stamp**

StandardName	Height (mm)	Length (mm)	Angle (deg)	Radius2 (mm)	Radius1 (mm)
C1	4	6	75	1	1
C2	5	7	80	1	1

- **Circular Stamp**

StandardName	Diameter (mm)	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
C1	10	6	80	2	2
C2	20	5	85	1	1

- **Bead**

StandardName	SectionRadius(mm)	EndRadius(mm)	Height(mm)	Radius1 (mm)
Bead04	4	6	4	2
Bead09	9	10	5	3

- **Bridge**

StandardName	Angle (deg)	PositioningAngle (deg)	Length (mm)	Radius1 (mm)	Radius2 (mm)	Height (mm)	Width (mm)
B1	80	5	10	2	2	6	5
B2	75	4	12	1	1	8	6

- **Flanged Cutout**

StandardName	Height (mm)	Angle (deg)	Radius1 (mm)
F1	6	80	2
F2	8	75	1

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

StandardName	Diameter (mm)	Height (mm)	Angle (deg)
D20	20	6	90
D15	15	6	70

- **Stiffening Rib**

StandardName	Angle (deg)	Radius2 (mm)	Length (mm)	Radius1 (mm)
S1	80	2	30	2
S2	75	1	35	2

Modifying the Bend Extremities

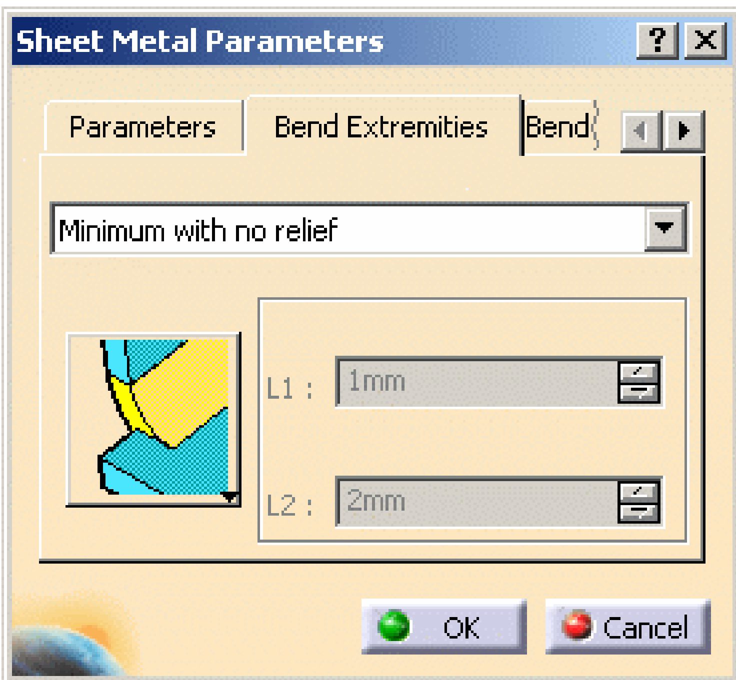
This section is only available with the SheetMetal Design products.

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.

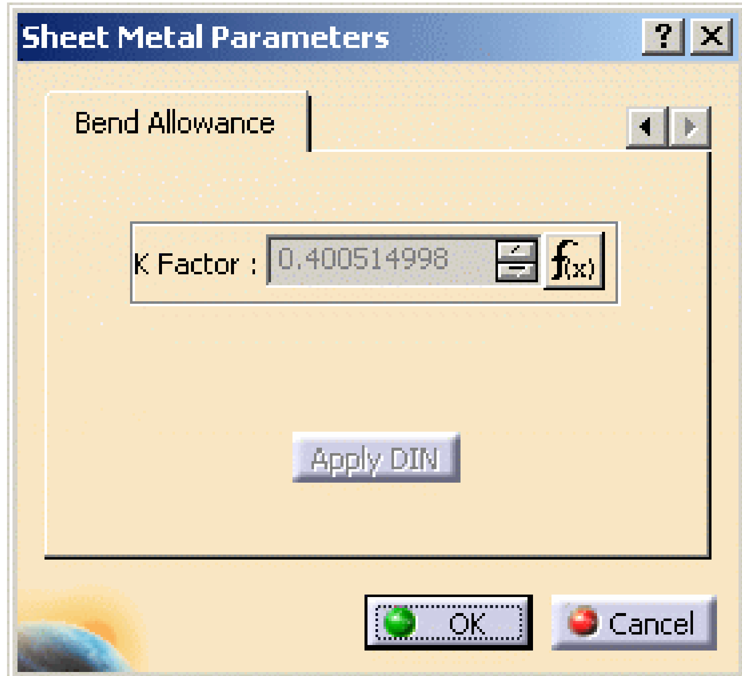
- **Maximum**: the bend is calculated between the furthest opposite edges of the supporting walls.

4. Click **OK** to validate.

Computing the Bend Allowance

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

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

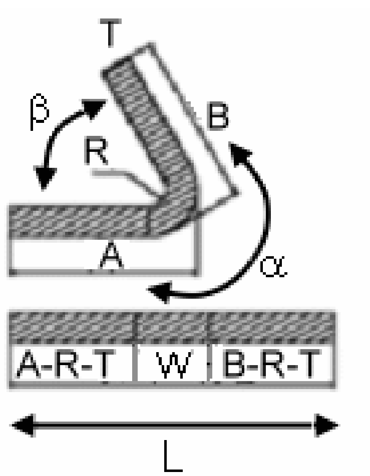


The Sheet Metal Parameters dialog box is displayed.

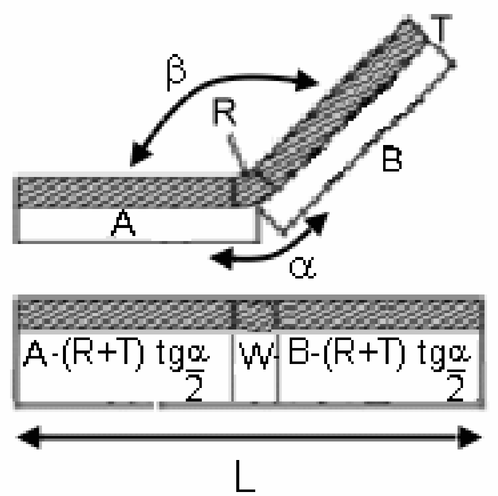
The fourth tab concerns the bend allowance.

- **Bend Allowance**

The bend allowance corresponds to the unfolded bend width.



bend < 90deg



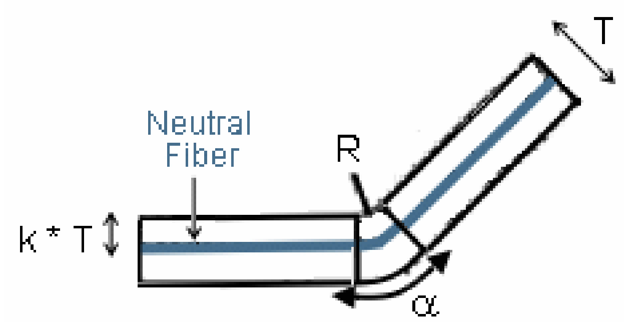
bend > 90deg

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.

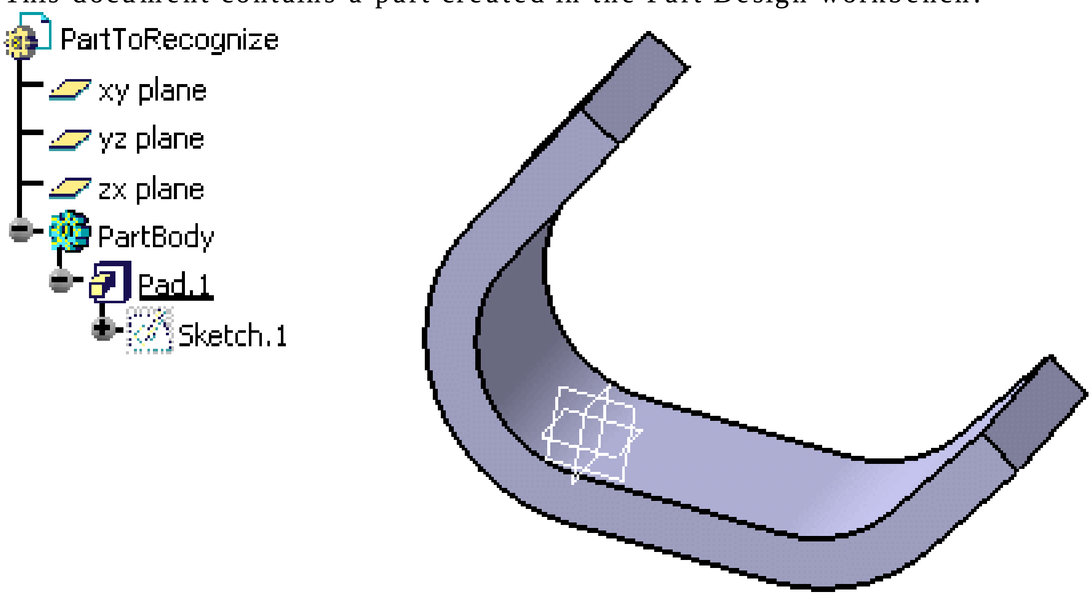
Recognizing Walls From an Existing Part

This task illustrates how to recognize an existing part as a sheet metal part, i.e. recognize as sheet metal features the thin part shapes of a part created using the Part Design workbench or of a CATIA Version 4 Solid, for example. You can also use this functionality to recognize parts created using the SheetMetal Design workbench as Generative Sheetmetal Design parts.

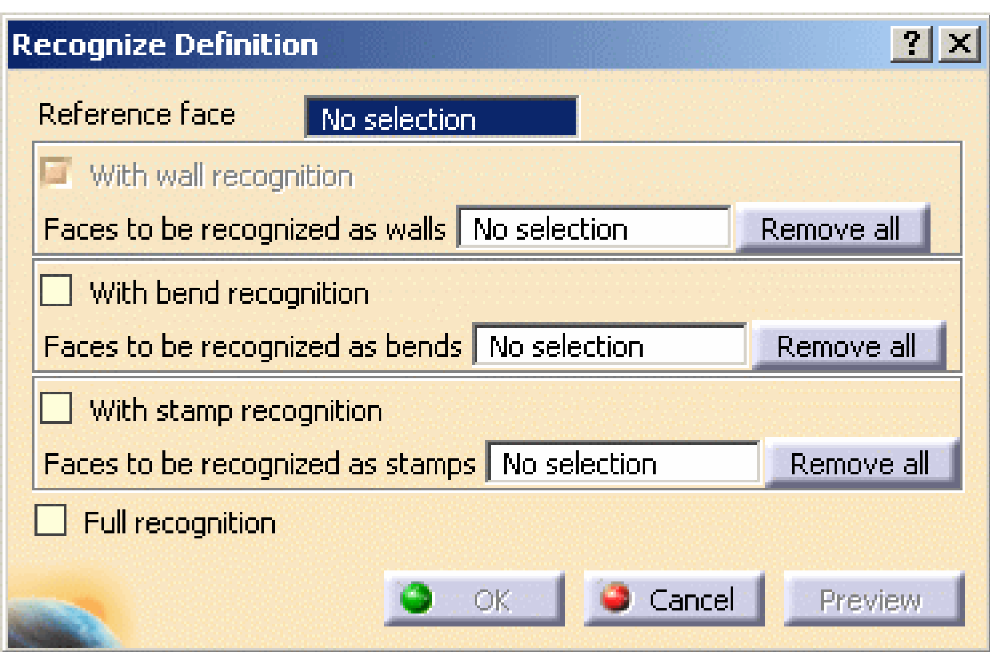
Currently, walls and bends can be recognized. Hems are recognized as walls.

Open the NEWRecognize01.CATPart document.

This document contains a part created in the Part Design workbench:



1. Click the **Recognize** icon. The Recognize Definition dialog box is displayed.



Note that the **With Wall recognition** option is already selected, and grayed out. This is because at least walls will be recognized, regardless of the other options you may choose.

2. Select a reference face. It will be the reference face for unfolding and for the definition of the sheet metal parameters (i.e. all default parameters will be based on this face).

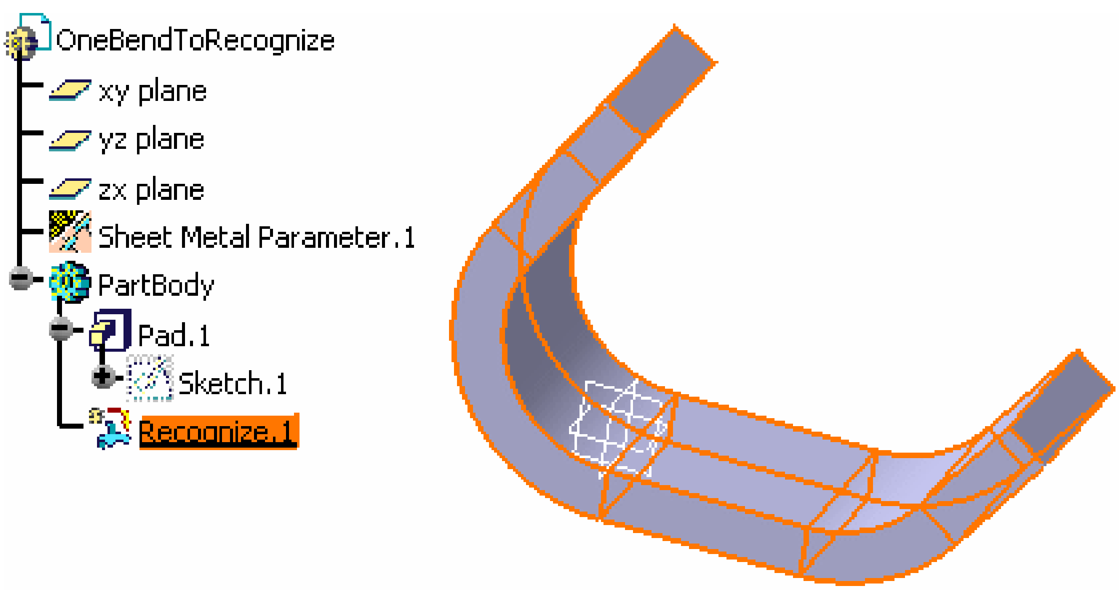
3. For the purpose of this scenario, select **Full recognition** to specify that you want as many features (walls, bends) as possible to be recognized. Doing this automatically selects the **With Bend recognition** option.

   You can also manually select the faces to be recognized as walls, after having activated the corresponding field. Then, select the **With Bend recognition** option, activate the associated field and manually select the faces to be recognized as bends.

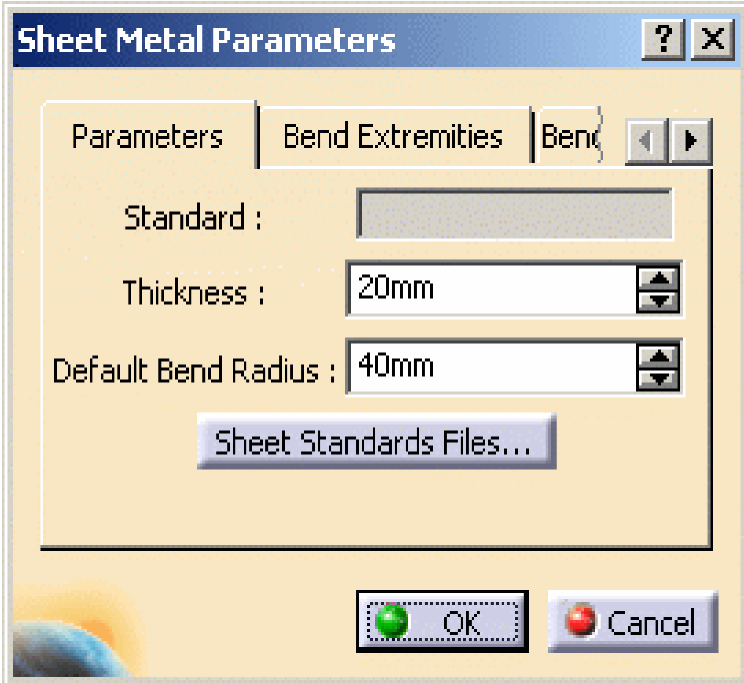
4. Click **OK** to validate.

   The walls are generated from the Part Design geometry. The **Recognize.1** feature is added to the tree view.

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



5. Select the **Sheet Metal Parameters** icon  to display the sheet metal parameters.



On the **Parameters** tab:

- the **Thickness** is equal to 20mm,
- the **Minimum Bend Radius** value is set to 0mm,
- the **Default Bend Radius** value amounts to twice that of the thickness.

On the **Bend Extremities** tab:

- the bend extremities are set to **Minimum with no 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 other parameters (minimum bend radius, default bend radius and bend extremities) in order for them 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 bend corner relief affect all features, and therefore can be edited even for "recognized" features.

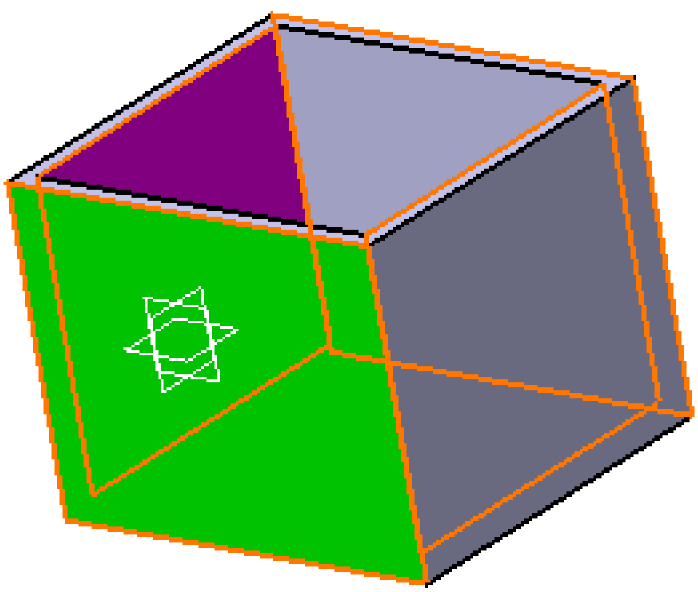
You can also define the sheet metal parameters prior to recognizing the part. In this case, you need to make sure that the **Thickness** parameter value corresponds to the part thickness.

6. When all parameters have been redefined as needed, click **OK** in the **Sheet Metal Parameters** dialog box.

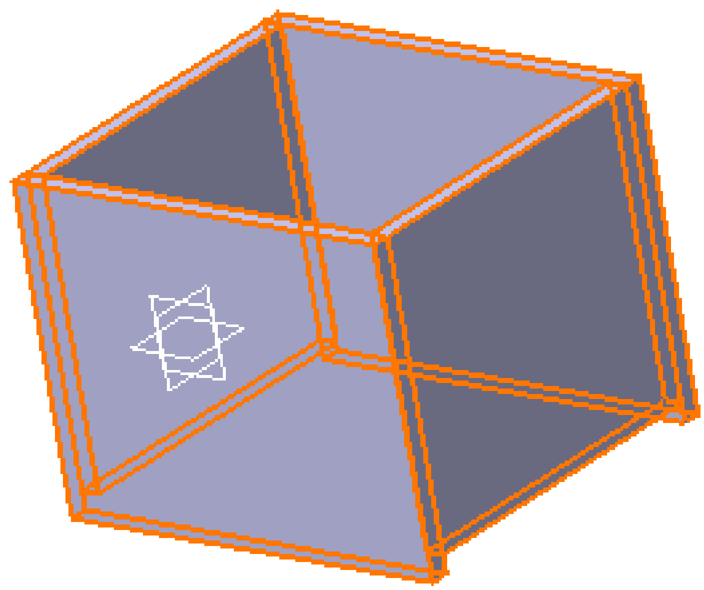
   The solid is now a Generative Sheetmetal Design part. You can now deal with it as with any other Generative Sheetmetal Design part, adding Generative Sheetmetal Design features to complete the design, or unfolding it.

In certain cases, there may be an ambiguity as regards the faces from which the walls are to be generated. For example, if the initial part is a box such as shown below, 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 recognizing the walls.

Faces to select



Recognition result



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 material sides.

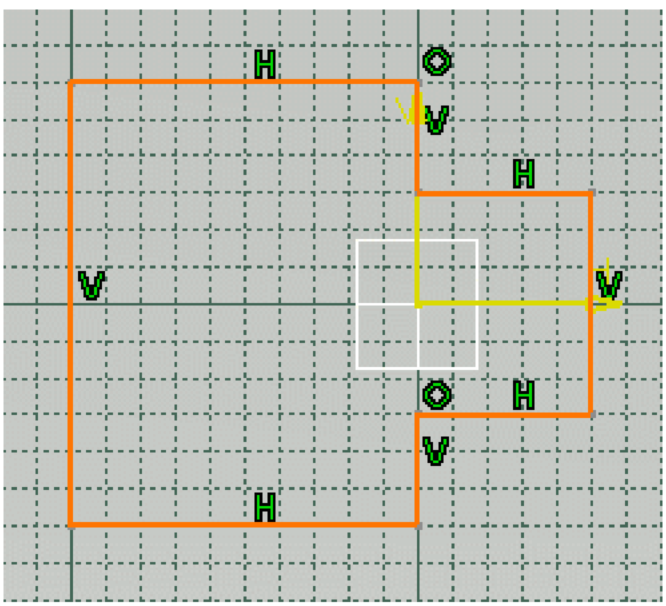
Creating Walls from a Sketch

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.

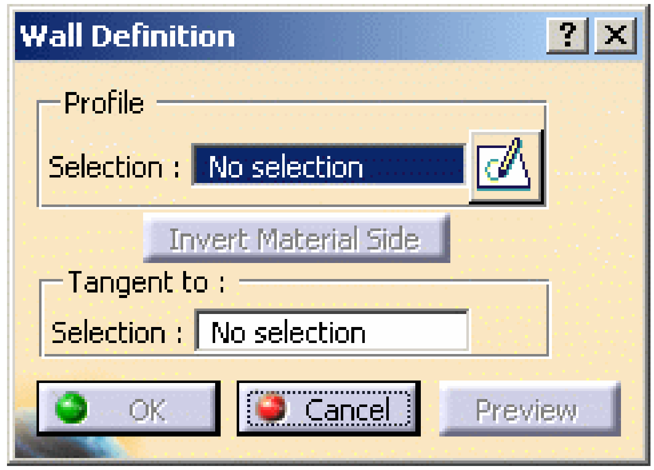
1. Click the **Sketcher** icon then select the **xy plane**.
2. Select the **Profile** icon .
3. Sketch the profile as shown below:



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

5. Click the **Wall** icon .

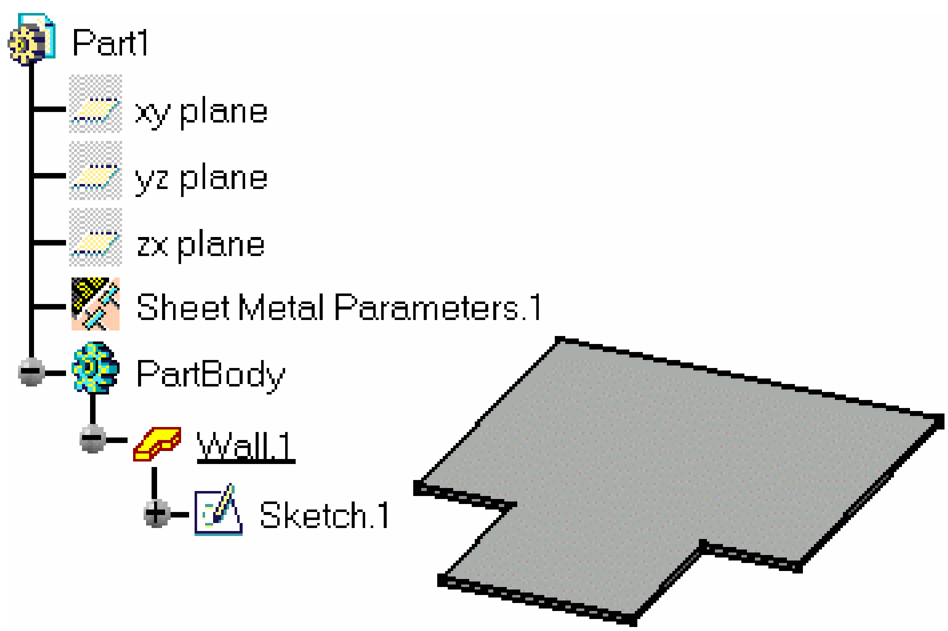
The Wall Definition dialog box opens.



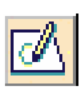
By default, the Material Side is set to the top.

6. Click OK.

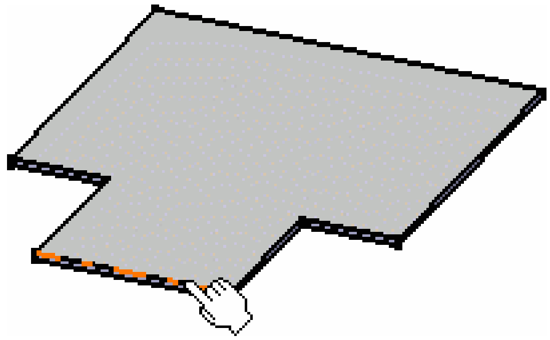
The **Wall.1** feature is added in the specification tree.



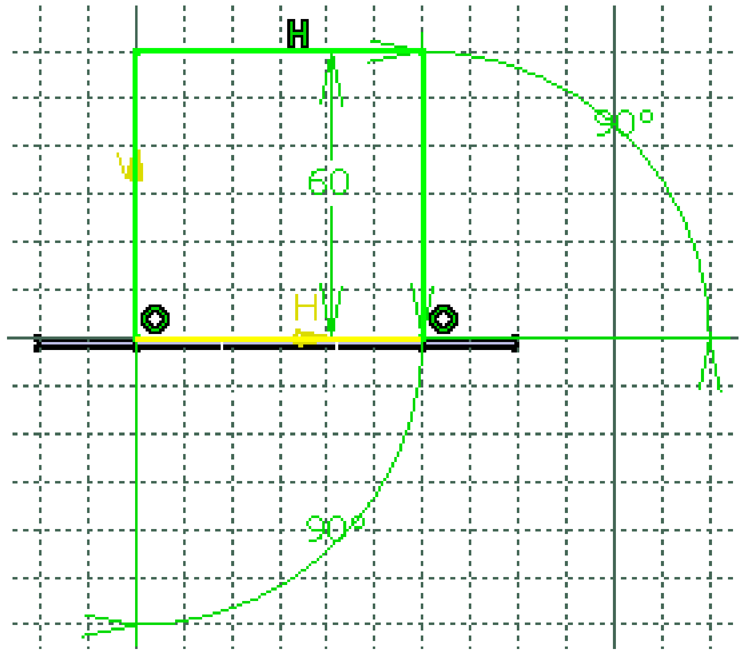
The first wall of the Generative Sheetmetal Design Part is known as the Reference wall.

- 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.

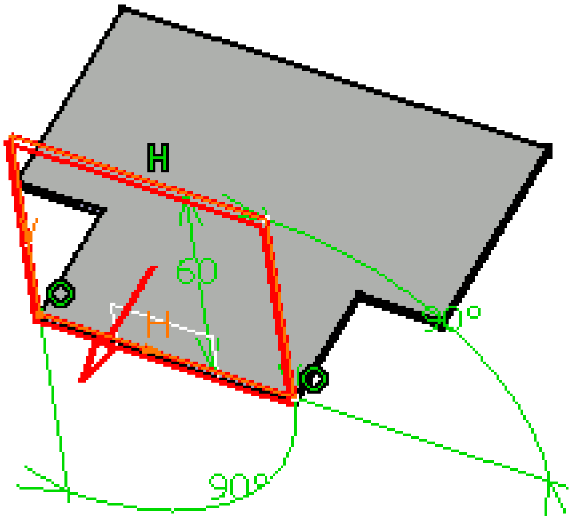
This is also very useful if you have selected an edge from a wall and clicked the **Wall** icon .



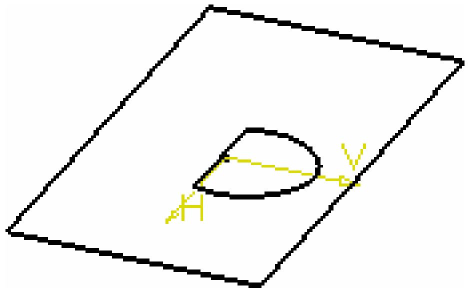
In this case, the sketcher is automatically activated displaying a square sketch constrained to the plane perpendicular to the support of the selected edge.



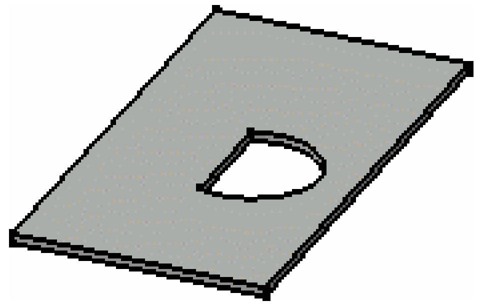
You can then directly draw a sketch, then exit the sketcher and return to the wall creation step.



- 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

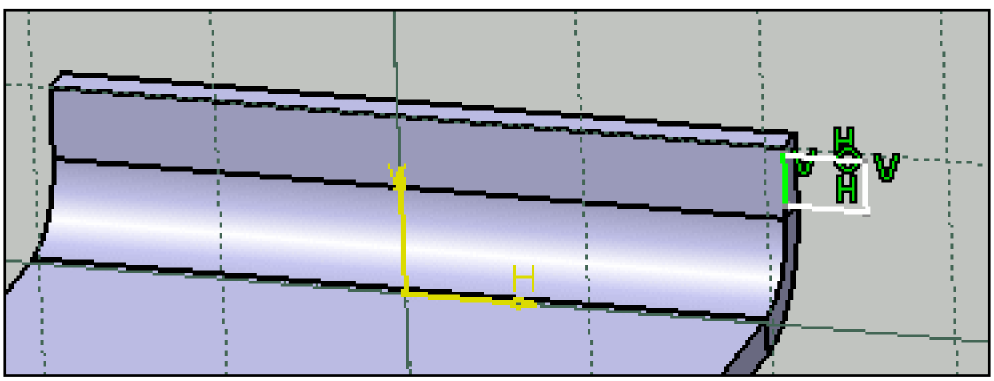
This task shows how to create a wall tangent to a planar part of another wall or of a flange. This capability will then allow this tangent wall to be seen when unfolding the part, even though there is no bend linking it to its tangent support, provided this support can also be unfolded.

Open the NEWTangentWall1.CATPart document.

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

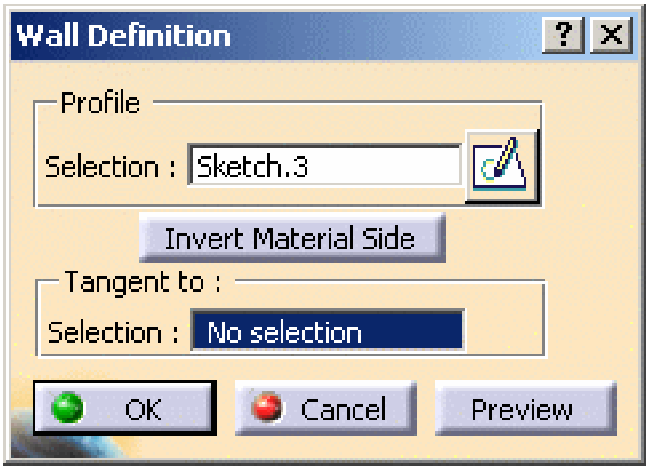
2. Select the **Profile** icon and sketch the profile as shown below.

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

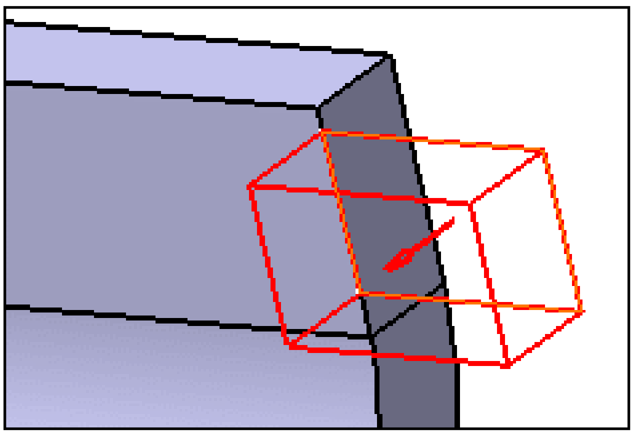


4. Click the **Exit workbench** icon to return to the 3D world.

5. Make sure the sketch is still active, then click the **Wall** icon . The Wall Definition dialog box is displayed.

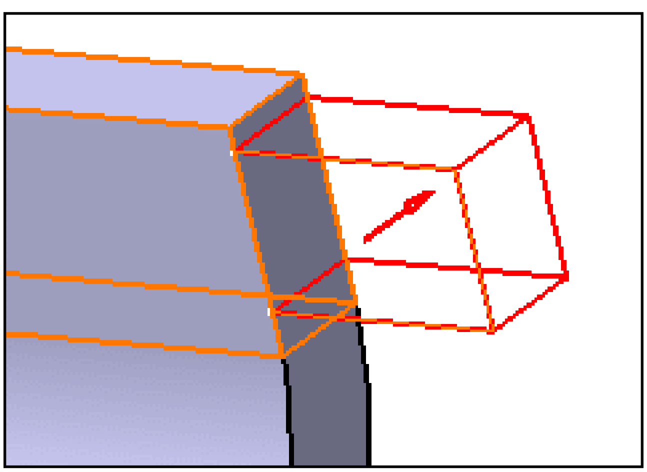


Note the arrow which indicates the orientation of the wall to be created.



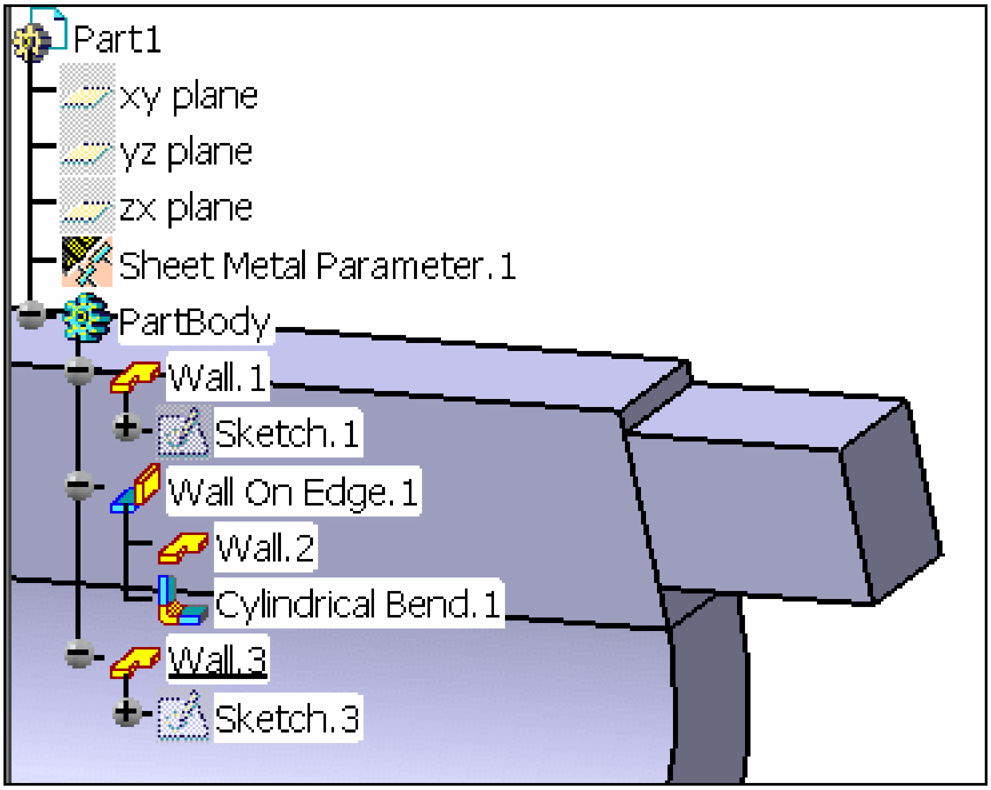
6. Make sure the **Tangent to** field is active, then select the wall to which the new wall has to be tangent. Here, you need to select the planar face of the flange.

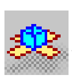
Note that the orientation automatically changes to conform to the material orientation already defined on the support wall.



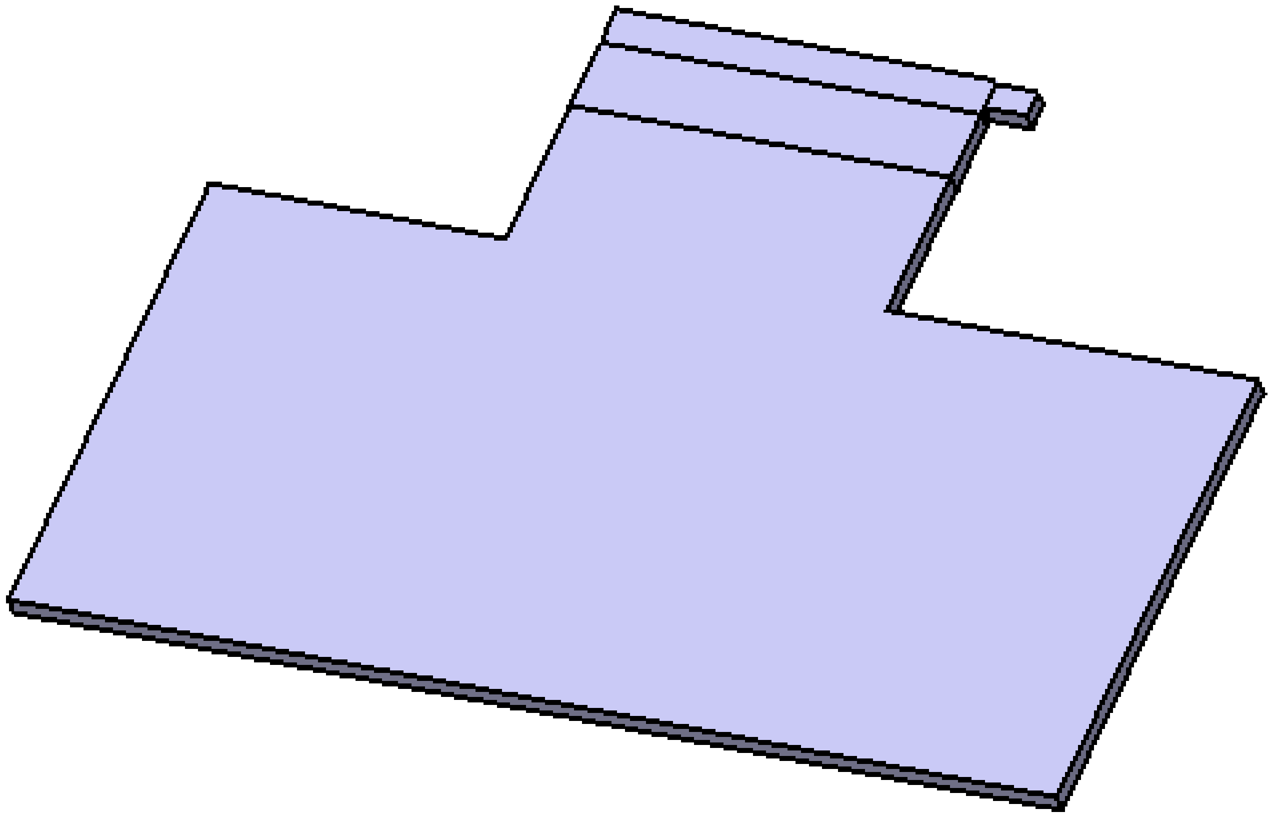
7. Click **OK**.

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

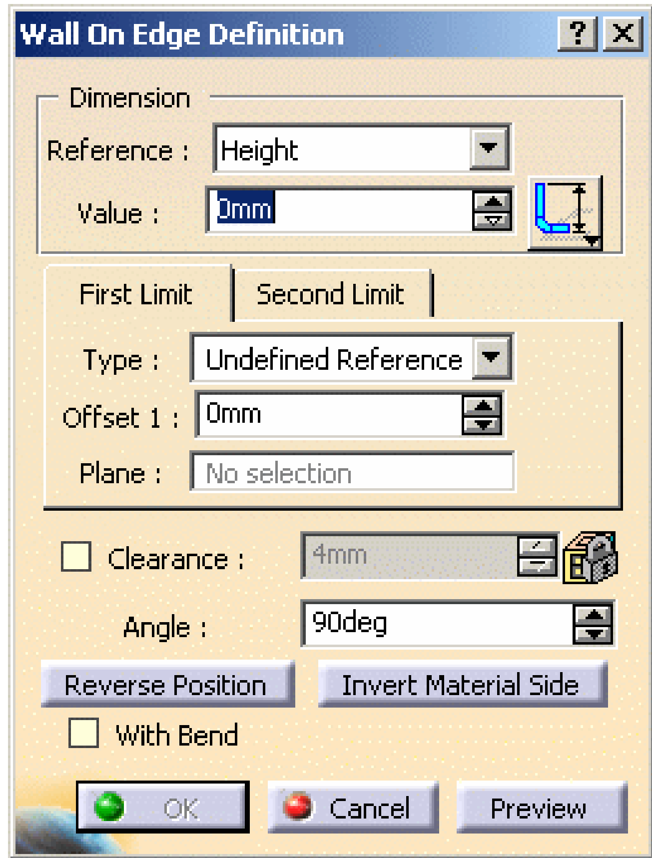
This task shows how to create walls from edges of an existing wall. This function is used to create a box in an easy and quick way from an existing reference wall.

At least one wall must already exist.

Open the NEWWall1.CATPart document from the samples directory.

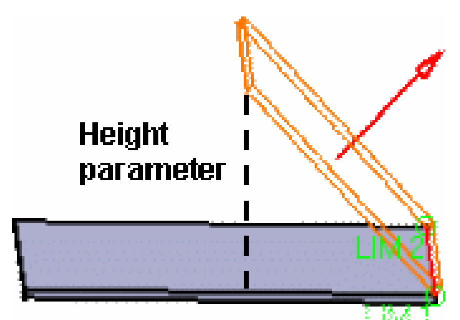
1. Click the **Wall on Edge** icon and select an edge of an existing wall.

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

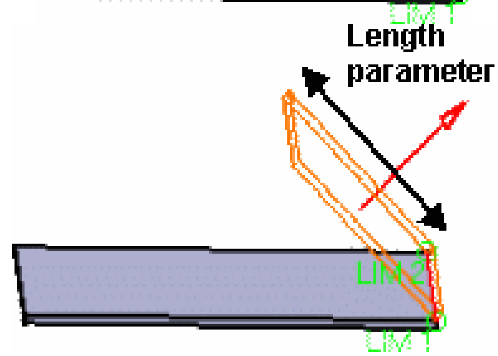


2. Define the type of wall to be created by specifying the Reference, that is:

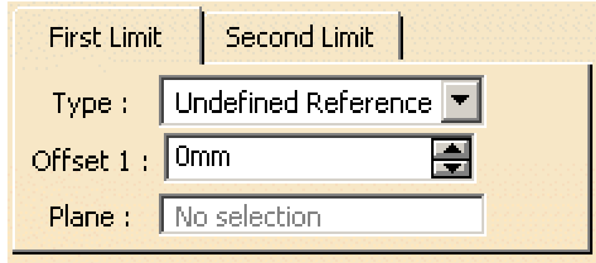
- the **Height** of the wall: that is the orthogonal projection from the top of the wall on edge to the reference wall.
  Select the icon to define the height of the wall from the bottom of the reference wall or the icon to define the height of the wall from the top of the reference wall.



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



3. You can now relimit the wall by selecting planes for the First and Second Limit:



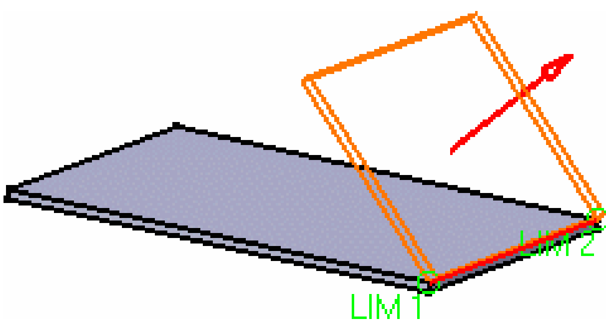
- type:
  - Undefined reference
  - Up To Plane
- Offset
- Plane

As a consequence, the generated wall is not necessarily rectangular.

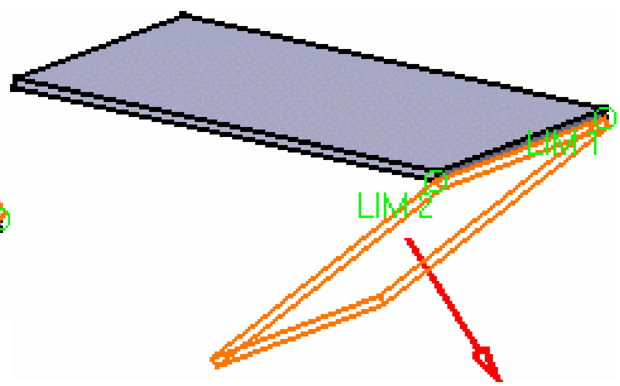
4. Define the angle of the wall: by default it is perpendicular to the plane containing the edge. You may modify it according to your needs. it is updated dynamically on the screen.

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.

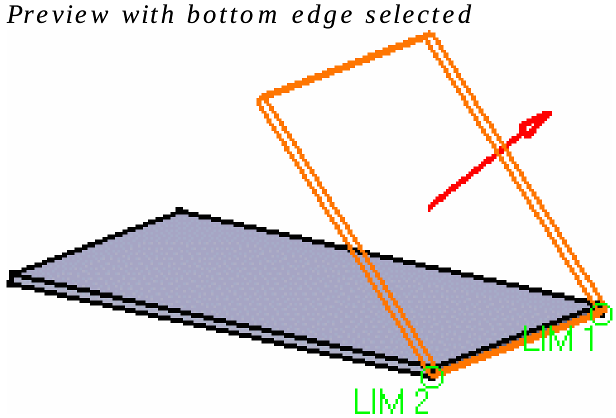


Preview with top edge selected



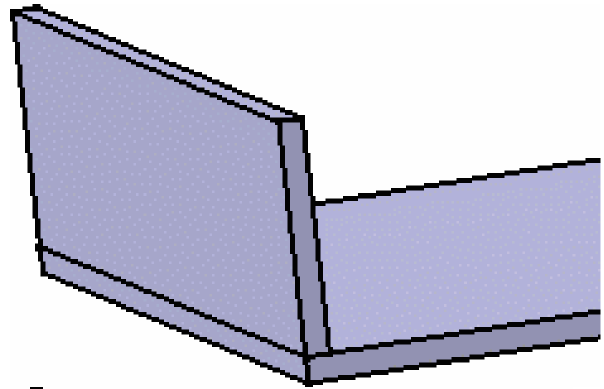
Preview with bottom edge selected

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

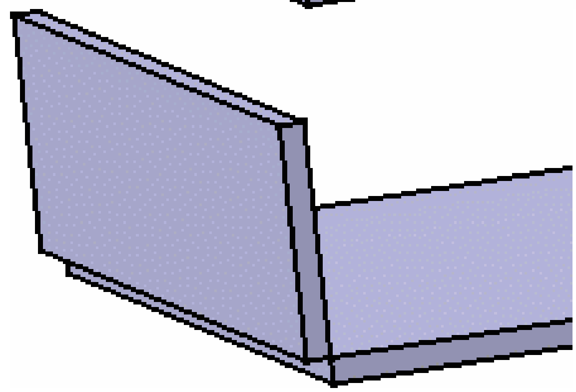


Preview with bottom edge selected and sketch profile inverted

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

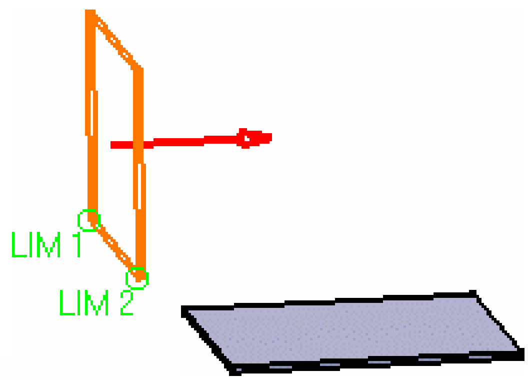


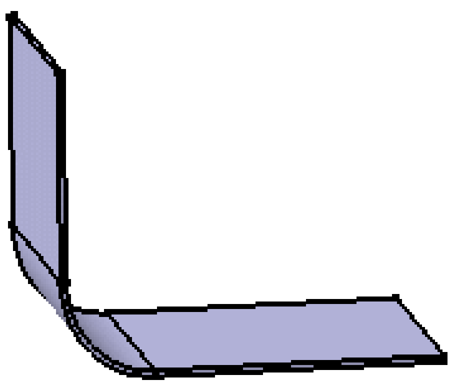
However, you can change it by clicking the red arrow or the **Reverse Side** button.



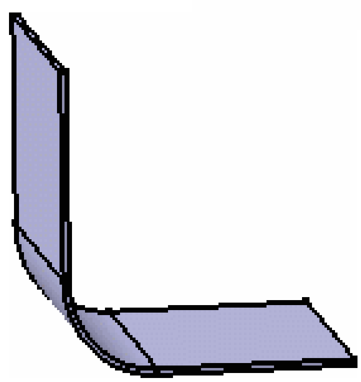
5. Check the **Clearance** option to offset the wall on edge from the selected edge.

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





Wall on edge with clearance



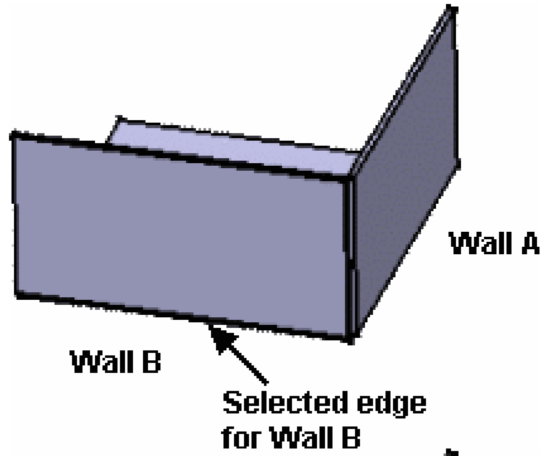
Wall on edge without clearance

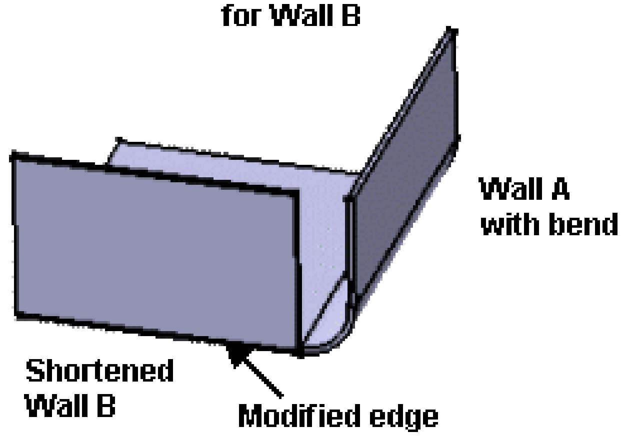
Note that the reference wall remains unchanged when changing the bend radius value. It is the Wall on Edge's length that is affected.

6. 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.

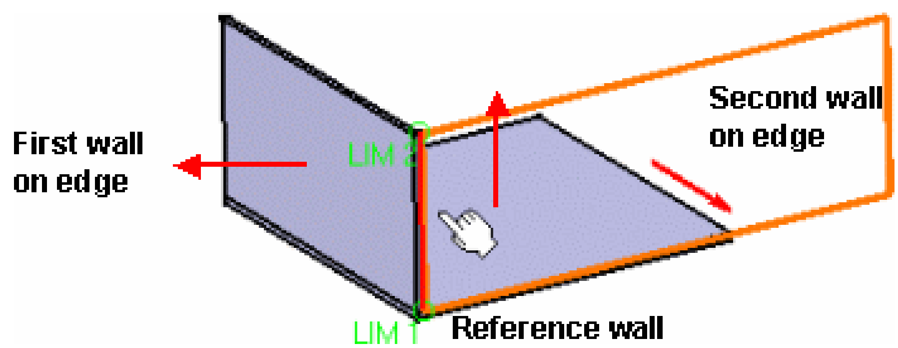
- A wall on edge is defined by the selected edge (reference edge). When the reference edge is modified, by adding any feature that shortens the edge (a bend to an adjacent wall on edge or a cutout for example) the wall on edge based on this reference edge is recomputed.
  To avoid this you may:
  - create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



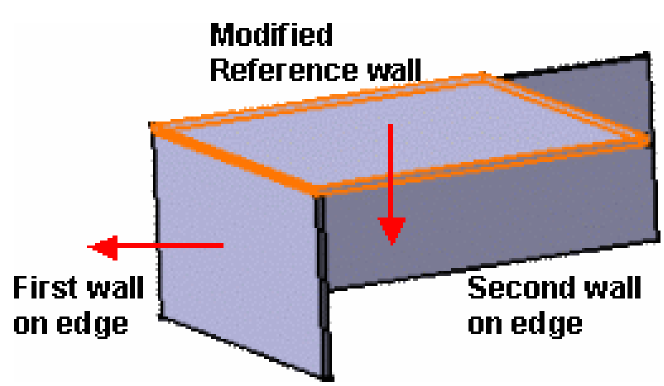


- Both limits are computed with the same reference icon.
- The bend is not previewed, even if the option **With Bend** is checked. However it will be created.
- The selected options are modal and will be proposed to create the next wall.
- Walls on edge being contextual features, if you break the profiles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

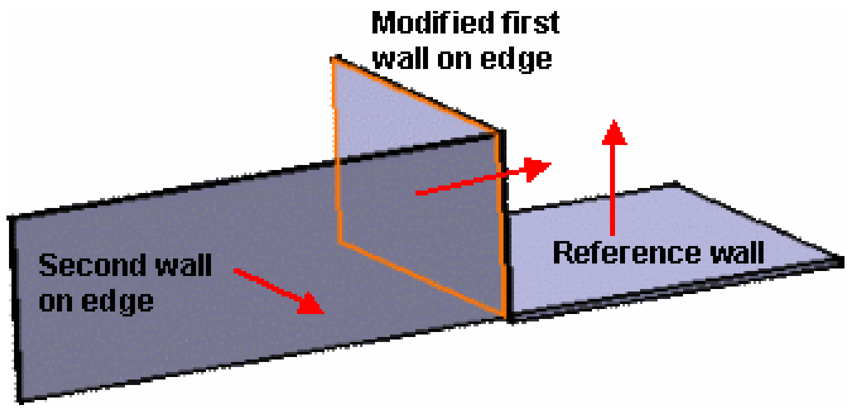
This is the case when creating a wall on edge from another wall on edge, for example:



- 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.



7. Click OK in the Wall On Edge Definition dialog box.

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

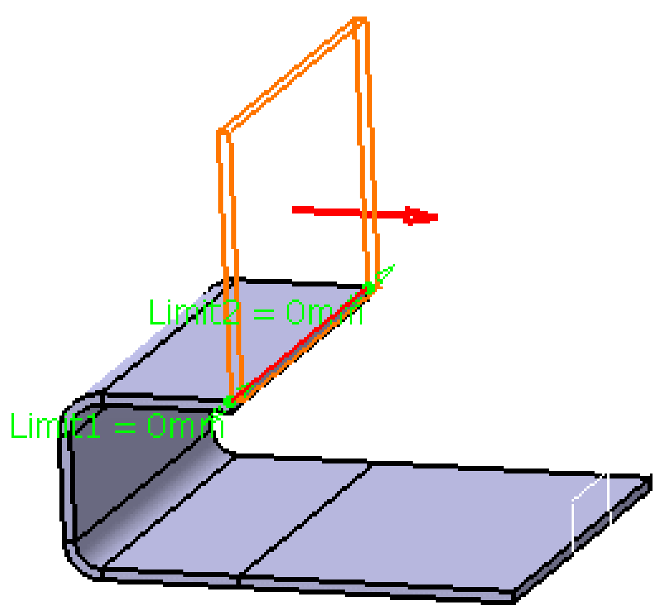
- A WallOnEdge.x element can be edited: double-click it in the graphic area or in the specification tree to display its creation dialog box and modify the parameters described above, including the edge from which it is created. However, the sketch of a wall on edge cannot be edited directly.
- You can 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.

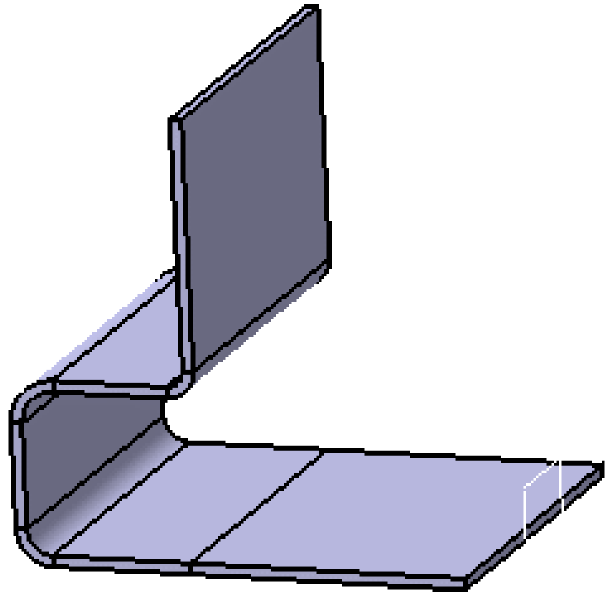
Creating a Wall On Edge on a Flange or a Stamp

Open the NEWWall3.CATPart document.
Let's take an example with a user flange.

1. Click the **Wall on Edge** icon and select the edge of the user flange.

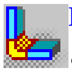


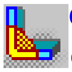
2. Click OK in the Wall on Edge Definition dialog box to create the wall on edge on the user flange.

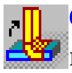


Creating Bends on Walls

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

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

 Create conical bends: select the two walls, set the right and left bend radius values as well as the bend extremities.

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

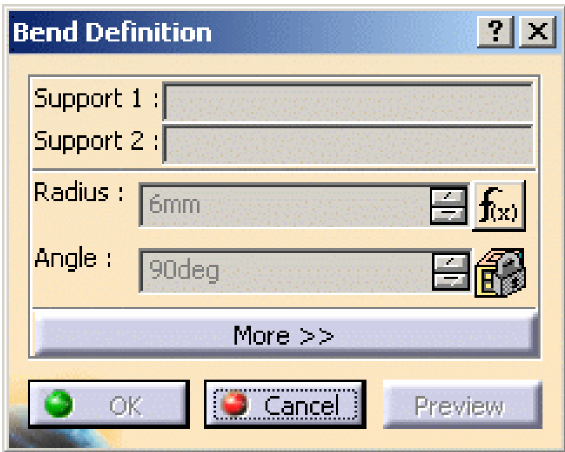
Create local fold and unfold of bends: select the two faces and the fold angle.

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 NEWBendExtremities01.CATPart document.

1. Select the **Bend** icon . The Bend Definition dialog box is displayed.



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 **Support 1** and **Support 2** in the specification tree or in the geometry area. The two supports must be connected by the edge of their internal faces.

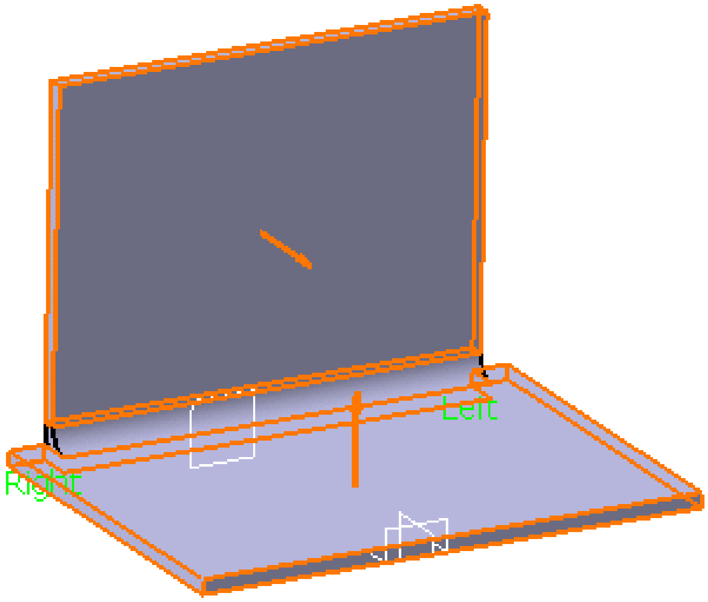
   The Bend Definition dialog box is updated, and arrows are displayed, indicating the walls orientation. When relevant (which is not the case in our example), you can click the arrows to invert the orientation.

   The left and right endpoints for the bend are identified using text.

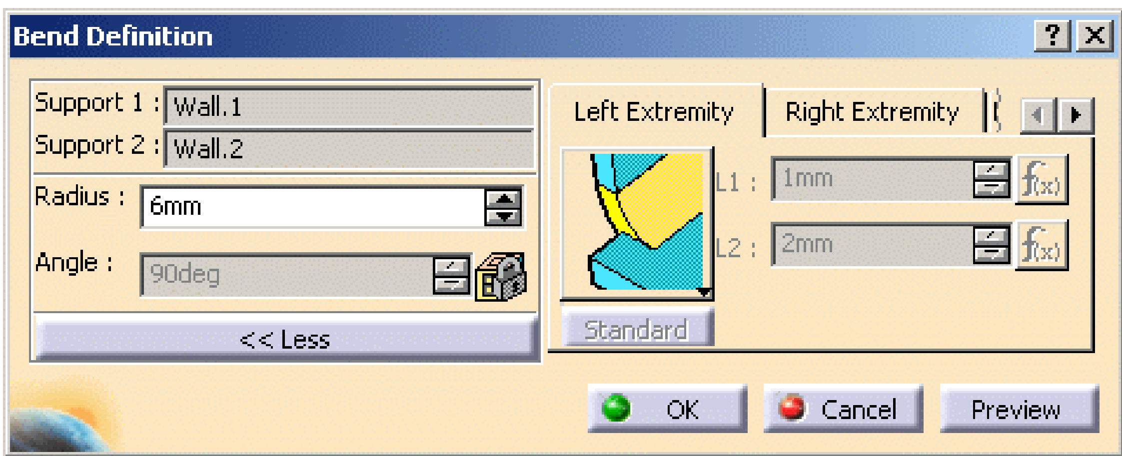
3. Right-click the **Radius** field and select **Formula -> Deactivate** from the contextual menu to change the value.

You can set the **Radius** value to 0.

4. Enter 4mm for the radius and click **Preview**. The bend is previewed, along with its orientation symbolized by arrows.



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

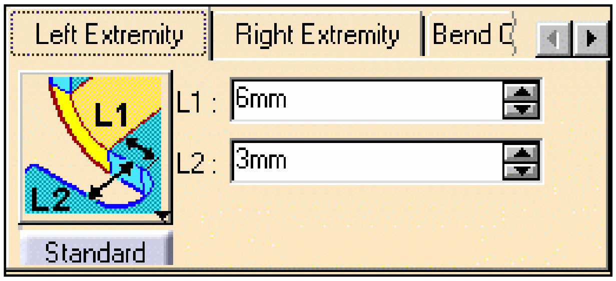


You can define:

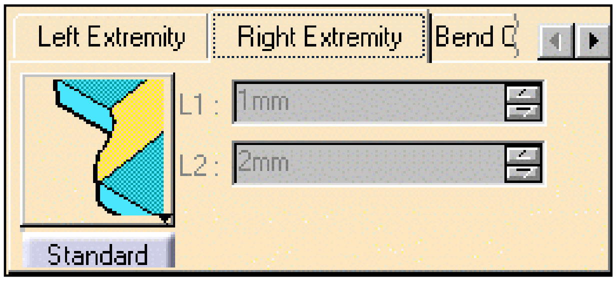
- the left and right extremity settings (see also extremities definition settings)
- the corner relief definition settings
- and the bend allowance settings.

6. Click the **Left Extremity** tab, and choose the **Mini with round relief** bend extremity type.

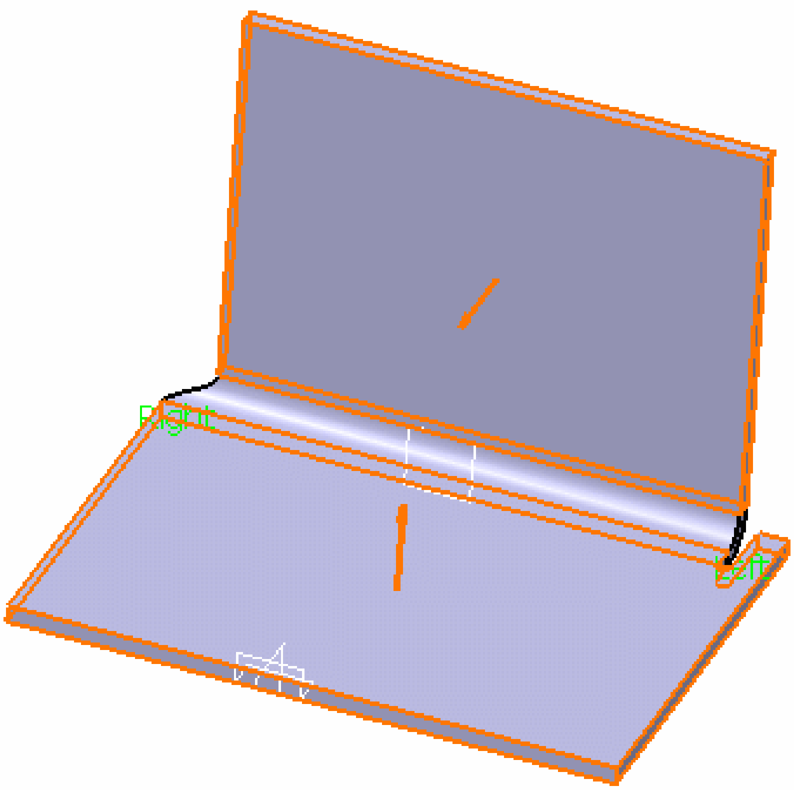
7. Then, deactivate the L1 and L2 length formulas (as previously explained in step 3), and set them to 6mm and 3mm respectively.



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

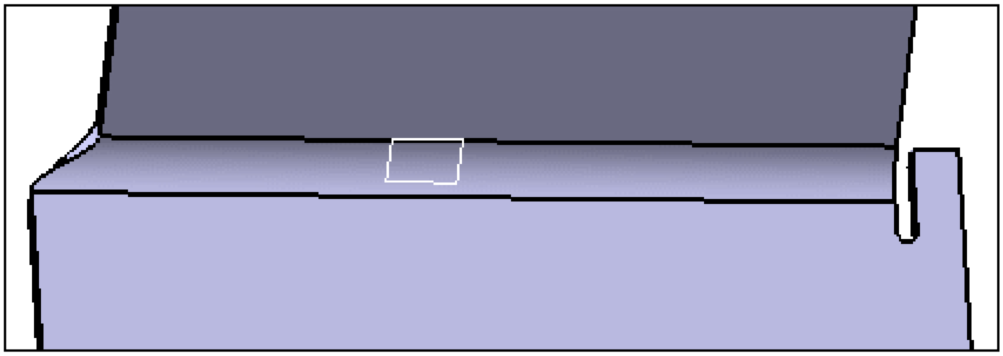


9. Click **Preview** to visualize the left and right extremities.

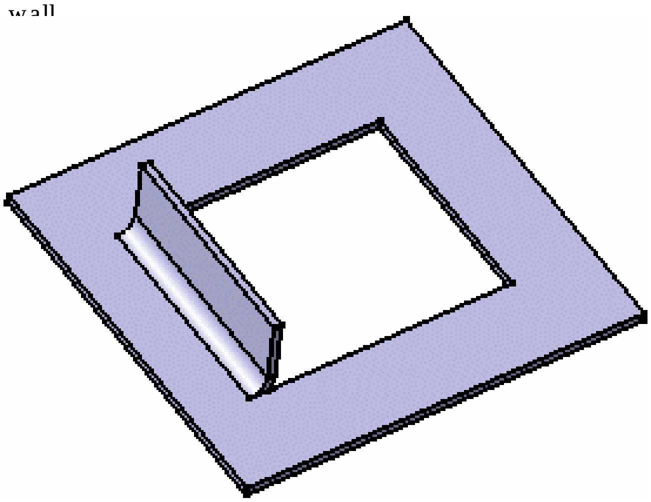


10. Click **OK** in the Bend Definition dialog box.

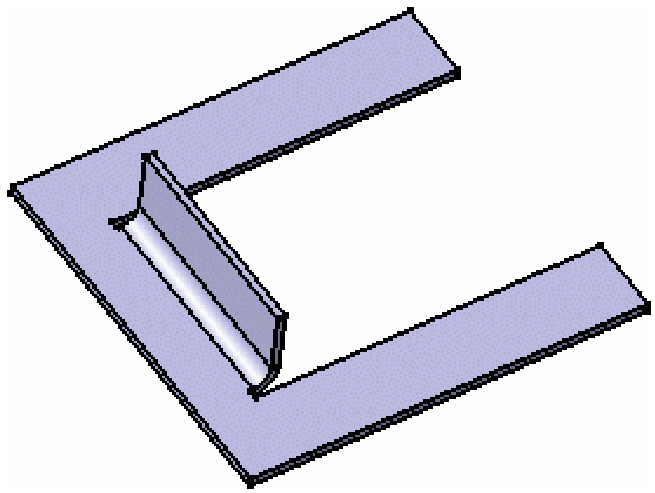
The bend (identified as Cylindrical Bend) is created with the specified extremity types and is added to the specification tree.



- 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

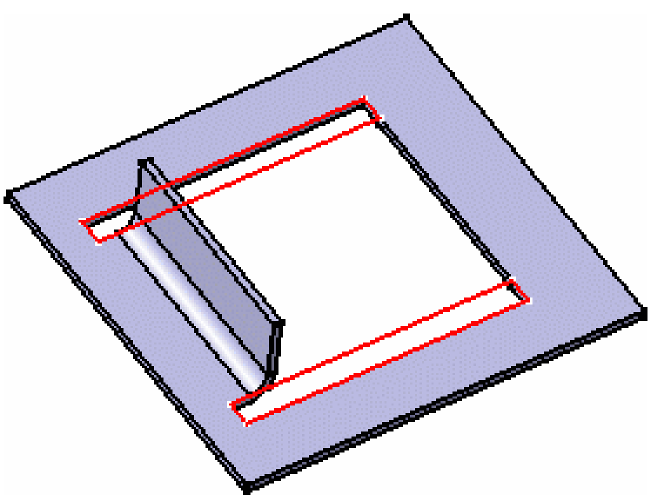


Bend with no relief



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.



Creating a Bend on a Flange or a Stamp

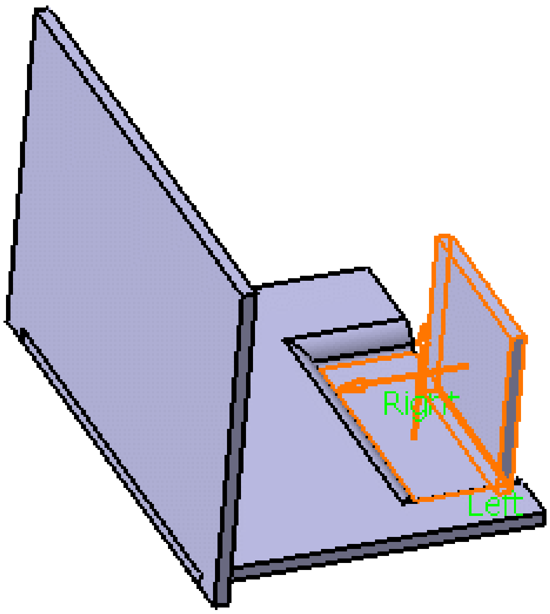
Let's take an example with a surface stamp.

Open the NEWBendExtremities02.CATPart document.

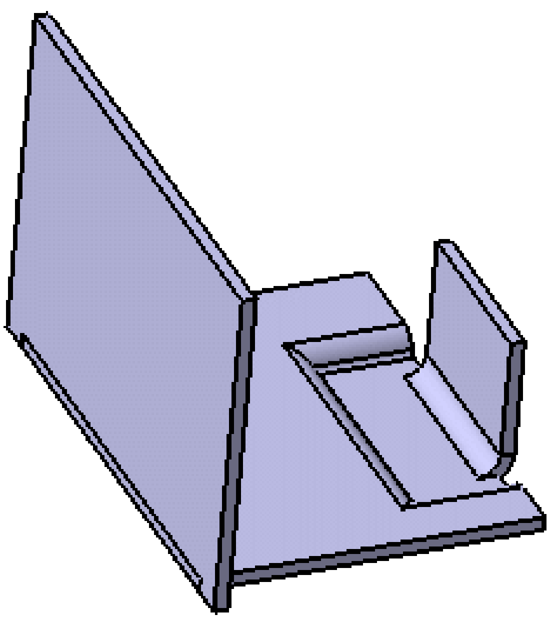
1. Select the **Bend** icon. The Bend Definition dialog box is displayed.

2. Select the surface stamp's face as **Support 1**.

3. Select Wall.3 in the specification tree as **Support 2**.



4. Click **OK** in the Bend Definition dialog box to create the bend on the surface stamp.



To deselect a wall, simply click on another wall available in the part.

The stamp must be open, contain a planar face, and have a radius higher than 0.

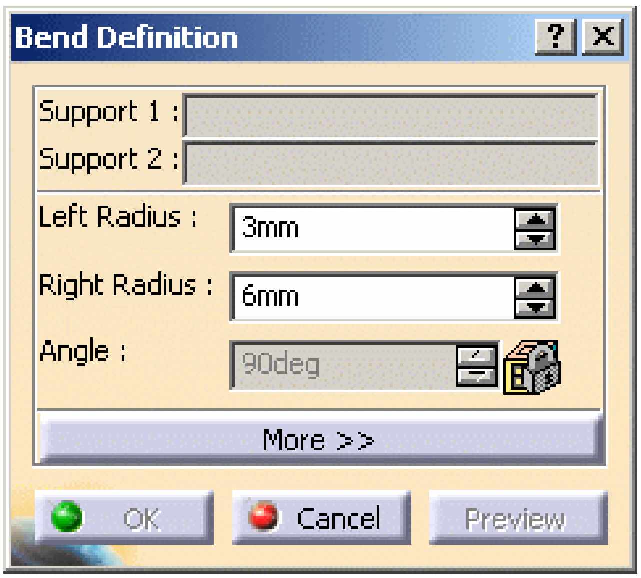
Creating Conical Bends

This task explains how to create conical bends between two walls in a Sheet Metal part.

Conical bends are different from the standard bend in that they allow different radius values at each end of the bend.

Open the NEWBendExtremities01.CATPart document.

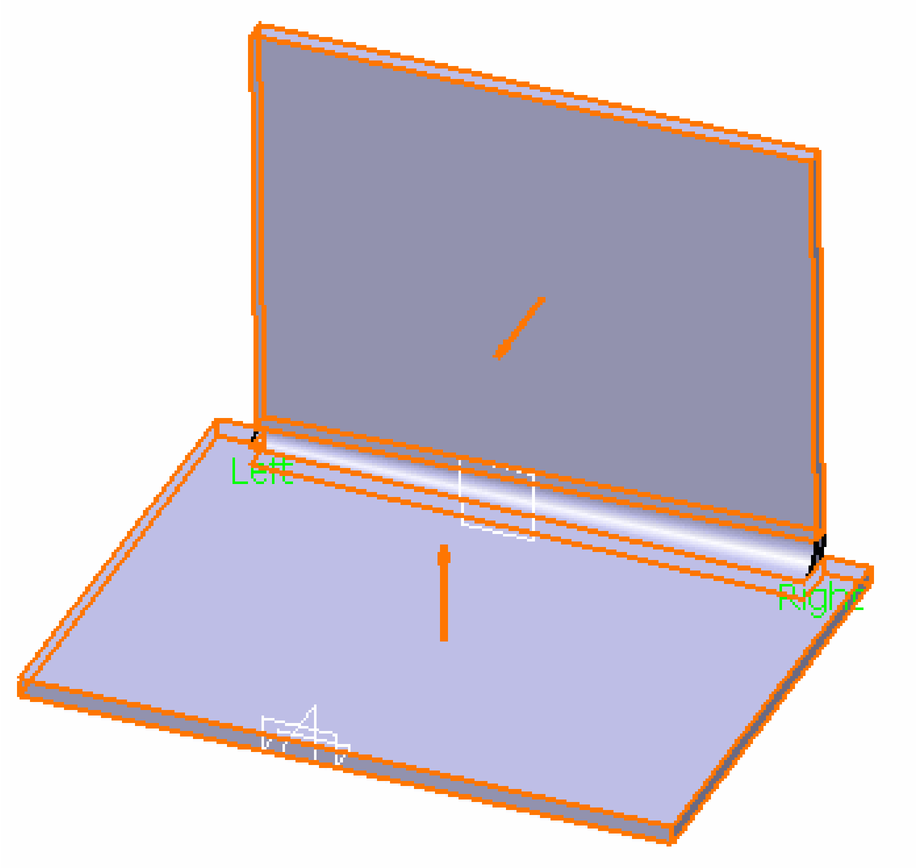
1. Select the **Conical Bend** icon . The Bend Definition dialog box appears.



2. Select **Support 1** and **Support 2** in the specification tree or in the geometry area. The two supports must be connected by the edge of their internal faces.

   The Bend Definition dialog box is updated, and arrows are displayed, indicating the walls orientation. When relevant (which is not the case in our example), you can click the arrows to invert the orientation.

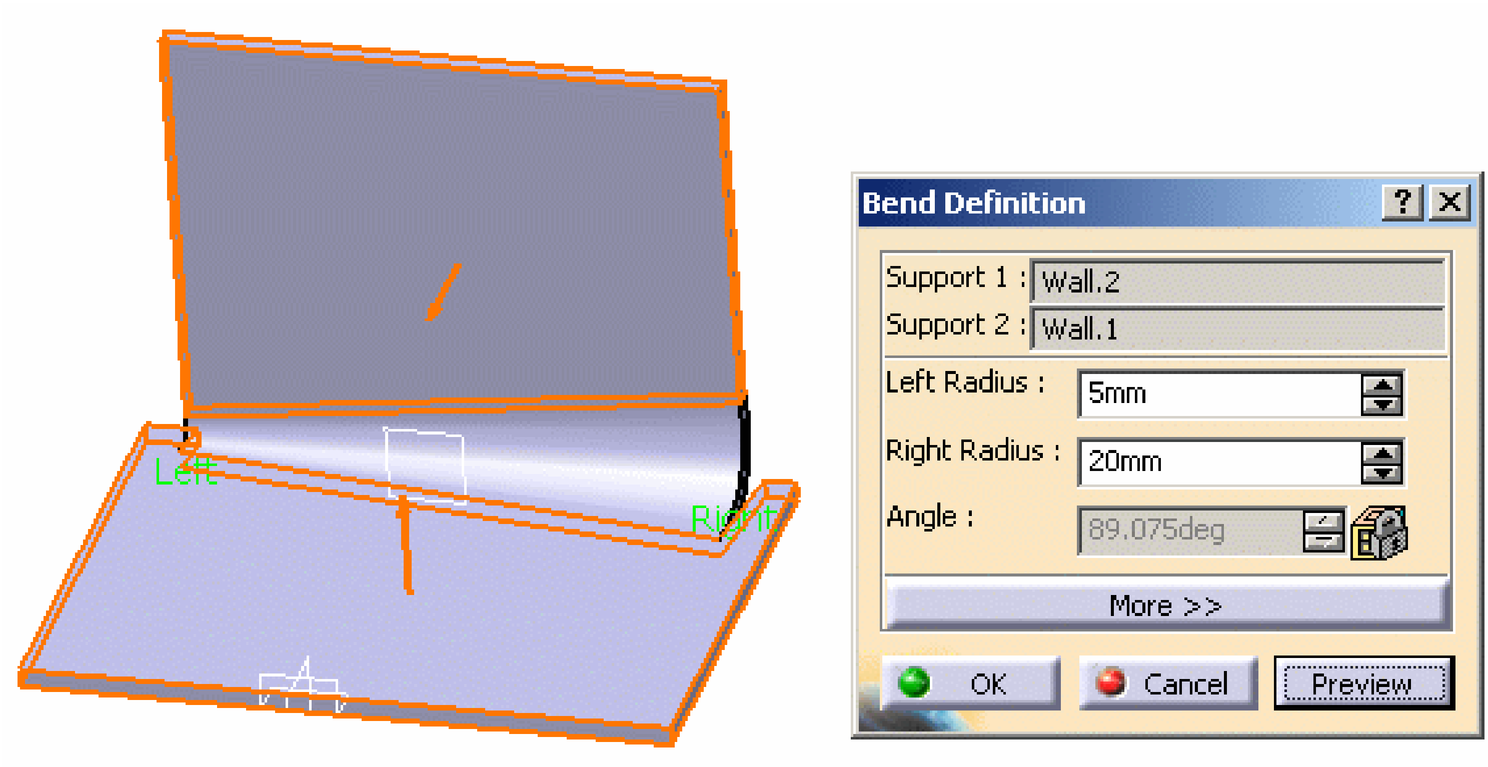
   The left and right endpoints for the bend are identified using text.



3. Enter the radius values for each end of the conical bend. For example, enter 5mm as the left radius and 20mm as the right radius.

- By default, **Right Radius** amounts to twice **Left Radius**.
- Note that the radius is computed based on the left and right endpoints for the bend. This radius, once computed, will be larger than the specified value.
- When the size of each support differs, the bend may exceed the top of the cone. In such a case, computing the radius value is impossible.

The **Angle** field is locked. If you then click the **Preview** button, it is updated, indicating the angle value between the two walls between which the bend is computed.

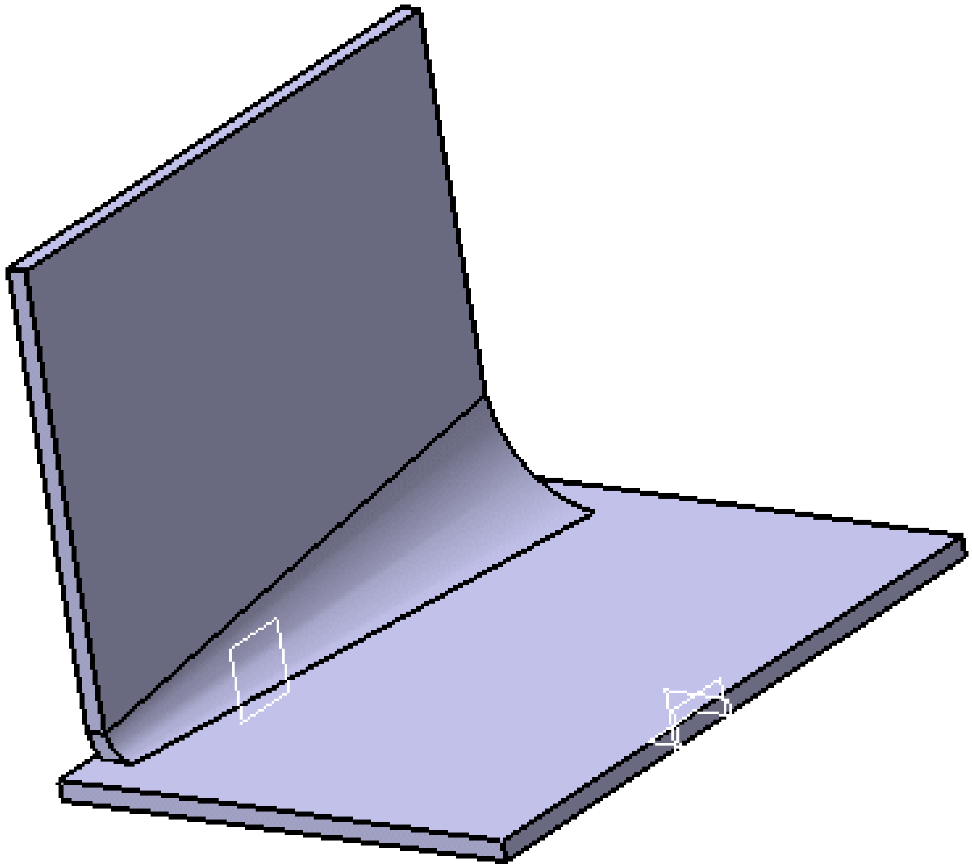


4. Click **More...** to display additional options.

5. Optionally, on the **Left Extremity** and **Right Extremity** tabs, click the graphical button to choose the bend extremities:

- **Mini with no relief** (default option): the bend corresponds to the common area of the supporting walls along the bend axis, and shows no relief.
- **Mini with 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.
- **Mini with 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 shape**: 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).
- **Curve shaped**: the edges of the bend are tangent to the edges of the supporting walls.
- **Maximum bend**: the bend is calculated between the furthest opposite edges of the supporting walls.

