



**S.M.O.**  
**Sheet Metal Optimizer**

**User's guide**

## Installation:

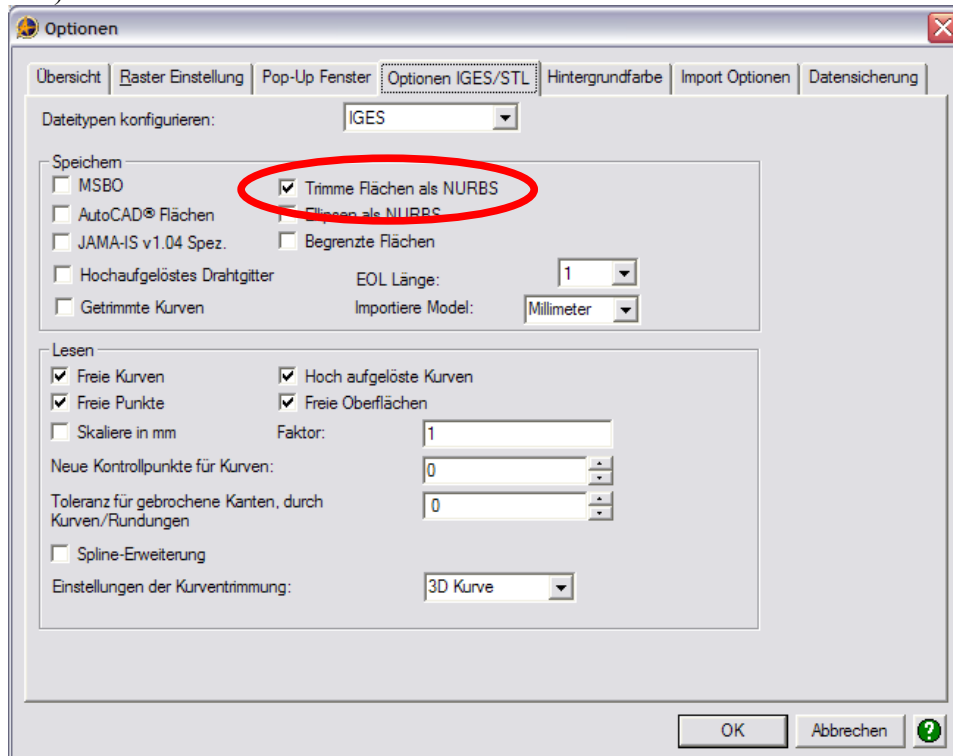
- 1) You must have received a USB dongle and a licence file named **catalcad.cod**.
  - 3) From CD, start the **setup.exe** in folder **SMO-Programm**.  
To simplify, please install the software on the **C:** drive.  
Accept all the default directories. In particular, for the use with Alibre, the folder **C:\smostore** must exist, so please don't change.
  - 4) The dongle may need a driver from the manufacturer. You can download and install it from the internet site of SafeNet Inc.  
<http://www.safenet-inc.com/support/tech/latestdriver.asp>
  - 5) Plug the USB dongle
  - 6) Copy the **catalcad.cod** that you got in the folder **C:\config-usr** which is automatically created at the install.
  - 7) Go to the Windows Start menu **USR** and run **S.M.O....** It should open normally. You can copy the short cut on the desktop. Don't create a short cut, it doesn't work.
-

## Now lets exam the specific issues for Alibre-->SMO

1) This version of SMO is normaly to be used in connection with Alibre, that's why the import possibility is limited to IGES items 144 (trimmed surfaces).

However, it is possible to use this SMO in stand alone way, but the IGES files must have been made with items 144 (trimmed surfaces). Normaly every CAD system has the possibility to tune the IGES output to do so.

So, in Alibre, use "**Werzeuge/Optionen/Optionen IGES**", set to Trim surfaces as NURBS (or JAMA-IS)" see below:



2) There is a simple way to check if SMO works well. Save a model as IGES file in Alibre and then, import it in SMO with "**U.S.R./import IGES**", you must see the model on the SMO display.

3) Complementary install:

The folder **C:\smostore** must exist, it is normaly created at the install. (If not, create it).

Start **SMO-Kopplung-Setup.exe** for Alibre-Linkage...

... or manually, copy in the folder

**"Programme/Alibre Design/Program"** on XP

**"Program Files/Alibre Design/Program"** on Vista, you must copy the following files lamatransf.exe, lamalif2.dll, Interop.AlibreX.dll .

In the folder **"Programe/Alibre Design/Progam/AddOns"**, you must copy the sub-folder **lasmusmo** in which you must find at least **lamaSMO.dll** and **lasmusmo.adc**.

4) On XP, eventually you may have to install the most recent Microsoft.NET framework.

5) Because we trust Alibre import possibilities and the Alibre-->SMO interface, we recommend to use Alibre & SMO together. The use is very simple as follows:  
Both programs must be open. In Alibre, just use "**Werkzeuge/Zusatzprogramme/SMO sheet metal optimizer**", the part automatically appears on the SMO display.

regards

catalCAD

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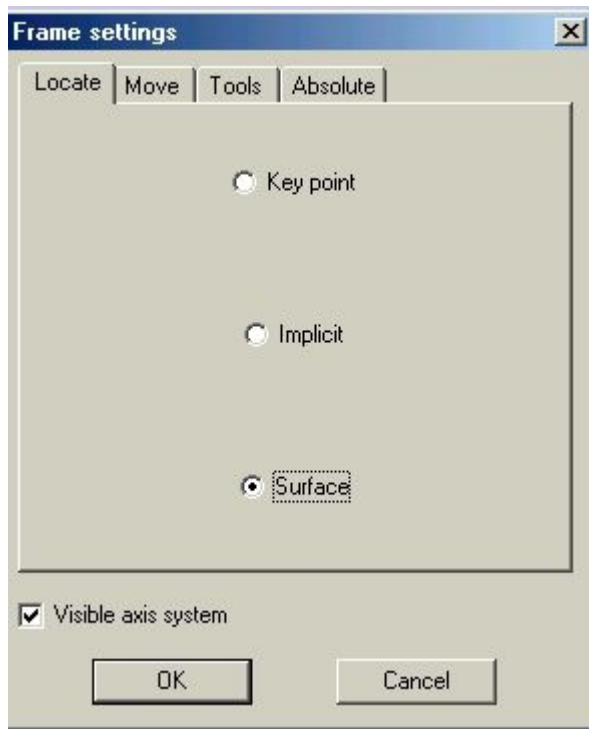
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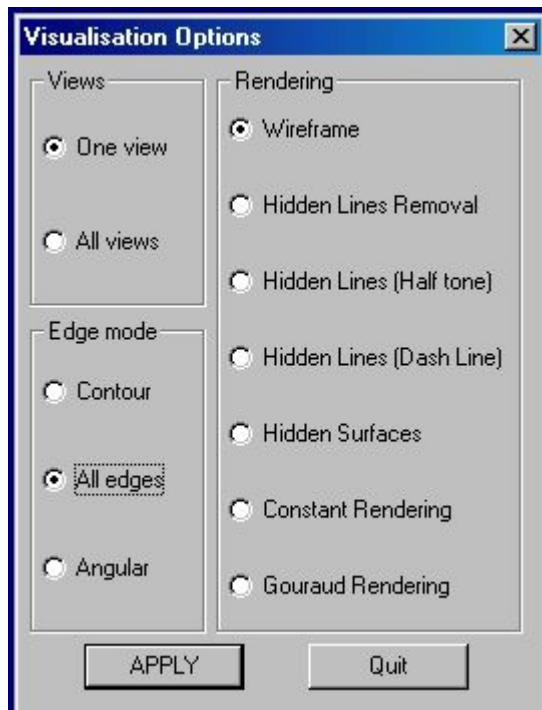


Choose “Surface” and click on a corner of the face you want to become the reference plan, then OK.

