Aerospace Sheetmetal Design

Overview

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

This overview provides the following information:

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

Aerospace SheetMetal Design in a Nutshell

The Aerospace Sheetmetal Design workbench provides an associative feature-based modeling, making it possible to design sheetmetal parts in concurrent engineering between an unfolded or folded part representation.

Aerospace Sheetmetal allows you to define a part using predefined features. Both folded geometry and flattened geometry can be computed from the feature specifications.

The *Aerospace SheetMetal Design User's Guide* has been designed to show you how to design aerospace 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.
- *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 Aerospace 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\aslug\samples folder. When samples belong to capabilities common to different SheetMetal products, those samples will be found 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.

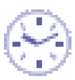
Graphic Conventions

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

Index

Text Conventions

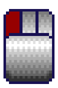
The following text conventions are used:

- 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.

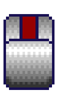
How to Use the Mouse

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

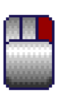
Use this mouse button... | **Whenever you read...**



- 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

Hybrid Design

You can now create wireframe and surfacic features within the same solid body which impacts the behavior of webs, surfacic flanges and corner reliefs.

User-stamp

You can now create user-stamps.

Enhanced Functionalities

Joggle on Web

You can now create joggles on a web.

Runout parameters

You can now define precisely the runout parameters of the joggles.

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.

Flange pattern

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

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 Version 5 CATIA - Aerospace Sheet Metal 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 Aerospace SheetMetal Design Workbench
Defining the Aerospace SheetMetal Parameters
Creating a Web from a Sketch
Creating a Surfacic Flange on a Web
Creating a Cutout
Extracting Drawings from the Aerospace SheetMetal Design Part

All together, these tasks should take about 20 minutes to complete.

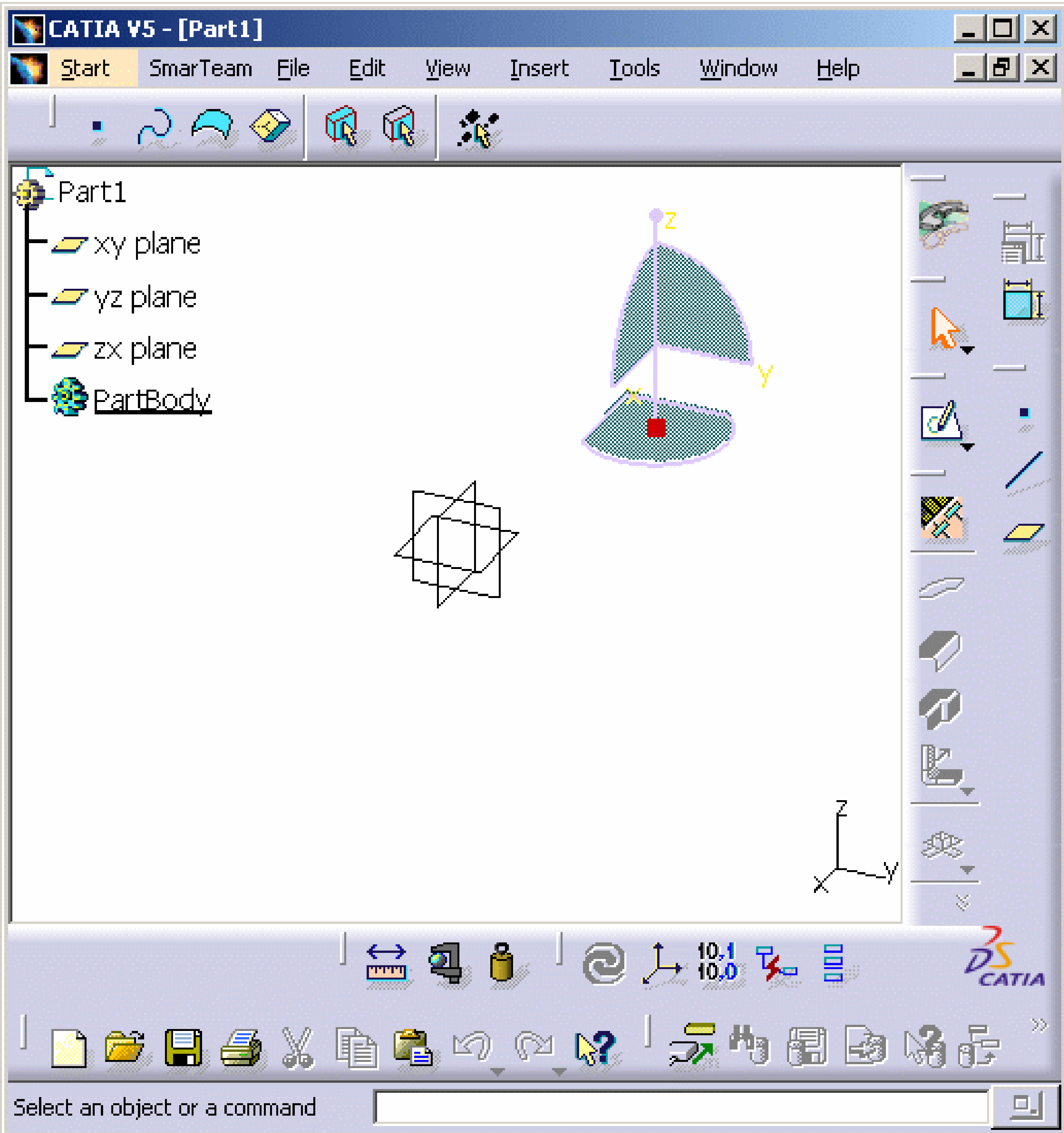
Entering the Aerospace SheetMetal Design Workbench

The Aerospace Sheet Metal 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.

Choose the **Mechanical Design -> Aerospace Sheet Metal Design** item from the **Start** menu.

The Aerospace Sheet Metal toolbar is displayed and ready to use.



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

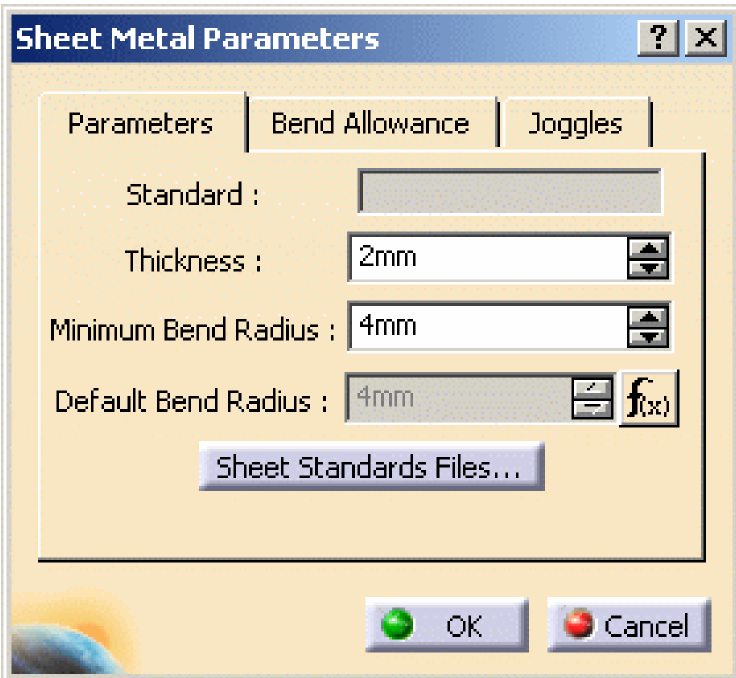
If you wish to use the whole screen space for the geometry, remove the specification tree clicking off the **View -> Specifications** menu item or pressing F3.

Defining the Aerospace SheetMetal Parameters

This task shows you how to configure the Aerospace SheetMetal parameters.

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

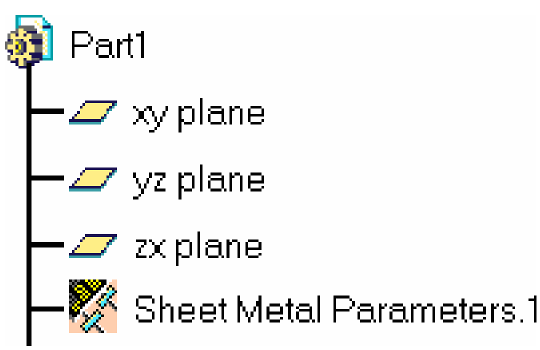
The Sheet Metal Parameters dialog box is displayed.



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.

4. Click **OK** to validate the parameters and close the dialog box. The **Sheet Metal Parameters** feature is added in the specification tree.



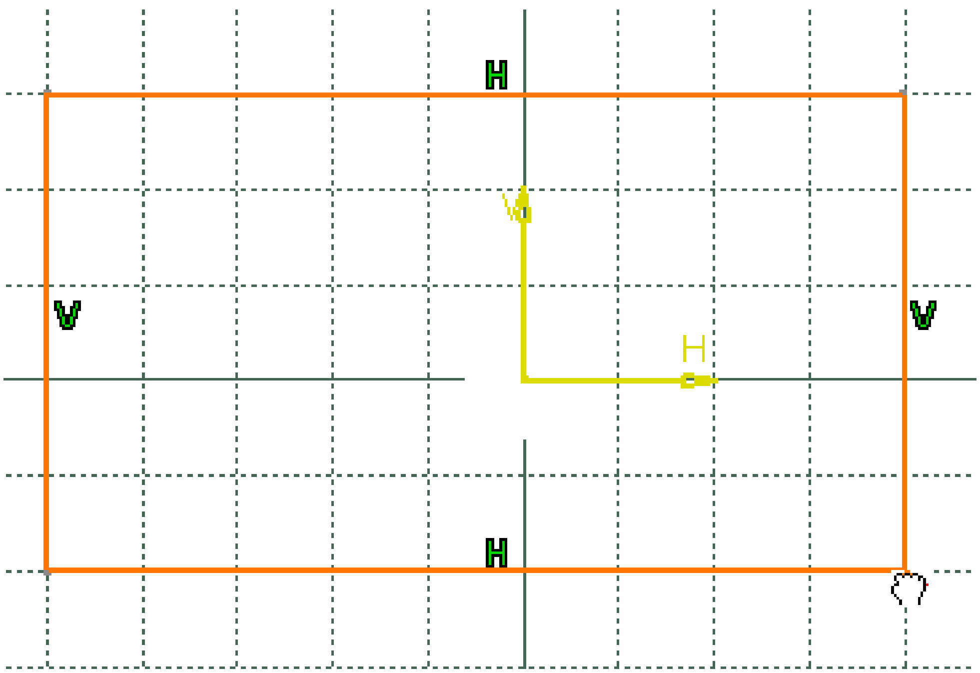
The other two tabs are not used in this scenario.

Creating a Web from a Sketch

This section explains how to create a web.

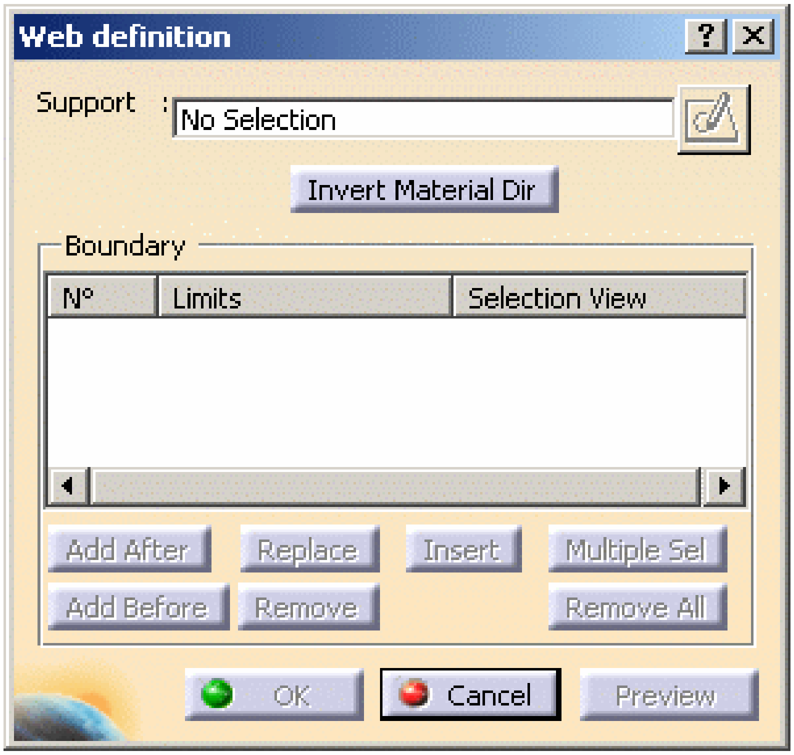
The web is the main feature of an Aerospace Sheetmetal Design part: there is always one (and only one) web.

1. Click the **Sketcher** icon .
2. Select the xy plane.
3. Click the **Rectangle** icon in the Profile toolbar to create the profile of the web.
4. Click to create the first point and drag the cursor.
5. Click to create the second point: the rectangle profile is displayed.
6. Click the **Exit workbench** to return to the 3D world.



7. Click the **Web** icon .

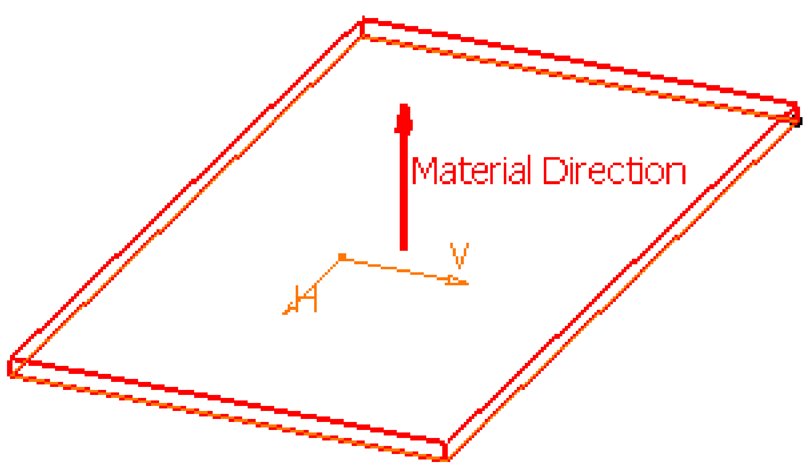
The Web definition dialog box is displayed.



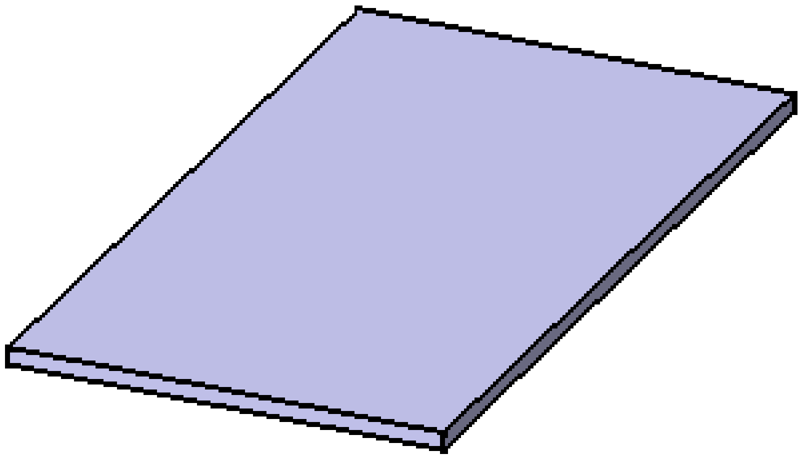
8. Select the sketch you just created as the support of the web.

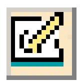
   A preview of the web appears.

9. Click OK to create the web.



Here is the web.



You can click the **Sketcher** icon  to edit the sketch.

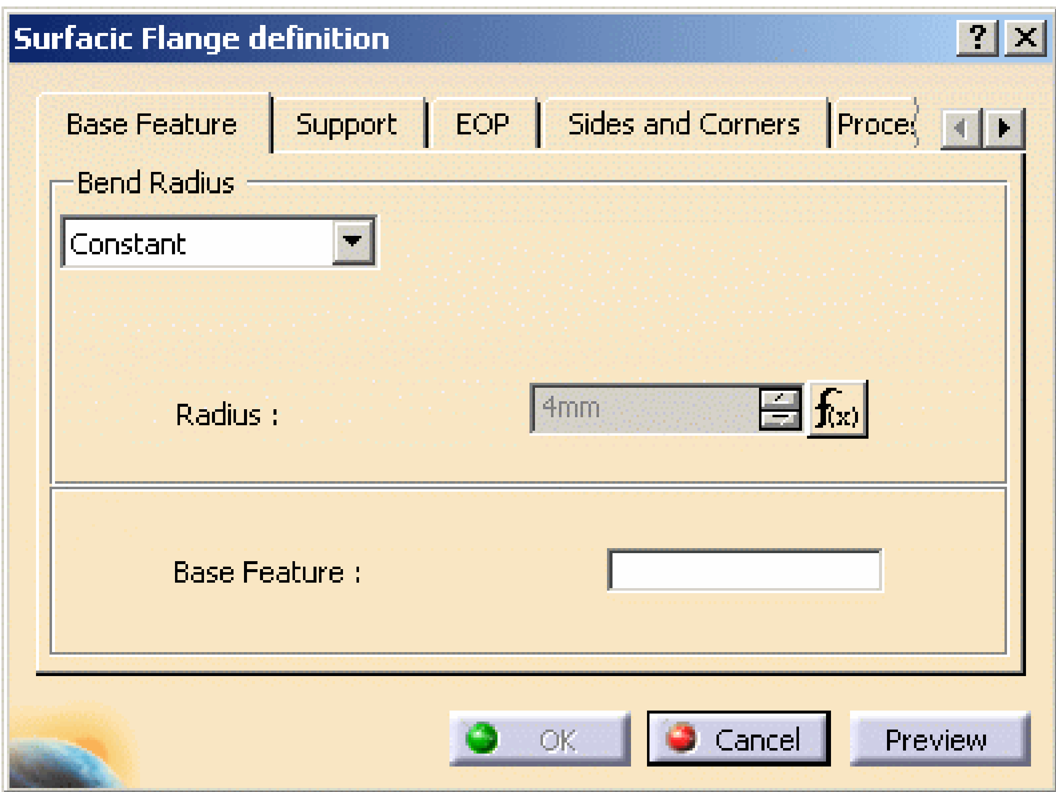
Creating a Surfacic Flange on a Web

This section explains how to create a surfacic flange on a web, that is a feature which enables to stiffen the part.

The web is still open from the previous task.

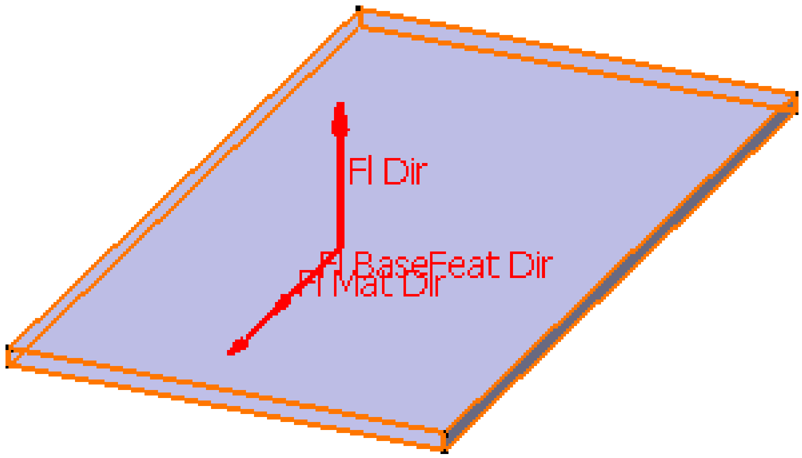
1. Click the **Surfacic flange** icon .

The Surfacic Flange definition dialog box is displayed.

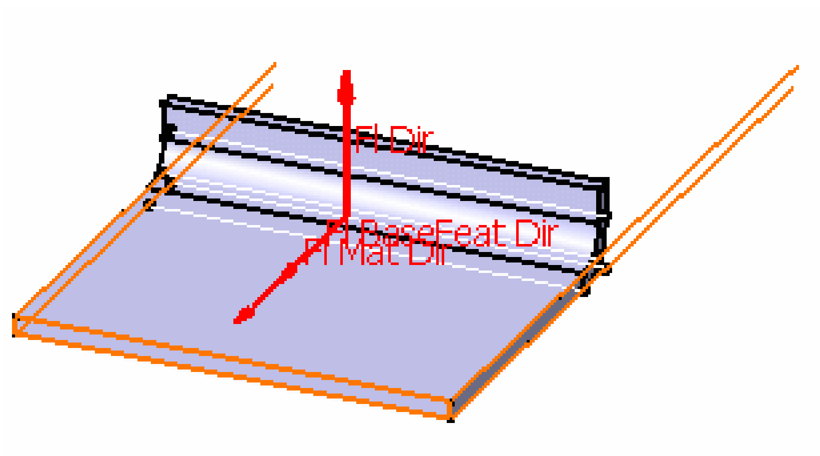


2. Choose the web as the Base Feature.

3. Choose the yz plane as support.

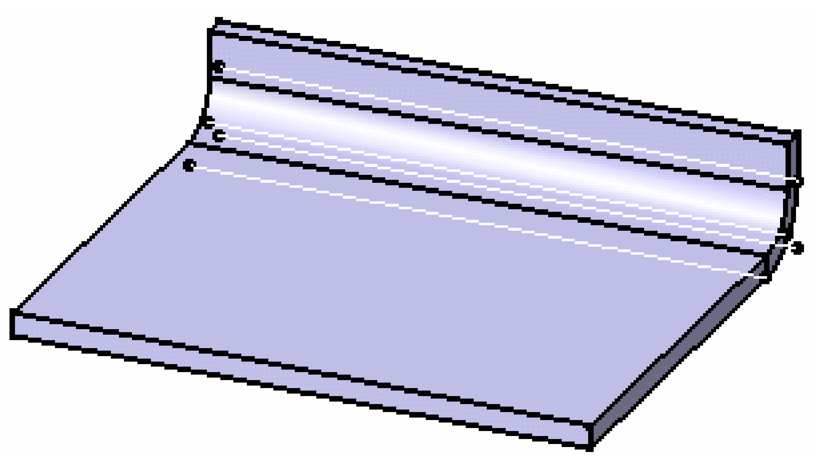


4. Click **Preview** to see the surfacic flange.



5. Click OK to create the surfacic flange.

Here is the surfacic flange.



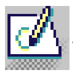
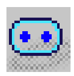
Creating a Cutout with a Sketch

In this task, you will learn how to:

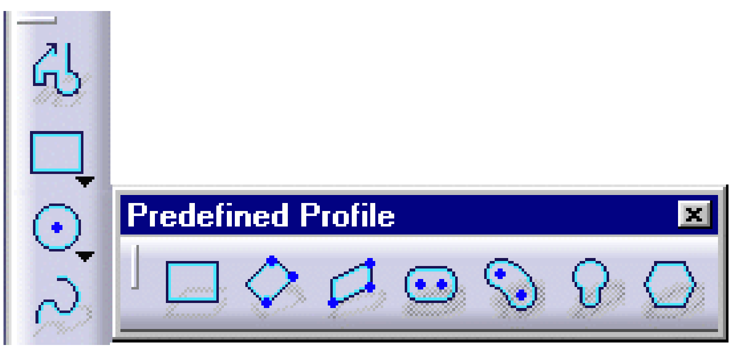
- open a sketch on an existing face
- define a profile on the face in order to create a cutout.

You can create a cutout defined either by a sketch or an open geometry.

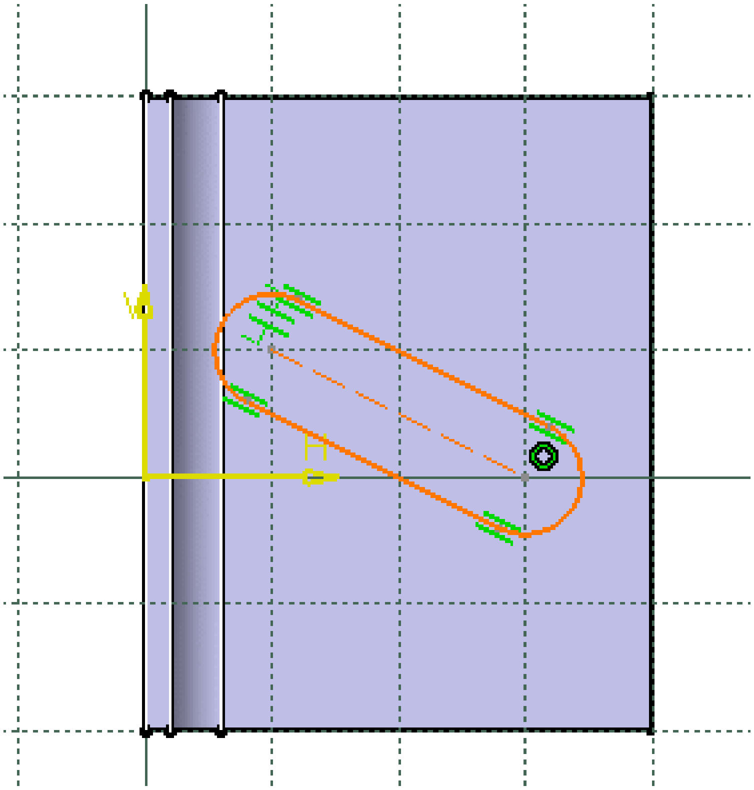
The surfacic flange is still open from the previous task.

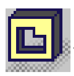
1. Select the surface from the geometry area to define the working plane.
2. Click the **Sketcher** icon .
3. Click the **Elongated Hole** icon  to create the contour.

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.

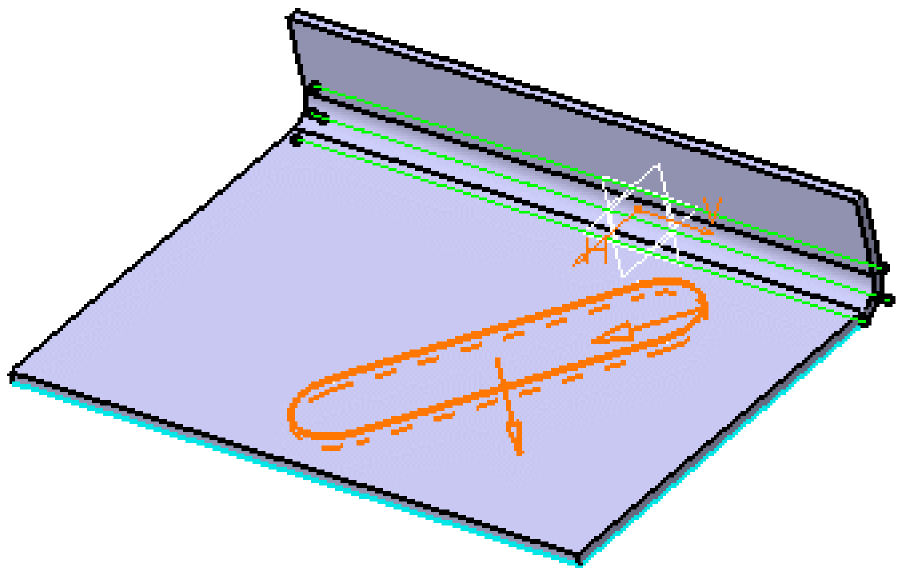


7. Click the **CutOut** icon .

7. Select the sketch.

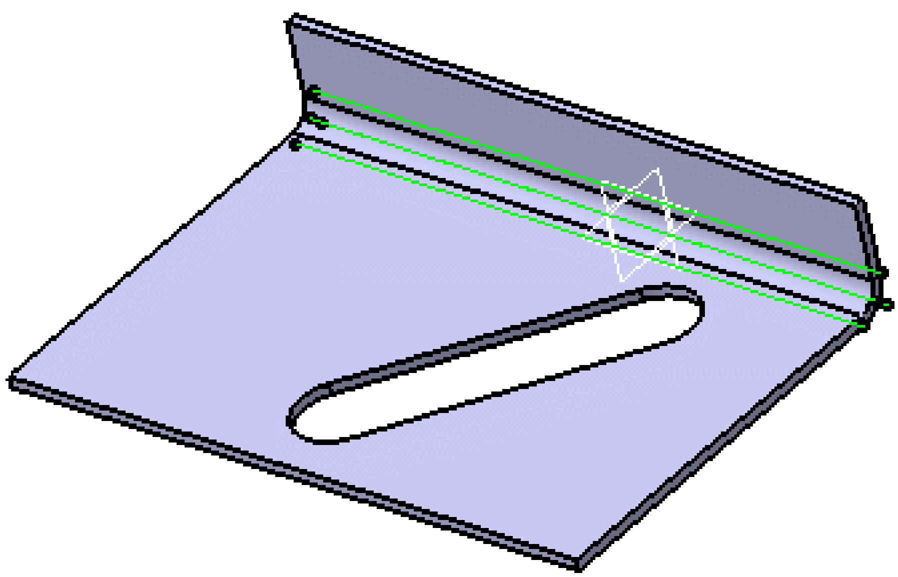
The CutOut Definition dialog box is displayed and a cutout is previewed with default parameters.

The vectors show the side and the direction of the cutout.



8. Select the **Dimension** type to define the limit of your cutout.

9. Click OK. Here is your cutout.



Extracting Drawings from the Aerospace SheetMetal Design Part

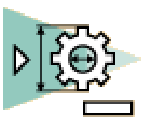
This task shows how to create the Aerospace SheetMetal Design Part views in the Generative Drafting workbench.

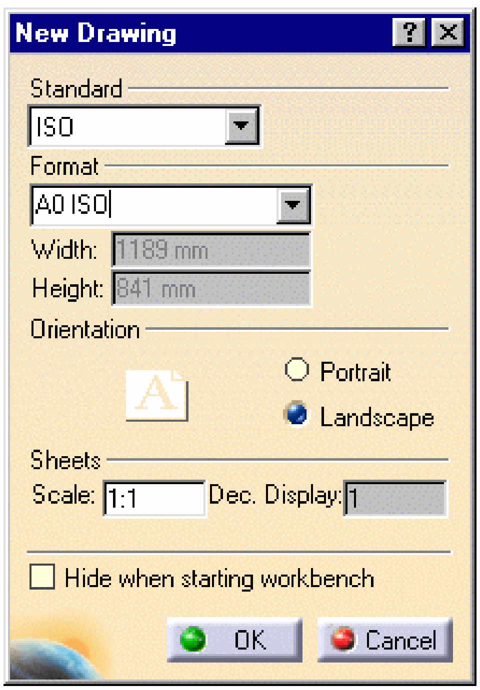
The Sheet Metal part is displayed.

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

   The Generative Drafting workbench is launched.

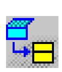
   The New Drawing dialog box opens.





3. Click OK.

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

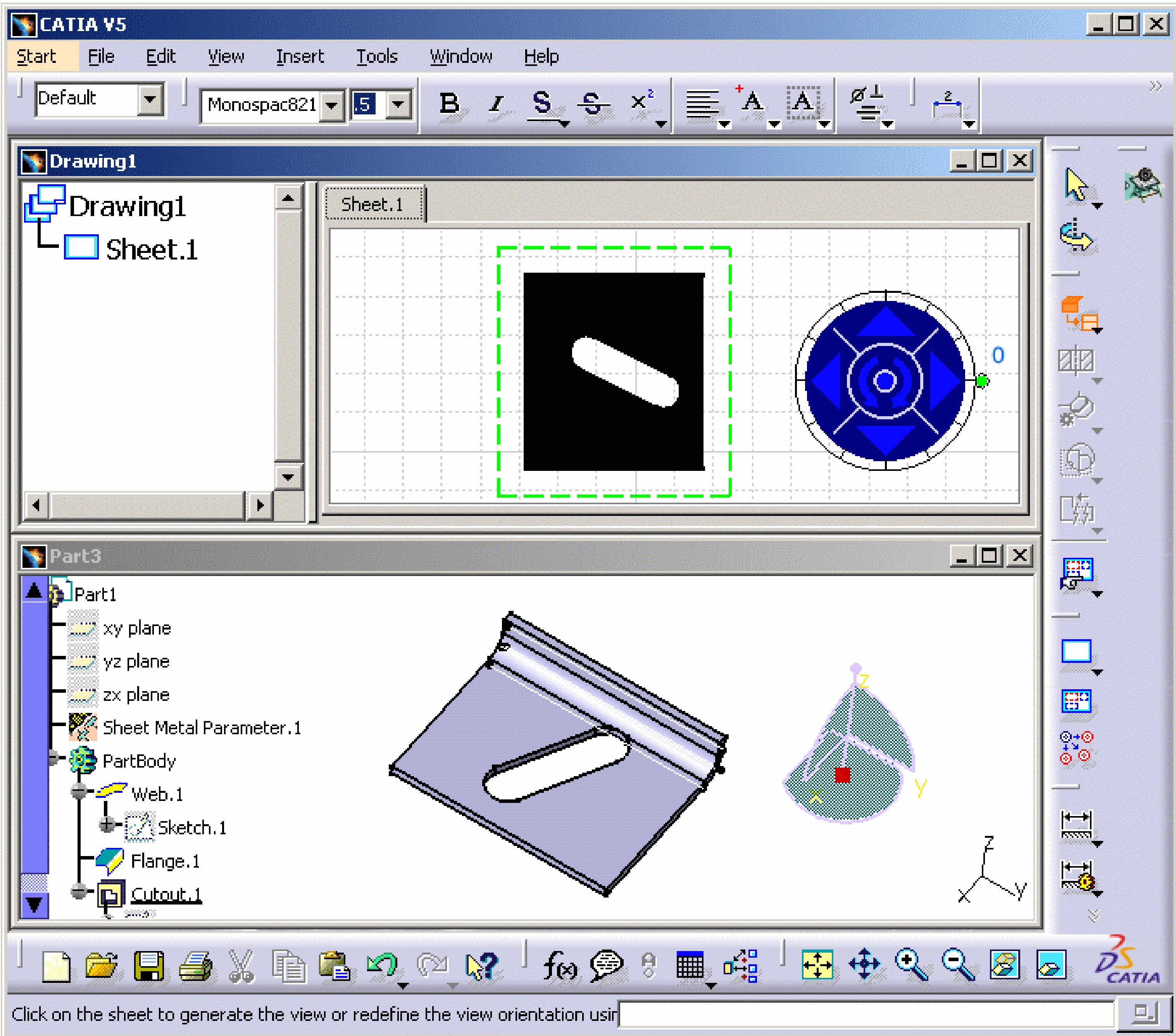
4. The drawing sheet appears.
5. Tile the windows horizontally using the **Window -> Tile Horizontally** menu item.
6. Select the Unfolded View icon  in the Projections toolbar from Generative Drafting Workbench.

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 displayed with the bends axes and limits.

   Eventually, the Drafting sheet looks like this:



User Tasks

Managing the Default Parameters

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

Using Aerospace Sheetmetal Design assumes that you are in a CATPart document.

Edit the parameters: select the Parameters tab and define the element thickness and bend radius values.

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

Define the compensations and runout: select the Joggles tab and define the compensations and the runout for the joggle.

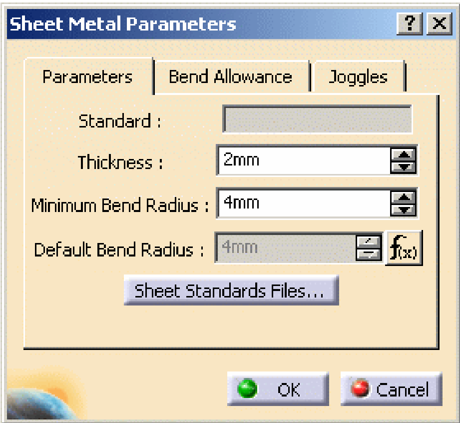
Please refer to the Customizing chapter to define the Sheet Standards Files.

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.

4. Change the **Default Bend Radius** if needed. To do this, you need to deactivate the formula first by right clicking the formula icon.

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

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.
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.

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

To know more about the different ways to access you files, refer to the Opening Existing Documents Using the Browse Window 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.

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
Stiffening Rib		

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

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

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

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

Surface Stamp

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

Curve Stamp

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

Circular Stamp

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

Bead

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

Flanged Cutout

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

Flanged Hole

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

Stiffening Rib

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

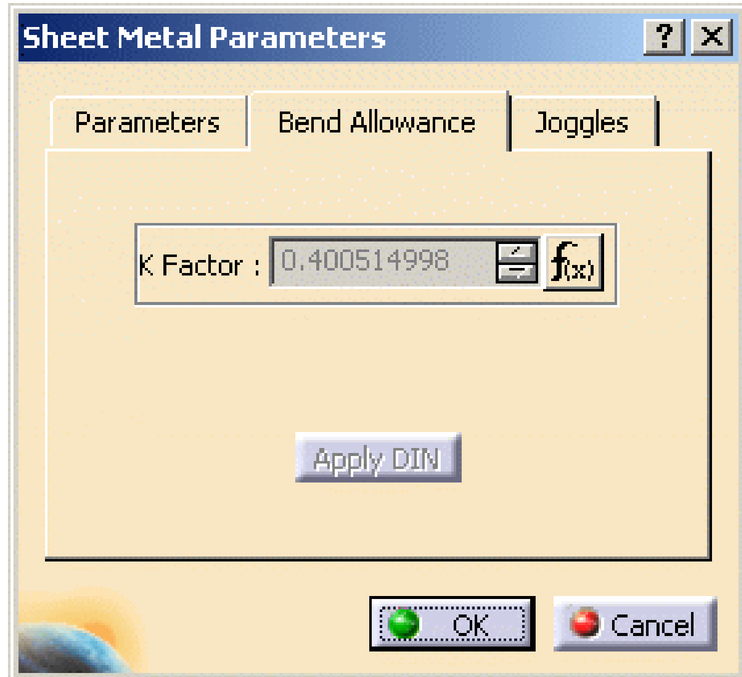
Computing the Bend Allowance

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

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

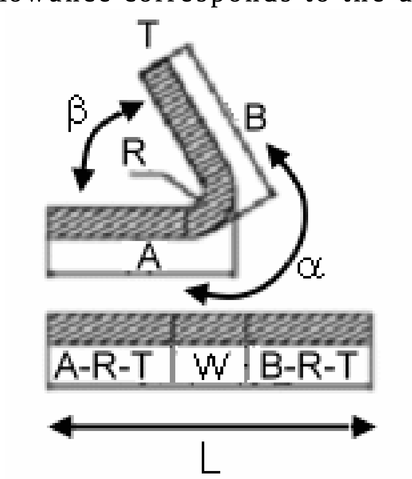
The Sheet Metal Parameters dialog box is displayed.

The fourth tab concerns the bend allowance.

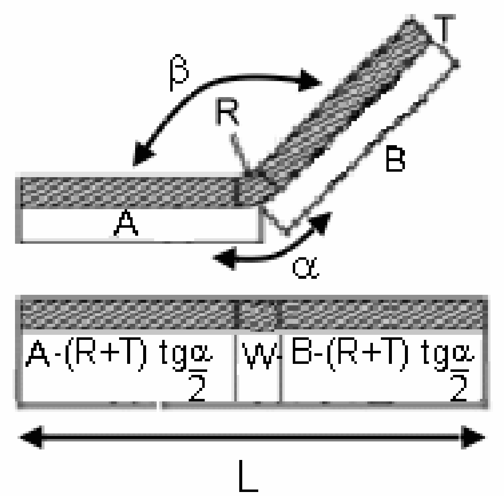


Bend Allowance

The bend allowance corresponds to the unfolded bend width.



bend < 90deg



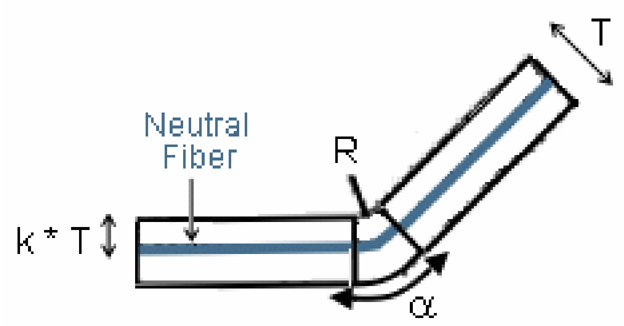
bend > 90deg

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:



W = α * (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:

α = π * (180 - β) / 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.

W = α * (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 = α * (R + k * T) - 2 * (R + T) * tan (min(π/2,α) / 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:

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

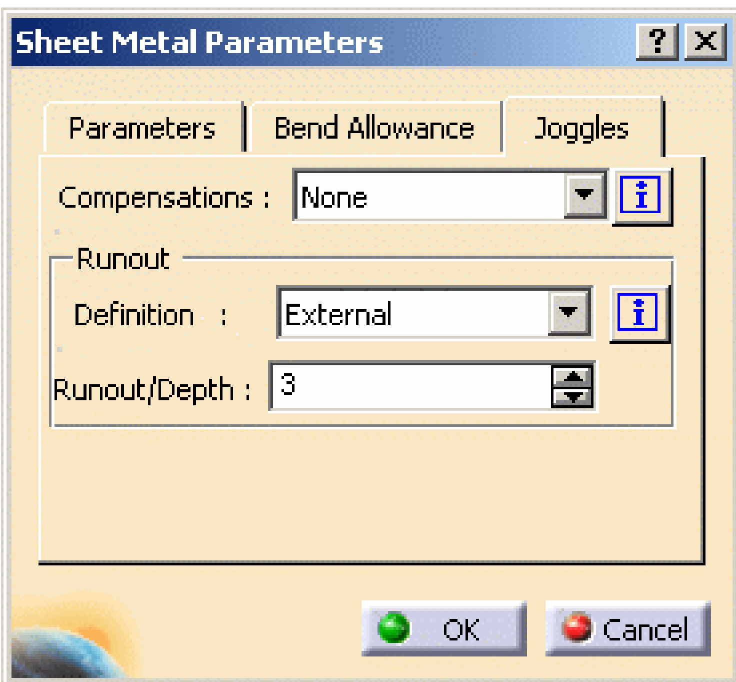
When the bend deduction is read in the bend table, the K factor is not used.

Defining the Compensations and Runout

This section shows how to select the appropriate method to define compensations when flattening a surfacic flange or a surfacic flange with a joggle.
Compensation is a modification of the standard calculation of the unfolding process which intends to best represent the material behavior.

You first need to define which method you want to apply by customizing design tables. To do so, proceed as explained in Customizing Standards Files To Define Methods for Compensations.

1. Click the **SheetMetal Parameters** icon . The Sheet Metal Parameters dialog box is displayed.



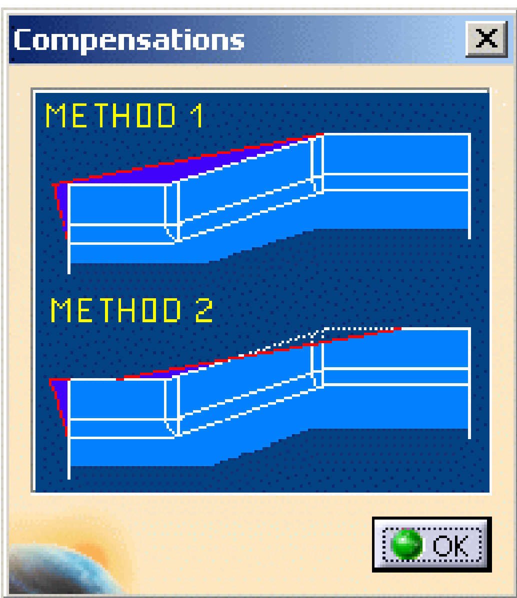
2. Click the **Joggles** tab.

3. In the **Compensations** combo list, select the method as defined in the SheetMetal Standard files:

- **None**: no compensation is applied
- **Method 1** (= Method V4)
- **Method 2**

If the method you chose is not the one defined in the SheetMetal Standard Files, a warning message is issued prompting you to select another method or the corresponding file.

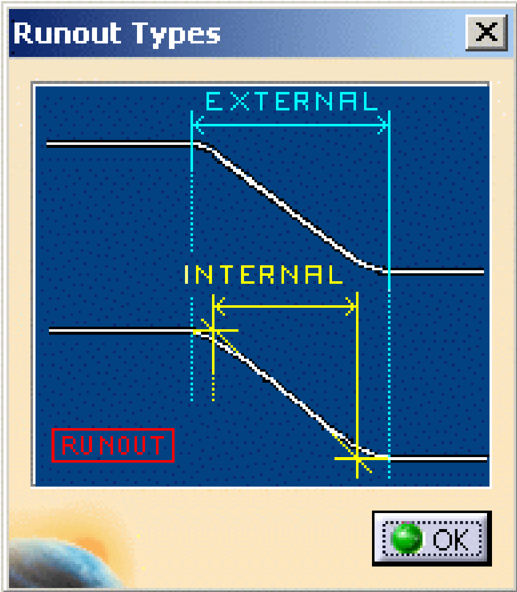
You can click the information icon to display a schema explaining both methods. More information is available in Customizing Standards Files To Define Methods for Compensations.



4. In the **Definition** combo list, select the runout definition:

- **External**: this type of runout includes the Joggle fillets.
- **Internal**: this type of runout excludes partially the Joggle fillets.

You can click the information icon to display a schema explaining both types of runout.



5. In the **Runout** section, modify the formula's coefficient by clicking the up and down arrows.

The Runout formula corresponds to the coefficient multiplied by the depth.
If you create a new part, you can modify the formula's coefficient.

If you work on a part created in a previous version, when you create a joggle a warning is displayed asking you to validate this parameter in the Sheetmetal Parameters dialog box.

6. Click OK in the dialog box to validate the compensations and runout parameters.

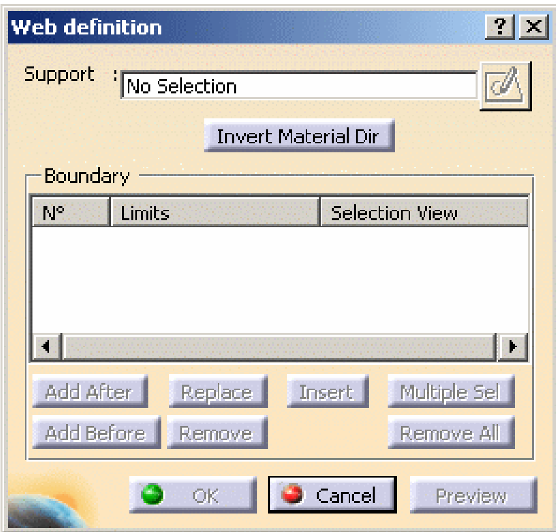
Creating a Web

This section explains how to create a web, that is the main feature of an Aerospace SheetMetal Design part.

Open the Web1.CATPart document.

1. Click the **Web** icon .

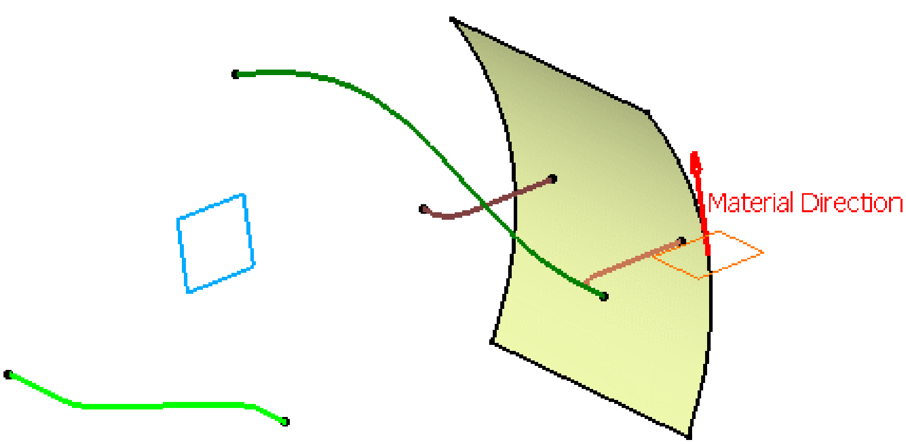
The Web definition dialog box is displayed.



2. In the **Support** field, select the support geometry in the specification tree. It can either be:

- a plane (example from the *Web from open geometry* geometrical set).

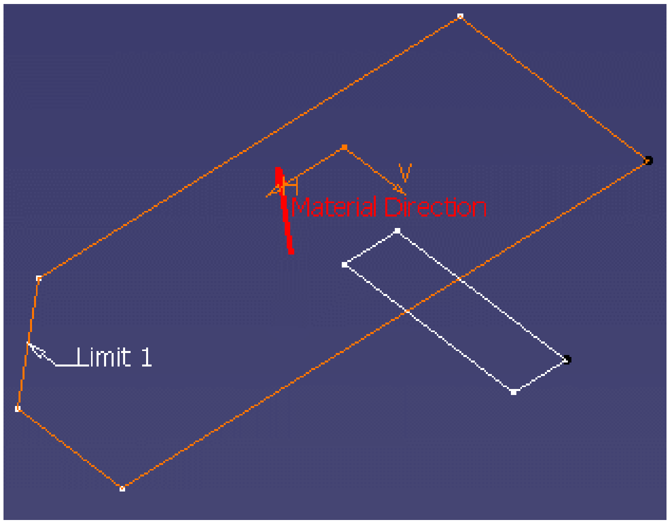
The Material Direction is displayed, perpendicular to the geometrical support. You can reverse the direction by clicking the arrow.



- a closed sketch (example from the *Web from closed sketches* geometrical set).

In our example, there are two closed sketches: the web will be calculated on the smallest part of the second sketch.

The Material Direction is displayed, perpendicular to the geometrical support. You can reverse the direction by clicking the arrow.



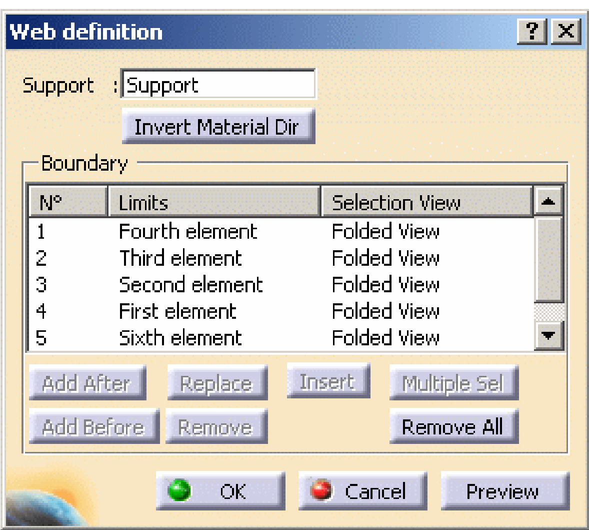
You can click the **Invert Material Dir** button to reverse the material direction of the web.

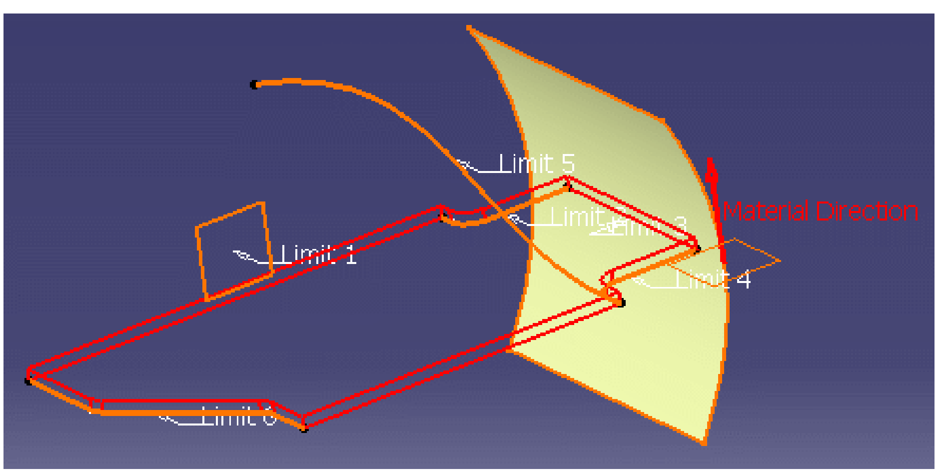
3. In the Boundary field, in the case of an open geometry, select the elements that limit the support geometry. It can either be:

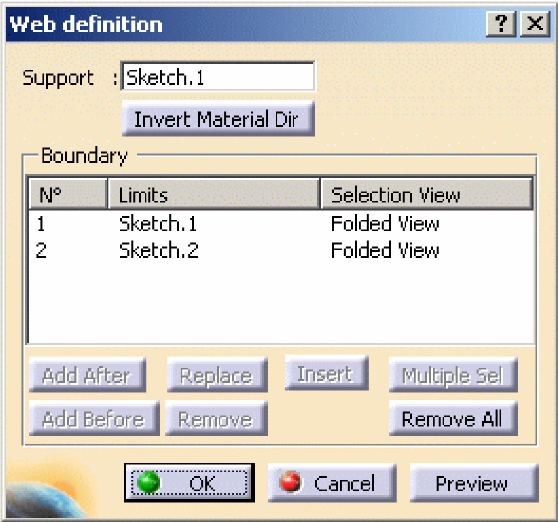
- a list of elements (curves, surfaces, or planes)
- one or more sketches

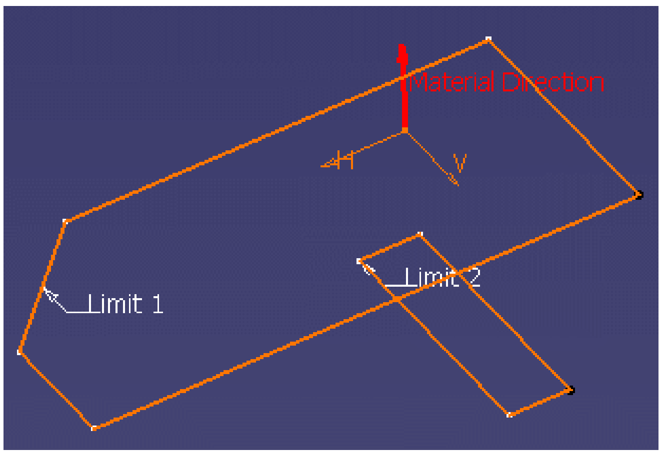
The elements must be selected consecutively.

They are displayed in the Boundary frame, in the order you have just chosen them, as well as in the 3D geometry.







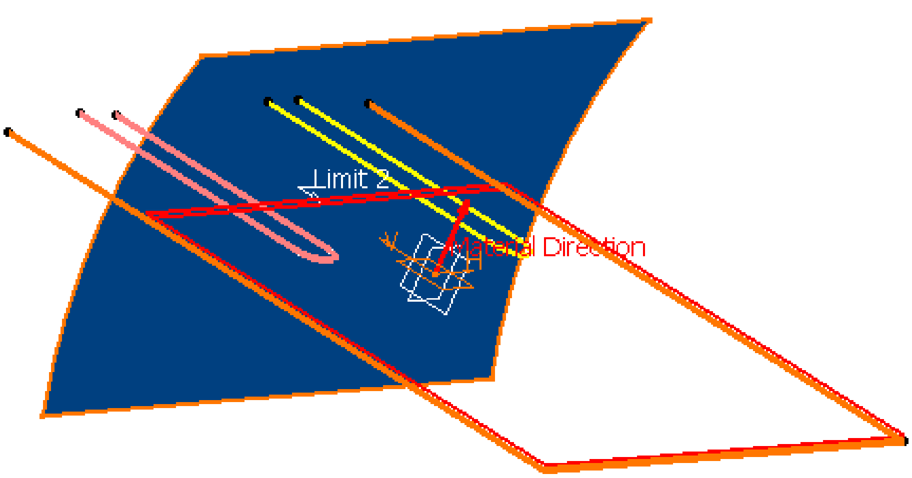


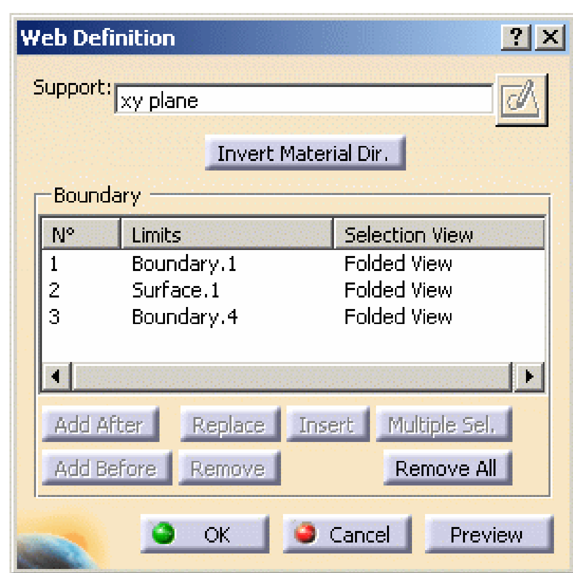
When a closed profile can be built, a light preview of the web is available. Otherwise, click **Preview**.

You can modify the selection by selecting an existing limit and using the following buttons to:

- add a limit after the selected limit (**Add After**)
- add a limit before the selected limit (**Add Before**)
- replace a limit (**Replace**)
- remove a limit (**Remove**)
- select a limit more than one time (**Insert**)

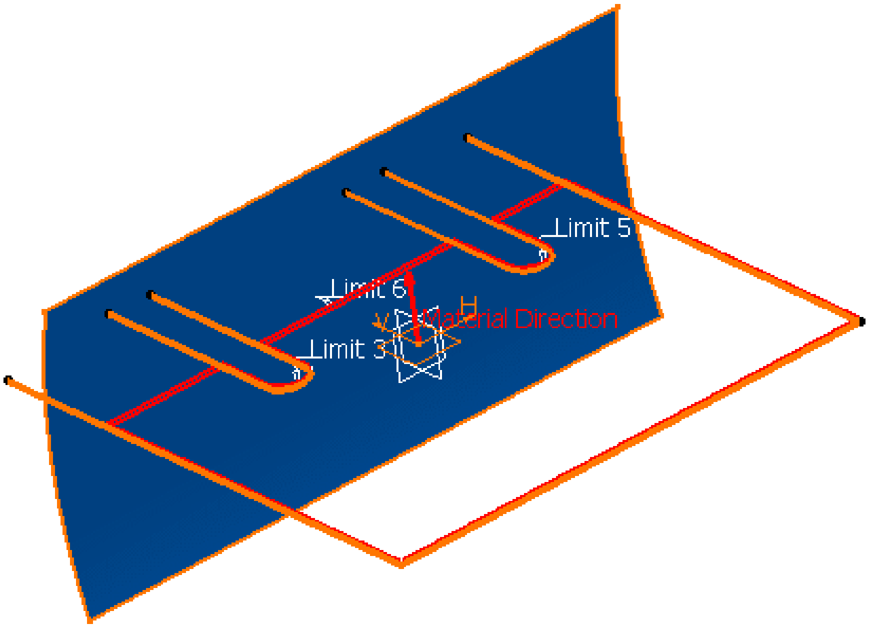
In the following example, we first select three elements to limit the web.

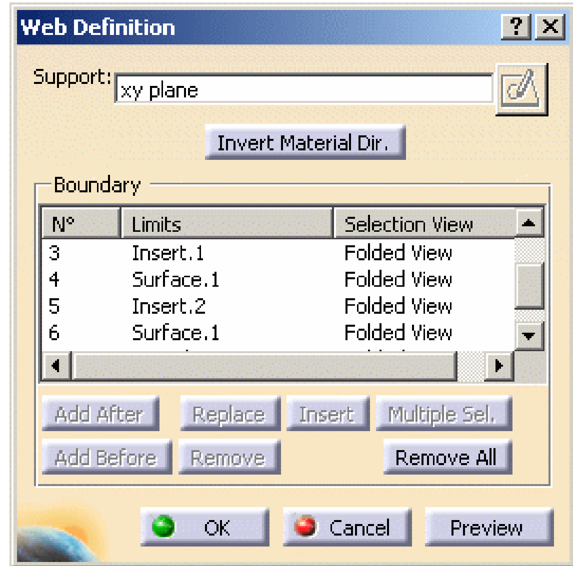




Then we select Surface.1 and click on the pink sketch to add it to the web's definition.

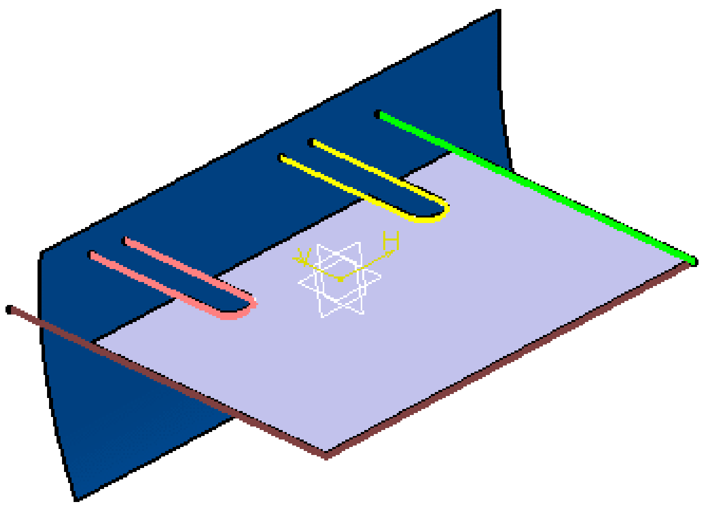
Then we select Surface.1 again and click on the yellow sketch to add it to the web's definition.





Note that Surface.1 is displayed twice in the Web Definition dialog box.

The Web is created according to several elements.



- select several limits to modify the existing limit (**Multiple Sel**). This option is available once you have selected **Add After**: the Limits to Add dialog box appears to let you select the limits.
- remove all limits (**Remove All**)

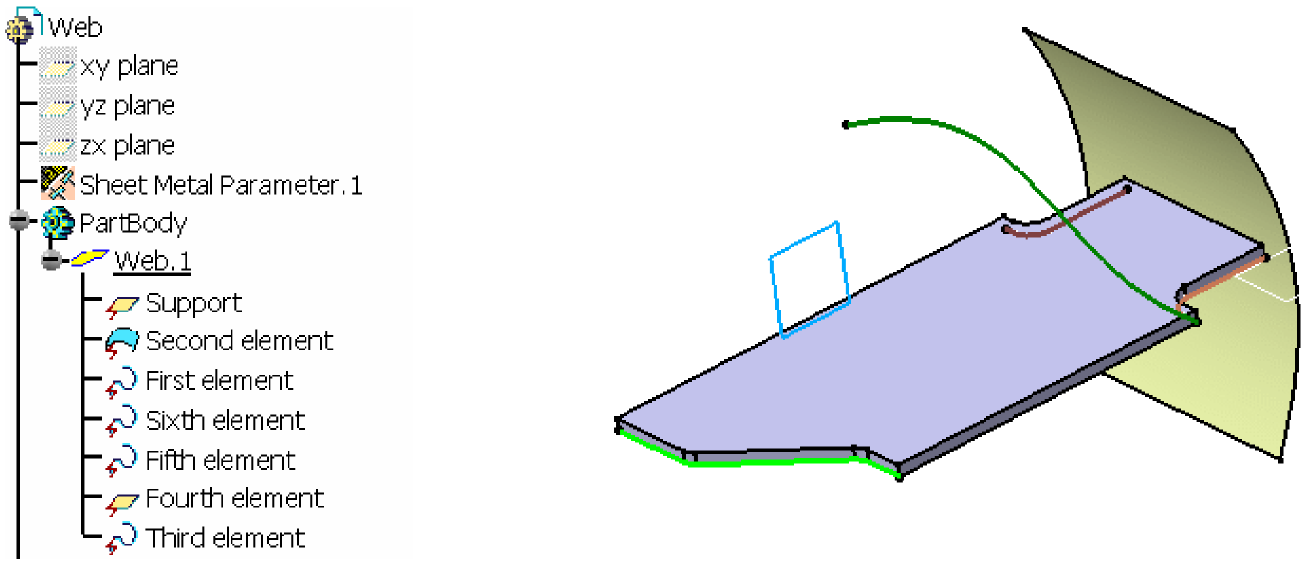
Once you have modified the selection, a light preview is available. You can click the Preview button to display the result of the web.

When the profile is defined by a list of geometrical elements, the following operations are performed:

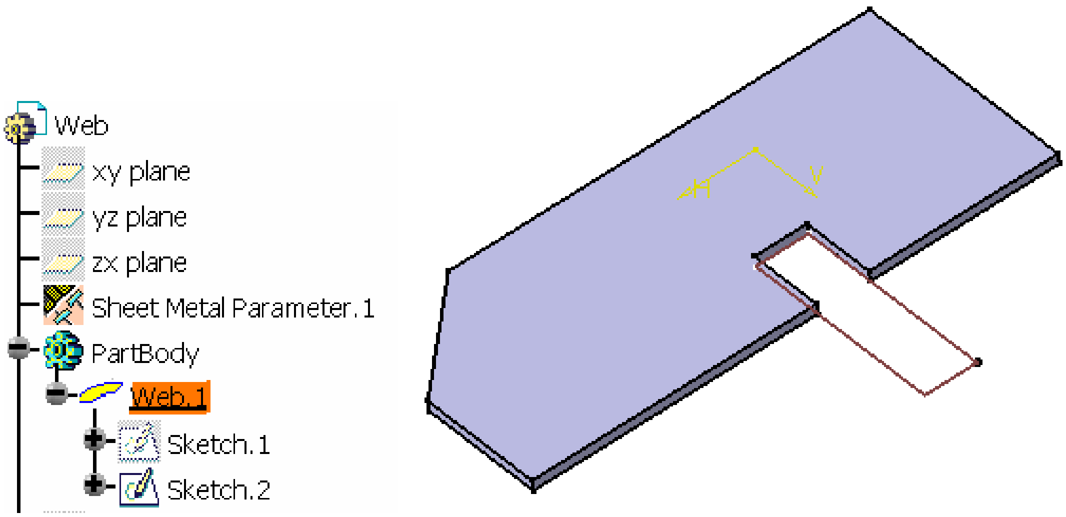
- the curves are projected on the web geometrical support
- the surfaces are intersected with the web geometrical support

4. Click OK.

The web (identified as Web.1) is created and the specification tree is updated accordingly.

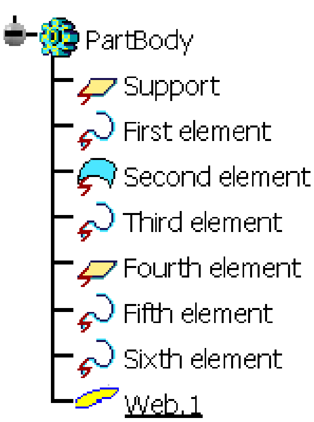


Features are displayed when editing Web.1's panel.

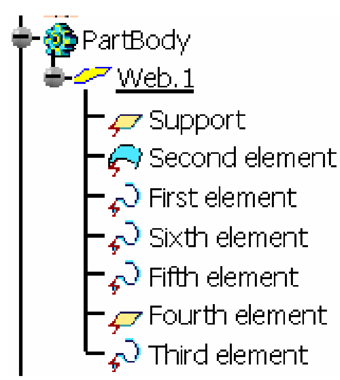


The sketches are aggregated under the web (identified as Web.xxx). Sketch.1 is displayed in No Show mode as it was only used to create the web.

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



Specification tree's behavior in hybrid context.



Specification tree's behavior in pre-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.

Creating a Surfacic Flange

This section explains how to create a surfacic flange on a web, or an existing surfacic flange (in that case, their fillets must not intersect).

Understanding the surfacic flange

When creating a surfacic flange, the bend is propagated along the whole base feature (with a continuity in tangency).

In certain cases this propagation prevents the surfacic flange from being relimited: it happens when the selected edge allows propagation of the bend.

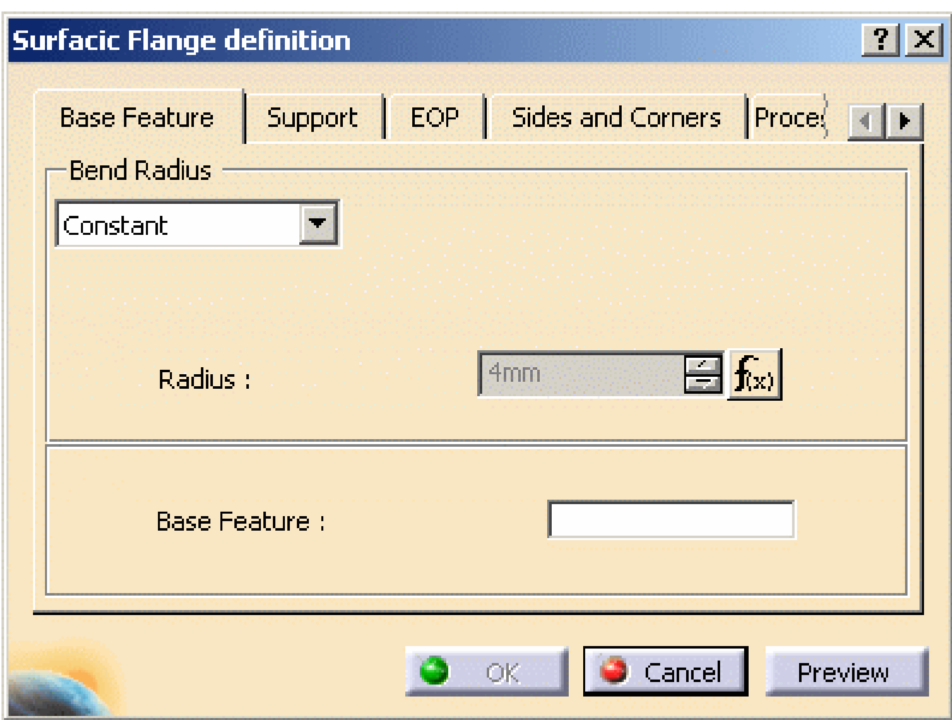
Open the SurfacicFlange1.CATPart document. Create a web as shown in the previous task.

Here are the different elements taken into account when creating a surfacic flange:

- Base Feature
- Support
- EOP
- Sides and Corners
- Process
- Compensations

1. Click the **Surfacic Flange** icon .

The Surfacic Flange definition dialog box is displayed.



Base Feature

In the Base Feature tab, the Bend Radius is of **Constant** type. It is set to the default bend radius of the part.

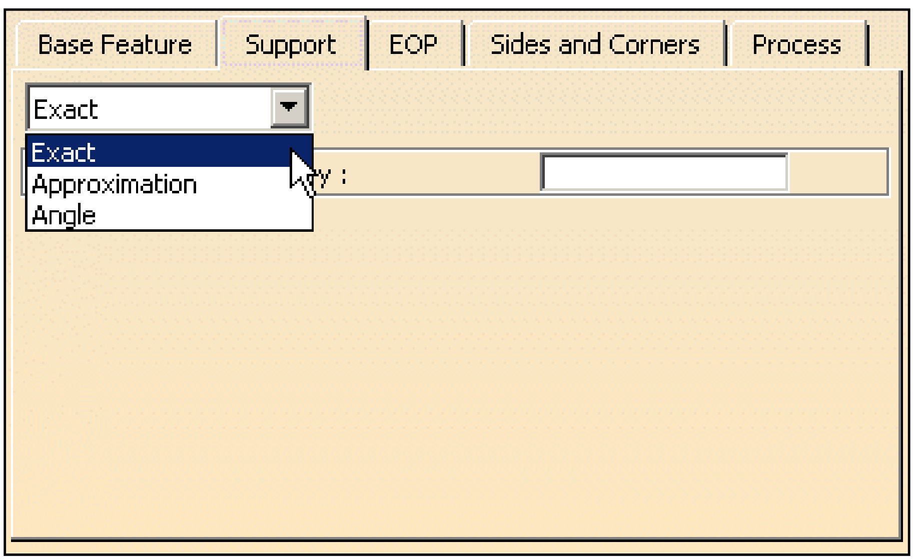
2. You can modify the fillet Radius value by changing the driving equation: click the  icon.
   The Formula Editor dialog box opens, you can modify the dictionary and the parameters.

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

3. Choose the web as the Base Feature.

Once you chose the base feature, the Support tab automatically displays.