6. Optionally, on the **Bend Allowance** tab, define the K factor (which performs calculations related to folding/unfolding operations). Refer to Computing the Bend Allowance for complete information.
7. Click **OK**. The conical bend is created.

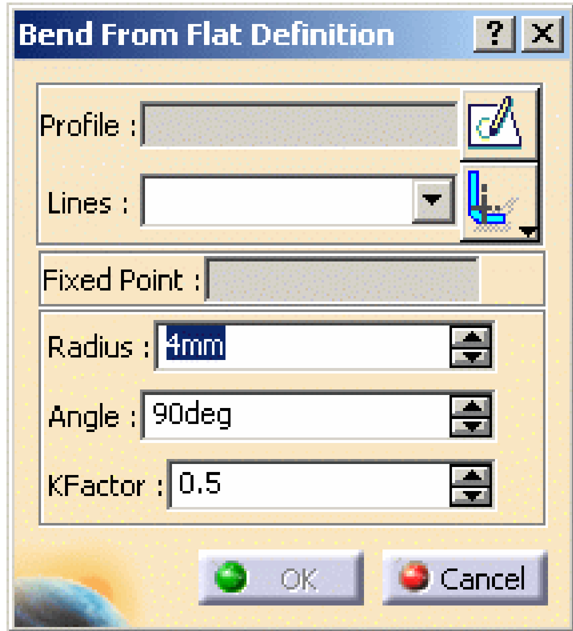


Creating Bends From a Line

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

Open the NEWFlatBend1.CATPart document.

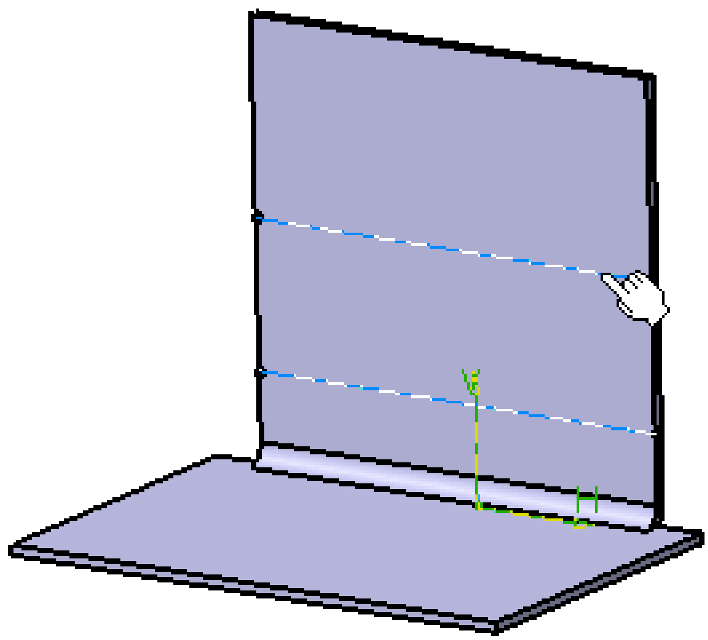
1. Select the reference wall then the **Bend From Flat** icon .



The Bend From Flat Definition dialog box opens.

2. Select a profile (Sketch.2 here) containing one or several lines.

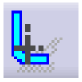
   This sketch must necessarily be a line.

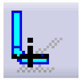


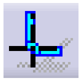
Selected lines appear in the Lines combo list.

The line(s) must not intersect an area where a 3D feature (such as a longitudinal chamfer, or a stamp created from punch and die) lies.

3. You can choose the line extrapolation option:

 Axis

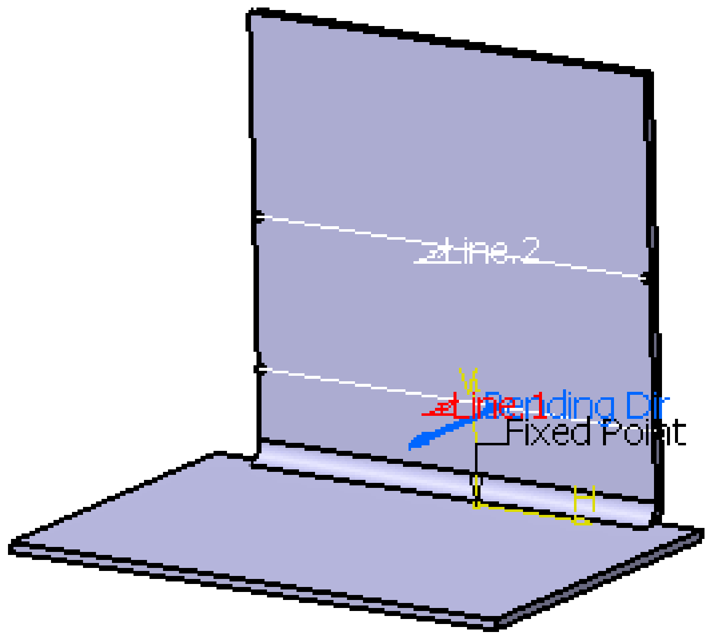
 BTL (Bent Tangent Line): line corresponding to the limits of the bend's fillet

 IML (Inner Mold Line): line created by intersecting the internal surfaces of the bend (before filleting) and the wall

 OML (Outer Mold Line): line created by intersecting the bend support and a plane perpendicular to the wall and normal to the OML.

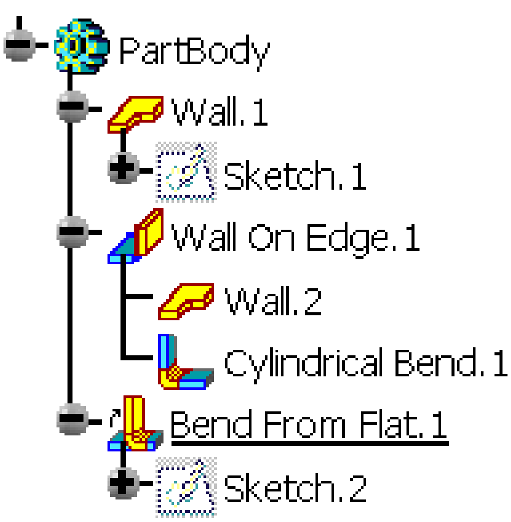
The Radius and the KFactor values are the one defined when editing the sheetmetal parameters: Right-click the **Radius** or the **KFactor** field and select **Formula -> Deactivate** from the contextual menu to change the value.

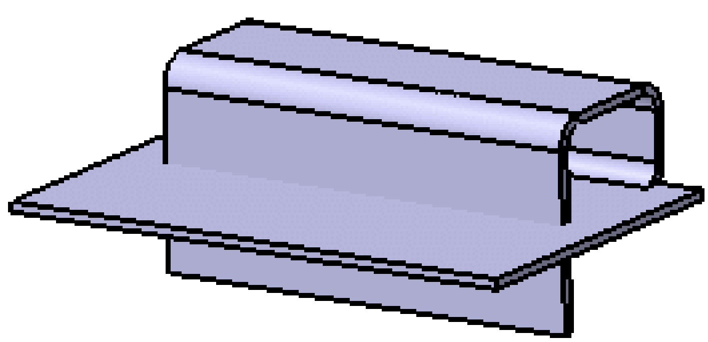
You can set the **Radius** value to 0.



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

Creating Local Fold and Unfold

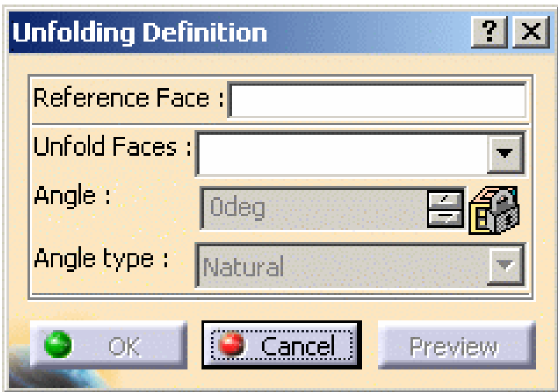
This task explains how to fold or unfold bends in the Sheet Metal part.

Local Fold and Unfold applies to cylindrical faces such as flange, bend and surfaces recognized as bend.

In our example, we chose local fold and unfold of bends.

Open the NEWFoldUnfoldBends.CATPart document.

1. Select the **Unfolding** icon. The **Unfolding Definition** dialog box is displayed.

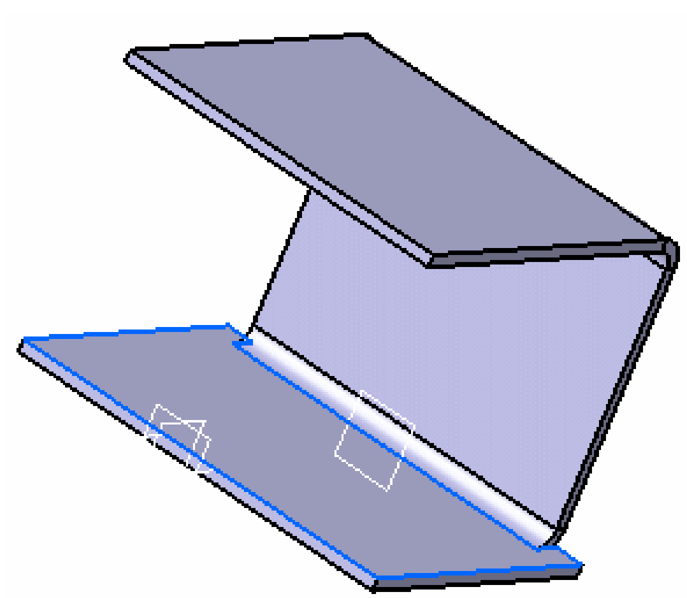


Unfold

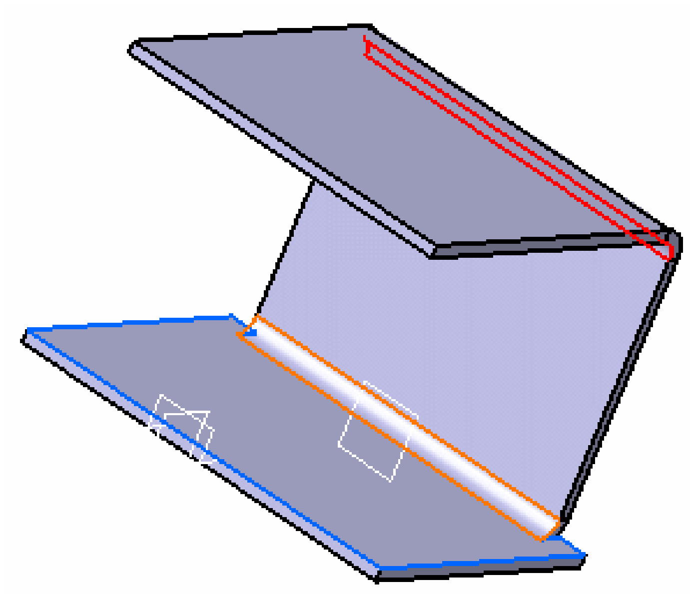
1. Select Wall.1 as reference face on the geometry.

   Angle type is disabled and set to Natural.

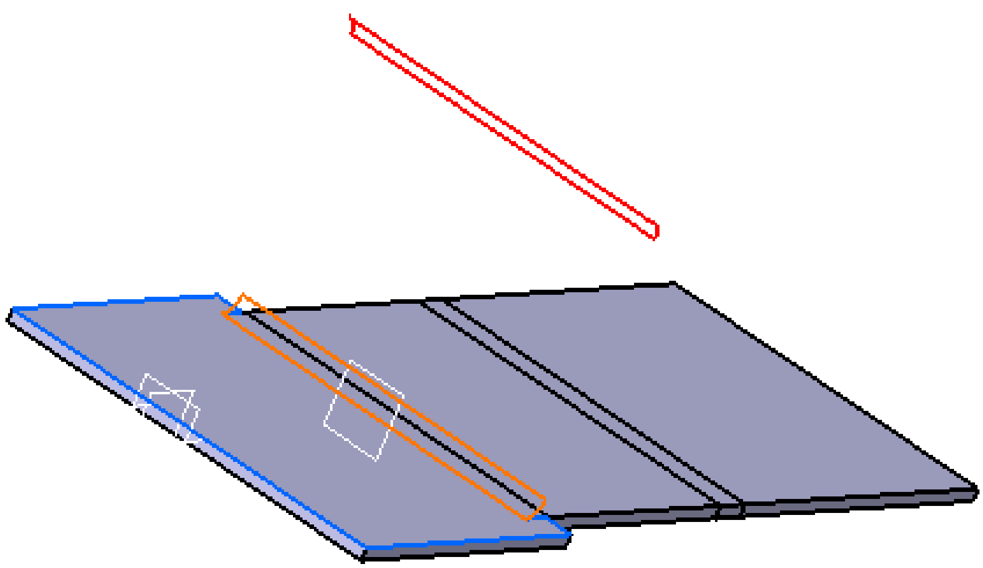
   Angle is at 90 degrees.



2. Select Cylindrical Bend.1 and Cylindrical Bend.2



3. Click on **Preview** to visualize the unfolding of the bends you selected.



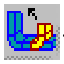
4. Click on **OK** to unfold the bends.

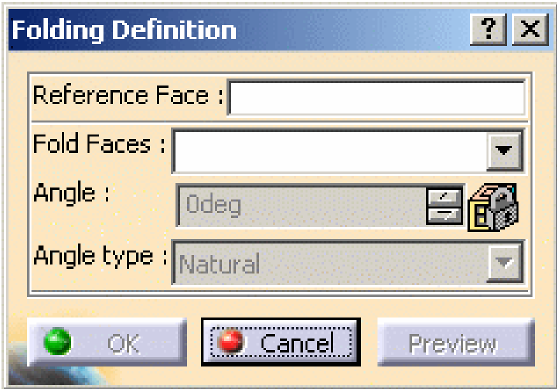
   The Unfolding.1 feature is created in the specification tree.

The following options are available in the **Unfolding Definition** dialog box:

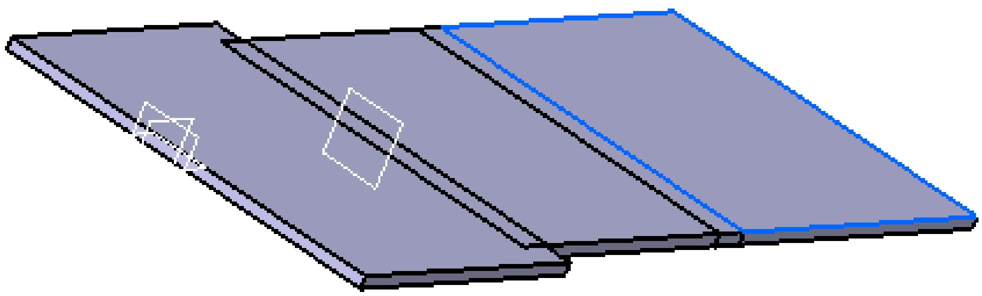
- Reference face: fixed part of the body
- Unfold Faces: bend(s) to unfold
- Angle: disabled, corresponds to the angle of the selected face to fold that is 90 degrees by defaut
- Angle type: disabled, set to natural by default

Fold

1. Select the **Folding** icon . The **Folding Definition** dialog box is displayed.

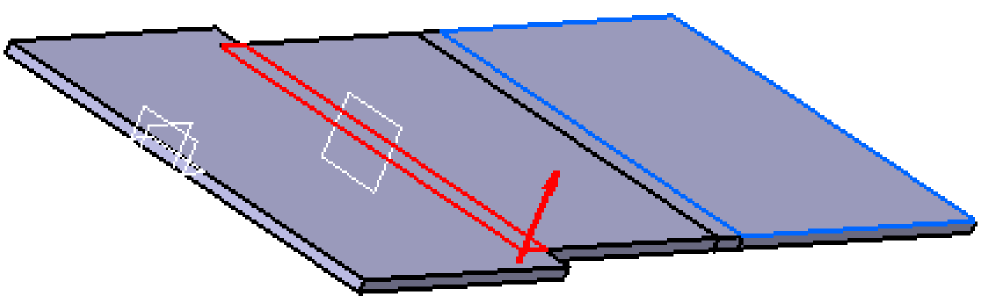


2. Select Wall.3 as reference face.



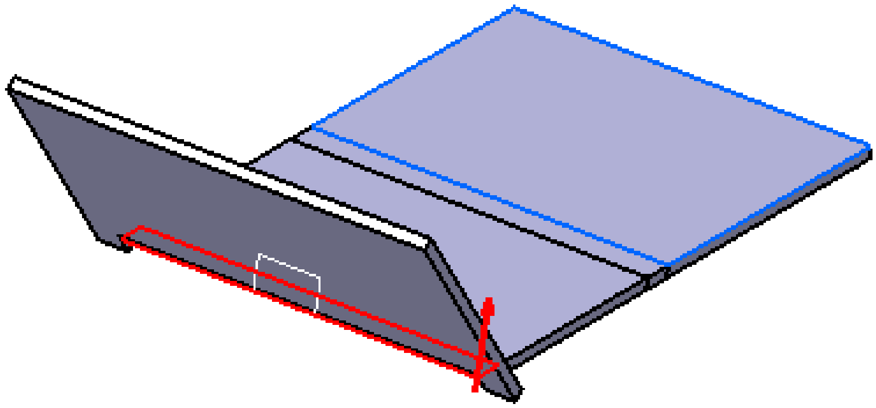
3. Select Cylindrical Bend.1

   A red arrow is displayed indicating the direction of the angle.



4. In the **Folding Definition** dialog box, select **Defined** as angle type and 45 degrees as angle.

5. Click on **Preview** to visualize the folding of the bend you selected.

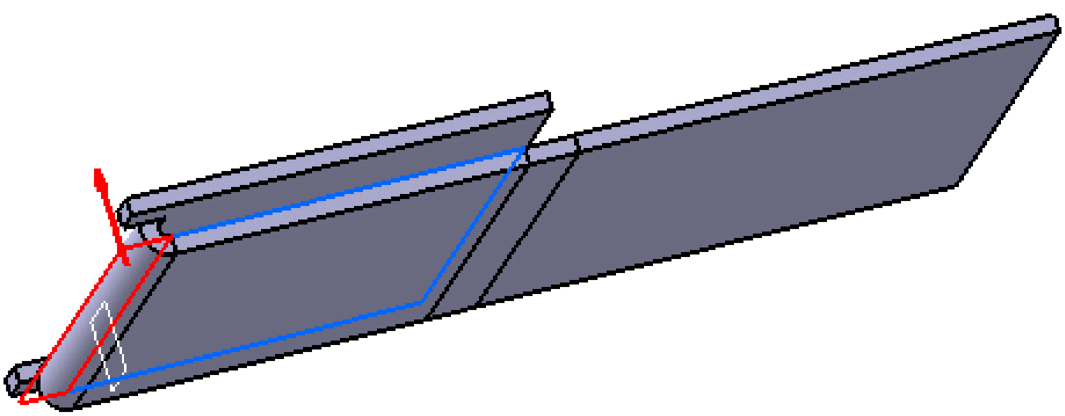


6. Click on **OK** to fold the selected bend.

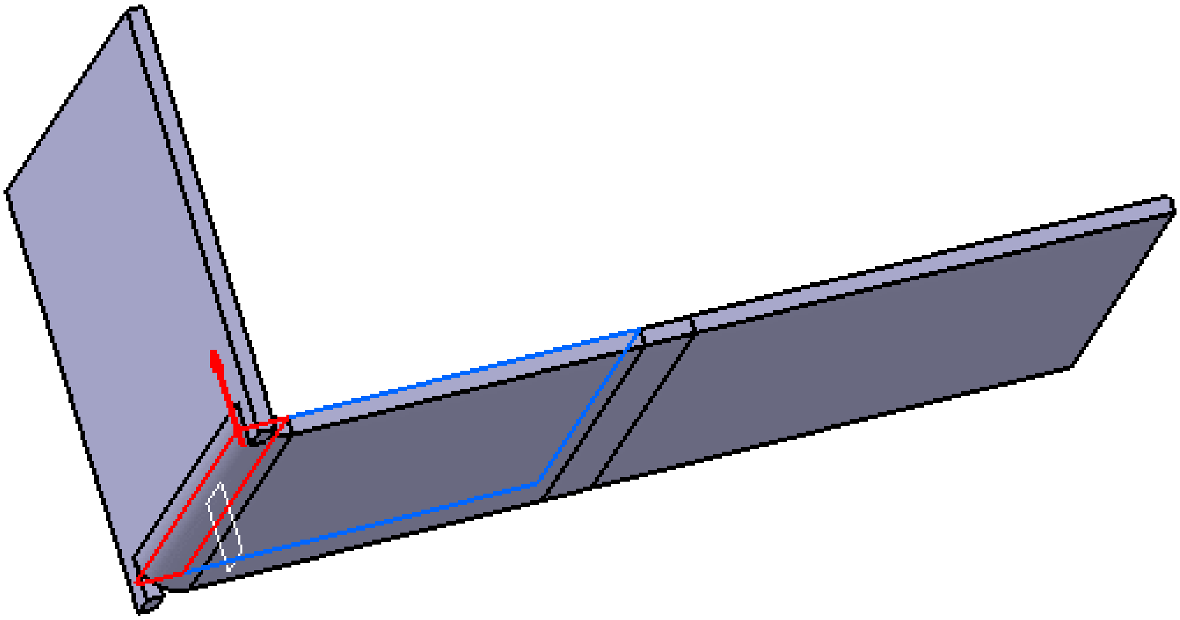
   The Folding.1 feature is created in the specification tree.

The following options are available in the **Folding Definition** dialog box:

- Reference face: fixed part of the body
- Fold faces: bend(s) to fold
- Angle: corresponds to the angle of the selected face to unfold
- Angle type
  - Natural: when selected, the angle is set to 90 degrees by default and is disabled
  - Defined: allows you to choose the bend angle with the up and down arrows
  - Spring back: the angle of the bend is based on the natural angle with additional degrees.

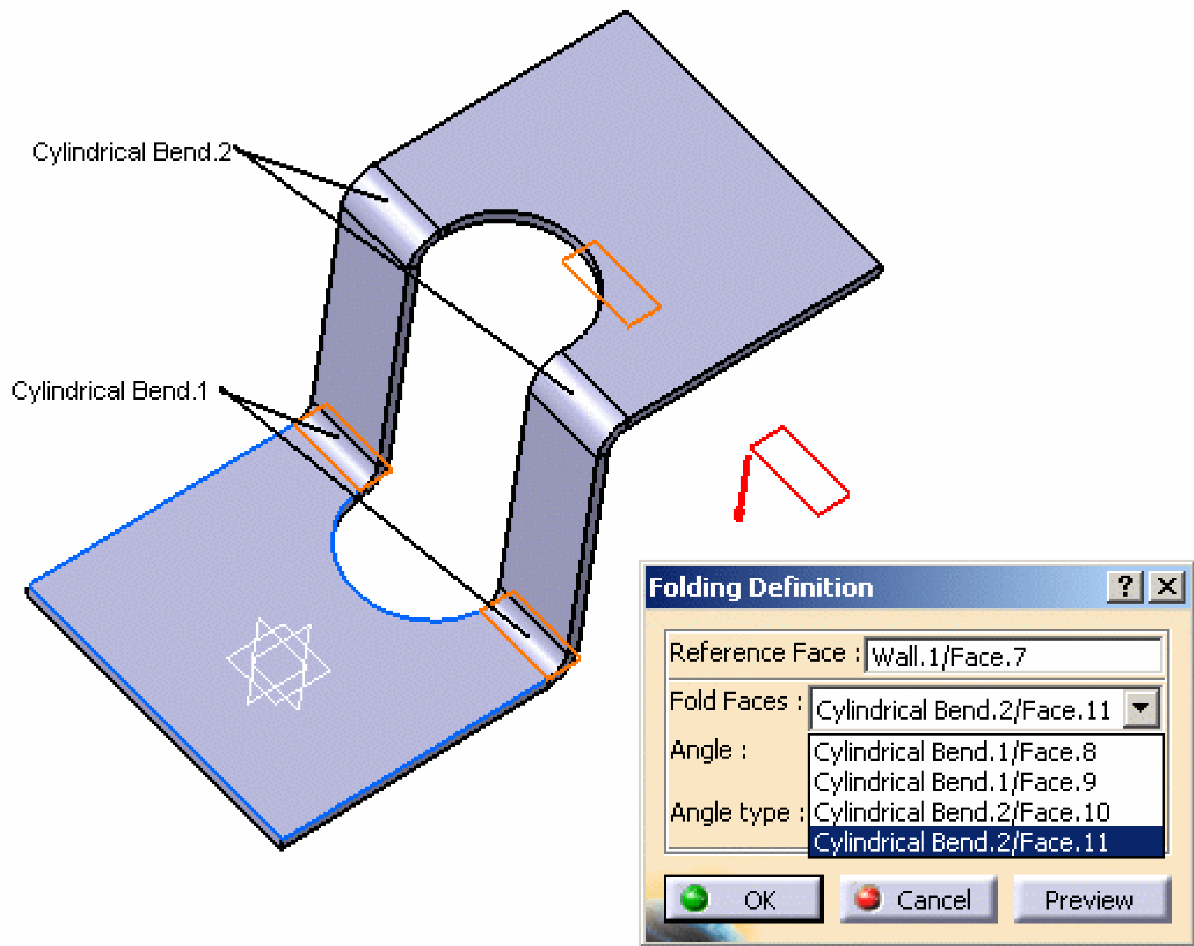


Bend folded with a Spring back angle at 90 degrees.



Bend folded with a natural angle at 90 degrees.

To fold or unfold a part with a bending cycle, each element of the cylindrical bend has to be selected individually.



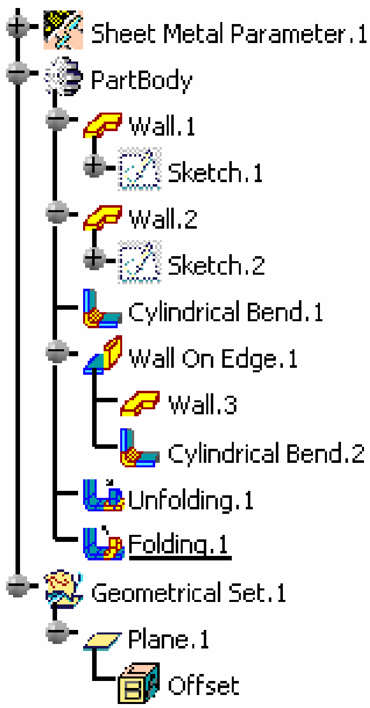
- Multiselection is possible but since unfold faces must be in the same domain you must fold/unfold the parts one after the other. You cannot create folded faces with null radius.

- The fold/unfold feature is only available in folded view, as the flattened view is not modified.

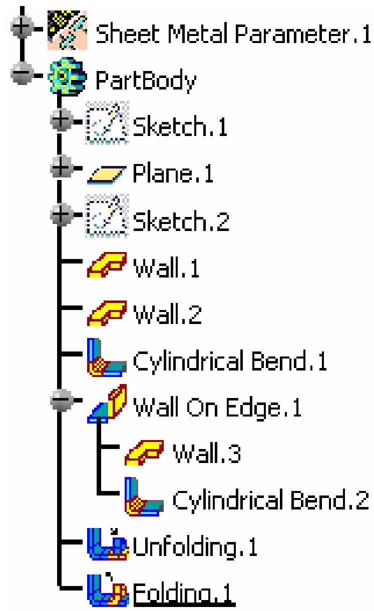
You cannot fold/unfold bends:

- if faces to fold are not in the same domain,
- if faces to unfold have a null radius,
- if cylindrical faces are generated from stamps or rolled walls.

In hybrid context, even though a wall is created with one or several features, none are aggregated under the wall in the specification tree.



Specification tree's behavior in pre-hybrid context.



Specification tree's behavior in hybrid context.

Yet, if you open a part created in a previous release, the specification tree will be displayed accordingly to the previous behavior.
Fore more information about Hybrid Design, refer to the Hybrid Design section.

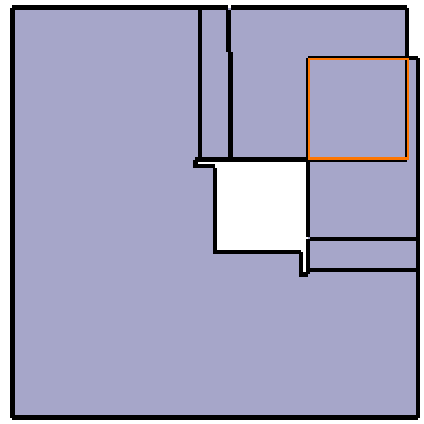
Checking Overlapping

This task shows how check the overlapping of different areas of the part. This is usually done when the part has been designed, bends created and the part is unfolded.
Overlapping between areas of the part are highlighted.

Open the NewOverlap01.CATPart document from the samples directory.

1. Click the **Unfold** icon to unfold the part.
2. Click the **Check Overlapping** icon .

The area with the overlapping is highlighted in red.

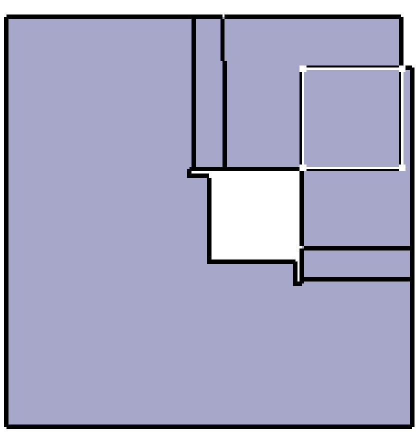


The Overlapping Detection dialog box is displayed indicating how many overlaps are present on the analyzed part.

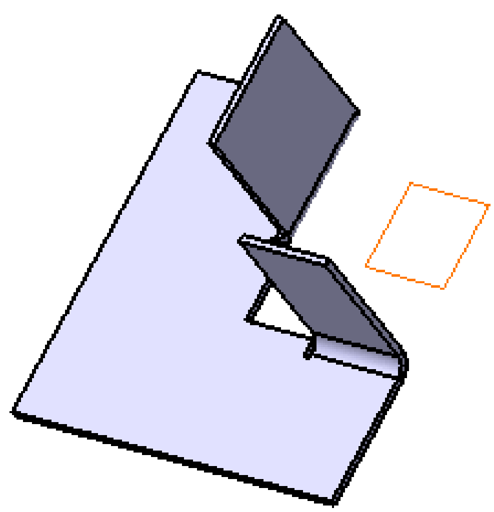
3. Click **OK**.

   A curve is generated. This curve can then be used to solve the overlapping, by redesigning the involved feature's sketches or specifications.

4. Click **OK**.



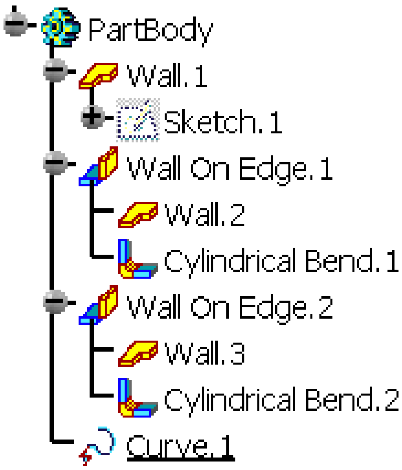
Generated curve on unfolded view



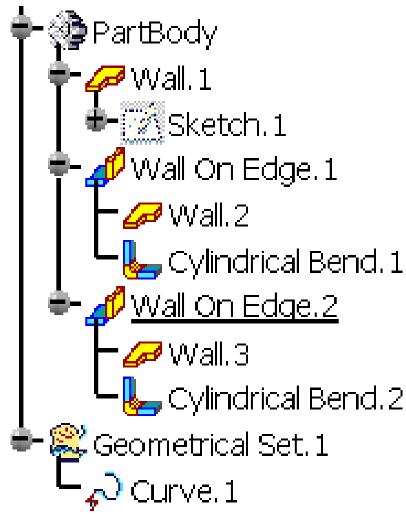
Generated curve on folded view

When no overlapping is detected, the Overlapping Detection dialog box indicates so:

In hybrid context, the curve generated after an overlapping detection is aggregated under the part body in the specification tree.



Specification tree's behavior in hybrid context.



Specification tree's behavior in pre-hybrid context.

Creating Extrusions

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

Extrude: choose the extrusion type, specify the length and choose additional options.

Create Rolled Walls: select a circular sketch, specify the rolled wall type and choose additional options.

Extruding

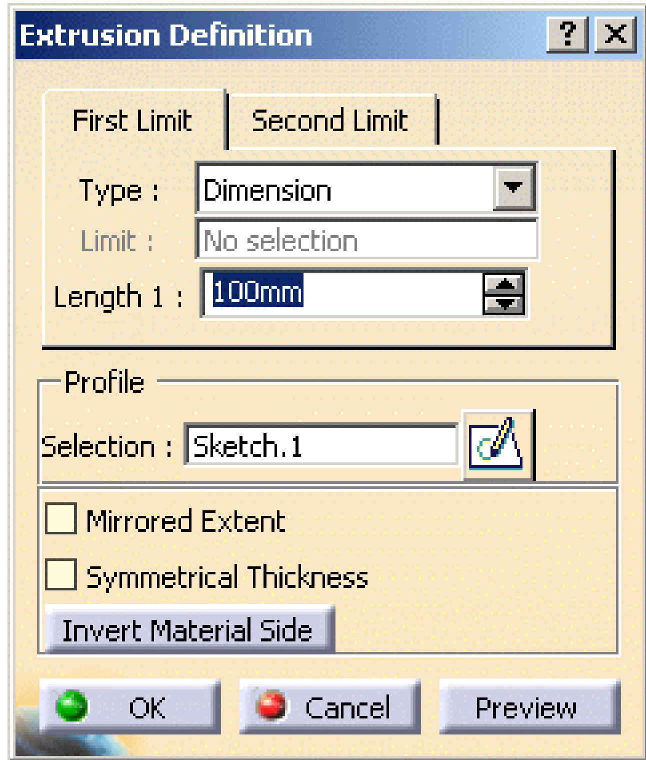
This task explains how to create a wall by extrusion.
You can extrude sketches composed of lines, circles, projection of lines, and projection of circles.

You must have defined the Sheet Metal parameters.
A model is available in the NEWExtrude1.CATPart from the samples directory.

1. Click the **Extrusion** icon.

2. Select a sketch.

The Extrusion Definition dialog box is displayed.

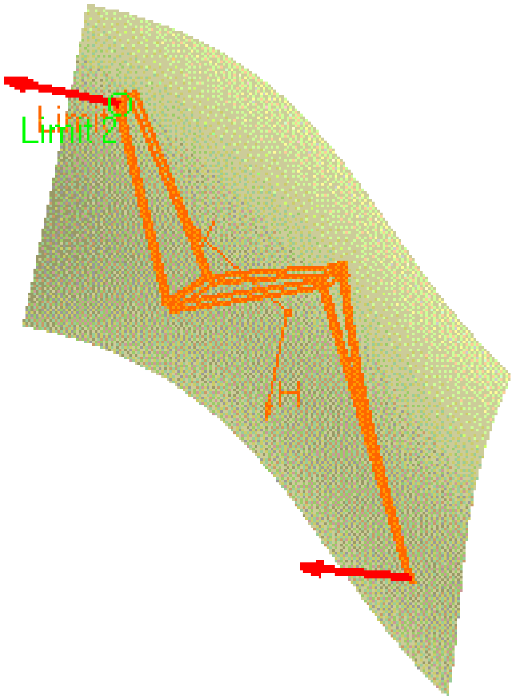


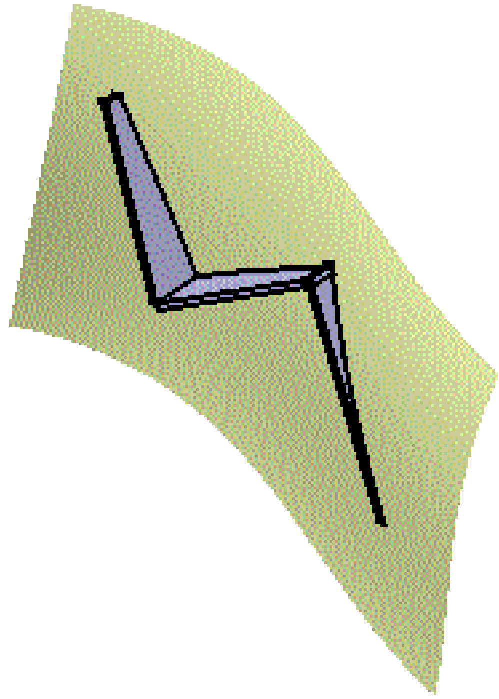
Several types of extrusion are available:

- **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**.

By default, the **Length1** value is positive.



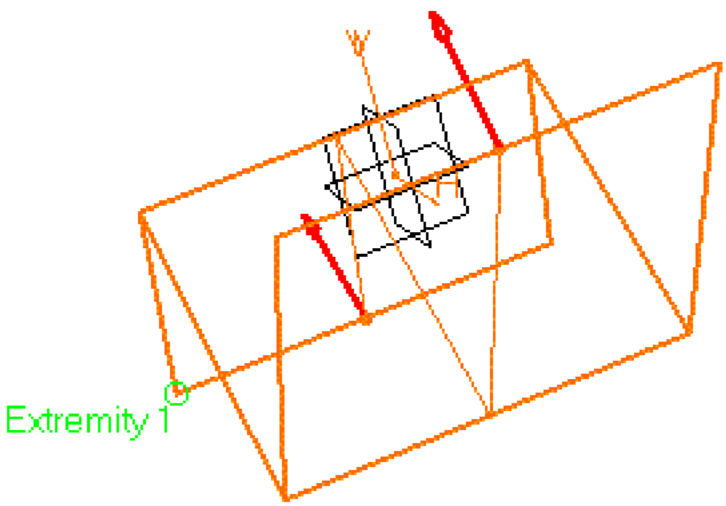


The sketch you selected appears in the **Selection** field

You can now edit it by clicking the **Sketcher** icon  if you wish to modify it.

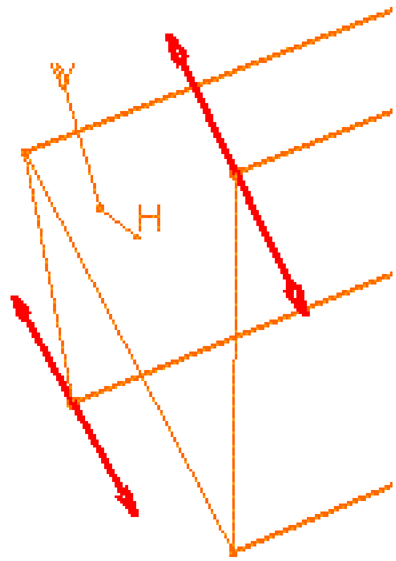
4. Define the options as needed:

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

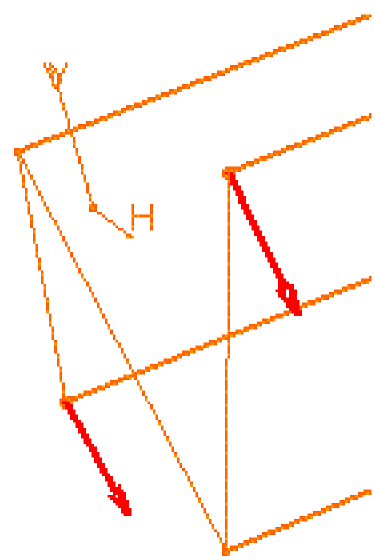


This option is only available if the type is set to **Dimension**.

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



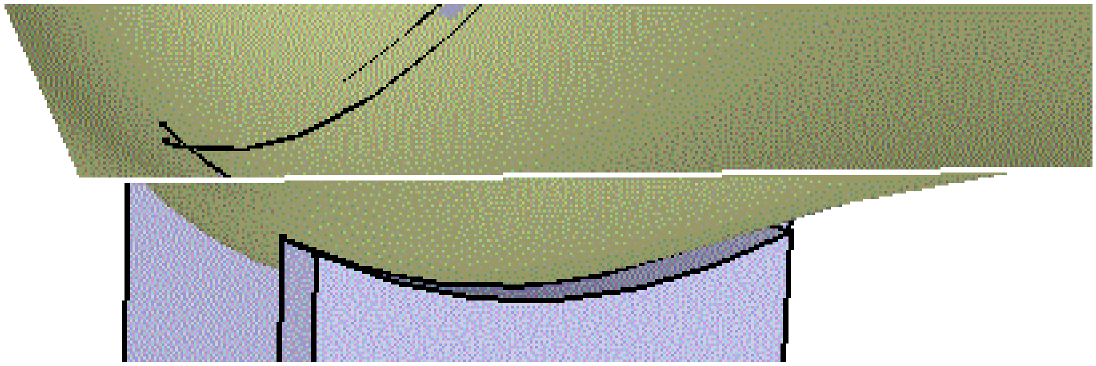
- Push **Invert Material Side** to invert the direction of the creation of the material.

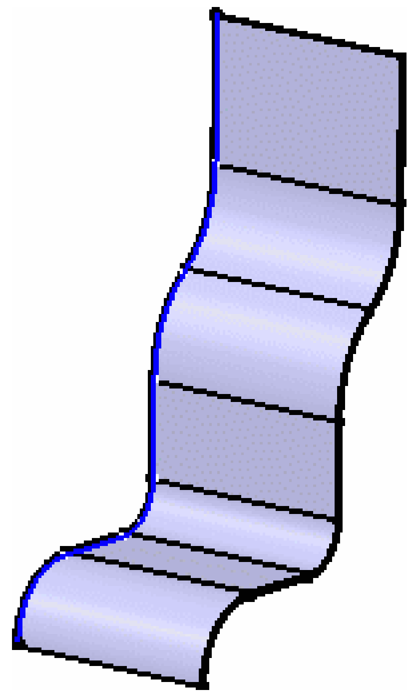


5. Click OK.

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.

The sketch may not be closed, and must contain at least a line.

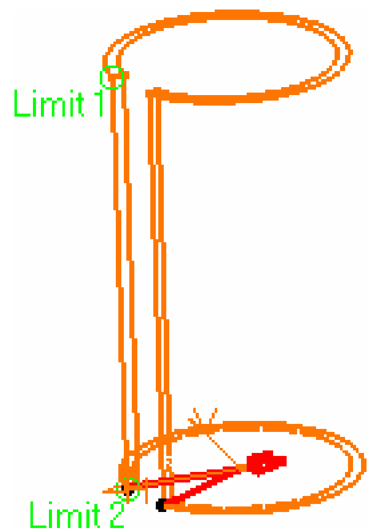
Creating Rolled Walls

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 **Rolled Walls** icon .

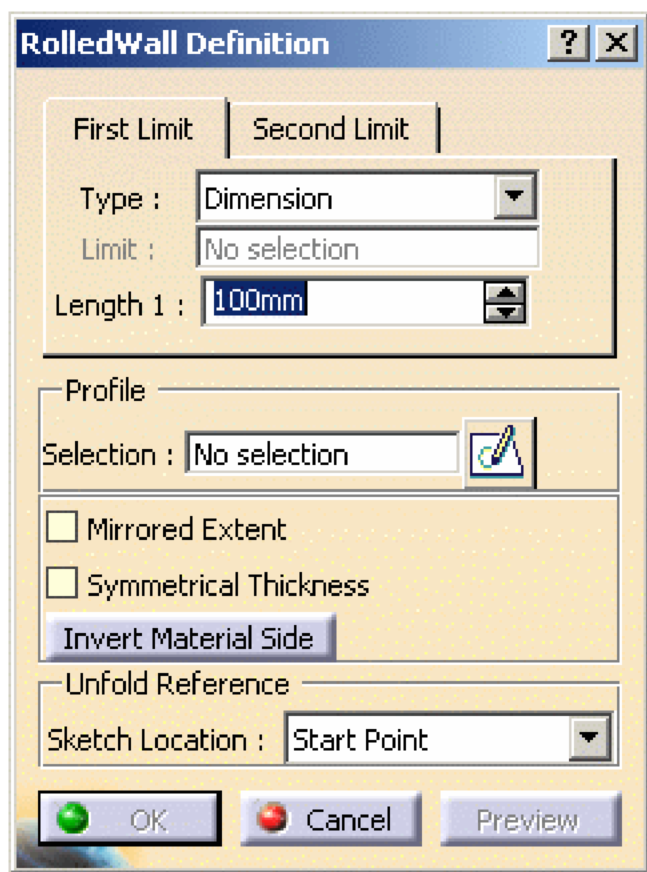
2. Select the circular sketch.



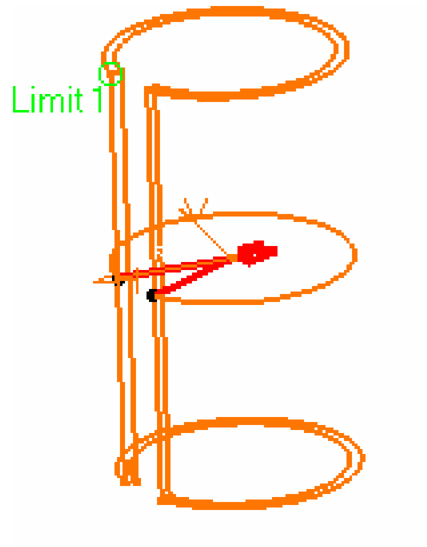
3. Make sure the type is set to **Dimension**.

4. **Length 1** and **Length 2** indicate the location of **Limit 1** and **Limit 2**.

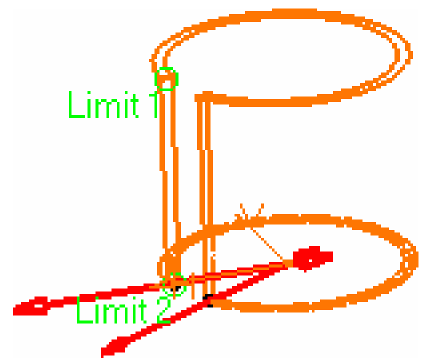
5. Define the options as needed.



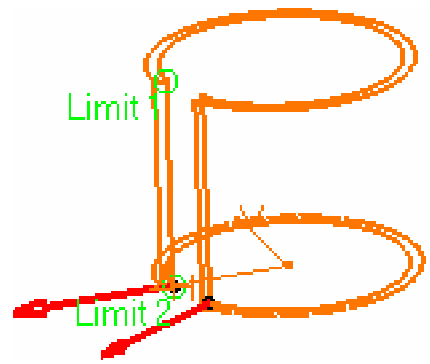
- Check the option **Mirrored extent** to extend the material on both sides of the sketch. In that case, only **Length1** can be edited.



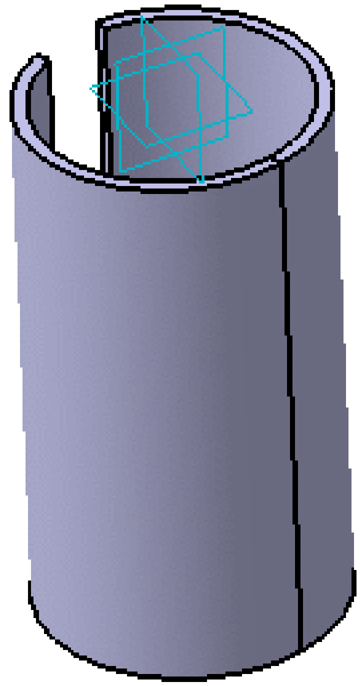
- Check **Symmetrical thickness** to create thickness inside and outside the sketch.



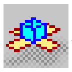
- Click **Invert Material Side** to invert the direction of the creation of the material.



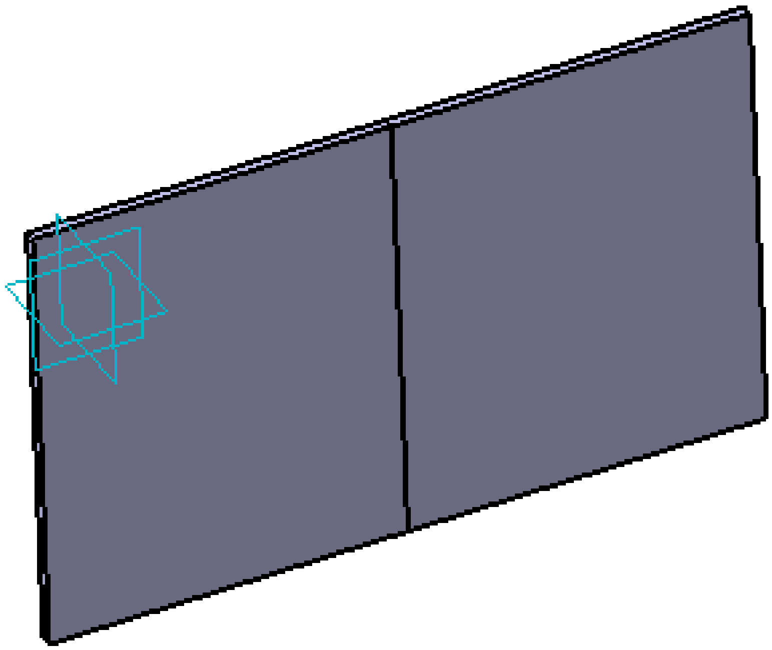
6. Click OK in the dialog box to create the rolled wall.



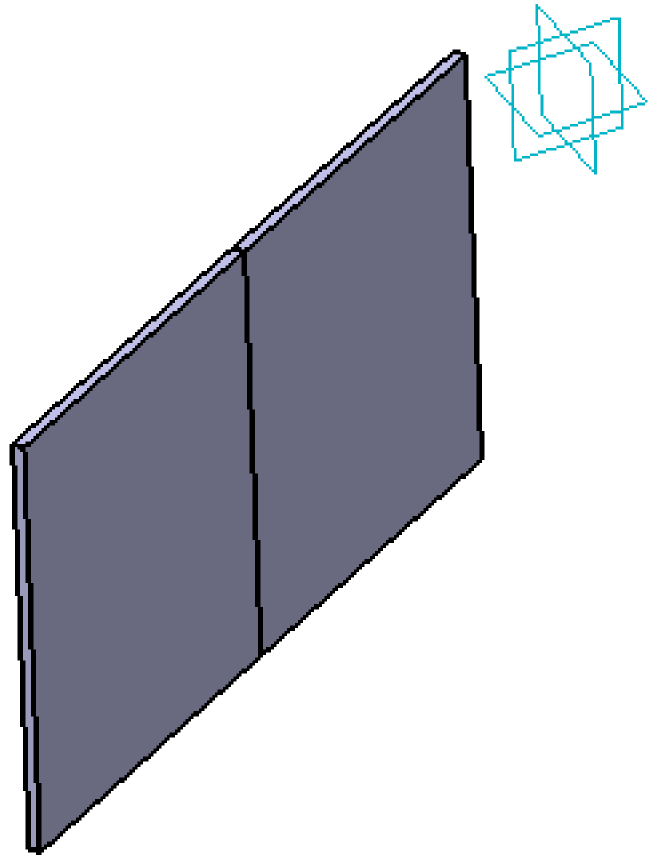
Unfold Reference

Two symmetrical faces are generated so that the rolled wall can be unfolded.
To do so, click the unfold icon .
You can unfold the rolled wall using three different references:

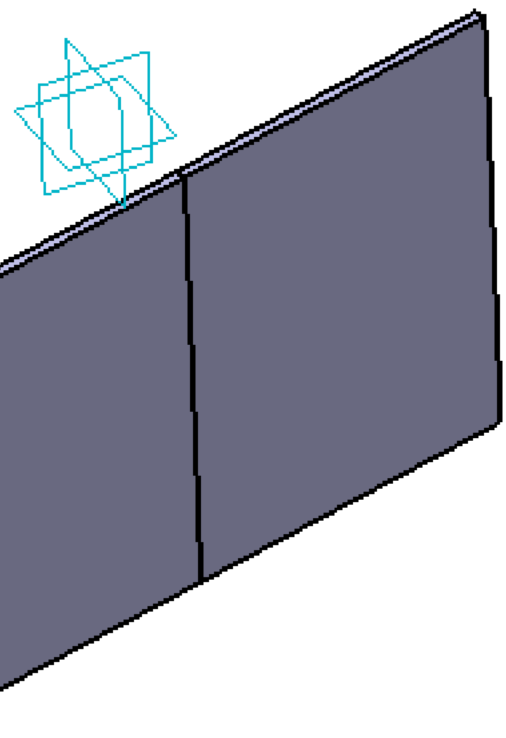
- **Start point**



- **End point**



- **Middle point**

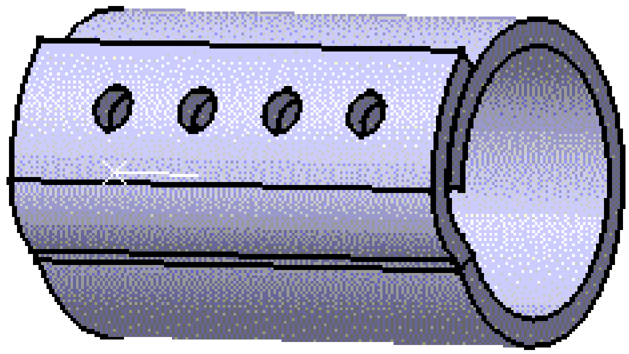


The rolled wall is a particular extrusion, indeed the input sketch is either a circular arc or a closed circle.

The sketch may be open. In that case, you can define where the opening should be.
The opening angle must be at least 0.5 degree.

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 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.

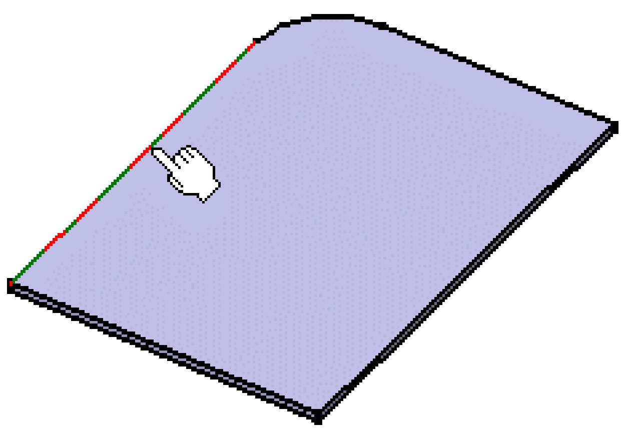
Create a tear drop: select a spine, and set the radius, and length values.

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

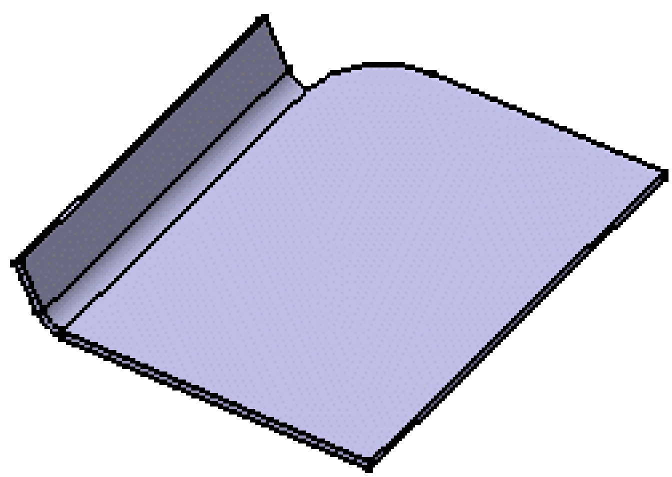
Selecting the Spine

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 contour, either pre- or user-defined, is to be swept. You can:

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

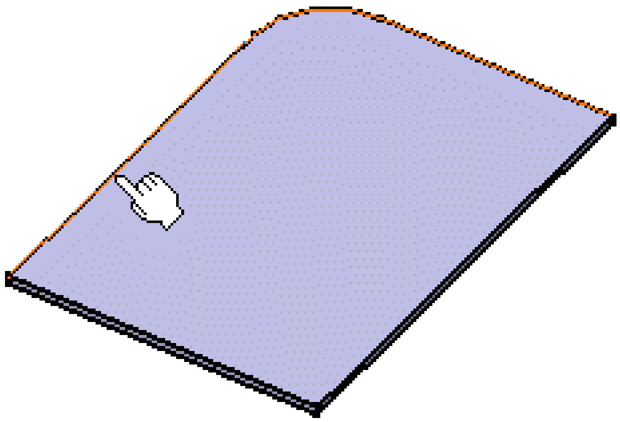


Selection without propagation

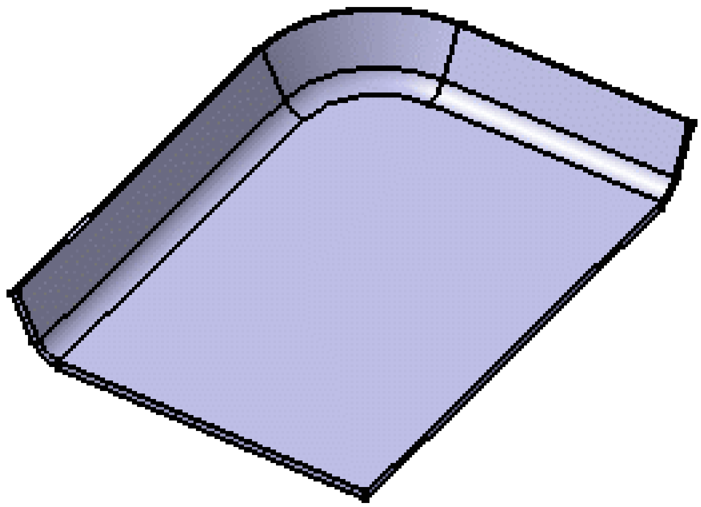


Resulting flange without propagation

- 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

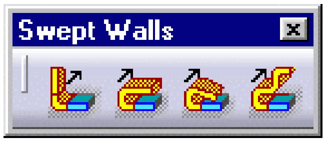
Creating a Flange

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

For the Generative Sheetmetal Design workbench, open the NEWSweptWall01.CATPart document.

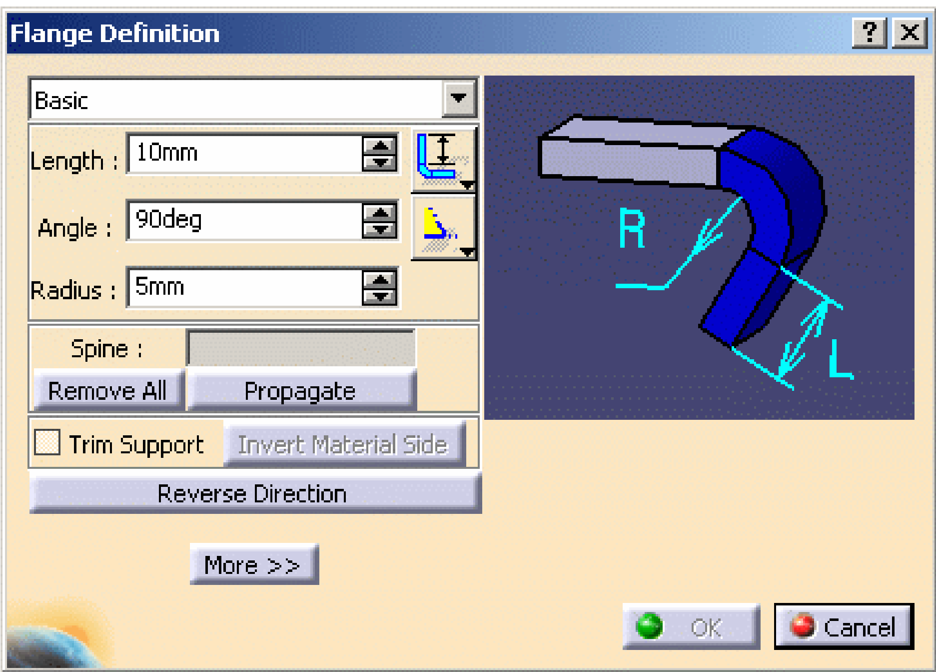
For the Aerospace SheetMetal Design workbench, open the Aero_SweptWall01.CATPart document.

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



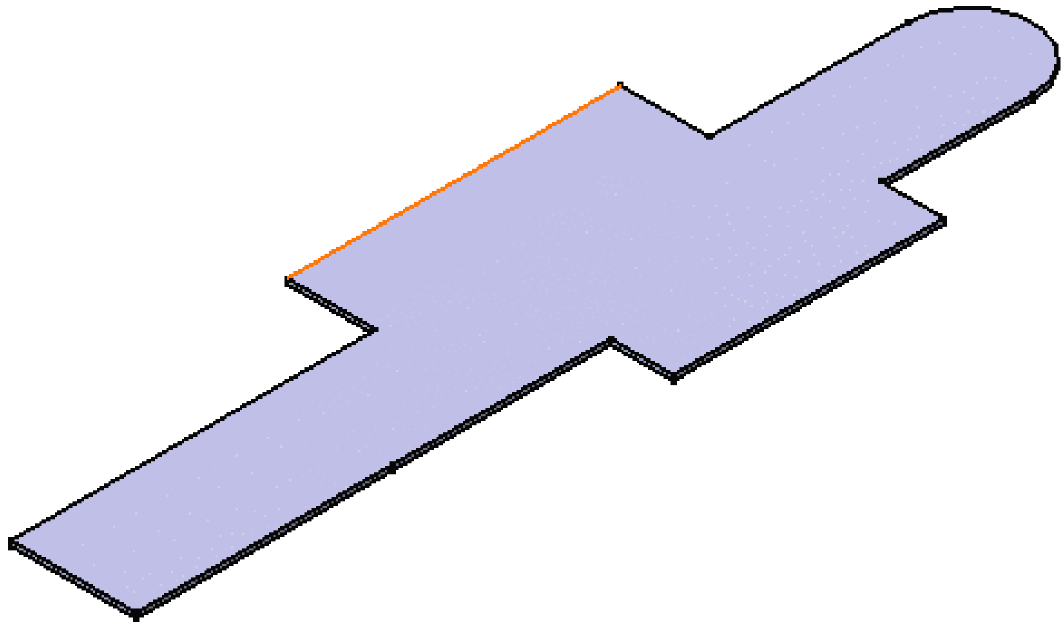
The Flange Definition dialog box is displayed.

Note that the image in the right-hand pane of the dialog box is updated as you choose your parameters and options, and provides a graphical explanation about the current selection.



By default, the icon which is pre-selected next to the Angle field corresponds to an acute angle for the Generative Sheetmetal Design workbench, and to an obtuse angle for the Aerospace SheetMetal Design workbench.

2. Select the edge as shown in red.



The **Spine** field is updated with the selected edge.

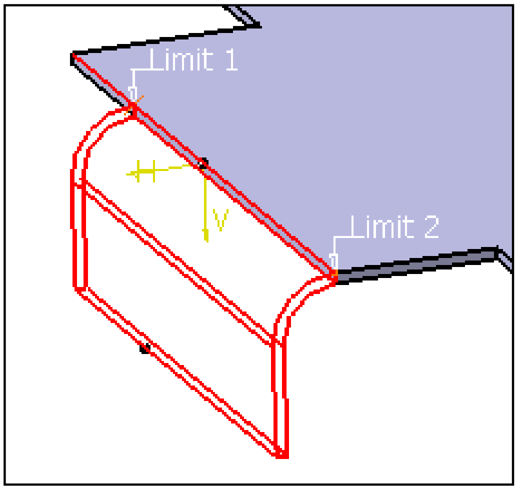
- You can use the **Remove All** button to remove the selected edge(s).
- You can use the **Propagate** button to select all tangentially contiguous edges forming the spine.

The drop-down list offers two choices:

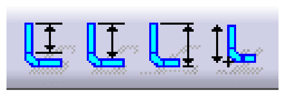
- **Basic**: the flange is created along the whole support.
- **Relimited**: the flange is created within limits you define on the support (points, for example).

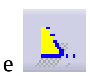
3. Leave **Basic** selected.

Selecting **Relimited** updates the dialog with two new fields (**Limit 1** and **Limit 2**) to let you specify the flange limits. You can then select as the limits two points, two planar faces, a point and a planar face, or a point and a vertex, as shown below, for example. Note that right-clicking in the **Limit 1** and **Limit 2** fields lets you create the limits (points, plane) or choose the X, Y or Z plane on-the-fly.



4. Choose the flange parameters:

- Enter 15mm in the **Length** field. Use the icons  next to the field to specify the type of length. Note that the length is always computed using the lowest external point of the flange.

- Enter 45deg in the **Angle** field. Use the icons next to the field to specify whether the angle is acute  or obtuse .

- Enter 2mm in the **Radius** field.

5. Check the **Trim Support** option to trim the selected edge.

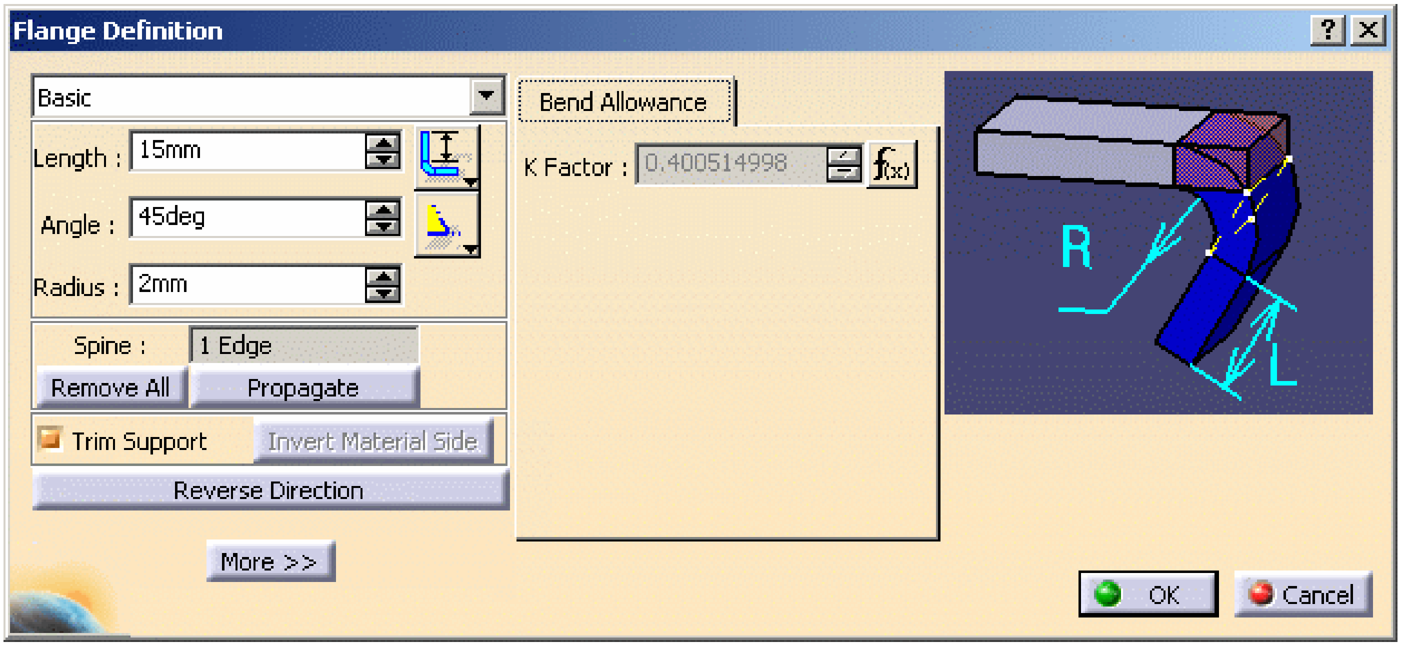
- The Trim Support option only works in the case of a planar support.

- You cannot select **Relimited** and **Trim Support** at once. Indeed, lateral cuts in the sheet metal part are currently not supported when the support is trimmed (i.e. the flange must be created from one edge of the sheet metal part to the other).

6. Click the **Reverse Direction** button to reverse the direction of the flange.

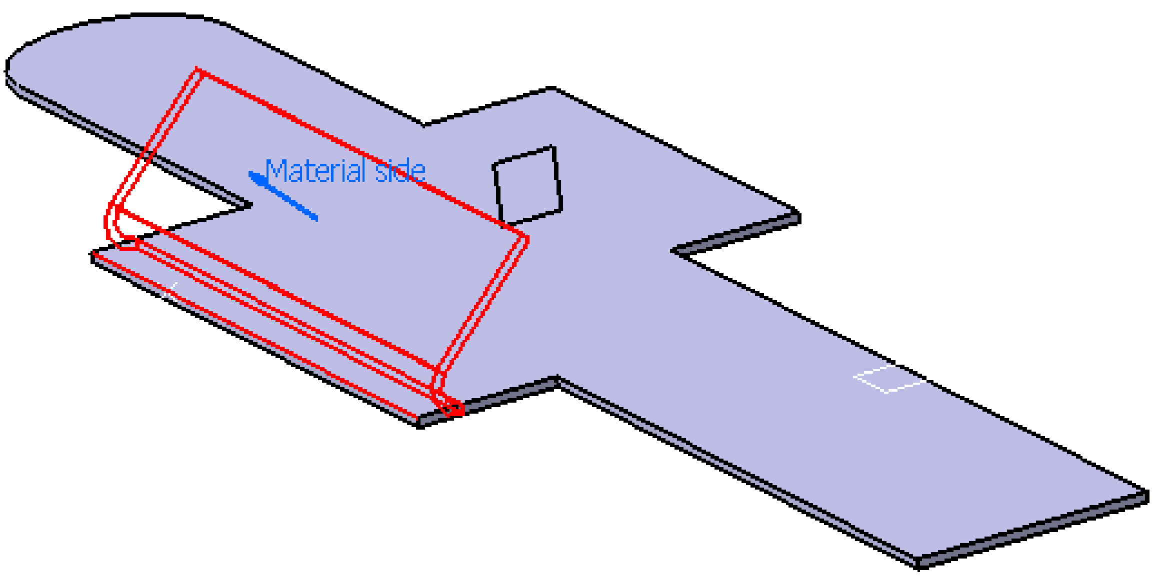
7. Click the **Invert Material Side** button to invert the material side. (This option is only available when the **Trim Support** option is checked, otherwise it is deactivated.)

8. 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.

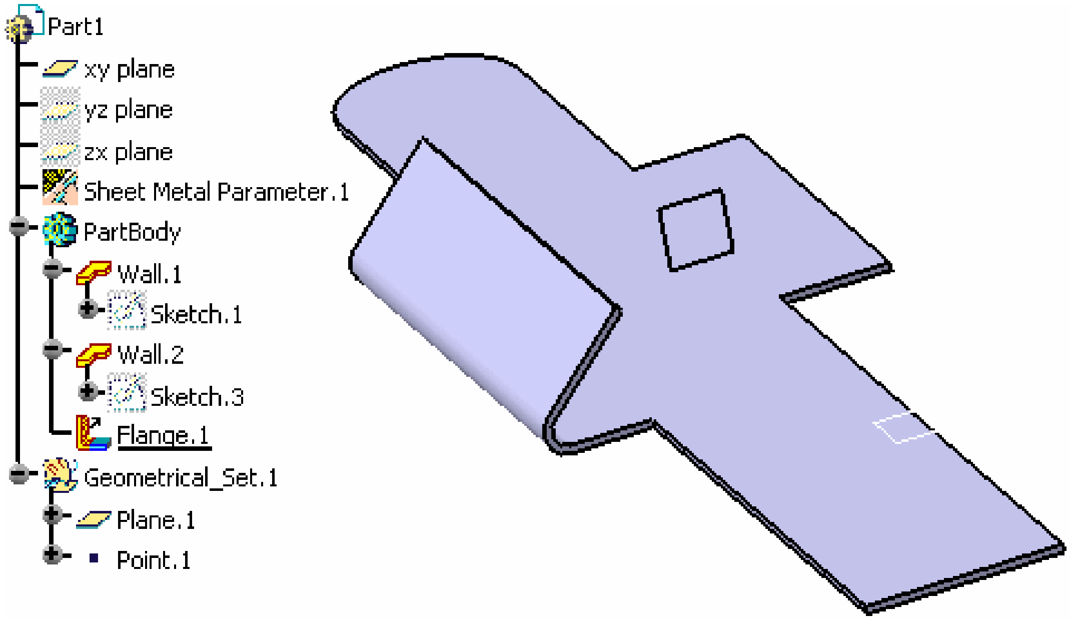


In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

A preview of the flange to be created is displayed in the geometry area.



9. When you are satisfied with the result, click **OK** to create the flange. The flange is created and the feature is added to the specification tree.

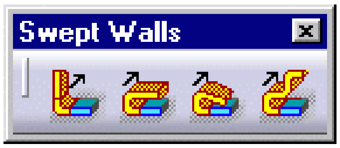

Part1
xy plane
yz plane
zx plane
Sheet Metal Parameter.1
PartBody
Wall.1
Sketch.1
Wall.2
Sketch.3
Flange.1
Geometrical_Set.1
Plane.1
Point.1

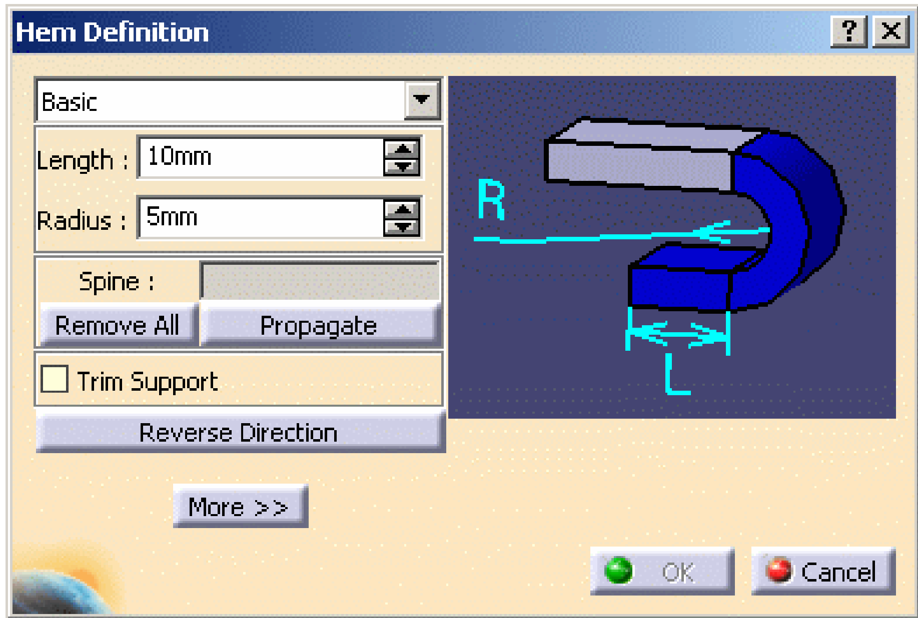
Creating a Hem

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

The NEWSweptWall01.CATPart document is still open from the previous task.
If not, open the NEWSweptWall02.CATPart document from the samples directory.

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

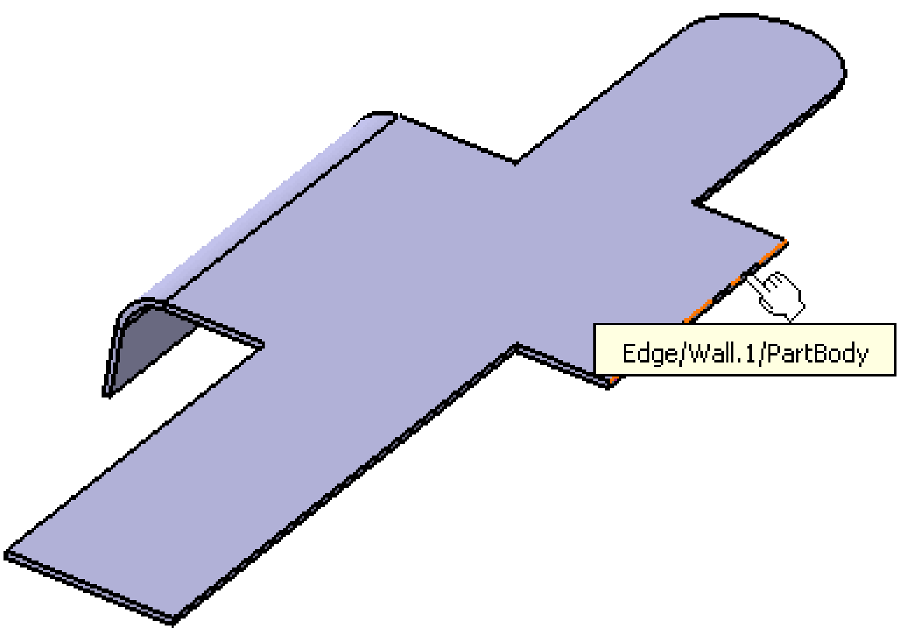




The Hem Definition dialog box opens.

Note that the image in the right-hand pane of the dialog box is updated as you choose your parameters and options, and provides a graphical explanation about the current selection.

2. Select the edge as shown.



The **Spine** field is updated with the selected edge.

- You can use the **Remove All** button to remove the selected edge(s).
- You can use the **Propagate** button to select all tangentially contiguous edges forming the spine.

The drop-down list offers two choices:

- **Basic**: the hem is created along the whole support.
- **Relimited**: the hem is created within limits you define on the support (points, for example).

3. Leave **Basic** selected.

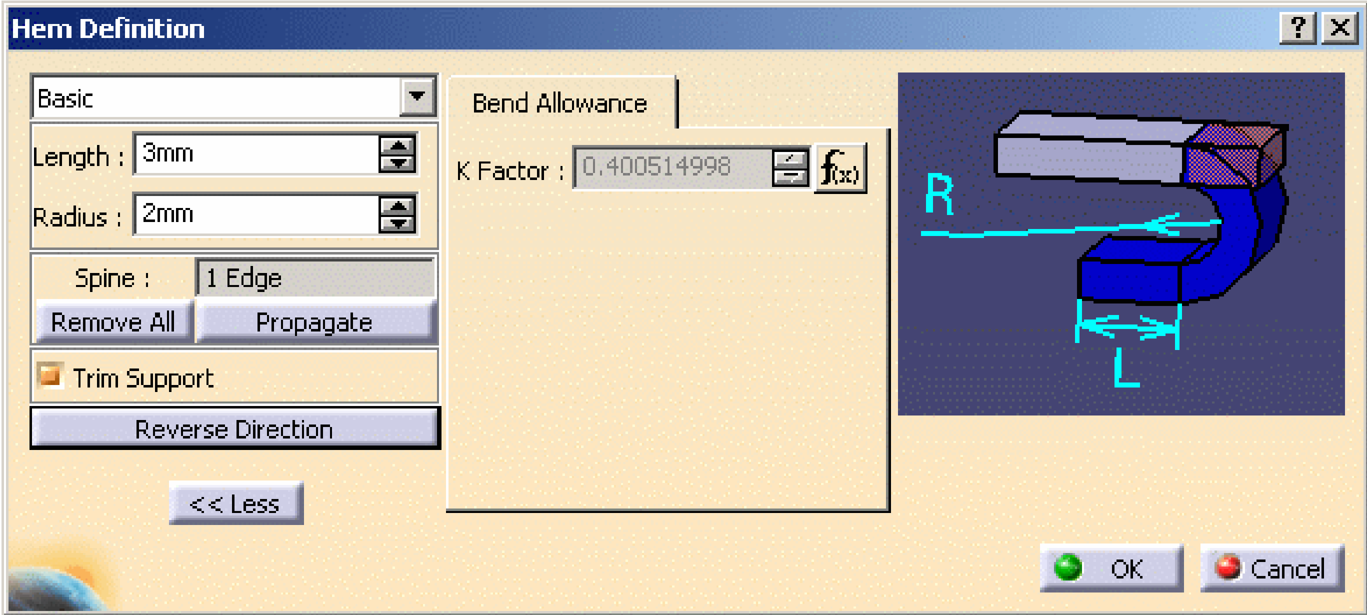
Selecting **Relimited** updates the dialog with two new fields (**Limit 1** and **Limit 2**) to let you specify the hem limits. You can then select as the limits two points, two planar faces, a point and a planar face, or a point and a vertex, for example. Note that right-clicking in the **Limit 1** and **Limit 2** fields lets you create the limits (points, plane) or choose the X, Y or Z plane on-the-fly.

4. Enter 3mm in the **Length** field, and 2mm in the **Radius** field.
5. Check the **Trim Support** option to trim the selected edge.

- The Trim Support option only works in the case of a planar support.
- You cannot select **Relimited** and **Trim Support** at once. Indeed, lateral cuts in the sheet metal part are currently not supported when the support is trimmed (i.e. the hem must be created from one edge of the sheet metal part to the other).

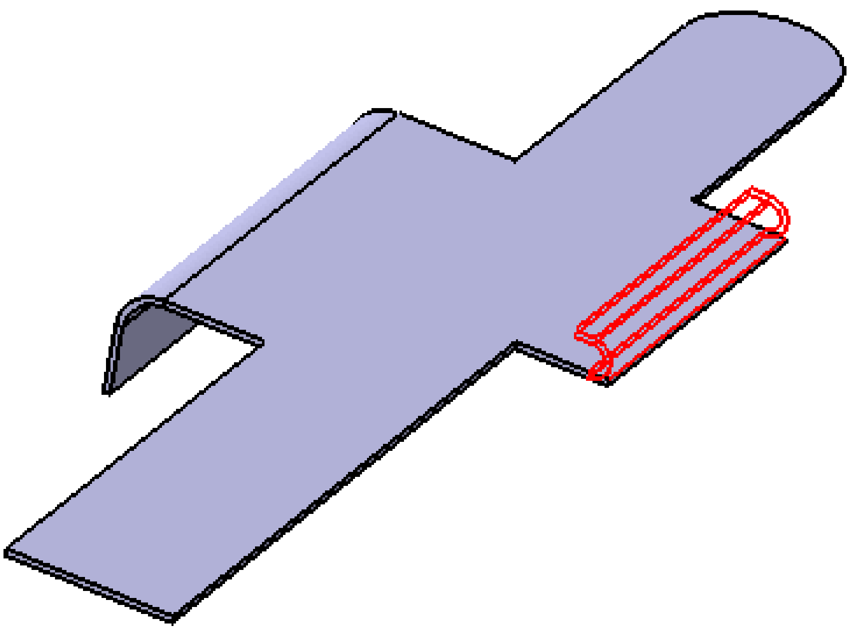
6. Click the **Reverse Direction** button to reverse the direction of the hem.
7. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

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

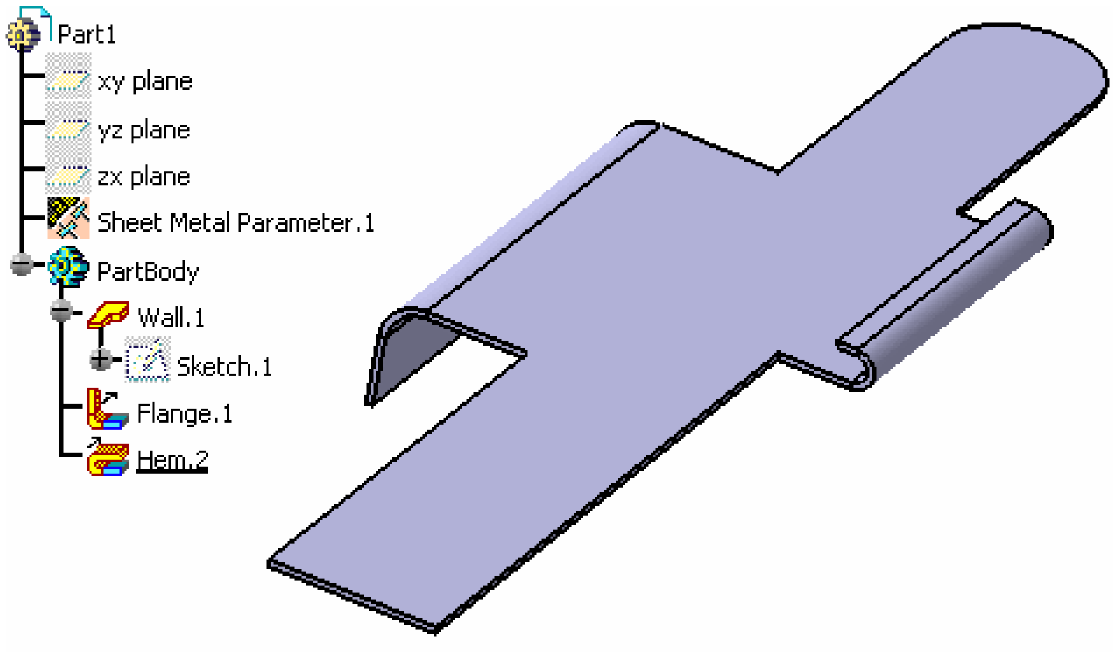


In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

A preview of the hem to be created is displayed in the geometry area.