Or choose “Key point” and click 3 points to define the working plan, then OK

## 1.3 Visualization

Use the visualization icon  on the left bar, this window appears:



Then you can choose different options.

You can also click on the rendering icon on the top bar to see the part in rendering mode.



Then, in clicking several times on the VALID button, you change for "hidden lines", "dash lines" then "wire frame" visualization modes.



You can also click on the transparent icon  to see the part in dashed hidden lines mode.


## 1.4 Dynamic display functions

Press <Ctrl> and move the mouse with the left button, you get the dynamic rotation around axis origin.

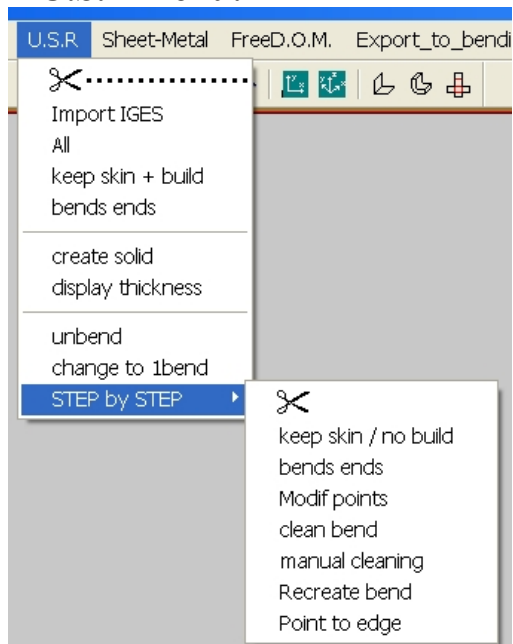
Press <Shift> and move the mouse with the left button, you get the dynamic pan.

Press <Ctrl> and <Shift> together and move the mouse with the left button from left to right, you get the dynamic zoom.

For rotating around X axis, hit one time the F5 key, then move the mouse from left to right.  
 For rotating around Y axis, hit one time the F6 key, then move the mouse from left to right.  
 For rotating around Z axis, hit one time the F7 key, then move the mouse from left to right.  
 To stop hit again the same key or <escape>

**Note:** the center of the rotation is the axis origin. To change the origin, use the "local axis system" icon  and click anywhere on the part, then OK. (See the \$1-2)

## 2-U.S.R menu:



**Note:** In the North American version, you have choice between "keep skin/mm" and "keep skin/inch". The default tolerances values are different.

Icons:



## 3-Recovering

The normal way to import and recover a 3D **solid** is based on only three commands: "Import", "keep skin" and "create solid".

### 3.1 Import

**Note:** Depending on the unit you use, you may have advantage to click on "load\_mm" or "load\_inch" in the menu "File" to set the default parameters for mm or inch.

Click in "USR/import" then select an IGES file to load and display it.

You can click on the perspective view icon




to see the model in perspective mode.

You can also click on the rendering icon to see the part in rendering mode.



Then, in clicking several times on the VALID button, you change for "hidden lines", "dash lines" then "wire frame" visualization modes.

You can also hit the F9 key to use the dynamic rotation

**Note:** the center of the rotation is the axis origin. To change the origin, use the "local axis system" icon  and click anywhere on the part, then OK.

### 3.2 Keep skin



This icon corresponds to the function "keep skin + build" of the USR menu.

This command will simplify the part in keeping the external or internal skin.

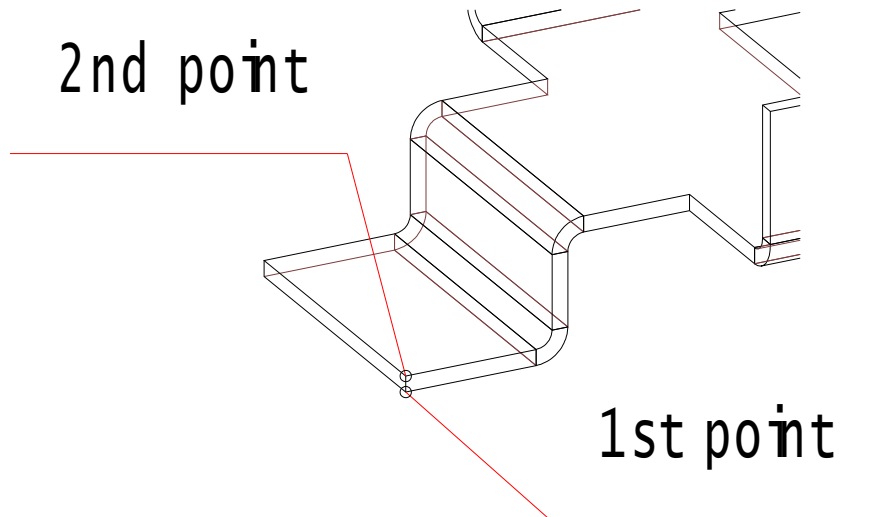
The system asks you to click a first point.



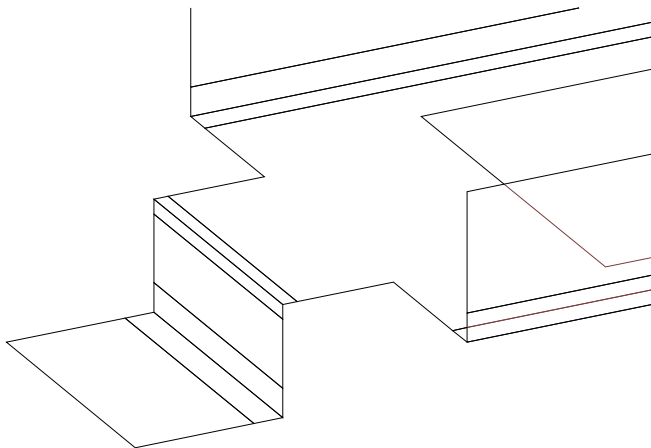
Display the part in perspective and zoom if necessary.

You must click a point located anywhere on a vertex. The point you choose indicates the reference side defining the skin you want to keep.

Then you must click a second point. It must be the corresponding point on the other side. The distance between the 2 points must be the thickness value. See the following sketch:



You get this:



The bends are changed in sharp edges or not, depending upon the "radius/thick." Parameter. See the § 2.2.1.

**Note:** In some rare cases, you cannot reach any opposite point. You can anyway give the second point in using relative coordinates .

In the example above, instead of clicking the 2<sup>nd</sup> point, you may write **&0,0,3** ( If 3 is the thickness value). It means that the second point is 3mm higher on the z axis.

**Warning:**


If the system beeps and displays (on the bottom white bar) **neither a surface nor a solid**, it means:

1- Or the model is made from several elements grouped in a set (or block). You can check it with "analyze/element"

In this case use the little bomb "explode"  then, with the left icons, "select/all"



2- Or there are some points, lines or something different than surface or solid.