Support

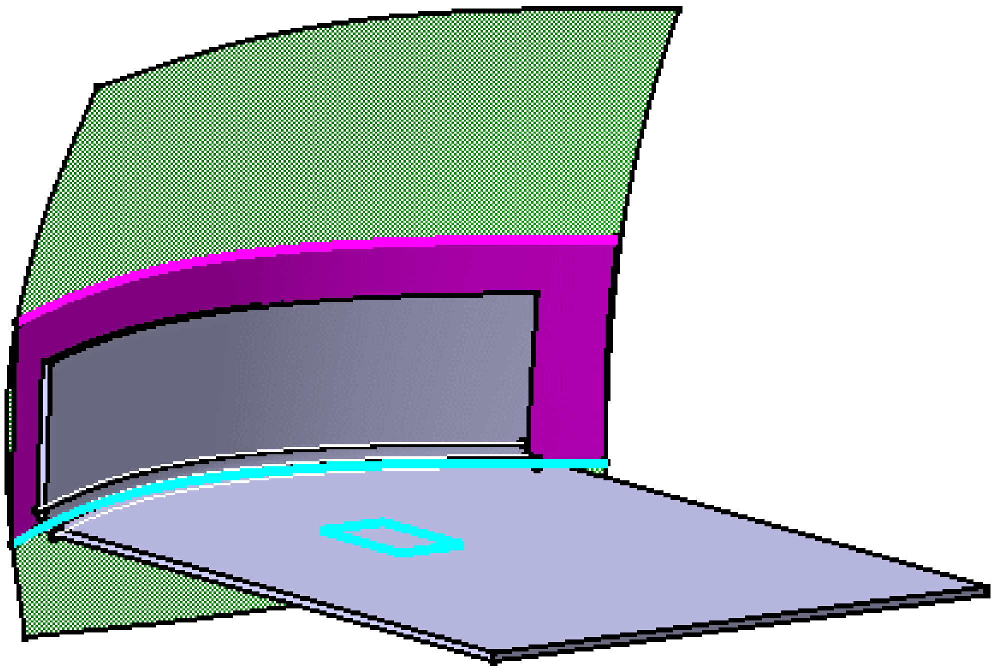


4. In the Support tab, choose the surfacic flange's geometrical support. It can either be a surface, a plane or a curve.

Make sure the support is big enough to be able to later define an EOP with a length from OML.
The OML is a curve created by intersecting the flange support and a plane perpendicular to the web and normal to the OML.

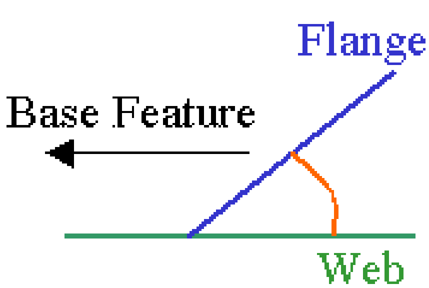
- Exact: the selected support is to be used for the creation of the surfacic flange.
- Approximation: the support surface is approximated using a ruled surface. This ruled surface is defined from two curves:
  - the OML (in light blue), computed at the intersection between the support surface and the web plane.
  - a curve parallel to the OML (in pink), computed at a distance equal to the approximation length

  This mode enables you to compute the maximum deviation between the support geometry and the approximated surface.



- Angle: the support of the surfacic flange can also be defined by a line, a curve, an angle or the edge of a base feature. The angle is constant and you can change its value using the spinners.

You can modify the Support Length generated by the curve and the angle. By default, the length is set to ten times the EOP (Edge of Part) length. If the default EOP length is higher than 100mm, you need to modify the surface length.

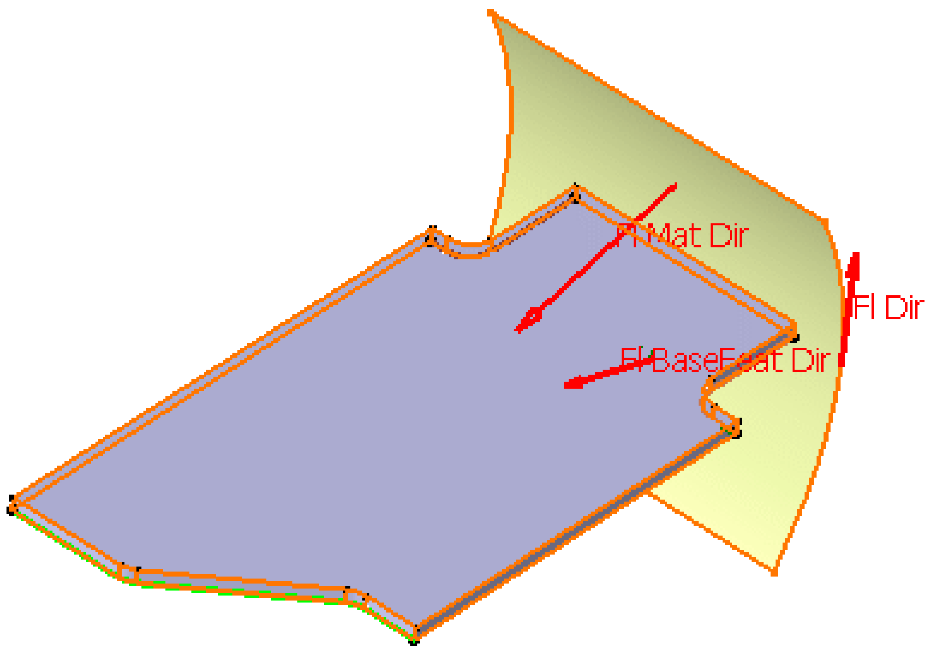


The red angle is the angle taken into account when creating the surfacic flange.

5. Define the vectors' directions.

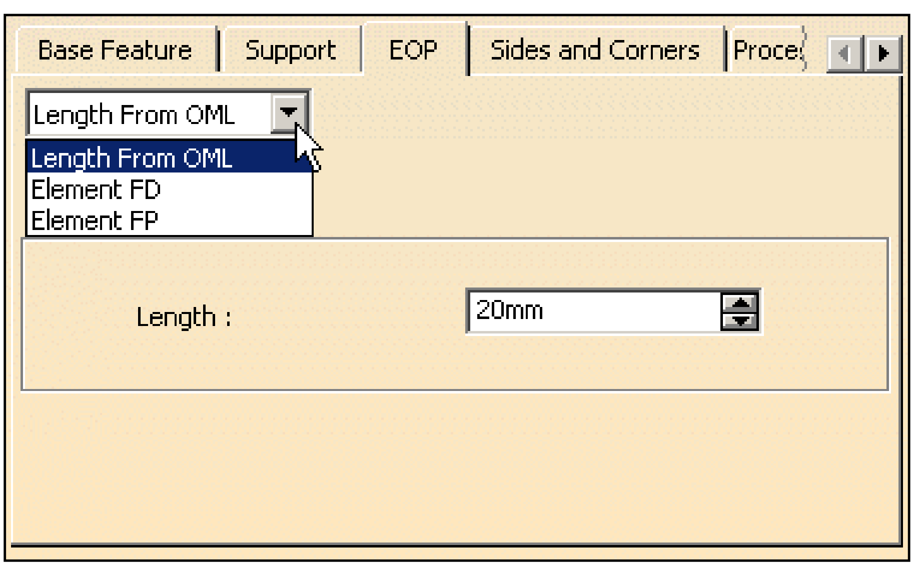
The vectors show the Base Feature Direction, the Direction and the Material Direction according to the direction of the geometrical support of the surfacic flange.

You can modify the directions by clicking the arrows.



The surfaces (or curves) used to define the support surface must be continuous in point and tangency.

EOP



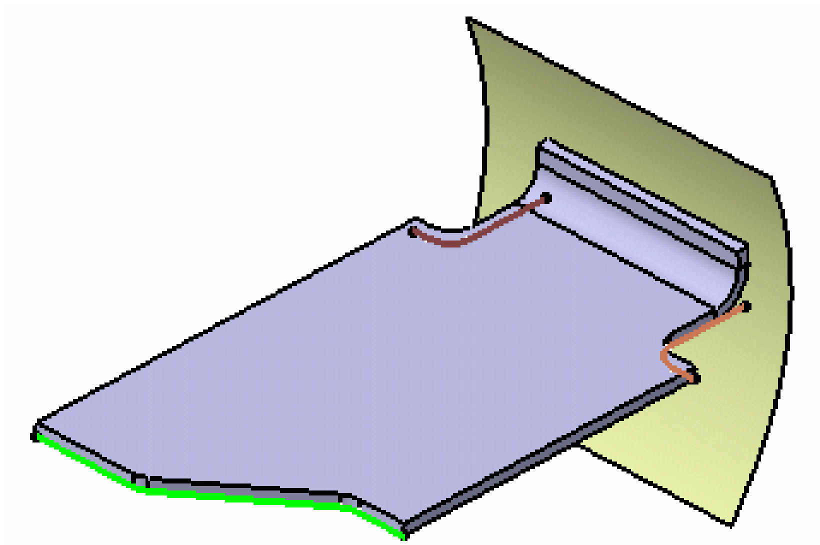
6. In the EOP (Edge Of Part) tab, you can define either:

- a length from OML (Outer Mold Line): length between the curve defining the top of the surfacic flange and the OML,
- an element FD (Folded): boundary element (either a surface that intersects with its surface, or a sketch, or a wire projected on its surface),
- an element FP (Flat Pattern): curve or sketch defining the flattened profile of the surfacic flange.

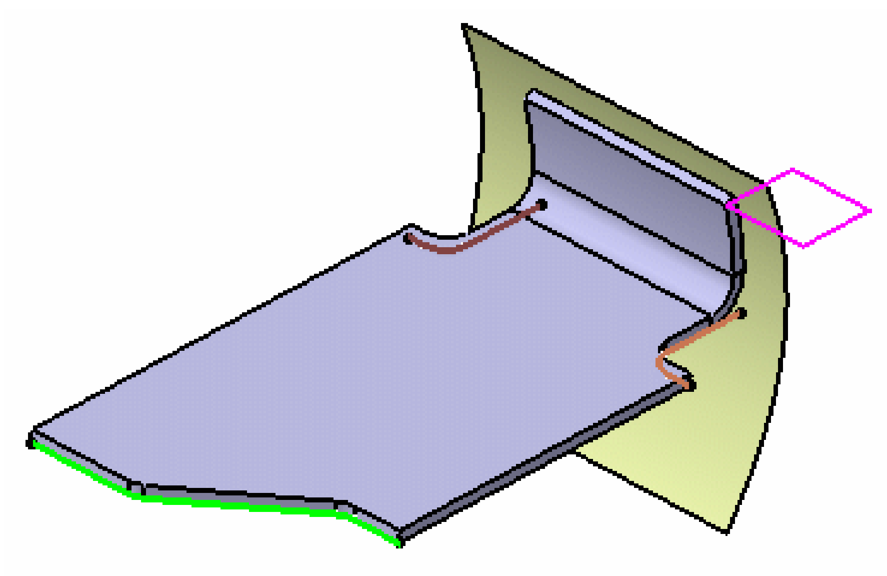
The element FP **must** be included within the limits of the surfacic flange support when folded.

7. Click OK.

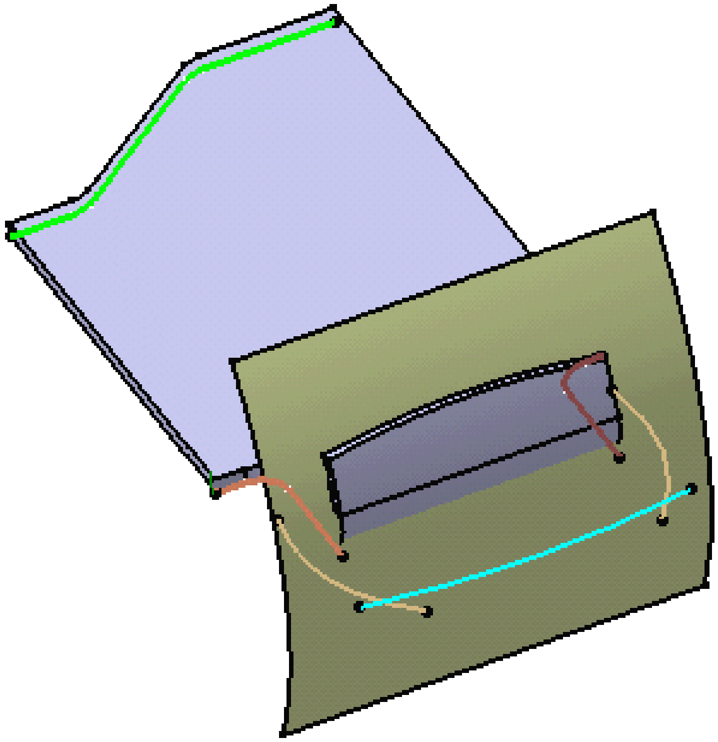
The Surfacic Flange (identified as Surfacic Flange.xxx) is created and the specification tree is updated accordingly.



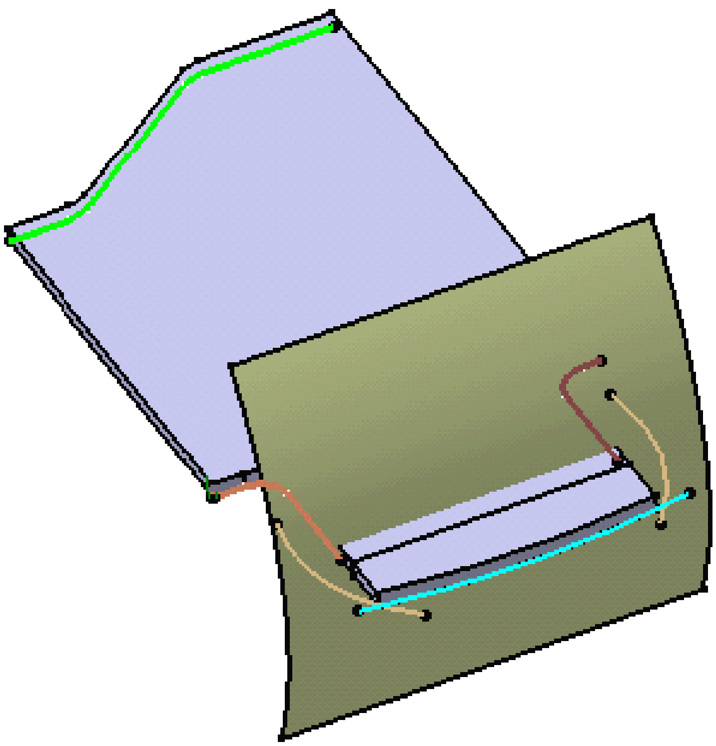
Surfacic Flange with a length from OML of 15 mm



Surfacic Flange with Plane.2 (in pink) as Element FD

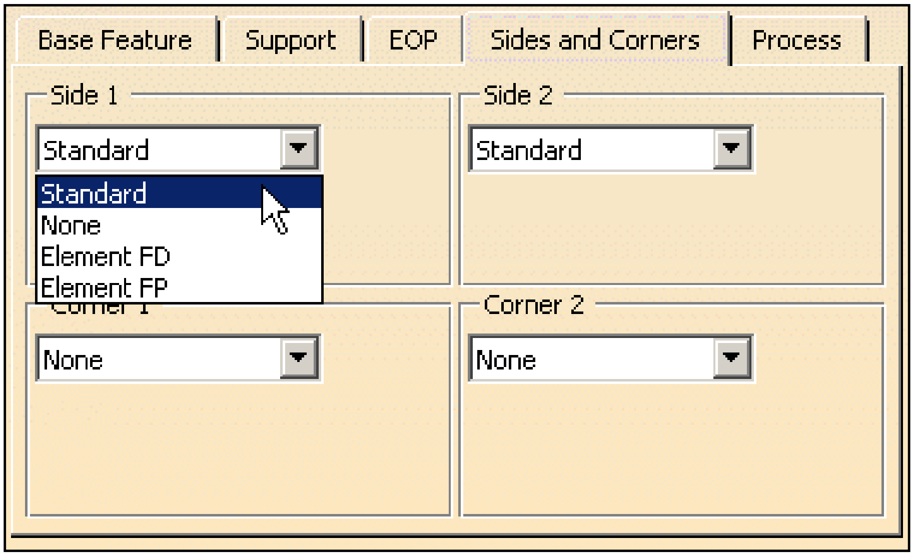


Surfacic Flange with EOP FP (in light blue) as Element FP



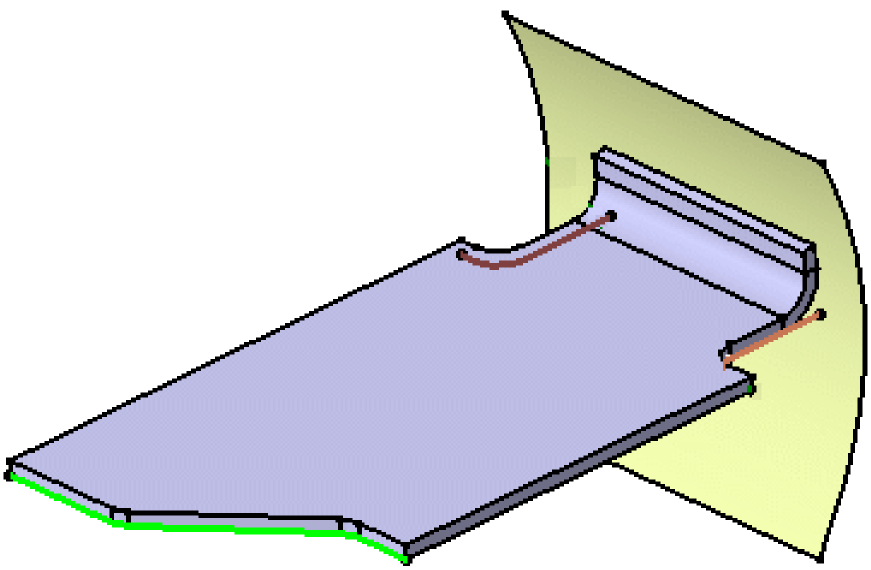
Unfolded view of the Surfacic Flange with EOP FP as Element FP. See Unfolding.

Sides and Corners

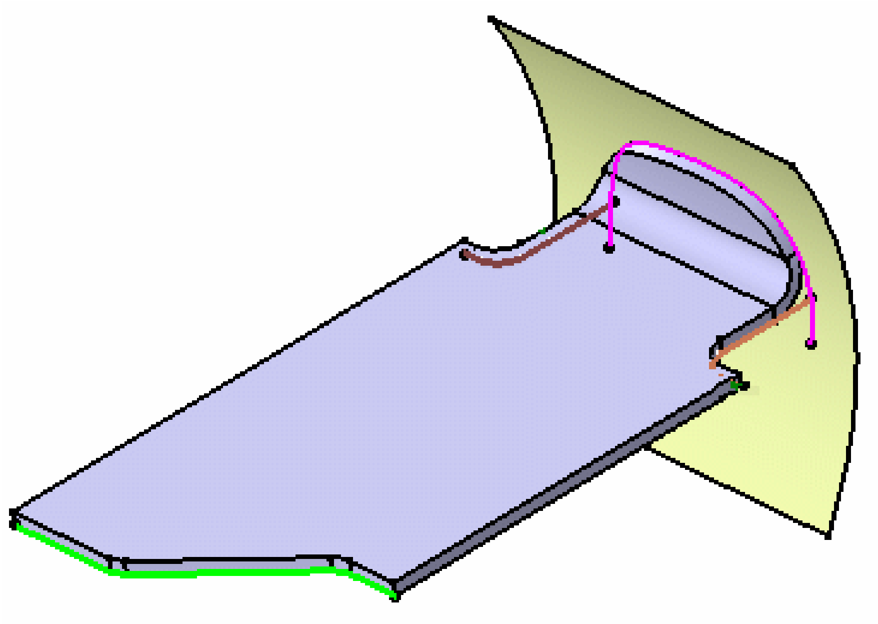


In the Sides and Corners tab, you can choose to define the following elements:

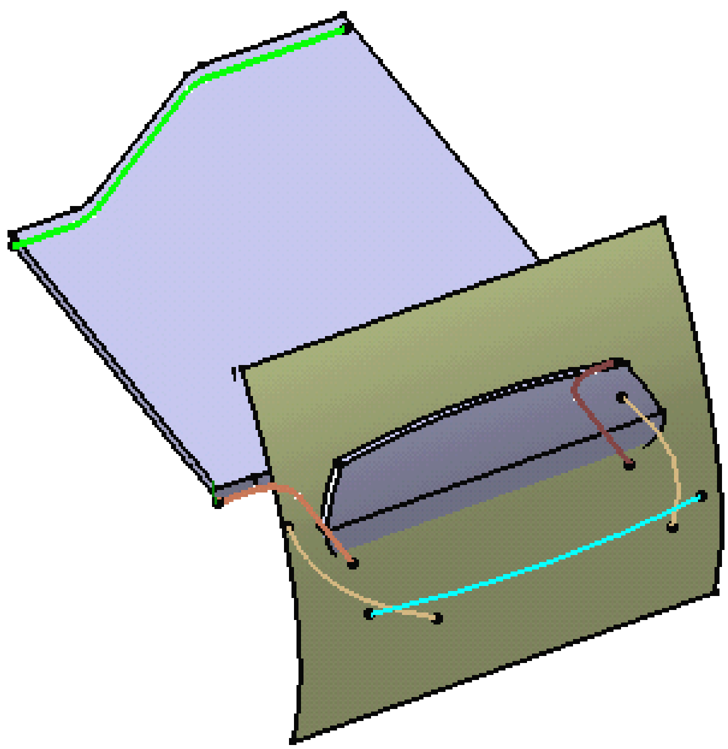
- **sides** (intersection between the Base Feature and a curve) as:
  - standard: they are automatically defined at the web limit and the perpendicular plans are kept (in this case, the user does not have to define them)
  - none: no side computed (only the EOP will define the profile of the Flange)
  - element FD (Folded): they are defined by a folded geometrical element (curve, plane or surface).
  - element FP (Flat Pattern): curve defining the flattened profile of the flange.



Surfacic Flange defined with a Length from OML of 10mm, and Side 1 and 2 as Standard

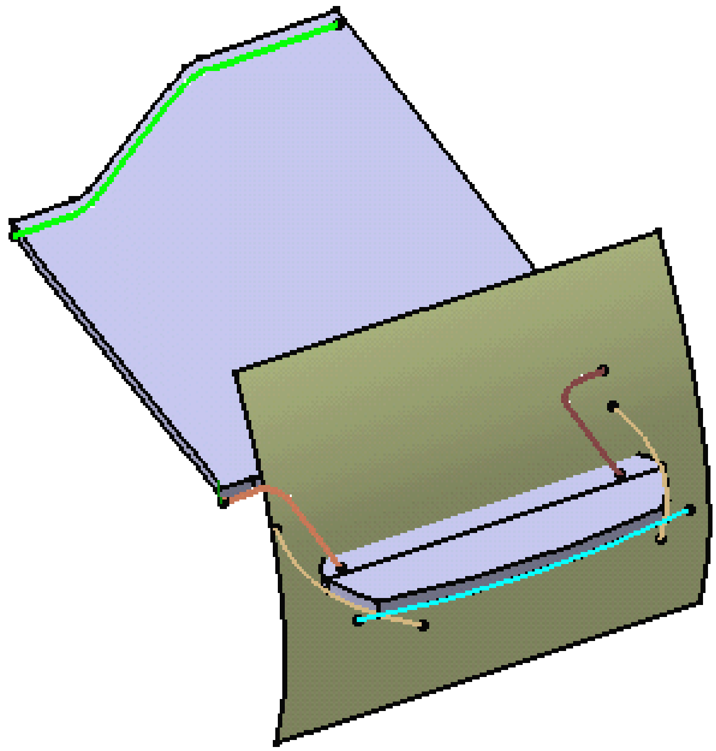


Surfacic Flange defined with EOP FD as Element FD, and Side 1 and 2 as None



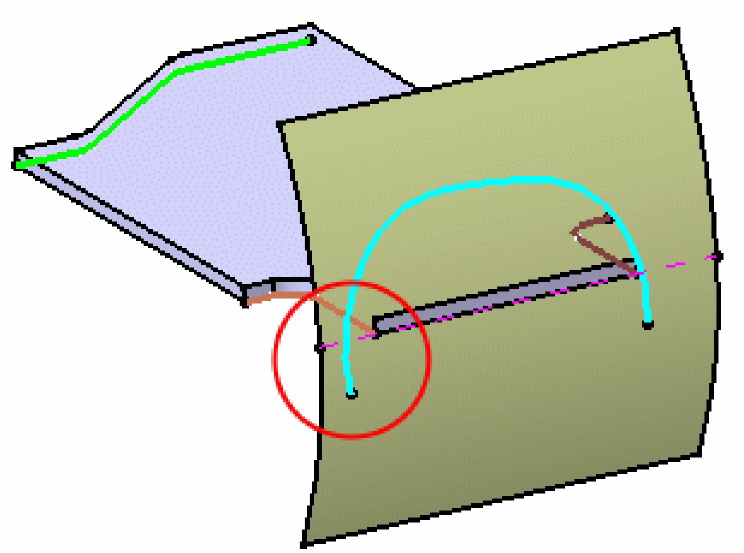
Folded view of the Surfacic Flange with EOP FP as Element FP, Side 1 FP and Side 2 FP (in light brown) as Side 1 and Side 2.

See Unfolding.



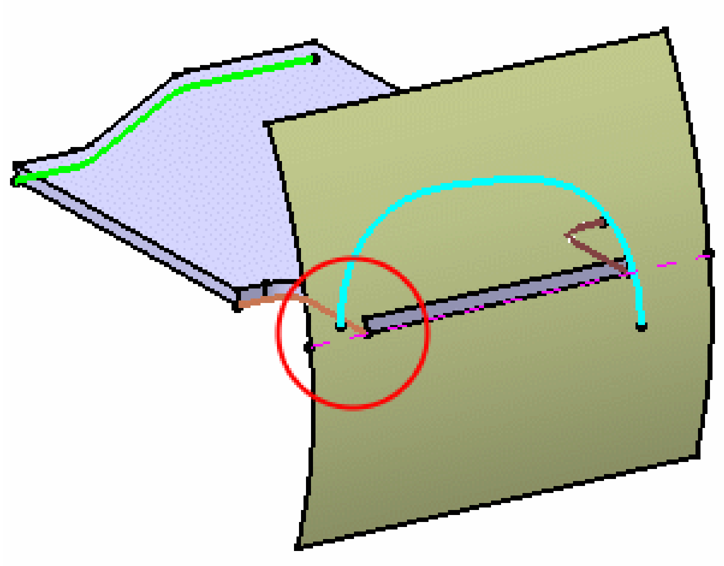
Unfolded view of the Surfacic Flange with EOP FP as Element FP, Side 1 FP and Side 2 FP (in light brown) as Side 1 and Side 2.
See Unfolding.

a. The following examples show two cases of a flange defined by an EOP FP or FD and Sides as None.



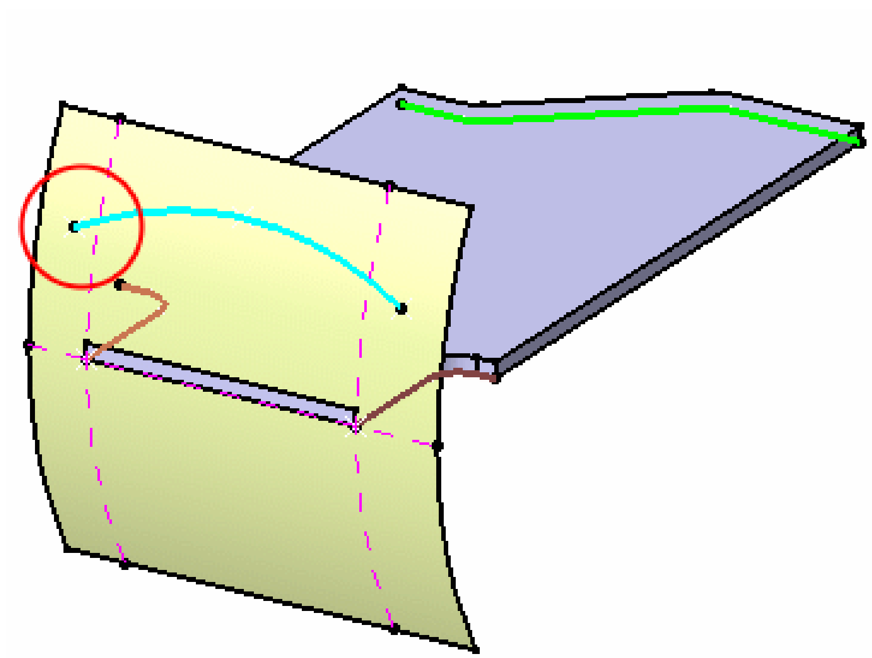
There is an intersection between the EOP and the web support -> the Surfacic Flange can be computed

There is no intersection between

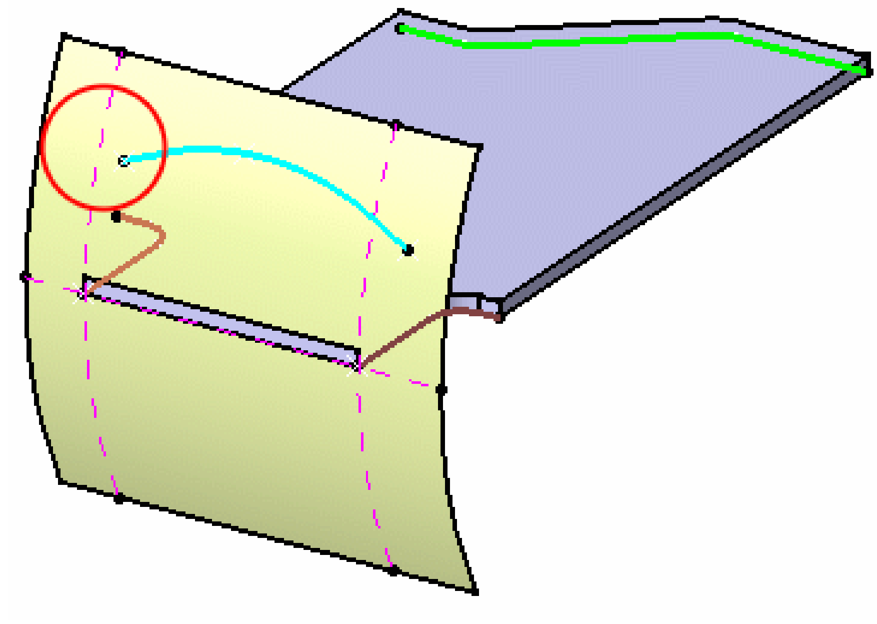


the EOP and the web support -> the Surfacic Flange cannot be computed

b. The following examples show two cases of a flange defined by an element FD as the EOP and Standard sides.

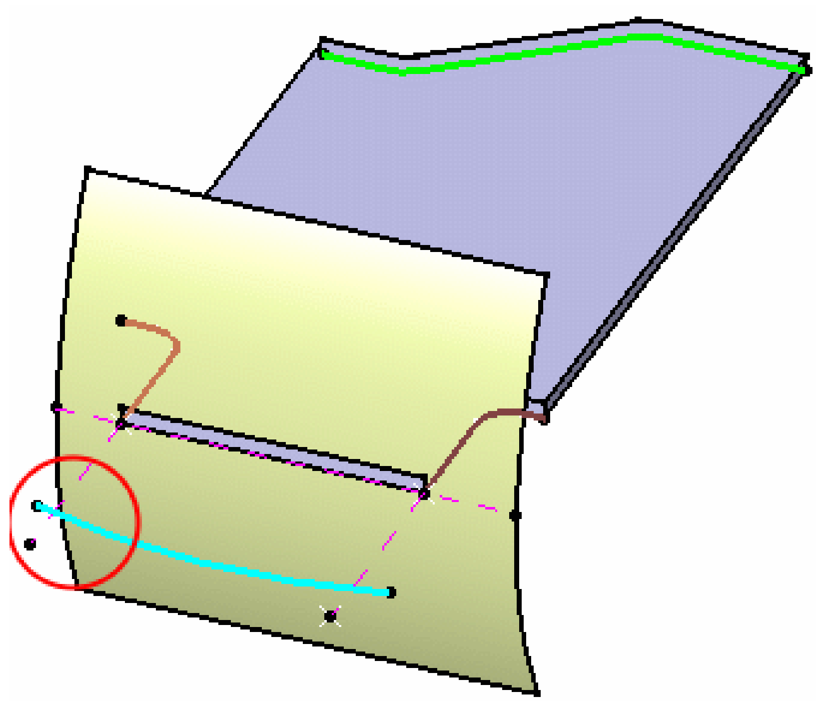


There is an intersection between the EOP and the side -> the Surfacic Flange can be computed

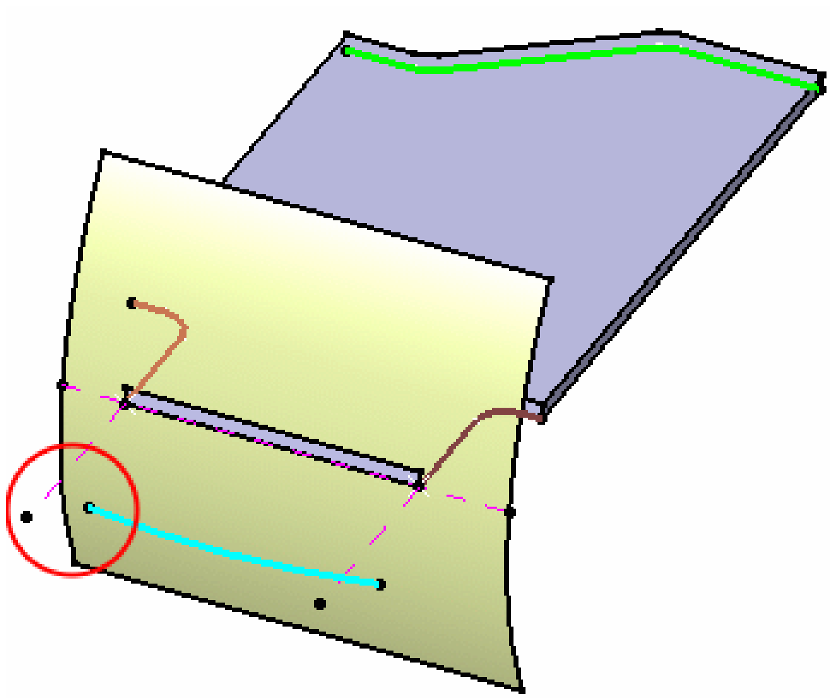


There is no intersection between the EOP and the side -> the Surfacic Flange cannot be computed

c. The following examples show two cases of a flange defined by an element FP as the EOP and Standard sides.

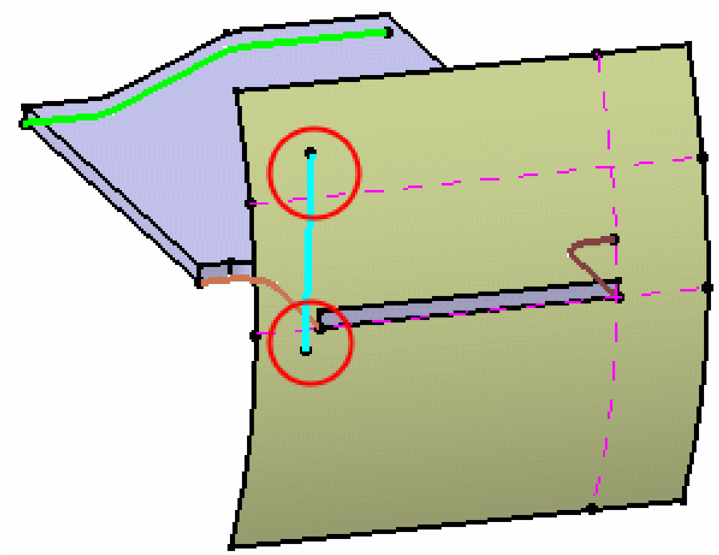


There is an intersection between the OML and the EOP -> the Surfacic Flange can be computed

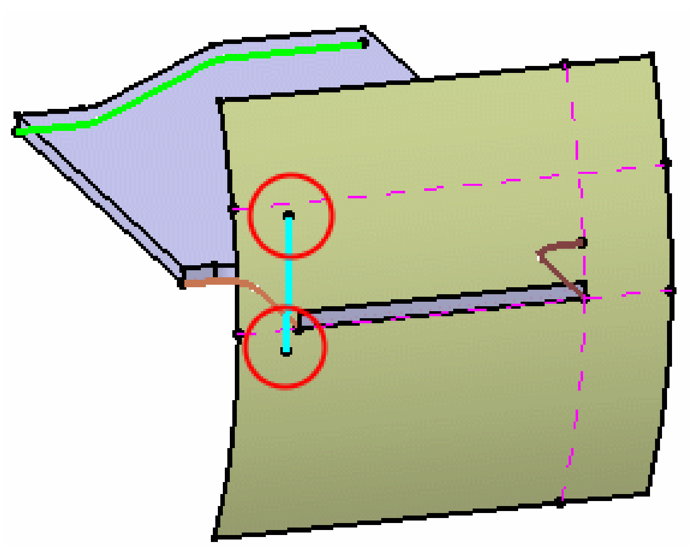


There is no intersection between the OML and the EOP -> the Surfacic Flange cannot be computed

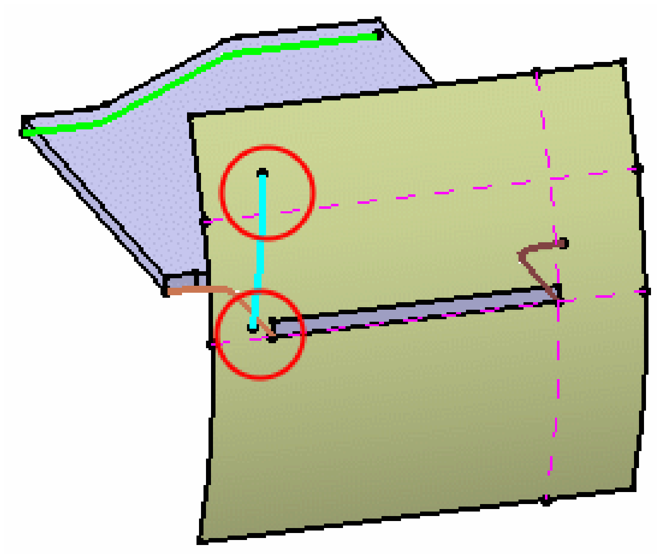
d. The following examples show three cases of a flange defined by an element FD as the side.



There is an intersection between the Element FD and the OML and between the side and the EOP -> the Surfacic Flange can be computed

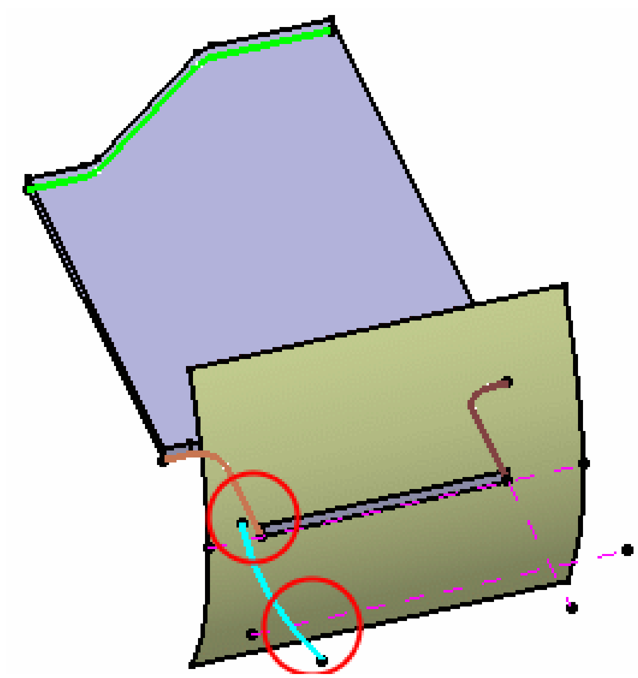


There is an intersection between the side and the OML but no intersection between the side and the EOP -> the Surfacic Flange cannot be computed



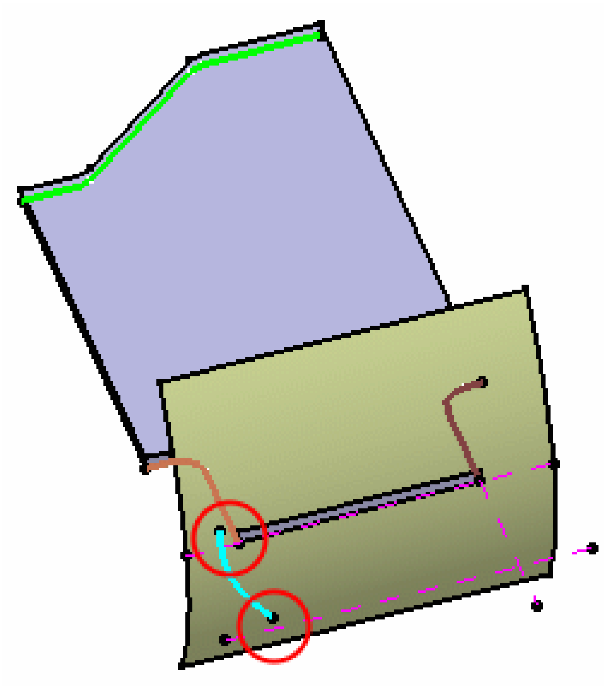
There is an intersection between the side and the EOP but no intersection between the side and the OML -> the Surfacic Flange cannot be computed

e. The following examples show three cases of a flange defined by an element FD as the side.

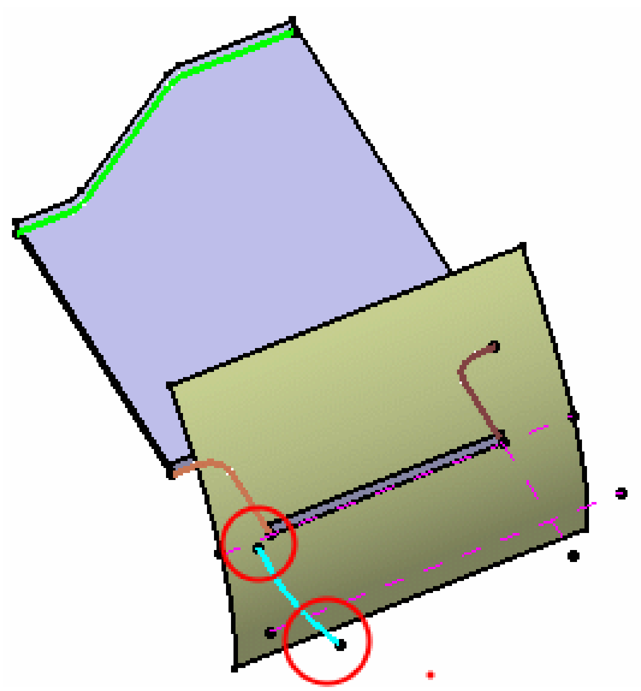


There is an intersection between the side and the EOP and between the side and the OML -> the Surfacic Flange can be computed

There is an intersection between

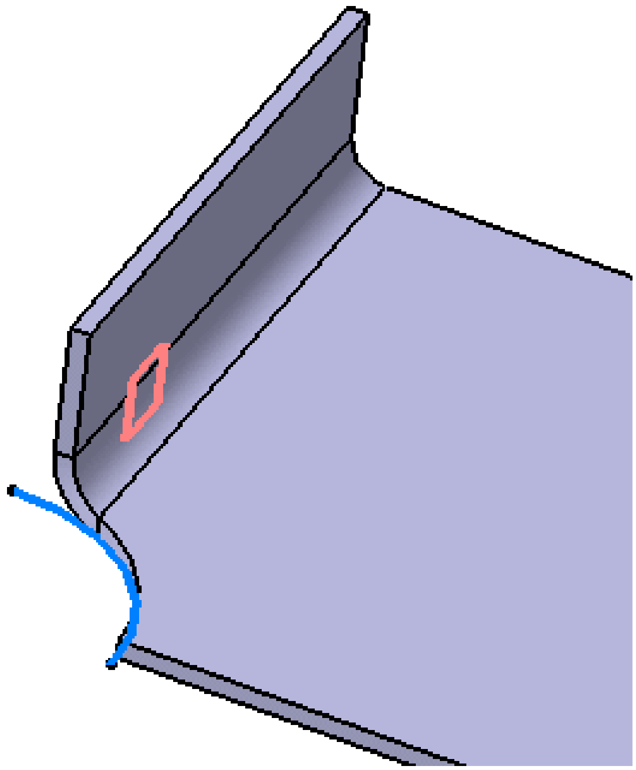


the side and the OML but no intersection between the side and the EOP -> the Surfacic Flange cannot be computed



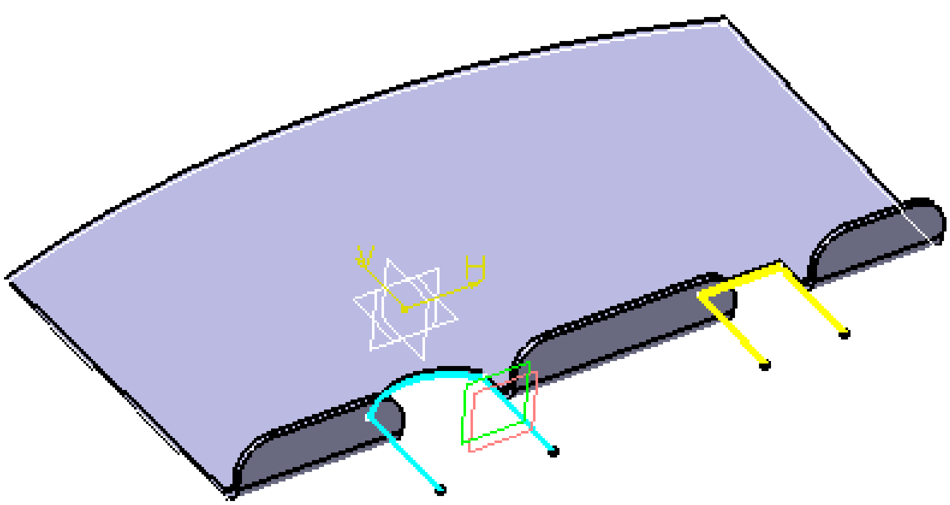
There is an intersection between the side and the EOP but no intersection between the side and the OML -> the Surfacic Flange cannot be computed

Any cutout on the web will be taken into account to create the surfacic flange's sides. For example, if you create a cutout on the web, then create a surfacic flange with standard sides, the latter will be calculated from the web's profile including the cutout.
For optimization reasons, we advise you to first create a sketch with the desired shape, then create the surfacic flange.



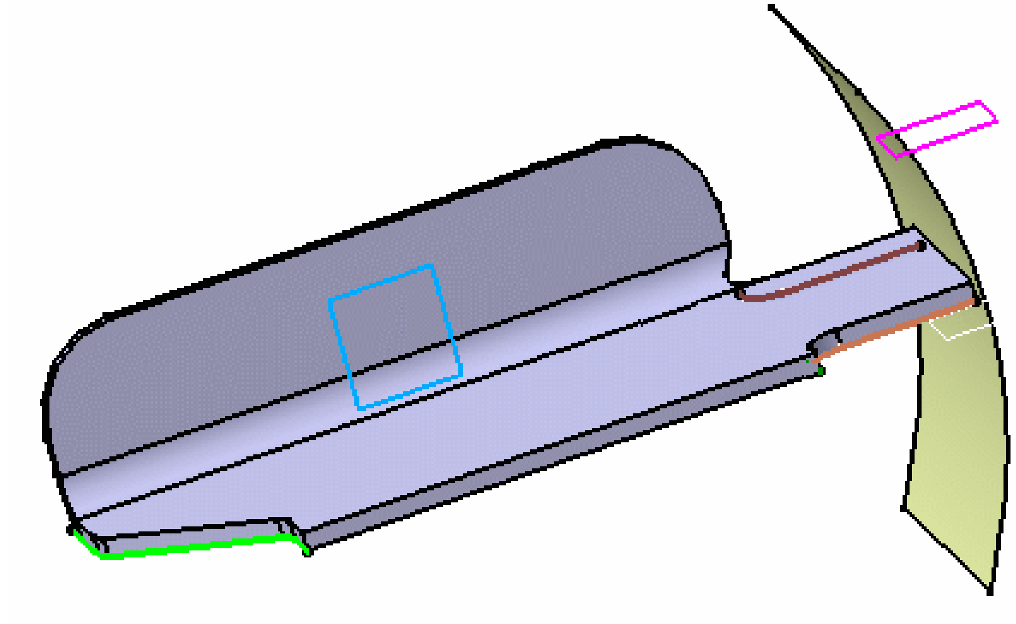
The example above shows a Surfacic Flange with standard sides calculated from the web modified by a cutout (the cutout, shown in blue, is taken into account)

You can create several surfacic flanges on a web already containing cutouts by selecting the web and the same support plane.



In such a case, each surfacic flanges can be opened and modified independently from the other surfacic flanges. Yet, if the support of one of the surfacic flanges was to be modified, the change would be propagated to the other surfacic flanges.

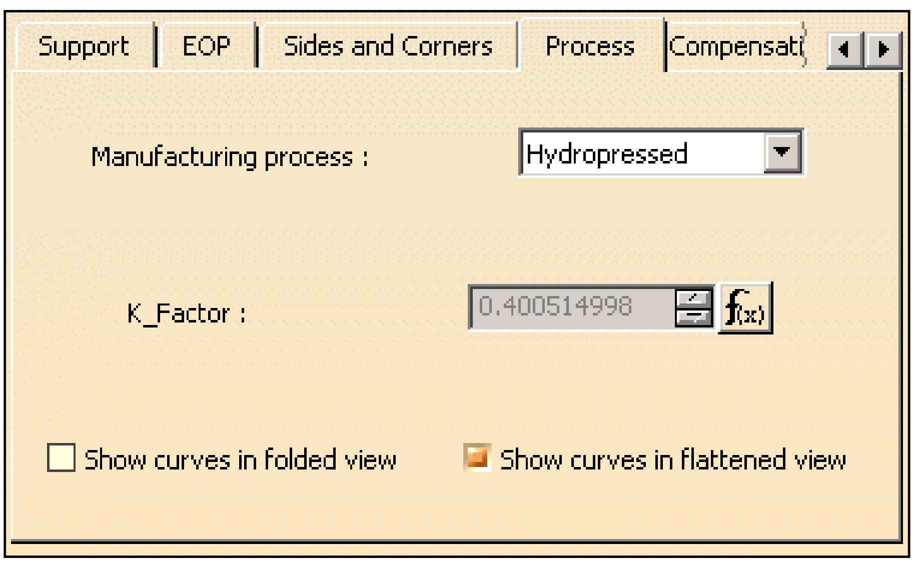
- **corners** (profile defined between the EOP and the sides) as:
  - none: no corner computed (only the EOP is able to define the profile of the Surfacic Flange)
  - corner: between the side and the EOP (defined with a radius value)



The example above shows a Surfacic Flange defined with Sides 1 and 2 as Standard, and Corners 1 and 2 of 10mm each.

- In the case the user does not define a surfacic flange side, the latter is automatically computed at the Web limit, perpendicular to the OML.
- In the case no corner is defined, the side and the EOP are simply relimiting each other.
- The sides of the fillet are continuous in tangency with the profile of the web and the sides of the surfacic flange.

Process



In the Process tab, you can define the:

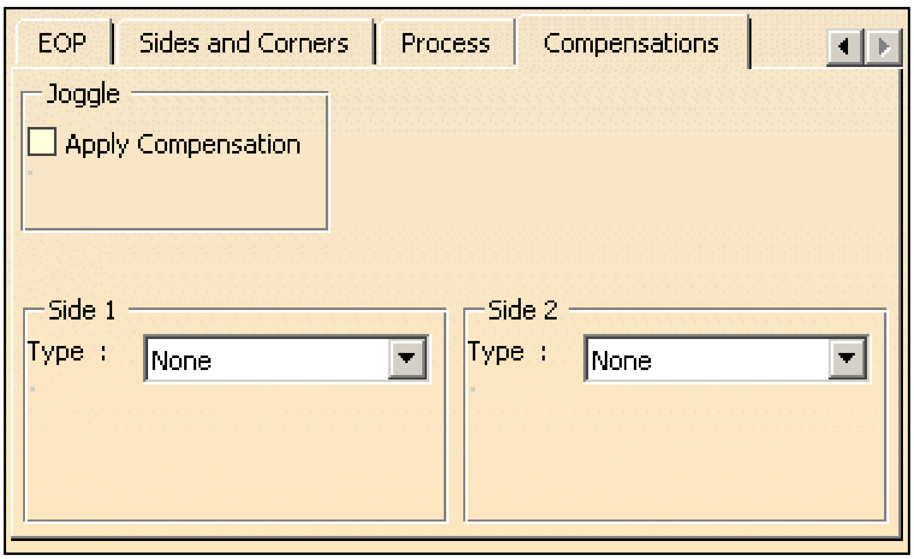
- Manufacturing process:
  - Hydropressed
  - BreakFormed

- K_Factor: you can modify the K Factor as defined in the SheetMetal Parameters dialog box by changing the driving equation.

  Click the icon. The Formula Editor dialog box opens, you can modify the dictionary and the parameters. Or you may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.

You can also choose to display the characteristic curves either on the folded view (**Show curves in folded views**), and/or on the flattened view (**Show curves in flattened view**) of the part.

Compensations



In the Compensations tab, you can define compensations for the:

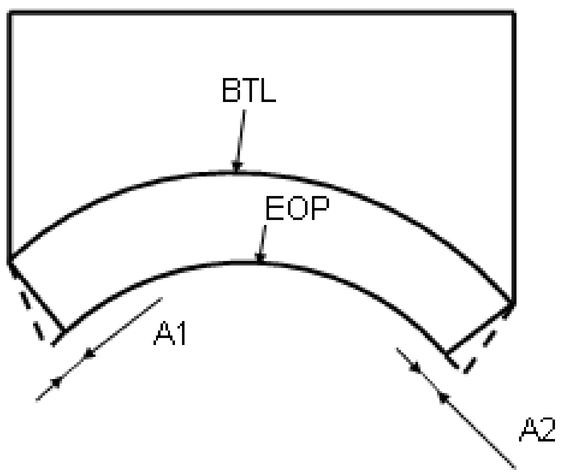
- Joggle:
  - check the **Apply Compensation** button when creating or editing the joggle.
    See Creating a Joggle for further information.

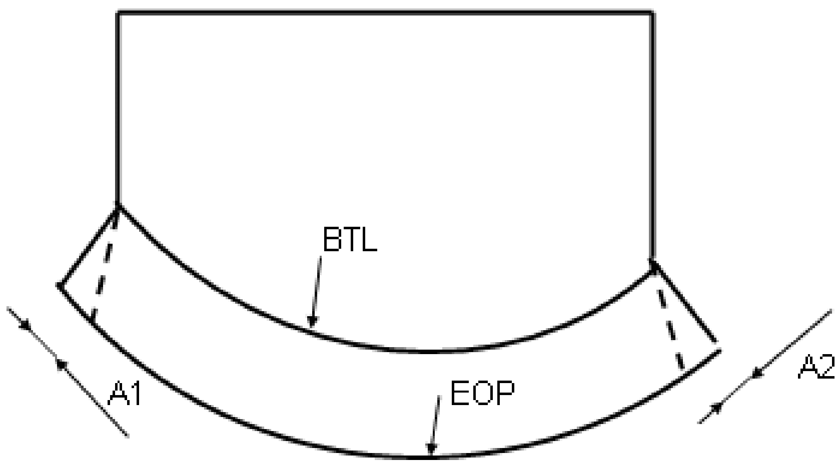
Standard files and methods must be previously defined from the SheetMetal Parameters dialog box to have access to the **Apply Compensation** button.

- Surfacic Flange Sides (Side 1 and Side 2).
  - Define the type:
    - **None:** no compensation is applied
    - **Automatic:** for symmetric flanges , A1=A2, so that the length of the flatten EOP = length of the folded EOP
    - **Manual: Angle**: the deformation is computed according to an angle

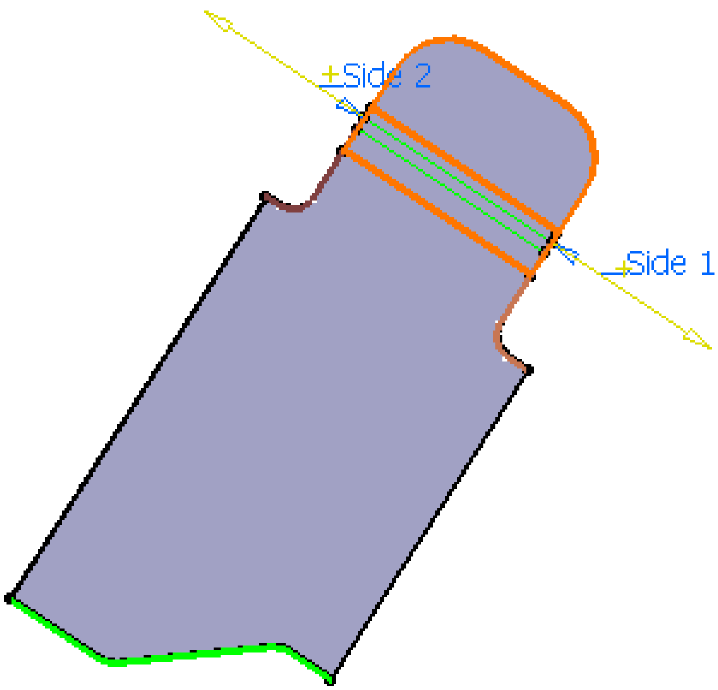
- **Manual: Length**: the deformation is computed according to a length parallel to the BTL.

- Define the **Angle** in the case of a **Manual: Angle** compensation. A negative angle adds material, and a positive angle removes material.
- Define the **Length** in the case of a **Manual: Length** compensation

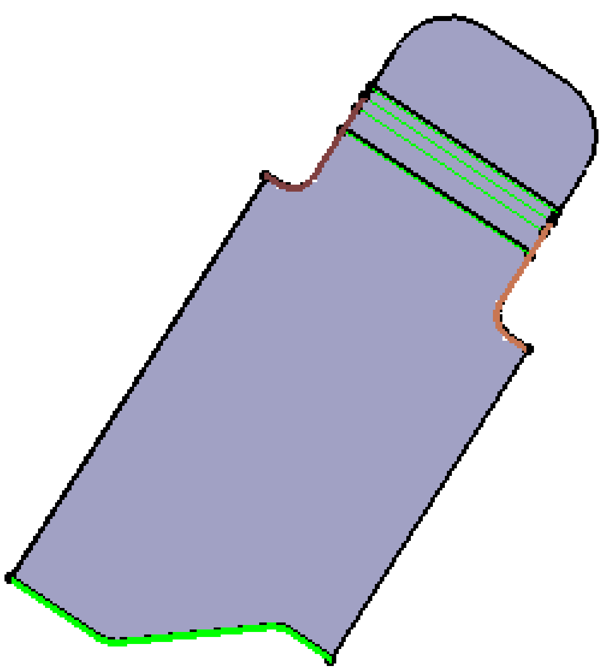




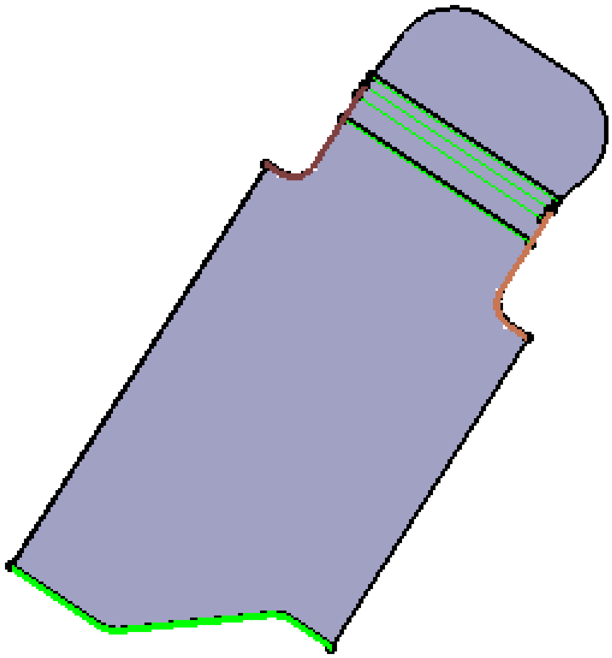
The values of the modification are the angles A1 and A2.



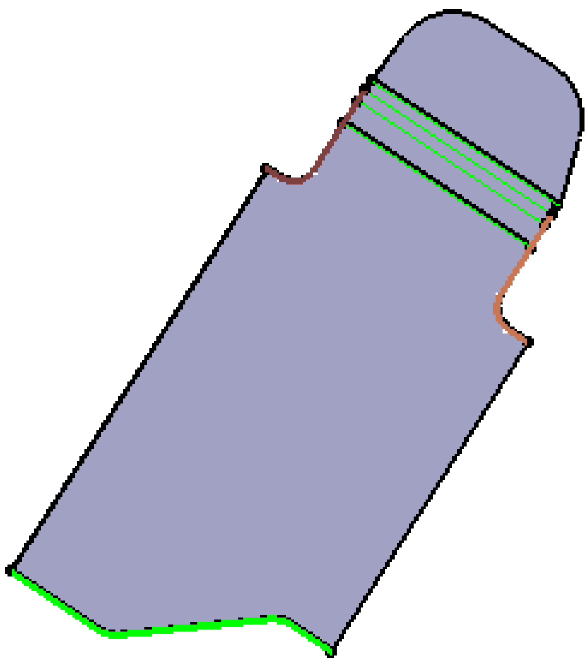
- Compensations can be created either on the folded or flattened part, but they only apply on the flattened part.
- Compensations can be modified independently on each flange.
- The + sign (in yellow in the 3D geometry) shows that material will be added to the sides.



Unfolded Surfacic Flange defined with Corners 1 and 2 of 10mm each, and no compensation for Side 1 and Side 2



Unfolded Surfacic Flange defined with Corners 1 and 2 of 10mm each, a Manual: Angle compensation of -20deg for Side 1 and no compensation for Side 2



Unfolded Surfacic Flange defined with Corners 1 and 2 of 10mm each, a Manual: Angle compensation of 20deg for Side 1 and -10deg for Side 2

Hybrid Design

- In hybrid design context, when the edge of a part and/or the surfacic flange are defined by a sketch, they follow the hybrid design aggregation rules.
- Yet, if you open a part created using an application release prior to Version 5 Release 14, the specification tree is displayed according to the rules implemented for that release.

For more information about hybrid design, refer to the Hybrid Design section.

Creating a Joggle

This task explains how to create a joggle, that is a feature which causes the main feature, that is, a surfacic flange or a web, to be locally deformed.

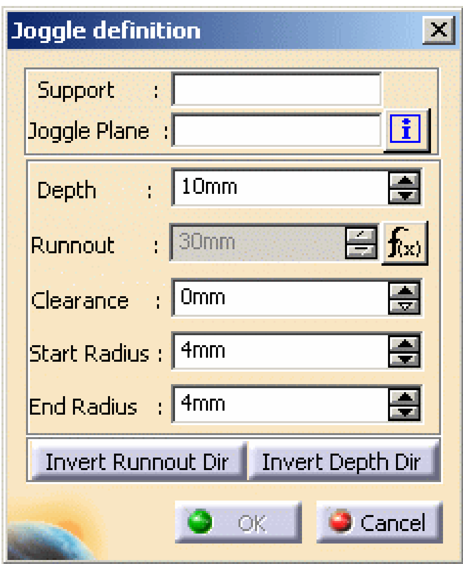
The joggle is a feature which cannot exist alone, it is always defined on a surfacic flange or a web.

In the following example, you will create a joggle defined on a surfacic flange, yet, this description is also valid for a joggle created on a web.

Open the Joggle1.CATPart document.

Create a surfacic flange as shown in the previous task.

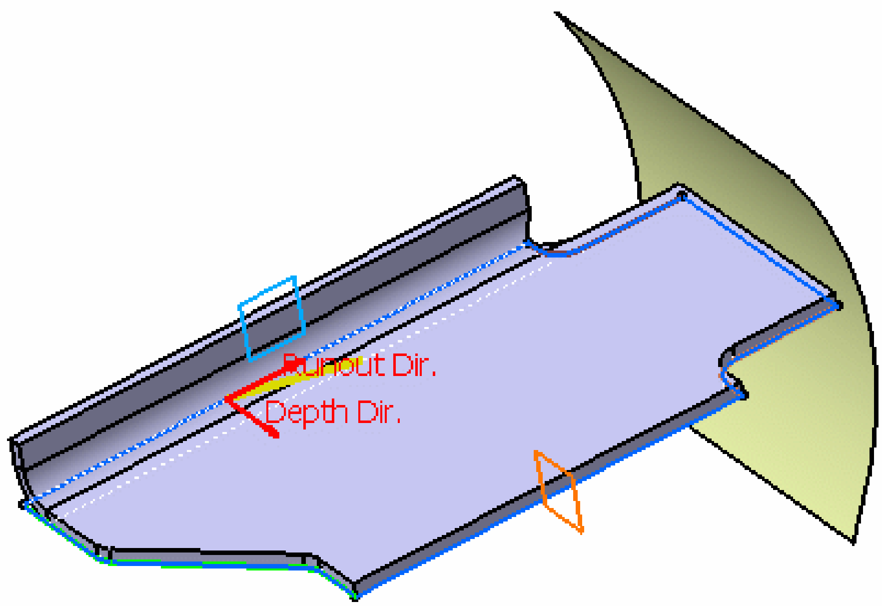
1. Click the **Joggle** icon .



2. Select the surfacic flange or the web as the support.

The **Support** of the joggle is not automatically set to the last created feature (surfacic flange or web).

3. Choose a plane as the joggle plane, here we choose Plane3.



The blue curve defines the boundary of the web.

The yellow line is a preview of the joggle.

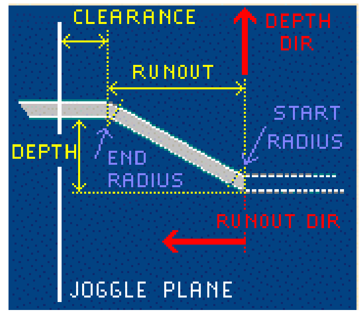
The vectors show you the joggle directions:

- The vector on the surfacic flange or the web support determines the depth direction

- The vector on the joggle plane determines the side on which the joggle is to be created

In case there are several intersections between the surfacic flange or the web and the plane, the closest intersection is chosen.

You can click the [i] icon to display a schema showing the joggle parameters to be defined.



4. You can modify the following parameters of the joggle by clicking the up and down arrows.

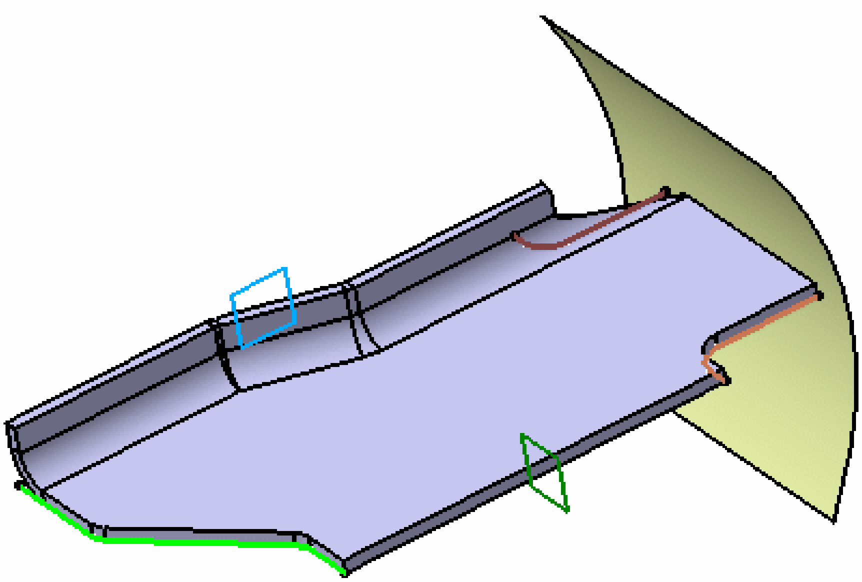
- depth: offset from the support surface (shown as dotted lines)

The dotted lines must remain inside the blue curve.

- runout: length of the offset, between the original surface of the surfacic flange or the web and the new surface (joggle)
- clearance: length added to the offset at the joggle starting plane
- start radius: fillet between the runout and the surfacic flange or the web
- end radius: fillet between the runout and the offset

5. Click OK.

The joggle (identified as Joggle.xxx) is created and the specification tree is updated accordingly.



- If you modify the depth, the runout adjusts automatically, thanks to the formula applied to the runout parameters. If you do not want it to be adjusted automatically, right click on the runout field, select **Formula ->Deactivate** in the contextual menu.
  If you want to have further information about the runout parameters, refer to the Defining the Compensations and Runout section.

- You can reverse the runout direction either by clicking the red arrow or by clicking the **Invert Runout Dir** button in the dialog box.

- You can reverse the depth direction either by clicking the red arrow or by clicking the **Invert Depth Dir** button in the dialog box.

You cannot have a runout intersect with another runout.

If you try to create a joggle on a web as well on the surfacic flange or if you create two joggles side by side that end up touching each other, an error message is displayed to prevent the runouts from intersecting with each other.

When creating a joggle on a web, you can choose the shape of the surfacic flange edge of part:

If you create an element above the part, for instance, a line, and want the surfacic flange edge of part to follow the line:

- Open the **Surfacic Flange Definition** dialog box,
- Go to **EOP** tab and select **Element FD** or **Element FP** in the combo box.

If you would rather the flange edge of part to follow the joggle runout:

- Open the **Surfacic Flange Definition** dialog box,
- Go to **EOP** tab and select **Length From OML** in the combo box.

Applying Compensations

You can apply compensations only on joggle defined on a surfacic flange.

In the Surfacic flange dialog box, you can check the **Apply Compensation** option either on the folded or flattened part, but they only apply on the flattened part.

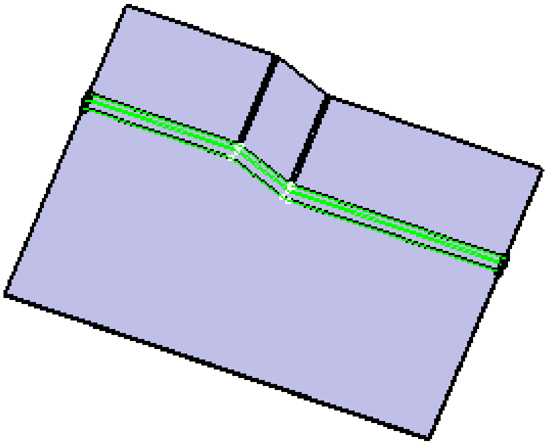
Open the Joggle2.CATPart document for a single joggle, Joggle.3CATPart for a twin joggle, and Joggle4.CATPart for a double joggle.

Standard Files must have been previously imported and a method for compensations defined.

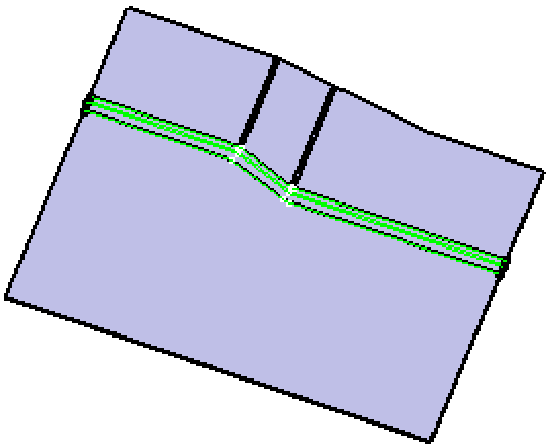
1. Double-click the **surfacic flange** supporting the joggle to edit it.