8. When you are satisfied with the result, click **OK** to create the hem. The hem is created and the feature is added to the specification tree.

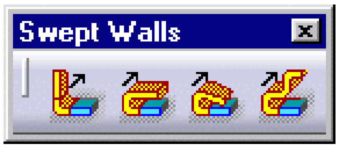

Part1
xy plane
yz plane
zx plane
Sheet Metal Parameter.1
PartBody
Wall.1
Sketch.1
Flange.1
Hem.2

Creating a Tear Drop

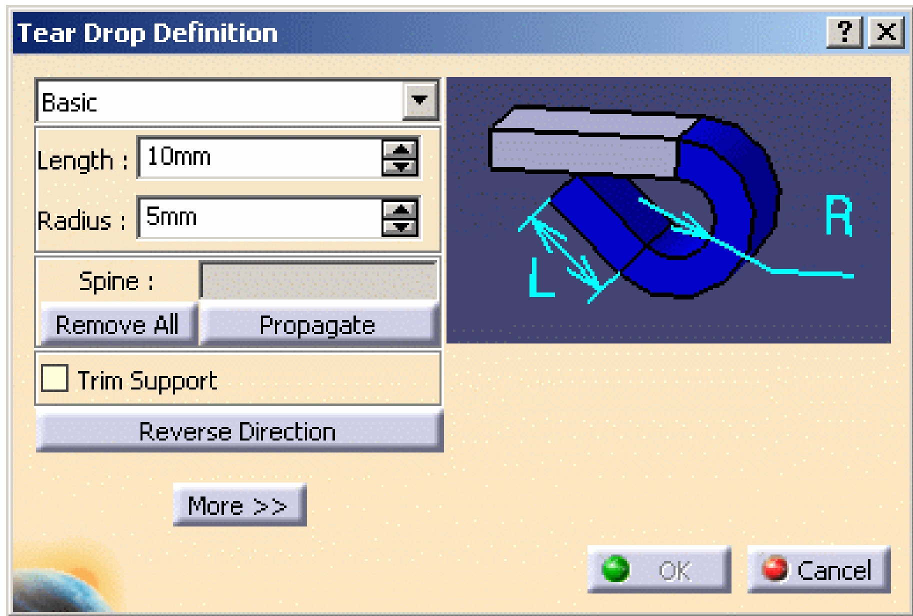
This task explains how to generate a tear drop from a spine and a profile.

The NEWSweptWall01.CATPart document is still open from the previous task.
If not, open the NEWSweptWall03.CATPart document from the samples directory.

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

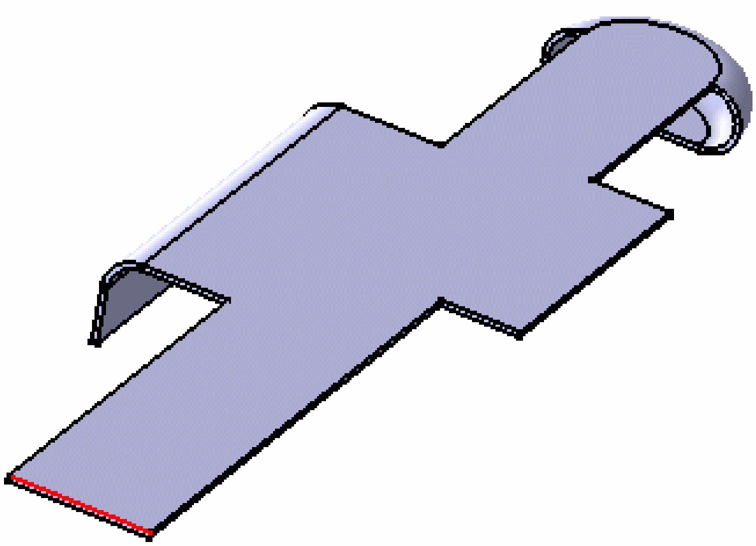


The Tear Drop Definition dialog box opens.



Note that the image in the right-hand pane of the dialog box is updated as you choose your parameters and options, and provides a graphical explanation about the current selection.

2. Select the edge as shown in red.



The **Spine** field is updated with the selected edge.

- You can use the **Remove All** button to remove the selected edge(s).

- You can use the **Propagate** button to select all tangentially contiguous edges forming the spine.

The drop-down list offers two choices:

- **Basic**: the tear drop is created along the whole support.

- **Relimited**: the tear drop is created within limits you define on the support (points, for example).

3. Leave **Basic** selected.

Selecting **Relimited** updates the dialog with two new fields (**Limit 1** and **Limit 2**) to let you specify the tear drop limits. You can then select as the limits two points, two planar faces, a point and a planar face, or a point and a vertex, for example. Note that right-clicking in the **Limit 1** and **Limit 2** fields lets you create the limits (points, plane) or choose the X, Y or Z plane on-the-fly.

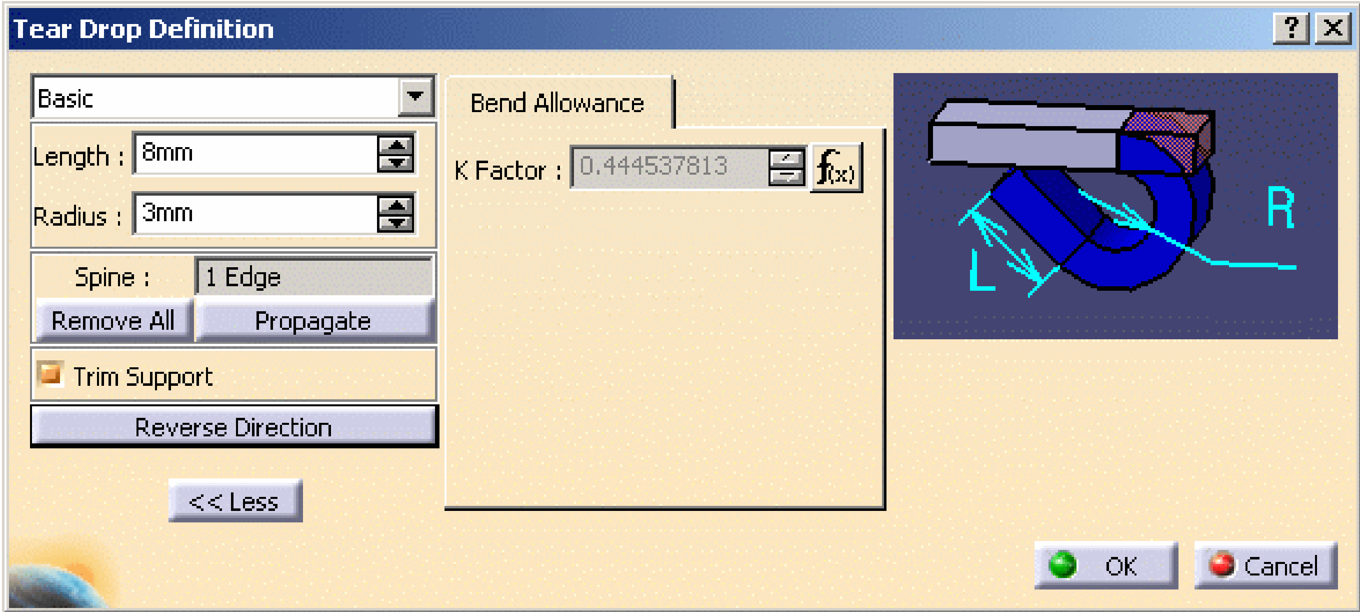
4. Enter 8mm in the **Length** field, and 3mm in the **Radius** field.
5. Check the **Trim Support** option to trim the selected edge.

- The Trim Support option only works in the case of a planar support.

- You cannot select **Relimited** and **Trim Support** at once. Indeed, lateral cuts in the sheet metal part are currently not supported when the support is trimmed (i.e. the tear drop must be created from one edge of the sheet metal part to the other).

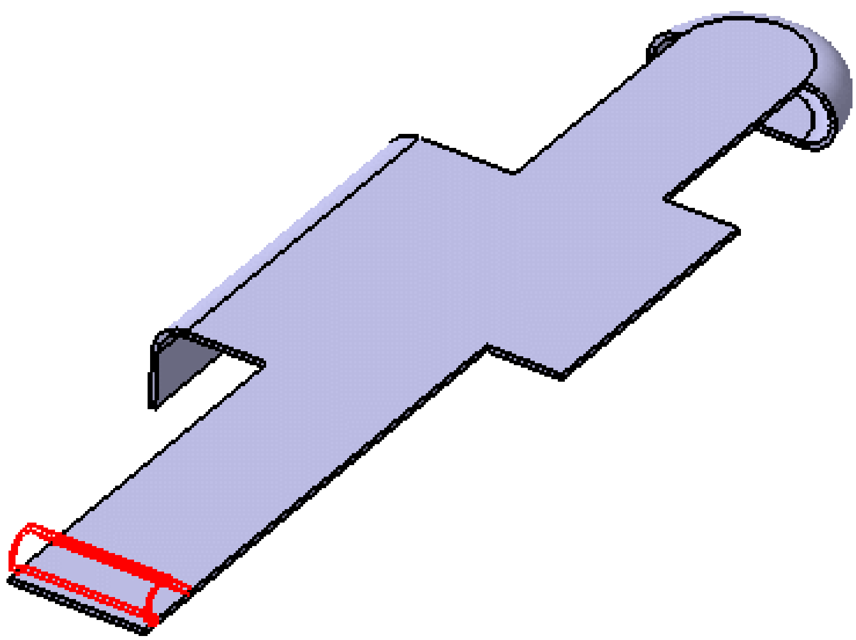
6. Click the **Reverse Direction** button to reverse the direction of the tear drop.
7. Click the **More>>** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

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

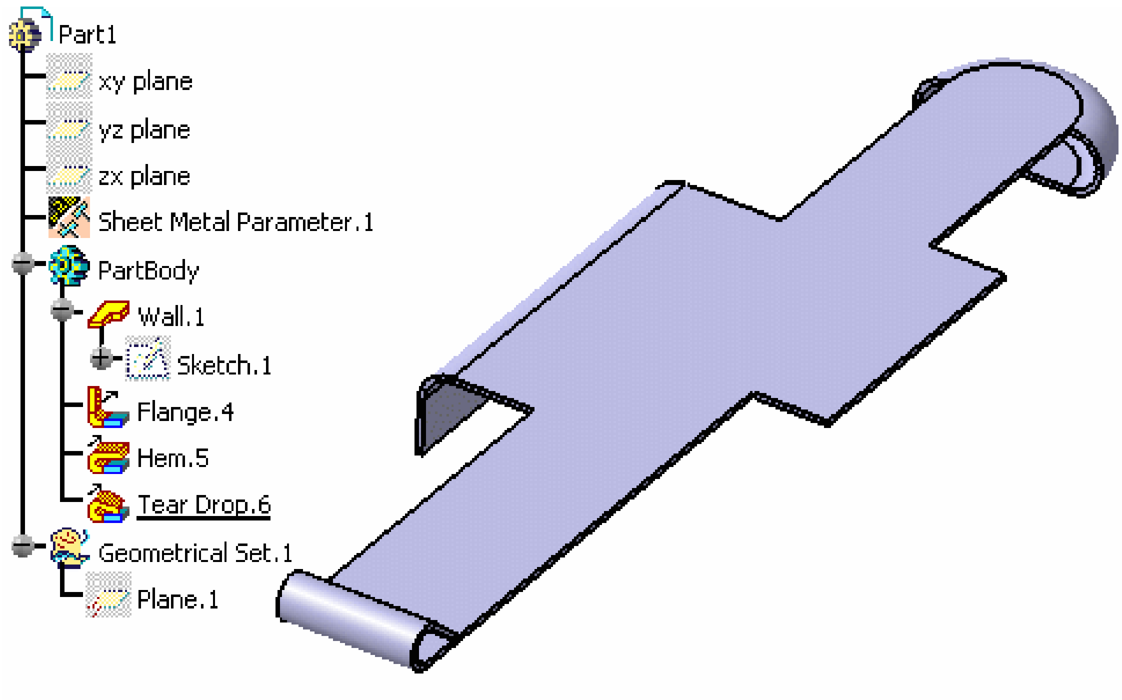


In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

A preview of the tear drop to be created is displayed in the geometry area.



8. When you are satisfied with the result, click **OK** to create the tear drop. The tear drop is created and the feature is added to the specification tree.

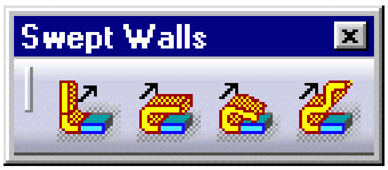

Part1
xy plane
yz plane
zx plane
Sheet Metal Parameter.1
PartBody
Wall.1
Sketch.1
Flange.4
Hem.5
Tear Drop.6
Geometrical Set.1
Plane.1

Creating a User Flange

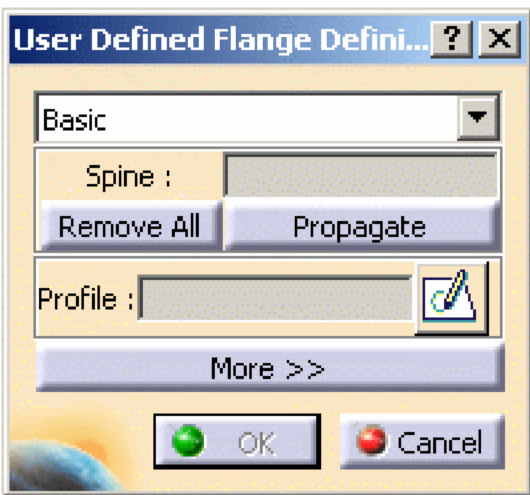
This task explains how to generate a user flange from a spine and a user-defined profile.

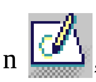
The NEWSweptWall01.CATPart document is still open from the previous task.
If not, open the NEWSweptWall04.CATPart document from the samples directory. As a profile is already defined on the part, you will be able to skip step 2 of the scenario.

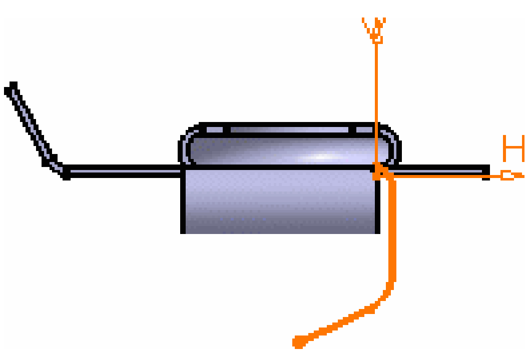
1. Select the **User Flange** icon  in the **Swept Walls** sub-toolbar.

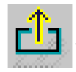


The **User Defined Flange** Definition dialog box opens.



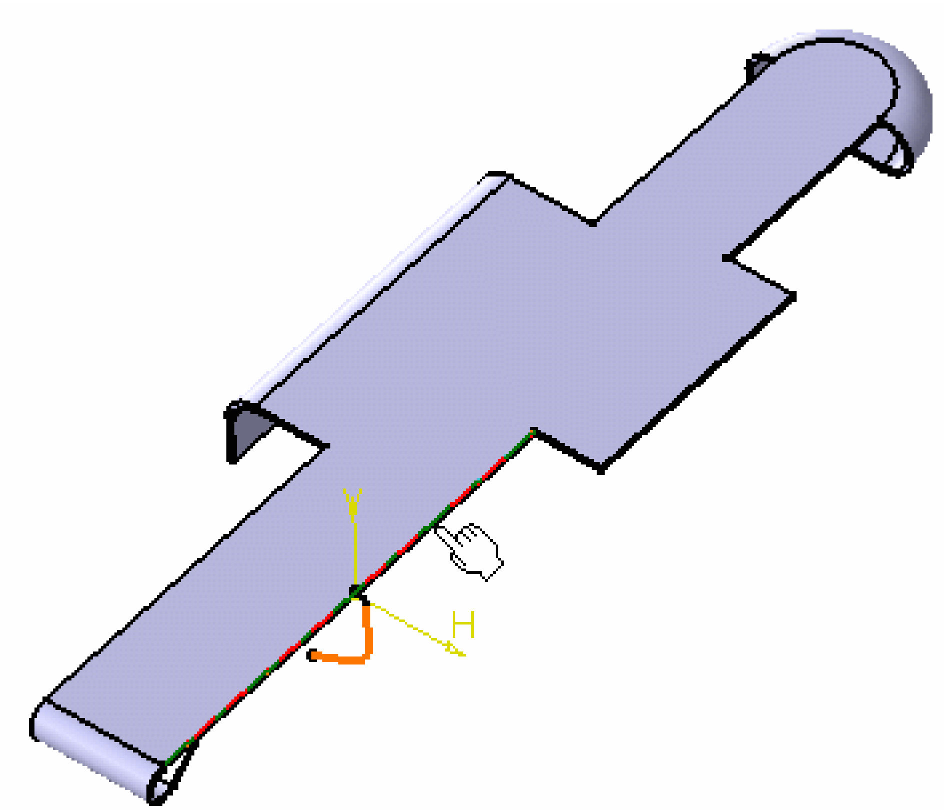
2. If you are using the NEWSweptWall01.CATPart document, click the **Sketcher** icon , and define a profile in the yz plane as shown below:



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

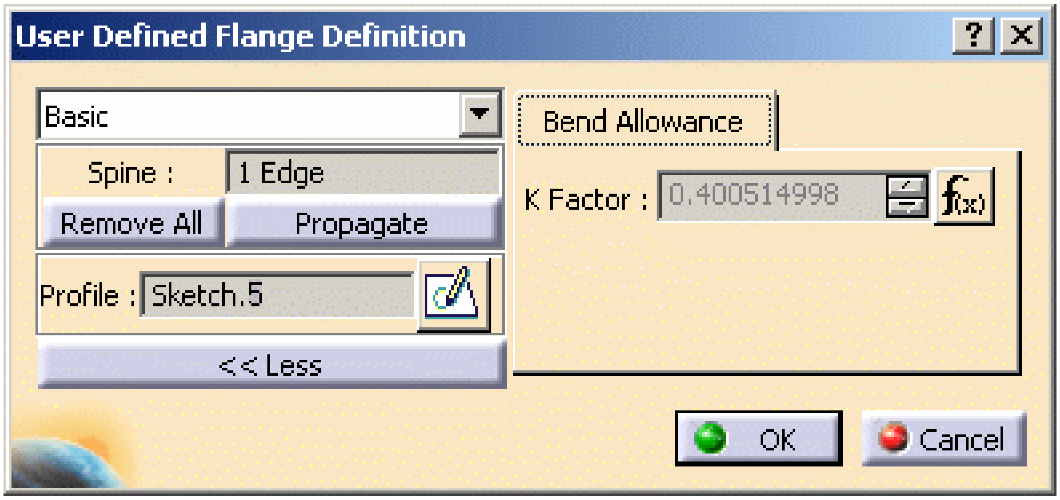
If you are using the NEWSweptWall04.CATPart, go directly to step 3 as the profile is already defined.

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



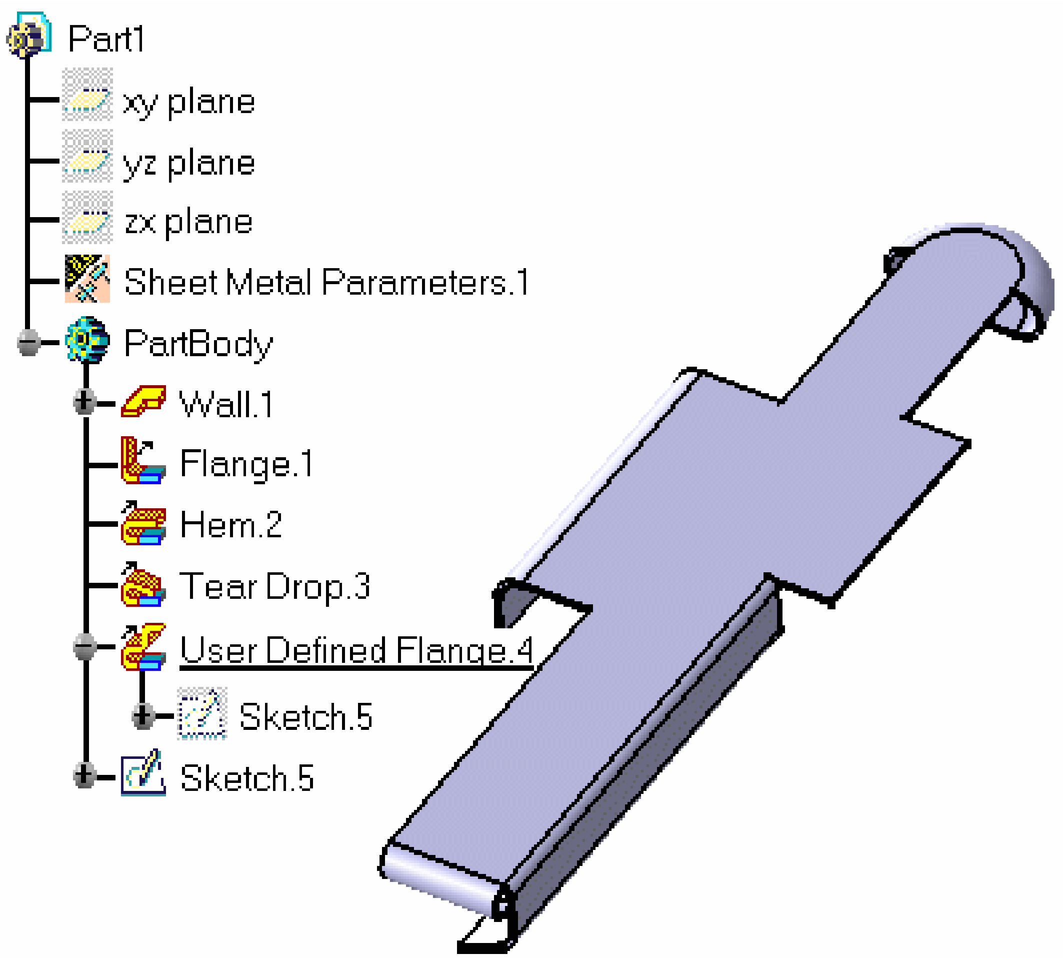
4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

   You may need to deactivate the formula using the contextual menu on the **K Factor** 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 user flange.



The feature is added in the specification tree.

- Use the **Remove All** button to remove the selected edge(s).
- Use the **Propagate** button to select all tangentially contiguous edges forming the spine.

As far as the profile is concerned, remember that:

- There must be a tangency continuity with the edge on which the flange is created,
- The plane must be normal to the spine.

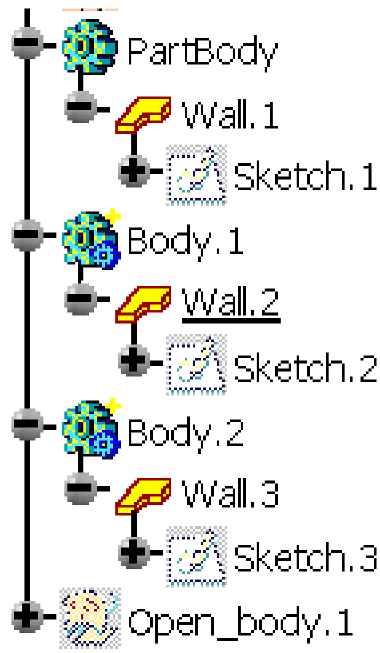
Aggregating Bodies

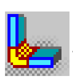
This task shows you, when creating a bend, to aggregate, under the current body containing a wall, the created bend as well as the body containing the second wall.

Open the NEWBendExtremities03.CATPart document.

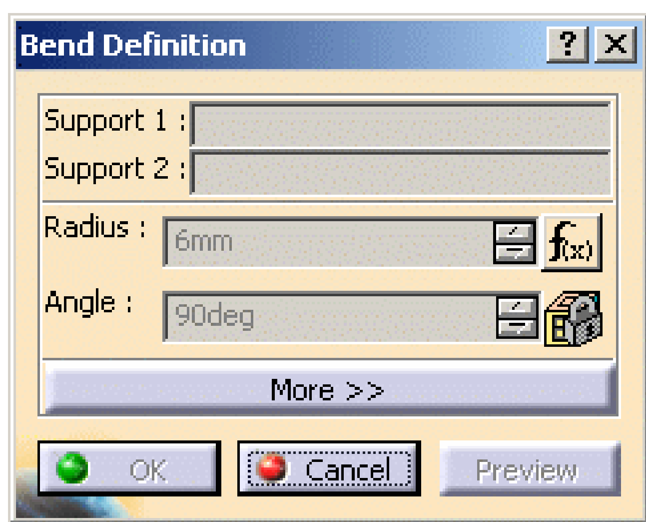
This capability is available with the Bend and Bend From Flat functionalities.
Let's take an example with the Bend.

1. In the specification tree, define **Wall.2** (in Body.1) as current: right-click and select **Define In Work Object** from the contextual menu.



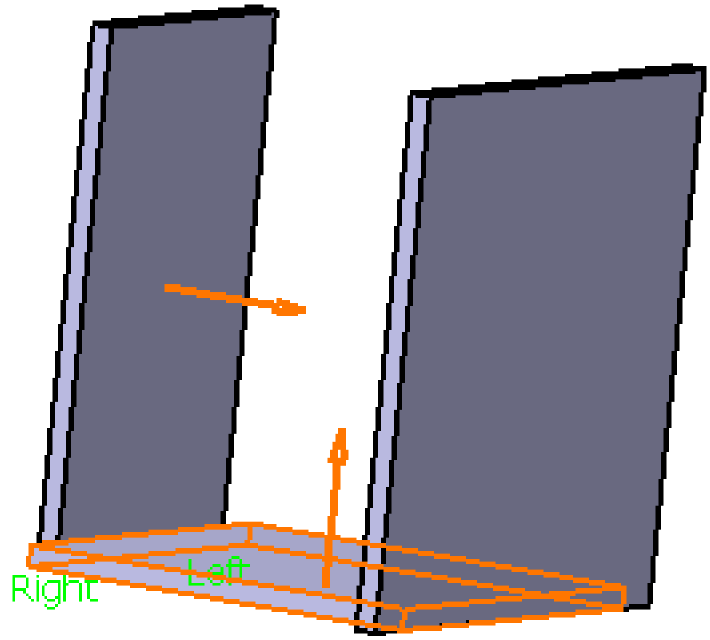
2. Select the **Bend** icon .

The Bend Definition dialog box opens.

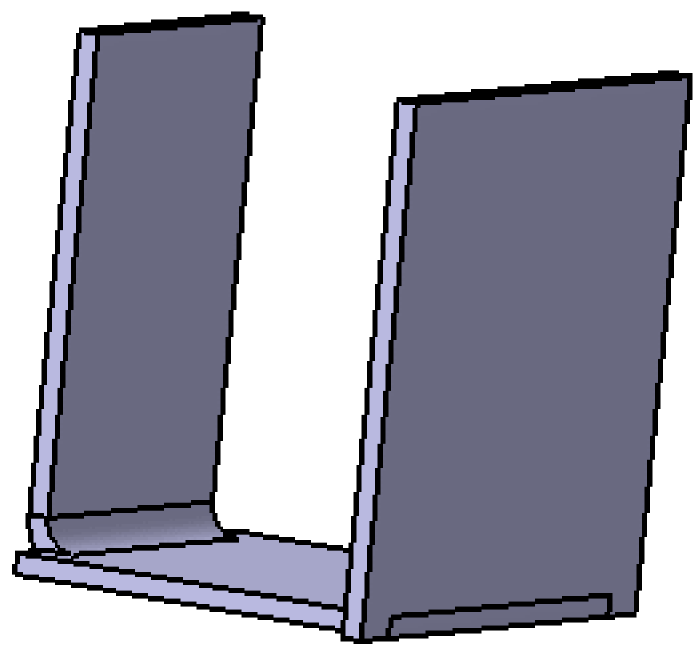


3. Select **Wall.2** and **Wall.3** in the specification tree.

   The Bend Definition dialog box is updated.

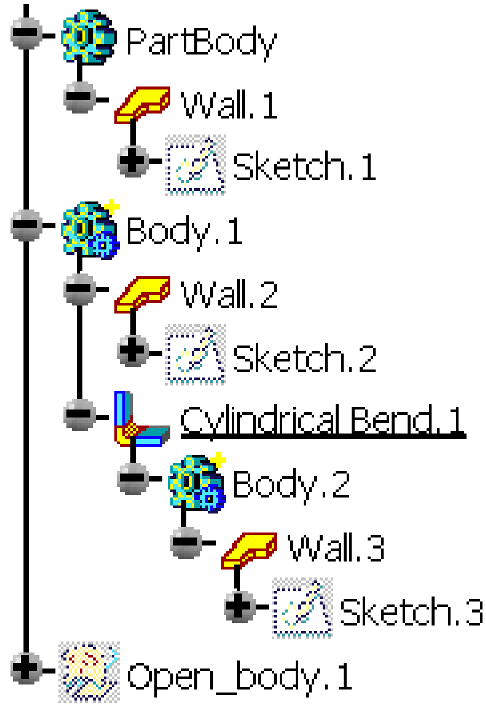


4. Click **Preview** to visualize the bend.
5. Click OK in the Bend Definition dialog box.



The Bend (identified as Cylindrical Bend.x) is created and added to the specification tree.

You can see that it appears under the Body.1 node and aggregates Body.2 containing Wall.3.



You can perform the same operation by defining PartBody as the current object and selecting Wall.1 and Wall.2 as inputs. PartBody contains the cylindrical bend, that aggregates Body.1 (itself containing Wall.2).

Creating a Hopper

This task illustrates how to create a surfacic and a canonic hopper between two sketched profiles, with an opening line (for unfolding operations) defined by an edge for surfacic hoppers or two points for canonic hoppers.

Open the NEWHopper01.CATPart document. This document contains two sketches, as well as a point on each sketch.

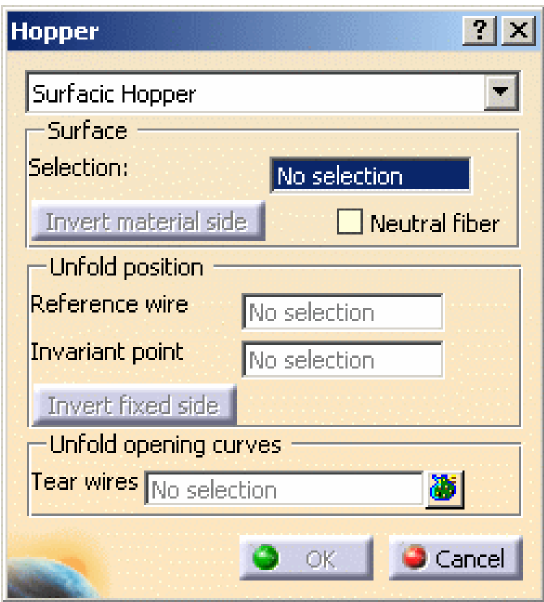
Surfacic hopper

Surfacic hoppers are defined by a ruled surface selected by the user or created thanks to the loft command. Defining a surfacic hopper via a loft is highly recommended since it allows detection of all canonical segments.

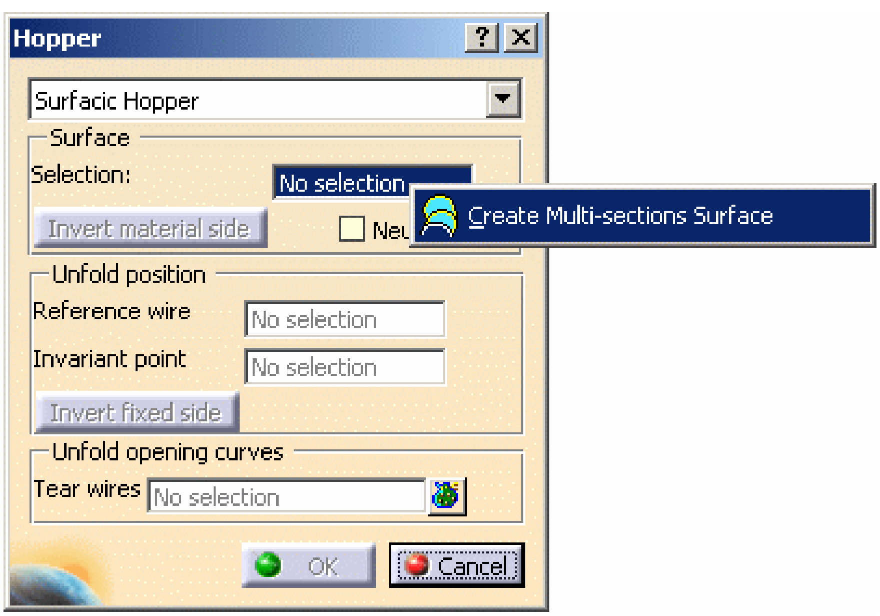
The two sketches used to define the loft can be on parallel or non parallel planes.

The reference wire and the invariant point, used to unfold the hopper, must lie on the surface, as well as the tear wire.

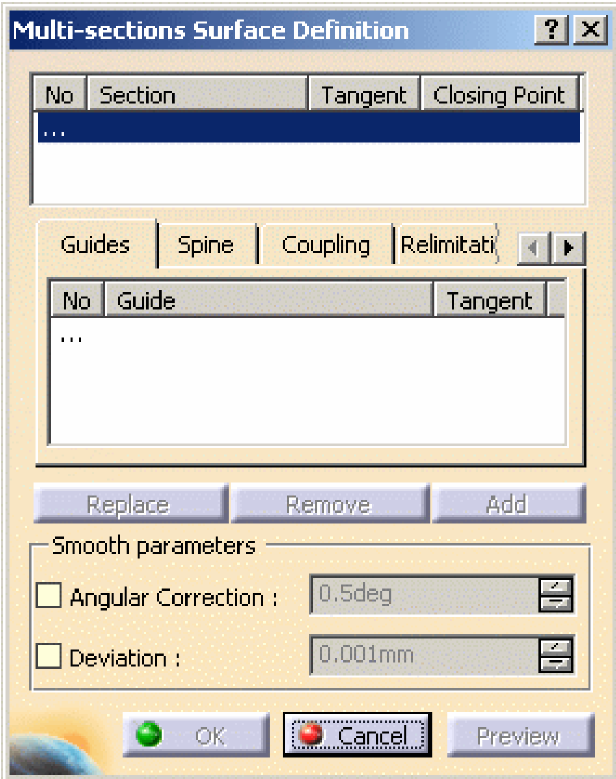
1. Click the **Hopper** icon . The Hopper dialog box is displayed.



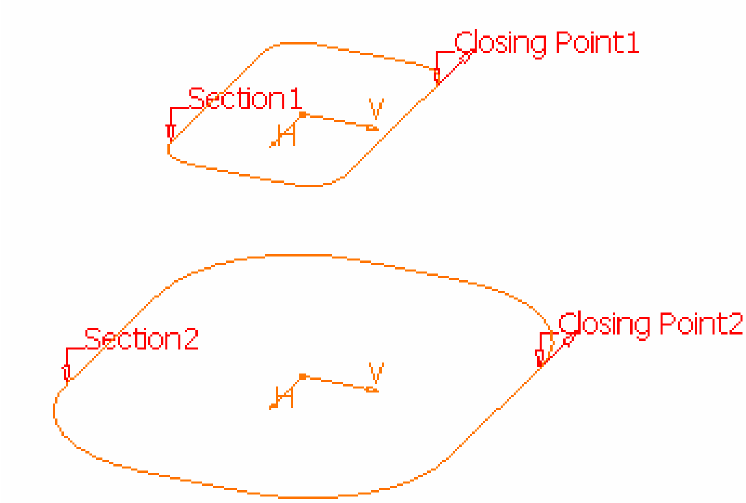
2. Right-click on No selection field and select Create Multi-sections surface.

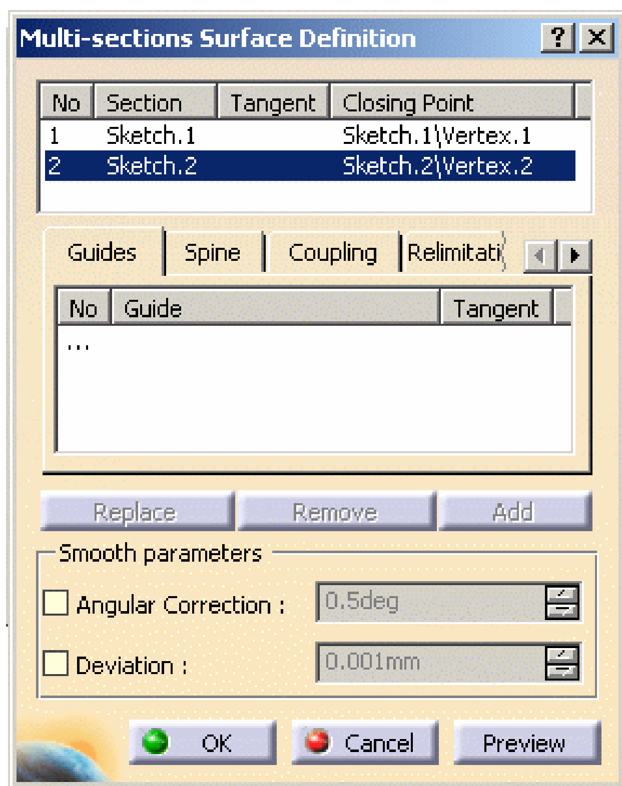


The Multi-sections Surface Definition dialog box is displayed.



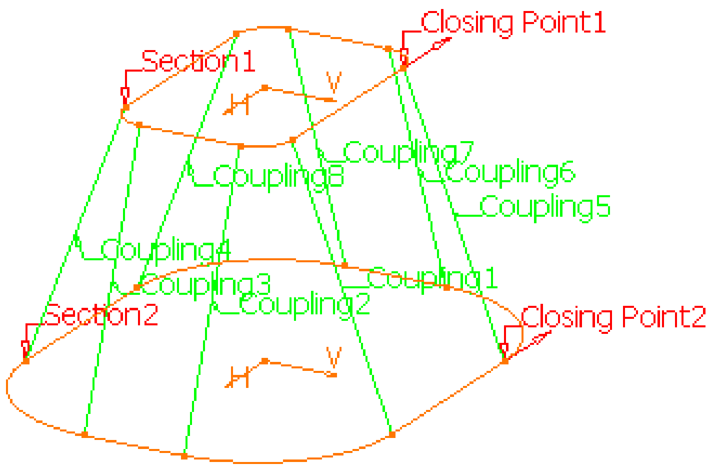
3. Select Sketch.1 and Sketch.2 in the specification tree as surfaces.





4. Click on the Coupling tab and start coupling sketches together.

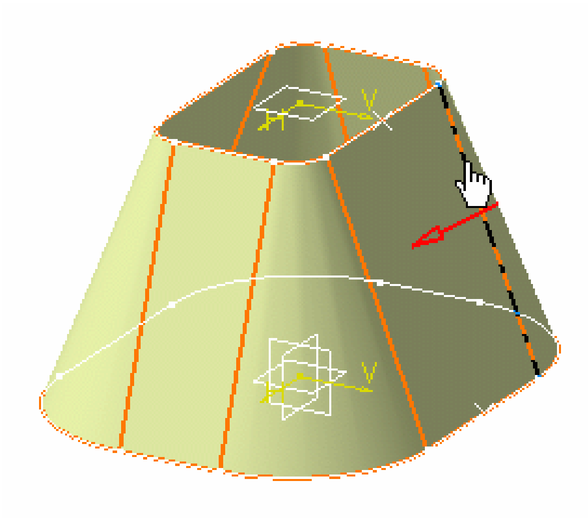
You should end up with eight couplings.



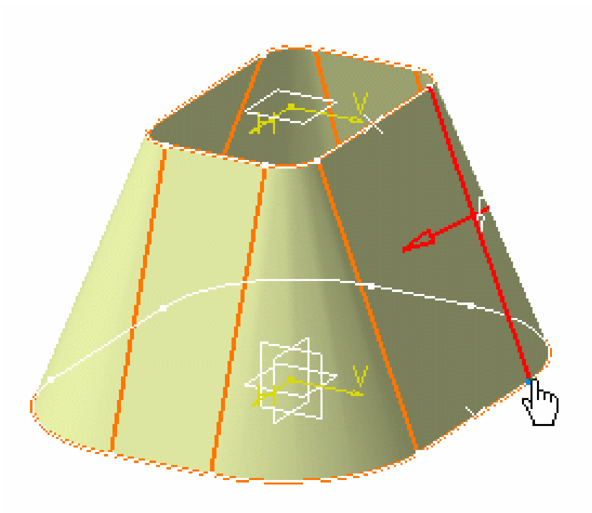
5. Click on **Preview** if you want to visualize the loft, then on **OK**.

6. Select a reference wire lying on the surface of the hopper.

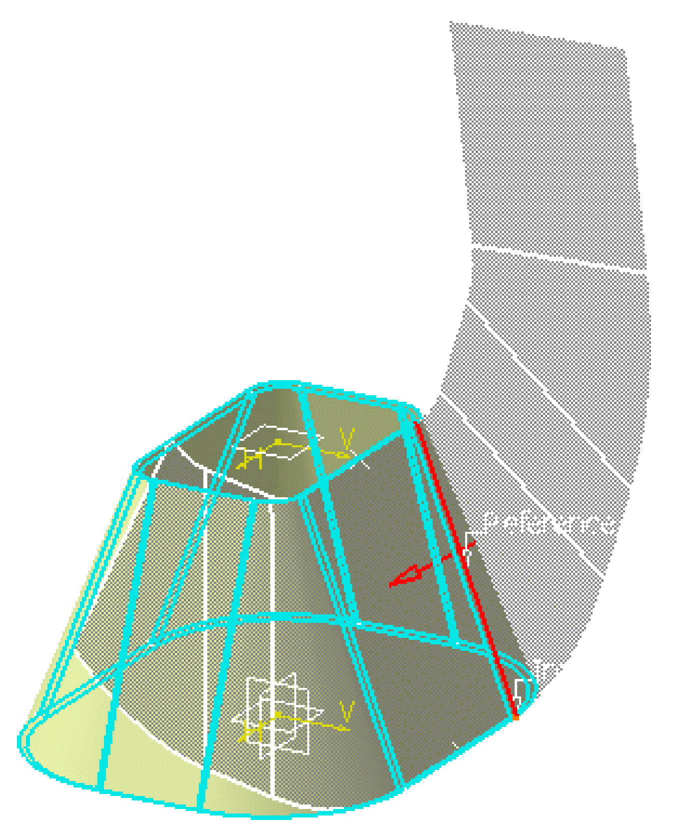
   In our example, we select an edge.



7. Select an invariant point lying both on the surface and on the reference wire.



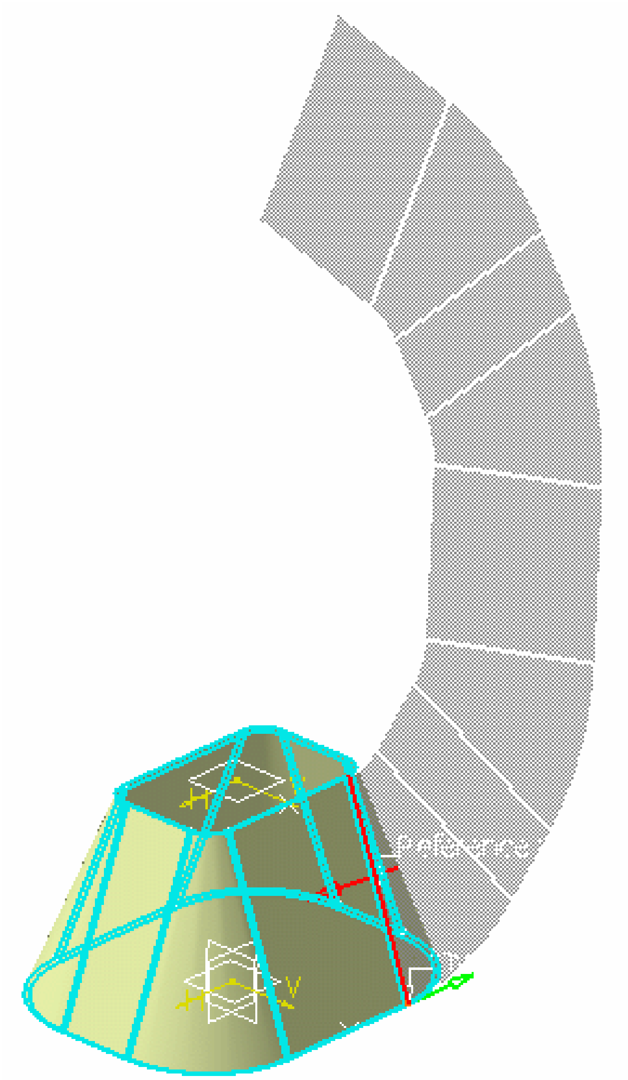
A preview of the unfolded hopper is displayed, as well as its thickening in light blue.



8. Select a tear wire.

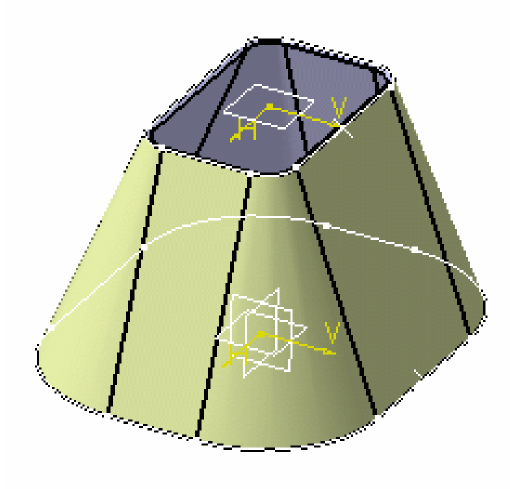
In our example, we select the same edge as the reference wire.

The unfolded view of the hopper starting from the tear wire is displayed.

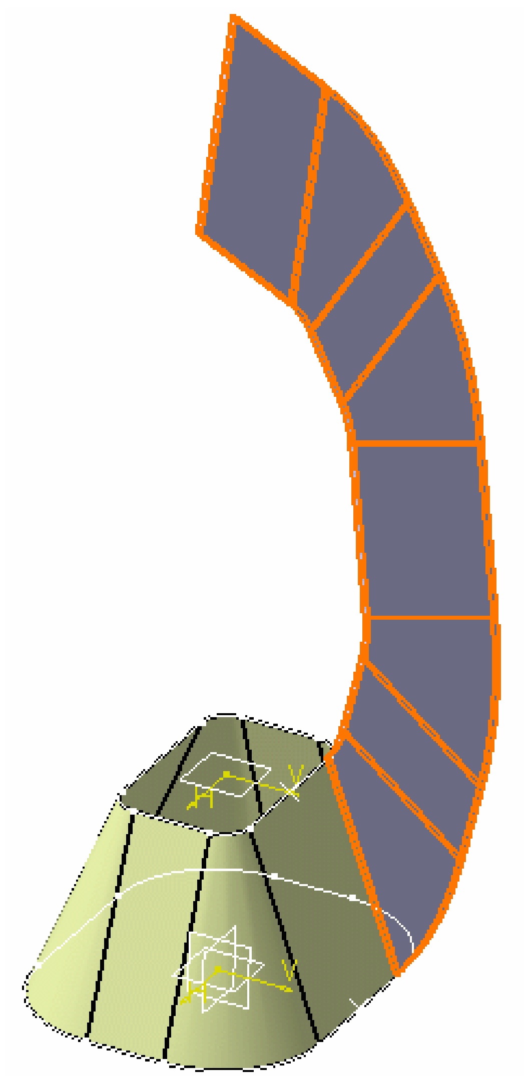


- The surface must be ruled otherwise you will not be able to create a hopper.
- If you want to modify your selections in the Hopper dialog box, you can right-click in the field and select **Clear selection** in the contextual menu.

9. Click **OK** to validate and exit the dialog box. The hopper is created.



10. If you now click the **Unfold** icon , the hopper is unfolded as follows:



You can modify the hopper you are creating thanks to the options available in the Hopper dialog box.

- **Invert material side**: the red arrow indicates the direction for thickening the hopper. You can change the thickening direction by clicking either this red arrow or the **Invert material side** button.
- **Neutral fiber**: the selected input surface is considered to be the hopper neutral fiber. This option proves to be useful when the offset input surface is not ruled.
- **Reference wire**: is an edge lying on the surface or a curve.
  When you right-click on the Reference Wire field, you can select other types of reference wires.
  - Create line
  - Create Intersection
  - Create Projection
  - Create Extract
- **Invariant point**: point lying on the surface and the reference wire.
  When you right-click on the **Invariant point** field, you can create you own invariant point.
- **Invert fixed side**: the green arrow indicates the fixed side for unfolding: the hopper may be unfolded from the side opposite to the arrow. You can change the unfolding direction by clicking either this green arrow or the **Invert fixed side** button.
- **Tear wire**: defines the opening line. It can be a curve going through the surface or several edges.

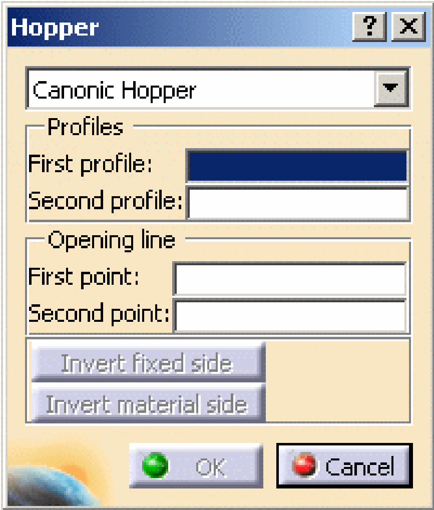
Canonic hopper

Canonic hoppers can be conical, cylindrical or planar. Their defining sketches can be on parallel planes when creating a canonical hopper. They may be open or closed; they should have similar shapes, and should be composed of arcs of circles and straight segments. For each segment or arc in a sketch, there should be a corresponding segment or arc in the other sketch. In the case of arcs, the projection of their centers onto the other sketch plane must match.

To sum up, each edge in a sketch must be an offset from a corresponding edge in the other sketch so that the resulting hopper may be conical, cylindrical or planar.

The two points defining the opening line (one point on each sketch) must have been created explicitly prior to creating the hopper. Each point should lie on a corresponding segment.

1. Click the **Hopper** icon . Select Canonic hopper in the Hopper dialog box that is displayed.



2. Select Sketch.1 as the first profile, either from the geometry area or from the specification tree.

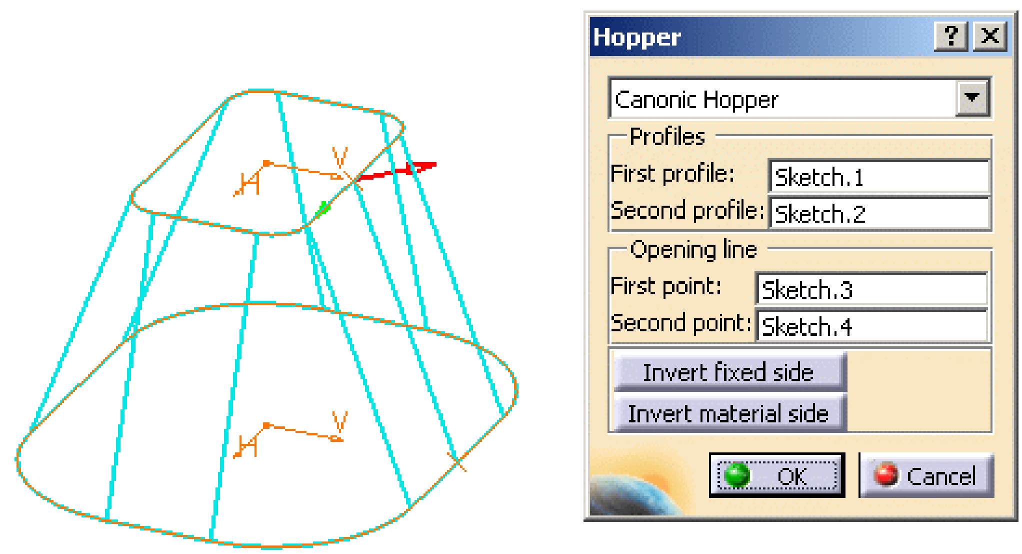
3. Select Sketch.2 as the second profile.

4. Select Sketch.3 as the first point for the opening line.

5. Select Sketch.4 as the second point for the opening line.

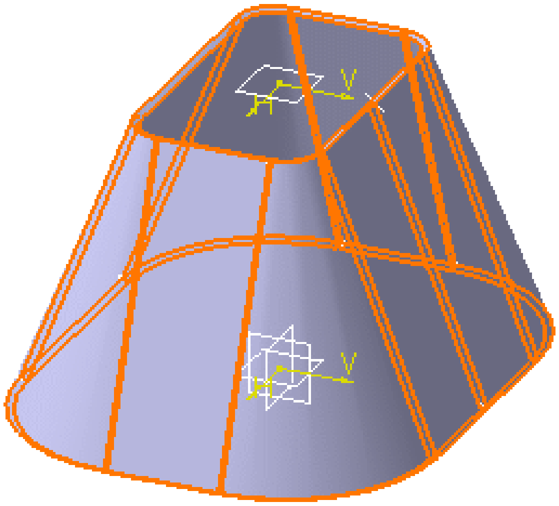
Note that the first point of the opening must be on the first profile, and the second point must be on the second profile.

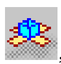
A preview of the hopper is displayed. You can notice two arrows starting from the first point.

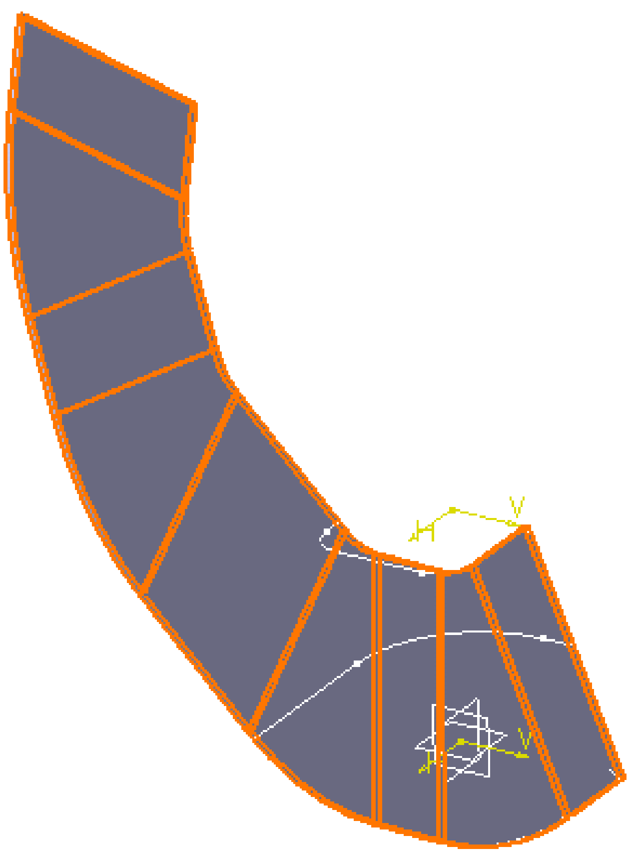


- The green arrow indicates the fixed side for unfolding: the hopper may be unfolded from the side opposite to the arrow. You can change the unfolding direction by clicking either this green arrow or the **Invert fixed side** button.
- The red arrow indicates the direction for thickening the hopper. You can change the thickening direction by clicking either this red arrow or the **Invert material side** button.

6. Click **OK** to validate and exit the dialog box. The hopper is created.



7. If you now click the **Unfold** icon , the hopper is unfolded as follows:



Unfolding

Unfolded Sheet Metal parts can be displayed in two ways:

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.

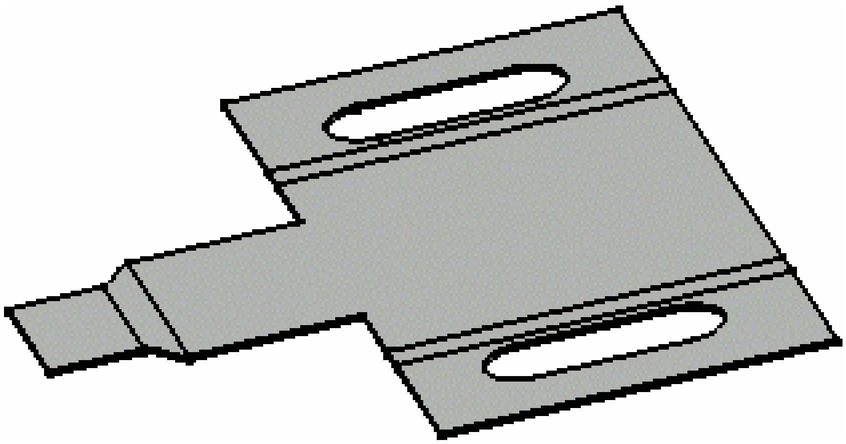
Folded/Unfolded View Access

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.

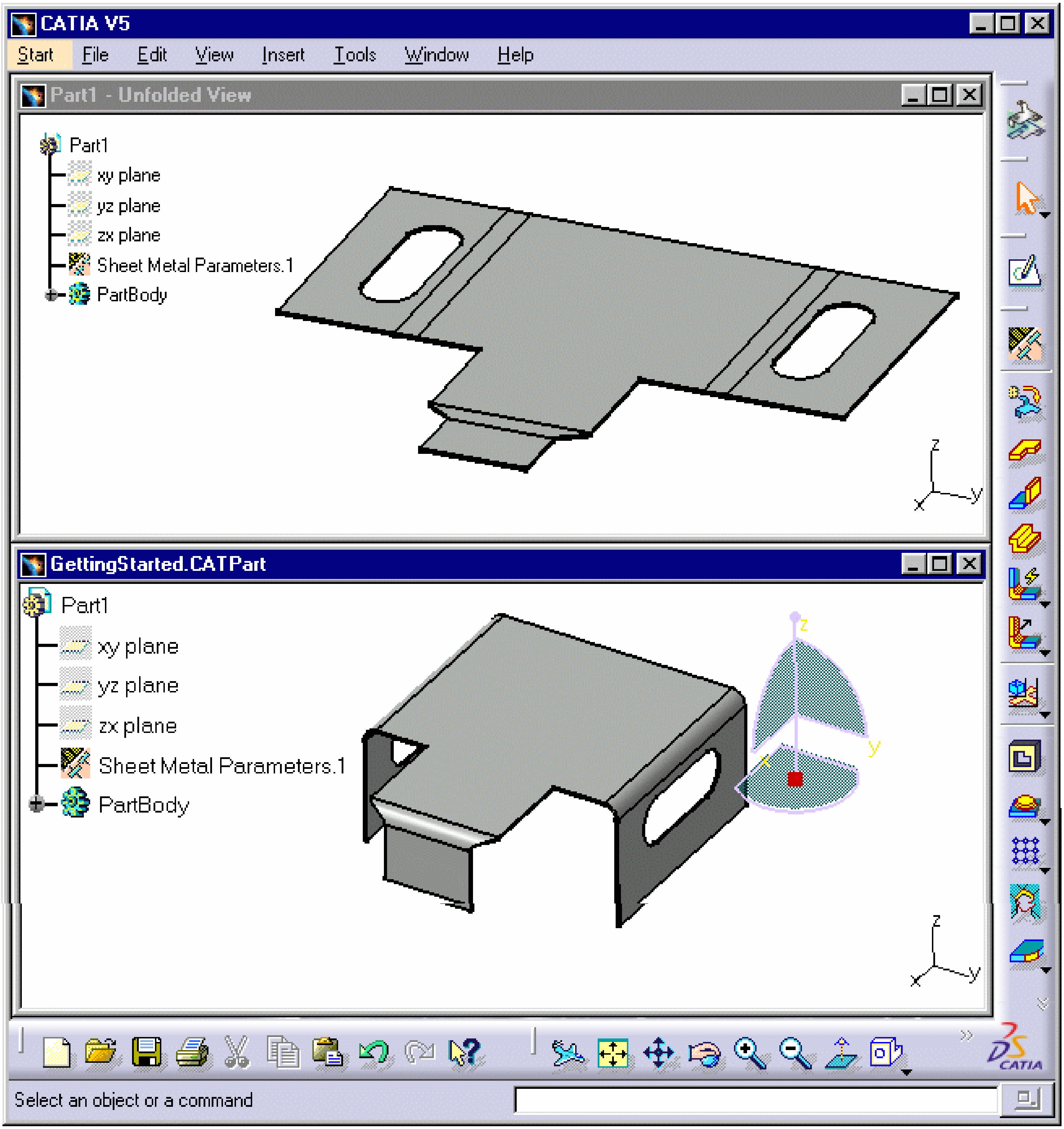
This task explains how to display the sheet metal part in two windows: one with the folded view, one with the unfolded view. Any modification in one window is displayed in the other window.

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)

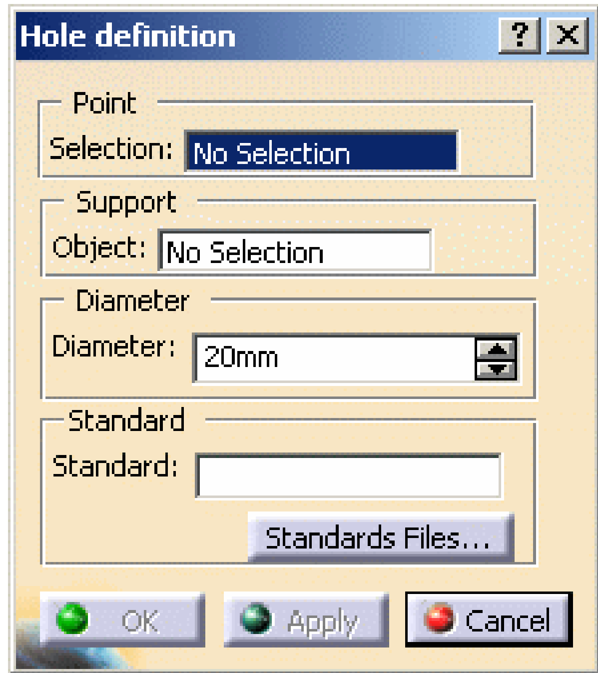
Creating a Hole

This task shows you how to create a hole, that consists in removing material from a body.

Open the Hole1.CATPart document.

1. Click the **Hole** icon .

The Hole definition dialog box opens.



2. Select the **Point** that will be the center of the hole.

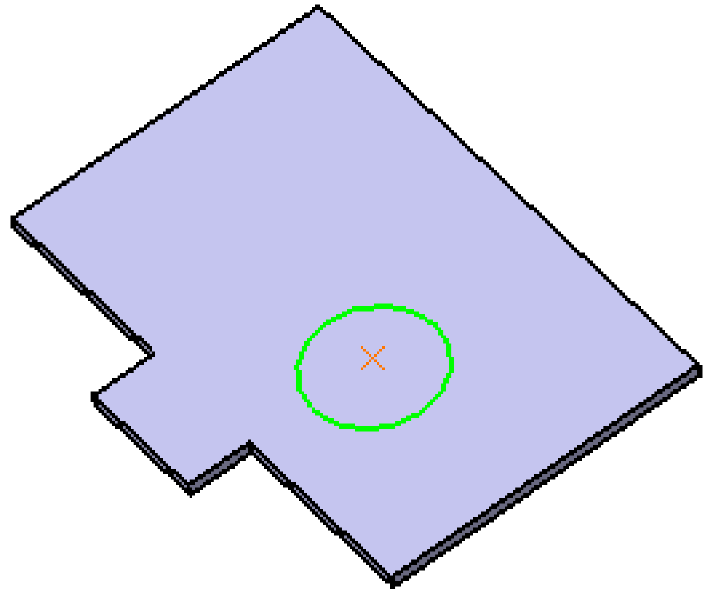
   It can be either a sketch containing one or more points, or a point, or several points. The points must be on the same support.

- The point can be selected anywhere in the geometry, not necessarily on a surface. In that case, an orthogonal projection will be performed.
- You can also directly click the surface: a point will be created under the pointer.
- To deselect a point, click it in the specification tree.

3. Select the **Support** object where the hole will be positioned.

- The support can be different from the support where the point lies. In that case, an orthogonal projection will be performed.

The hole is previewed with default parameters.

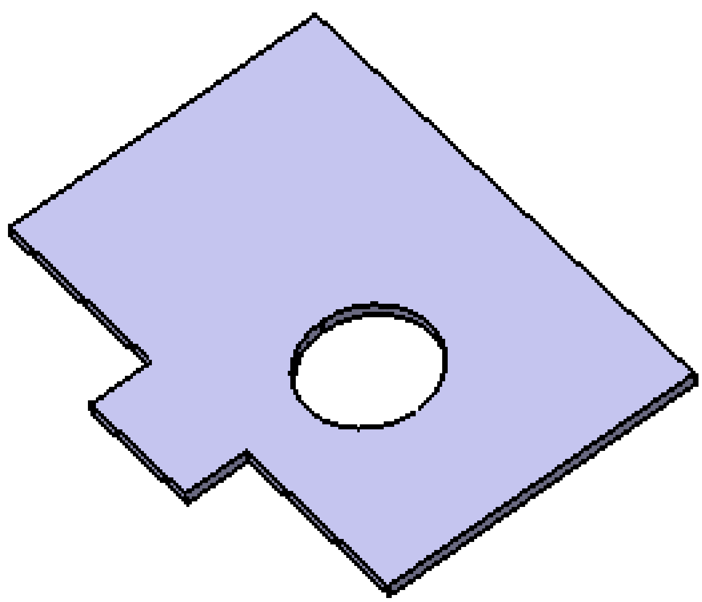


4. Define the value for the diameter of the hole in the **Diameter** field.

If you change the **Diameter** value using the spinners, the preview of the hole automatically updates. However, if you enter a value directly in the field, you need to click the **Apply** button to update the preview.

5. Click OK to validate.

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



Holes can be created on the unfolded parts and on bends.

For have further information on standard files, refer to the Customizing section.

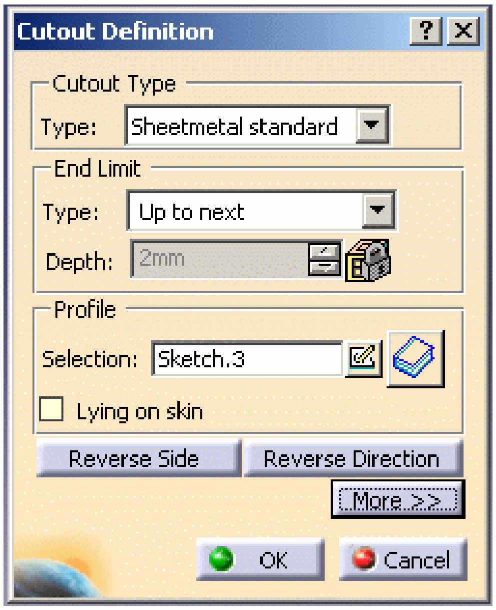
Creating a Cutout

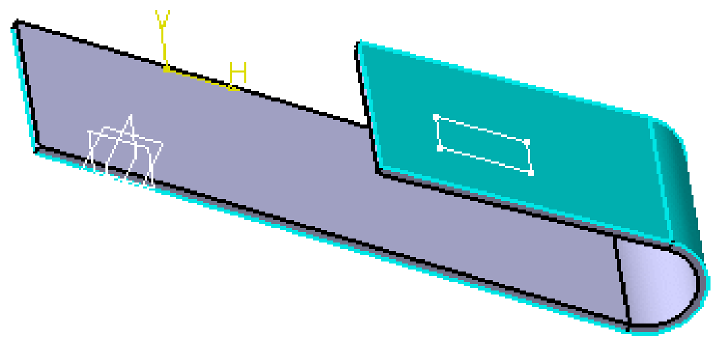
This task explains how to create a cutout in a wall.
Creating a cutout consists in extruding a profile and removing the material resulting from the extrusion.

Open the NEWCutout1.CATPart document.

1. Click the **Cutout** icon.

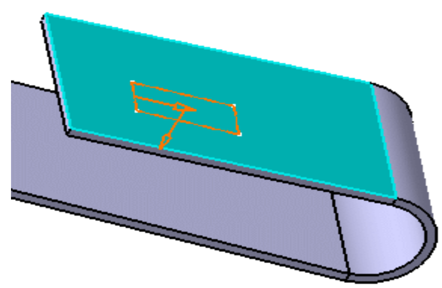
The Cutout Definition dialog box is displayed and the skin to be impacted by the cutout is displayed in a different color.





2. Select a profile (sketch.3 in our example).

A preview of the projected cutout is displayed.

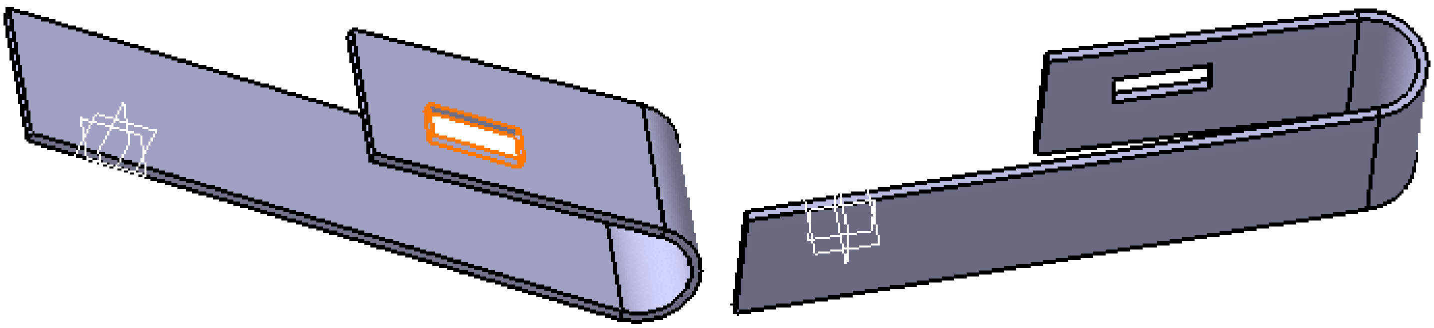


Once the sketch is selected, you can modify it by clicking the Sketcher icon.

- 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.

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

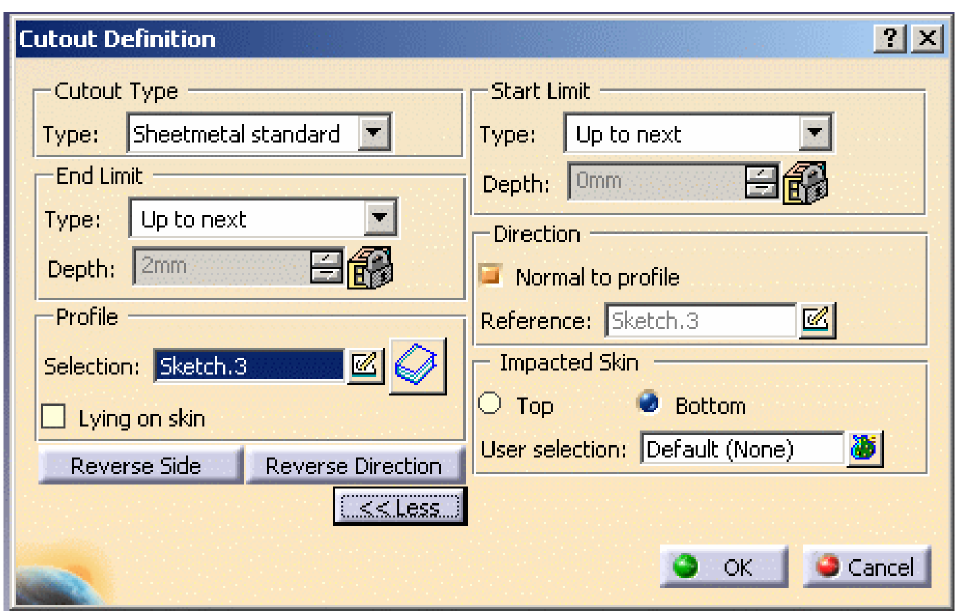
The cutout is created.



Several end limit types are available:

- **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.
- **Dimension**: the cutout depth is defined by the specified value.

4. In the specification tree, double click on Cut Out.1 to display the Cutout Definition dialog box.
5. Click **More>>** to display the maximum information.

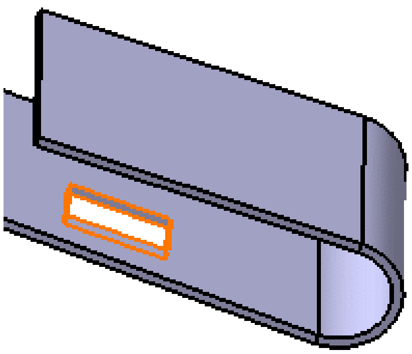


Here the Cutout's impacted skin is set to Default, that is, the surface on which lies Sketch.3

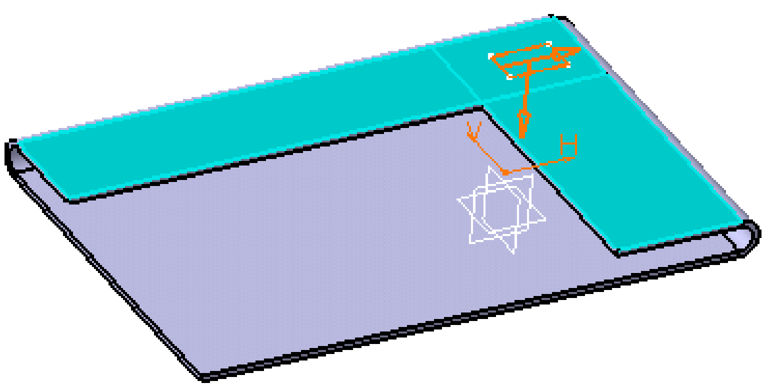
7. Click on  and select the support for the cutout.
   The **Support Selection** dialog box is displayed.
8. Select Wall.1 as your new support for the cutout.

9. Close the **Support Selection** dialog box and click OK in the **Cutout Definition** dialog box.

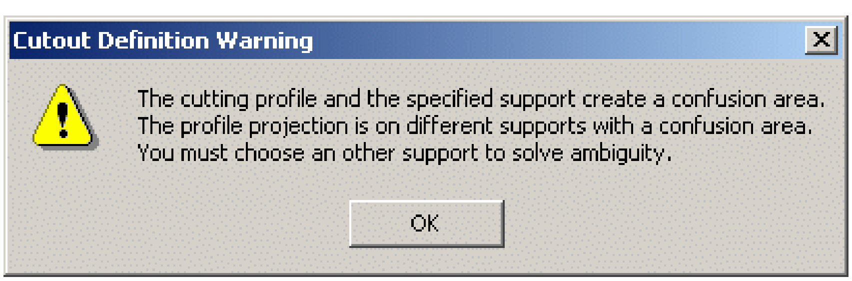
The cutout is created on Wall.1.



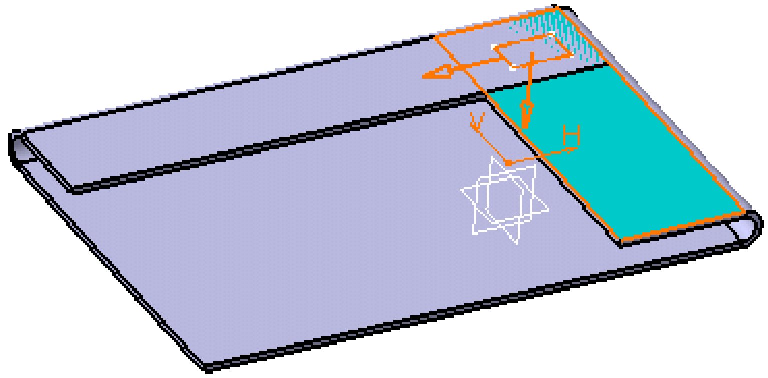
Specifying the support for the cutout avoid confusions in case of overlaps.



For instance, if you try to create a cutout on the part below, the following message is displayed:



To avoid this, you have to select the exact support for the cutout.



- When **Lying on skin** is checked,
  - The End limit and Start limit types are automatically set to Dimension and disabled,
  - The Depth is set to 0mm and disabled,
  - The skin to be impacted is displayed on the part.

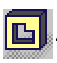
The cutout is not projected anymore on the skin. It is based on a sketch that inevitably lies on a surface.

This option is available only when creating a standard cutout.

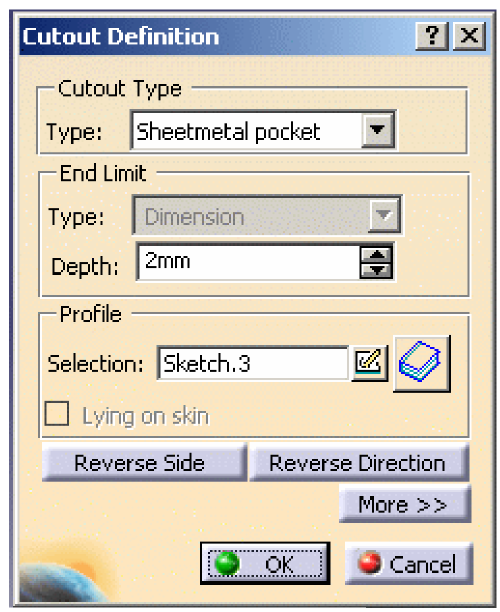
In case the prism resulting from the cutout's profile and direction is tangent to the impacted skin, the projection is non-valid and the cutout cannot be created.

To avoid this, check **Lying on skin** or select a wall as support to be able to create your cutout.

Open the NEWCutout1.CATPart document again.

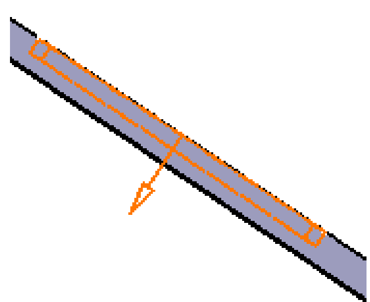
1. Click the **Cutout** icon .
2. Select **Sheetmetal pocket** as Cutout type in the combo box.

   The skin to be impacted remains grey and the End limit type is disabled.
3. Set the Depth to 1mm.



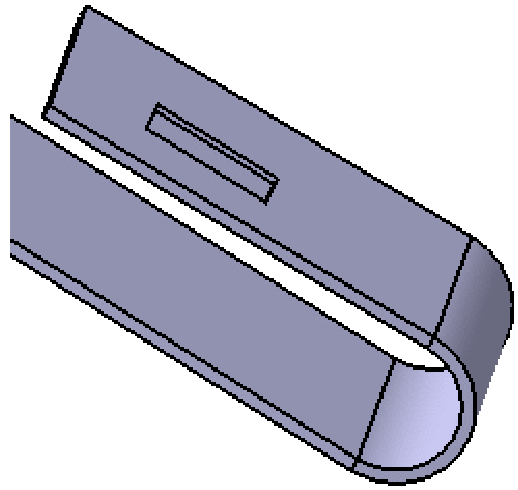
4. Select Sketch.3 as profile.

   A preview of the cutout is displayed. In our example, the cutout will impact only half the wall.



5. Click OK in the **Cutout Definition** dialog box.

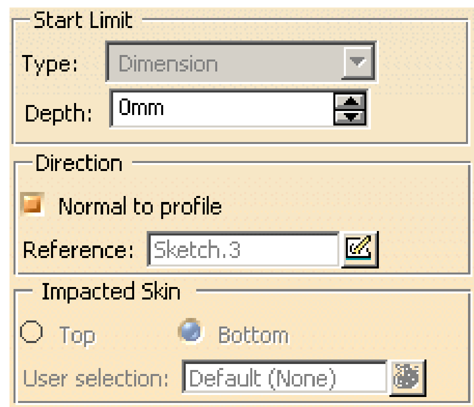
The cutout is created.



4. In the specification tree, double click on Cut Out.1 to display the Cutout Definition dialog box.

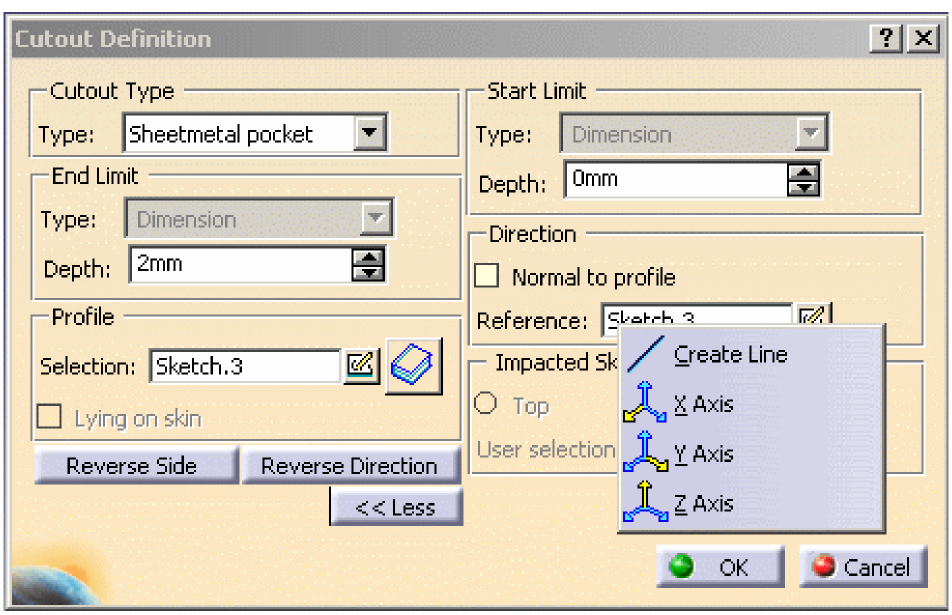
5. Click **More>>** to display the maximum information.

The Direction is already selected (Sketch.3). By default, it is set as normal to the profile.

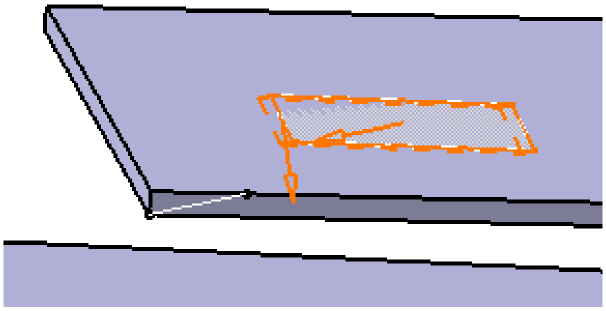


7. Uncheck **Normal to profile**.

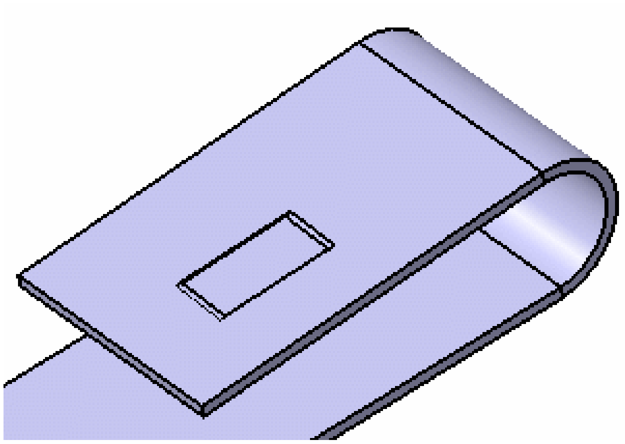
8. In the **Reference** field, right-click on Sketch.3 and select Create line.



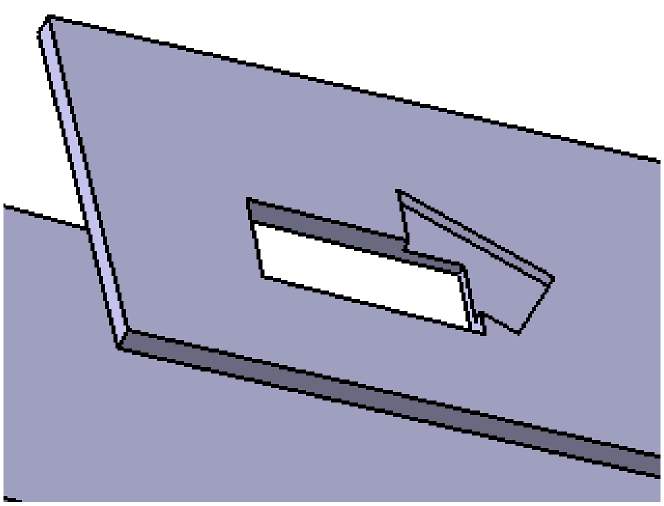
9. Select the line to perform a cutout normal to the line direction.