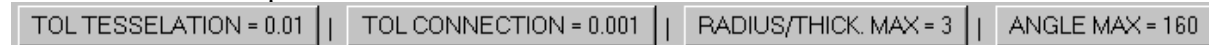
In this case you must use the "delete" icon , then with the left icons, "select/all/except/surface"




***ANYWAY, after "keep skin", If you don't get the expected result, see the chapter 5- Particular cases***

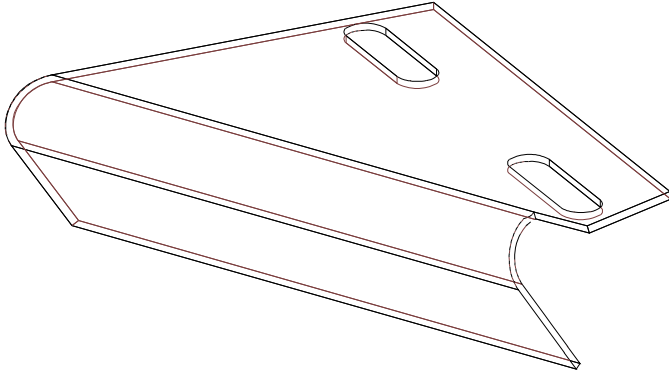
### 3.2.1 Tuning parameters of the function "keep skin"

There are 4 tunable parameters

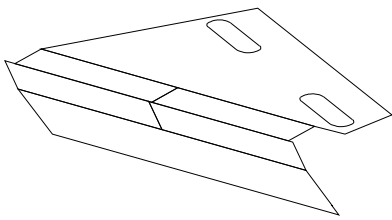


- The "tolerance of tessellation" changes the mesh density. You must decrease it if the system doesn't recognize the bends.  
It is useful when the part is small, or if the inside bend angle is large.
- The "tolerance of connection" checks if 2 surfaces are considered as connected.  
In an IGES transfer, the surfaces are often disconnected with a very small gap.
- With the "Radius/Thickness max = R/T" parameter, you can choose to keep the original bend shape or change it to a sharp edge if the bend radius is smaller than R/T\*thickness.  
See the ScreenCam *radius.scm* 

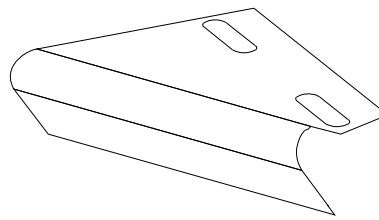
Example:



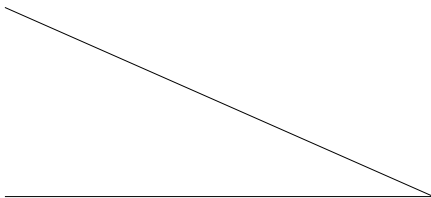
becomes



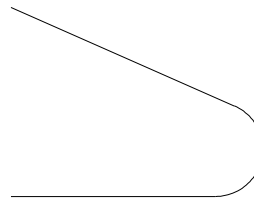
OR



- The "Angle max" parameter allows to keep the original bend shape when the angle is closed (almost hemming operation).



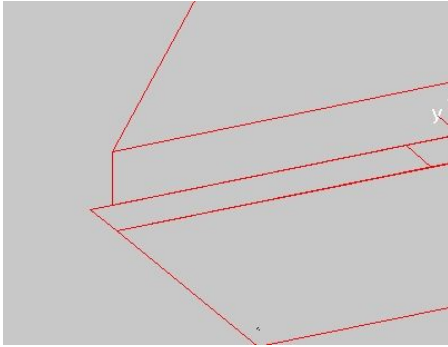
OR



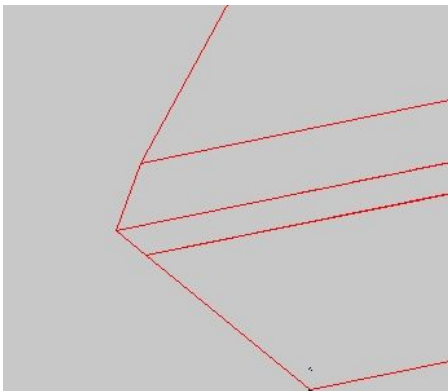
### 3.2.2 “Bends ends” and “Connect bends”

After the result of the “keep skin” process, the system proposes the button <BENDS ENDS> .

- BENDS ENDS changes the geometry of the ends as shown:



becomes



### 3.2.3 Keep skin / no build

This function is almost the same as “keep skin + build” except that the cylindrical surfaces of the bends, when they are kept, are not rebuilt.

So, it can be necessary to use “USR/STEP by STEP/manual cleaning” function to make those bends unfoldable.

This function is to be used when the “keep skin + build” doesn’t work.


### 3.3 Create Solid



Just click on the icon and the part becomes a solid 3D model. This model is now modifiable with the functions of the "sheet-metal" menu, see “Modification guide”.

Actually, it is now a S.M.O. model.



You can display the part in S.M.O. special mode in clicking on the eye  then on the part. The free edges and the bends are in different colors and the arrows show the thickness side. This is the most convenient display mode while using the "sheet metal" menu.

The eye is also available to come back to the solid display mode.

Sometimes, after "Create Solid" the part is displayed directly in S.M.O. special display mode, without thickness. It means there is a material conflict somewhere, like a missing cutout for example. Don't worry, the unfolding result will be good anyway.

## 4-Unbending

### 4-1 Unbend auto

To get the flat pattern, just click on the "unbend auto" icon .



Then choose the bend allowance computation method and click on a free edge of the reference face. The part is unfolded.

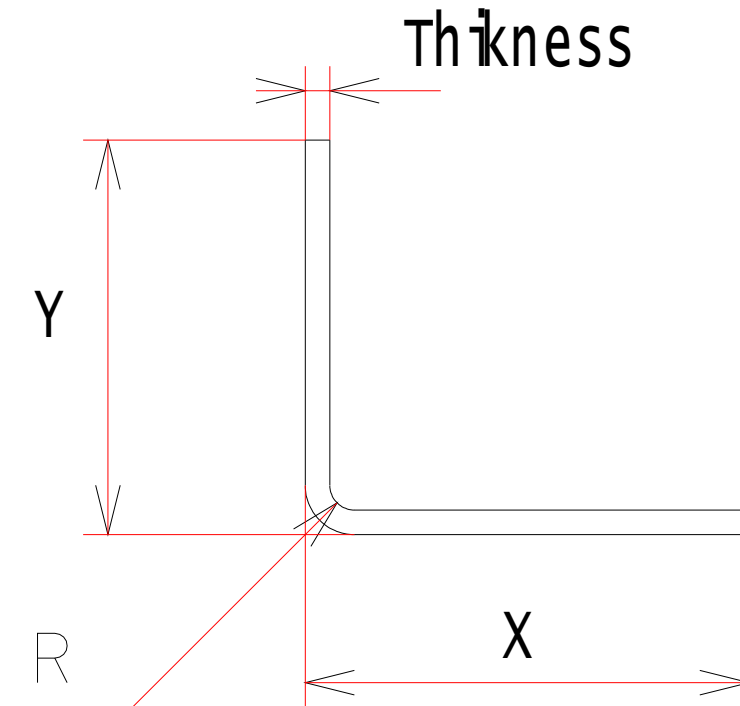
If you click on a bend line, it is ambiguous, so the system asks for a second edge.

You can also use the "Sheet metal/UNBEND" menu, in this case the 3D part is kept in memory and you can get it back with the "bend/unbend" function.

### 4-2 Bend allowance computation

To understand correctly the different bending allowance computation methods. It is mainly necessary to understand the theoretical method which is the reference.

Basically we consider just a simple part with only one 90 degrees bend as on the following sketch.



If the inside bend radius has the same value than the thickness, generally the professionals are used to saying that the neutral fibre pass in the bend approximately at 1/3 (or 0.333) of the thickness.