2. In the Compensations tab, click the **Apply Compensation** button.

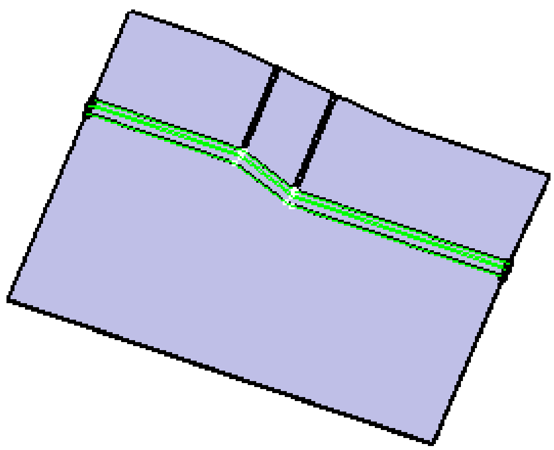
On a single joggle



Unfolded single joggle without compensations

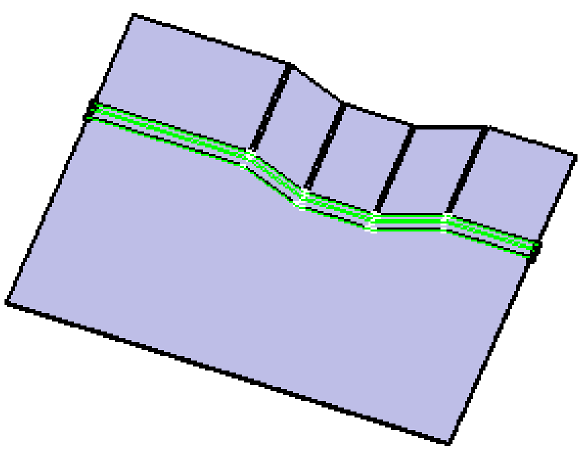


Unfolded single joggle with compensations relying on Method 1

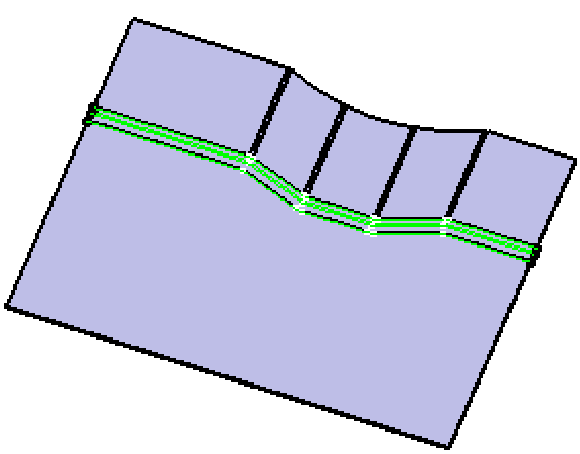


Unfolded single joggle with compensations relying on Method 2

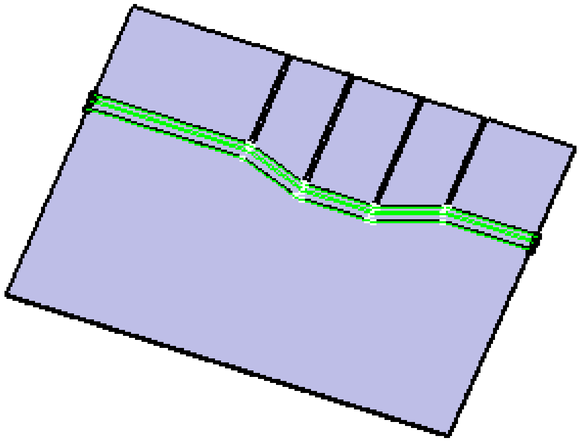
On a twin joggle



Unfolded twin joggle without compensations

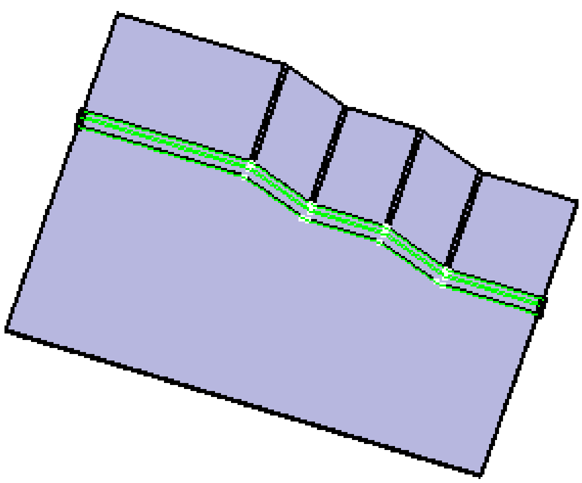


Unfolded twin joggle with compensations relying on Method 1

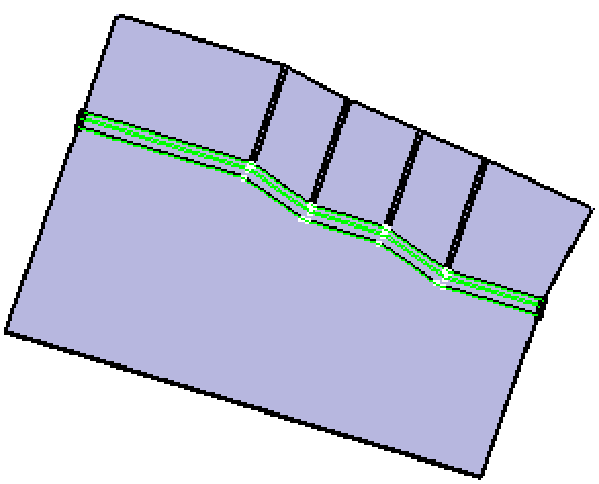


Unfolded twin joggle with compensations relying on Method 2

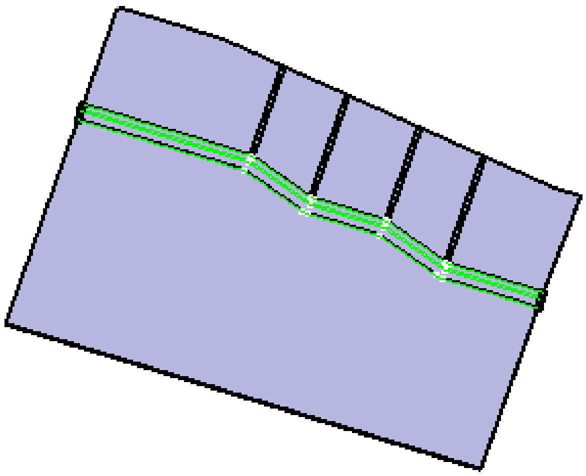
On a double joggle



Unfolded double joggle without compensations



Unfolded double joggle with compensations relying on Method 1



Unfolded double joggle with compensations relying on Method 2

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.

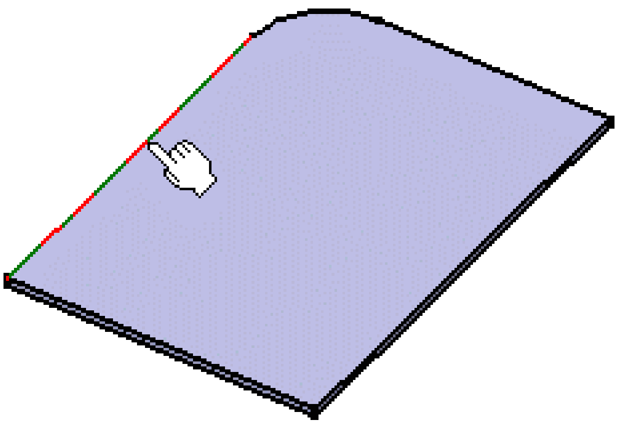
 Create a tear drop: select a spine, and set the radius, and length values.

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

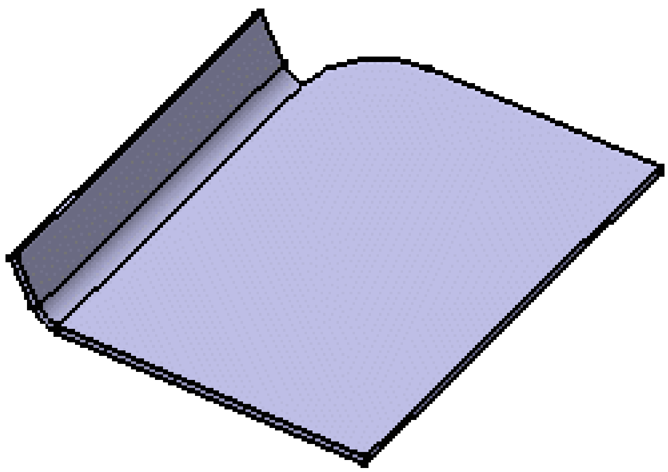
Selecting the Spine

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

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

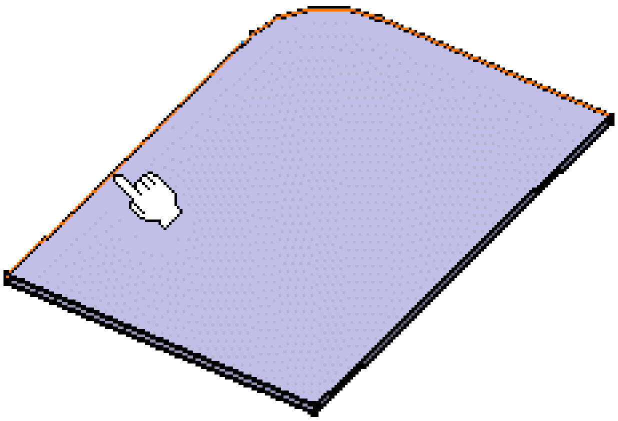


Selection without propagation

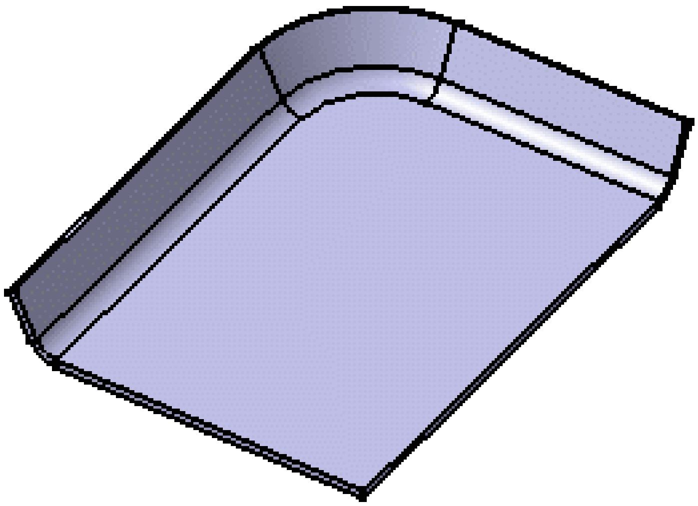


Resulting flange without propagation

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



Selection with propagation



Resulting flange with propagation

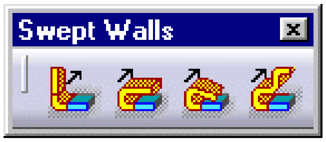
Creating a Flange

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

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

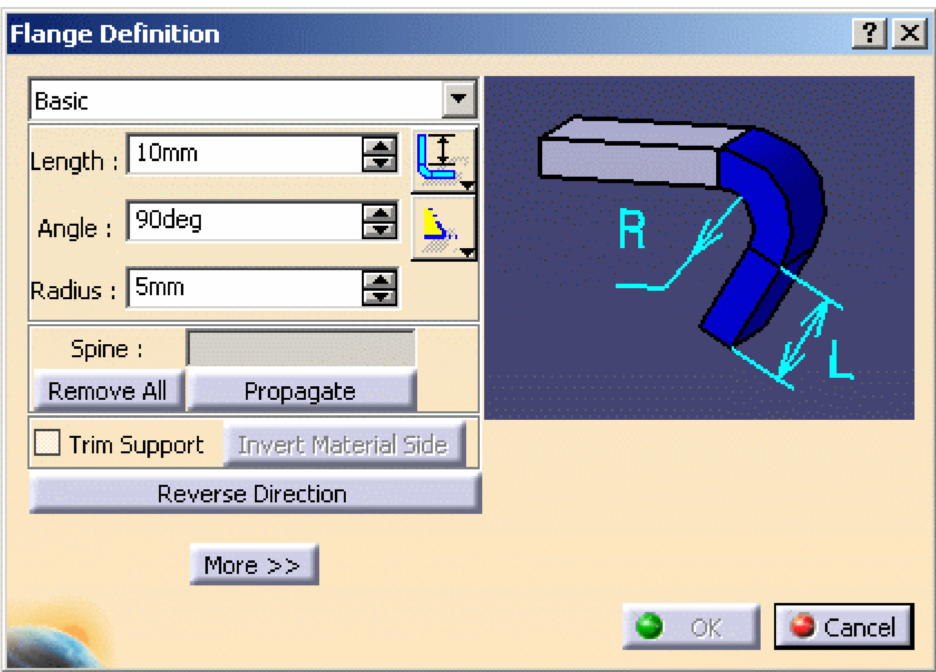
For the Aerospace SheetMetal Design workbench, open the Aero_SweptWall01.CATPart document.

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



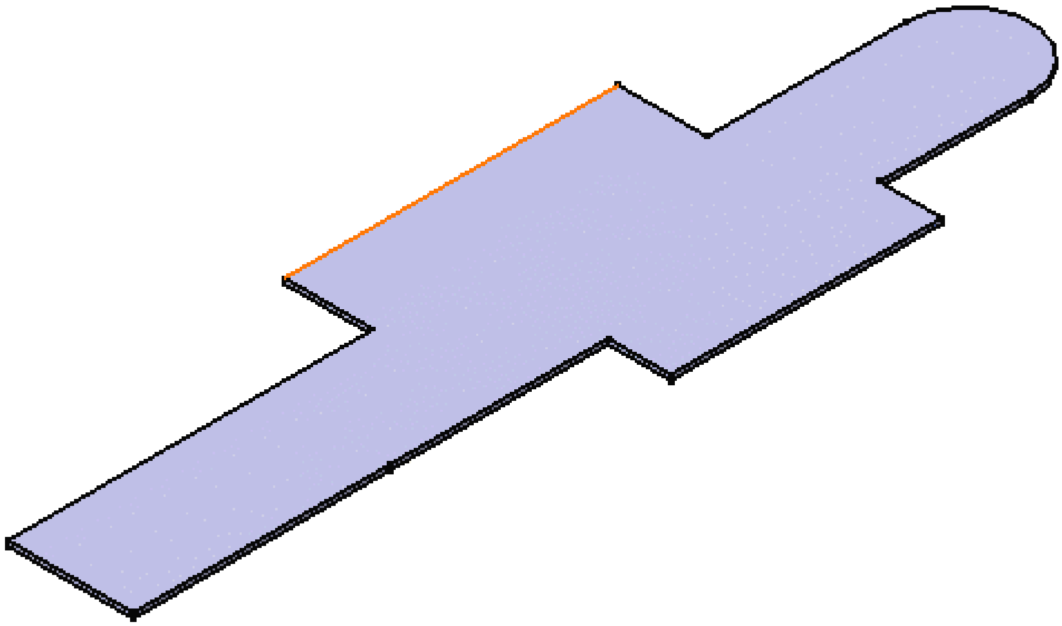
The Flange Definition dialog box is displayed.

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.

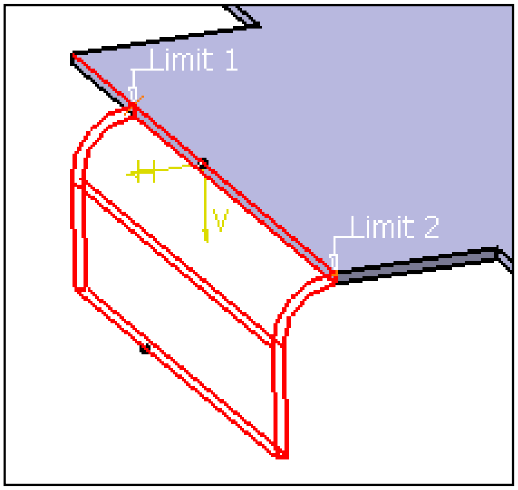
- You can use the **Remove All** button to remove the selected edge(s).
- You can use the **Propagate** button to select all tangentially contiguous edges forming the spine.

The drop-down list offers two choices:

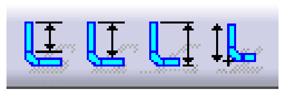
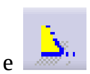
- **Basic**: the flange is created along the whole support.
- **Relimited**: the flange is created within limits you define on the support (points, for example).

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.



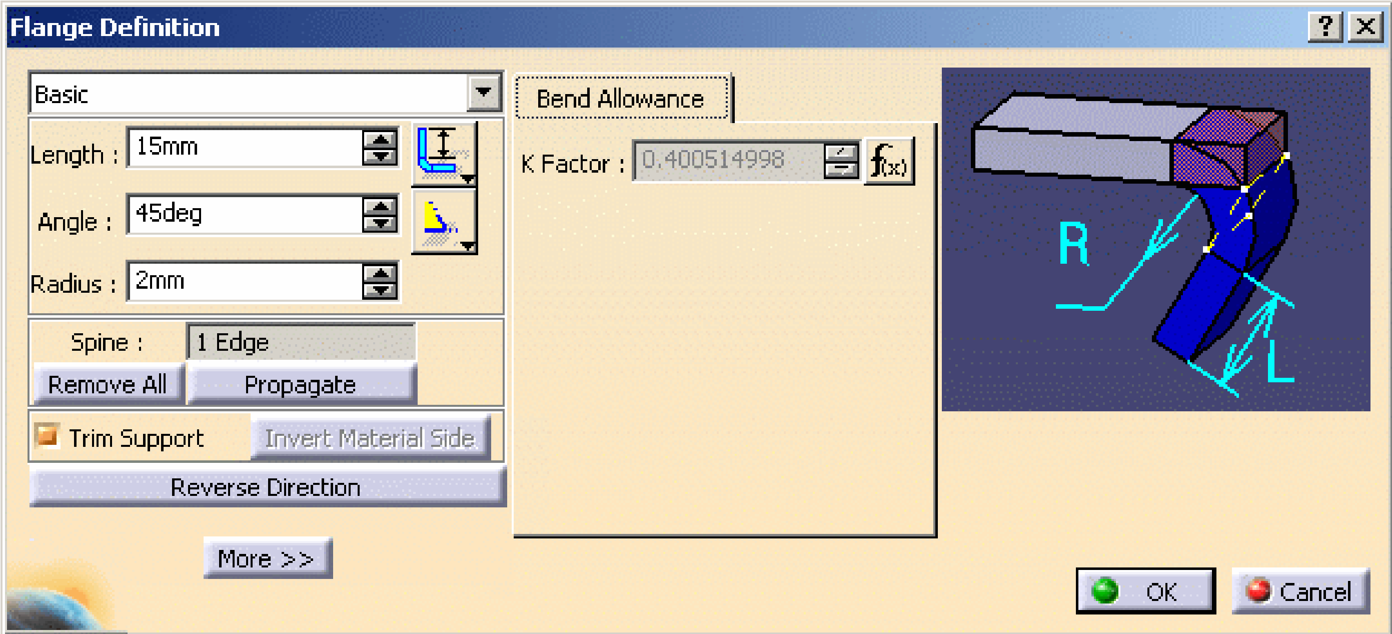
4. Choose the flange parameters:

- 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.

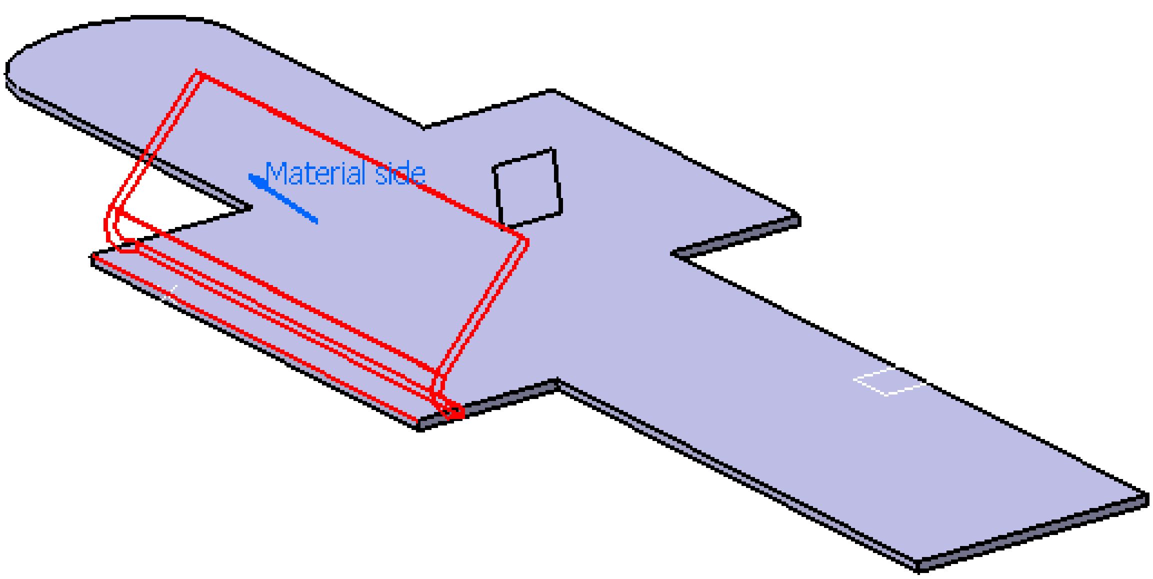
- The Trim Support option only works in the case of a planar support.
- You cannot select **Relimited** and **Trim Support** at once. Indeed, lateral cuts in the sheet metal part are currently not supported when the support is trimmed (i.e. the flange must be created from one edge of the sheet metal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

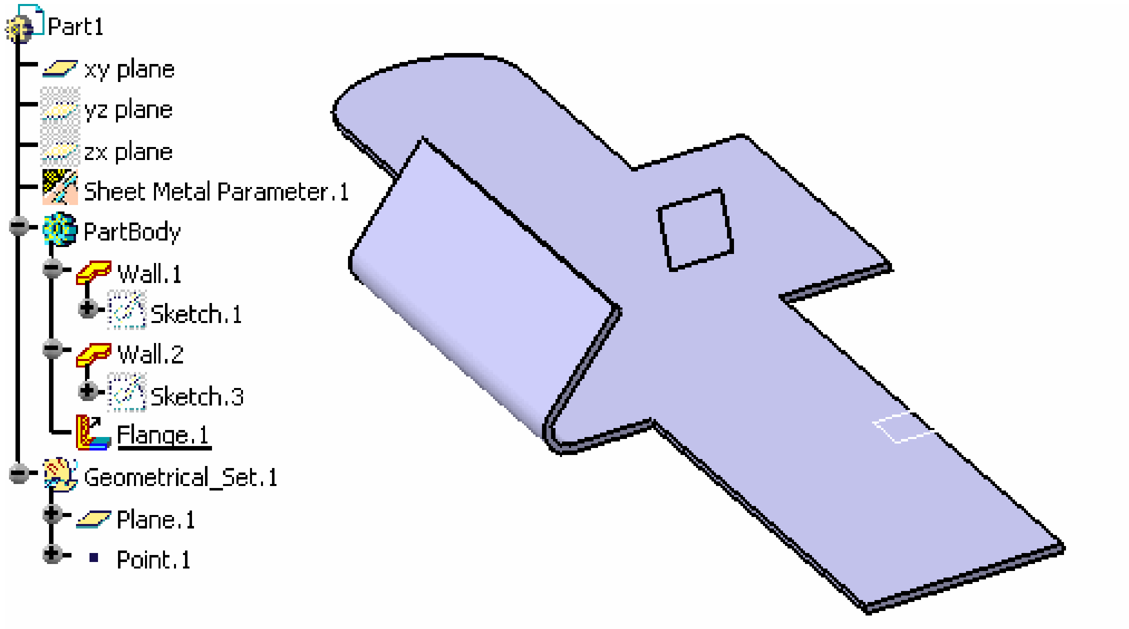


In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

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.

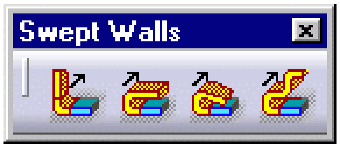

Part1
xy plane
yz plane
zx plane
Sheet Metal Parameter.1
PartBody
Wall.1
Sketch.1
Wall.2
Sketch.3
Flange.1
Geometrical_Set.1
Plane.1
Point.1

Creating a Hem

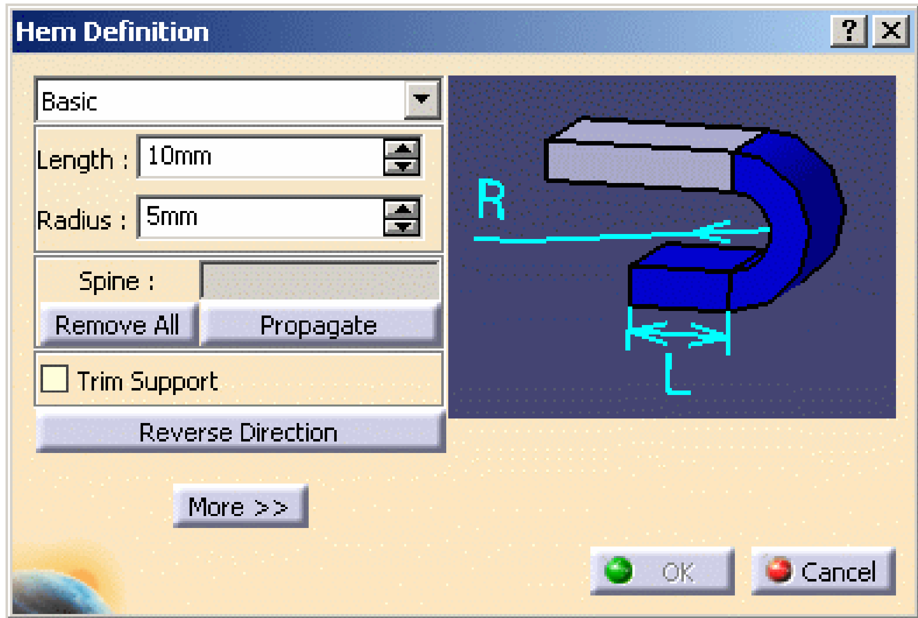
This task explains how to generate a hem from a spine and a profile.

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.

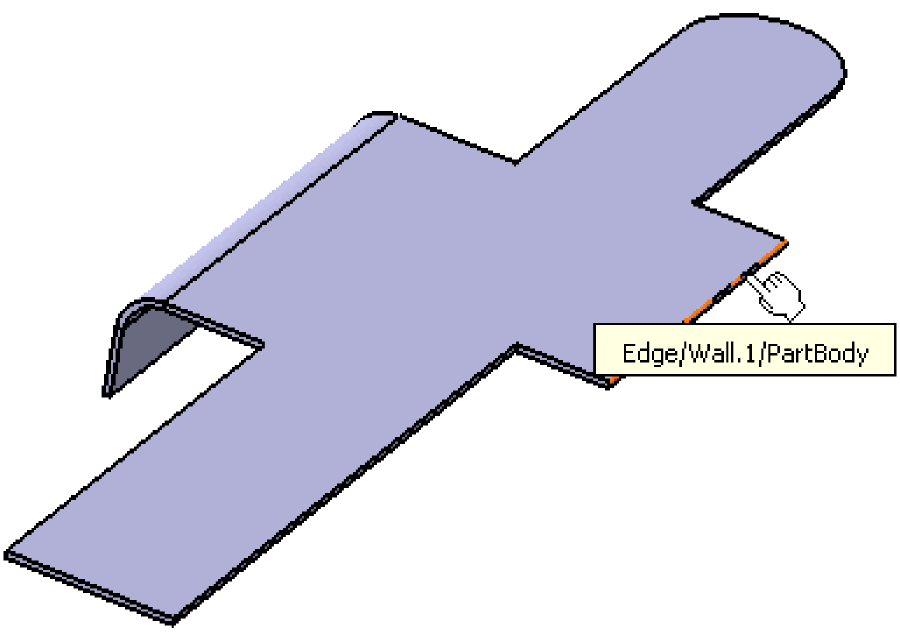


The Hem Definition dialog box opens.



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.

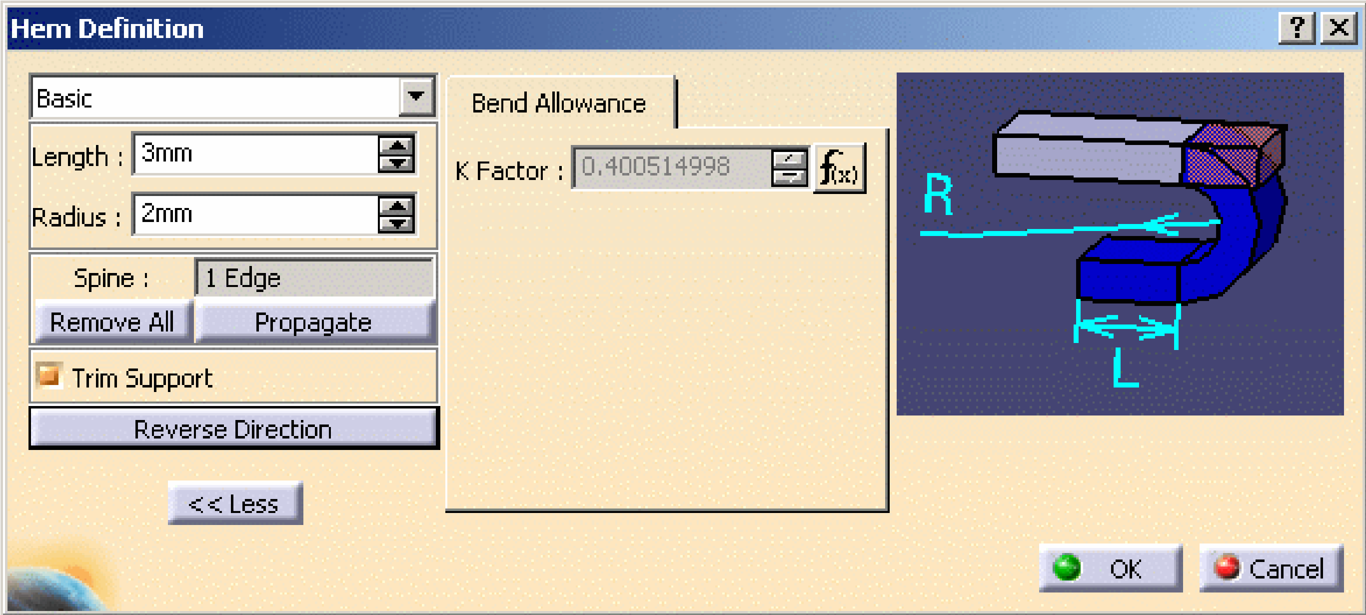
4. Enter 3mm in the **Length** field, and 2mm in the **Radius** field.
5. Check the **Trim Support** option to trim the selected edge.

- 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).

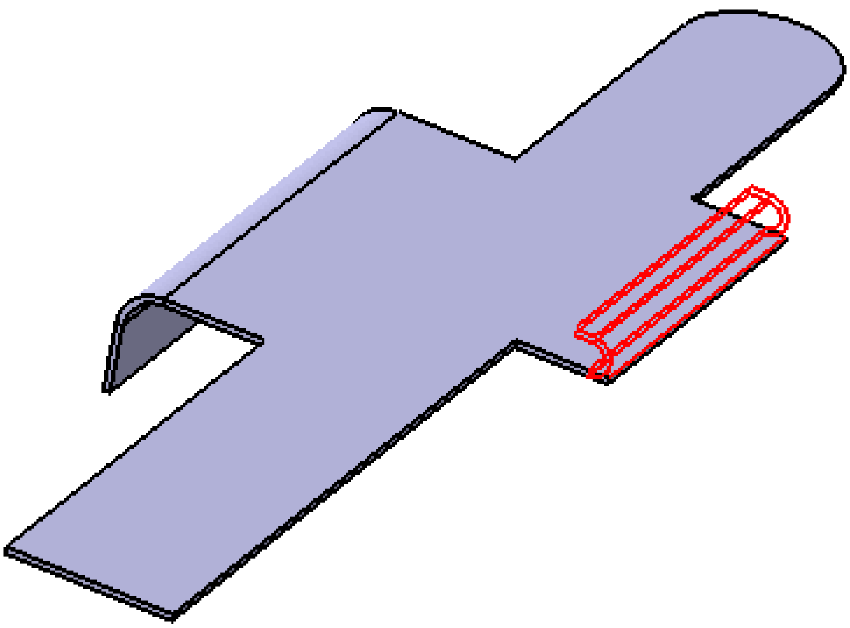
6. Click the **Reverse Direction** button to reverse the direction of the hem.
7. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

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

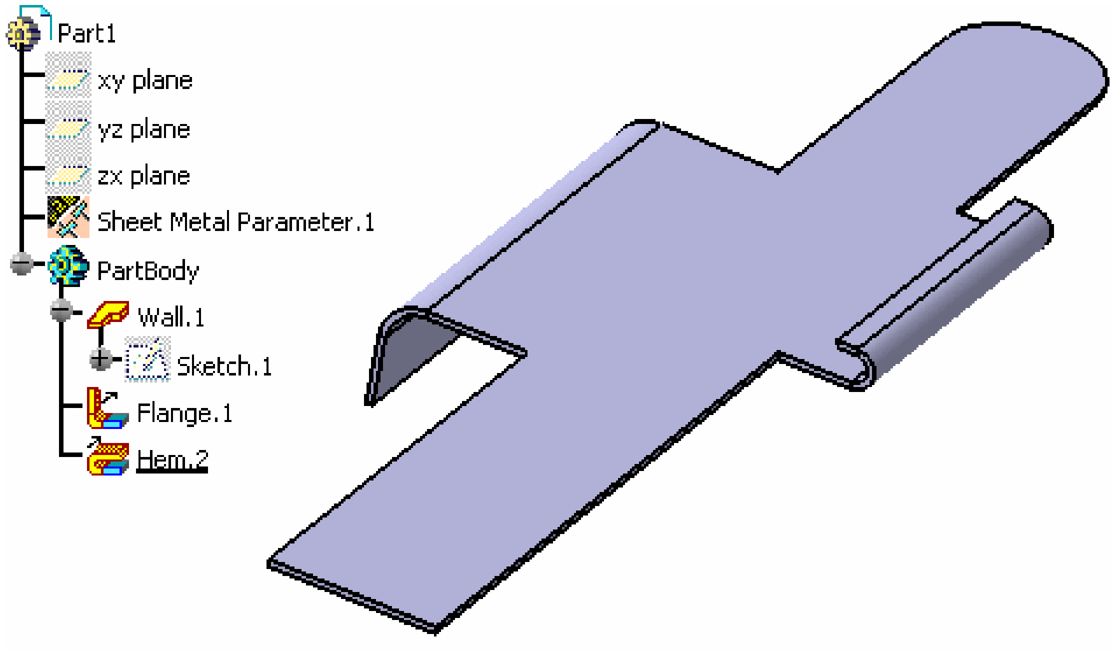


In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

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.

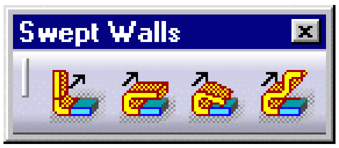

Part1
xy plane
yz plane
zx plane
Sheet Metal Parameter.1
PartBody
Wall.1
Sketch.1
Flange.1
Hem.2

Creating a Tear Drop

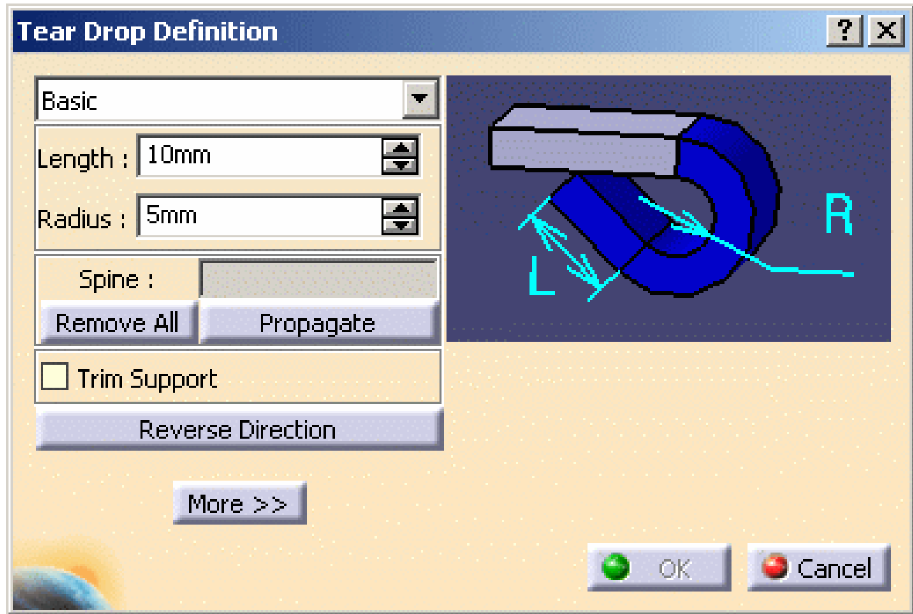
This task explains how to generate a tear drop from a spine and a profile.

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.

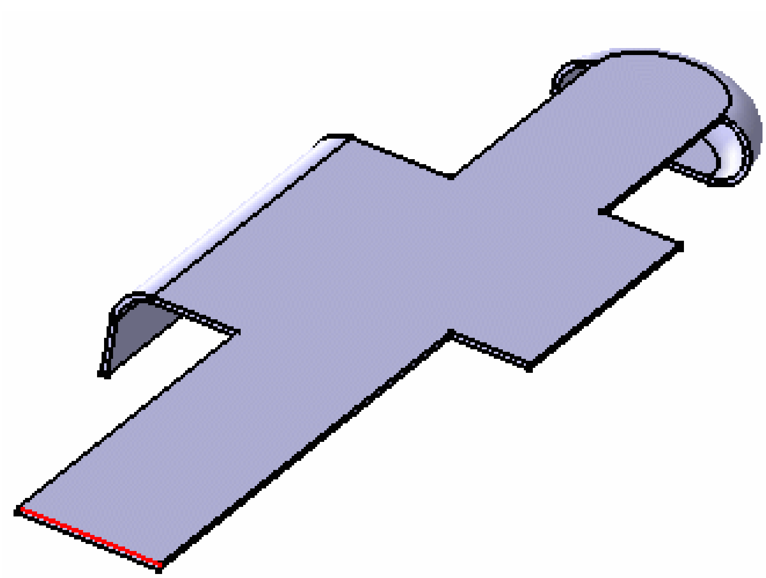


The Tear Drop Definition dialog box opens.



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.

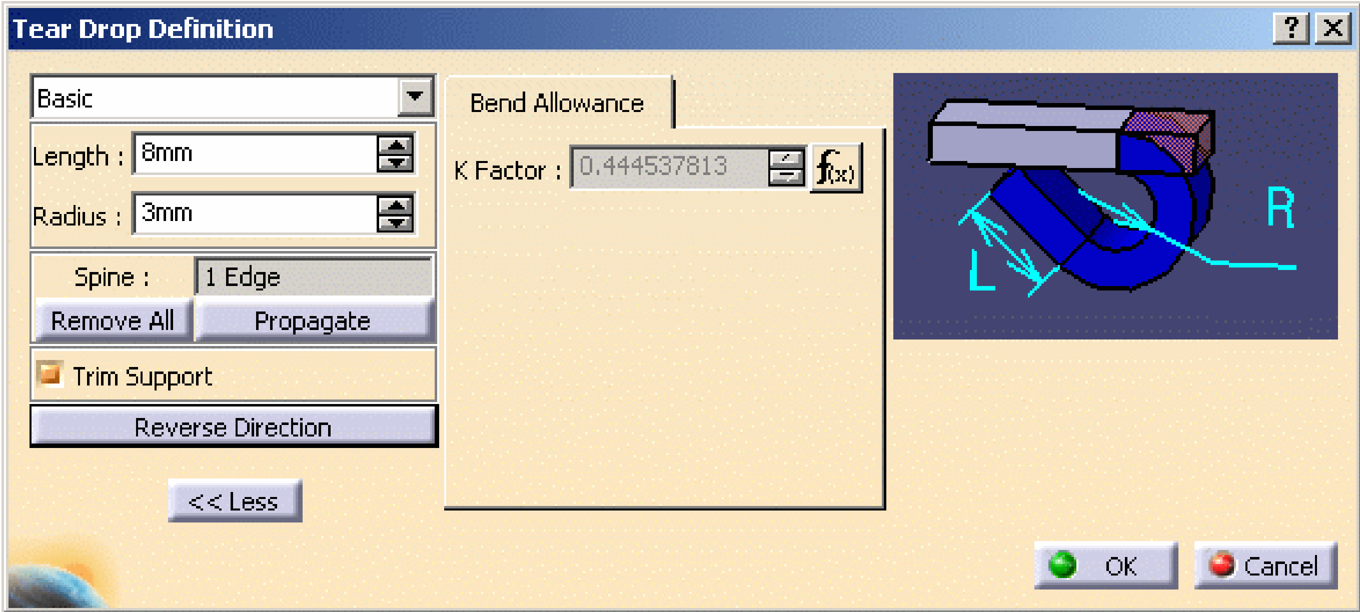
Selecting **Relimited** updates the dialog with two new fields (**Limit 1** and **Limit 2**) to let you specify the tear drop limits. You can then select as the limits two points, two planar faces, a point and a planar face, or a point and a vertex, for example. Note that right-clicking in the **Limit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

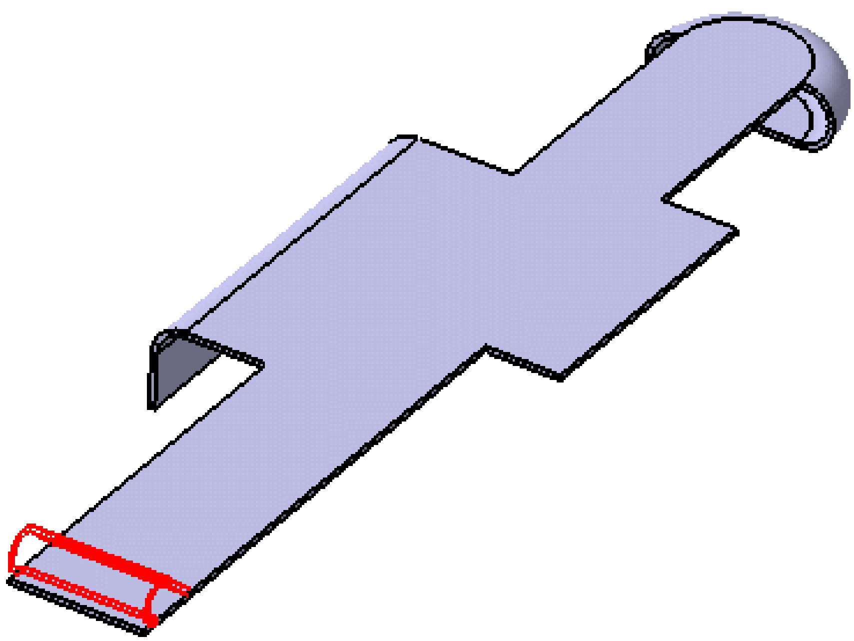
6. Click the **Reverse Direction** button to reverse the direction of the tear drop.
7. Click the **More>>** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.

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

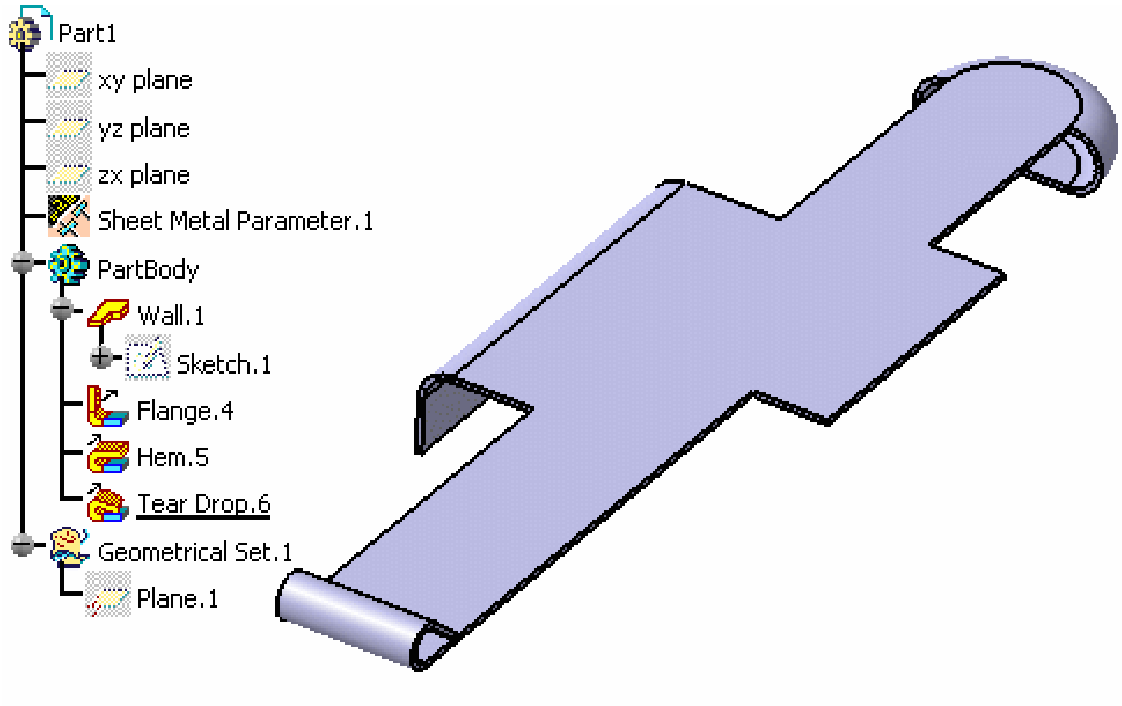


In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

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.

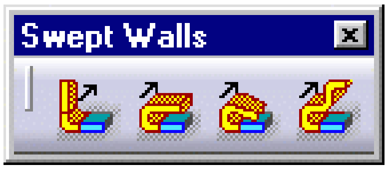

Part1
xy plane
yz plane
zx plane
Sheet Metal Parameter.1
PartBody
Wall.1
Sketch.1
Flange.4
Hem.5
Tear Drop.6
Geometrical Set.1
Plane.1

Creating a User Flange

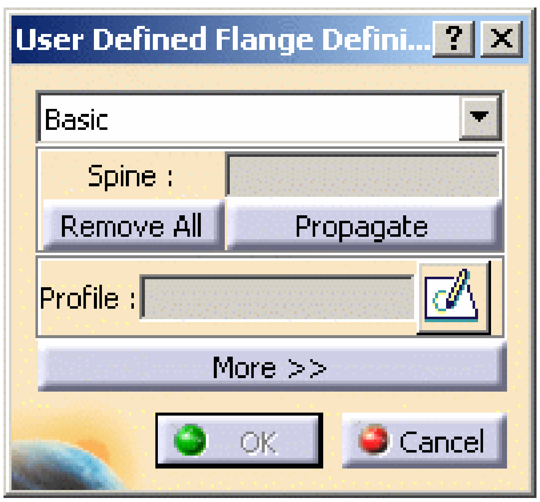
This task explains how to generate a user flange from a spine and a user-defined profile.

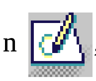
The NEWSweptWall01.CATPart document is still open from the previous task.
If not, open the NEWSweptWall04.CATPart document from the samples directory. As a profile is already defined on the part, you will be able to skip step 2 of the scenario.

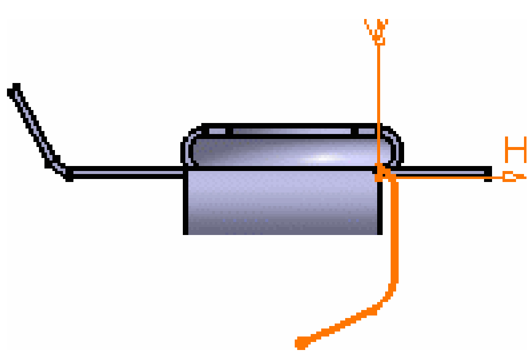
1. Select the **User Flange** icon  in the **Swept Walls** sub-toolbar.



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



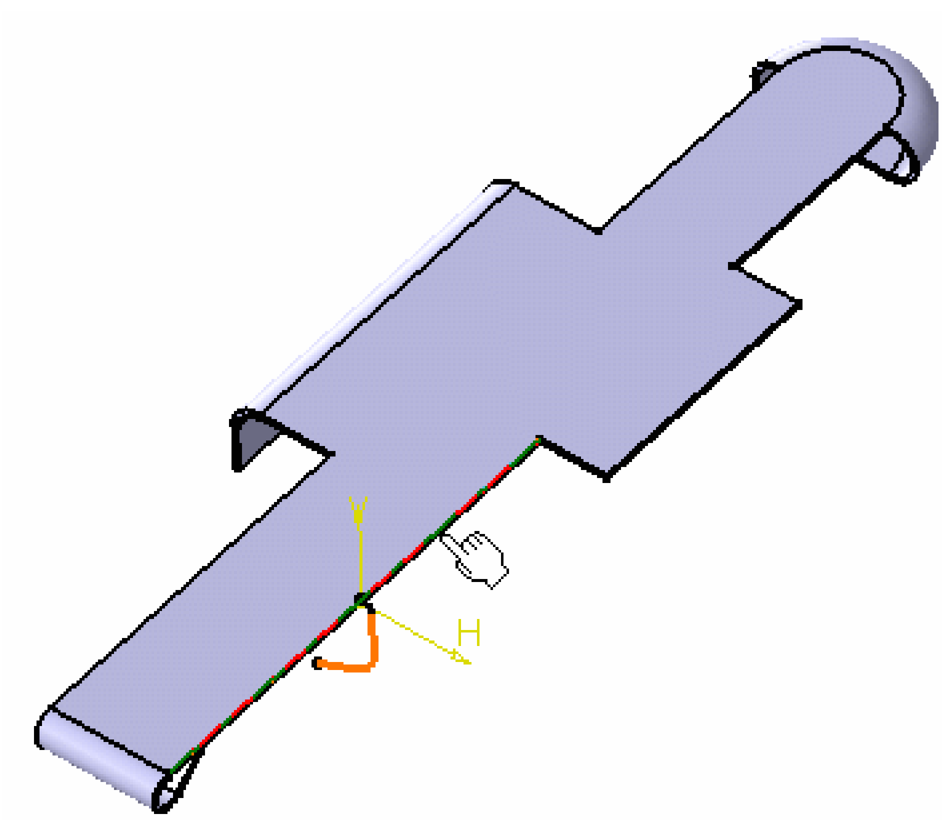
2. If you are using the NEWSweptWall01.CATPart document, click the **Sketcher** icon , and define a profile in the yz plane as shown below:



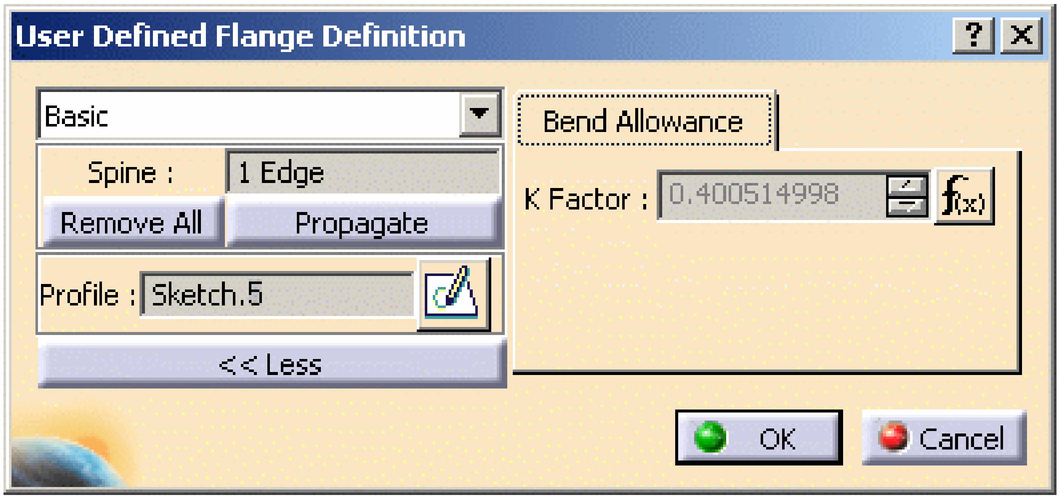
Then quit the Sketcher, using the **Exit** icon .

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.

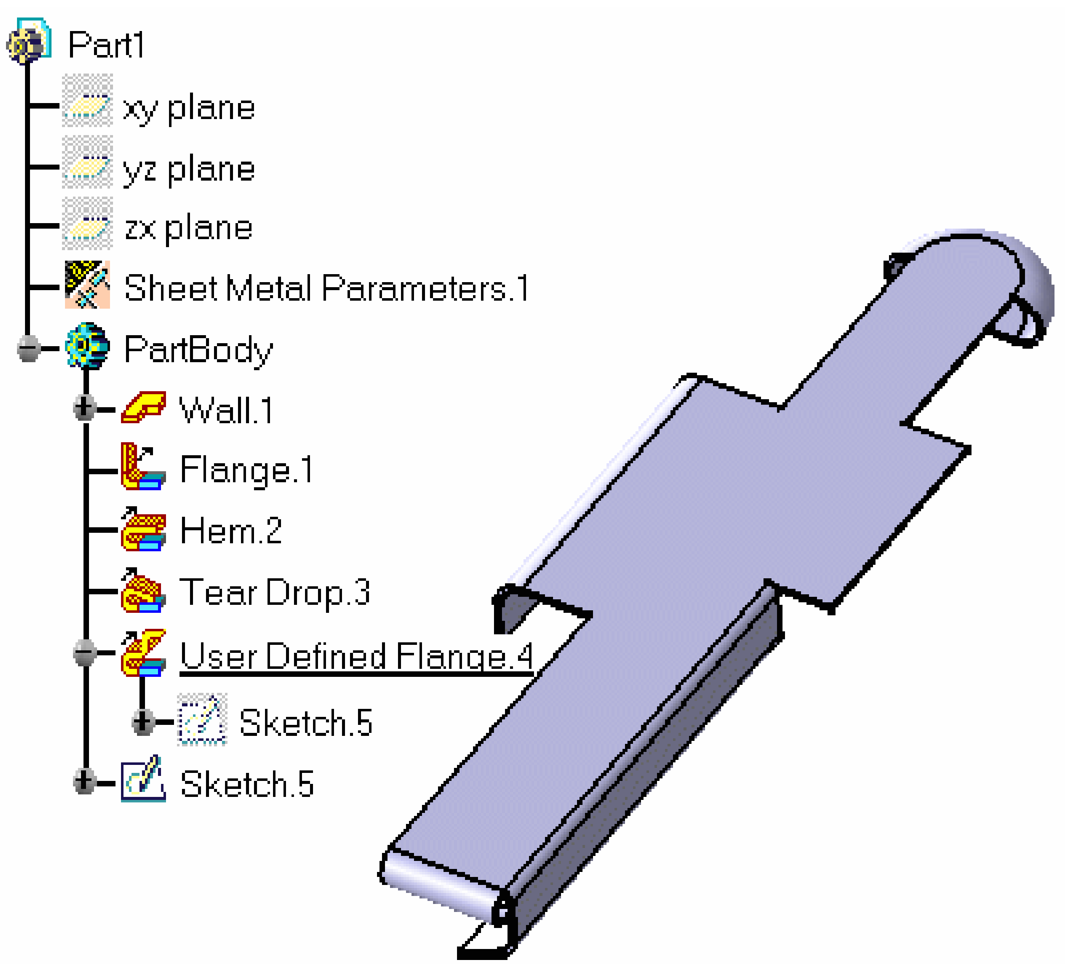


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
   You may need to deactivate the formula using the contextual menu on the **K Factor** field and choosing **Formula -> Deactivate** before editing the value.



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.

Unfolding

Unfolded Aerospace Sheet Metal parts can be displayed in two ways:

Folded/Unfolded View Access
Concurrent Access

Each Aerospace 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.

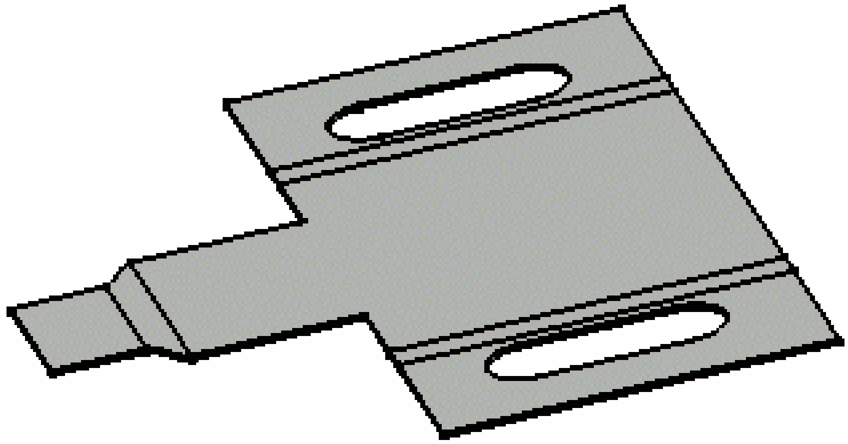
Folded/Unfolded View Access

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.

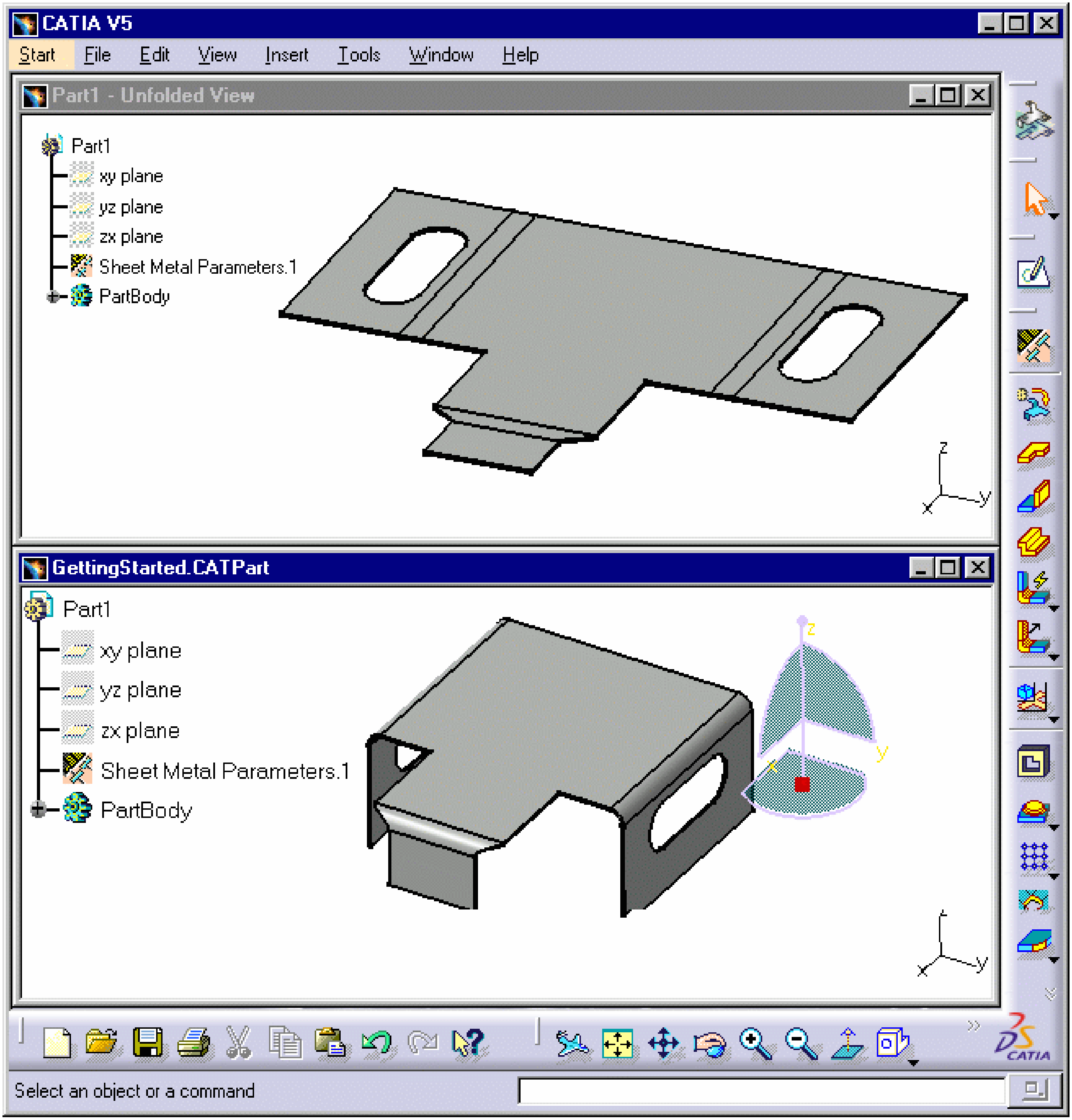
This task explains how to display the sheet metal part in two windows: one with the folded view, one with the unfolded view. Any modification in one window is displayed in the other window.

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 Cutout

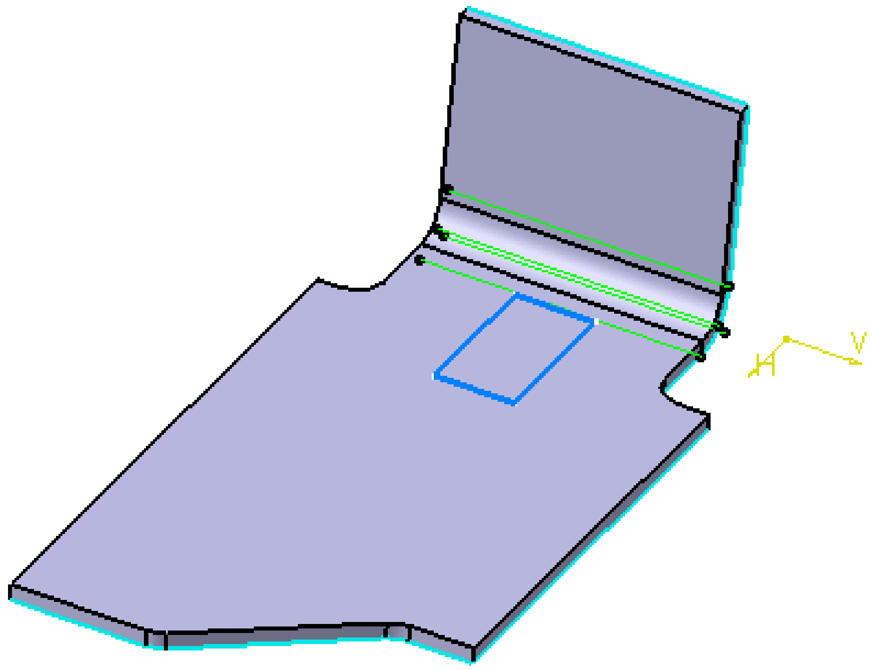
This task explains how to create a cutout.
Creating a cutout consists in extruding a profile and removing the material resulting from the extrusion.

You can create a cutout defined either by a sketch or an open geometry.

Open the CutOut1.CATPart document.

1. Click the **CutOut** icon.

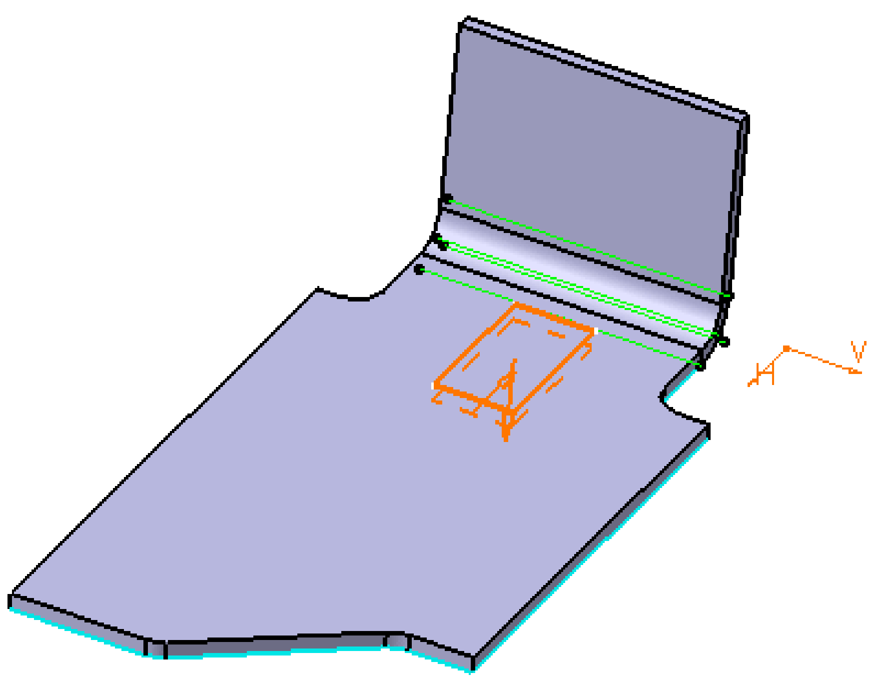
   The Cutout Definition dialog box is displayed and the skin to be impacted by the cutout is displayed in a different color.



2. Select a profile (sketch.1 in our example).

   It can be either a sketch containing one or more shapes, a wire, or a part.

   A preview of the projected cutout is displayed. The vectors show the side and the direction of the cutout.

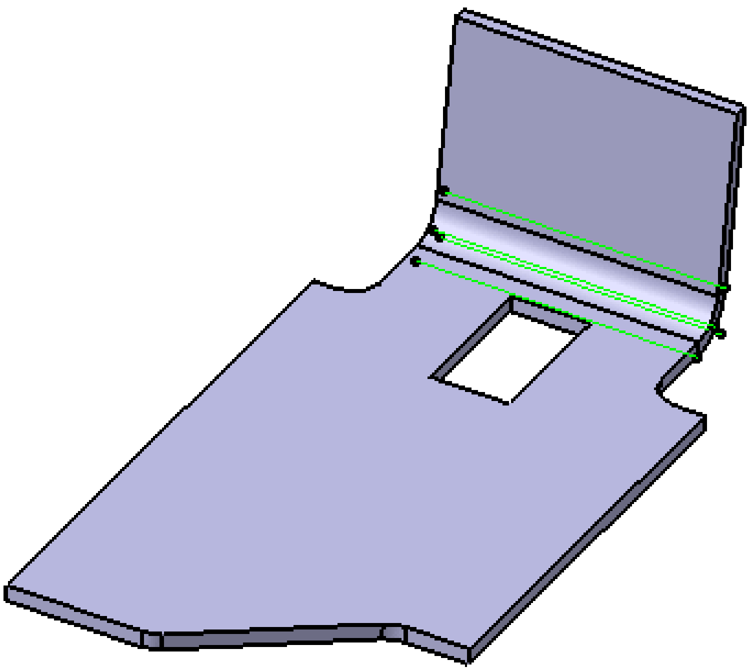


Once the sketch is selected, you can modify it by clicking the Sketcher icon.

- 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:

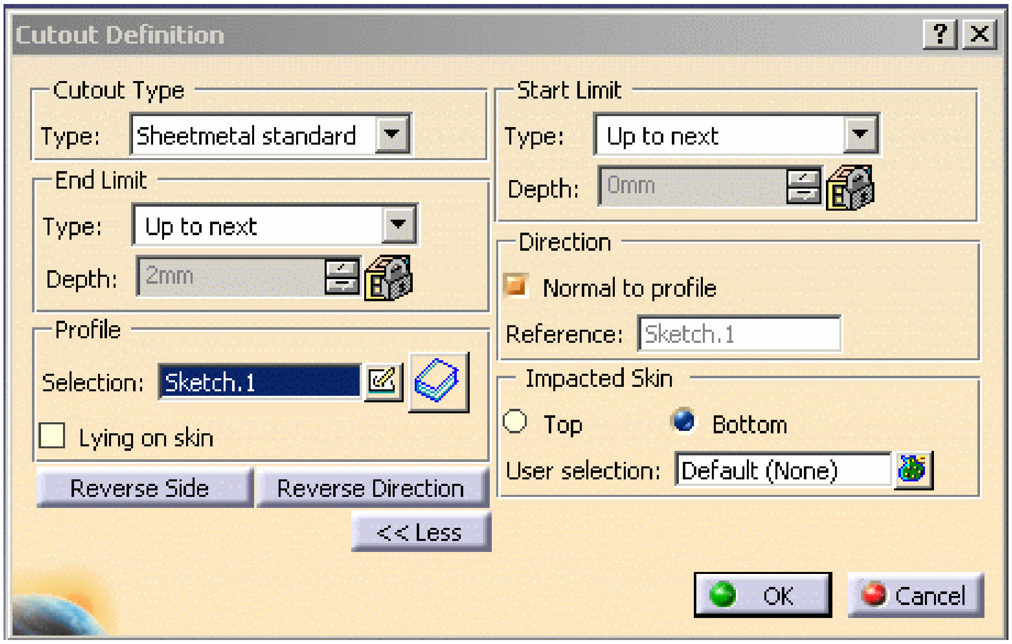
- **Dimension**: the cutout depth is defined by the value measured along the direction.

The depth corresponds to the base feature thickness.
Please refer to Editing the Sheet and Tool Parameters.

- **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 application will limit the cutout onto the last possible face encountered by the extrusion.

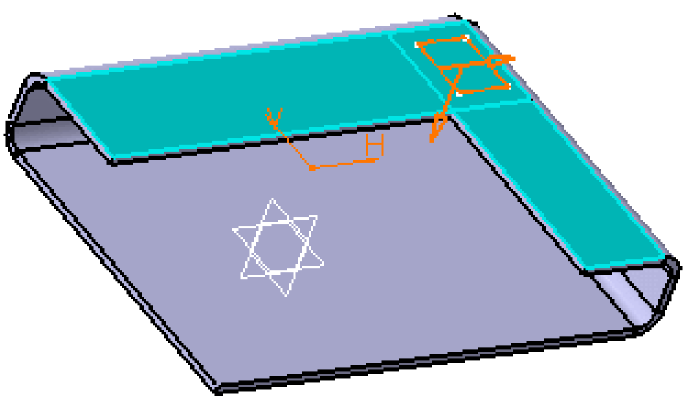
4. In the specification tree, double click on Cut Out.1 to display the Cutout Definition dialog box.
5. Click **More>>** to display the maximum information.

   The Direction is already selected (Sketch.1), that is perpendicular to the base feature.



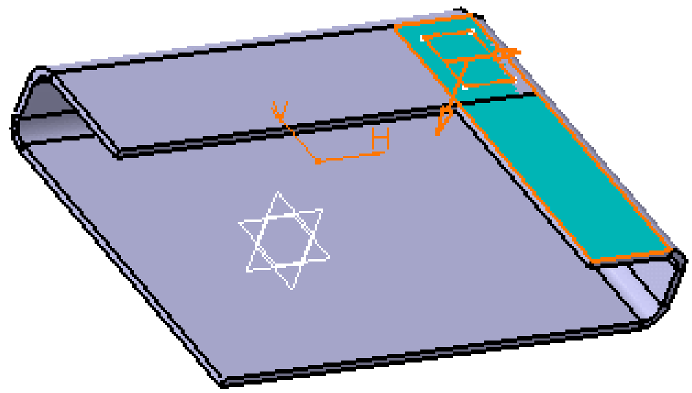
Here the Cutout's impacted skin is set to Default, that is, the surface on which lies Sketch.1.

If you want to select another support for the cutout, click on  and select your new support. It can be a web, a flange or the planar part of the surfacic flange.
Specifying the support for the cutout avoid confusions in case of overlaps.



In the following example two flanges are overlapping each other. If you try to create a cutout on such a part, 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.

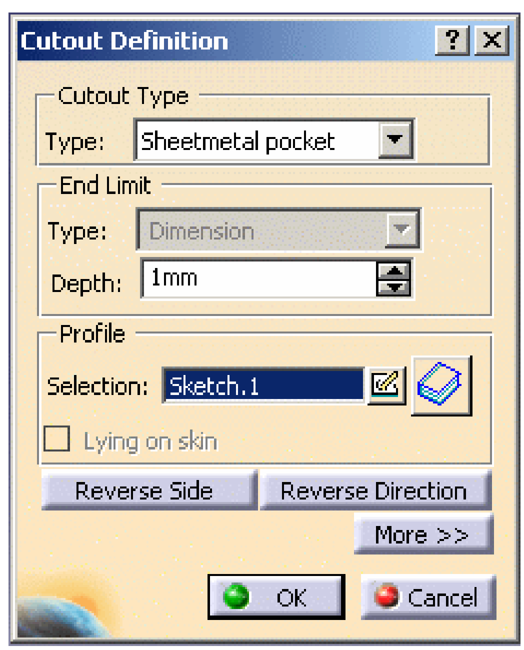
  This option is available only when creating a standard cutout.

- In case the profile's edges and the impacted skin to extrude are tangent, the sketch becomes non-valid and the cutout cannot be created.

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

Open the CutOut1.CATPart document again.