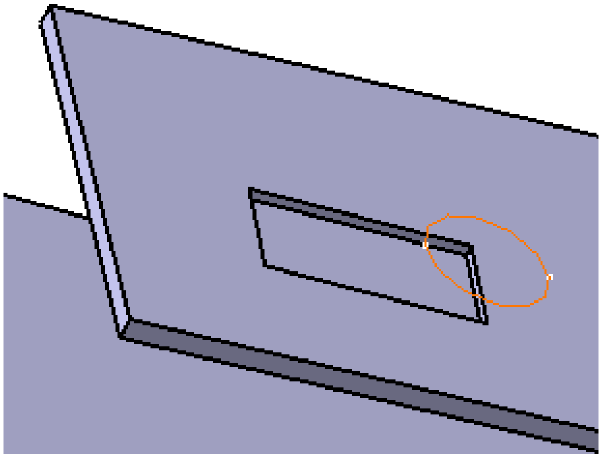
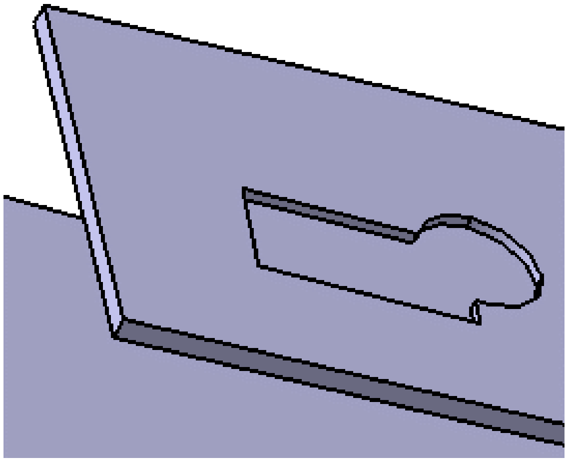
10. Click on OK to create the cutout normal to the line direction.



- The pocket cutout can be created only on a planar and monosupport surface (i.e. a wall or the planar face of a flange).
- 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 (as shown in the picture above), or unfold the part to create the cutout.
- You cannot create a pocket cutout on a stamp or a surfacic flange.
- You cannot create
  - a standard cutout on a pocket cutout
  - a standard cutout on a feature impacting a pocket cutout.
- You can create
  - a pocket cutout on a standard cutout.



  - a pocket cutout on a pocket cutout,

You can use the Catalog icon to open the Catalog Browser.

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

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

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 flanged hole: select a point on a face, and set the stamping parameters.

Create a bead: select a profile, and set the stamping parameters.

Create a circular stamp: select a point on a face, 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 flanged cutout: select a profile, and set the stamping parameters.

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

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

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

Creating a Flanged Hole

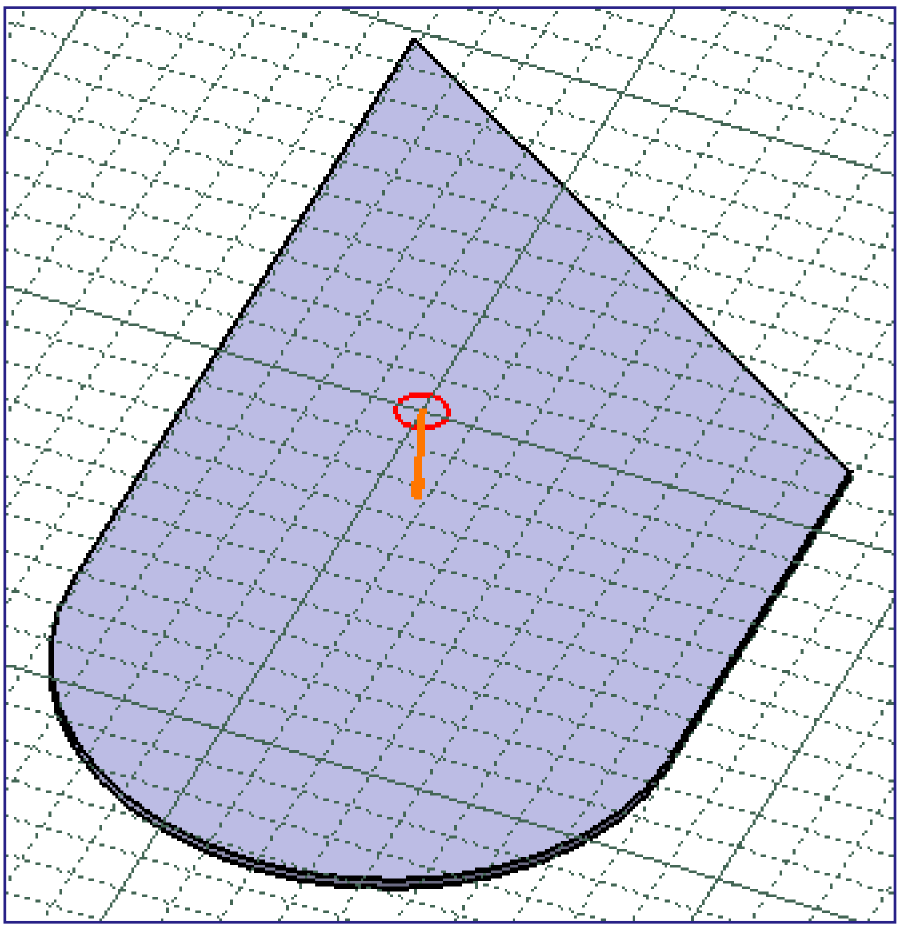
This task shows you how to create a flanged hole by specifying the punch geometrical parameters.

For the Generative Sheetmetal Design workbench, open the NEWStamping.CATPart document.
For the Aerospace SheetMetal Design workbench, open the Aero_Stamping.CATPart document.

1. Click the **Flanged Hole** icon .

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

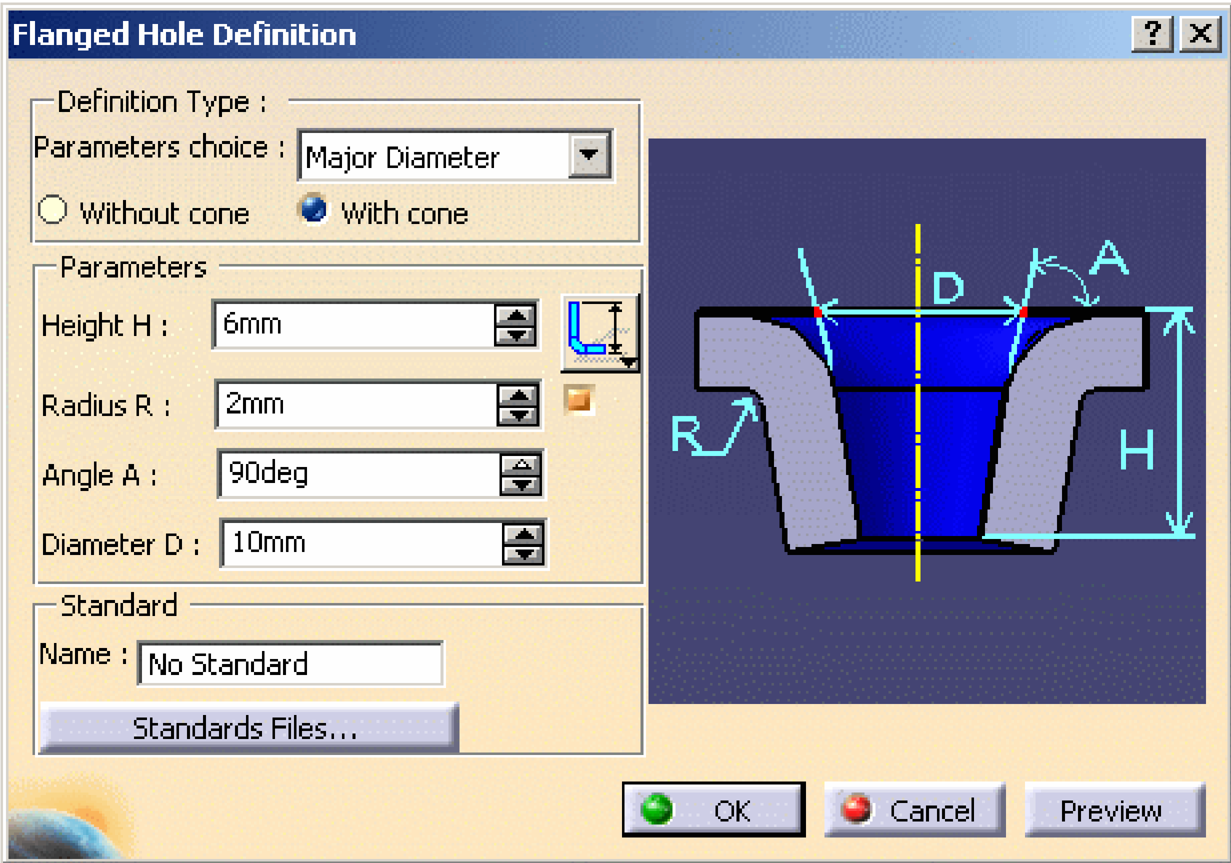
   A grid is displayed to help you position the flanged hole.



The Flanged Hole Definition dialog box is displayed, providing default values.

Note that the image in the right-hand pane of the dialog box is updated as you choose your parameters and options, and provides a graphical explanation about the current selection.

Also note that the options available in the dialog box are updated according to the items selected in the **Definition Type** area.

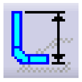
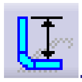


3. Choose the diameter that should be dimensioned from the **Parameters choice** list:
   - **Major Diameter**
   - **Minor Diameter**
   - **Two Diameters** (major and minor diameters)
   - **Punch & Die**
4. Specify whether the flanged hole should be created without a cone (i.e. only with the filleted portion of the flanged hole) or with a cone (i.e. with the filleted portion of the flanged hole and with a cone).

Note that selecting the **Without cone** option has the following consequences:

- The **Height H** field is disabled, the height being automatically computed in this case.
- Deactivating the **Radius** field is impossible, because the radius value for the flanged hole external curvature must be specified in this case.

5. If you want to use a standard, click the **Standard File** button and browse to select a standard file. In this case, the standard parameters will be used, and you do not need to specify the flanged hole parameters. You can skip the next step.
6. Choose the flanged hole parameters:

- In the **Height H** field, specify the height value for the flanged hole. Use the icon next to the field to specify the reference from which the height is defined:  or .
- In the **Radius R** field, specify the radius value for the flanged hole external curvature. Use the icon next to the field to disable this option.
- In the **Angle A** field, specify the angle value for the flanged hole.

  This option is not available for the **Two Diameters** or **Punch & Die** parameters, as the angle is automatically computed in these cases.

- In the **Diameter D** field, specify the major diameter value for the flanged hole.

  This option is not available for the **Minor Diameter** parameter, as the major diameter is automatically computed in this case.

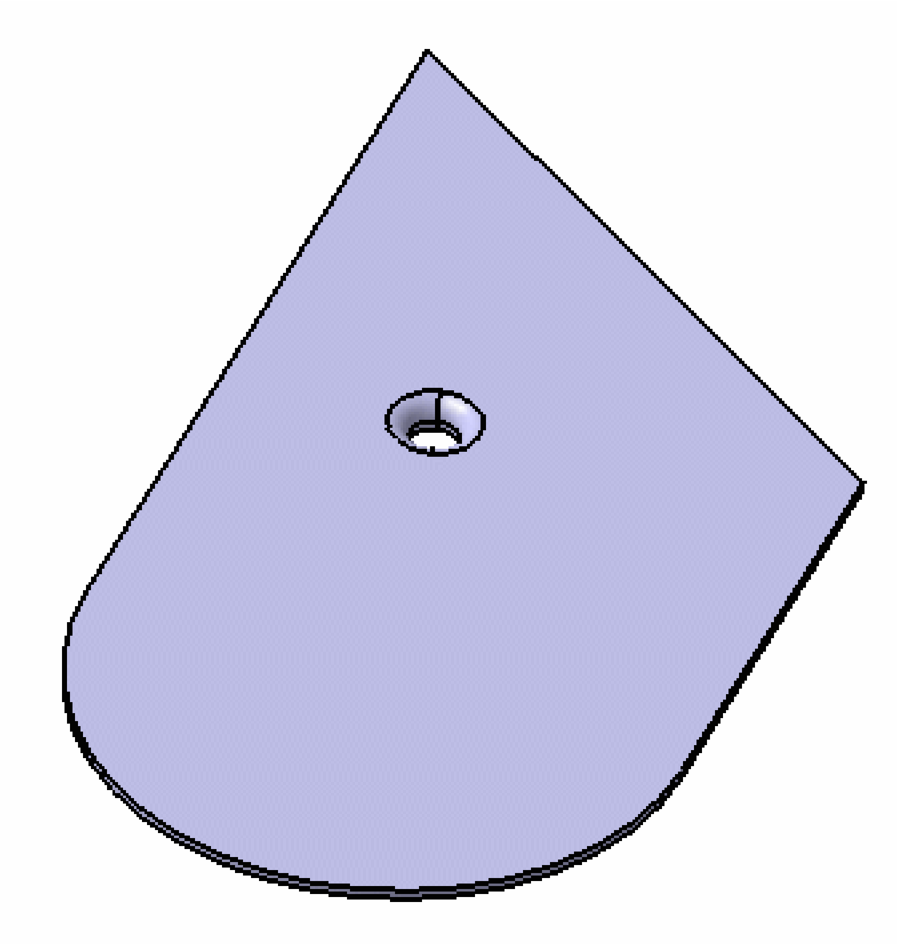
- In the **Diameter d** field, specify the minor diameter value for the flanged hole.

  This option is not available for the **Major Diameter** parameter, as the minor diameter is automatically computed in this case.

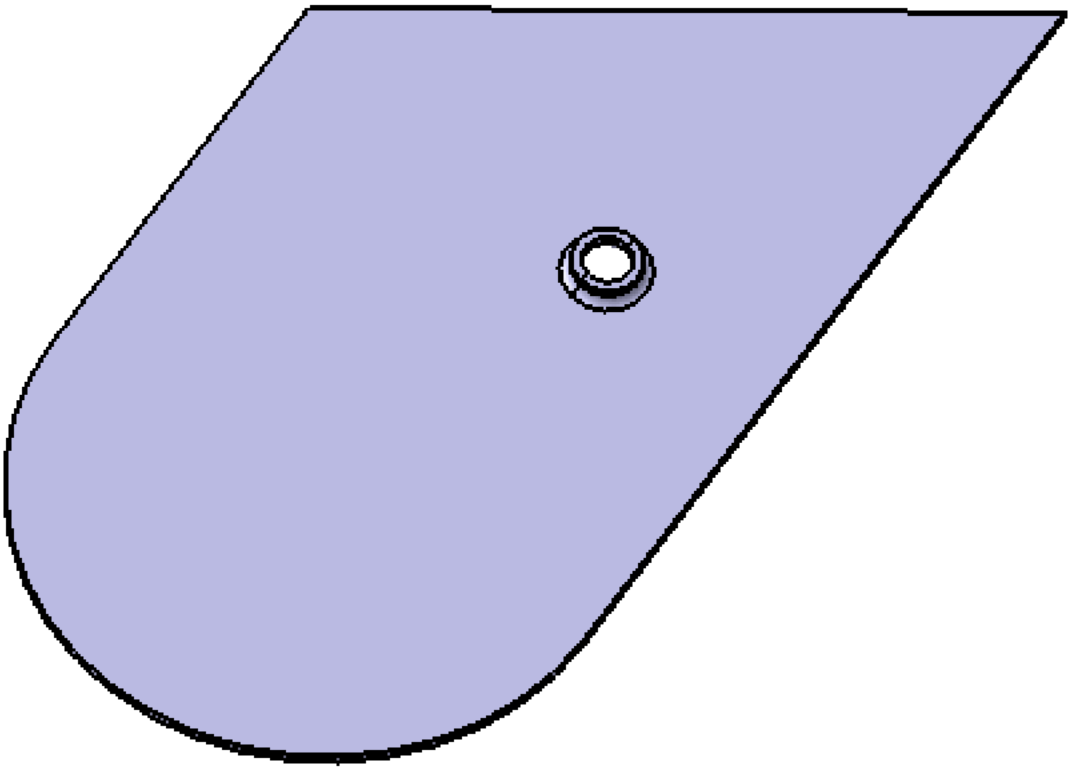
7. Click **Preview** to visualize the flanged hole.

8. Click **OK** to validate.

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



Flanged hole viewed from the front



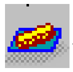
Flanged hole viewed from the back

Refer to the Customizing Standard Files chapter for more information about defining the Standards Files.

Creating a Bead

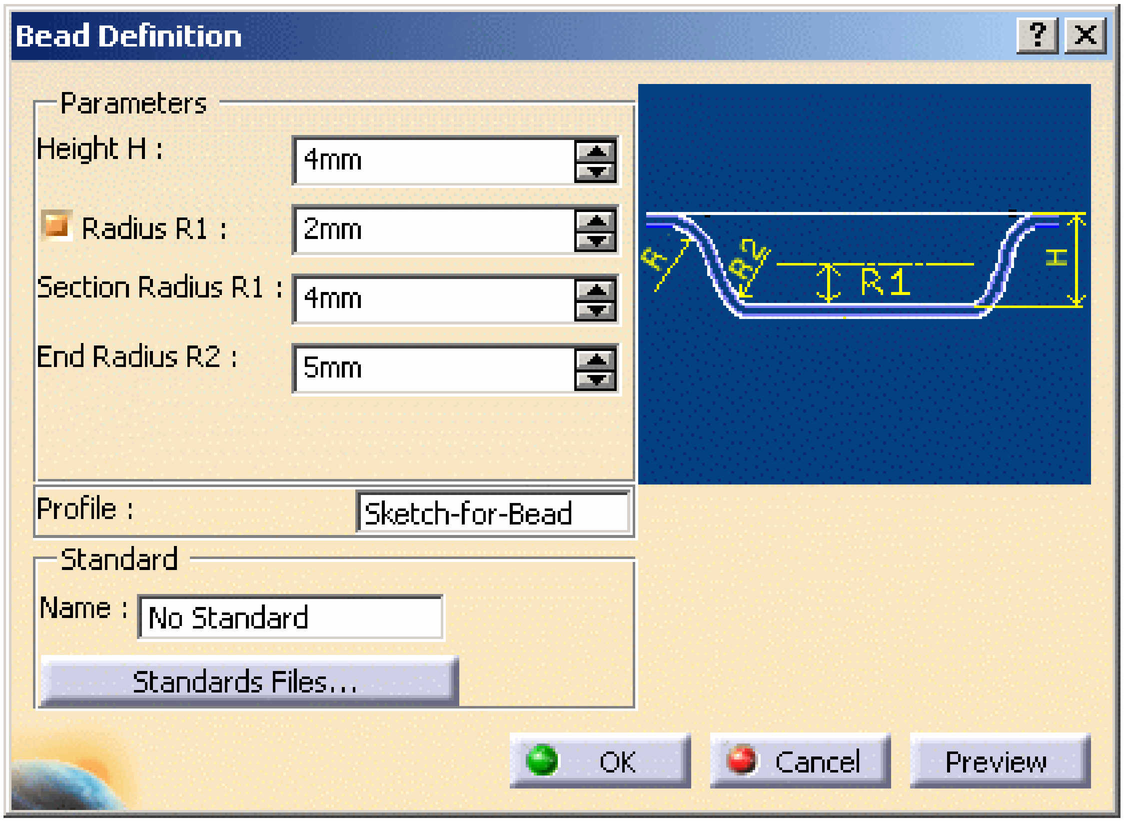
This task shows you how to create a bead, that is a local deformation in the web.

Open the NEWStamping6.CATPart document.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping6.CATPart document.

1. Click the **Bead** icon .

2. Select the spine profile where you want to place the bead.

The Bead definition dialog box is displayed, providing default values.

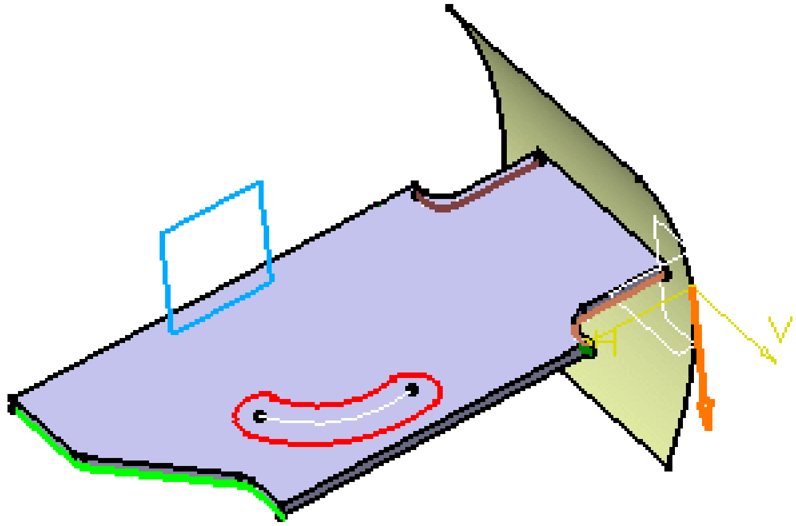


3. Change the value in the different fields, if needed:

- Height H
- Radius R
- Section Radius R1 (corresponding to the cross section value)
- End Radius R2

The **Sketch** is automatically set to the sketch you chose.

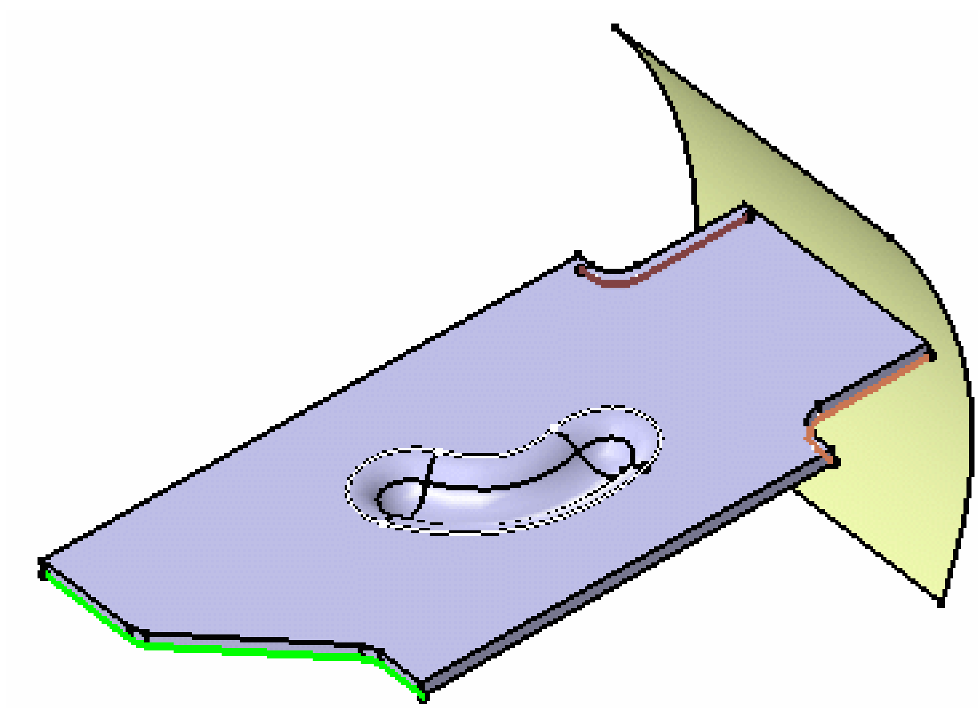
The vector for the direction of the bead is shown in the model and a preview of the bead appears and a vector shows its direction.



4. Click **Preview** to visualize the bead.

5. Click **OK** to validate.

The bead (identified as Bead.xxx) is created and the specification tree is updated accordingly.



The vector cannot be reverted until the bead spine is defined.

You can use 0 as the Radius value to deactivate the Radius R value, and to create the bead without a fillet.

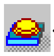
Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Circular Stamp

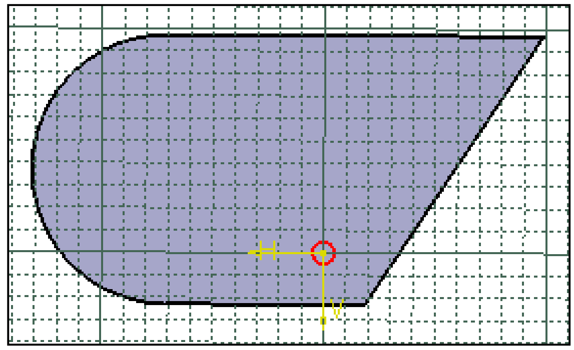
This task shows you how to create a point stamp by specifying the punch geometrical parameters.

Open the NEWStamping.CATPart document from the samples directory.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping.CATPart document.

You have now the choice between several parameters to dimension the diameter of your circular stamp.

1. Click the **Circular Stamp** icon .

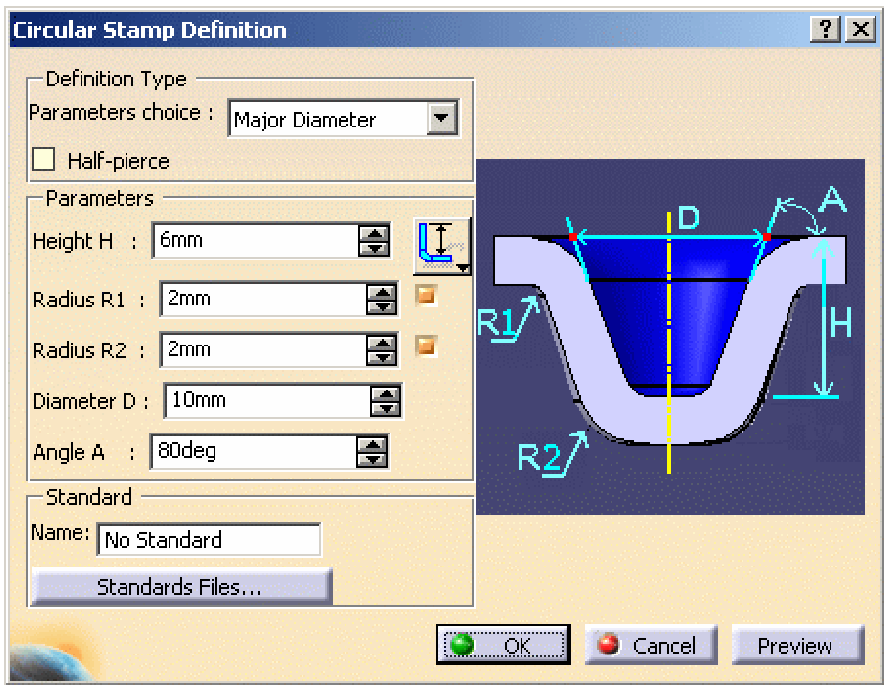
2. Select a point on the top face.



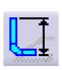
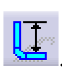
A grid is displayed to help you position the circular stamp.

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

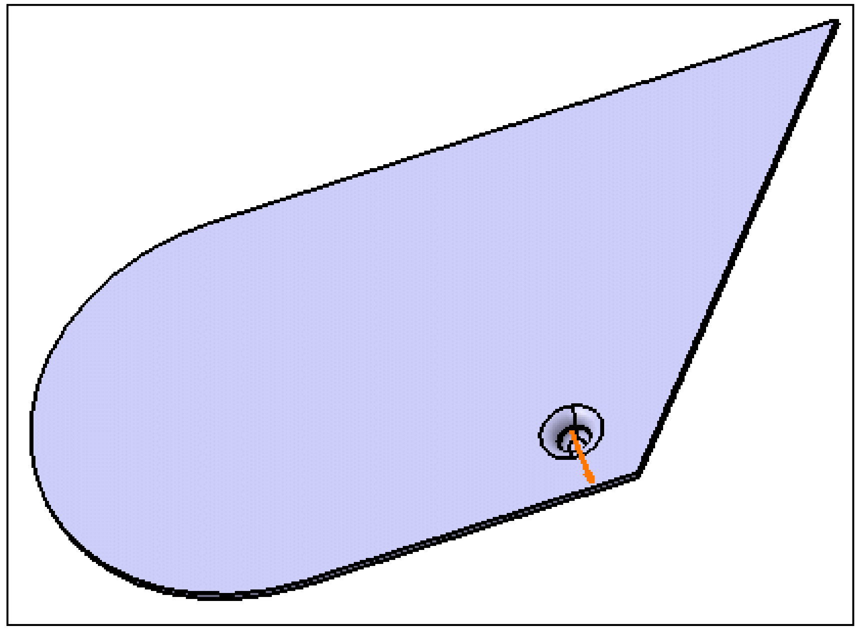
3. Choose the diameter that should be dimensioned from the **Parameters choice** list:
   - **Major Diameter**
   - **Minor Diameter**
   - **Two Diameters** (major and minor diameters)
   - **Punch & Die**



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

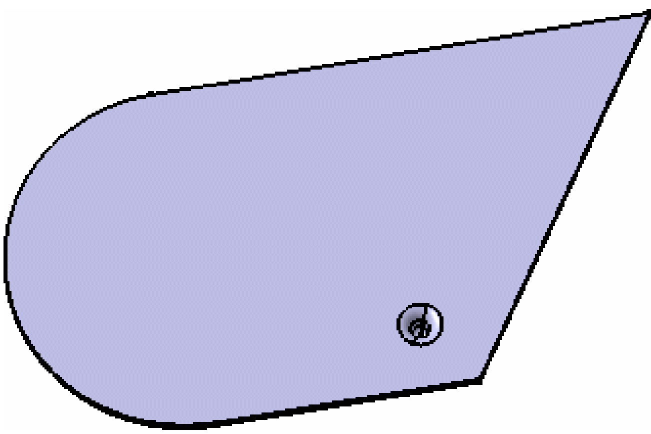
- Height H: use the icon next to the field to specify the reference from which the height is defined:  or .
- Radius R1
- Radius R2
- Angle A
- Diameter D

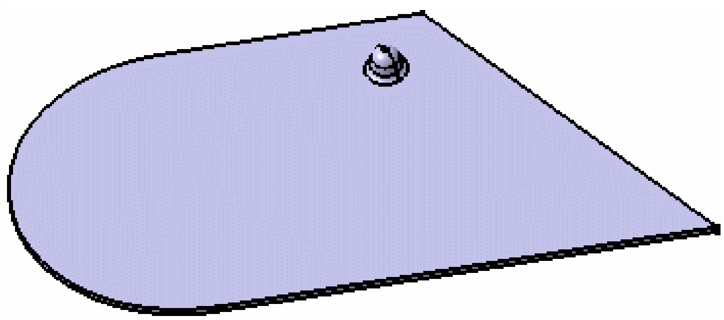
4. Click **Preview** to visualize the circular stamp.



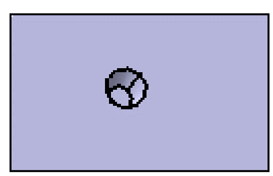
5. Click **OK** to validate.

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





To create the point stamp without a fillet, unselect the Radius R1 and Radius R2 checkbox in the Circular Stamp Definition dialog box.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Surface Stamp

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

Open the NEWStamping4.CATPart document from the samples directory.

If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping4.CATPart document.

Stamps can be based on different types of profile:

- a profile containing a sketch with several inner contours
- a profile containing a 3D curve sketch
- a profile containing a punch and die sketch intersecting with the part

You can now apply parameters, such as height, radius, angle etc. on the top or on the bottom of the surface stamp:

- When selecting , parameters are applied to the top of the surface stamp
- When selecting , parameters are applied to the bottom of the surface stamp

You can now create Half pierce stamps. For more information, refer to the Creating a Half Pierce Stamp section.

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

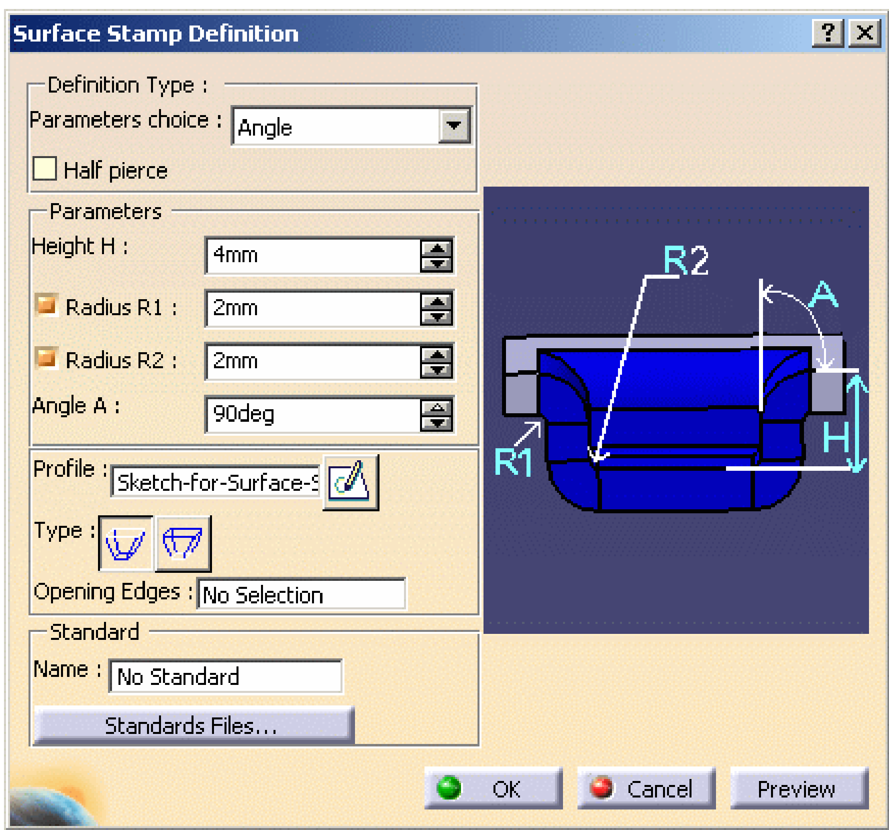
2. In the specification tree, select **Sketch-for-Surface-Stamp**, the profile previously defined.

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

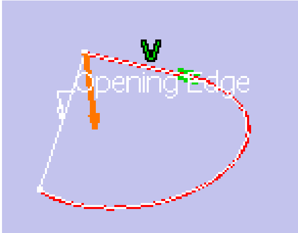
3. Change the value in the different fields, if needed.

   In our example, we chose the following values:

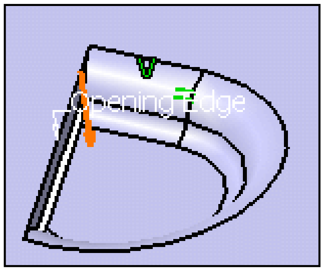
- Height H: 4mm
- Radius R1: 2mm
- Radius R2: 2mm
- Angle A: 90deg



4. Click in the **Opening Edges** field and select a sketch's edge.

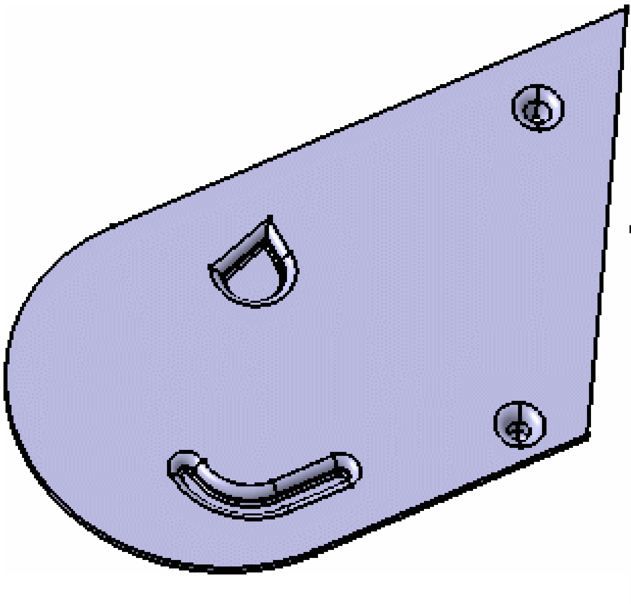


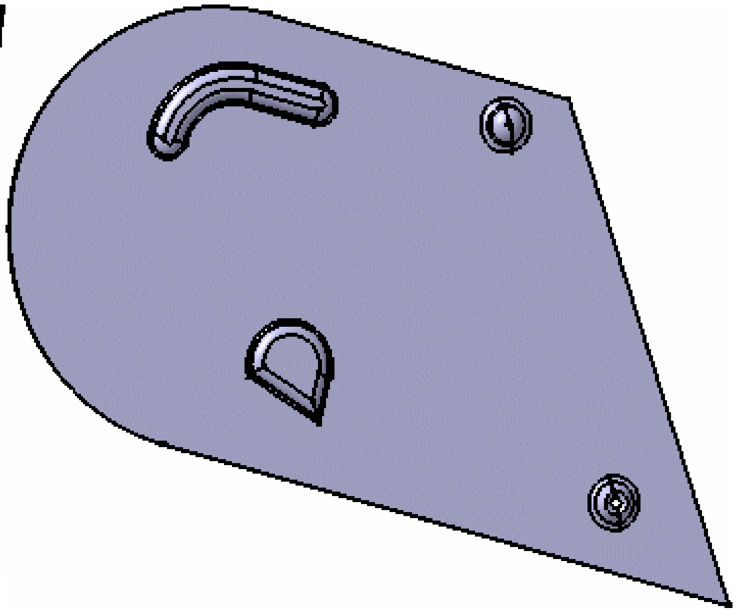
5. Click **Preview** to visualize the surface stamp with an opening edge.



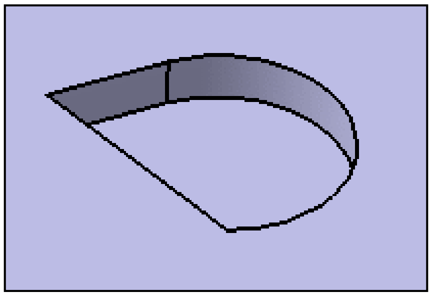
6. Click **OK** to validate.

The surface stamp (identified as Surface Stamp.1) is created and the specification tree is updated accordingly.

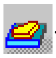




You can disable Radius R1 and Radius R2 if you want to create the surface stamp without a fillet.



Now let's create a stamp of type 1 based on a profile containing several inner contours: one stamp in a direction and other stamps in opposite direction.

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

2. In the specification tree, select **Sketch-multicontour-type1**

   The Surface Stamp Definition dialog box opens.

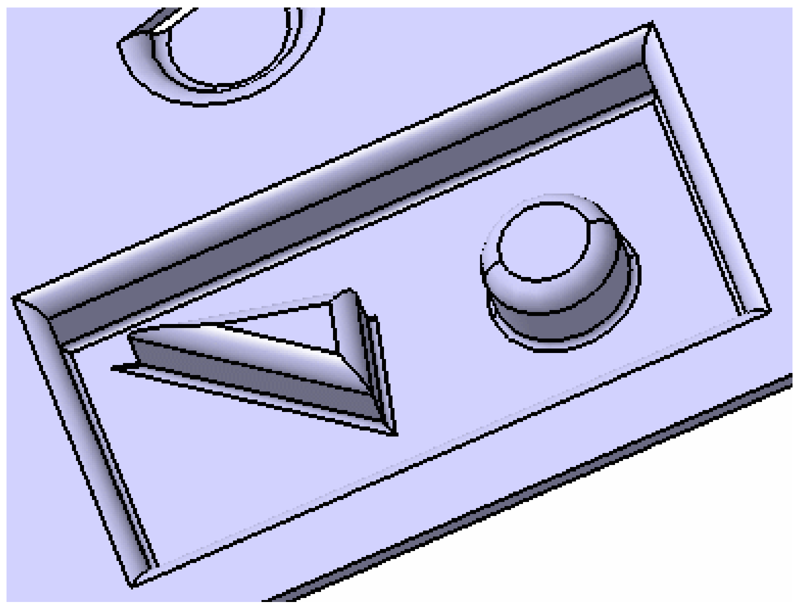
3. Select Angle as Definition type and the first type of stamp 

4. Change the value in the different fields, if needed.

   In our example, we chose the following values:

- Height H: 10mm
- Radius R1: 1mm
- Radius R2: 1mm
- Angle A: 90deg

5. Click **Preview** to visualize the surface stamp.



6. Click **OK** to validate.

The surface stamp with several inner contour (identified as Surface Stamp.2) is created and the specification tree is updated accordingly.

Now, let's create another stamp based on a profile containing several inner contours of type 2.

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

2. In the specification tree, select **Sketch-multicontour-type2**

   The Surface Stamp Definition dialog box opens.

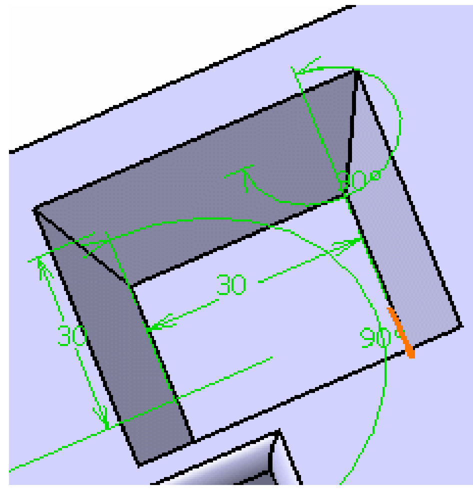
3. Select Angle as Definition type and the second type of stamp 

4. Change the value in the different fields, if needed.

   In our example, we chose the following values:

- Height H: 13mm
- Radius R1: disabled
- Radius R2: disabled
- Angle A: 60deg

5. Click **Preview** to visualize the surface stamp.



6. Click **OK** to validate.

The surface stamp with several inner contour (identified as Surface Stamp.3) is created and the specification tree is updated accordingly.

Now, let's create a stamp based on a profile containing 2 contours (one inside the other)

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

2. In the specification tree, select **Sketch-punch&die**

   The Surface Stamp Definition dialog box opens.

3. Select Punch and Die as Definition type.

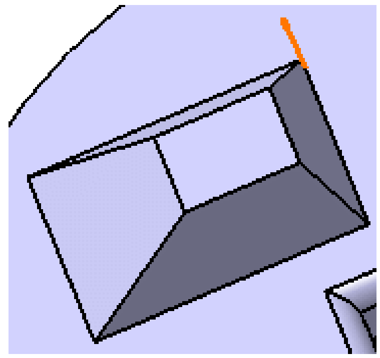
 When **Punch and Die** is selected, only the second type of stamp is enabled 

4. Change the value in the different fields, if needed.

   In our example, we chose the following values:

- Height H: 13mm
- Radius R1: disabled
- Radius R2: disabled
- Angle A: disabled

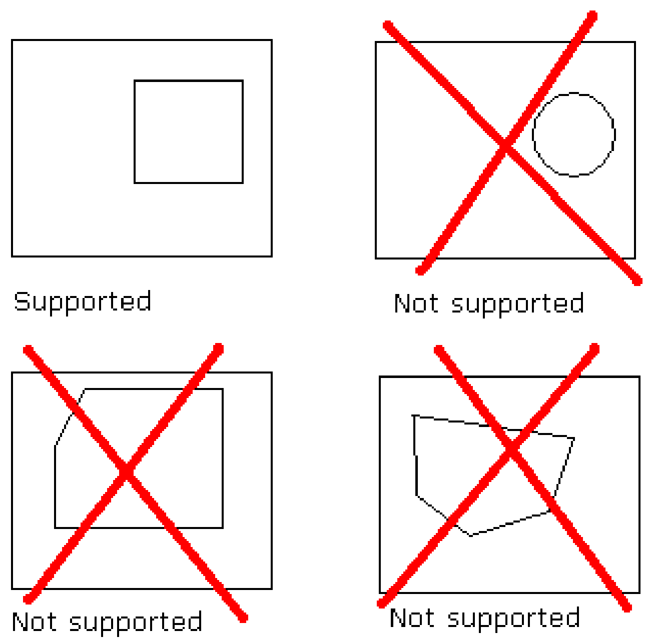
5. Click **Preview** to visualize the surface stamp.



6. Click **OK** to validate.

The surface stamp with two contours (identified as Surface Stamp.4) is created and the specification tree is updated accordingly.

Each punch edge must be parallel to the corresponding die edge.



Now, let's create a stamp based on a profile containing a 3D curve.

To do this, open the XXX.CATPart document from the samples directory.
If you use the Aerospace SheetMetal Design workbench, open the XXX.CATPart document.

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

2. In the specification tree, select **Folded curve.1**.

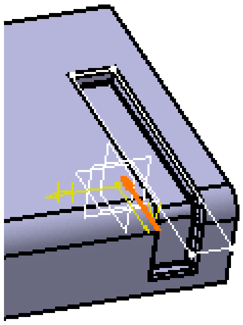
The Surface Stamp Definition dialog box opens.

Make sure that Angle is selected as parameter, as well as the first type of stamp.

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

- Height H:20mm
- Radius R1: 2mm
- Radius R2: 2mm
- Angle A: 90deg

4. Click **Preview** to visualize the surface stamp.

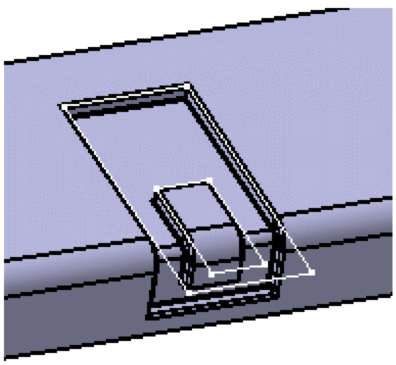


5. Click **OK** to validate.

The surface stamp with a 3D curve profile (identified as Surface Stamp.xxx) is created and the specification tree is updated accordingly.

You can also create a stamp based on a 3D multicurve profile.

For instance, you can create a surface stamp based on the Folded curve.5 and obtain the following result:



Avoid as much as possible a coincidence between the edge of the sketch profile and the edge of the wall. Instead, let the sketch profile exceed the edge of the wall.

Insert a screen capture

Creating a Bridge

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

Open the NEWStamping5.CATPart document from the samples directory.

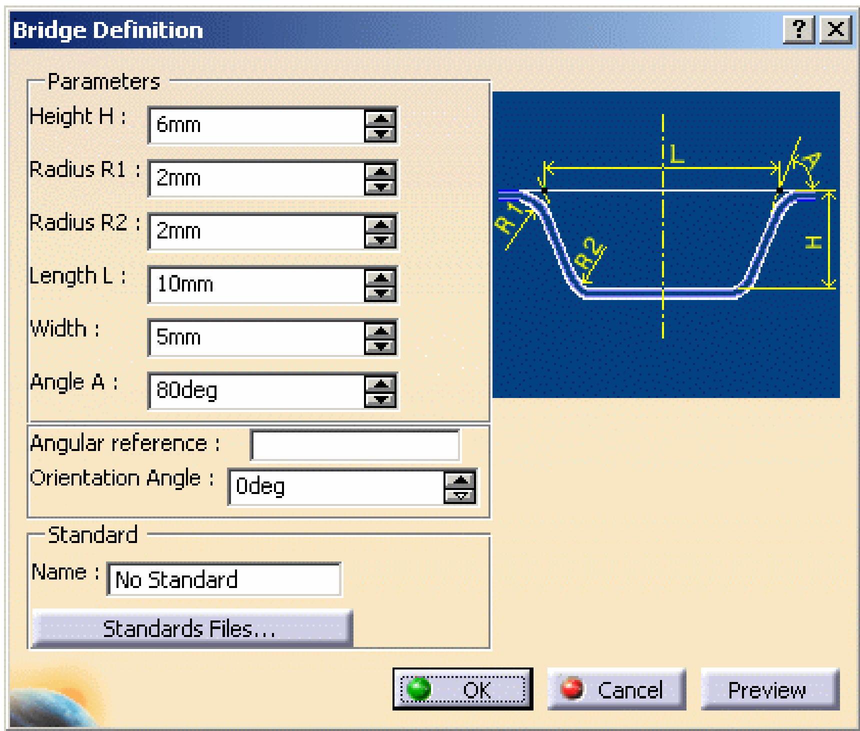
1. Click the **Bridge** icon .

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

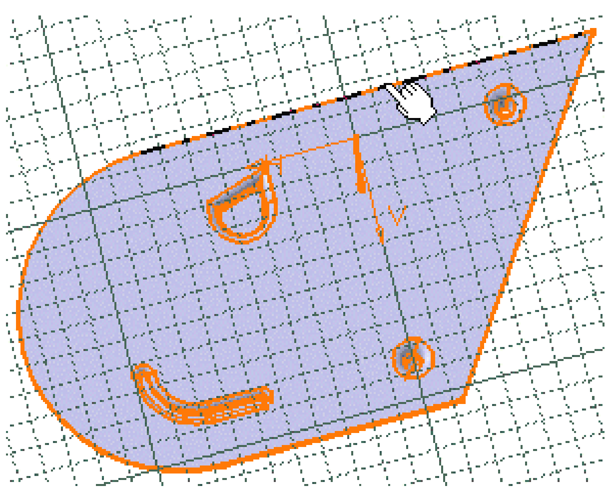
The Bridge Definition dialog box opens, providing default values.

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

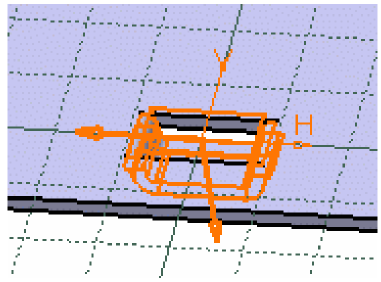
- Height H
- Radius R1
- Radius R2
- Angle A
- Length L1
- Length L2



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

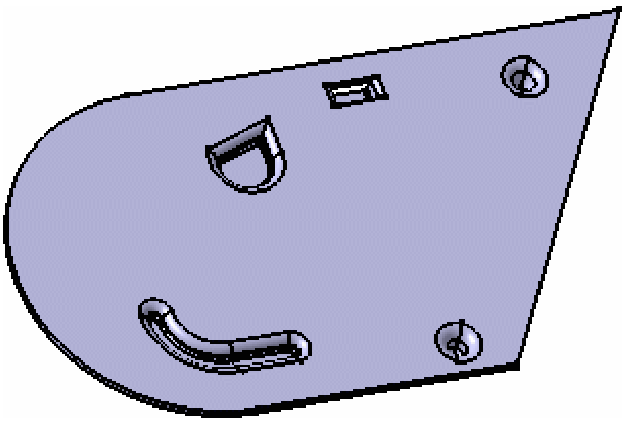


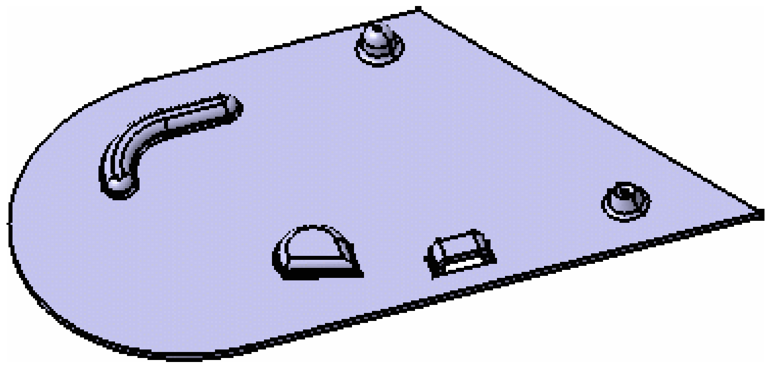
5. Click **Preview** to visualize the bridge.



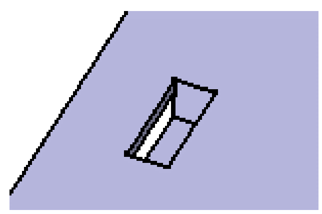
6. Click **OK** to validate.

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





You can use 0 as the Radius value to deactivate the Radius R value, and to create the bridge without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Flanged Cutout

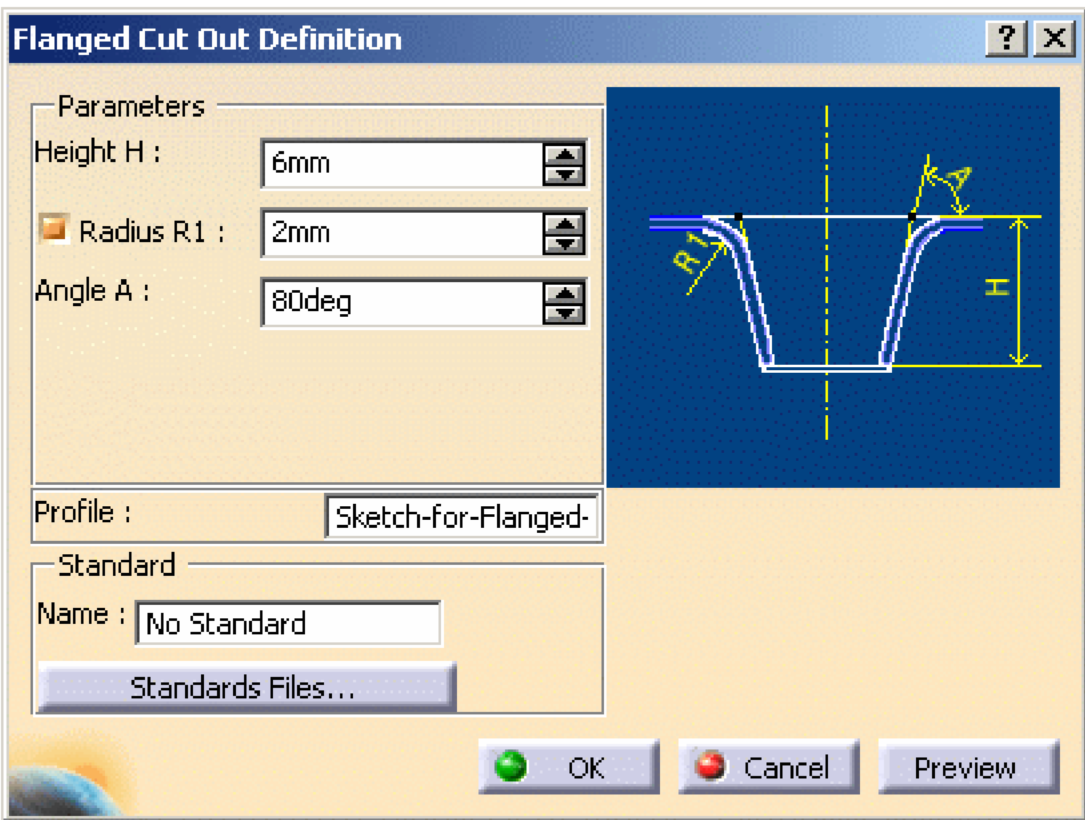
This task shows you how to create a flanged cutout by specifying the punch geometrical parameters.

Open the NEWStamping8.CATPart document.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping8.CATPart document.

1. Click the **Flanged Cutout** icon .

2. Select a profile.

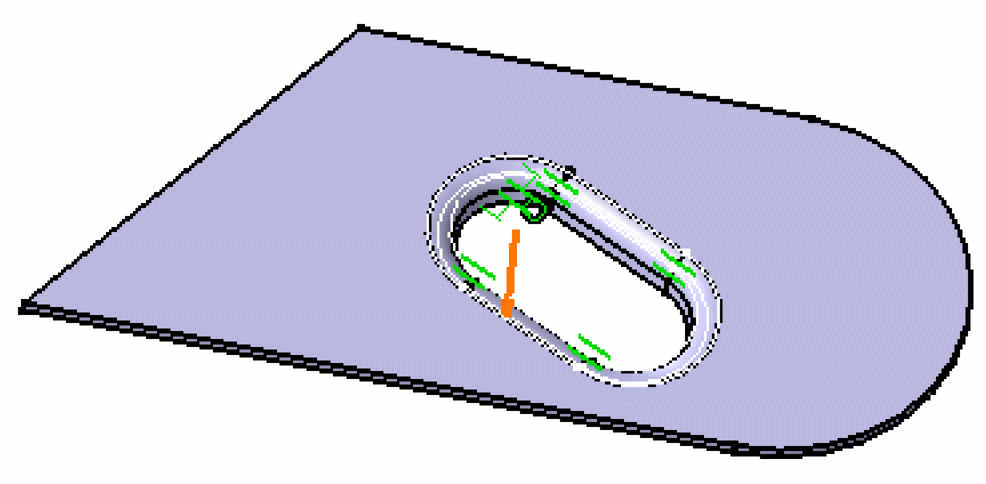
The Flanged Cutout Definition dialog box is displayed, providing default values.



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

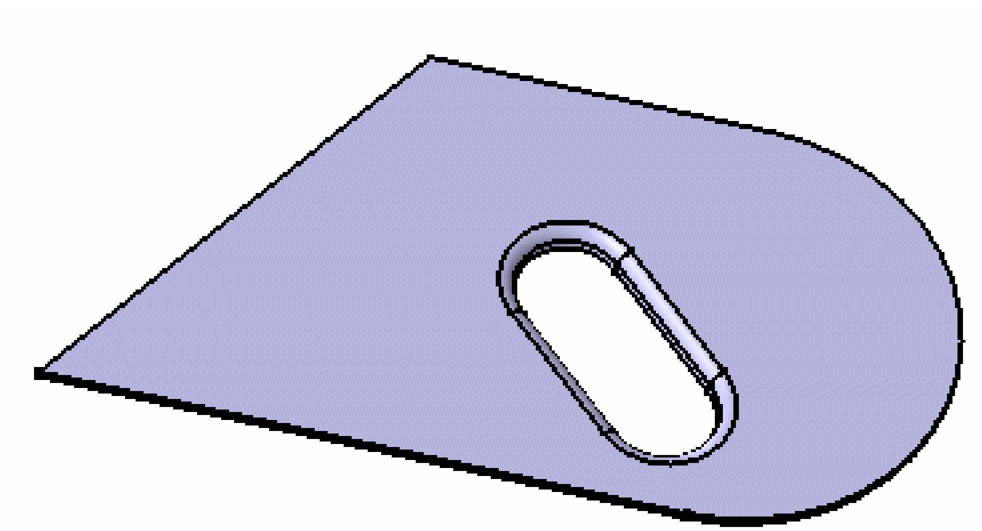
- Height H
- Radius R
- Angle A

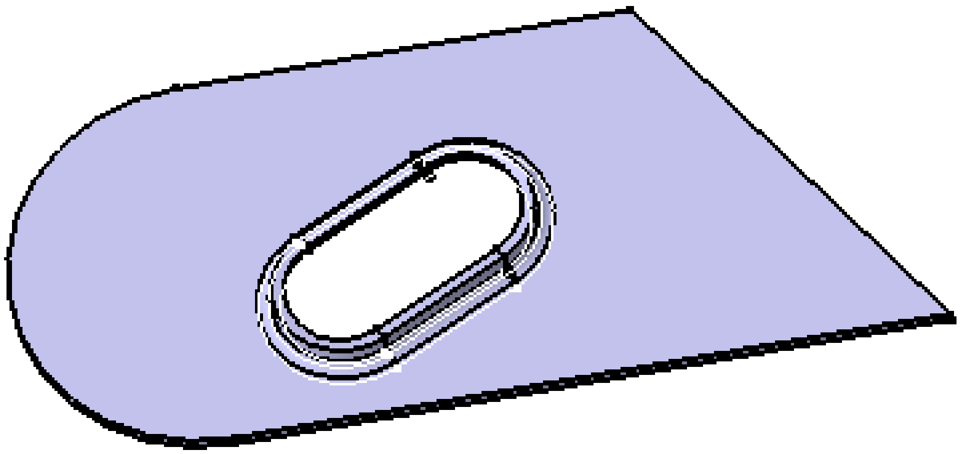
4. Click **Preview** to visualize the flanged cutout.



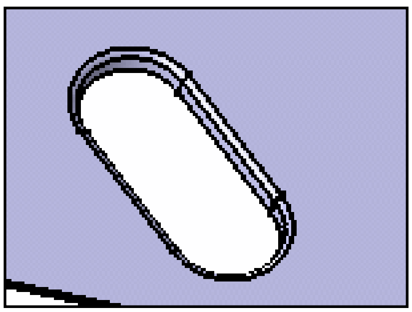
5. Click **OK** to validate.

The flanged cutout (identified as Flanged Cutout.xxx) is created and the specification tree is updated accordingly.





You can use 0 as the Radius value to deactivate the Radius R value, and to create the flanged cutout without a fillet.



Note that if you create a flanged cutout from a sketch that is not tangent continuous, you cannot design any other feature on it (such as bend, cutout, hole).

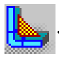
Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Stiffening Rib

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

Open the NEWStamping7.CATPart document from the samples directory.

If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping7.CATPart document.

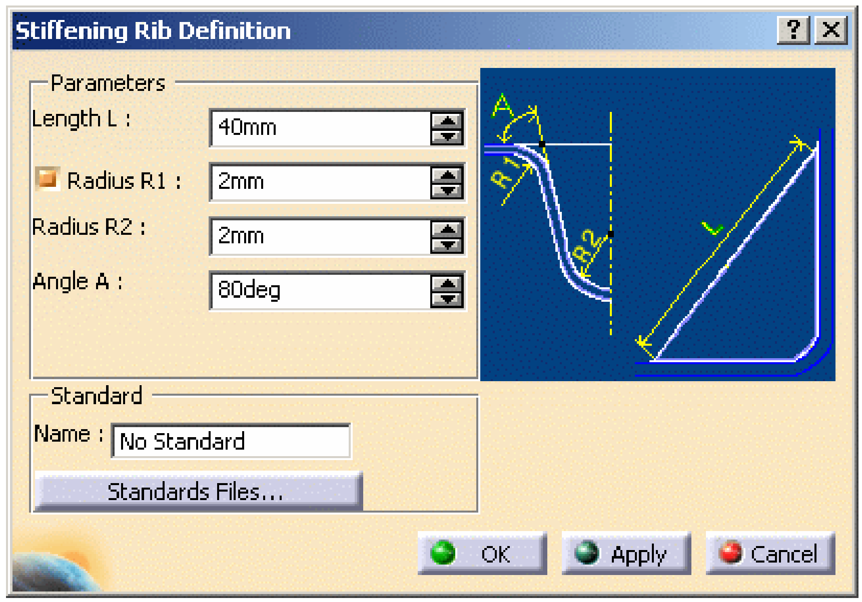
1. Click the **Stiffening Rib** icon .

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.

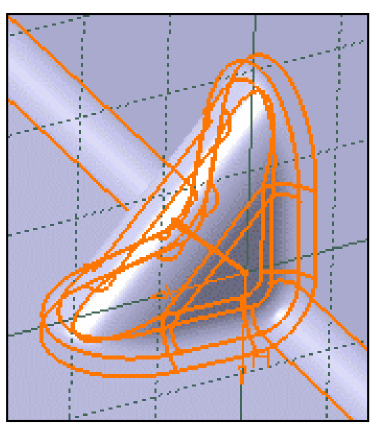
The Stiffening Rib Definition dialog box opens, providing default values.



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

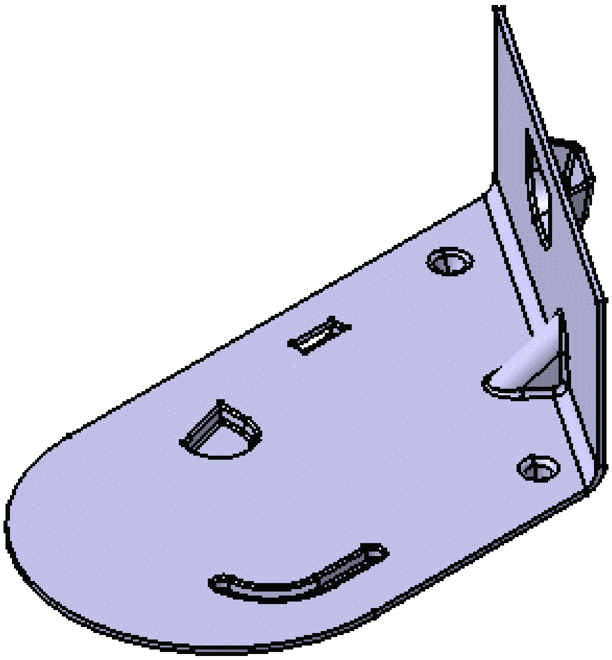
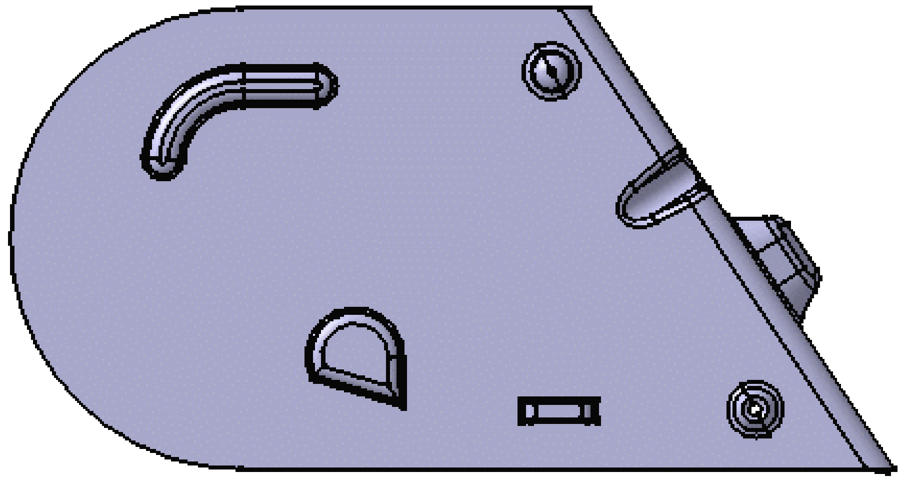
- Length L
- Radius R1
- Radius R2
- Angle A

4. Click **Preview** to visualize the stiffness rib.

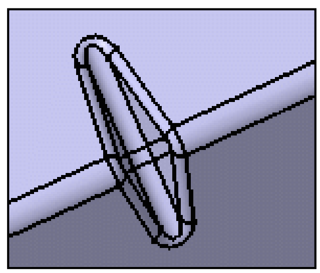


5. Click **OK** to validate.

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

You can use 0 as the Radius value to deactivate the Radius R1 value, and to create the stiffening rib without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Curve Stamp

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

Open the NEWStamping3.CATPart document.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping3.CATPart document.

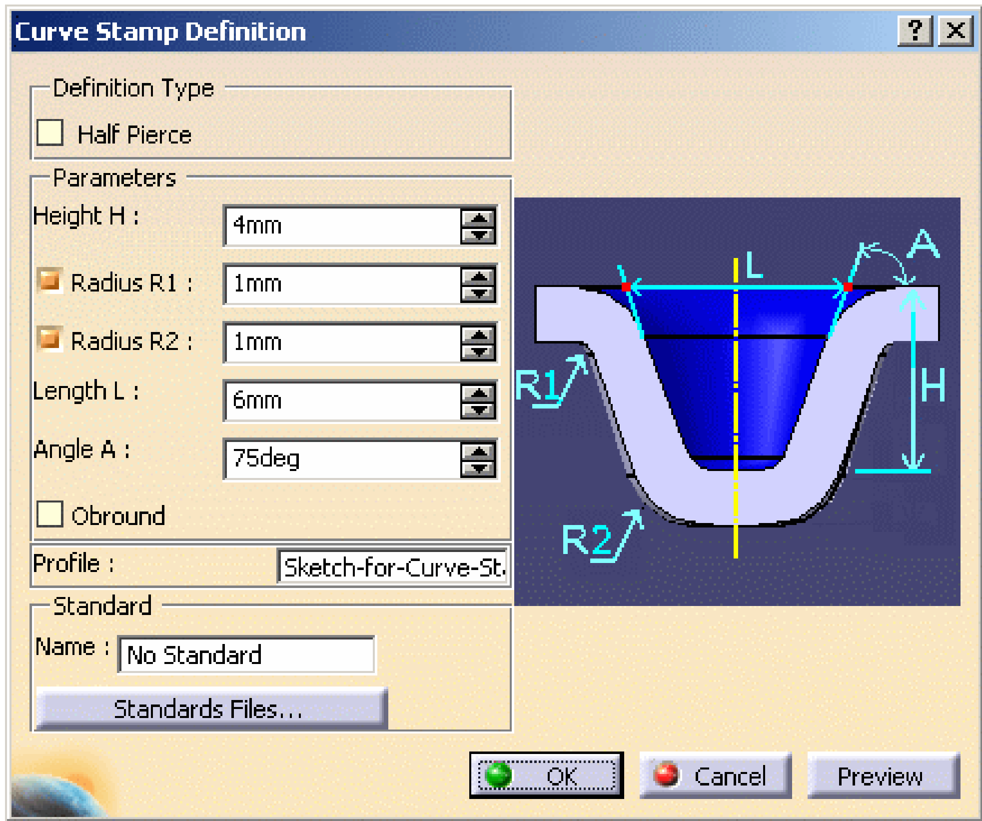
1. Click the **Curve Stamp** icon.

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

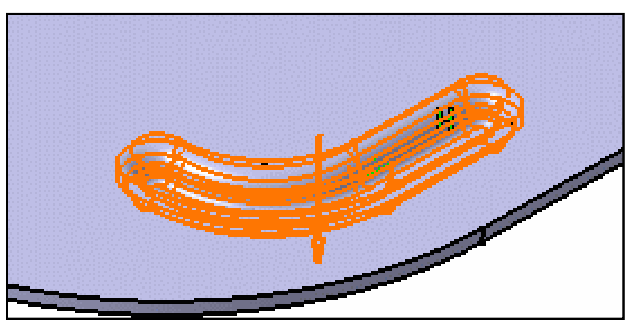
The Curve Stamp Definition dialog box opens, providing default values.

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

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

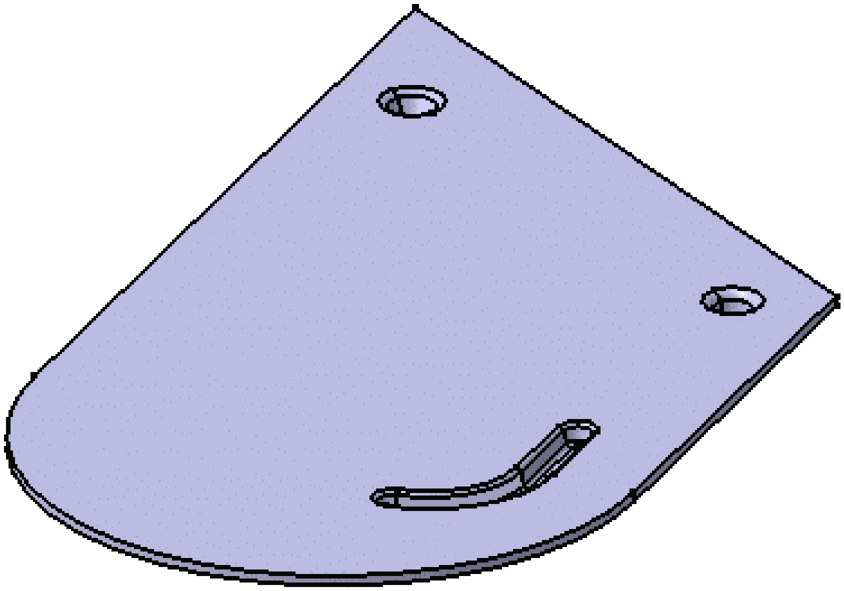


4. Click **Preview** to visualize the curve stamp.

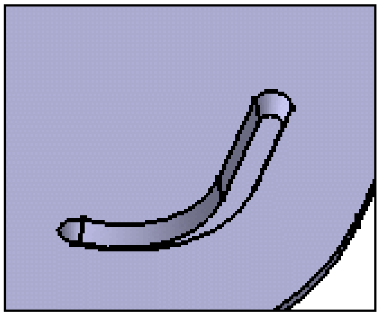


5. Click **OK** to validate.

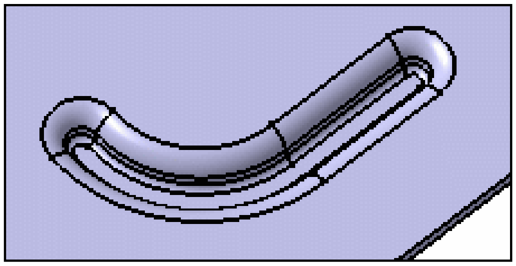
The curve stamp (identified as Curve Stamp.xxx) is created and the specification tree is updated accordingly.



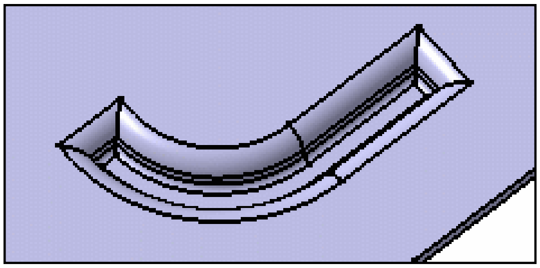
- You can use 0 as the Radius value to deactivate the Radius R and Radius R2 values, and to create the curve stamp without a fillet.



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



Obround option checked



Obround option unchecked

Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Louver

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

Open the NEWStamping11.CATPart document.

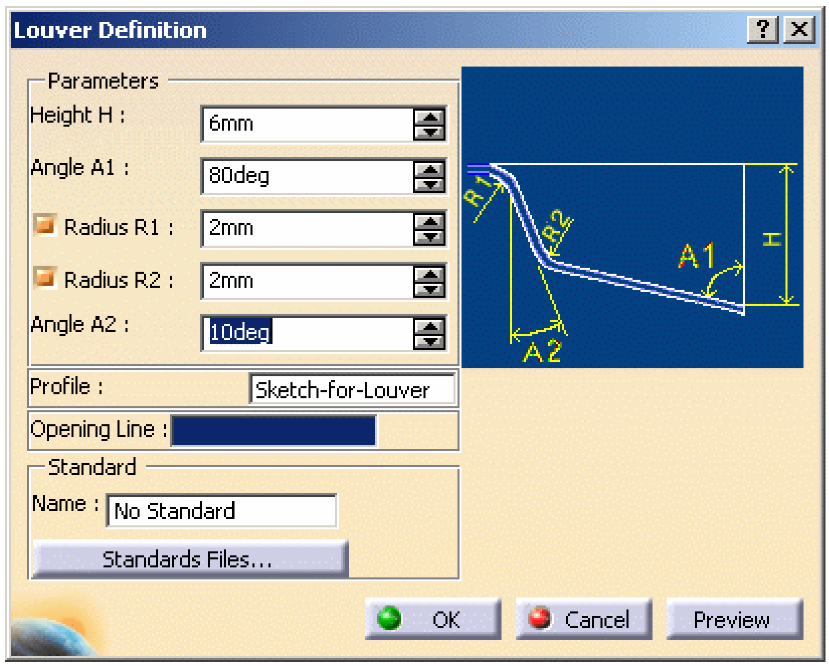
1. Click the **Louver** icon .

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.

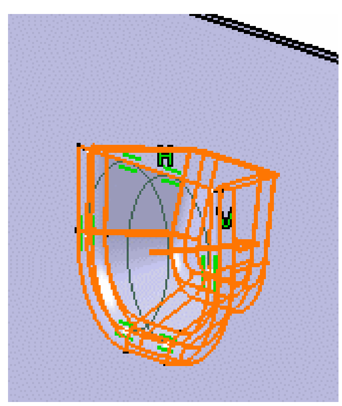
3. Change the value in the different fields, if needed:

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



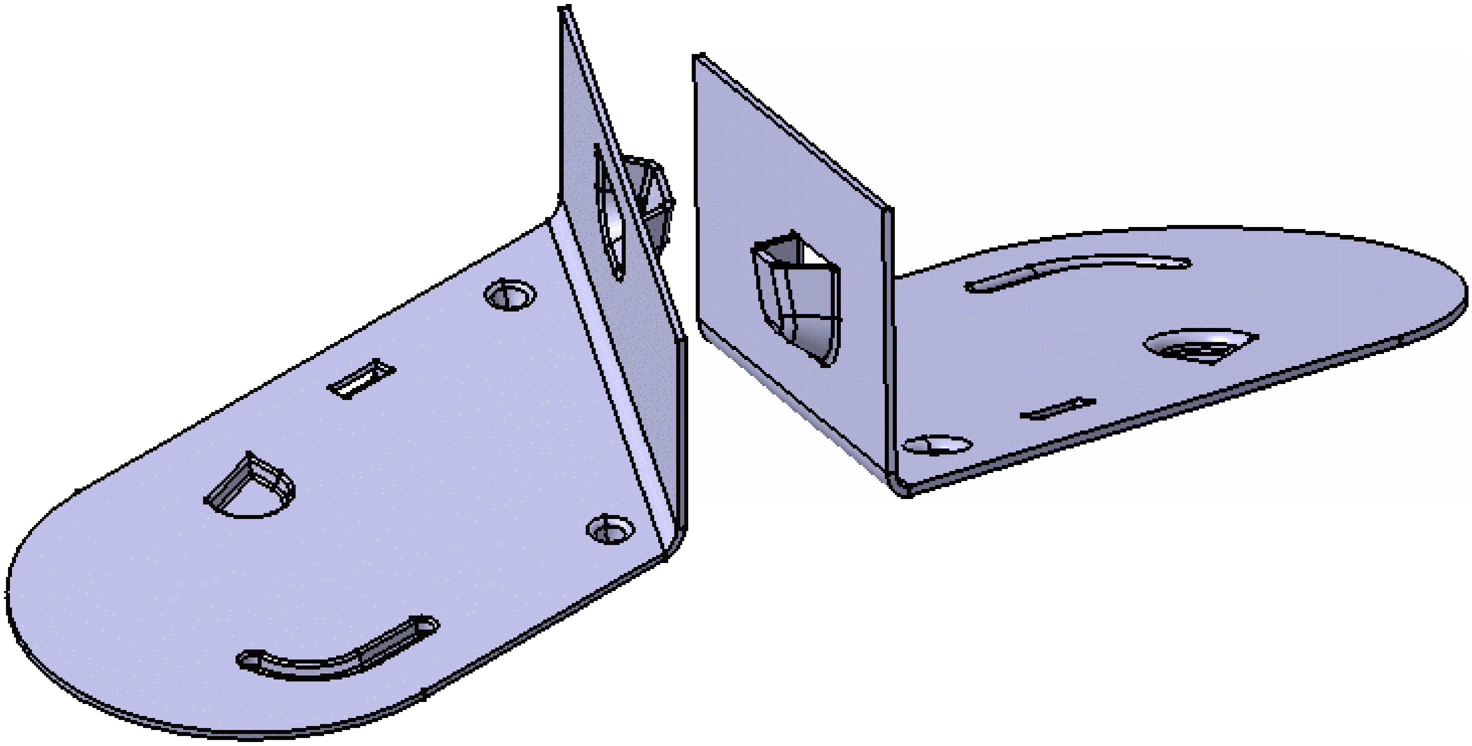
4. Select an edge of the sketch as the **Opening Line**: it indicates the position of the opening face of the louver.

5. Click **Preview** to visualize the louver.



6. Click **OK** to validate.

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



Recognizing Stamping Features

This task illustrates how to recognize a stamp geometry in order to create a Generative Sheetmetal Design stamping feature provided it is on a planar and single support.

Consequently, the following types of stamps can be recognized:

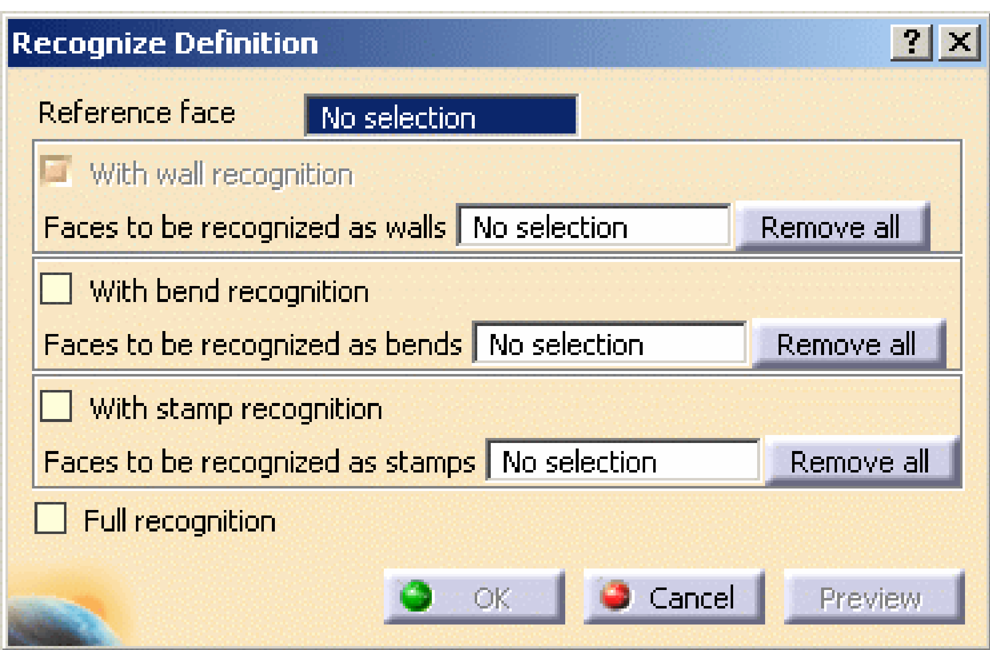
- Circular stamp
- Curve stamp
- Surface stamp
- Bead
- Bridge
- Louver

The recognize feature enables to create a Generative Sheetmetal Design stamping feature from a V4 model or parts created with Sheetmetal Design.

The Part Feature Recognition license is required to activate this feature in the Generative Sheetmetal Design Workbench.

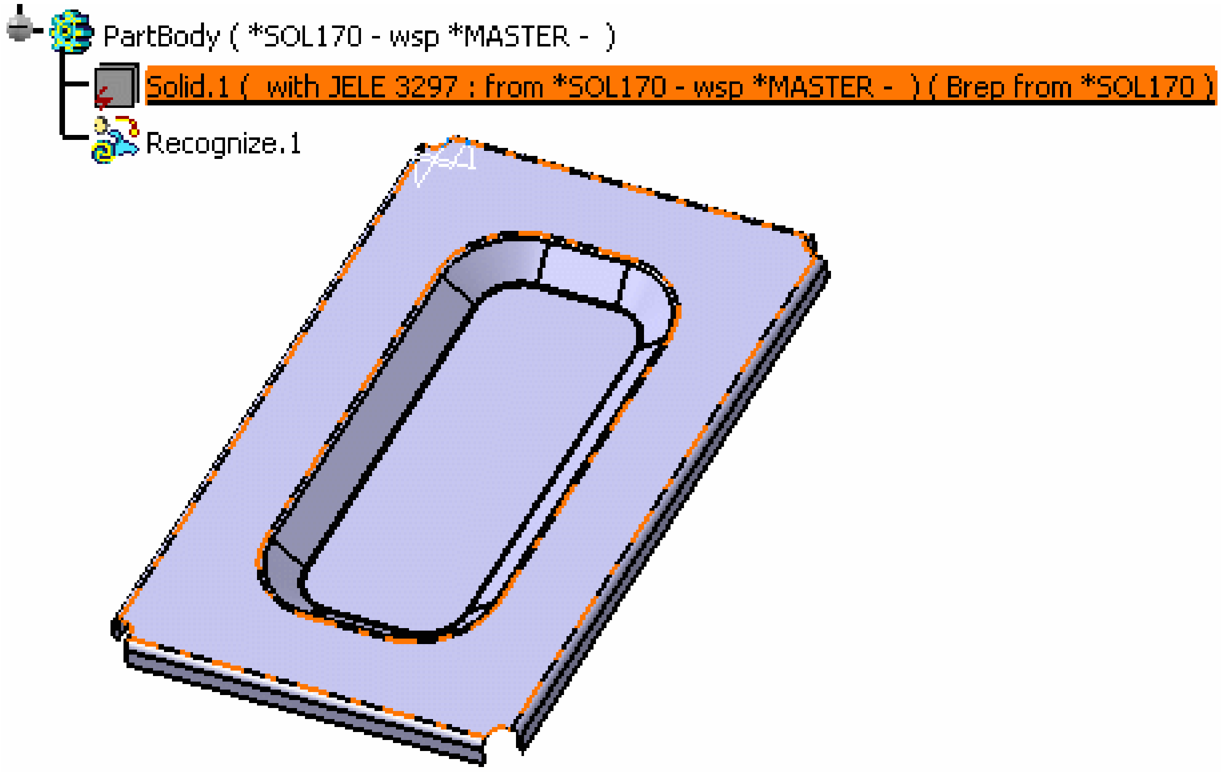
Open the NEWRecognize03.CATPart document. This document contains a part created from a V4 model.