And when the inside bend radius is larger than 4 times the thickness, the neutral fibre pass in the middle ( or 0.5).

So we have put in C:\config-usr\topa.cfg a simple curve defined with 3 points as follow for instance:

	Order nb	Radius/Thickness	Ratio of Neutral fibre Position (K factor)
D3ZA_BEND_CURVE_RATIO	0	0.	0.1
D3ZA_BEND_CURVE_RATIO	1	1.	0.333
D3ZA_BEND_CURVE_RATIO	2	4.	0.5

This is the basic tuning off the system.

By default SMO is delivered with the following values:

D3ZA_BEND_CURVE_RATIO	0	0.	0.
D3ZA_BEND_CURVE_RATIO	1	1.	0.2732395
D3ZA_BEND_CURVE_RATIO	2	4.	0.5

We recommend the value of 0.2732395 because, as developed length, we get exactly the pragmatic value  $(X+Y-2*\text{thickness})$ , which is very usual.

If the bend radius increases, the value goes up until 0.5 when  $R = \text{or } > 4$  times thickness.

This curve is more powerful than a simple k factor (like most systems) because SMO adjust automatically the good value when the radius changes, without manual manipulation.

In addition when the angle change, the result is automatically adjusted.

**Table:**

Sometimes, according to the user's experience, or a bend test, the real needed value is a bit different:  $(X+Y-2*\text{thickness} \pm \text{complementary allowance})$  the complementary allowance can be positive or negative.

It is possible to add this "bend allowance" with "BENDBREAK/default\_allow" for the part or "1\_bend\_allow" for one bend.

Example:

$X=100, Y=100, \text{Thickness}=3, R=3$

The theoretical unbending with the default delivered parameters will give the following developed length:

$$100 + 100 - 2*3 = 194 \text{ mm}$$

If, after a bend test, you get a part with two faces of 99.8 mm each, you can adjust with  $\text{default\_allow}=0.4$  .

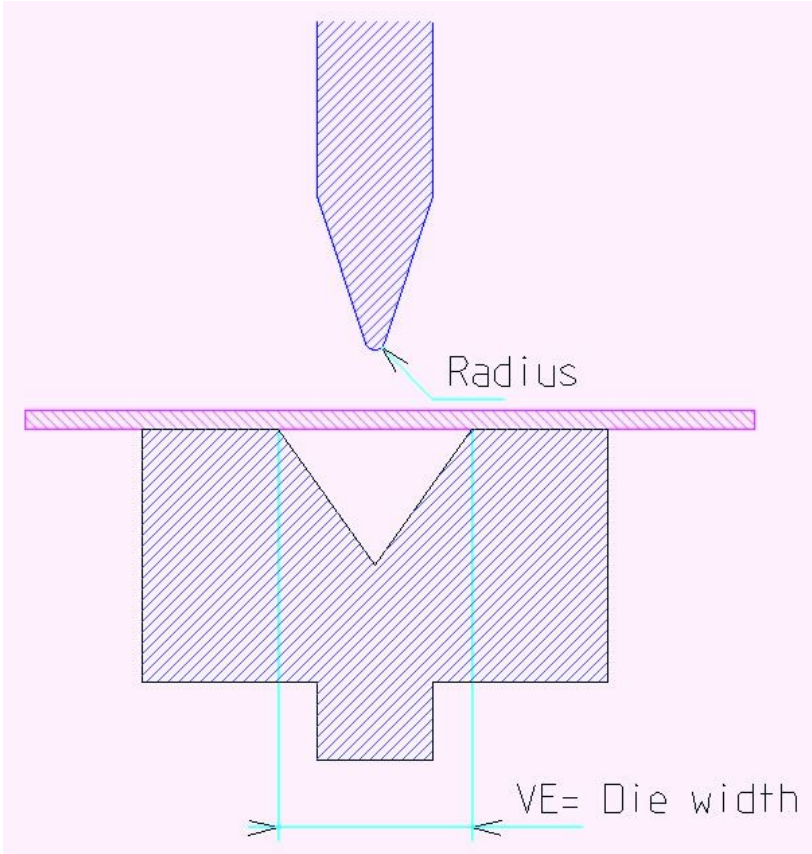
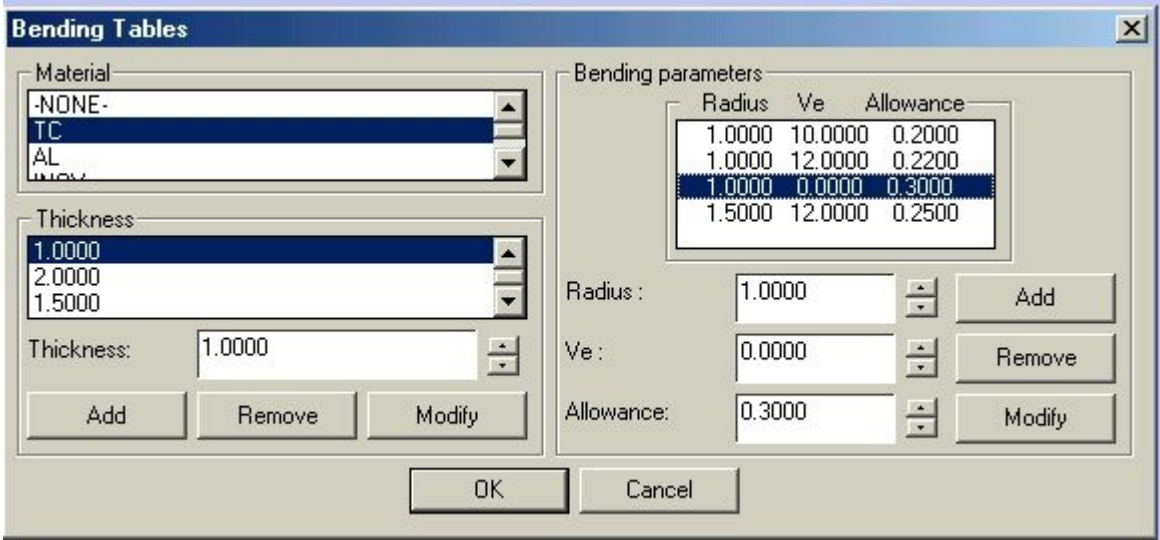
If you want to save this value, use "Sheet metal/PARAMETERS/tables". There you can record a value for any material, any thickness, any bend radius and any die width(Ve). Be careful, the material must be pre-existing. For this, use "Sheet metal/PARAMETERS/materials" and input the new material.

Don't forget that this complementary allowance must be recorded only for an angle of 90 degrees. All other values are automatically computed by the system, taking in account this reference.

Then if you want to unbend according to the table, you must assign a material to the part with "Sheet metal/SHEET METAL/material".

You must also assign a die width(Ve) by default or for each bend with "Sheet metal/BEND&BREAK/Default\_Ve" or "Sheet metal/BEND&BREAK/1\_bend\_Ve".

**Important tip:** To avoid assigning the Ve value every time, we recommend to add in the table, for each couple radius/thickness, a line with Ve=0, which is the default value.



In our tables, the (complementary) Allowance is the difference between the theoretical unfolded length and the real result according to the user's experience.

## 5-Adjusting functions (STEP by STEP menu)

Importing file doesn't guaranty a clean and precise geometry. It depends upon the CAD which it comes from.

For some of them, it can be necessary to adjust the drawing after "keep skin".