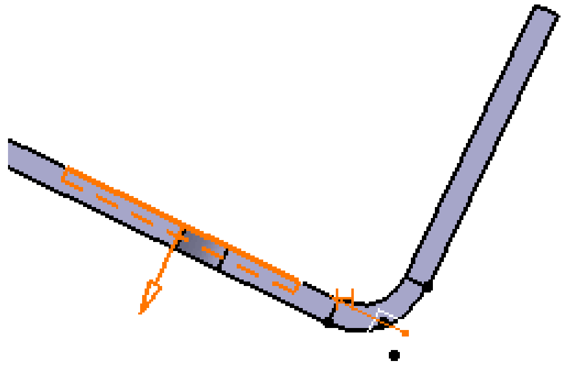
1. Click the **Cutout** icon.
2. Select **Sheetmetal pocket** as Cutout type in the combo box.
   The skin to be impacted remains grey and the End limit type is disabled.
3. Set the Depth to 1mm.



4. Select Sketch.1 as profile.

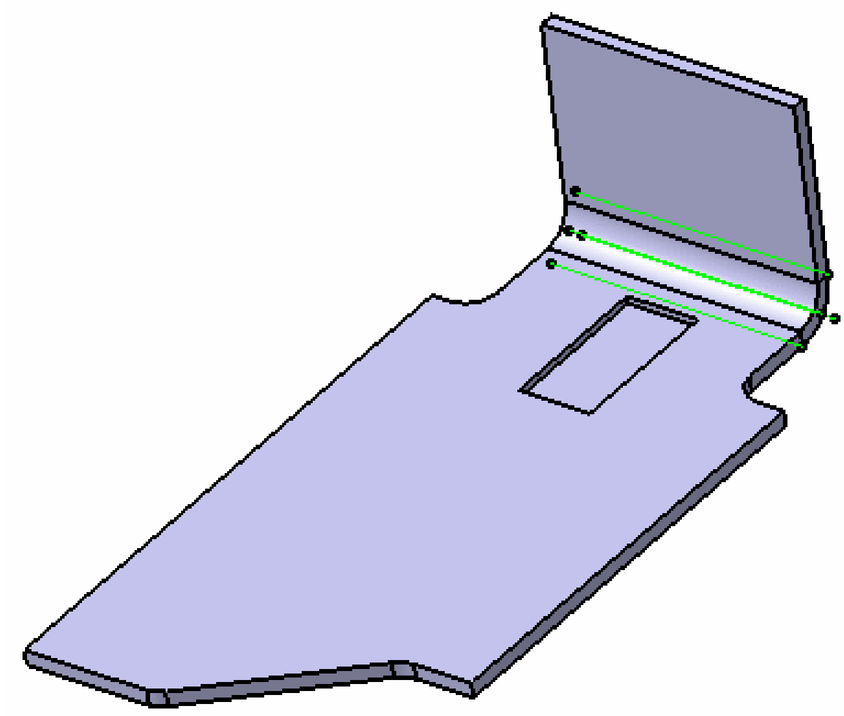
   A preview of the cutout is displayed.

   In our example, the cutout will impact only half the base feature.



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

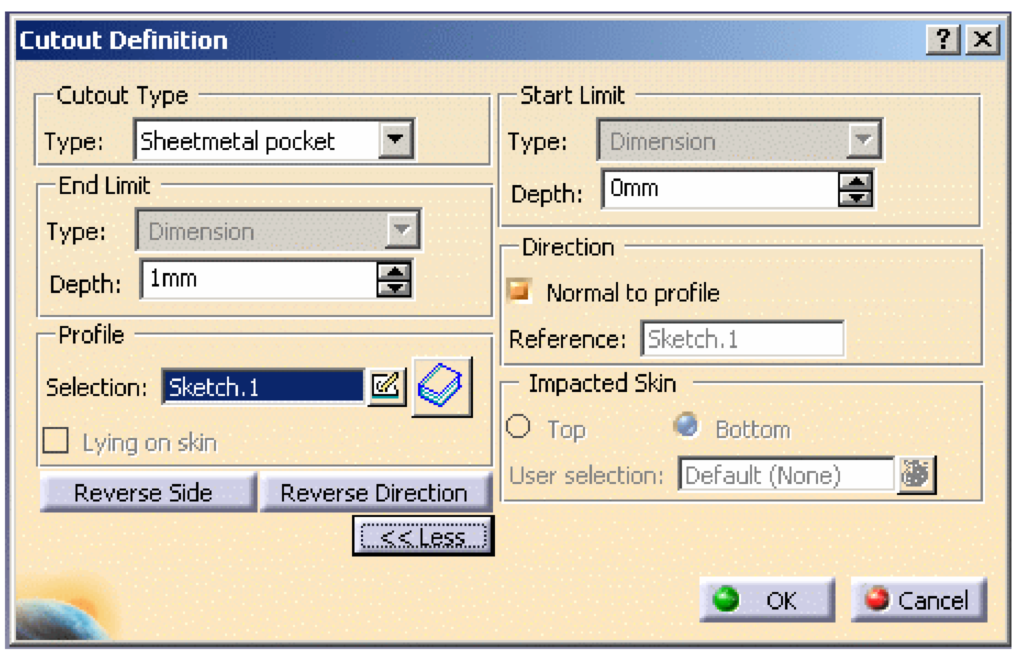
The cutout is created.



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

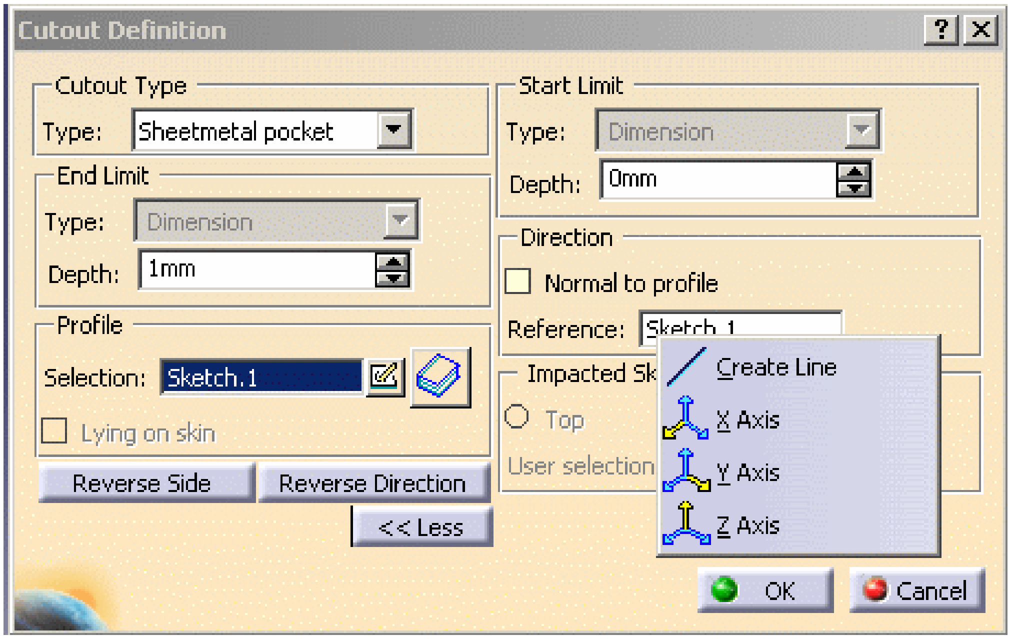
5. Click **More**>> to display the maximum information.

The Direction is already selected (Sketch.1). By default, it is set as normal to the profile.



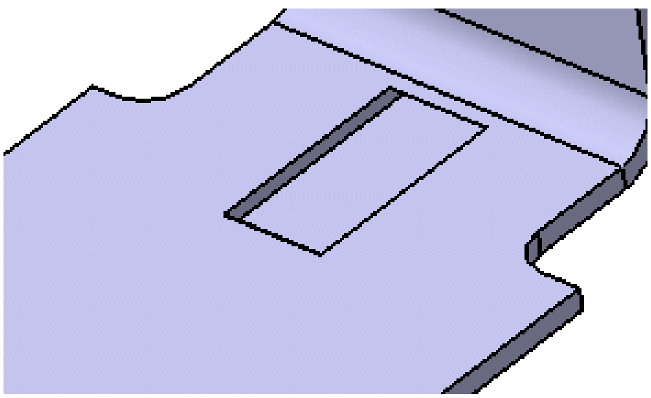
7. Uncheck **Normal to profile**.

8. In the **Reference** field, right-click on Sketch.1 and select Create line.

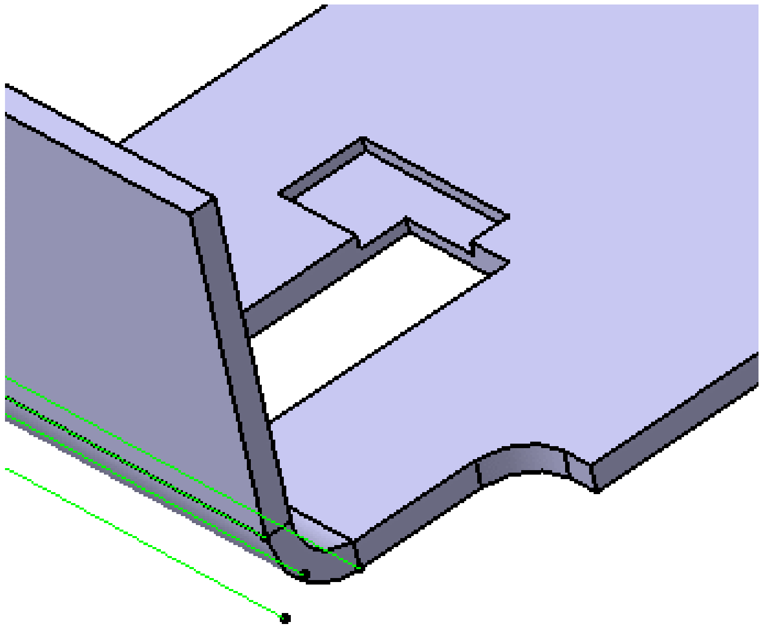


9. Select the line to perform a cutout normal to the line direction.

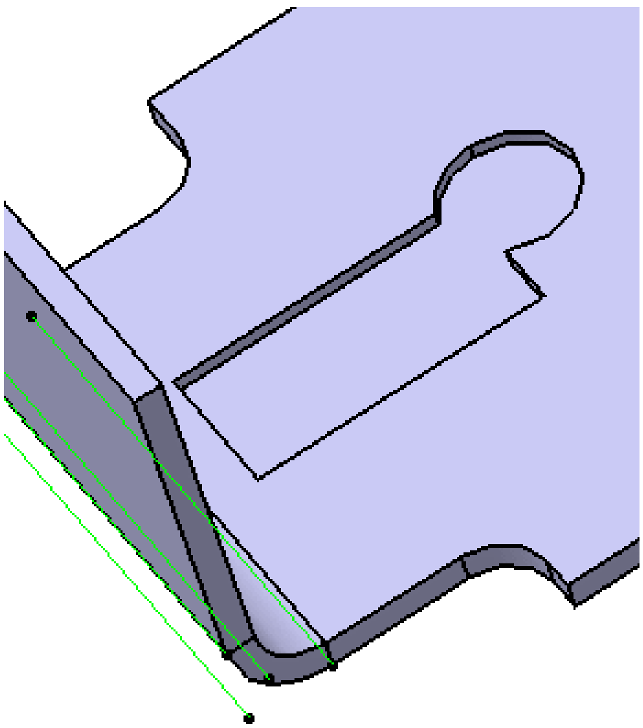
10. Click on OK to create a cutout normal to the line direction.



- The pocket cutout can be created only on a planar and monosupport surface (i.e. a web a flange or the planar face of a surfacic 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,



Once the **Reference Direction** and the **Objects Support** fields are filled in, the selection can be modified but cannot be cleared.

Cutouts can be created directly on the unfolded view of the part.

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

- Refer to *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.

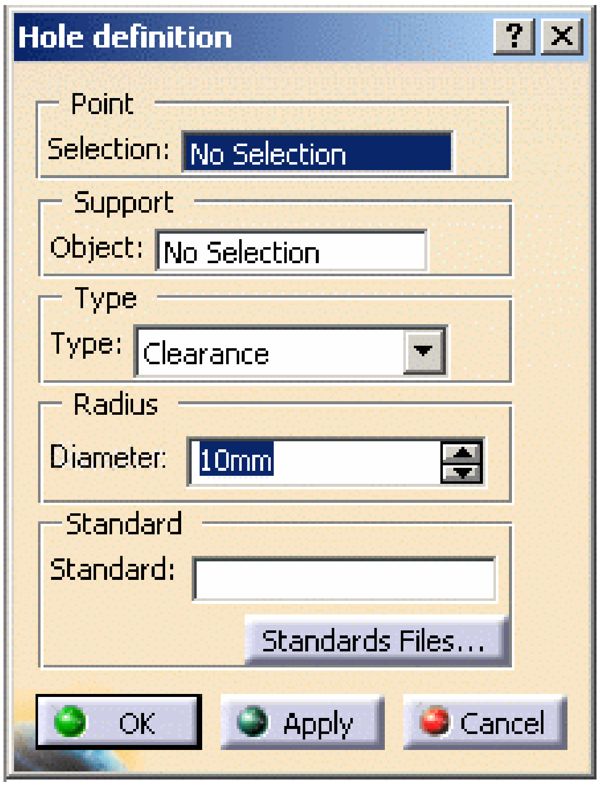
Creating a Hole

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.

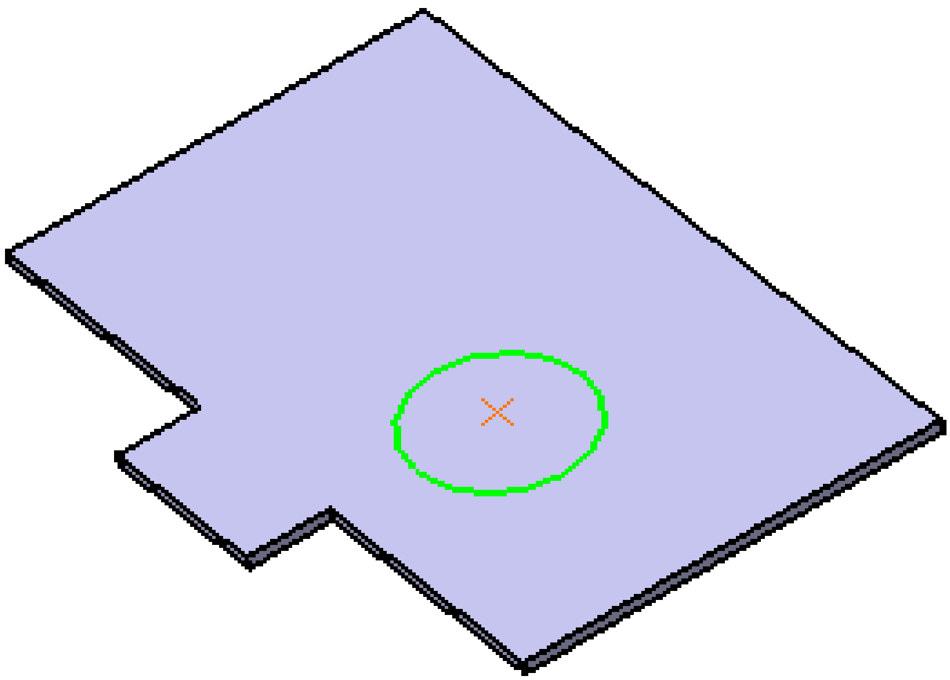
   It can be either a sketch containing one or more points, or a point, or several points.

- The 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. Select hole type:

- Clearance: defined with a center (point) and a radius
- Index: used to measure and validate parts
- Manufacturing: used for manufacturing (for example to fasten a part on an equiment
- Fastener: used as a rivet

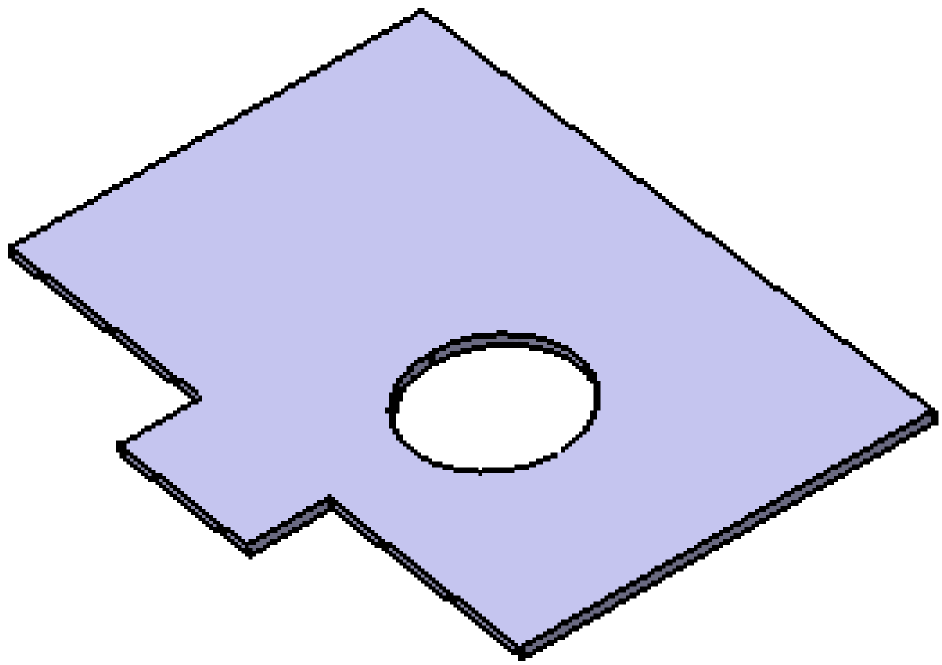
Hole types do not affect the hole geometry.

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

If you change the **Radius** 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.

6. Click OK to validate.

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

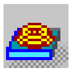


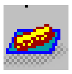
- To have further information on **Standard Files...**, please refer to the Customizing section.

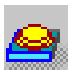
Holes can be created on the flattened part and on the bend in case of a flange.

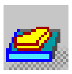
Creating Stamping Features

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 flanged cutout: select a profile, and set the stamping parameters.

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

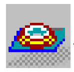
 Create a curve stamp: select a sketch, and set the stamping parameters.

 Create a user-stamp: select a face, and set the stampings parameters.

Creating a Flanged Hole

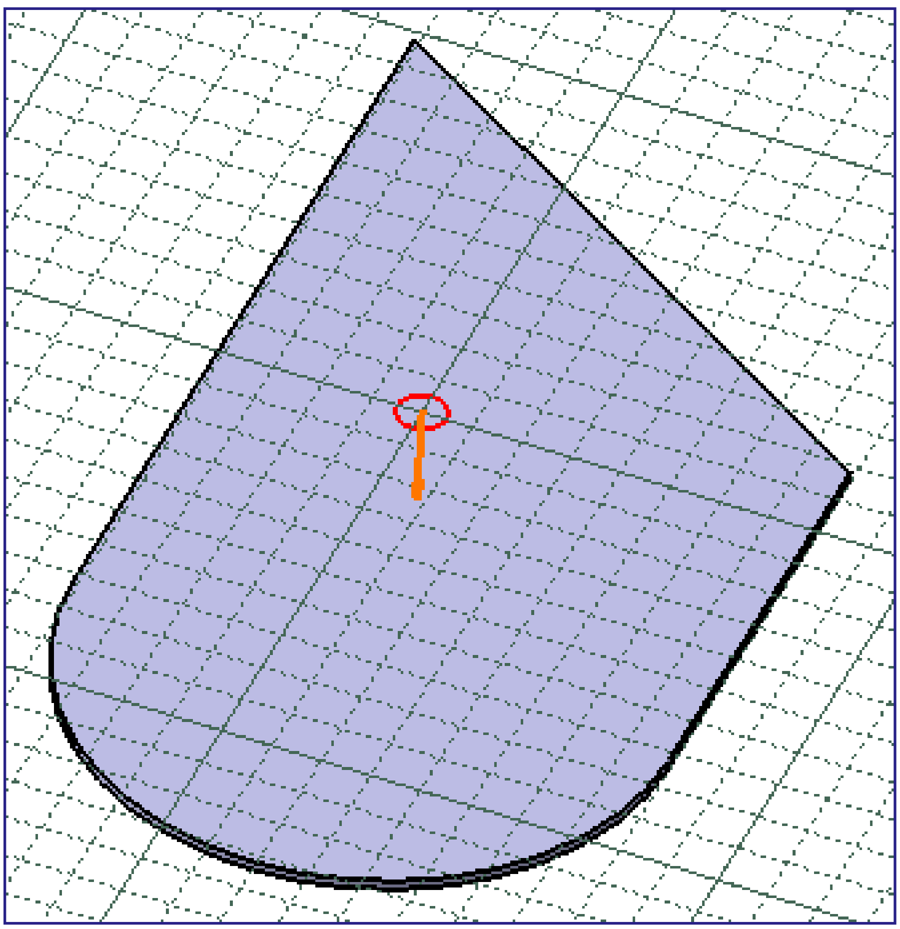
This task shows you how to create a flanged hole by specifying the punch geometrical parameters.

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.

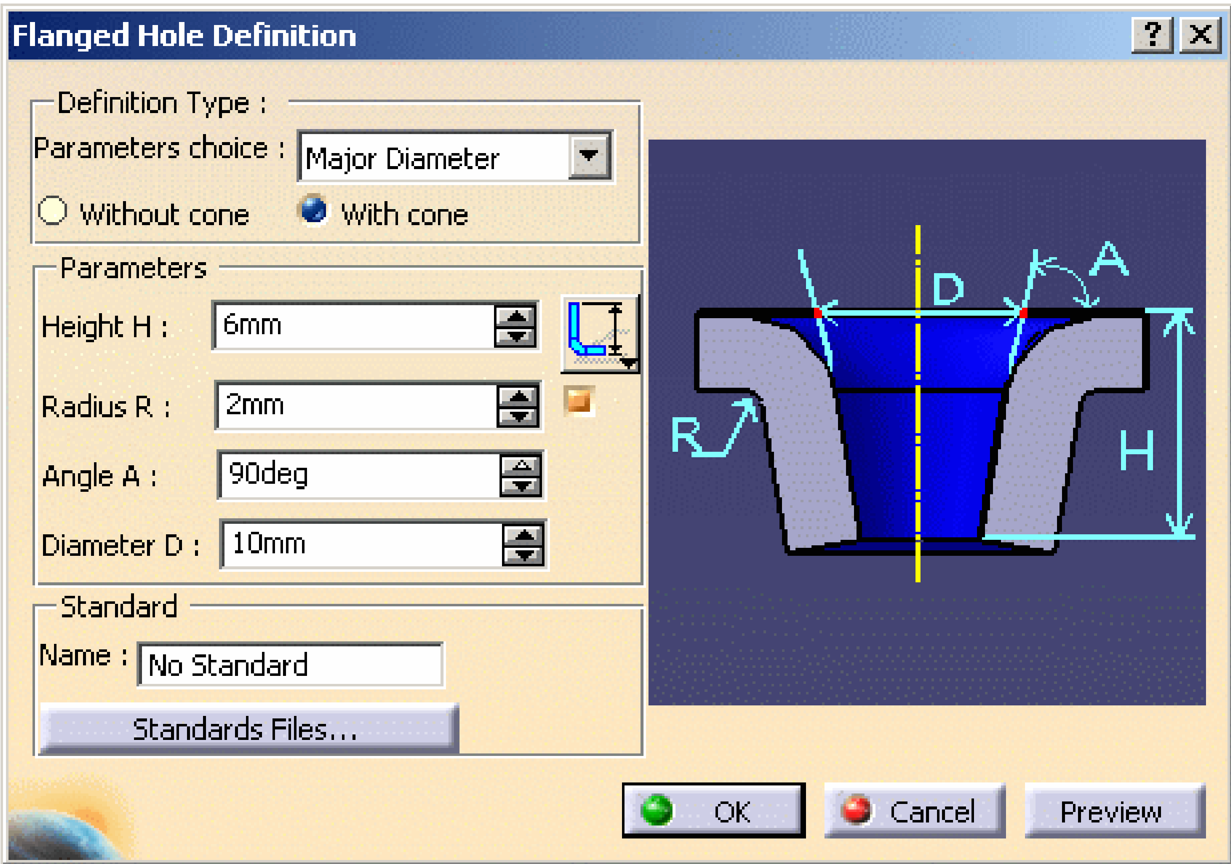
   A grid is displayed to help you position the flanged hole.



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:

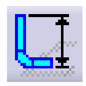
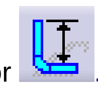
- **Major Diameter**
- **Minor Diameter**
- **Two Diameters** (major and minor diameters)
- **Punch & Die**

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.

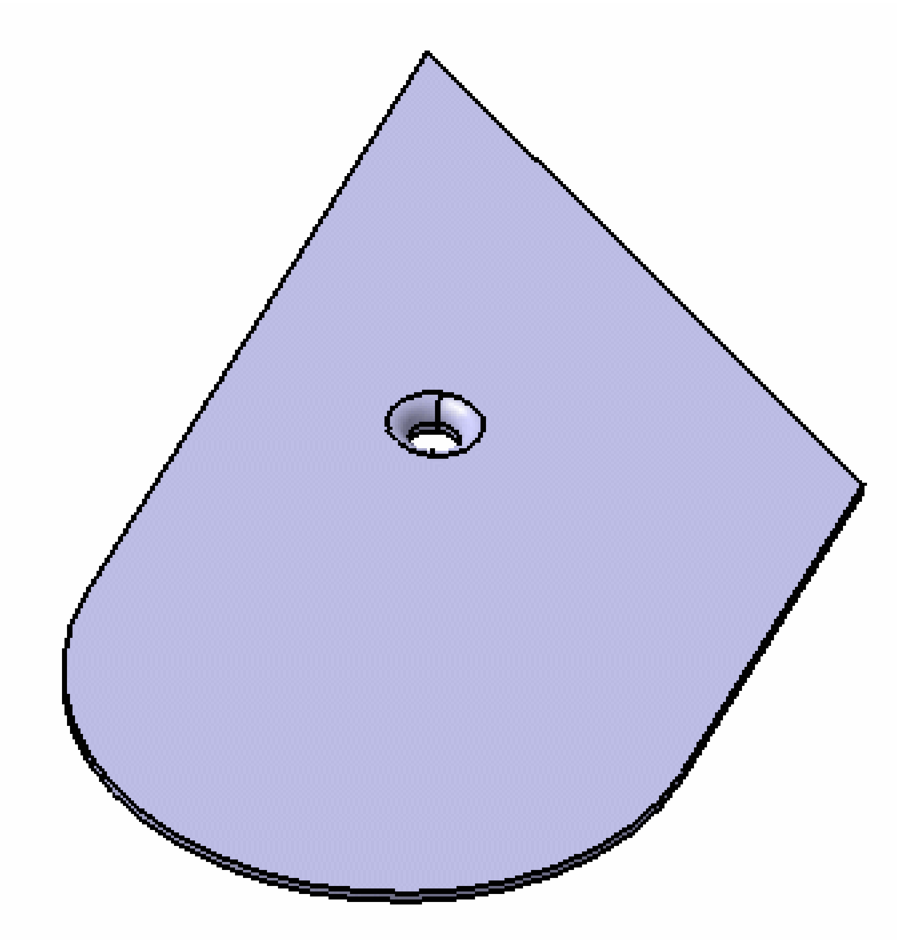
- In the **Diameter d** field, specify the minor diameter value for the flanged hole.

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

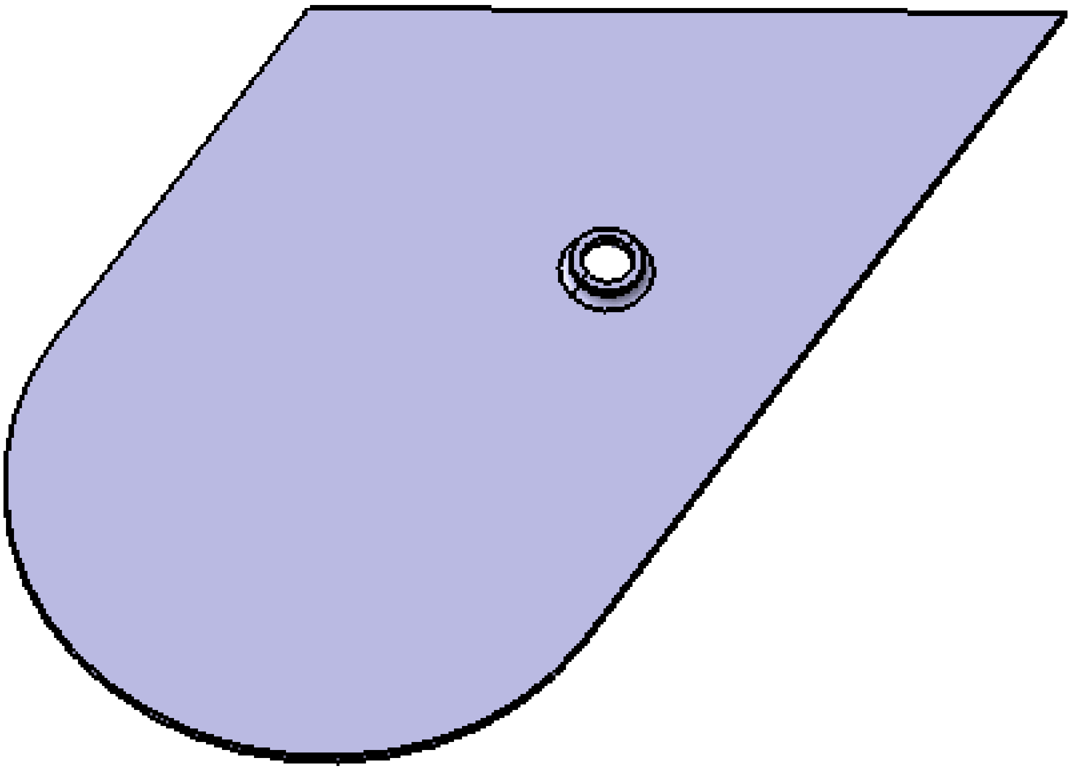
7. Click **Preview** to visualize the flanged hole.

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

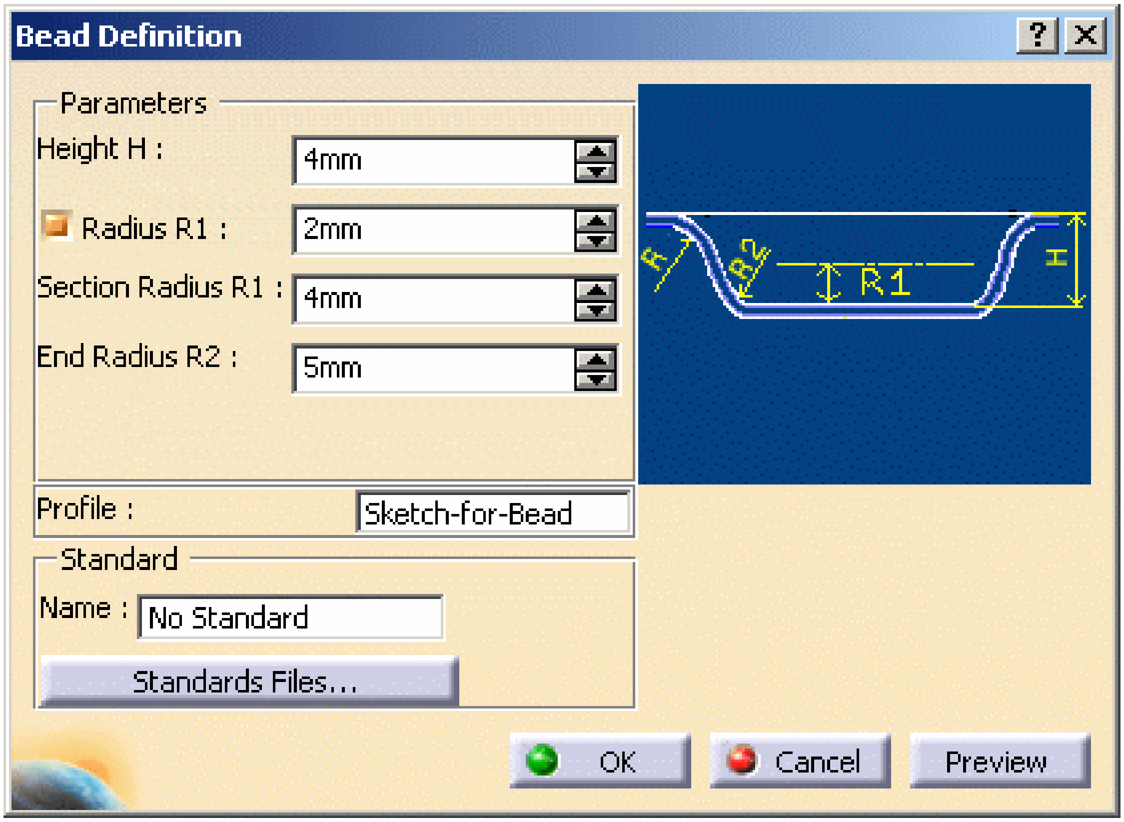
This task shows you how to create a bead, that is a local deformation in the web.

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.

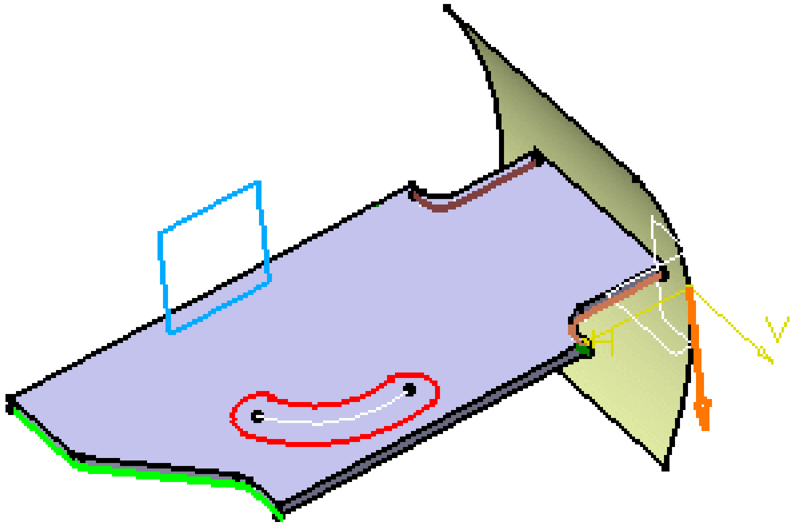


3. Change the value in the different fields, if needed:

- 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.

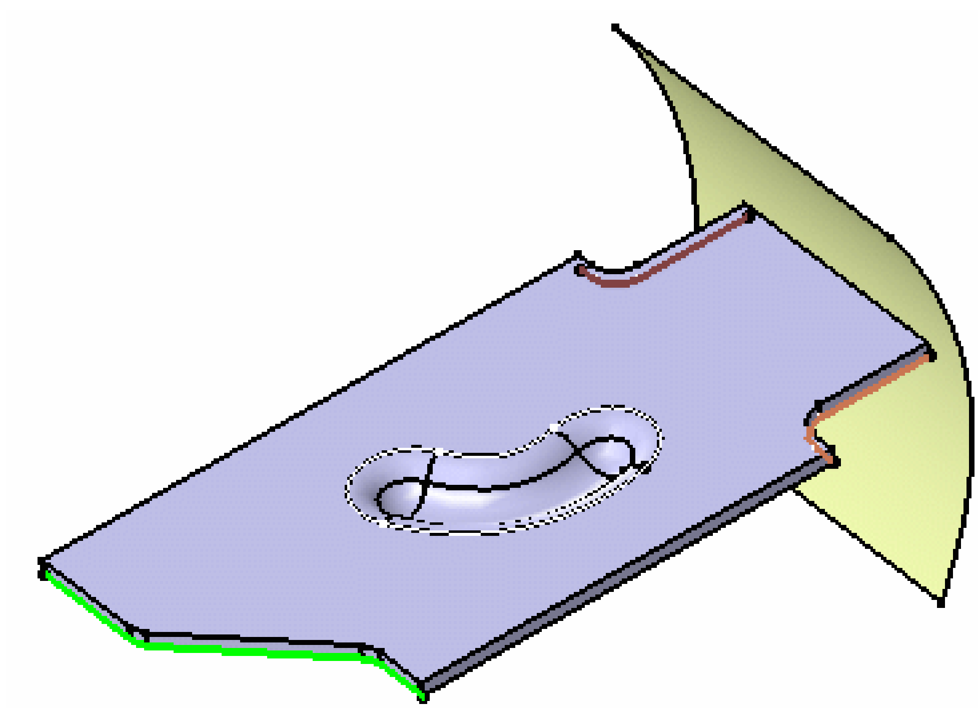
The vector for the direction of the bead is shown in the model and a preview of the bead appears and a vector shows its direction.



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

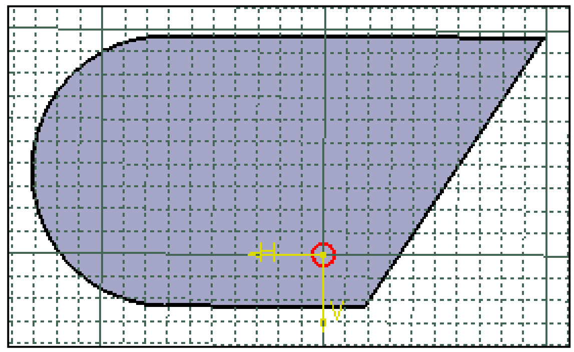
This task shows you how to create a point stamp by specifying the punch geometrical parameters.

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 .

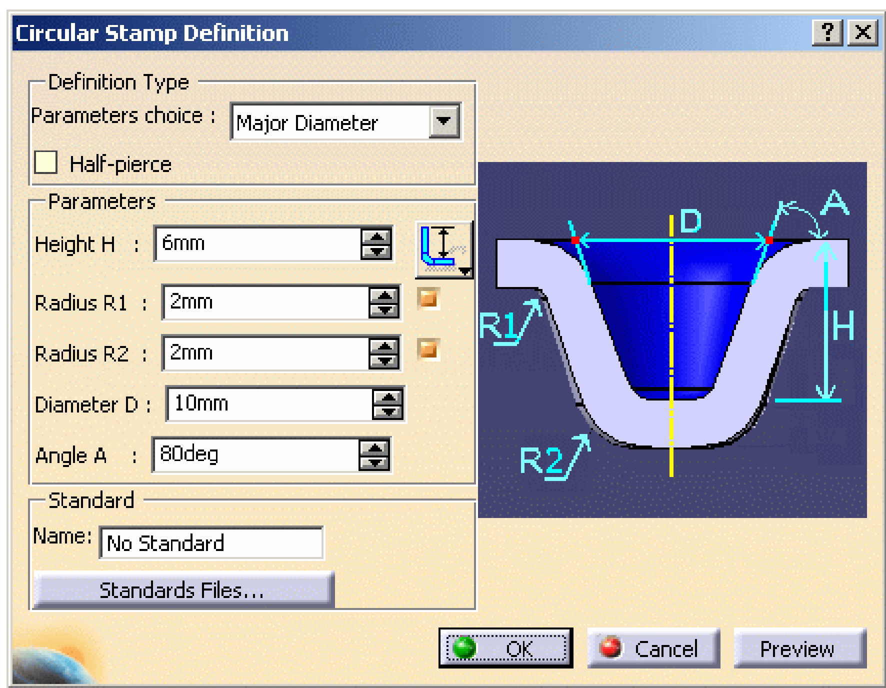
2. Select a point on the top face.



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

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

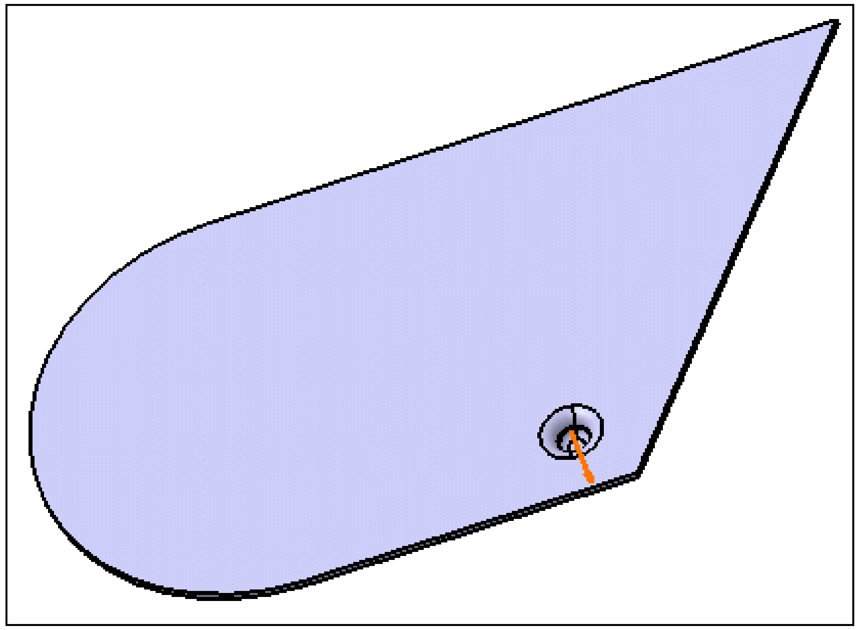
3. Choose the diameter that should be dimensioned from the **Parameters choice** list:
   - **Major Diameter**
   - **Minor Diameter**
   - **Two Diameters** (major and minor diameters)
   - **Punch & Die**



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

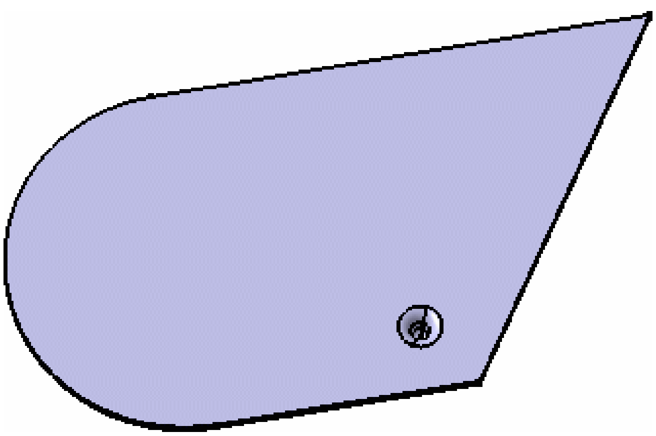
- Height H: use the icon next to the field to specify the reference from which the height is defined: or .
- Radius R1
- Radius R2
- Angle A
- Diameter D

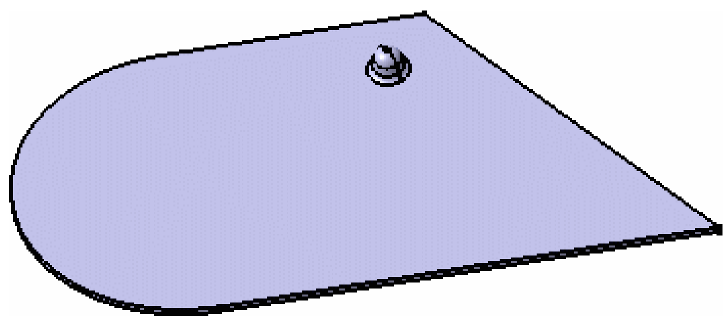
4. Click **Preview** to visualize the circular stamp.



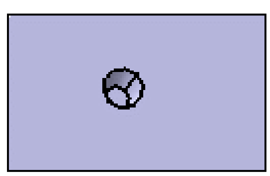
5. Click **OK** to validate.

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 .

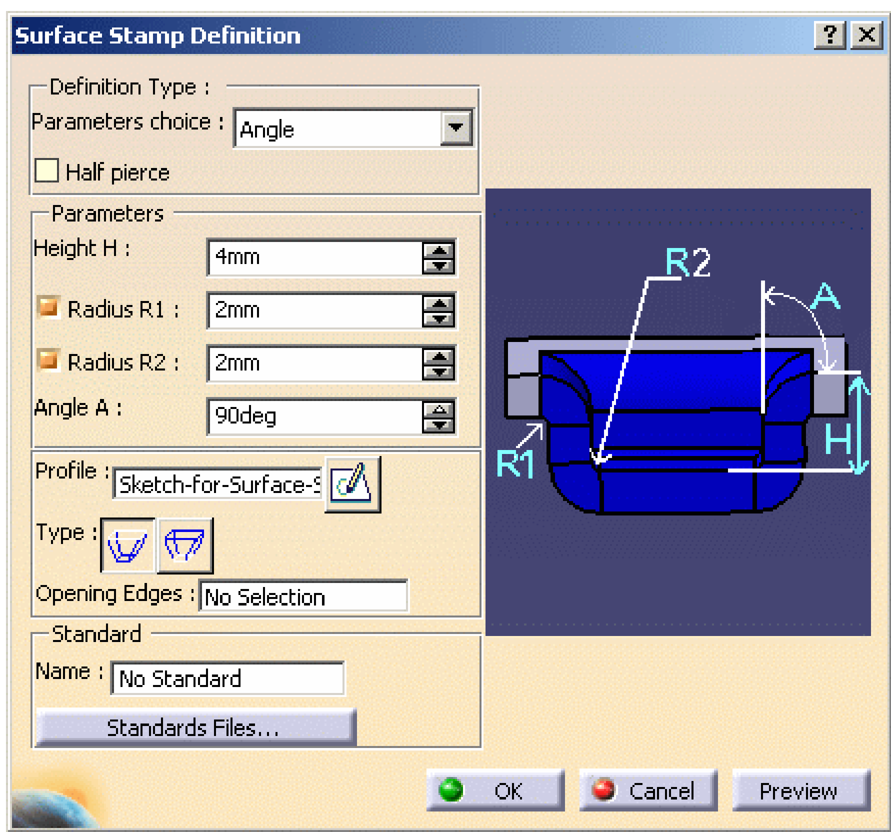
2. In the specification tree, select **Sketch-for-Surface-Stamp**, the profile previously defined.

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

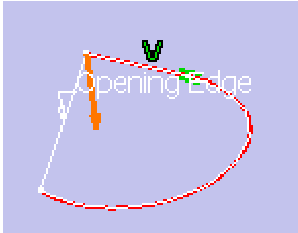
3. Change the value in the different fields, if needed.

   In our example, we chose the following values:

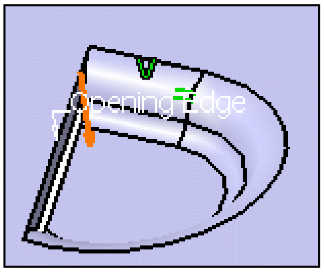
- Height H: 4mm
- Radius R1: 2mm
- Radius R2: 2mm
- Angle A: 90deg



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

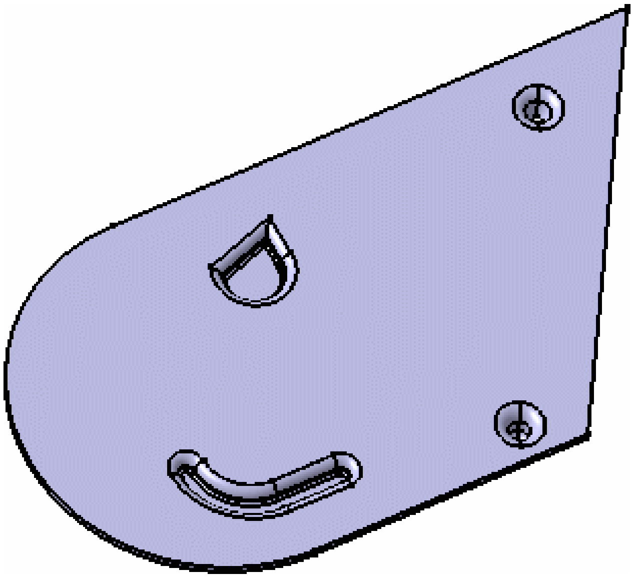


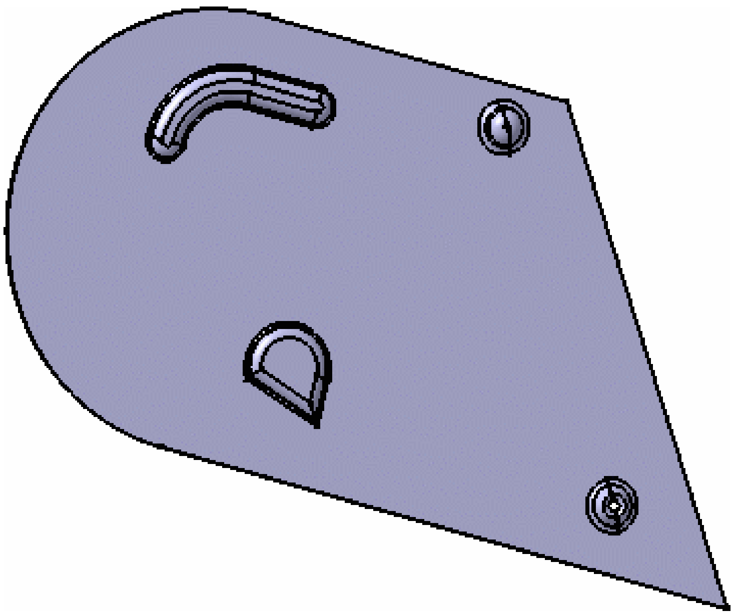
5. Click **Preview** to visualize the surface stamp with an opening edge.



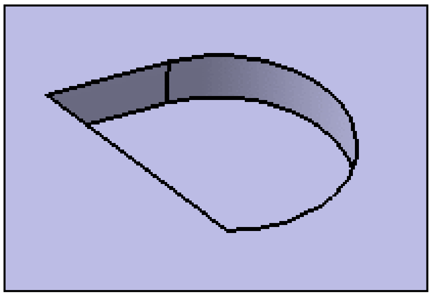
6. Click **OK** to validate.

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.

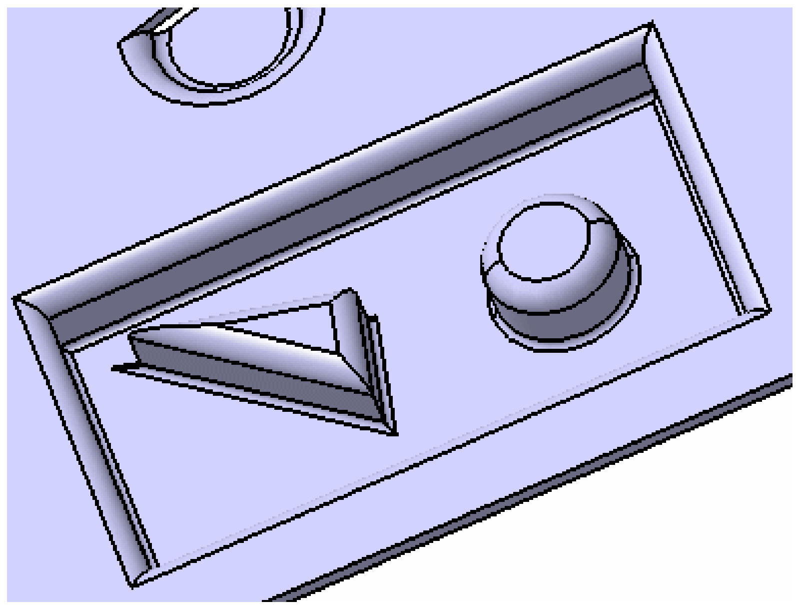
3. Select Angle as Definition type and the first type of stamp 

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.

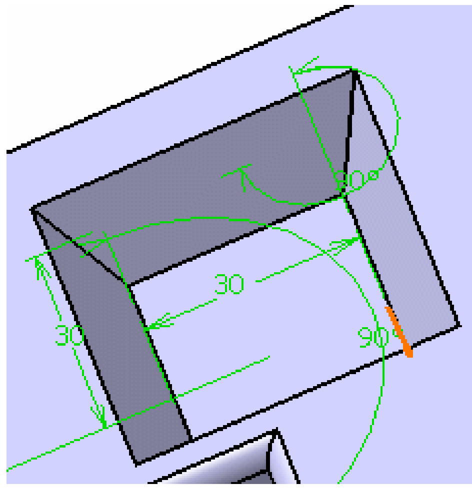
3. Select Angle as Definition type and the second type of stamp 

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

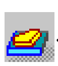
5. Click **Preview** to visualize the surface stamp.



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.

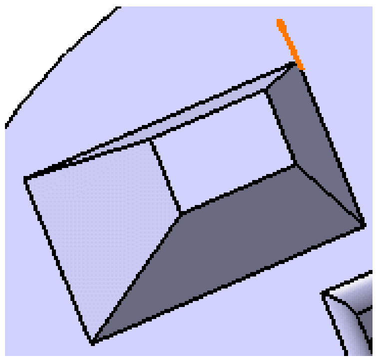
 When **Punch and Die** is selected, only the second type of stamp is enabled 

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

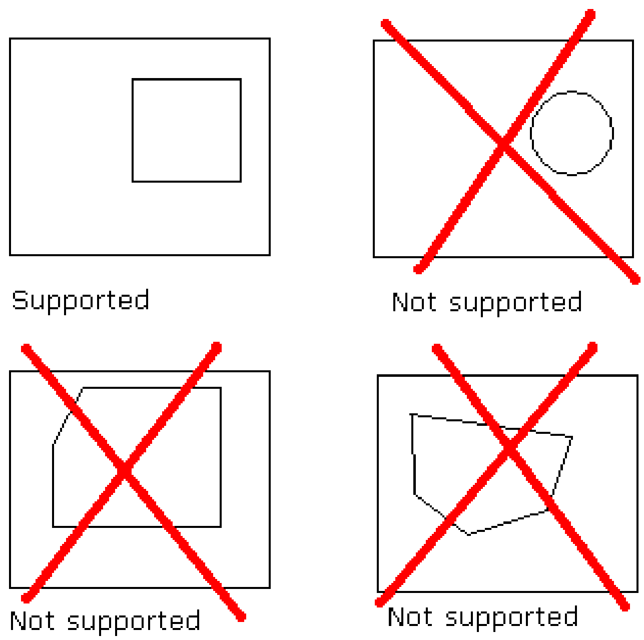
5. Click **Preview** to visualize the surface stamp.



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**.

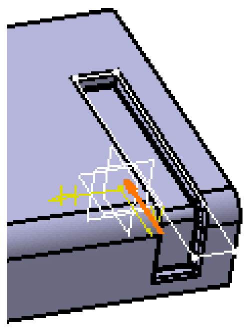
The Surface Stamp Definition dialog box opens.

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.

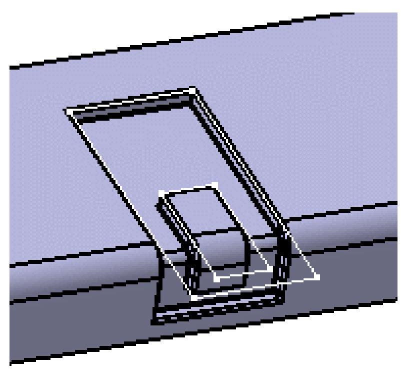


5. Click **OK** to validate.

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 Flanged Cutout

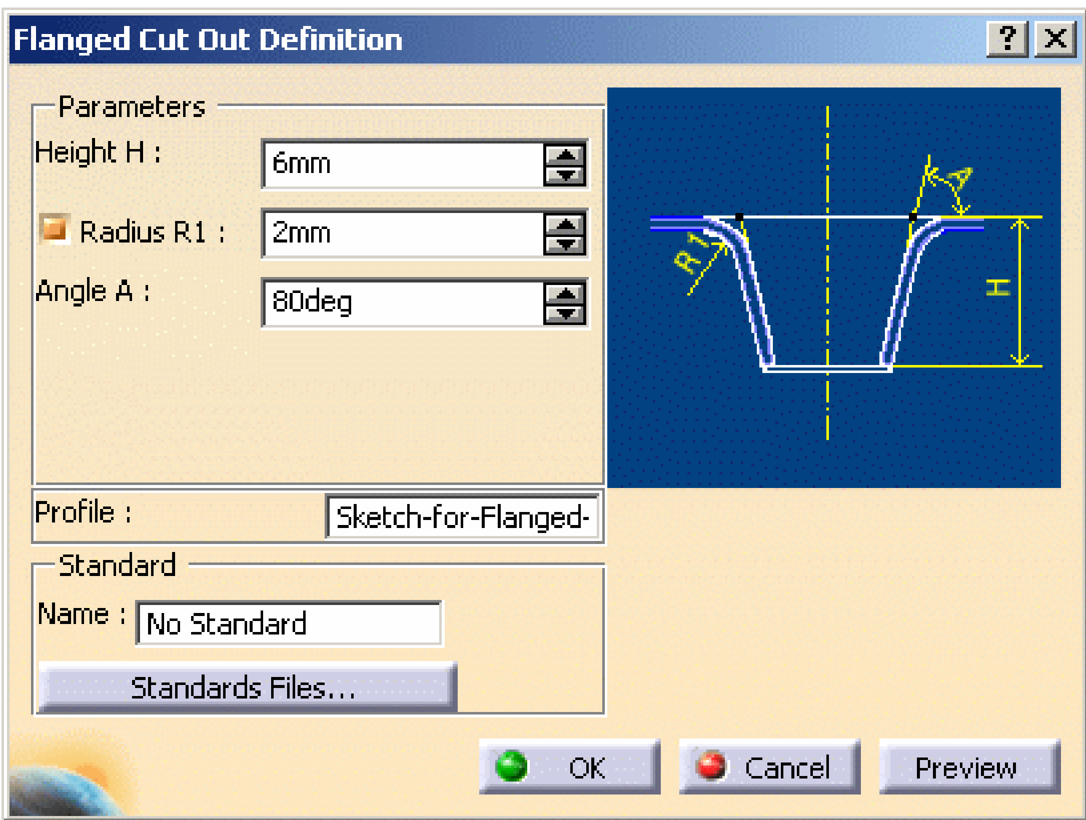
This task shows you how to create a flanged cutout by specifying the punch geometrical parameters.

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.

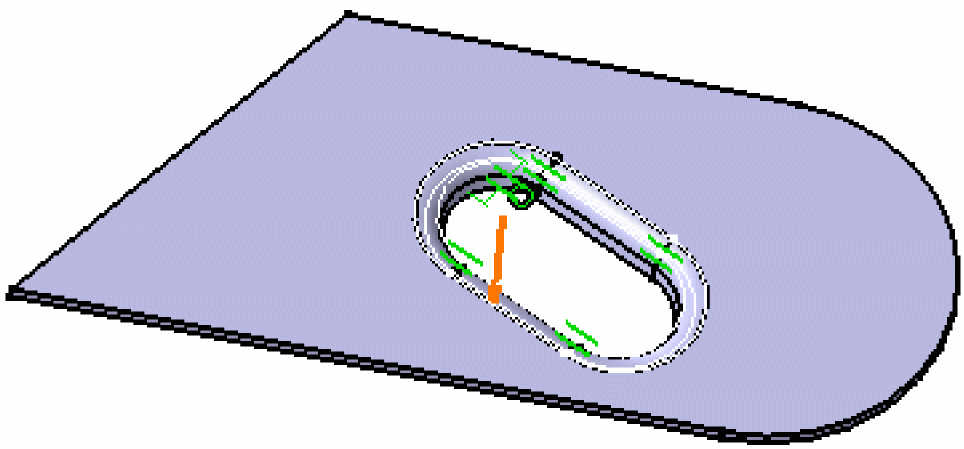
The Flanged Cutout Definition dialog box is displayed, providing default values.



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

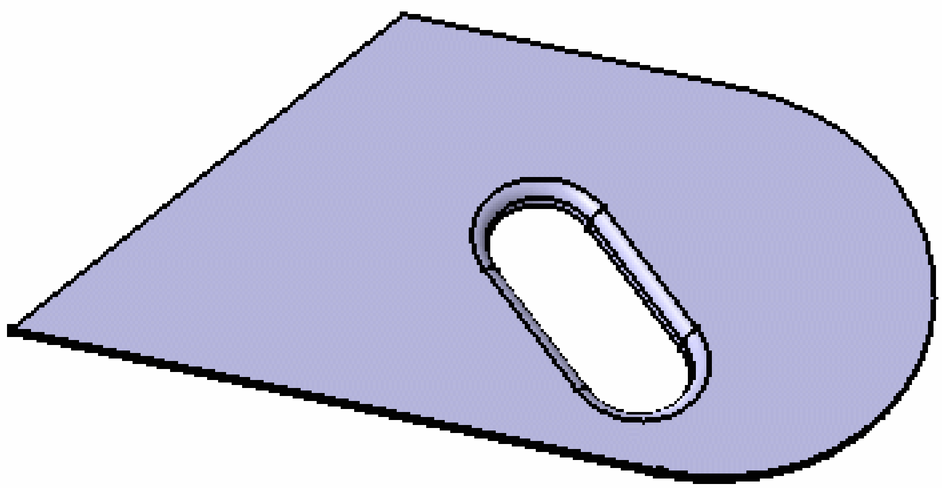
- Height H
- Radius R
- Angle A

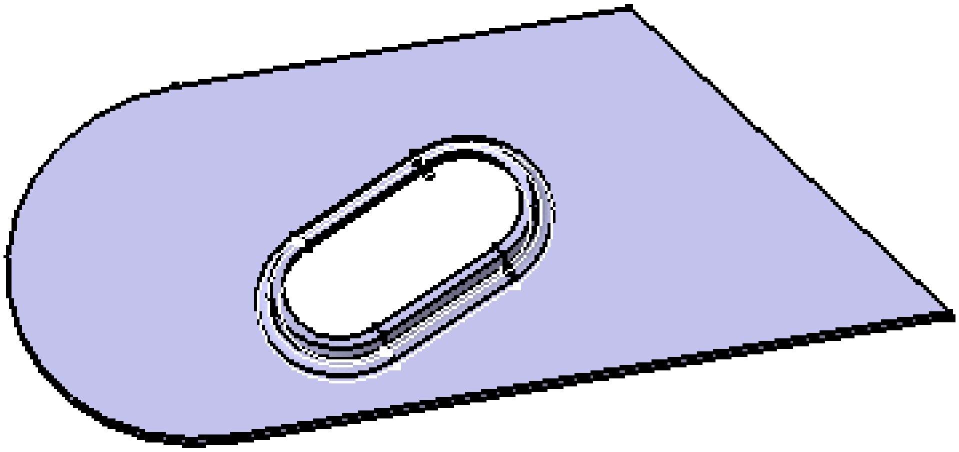
4. Click **Preview** to visualize the flanged cutout.



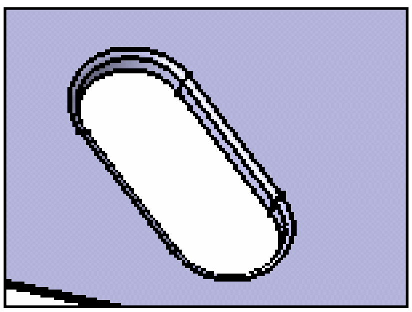
5. Click **OK** to validate.

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.

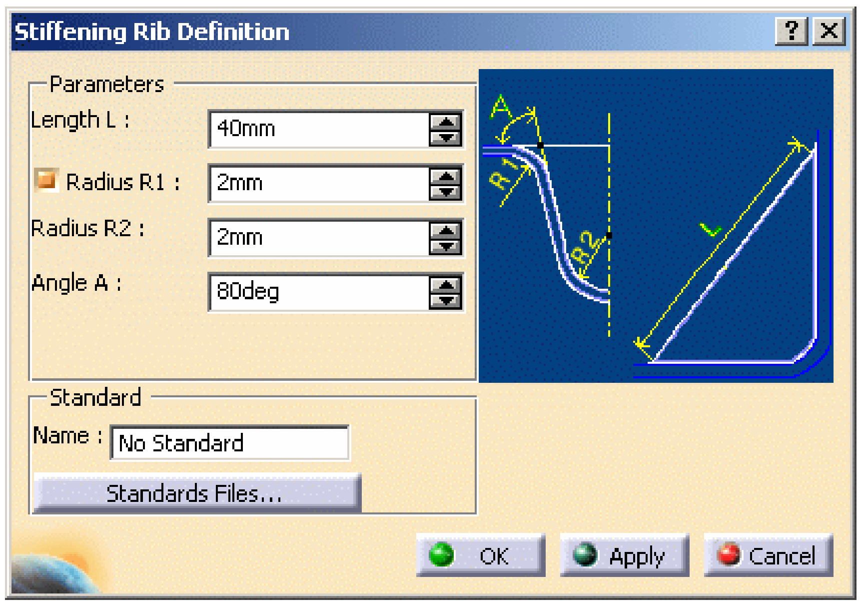
1. Click the **Stiffening Rib** icon .

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.

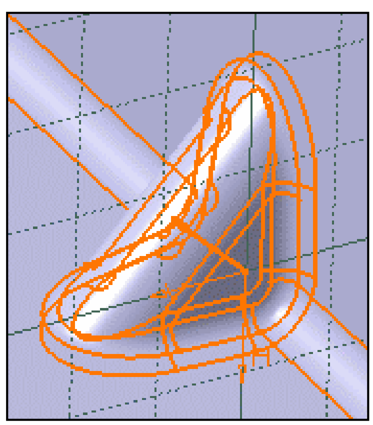
The Stiffening Rib Definition dialog box opens, providing default values.



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

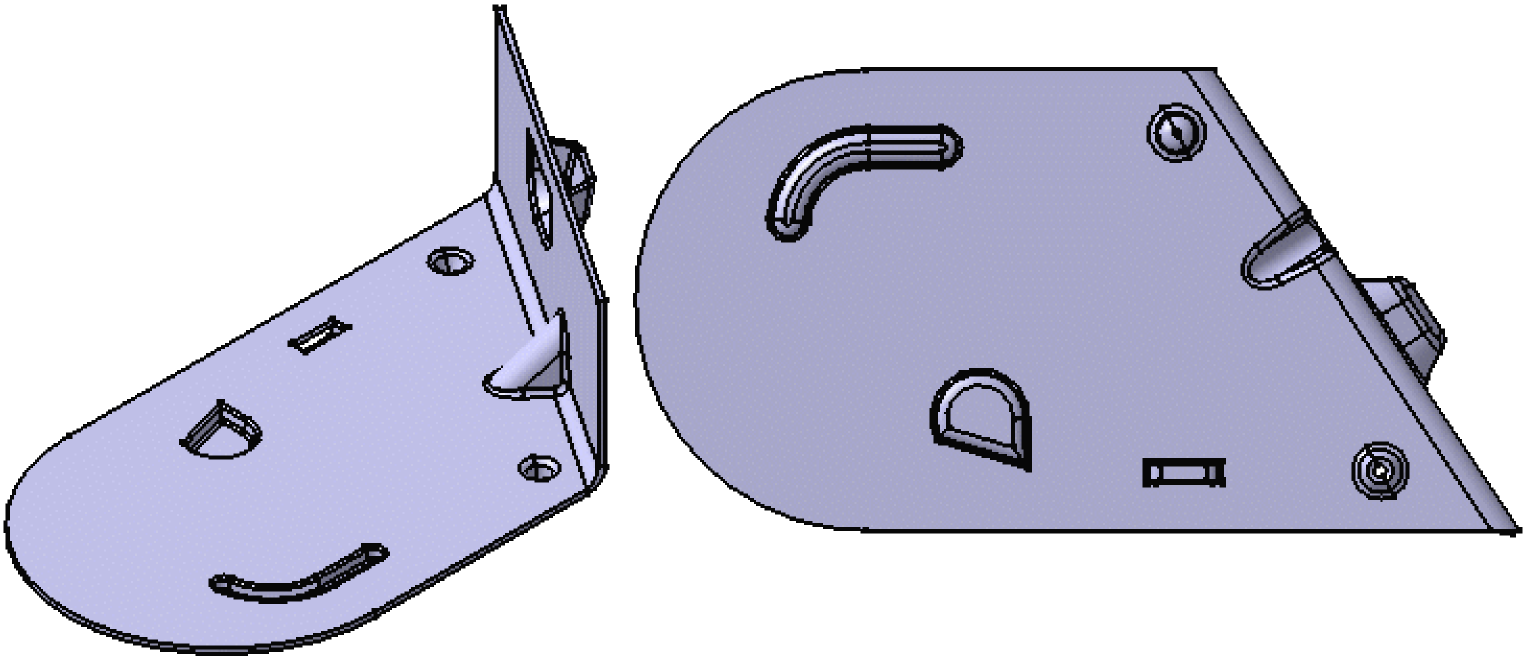
- Length L
- Radius R1
- Radius R2
- Angle A

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

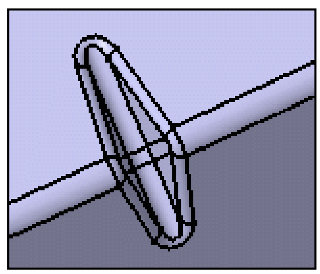


5. Click **OK** to validate.

   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.

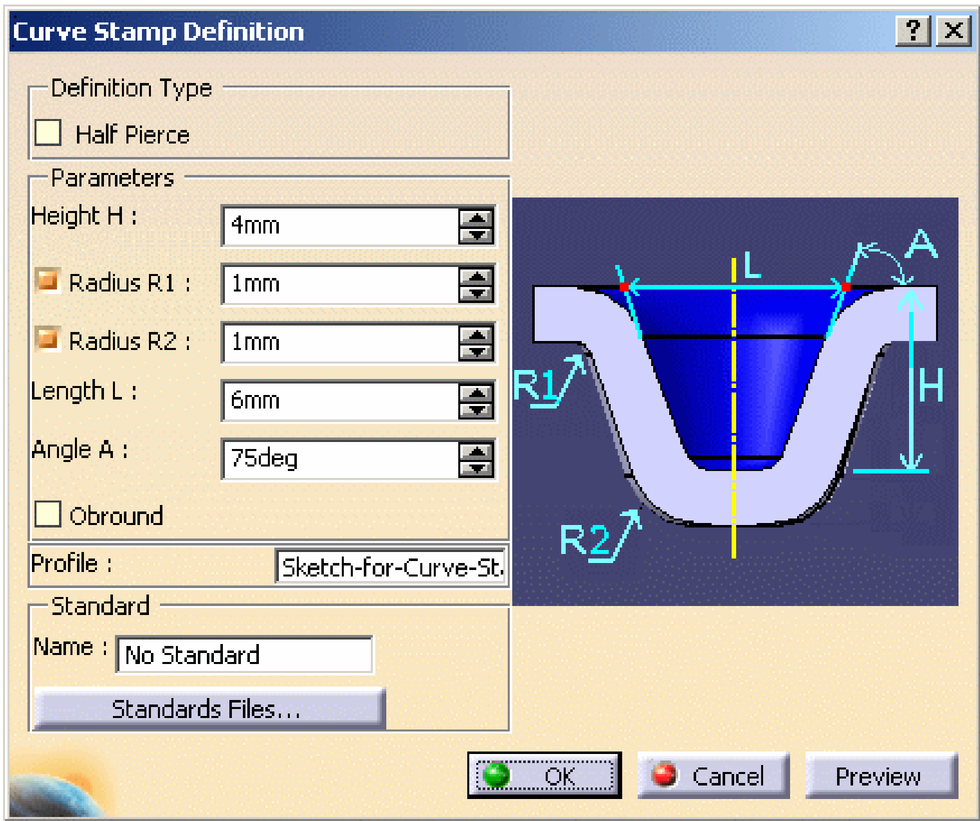
1. Click the **Curve Stamp** icon.

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

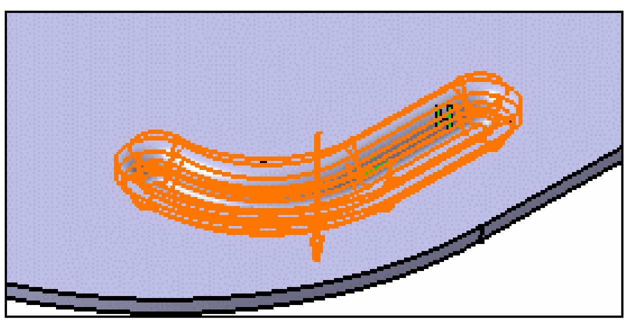
The Curve Stamp Definition dialog box opens, providing default values.