1. Click the **Recognize** icon . The Recognize Definition dialog box is displayed.

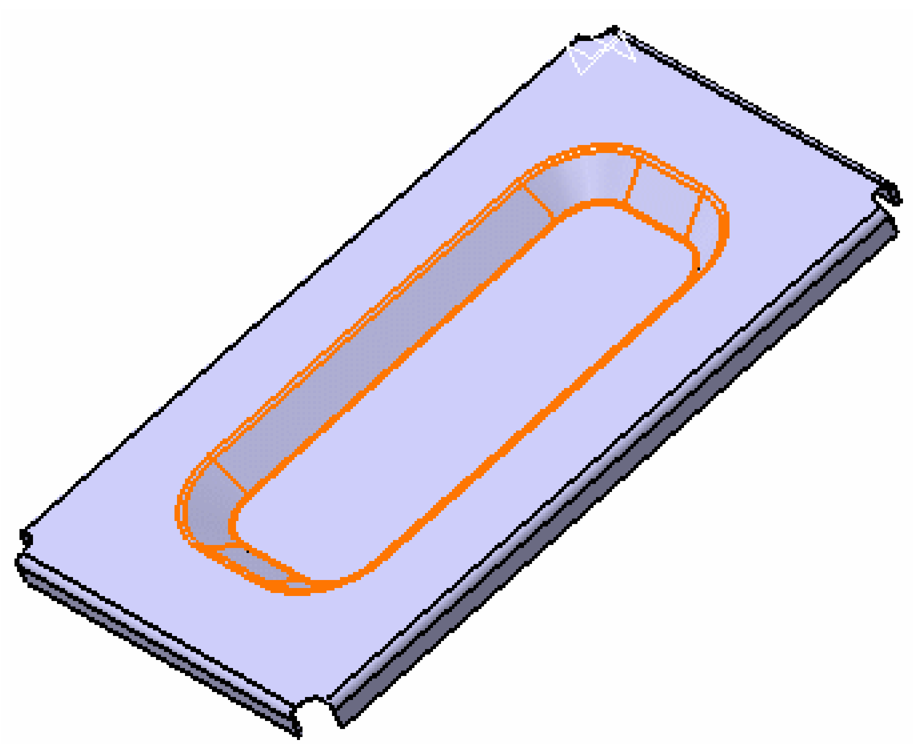


Note that the **With Wall recognition** option is already selected, and grayed out. This is because at least walls will be recognized, regardless of the other options you may choose.

2. Select a reference face. It will be the reference face for unfolding and for the definition of the sheet metal parameters (i.e. all default parameters will be based on this face).



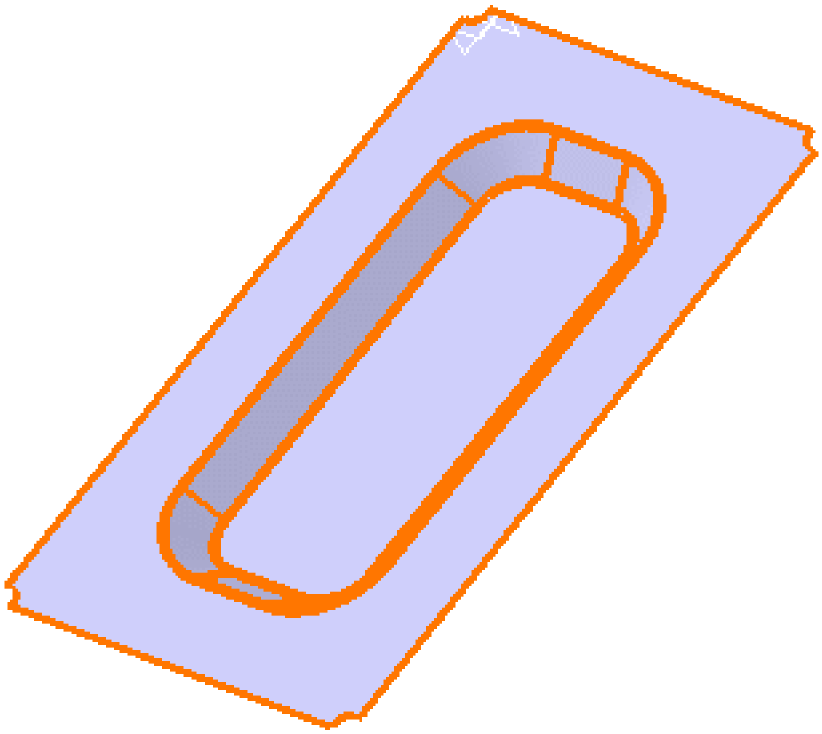
3. Select **With stamps recognition** and manually select all the faces to be recognized as stamps.



4. Select the reference face as a face to be recognized as wall.

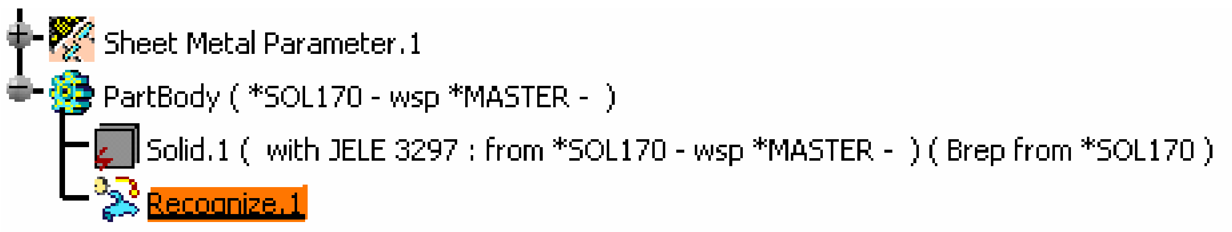
5. Click **OK** to validate.

The stamps are generated from the geometry.

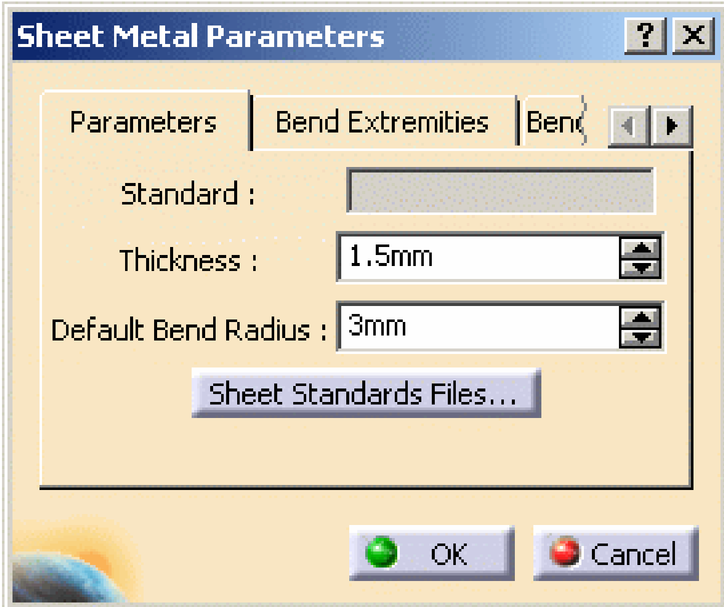


The **Recognize.1** feature is added to the tree view.

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



6. Select the **Sheet Metal Parameters** icon  to display the sheet metal parameters.



On the **Parameters** tab:

- the **Thickness** is equal to 1.5mm,
- the **Default Bend Radius** value amounts to twice that of the thickness.

On the **Bend Extremities** tab:

- the bend extremities are set to **Minimum with no 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 other parameters (default bend radius and bend extremities) in order for them 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 bend corner relief affect all features, and therefore can be edited even for "recognized" features.

You can also define the sheet metal parameters prior to recognizing the part. In this case, you need to make sure that the **Thickness** parameter value corresponds to the part thickness.

6. When all parameters have been redefined as needed, click **OK** in the **Sheet Metal Parameters** dialog box.

   The solid is now a Generative Sheetmetal Design part. You can now deal with it as with any other Generative Sheetmetal Design part, adding Generative Sheetmetal Design features to complete the design, or unfolding it.

- There is no stiffening rib recognition, since the support feature for the stamp must be planar.
- Stamps containing inner contours such as flanged hole, flanged cutout cannot be recognized.

Creating User-Defined Stamping Features

Two user-defined stamping features are available:

Create a punch with 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.

Create a punch with opening faces: define the punch, select a wall, define the opening faces of the punch, select an edge on the wall and give an angle for orientation purposes.

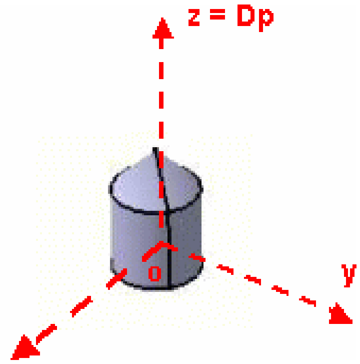
Edit a user-defined stamp: double-click the existing stamp and change its type, or select, or remove cutting and opening faces

What You Should Know

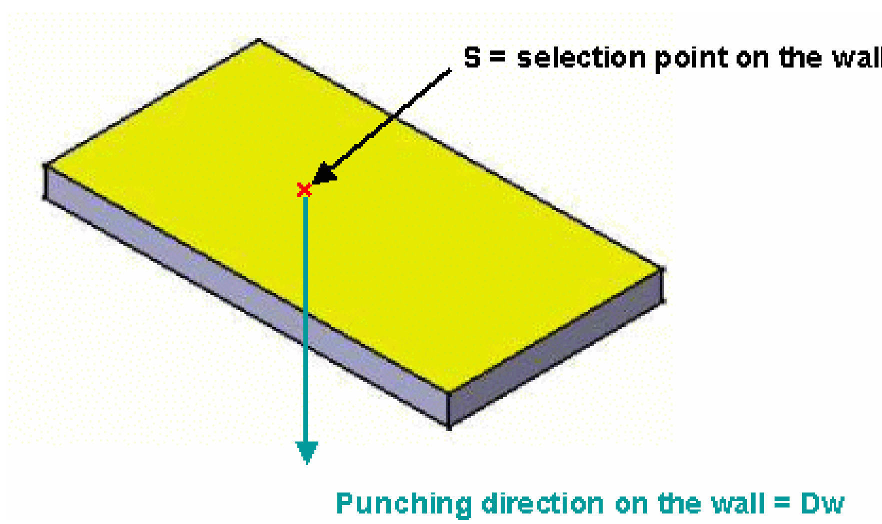
In both tasks illustrating either a stamp based on a punch and a die, or a punch with cutting and opening faces, the punch positioning is defined as below:

Defining the Punch in Relation to the Wall to be Stamped

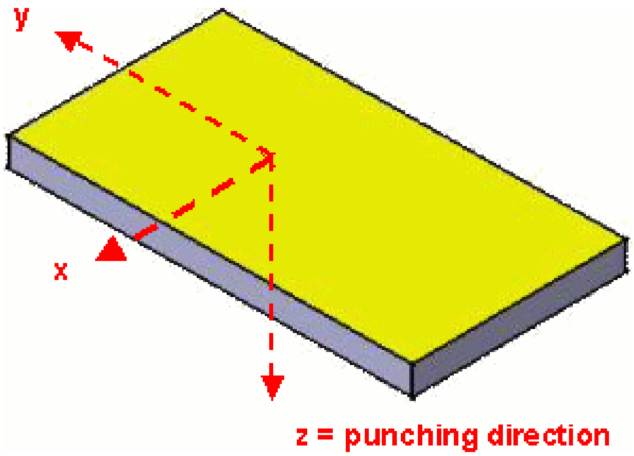
The punch is defined within the absolute (default) axis-system of the .CATPart document. (o, x, y, z) is the axis associated with the punch. The punching direction on the punch (*Dp*) must be equal to z.

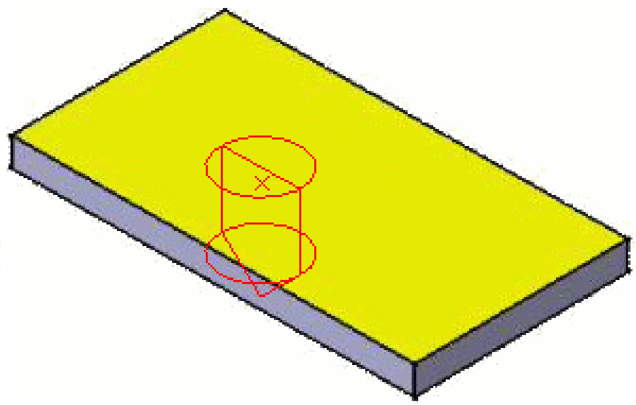


The punching direction on the wall (*Dw*) is normal to the selected wall face, and is oriented from the selected wall face towards the opposite face.



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



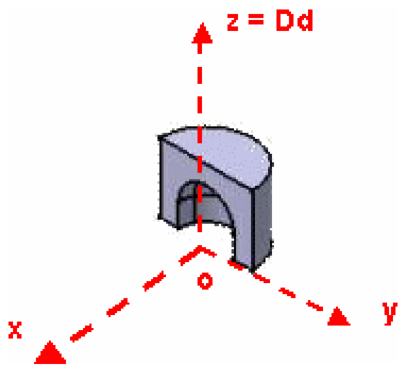


Defining the Die in Relation to the Wall to be Stamped

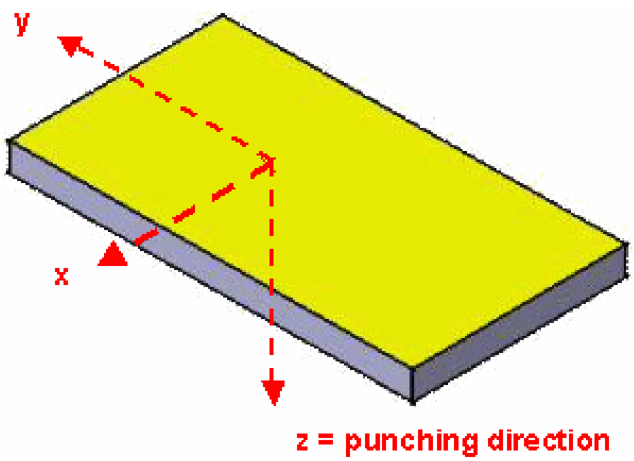
This is useful only when defining a punch a die, and does not apply to punches with cutting and opening faces.

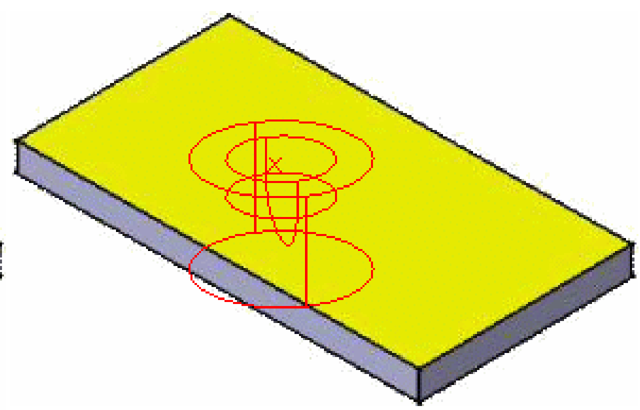
The die is also defined within the absolute (default) axis-system of the .CATPart document. (o, x, y, z) is the axis associated with the punch. The punching direction on the die (*Dd*) must be equal to z.

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.

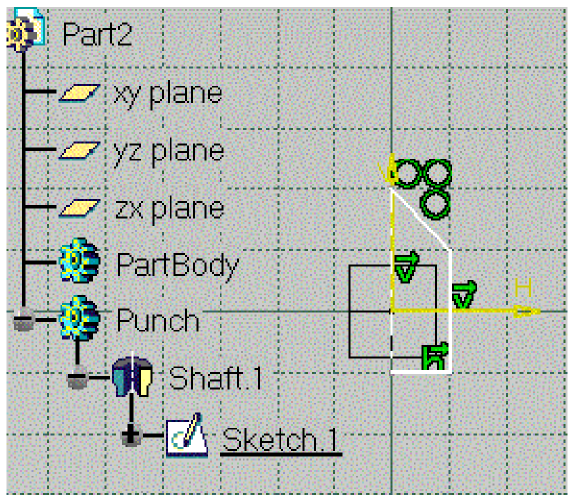
First, you will define a punch and a die in Part Design, in the absolute axis-system.
Then, in a Sheet Metal part, you will bring the punch and the die features (and their axis system) to a point you have selected. If necessary, you will define a rotation of the axis system from a reference line.

You can also position the stamp according to the punch and die features positioning and direction thanks to the **On context** option.

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

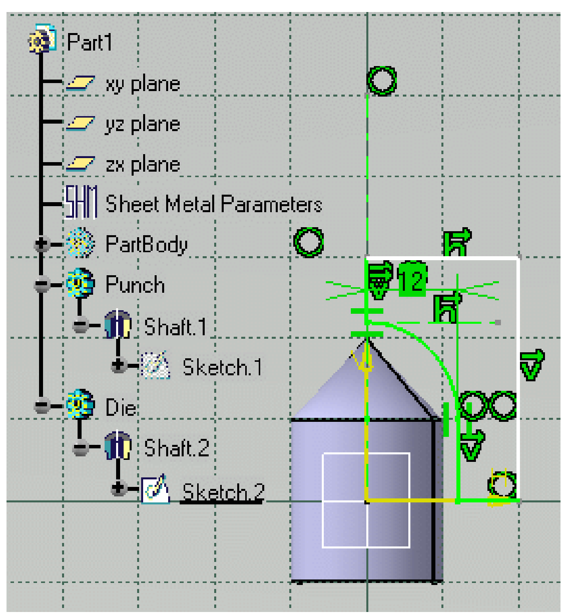
All .CATParts are available from the samples directory (PunchDie1.CATPart, Punch1.CATPart and Die1.CATPart or NEWPunchDie1.CATPart, NEWPunch1.CATPart and NEWDie1.CATPart for Generative Sheetmetal Design or Aero_PunchDie1.CATPart, Aero_Punch1.CATPart and Aero_Die1.CATPart for Aerospace Sheetmetal Design).

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 .

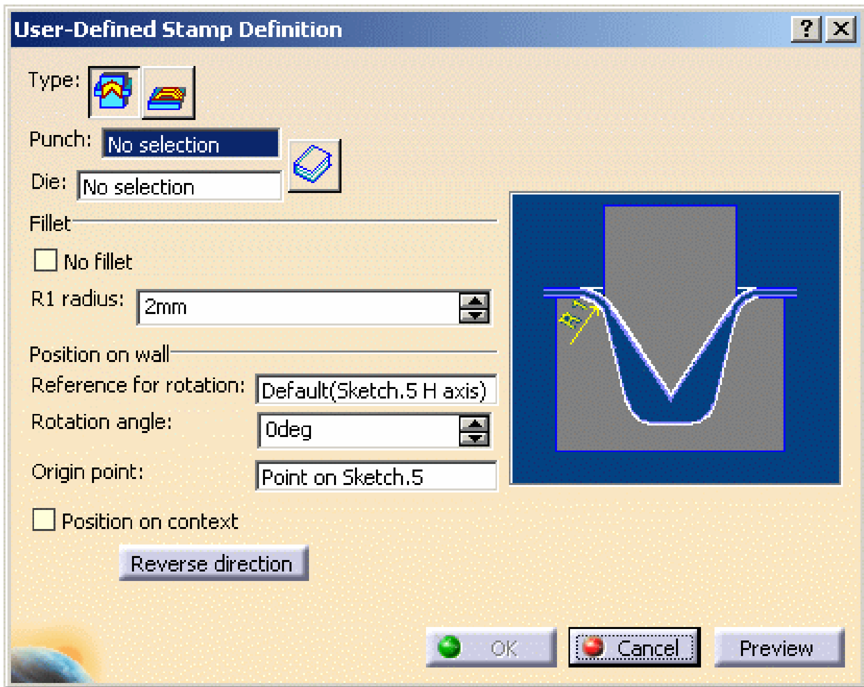


5. Repeat from step 2 to step 4 to define the die, making sure that it is oriented as described in Defining the Die in Relation to the base feature to be Stamped.

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.

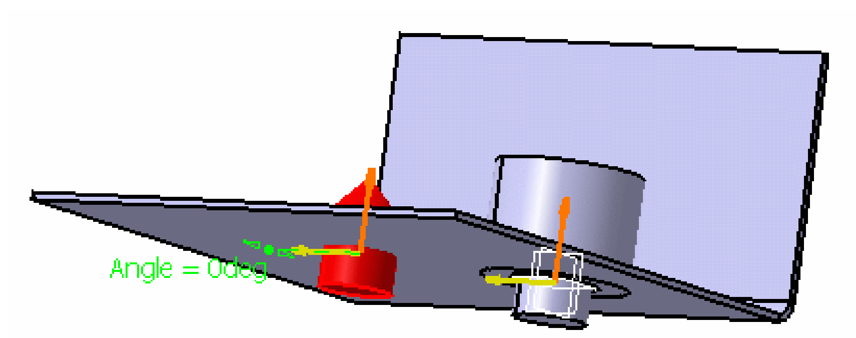
7. Click the **User Stamping** icon from the Stamping tool bar and select a base feature, or a face where the stamping is to be created. This base feature or face is used to define the stamping location and direction, by matching the punch's origin to the selected point on the base feature.

The User Defined Stamp Definition dialog box is displayed:



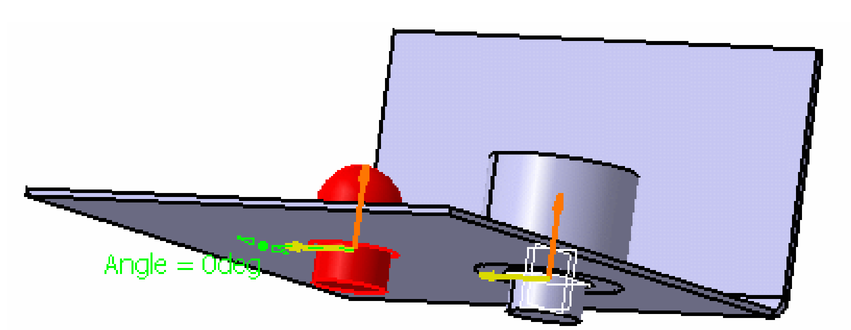
8. Make sure the **With die** icon is pressed down and select the Punch feature from the specification tree.

The punch's positioning is previewed in the geometry.

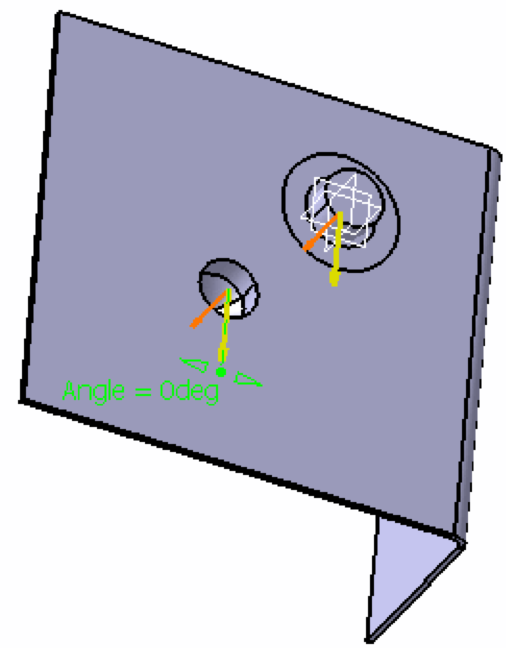


9. Select the Die feature.

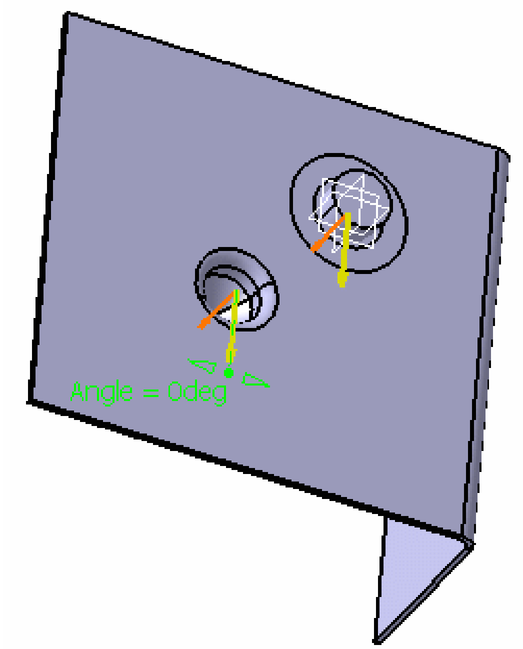
The die's positioning is previewed in the geometry as well.



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

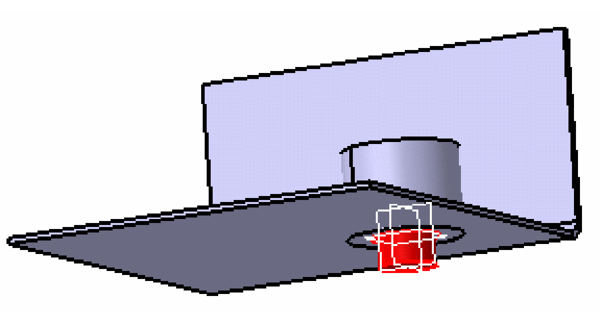
11. If needed, define the stamp's positioning on the selected base feature by choosing:

- 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.

12. If needed, select the **In Context** check box.

The punch and die's positioning is previewed on the geometry.

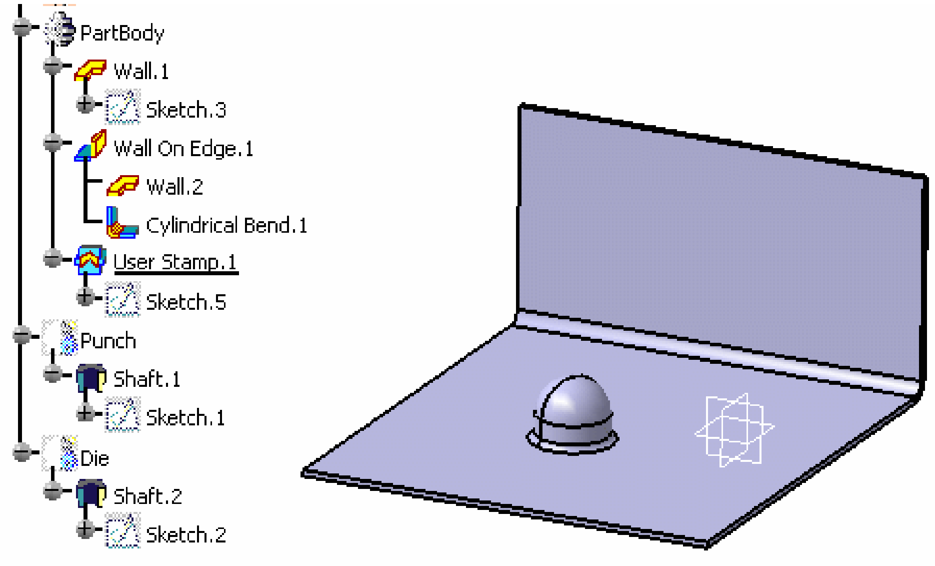


When selecting the **On Context** check box, the stamp's positioning and direction are not defined in relation to the base feature anymore.

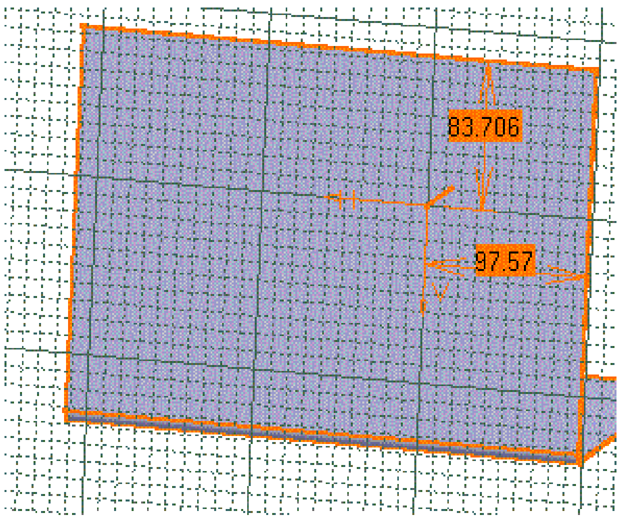
Only the punch and die's axis system is taken into account and the stamp is created according to their positioning and direction.
Once **On Context** is selected, the position on wall cannot be modified nor the direction of the stamp: the fields available in **Position on wall** section and the **Reverse direction** button are disabled.

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.

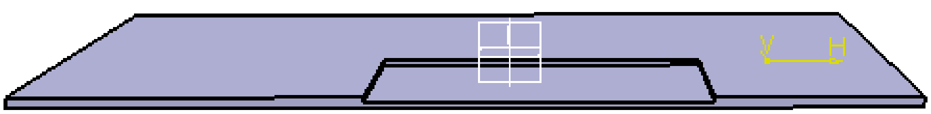


- **Radius** is the radius of the bend between the stamping and the base feature.
- **Punch** and **Die** are the bodies you have defined previously. If the punch and the die are in another CATPart document, activate this document before clicking the punch or the die.
- If you select two reference lines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



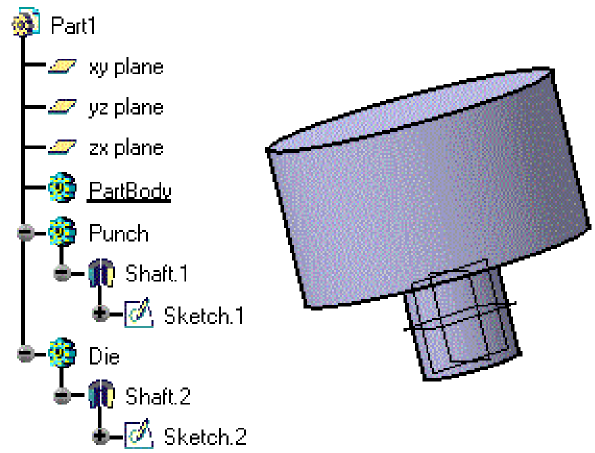
- A user-defined stamping can be edited (punch, die, position, constraints).
- As the punch and die are not symmetrical, you cannot create such features as a cutout, a hole, a corner, etc., on this kind of stamping.

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



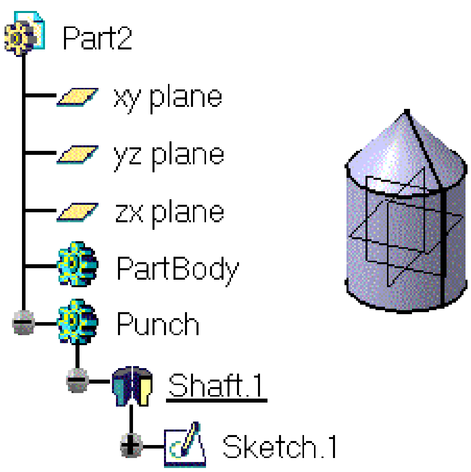
- The punch height cannot be superior to the base feature height, otherwise it is considered as a cutout.

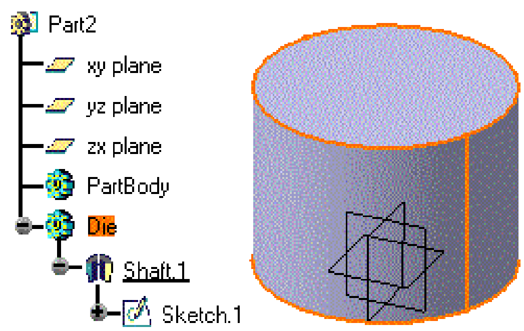
- You may create a user-defined stamping from a punch only but you cannot create a fillet.
- Only the stamping sketch is displayed in unfolded views.
- The punch and die bodies can be defined in the Sheet Metal part where the stamping is to be created (see PunchDie1.CATPart or NEWPunchDie1.CATPart in 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.

Creating a Punch with Opening Faces

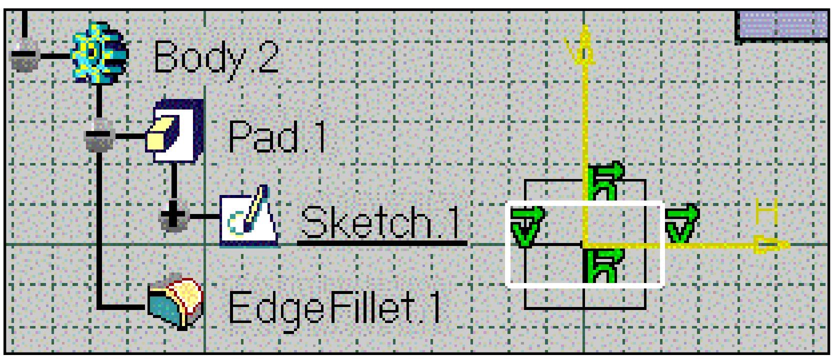
This task explains how to create a stamp from a punch feature with opening faces.

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.
You can also position the stamp according to the punch and die features positioning and direction thanks to the **On context** option.

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

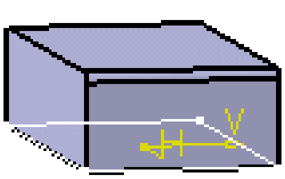
The CATPart documents are available from the samples directory, NEWOpenFaces1.CATPart for is Generative Sheetmetal Design or Aero_OpenFaces1.CATPart for Aerospace Sheetmetal Design.

1. Start the Part Design application.
2. Insert a PartBody (menu **Insert -> Body**) to define the punch.
3. Enter the Sketcher workbench , 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 .

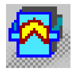


The punch must be oriented as described in Defining the Punch in Relation to the base feature to be Stamped.

- 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 base feature to be stamped is located.
  A link is retained between the initial punch feature and its copy.



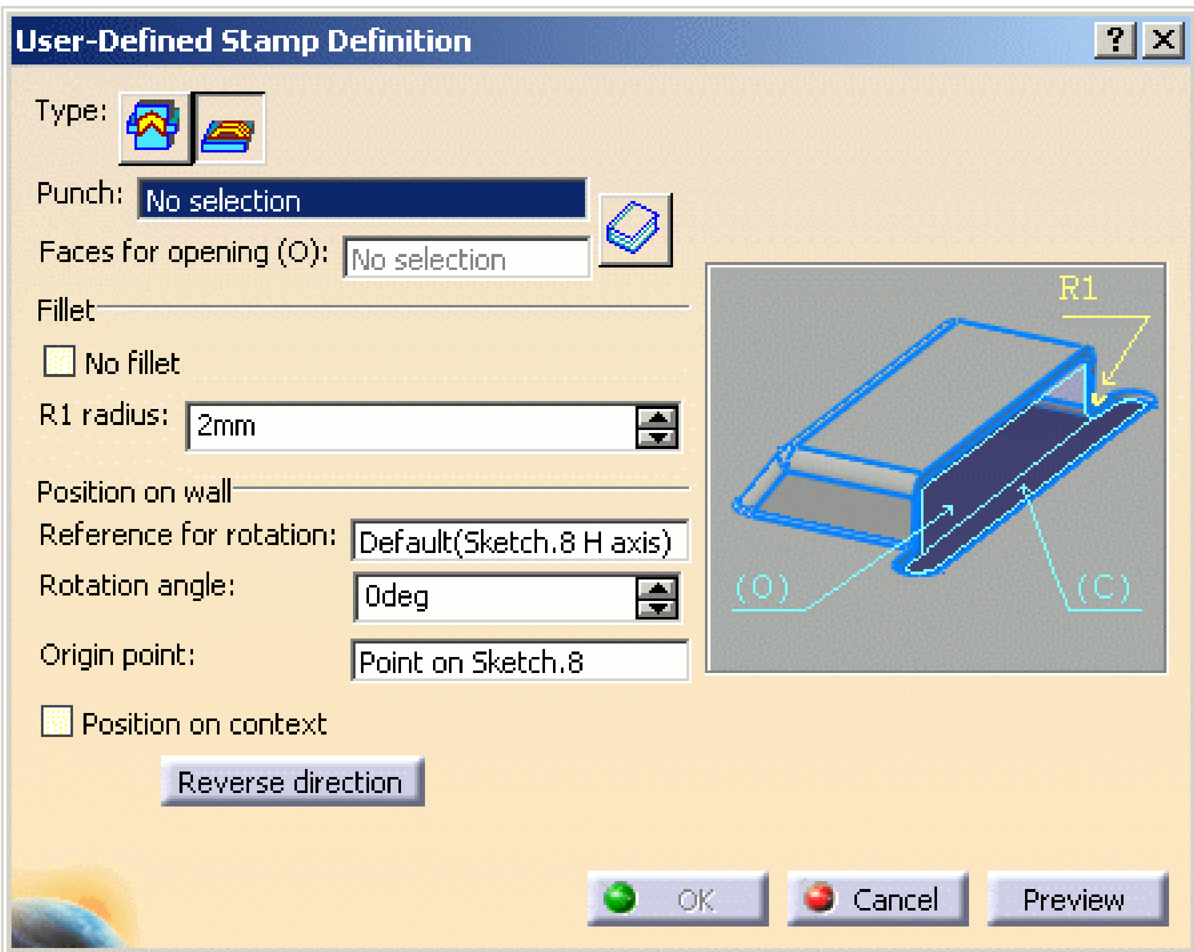
5. Return to the Sheet Metal application, and if needed, use the **Define In Work Object** on the PartBody containing the base feature to be stamped.

6. Click the **User Stamp** icon  from the Stamping toolbar and select a base feature where the stamping is to be created.

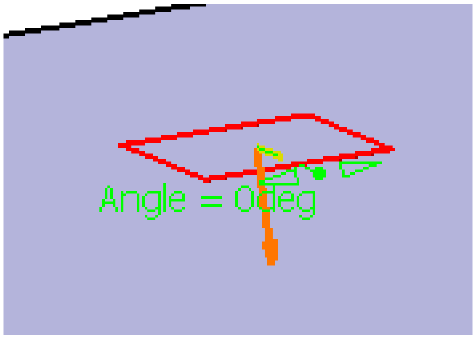
This base feature is used to define the stamping location and direction, by matching the punch's origin to the selected point on the base feature.

The User Defined Stamp Definition dialog box is displayed.

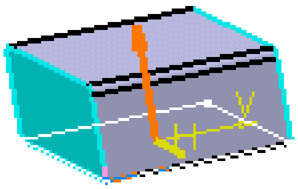
7. Click the **With opening**  icon.



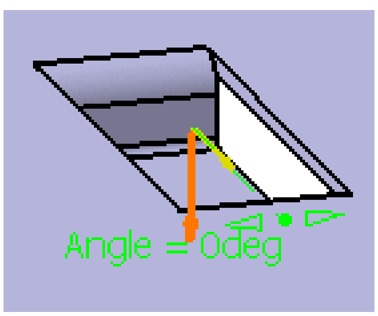
8. Select the punch (Body.2). The punch is previewed on the base feature.



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

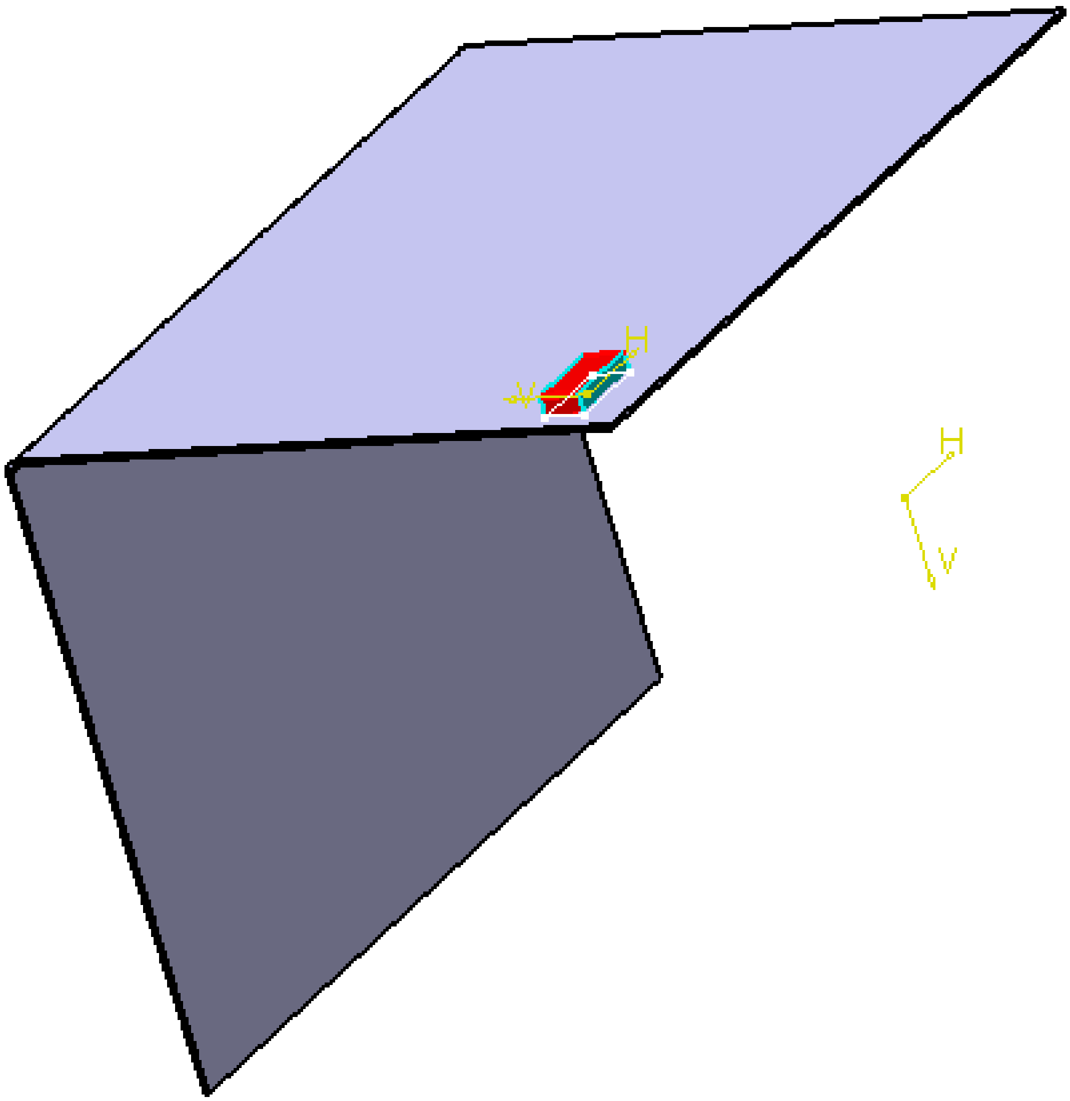


10. Click Preview. The stamp is previewed with the opening faces:



10. Select the **On Context** check box if you wish to position the stamp according to the positioning and direction of the punch and die features.

11. Click **Preview**. The stamp is previewed with the opening faces at the point where the punch and die were created on the part:



For more information about the **On context** check box, refer to the Creating a Punch with a Die section.

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

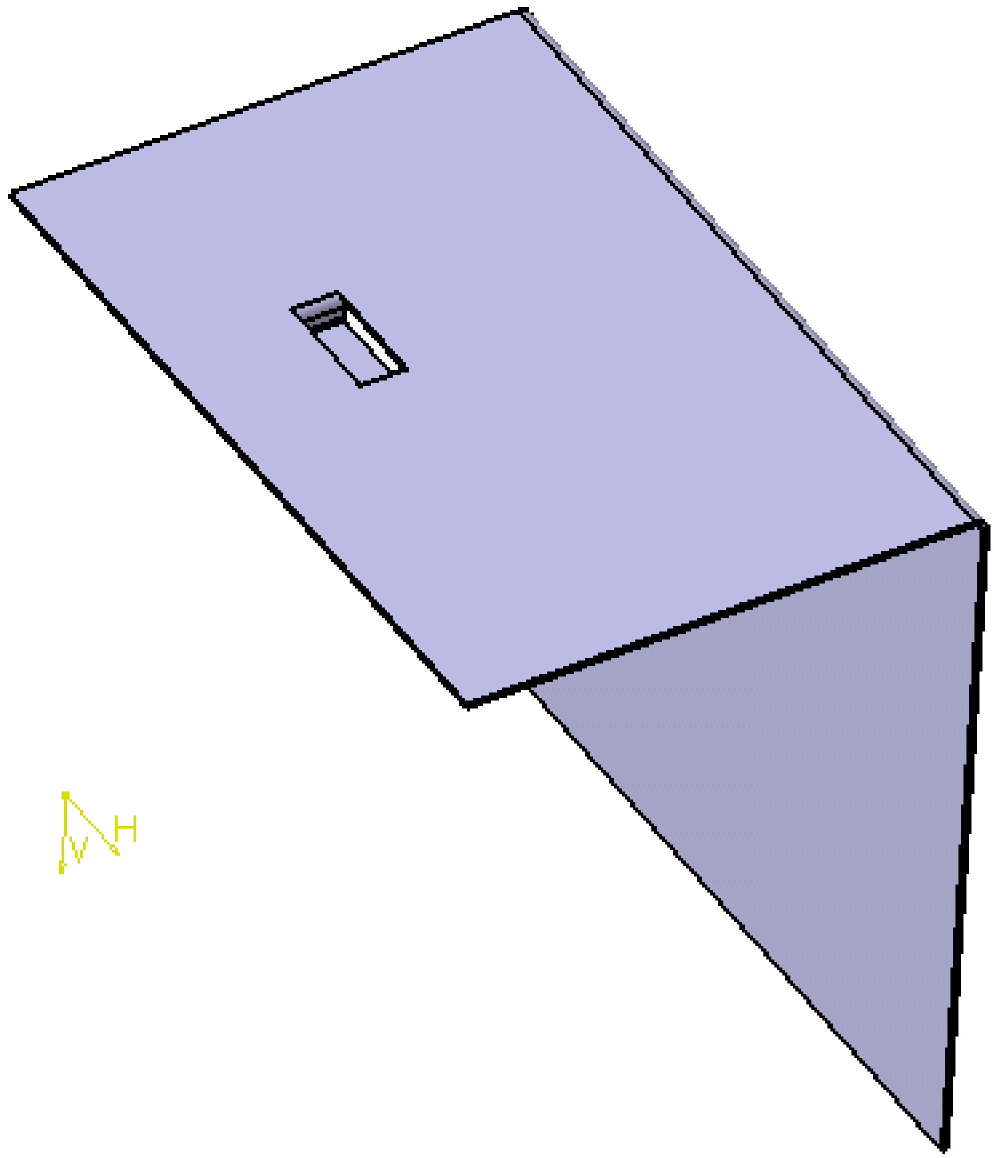
13. If needed, define the stamp's positioning on the selected base feature by choosing:

- a **Reference for rotation**: by default, it is the sketch axis, but you can also select any line or edge on the 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.

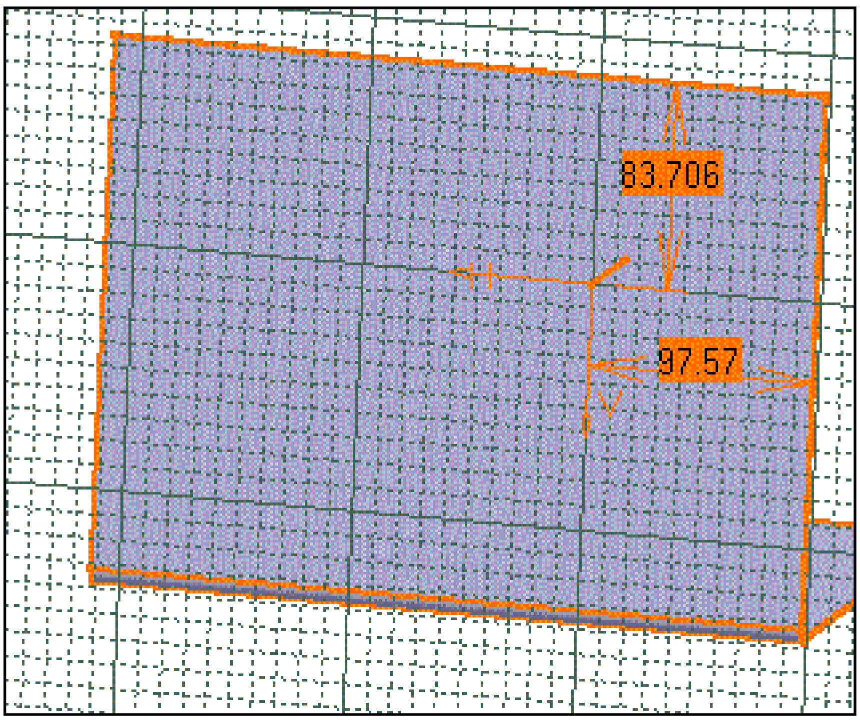
This is especially useful for non-circular stamps, but you can very well create the stamp as is, without further positioning.

14. Click **OK** to validate and create the stamping.

The stamp is automatically set in No Show mode.



- **Radius** is the radius of the bend between the stamping and the base feature.
- **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 opening** must be picked on the punch, not on the base feature.
  If the punch is located into another .CATPart document, these faces must be picked on the copy of the punch where the base feature to be stamped is located.
- If you modify the selected punch, the user-defined stamp with the opening faces will not be updated accordingly, nor will it be updateable. If you want to update the user-defined stamp, you will need to edit it; in the User Defined Stamp Definition dialog box, clear the **Faces for opening** field and re-select the lateral faces of the modified punch.
- Avoid using stamps with faces merging with the face of the base feature to be stamped, as it would be difficult to remove afterwards, especially on a curved part.
  If you do use such a stamp, select the stamp face tangent to the base feature and define it as an open face.
- 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

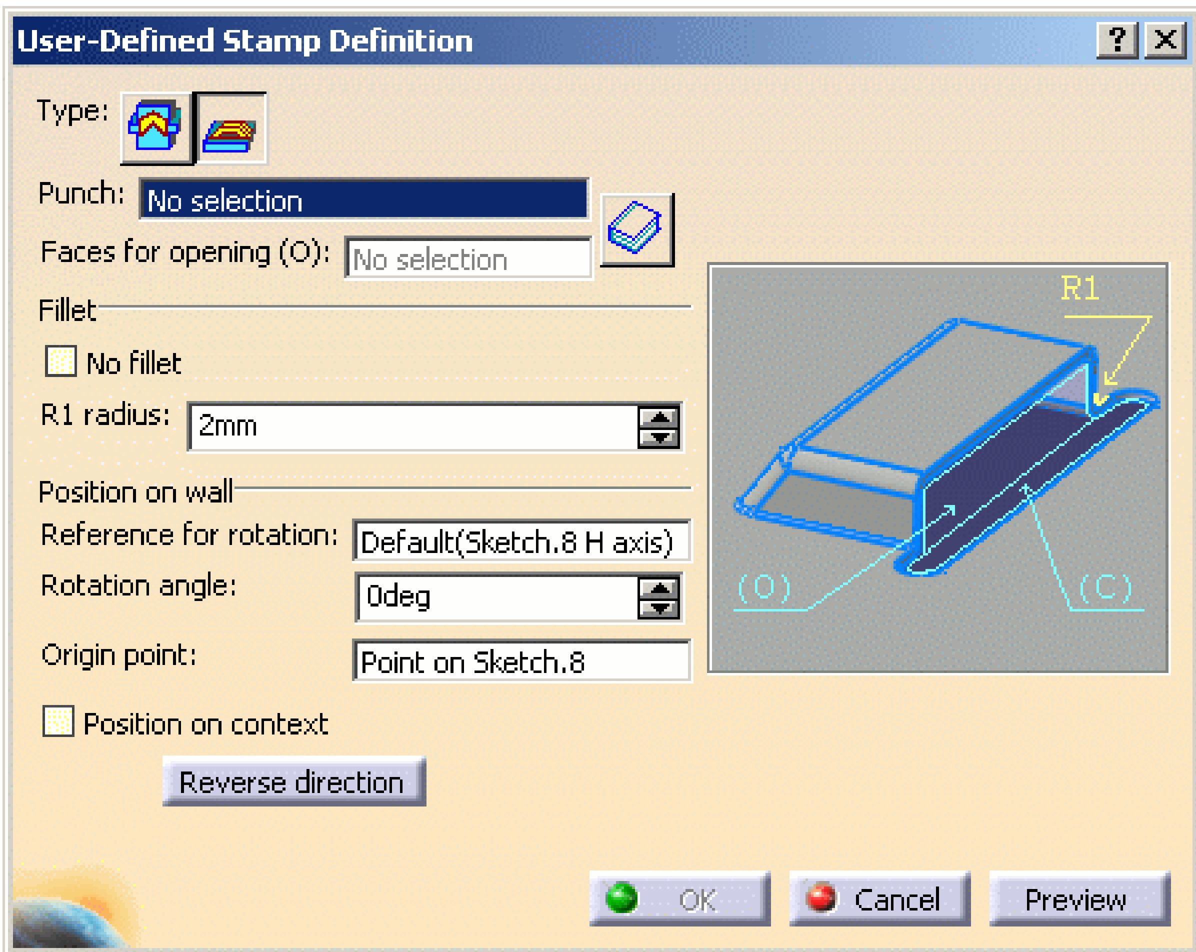
This task explains how to edit a user-defined stamp, that is:

- 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.

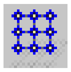
4. Modify any other parameter as needed.

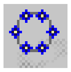
5. Click OK in the User Defined Stamp Definition dialog box to take these modifications into account.

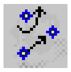
The stamp is updated accordingly.

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*.

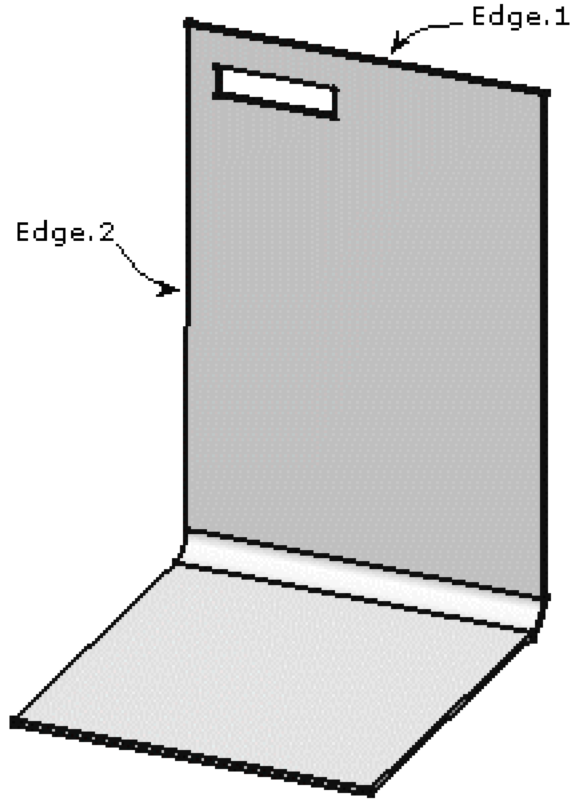
Creating Rectangular Patterns

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

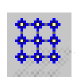
In the Generative Sheetmetal Design workbench, you can only duplicate flanges, cutouts, holes, stamps (except stiffening ribs), stamps without fillet (radius=0) and Generative Sheetmetal Design patterns.
These features must lie on a unique and planar wall.

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:



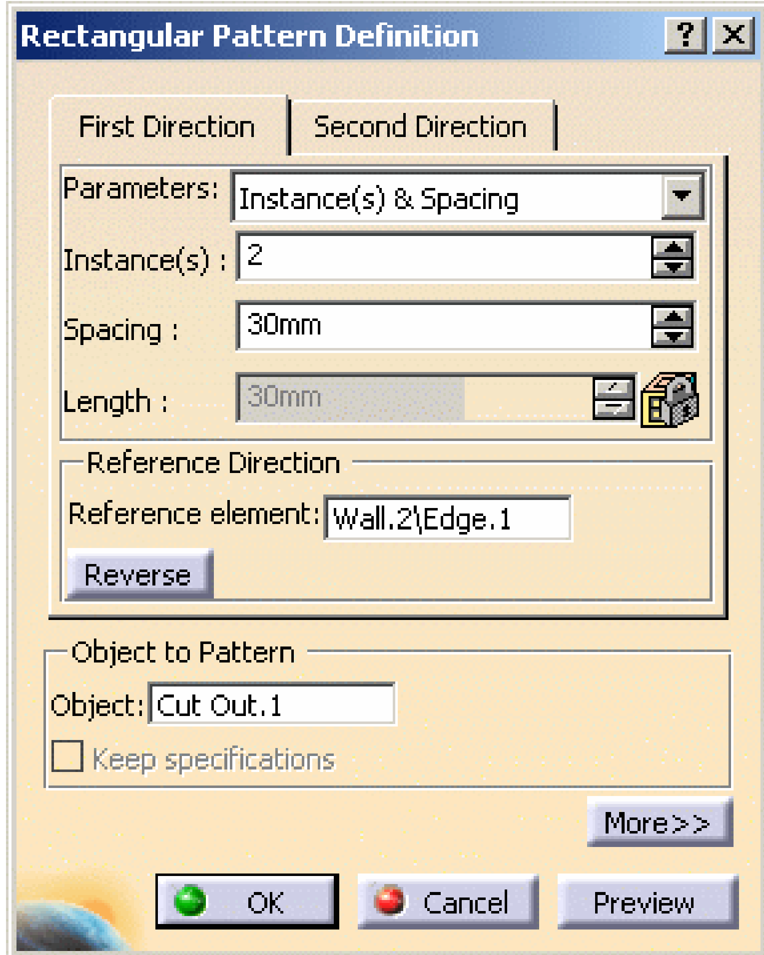
1. Select the rectangular cutout you want to duplicate.

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.

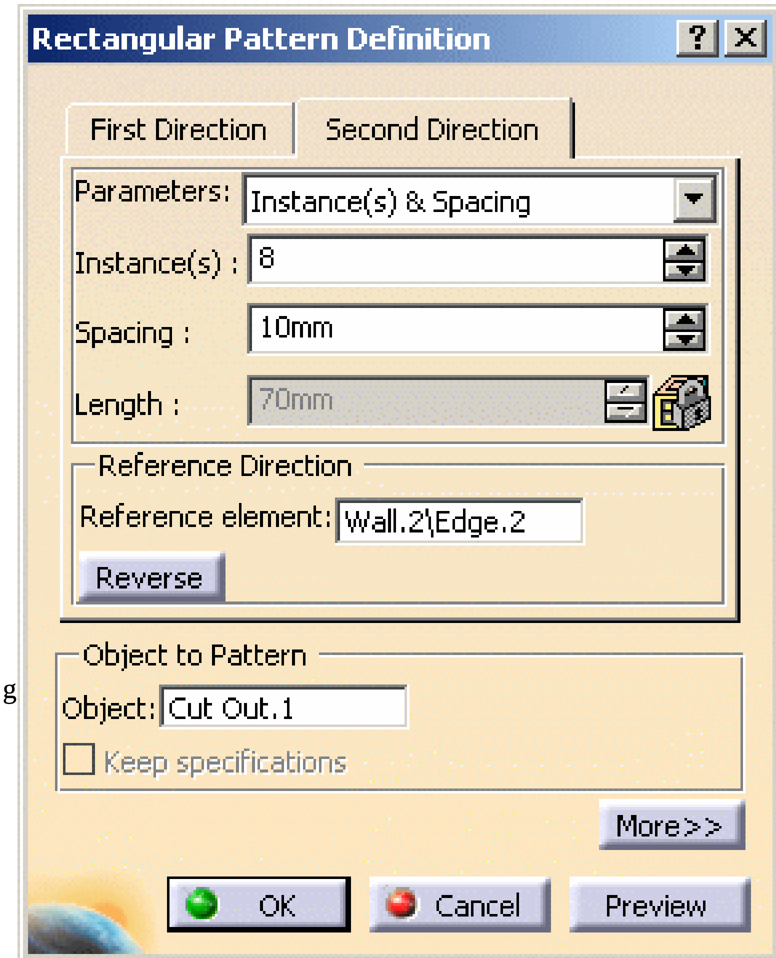
Patterns should not go beyond the model.

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.

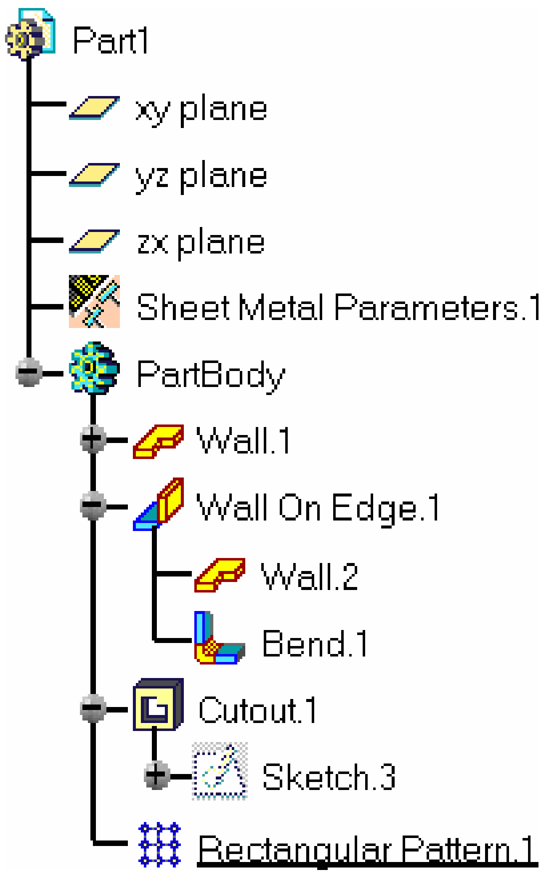
8. Select the second edge (**Edge.2**), as shown, to define the second direction.

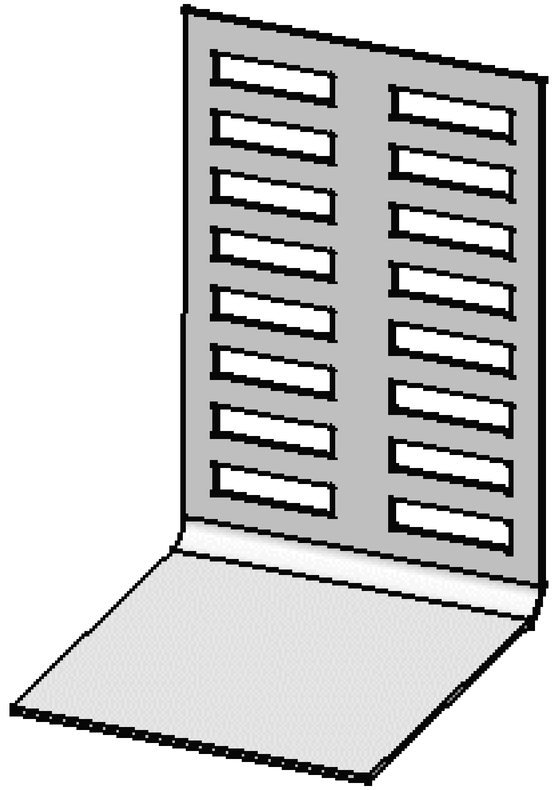
9. Keep the **Instances & Spacing** option: enter 8 and 10 mm in the appropriate fields.

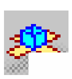
Additional cutouts have been aligned along this second direction.

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

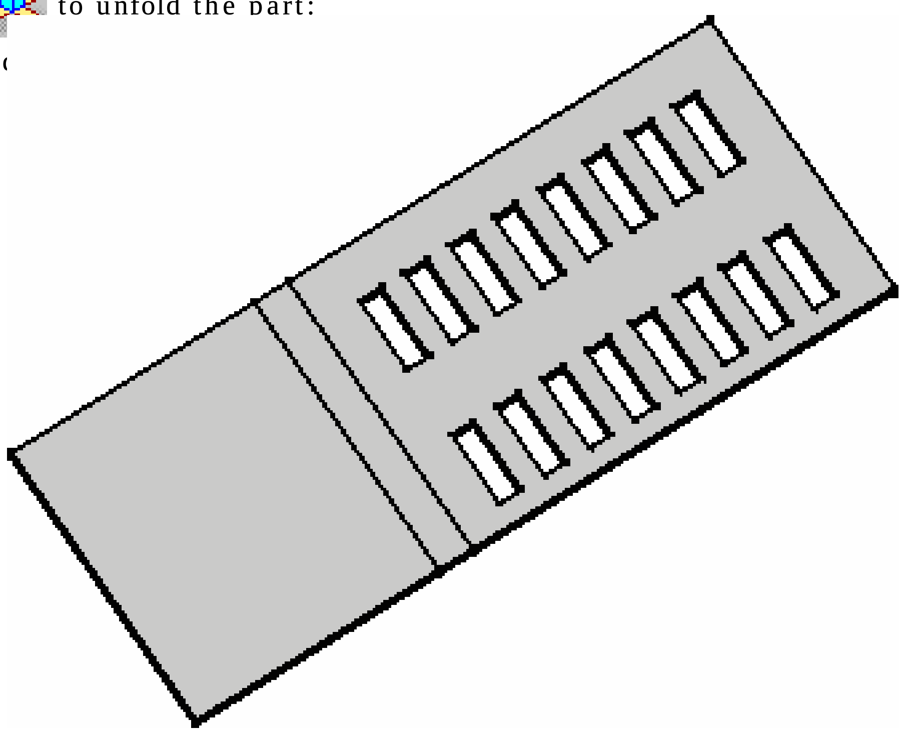
After the update, the Sheet Metal part looks like this:





11. Select this icon  to unfold the part:

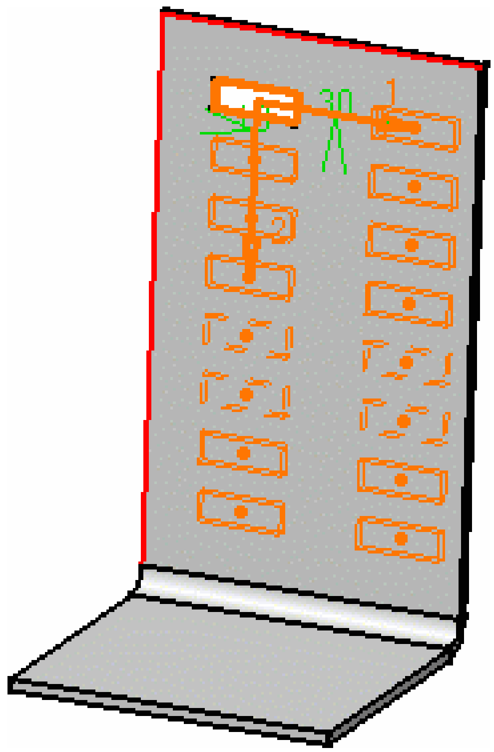
The pattern is up



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.



- When you duplicate a pattern of flange, the edge of the flange spine and its instances have to be tangent to the wall edge: you cannot choose a direction of patterning not parallel to the flange spine.
- All instances of the flange pattern must lie on the same face as the flange pattern.

Creating Circular Patterns

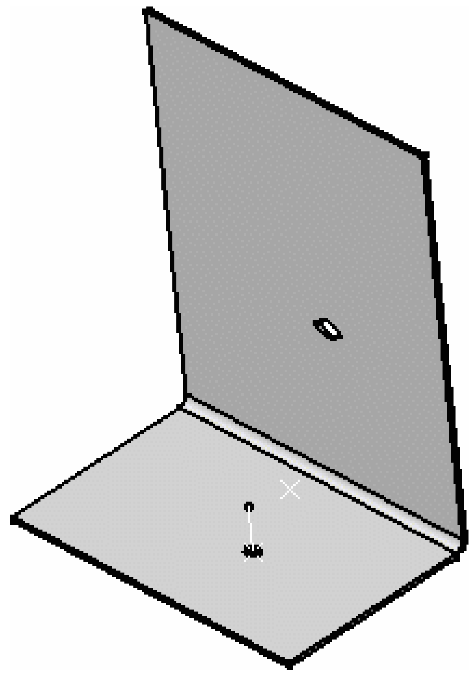
In this task, you are going to create circular cutouts according to a pattern.
These features make the creation process easier.

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:

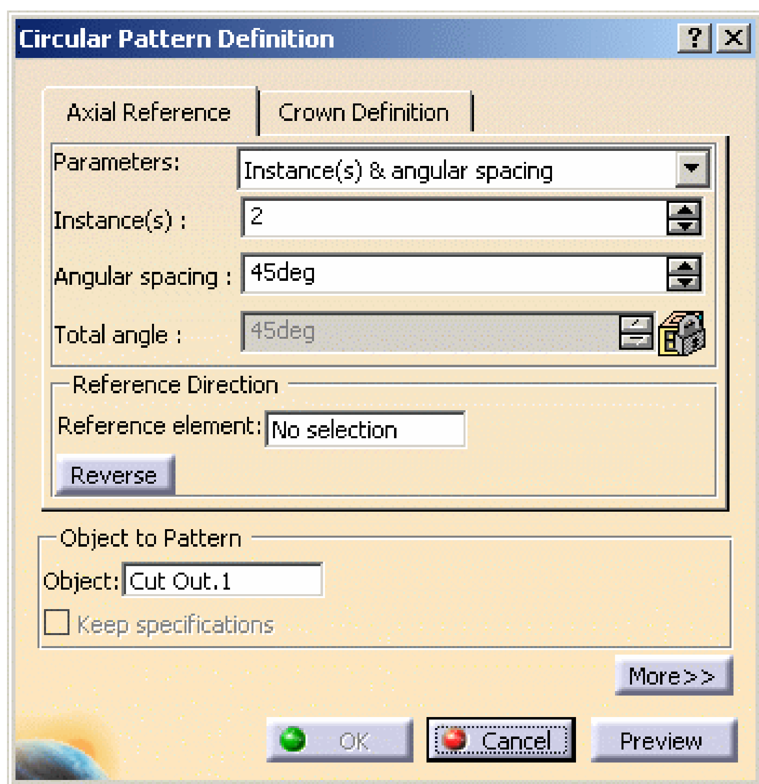


1. Select the circular cutout you want to duplicate.

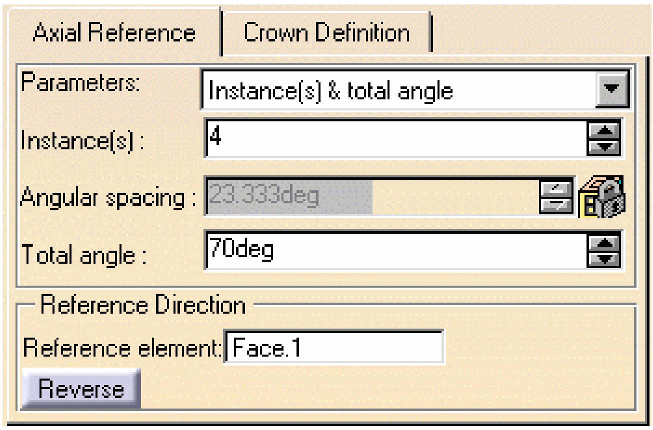
2. Click the **Circular Pattern** icon .

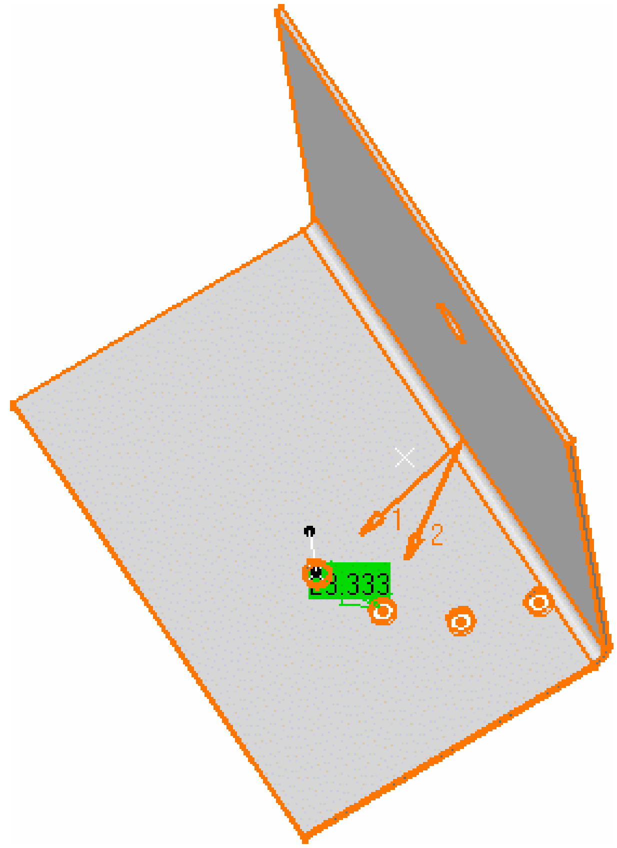
The **Circular Pattern Definition** dialog box is displayed.

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

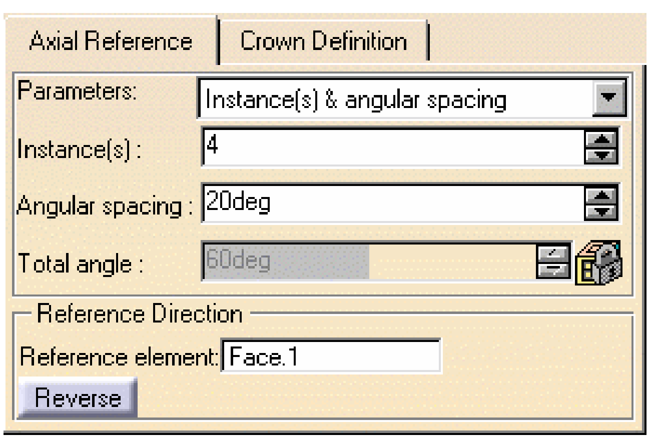


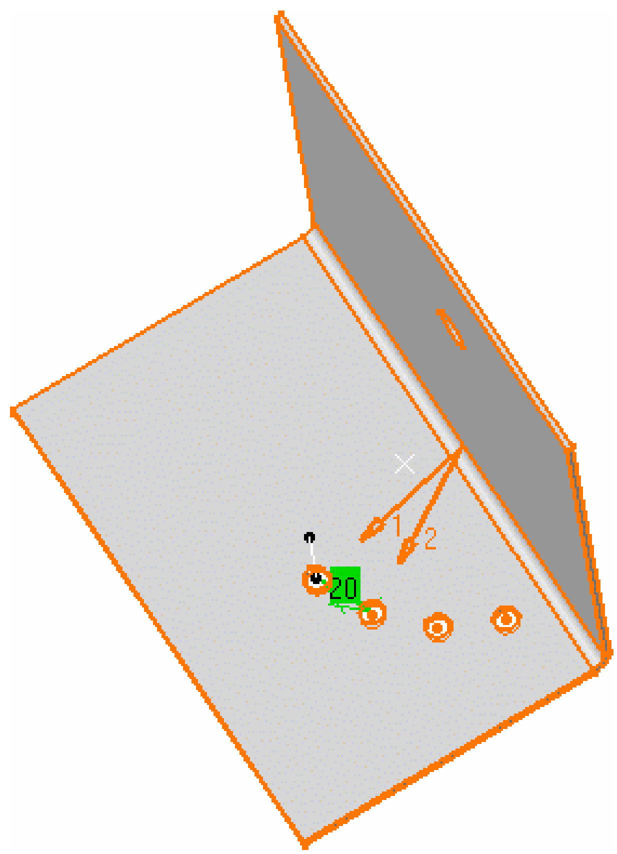
- **Instance(s) & total angle**: the number of patterns as specified in the instances field are created, in the specified direction, and evenly spread out over the total angle.



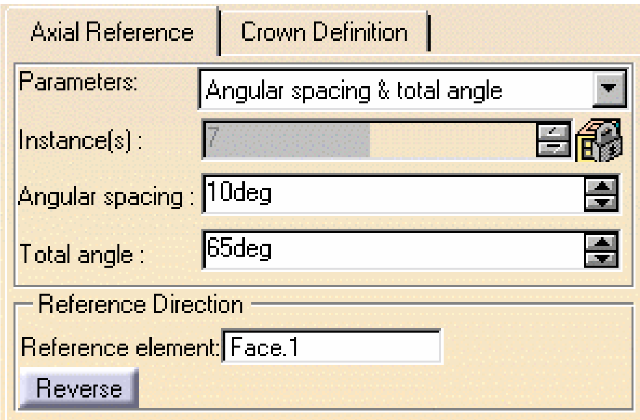


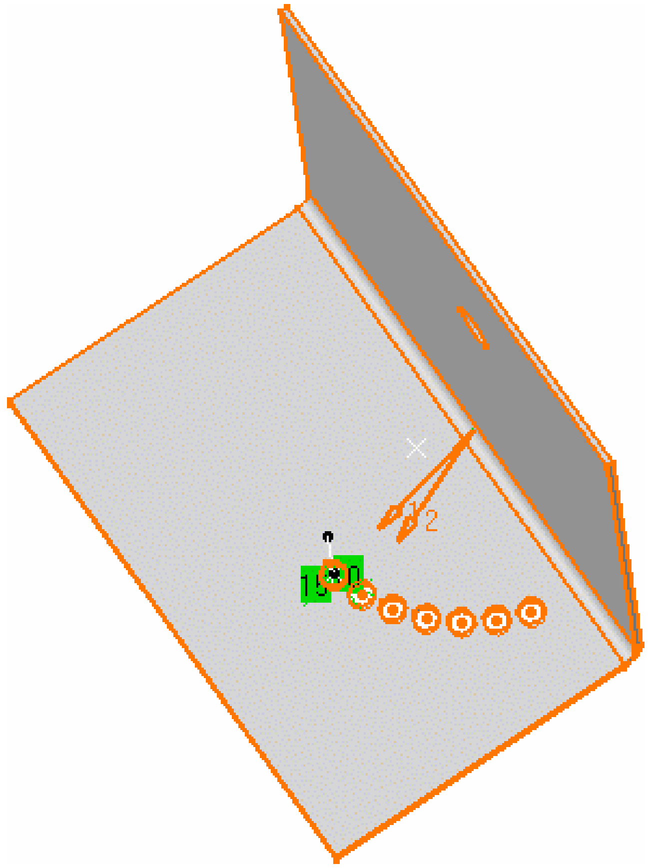
- **Instance(s) & angular spacing**: the number of patterns as specified in the instances field are created in the specified direction, each separated from the previous/next one of the angular angle value.



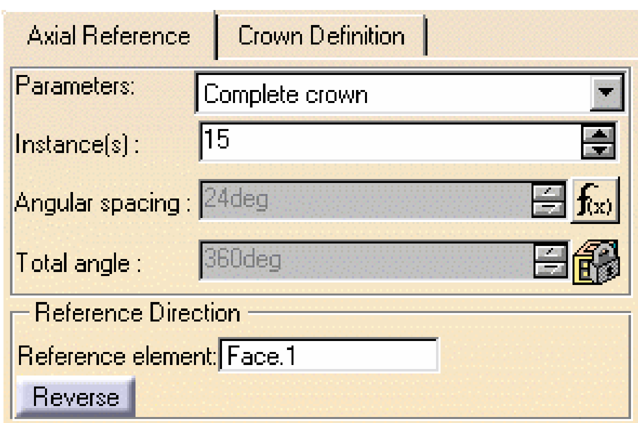


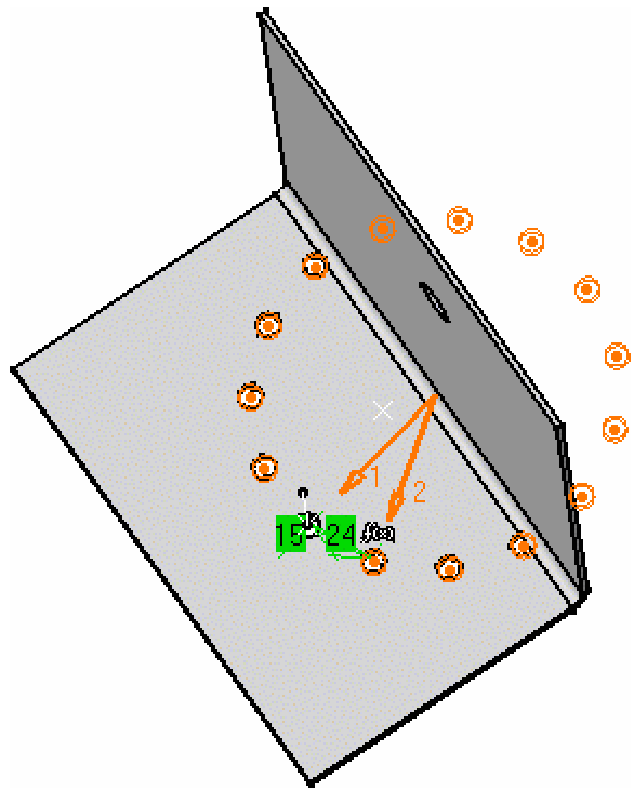
- **Angular spacing & total angle**: as many patterns as possible are created over the total angle, each separated from the previous/next one of the angular angle value.





- **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.

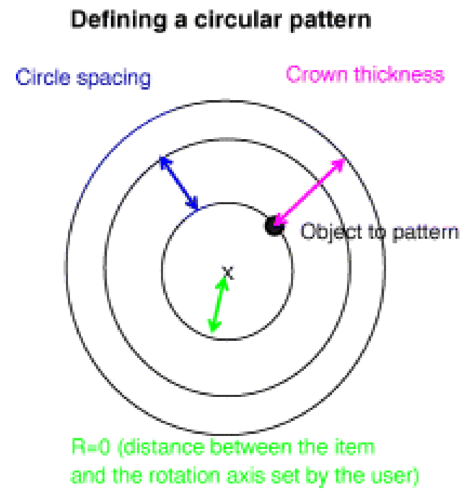
   Here select the face on which lies the circular cutout.

- 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.

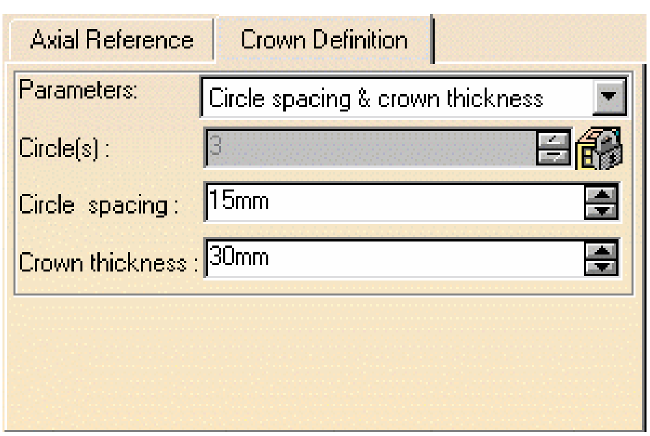
5. Click the **Crown Definition** tab, and choose which parameters you wish to define the crown.

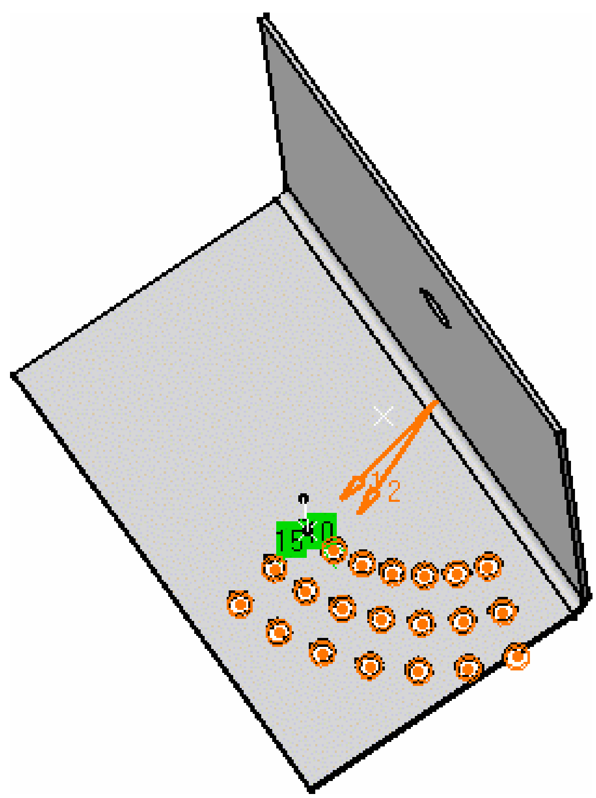
This figure may help you define these parameters:



- 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:

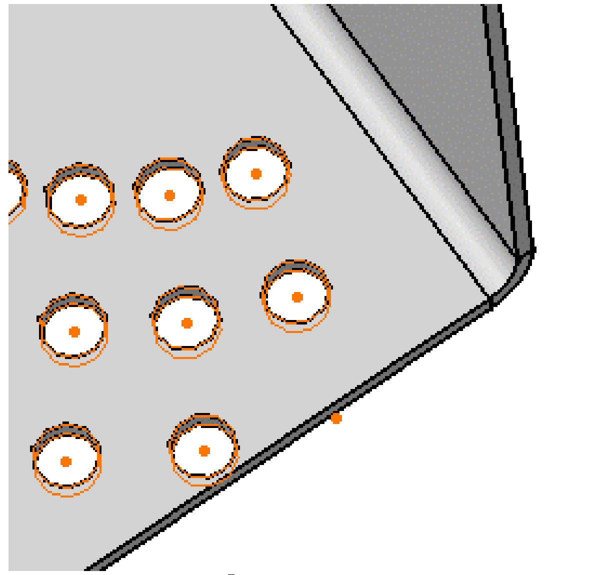




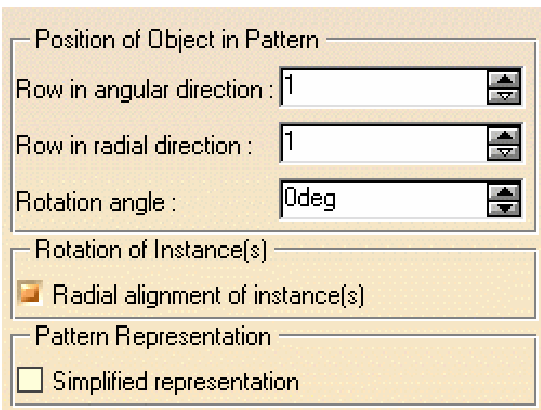
Note that one of the pattern is created beyond the wall.

