CADSolutions

SPI SheetMetalWorks 2021

Tutorial for the design of sheet metal parts

© Copyright 2020 SPI GmbH

All rights in this documentation, in particular the right of the duplication and spreading as well as the translation, are reserved. No part of the documentation may be processed in any form (by photocopy, microfilm or another procedure) without written approval of the SPI GmbH reproduced or under use of electronic systems - except for the house-internal use-, be multiplicated or be spread. All technical information was written by SPI with the biggest care and was controlled. However, mistakes cannot be still excluded. A liability for the indirect and immediate damages which originate in connection with the delivery or the use of this documentation is excluded, as far as this is allowed legally. Missing tips to registered trade mark do not entitle to the acceptance, these are freely available.

Table of Contents

Introduction *5*
Some theory to begin with 6
SPI unfolding engine 8
The purpose of SPI SheetMetal Works *14*
Data export *15*
Activating SPI Sheet MetalWorks *16*
Exercise 1 ..... *17*
Make a solid body a SPI sheet metal part 20
Create Unfolding 21
Unfolding Log 24
Error Protocol 24
Exercise 2 ..... *27*
Exercise 3 ..... *33*
Base Sheet – New 34
Flange parameters 36
Design with SPI Sheet Metal Attributes *43*
Exercise 4 ..... *44*
Split Face 48
Corner Split Attributes 50
Bend Radius 52
Relief 54
Bend Marks 57
Exercise 5 ..... *61*
Profile – New 61
Flange Relief 62
Fillet 65
Shortening Attributes 68
Update Unfolding 69
Exercise 6 ..... *71*
Facet 75
Attributes 75
Cost Module 77
Exercise 7 ..... *78*
Unfold Parameters *81*

© SPI GmbH 2020

General **82**
Display **85**
Freeform surfaces **87**
Precision **90**
Post Processing **93**
Relief **96**
Bend Mark **103**
NC **107**
Bend Table **109**
Drawing **110**
DXF **111**
GEO **112**
WiCAM **113**
PDF **114**
Actions **115**
Index ***116***

© SPI GmbH 2020

Introduction

This tutorial is dedicated to SOLIDWORKS users who intensely deal with sheet metal construction and/or manufacturing.

About this Tutorial

This tutorial is supposed to show and explain how **SPI SheetMetalWorks** software helps you to design sheet metal parts and how it helps to prepare your data for the NC-controled manufacturing. The tutorial is divided into the following sections:

Section 1

Section 1 treats the mathematical bases of the sheet metal unfolding and explains the concept and theoretical function way of **SPI SheetMetalWorks**.
You should read over this paragraph in particular completely when you deal for the first time with sheet metal construction software of SPI; but also if you are long-standing SPI user you will bump into new functions. No fear of the theory: Even if you would like to begin as a "man of the action" rather directly with the practical exercises - you will profit from the reading of this section!

Section 2

In section 2 you get to know the practical use of **SPI SheetMetalWorks** with the help of different examples. All mentioned examples are located in the SPI installation subfolder `Documentation\Tutorial Exercises`. During the exercises you will also get to know the numerous options of the software in detail. This section complements section 1 and is dedicated to all users of **SPI SheetMetalWorks**.

Requirements

As a user you should fulfill following requirements:

- Experience with the CAD software SOLIDWORKS
- Experience in the design of sheet metal parts
- Knowledge of the Windows™ operating system

Typographic conventions

In this tutorial the following typographic conventions are used:

Convention	Meaning
Verdana Bold	Commands and options of SOLIDWORKS and **SPI SheetMetalWorks** are displayed in this character font. For example **Material > Add** means that you are supposed to choose the command **Add** within the menu **Material**.
`Courier New`	Names of features and files are displayed in this style. Example: `Enclosure.sldprt`
1 2 Schritt	Double lines separate sections of explaining text from section that describe practical exercises. The steps within the exercise are numbered in **Verdana, bold**.

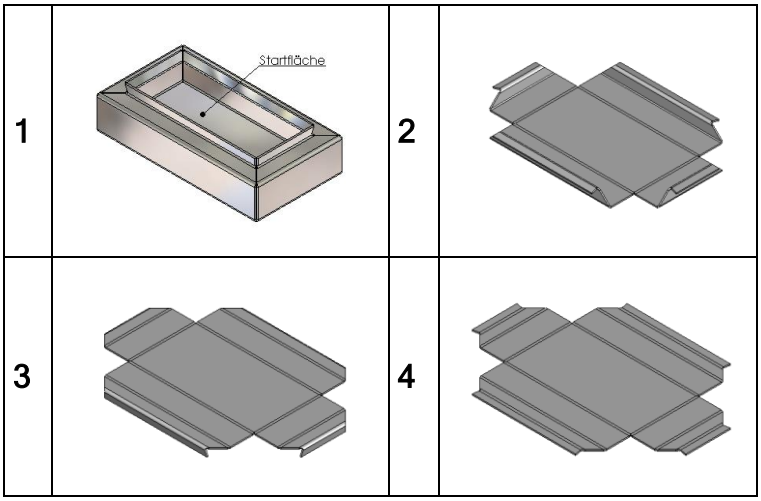
© SPI GmbH 2020

Some theory to begin with

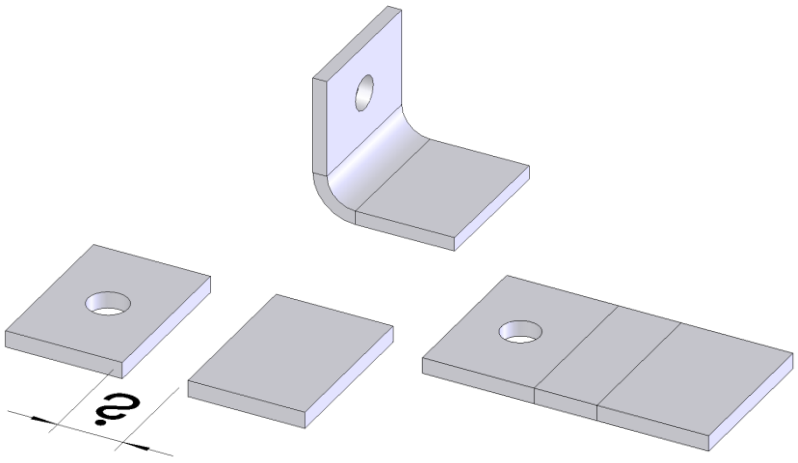
To be able to understand easier the different concepts in **SPI SheetMetalWorks** and SOLIDWORKS, we would like to look at first at the basic questions: „How does one create a sheet metal unfolding?" and „To what extent the behavior of a real sheet metal corresponds in manufacturing with the theoretical consideration?"

Unfolding a sheet metal part by „laying down"

The shape of the final sheet metal part as well as further technical properties arises from the constructive conceptual formulation, as for example material, stability, surface, etc. In order to be able to produce the desired part, the accompanying flat pattern is to be determined at first from the form of the finished part. Moreover, the level ranges are folded step-by-step in the level of the selected start surface:



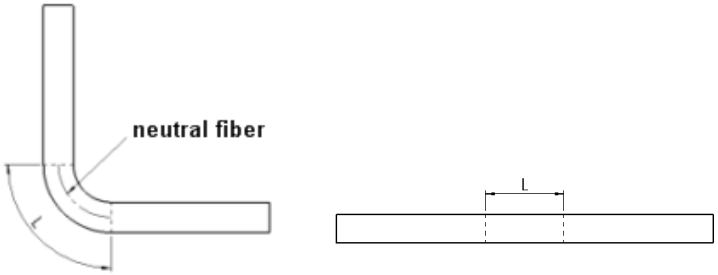
Indeed, determining the position of the single surfaces, in the flat pattern in relation to the start surface can be labour-intensive, however, throws up no questions to the approach because the level surfaces are not changed with the bending. Quite differently it looks with the bend zones: The bend zone is a level surface in the flat pattern, has in the crooked part, nevertheless, in possibly cylinder form. With it the question positions itself: How must the bend zone look in the flat pattern, so that the desired finished part arises after the bending?



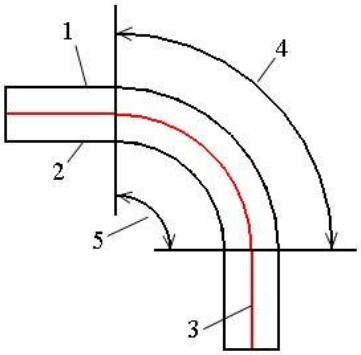
© SPI GmbH 2020

Determine the bend zone using the "neutral fiber"

In a first theoretical approach it seems reasonable to determine the stretched length of the bend zone, using simply the length of the arc in the middle. This imaginary middle is called „neutral fiber"

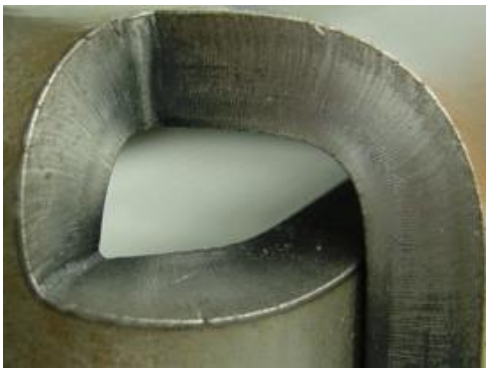


However, the bending is more then "lay into a curve" of the neutral fiber. The sheet metal is transformed: On the inside it is crushed, on the outside stretched.



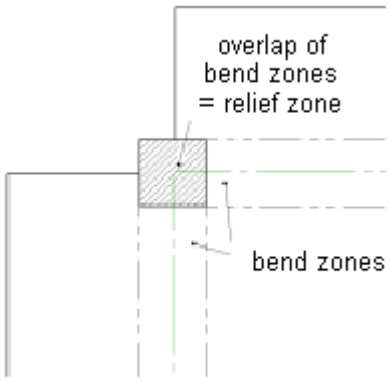
1 outer fiber
2 inner fiber
3 neutral fiber
4 stretch
5 crush

This affects in two things: On the one hand, the stretched length does not correspond in practice with the computational ones, - as nevertheless in theory and practice the stretched lengths can be determined - to the other it can come at the ends of the bend zones to deformations which can not be described mathematically.



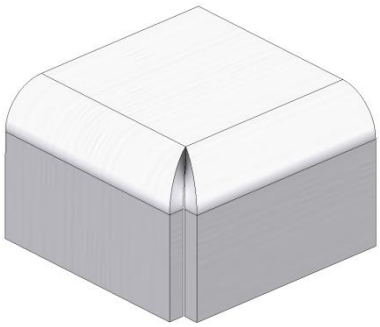
© SPI GmbH 2020

SOLIDWORKS follows a widely used approach to represent the bend zone's end surfaces: The surfaces are and remain plane. This succeeds especially when the bending crosses the complete part or when the ending of the bend zone is relieved. The relief leads to a free space at the end of the bend zone, for example by removing the material of the intersecting bend zones. (See picture)



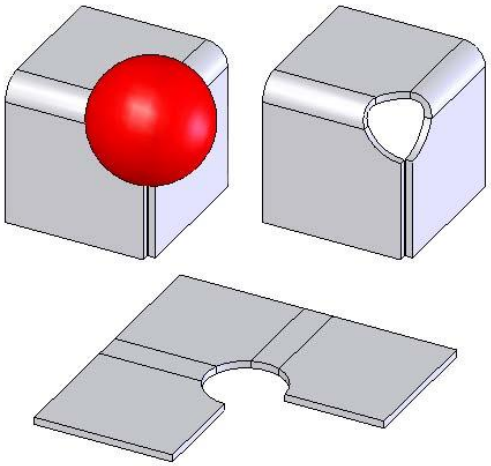
To relieve a corner is a common procedure when manufacturing sheet metal products. This avoids undesired deformations during the bending.

Computing the unfolding from the desired final geometry runs into problems, when the corners are not or only incompletely relieved. It is often wanted to avoid reliefs to obtain a bended part without holes in the corner, since this minimizes the effort to weld the edges to close the enclosure.



SPI unfolding engine

The unfolding engine of **SPI SheetMetalWorks** produces flat patterns that are optimized for processing in NC software for bending and cutting. Of course it is able to unfold any sheet metal part that has been designed in SOLIDWORKS. Additionally, it can calculate unfoldings of a variety of imported bodies, even if they are suffering precision problems.



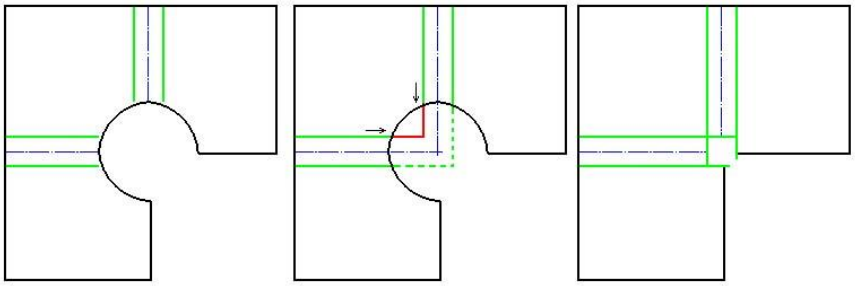
The calculation of bend deduction can be evaluated using one of the following methods: Table, K-factor, formula or TRUMPF TruTops Table. All calculation methods are maintained within the SPI material management.

© SPI GmbH 2020

Bend zone calculation

The calculation of a flat pattern starts with analysing the bend zones. The first step is to remove all corners. For this the volume of a sphere is subtracted. (See picture, left). This reduces the complexity of the calculation of the bend zones since all inaccuracies of the corner situation of the 3D model do not need to be considered. The result is a fast calculation of all bend zones without errors.

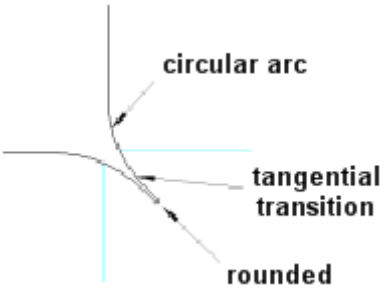
Corner relief



Picture 1: Insert of a rectangular relief

Having calculated the bend zones, the corner geometry is inserted. This is usually a relief. To achieve this, the lines of the bend zones are extended until they meet each other (see picture, middle). The intersection point is used to position the relief, according to the relief attributes. These attributes are stored during the design in the 3D sheet metal part. The user can choose from the most important relief types (rectangle, circle, and triangle). The relief types are maintained in a dedicated tool library. Also the tool data from the TruTops punch database can be read to guarantee, that only reliefs are used that can be manufactured in the workshop.

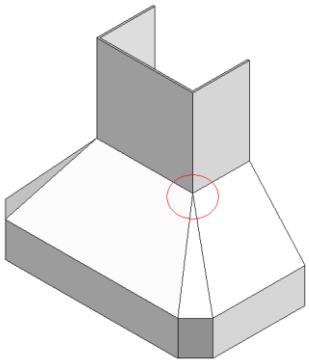
The usage of particular Laser Edge relief leads to a "watertight" corner after bending the flat pattern in the real world. Additionally, the laser edge is detailed in a way that a perfect cut is realized with a simple NC program, composed of lines and tangent circular arcs (see picture)

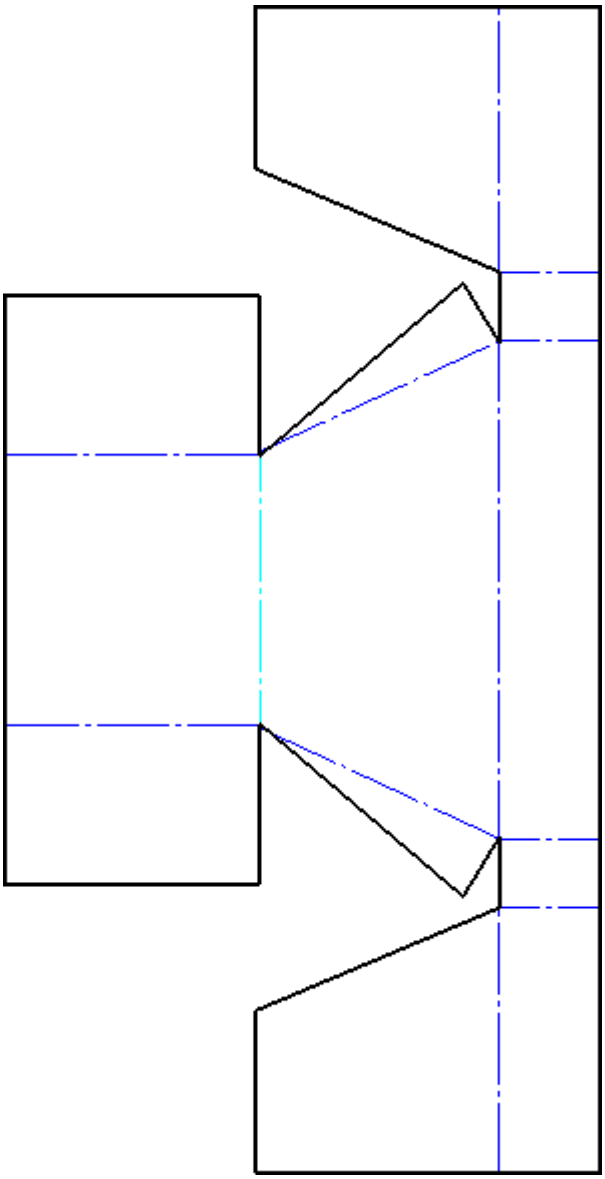


© SPI GmbH 2020

Multiple bend zones meeting in single corner

In the production of sheet metal products, it is not unusual when multiple bend zones meet in one point. This is one of the cases that will be processed by **SPI SheetMetalWorks** without pain and errors.

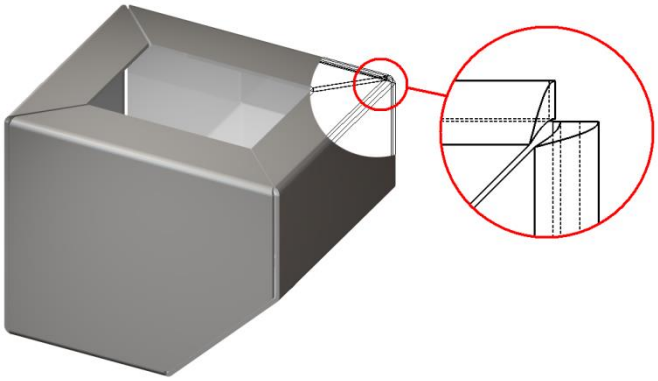


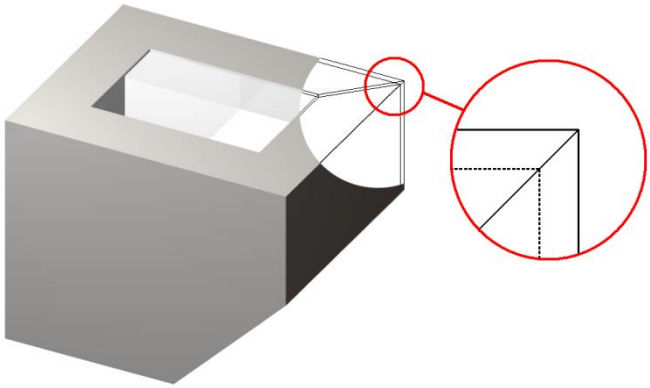


© SPI GmbH 2020

Sharp edged sheet metal parts

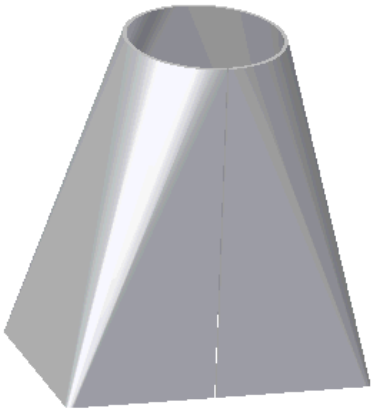
Another advantage comes up in the construction by the use of sharp edged models. The 3D representation is easier and more robust compared with filleted sheet metal parts and offers more freedom in designing new parts.

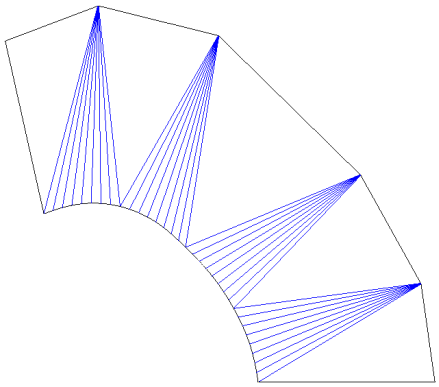




Unfolding freeform surfaces with bend lines

The design of transition parts connects two different contours. For example, a circle is to be transferred to a rectangle.





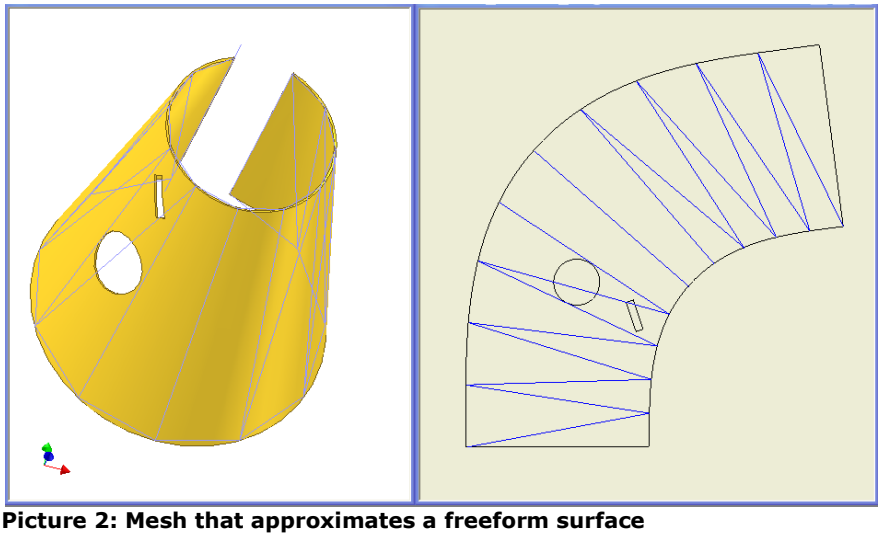
© SPI GmbH 2020

Parts like this can be manufactured by roll forming. However, it is possible to make such products with multiple bends.