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

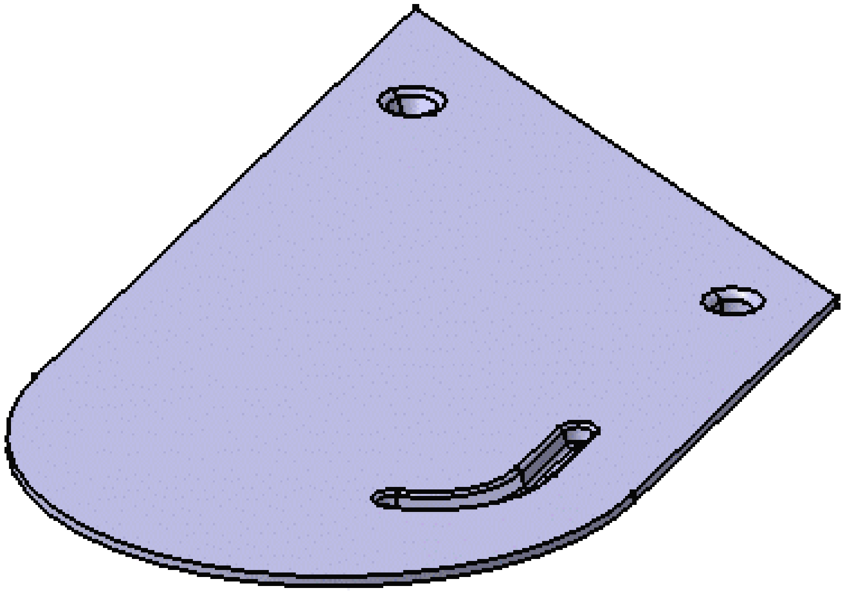


4. Click **Preview** to visualize the curve stamp.

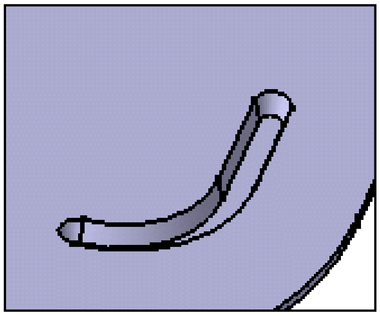


5. Click **OK** to validate.

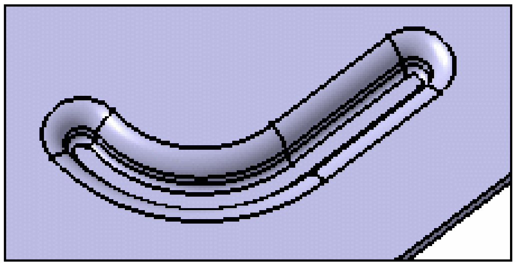
The curve stamp (identified as Curve Stamp.xxx) is created and the specification tree is updated accordingly.



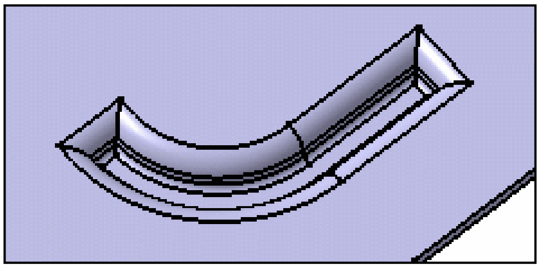
- You can use 0 as the Radius value to deactivate the Radius R and Radius R2 values, and to create the curve stamp without a fillet.



- 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 User-Defined Stamping Features

Two user-defined stamping features are available:

Create a punch with die: define the punch and die features, select a base feature, choose the punch and die as stamping elements, select an edge on the base surface and give an angle for orientation purposes.

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

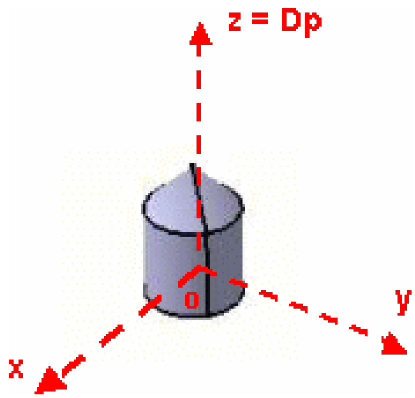
Edit a user-defined stamp: double-click the existing stamp and change its type, or select, or remove opening faces

What You Should Know

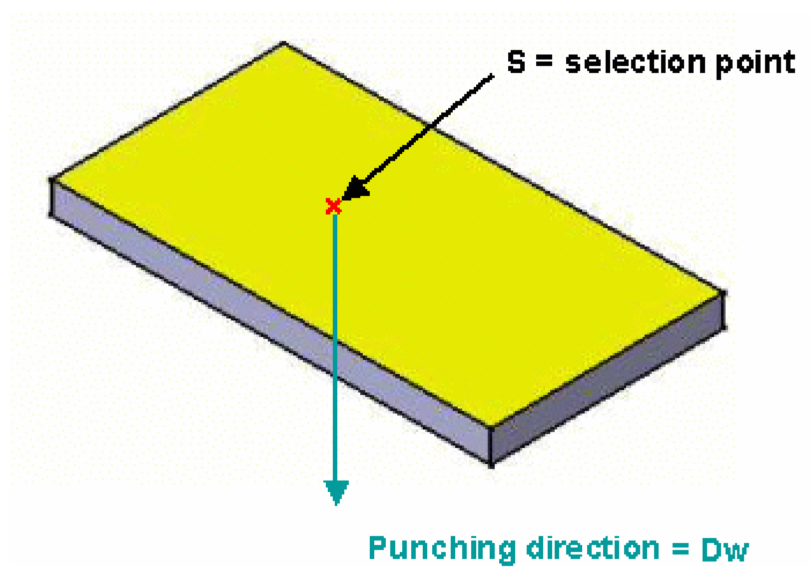
In both tasks illustrating either a stamp based on a punch with die, or a punch with opening faces, the punch positioning is defined as below:

Defining the Punch in Relation to the base feature to be Stamped

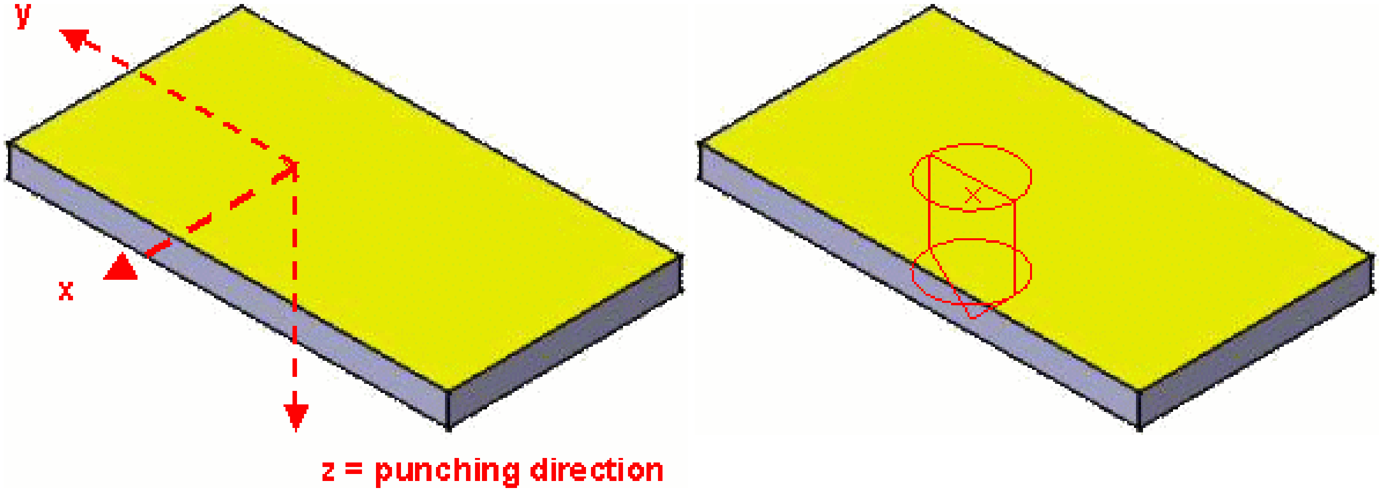
The punch is defined within the absolute (default) axis-system of the .CATPart document. (o, x, y, z) is the axis associated with the punch. The punching direction on the punch (*Dp*) must be equal to z.



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



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

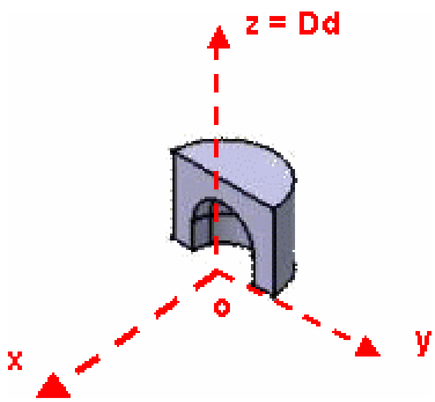


Defining the Die in Relation to the base feature to be Stamped

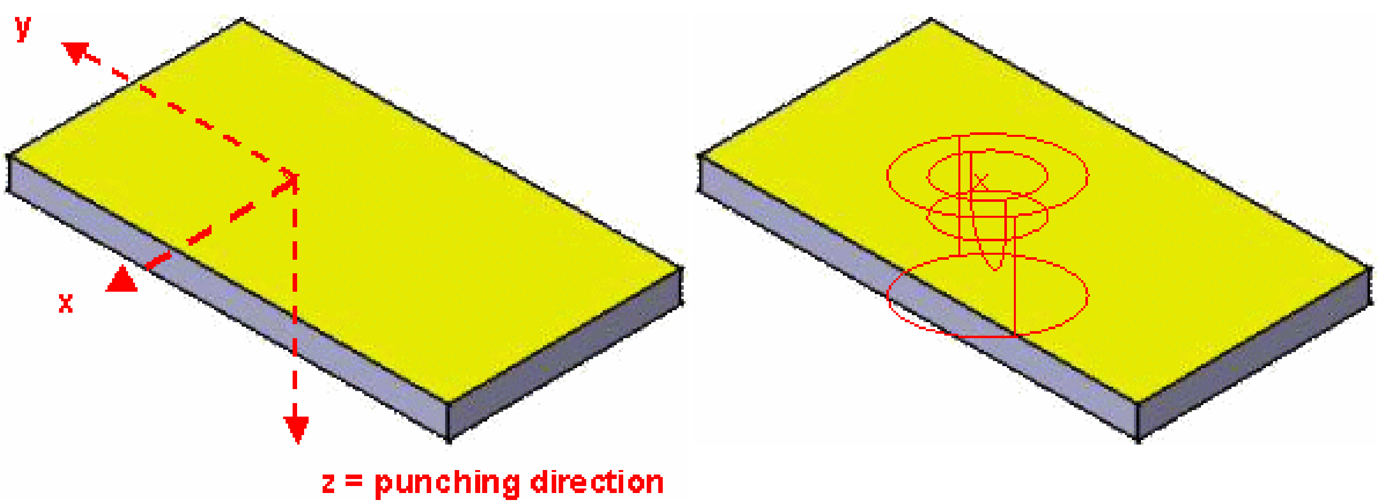
This is useful only when defining a punch with die, and does not apply to punches with opening faces.

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 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.

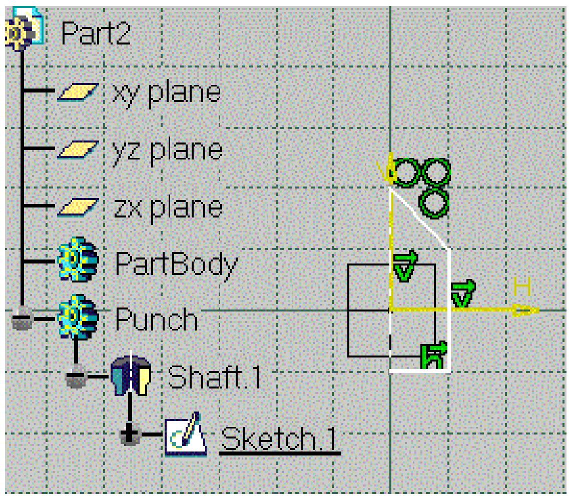
This user-defined stamping cannot be combined with the Opening and Cutting Faces approach.

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.

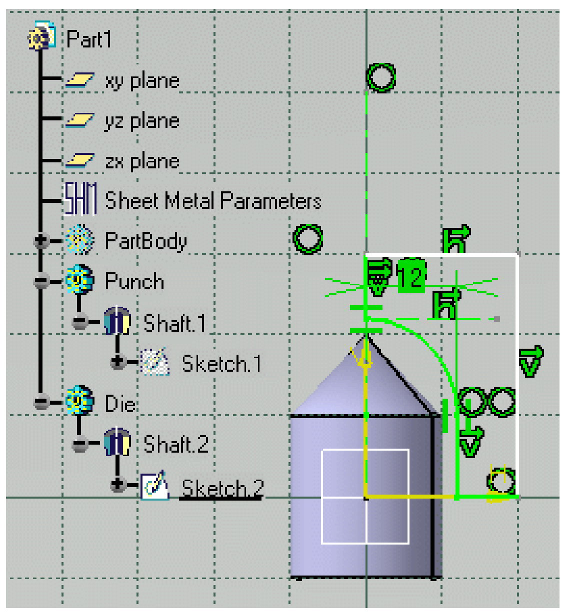
2. Insert a PartBody (menu **Insert -> Body**) to define the punch.

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 .

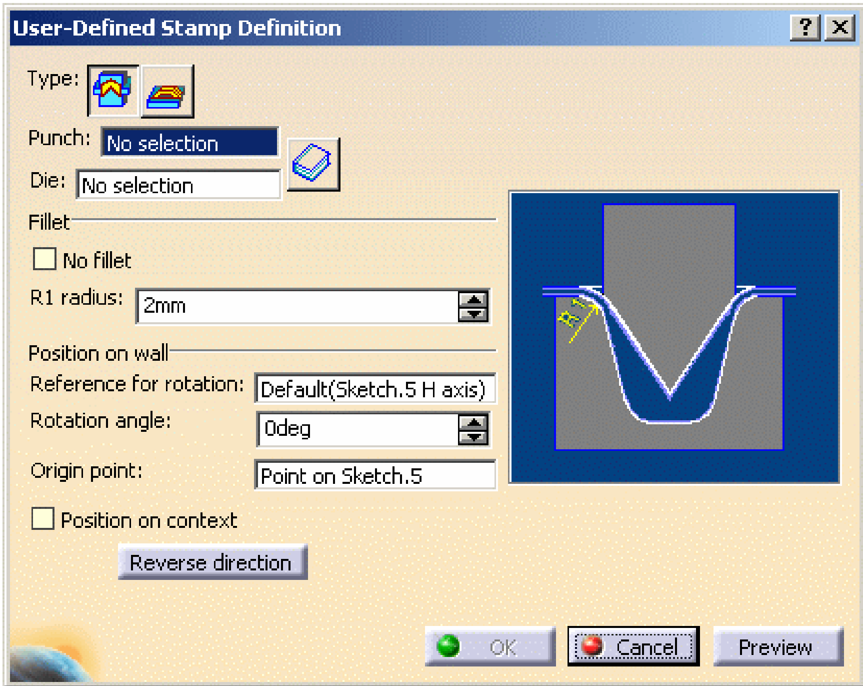


5. Repeat from step 2 to step 4 to define the die, making sure that it is oriented as described in Defining the Die in Relation to the base feature to be Stamped.

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.

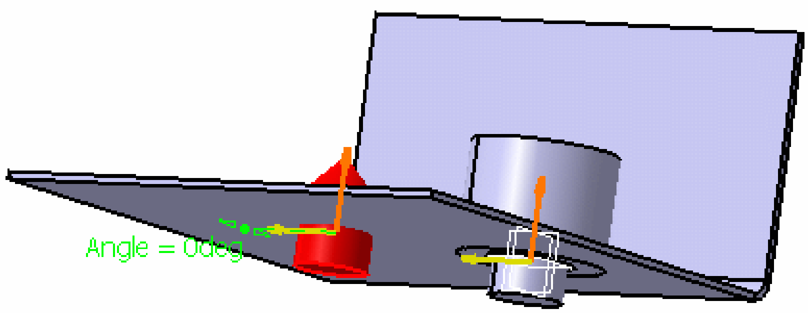
7. Click the **User Stamping** icon from the Stamping tool bar and select a base feature, or a face where the stamping is to be created. This base feature or face is used to define the stamping location and direction, by matching the punch's origin to the selected point on the base feature.

The User Defined Stamp Definition dialog box is displayed:



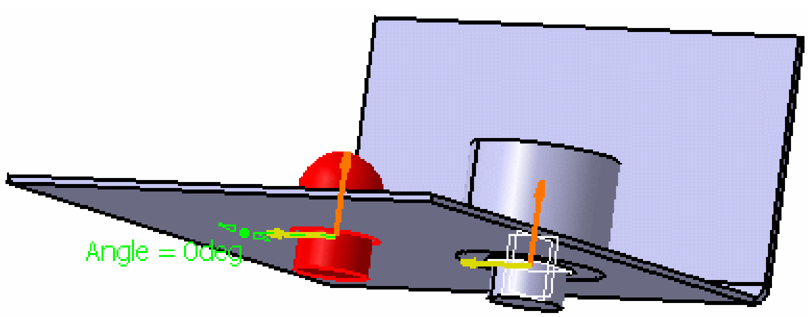
8. Make sure the **With die** icon is pressed down and select the Punch feature from the specification tree.

The punch's positioning is previewed in the geometry.

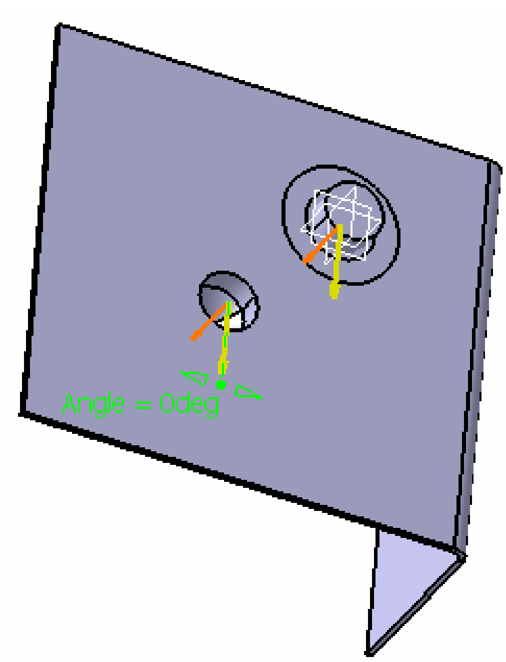


9. Select the Die feature.

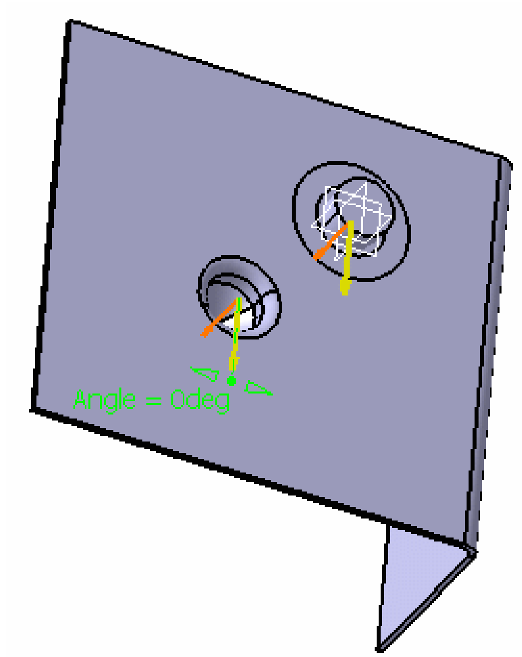
The die's positioning is previewed in the geometry as well.



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

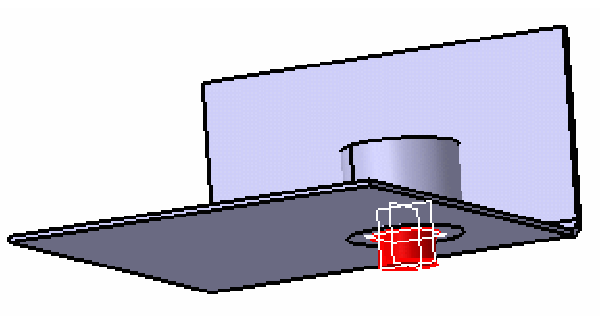
11. If needed, define the stamp's positioning on the selected base feature by choosing:

- 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.

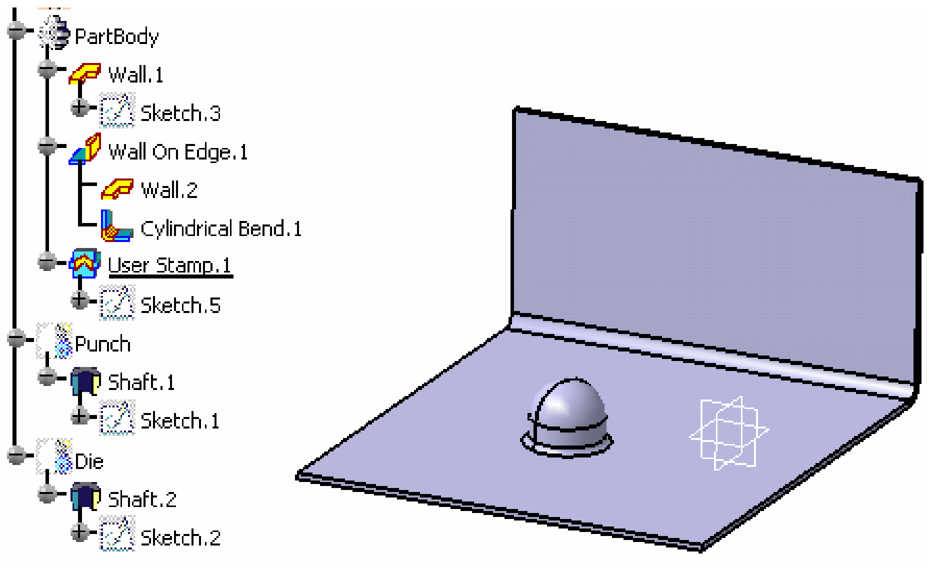


When selecting the **On Context** check box, the stamp's positioning and direction are not defined in relation to the base feature anymore.

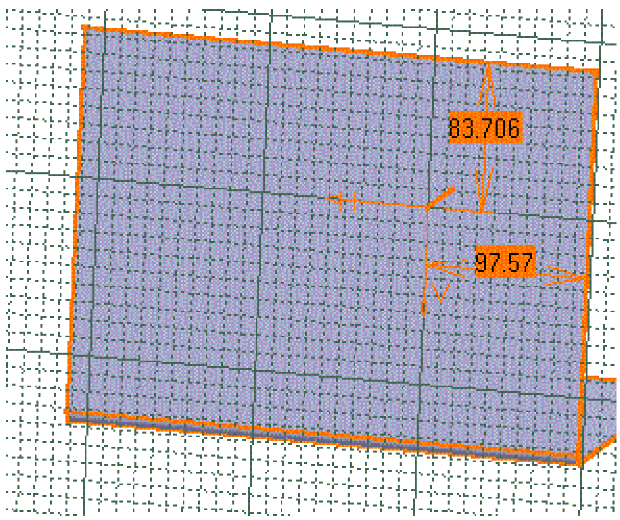
Only the punch and die's axis system is taken into account and the stamp is created according to their positioning and direction.
Once **On Context** is selected, the position on wall cannot be modified nor the direction of the stamp: the fields available in **Position on wall** section and the **Reverse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



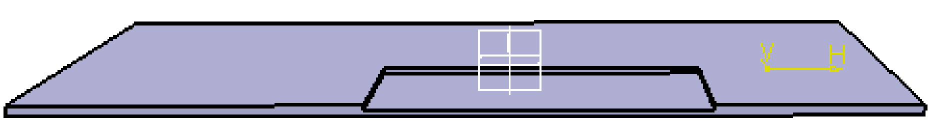
- **Radius** is the radius of the bend between the stamping and the base feature.
- **Punch** and **Die** are the bodies you have defined previously. If the punch and the die are in another CATPart document, activate this document before clicking the punch or the die.
- If you select two reference lines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



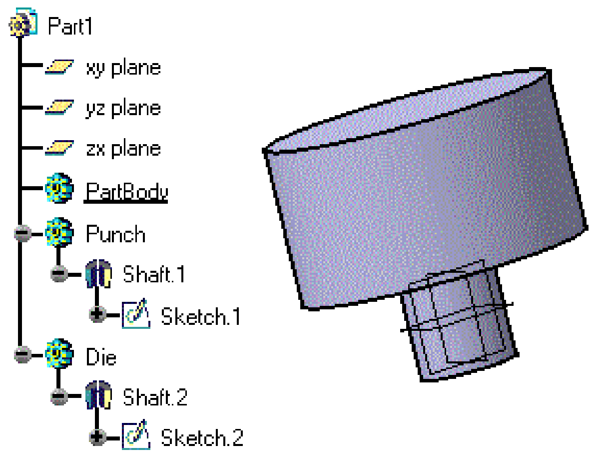
- A user-defined stamping can be edited (punch, die, position, constraints).
- As the punch and die are not symmetrical, you cannot create such features as a cutout, a hole, a corner, etc., on this kind of stamping.

- 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.

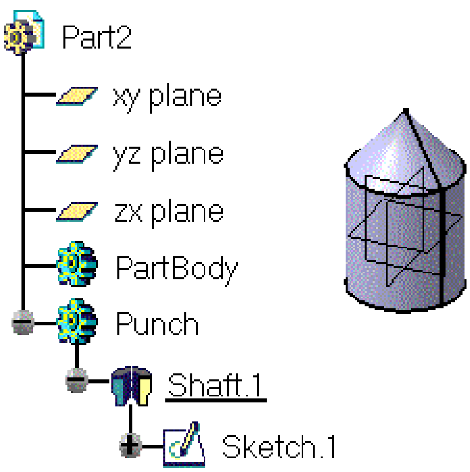


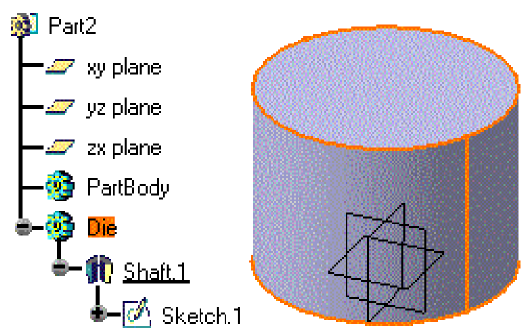
- You may create a user-defined stamping from a punch only but you cannot create a fillet.
- Only the stamping sketch is displayed in unfolded views.
- The punch and die bodies can be defined in the Sheet Metal part where the stamping is to be created (see PunchDie1.CATPart or NEWPunchDie1.CATPart in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

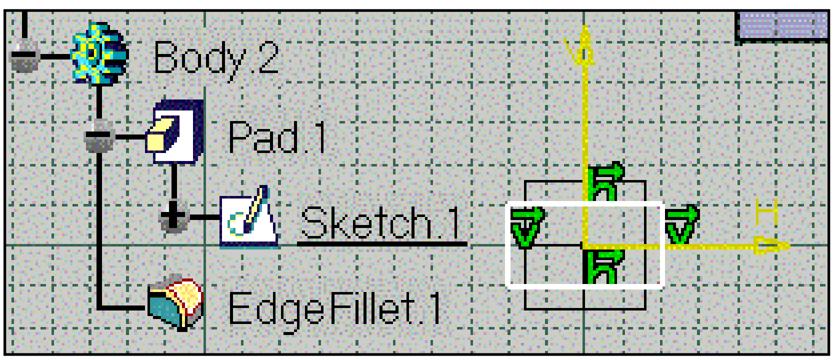
First, you will define a punch in Part Design, in the absolute axis system.
Then, in a Sheet Metal part, you will bring the punch feature (and its axis system) to a point you have selected. If necessary, you will define a rotation of the axis system from a reference line.

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.

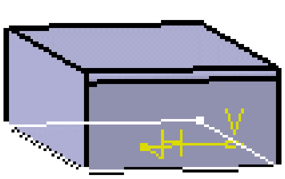
The CATPart documents are available from the samples directory, NEWOpenFaces1.CATPart for is Generative Sheetmetal Design or Aero_OpenFaces1.CATPart for Aerospace Sheetmetal Design.

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.

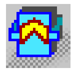


The punch must be oriented as described in Defining the Punch in Relation to the base feature to be Stamped.

- 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.



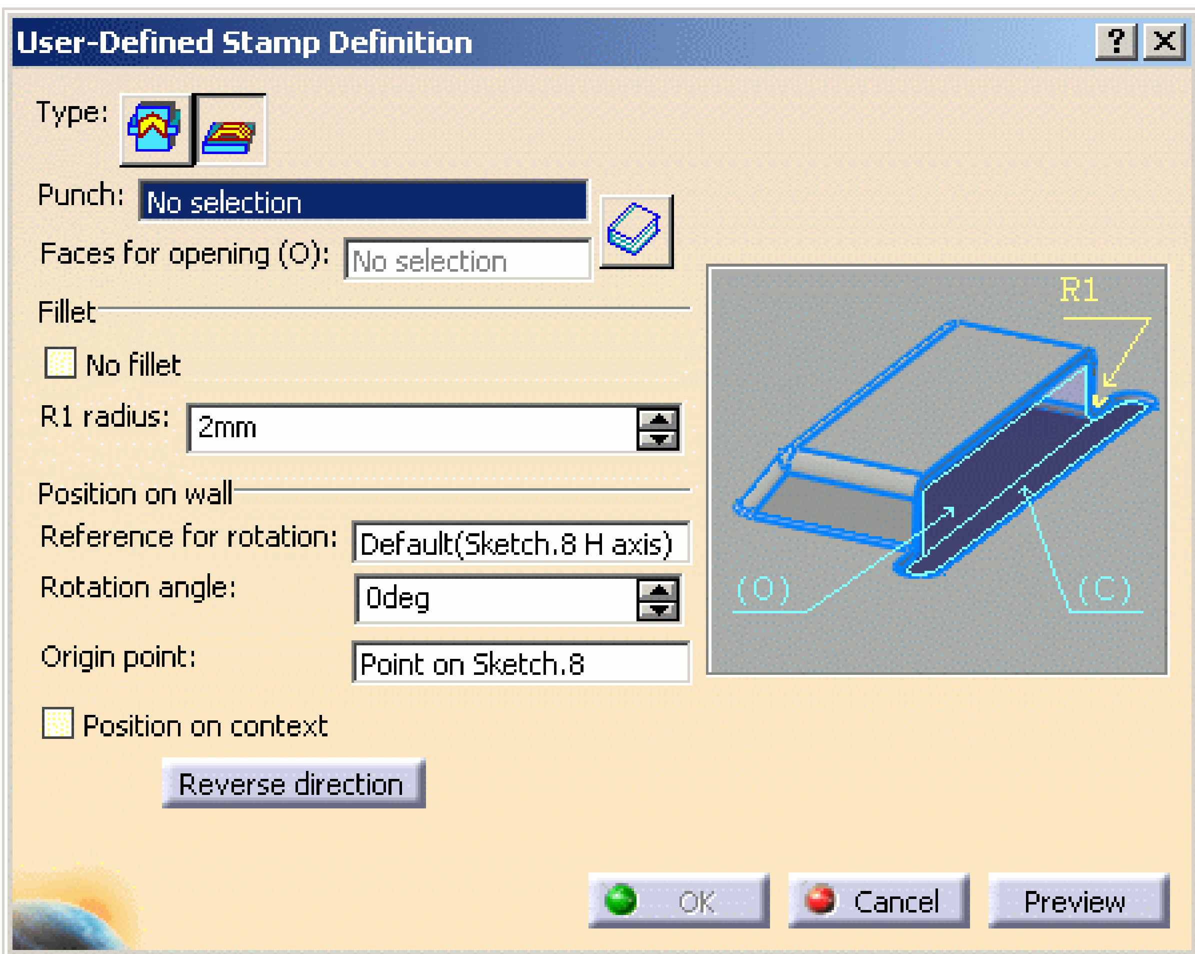
5. Return to the Sheet Metal application, and if needed, use the **Define In Work Object** on the PartBody containing the base feature to be stamped.

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

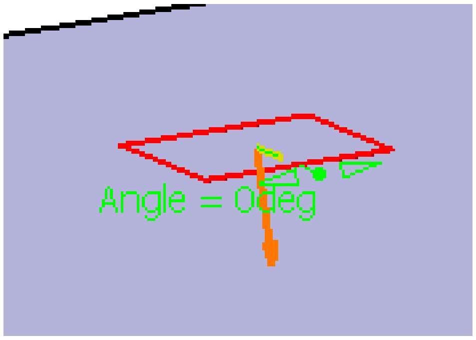
This base feature is used to define the stamping location and direction, by matching the punch's origin to the selected point on the base feature.

The User Defined Stamp Definition dialog box is displayed.

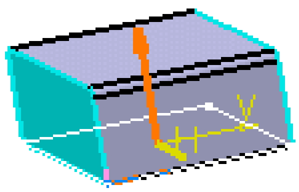
7. Click the **With opening**  icon.



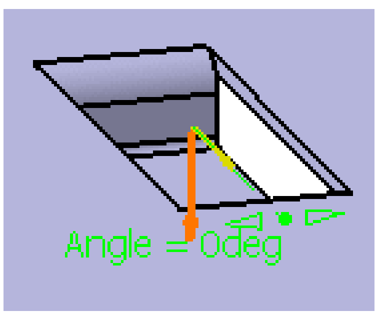
8. Select the punch (Body.2). The punch is previewed on the base feature.

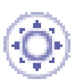


9. Click the **Faces for opening** field and select the lateral faces of the punch (Pad.1).



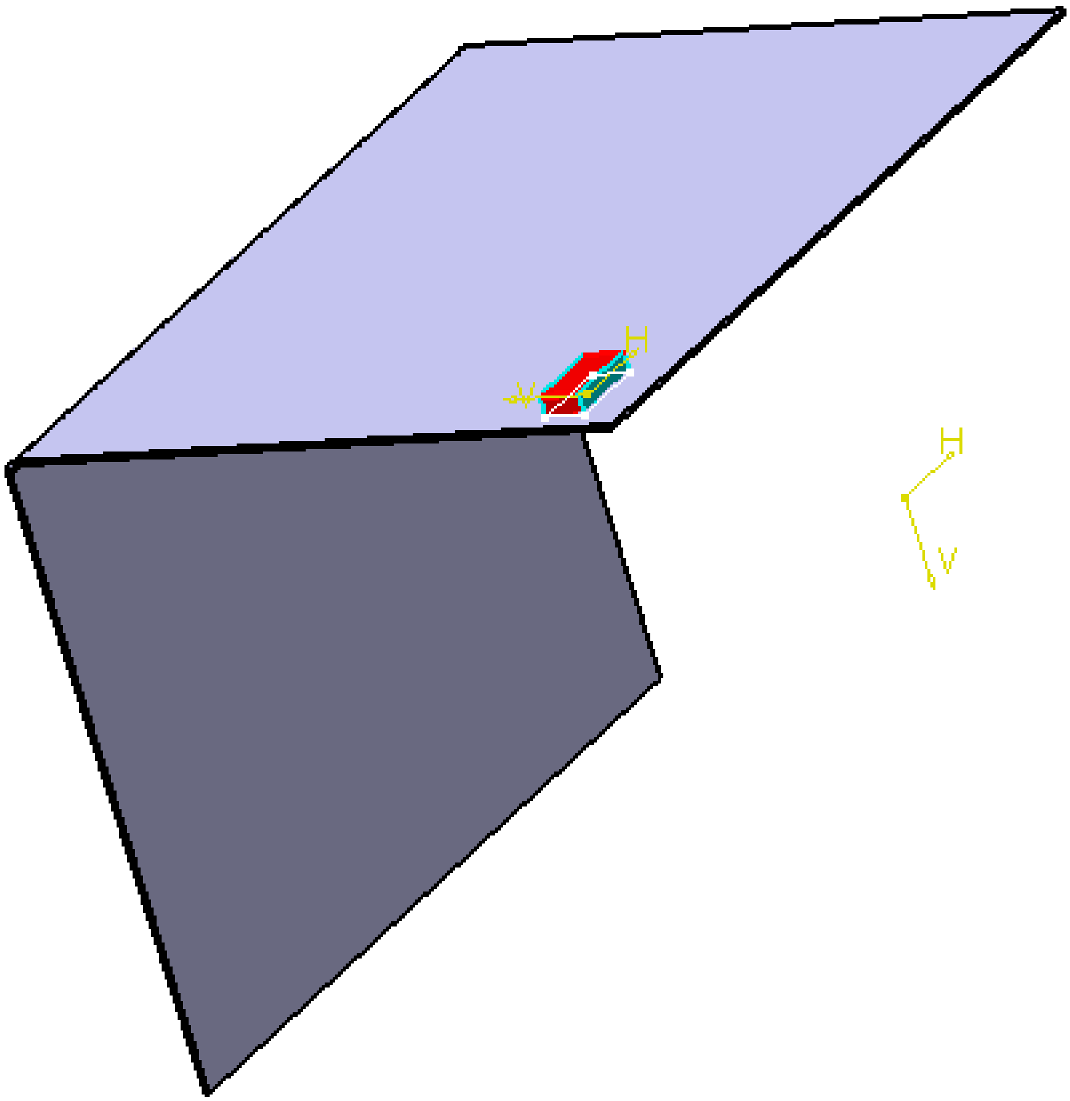
10. Click Preview. The stamp is previewed with the opening faces:





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.

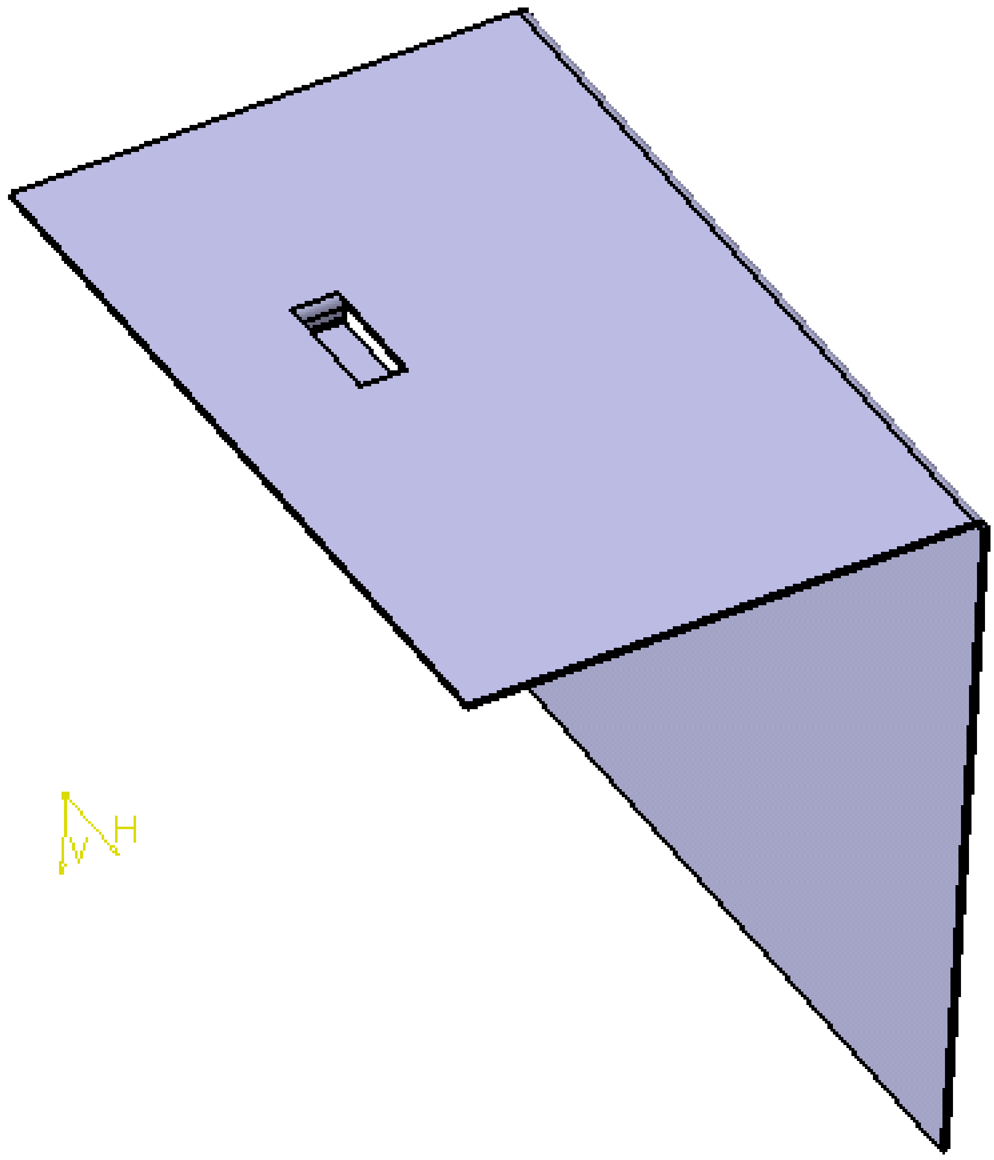
13. If needed, define the stamp's positioning on the selected base feature by choosing:

- 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.

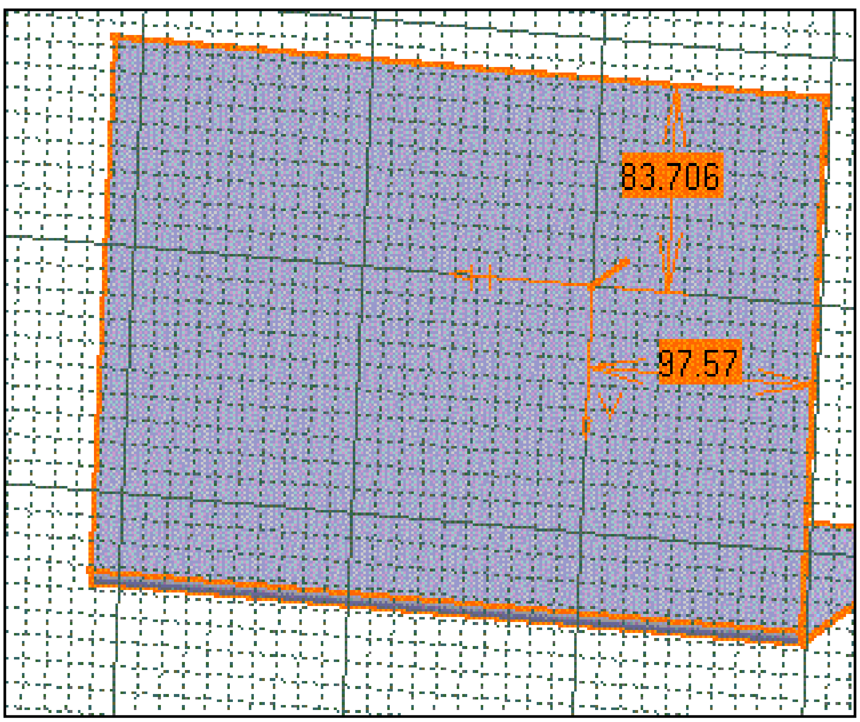
This is especially useful for non-circular stamps, but you can very well create the stamp as is, without further positioning.

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

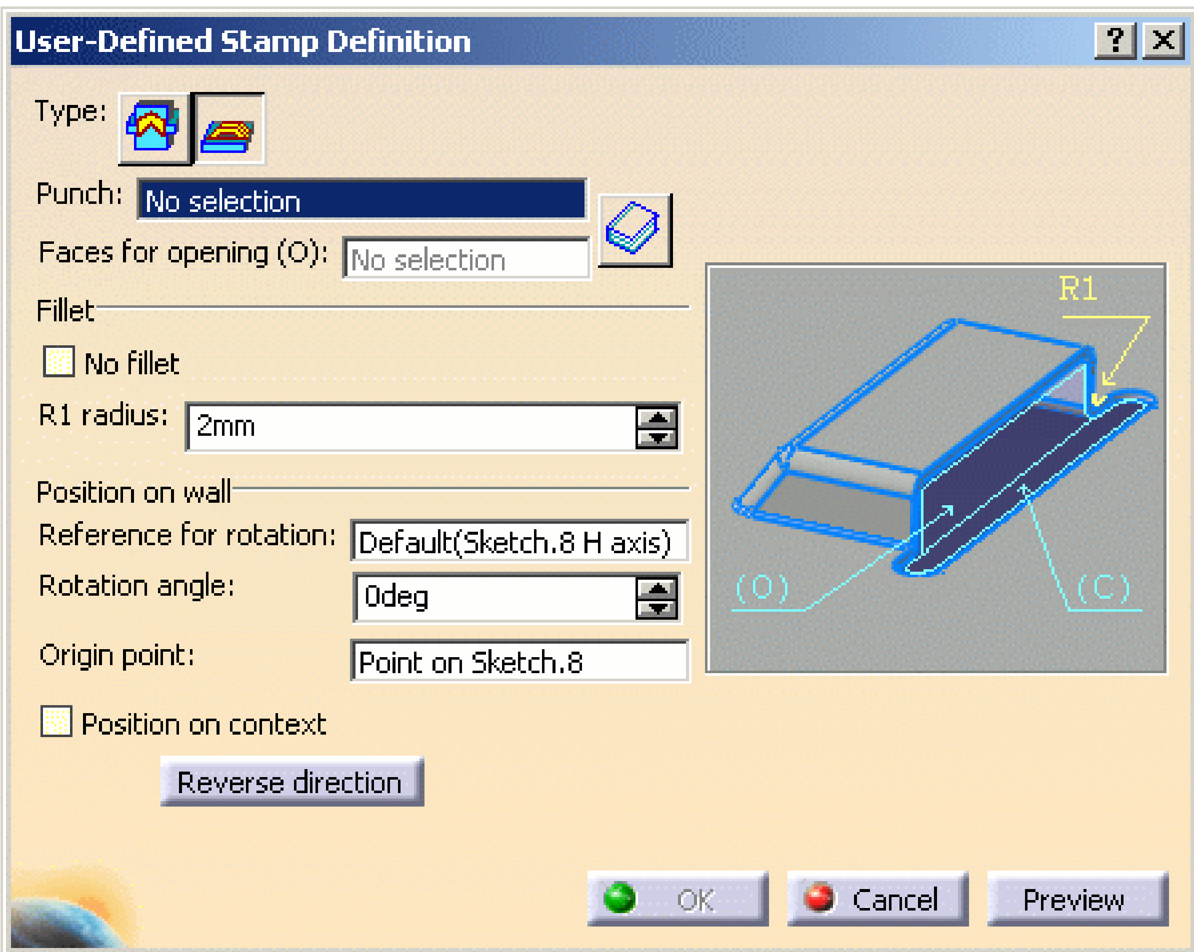
This task explains how to edit a user-defined stamp, that is:

- 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.

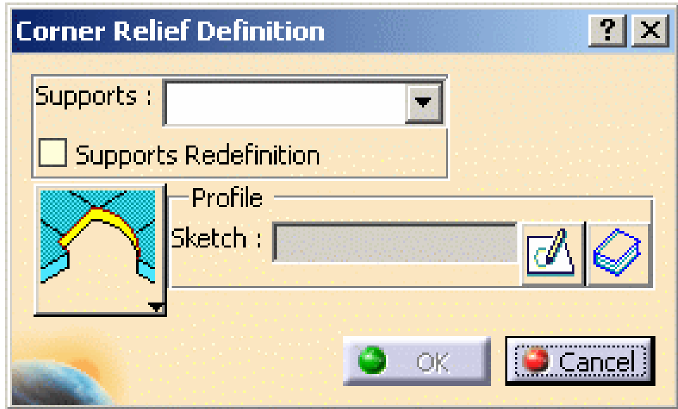
The stamp is updated accordingly.

Creating a Local Corner Relief

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

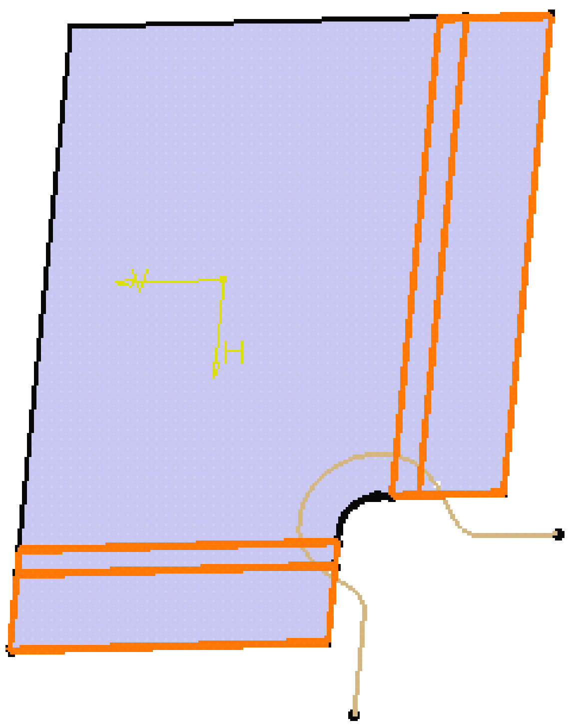
Open the CornerRelief01.CATPart document from the samples directory.
The part needs to be unfolded prior to creating the corner relief.

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

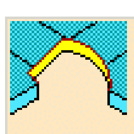


The Corner Relief Definition dialog box is displayed.

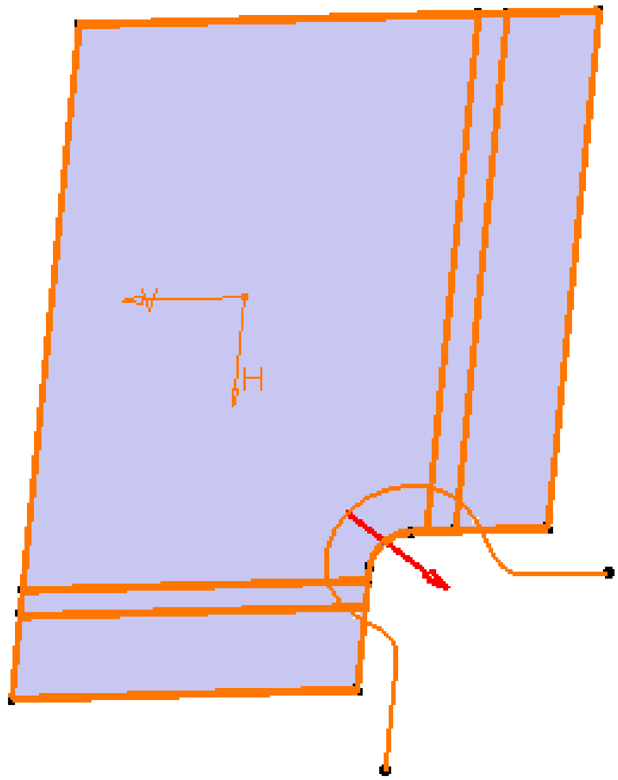
2. Select the supports on which a corner relief should be created (here we chose Surfacic Flange.1 and Surfacic Flange.2)

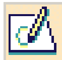


A notch was defined on the web profile between the two fillets' flanges; so that flanges do not intersect. This operation enables to prepare the web as to create the flanges that will be later used to define the corner relief.

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

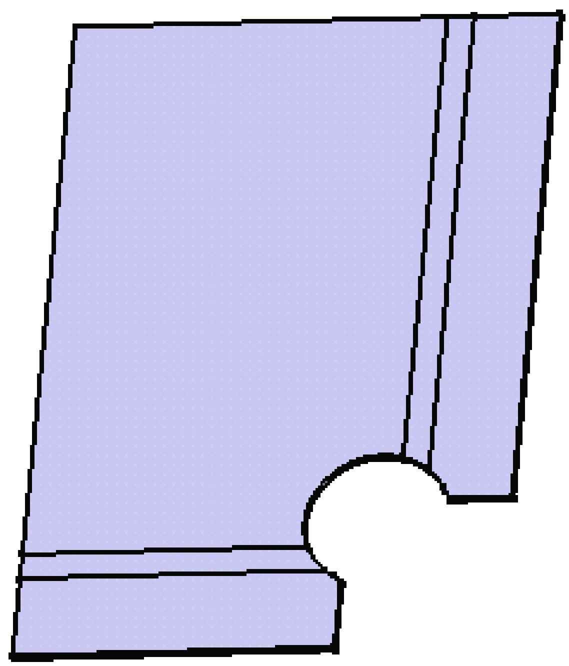
1. Select the sketch, directly in the 3D geometry.

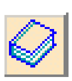


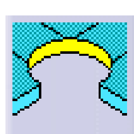
As soon as the sketch has been selected, the **Sketcher**  icon is displayed in the dialog box allowing you to edit the selected sketch, if needed.

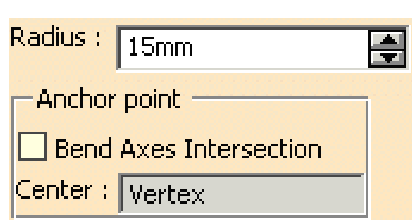
The red arrow lets you choose the direction of matter to remove. Click it to reverse the direction.

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.

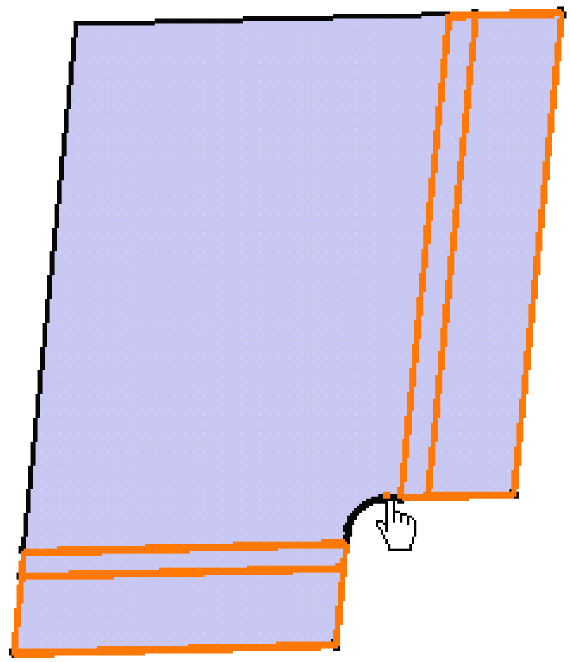


1. Define the default radius: it is equal to the bend radius + the thickness.

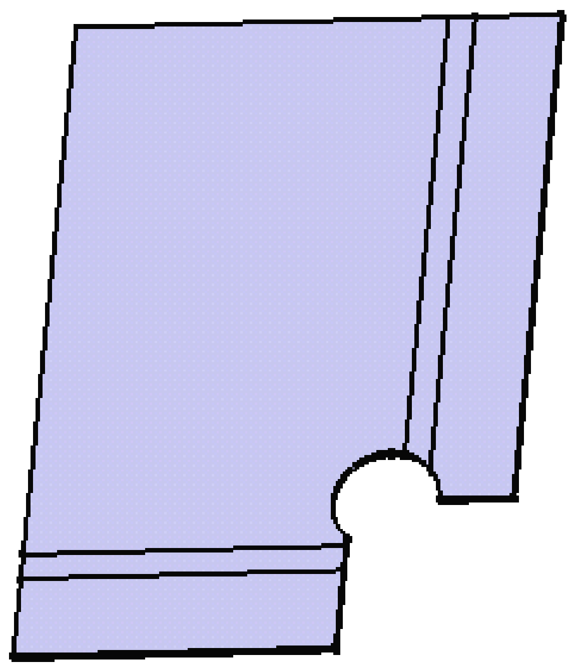
   In our example, we defined a radius of 15 mm.

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.

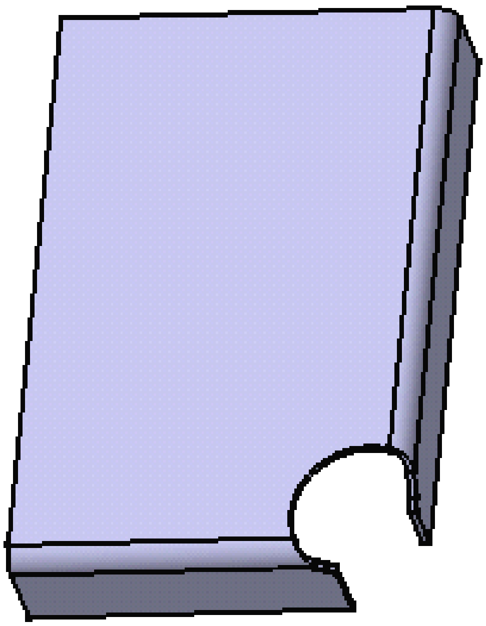


3. Click OK in the Corner Relief Definition dialog box.

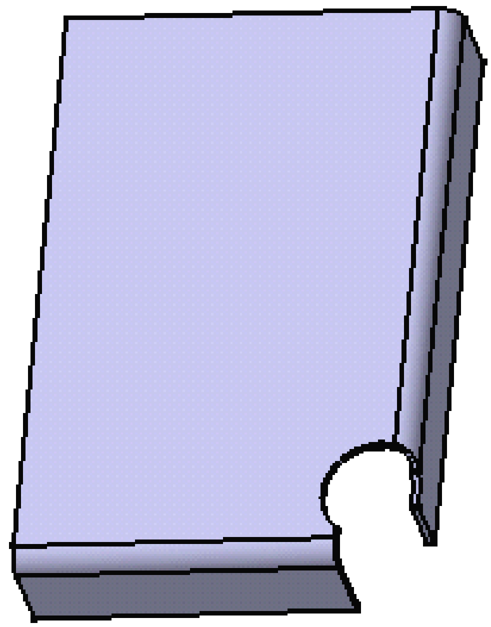


The created element (identified as Corner Relief.xxx) is added to the specification tree.

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

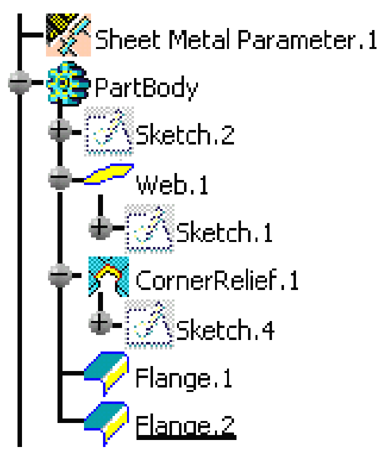


Folded user corner relief



Folded circular corner relief

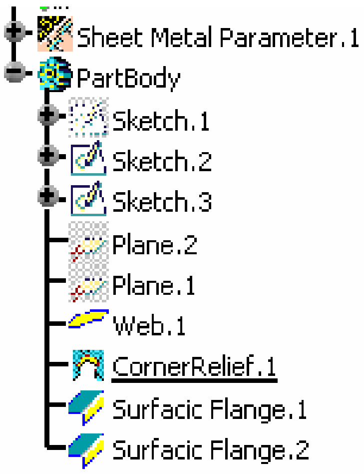
The **Supports Redefinition** checkbox enables to redefine the supports' sides thus adding matter to these supports.
In that case, the created element (identified as Corner Relief.xxx) appears before the supports in the specification tree.



- 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**.

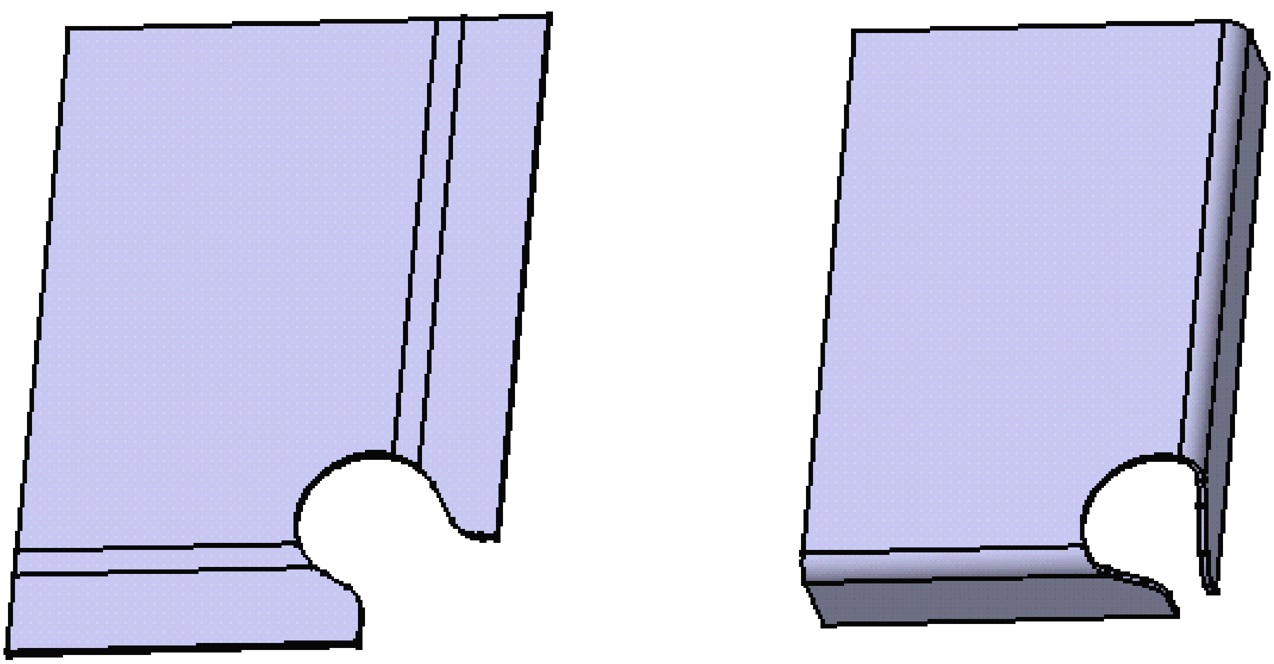
- In hybrid context, when checking **Supports Redefinition**, the Surfacic flanges are hidden in the 3D since the define in work object parameter is applied to the corner relief.

- 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.

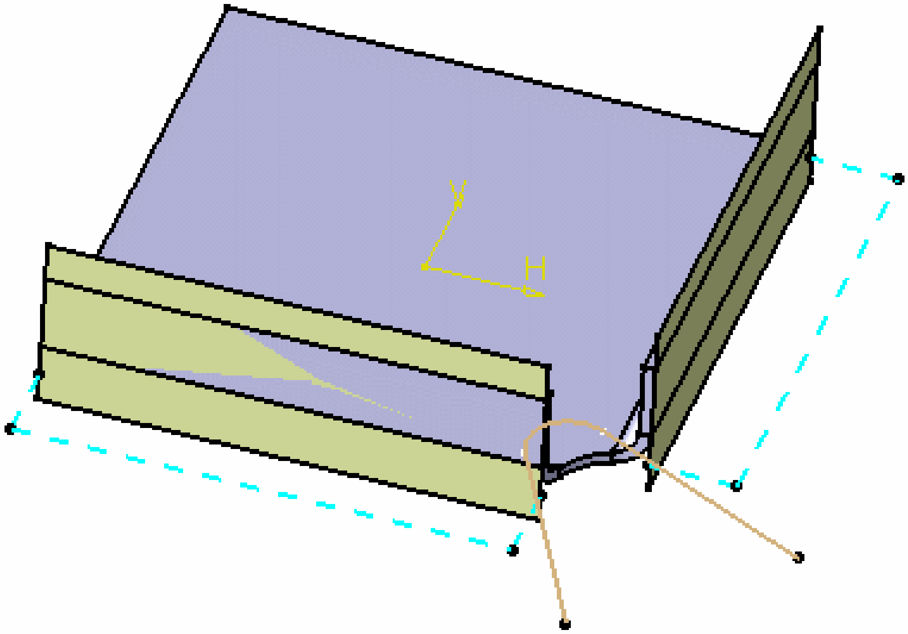
Fore more information about Hybrid Design, refer to the Hybrid Design section.



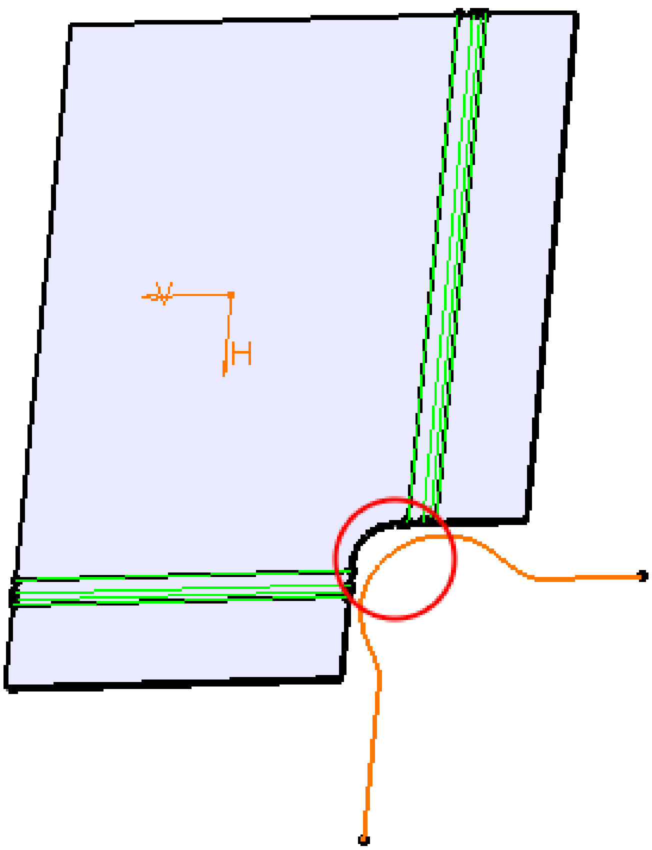
Unfolded user corner relief with redefined supports

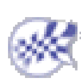
Folded user corner relief with redefined supports

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.





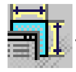
Creating Constraints

This task shows how to set geometric constraints on geometric elements.

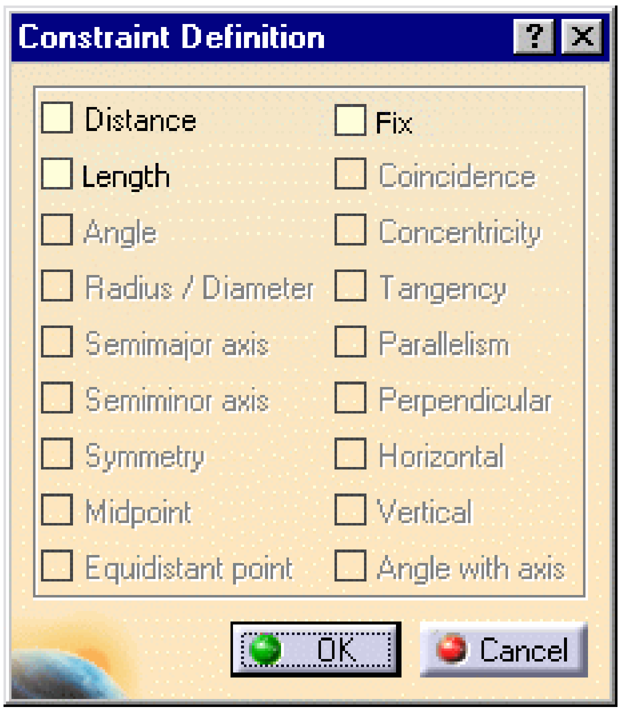
Such a constraint forces a limitation. For example, a geometric constraint might require that two lines be parallel.

To set a constraint between elements:

1. Multi-select two or three elements to be constrained.

2. Click the **Constraint defined in dialog box** icon .

The Constraint Definition dialog box appears indicating the types of constraint you can set between the selected elements.

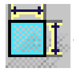


3. Select one of the available options to specify that the corresponding constraint should be made.

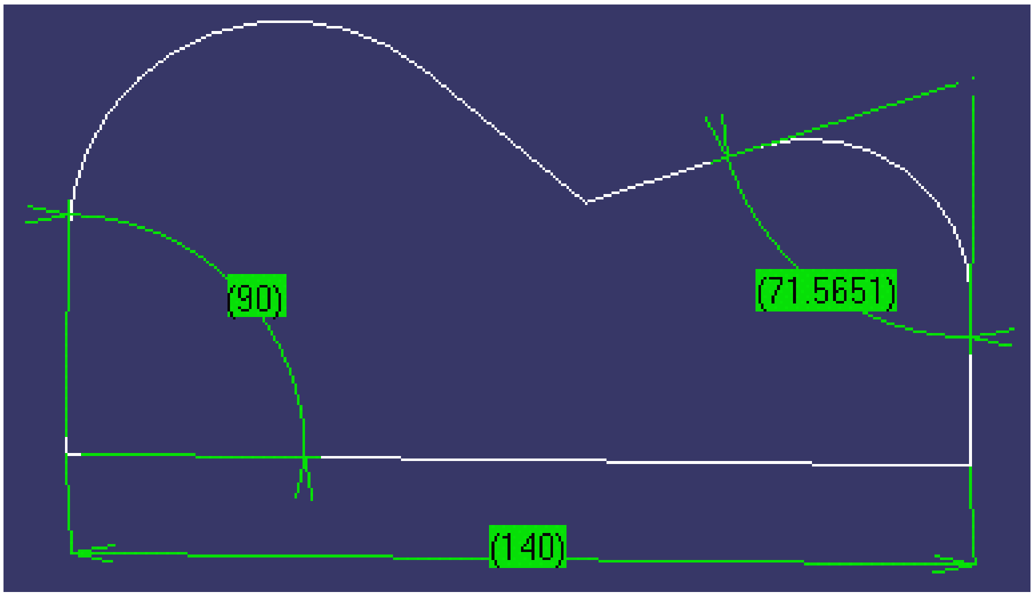
4. Click OK.

The corresponding constraint symbol appears on the geometry.

To set a constraint on a single element:

1. Select the element to be constrained.
2. Click the **Constraint** icon .

The corresponding constraint symbol appears on the geometry.



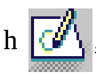
Mapping Elements

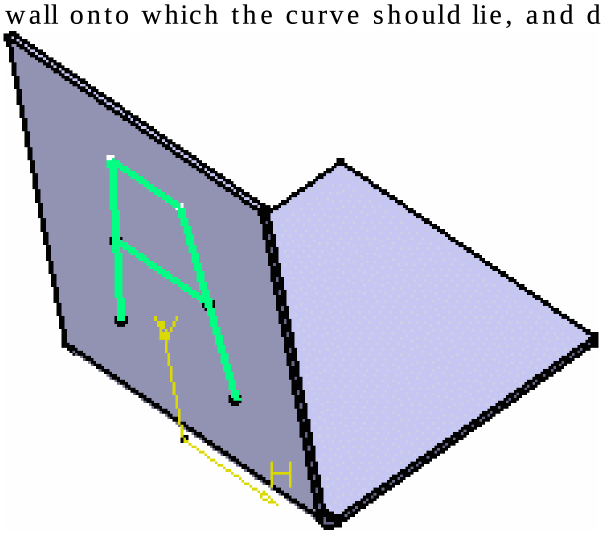
This task shows how to create curves or points from a sketch (as designed using the Sketcher) or from existing curves or points, onto a Sheet Metal part; and to fold/unfold it, just as other Sheet Metal elements.
This is especially useful when:

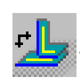
- you want to generate a logotype
- you want to define an area for chemical milling
- you want to create a cutout (pocket) to solve the overlapping of walls for example (the overlapping can be checked with the Sheet Metal Production product).

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

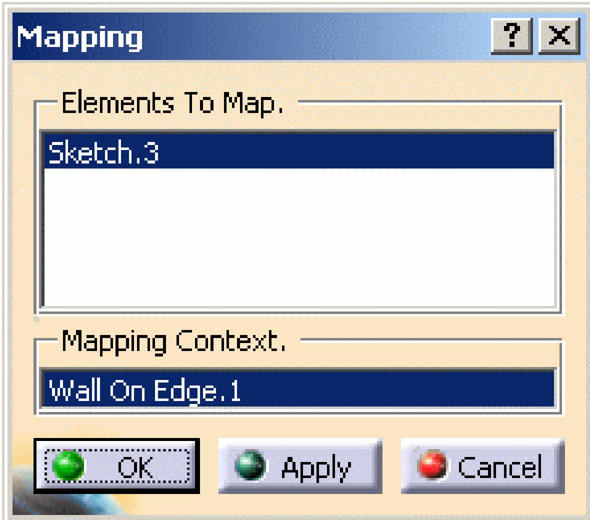
These samples already contain a pre-defined sketch that will be mapped onto the part.