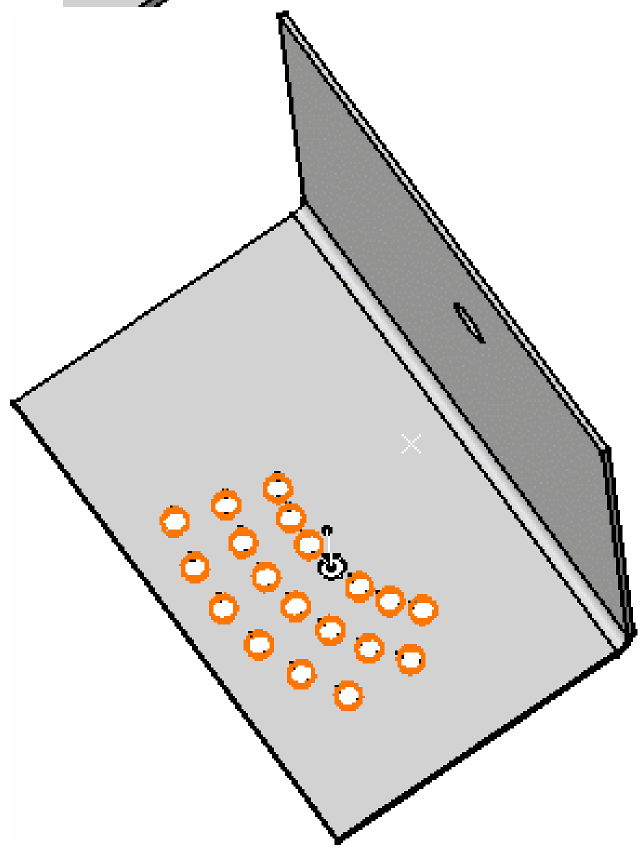
You can delete instances of your choice when creating or editing a pattern. To do so, just select the points materializing instances in the pattern preview.

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.

- When you duplicate a pattern of flange, the edge of the flange spine and its instances have to be tangent to the wall edge: you cannot choose a direction of patterning not parallel to the flange spine.
- All instances of the flange pattern must lie on the same face as the flange pattern.

Creating User-Defined Patterns

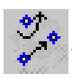
The User Pattern command lets you duplicate a cutout, a stamp, or any other feature as many times as you wish at the locations of your choice.
Locating instances consists in specifying anchor points. These points are sketches.

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.

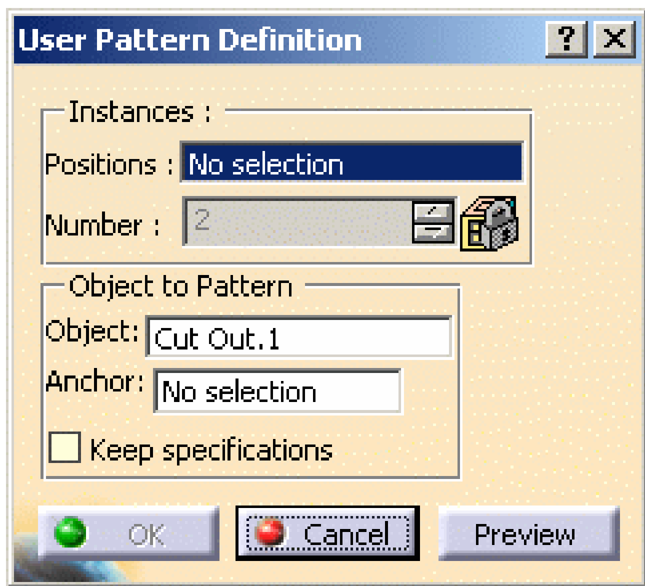
For the SheetMetal Design workbench, open the UserPatterns1.CATPart document.
For the Generative Sheetmetal Design workbench, open the NEWUserPatterns1.CATPart document.

1. Select the feature to be duplicated.

   Here we selected the cutout.

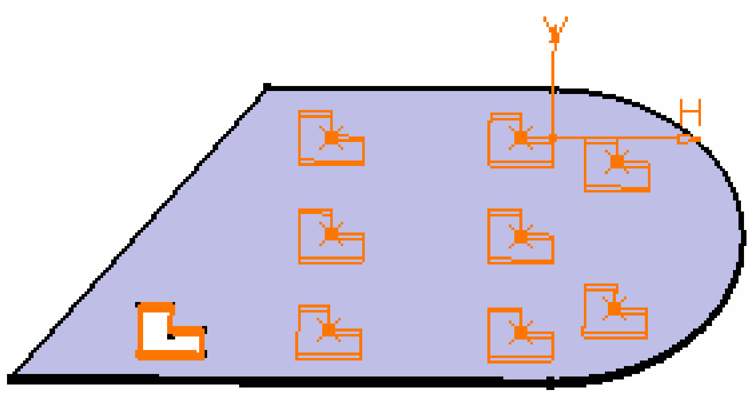
2. Click the **User Pattern** icon .

   The User Pattern Definition dialog box is displayed.



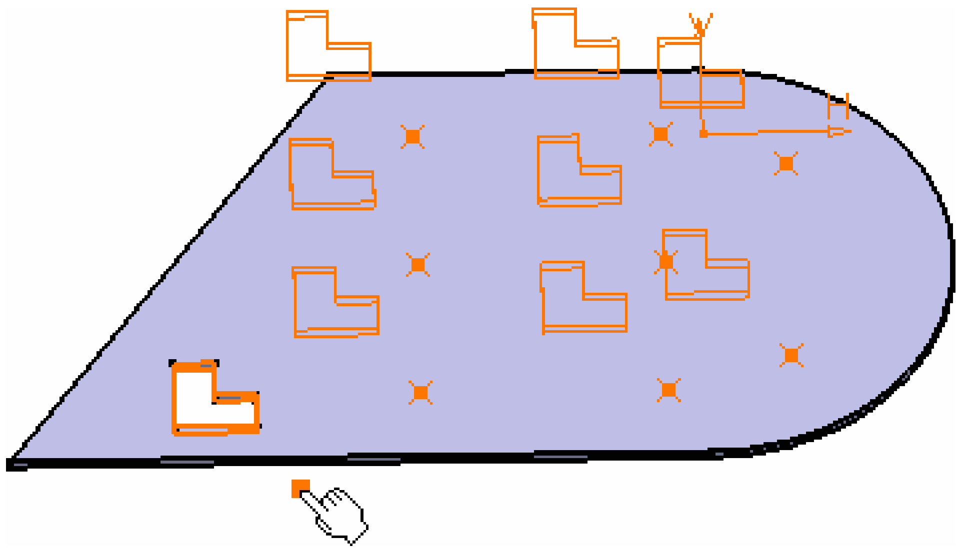
3. Select '**Sketch 3**' in the specification tree and click **Preview**.

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

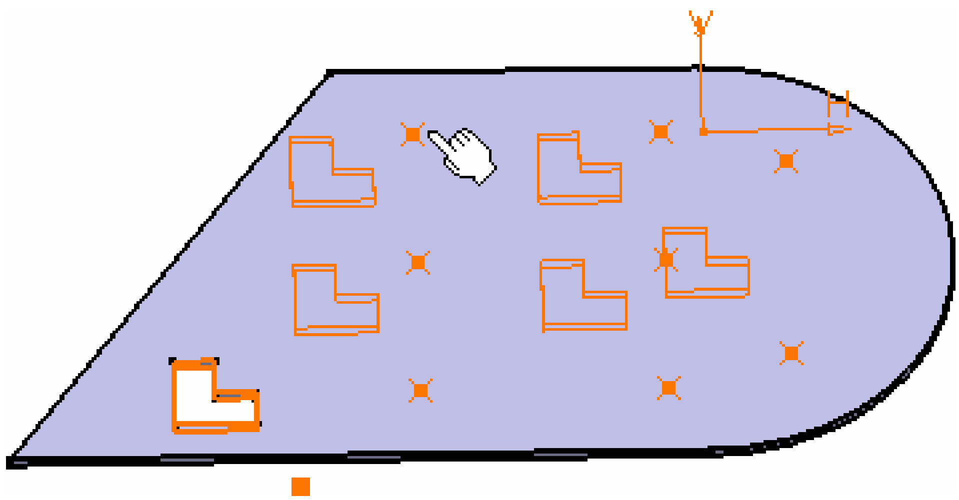


By default, the application positions each instance with respect to the center of gravity of the element to be duplicated. To change this position, use the anchor field: click the anchor field and select a vertex or a point.

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

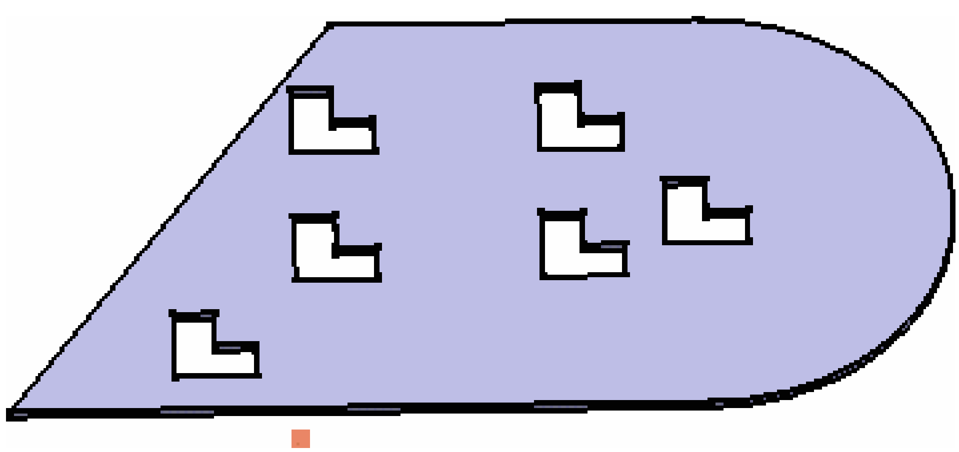


5. You can then click the points corresponding to the pattern instances to be removed.



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.

- When you duplicate a pattern of flange, the edge of the flange spine and its instances have to be tangent to the wall edge: you cannot choose a direction of patterning not parallel to the flange spine.
- All instances of the flange pattern must lie on the same face as the flange pattern.

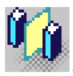
Mirroring

In this task, you are going to mirror a given feature, that is, to duplicate it symmetrically with respect to a plane.

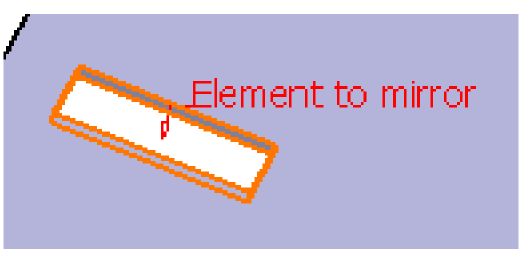
- You can mirror a hole, a cutout, a flange, a stamp, a pattern or another mirror.
- Whatever feature you choose to mirror, it must lie on a unique, planar wall.
- It is mandatory that the result of the mirroring should lie on the part, otherwise it cannot be mirrored.

Open the NEWMirror01.CATPart document.

A cutout and a relimited flange are displayed on the wall.

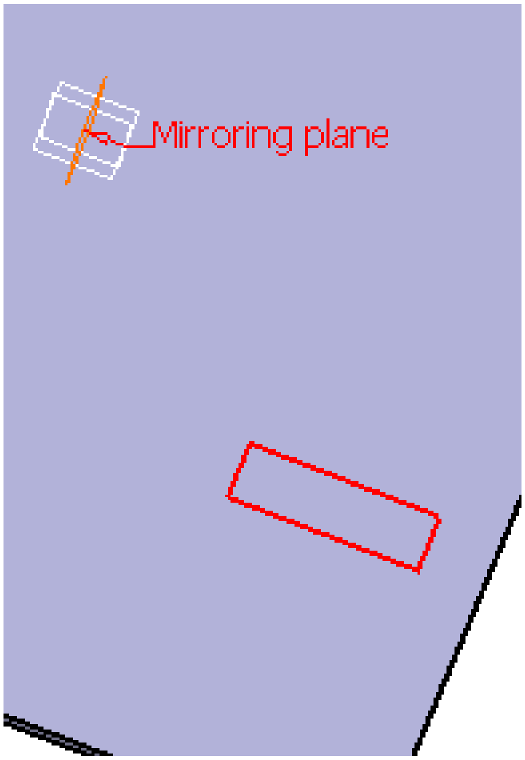
1. Click on the Mirror icon .
   The Mirror definition dialog box is displayed.

2. Select the feature to mirror, that is, Cut Out.1 in our example.

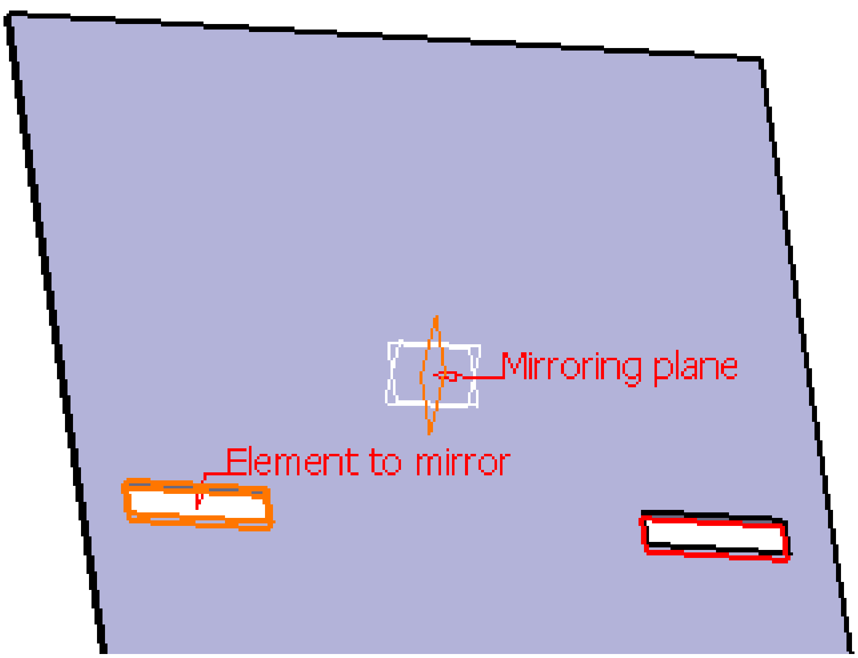


3. Select the zx plane in the 3D window.

The selected plane and a preview of the cutout to be mirrored is displayed.

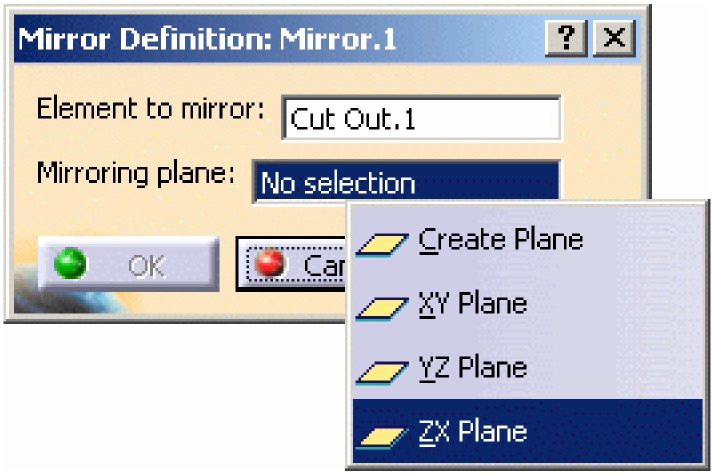


4. Click on Preview to visualize the mirrored cutout.

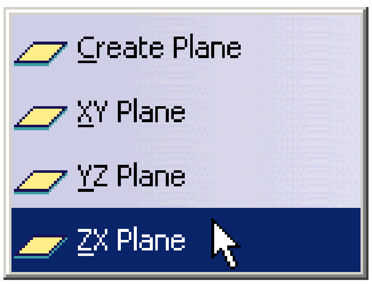


There is three different ways to select a mirroring plane :

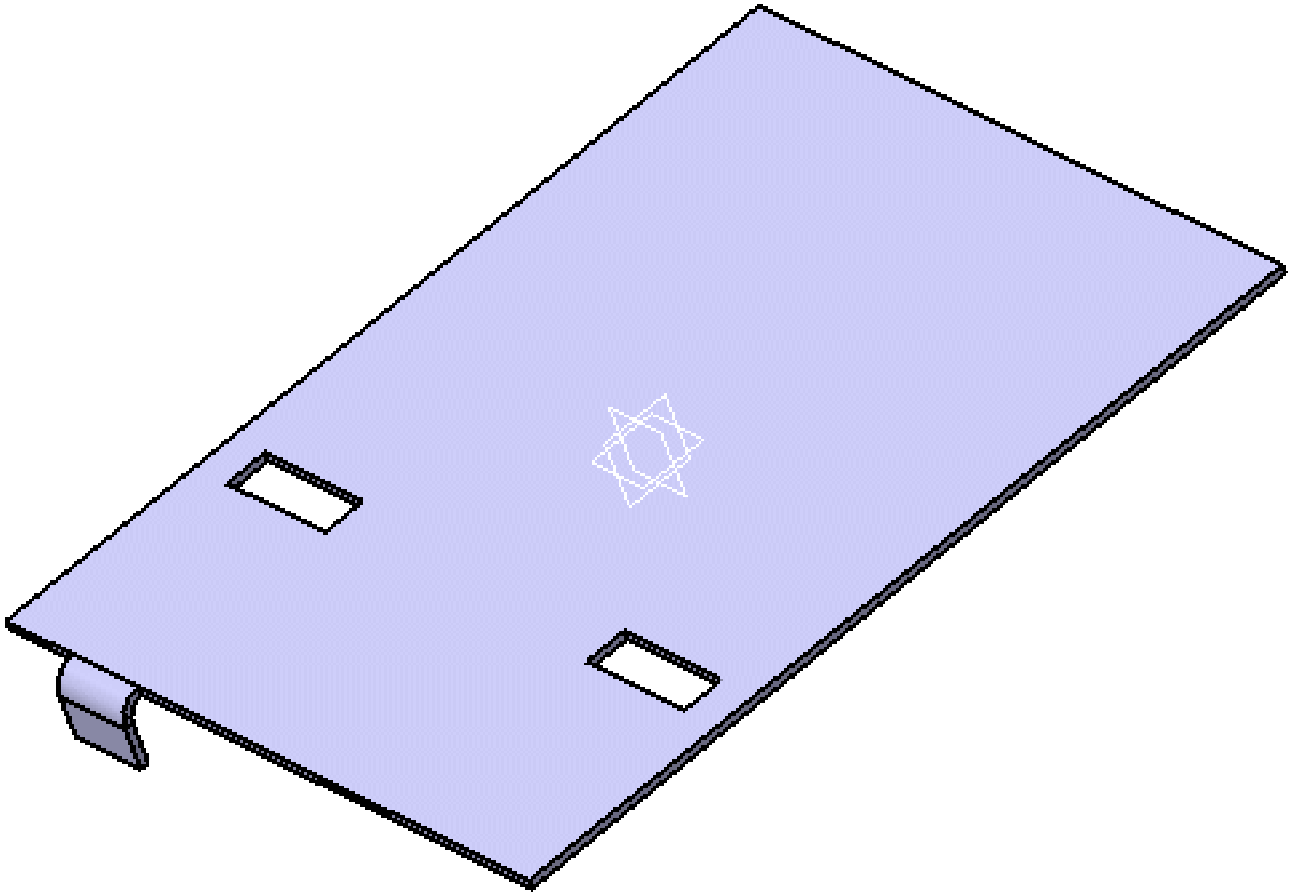
- In the 3D window, as in our example.
- with a right click on the Mirroring field in the Mirror definition dialog box.



- with a right click in the workbench background.



5. Click on Ok to create the mirrored cutout.

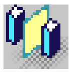


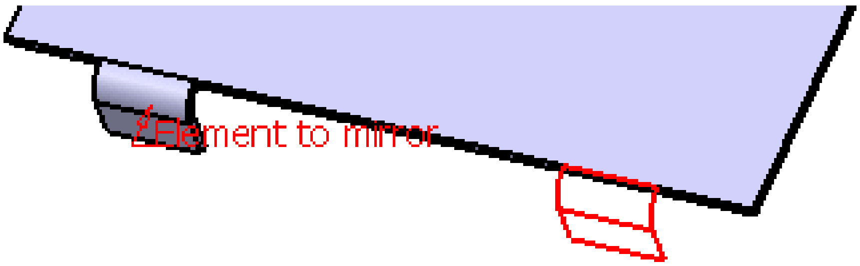
The mirror feature is displayed in the specification tree.

If you selected the mirroring plane with a right click, the plane feature is also displayed in the specification tree, allowing you to double click on it should you need to edit it.

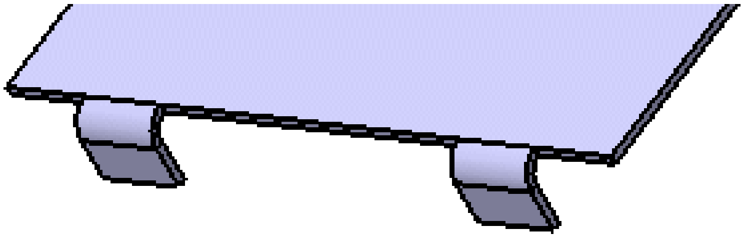
If you selected the mirroring plane with a right click, an icon is displayed in the **Mirror definition** dialogue box so that you can modify it if needed.

6. Select Flange.3.

7. Click on the Mirror icon  and select the zx plane.
   The flange to be mirrored is displayed.



8. Click on Preview if you want to visualize the mirrored flange, then on **OK** to create it.



When mirroring a flange, it is mandatory that the result of the mirror should lie on an edge.

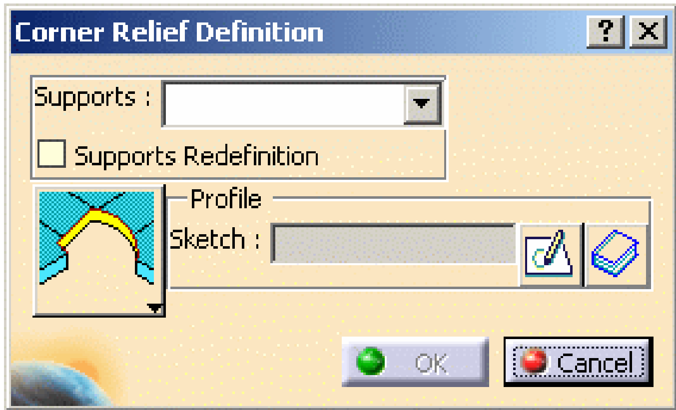
Creating a Local Corner Relief

This task explains how to define a corner relief locally on a set of supports.

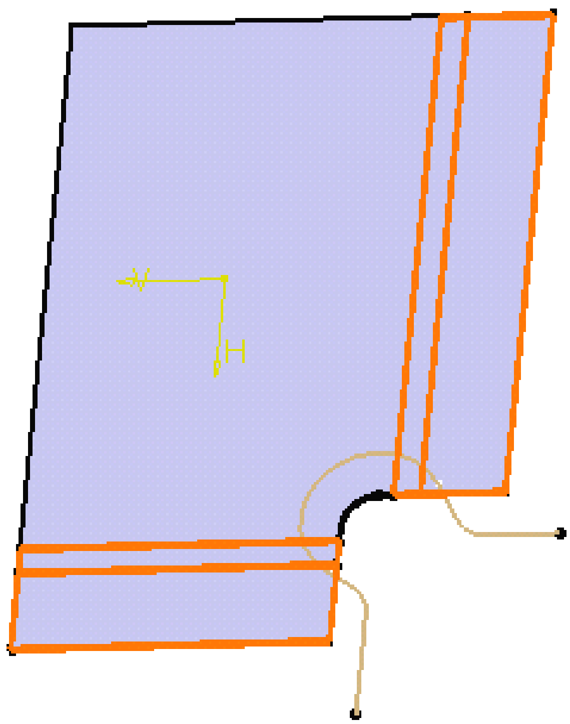
Open the CornerRelief01.CATPart document from the samples directory.
The part needs to be unfolded prior to creating the corner relief.

1. Click the **Corner Relief** icon.

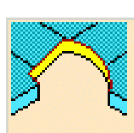
   The Corner Relief Definition dialog box is displayed.



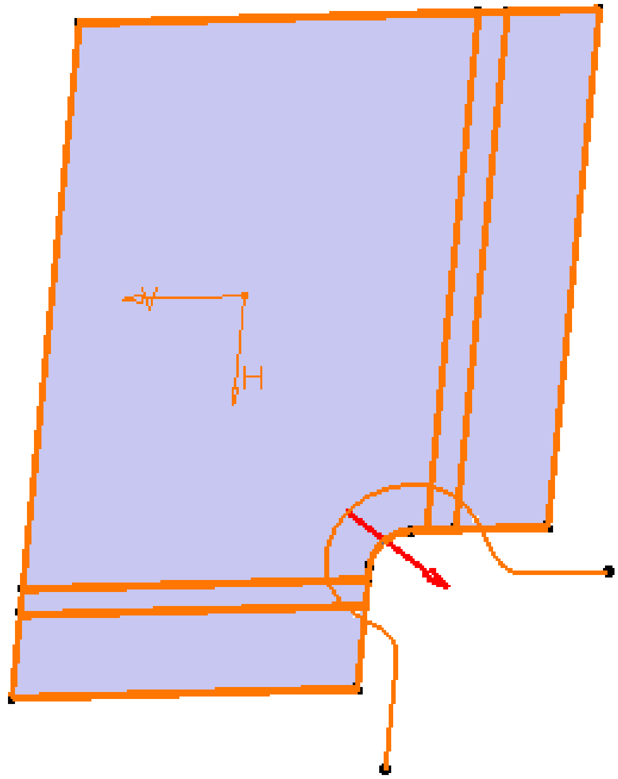
2. Select the supports on which a corner relief should be created (here we chose Surfacic Flange.1 and Surfacic Flange.2)

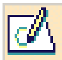


A notch was defined on the web profile between the two fillets' flanges; so that flanges do not intersect. This operation enables to prepare the web as to create the flanges that will be later used to define the corner relief.

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

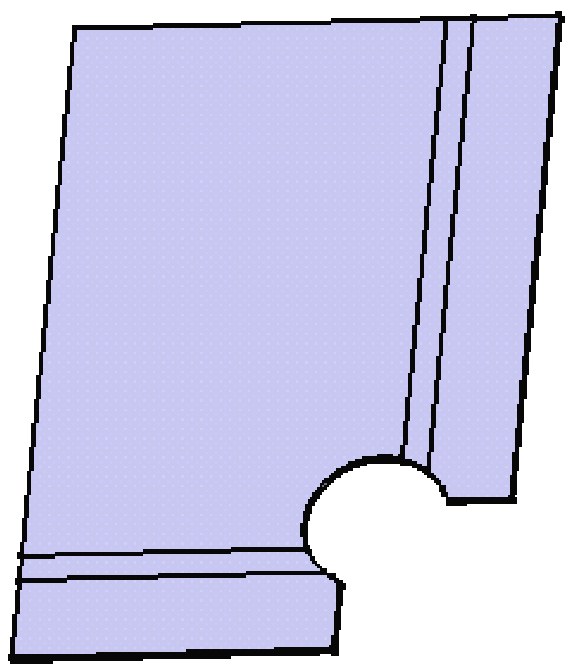
1. Select the sketch, directly in the 3D geometry.

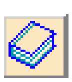


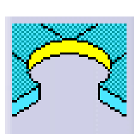
As soon as the sketch has been selected, the **Sketcher**  icon is displayed in the dialog box allowing you to edit the selected sketch, if needed.

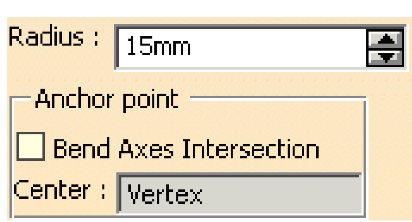
The red arrow lets you choose the direction of matter to remove. Click it to reverse the direction.

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



You can use the **Catalog** icon  to open the Catalog Browser.
For more information on catalogs, please refer to the Using Catalogs chapter in the *CATIA Infrastructure User Guide*.

- Select the **Circular Profile**  using the down arrow.

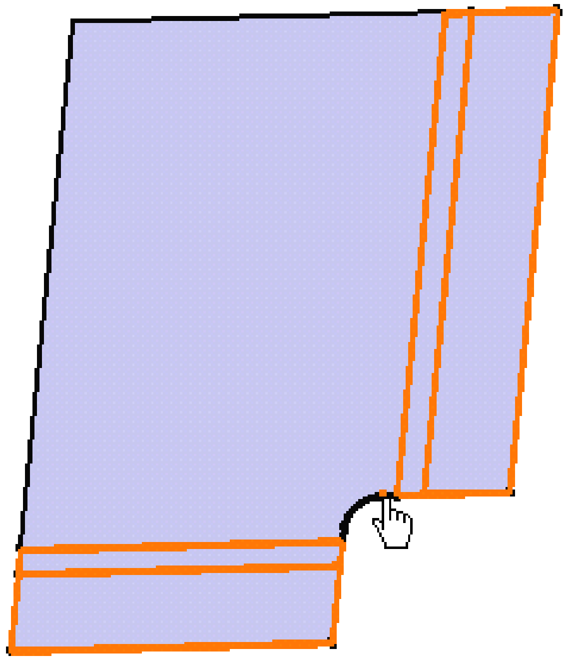


1. Define the default radius: it is equal to the bend radius + the thickness.

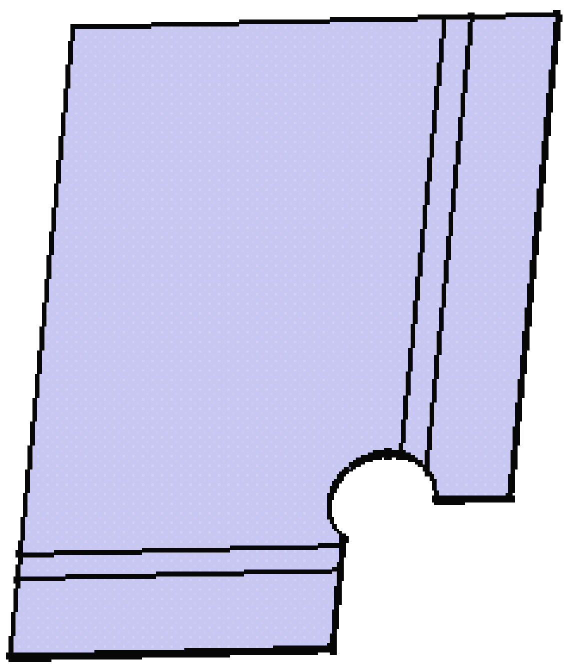
   In our example, we defined a radius of 15 mm.

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.

2. Select the vertex between the two flanges: it will be the center of the corner relief.

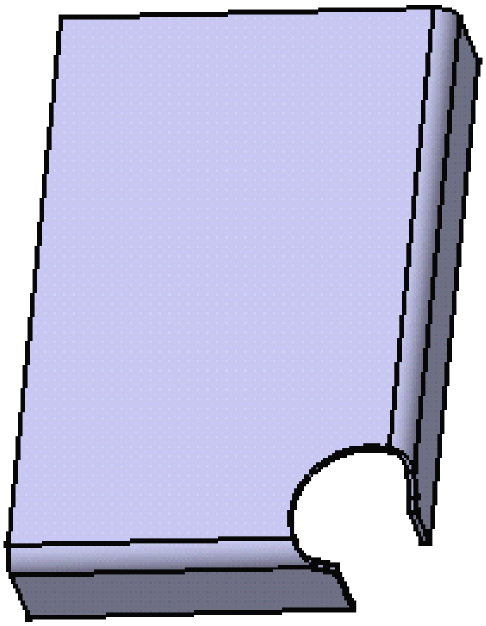


3. Click OK in the Corner Relief Definition dialog box.

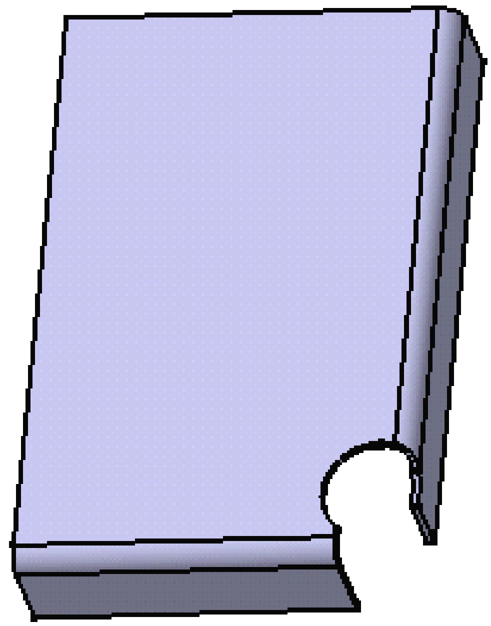


The created element (identified as Corner Relief.xxx) is added to the specification tree.

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

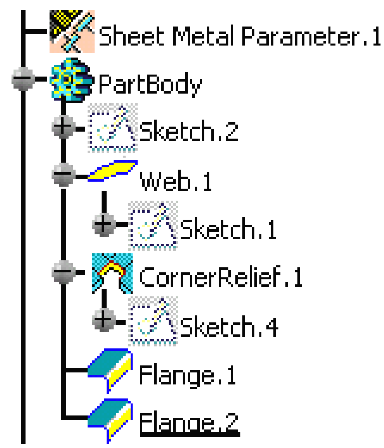


Folded user corner relief



Folded circular corner relief

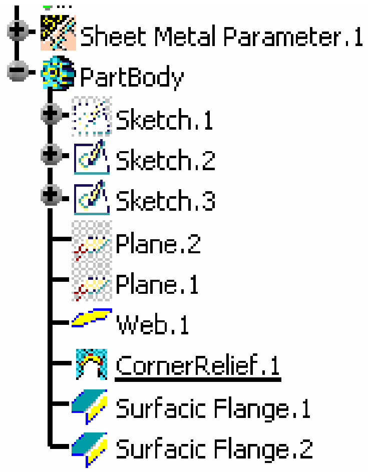
The **Supports Redefinition** checkbox enables to redefine the supports' sides thus adding matter to these supports.
In that case, the created element (identified as Corner Relief.xxx) appears before the supports in the specification tree.



- Please note that checking this button means that the corner relief replaces the surfacic flange's side. This side must therefore exists: when creating the surfacic flange, do not define the side as **None**.

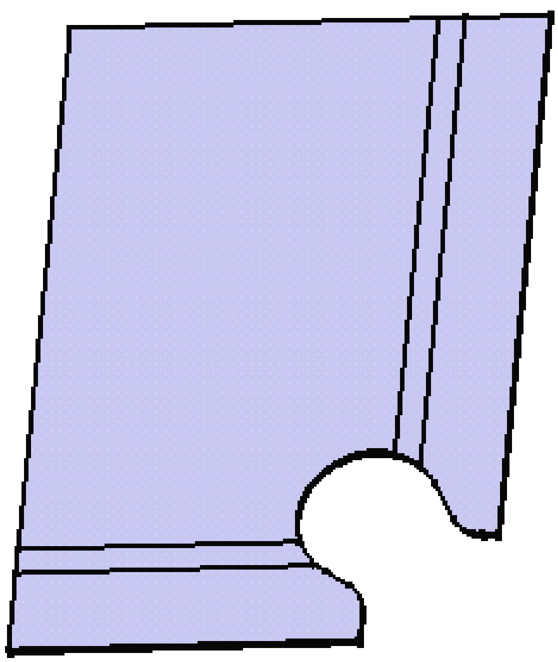
- In hybrid context, when checking **Supports Redefinition**, the Surfacic flanges are hidden in the 3D since the define in work object parameter is applied to the corner relief.

- Moreover, the sketch is not aggregated anymore under the corner relief in the specification tree.

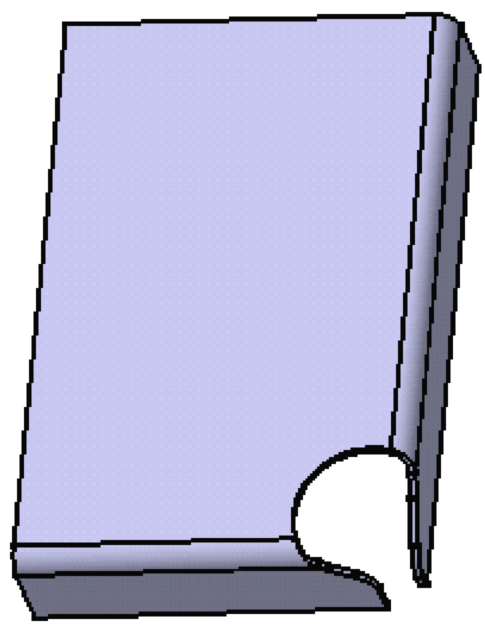


Yet, if you open a part created in a previous release, the specification tree will be displayed accordingly to the previous behavior.

Fore more information about Hybrid Design, refer to the Hybrid Design section.

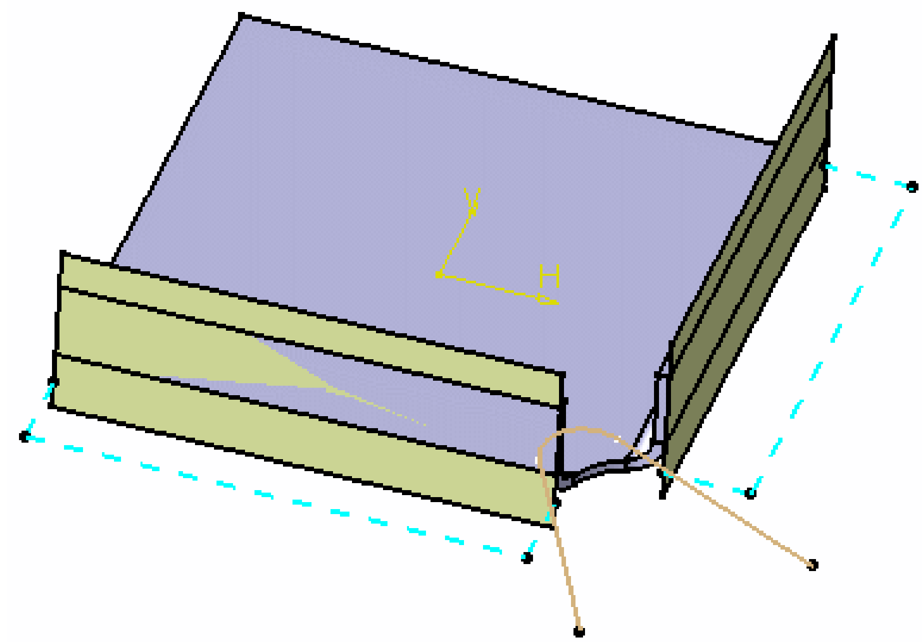


Unfolded user corner relief with redefined supports

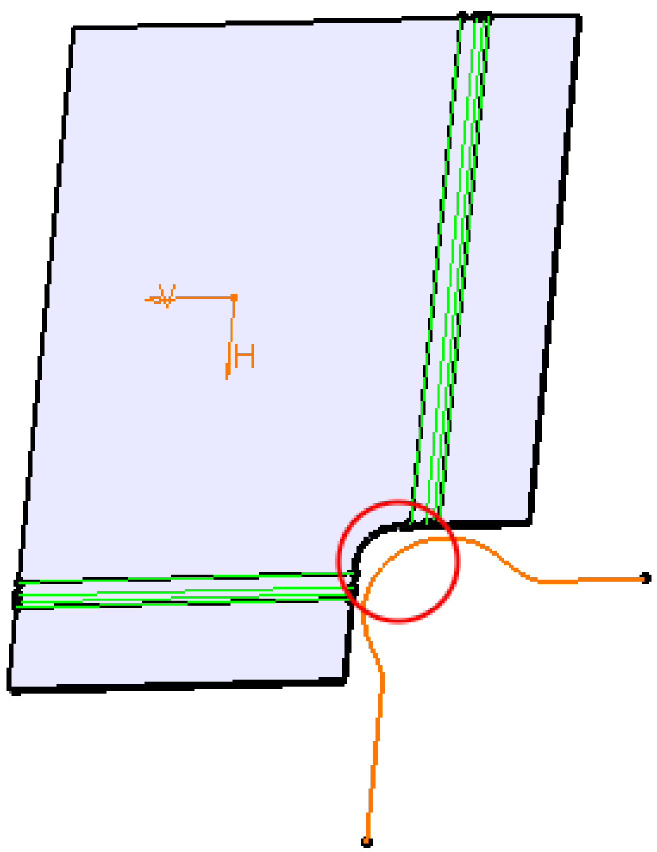


Folded user corner relief with redefined supports

The image besides shows two surfacic flanges creating with Angle as support type. The two blue dotted lines represent the limits of the unfolded surfacic flanges.



- The creation of a corner relief with supports redefined is not possible as it is not located within the limits of the unfolded flanges.
- A corner relief with supports redefined cannot be created if its profile implies adding matter to the web.

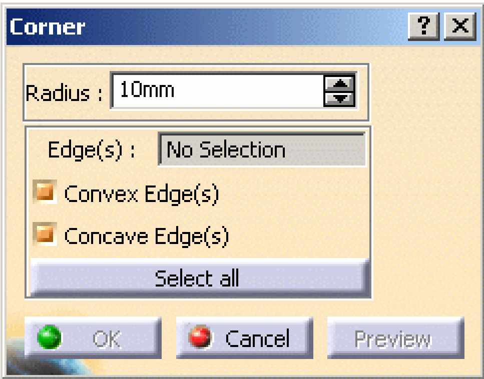


Creating Corners

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 Corner1.CATPart document.
If you use Aerospace SheetMetal Design, open the Corner_Aero1.CATPart document.

1. Click the **Corner** icon . The Corner dialog box is displayed.



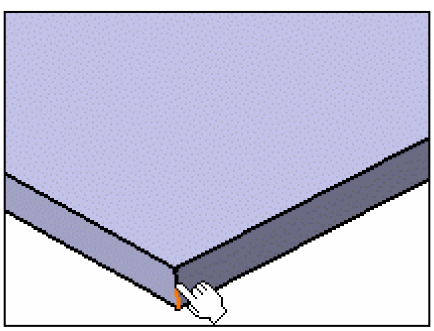
2. Set the radius value.

3. Choose the type of edge you wish to round off: **Convex Edge(s)** and/or **Concave Edge(s)**. For the purpose of this scenario, leave both options selected.

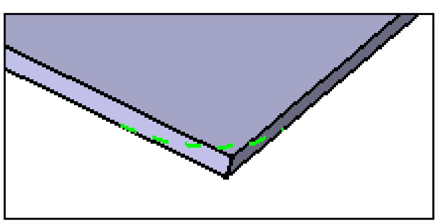
Once you have selected an edge, you can no longer modify the chosen options (they are grayed out), unless you cancel the selection.

4. Click to select a convex edge on a part.

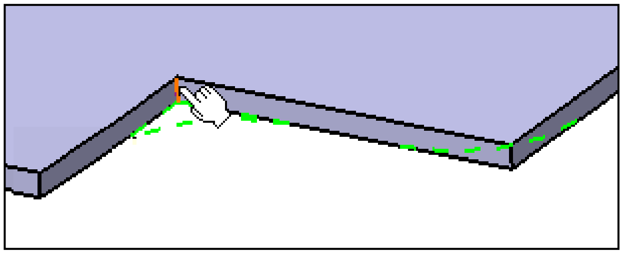
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.



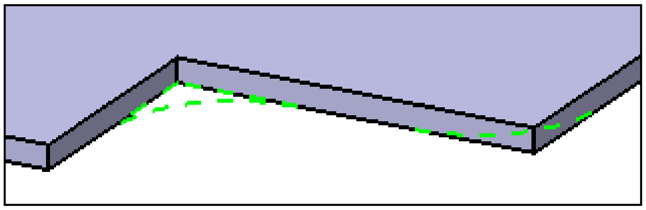
The corner is previewed on the edge, with the current radius value.



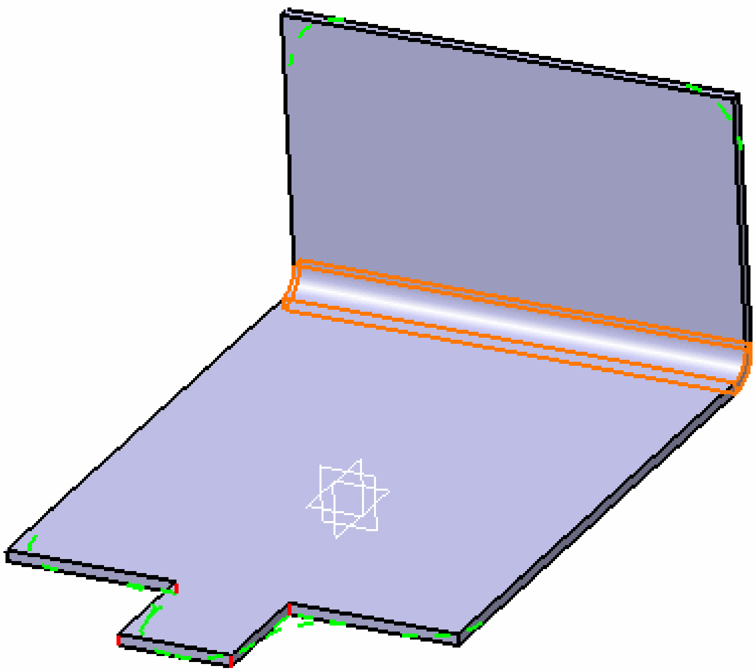
5. Click to select a concave edge on a part.



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

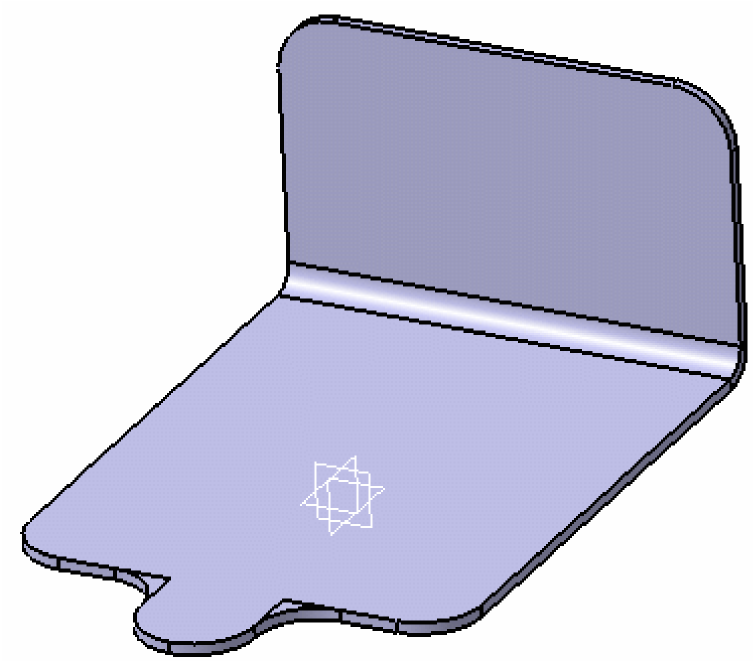


6. Click **Cancel Selection** then click the **Select All** button. All edges of the part are selected and the corners previewed.



7. 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 present at the time. If when modifying the part, new edges are created, these will not be automatically rounded off.

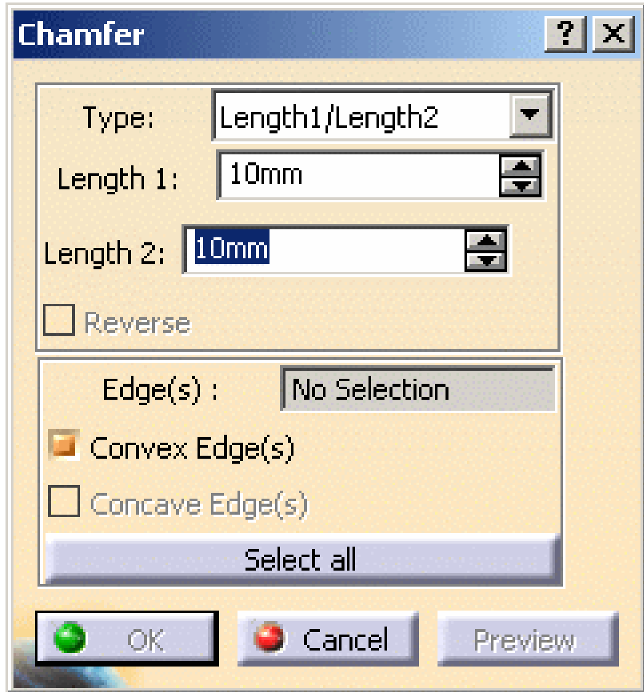
Creating Chamfers

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 Corner1.CATPart document.
If you use Aerospace SheetMetal Design, open the Corner_Aero1.CATPart document.

1. Click the **Chamfer** icon .

The Chamfer Definition dialog box is displayed.



You can choose the type of edge you wish to chamfer:

- using the **Select All** button, you can select all convex edges on the part
- any edge you select manually.

2. Leave the **Convex Edge(s)** option selected.

3. Select a sharp edge on a part.

If you want to create a longitudinal chamfer, you can select a single long edge. This allows you to create a welding chamfer, for example.

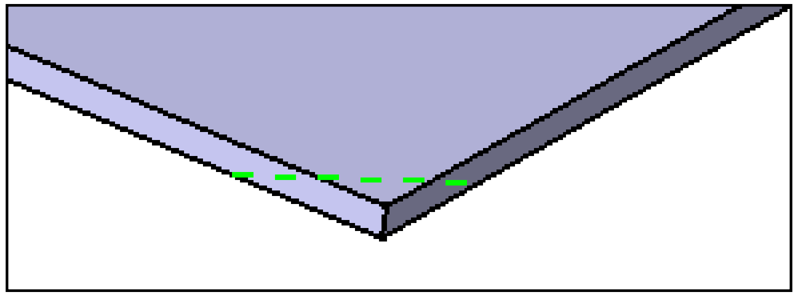
The chamfer is previewed on the edge.

Remember that when you create a chamfer on one edge it is automatically propagated on the tangent edge.

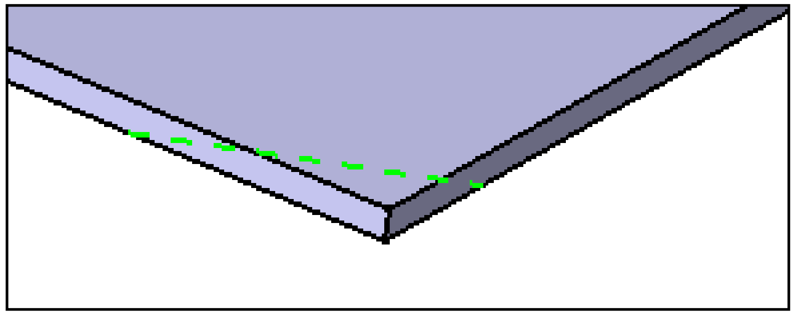
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.

3. Choose a chamfer mode. You can either enter:

- two lengths: these lengths are computed from the selected edge on both sides.
  Here, we chose two lengths of 10mm.



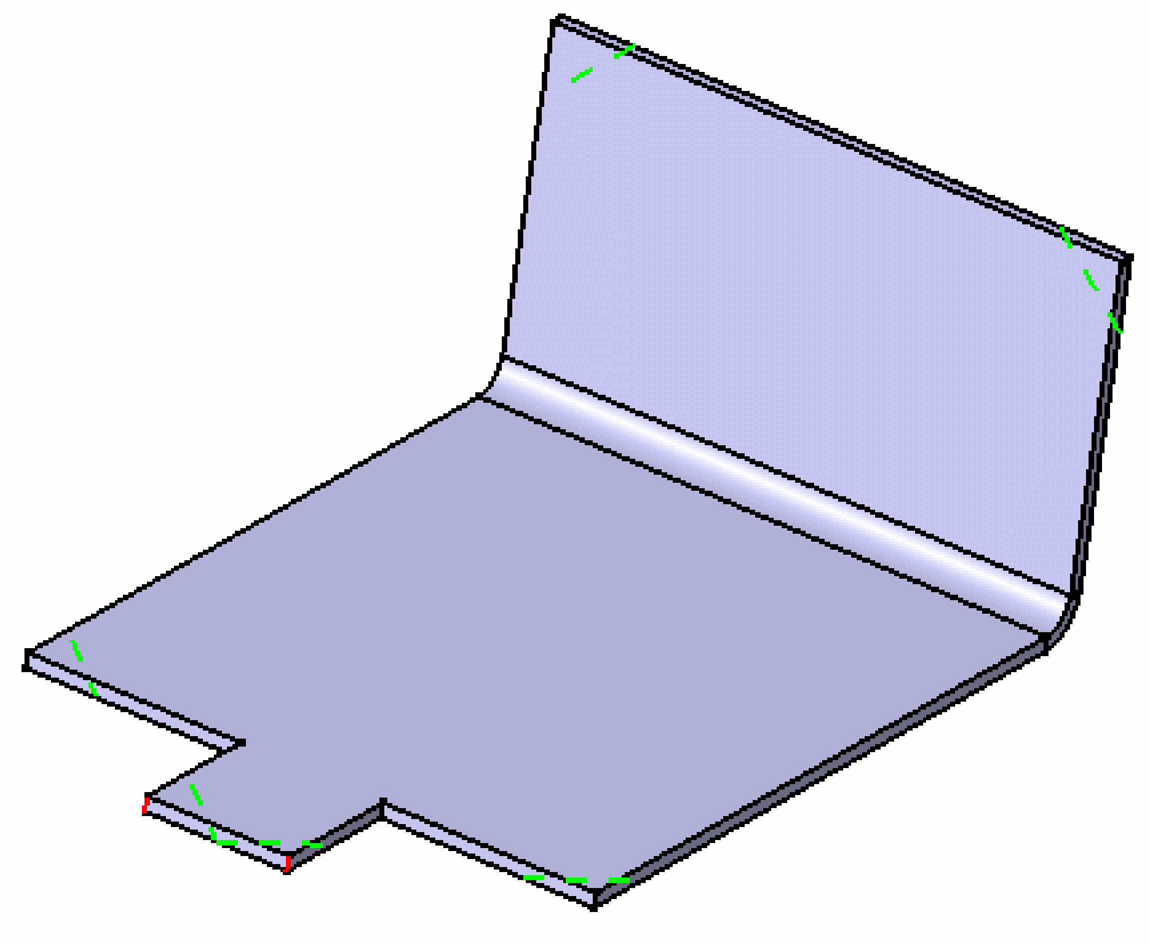
- a length value and an angle: the length is computed on one side of the edge and the angle from the chamfer's limit on the same side.
  Here, we chose a length of 10mm and an angle of 60deg.



You can use the **Reverse** button to inverse all edges' side, on which the values are taken into account.

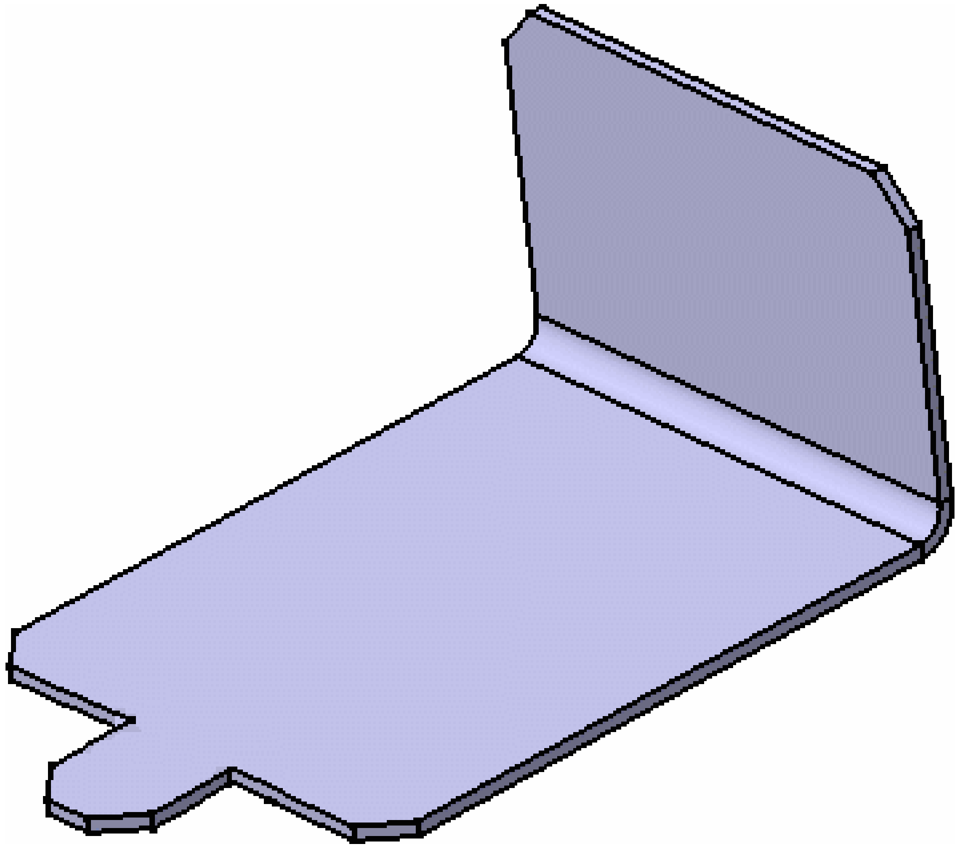
4. Click **Cancel Selection** then click the **Select All** button.

All sharp edges of the part are selected, the **Select All** button taking into account the chosen type and the chamfers previewed.



5. 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 present at the time. If when modifying the part, new edges are created, these will not be automatically chamfered.
- When the sharp edge is selected in the thickness of the wall, its length has to be equivalent to the wall's thickness.
- If the sharp edge is not selected in the thickness of the wall, it has to limit the faces of the wall.

Mapping Elements

P2

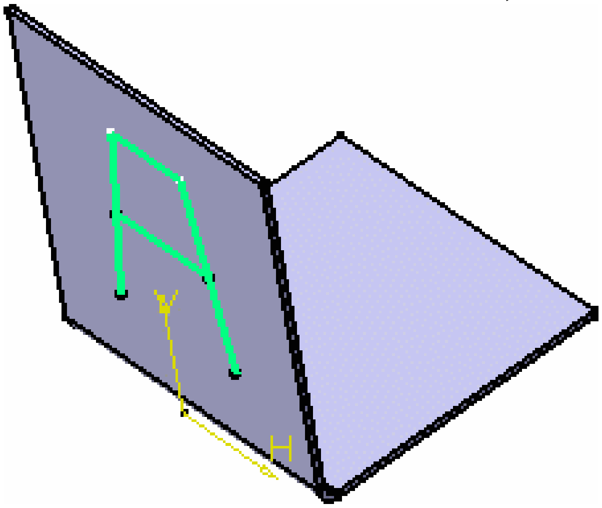
This task shows how to create curves or points from a sketch (as designed using the Sketcher) or from existing curves or points, onto a Sheet Metal part; and to fold/unfold it, just as other Sheet Metal elements.
This is especially useful when:

- 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.

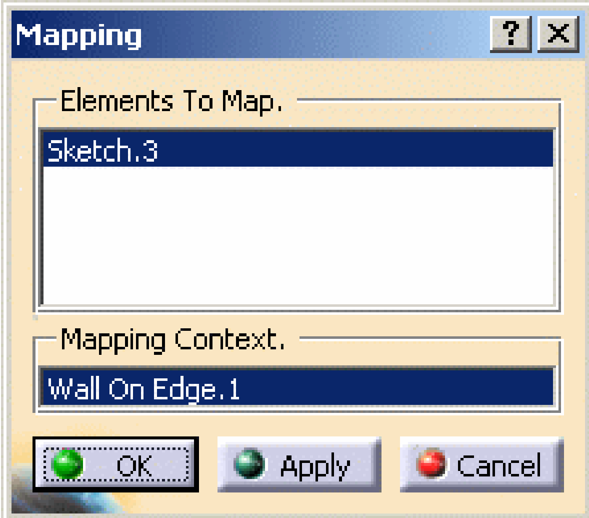
These samples already contain a pre-defined sketch that will be mapped onto the part.

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.



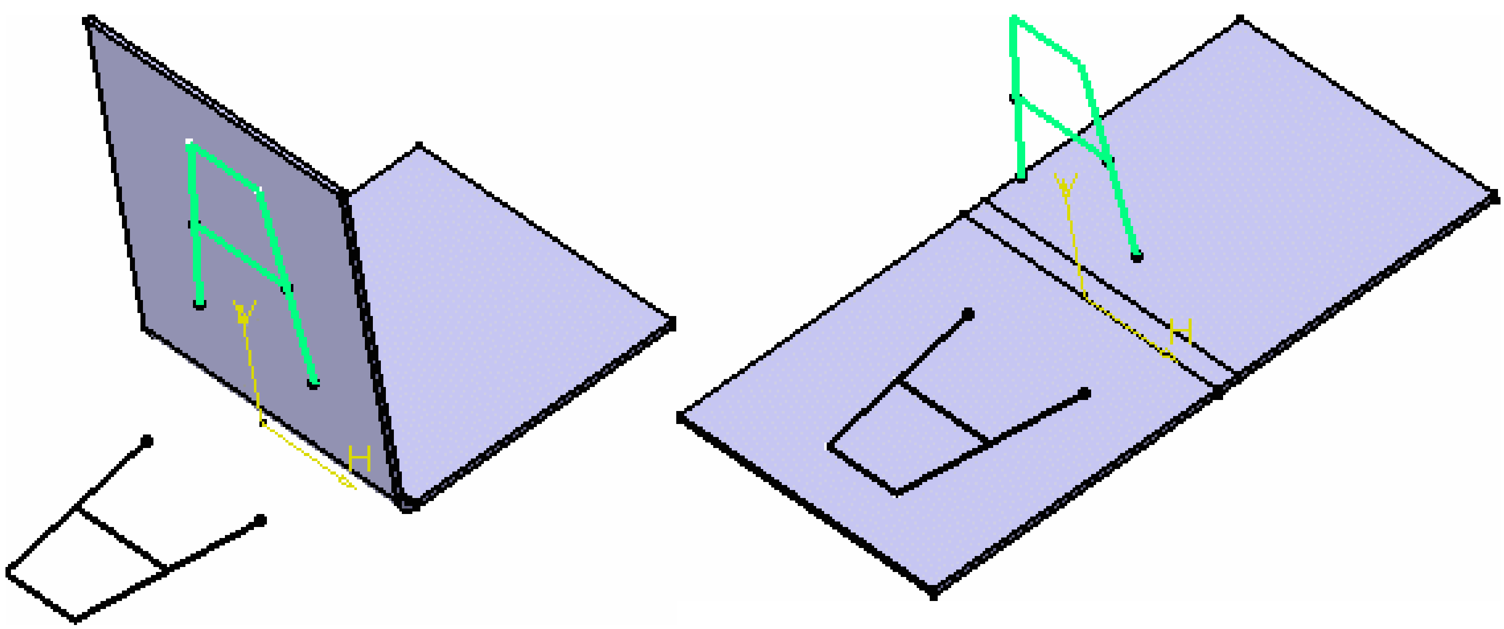
You can manage the list of elements:

- 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.

Interoperability with Wireframe

Creating Points

This task shows the various methods for creating points:

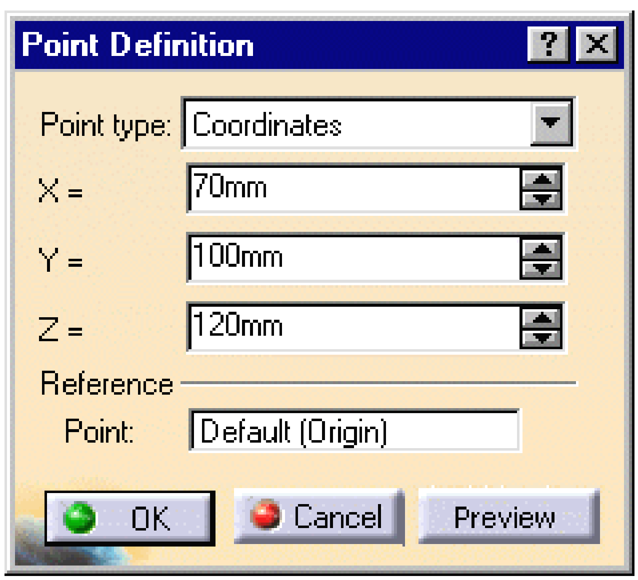
- by coordinates
- on a curve
- on a plane
- on a surface
- at a circle/sphere center
- tangent point on a curve
- between

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.



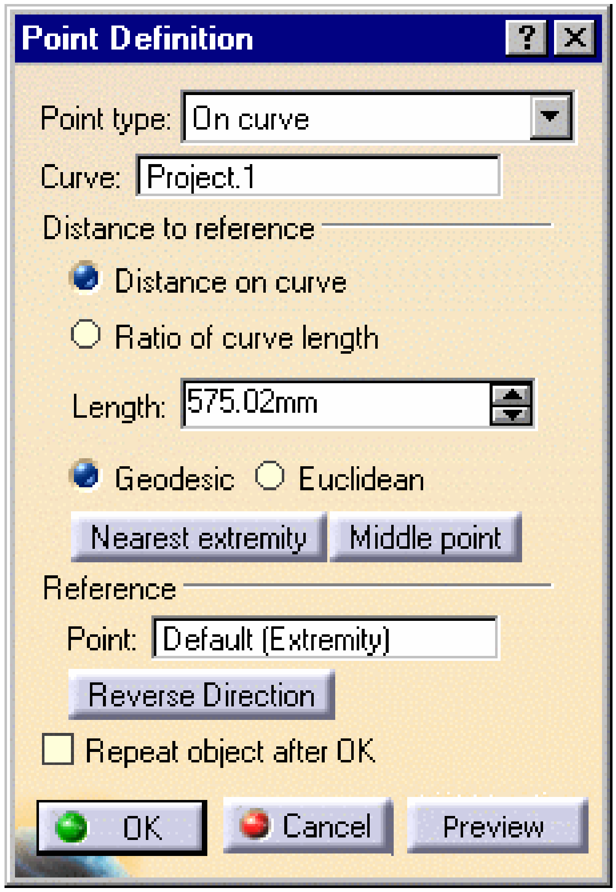
Coordinates

- 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.

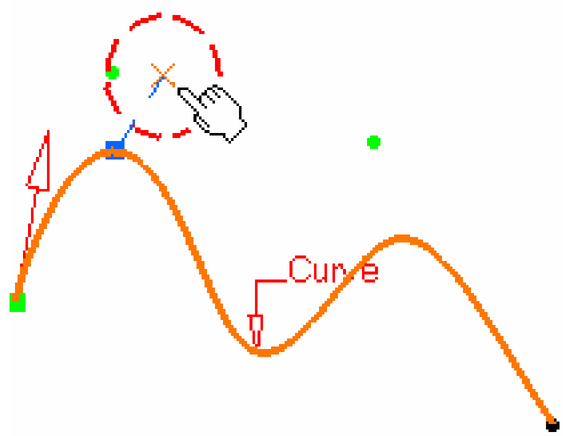


On curve

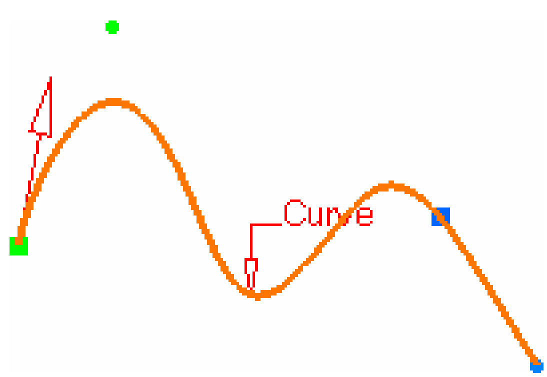
- Select a curve
- Optionally, select a reference point.

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:

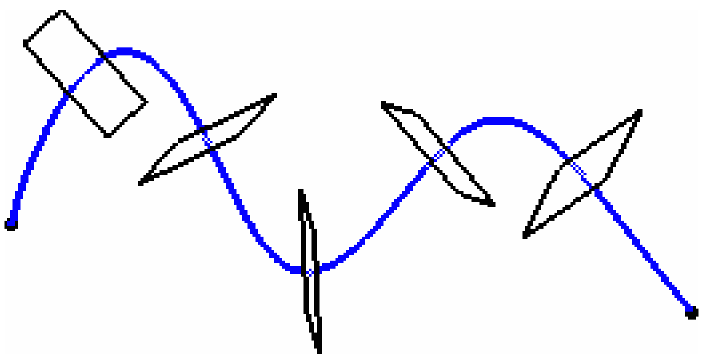
- click the **Nearest extremity** button to display the point at the nearest extremity of the curve.
- click the **Middle Point** button to display the mid-point of the curve.

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).

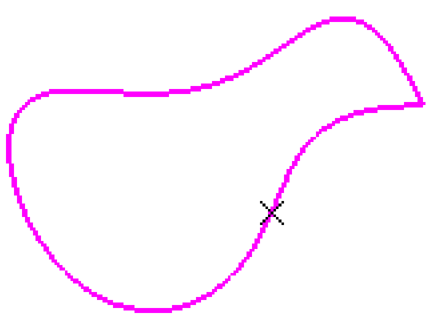
- click the **Repeat object after OK** if you wish to create equidistant points on the curve, using the currently created point as the reference, as described in Creating Multiple Points in the Wireframe and Surface User's Guide.

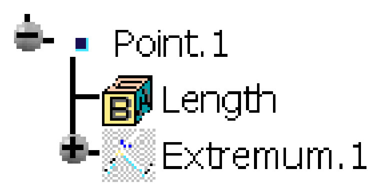
You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new geometrical set by checking the **Create in a new geometrical set** button.
If the button is not checked the instances are created in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

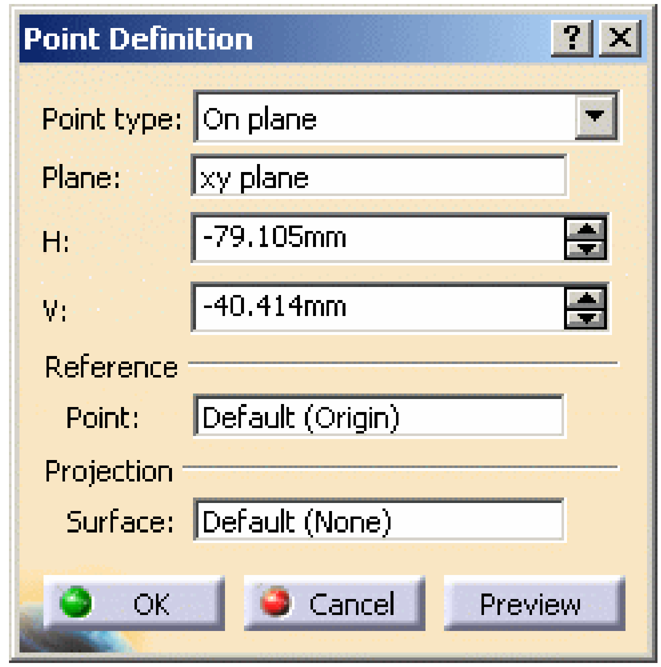
Extremum points created on a closed curve are now aggregated under their parent command and put in no show in the specification tree.





On plane

- Select a plane.



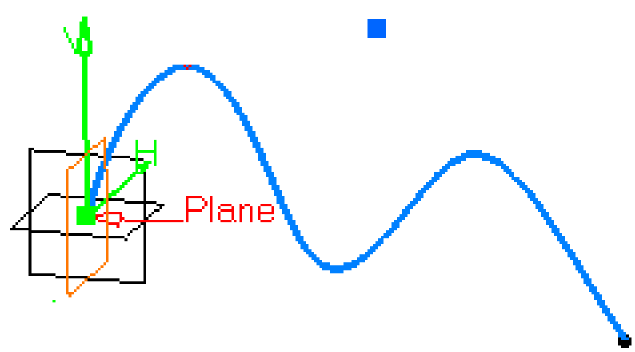
- Optionally, select a point to define a reference for computing coordinates in the plane.

  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.

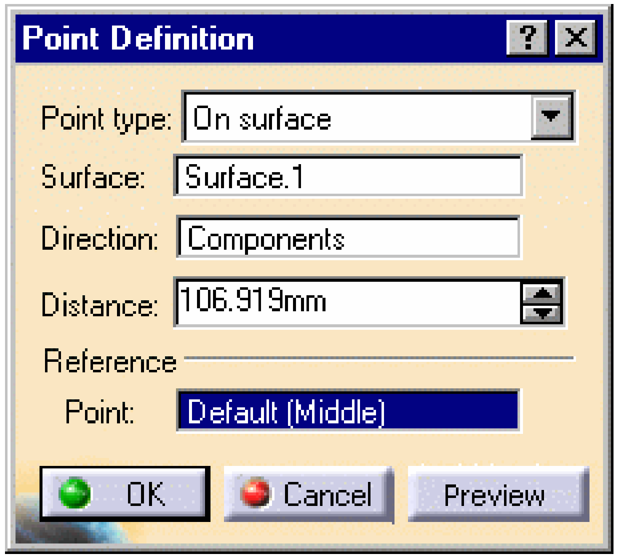


  Would the plane move, during an update for example, the reference direction would then be projected on the plane.

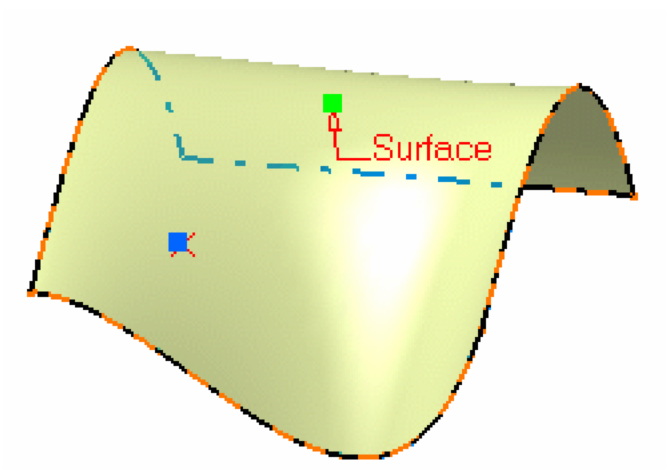
- Click in the plane to display a point.

On surface

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



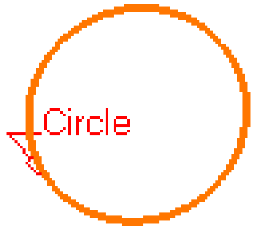
- Optionally, select a reference point. By default, the surface's middle point is taken as reference.
- You can select an element to take its orientation as reference direction or a plane to take its normal as reference direction.
  You can also use the contextual menu to specify the X, Y, Z components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

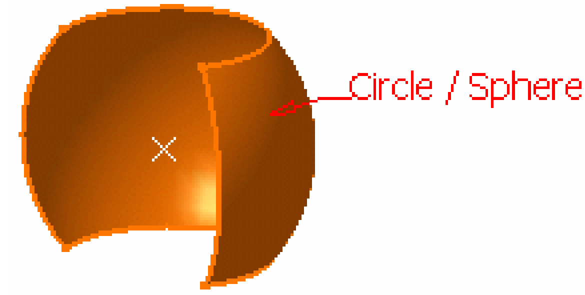


Circle/Sphere center

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

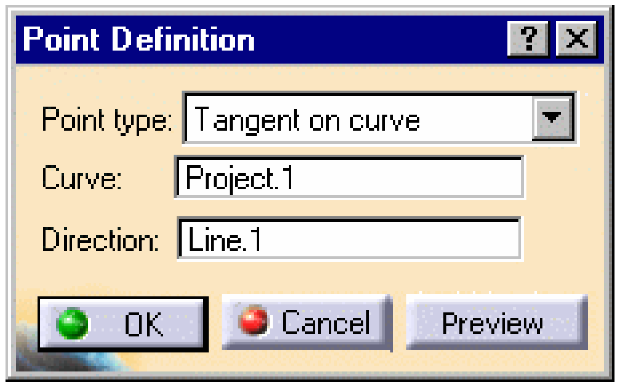
A point is displayed at the center of the selected element.





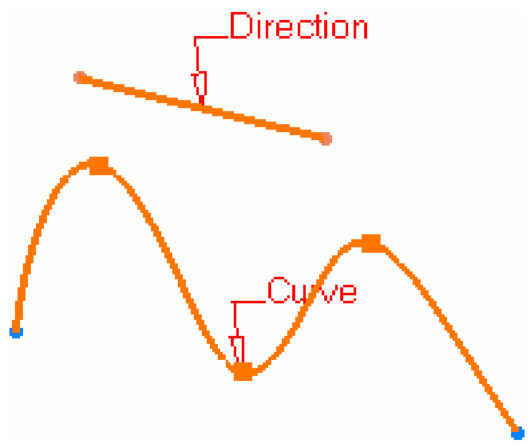
Tangent on curve

- 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.

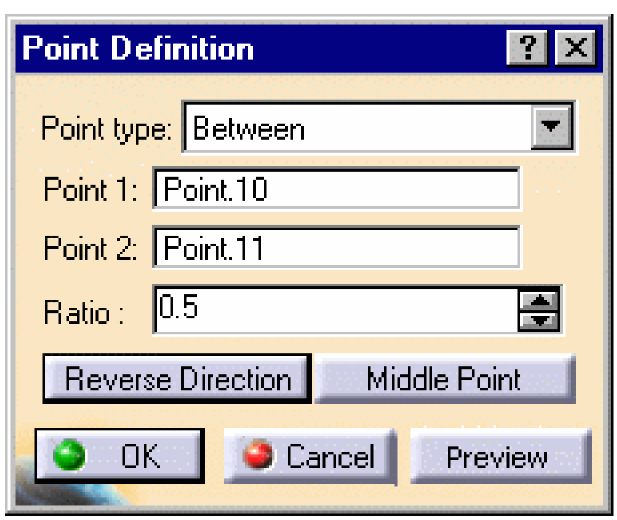


- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.

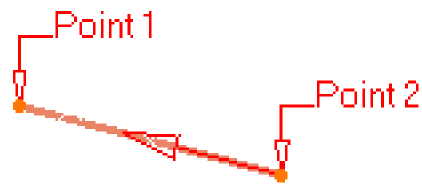
For further information, refer to the Managing Multi-Result Operations chapter.

Between

- Select any two points.

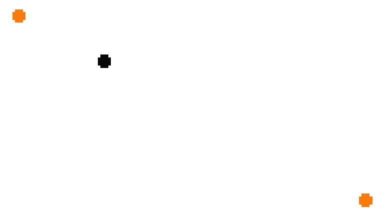


- Enter the ratio, that is the percentage of the distance from the first selected point, at which the new point is to be.
  You can also click **Middle Point** button to create a point at the exact midpoint (ratio = 0.5).



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.

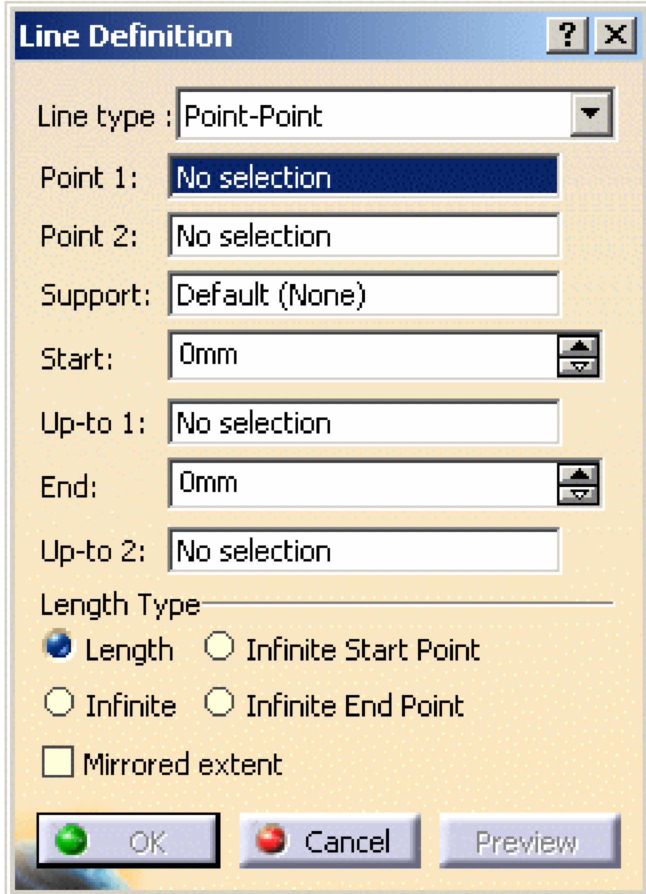
Defining the line type

Point - Point

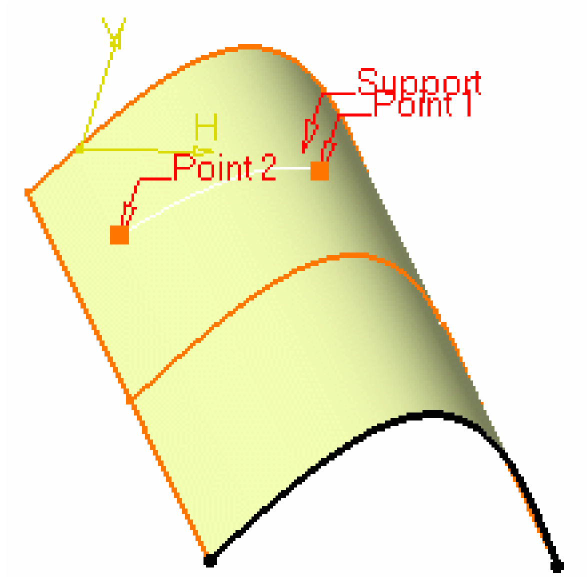
This command is only available with the Generative Shape Design 2 product.

- Select two points.

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

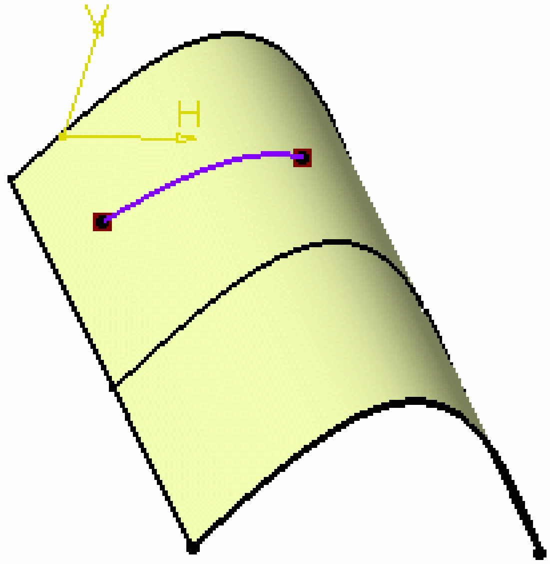


- If needed, select a support surface.
  In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below).
  If no surface is selected, the line is created between the two points based on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

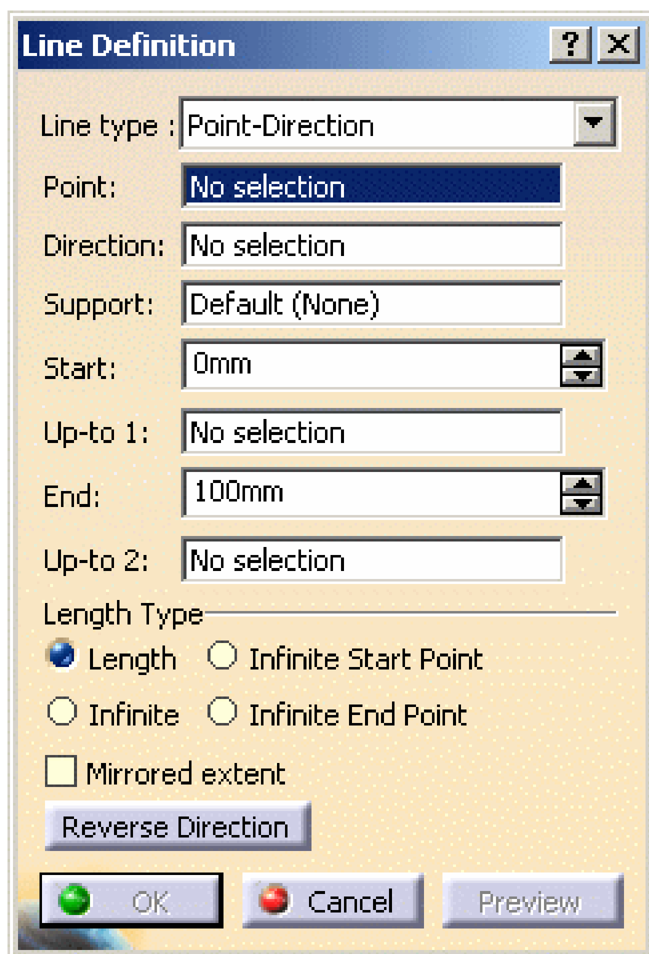


- Specify the **Start** and **End** points of the new line, that is the line endpoint location in relation to the points initially selected. These **Start** and **End** points are necessarily beyond the selected points, meaning the line cannot be shorter than the distance between the initial points.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

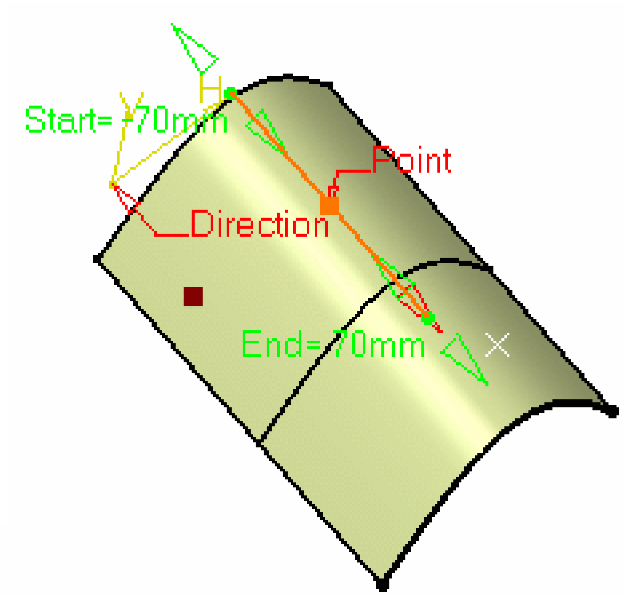
The projections of the 3D point(s) must already exist on the selected support.