The surfaces that form the transition are represented as freeform surfaces within the modeling core. The SPI unfolding engine can easily unfold faces like these. The flat pattern can be used to manufacture the product by roll forming or bending.
Besides the contour the unfolding engine is able to calculate the bend lines that are necessary for bending on a bend press. This is done mathematically with no modification of the 3D model. The SPI unfolding engine takes into consideration the bend angles and bend radii.

Calculating an unfolding

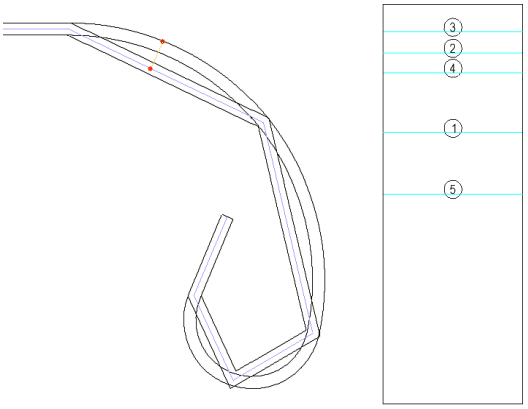
To unfold a freeform surface, the SPI unfolding engine creates a mesh of plane triangular faces that approximates the shape of the freeform surface. The parameter "Net size" controls the number of straight lines that are positioned onto the surface following the minimum curvature. Surfaces that can be unfolded in a mathematical sense need to have straight lines in which one of the two main curvatures has the value 0. In some cases, these lines assimilate quadrangles. In other cases, the quadrangles need to be divided into two triangles to obtain plane faces.



Picture 2: Mesh that approximates a freeform surface

The SPI unfolding engine calculates the flat pattern of the faceted model. During the calculation the border contour of the freeform faces are projected onto the plane faces. This allows unfolding holes in the faces, too.

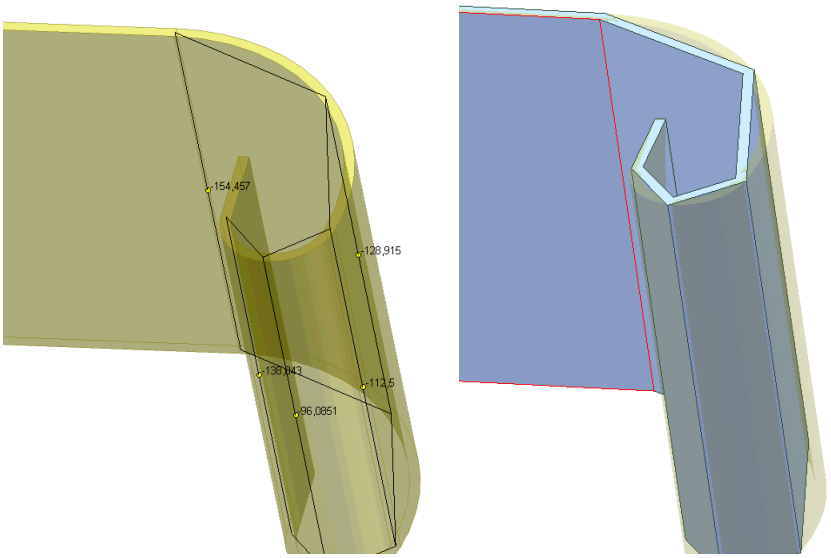
© SPI GmbH 2020



Name :	loft_tang		
Material :	Neutral		
Press Brake :	Neutral		
Thickness :	0.5 mm		
No.	Radius	Shortening	Inclusive Angle
1	2 mm	-0.3833 mm	129°
2	2 mm	-1.2 mm	96.1°
3	2 mm	-0.2695 mm	138°
4	2 mm	-0.6902 mm	113°
5	2 mm	-0.1303 mm	154°

Picture 3: On the left a 2D section of a freeform surface is displayed with its faceted representation. The right side shows the resulting unfolding.

The calculation of the unfolding assumes that the sharp edges are sheet metal bends. Thus a bend deduction can be incorporated as if there were bend radii.



Picture 4: The left side show the freeform surface part with the approximated facets. The calculation of the bend deduction is identic with the blue part on the right side. All bendings are calculated using the "standard bend radius".

© SPI GmbH 2020

The purpose of SPI SheetMetal Works

The add-on **SPI SheetMetalWorks** is to assist a parametric design and calculation of an unfolding. The calculation of an unfolding of a 3D body leads to a 2D contour that can be transferred directly to any NC software to pre-process the following manufacturing (punching, laser cutting or bending).

Design

SPI SheetMetalWorks supports the design engineer. The software offers commands that are easy to use, like "Create Base Sheet" or "Sharp Cornered Flange".

Examination

The design engineer can unfold a sheet metal part with a single click. The software verifies the geometry of the solid body. Topologic errors are recognized and – depending on the error quality – displayed as warning or error message. This helps to discover faults in an early design phase to avoid unnecessary costs during manufaction.

Unfolding

One of the main features of **SPI SheetMetalWorks** is to calculate an accurate drawing of the unfolding leading to the desired sheet metal product after bending.

The bending of sheet metal deforms the material. This might cause a considerable difference between a mathematical and a practical unfolding. **SPI SheetMetalWorks** maintains the necessary correction of the bend zones in the SPI material database. The user can choose one of these five options to determine the method of calculation:

- K-factor
- standard formula according DIN
- custom formula
- table containing empiric correction values
- TruTops table
- calculation using DIN 6935

The maintenance of material data takes place in an independent module, not inside the CAD system. During the drafting within the CAD system, the user can access the material data, like material type, thickness and bending tool and assign these data to the 3D body. The choice of material data is always the first step when using **SPI SheetMetalWorks** inside SOLIDWORKS.

© SPI GmbH 2020

Data export

TruTops export

SPI SheetMetalWorks can write an unfolding directly into a TruTops GEO file. This bypasses the conversion in the TruTops drawing editor. The GEO file can be read directly into TruTops Bend, TruTops Punch and TruTops Laser.
The configuration of the unfolding parameters, which refer to the TruTops export, is described in chapter **GEO** (See p. 112).

DXF export

Most NC software packages are able to read in DXF12 2D data. To export a SOLIDWORKS drawing please use the **Save As** command. It is recommended to use a drawing sheet that only contains a flat pattern with the related NC data.
SPI SheetMetalWorks accelerates the generation of DXF files. The software can write directly a DXF file when calculating an unfolding.

Unfolding with Cybelec and Delem data

SPI SheetMetalWorks can generate NC information for Cybelec- and Delem controls. It writes additional text elements at the bend lines. They contain the bend angle, radius, etc. To make **SPI SheetMetalWorks** create the NC data for your bend press some options needs to be set in the SPI unfolding parameters. This makes it easier for the NC control to read the DXF file of the unfolding.
Having activated one of the NC option sets, **SPI SheetMetalWorks** verifies all unfolding settings. When running into a conflict the software will prompt the user if the settings shall be corrected automatically. The table below shows the relation between NC program and related parameter settings.

NC press brake control	Cybelec	Delem
Post Processing	Distance to contour = 0 Join bends = Off	Distance to contour = 0 Join bends = Off
Display	Bend Lines = On Bend Zones = Off	Bend Lines = On Bend Zones = Off Bend Two Unfoldings = Off
Lines	All bend lines on the same layer All upper contour lines on the same layer Different layers for contour and bend lines.	All bend lines on the same layer All upper contour lines on the same layer Different layers for contour and bend lines.
Bend Table		Bend Table = Off

Please read the documentation of your NC software, too.

© SPI GmbH 2020

Activating SPI Sheet MetalWorks

When **SPI SheetMetalWorks** is installed correctly but does not appear in SOLIDWORKS, please check in the menu **Tools, Add-ins** that it is set up to load automatically.
Please read also the document **Getting Started** for further details of the installation and registration of **SPI SheetMetalWorks**.

User interface

The SOLIDWORKS Command Manager contains the tab **SPI SheetMetalWorks**, where the mostly used commands are available. These and more commands are also available in the menu **Tools > SPI SheetMetalWorks**.

Designing with SPI commands

SPI SheetMetalWorks differentiates between **rounded** and **sharp edged** sheet metal parts. When creating a rounded SPI sheet metal part a SOLIDWORKS Base Flange feature and all standard sheet metal commands can be used without any restrictions. When creating a sharp edged SPI sheet metal part only SPI sheet metal commands are available. Of course all ordinary features like holes can be applied. (However it is possible to convert it into a SOLIDWORKS sheet metal part like it can be done with imported bodies.)

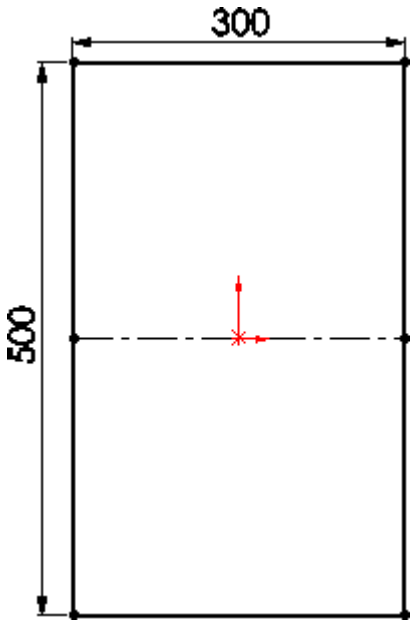
© SPI GmbH 2020

Exercise 1

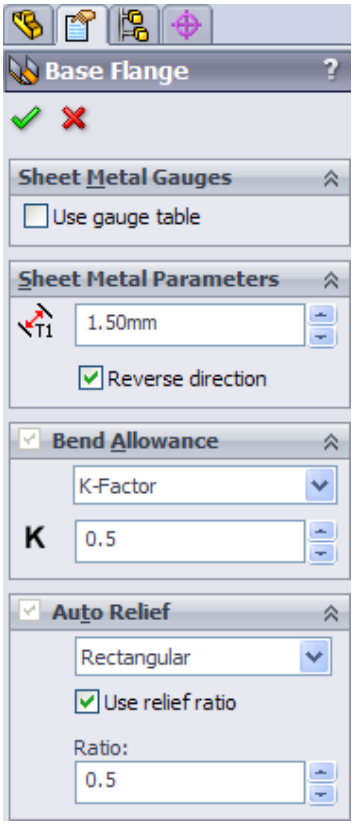
In exercise 1 the handling of SOLIDWORKS sheet metal parts with an additional use of **SPI SheetMetalWorks** is explained.

1 Create a new part using **mm** as units. **Save** and name it `enclosure.sldprt`.

2 Create a sketch on the **Front plane**. Alternatively, you can open the file `Exercise1.sldprt`.



3 Select the sketch and apply the command:
Base Flange

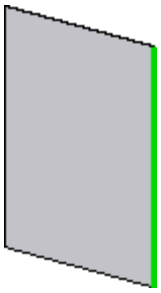


4 **Settings for the Base Flange**
Please use the following settings:

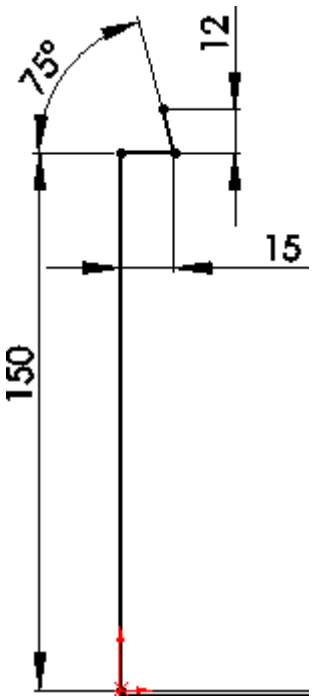
- Thickness = 1.50 mm
- Reverse direction = activated
- Bend Allowance = K-Factor
- K-Factor = 0.5
- Auto Relief: Rectangular
- Ratio = 0.5

© SPI GmbH 2020

5 Create a sketch on the side face according to the figure.



6 Now look **Normal To** the sketch plane and draft the sketch entities according the following image:

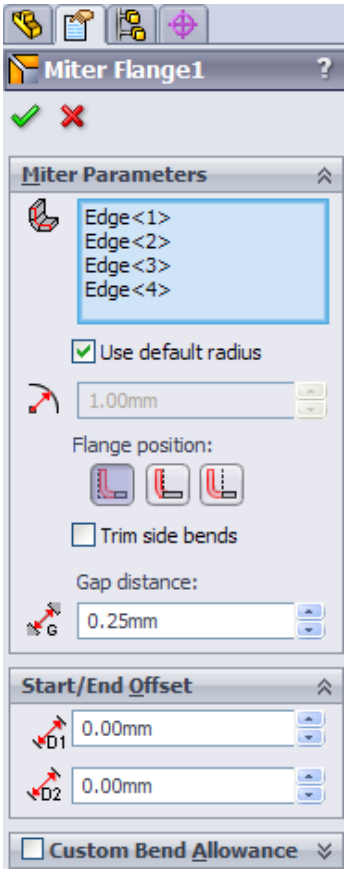


7 Select the sketch and apply the command **Miter flange**.

© SPI GmbH 2020

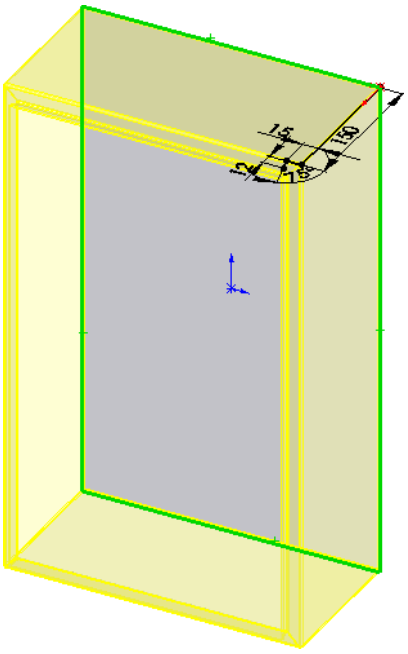
8 **Settings for the Miter Flange**
Please set the Miter Flange parameters like this:

- Flange position = inside
- Flange position = Material Inside
- Trim side bends = off
- Gap distance = 0.25mm
- Start/End Offset = 0.00mm
- Custom Bend Allowance =deactivated



9 Apply the command.

10 Use the command **SPI SheetMetalWorks Sheet Metal Wizard**.



© SPI GmbH 2020

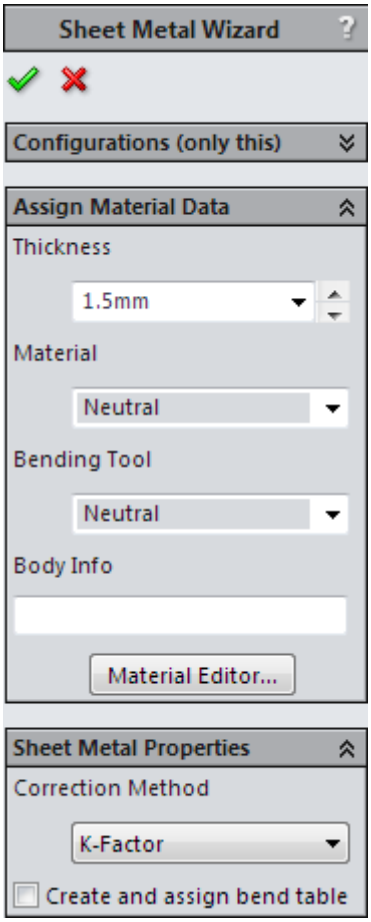
Make a solid body a SPI sheet metal part

To use all SPI commands and attributes the SPI material data are required. Thus the first step is using the **SPI Sheet Metal Wizard**.

The Property Manager displays the parameter settings of the Sheet Metal Wizard and prompts you to choose your material data and –property selection.

The choice and assignment of material data and properties is similar to the value settings when creating a new base sheet (see p. 34).

In this case the thickness is read from the part. However, it is possible to modify the thickness.



11 **Settings for the SPI Sheet Metal Wizard**
Please set up the following values:

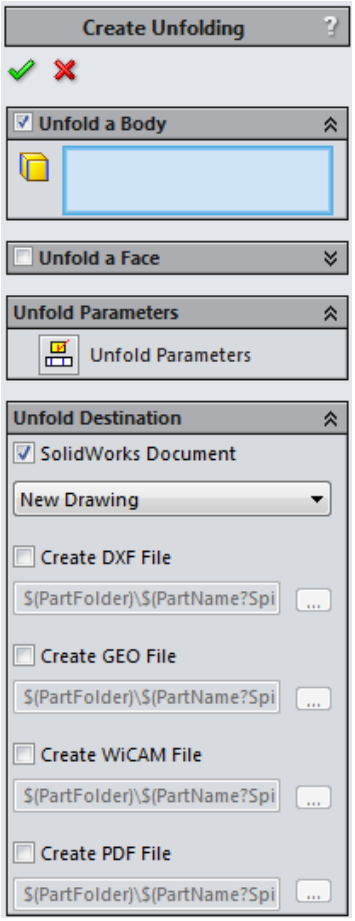
- Thickness = 1.50mm
- Material = Neutral
- Bending Tool = Neutral
- Correction method = K-Factor

12 Now let's verify the design by unfolding the part!
Please use the **SPI SheetMetalWorks** command **Create Unfolding**

© SPI GmbH 2020

Create Unfolding

The Property Manager displays the dialog of **Create Unfolding** that allows setting the parameters for the unfolding (**Unfold a Body, Unfold a Face and Unfold Destination**).
The command Create Unfolding allows unfolding either a complete solid body or only a single face. The SPI unfold options will be respected in both cases. Please see chapter **Unfold Parameters** (page 81) for further details.



Unfold solid body

The user has to select a start edge when the option **Unfold a Body** is activated. This edge is to control the orientation of the unfolding. The start edge will be horizontal in the unfolding.

Unfold surface

When the option **Unfold a Face** is activated, the user has to select the face, which shall be unfolded only. A start edge can be optionally selected to control the orientation of the unfolding.

Unfold Destination

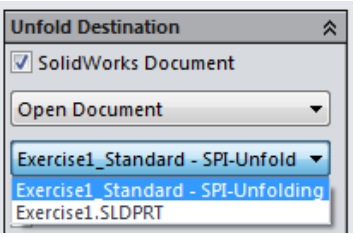
Depending on the selection for Unfold Destination **SPI SheetMetalWorks** will put the unfolding into a view of a **new drawing**, into a sketch of a **new part**, or into an existing **open document**.
It is also possible to create directly a **GEO file**, **DXF file, WiCAM file** or **PDF file**. These unfold destinations may be combined individually. When the part has been unfolded before, the user has also the option to update the prior unfolding.

Caution

The creation of an unfolding will always generate a new unfolding. The unfolding does not refer to the 3D sheet metal part.

SOLIDWORKS Document - Destination Open document

When the Unfold Destination **Open Document** is selected, an additional dialog is displayed that shows all documents which are open in this moment. It is possible to put the unfolding into a **SOLIDWORKS part or drawing** (.sldprt, .slddrw). When unfolding into an open part, it will be drawn in a sketch on the front plane. Unfolding into an open drawing will create the unfold view on the sheet which was configured as sheet name in the drawing unfold parameters. If this sheet does not exist, the current sheet will be taken. If an unfold view of this sheet metal part already exists, it will be replaced.



© SPI GmbH 2020

Unfold Destination GEO file

The unfold destination **Create GEO File** will lead directly to a GEO file. The destination file path can be set globally in the **Application Options** or especially only for this part in the **Unfold Parameters**.

Unfold Destination DXF file

The unfold destination **Create DXF File** will lead directly to a DXF file. The destination file path can be set globally in the **Application Options** or especially only for this part in the **Unfold Parameters**.

Remark

To view a GEO file a specific viewer is required. DXF files can be viewed with the SOLIDWORKS DWG Editor or eDrawings.

Unfold Destination WiCAM file

The unfold destination **Create WiCAM file** will write a WiCAM XML file. The destination file path can be set globally in the **Application Options** or especially only for this part in the **Unfold Parameters**.

Unfold Destination PDF file

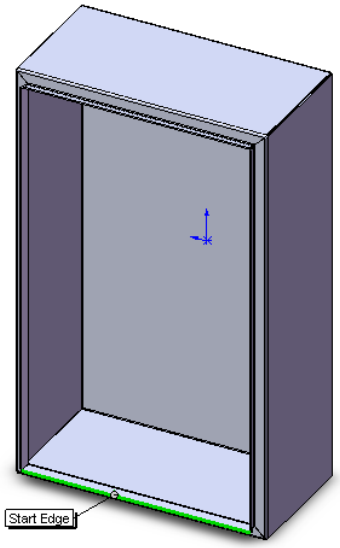
The unfold destination **Create PDF file** will first create the unfolding in a SOLIDWORKS drawing and then save it as PDF file. The destination file path can be set globally in the **Application Options** or especially only for this part in the **Unfold Parameters**.