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



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

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



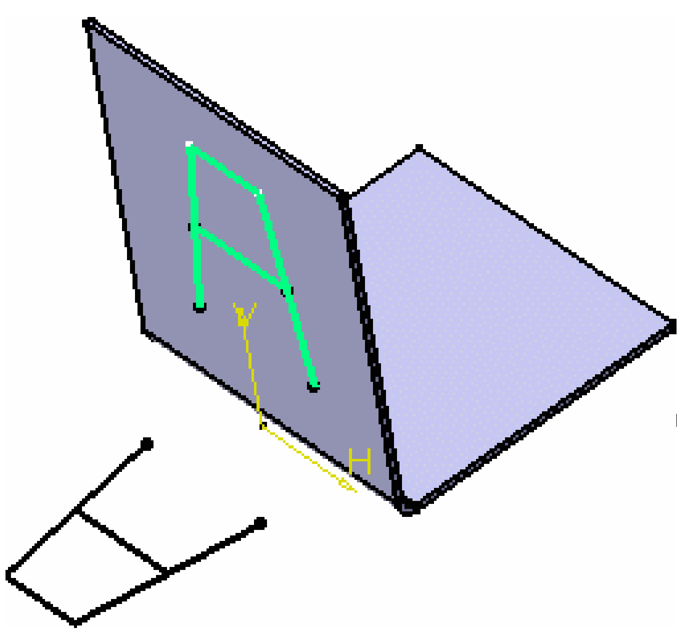
You can manage the list of elements:

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

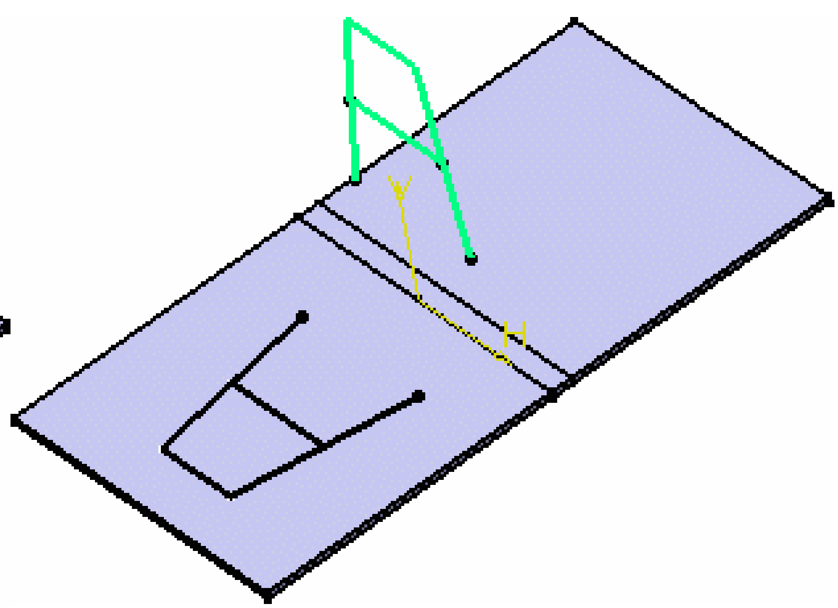
2. Select the **Mapping Context**, that is the element of the part on which the curve should be generated when folding or unfolding.

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

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



Folded view of the curve mapping



Unfolded view of the curve mapping

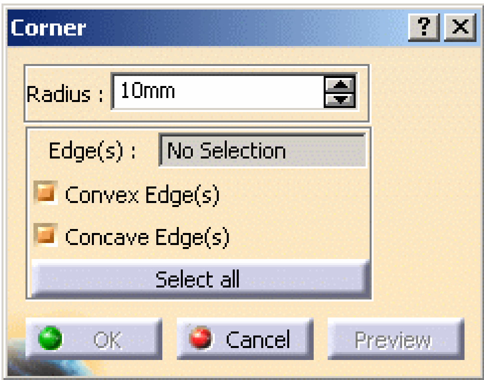
- You can select several sketches/curves/points to be mapped at a time.
- Mapped curves can be created across several walls and bends.

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.

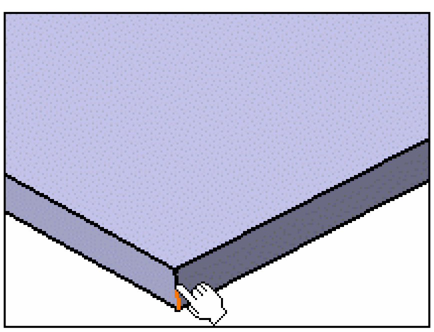


2. Set the radius value.
3. Choose the type of edge you wish to round off: **Convex Edge(s)** and/or **Concave Edge(s)**. For the purpose of this scenario, leave both options selected.

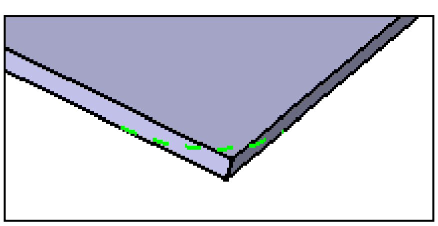
Once you have selected an edge, you can no longer modify the chosen options (they are grayed out), unless you cancel the selection.

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

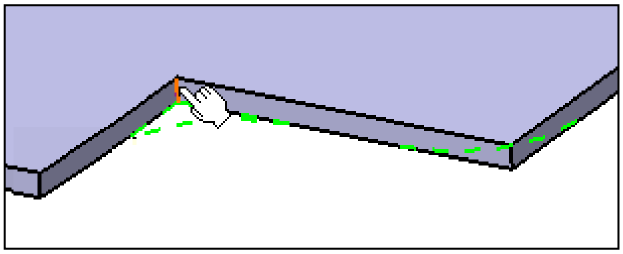
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.



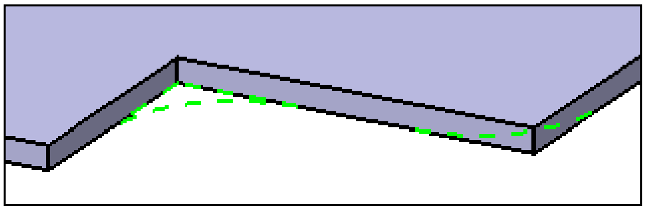
The corner is previewed on the edge, with the current radius value.



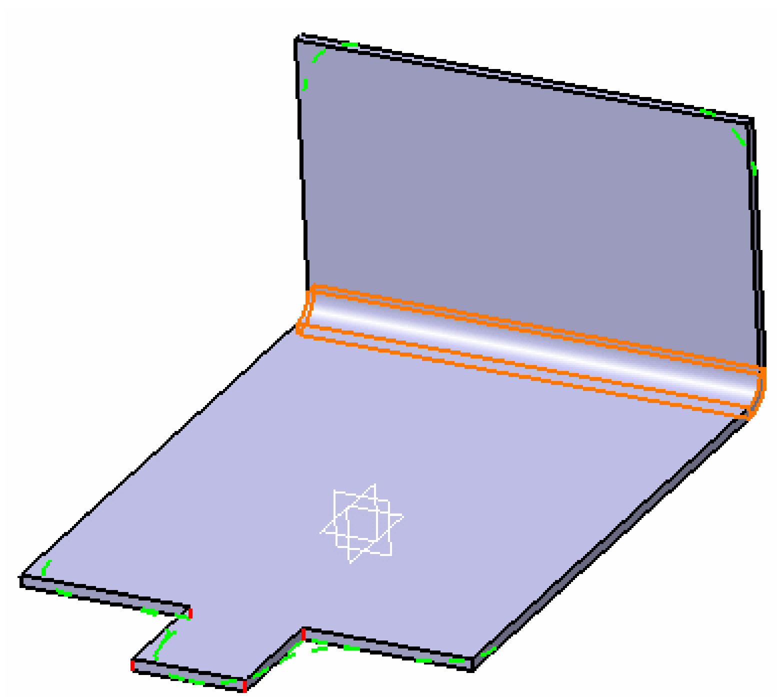
5. Click to select a concave edge on a part.



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

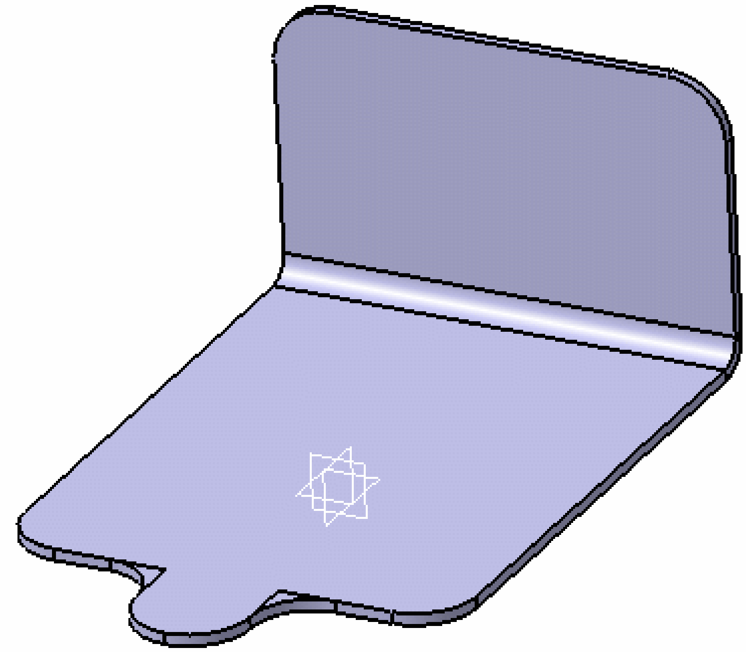


6. Click **Cancel Selection** then click the **Select All** button. All edges of the part are selected and the corners previewed.



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.

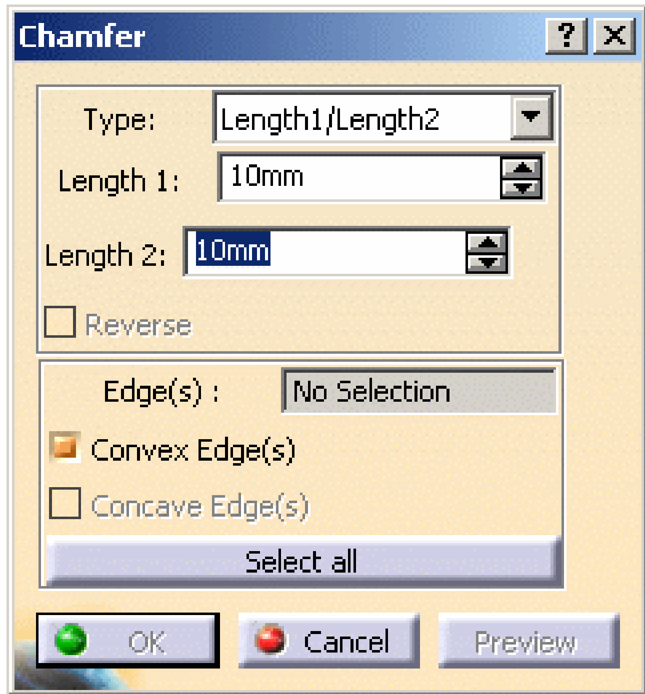
Creating Chamfers

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.

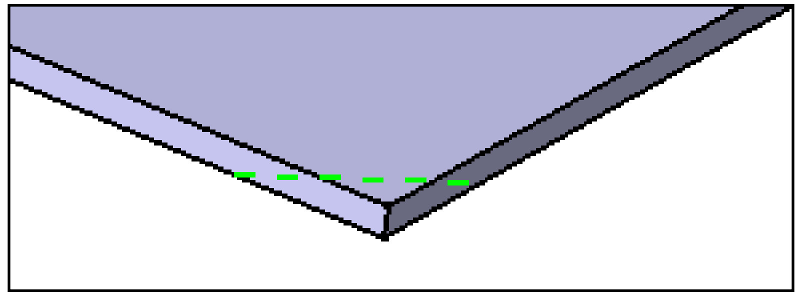
The chamfer is previewed on the edge.

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

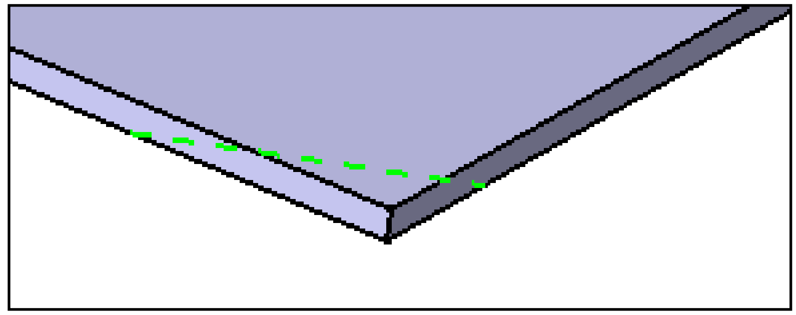
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.

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.



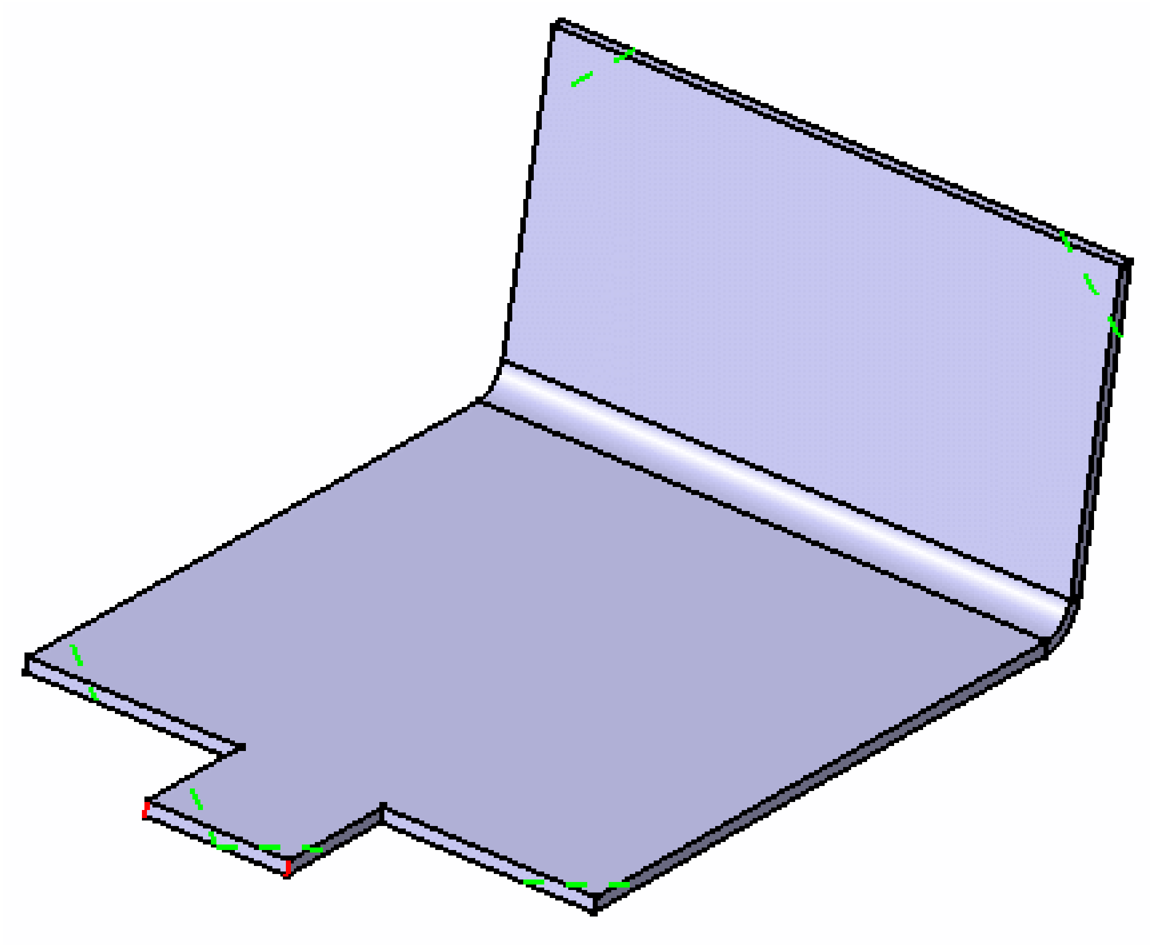
- a length value and an angle: the length is computed on one side of the edge and the angle from the chamfer's limit on the same side. Here, we chose a length of 10mm and an angle of 60deg.



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

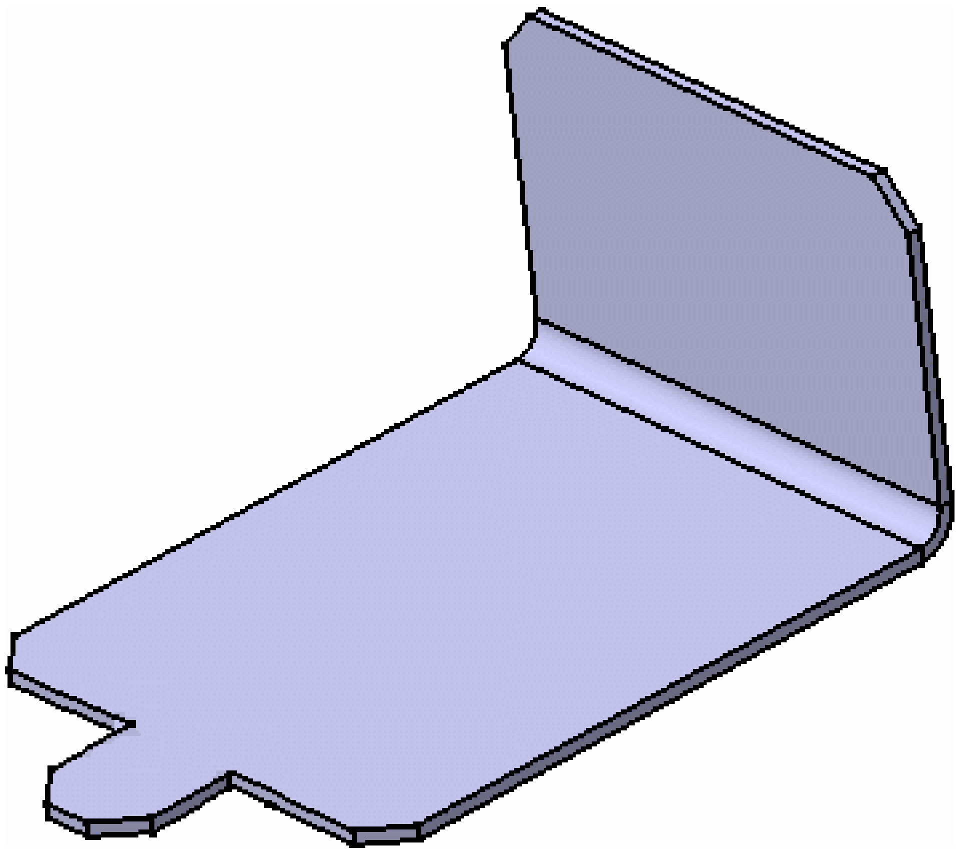
4. Click **Cancel Selection** then click the **Select All** button.

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.

Patterning

This section explains and illustrates how to create various kinds of patterns on Aerospace 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 have further information about patterns, refer to *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.

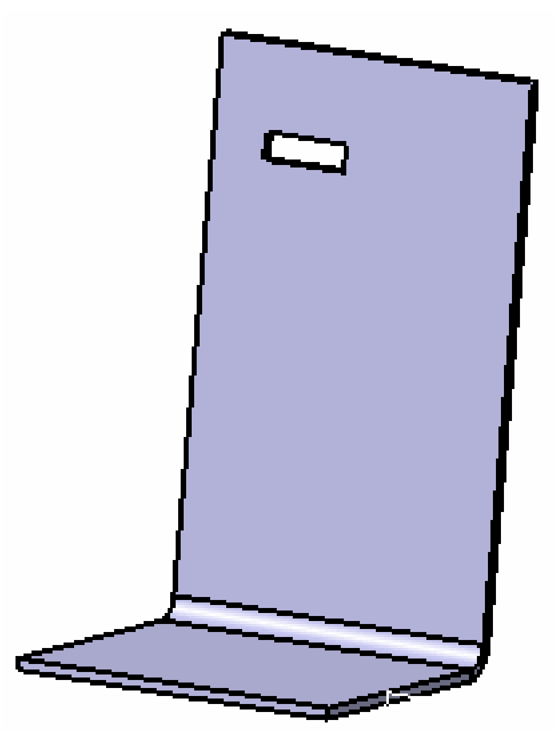
You can only duplicate the following items:

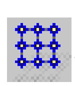
- cutouts
- holes
- beads
- flanges
- flanged holes
- stamps (except stiffening ribs)
- Aerospace Sheetmetal Design patterns

These features must lie on a unique and planar surface.

Open the RectangularPattern1.CATPart document.

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.2**) as shown, to specify the first direction of creation.

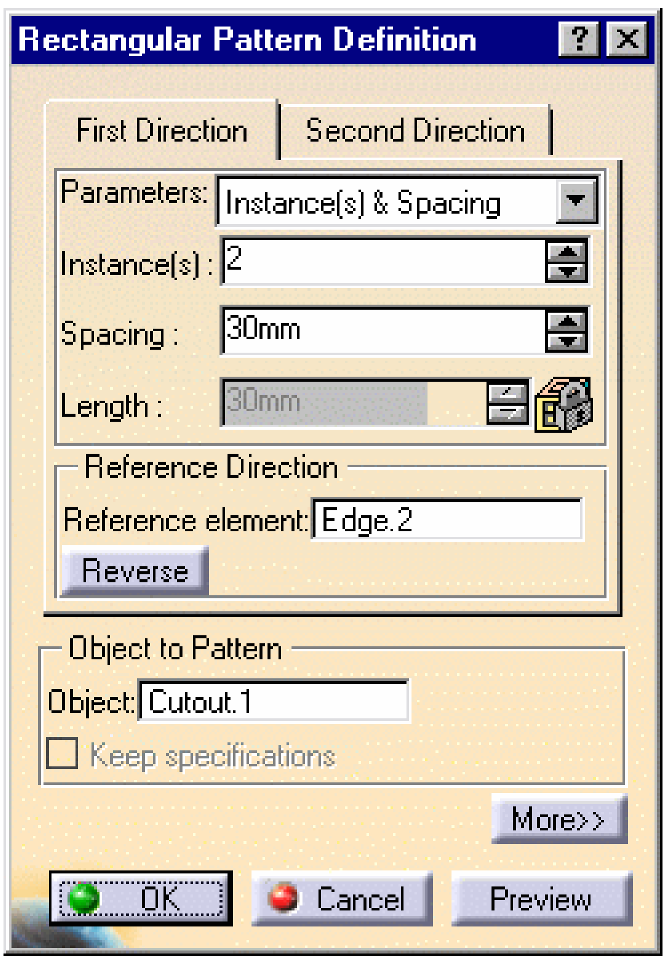
   An arrow is displayed on the flange.

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 surface (this can be checked using the preview).

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.

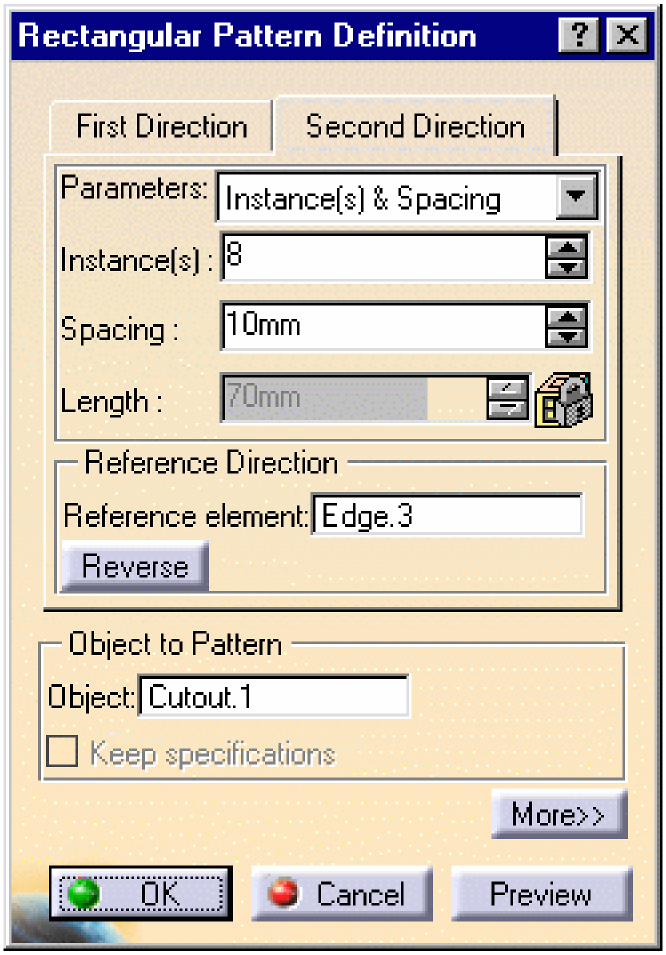
Defining the spacing along the grid and the length of your choice, would make the application compute the number of possible instances and space them at equal distances.

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.

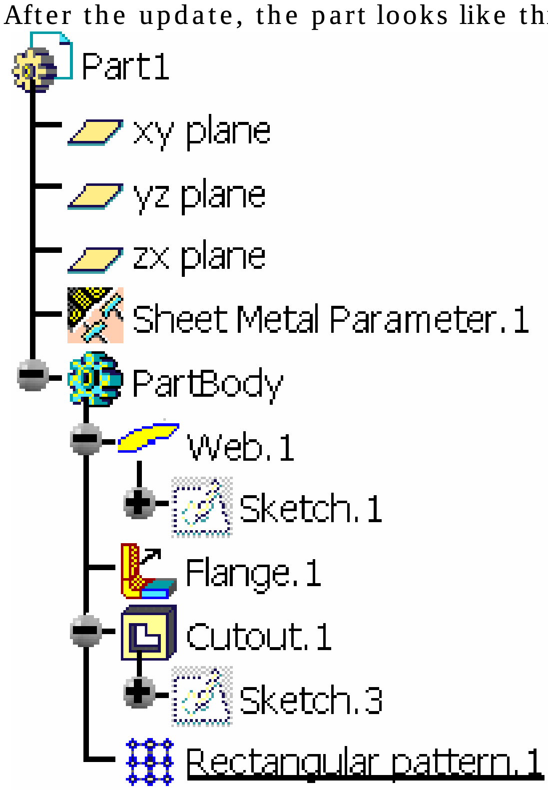
8. Select the second edge (**Edge.3**), as shown, to define the second direction.
9. Keep the **Instances & Spacing** option: enter 8 and 10 mm in the appropriate fields.

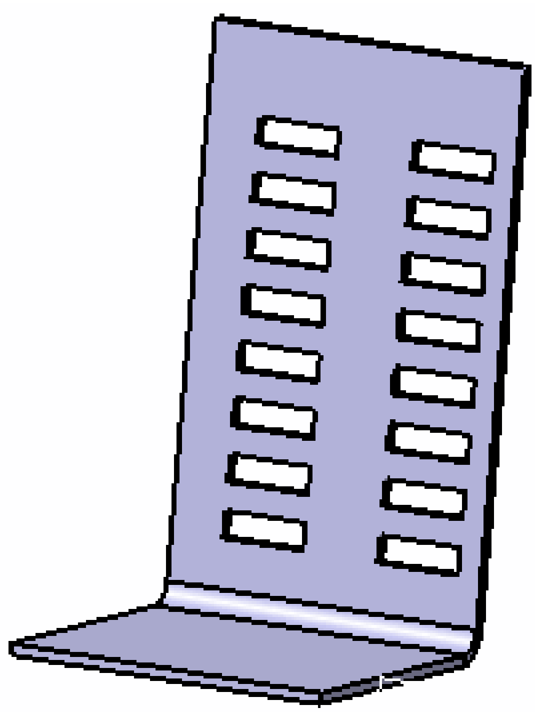
Additional cutouts have been aligned along this second direction.

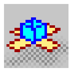


10. Click **OK** to repeat the cutouts.

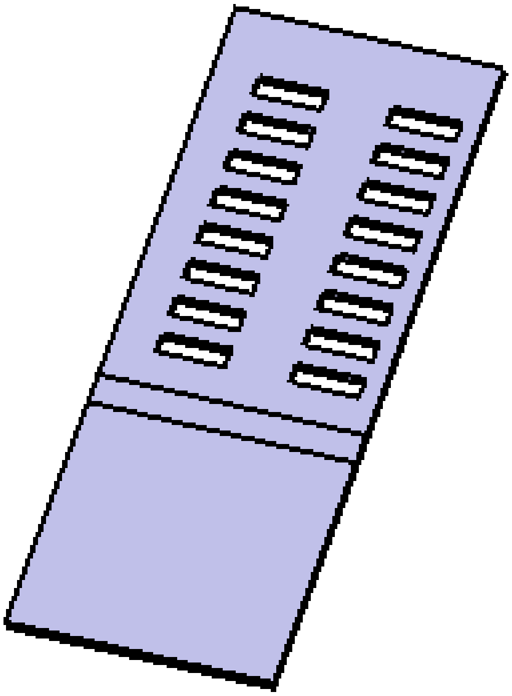
After the update, the part looks like this:





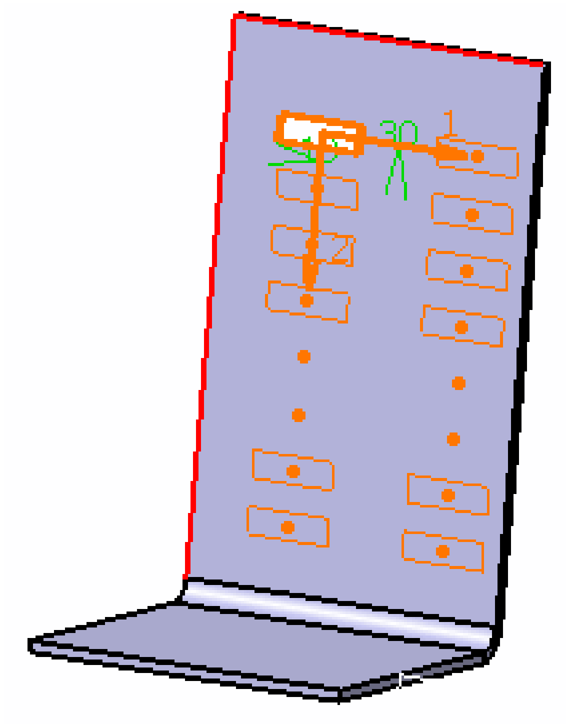
11. Select this icon  to unfold the part:

The pattern is updated on the unfolded view.



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

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



- 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.

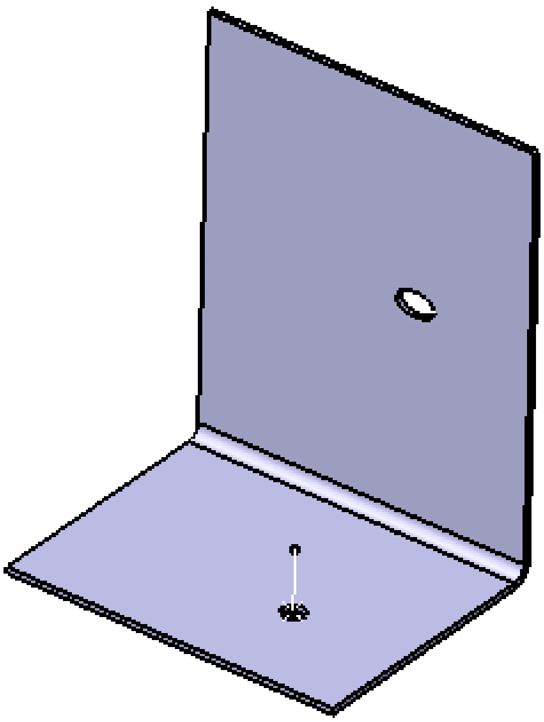
You can only duplicate the following items:

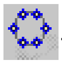
- cutouts
- holes
- beads
- flanges
- flanged holes
- stamps (except stiffening ribs)
- Aerospace Sheetmetal Design patterns

These features must lie on a unique and planar surface.

Open the CircularPatterns1.CATPart document.

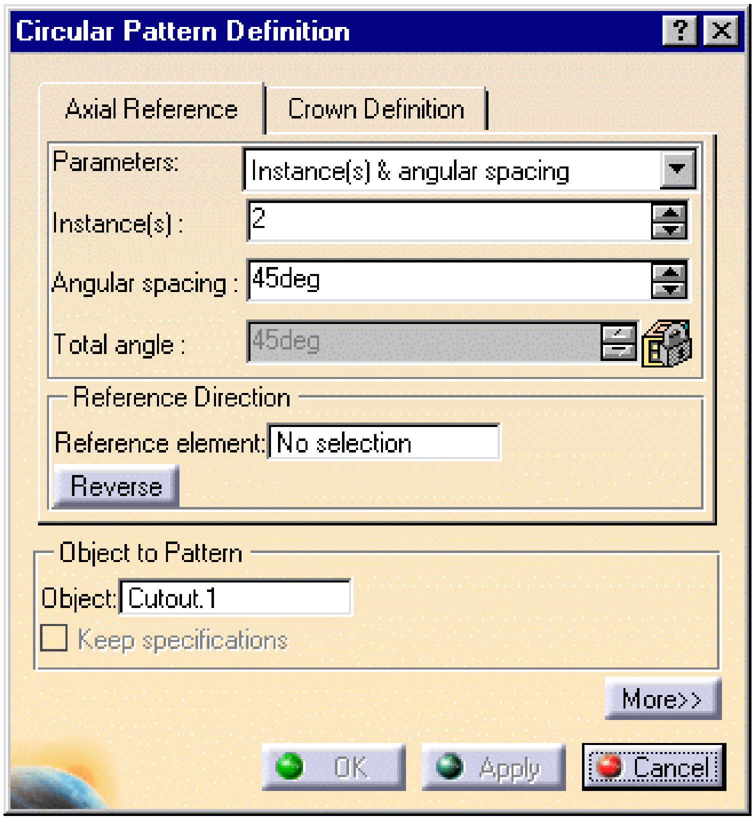
1. Select the circular cutout you want to duplicate.



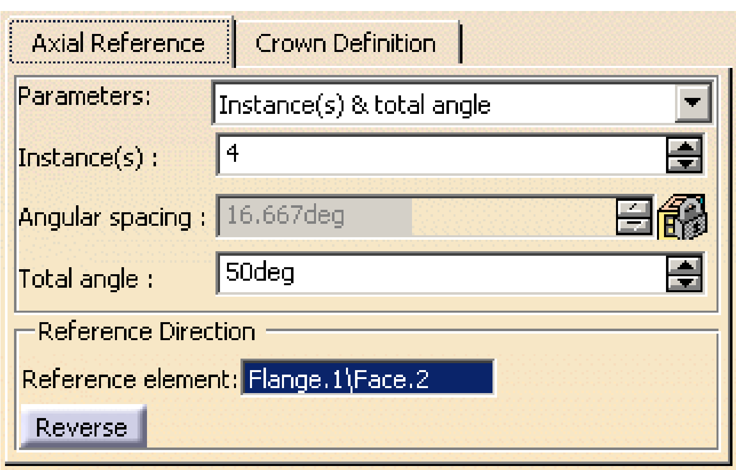
2. Click the **Circular Pattern** icon .

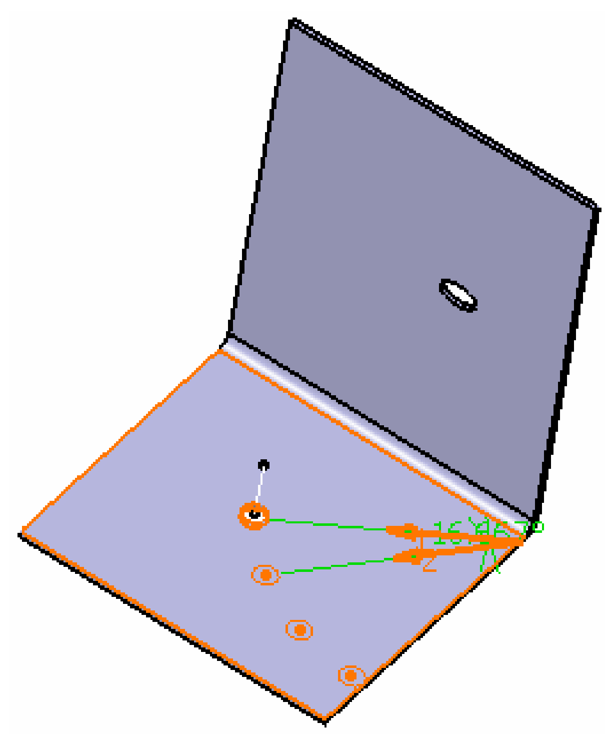
The **Circular Pattern Definition** dialog box is displayed.

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

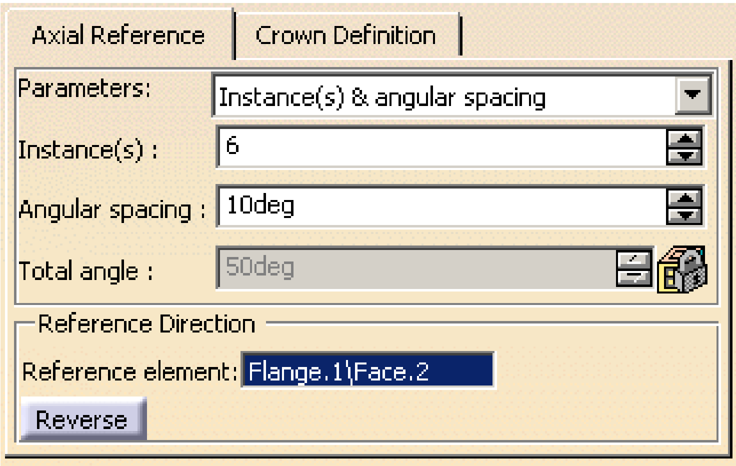


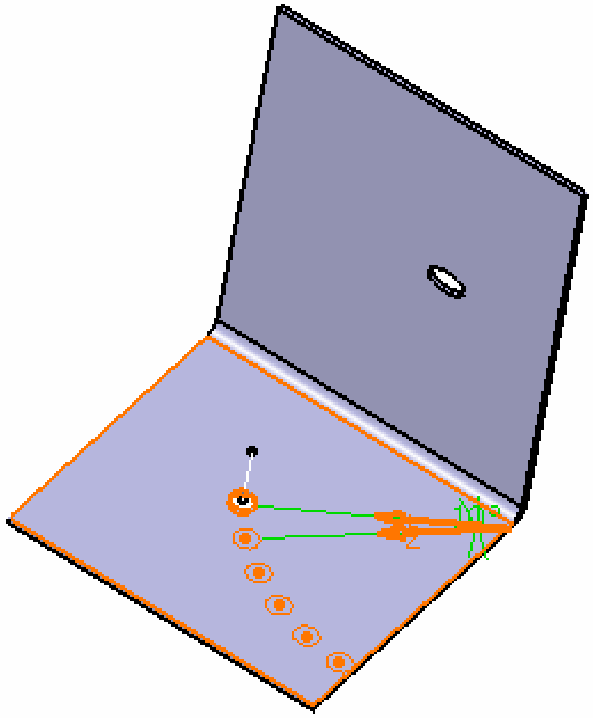
- **Instance(s) & total angle**: the number of patterns as specified in the instances field are created, in the specified direction, and evenly spread out over the total angle.



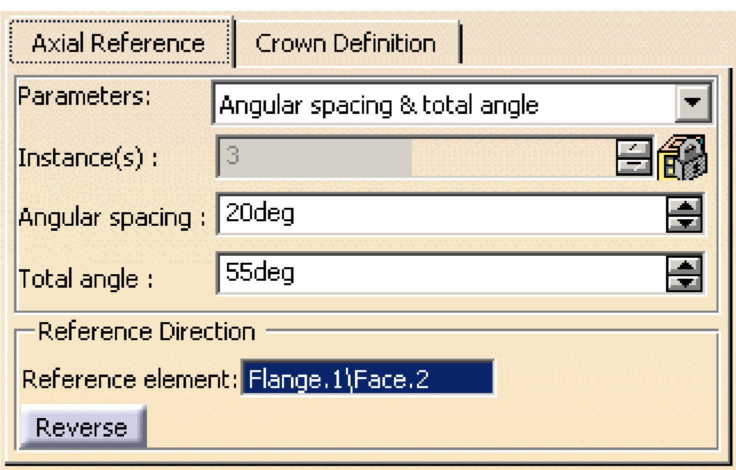


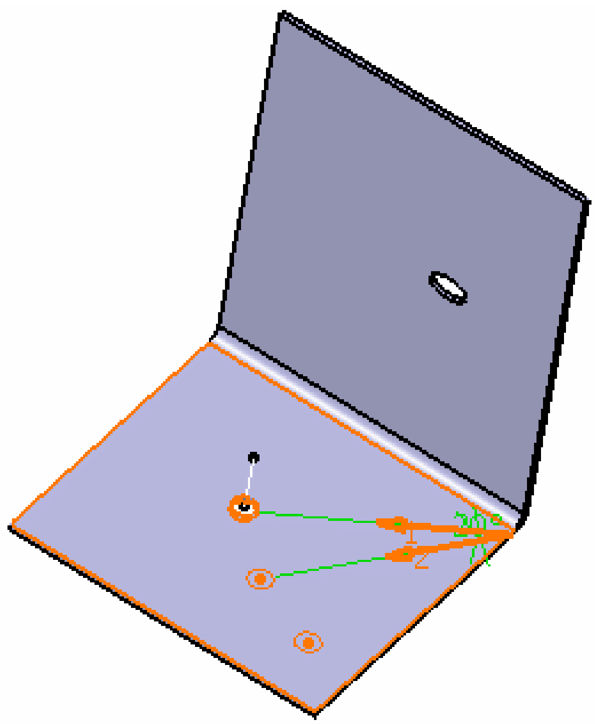
- **Instance(s) & angular spacing**: the number of patterns as specified in the instances field are created in the specified direction, each separated from the previous/next one of the angular angle value.



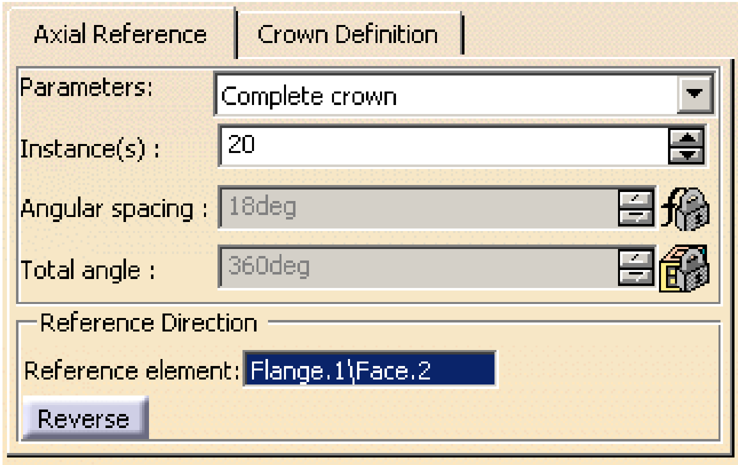


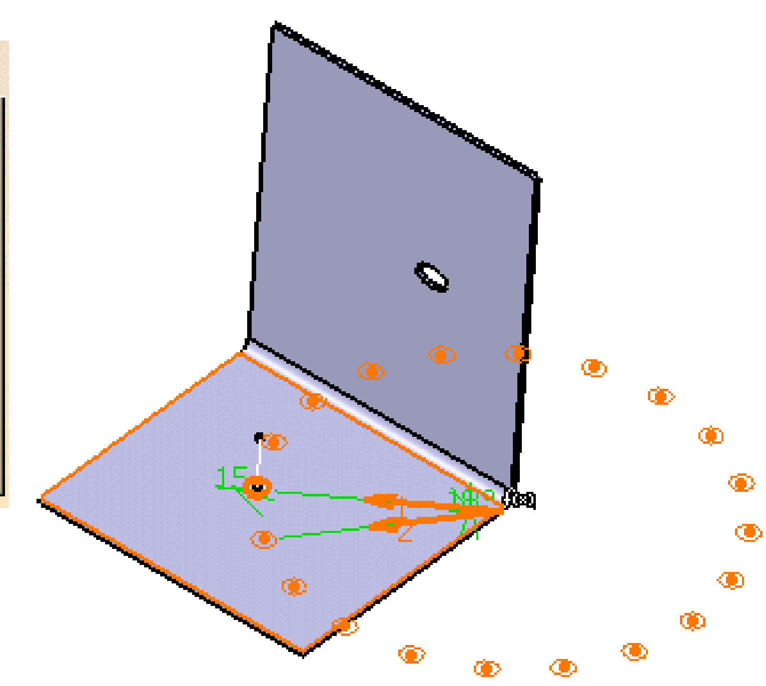
- **Angular spacing & total angle**: as many patterns as possible are created over the total angle, each separated from the previous/next one of the angular angle value.





- **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.

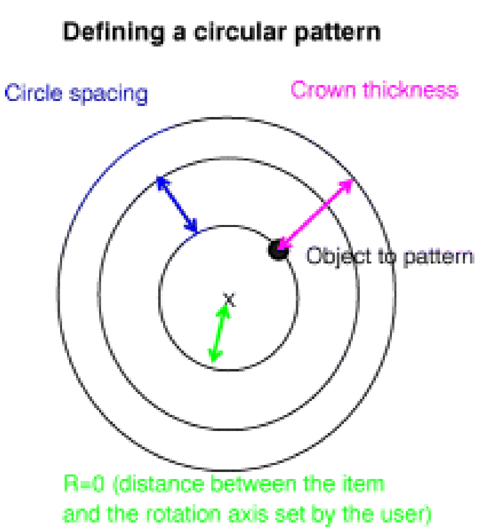
   Here select the face on which lies the circular cutout.

- To define a direction, you can select an edge or a planar face.
  Should you select the face of a web, 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.

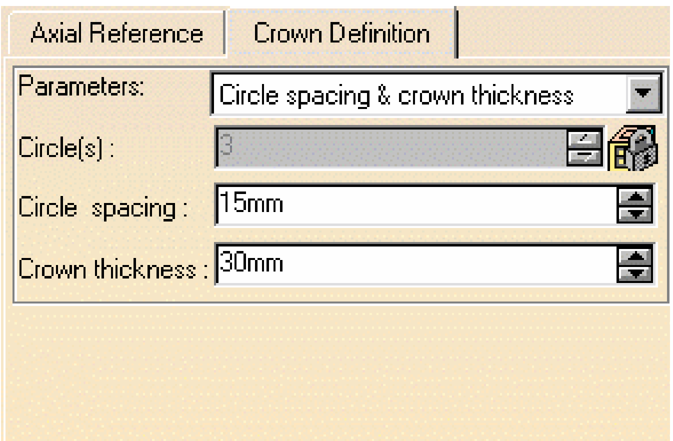
5. Click the **Crown Definition** tab, and choose which parameters you wish to define the crown.

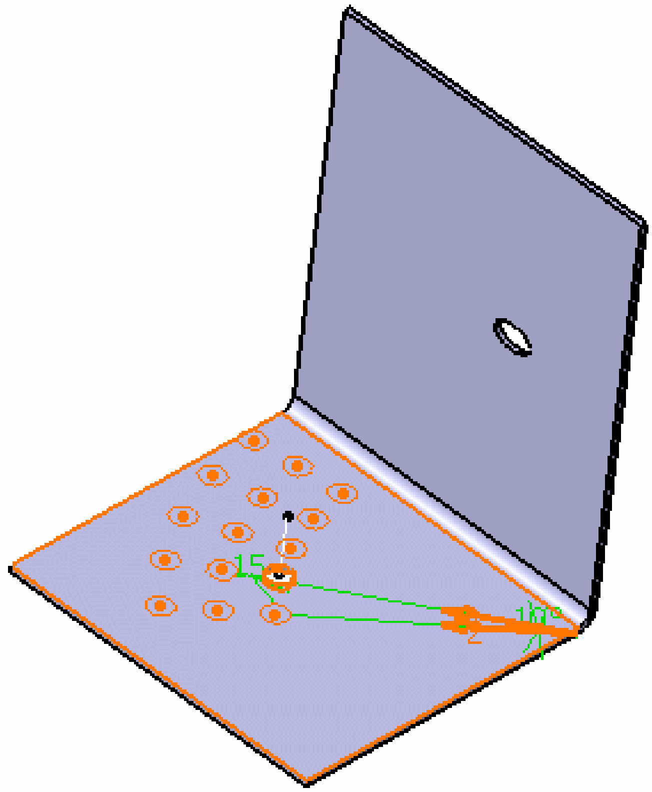
This figure may help you define these parameters:



- 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:

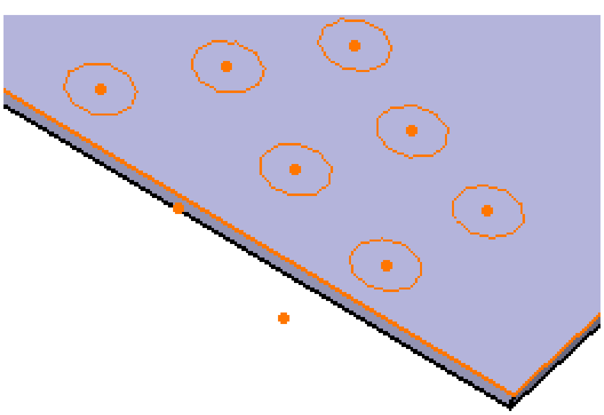




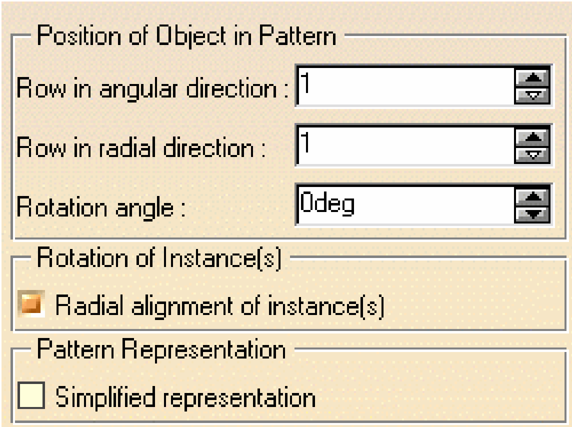
Note that one of the pattern goes beyond the flange (this can be checked using the preview).

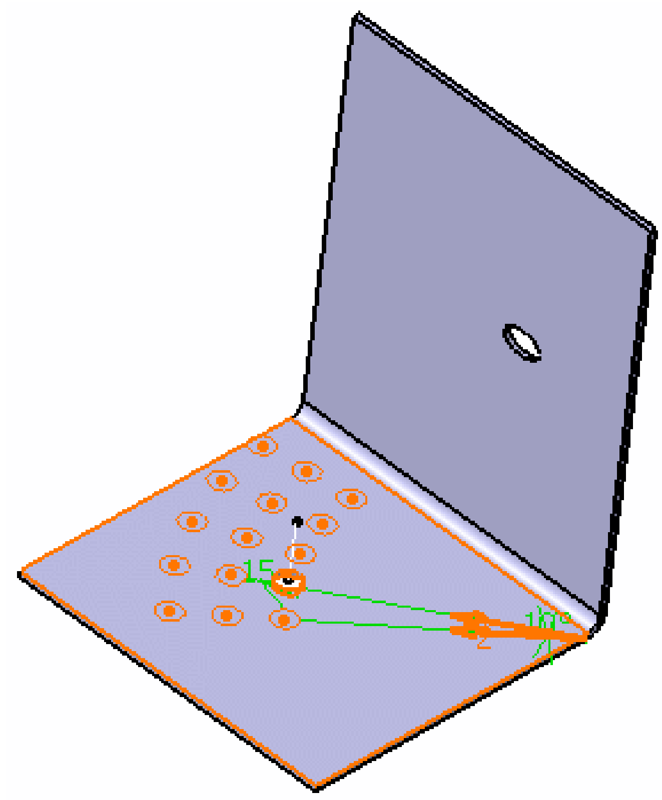
You can delete instances of your choice when creating or editing a pattern. To do so, just select the points materializing instances in the pattern preview.

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.

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.
You can only duplicate the following items:

- cutouts
- holes
- beads
- flanges
- flanged holes
- stamps (except stiffening ribs)
- Aerospace Sheetmetal Design patterns

These features must lie on a unique and planar surface.

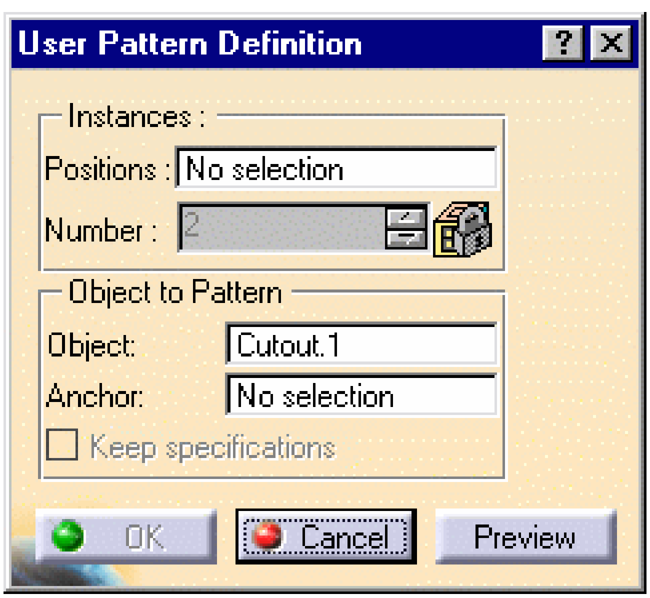
Open the UserPatterns1.CATPart document.

1. Select the feature to be duplicated.

   Here we selected the cutout.

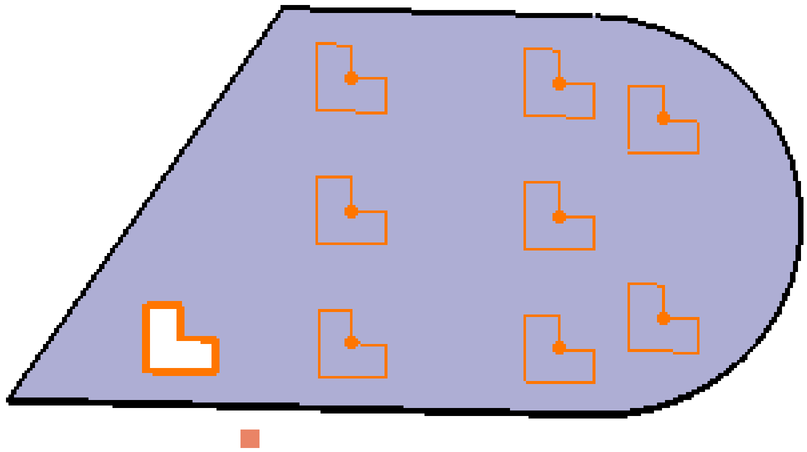
2. Click the **User Pattern** icon .

The User Pattern Definition dialog box is displayed.



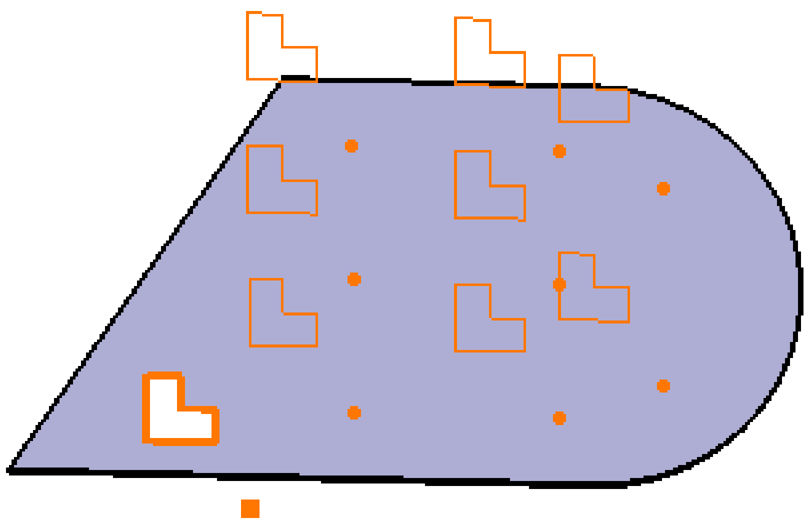
3. Select '**Sketch 3**' in the specification tree and click **Preview**.

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



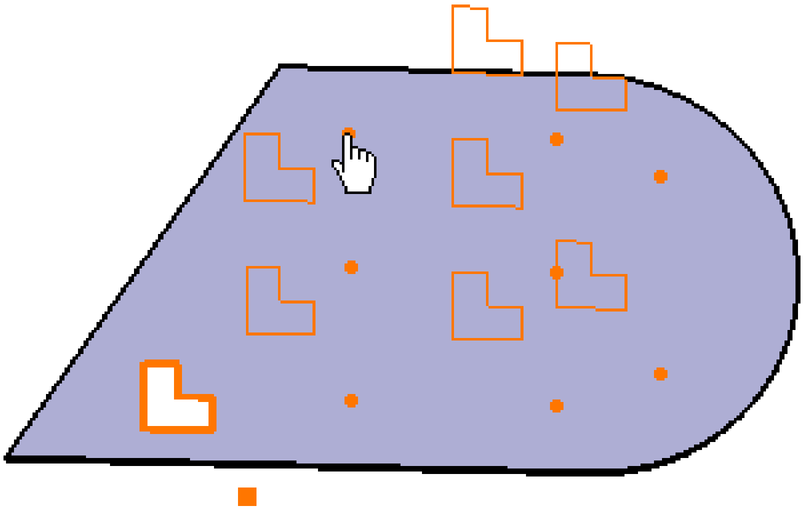
By default, the application positions each instance with respect to the center of gravity of the element to be duplicated. To change this position, use the anchor field: click the anchor field and select a vertex or a point.

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



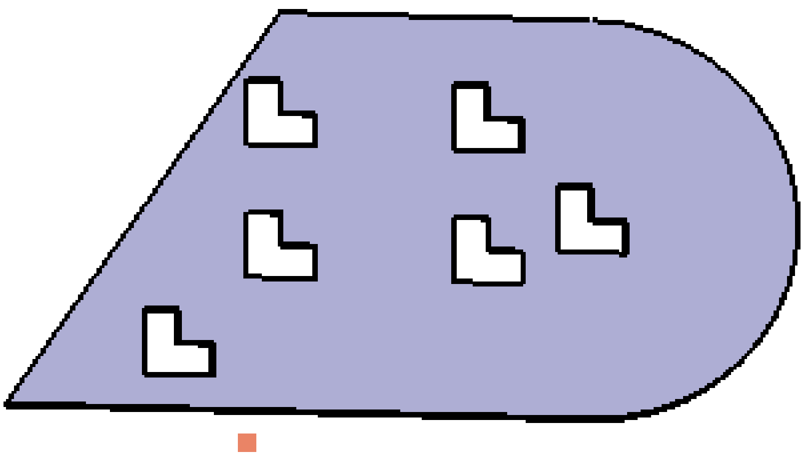
5. You can then click the points corresponding to the pattern instances to be removed.

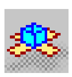
Patterns should not go beyond the surface (this can be checked using the preview).



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

Cutouts are created at the points of the sketch.

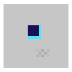


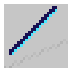
Would you need to unfold the part using the  icon, you would notice that the pattern is updated.

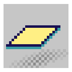
- When you duplicate a pattern of flange, the edge of the flange spine and its instances have to be tangent to the wall edge: you cannot choose a direction of patterning not parallel to the flange spine.
- All instances of the flange pattern must lie on the same face as the flange pattern.

Reference Elements

You can create wireframe elements within the Aerospace Sheetmetal Design workbench:

 Create points: click this icon, choose the point creation type, and specify parameters

 Create lines: click this icon, choose the line creation type, and specify parameters

 Create planes: click this icon, choose the plane creation type, and specify parameters

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 .

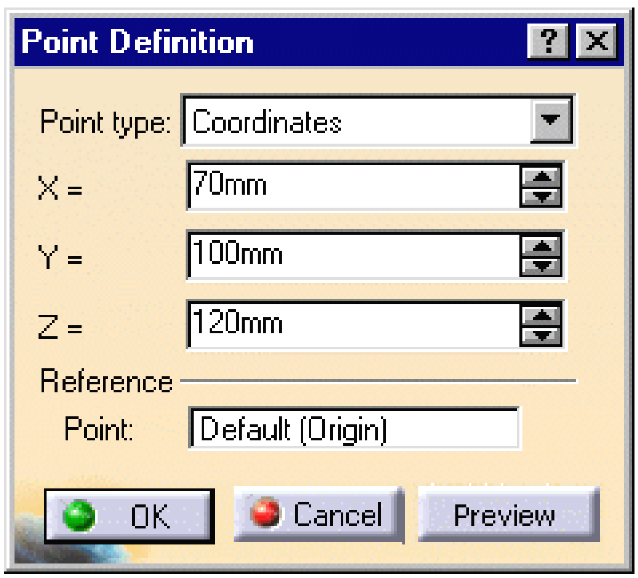
The Point Definition dialog box appears.

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.

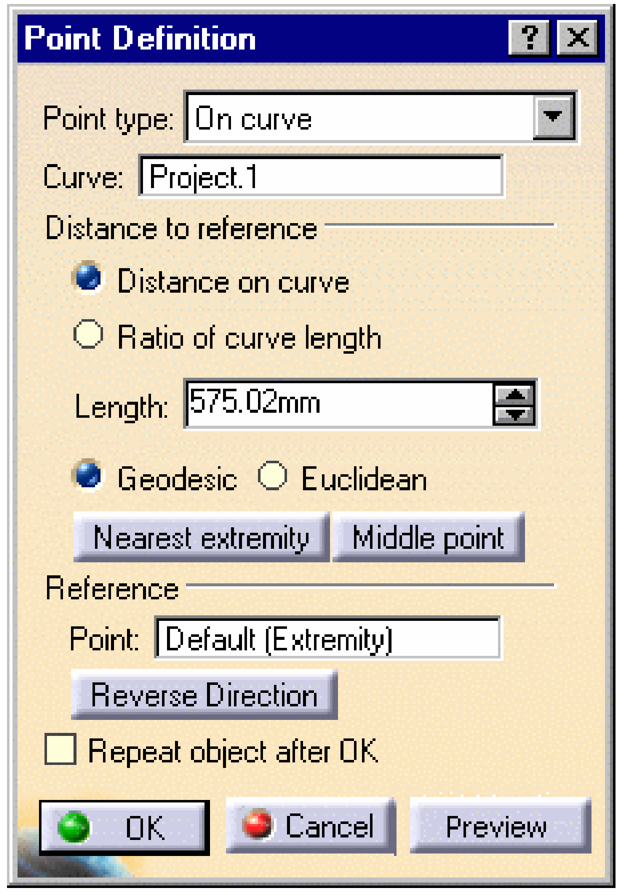


When creating a point within a user-defined axis-system, note that the **Coordinates in absolute axis-system** check button is added to the dialog box, allowing you to be define, or simply find out, the point's coordinates within the document's default axis-system.
If you create a point using the coordinates 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.

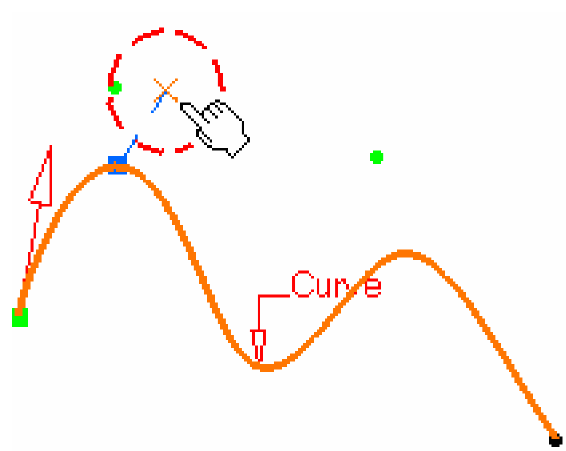
On curve

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

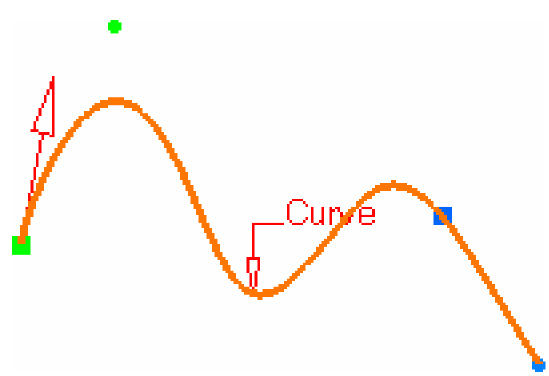


If this point is not on the curve, it is projected onto the curve.
If no point is selected, the curve's extremity is used as reference.

- 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.

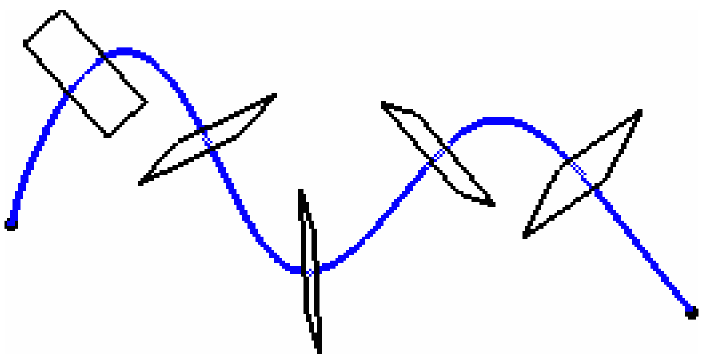
You can also:

- 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.

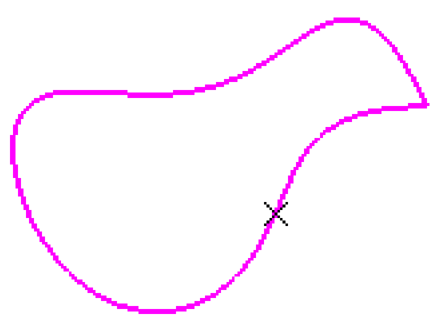
- use the **Reverse Direction** button to display:
  - the point on the other side of the reference point (if a point was selected originally)
  - the point from the other extremity (if no point was selected originally).
- click the **Repeat object after OK** if you wish to create equidistant points 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.

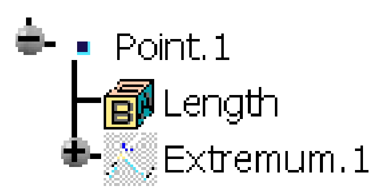
You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new geometrical set by checking the **Create in a new geometrical set** button.
If the button is not checked the instances are created in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



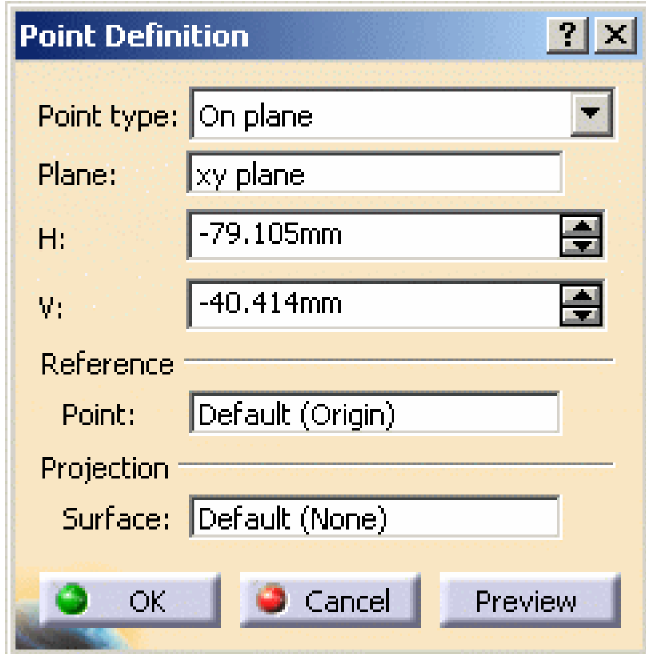
- If the curve is infinite and no reference point is explicitly given, by default, the reference point is the projection of the model's origin
- If the curve is a closed curve, either the system detects a vertex on the curve that can be used as a reference point, or it creates an extremum point, and 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.

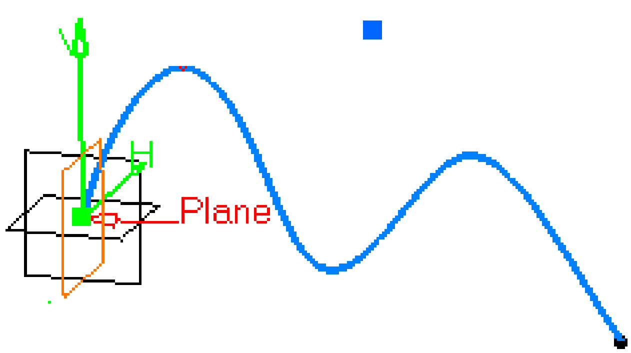
  If no point is selected, the projection of the model's origin on the plane is taken as reference.

- 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.

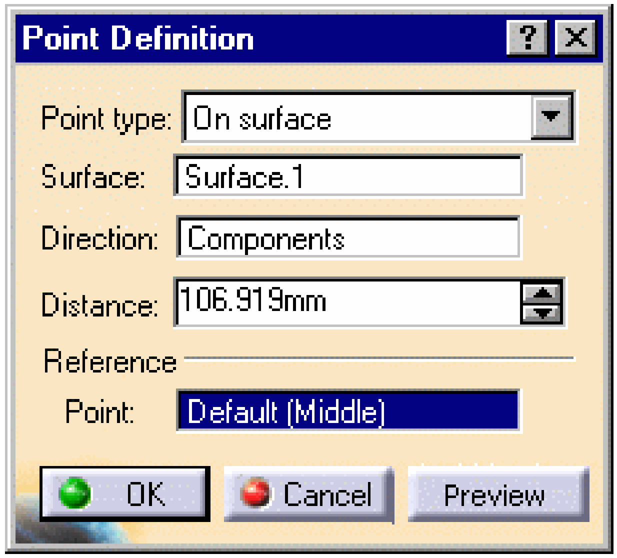
  Would the plane move, during an update for example, the reference direction would then be projected on the plane.



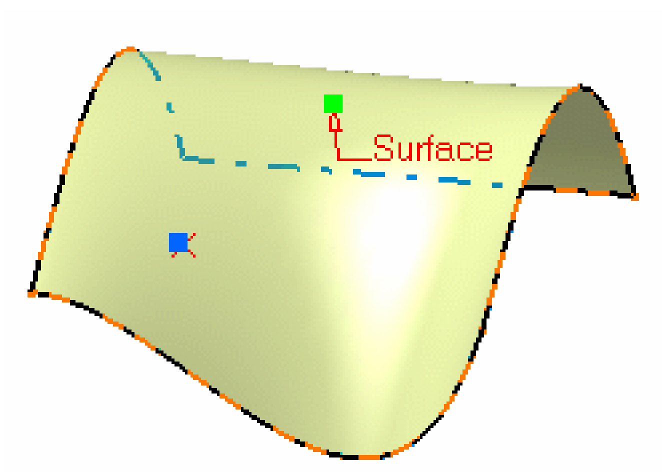
- Click in the plane to display a point.

On surface

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



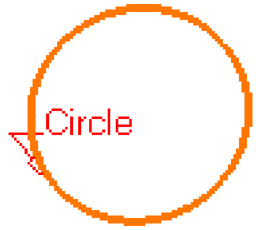
- Optionally, select a reference point. By default, the surface's middle point is taken as reference.
- You can select an element to take its orientation as reference direction or a plane to take its normal as reference direction.
  You can also use the contextual menu to specify the X, Y, Z components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

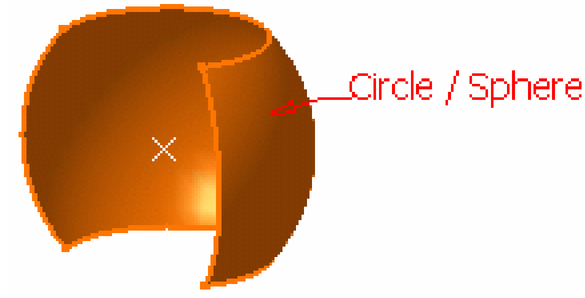


Circle/Sphere center

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

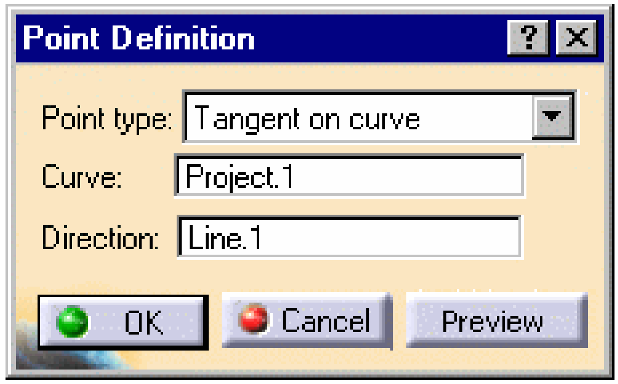
A point is displayed at the center of the selected element.