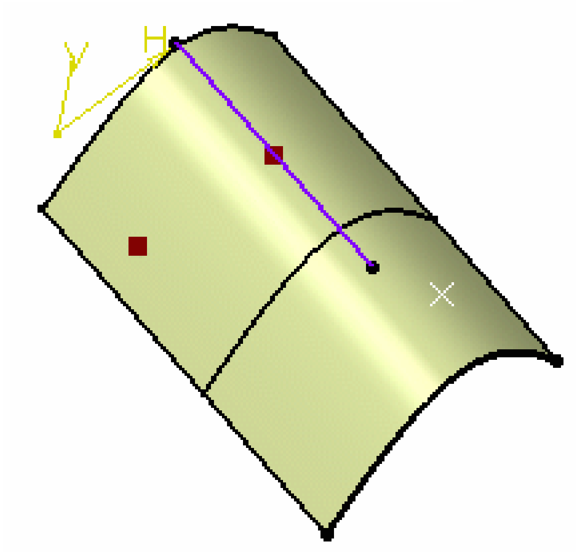
Point - Direction

- 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.





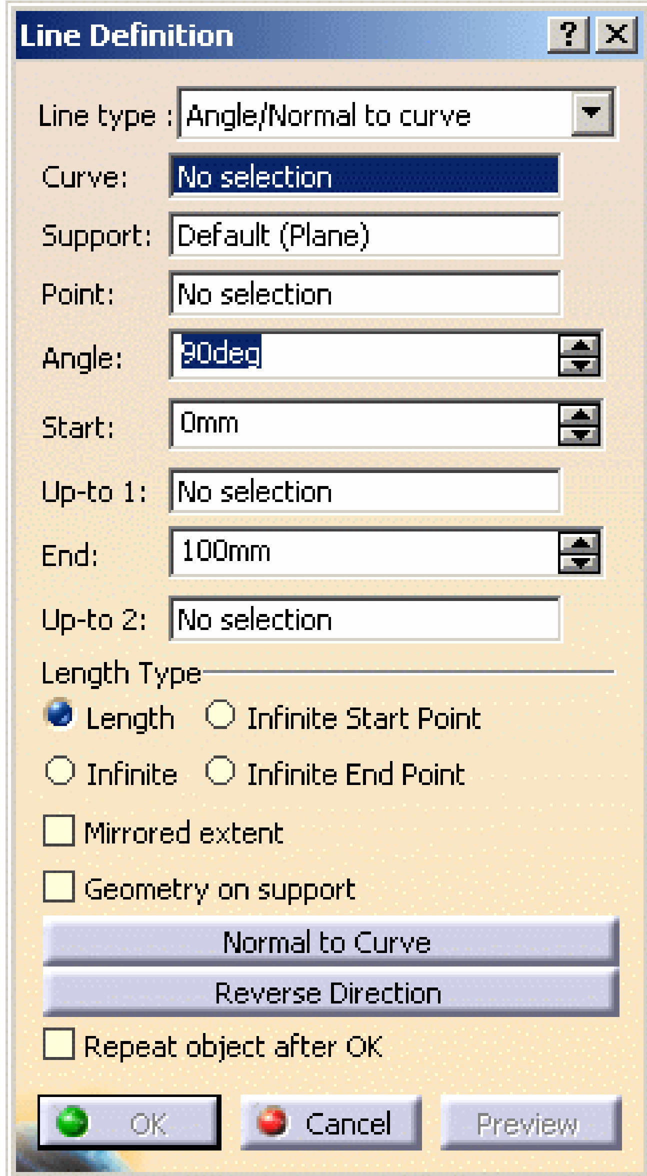
The projections of the 3D point(s) must already exist on the selected support.

Angle or Normal to curve

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

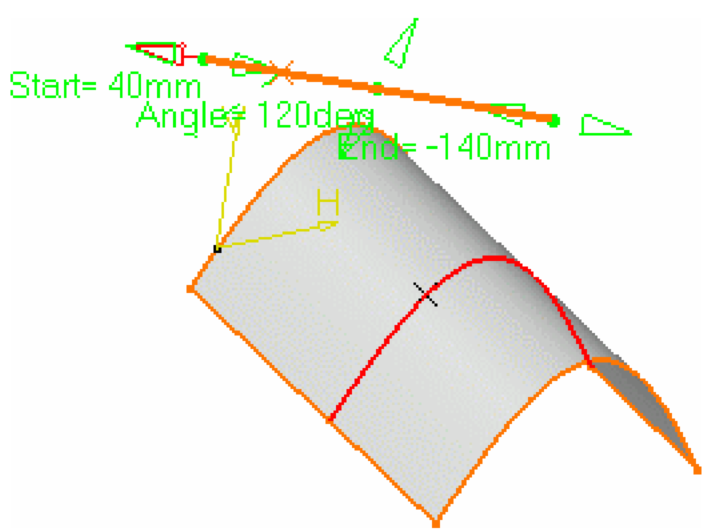
- If the selected curve is planar, then the **Support** is set to Default (Plane).

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



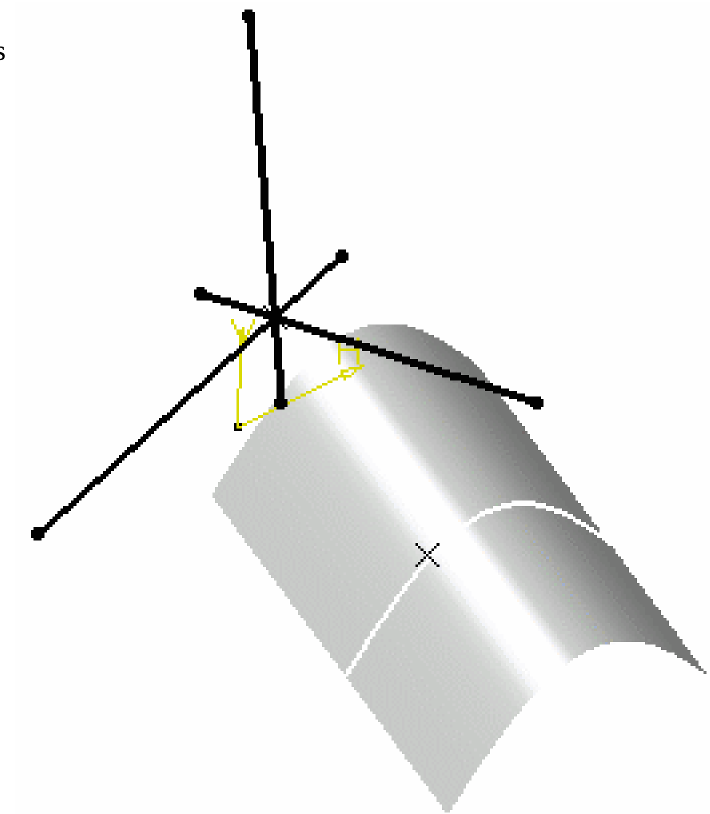
- Select a **Point** on the curve.
- Enter an **Angle** value.

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.

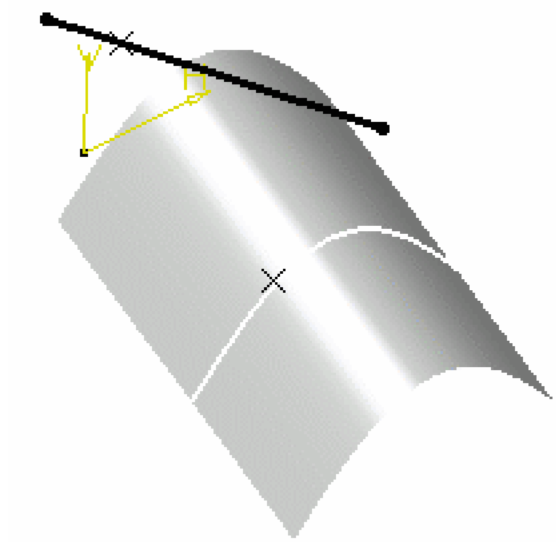
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.

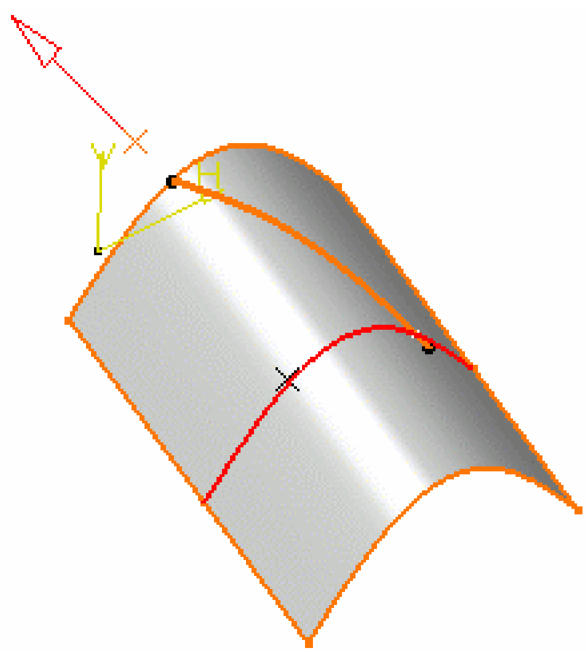


As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.

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.

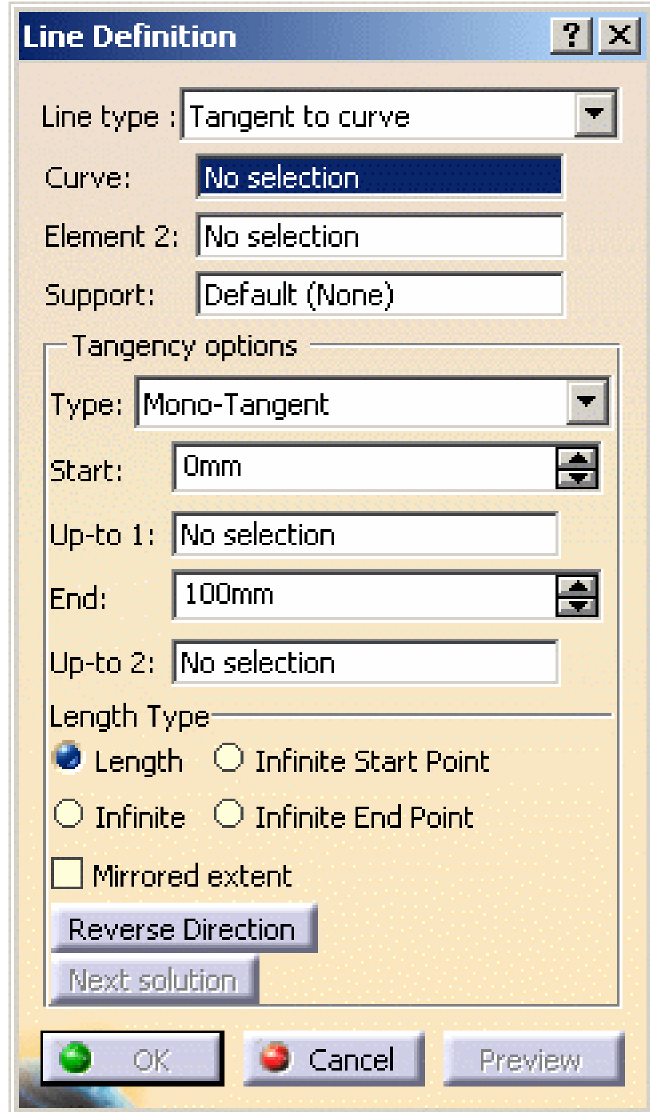




Geometry on support option not checked *Geometry on support option checked*

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

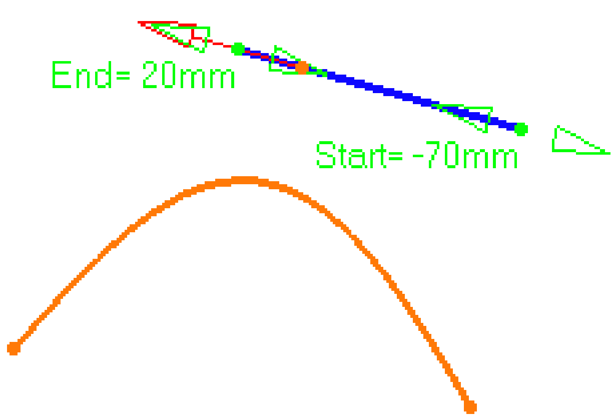


- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
  - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
  - If a second curve is selected (or a point in bi-tangent mode), you need to select a support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

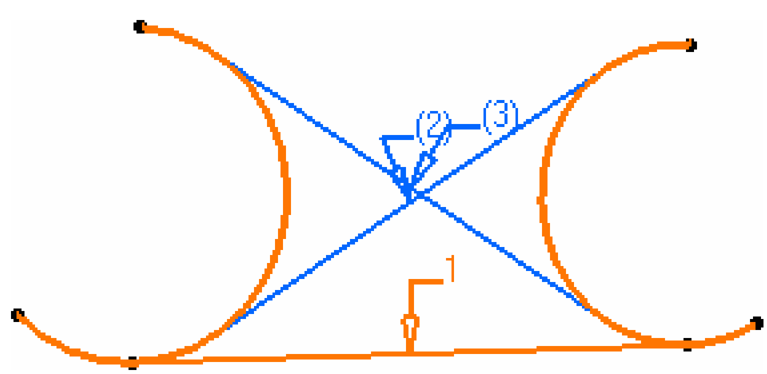
- If the selected curve is a line, then the **Support** is set to Default (Plane).

- 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.

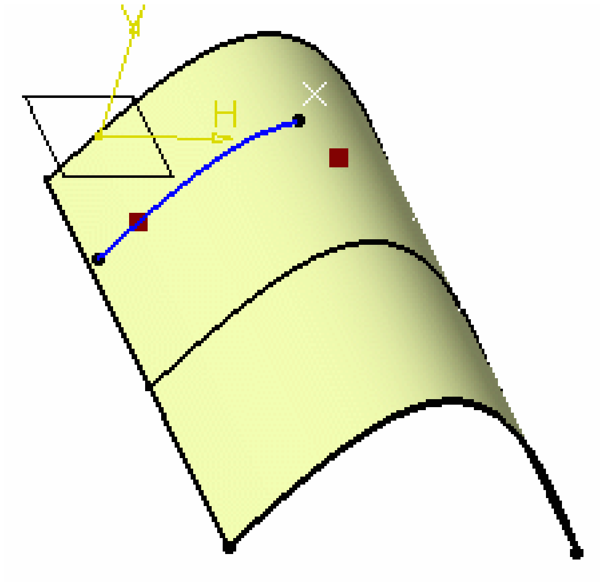


Line tangent to curve at a given point



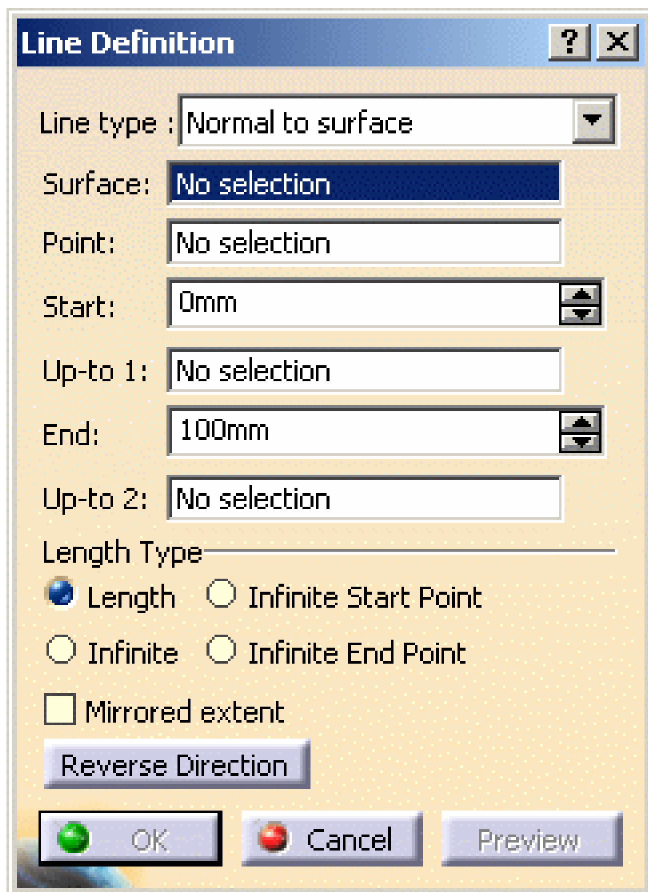
Line tangent to two curves

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



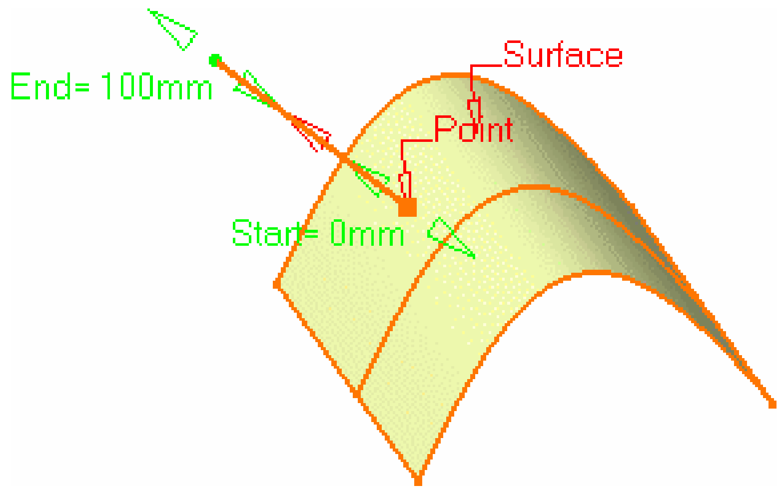
Normal to surface

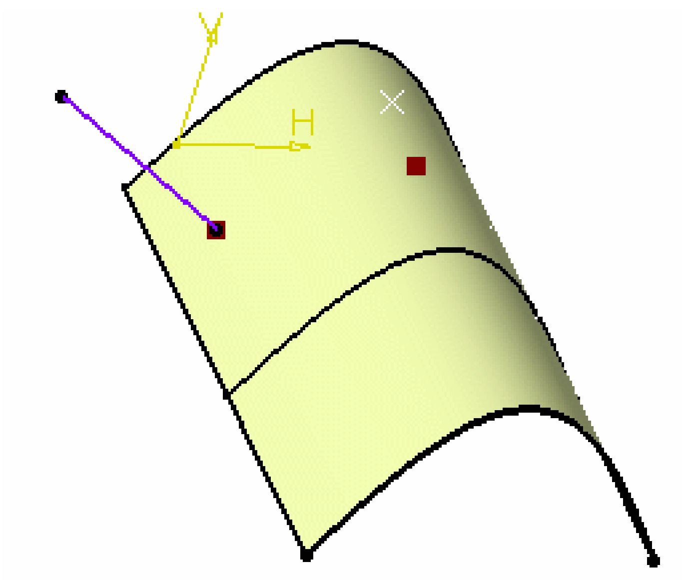
- Select a reference **Surface** and a **Point**.
  A vector normal to the surface is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.



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.

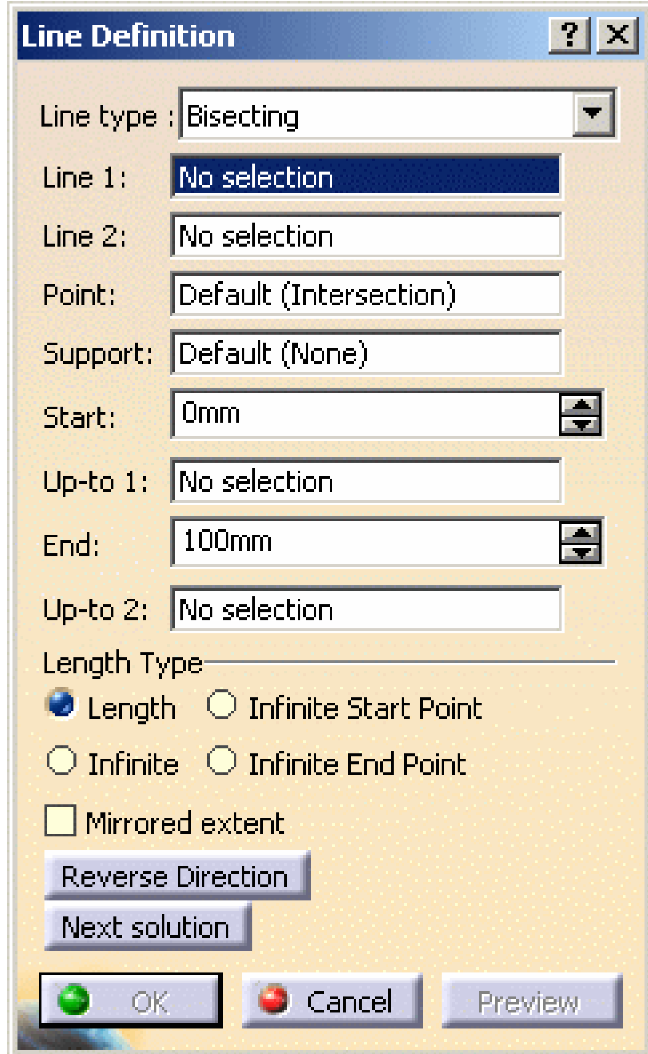
- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.



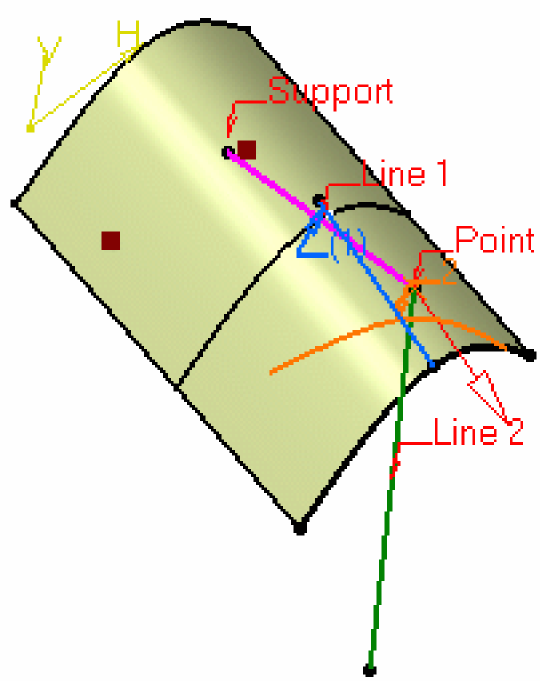


Bisecting

- 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

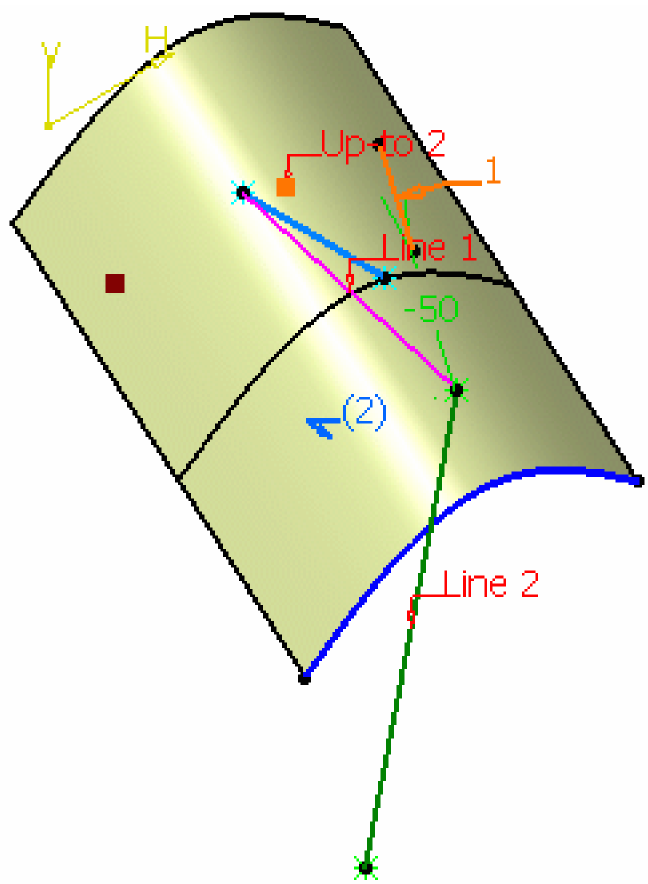
This capability allows you to create a line up to a point, a curve, or a surface.

- 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.

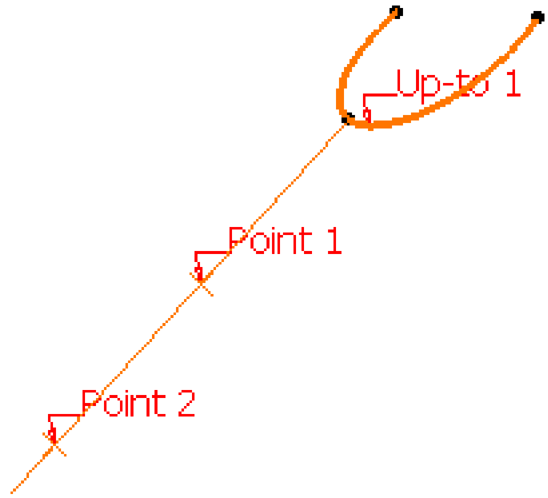
  Here is an example with the Bisecting line type, the **Length** Length type, and a point as **Up-to 2** element.



Up to a curve

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

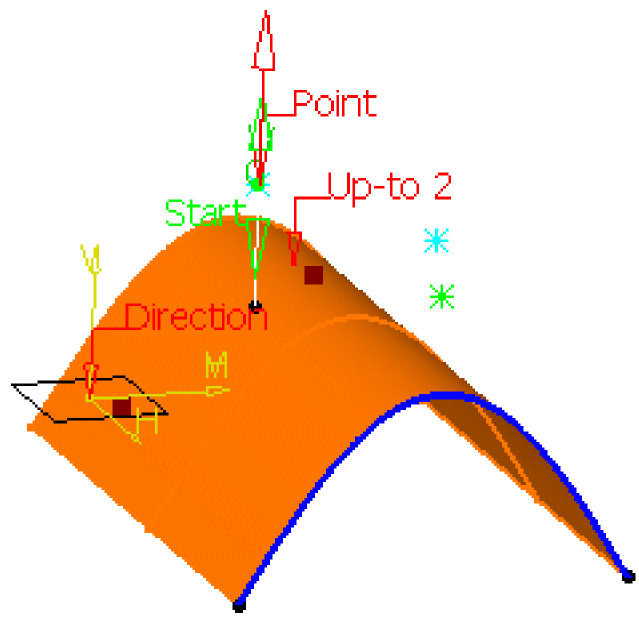
  Here is an example with the Point-Point line type, the **Infinite End** Length type, and a curve as the **Up-to 1** element.



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

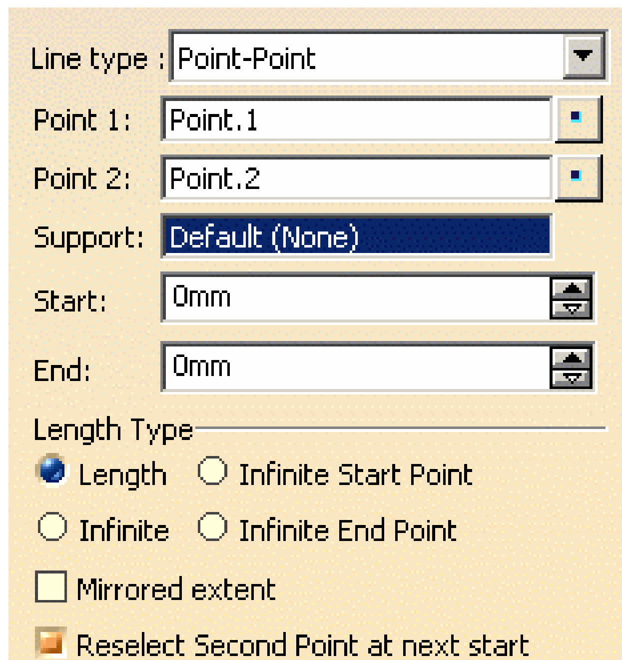
This capability is only available with the Point-Point line method.

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.



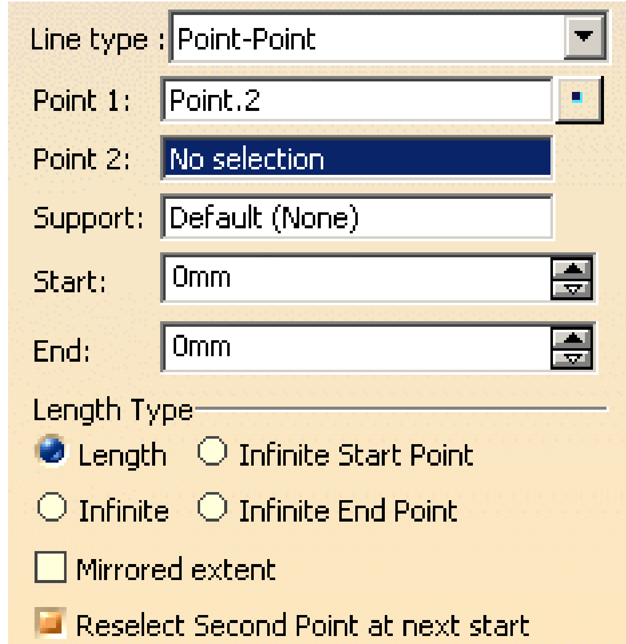
3. Check it to be able to later reuse the second point.

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.

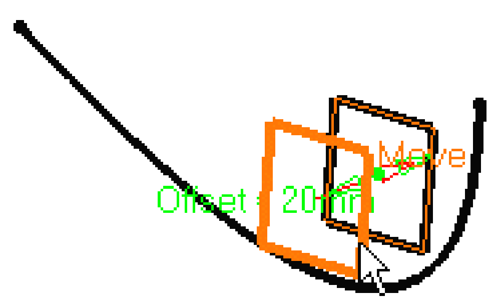
1. Click the **Plane** icon .

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.

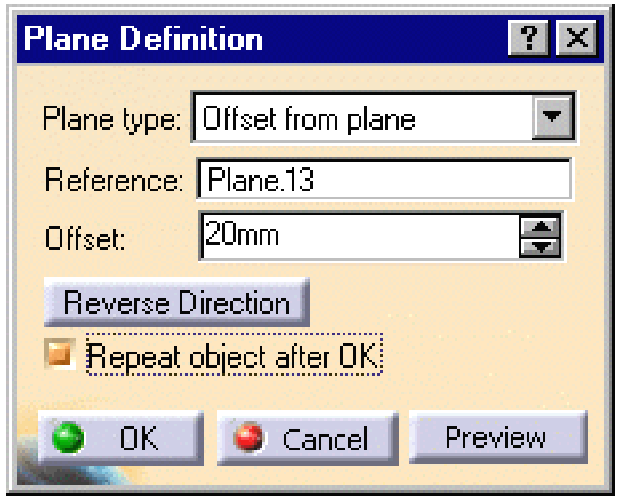
Offset from plane



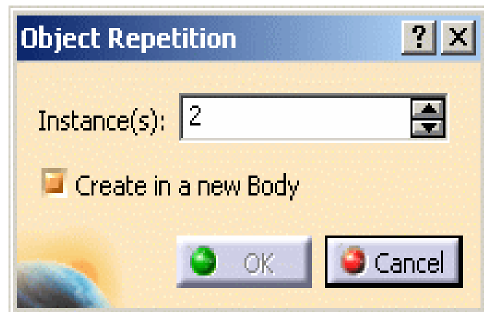
- Select a reference **Plane** then enter an **Offset** value.

A plane is displayed offset from the reference plane.

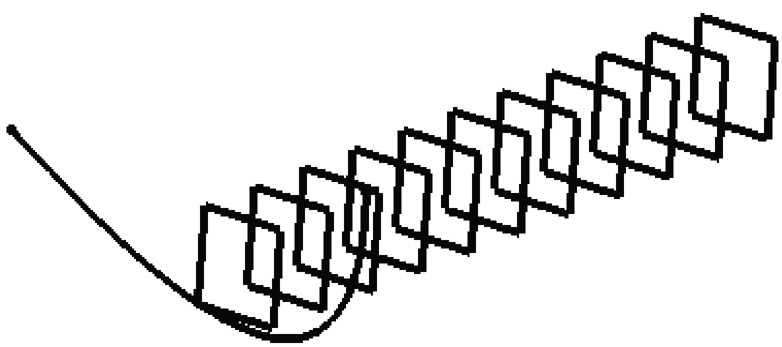
Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.



- 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.

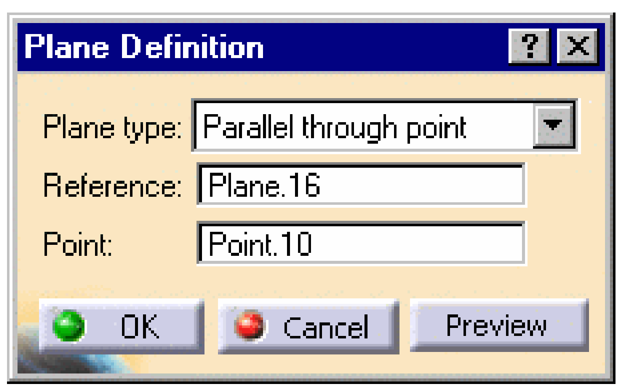


As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Offset** value.

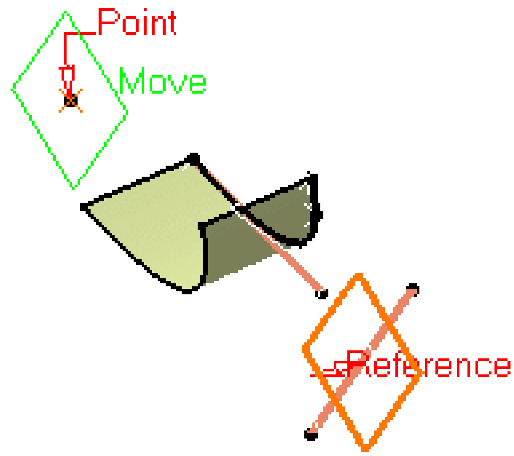


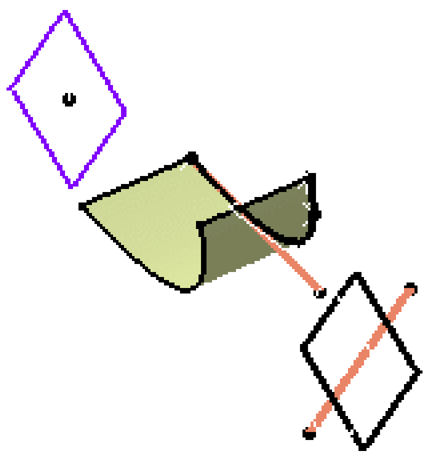
Parallel through point

- Select a reference **Plane** and a **Point**.



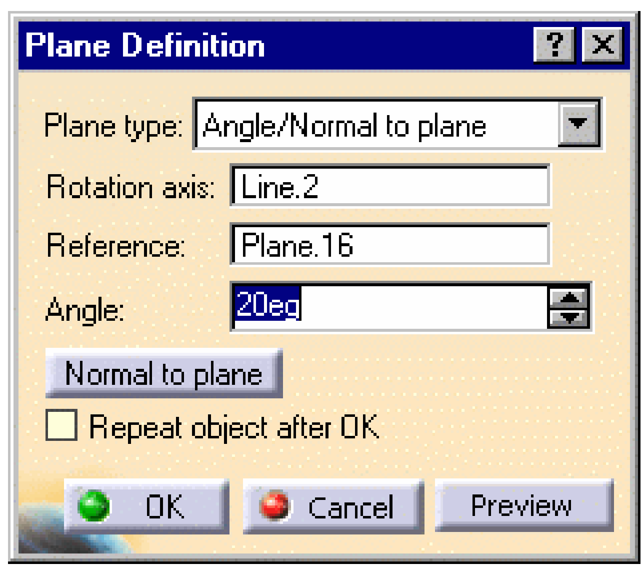
A plane is displayed parallel to the reference plane and passing through the selected point.



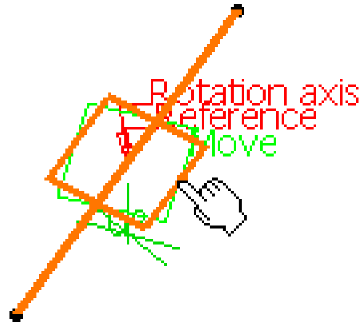


Angle or normal to plane

- 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.



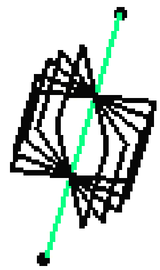
A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



- 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.

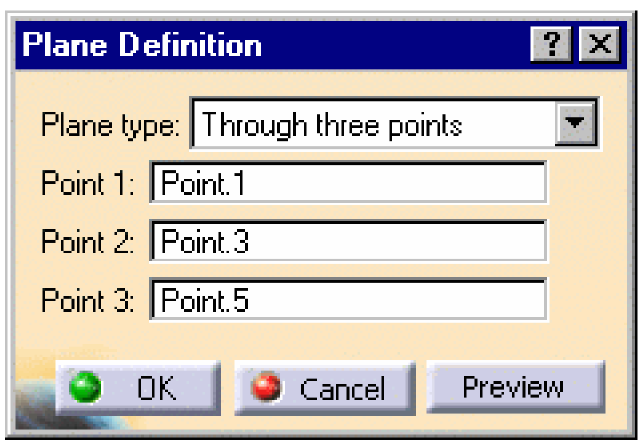
Here we created five planes at an angle of 20 degrees.



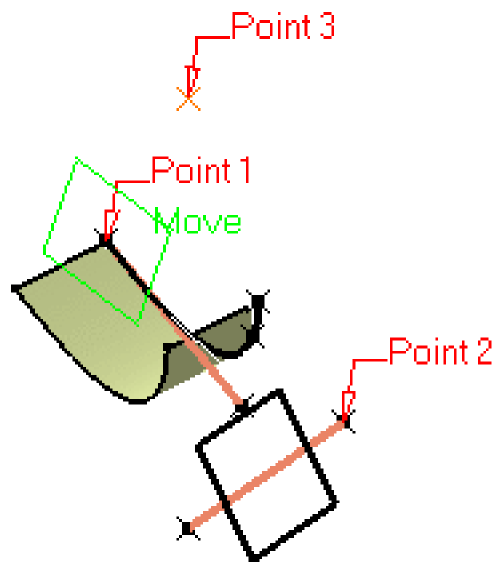
This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

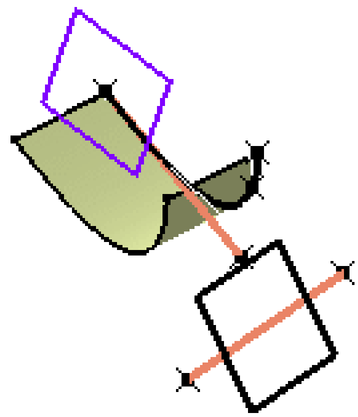
Through three points

- Select three points.



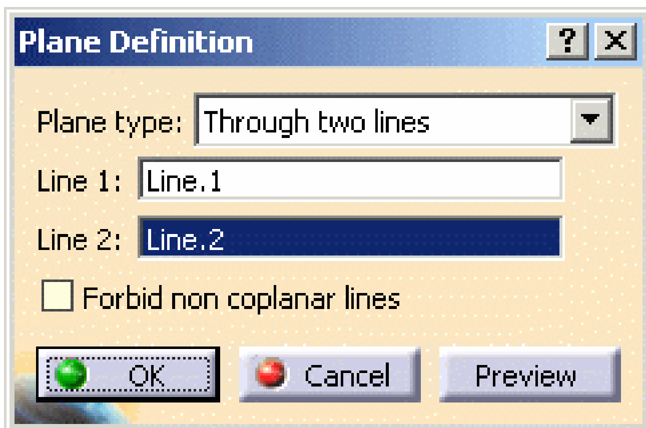
The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.



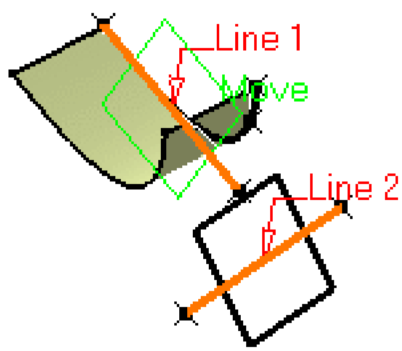


Through two lines

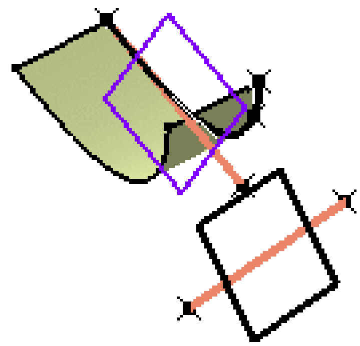
- Select two lines.



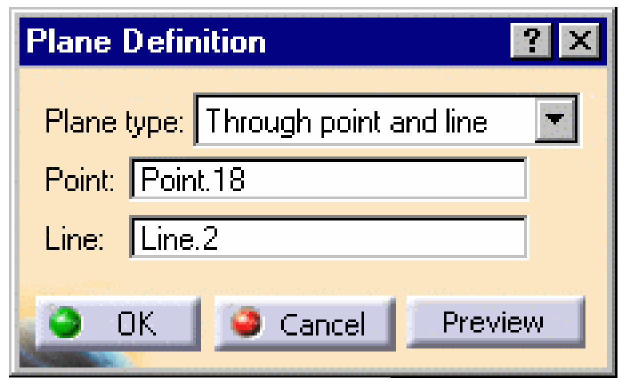
The plane passing through the two line directions is displayed.
When these two lines are not coplanar, the vector of the second line is moved to the first line location to define the plane's second direction.



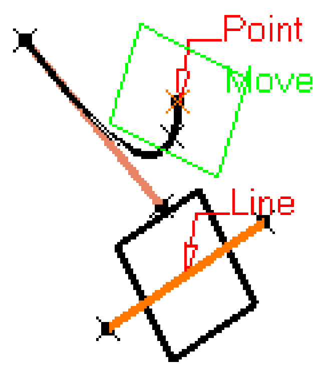
Check the **Forbid non coplanar lines button** to specify that both lines be in the same plane.



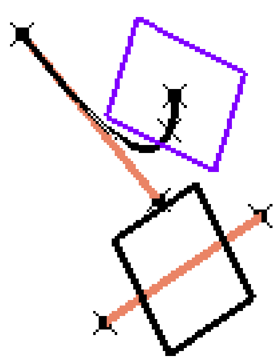
Through point and line



- Select a **Point** and a **Line**.

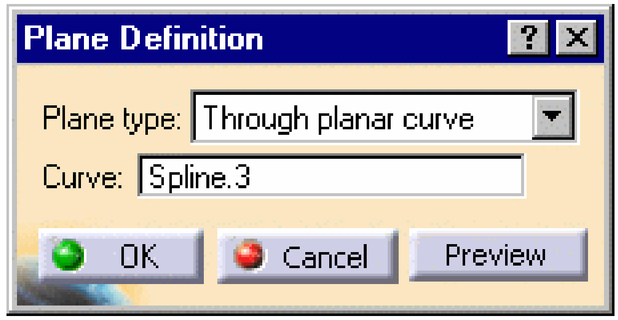


The plane passing through the point and the line is displayed.

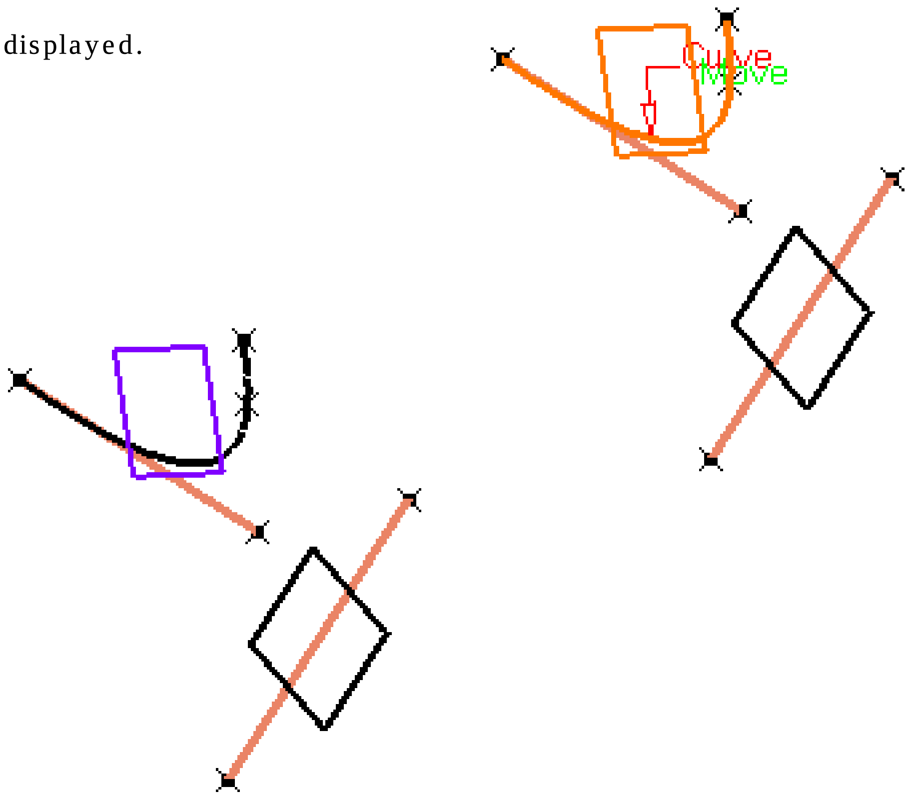


Through planar curve

- Select a planar **Curve**.

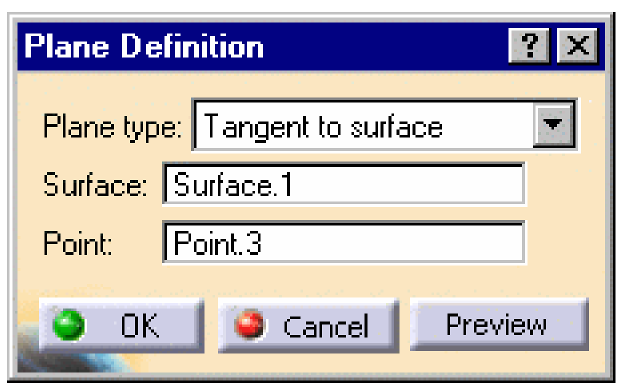


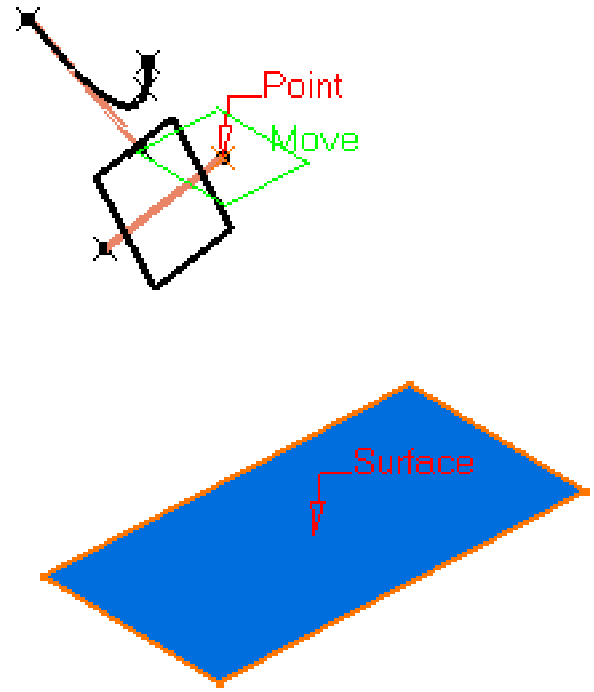
The plane containing the curve is displayed.



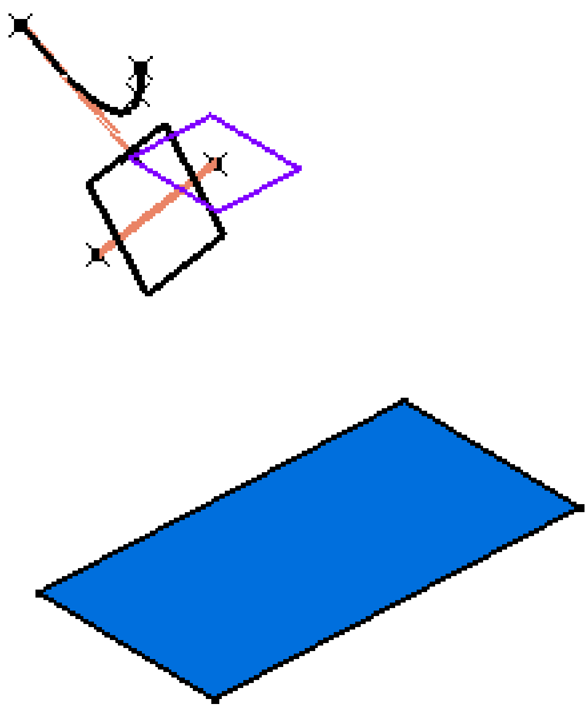
Tangent to surface

- Select a reference **Surface** and a **Point**.



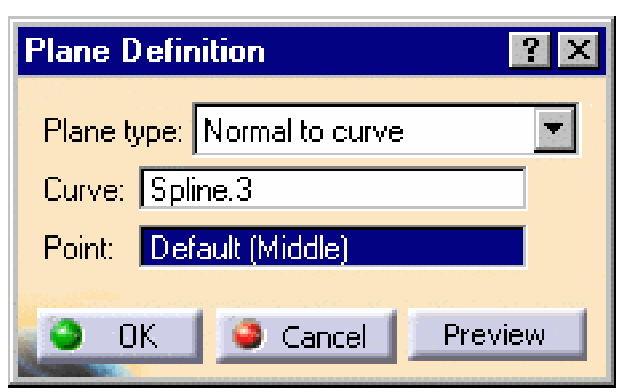


A plane is displayed tangent to the surface at the specified point.

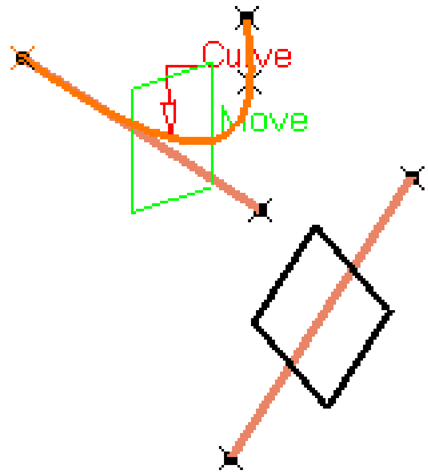


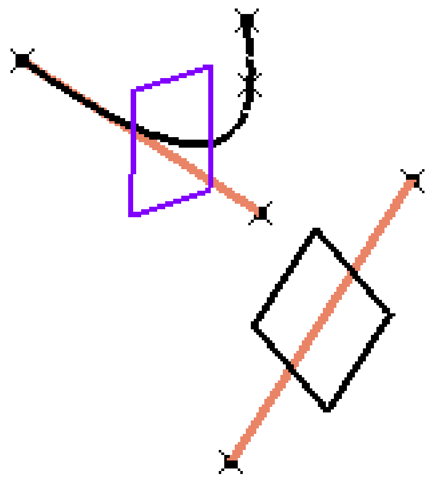
Normal to curve

- Select a reference **Curve.**
- You can select a **Point**. By default, the curve's middle point is selecte.



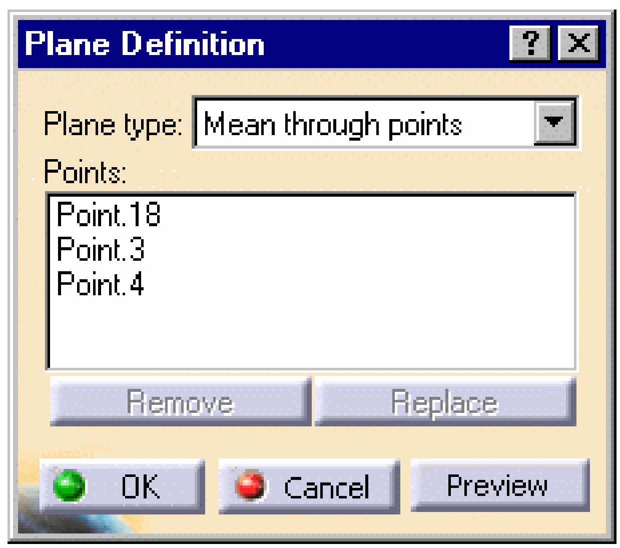
A plane is displayed normal to the curve at the specified point.





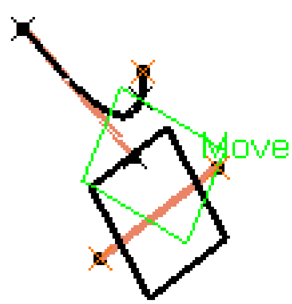
Mean through points

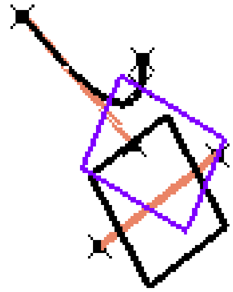
- Select three or more points to display the mean plane through these points.



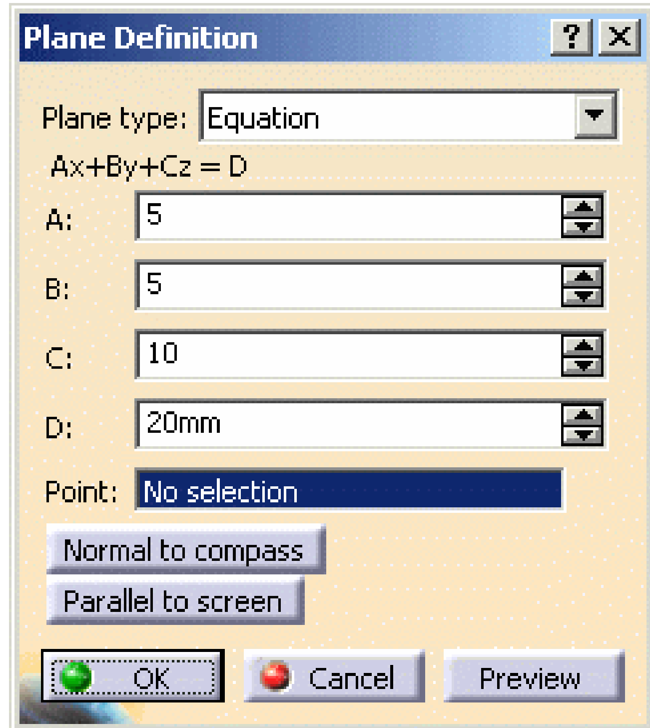
It is possible to edit the plane by first selecting a point in the dialog box list then choosing an option to either:

- **Remove** the selected point
- **Replace** the selected point by another point.



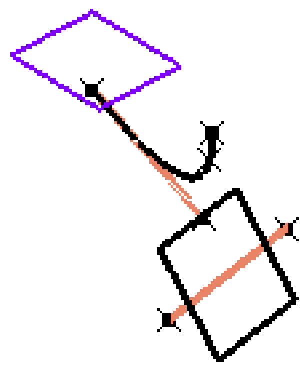


Equation



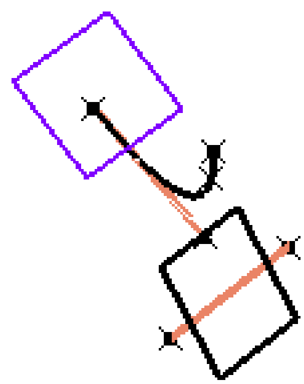
- Enter the **A**, **B**, **C**, **D** components of the **Ax + By + Cz = D** plane equation.

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.

Integration with Other Workbenches

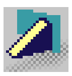
Integration With Part Design

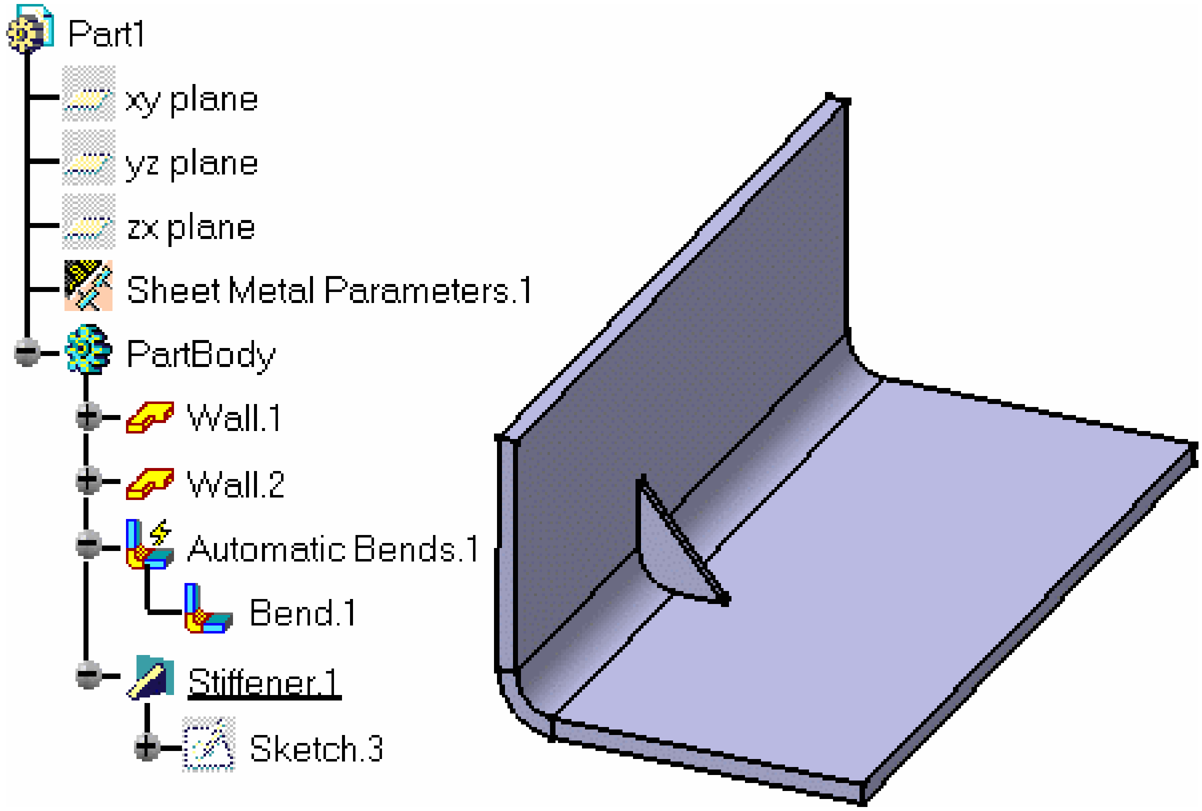
For the Sheet Metal Design workbench, open the Stiffener1.CATPart document.
For the Generative Sheetmetal Design workbench, open the NEWStiffener1.CATPart document.

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:

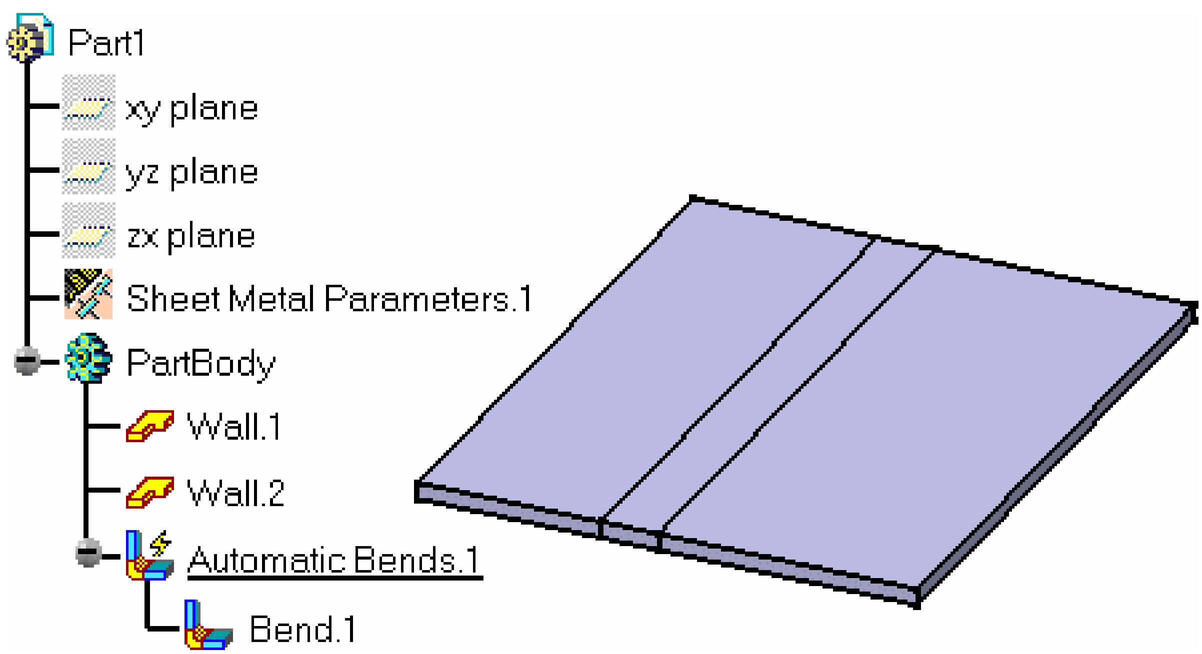
- Part Design features can be created before Sheet Metal features.
- a Part Design feature can also be created after Sheet Metal features as long as the part is in folded view.
- in the unfolded view, the Part Design feature will not be displayed.
- it is no longer possible to create Sheet Metal features 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.

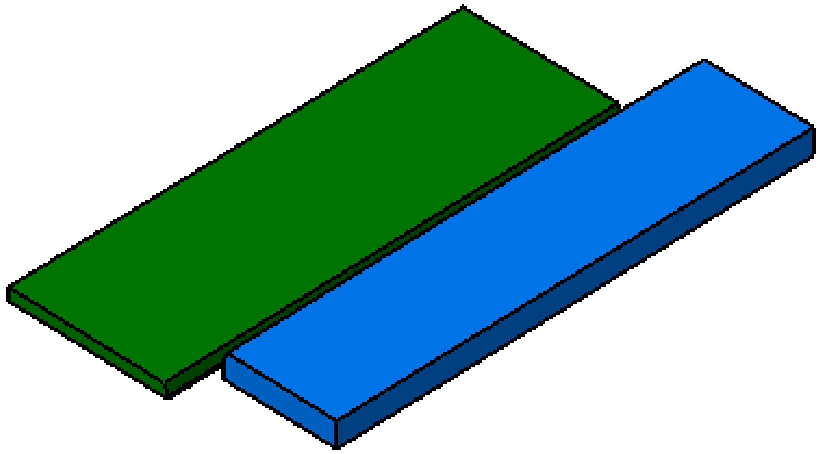
To add a Generative Sheetmetal Design feature and switch to the Sheet Metal Design workbench or to the Generative Sheetmetal Design Design workbench, select the Bend for example and right-click the **Define In Work Object** item. The Generative Sheetmetal Design feature will be added after the Bend but before the Stiffener.

Integration With Weld Design

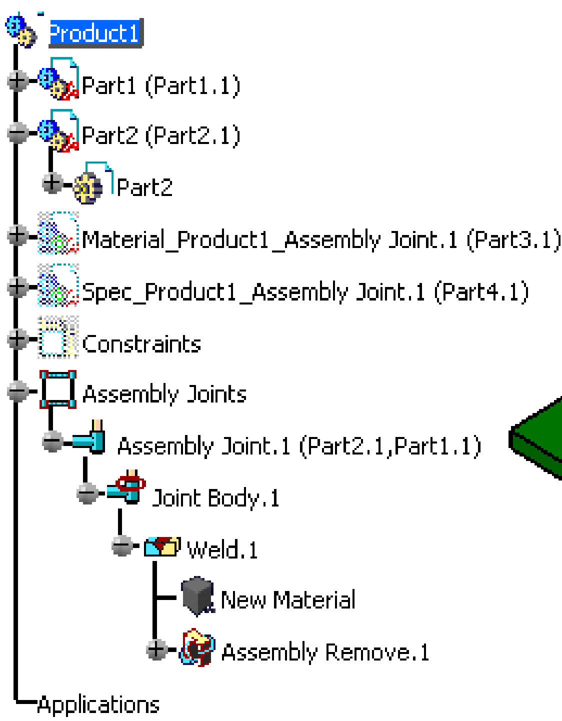
You can open the Integration.CATProduct document from the samples directory to replay the scenario.
If you use the Generative Sheetmetal Design workbench, open the NEWIntegration.CATProduct document.
In a CATProduct document, you may have Weld Design features and Sheet Metal features according to the following rules:

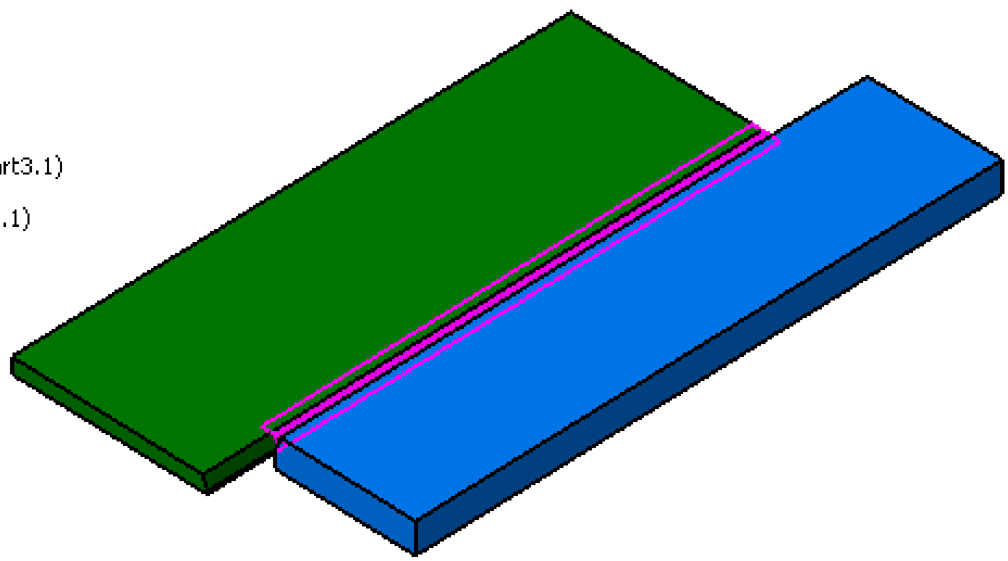
- Weld Design features are created after Sheet Metal features.
- in the unfolded view, the Weld Design feature will not be displayed.
- it is no longer possible to create Sheet Metal features after this last Weld Design feature.

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

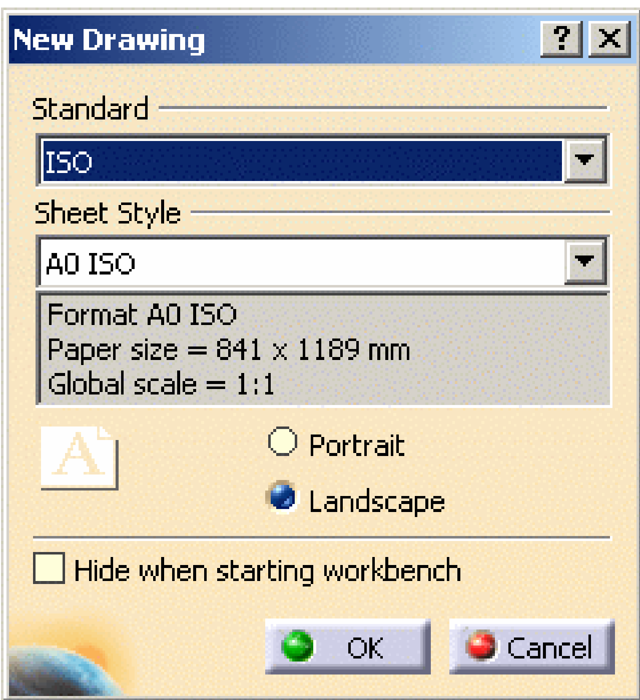
This task explains how to generate an unfolded view of a part with swept wall in a drawing sheet. The axes of planar hems, tear drops, and flanges are displayed within the drawing.

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.**

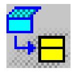


3. Click **OK**.

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

4. The drawing sheet appears.

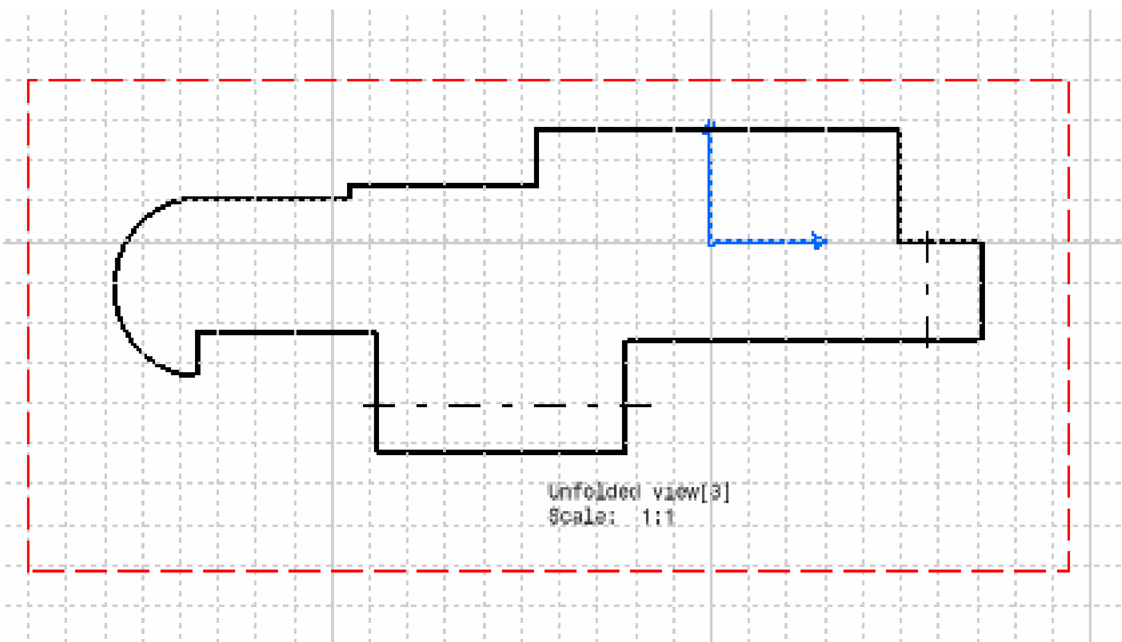
5. Tile the windows horizontally using the **Window -> Tile Horizontally** menu item.

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.



Defining Generative View Styles

This section intends to give you information about Generative Sheetmetal Design-specific generative view style parameters.

Generative view styles make it possible to customize the appearance of drawings via a set of parameters defined in an XML file. Some generative view style parameters are specifically available to produce customized drawings from parts designed in the Generative Sheetmetal Design workbench.

Customizing these parameters is an administrator task. By default, a pre-defined generative view style file, DefaultGenerativeStyle.xml, is delivered. This file is located in install_root/resources/standard/generativeparameters. It should not be deleted.

Administrators can customize this file to define their default generative view styles. They can also use this file as a template for creating new generative view styles.

When finished, generative view style files must be placed by default in install_root/resources/standard/generativeparameters to make them available to the user.
For additional information on how to customize parameters, or more generally on generative view styles, see the *Generative Drafting User's Guide*. The Administration Tasks chapter deals with the administration of generative view styles. The Basic Tasks > Generative View Styles chapter explains how to use generative view styles.

An appropriate Generative Drafting license is required.

SheetMetal parameters

To customize SheetMetal parameters, launch the Standard Definition dialog box, available via **Tools -> Standards**. Choose the **generativeparameters** category, and then open the **DefaultGenerativeStyle.xml** file from the drop-down list. Expand the **Standard -> DefaultGenerativeStyle -> SheetMetal -> SheetMetalDesign** nodes of the tree.

The parameters available under the **Bend** node specify how specific sheet metal elements should be projected in a Generative Drafting view.

Parameter	Value	Description
OML	1 = Visible 0 = Hidden	Specifies whether the OML (Outer Mold Line) should be projected.
BTLs	1 = Visible 0 = Hidden	Specifies whether BTLs (Bend Tangent Lines) should be projected.

Producing Drawings with Generative View Styles

This tasks shows you how to produce drawings using generative view styles.

Generative view styles make it possible to customize the appearance of drawings via a set of parameters defined in an XML file, DefaultGenerativeStyle.xml . Some generative view style parameters are specifically available to produce customized drawings from parts designed in the Generative Sheetmetal Design workbench. It is the administrator's job to provide suitable styles. Refer to Defining Generative View Styles for more information.

For more information on generative view styles and other drafting capabilities, see the *Generative Drafting User's Guide*.

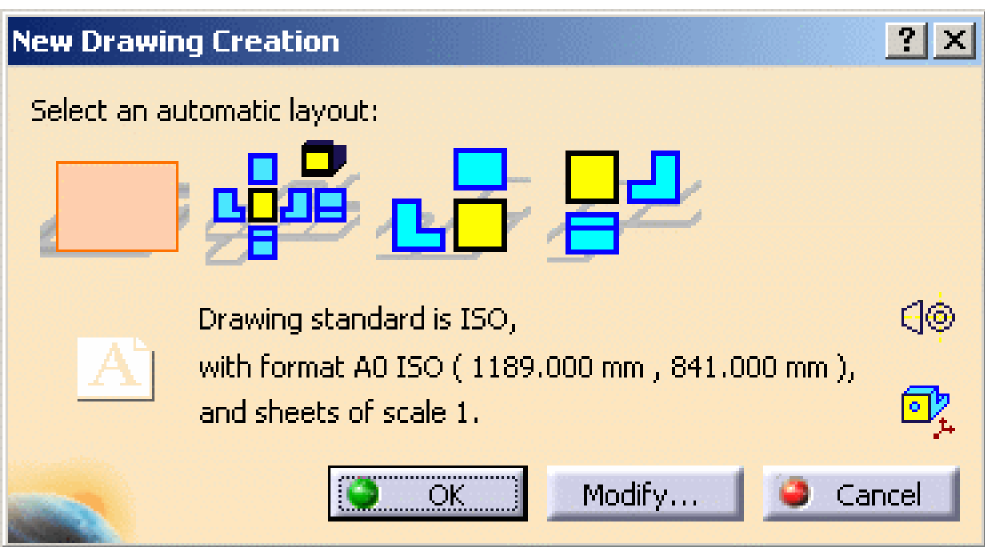
Open the NEWSweptWall1.CATPart document.
Make sure you have an appropriate Generative Drafting license.

1. Specify your settings before you start: select **Tools -> Options -> Mechanical Design -> Drafting -> Administration** tab, and uncheck **Prevent generative view style creation**.

   This activates generative view style functionalities.

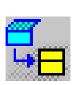
2. Select **Start -> Mechanical Design -> Drafting.**

   The New Drawing Creation dialog box appears. The empty sheet layout is pre-selected.



3. Simply click **OK**. You switch to the Drafting workbench and an empty drafting sheet is created.

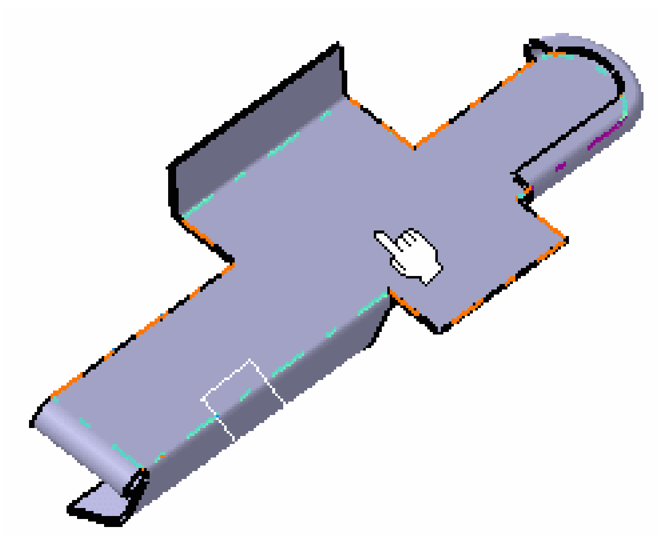
4. For ease of use, tile the windows horizontally using the **Window -> Tile Horizontally** menu item.

5. Select the **Unfolded View** icon  in the Projections toolbar from the Generative Drafting

workbench. The Generative View Style toolbar is automatically displayed.

6. Select one of the available styles from the list, DefaultGenerativeStyle in this case.

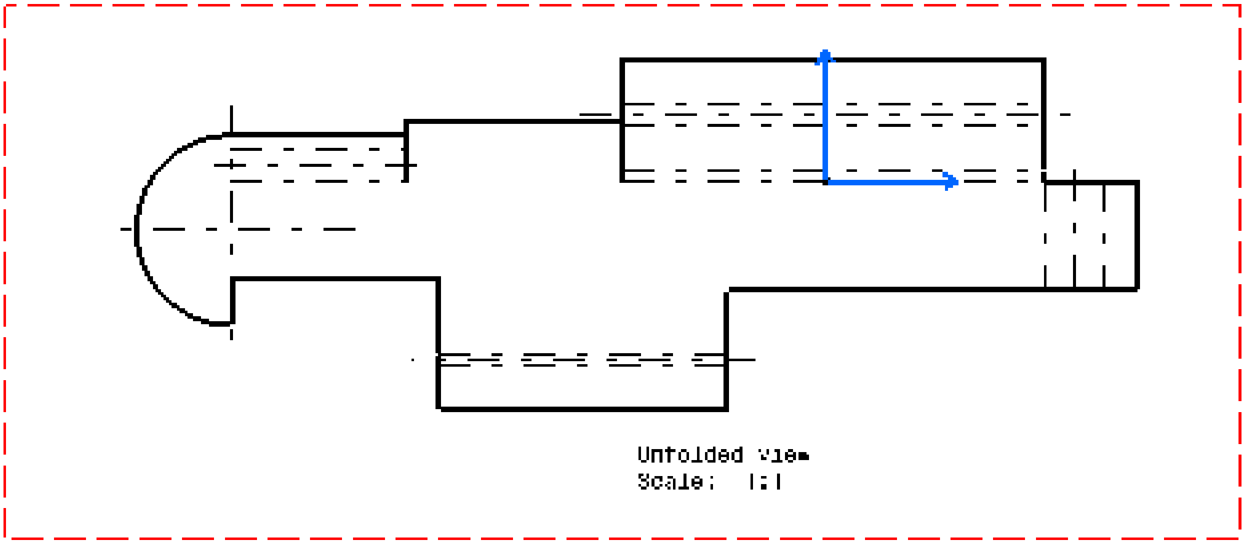
7. Return to the 3D document and select a plane surface on the sheet metal part. The sheet metal reference wall plane will be used automatically.



If you prefer to specify which plane should be used, you can also select the 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 created, using the sheet metal-specific styles defined by the chosen generative view style. Here, the OML (Outer Mold Line) and the BTLs (Bend Tangent Lines) are projected in the view, as specified in the DefaultGenerativeStyle.xml file.


Unfolded View
Scale: 1:1

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

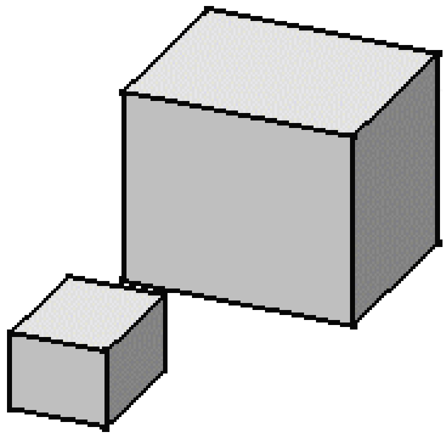
This task explains how to create a Sheet Metal part in an Assembly context.

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.

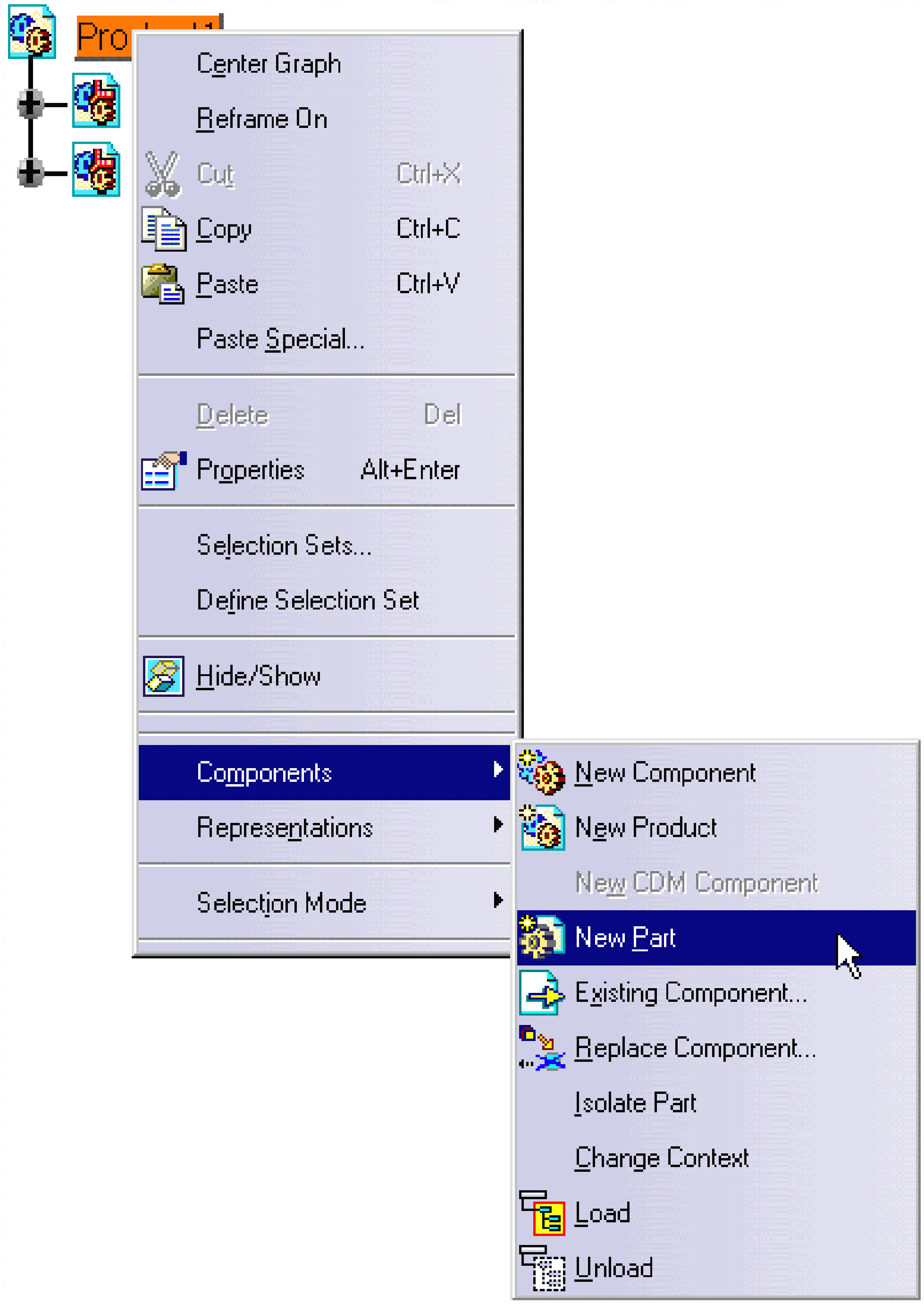
The document contains two parts.