© SPI GmbH 2020

13 **Create Unfolding.**
Please set up in the Property Manager these values:

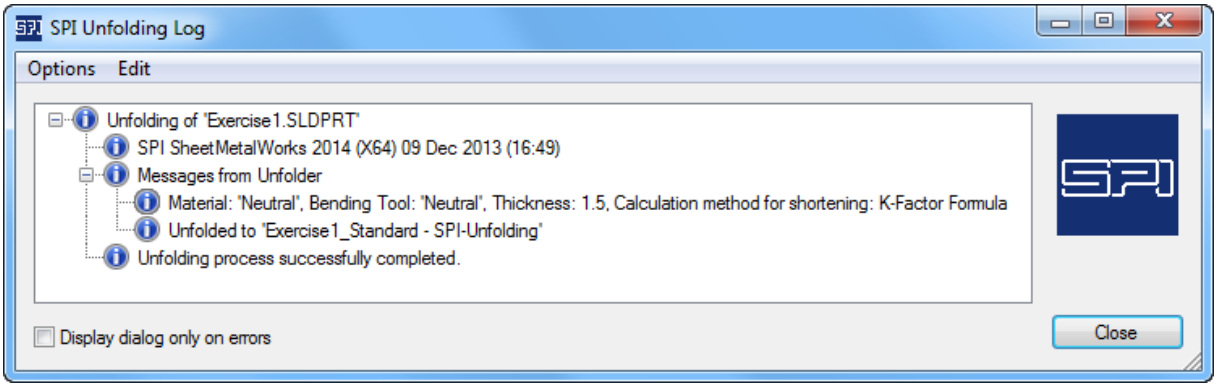
- Unfold a Body
- Unfold Destination = New Drawing

14 **Please pick a start edge according to the following figure:**



15 Click **OK,** to start the calculation of the unfolding.

16 The SPI Unfolding Log is displayed:



© SPI GmbH 2020

Unfolding Log

If the calculation of the unfolding encounters errors, these errors are stored and displayed in the Unfolding Log. Additionally, the errorous edges or faces are numbered and can be highlighted in the graphics window of the 3D sheet metal part. This is to help you finding problematic design details. This information is available in the Unfolding Log.

In the unfolding log dialog, the **Menu > Options** allows setting the display level. You may suppress warnings or info messages so that **SPI SheetMetalWorks** will only display the errors.

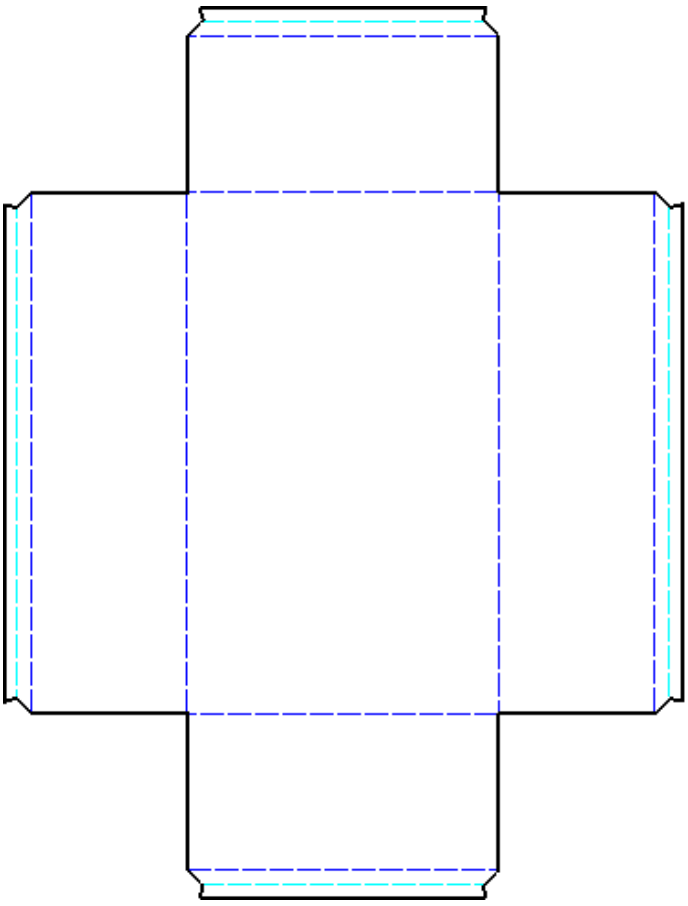
It is not recommended to suppress the display of these messages.

The command **Menu > Edit** allows copying the Unfolding Log content into the clipboard, so you can save it as text file or copy it into an email.

Error Protocol

The command **Error Protocol** displays the most recent Unfolding Log of the current document. You can open the error protocol at any time during the design with the command **Error Protocol**.

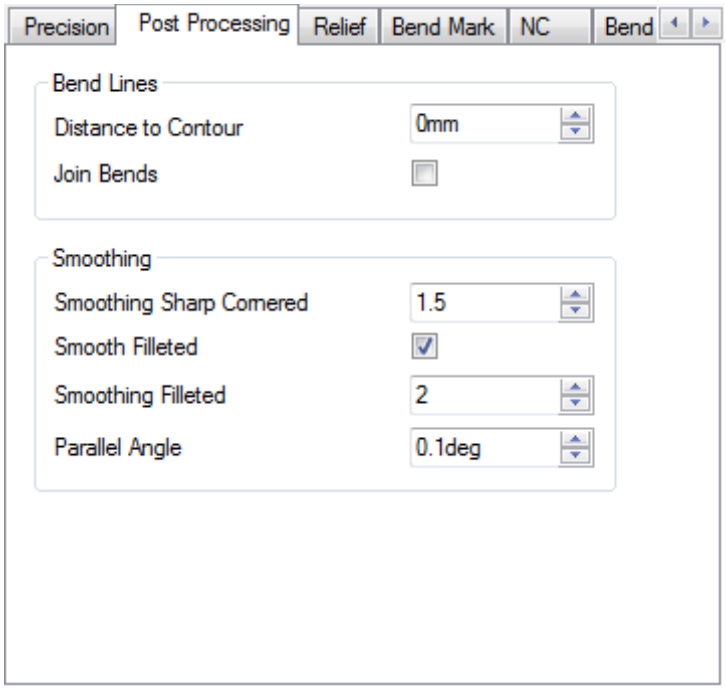
17 Use the button **Close** to exit the **SPI Unfolding Log**. Please compare your result with the figure below:



© SPI GmbH 2020

18 **Save** the part as `enclosure.sldprt`.

To modify the reliefs some settings in the **Unfold Parameters** have to be changed. The settings, that are used here, are described in section **Unfold Parameters** (S. 81).



19 Unfold Parameters **Post Processing:**

- **Bend Lines**

  Distance to Contour = 0 mm
  Join Bends = deactivated

- **Smoothing**

  Smoothing Sharp Cor. = 1.5
  Smooth Filleted = activated
  Smoothing Filleted = 2
  Parallel Angle = 0.1 degrees

© SPI GmbH 2020

20 Unfold Parameters **Relief.**

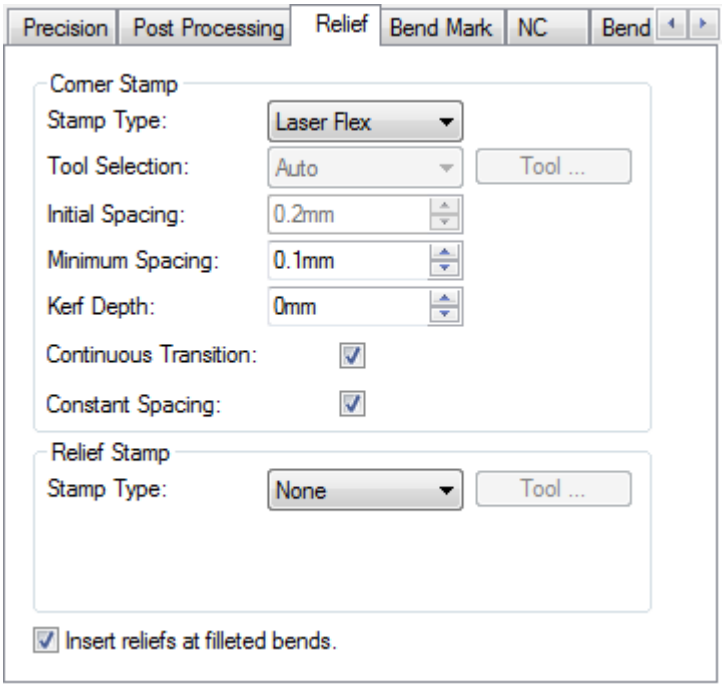
- Corner Stamps

  Stamp Type = Laser Flex
  Minimum Spacing = 0.1 mm Continous Transition = activated
  Constant Spacing = activated

- Relief Stamp

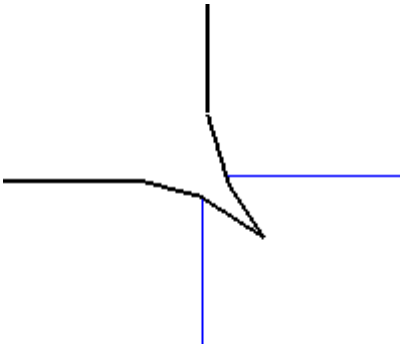
  Stamp Type = None

Important! **Insert reliefs at filleted bends = activated**

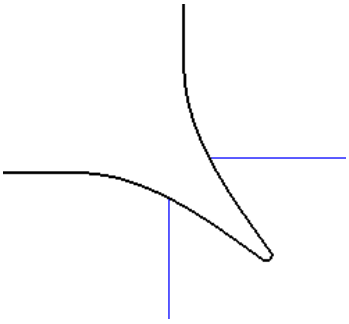


21 Please execute the command

Create Unfolding



Corner relief **before** modifying the Unfold Parameters:



Corner relief **after** modifying the Unfold Parameters:

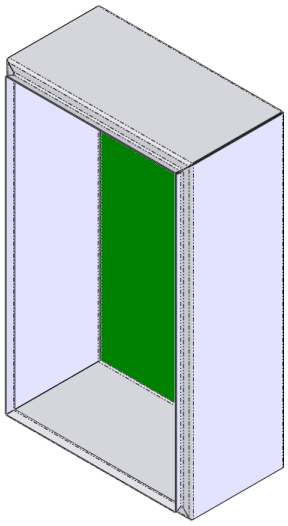
© SPI GmbH 2020

Exercise 2

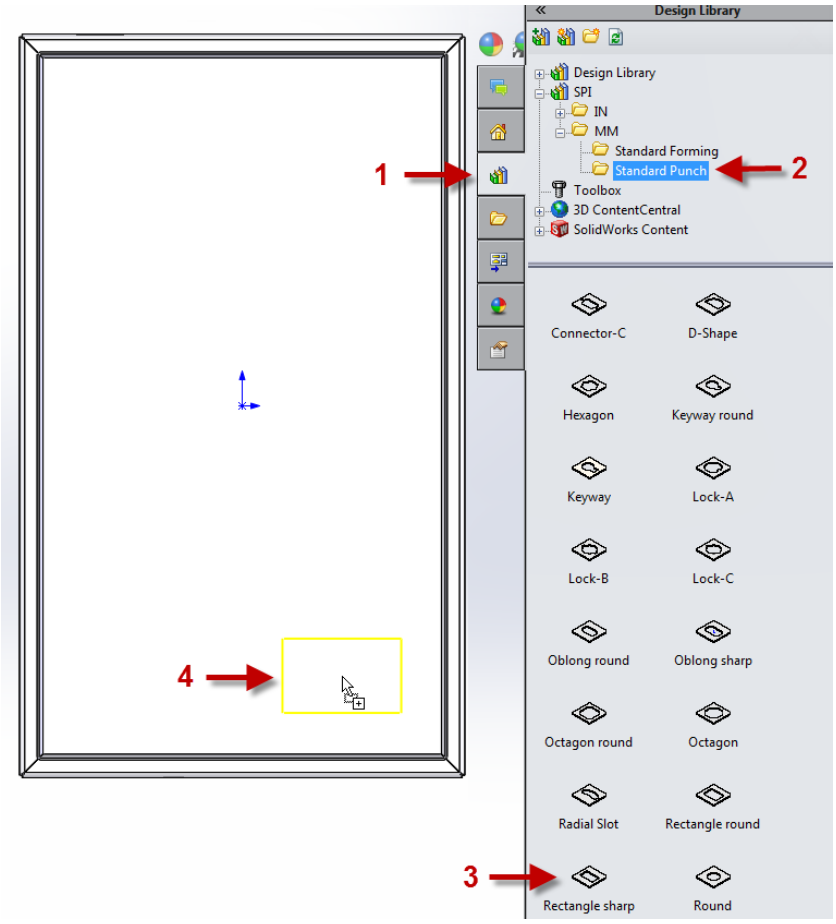
This exercise demonstrates the usage of the SPI punch and forming tools.

1 Open the file `Exercise2.sldprt`

2 Choose the inner back side of the box as shown and use Normal To.



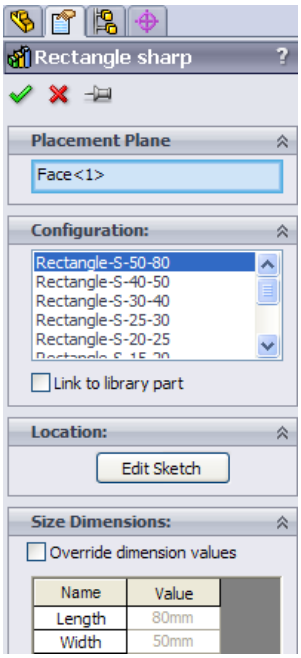
3 Place the SPI punch tool Rectangle sharp as shown in the figure.



(1) Open the SOLIDWORKS Design Library
(2) Open the folder *SPI\MM\Standard Punch*
(3) Click and hold down the right mouse button on the tool *Rectangle sharp*.
(4) Drag the tool in the lower area of the box.

© SPI GmbH 2020

4 In the Property Manager of the Library Feature choose the **Configuration** Rectangle-S-50-80.



The configurations are inside the tool database as delivered from SPI.

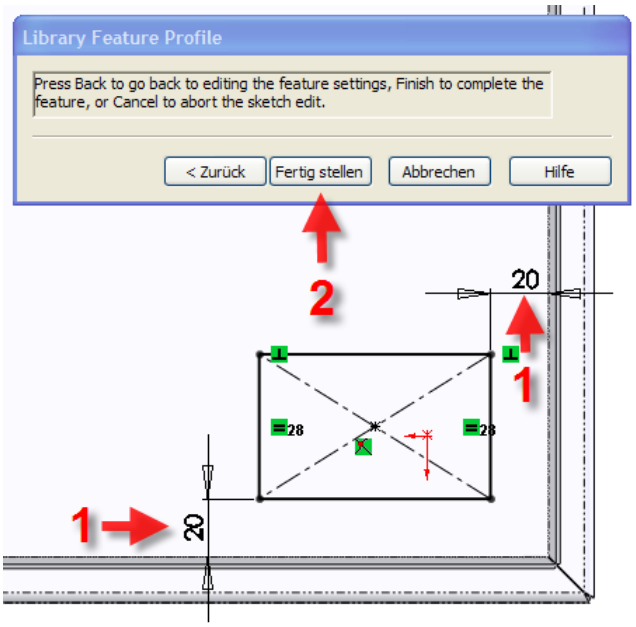
See the help of the **SPI Tool Editor** to learn how to manage the tool configurations.

5 In the Property Manager click on **Edit Sketch**.

You can position the tool on the model by adding dimensions or relations between the outline of the tool and the model.
Please consider some general rules:

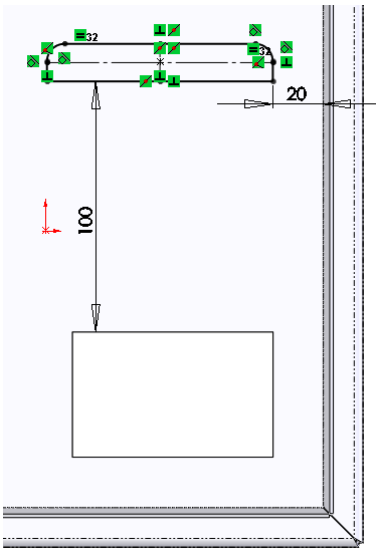
- Do not change elements of the outline sketch of the tool.
- Rotate the tool by adding an angular dimension or a perpendicular or parallel relation between an element of the tool outline and an element of the model.
- In all cases the sketch should be **fully defined**.
- The outline sketch of every tool includes a special point: The midpoint of the enclosing rectangle.

Now position the tool as follows:
(1) Add two dimensions of 20mm distance as shown below.
(2) Click the **Finish** button.



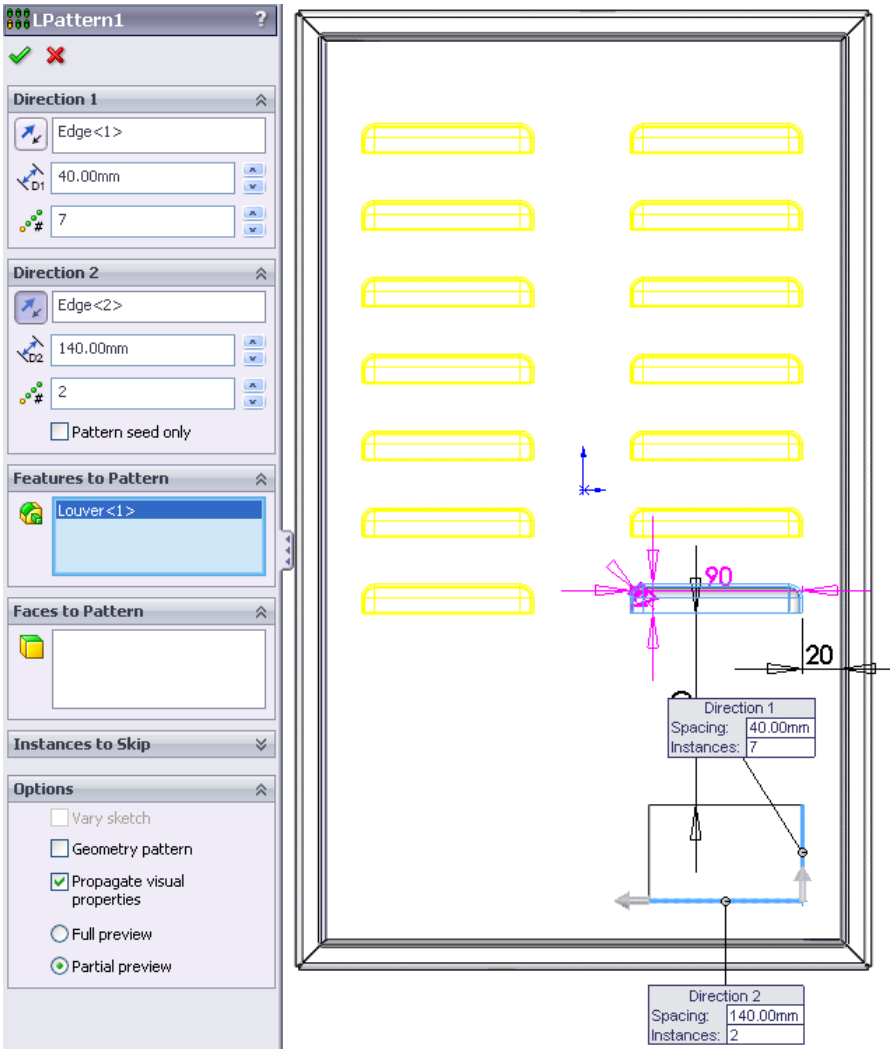
© SPI GmbH 2020

6 Now place the SPI forming tool *Louver* above the rectangular punch and choose the configuration Louver-15-7-90. Click on **Edit Sketch** and add dimensions as shown.