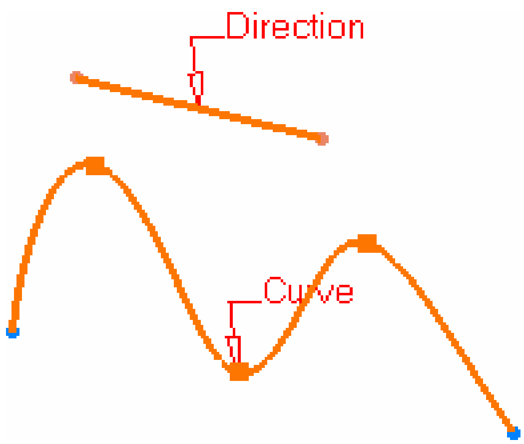
Tangent on curve

- 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.

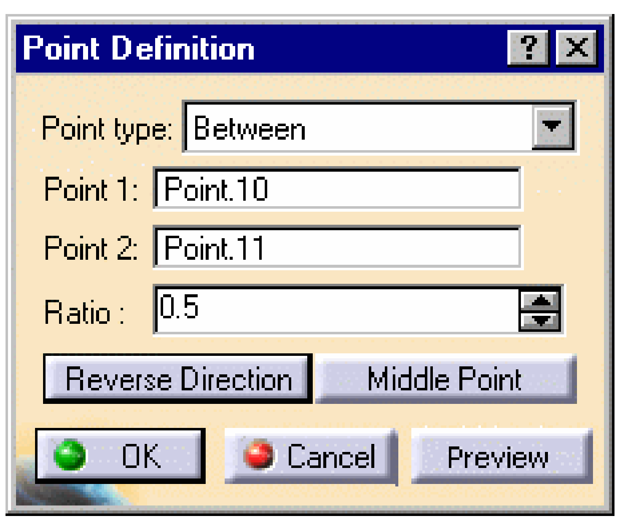


- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.

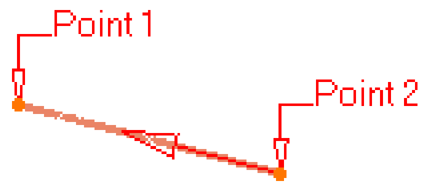
For further information, refer to the Managing Multi-Result Operations chapter.

Between

- Select any two points.

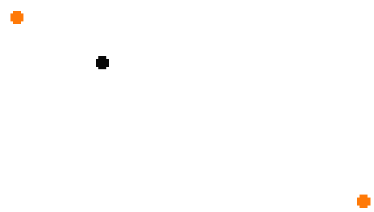


- Enter the ratio, that is the percentage of the distance from the first selected point, at which the new point is to be.
  You can also click **Middle Point** button to create a point at the exact midpoint (ratio = 0.5).



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.

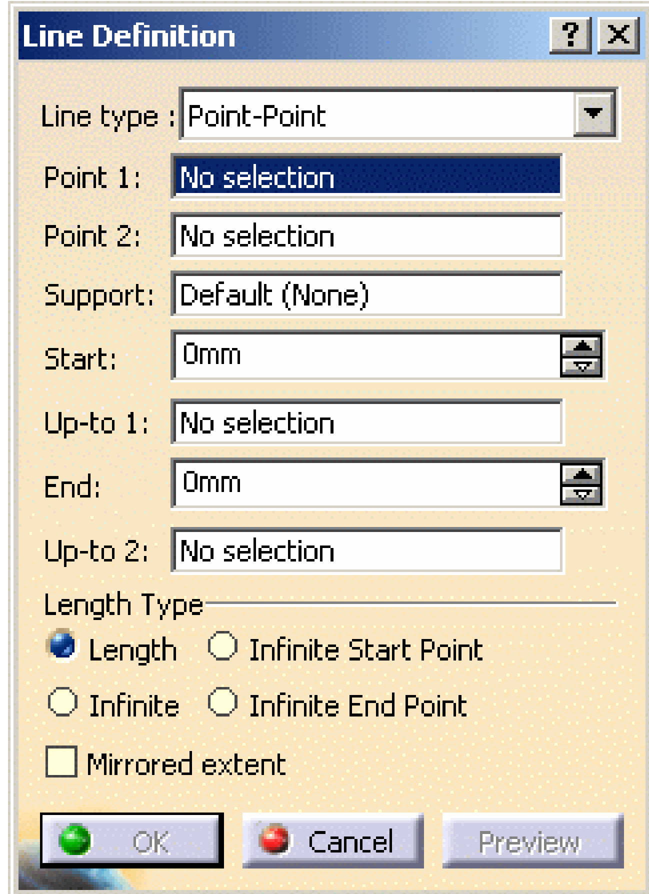
Defining the line type

Point - Point

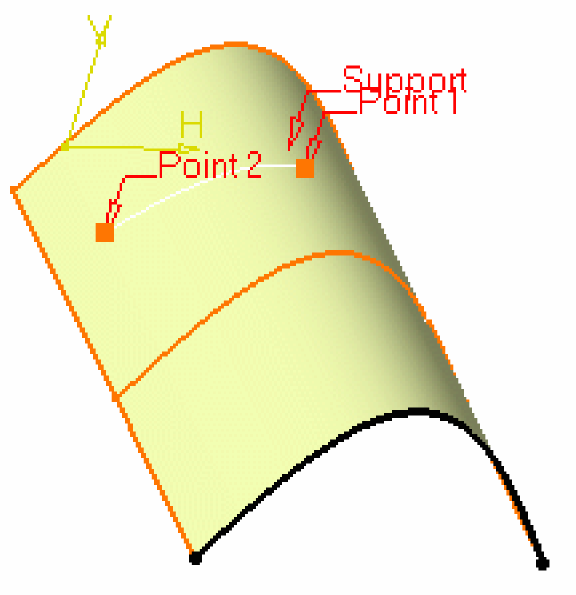
This command is only available with the Generative Shape Design 2 product.

- Select two points.

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

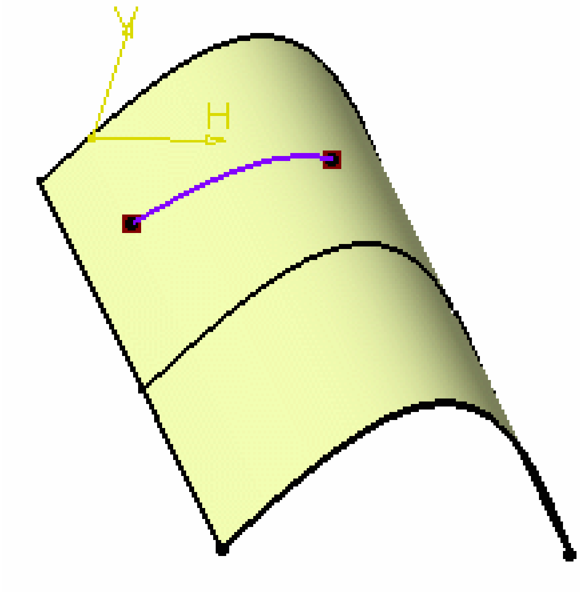


- If needed, select a support surface.
  In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below).
  If no surface is selected, the line is created between the two points based on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

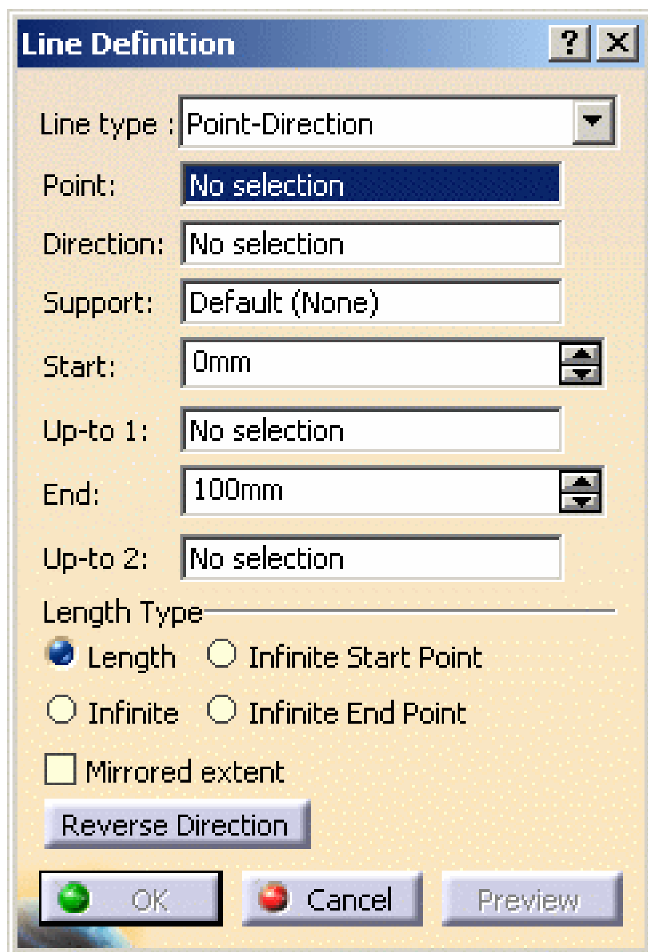


- Specify the **Start** and **End** points of the new line, that is the line endpoint location in relation to the points initially selected. These **Start** and **End** points are necessarily beyond the selected points, meaning the line cannot be shorter than the distance between the initial points.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

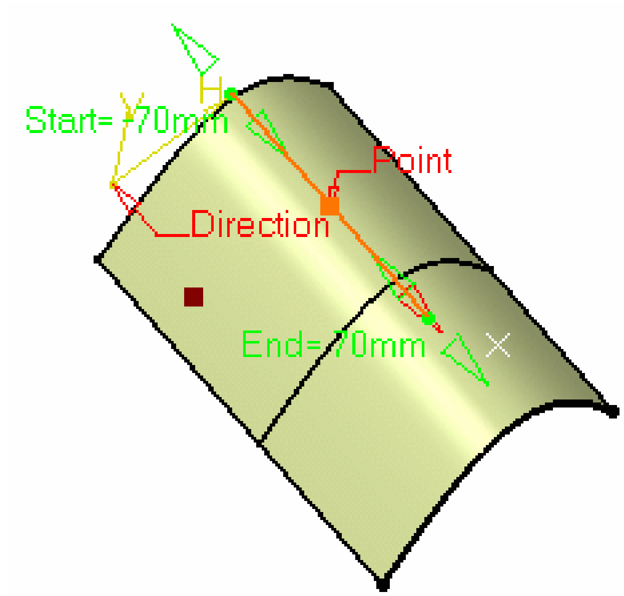
The projections of the 3D point(s) must already exist on the selected support.

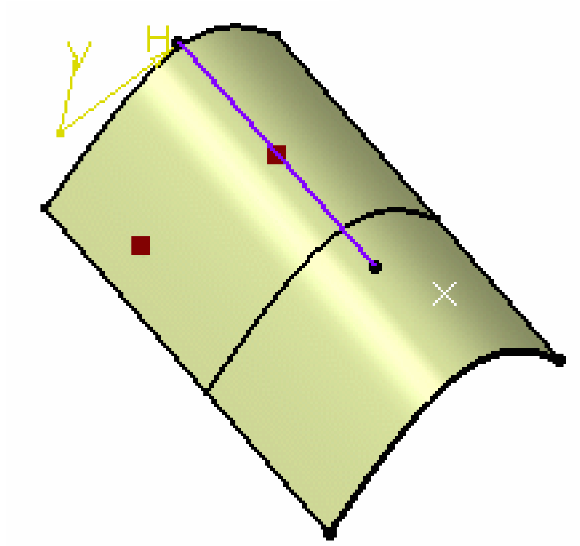
Point - Direction

- 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.





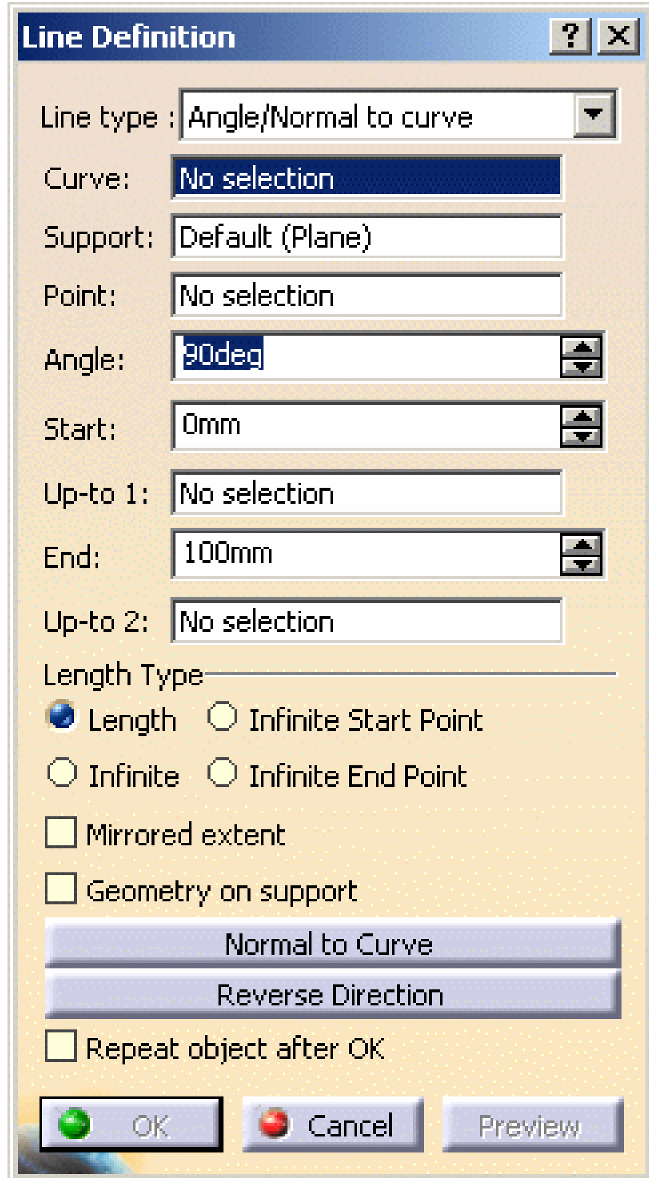
The projections of the 3D point(s) must already exist on the selected support.

Angle or Normal to curve

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

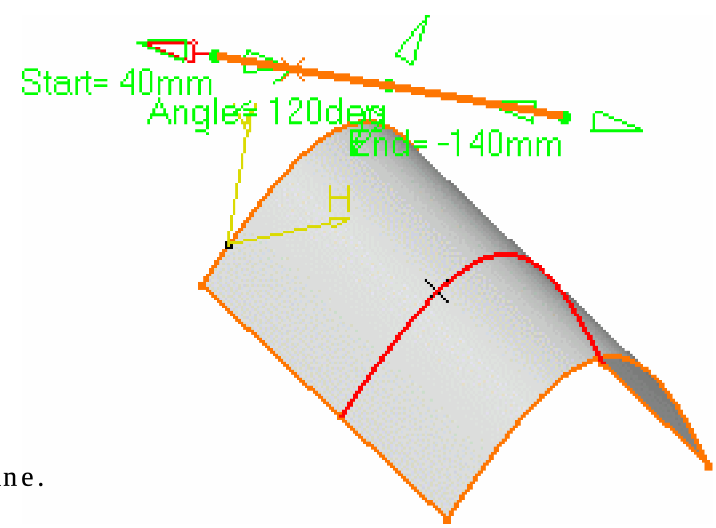
- If the selected curve is planar, then the **Support** is set to Default (Plane).

- 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.

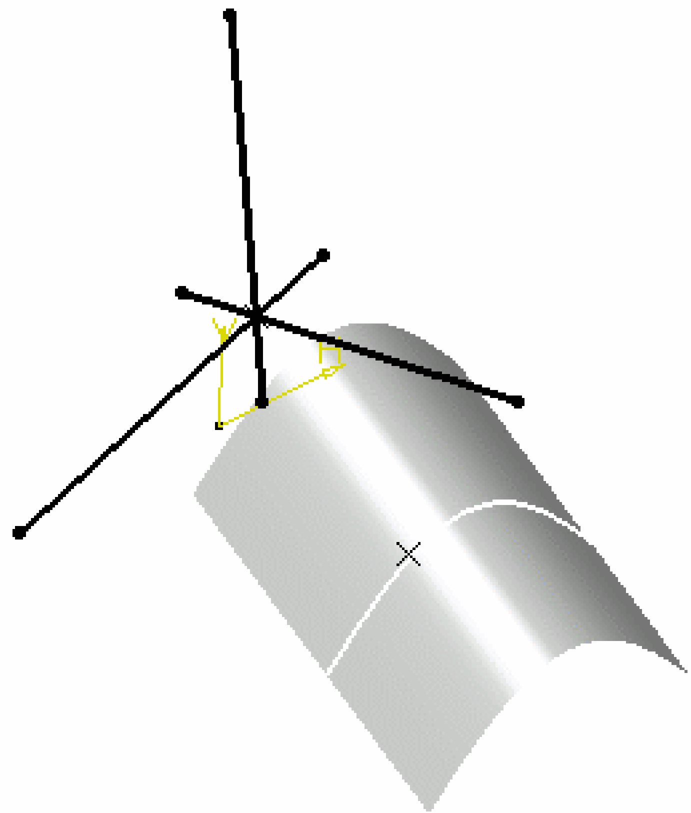
A line is displayed at the given angle with respect to the tangent to the reference curve at the selected point. These elements are displayed in the plane tangent to the surface at the selected point.
You can click on the **Normal to Curve** button to specify an angle of 90 degrees.
Proposed **Start**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



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

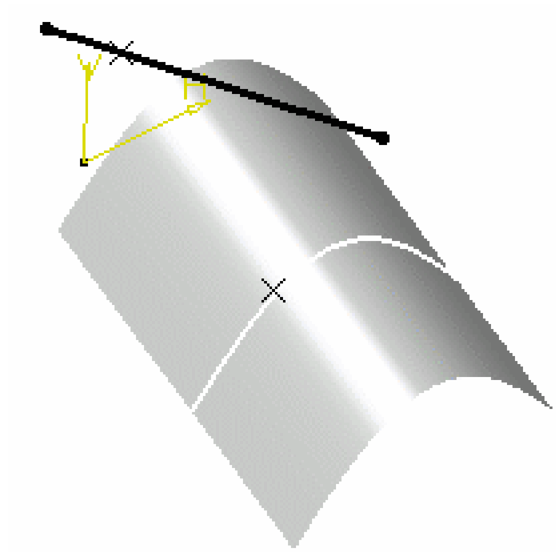
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.

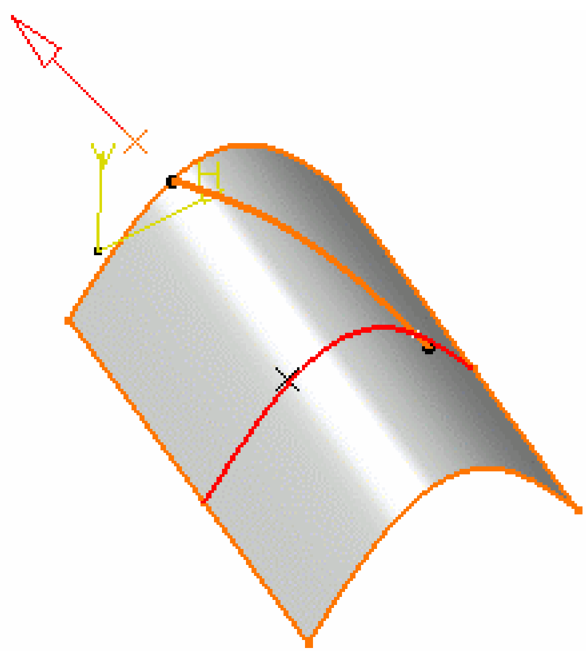
As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.



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.

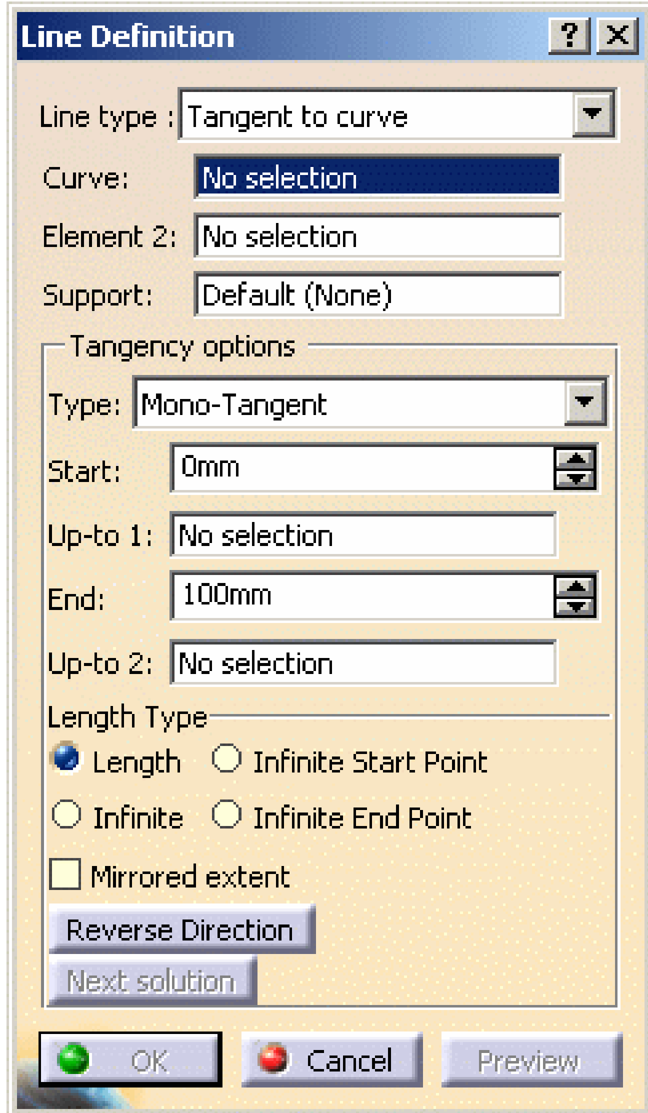




Geometry on support option not checked *Geometry on support option checked*

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

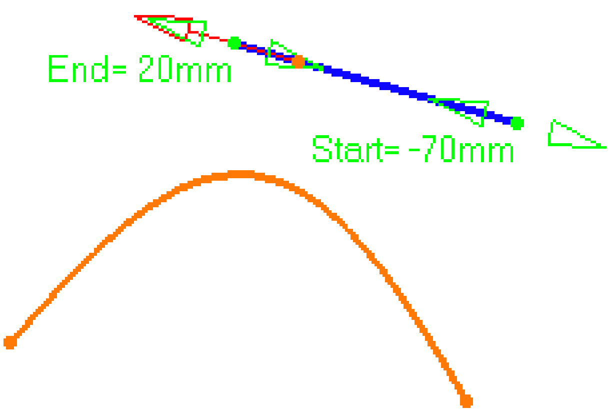


- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
  - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
  - If a second curve is selected (or a point in bi-tangent mode), you need to select a support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

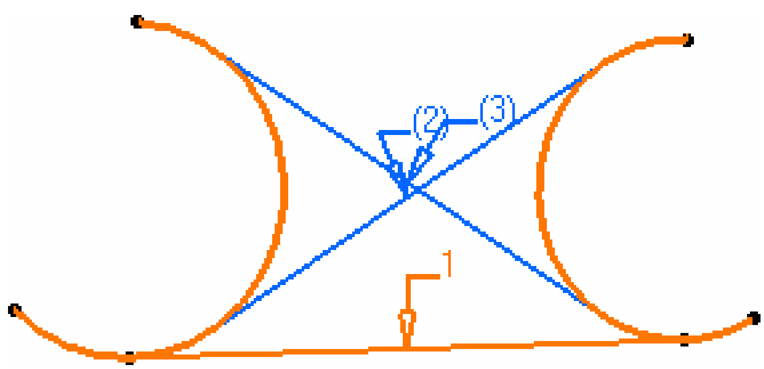
- If the selected curve is a line, then the **Support** is set to Default (Plane).

- 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.

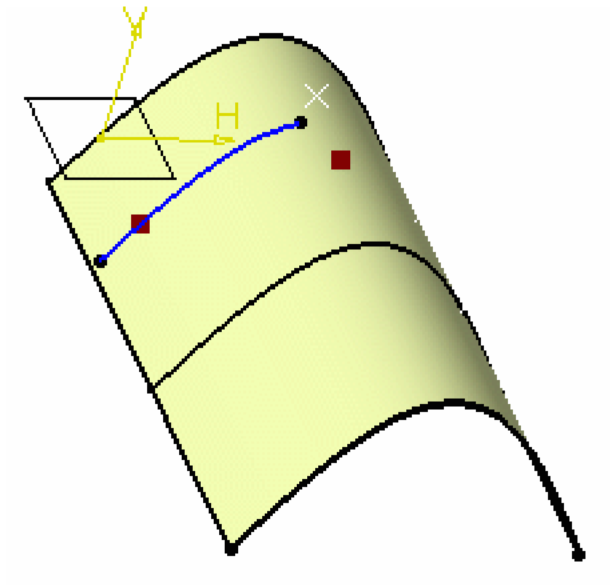


Line tangent to curve at a given point



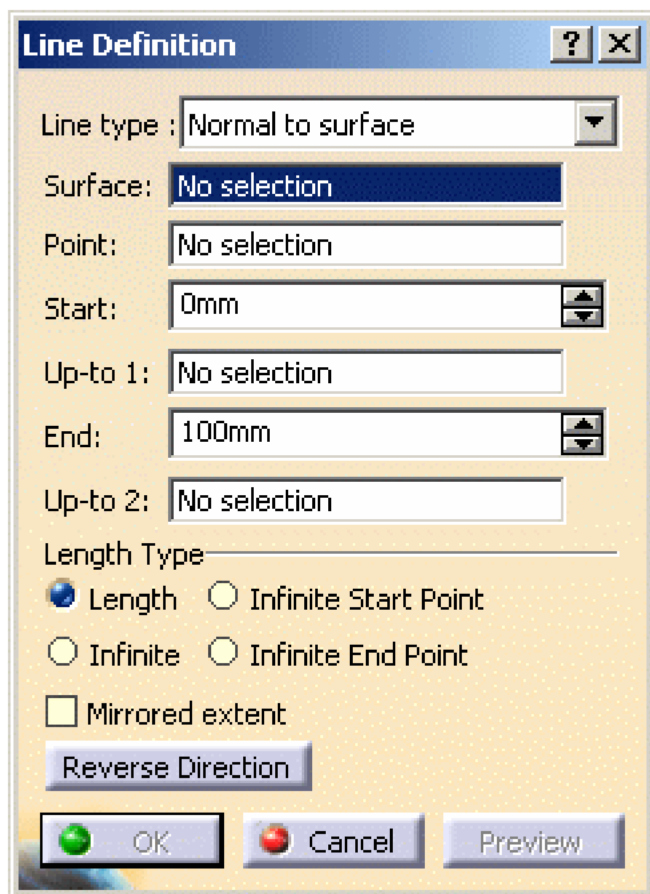
Line tangent to two curves

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



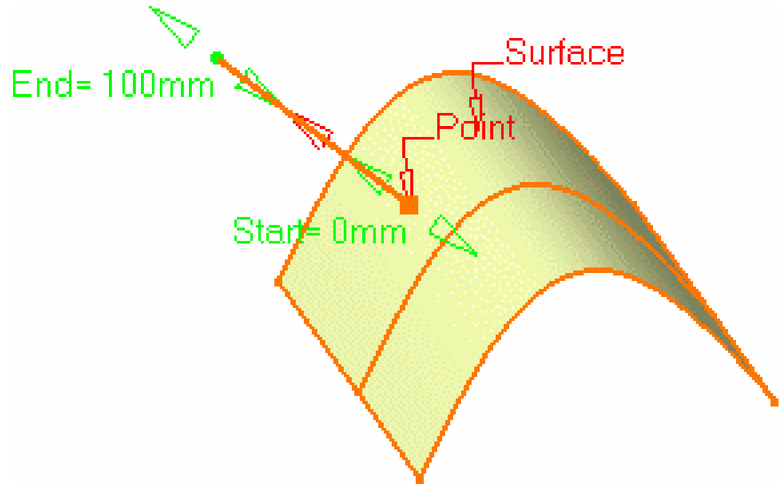
Normal to surface

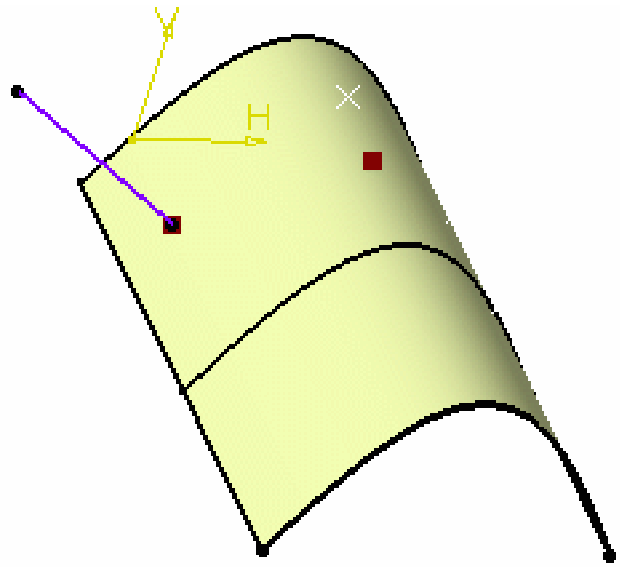
- Select a reference **Surface** and a **Point**.
  A vector normal to the surface is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.



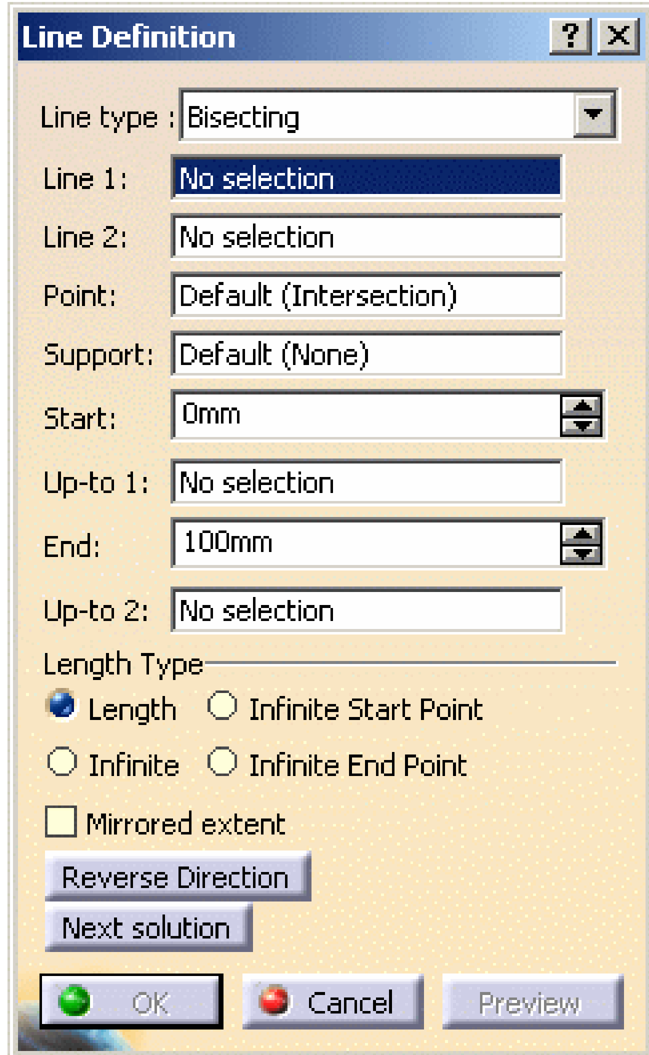
If the point does not lie on the support surface, the minimum distance between the point and the surface is computed, and the vector normal to the surface is displayed at the resulted reference point.

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

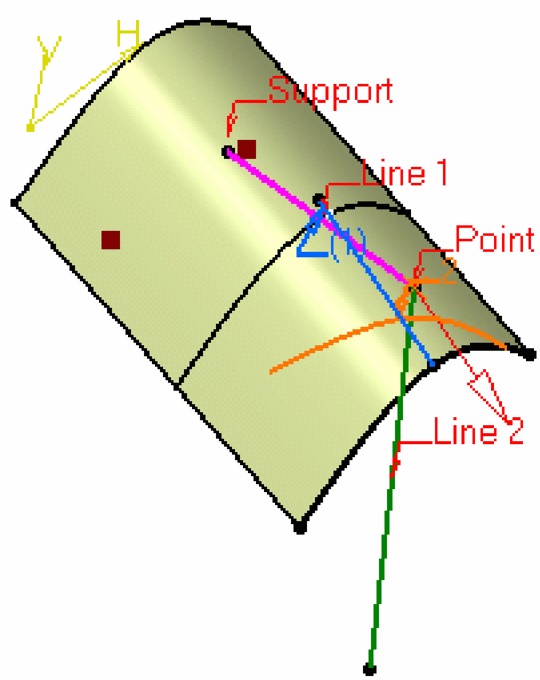




Bisecting



- 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

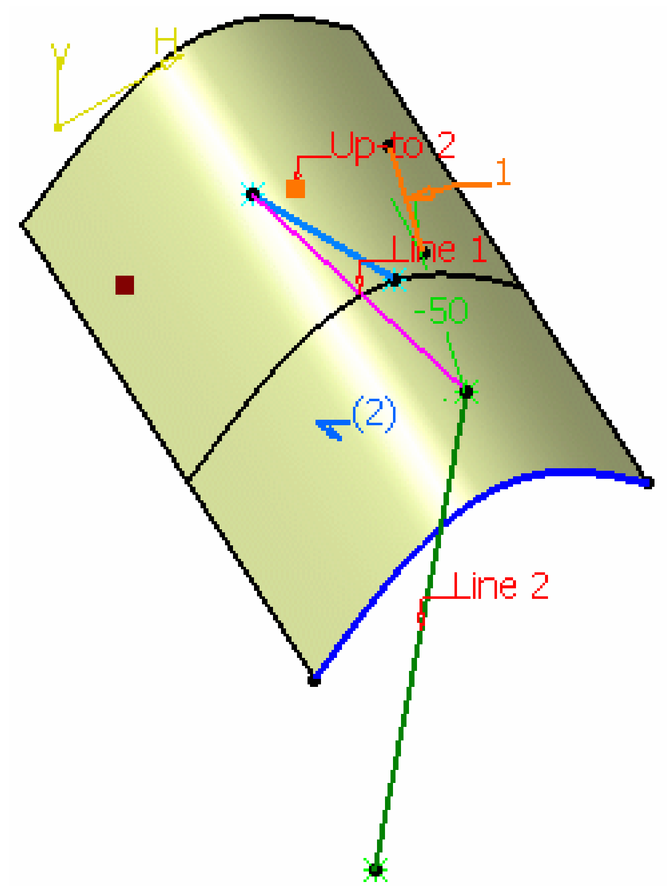
This capability allows you to create a line up to a point, a curve, or a surface.

- 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.

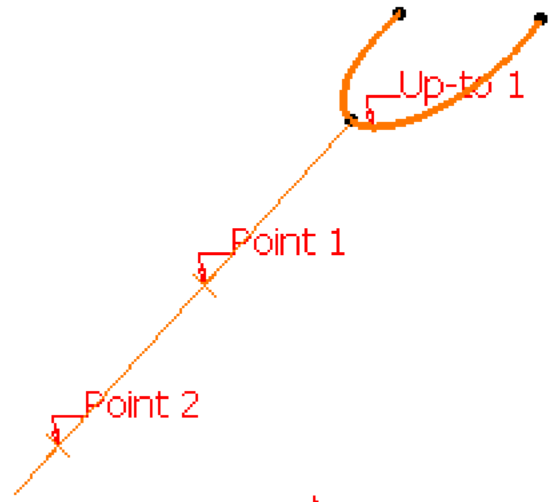
Here is an example with the Bisecting line type, the **Length** Length type, and a point as **Up-to 2** element.



Up to a curve

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

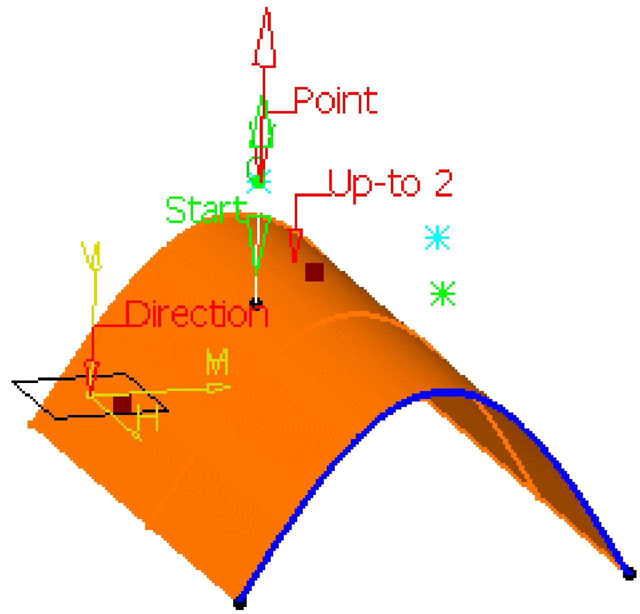
  Here is an example with the Point-Point line type, the **Infinite End** Length type, and a curve as the **Up-to 1** element.



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.

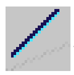
Defining the length type

- 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.

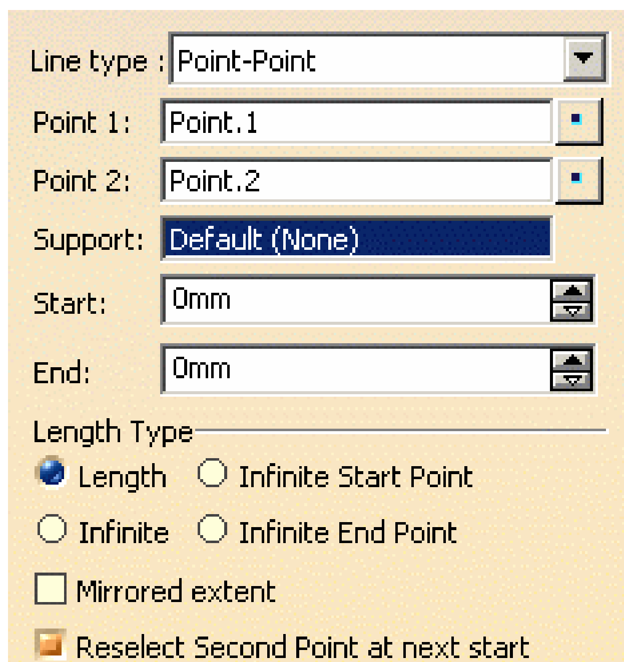
Reselecting automatically a second point

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.



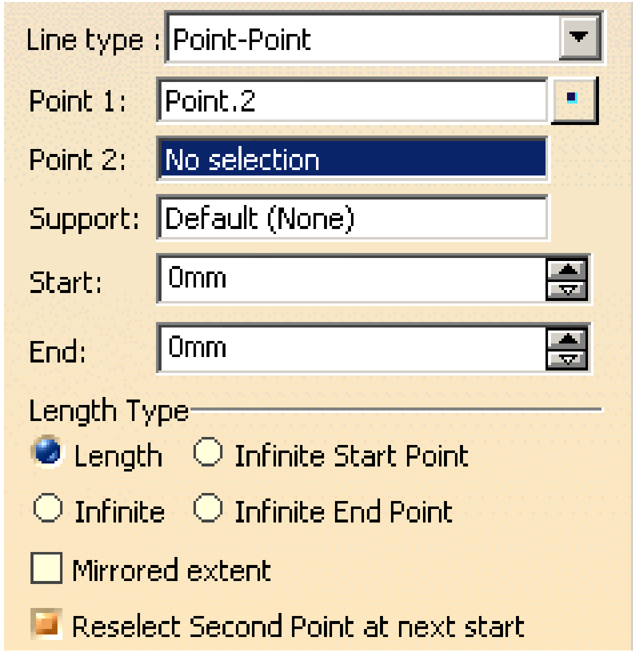
The **Reselect Second Point at next start** option appears in the Line dialog box.

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.

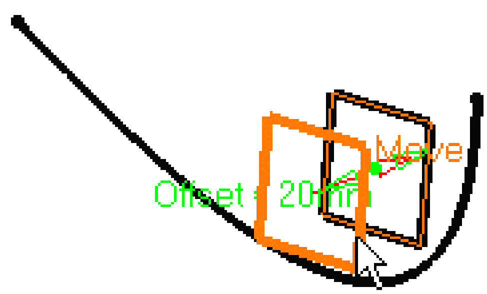
2. Use the combo to choose the desired **Plane type**.

Once you have defined the plane, it is represented by a red square symbol, which you can move using the graphic manipulator.

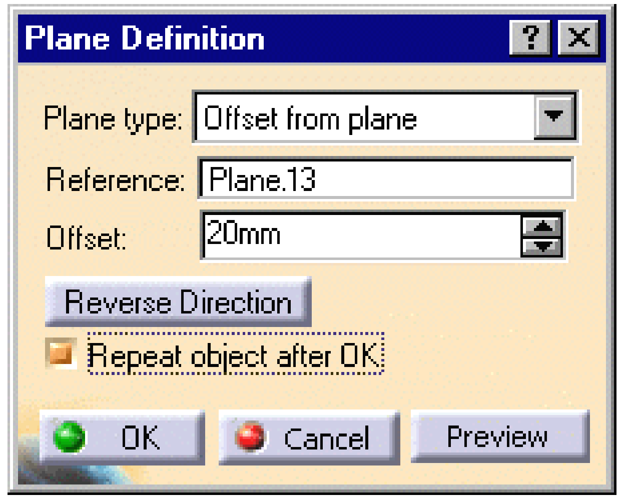
Offset from plane

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

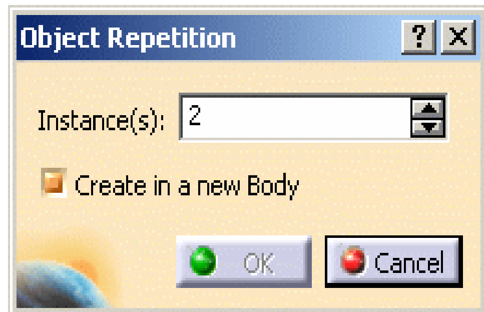
A plane is displayed offset from the reference plane.



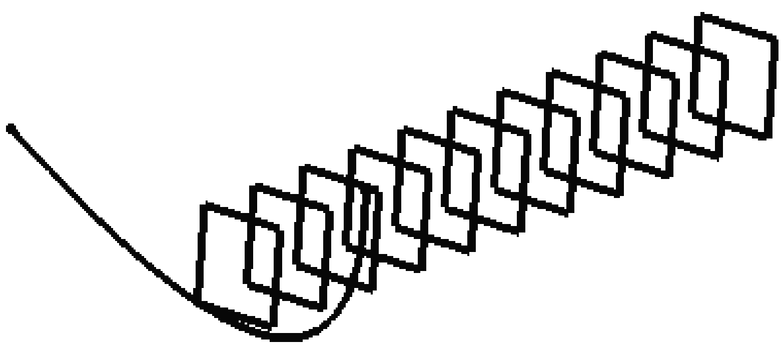
Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.



- 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.

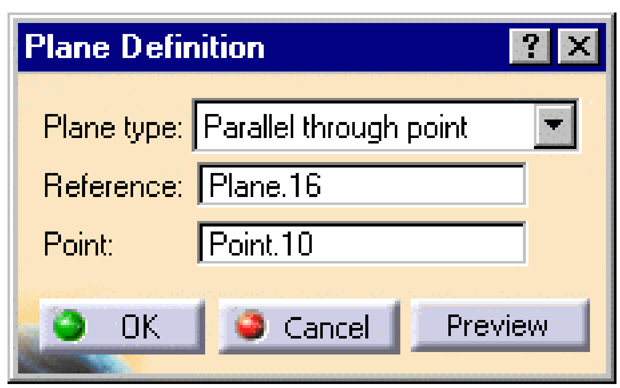


As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Offset** value.

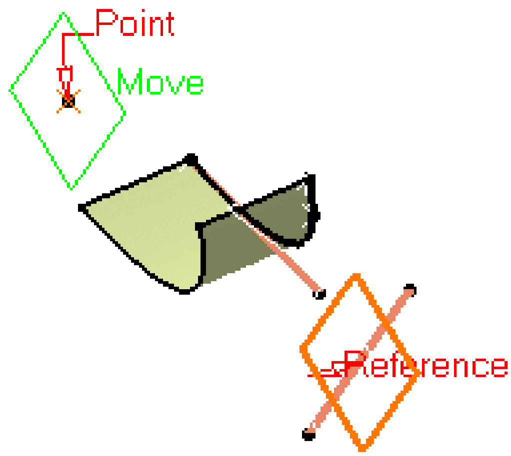


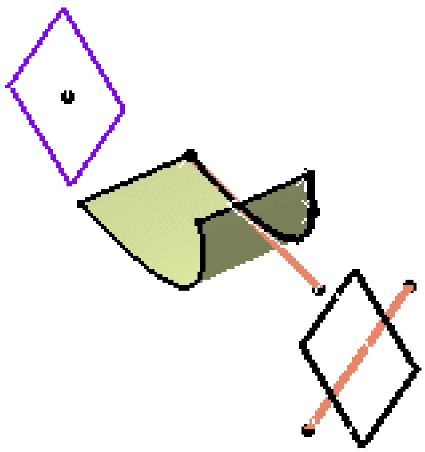
Parallel through point

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



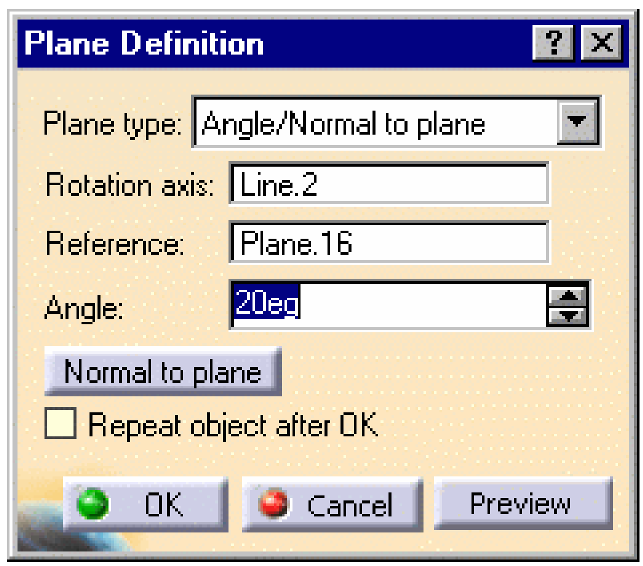
A plane is displayed parallel to the reference plane and passing through the selected point.



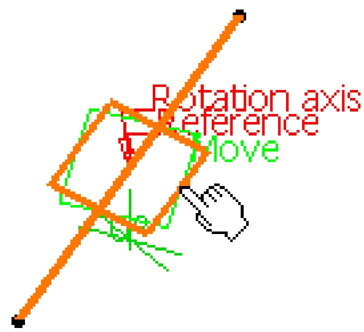


Angle or normal to plane

- 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.



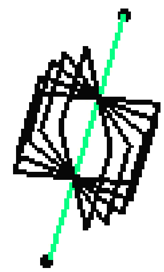
A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



- 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.

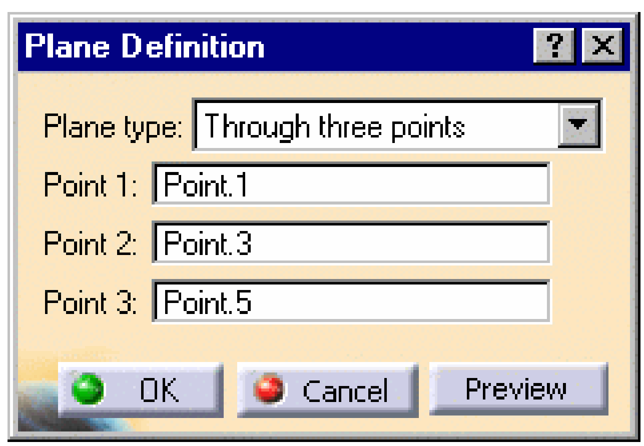
Here we created five planes at an angle of 20 degrees.



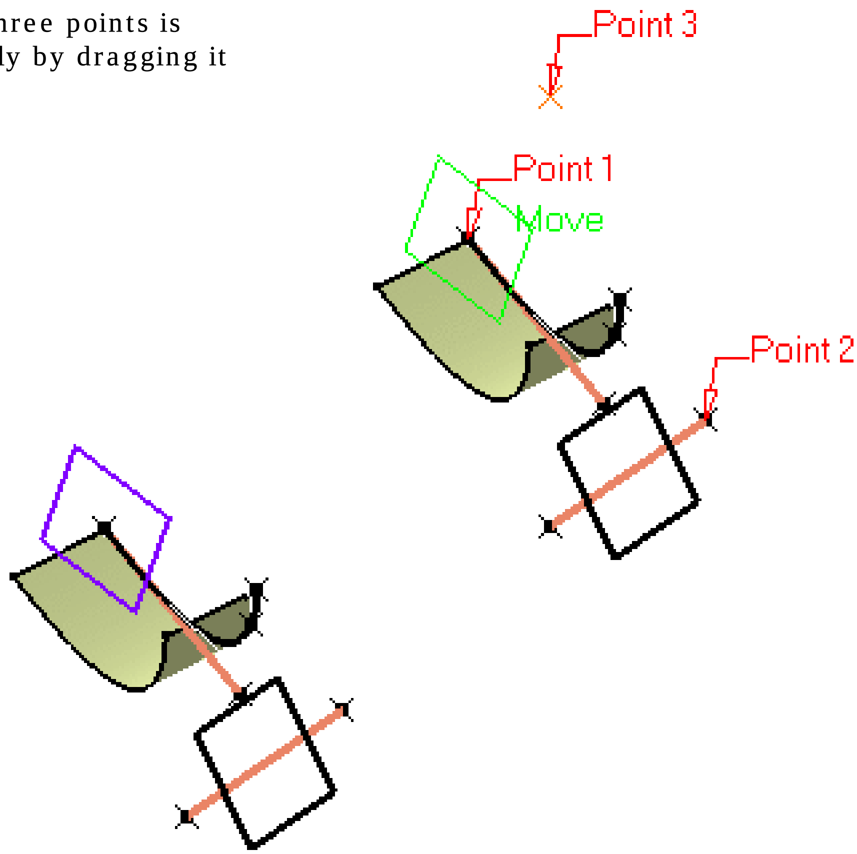
This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

Through three points

- Select three points.

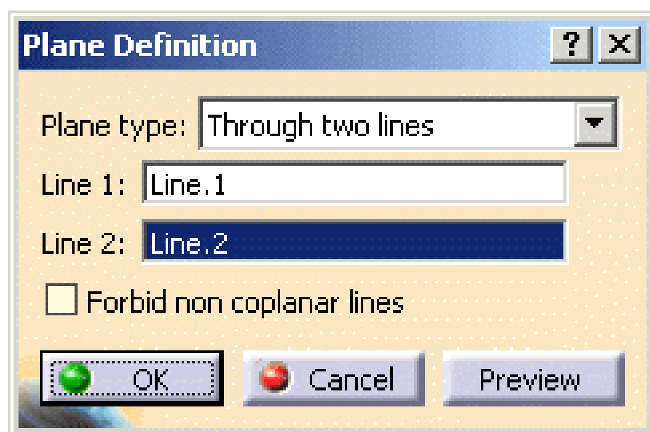


The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.

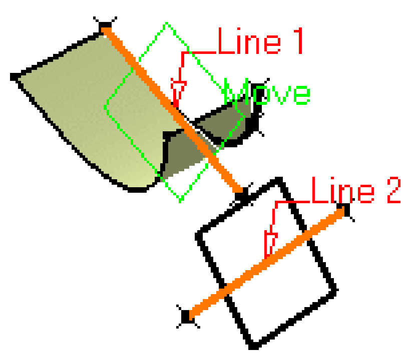


Through two lines

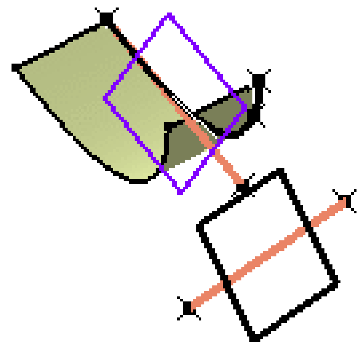
- Select two lines.



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.

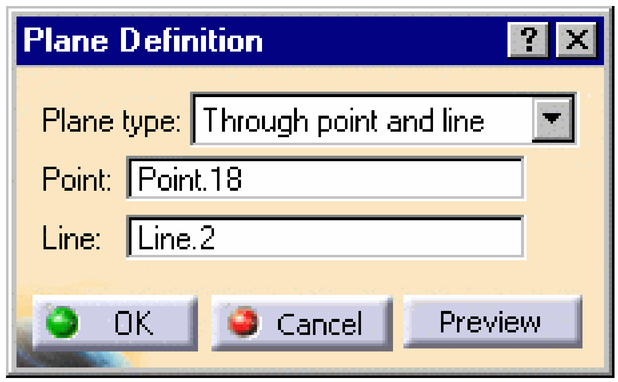


Check the **Forbid non coplanar lines button** to specify that both lines be in the same plane.

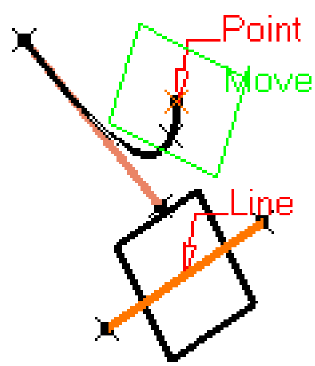


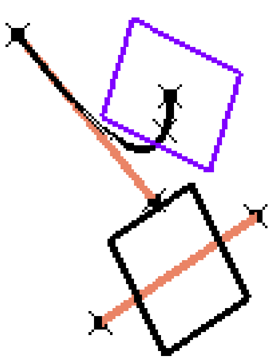
Through point and line

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



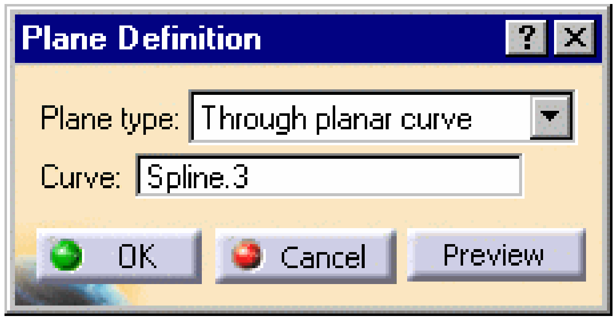
The plane passing through the point and the line is displayed.



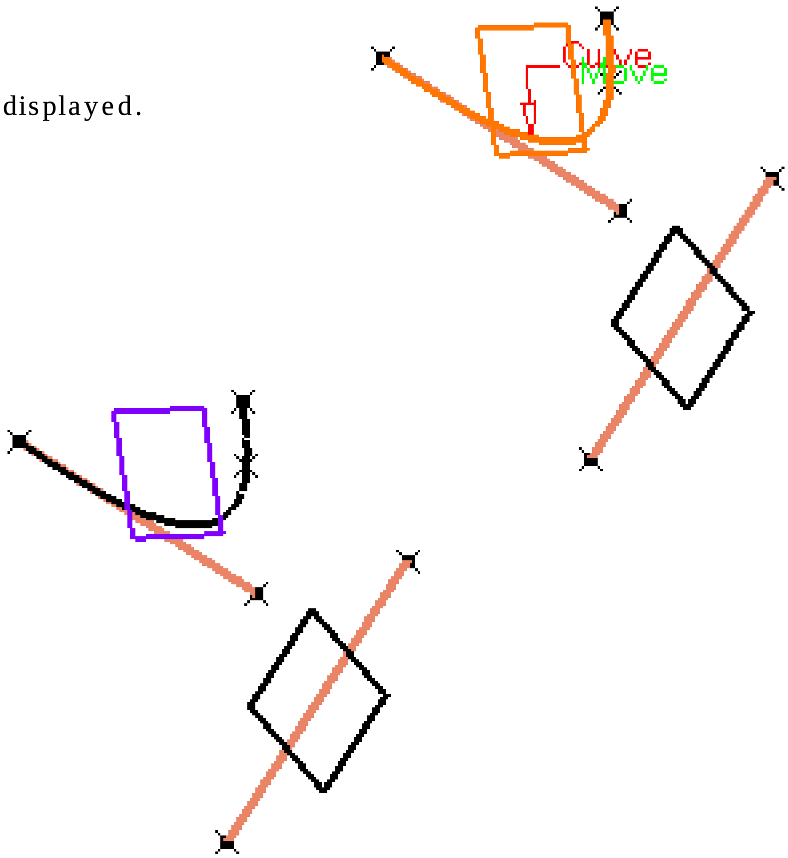


Through planar curve

- Select a planar **Curve**.

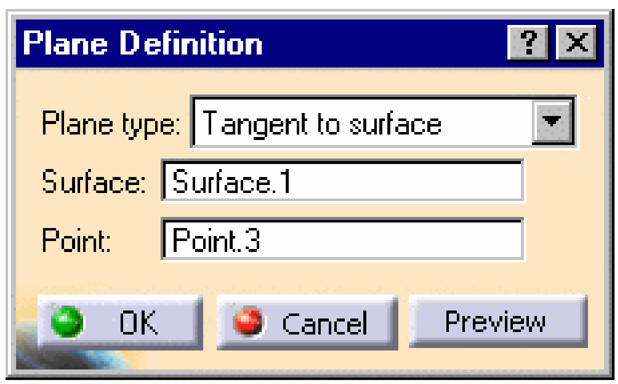


The plane containing the curve is displayed.

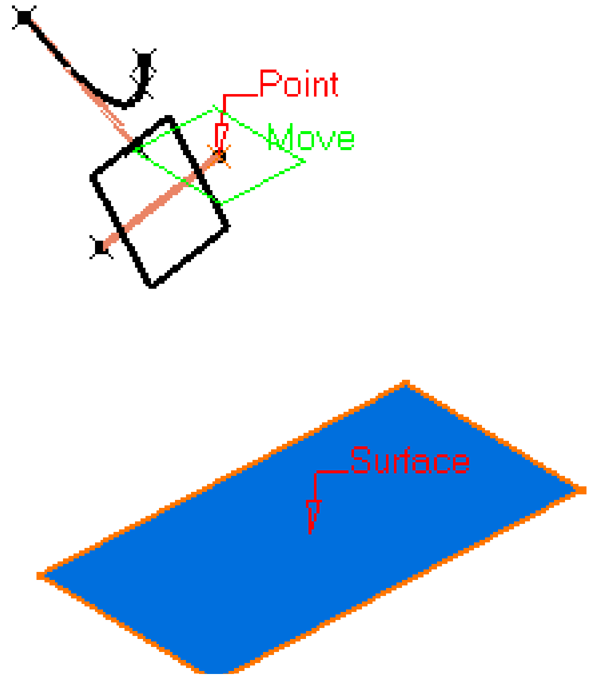


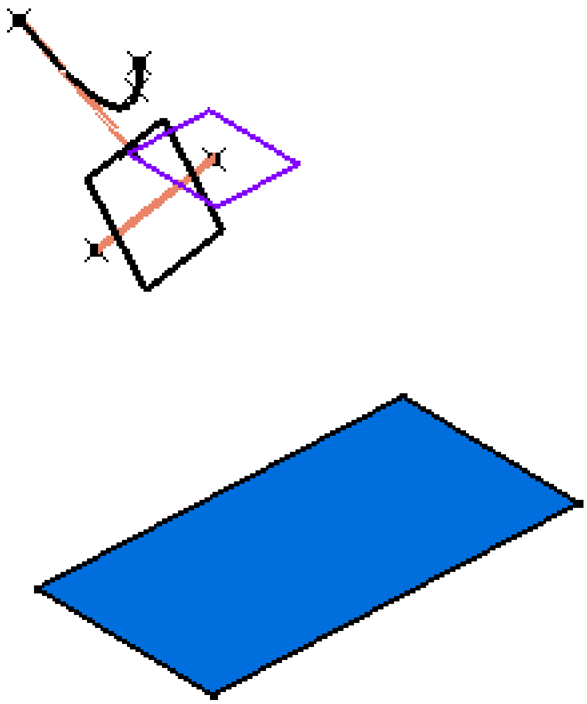
Tangent to surface

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



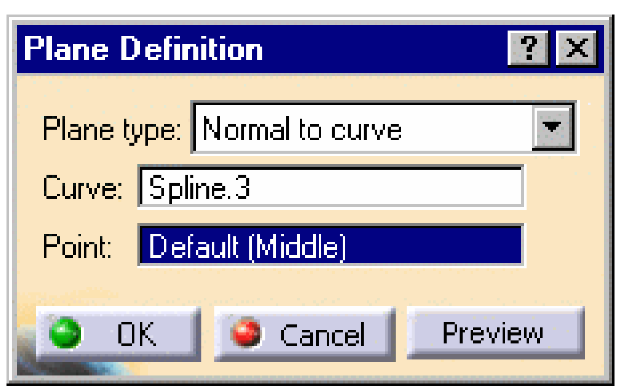
A plane is displayed tangent to the surface at the specified point.



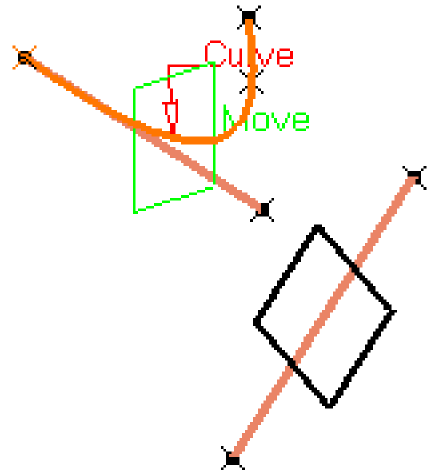


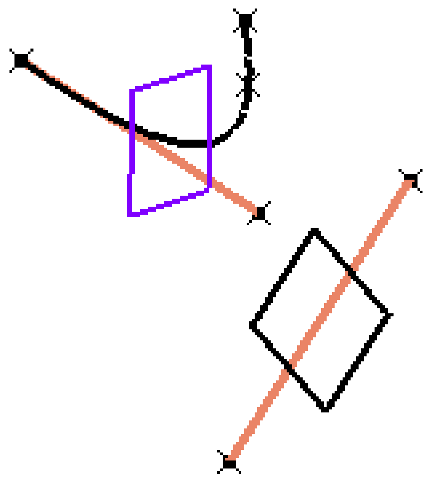
Normal to curve

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



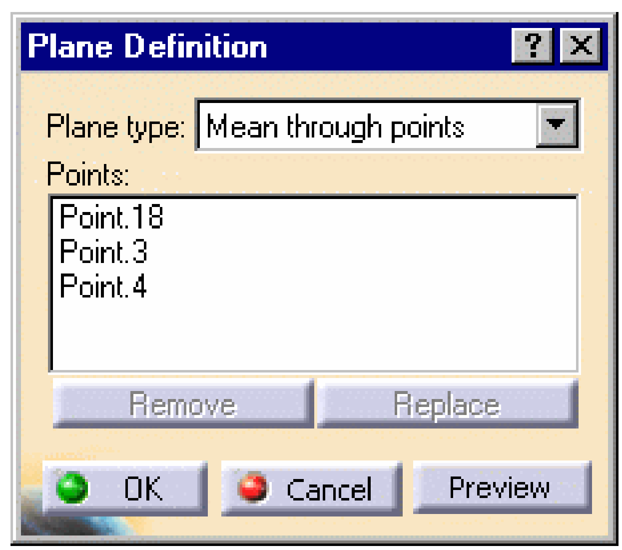
A plane is displayed normal to the curve at the specified point.





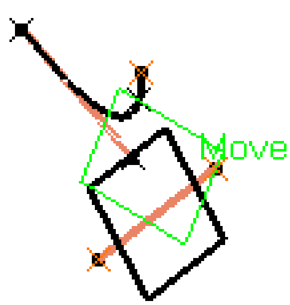
Mean through points

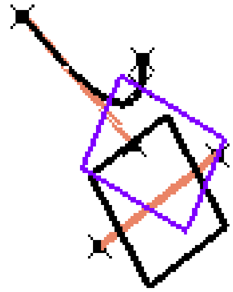
- Select three or more points to display the mean plane through these points.



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

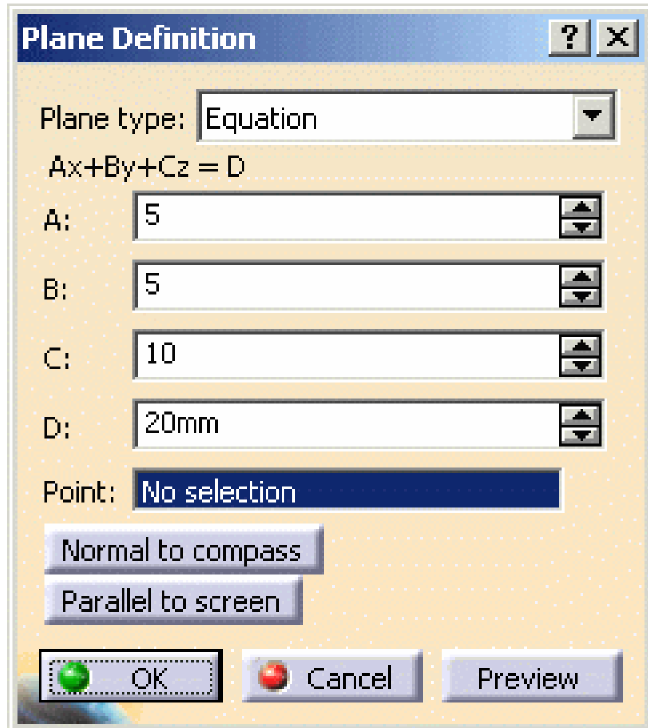
- **Remove** the selected point
- **Replace** the selected point by another point.





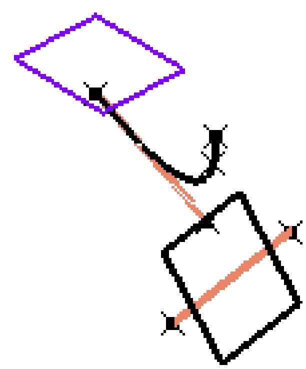
Equation

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

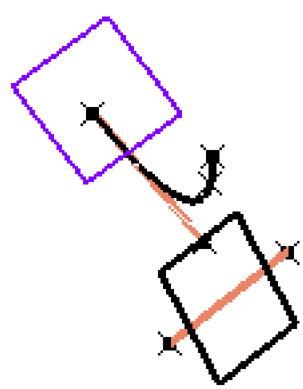


Select a point to position the plane through this point, you are able to modify **A**, **B**, and **C** components, the **D** component becomes grayed.

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.

Displaying Characteristic Curves

This task shows you how to manage characteristic curves.
Characteristic curves are displayed in the folded view of the part, as well as in the flattened view. They can be selected, though not edited, and be used as a support (to create points for example),

A surfacic flange or a stamp with a fillet must be created.

The following curves can be computed:

- **OML**: Outer Mold Line
  Intersection between external surfaces of the feature (before filleting) and the part
- **IML**: Inner Mold Line
  Intersection between internal surfaces of the feature (before filleting) and the part
  Computed only on a hydropressed flange
- **BTL**: Bent Tangent Line
  Limits of the fillet

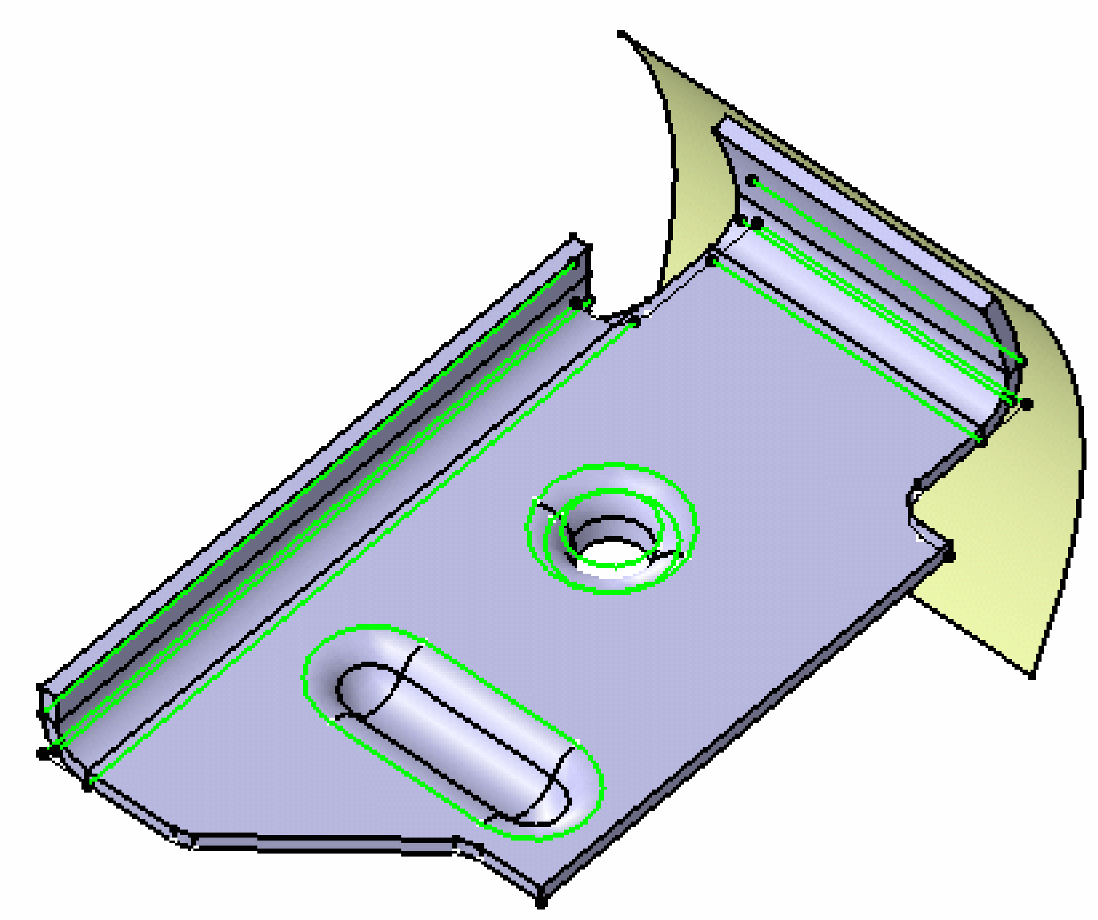
They must be computed on the following features:

- **Surfacic Flange**: the following curves are computed on both folded and flattened view (as defined in the Process tab of the Surfacic Flange Definition dialog box)
  - BTL (two BTL for each limit of the fillet)
  - OML
  - IML
- **Stamps with an internal cutout** (flanged hole, flanged cutout, etc): the following curves are computed on both folded and flattened view
  - BTL on the base feature
  - IML (in no show in the folded view)
  - OML (in no show in the folded view)
- **Stamps without an internal cutout** (bead, curve stamp, etc): the following curve is computed on both folded and flattened view
  - BTL on the base feature

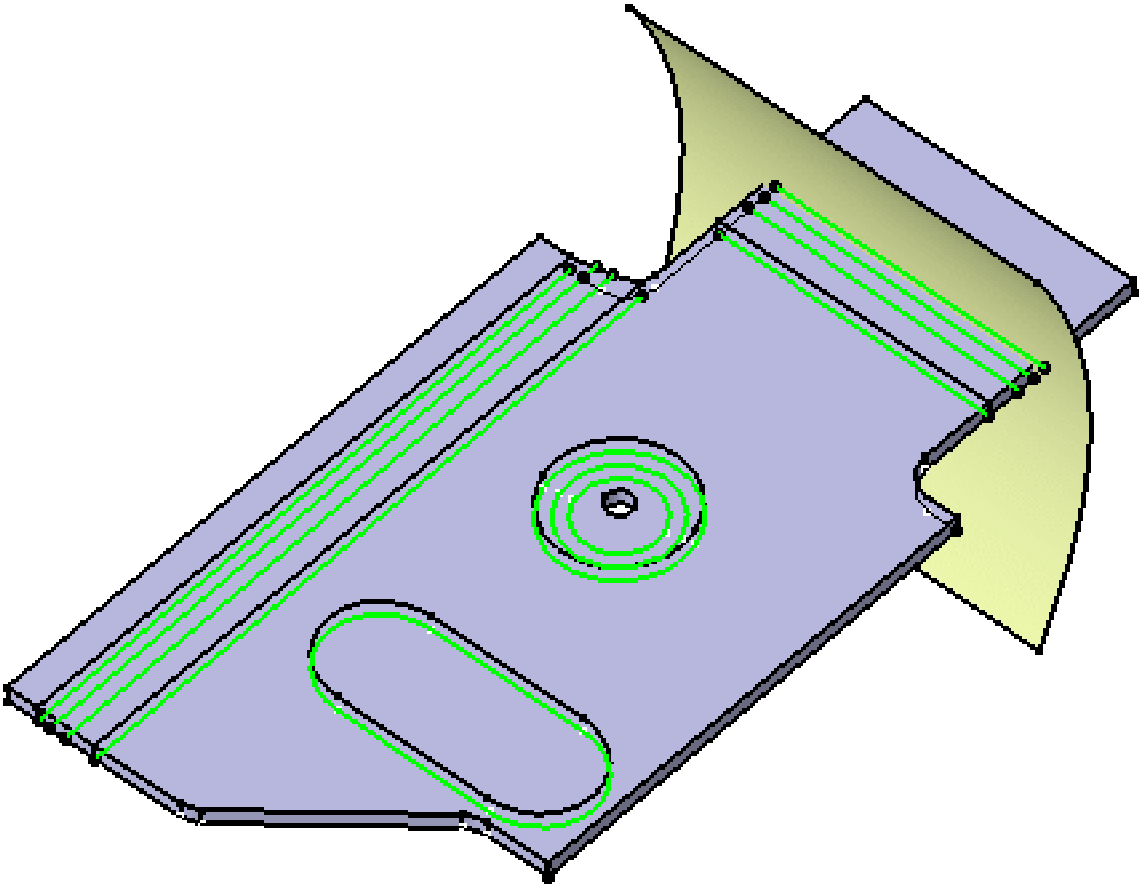
Open the CharacteristicCurves1.CATPart document.