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.

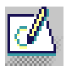
5. Switch to the Sheet Metal Design workbench or to the Generative Sheetmetal Design workbench.

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.

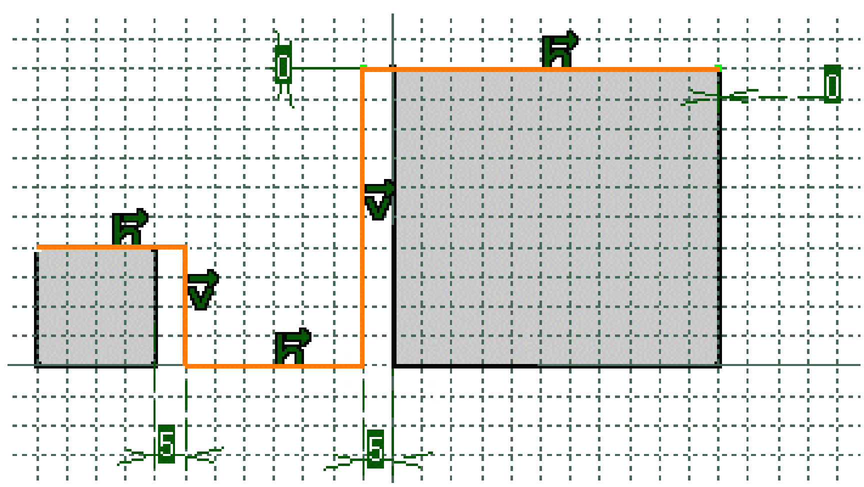
7. Choose the **Tools -> Options -> Infrastructure -> Part Infrastructure, General** tab and check the **Keep link with selected object** option, then click OK.

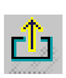
8. Click the **Sketcher** icon  and select the **zx plane**.

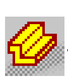
9. Select the **Profile** icon .

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.

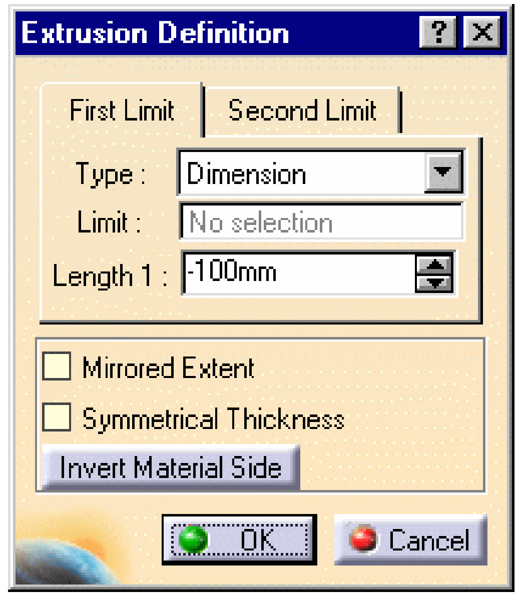
12. Select the **Extrusion** icon .

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.

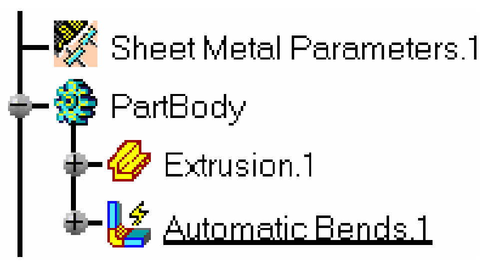


15. Perform this step only if you are using the Sheet Metal Design workbench: select the **Automatic Bends** icon .

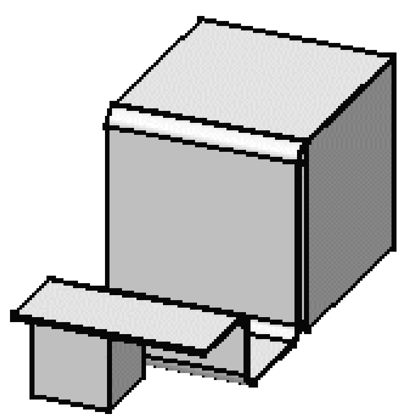
The bends are created.

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.

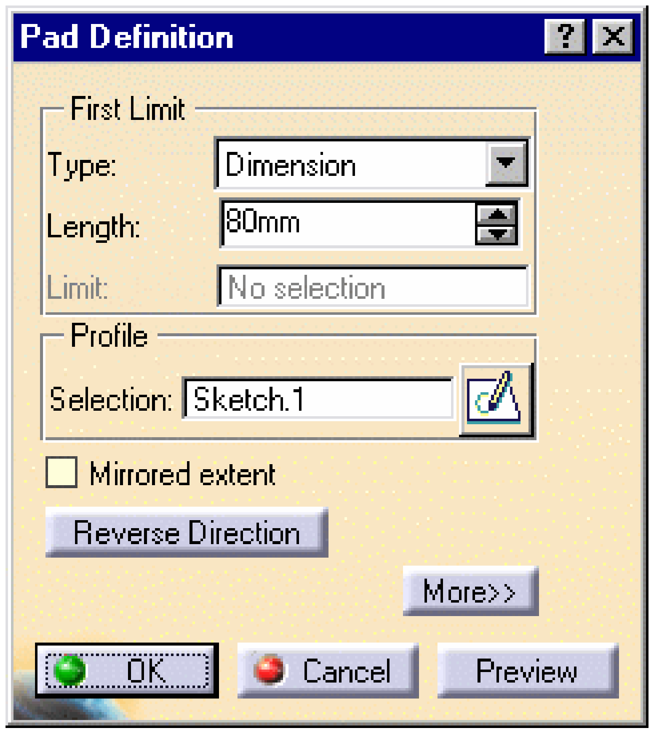
For the Sheet Metal Design workbench, the Scenario2.CATProduct document is open from the previous task. If not, open the Scenario2_2.CATProduct document from the samples directory.

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.



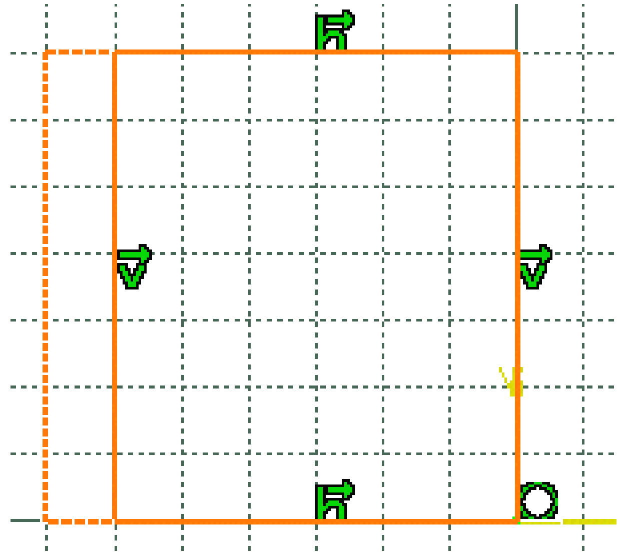
2. Enter 40mm for the **Length** and click **OK**.

   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**.

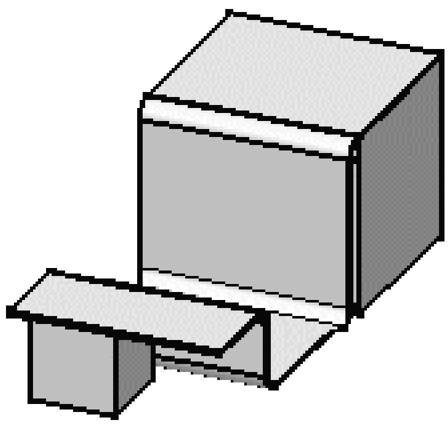
5. Modify the sketch:



6. Click the **Exit** icon  to return to the 3D world.

The constraints are respected.

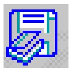
After the Part3 update, the document looks like this:



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.

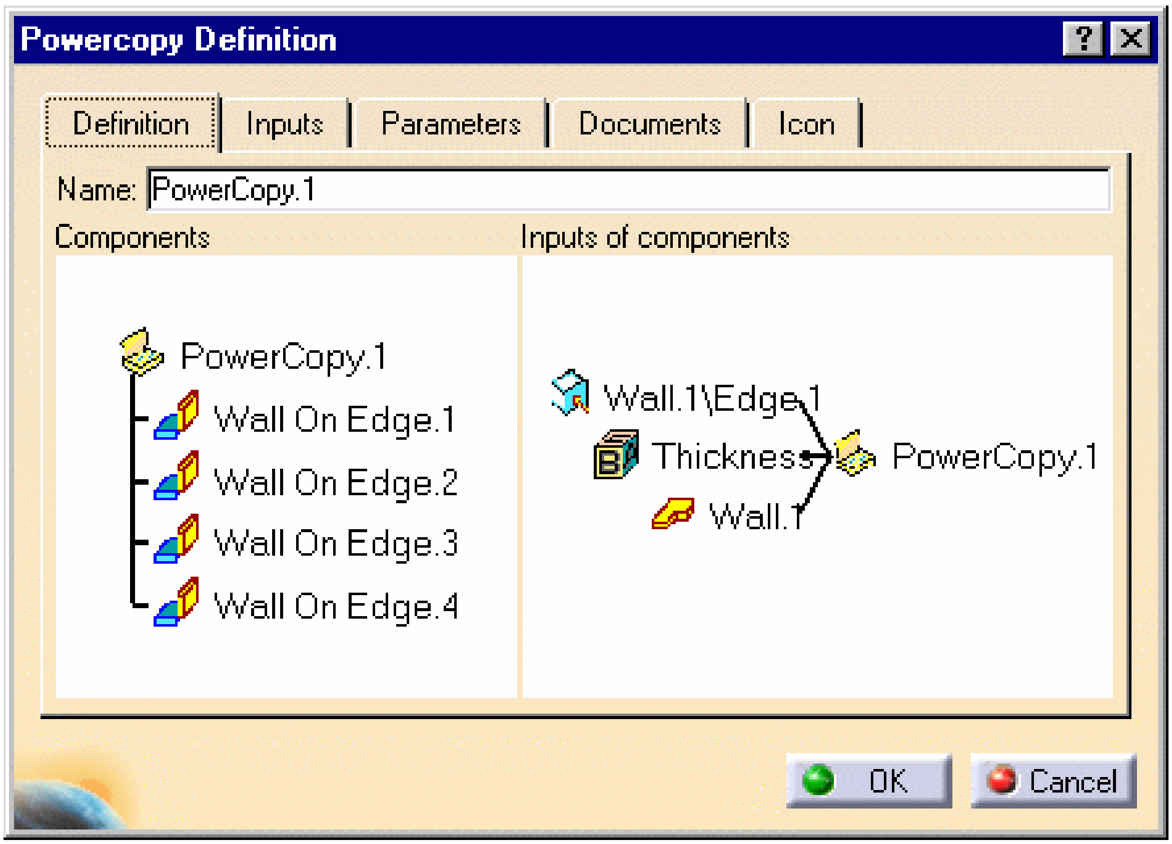
Open the PowerCopyStart.CATPart document.

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.

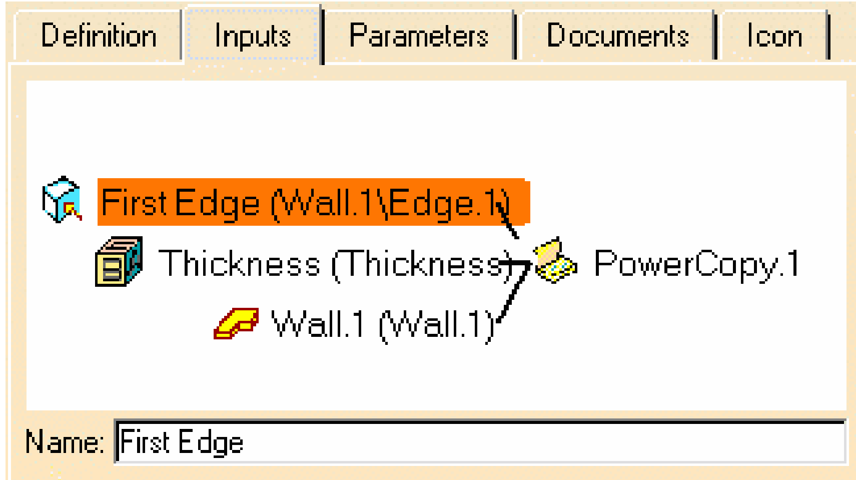
The **PowerCopy Definition** dialog box is automatically filled with information about the selected elements.



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.

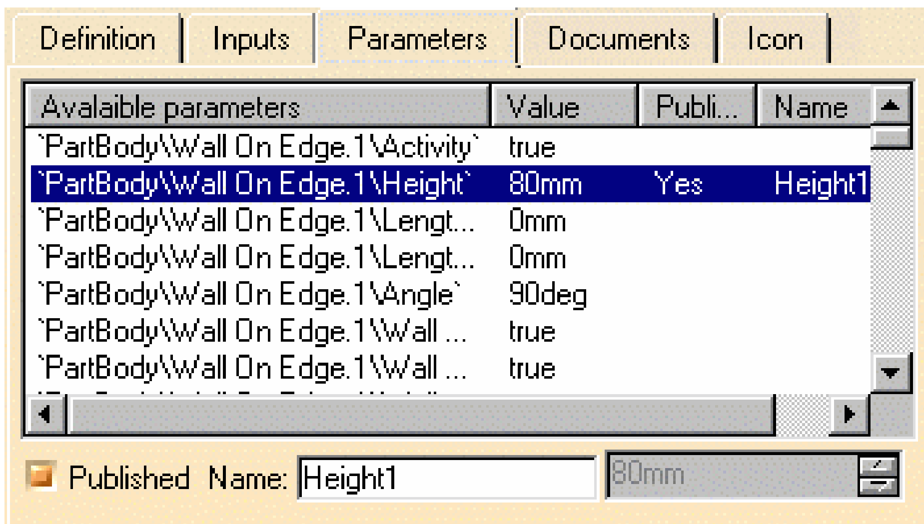
The **Inputs** tab lets you rename the reference elements making up the PowerCopy.



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.

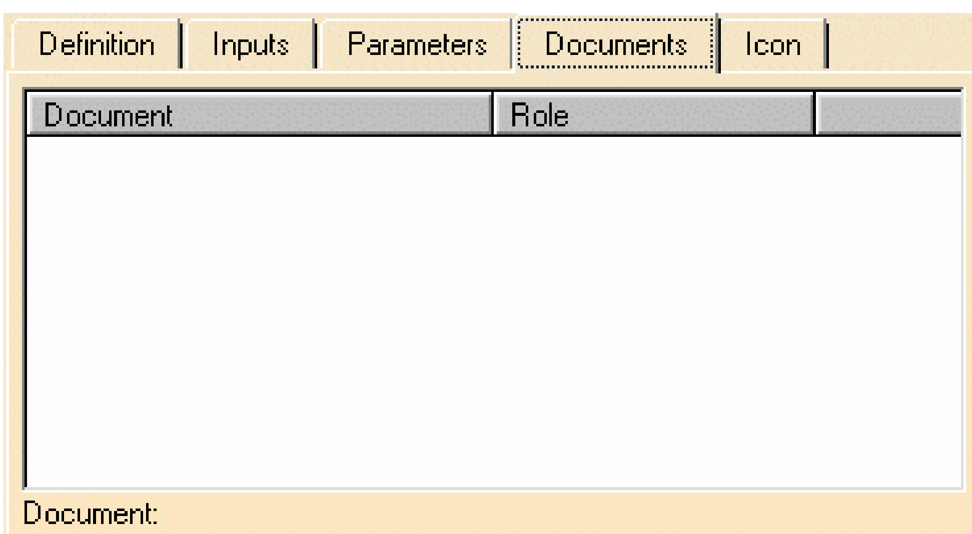
The **Parameters** tab lets you define which of the parameter values used in the PowerCopy you will be able to modify at instantiation time.



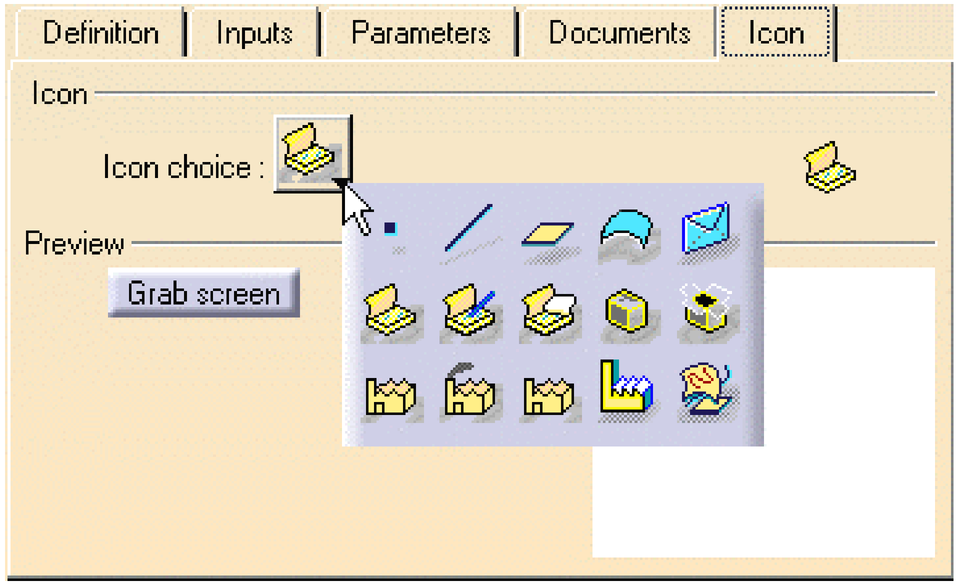
Simply check the Published button.

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.

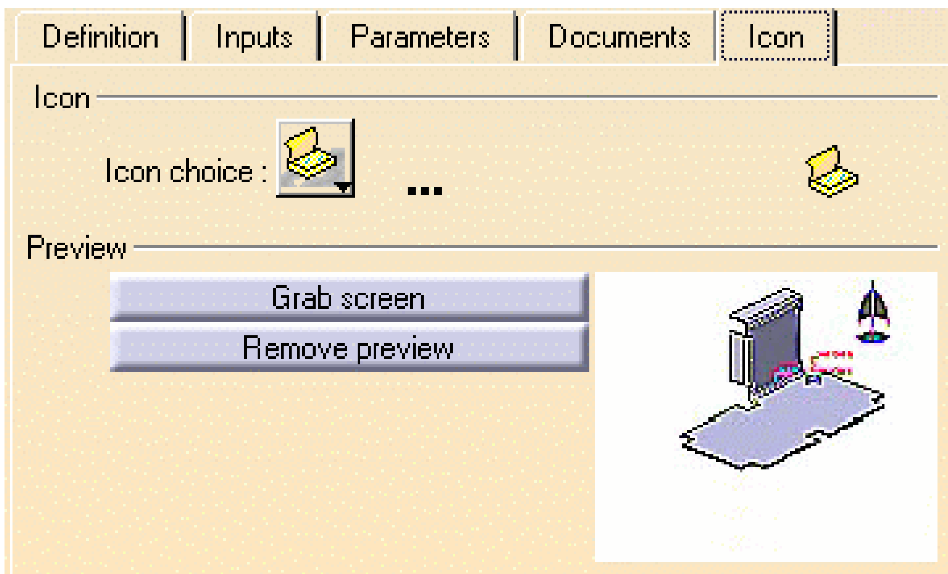


The **Icon** tab lets you modify the icon identifying the PowerCopy in the specifications tree.



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:

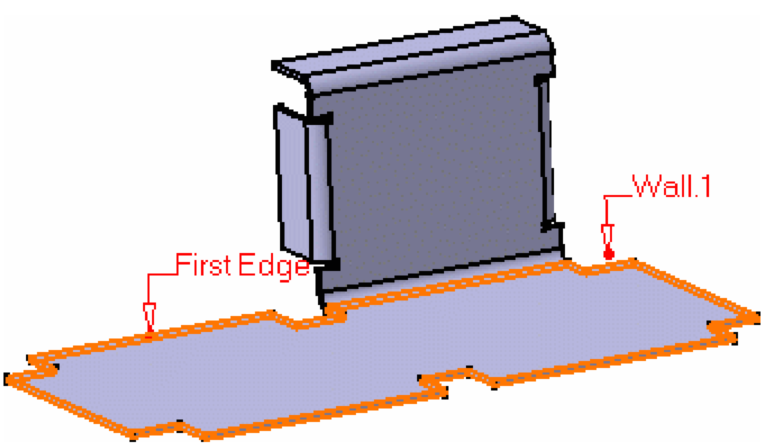
1. Select the PowerCopy feature from the specification tree.

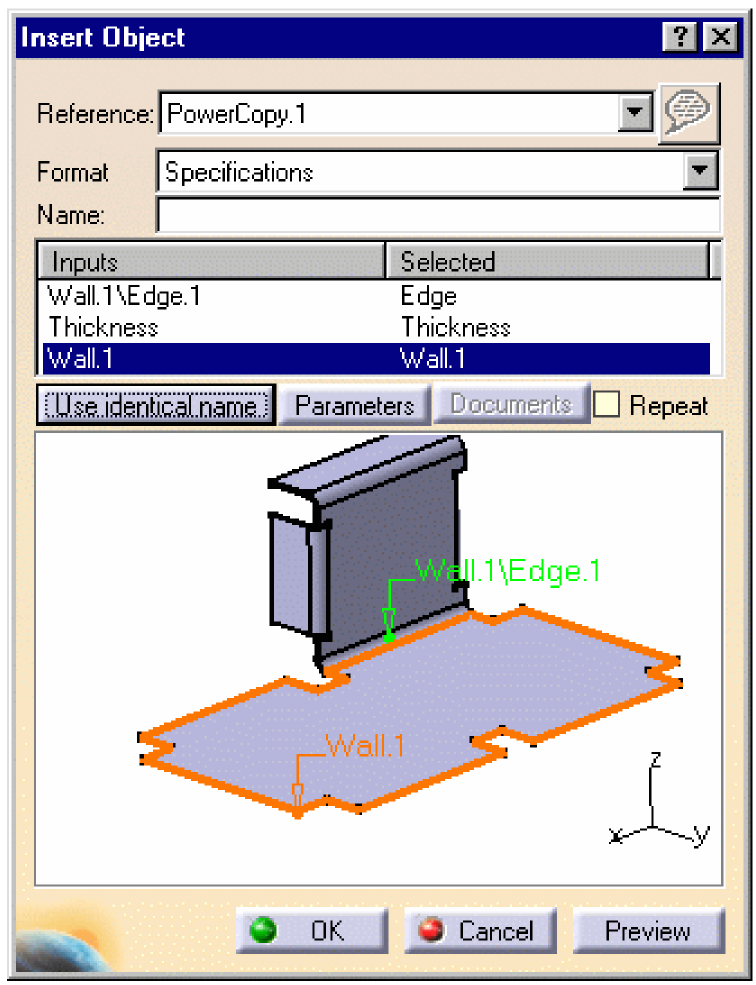
2. Right-click to display the contextual menu, and choose the **PowerCopy.1 object -> instantiate** menu item.

The **Insert Object** dialog box is displayed.

3. Complete the **Inputs** within the dialog box by selecting the adequate element in the geometric area.

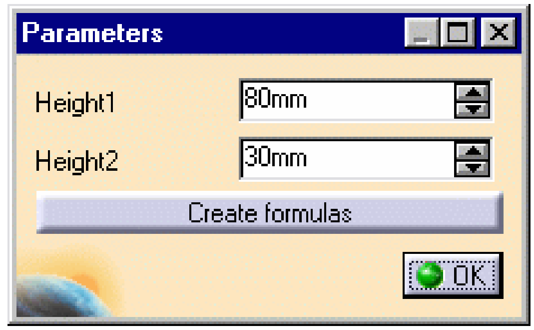
You need to click the arrow to invert the First Edge orientation.





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.



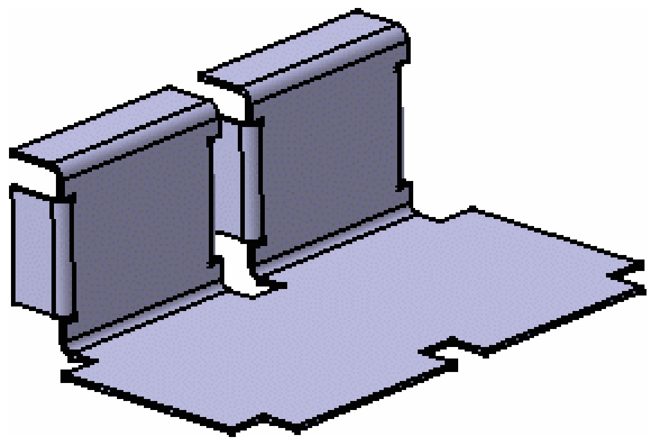
6. Use the Create formulas button to automatically create a formula on every parameters with the same name provided there are any.

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.

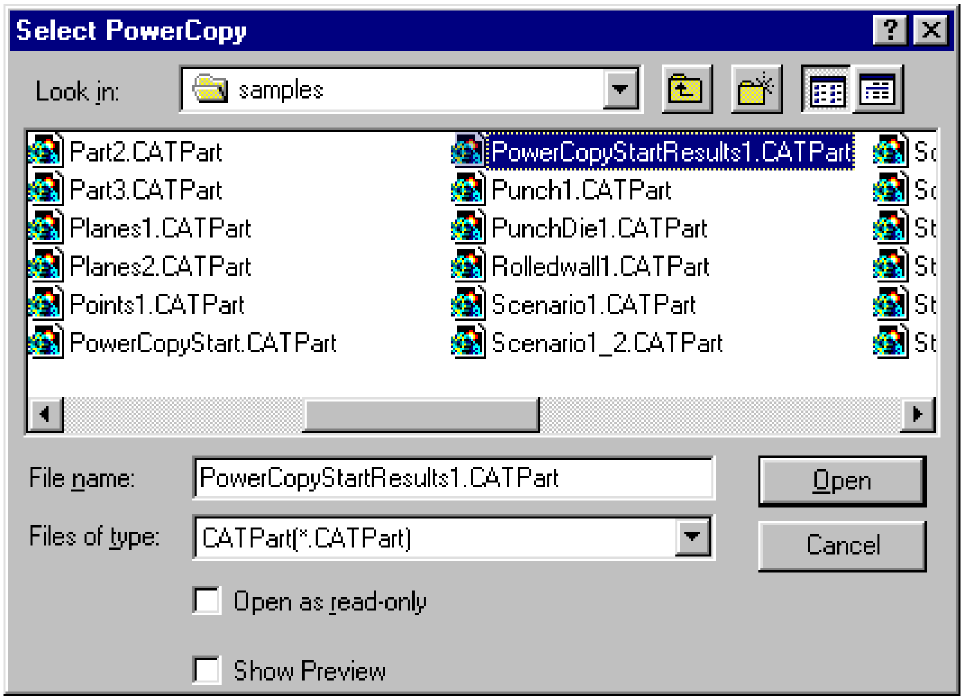
The PowerCopy is instantiated in context, meaning its limits are automatically re-defined taking into account the elements on which it is instantiated.



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.

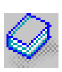
Use the **Reference** list to choose the correct PowerCopy when several have been defined in the document.

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.

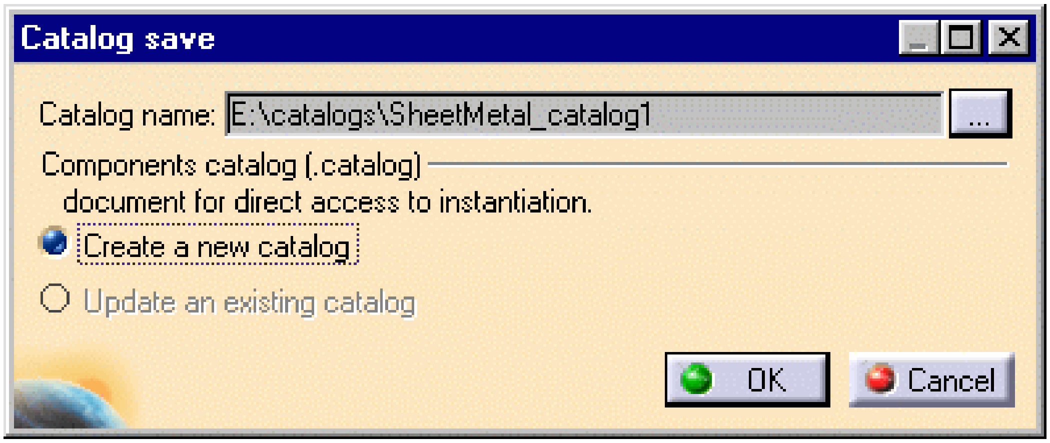
Saving PowerCopy Features

This task shows how to use store Power Copy elements into a catalog, for later use as described in Instantiating PowerCopy Features.

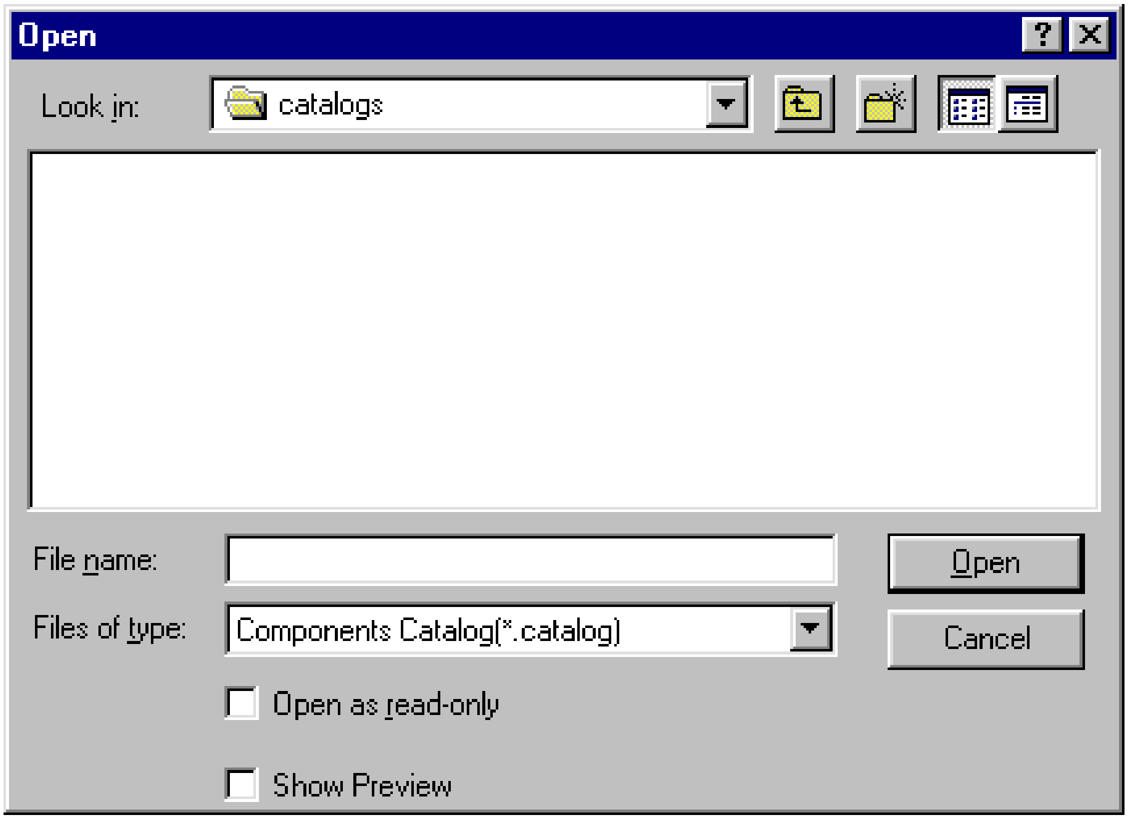
Open the PowerCopyStartResults1.CATPart document.

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.

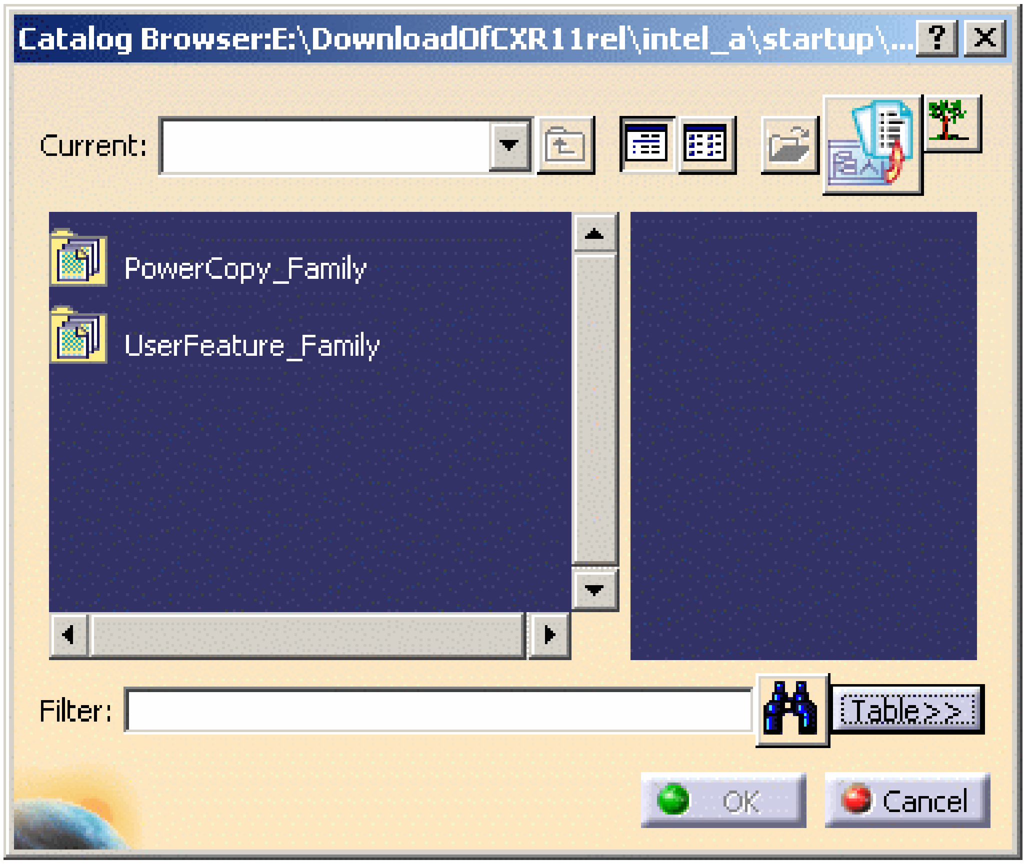
3. Click **OK**.

The PowerCopy has been stored in the catalog.

Browsing the Sheet Metal Catalog

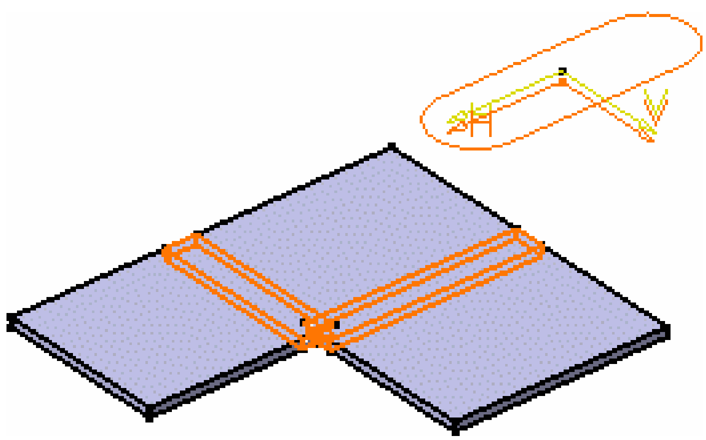
Open the CornerRelief03.CATPart document from the samples directory.

In this case, the Catalog Browser opens, and lets you choose from a number of sketches.

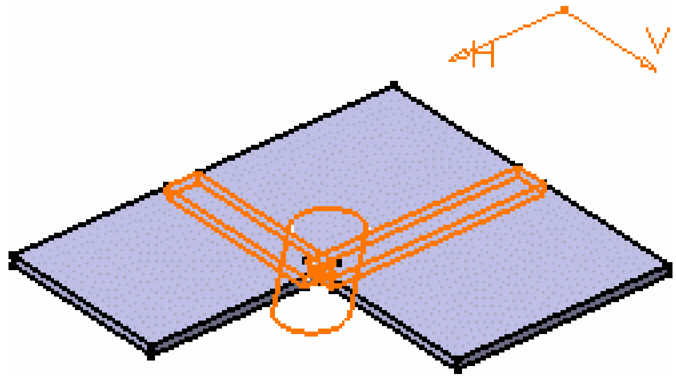


1. Select a sketch and click OK in the Catalog Browser.

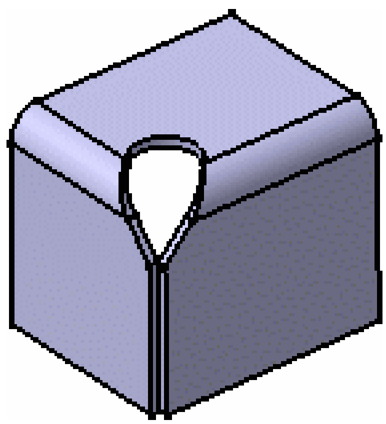
You return to the Corner Relief Definition dialog box, that has been updated with the Sketcher icon.

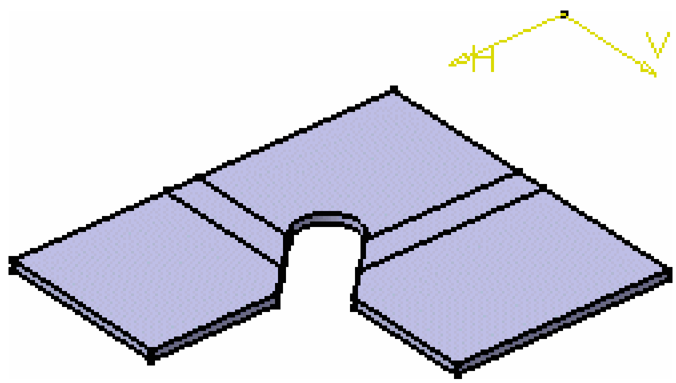


2. Click the sketcher icon, and move the sketch to position it properly in relation to the part.



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.

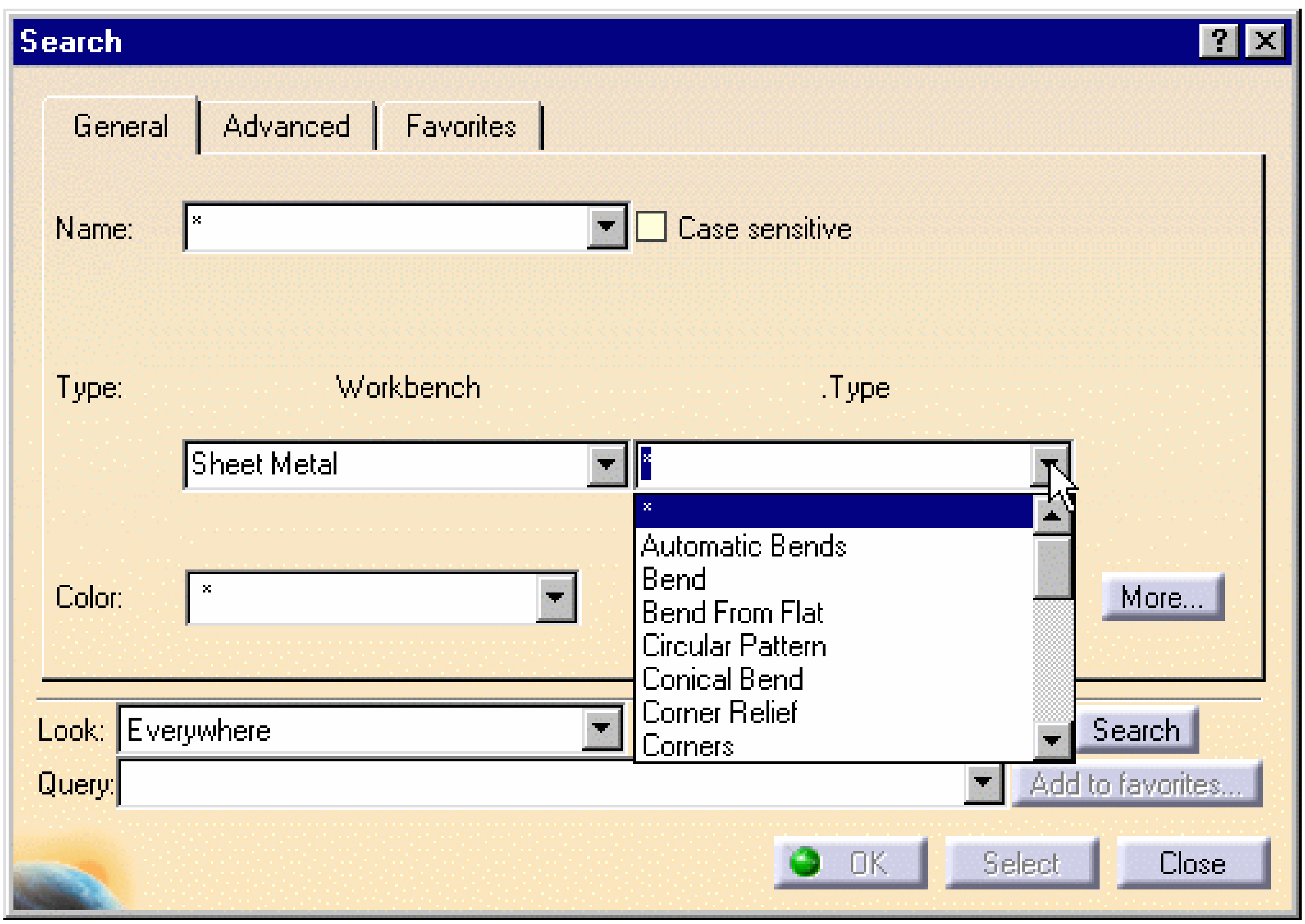
Open the PowerCopyStart.CATPart document.

1. Select the **Edit** -> **Search** menu item.

The Search dialog box is displayed.

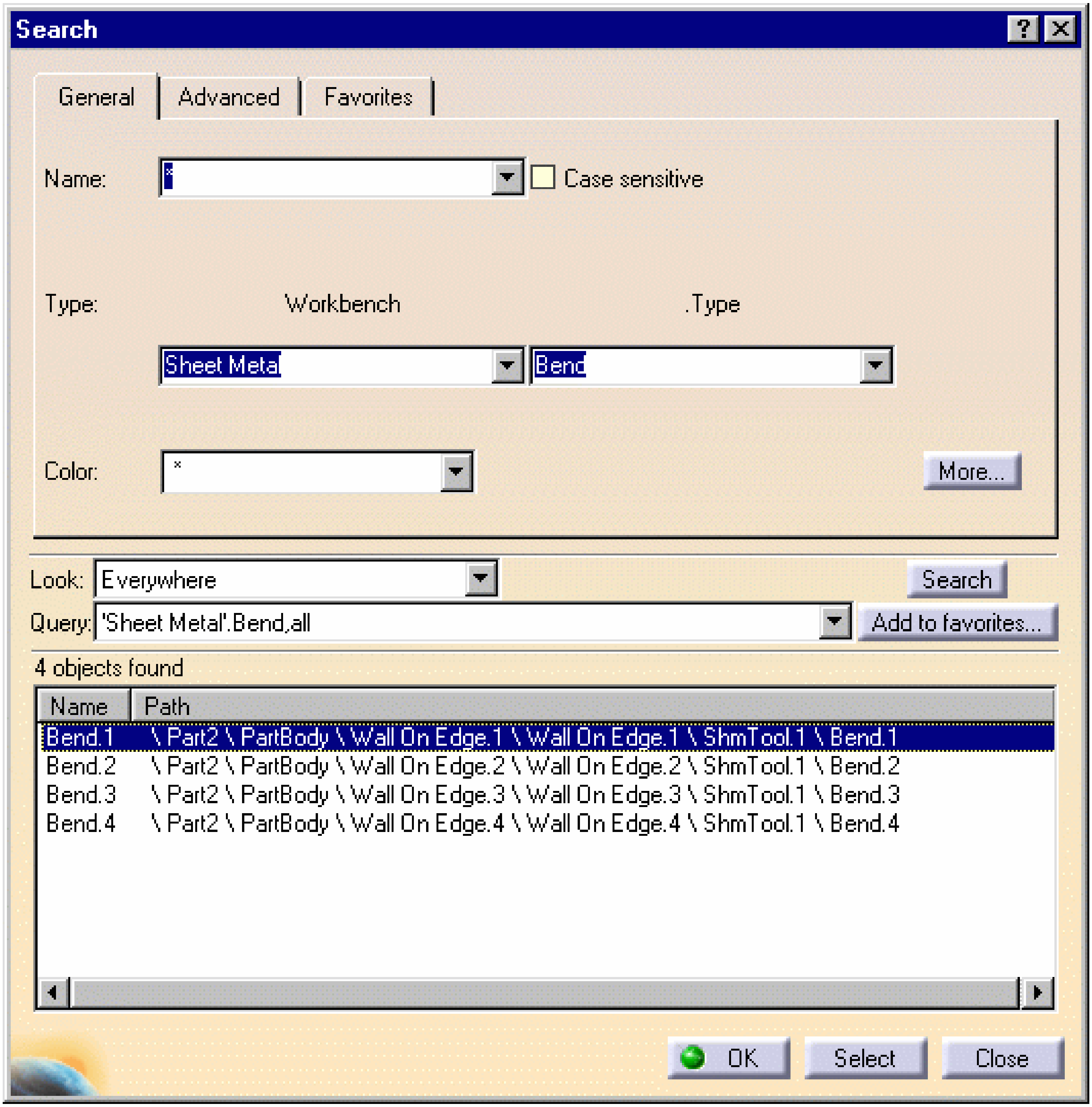
2. From the Type Workbench list choose Sheet Metal.

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*.

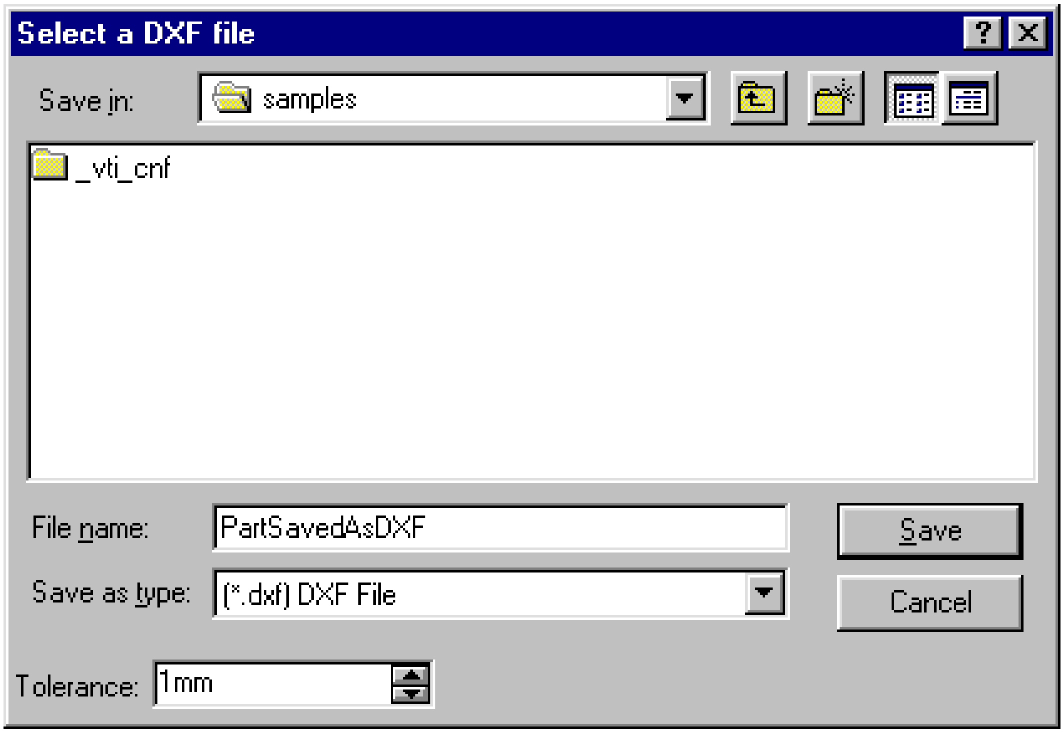
Saving As DXF

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.

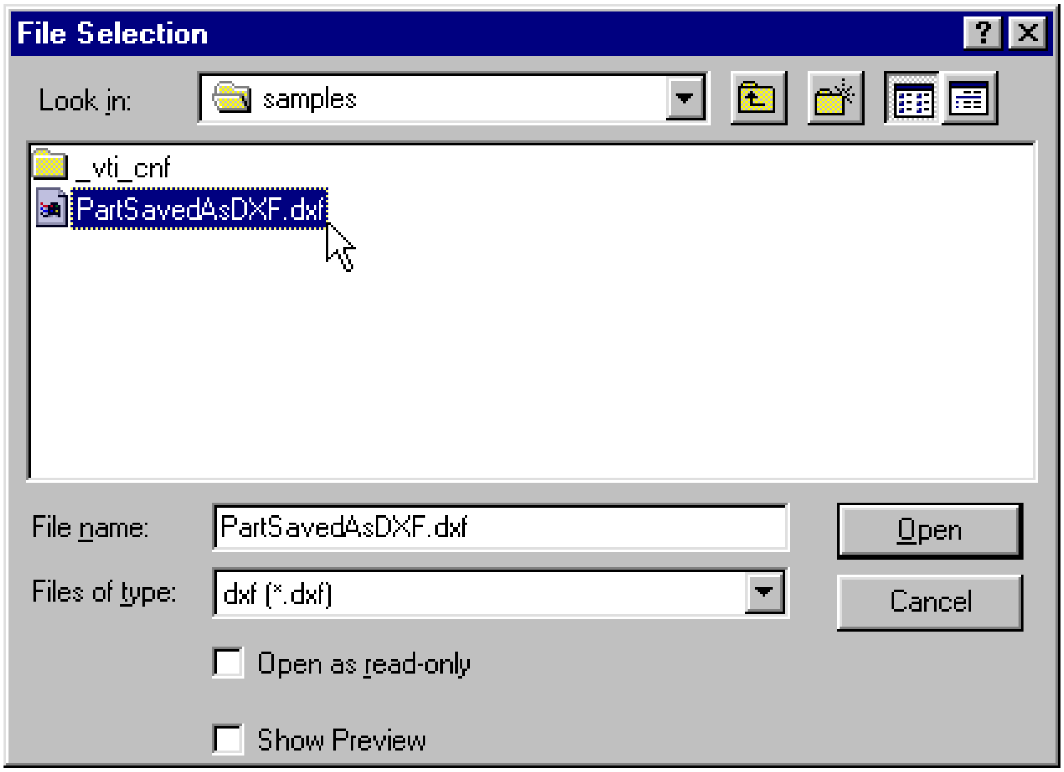


2. Indicate the correct path and file name (we saved it as PartSavedAsDXF.dxf).
3. Click **Save**.

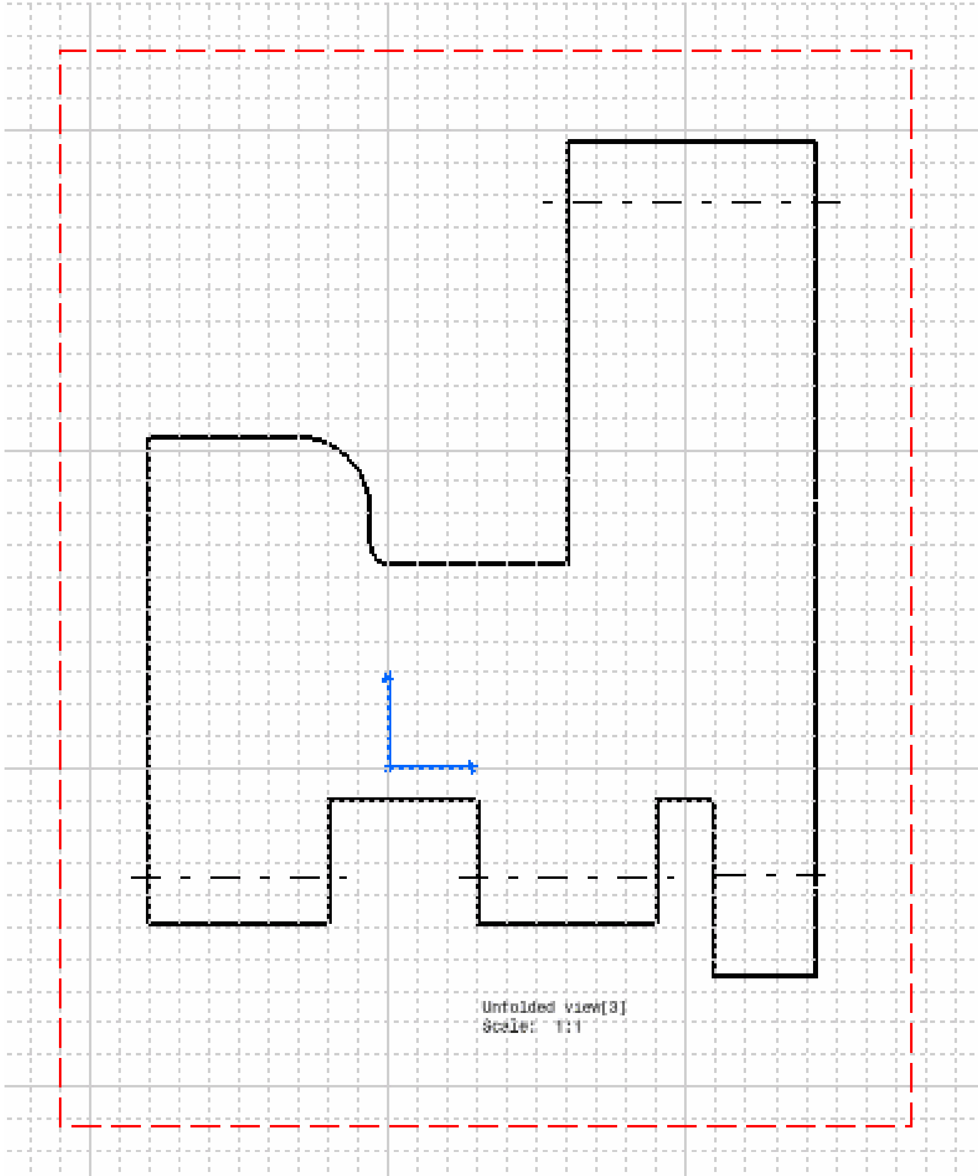
The geometry has been saved, and can be imported as a DXF file in any system supporting this type.

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.

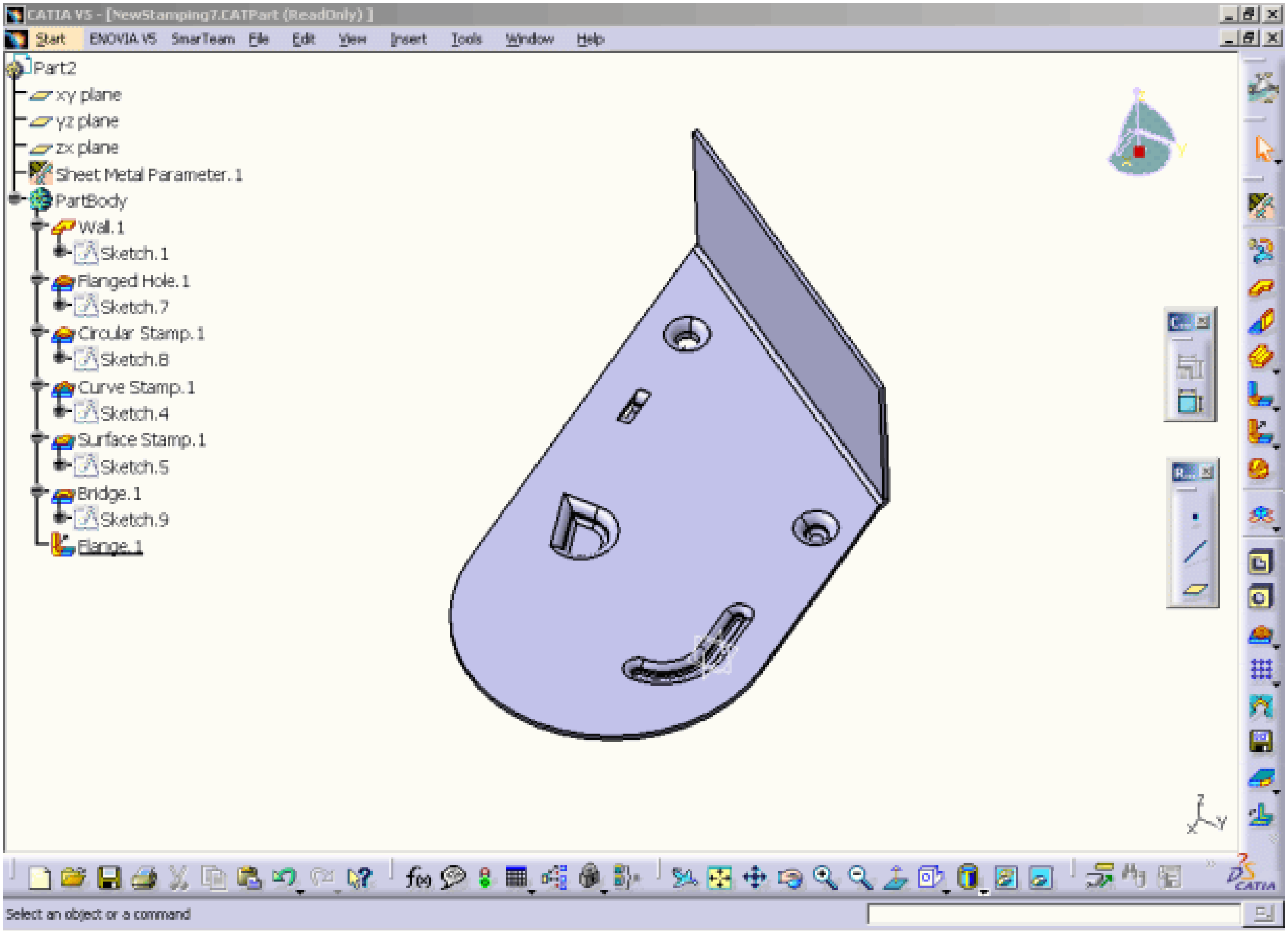


Note that the axes of bends and planar hems, tear drops, or flanges are automatically displayed on the drawing.

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

Workbench Description

The Generative Sheetmetal Design application window looks like this. Click the hotspots to display the related documentation.



Menu Bar
Generative Sheetmetal Toolbar
Constraints Toolbar
Reference Elements Toolbar
Specification Tree

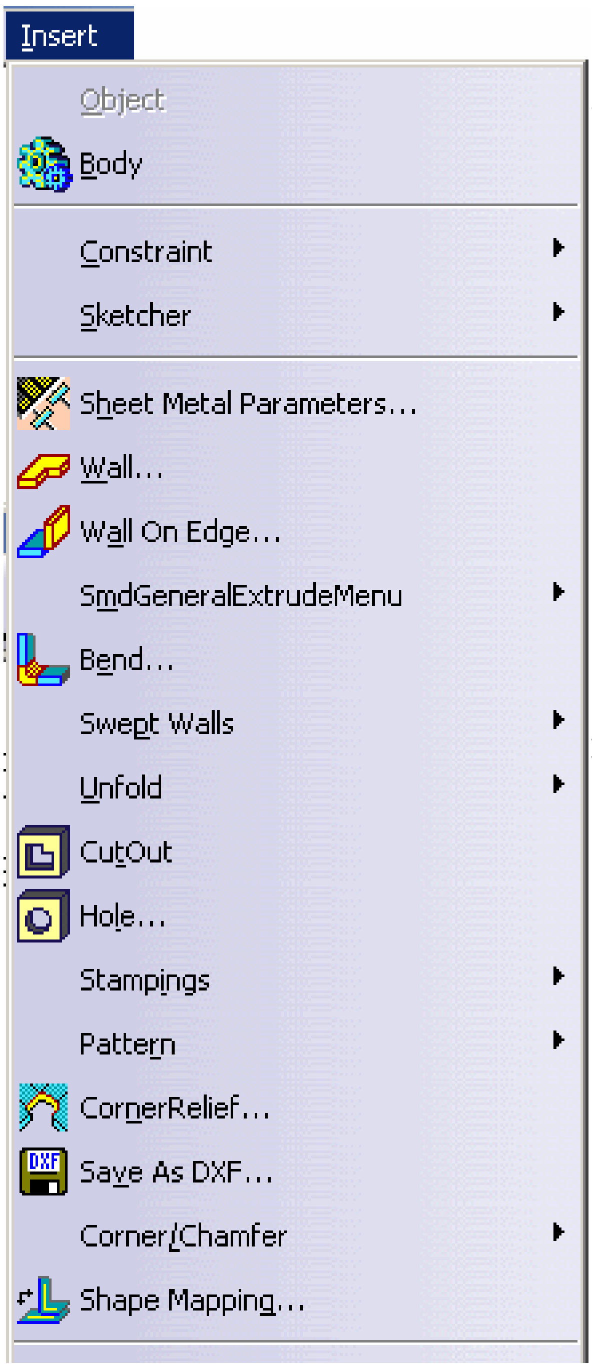
Menu Bar

The various menus and menu commands that are specific to Sheet Metal Design are described below.

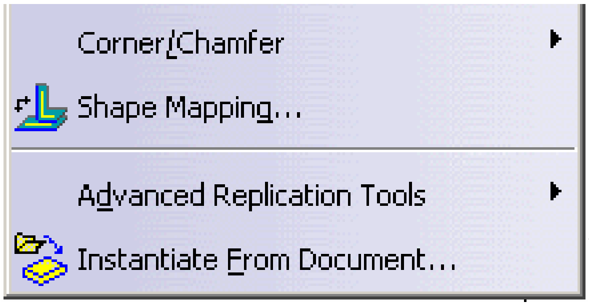
Start	File	Edit	View	Insert	Tools	Windows	Help

Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*. Refer to the Menu Bar section.

Insert



For...	See...
Constraints	Setting Constraints in the *Part Design User's Guide*
Sketcher	Refer to Sketching in the *Sketcher User's Guide*.
Sheet Metal Parameters...	Managing the Default Parameters
Wall...	Creating Walls from a Sketch
Wall on Edge...	Creating Walls From An Edge
Extrudes	Insert -> Extrudes
Bend...	Creating Bends on Walls
Swept Walls	Insert -> Swept Walls
Unfold	Insert -> Unfold
CutOut	Creating a Cutout
Hole...	Creating a Hole
Stampings	Insert -> Stampings
Patterns	Insert -> Patterns
CornerRelief...	Creating a Local Corner Relief
Save As DXF...	Saving As DXF



Corner/Chamfer	Insert -> Corner/Chamfer
Shape Mapping...	Mapping Elements
Advanced Replication Tools	Insert -> Replication Tools
Instantiate From Document...	Instantiating PowerCopy Features

Insert -> Extrudes

For...	See...
Extrude...	Extruding
Rolled Wall...	Creating Rolled Walls

Insert -> Swept Walls

For...	See...
Flange	Creating a Flange
Hem	Creating a Hem
Tear Drop	Creating a Tear Drop
User Flange	Creating a User Flange

Insert -> Unfold

For...	See...
Unfold	Folded/Unfolded View Access
MultiView	Concurrent Access

Insert -> Stampings

For...	See...
Flanged Hole...	Creating a Flanged Hole
Bead...	Creating a Bead
Circular Stamp...	Creating a Circular Stamp
Surface Stamp...	Creating a Surface Stamp
Bridge...	Creating a Bridge
Flanged CutOut...	Creating a Flanged Cutout
Stiffening Rib...	Creating a Stiffening Rib
Curve Stamp...	Creating a Curve Stamp

User Stamp...	Creating User-defined Stamping Features
Louver...	Creating a Louver

Insert -> Patterns

For...	See...
Rectangular Pattern	Creating Rectangular Patterns
Circular Pattern	Creating Circular Patterns
User-Defined Pattern	Creating User-Defined Patterns

Insert -> Corner/Chamfer

For...	See...
Corner...	Creating Corners
Chamfer...	Creating Chamfers

Insert -> Replication Tools

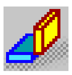
For...	See...
PowerCopy Creation	Creating PowerCopies
PowerCopy Save in Catalog	Saving PowerCopies

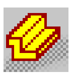
Generative Sheetmetal Toolbar

 See Managing the Default Parameters

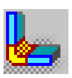
 See Recognizing Walls From an Existing Part

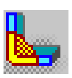
 See Creating Walls from a Sketch

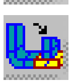
 See Creating Walls From An Edge

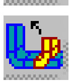
 See Extruding

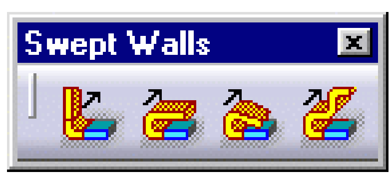
 See Creating Rolled Walls

 See Creating Bends From a Line

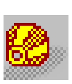
 See Creating Conical Bends

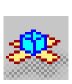
 See Local fold and unfold of bends

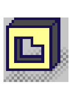
 See Local fold and unfold of bends

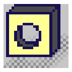


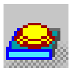
 See Creating Swept Walls

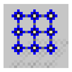
 See Creating Hoppers

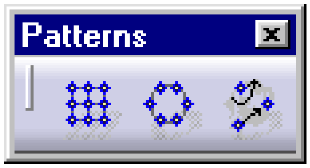
 See Unfolding

 See Creating a Cutout


see Creating a Hole

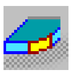

See Stamping

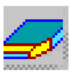

See Patterning

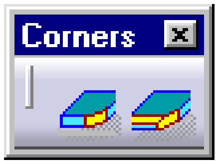


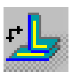

See Creating a Local Corner Relief

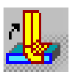
See Saving As DXF


See Creating Corners

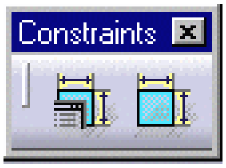

See Creating Chamfers

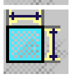



See Mapping Elements

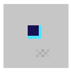

See Creating Bends from a Line

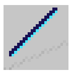
Constraints Toolbar

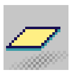



 See Setting Constraints from the *Part Design User's Guide*

Reference Elements Toolbar

 See Creating Points

 See Creating Lines

 See Creating Planes

Specification Tree

Within the Sheet Metal 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
- Rolled Wall
- Bend
- Flange
- Hem
- Tear Drop
- Swept Flange
- Cutout
- Hole
- Flanged Hole
- Bead
- Circular Stamp
- Surface Stamp
- Bridge
- Flanged Cutout
- Stiffening Rib
- Rectangular Pattern
- Circular Pattern
- User-Defined Pattern
- Corner Relief
- Corner
- Chamfer
- Mapping
- Point
- Line
- Plane
- Flat Bend

Curve Stamp

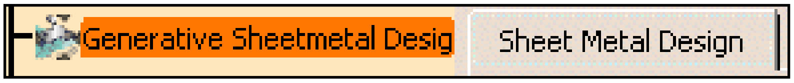
User-defined Stamp

Louver

Customizing

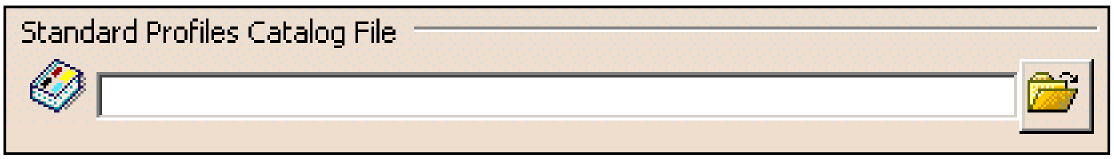
This section describes how to customize standards files and settings specific to the Generative Sheetmetal Design workbench.

Customizing settings



This page deals with the following category of options in the **Aerospace Sheet Metal Design** tab: Standard Profiles Catalog File.

Standard Profiles Catalog File



Enter the default path in this field. You may click the **Browse** icon .

By default, this field is empty.

If no catalog path has been defined prior to entering the Catalog Browser command, the default catalog is selected and its path is automatically added to the Standard Profiles Catalog File field.

Customizing Standard Files

This section describes how to customize settings.
The task described here deals with permanent setting customizing.

Using Sheet Metal Standards Files

This task explains how to access company standards files.

Open a new document.

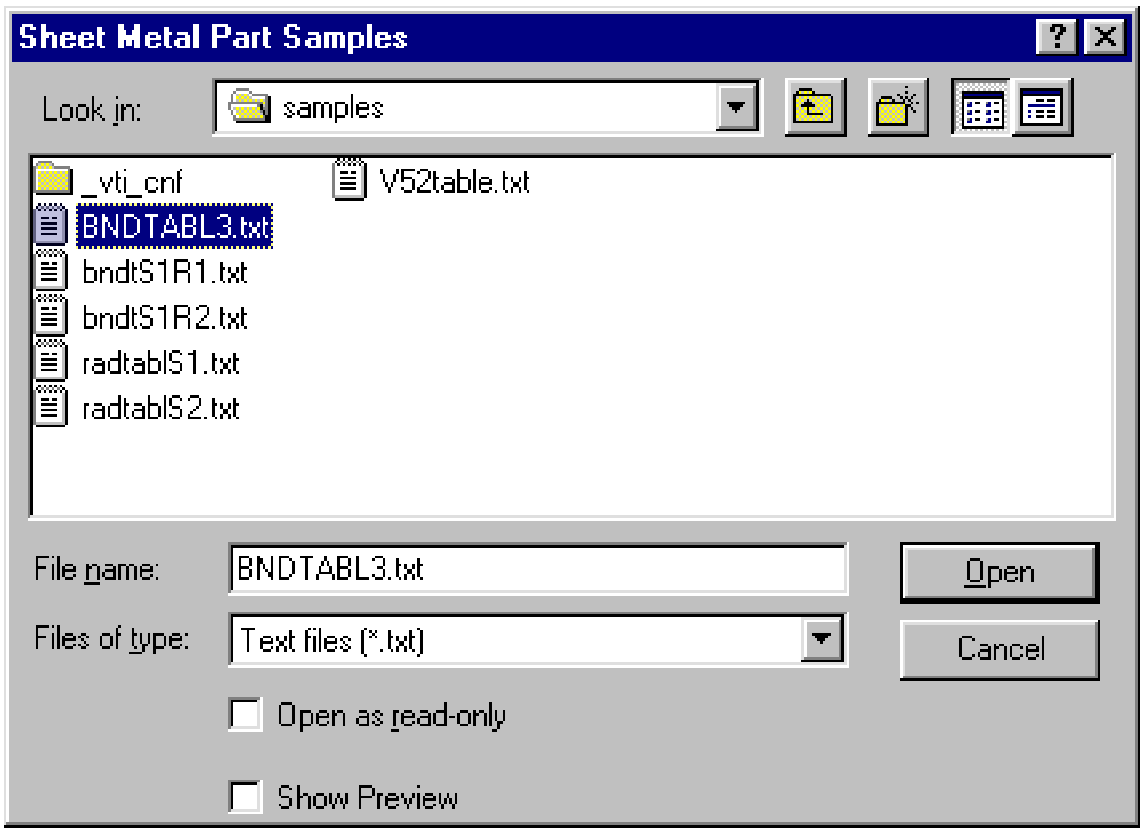
1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box opens.

2. Select the **Sheet Standards Files...** button.

   The Sheet Metal Part Samples window is displayed.

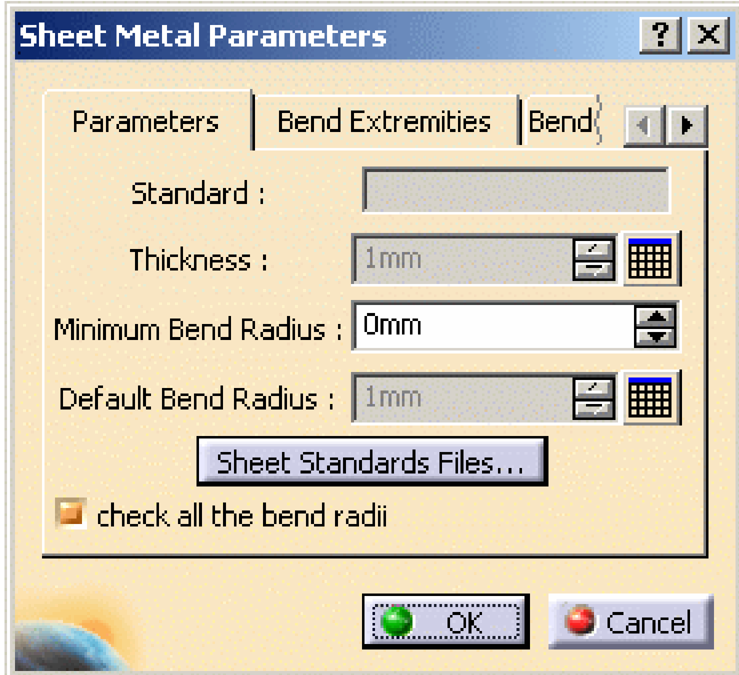
3. Indicate the path to the Sheet Metal table.



These files are available under .xls or .txt format.

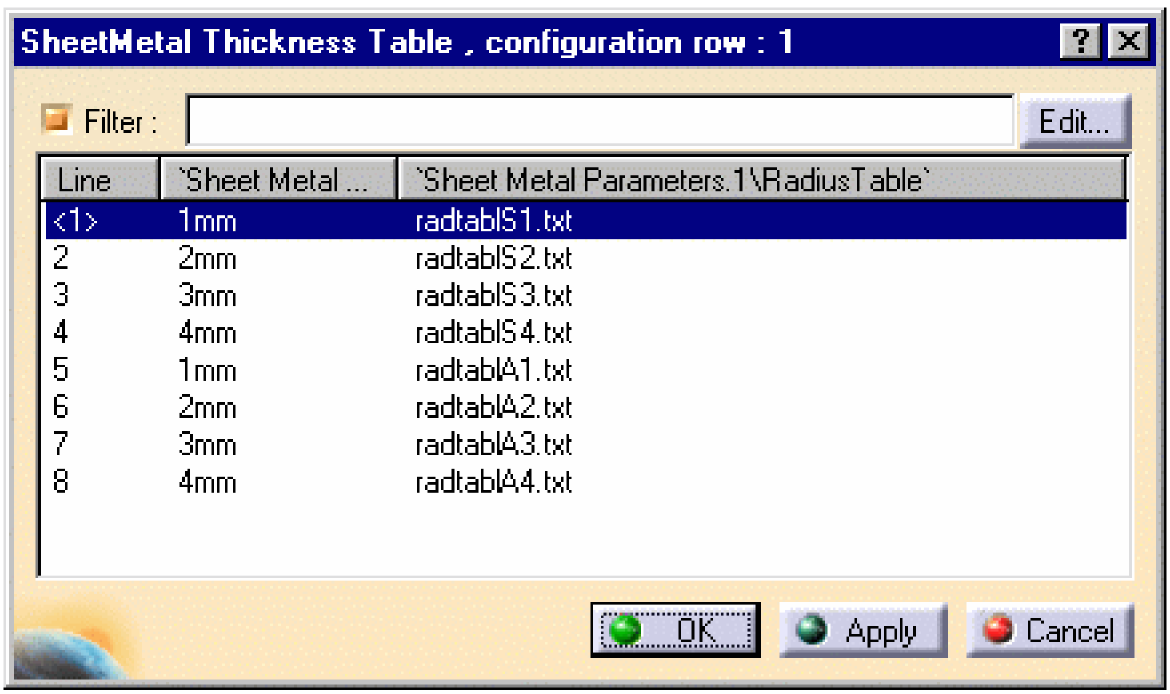
4. Click **Open**.

In the Sheet Metal Parameters dialog box, the **Design Table** icon appears opposite the Thickness and Bend radius fields.



The parameters are now in gray, indicating that you can no longer modify the values.

5. Click the Thickness Design Table icon and select the line containing the appropriate parameters (for example Line 1).

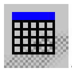


Using the **Tools -> Options -> General -> Document** tab, **Other Folders** option, you can specify where the files are located. Refer to Document.

This scenario can work when the .CATPart document and all reference table files (Design & Radius) are located in the same directory. This directory is the current one when the Design table is created, and also when the .CATPart is open.
However, generally speaking, you must reference the complete path indicating where the radius table files are to be found in the RadiusTable column. In this case, regardless of the current directory, the correct tables are located when re-opening the .CATPart document.

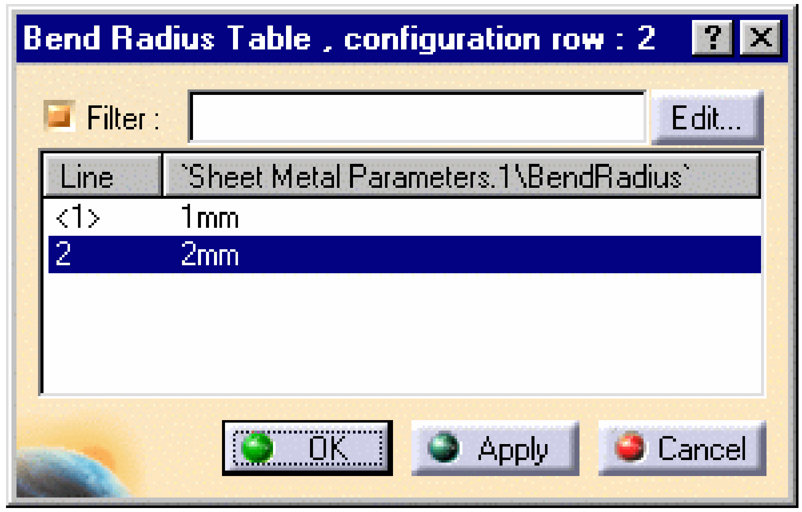
6. Click **OK**.

The parameter values are updated in the Sheet Metal Parameters dialog box.

7. Click the Bend Radius Design Table icon .

8. Select line 2 and click OK.

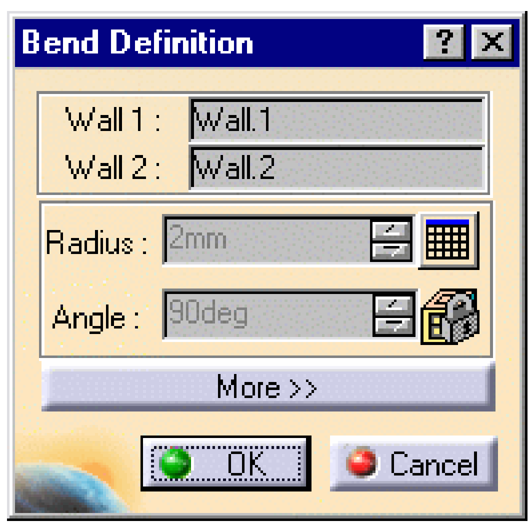
The parameter values are updated in the Sheet Metal Parameters dialog box.



9. Create a bend.

The Bend Definition dialog box displays a design table for the Bend Radius.

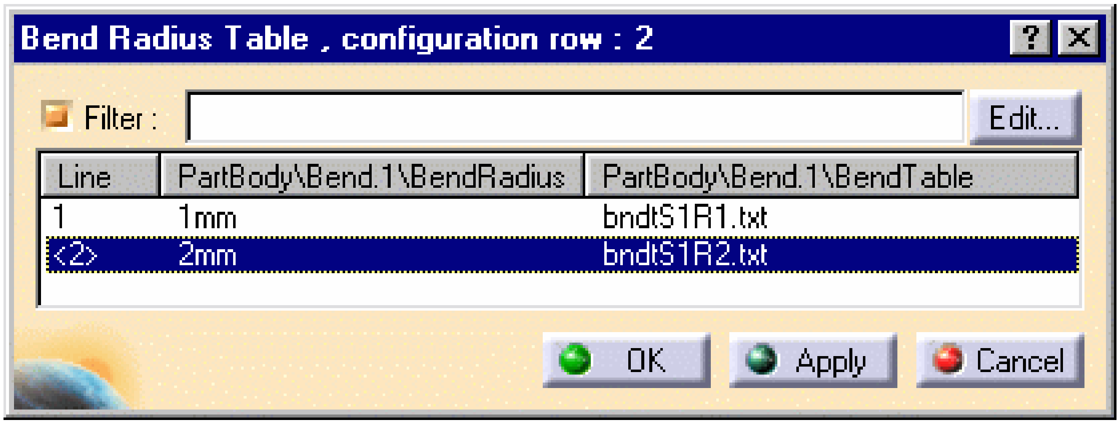
The default mode, that is to say the formula: **Bend Radius = Part Radius**



is

deactivated.

Let's see the Bend Radius Table, using this icon .



It shows the Bend Radius and the corresponding Bend Table.

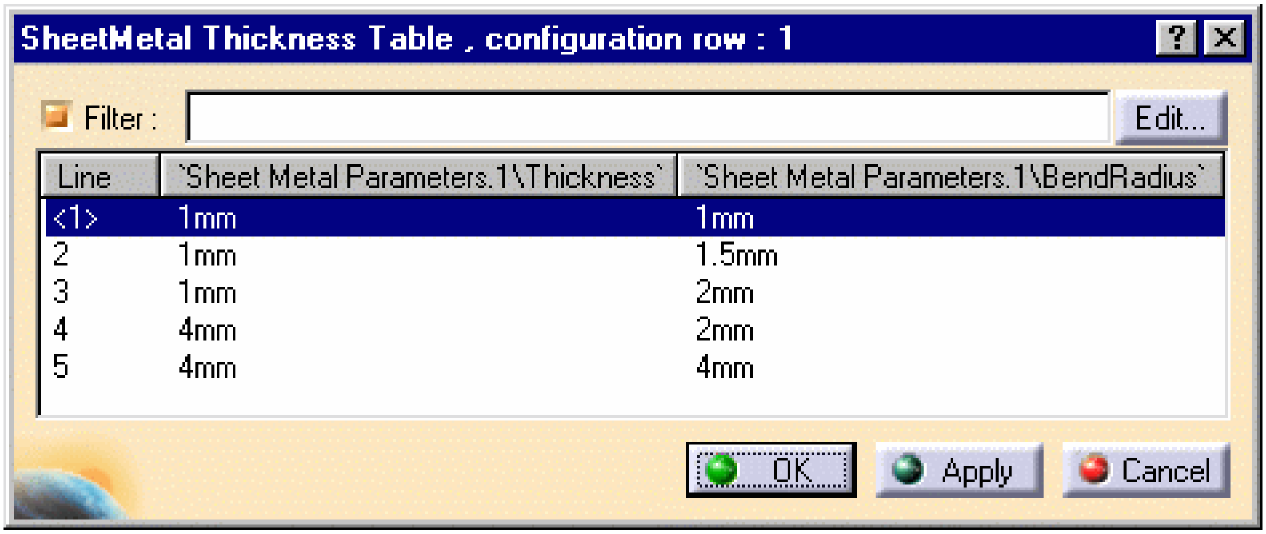
10. Click OK.

If the **Angle** value is contained in the **Bend Table**, the **Bend Allowance** uses the corresponding value.
If not, the **Bend Allowance** is computed according to the KFactor.

Using the Sheet Metal Design Tables:

Steps 1 to 4 are identical.

5. Click the Design Table icon and select a line.



6. Click **OK**.

   The parameter values are updated in the Sheet Metal Parameters dialog box.

At that time, the parameters Thickness and Bend radius are driven by the design table.

They are now in gray, indicating that you can no longer modify the values.

Note that if you create a bend, there is no design table: it's the formula which is used.

To disable the access to design tables:

- Select the **Tools -> Options -> Part -> Display** tab and check **Relations:**

  the **Design Table** icon is displayed in the specification tree.

- Right-click this icon: the contextual menu appears.

- Select **SheetMetal Thickness Table object -> Deactivate**

The relation is no longer used but still exists.

It can be activated at any time.

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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Generative Drafting

Sheet Metal Design