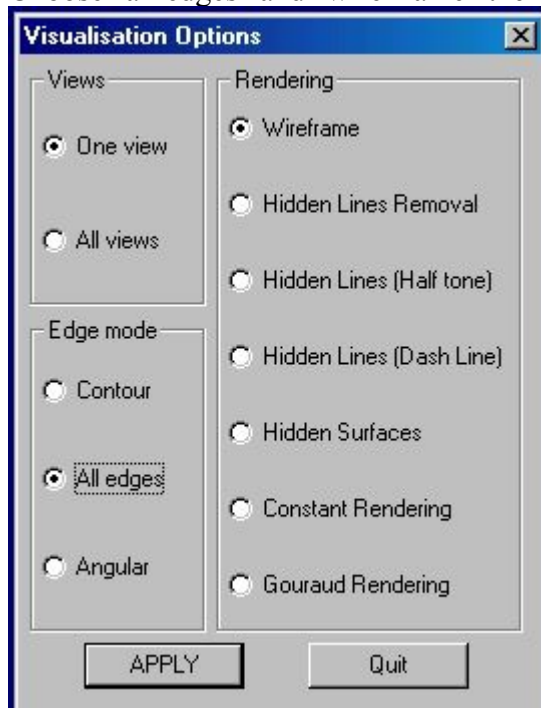
Generally, you see clearly what happens after the "create solid" function. When the expected bends don't appear in the usual color, you must click the "undo" function and check the connections

### 5.1 Clean bend

This is useful only when using **keep skin / no build**.

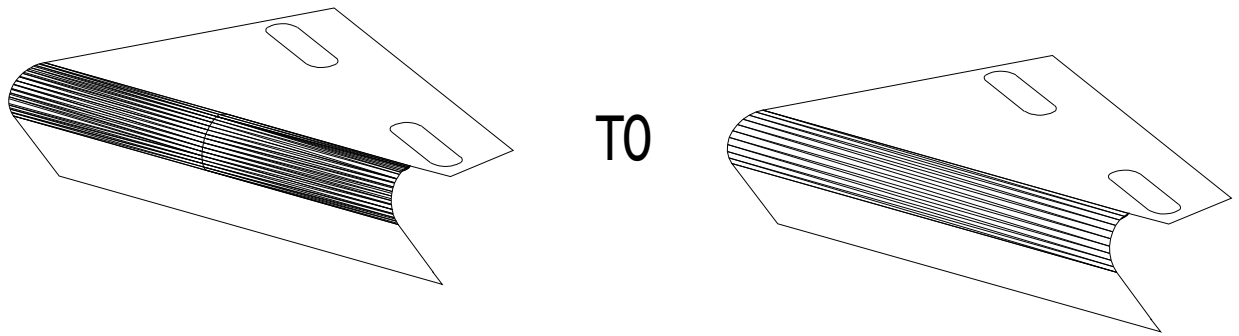
To be unbendable, a bend must be a sharp connection of 2 faces or a portion of cylinder ruled between 2 curves.

You can visualize the mesh structure of a surface in clicking on the "visualisation" icon . Choose "all edges" and "wire frame" then APPLY in the following window.



After using **keep skin / no build** , If the portion of cylinder has a complex mesh structure, you can use the "Clean bend" or "Manual cleaning" function:

Choose the mesh density and click the bend surface to get a regular clean mesh structure:



## 5.2 Recreate bend

It allows to recreate a sharp bend from a bend which has kept its original shape.

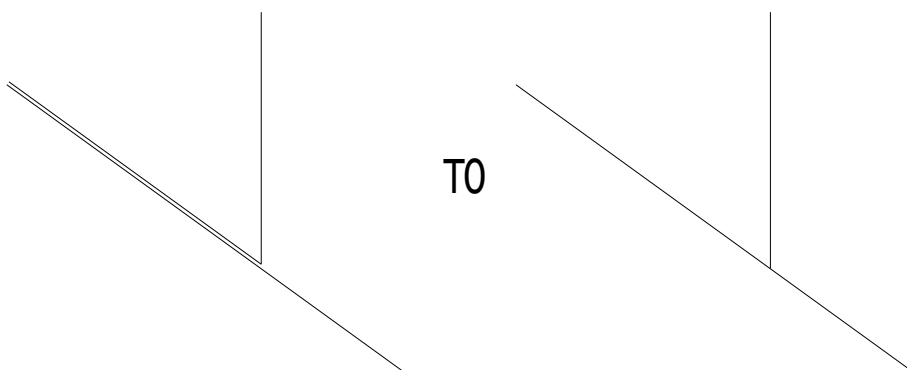
## 5.3 Connect points

When the geometry is not precise you can have a very tiny distance between 2 points. The "Connect points" function checks all the part and connects all the points which have a gap smaller than the chosen value.

Be careful, this function doesn't work for a point which is not connected to an edge. In this case you must use the next function "Point to edge".

## 5.4 Point to edge

This function allows to connect the clicked point to the nearest edge:

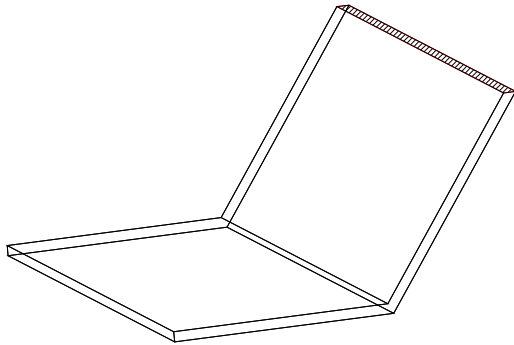


## 6- Particular cases

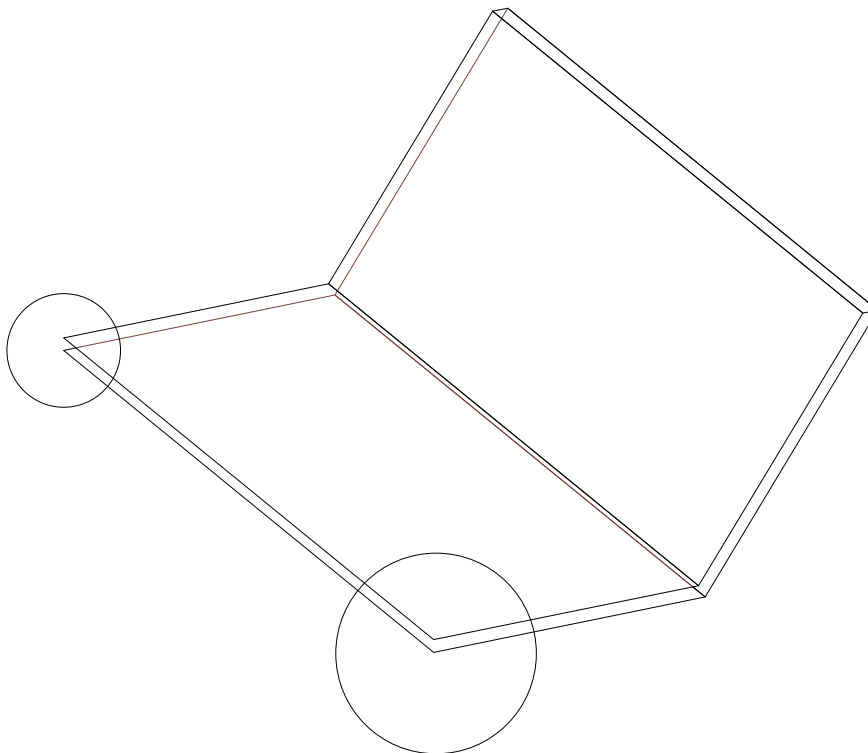
**When you don't get the expected result after "keep skin", before doing anything else, try to invert the two points you've just clicked or choose another couple of points.**

### 6.1 After "keep skin", the internal and external skins are both kept.

It normally means that there is an edge which is not perpendicular to the flange as the hatched edge in the following sketch.