7 Click the **Finish** button.

8 Insert a Linear Pattern and set these properties:



© SPI GmbH 2020

Direction 1

- Pattern Direction = vertical model edge
- Spacing = 40mm
- Number of Instances = 7

Direction 2

- Pattern Direction = horizontal model edge
- Spacing = 140mm
- Number of Instances = 2

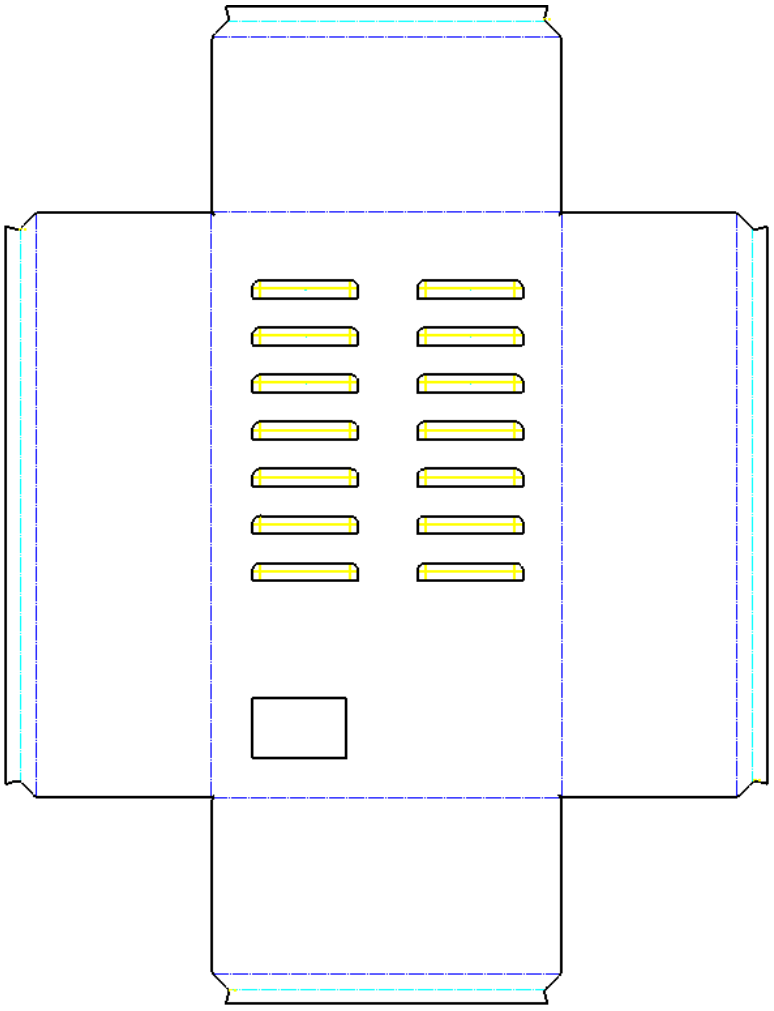
Features to Pattern

- Features to Pattern = Louver<1>

9 Finish the Linear Pattern by clicking the **OK** button.

10 Click on **Create Unfolding** and in the Property Manager click on **OK**.

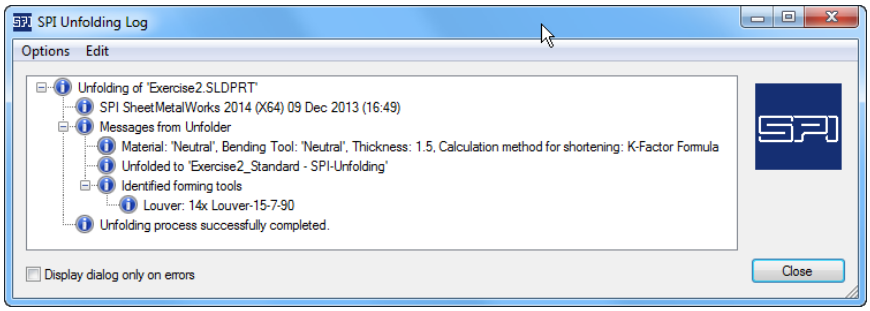
11 Compare your result with this unfold drawing:



© SPI GmbH 2020

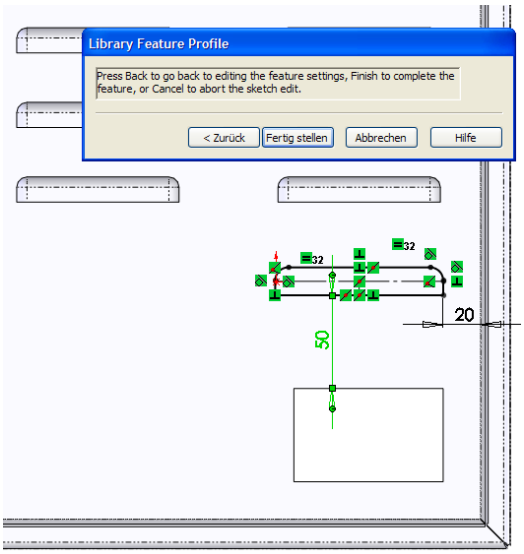
Remark that all the louvers are drawn in form of a special symbol. This identification is possible for all SPI forming tool classes.
In contrast the rectangular punch tool was unfolded normally.
In the middle of the enclosing rectangle of a tool symbol there is a small positioning marker.

12 Close the drawing and open the **Error Protocol**.
The **SPI Unfolding Log** includes a little statistic over all identified forming tools:



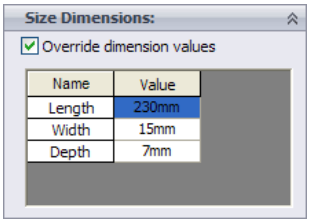
13 Place another Louver above the rectangular punch and choose the **Configuration** Louver-15-7-90 again.
Click the **Edit Sketch** button and add two dimensions as shown in the figure.

14 Click the **< Back** button.



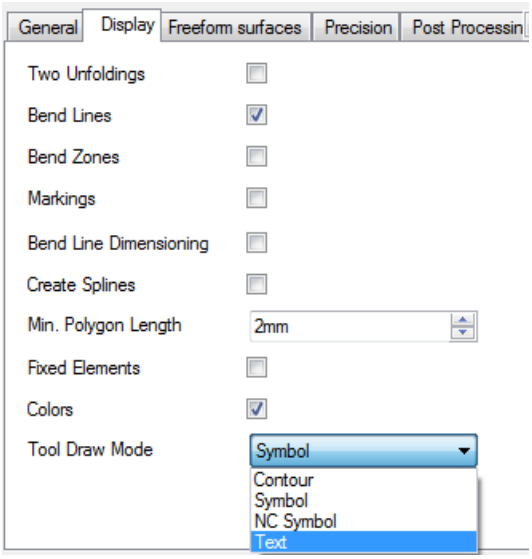
15 Activate the option **Override dimension values** and set **Length** = 230mm

16 Click on **OK**



© SPI GmbH 2020

17 Open the dialog **Unfold Parameters** and on the tab **Display** set the option **Tool Draw Mode** = Text

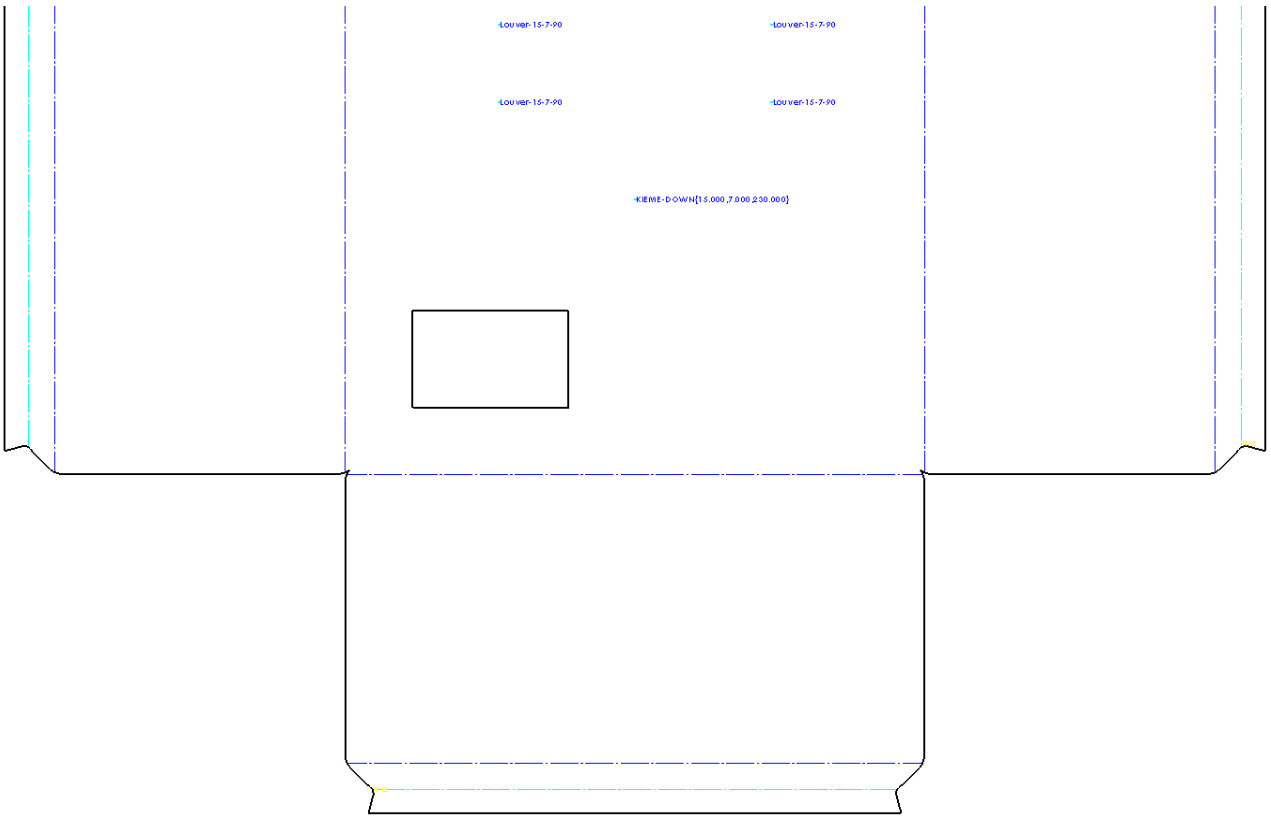


18 Change to the tab **Drawing** and set the color for **forming symbol – down** to blue.

19 Close the dialog by **OK**.

20 Start the Command **Create unfolding** and click on **OK**.

21 Compare your result with the following figure.



On the right hand of the positioning markers now the name of the tool was plotted instead of the tool symbol.
Since the lower louver is not inside the SPI tool database instead the generated name **KIEME-DOWN{15.000,7.000,230.000}** was drawn.

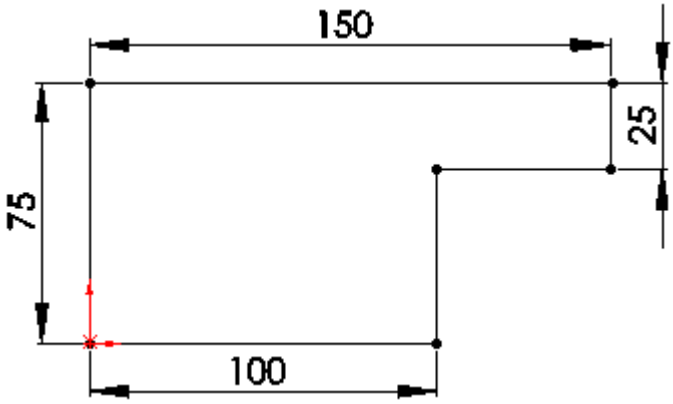
© SPI GmbH 2020

Exercise 3

Exercise 3 is to describe how to create a sheet metal part using the SPI Base Sheet command.

1 Please create a new part using **mm** as units. **Save** and name it `flanges.sldprt`

2 Please make a sketch on the **Front plane**. Alternatively, you can open the file `Exercise3.sldprt`



Remark

Within a closed outer contour, you can sketch one or multiple inner contours which will be recognized as holes.

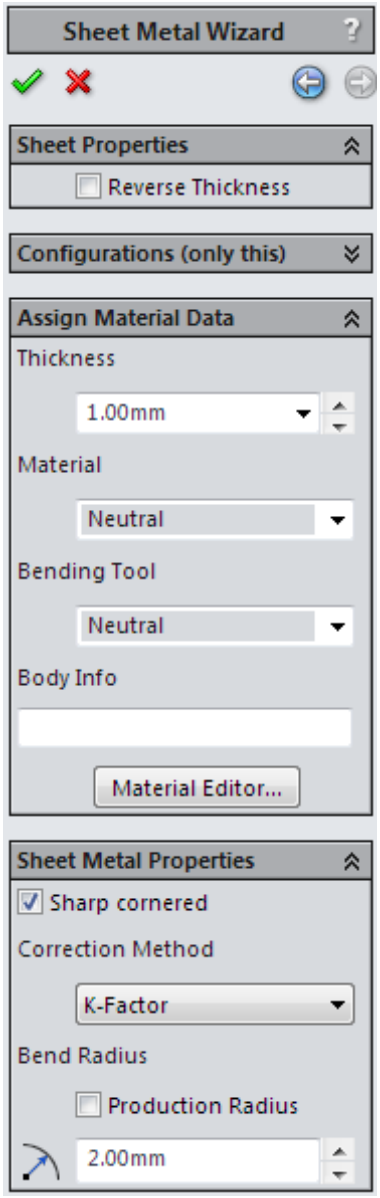
3 Apply the command **Base Sheet - New**

© SPI GmbH 2020

Base Sheet – New

The Property Manager displays the dialog **Base Sheet New**. Now you can enter the Sheet Properties, material data and sheet metal properties.

Please consider, that only Piston and Die Tools as well as material data and thicknesses are available that are stored in the SPI material database. See the help of the **SPI Material Editor** for more details.



Sheet Properties

The Sheet Properties are relevant for the geometry of the 3D model. The parameter **Reverse Thickness** sets up the direction of the first boss extrude feature.

Material Data

The Material Data are read from the SPI material database. Consequently, all values (**Thickness, Material, and Bending Tool**) are linked to each other. Thus only values are available that fit.
The **Thickness** is equal to the length of the linear extrusion.

If you are using bending tool pairs in the SPI material database, you have to choose the **Piston Tool** and combine it with a **Die Tool** here.

The maintenance of the material database is described in the help of the **SPI Material Editor**.

Sheet Metal Properties

To obtain a sharp edged solid body the option **Sharp cornered** is to be activated.

The choice of the **Correction Method** controls how the SPI unfolding engine calculates the bend deduction. It is possible to use experience values from table, K-factor values, a formula or data from a TruTops Bend table. The **Bend Radius** is a result of the chosen correction method.
If the option **Production Radius** is activated an additional selection for the calculation is required. You have the choice between:
Formula, Table and TruTops Table.

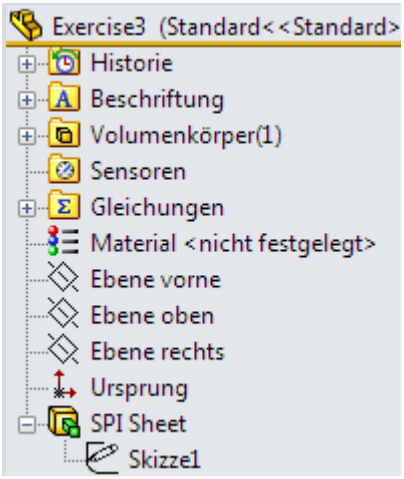
If the option **Sharp cornered** is deactivated, the default radius is identical with the SOLIDWORKS **default radius**.

© SPI GmbH 2020

4 **Settings for Base Sheet – New:**
Please enter the following values:

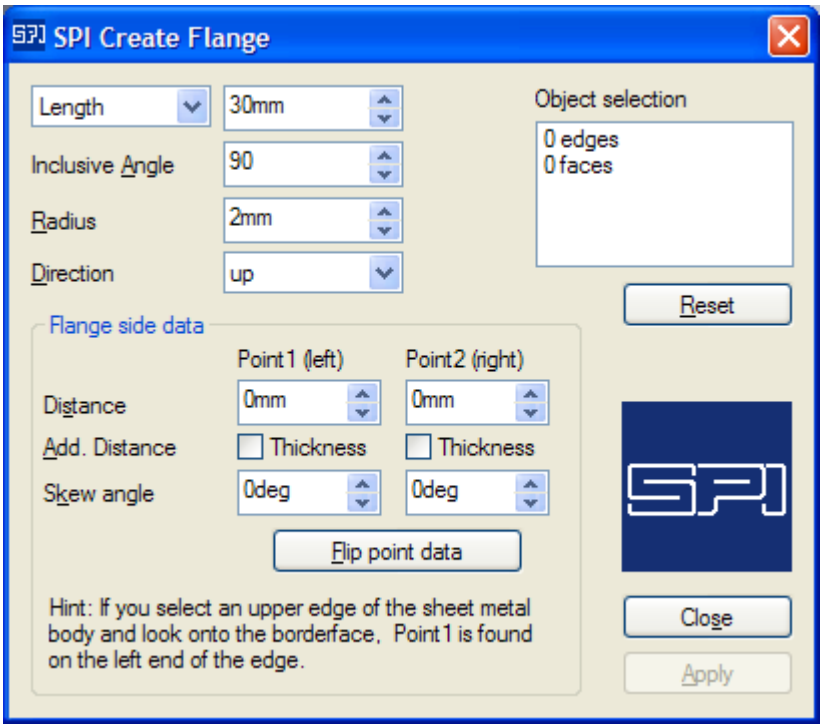
- Thickness = 1mm
- Material = Neutral
- Bending Tool = Neutral
- Sharp cornered = Activated
- Correction Method = K-Factor
- Bend Radius = 2.00 mm

5 Click **OK** to create the Base Sheet. In the Feature Manager a new feature `SPI Sheet` will be displayed:



© SPI GmbH 2020

Flange parameters



Add flanges

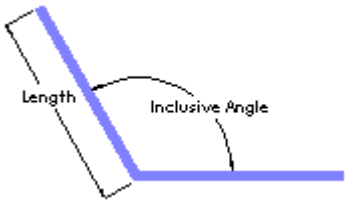
SPI SheetMetalWorks allows attaching different types of flanges to straight edges to perpendicular bordered, parametric sheet metal bodies.

To add a flange, the user must pick a surface, hold down the CTRL-key and pick an edge, too. The edge and the surface are listed in the flange dialog box.
A preview of the upcoming flange is displayed to verify the value before applying the command.

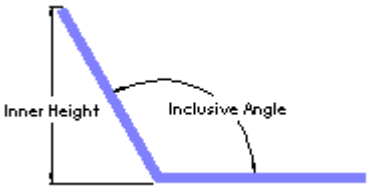
Length and Inclusive Angle

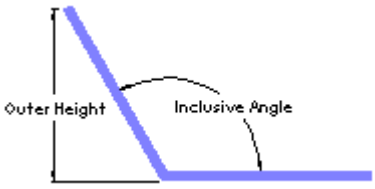
The length of the flange can be calculated in three different ways:

- Absolute length
- Inner height
- Outer height



The **Inclusive Angle** is defined as angle between the face where the flange is to be attached and the flange itself. The valid value range for this parameter is from 90° to 180°.





© SPI GmbH 2020

Radius

The **Bend Radius** is defined as inner radius of the bend zone at a rounded flange. The value must be equal or bigger than 0. **SPI SheetMetalWorks** attaches the value for the bend radius as attribute at the bend edge of the flange. This attribute will be interpreted by the command **SPI Flange Relief**, **SPI Fillet** and **Create Unfolding**.

The value 0 has a special meaning: If the user enters the value 0, the production radius will be read from the SPI material database for the material of this sheet metal part. This radius will be used.

Direction

The selected surface determines the direction of the Flange. The setting **up** will create the flange into the direction of the normal vector of the selected face, **down** in the opposite direction.

Distance and Additional Distance Thickness

The selected border edge is responsible for the orientation and position of the flange. The broadness of the flange depends on the two end points of the edge and the entered distances values. The orientation of the end points (left and right) depends on the orientation of the edge. Use the button **Flip Point Data** to toggle the settings for the right and left side of the flange.

You can enter **positive and negative values for the distances**. A **positive value reduces** the broadness of the flange. A **negative value enlarges** the flange beyond the end point of the border edge.

The option button **Add. Distance** adds automatically the thickness to the distance. This is very useful when designing corner details.

Skew angle

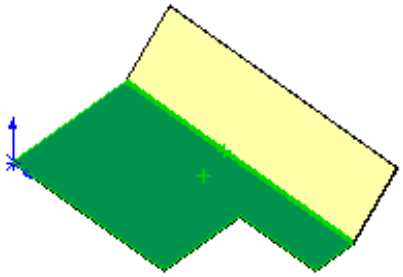
If one or both sides of the flange are supposed to be beveled, you can enter a Skew angle different to 0°. The value has to be in the range between -90° and 90°. If the value is bigger than 0°, the skew will be into the flange. Values that are smaller than 0° will bevel to the outside.

© SPI GmbH 2020

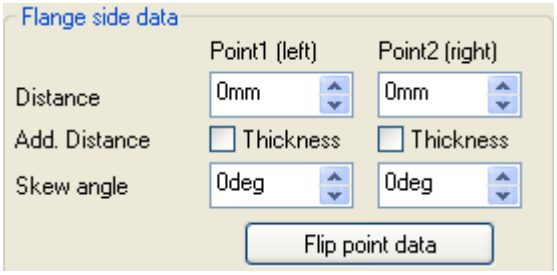
6 Please click the icon **Sharp Cornered Flange.**
This opens the dialog box **SPI Create Flange**.

7 **Creation of the 1st flange.**
Please pick the face and the right border edge (Length = 150 mm).

Settings for the 1st flange.
Please enter the following values:



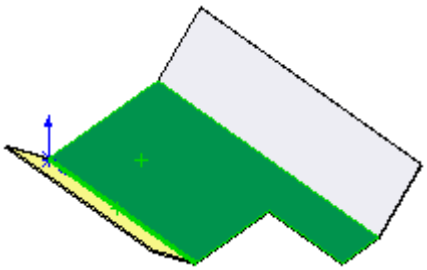
- Inner Height= 30 mm
- Inclusive Angle = 135°
- Radius = 3.7 mm
- Direction = up
- Set up the parameters for the flange sides according to the following figure:



8 Please hit the button **Apply** to create the flange.

9 **Settings for the 2nd flange**
Select the face and the left border edge (Length = 100 mm).

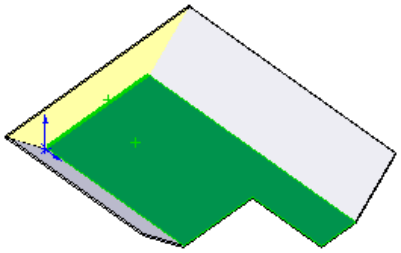
Keep the settings of the first Flange, do not make any modifications!



10 Please hit the button **Apply** to create the flange.

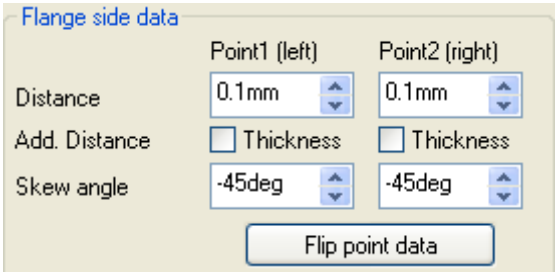
© SPI GmbH 2020

11 **Creation of the 3rd flange with skew angle on both sides.**
Please pick the face and the rear border edge (Length = 75 mm).