Here is an example with two surfacic flanges, a bead and a flanged hole.
All characteristic curves are put in show in the following images.

Characteristic curves computed on the folded view



Characteristic curves computed on the flattened view



If you want to edit and modify a curve color, select it in the 3D geometry, right-click and choose the **Properties** contextual command. In the Properties dialog box, select the Graphic tab to access the graphic properties of the curve.

Looking For Aerospace SheetMetal Features

This task shows how to use the Search capabilities on Aerospace SheetMetal Features, in order to detect any specific kind of feature.

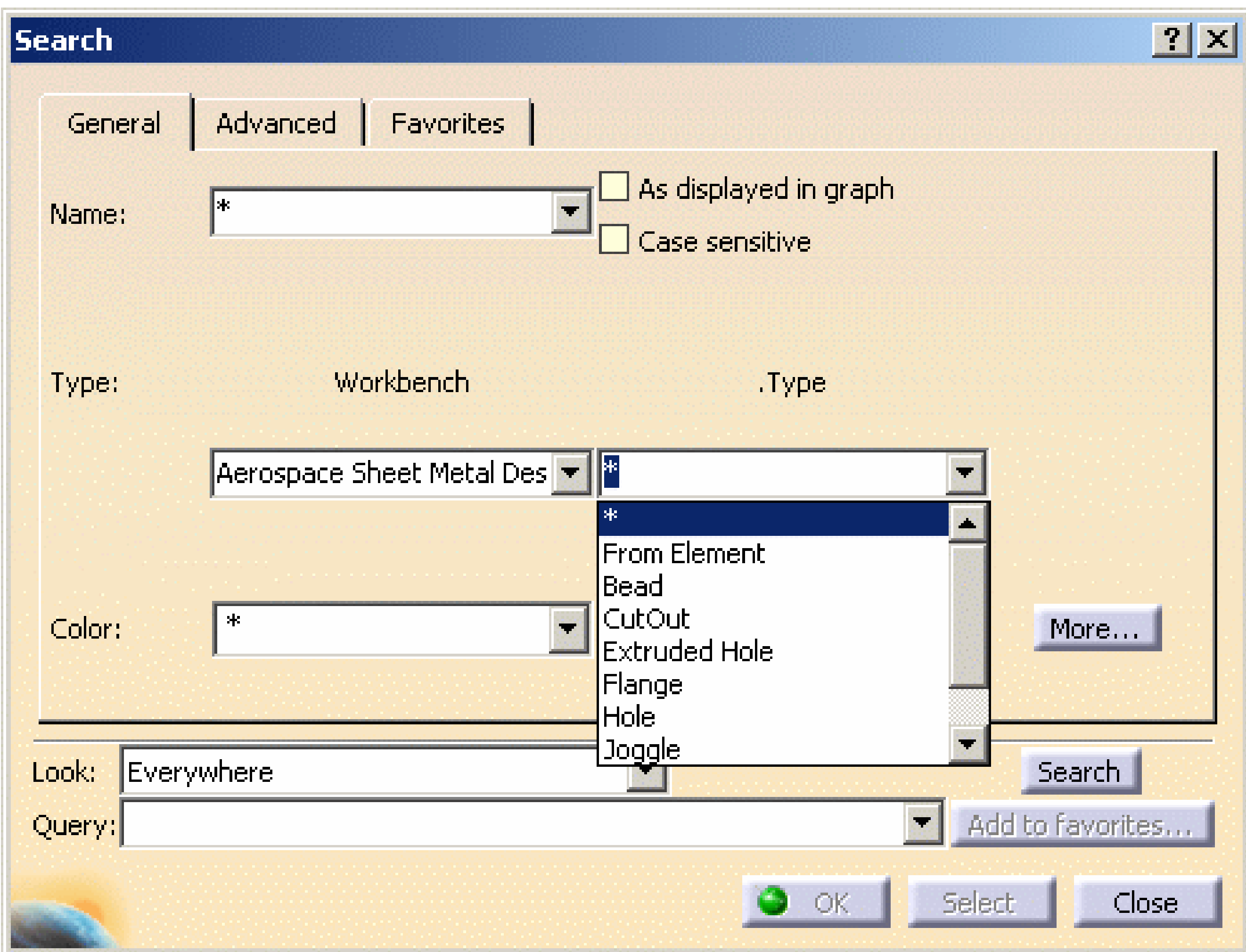
Open the PowerCopyStart.CATPart document.

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 Aerospace Sheet Metal Design features from the **Type** list:

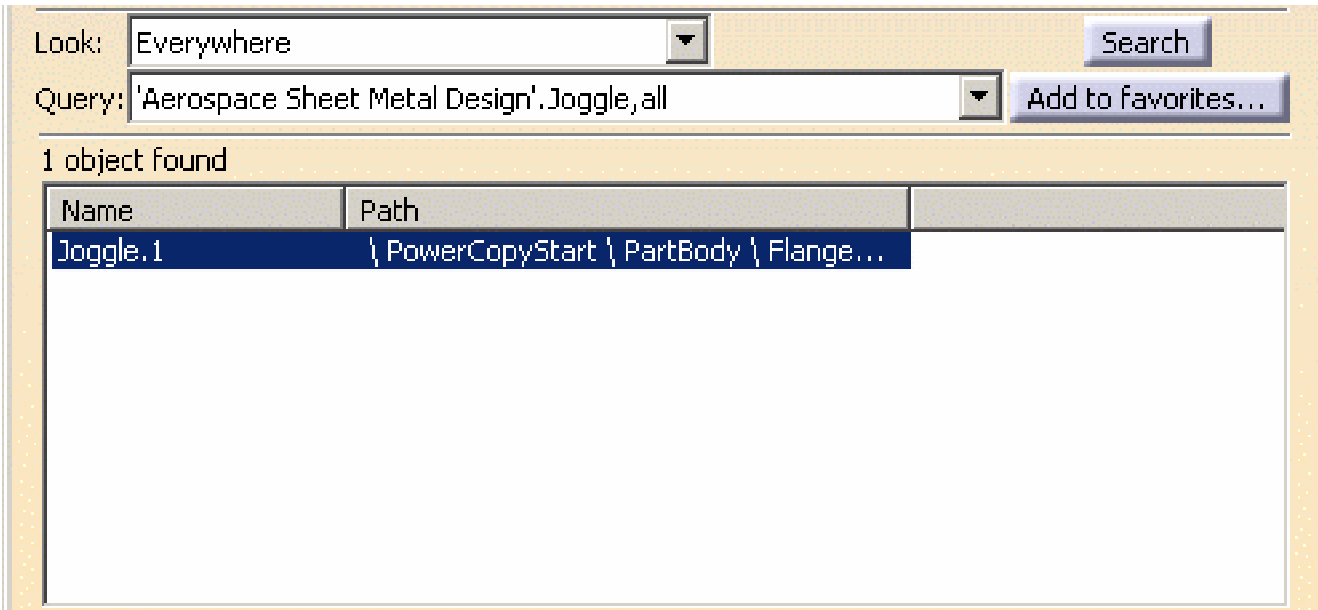


3. Select the type of feature you wish to find within the open .CATPart document.

   Here we chose *Joggle*.

4. Click the **Search** button.

The list of all elements of the selected type is displayed in the Objects found field:



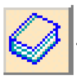
You can select an element from the list, it will be highlighted in the geometry area.
To find out more on the search capabilities, refer to Selecting Using the Search... Command (General Mode) and Selecting Using the Search... Command (Favorites and Advanced Modes) from the *Infrastructure User's Guide*.

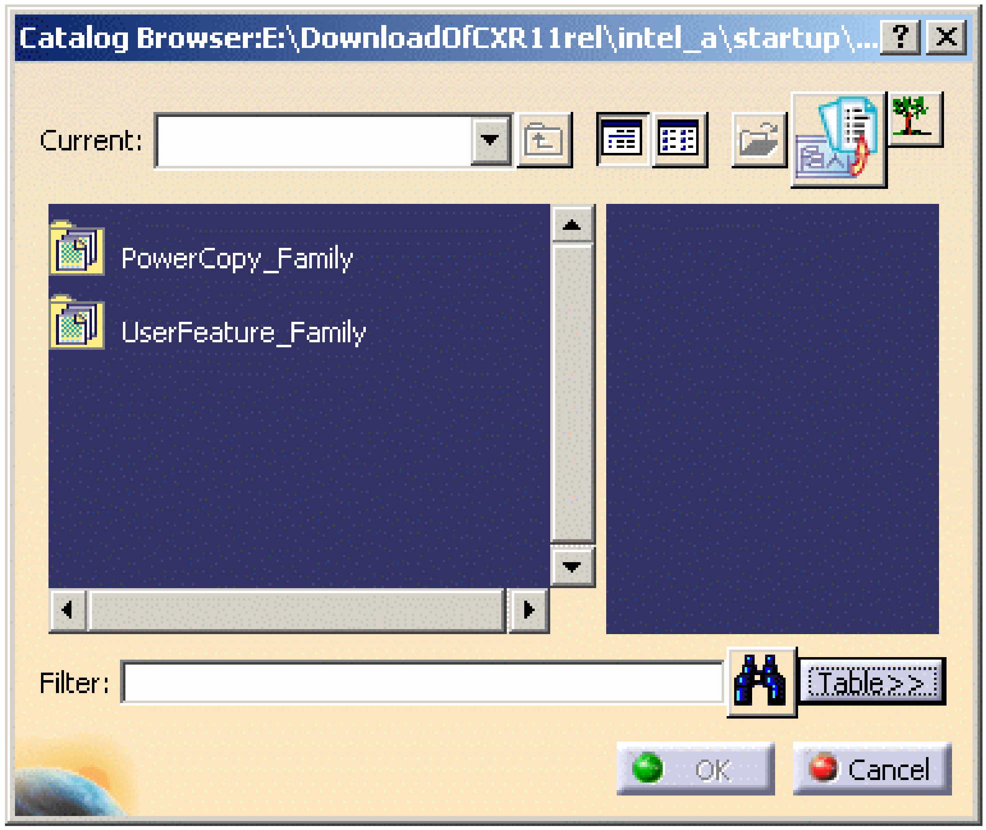
Browsing the SheetMetal Catalog

This task explains how to browse the SheetMetal catalog and instantiate its components. The catalog enables to store the available profiles, therefore providing a method to position the profile in the part.
This command is available with the CutOut and the Corner Relief functionalities.

Let's take an example with the CutOut functionality.
For more information on catalogs, please refer to the Using Catalogs chapter in the *CATIA Infrastructure User Guide*.

Open the CutOut1.CATPart document.

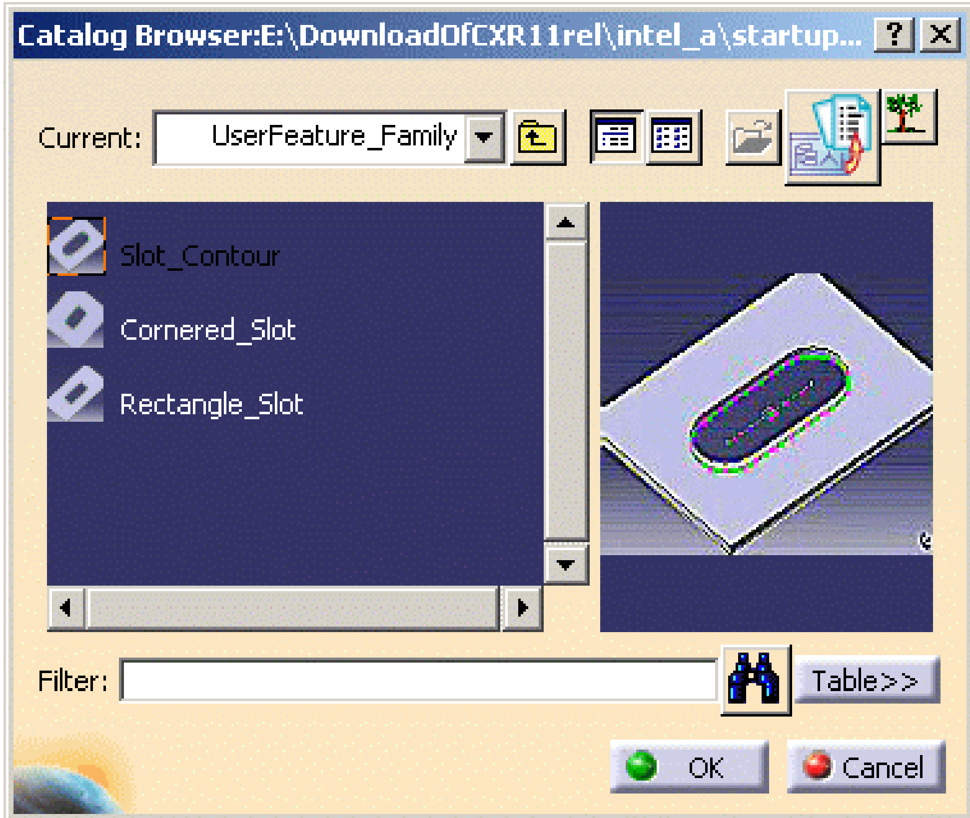
1. Once in the CutOut Definition dialog box, click the **Catalog** icon .



2. Double-click a family from the list to display its components.

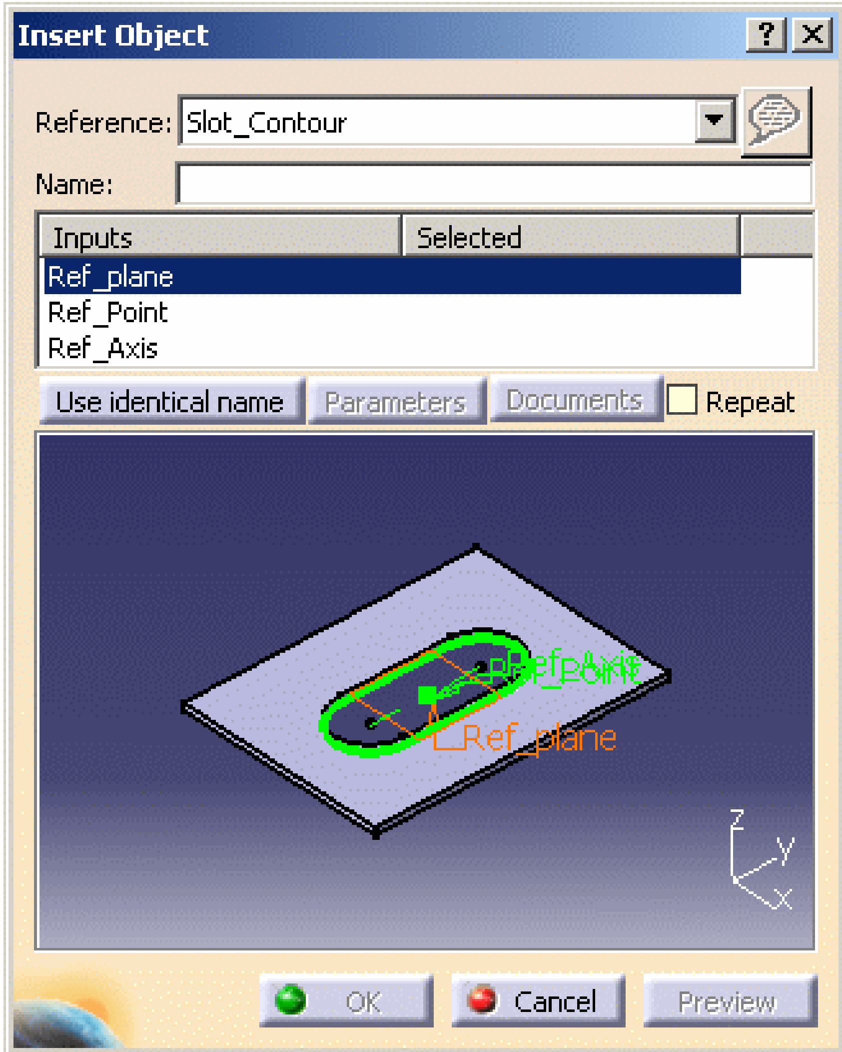
   Here we chose the UserFeature_Family.

3. Click a component to see its preview. Here we chose Slot_Contour.



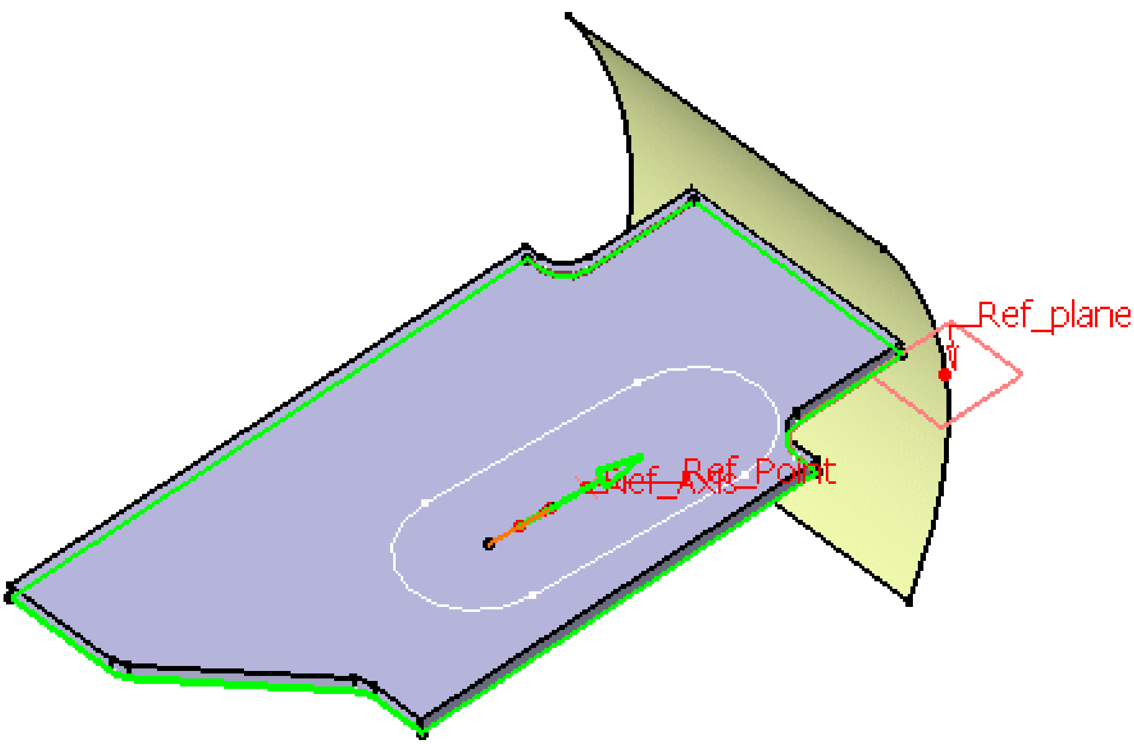
4. Click the **Table>>** button to show/hide the catalog descriptions and keywords. By default, the table is hidden.

5. Instantiate the component by double-clicking it.



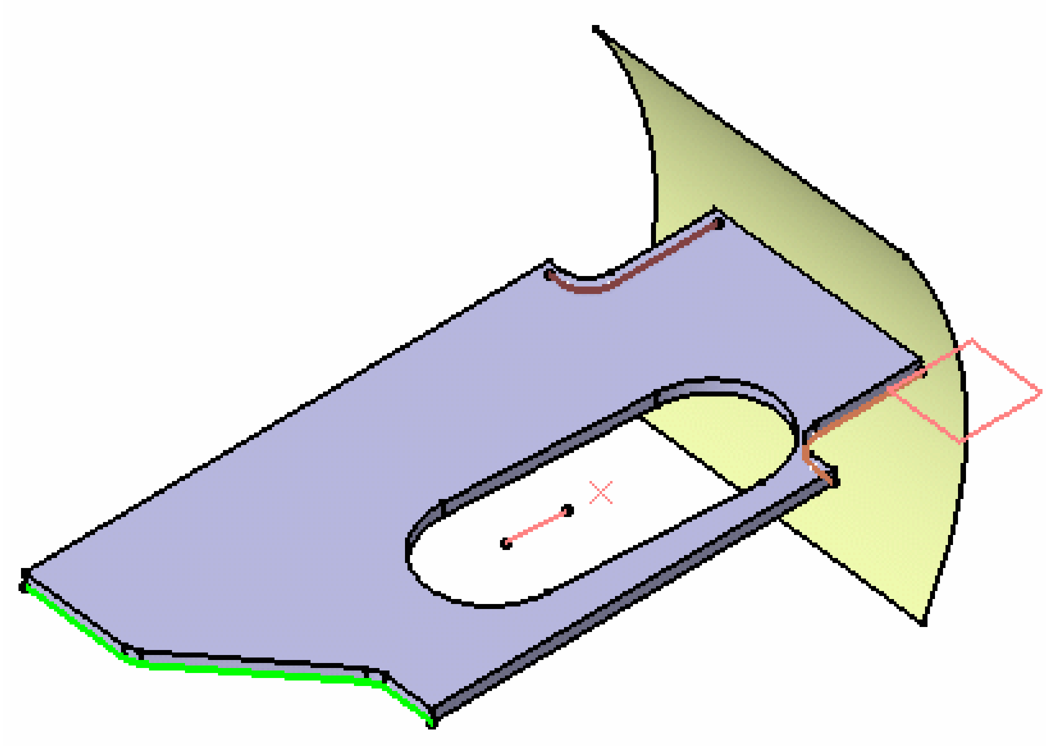
The Insert Object dialog b

6. Select the required inputs: plane, point, and axis.
7. Click Preview to see the resulting profile in the 3D geometry.
8. Click OK in the Insert dialog box.



The selected profile appears in the Selection field.

9. Click OK.



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

You may need to reverse the direction of the cutout to create it.

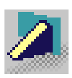
Once the profile is instantiated in the default catalog, its path is automatically set in the Standard Profiles Catalog Files field.
See Customizing General Settings.
A new panel now allows you to select alternate document access methods.
See Opening Existing Documents Using the Browse Panel in *CATIA Infrastructure User Guide*.

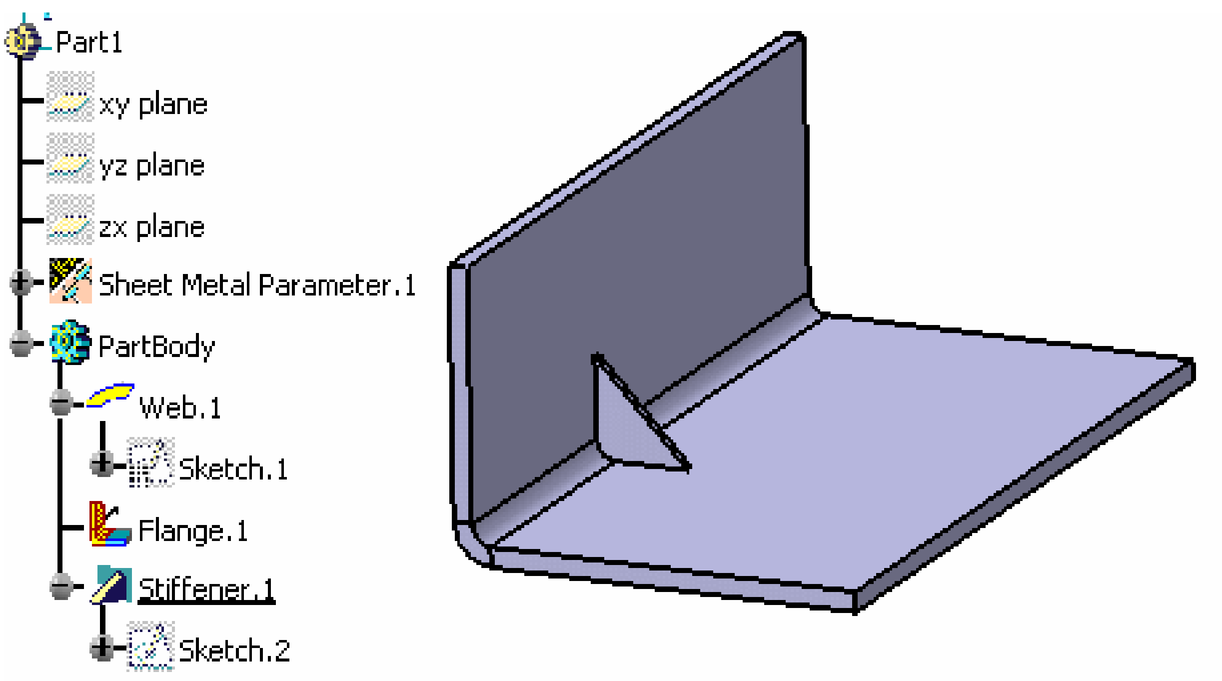
Integration With Part Design

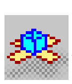
You can open the Integration1.CATPart document.

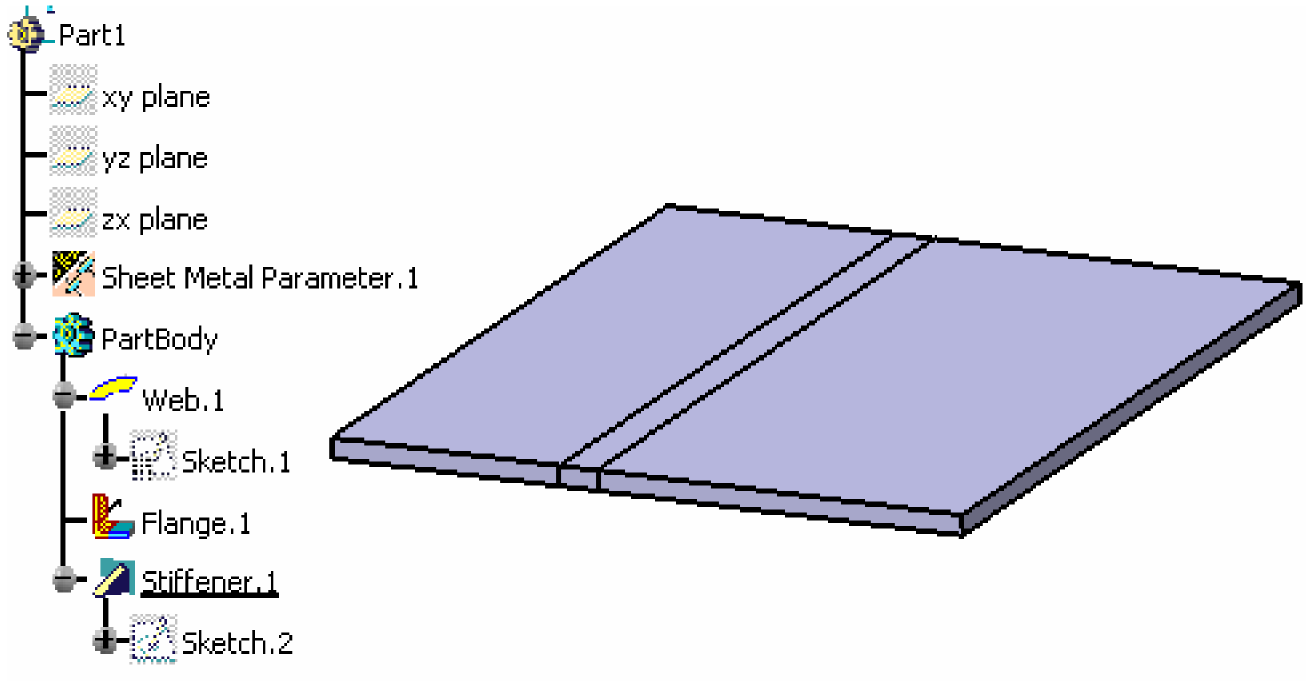
In a CATPart document, you may have Part Design features and Aerospace Sheet Metal features according to the following rules:

- Part Design features can be created before Aerospace Sheet Metal features.
- a Part Design feature can also be created after Aerospace 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 Aerospace Sheet Metal features after this last Part Design feature in folded view.

1. Create a web and a flange.
2. Switch to Part Design workbench.
3. Launch the Sketcher and draw an oblique line in the yz plane.
4. Click the icon  to create a **Stiffener**.

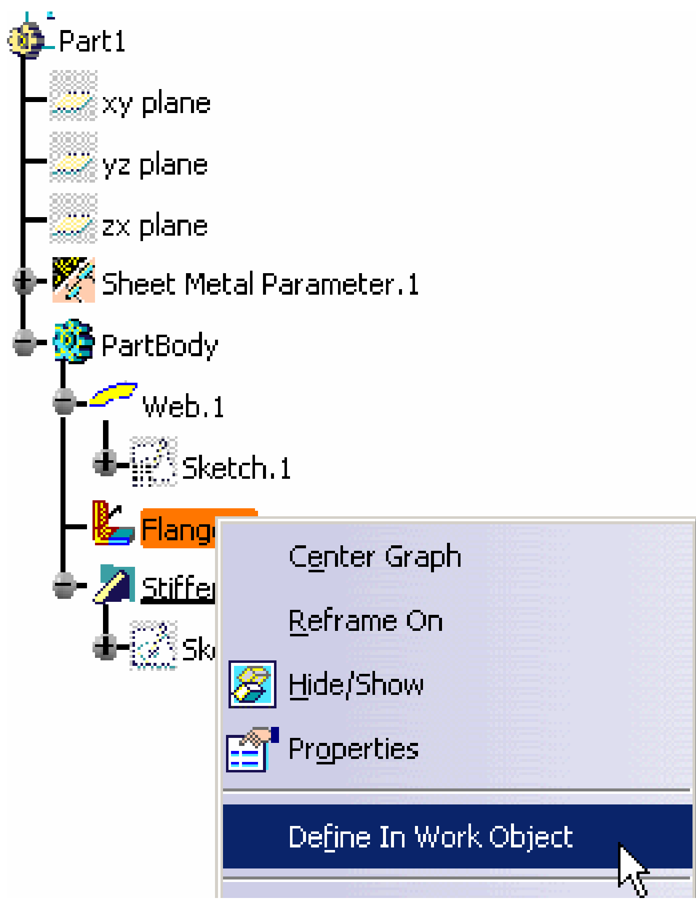


5. Switch to the Aerospace Sheet Metal workbench.
6. Click the **Unfold** icon .



The stiffener is not displayed on the unfolded view.

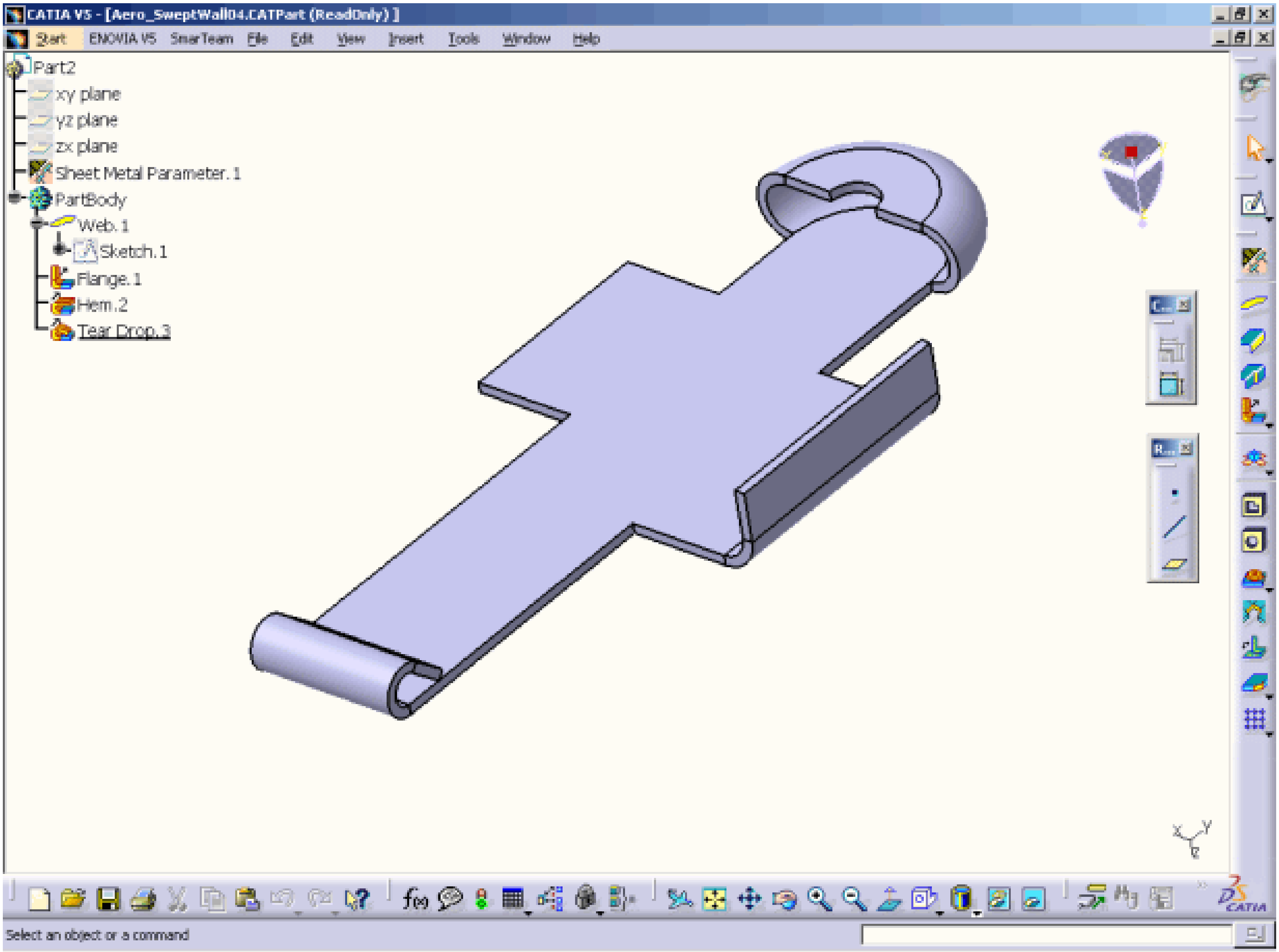
To add a new Aerospace Sheet Metal feature, select the Flange for example and right-click the **Define In Work Object** item.



The new Aerospace Sheet Metal feature will be added after the Flange but before the Stiffener.

Workbench Description

The Aerospace Sheet Metal Design application window looks like this:
Click the hotspots to display the related documentation.



Menu Bar
Aerospace SheetMetal Toolbar
Stampings Toolbar
Constraints Toolbar
Reference Elements Toolbar
Specification Tree

Menu Bar

The various menus and menu commands that are specific to Aerospace Sheetmetal 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.

Edit

The Edit menu lets you manipulate selected objects. Refer to the *Infrastructure User's Guide* and *Part Design User's Guide*.

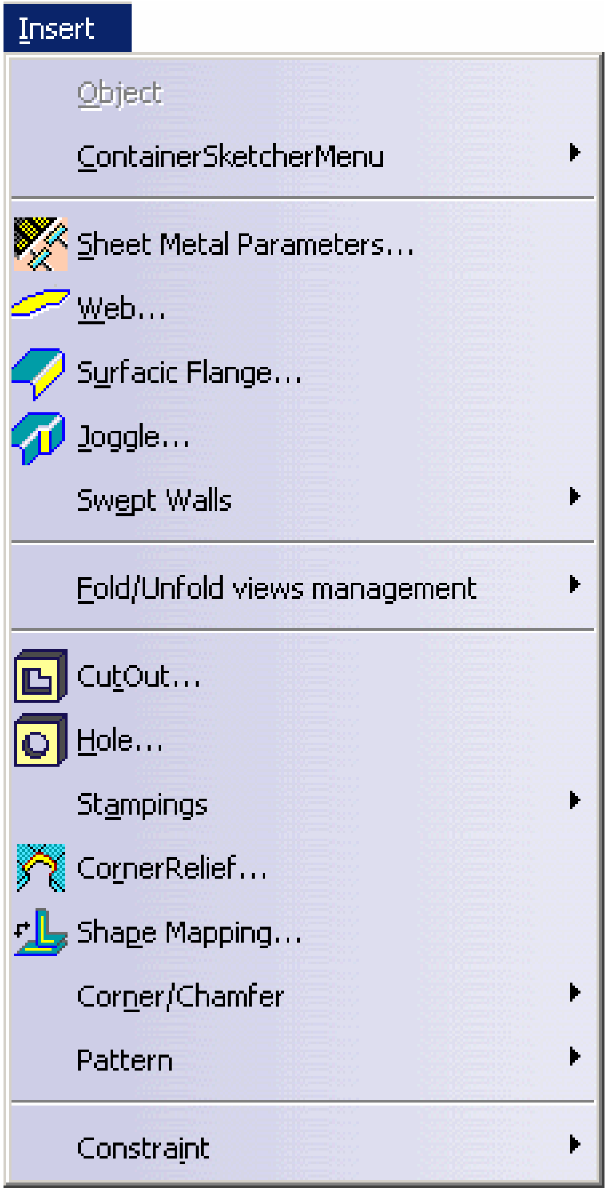
View

The View menu lets you view document contents Refer to the *Infrastructure User's Guide*.

The **Search** capability is available.

Insert

For...	See...
ContainerSketcherMenu	Refer to the *Sketcher User's Guide*.
Sheet Metal Parameters...	Managing the Default Parameters
Web...	Creating a Web
Surfacic Flange...	Creating a Surfacic Flange
Joggle...	Creating a Joggle
Swept Walls	Insert -> Swept Walls
Fold/Unfold views management	Insert -> Unfold



CutOut...	Creating a Cutout
Hole...	Creating a Hole
Stampings	Insert -> Stampings
Corner Relief...	Creating a Local Corner Relief
Shape Mapping	Mapping Elements
Corner/Chamfer	Insert -> Corner/Chamfer
Pattern	Insert -> Pattern
Constraint	Setting Constraints in the *Part Design User's Guide*

Insert -> Swept Walls

For...	See...
Flange	Creating a Flange
Hem	Creating a Hem
Tear Drop	Creating a Tear Drop
User Flange	Creating a Swept Flange

Insert -> Unfold

For...	See...
Unfold...	Folded/Unfolded View Access
Multi Viewer...	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
Flanged CutOut...	Creating a Flanged Cutout
Curve Stamp...	Creating a Curve Stamp
Stiffening Rib...	Creating a Stiffening Rib
User Stamp...	Creating User-Defined Stamping Features

Insert -> Corner/Chamfer

For...	See...
Corner...	Creating Corners
Chamfer...	Creating Chamfers

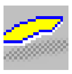
Insert -> Pattern

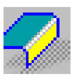
For...	See...
Rectangular Pattern...	Creating Rectangular Patterns
Circular Pattern...	Creating Circular Patterns
User Pattern...	Creating User-Defined Patterns

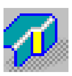
Aerospace SheetMetal Toolbar

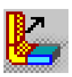
The Aerospace Sheet Metal Toolbar contains the following tools:

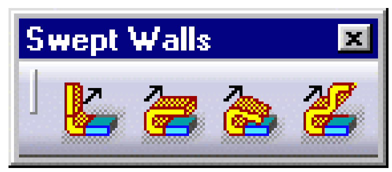
 See Managing the Default Parameters

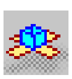
 See Creating a Web

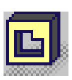
 See Creating a Surfacic Flange

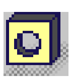
 See Creating a Joggle

 See Creating Swept Walls



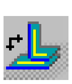
 See Unfolding the Part

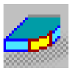
 See Creating a Cutout

 See Creating a Hole

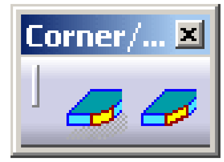
 See Creating Stamping Features

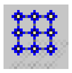
 See Creating a Corner Relief

 See Mapping Elements

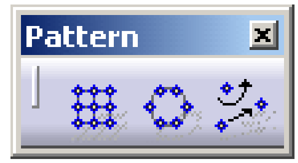


See Creating Corners
See Creating Chamfers



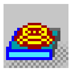


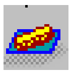
See Creating Patterns

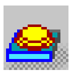


Stampings Toolbar

The Stampings toolbar contains the following tools:

 See Creating a Flanged Hole

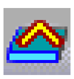
 See Creating a Bead

 See Creating a Circular Stamp

 See Creating a Surface Stamp

 See Creating a Flanged Cutout

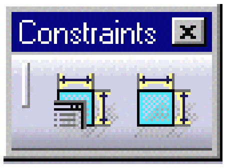
 See Creating a Stiffening Rib

 See Creating a Curve Stamp

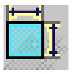
 See Creating User-Defined Stamping Features

Constraints Toolbar

The Constraints Toolbar contains the following tools:

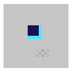


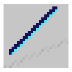

See Setting Constraints from the *Part Design User's Guide*

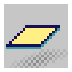


Reference Elements Toolbar

The Reference Elements Toolbar contains the following tools:

 See Creating Points

 See Creating Lines

 See Creating Planes

Specification Tree

Within the Aerospace Sheetmetal Design workbench, you can generate a number of features that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

- Sketch
- Sheet Metal Parameters
- Web
- Surfacic Flange
- Joggle
- Flange
- Hem
- Tear Drop
- Swept Flange
- Cutout
- Hole
- Flanged Hole
- Bead
- Circular Stamp
- Surface Stamp
- Corner Relief
- Mapping
- Corner
- Chamfer
- Rectangular Pattern
- Circular Pattern
- User-Defined Pattern
- Point
- Line
- Plane

Flanged Cutout

Curve Stamp

Stiffening Rib

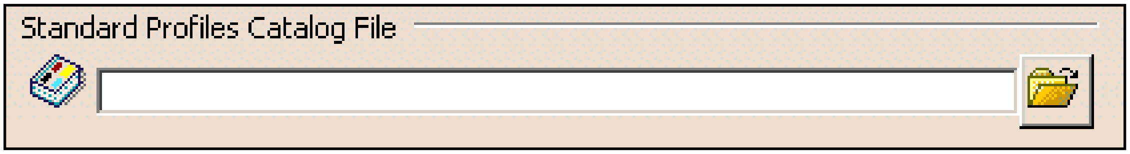
Customizing

This section describes how to customize standards files and settings specific to the Aerospace Sheet Metal Design workbench.

Aerospace Sheet Metal Design

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 Standards Files to Define Design Tables

This section describes how to customize company standards files to define design tables.

To know more about the different ways to access you files, refer to the Opening Existing Documents Using the Browse Window section.

Using Sheet Metal Standards Files

This task explains how to access company standards files in order to access and define design tables.

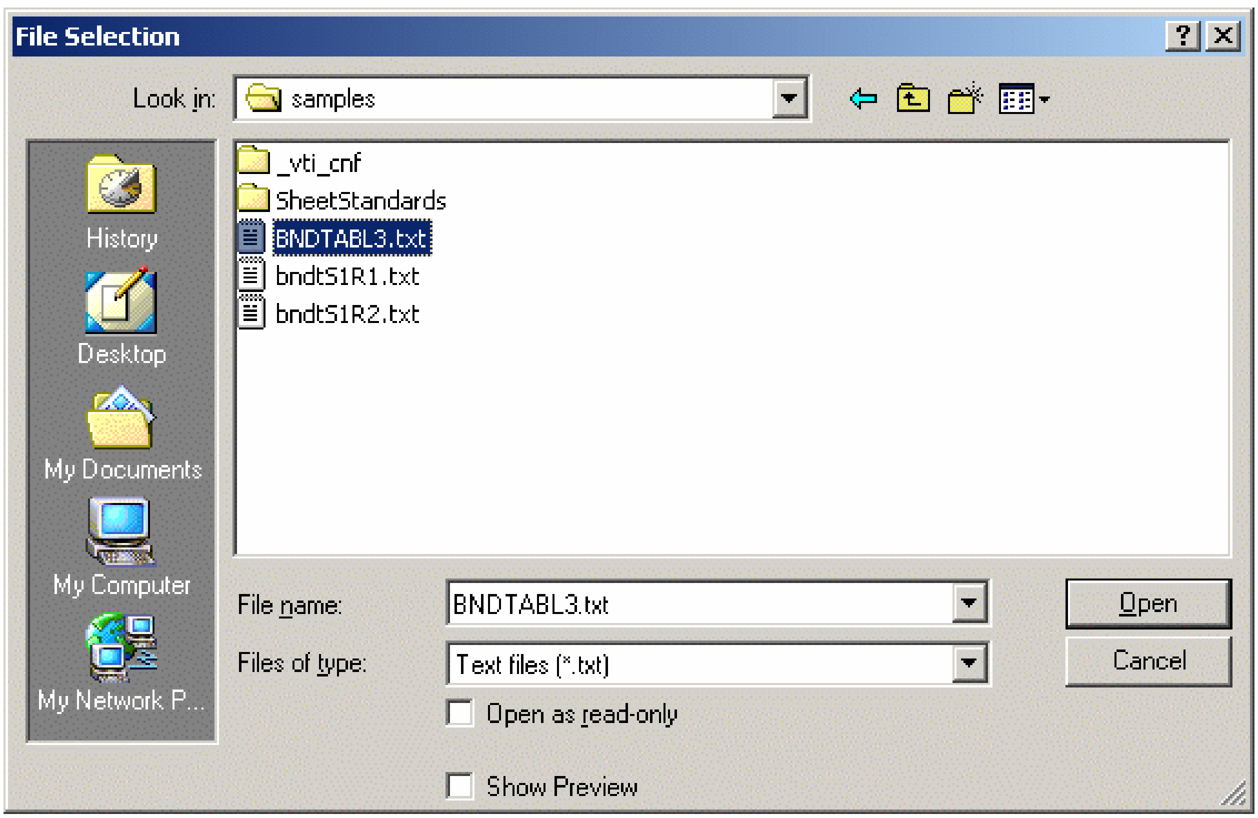
Open a new document.

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

   The Sheet Metal Parameters dialog box opens.

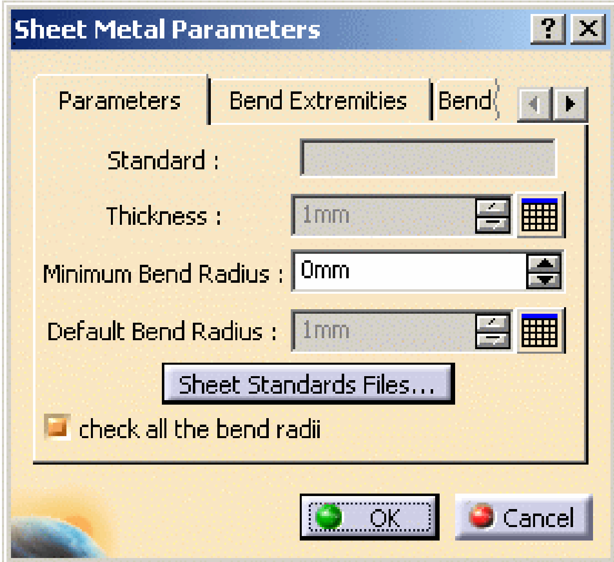
2. Select the **Sheet Standards Files...** button. The File Selection window is displayed.

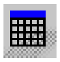
3. Indicate the path to the Sheet Metal table.



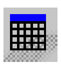
These files are available under .txt format or .xls format (only for Windows)

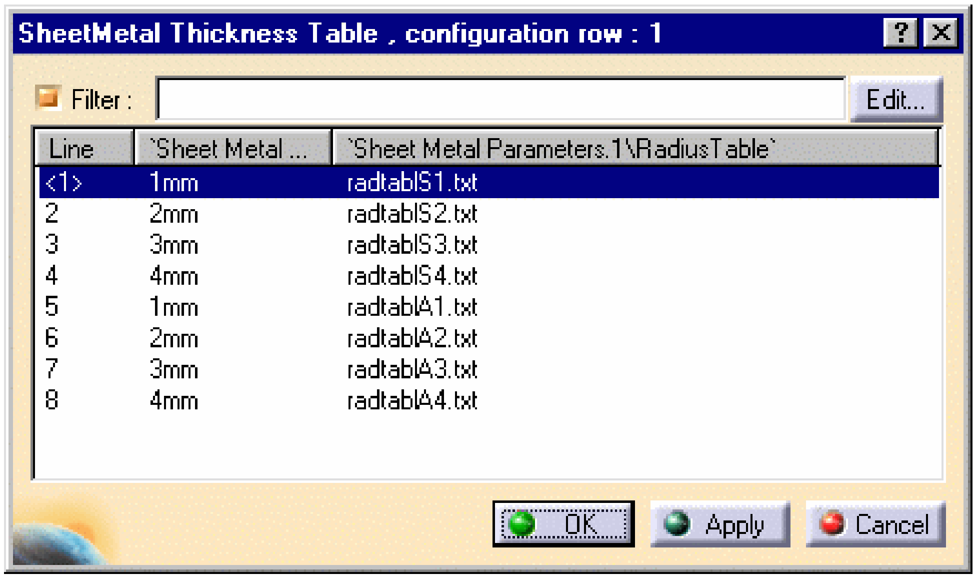
4. Click **Open**.



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).

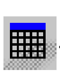


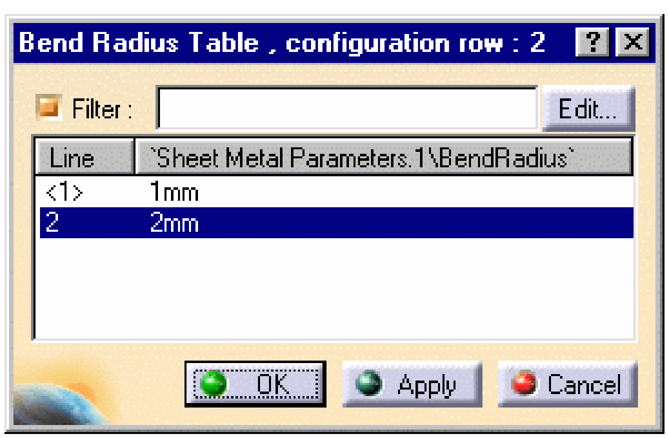
Using the **Tools -> Options -> General -> Document** tab, **Other Folders** option, you can specify where the files are located. Refer to Document.

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 .

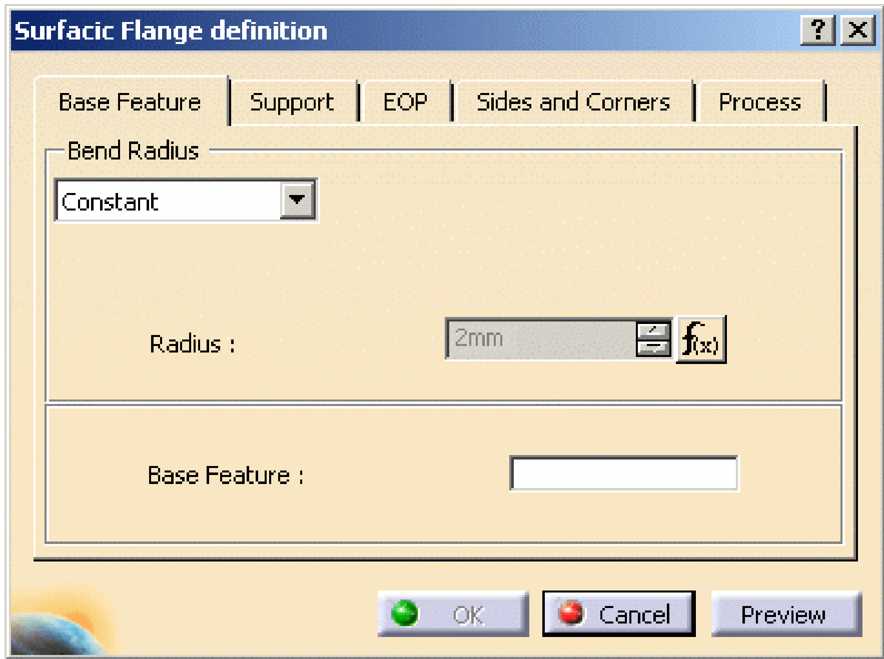


8. Select line 2 and click OK.

   The parameter values are

updated in the Sheet Metal Parameters dialog box.

9. Create a flange.

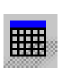


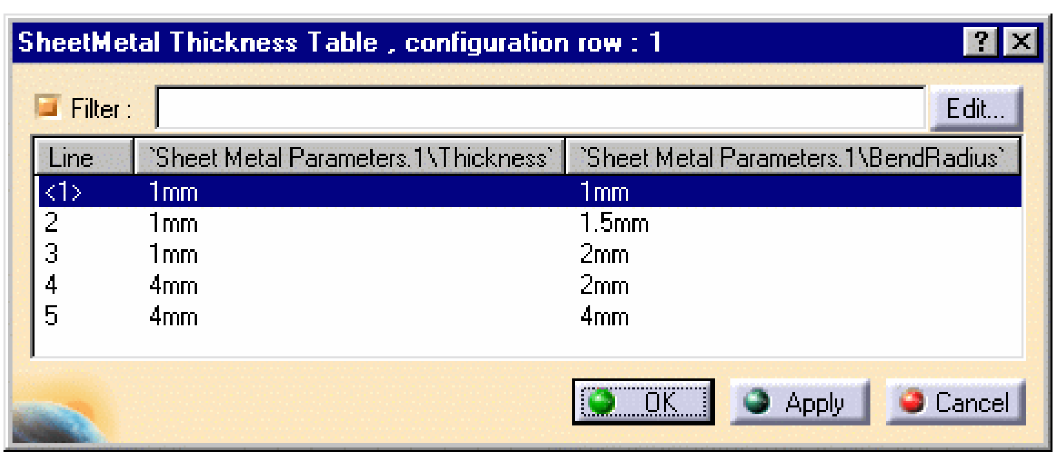
The Flange definition dialog box reflects the modification for the Radius.

The default mode, that is to say the formula: **Bend Radius = Part Radius** is deactivated.

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.

Customizing Standards Files to Define Methods for Compensations

This task explains how to access company standards files (available in .xls format) in order to define methods for joggle and sides compensations. The methods described in this task apply to joggles relying on a flange with a base feature (either a web or a flange) without joggles.

Open a new document.

1. Click the **Sheet Metal Parameters** icon. The Sheet Metal Parameters dialog box is displayed.
2. Select the **Sheet Standards Files...** button. The File Selection window is displayed.
3. Indicate the path to the Sheet Metal methods (Std_Method1.xls or Std_Method1_2.xls).

Two methods are available to enable the joggle compensations. Both use Design Tables.

Method 1

This method is the method which was used in V4.

The type of modifications performed depends on the position of the joggle with regards to:

- the end of the part:
  - near the end of part (case 1)
  - not near the end of part (case 2)
- the position of other joggles:
  - twin joggles (case 3)
  - double joggle near the end of part (case 4)
  - double joggle not near the end of part (case 5)

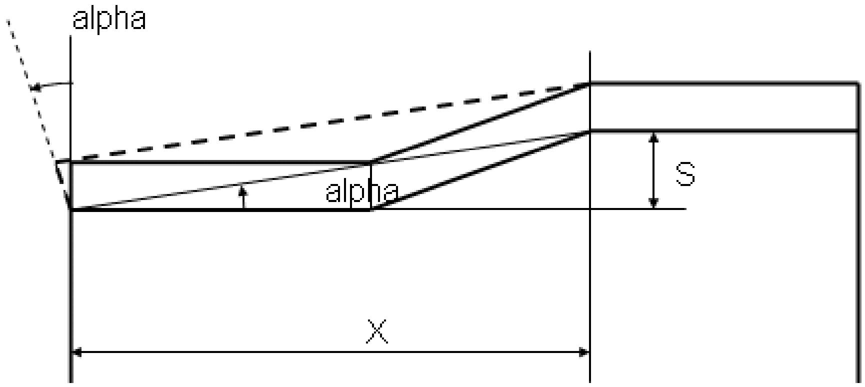
How the different cases and the deformation are managed

Six values are used in the following description: C1, C2, C3, C4, C5, Ra. These values are defined in a design table. The path to this design table is defined in the Std_Method1.xls file.

Here is an example of a design table:

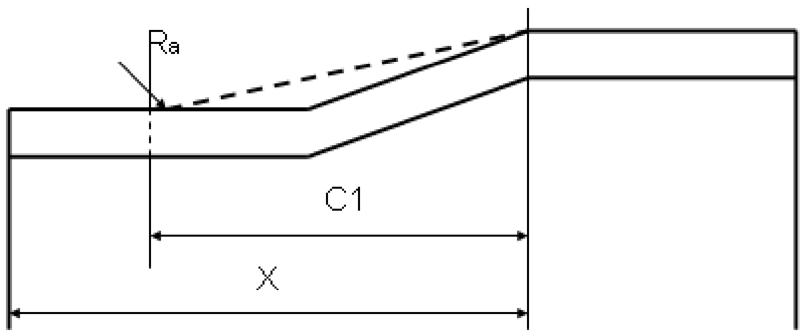
C1	C2	C3	C4	C5	Ra
75	5	150	10	75	3

Case 1



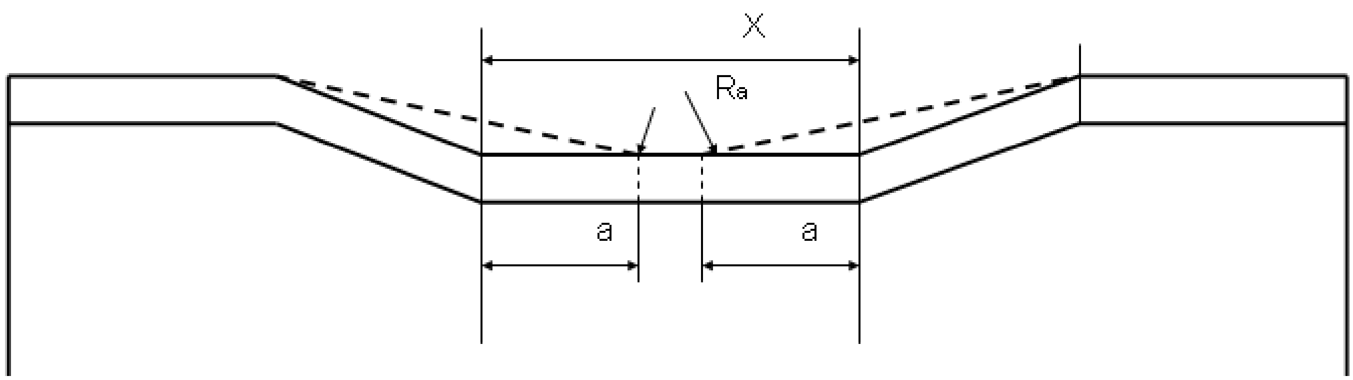
- The parameters necessary for performing the modifications are as follows:
  - The parameter of the feature is: S (joggle depth)
  - Some values are computed to define the type of deformation: alpha, X.
- The parameter definition and links are as follows:
  - This case is applied if X < C1.
  - C1 is defined with a constant value.
- This deformation is performed as follows:
  - The alpha angle is computed from S and X (tangent(alpha) = S / X).
  - The side and the EOP are then rotated with the alpha angle. The center of rotation is the intersection between the BTL on the flange and the side.

Case 2



- The parameters necessary for performing the modifications are as follows:
  - The values of the modification are: C1, Ra.
- The parameter definition and links are as follows:
  - This case is applied if X > C1.
  - C1 is defined with a constant value.
  - Ra is defined with a constant value.

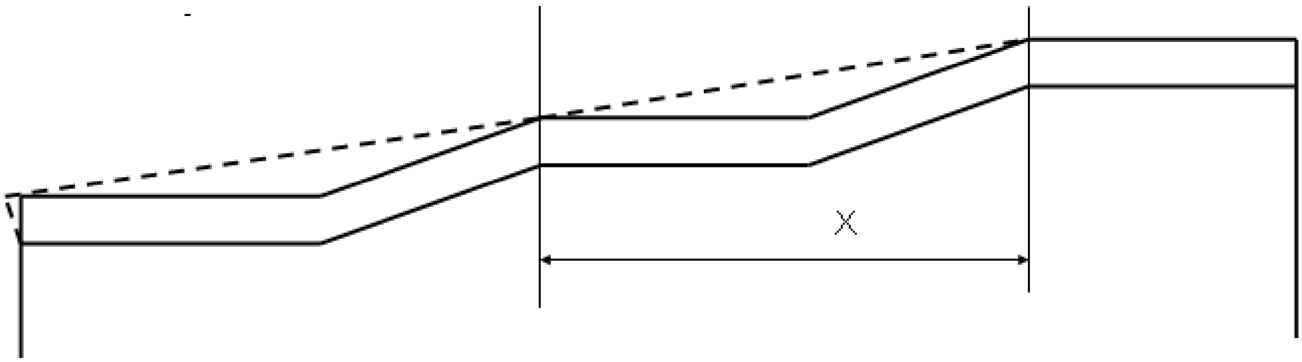
Case 3



- The parameters necessary for performing the modifications are as follows:
  - The values of the modification are: C2, C3, C4, Ra.
  - A value is computed to define the type of deformation (table) to apply: X.
- The parameter definition and links are as follows:
  - This case is applied if C3 > X > C2.
  - Then: a = (X - C4) / 2.
  - C2, C3 and C4 are defined with a constant value.
  - Ra is defined with a constant value.

Case 4

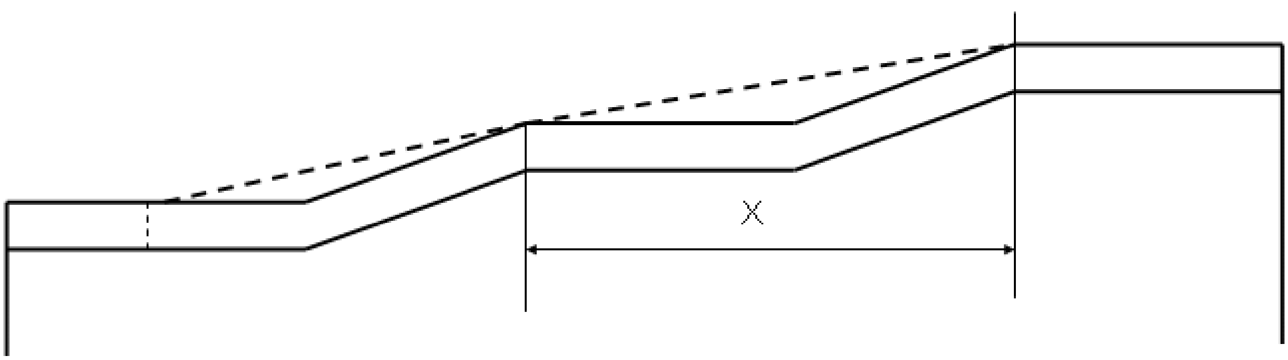
This case applies only to the joggle positioned on the right; the joggle positioned on the left follows case 1.



- The parameters necessary for performing the modifications are as follows:
  - A value is computed to define the type of deformation (table) to apply: X.
- The parameter definition and links are as follows:
  - This case is applied if C4 > X.
  - C4 is defined with a constant value.

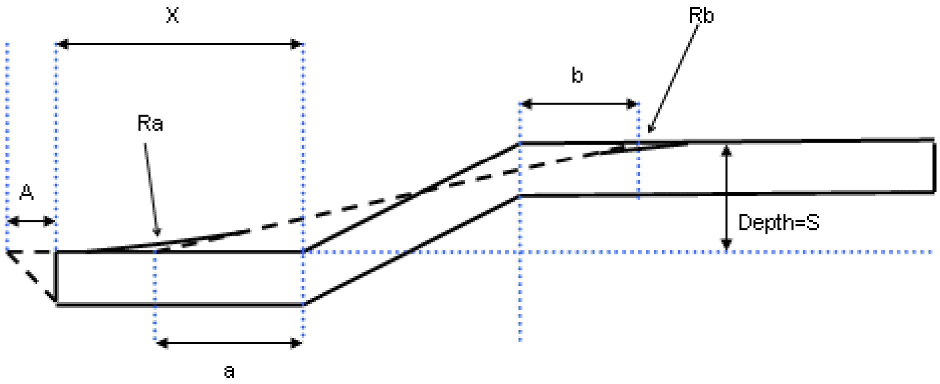
Case 5

This case applies only to the joggle positioned on the right; the joggle positioned on the left follows case 2.



- The parameters necessary for performing the modifications are as follows:
  - A value is computed to define the type of deformation (table) to apply: X.
- The parameter definition and links are as follows:
  - This case is applied if C5 > X.
  - C5 is defined with a constant value.

Method 2



- The parameters necessary for performing the modifications are as follows:
  - Input parameters for joggle compensation:
    - Material Thickness: t
    - Joggle parameters: S (joggle depth)
    - Distance between the joggle and the side of the flange: X
  - Output parameters:
    - Offset from start of joggle: a
    - Offset from end of joggle: b
    - Flange side compensation: A
    - Radius on start and end of joggle compensation: Ra, Rb

- The parameter definition and links are as follows:
  - A, a, b, Ra, Rb are defined from two design tables. The paths to these design tables are defined in the Std_Method1_2.xls file. Std_Method1_2.xls also contains the path to each defined thickness.

    For each value of a material (therefore for each thickness) in Std_Method1_2.xls, two tables defining compensation values may be defined:

    A first table defining A, depending on S:

SMax	a	b	Ra	Rb
0.8	a (0.8)	b (0.8)	Ra (0.8)	Rb (0.8)
1.1	a (1.1)	b (1.1)	Ra (1.1)	Rb (1.1)
1.5	a (1.5)	b (1.5)	Ra (1.5)	Rb (1.5)

The first line should read as follows: if (0<SMax≤0.8) then a=a (0.8); b=b(0.8); Ra=Ra (0.8) and Rb=Rb (0.8)
The third line should be read as follows: if (1.1<SMax≤1.5) then a=a (1.5); b=b(1.5); Ra=Ra (1.5) and Rb=Rb (1.5)

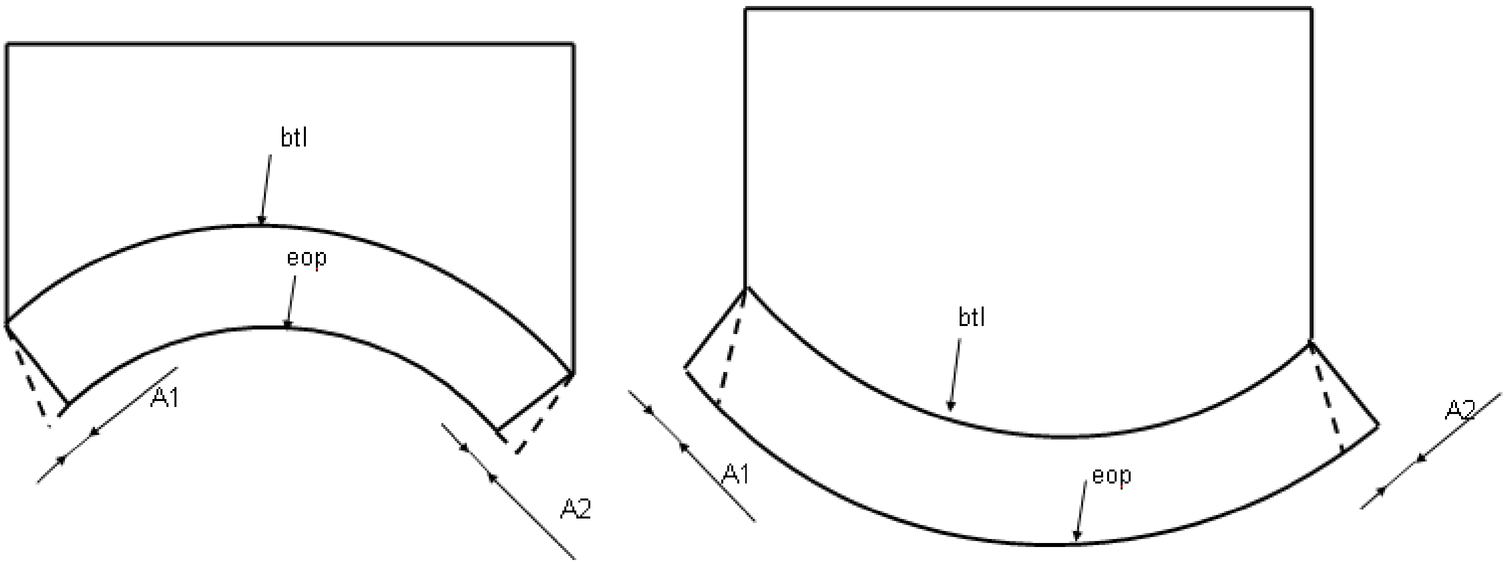
A second table defining A, depending on S and X:

SMax	XMax	A
0.8	X (0.8)	A (0.8)
0.8	X' (0.8)	A (0.8)
0.8	X" (0.8)	A (0.8)
1.1	X (1.1)	A (1.1)

1.1	X' (1.1)	A (1.1)
1.1	X" (1.1)	A (1.1)

The first line should be read as follows: if (0<SMax≤0.8) and (0<XMax≤X(0.8)) then A=A (0.8)
The fourth line should be read as follows: if (0.8<SMax≤1.1) and (X" (0.8)<XMax≤X(1.1)) then A=A (1.1)

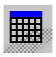
Definition of Side Compensation



- The parameters necessary for performing the modifications are as follows:
  - The values of the modification are: A1, A2
  - The type of compensation can be:
    - None: no compensation is applied
    - Automatic (for symmetric flanges): A1 = A2, so that the length of the flatten eop = the length of the folded eop
    - Manual: Angle: the deformation is computed according to an angle
    - Manual: Length: the deformation is computed according to a length parallel to the btl.

4. Choose the appropriate type of compensation.

5. Click **Ok**.

In the Sheet Metal Parameters dialog box, if you chose Method 2, the **Design Table** icon  appears opposite the Thickness field.

You can click this icon to edit the design table.

Glossary

B C D E F J K P S W

B

bead A local deformation in the web or a flange.

bend A feature joining two walls

C

corner relief A feature defined on two flanges, which forms a corner.
It relimits the two flanges and redefines the outer profile of the web between the two flanges.

cutout A feature corresponding to an opening through a feature.
The shape of the opening corresponds to the extrusion of a profile.

D

depth Dimension specifying the geometry of a bead or a joggle.

E

edge of part Element (usually a curve), which defines the length/height of a surfacic flange.

F

feature Characteristic form. Features are used to define a part.

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.

J

joggle Feature which causes the flange to be locally deformed.
Usually because the skin which is connected to the web is locally enforced by a strip or stringer (L or T profile).

K

K factor Determines the computation of the unfolded length of the surfacic flanges.

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

S

stamping A feature created by embossing sheet metal.
The different stampings available are:
bead, circular, flanged hole, flanged cutout, curve, surface, and stiffening rib.

surfacic flange A feature along the outer section of the web or an existing flange. It is used to connect to another product or to stiffen the part.

W

web Main constituent of a part. Many other features (flanges, holes, etc.) are defined onto this feature.

Index

A B C D E F G H I J L M P R S T U W

A

B

C

D

E

F

M

P

R

S

T

U

W