So you get this:



In this case, just remove the non-perpendicular edge with the icon "delete"

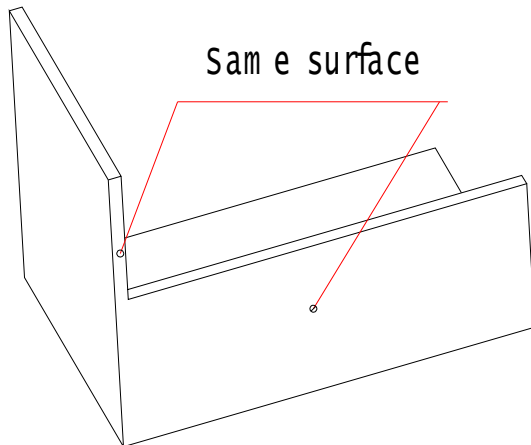


Then run again the "keep skin" function, you will get the good result.

## 6.2 Ambiguous cases.

Depending on the way the part was designed, especially when it is not drawn as a real sheet metal model, you can find some ambiguities. The most common is when an edge and a flange are on the same surface.

Example:



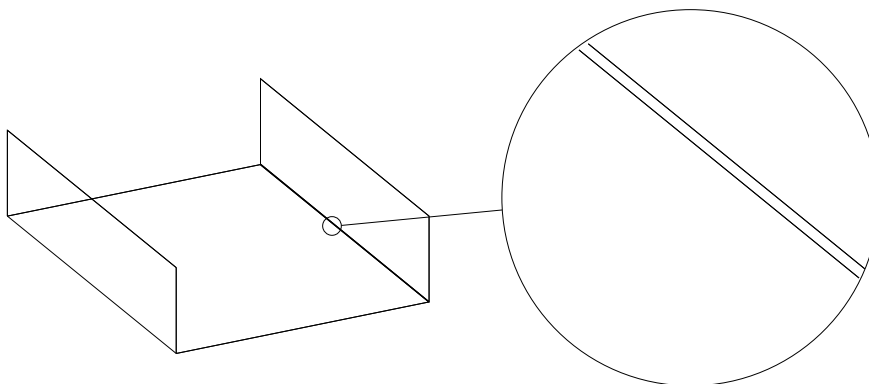
Generally, you can avoid the ambiguity in clicking the first point on the non-ambiguous side. Inside in the above example.

If the model is ambiguous on both sides, you must split the surface in 2, for example with the "break path" function.

## 6.3 Edges not connected

Sometimes, after "create solid", when displaying in simple mode, a bend line appears with the same color then the free edges.


It means that the two edges are not connected. You can see it in zooming.



In this case you must use the "connect edges" function. See in the **modifications guide** the § 2-1-10

## 7-Ruled shapes menu

### 7-2 Keep only

With this function, you can select the surface you want to unfold. If there are several surfaces, use the select icon . In this case all the selected surfaces must be on the same side of the material.

This function memorize the value of the thickness which is necessary for “Create smooth solid” function.

### 7-3 Make unbendable

#### 7-3-1 Ruled

Clicking the surface to unfold, eventually choose the bends density, the mesh structure becomes regular. So the surface becomes unfoldable.

The surface must have 2 straight edges between 2 edges to be ruled.

#### 7-3-2 Extruded

This function has to be used when the surface is a straight extrusion from an edge.

The length of extrusion must be  $>$  to the length of the part.

You can choose the number of bend lines.

### 7-4 Create smooth solid

This function creates a solid model which has the thickness of the original model. In case of several surfaces, the system will union and create one part. If it is necessary to make a cutout for unbending, use “TOOL/edges\_corners”, see the **Modifications Guide §2-1-5**

### 7-5 Show/Hide thickness


This function change the display mode with or without thickness.

# Modifications guide

## 1-Structure

S.M.O. can manage 2 different data structures of the part:

1) A complete solid structure with all sheet metal information as thickness, bend radii and so

on. This structure is automatically obtained when clicking the “create solid” function . In this structure, you have the choice between two display modes in using the “eye” icon, thickness hidden or not.

2) A simplified surface structure called 3D sketch. You can obtain it in using “Sheet-Metal/SHEET-METAL/back\_to\_3Dsketch”.

Then you loose sheet metal specific attributes. You can go back to the sheet metal structure in using “Sheet-Metal/SHEET-METAL/Init+show”, of course the system will ask for the thickness and the material side.

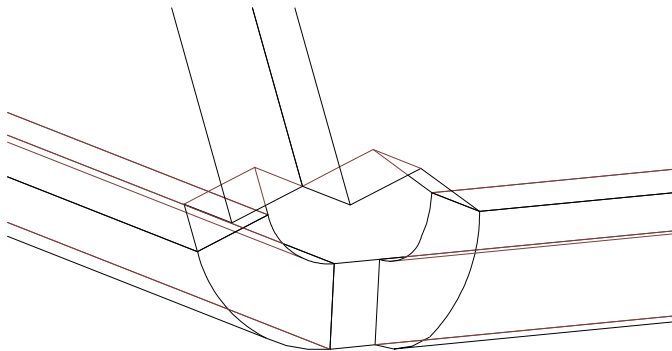
## 2- Sheet Metal/TOOLS menu

### 2-1 Relief and cutout.

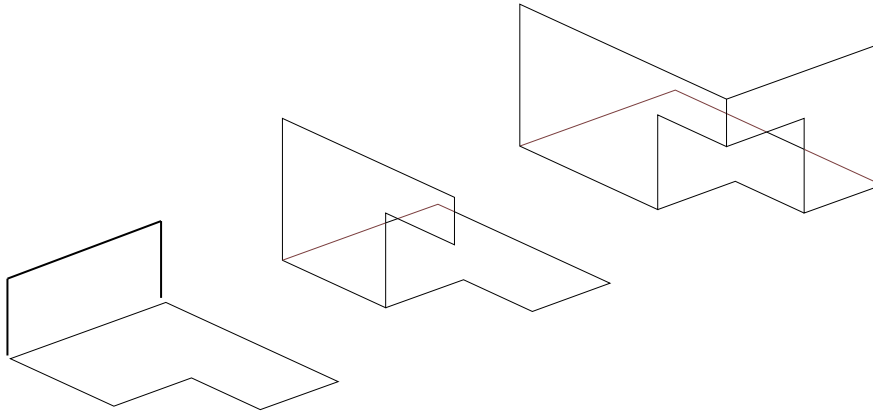
#### 2-1-1 Corner notch\*

This function creates a hole on a corner. It is necessary if you want to give enough space for the bends to avoid material conflict.

The system accepts any value, but the default value= $\text{thickness} + \text{bend radius}$ .