Settings for the 3rd flange
Please enter the following values:

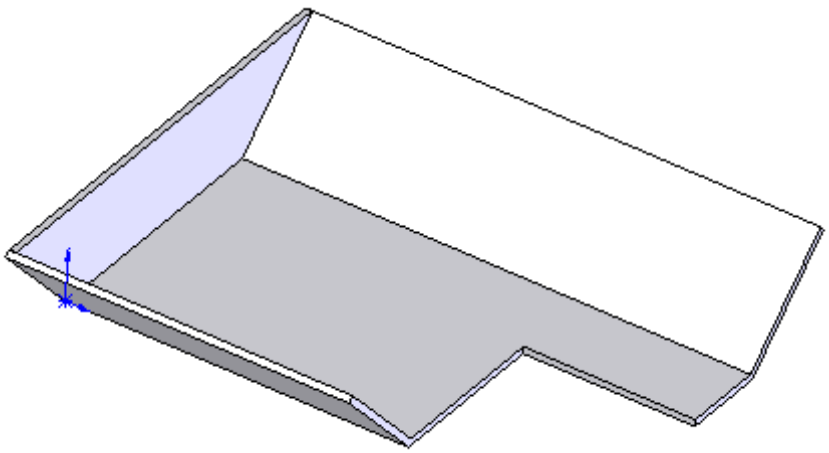
- Inner Height= 30 mm
- Inclusive Angle = 90°
- Radius = 1.6 mm
- Direction = up
- Set up the parameters for the flange sides according to the following figure:



12 Please hit the button **Apply** to create the flange.

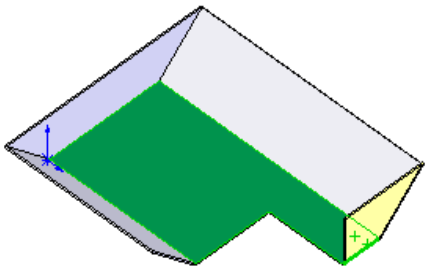
Intermediate result

Compare your result with the figure below:



© SPI GmbH 2020

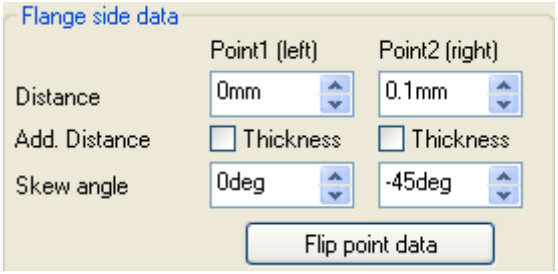
13 **Settings for the 4th flange with skew angle one side**



Pick the face and the small border edge (Length = 25 mm).

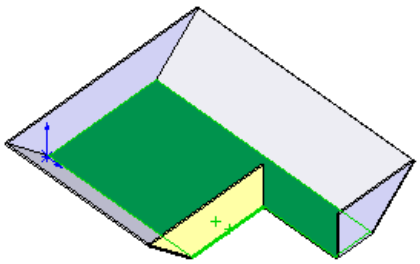
Please keep the settings of the prior flange. Only the settings for the flange side data are to be modified.

Set up the parameters for the flange sides according to the following figure:



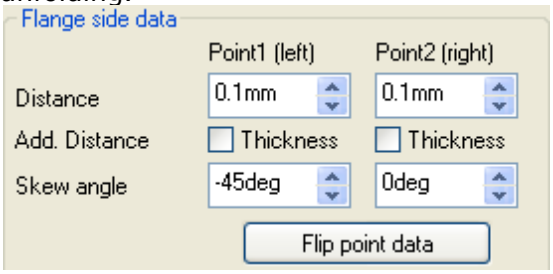
14 Please hit the button **Apply** to create the flange.

15 **The 5th flange**



Please pick the face and the next edge (Length = 50 mm).

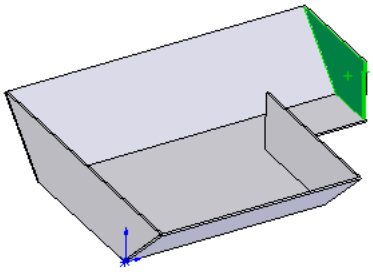
The settings are now as shown in the figure. A Distance > 0 at Point 2 is recommended The small Distance of 0.1 mm at Point 2 is necessary to add a kerf at the right flange side in the unfolding.



16 Please hit the button **Apply** to create the flange.

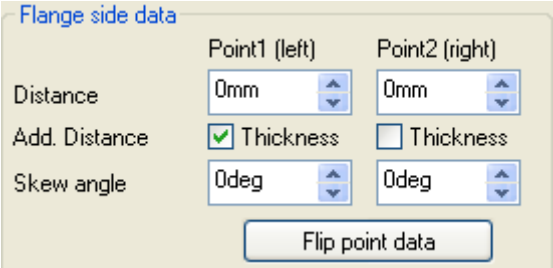
© SPI GmbH 2020

17 **Creation of the 6th flange without skew angle**



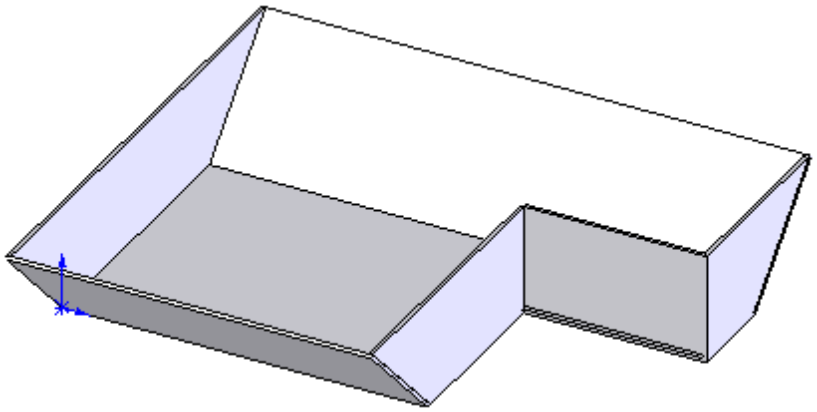
Pick the inner face and the vertical edge of the 4th flange.

- Inner Height= 50 mm
- Inclusive Angle = 90°
- Radius = 2 mm
- Direction = up
- Set up the parameters for the flange sides according to the following figure:



18 Please hit the button **Apply**, to create the flange.

Completed design Please compare your result with the figure below:



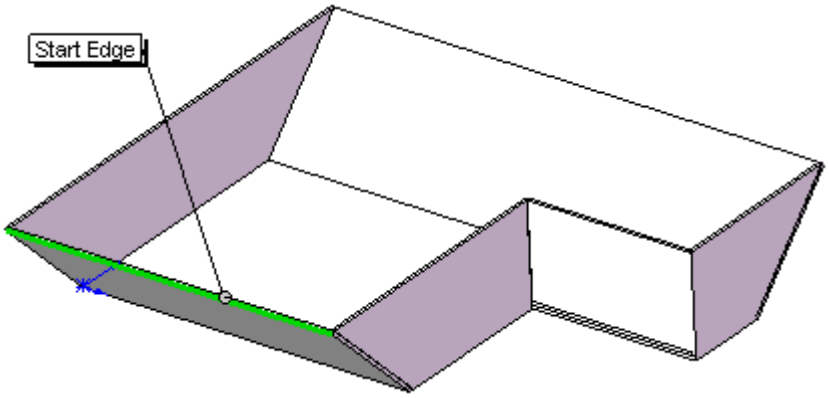
19 Now let's make an unfolding to verify the design.
Please use the command **Create Unfolding.**

20 **Settings for the Unfolding**
Please set the following values:

- Unfold a Body
- Unfold Destination = New Drawing

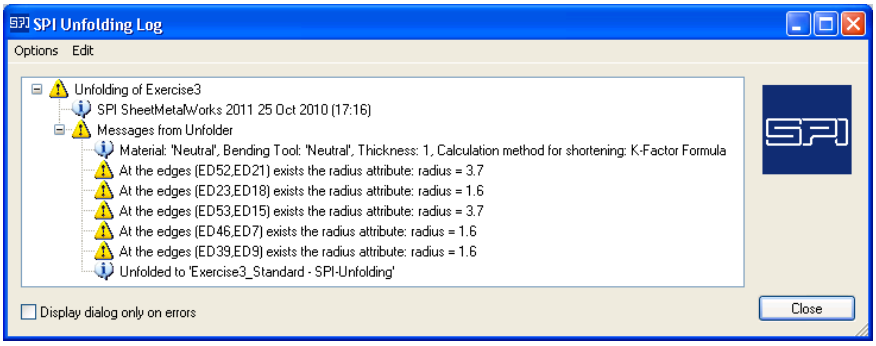
© SPI GmbH 2020

21 **Pick the Start Edge according to the following figure:**



22 Please hit the OK button to launch the calculation of the unfolding.

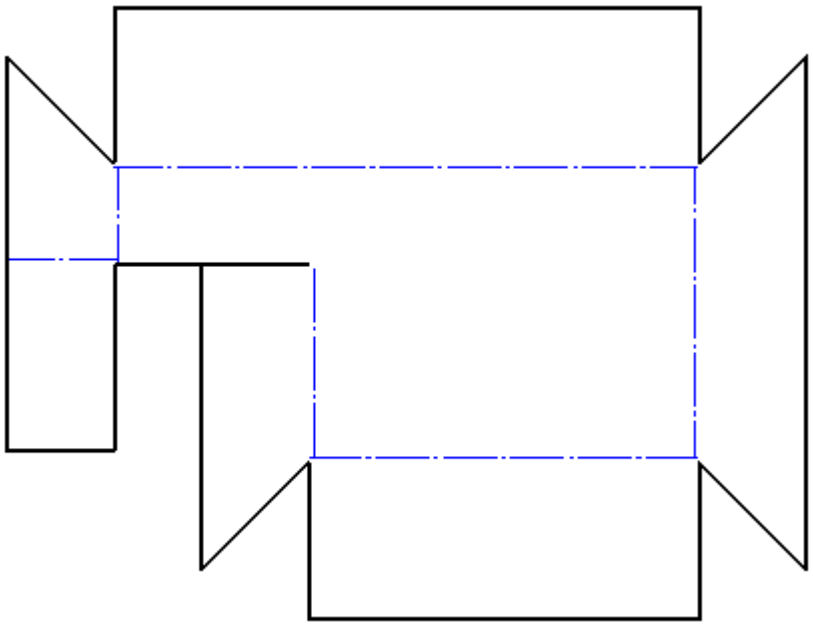
23 The SPI Unfolding Log is displayed:



24 Please close the SPI Unfolding Log.

Unfolding

SPI SheetMetalWorks has created a new drawing with the unfolding:



© SPI GmbH 2020

Design with SPI Sheet Metal Attributes

Corner Split Attributes

Corner Split Attributes can be assigned to sharp cornered flanges or edges that split a plane face. The calculation of the unfolding treats edges with a Corner Split Attribute as if it was geometrically ripped with a gap width.

Offset Attributes

Edges which represents border edges in the unfolding can be moved in the unfolding by attaching Offset Attributes.

Bend Radius Attributes

Bend Radius Attributes can be assigned to sharp cornered bends. The calculation of the unfolding will evaluate the Bend Radius Attributes for the calculation of the shortening instead of the standard bend radius from the SPI Unfold Parameters. This is helpful when a part is to be manufactured on two different machines.

Relief Attributes

Relief Attributes can be assigned to sharp cornered, half sharp cornered or filleted bends. The attribute allows using a relief that differs from the standard reliefs that are valid for the whole body. A relief attribute can be assigned to each side of a bend separately.

Bend Mark Attributes

Some methods of manufacturing need bend marks on the flat pattern for bending. The bend marks are often small notches in the outer contour that are used on the bend press to position the sheet metal on the bend die. **SPI SheetMetalWorks** can set automatically bend marks to all bend lines. Different types of marks are available: Notches or half circles in the outer contour as well as little circles or lines on the bend line.

Shortening Attributes

Shortening Attributes can be assigned to sharp cornered or filleted bends. The calculation of the unfolding will evaluate the Shortening Attributes for the calculation of the shortening instead of the standard value from the SPI Unfold Parameters. This is helpful when a part is to be manufactured on two different machines.

Facet Attributes

Facet Attributes can be assigned to **Freeform, cylindrical, toroidal, spherically and cone surfaces** of sheet metal bodies. The calculation of the unfolding will use this facet information of the attribute for the unfolding of the freeform surfaces instead of the general facet data that are defined in the SPI Unfold Parameters.

Manufacturing Information Attributes

In case of holes or formings it requires often that a text is displayed in the unfolding instead of the contour. For further information about this please refer to the **SPI SheetMetalWorks** Online-Help.

Sketch Attributes

You may attach attributes to unconsumed sketches to define the way of displaying them in an unfolding. . For further information about this please refer to the **SPI SheetMetalWorks** Online-Help.

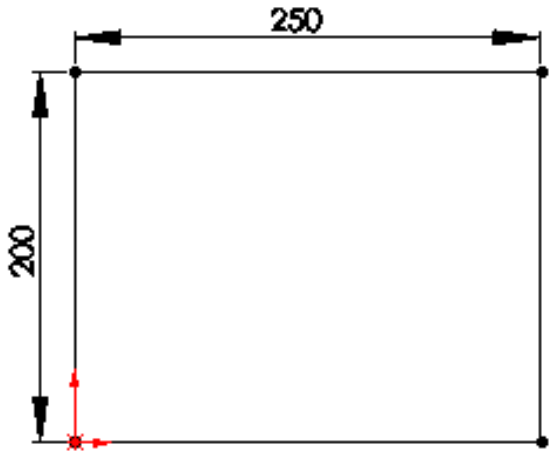
© SPI GmbH 2020

Exercise 4

This exercise is to explain the usage of the **SPI SheetMetalWorks** attributes.

1 Create a new part using **mm** as units. **Save** and name it `box.sldprt`.

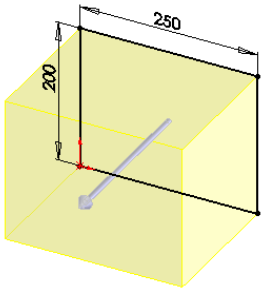
2 Make a sketch with the content displayed in the figure below on the **Front plane**. Alternatively, you can open the file `Exercise4.sldprt`.



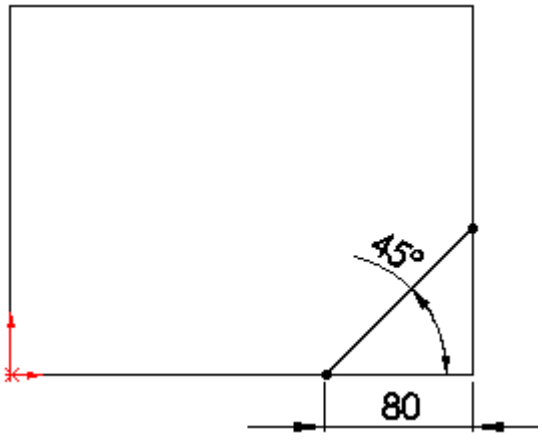
3 Ccreate a **linear extrusion**.

Enter the following values and settings:

- Type = Blind
- Length = 200 mm



4 Please sketch on the **Front plane** this contour:



The endpoints of the line should be coincident with the edges of the solid body.

© SPI GmbH 2020

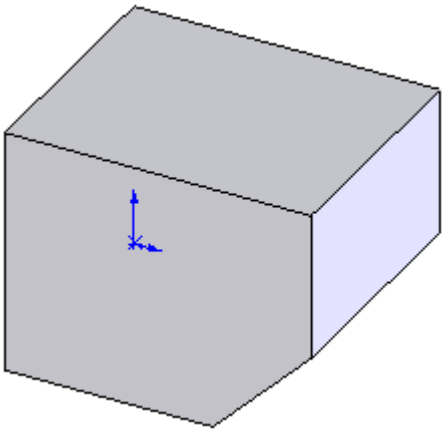
5 Please create a **linear cut**.

Set up the following values and settings:

- Direction1 = Activated
- End condition = Through all
- Flip side to cut= Activated
- Direction2 = Activated
- End Condition = Through all

Intermediate Result

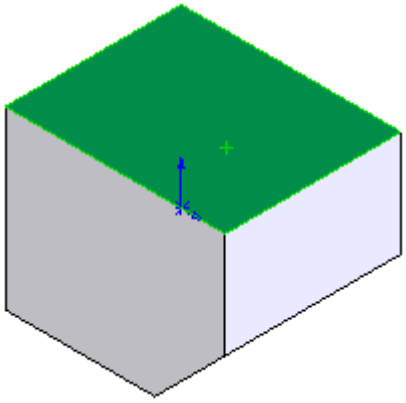
Please compare the result with the figure:



6 Create a **Shelling**.

Enter the following values and settings:

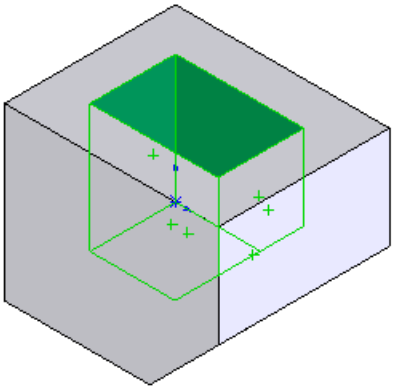
- Thickness = 50 mm
- Face to remove - see picture



7 Create another **Shelling.**

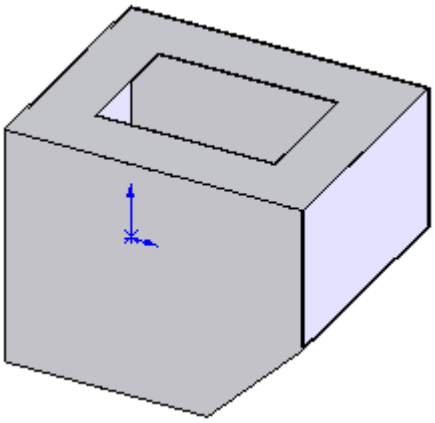
Enter the following values and settings.

- Thickness = 1 mm
- Faces to remove = all inner faces that were made in prior step 6 (see figure).



© SPI GmbH 2020

Preliminary result Please compare your result with the picture:

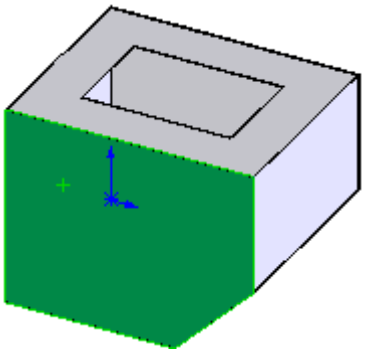


The shelling command in step 7 made a collar face. This needs to be split to allow unfolding. However, the next step is to assign SPI material data to the 3D part.

8 **Make a SPI sheet metal part from this 3D model.**
Please execute the SPI command **Sheet Metal Wizard**.

9 **Select one planar face.**

The selected face is listed in the **Selection** within the Property Manager of the **SPI Sheet Metal Wizard**.

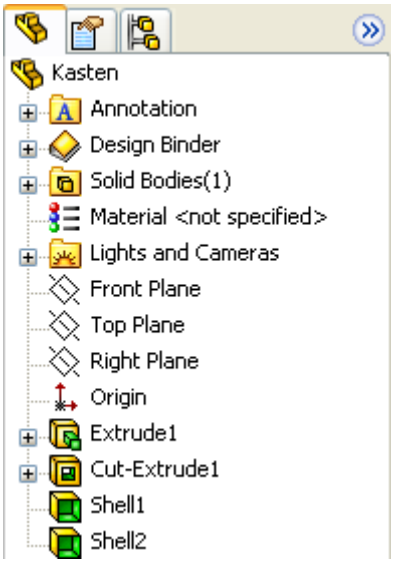


10 **SPI Sheet Metal Wizard's settings.**
Continue with the next page of the Wizard by clicking on the rightward arrow button.
Please assign the following values:

- Material = Neutral
- Bending Tool = Neutral
- Correction Method = K-Factor
- Bend Radius = 2.00 mm

© SPI GmbH 2020

11 Click **OK** to define a SPI sheet metal part by assigning the SPI material data. The feature tree looks as follows:

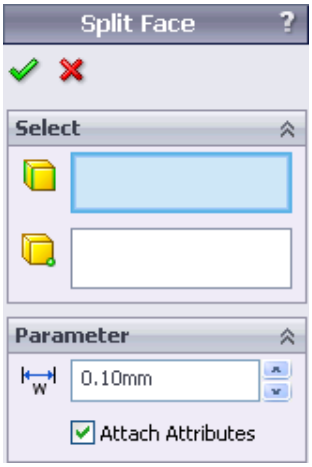


12 **Split collar face**
Please use the command **Split face**

© SPI GmbH 2020

Split Face

In a sharp edged model, it might be necessary to define split lines on a plane surface. This can happen in parts with profiles containing perpendicular lines. Parts like this can have so called "collar faces". The command **Split Face** allows defining the split lines and assigns Corner Split Attributes in a single step.



Selection Edge

It is easy to split a complete face by selecting two opposite edges. **SPI SheetMetalWorks** analyzes the face, finds the edges and generates a preview where the face will be separated.

Selection Vertices

You have more control on the **Split Face** command by the use of the vertex option. The software will create a line between the two selected vertexes. Please notice, that the face can only be processed if the split line will split the face into two complete separated faces.

Parameters

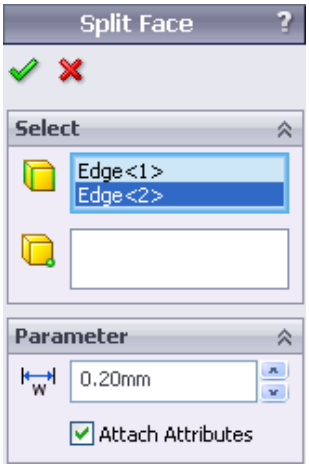
The parameter **Spacing** controls the width of the virtual gap between the two flanges.

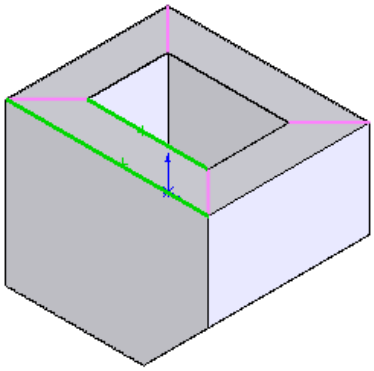
The option **Attach Attributes** sets up, if a **Corner Split Attribute** is to be assigned to the split line (recommended).

© SPI GmbH 2020

13 **Settings for Split Face**
Please enter the following values and settings:

- Spacing = 0.2 mm
- Attach Attributes = Activated
- Selection of two edges according to the following figure:

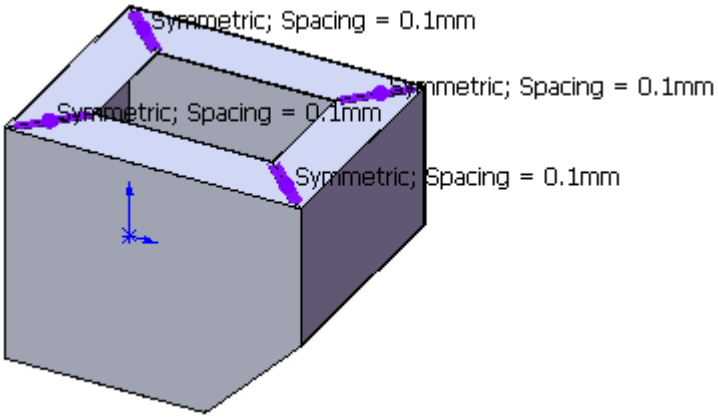




14 Please click **OK** to split the collar face.

15 Please use the command **Corner Split Attributes**.

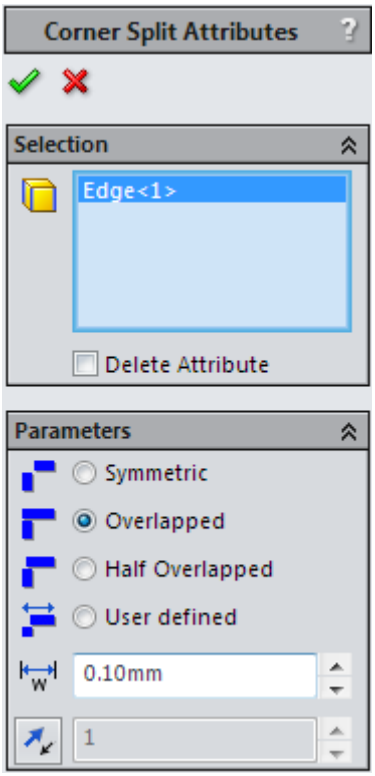
The part looks like in the picture below:



© SPI GmbH 2020

Corner Split Attributes

When launching the command **Corner Split Attributes**, all already existing Corner Split Attributes are displayed.



Selection

To assign attributes or to delete existing attributes you have to select the referred edges. The new and the existing attributes are displayed in different colors.
Please check the option **Delete Attribute** with this command to erase an attribute.

Parameters

The command offers different types for the edge situation:

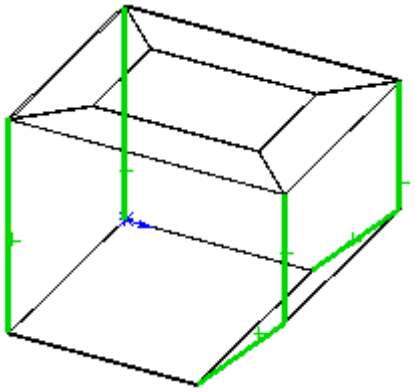
- Symmetric
- Overlapped
- Half Overlapped
- User Defined

The preview displays the edge situation that will be assigned.

The parameter **Spacing** is responsible for the width of the gap that will be calculated. Please note that the gap is not visible in the 3D model.

© SPI GmbH 2020

16 **Please select the six edges according to the following picture:**



17 **Settings for the Corner Split Attributes.**
Please set up the following values and settings:

- Corner Type = Symmetric
- Spacing = 0.2mm

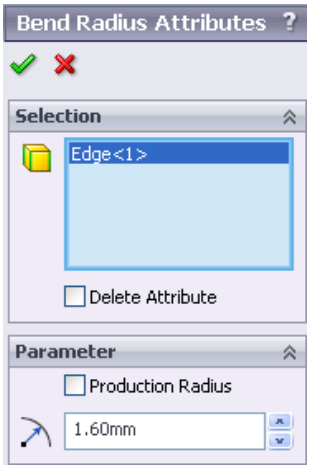
18 Hit the OK button to assign the **Corner Split Attributes**.

19 Execute the command **Bend Radius Attributes**.

© SPI GmbH 2020

Bend Radius

When running the command **Bend Radius Attributes**, all already existing Bend Radius Attributes are displayed.



Selection

You have to select the edges where attributes are to be assigned, modified or deleted. The preview for new attributes is shown in a different color.
When finishing the command all selected edges are equipped with the values that are displayed in the dialog.

Bend Radius

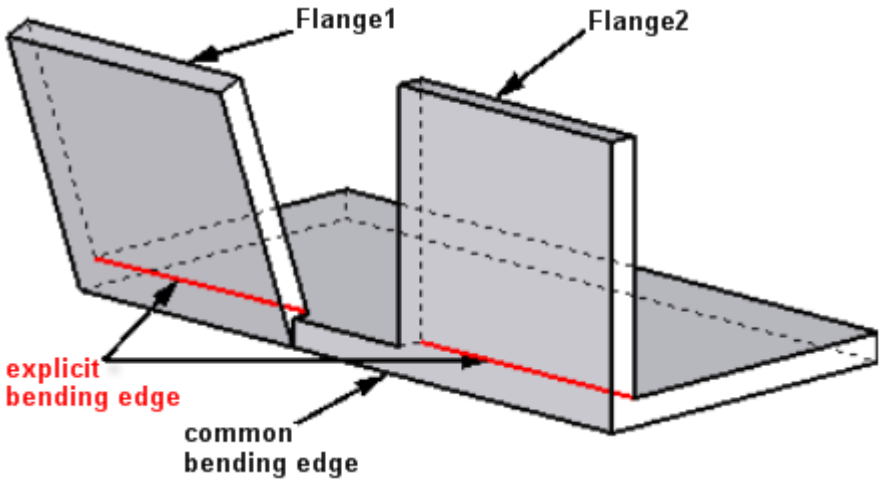
In the dialog element **Bend Radius** you can enter a value for the bend radius to be used.

Production Radius

If the option **Production Radius** is activated **SPI SheetMetalWorks** will determine the bend radius from the SPI material database.

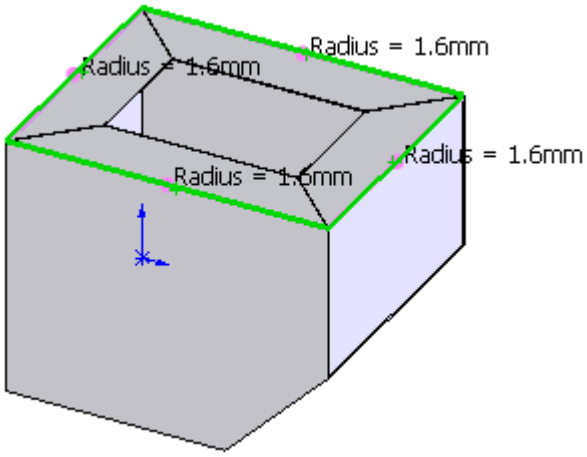
Caution!

It is important to select an edge that identifies the bend zone unambiguously. Please do not use a common edge. In cases like displayed below the inner edge is preferable.



© SPI GmbH 2020

20 Select the upper, outer 4 edges according to the following figure:



21 **Settings for the Bend Radius Attributes**
Set up the following values and settings:

- Radius = 1.6 mm

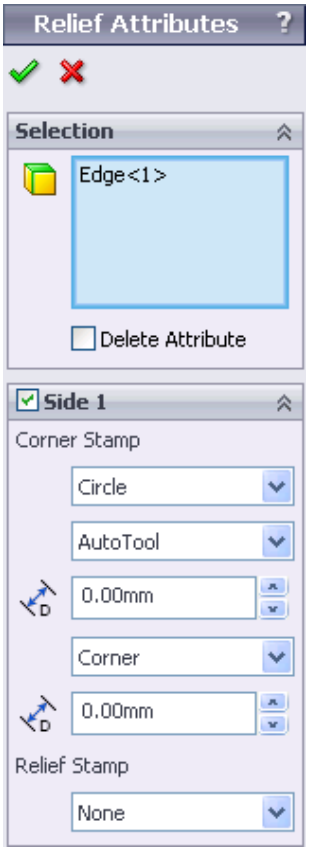
22 Hit the **OK** button to assign the **Bend Radius Attributes**.

23 Execute the command **Relief Attributes**.

© SPI GmbH 2020

Relief

When running the command **Relief Attributes**, all already existing Relief Attributes are displayed.



Selection

To assign new attributes or to delete or modify existing ones the referred edges must be selected. New and existing attributes are displayed in different colors. When running the command, all selected edges get the same values and settings.

Side 1/Side 2

The groups of options **Side 1** and **Side 2** allow setting up the parameters for reliefs for each end of the bend zone separately. If the option is activated, an attribute is set that overrides the general settings for the part. The software differentiates between Corner Stamps and Relief Stamps.

Corner Stamp

These types of corner stamps are available:

- Circle
- Square
- Diag
- Oblong
- Rhomb
- Generic
- (Laser Ed)
- (Laser Sym)
- Laser Lin
- Laser Flex
- Smooth
- Corner Blend

The types **Laser Ed** and **Laser Sym** are outdated and were improved by the new Type **Laser Flex**.
For some types of stamps, the size of the relief can be set.

Relief Stamp

There are 4 types of relief stamps:

- Rectangle
- Oblong
- Circle
- Smooth

Dialog

Depending on the tool type the dialog allows entering different values and settings.

© SPI GmbH 2020

Tool selection

For some types of corner stamp tools, you can set up how the size of the stamp has to be calculated. You have the choice between three options:

- Tool
- Auto
- AutoTool

Tool

When the option **Tool** is selected, a new button **Tool** will be displayed. Press this button to select a particular tool from the SPI tool database or enter the values for the size manually. Depending on the shape of the stamp the size values are to control:

- Diameter
- Length
- Width
- Reference Type/Displacement

Auto

The setting **Auto** calculates automatically the necessary dimensions for the selected relief type. You can improve the result by entering an offset for the size.

AutoTool

The setting **AutoTool** selects the smallest tool from the tool database that is bigger than the necessary dimensions. An extra offset can be added.

Offset

When calculating the necessary size for a relief, the size will be increased by the offset value.

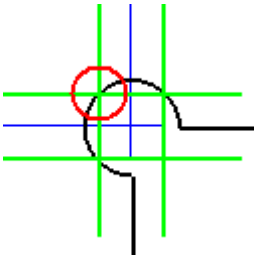
Reference Type

The **SPI SheetMetalWorks** unfolding engine uses the reference type to place the center point for the relief stamp. Two settings for the reference type are available:

- Corner
- Middle

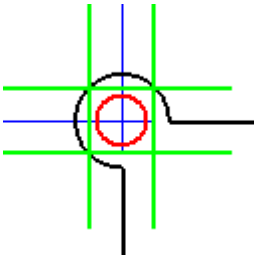
Corner

The **Reference Type Corner** refers to the intersection point of the inner borders of the bend zone.



Middle

The **Reference Type Middle** refers to the intersection point of the bend lines.

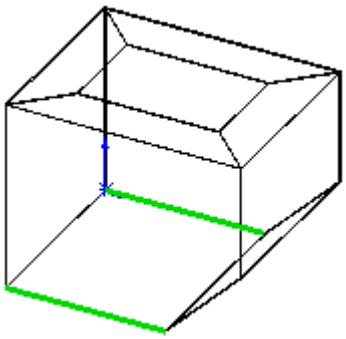


Displacement

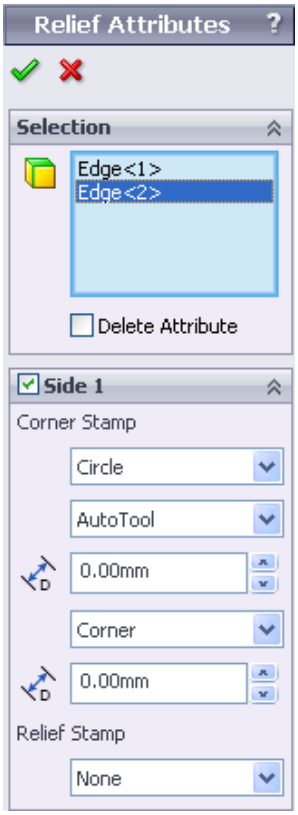
The displacement value allows shifting the center point about a dedicated value.

© SPI GmbH 2020

24 **Pick the lower 2 edges according to the following figure:**



25 **Settings for Relief Attributes.**
Enter the following values and settings:



- **Side 1** = Activated
- Corner Stamp = Circle
- Tool Selection = AutoTool
- Offset = 0.00mm
- Reference Type = Corner
- Displacement = 0.00mm
- Relief Stamp = None

- **Side 2** = Activated
- Corner Stamp = Circle
- Tool Selection = AutoTool
- Offset = 0.00mm
- Reference Type = Corner
- Displacement = 0.00mm
- Relief Stamp = None

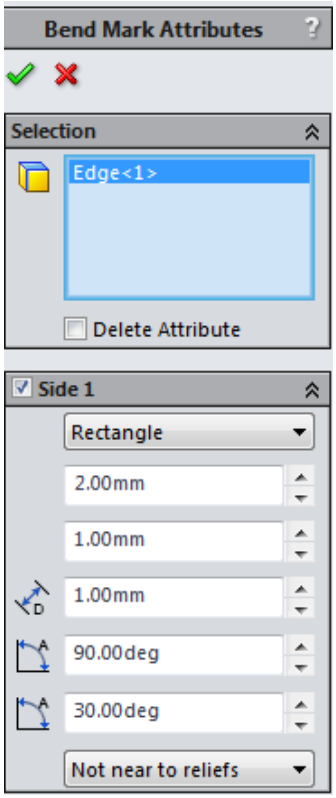
26 Hit the **OK** button to assign the **Relief Attributes**.

27 Use the command **Bend Mark Attributes**.

© SPI GmbH 2020

Bend Marks

When running the command **Bend Mark Attributes** all Bend Mark Attributes are displayed.



Selection

To attach new attributes or to delete or modify the existing ones, you have to select the referred edges from the 3D part. The new attributes are displayed in a different color. When finishing the command all selected edges are equipped with the currently selected values.

Side 1/Side 2

The groups of options **Side 1** and **Side 2** allow setting up the parameters for each end of the bend zone separately. If the option is activated, an attribute is set, that overrides the general settings for the part.

Type of Bend marks

There are different types of bend marks. You can choose from these options:

Notch in outer contour:

- Rectangle
- Parallel
- Circle
- Rhomb

Engraving options:

- Etch Line
- Etch Circle
- Center Mark
- Centerline

See also chapter **Unfold Parameters / Bend Marks** (p. 103).

Parameters

Depending on the selected bend mark type you can enter the following values:

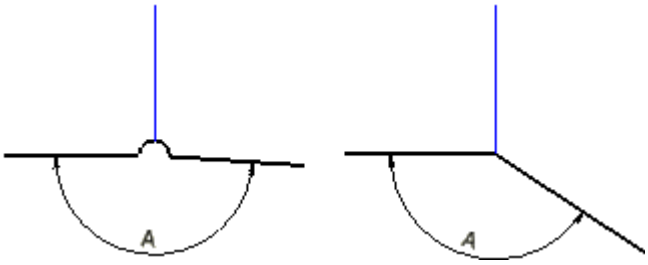
- Depth
- Width
- Distance

Caution! The setting for **Distance** overrides the value for **Distance to Contour**, which is set in the general **Unfold Parameters Post Processing**.

© SPI GmbH 2020

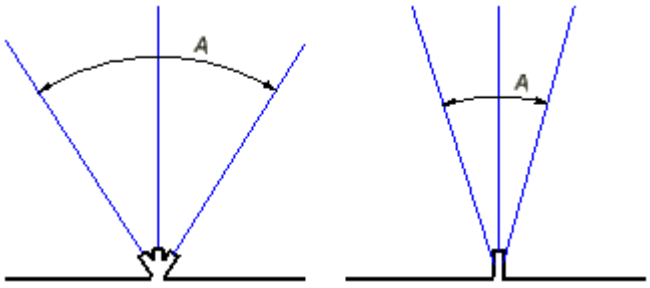
Border Angle

Often it is not necessary to put a bend mark at those places, where the bend line meets the outer contour in a position where the contour makes a sharp bend (kink). This kink is often a good guide that shows the craftsmen where to bend. The Border Angle sets up a threshold that has to be passed to make the SPI unfolding engine insert a bend mark. The two images in the following figure show the two cases. On the left the border angle is bigger or equal to the threshold angle, thus a bend mark is created. The other image shows the situation that a bend mark is not necessary.



Simplify Angle

In cases when multiple bend lines meet each other at the contour it might be sensible to join the related bend marks from them. The Simplify Angle sets up the threshold that makes the unfolding engine join the bend marks to a single one. When two or more bend lines meet at the contour, the angle between the two outmost lines is compared with the threshold angle. If this angle is smaller than the chosen Simplify Angle only one bend mark is created. The difference is shown in the following figure.

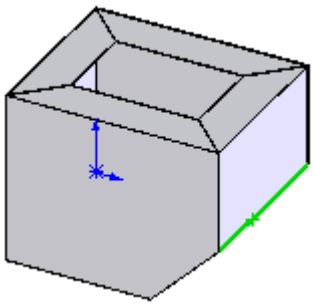


Placing Rule

The Placing Rule controls a condition for the creation of bend marks. The value **Always** makes the unfolding engine produce the bend marks in all cases that are not overruled by a particular condition. The rule **Not near to Reliefs** suppresses the bend marks at positions that are close to a corner stamp or bend relief.

© SPI GmbH 2020

28 **Pick the front edge according to the following figure.**



29 **Settings for Bend Mark Attributes.**
Enter the following values and settings:

- **Side 1** = Activated
- Type of Bend Mark = Rectangle
- Depth = 2 mm
- Width = 1 mm
- Distance = 1 mm
- Border Angle = 90°
- Simplify Angle = 30°
- Placing Rule = Not near to Reliefs
-
- **Side 2** = Activated
- Type of Bend Mark = Rectangle
- Depth = 2 mm
- Width = 1 mm
- Distance = 1 mm
- Border Angle = 90°
- Simplify Angle = 30°
- Placing Rule = Not near to Reliefs

30 Please hit the **OK** button to assign the **Bend Mark Attributes**.

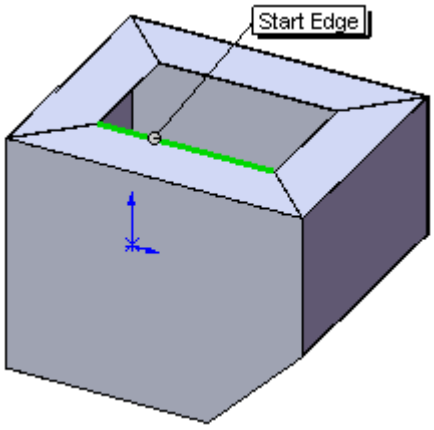
31 Verify your design with an unfolding.
Please use the command Create Unfolding

© SPI GmbH 2020

32 Settings for **Create Unfolding.**
Enter the following values and settings:

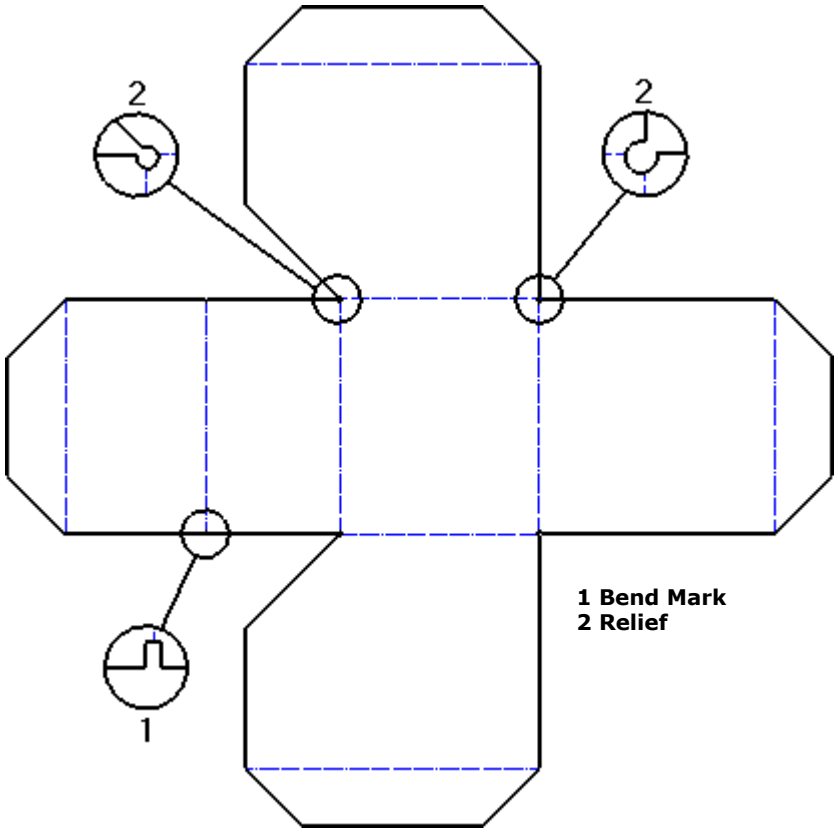
- Unfold a Body
- Unfold Destination = New Drawing

33 **Select a Start edge according to the following figure.**



34 Please hit the **OK** button to launch the calculation of the unfolding.

35 Close the **SPI- Unfolding Log**. Compare your result with the following unfold drawing.

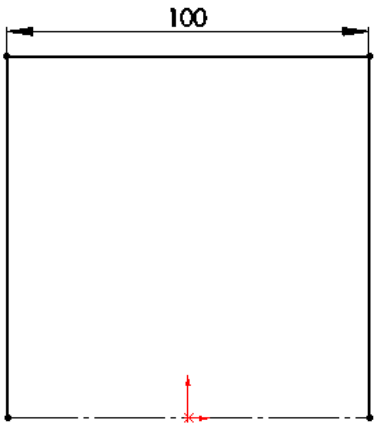


© SPI GmbH 2020

Exercise 5

1 Please create a new part using **mm** as units. **Save** and name it `profile.sldprt`.

2 Please make a sketch with the content displayed in the figure below on the **Front plane**. Alternatively, you can open the file `Exercise5.sldprt`.