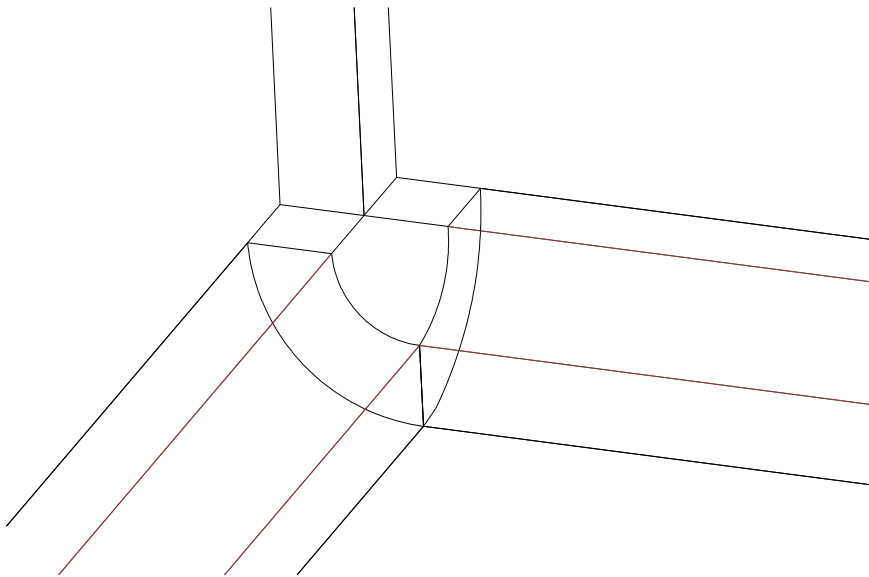
It can be used for one, two or three face



**Note:** For this function, the part must have at least two flanges

### 2-1-2 Break corner

This is simpler than 2-1-1, the hole is created by the subtraction of a parallelepiped.



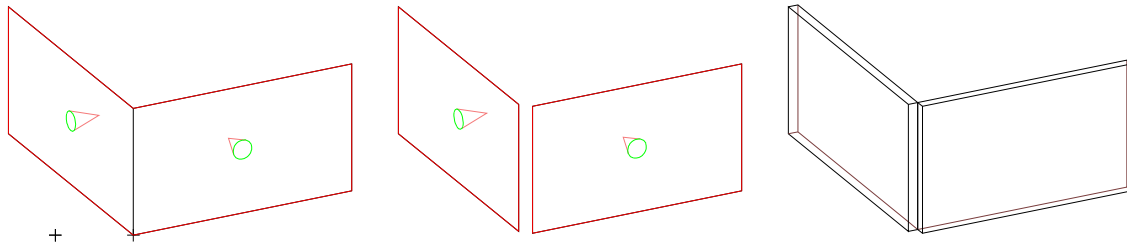
### 2-1-3 Round break corner

The hole is created by a sphere. You can use it only if you have chosen to keep the inside skin.

### 2-1-4 Break edge

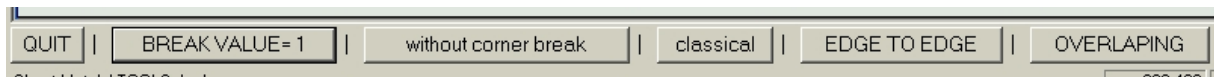
This is the simplest tool to break the common edge of 2 faces. The result is got by subtraction of a rectangular solid feature which have the same length than the length of the common line + the "Overstep value".

The system accepts any break value but the default value=thickness.



### 2-1-5 Edges corners

This is a more sophisticated tool to break the common edge of 2 faces. The system allows you different choices.

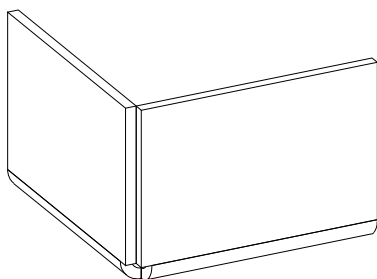


Clicking on the “break value” button, you can tune the gap value which is =thickness by default.

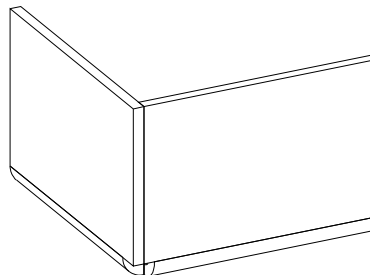
You can choose to add or not a “break corner” automatically in clicking the button “without corner break”. If “with break corner” is displayed, the “break corner” is the simple one ( see 2-1-2 ).

In clicking on the “edge to edge” button you got a symmetrical gap as in “break edge”

In clicking on the “overlapping” button you can allow a face to overlap the other. A small arrow shows the current overlapping side, and you can accept or invert.



Edge to edge

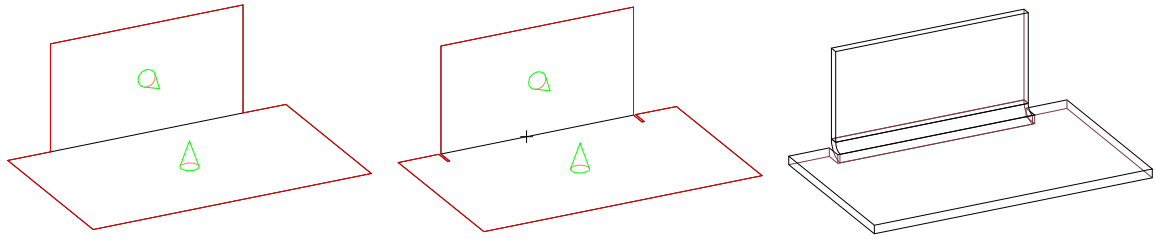


Overlapping

### 2-1-6 Notch

This function creates the necessary relief to avoid a tear. You can indicate the values of the depth and the width.

If you click the button “2 holes” , it changes for “1 hole”. Then, as the system asks for the second click, you will have to click near the concerned bend line end.



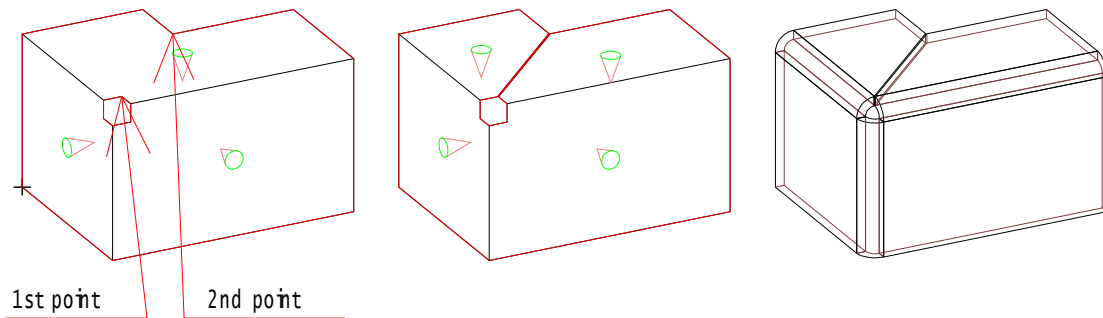
### 2-1-7 Groove

This function splits a face in two, using a groove.

You can change the width in clicking on the button “width=0.1”.

This groove is created between the first requested point and the second.

The system uses a virtual tool longer than the distance between the 2 points. This is tunable with the “overstep” button.