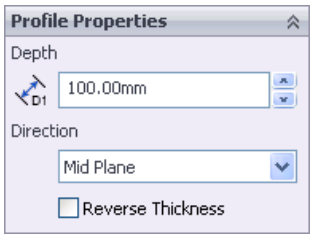


3 Now select this contour to create a SPI sheet metal part.
For this please start **Profile – new**.

Profile – New

The Property Manager **Profile – New** is opened, asking you to enter **Profile Properties**, **Material Data** and **Sheet Metal Properties**.
Please take notice, that the offered bend tools are related to the material and thickness. The material data and correction methods can be maintained in the **SPI Material Editor**.

Profile Properties



The **Profile Properties** control the creation of the 3D solid body. The **Depth** value it the length of the profile.
There are three options that control the side to which the sketch elements are thickened:

- Normal
- Reverse
- Mid Plane

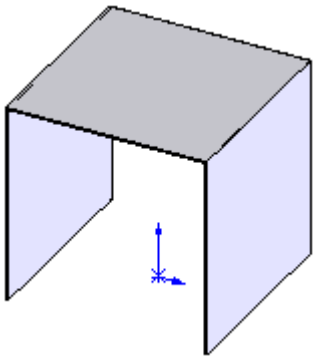
You can flip the side with the option button **Reverse Thickness**.

Material Data and Sheet Metal Properties

To have a sharp cornered model, activate the option **Sharp cornered**. Otherwise the sheet metal profile is filleted with the default radius.
The settings for material data and sheet metal properties are identical with the creation of a base plate (see **Base Sheet-New p. 34**).

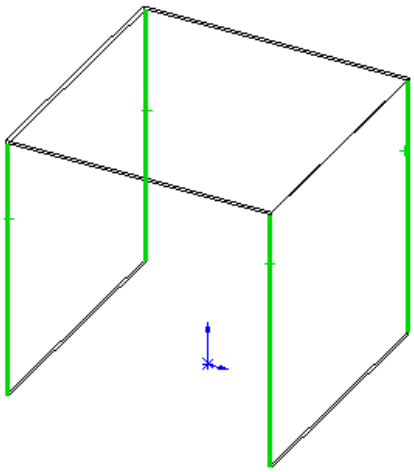
© SPI GmbH 2020

4 Settings for the command Profile – New.
Set up the following values and settings:



- Depth = 100 mm
- Direction = Mid plane
- Reverse Thickness = Deactivated
- Thickness = 1 mm
- Material = Neutral
- Piston Tool = Neutral
- Die Tool = Neutral
- Sharp cornered = Activated
- Correction Method = K-Factor
- Bend Radius = 2.0 mm

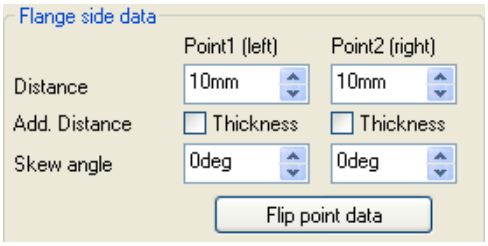
5 Create some Sharp Cornered Flanges according to the following figure, at the inside edges of the profile.



Please use the command **Sharp Cornered Flange**

Enter the following values.

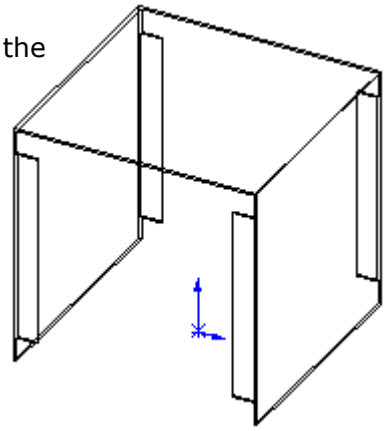
- Length = 10 mm
- Inclusive Angle = 90°
- Radius = 2.0 mm
- Direction = up
- Enter the values for the flange side data according to the following figure



Intermediate Result 1

Please compare the result with the figure:

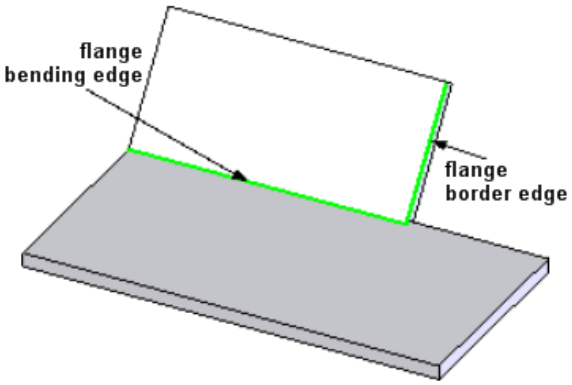
The display has been toggled to wireframe mode to improve the printed picture.



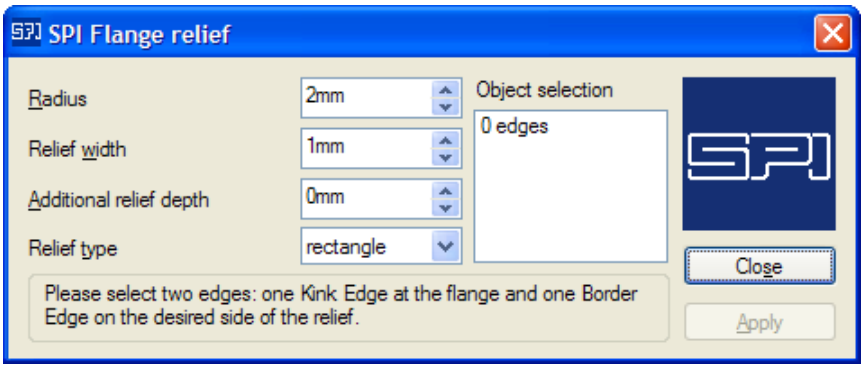
© SPI GmbH 2020

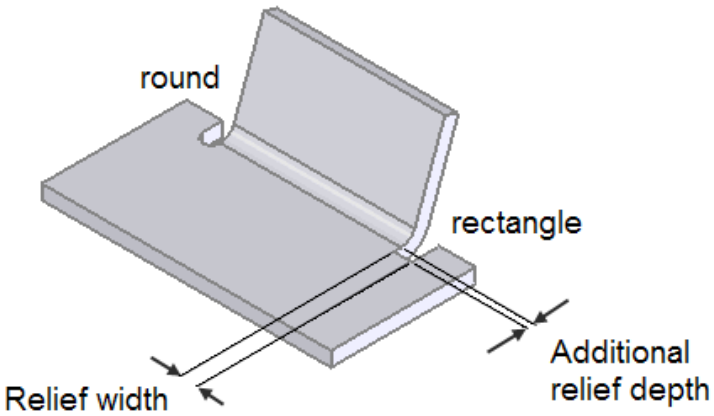
Flange Relief

There are two ways to create a flange relief. You can use the command **Flange Relief** in order to model a relief on a sharp cornered flange directly at the model. The second method is to assign a **Relief Attribute**. Such a relief is visible only in an unfolding created by the SPI Unfolder. After inserting a physical relief to a sharp cornered flange with the command **Flange Relief** it is possible to round the bends of the flanges. To apply the command, the bending edge and the referred border edge must be selected. Hold down the CTRL-Key while picking the two edges.



The dialog of the SPI Flange Relief contains the parameters **Radius**, **Relief width**, **Additional relief depth** and **Relief type**.





© SPI GmbH 2020

Radius When a bend radius attribute is already assigned to the bend edge, this radius will be used. Otherwise you'll have to enter a value. The numeric value defines the radius on the inner side of the bend zone

Relief width This value specifies the width of the relief.

Additional relief depth The depth of the relief is calculated according to the bend radius. It ends at the edge of the bend zone. To create a deeper relief an additional offset can be entered.

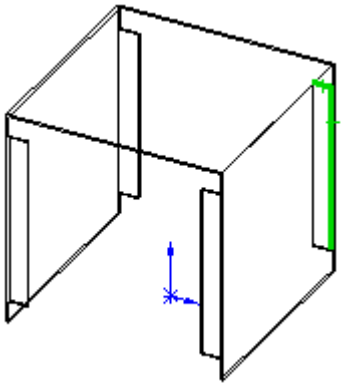
Relief type You have the choice between rectangular or round (oblong) relief.

Caution! In order to round the bend zone, you have to use a smaller radius than the total depth of the relief!

---

6 Launch the **Flange Relief** command.

7 Select the inner edge and a border edge of a flange as shown.



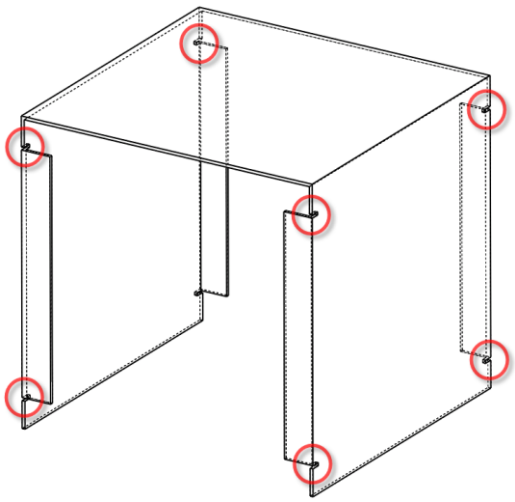
8 Enter the following values in the dialog **SPI Flange Relief**:

- Radius = this option is disabled due to the Bend Radius Attribute at the edge.
- Relief width = 1 mm
- Additional Relief Depth 0 mm
- Relief Type = rectangle

9 Click **Apply** to create the flange relief.

© SPI GmbH 2020

10 Insert the rest of the flanges by using the same procedure as in step 7 and with the settings from step 8. Totally there should be eight reliefs in the model:



11 Close the dialog **SPI Flange relief**.

12 Now use the SPI command **Fillet** in order to round the bends.

Fillet



One or multiple edges of sharp cornered flanges can be fillet if they are relieved in the model (a relief attribute is not sufficient).

Please select an edge of the sharp cornered bend zone.

The dialog of the command **Fillet** shows the **bend radius** that is set up in the general parameters for the sheet metal part. If a dedicated bend radius attribute is assigned to the edge, that value is displayed.

Click the button to fillet the bend zone.

To modify a fillet select a face of a bend zone that is already rounded, enter a different value for radius and click on the button.

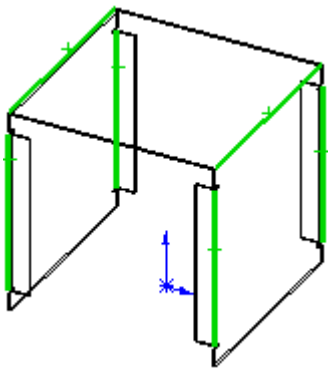
Use to delete the fillet feature.

Please pay attention that the flange contains reliefs with necessary depth for the desired bend radius before you create a Fillet.

The button ends the command.

© SPI GmbH 2020

13 Select according to the following figure the outside edges of the four flanges and the two outside edges of the profile. (Hold down CTRL).



14 Settings for the command **Fillet**:

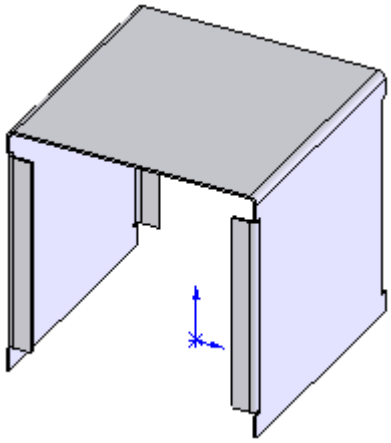
- Radius = 2.0mm

15 Please click the button **Fillet** to create the fillet features.

16 End the Fillet command with **End**.

Intermediate Result 2

Please compare the result with the figure:

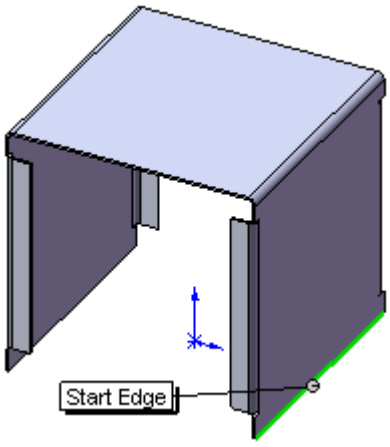


17 Verify your design with an unfolding.
Please use the command **Create Unfolding**

18 Set up the following values for the command:

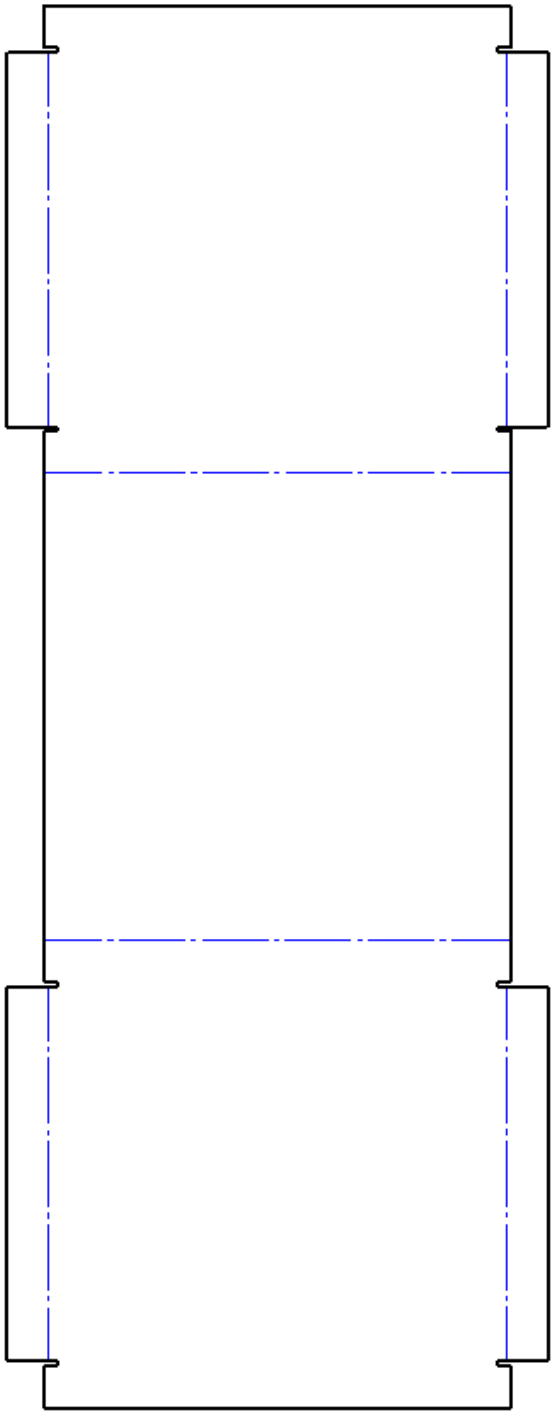
- Unfold a Body
- Unfold Destination = New Drawing.

19 Pick a start edge according to the figure.



© SPI GmbH 2020

20 Please hit the **OK** button to launch the unfolding calculation. Please compare the result with the figure below:



21 Switch back to the 3D sheet metal part.

22 Launch the command **Shortening Attributes**.

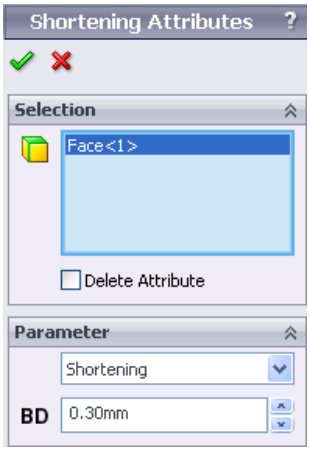
© SPI GmbH 2020

Shortening Attributes

When running the command **Shortening Attributes** all already assigned shortening attributes are displayed.

Selection

To attach new attributes or to delete or modify the existing ones, you have to select the referred edges or bend zone faces from the 3D model. The new attributes are displayed in a different color than the existing ones. When finishing the command all selected edges are equipped with the currently selected values.



Parameters

You can enter the method for the calculation and the referred value. The Shortening Attribute allows the following types to define calculation of the bend zone:

- Shortening
- Bend Zone Width
- K-Factor
- Bending tool
- Correction Method

Shortening

Define an explicit shortening value for the bend zone. .

Bend Zone Width

This will define exactly the width of the unfolded bend zone.

K-Factor

Let the SPI unfolding engine calculating this bend zone according to the k-Factor method and the assigned K-Factor value.

Bending tool

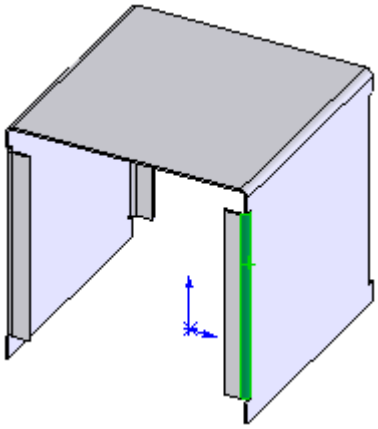
Assign a different bending tool for this bending. This reflects the workshop reality when a part is manufactured on two different machines or with two different sets of tools.

Correction Method

The option Correction Method allows accessing the SPI material database and assigning a different behavior for this bend zone.

---

23 Select a bend zone face according to the figure.



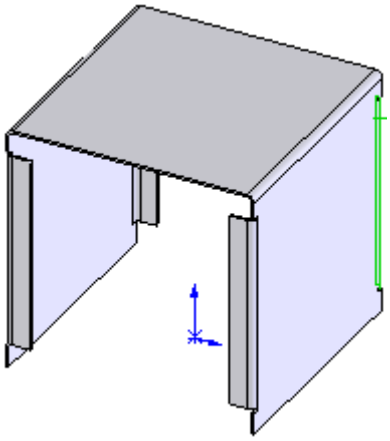
© SPI GmbH 2020

24 Set up the following values and settings.

- Parameter = Shortening
- Shortening = 1.0 mm

25 Please hit **OK** in order to assign the **Shortening Attributes**.

26 Select a bend zone face according to the figure.



27 Set up the following values and settings.

- Parameter = Bend Zone Width
- Bend Zone Width = 0.5mm

28 Please hit **OK** in order to assign the **Shortening Attributes**.

29 Launch **Update Unfolding**.

Update Unfolding

If a sheet metal body has been unfolded successfully before, the unfolding can be update directly using the command

Update Unfolding.

The command can be used from the 3D model to update a drawing or can be used in the context of a drawing document.

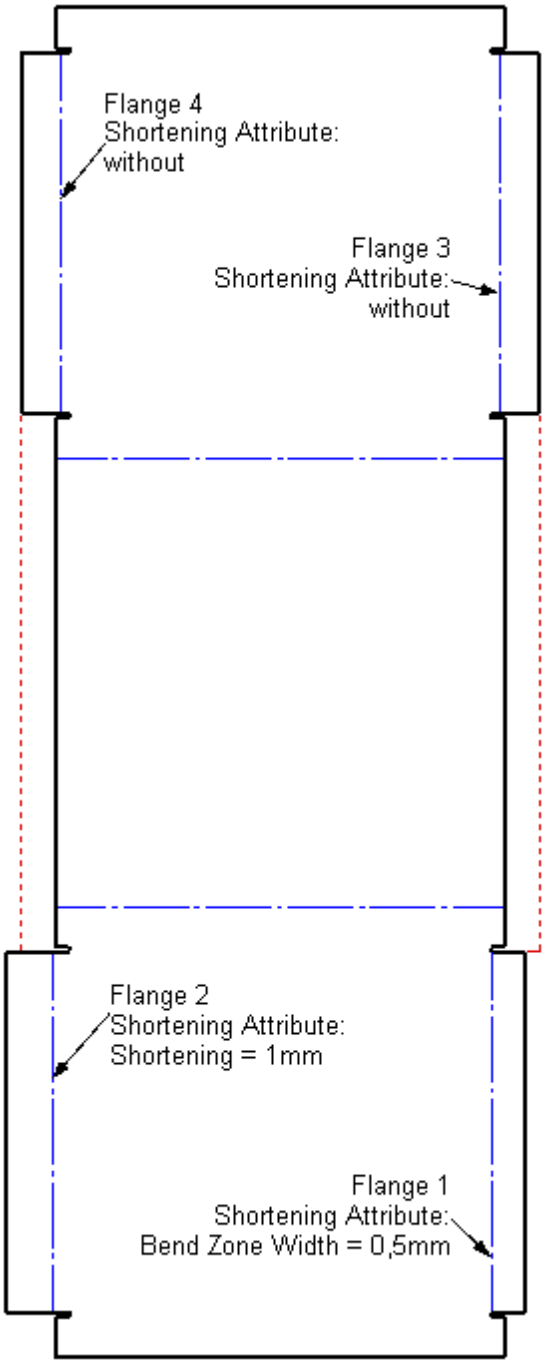
The update will not only modify the drawing document but also the 3D model document. For example, the updated values of the cost module are stored in user defined properties. Therefore, SOLIDWORKS will prompt you to save the file. It is recommended to do so.