### 2-1-8 Groove straight\*

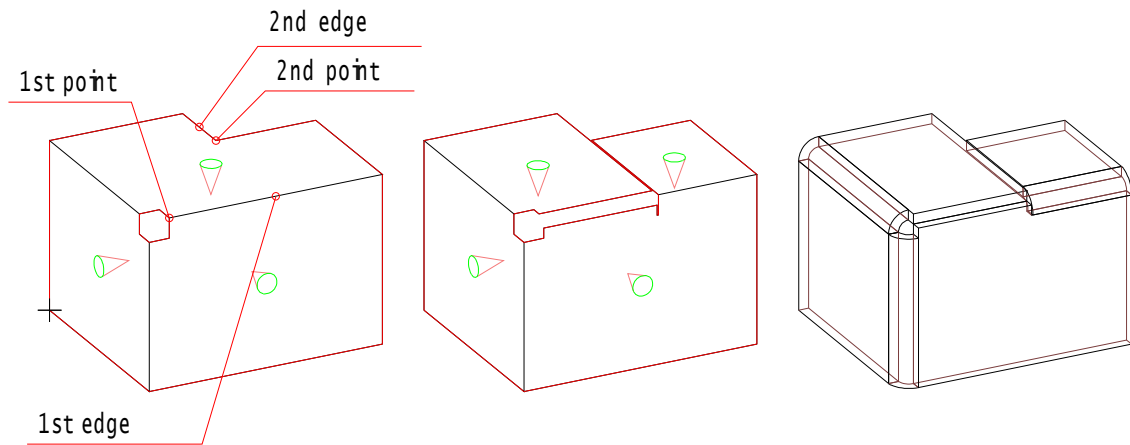
This is a more sophisticated way to split a face.



The function creates 2 cutouts. A “break edge” along the bend line and a “groove” at 90°.

The tunable values are the same than in “groove” and “edges corners” functions.

Click as indicated on the following drawing.

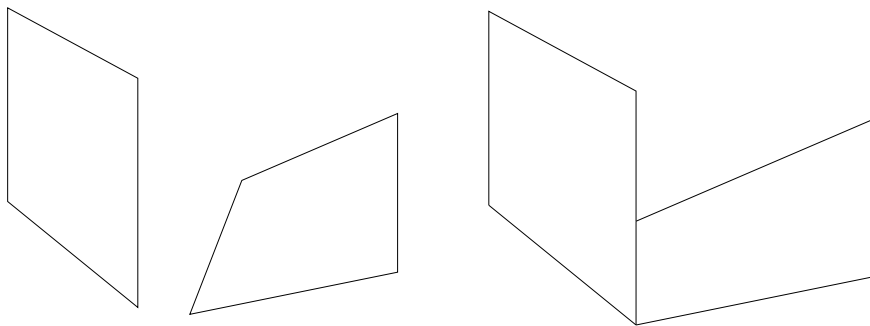


### 2-1-9 Gr\_remove\*

This function removes a groove. You must click on the 2 opposite edges. The two faces to be merged must be on the same plane. Those edges can be parallels or not. If it doesn't work, you can try the next function, "connect edges".

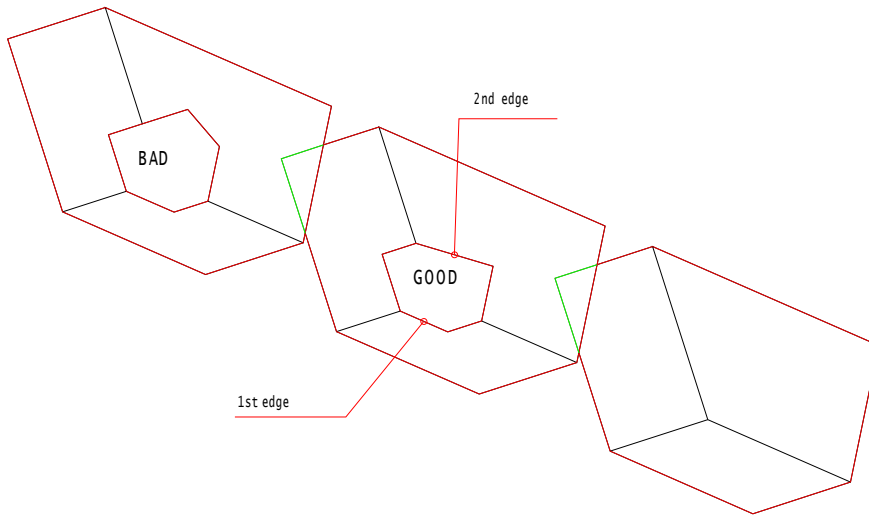
### 2-1-10 Connect edges\*

This function connects 2 faces in extending them to a common line. The edges to be connected can be parallels or not.



This function is perfect to cancel or remove the result of "Break edge" or "Edges corners". This is also available to remove a "break corner", but the number of segments on each face must be 2. If there are more, use the function "Sheet-metal/TOOLS/ext\_edge" to remove the necessary number of segments for keeping only 2, then click 2 opposite edges, not necessary parallels.

See below:



**Warning:** For some of those functions the bend radii are automatically set to the default value (radius=thickness). You can change them with “Sheet\_metal/BENDBREAK/default\_rad” or “Sheet\_metal/BENDBREAK/1\_bend\_rad”

### 2-1-11 Split part

If, after using some of those tools the part becomes two parts or more, you may have to use “Sheet metal/split part” to allow unbending or saving separately.

**Note:** To save the parts separately, use “file/W\_block” and follow the instructions in the bottom bar.

The parts can be reloaded in using “file/open”.

### 2-1-12 Modif angle

Click a bend line, give the new angle value, then the fixed side.

## 2-2 Creation tools.

### 2-2-1 Starting face



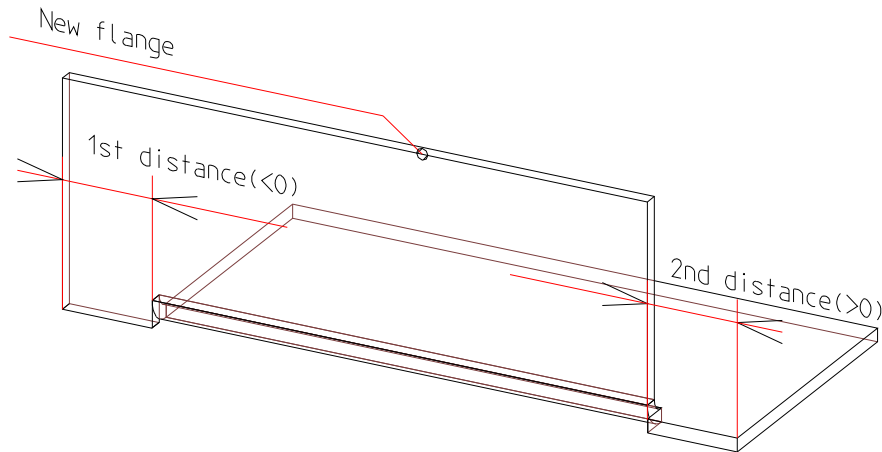
This function creates the 1<sup>st</sup> rectangular flange with a chosen thickness. Input the length and the width then input the thickness or click on a proposed thickness.

### 2-2-2 Create face



This function creates a new rectangular flange starting from the edge of another flange. The system asks for the distances from the ends of the chosen edge. Those distances can be positive or negative.

Then you must input the height and the angle.



### 2-2-3 Create face poly.

This function creates a new flange not necessarily rectangular. The system asks for the distance from each end of the chosen edge.

Then you must input several point according to the new reference axis. Those coordinates can be absolute (x,y) or relative (&x,y). After clicking <END> you must input the angle.

### 2-2-4 Create faces

This function creates several faces in extruding a profile perpendicular to the chosen edge. . The system asks for the distance from each end of the chosen edge.

Then you must input a distance and an angle for each face. The system will extrude this profile.

### 2-2-5 holes (circle, rectangular, slot)



You get those functions in the **holes** menu or clicking on this icon:

Before using them, you must put the origin on the surface you want to work on. See the § 1-2.

### 2-2-6 Linear pattern (circles, rectangles,slot)

You get those functions in the **holes** menu.

Before using them, you must put the origin on the surface you want to work on.  
See the § 1-2.