© SPI GmbH 2020

End result

Compare your result with the figure below:

The description and the stitched lines are added manually to improve the display.



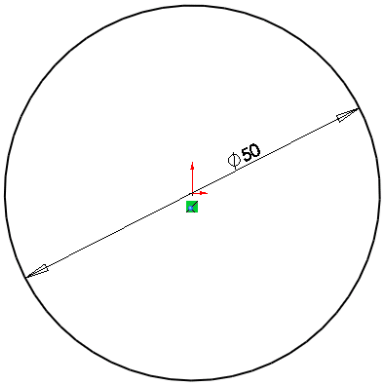
© SPI GmbH 2020

Exercise 6

This exercise shows how to create a typical *round-to-square transition* and how to set up the parameters for the unfolding.

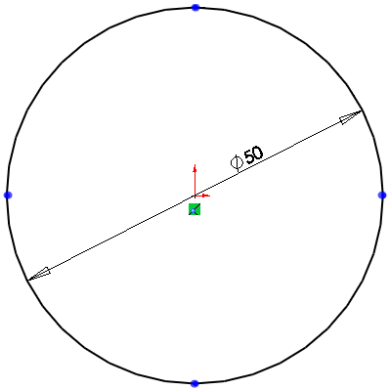
1 Please create a new part using **mm** as units. **Save** and name it `facetted.sldprt`.

2 Create a sketch as shown in the figure on the **Top plane.** Alternatively, you can open the file `Exercise6.sldprt`.



3 Use the sketch command **Split Element** to split the circle into four circular segments as shown.

4 Add **Sketch Relations:** Select the horizontal points and the origin (hold down CTRL) and in the Property Manager add a **horizontal Relation**. Between the vertical points and the origin add a **vertical Relation**.

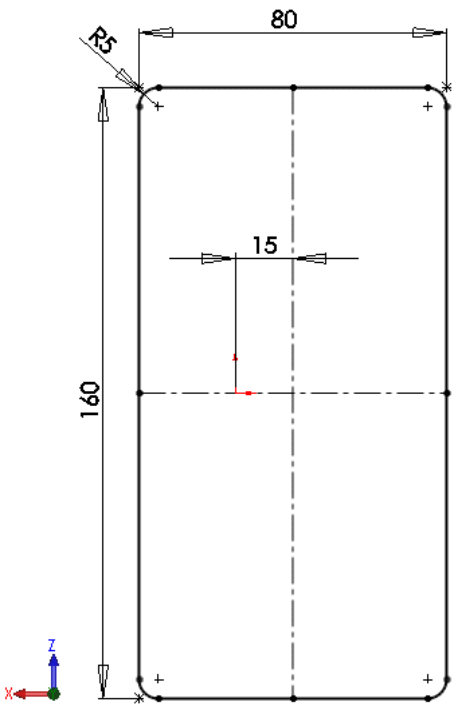


5 **Close** the sketch.

6 Create a new **Plane**, parallel to the **Top plane** with a distance of **150 mm**.

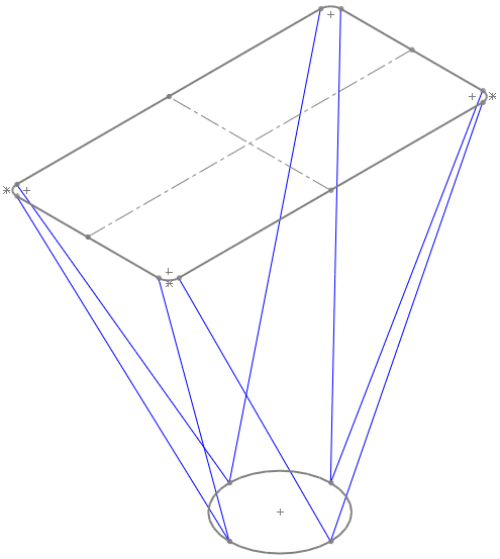
7 Draft the following contour on the new plane. Be sure that you add all sketch relations to make it tangential and **Fully Defined**.

8 **Close** the sketch.



© SPI GmbH 2020

9



Use the command **Curve Trough Reference Points** to create the guide curves as shown.

10 Make a **Lofted Boss / Base**.
Set up the following parameters:

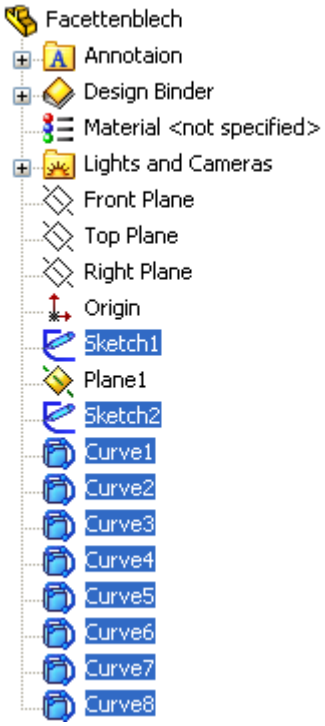
- Profiles:

  Sketch1
  Sketch2

- Start Constraint = None
- End Constraint = None
- Guide curves:

  Curve1
  Curve2
  Curve3
  Curve4
  Curve5
  Curve6
  Curve7
  Curve8

- Merge tangent faces = Deactivated
- Close loft = Deactivated
- Show preview= Activated

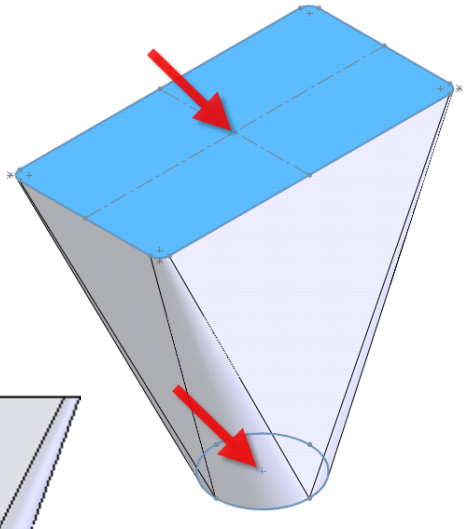


11 Please hit the **OK** button to create the solid body.

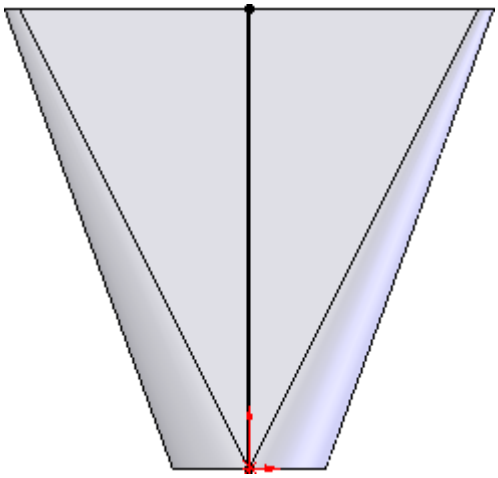
© SPI GmbH 2020

12 Make a **Shell**.
Set up the following parameters:

- Thickness = 1mm
- Faces to remove = see picture



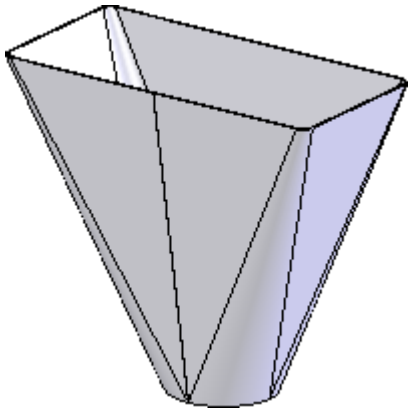
13



Make a new sketch on the **Plane right** according to the figure.

14 Please create an **Extruded Cut**.
Set up the following parameters:

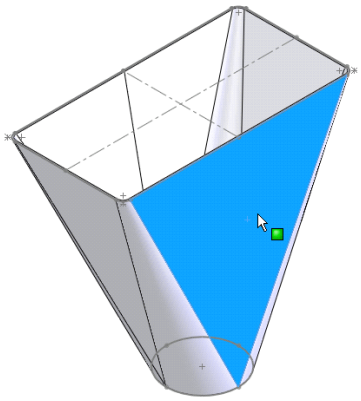
- Direction1 = Activated
- End Condition = Through all
- Flip side to cut= Deactivated
- Direction2 = Deactivated
- Thin Feature = Activated
- Type = Mid Plane
- Thickness = 0.1mm
- Auto-fillet corners = Deactivated



15 In Order to make a SPI sheet metal part from the solid body use the command **Sheet Metal Wizard**.

© SPI GmbH 2020

16 In the Property Manager activate the **Selection** area and pick a side face of the part.



17 Continue with the second page of the wizard by click on the right arrow button and set the parameters:

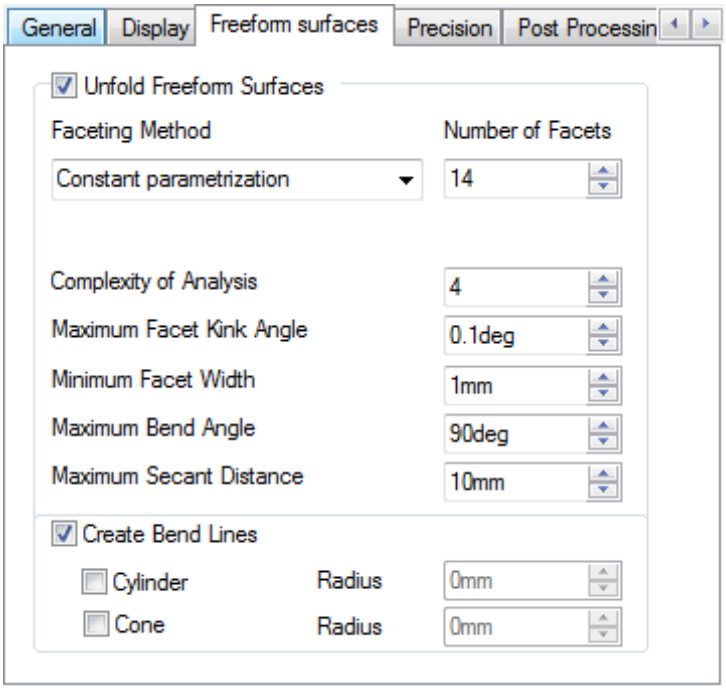
- Material = Neutral
- Bending Tool = Neutral
- Correction Method = K-Factor
- Bend Radius = 2.00 mm

18 Please click on **OK**, to assign the material data.

19 Launch the command **Unfold Parameters**.

20 Go to the tab **Freeform surfaces** and enter the following values:

- Unfold Freeform Surfaces= Activated
- Faceting Method = Constant parametrization
- Number of Facets = 14
- Create Bend Lines = Activated
- Cylinder = Deactivated
- Cone = Deactivated



21 Execute the command **Facet Attributes**.

© SPI GmbH 2020

Facet Attributes

When running the command **Facet Attributes**, all already assigned **Facet Attributes** are labeled by a callout.

Selection

To attach new attributes or to delete or modify the existing ones, you have to select the corresponding faces from the 3D model.

Parameters

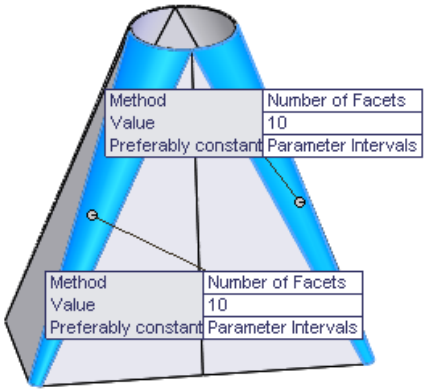
You can choose between different methods for the calculation of the facets. More detailed information you will find in the Online-Help of **SPI SheetMetalWorks**.

The facet attribute is only evaluated if the Unfold Parameter **Unfold Freeform surfaces** on the tab **Freeform surfaces** is activated. Facet attributes attached to cylindrical or conical faces will only be evaluated if the option **Create Bend Lines** is activated too.

22 Select the two conical faces as shown.

23 Set the following parameters:

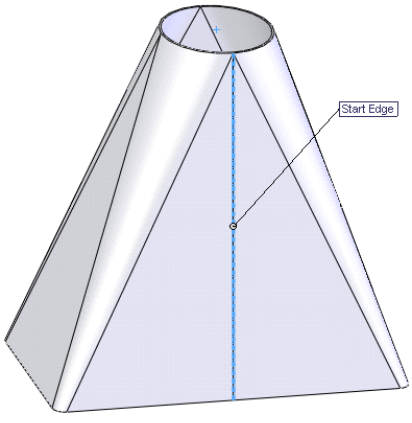
- Method = Number of Facets, Parameter Intervals
- Number of facets = 10



24 Verify your settings by clicking on the **Preview** button to calculate and display the preview of the facets.

25 Please click on **OK** to assign the Facet Attributes.

26 Verify your design with an unfolding. For this use the command Create Unfolding.



27 Set the following parameters:

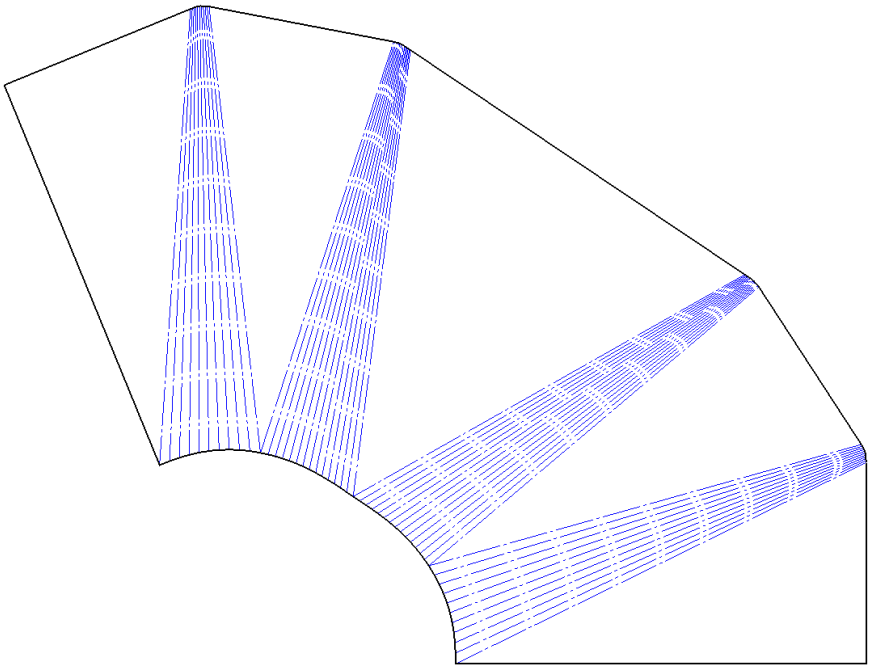
- Unfold a Body
- Unfold Destination = New Drawing

28 Select the start edge as shown in the figure

© SPI GmbH 2020

29 Please hit the **OK** button to launch the calculation of the unfolding.

Compare your result:



30 Save the unfolding with the file name `facetsheet.slddrw`.

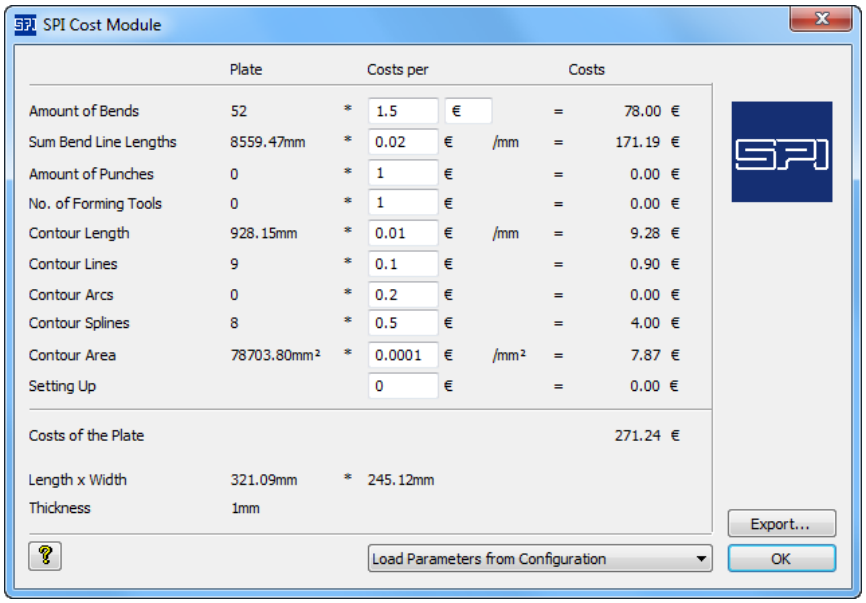
31 Go back to the document window `facetted.sldprt`.

32 Execute the command **Cost Module**.

© SPI GmbH 2020

Cost Module

The command **Cost Module** shows the data of the analysis of an unfolded sheet metal body. Estimate the costs for the manufacturing by assigning cost factors to calculculated data like Amount of bends, amount of forming tools, etc. These values were calculated while the last unfolding process of the part, so it can only be displayed when a part was unfolded successfully.



For each cost parameter you can enter dedicated values.

Additionally, the area of the raw flat pattern size (Length * Width) is calculated and displayed.

Export

The cost module allows storing the data into a text file. You will be prompted for a file name.

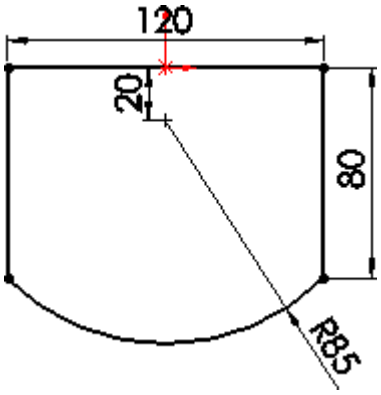
© SPI GmbH 2020

Exercise 7

Exercise 7 shows how to create and unfold a sheet metal part with a cylindrical surface and which Unfold Parameter needs to be set up in this case.

1 Please create a new part using **mm** as units. **Save** and name it `gap.sldprt`.

2 Make a sketch with the content displayed in the figure below on the **Front plane.** Alternatively, you can open the file `Exercise7.sldprt`.

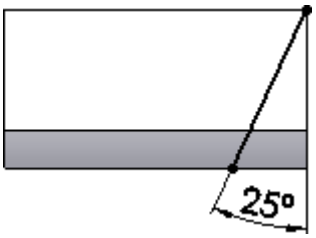


3 Please create a **Extruded Boss/Base**.

Set up the following values:

- End Condition = Blind
- Depth = 200 mm

4 Make a sketch on the **Plane right** as shown.



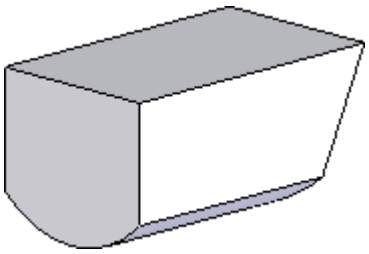
5 Please create a **Extruded Cut**.

Please enter the following values and settings:

- Direction1 = Activated
- End Condition = Through all
- Flip side to cut= Deactivated
- Direction2 = Activated
- End Condition = Through all

Intermediate Result

Please compare the result with the figure:

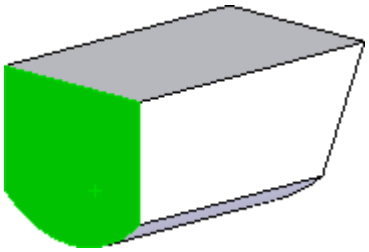


© SPI GmbH 2020

6 Please create a **Shell**.

Please enter the following values:

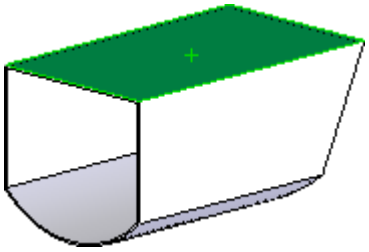
- Thickness = 1.5 mm
- Faces to remove = see picture



7 **Assign the SPI Material data to the 3D model.**
Use the command **Sheet Metal Wizard**.

9 Select a planar face of the part.

The face is listed in the **Selection** area of the **Sheet Metal Wizard** dialog.

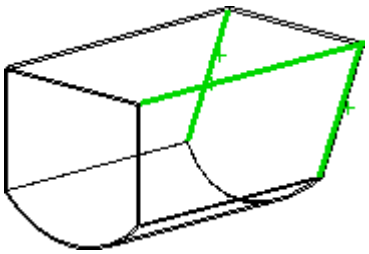


10 Continue with the second page of the wizard (click on the right arrow) and set these values:

- Material = Neutral
- Bending Tool = Neutral
- Correction Method = K-Factor
- Bend Radius = 2.0 mm

11 Execute the command **Corner Split Attributes**.

12 Select the edges according to the figure.



13 Set these parameters:

- Corner Type = Symmetric
- Spacing = 0.6 mm

14 Click on **OK** in order to assign the **Corner Split Attributes**

15 Execute the command **Unfold Parameters**.

© SPI GmbH 2020

16 Settings for the unfolding of bodies with **Corner Spacing**:

Enter the following values on the tab **Unfold Parameters General**:

- Corner Spacing = 0,6 mm

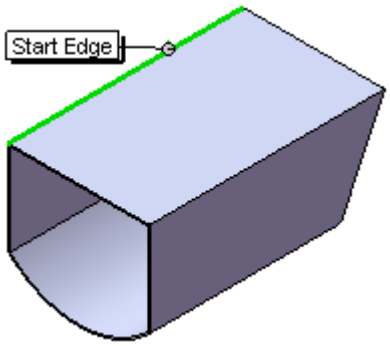
The parameter **Corner Spacing** is required in cases the part has non-straight edges that are supposed to be ripped during the unfolding. It is recommended to use the same value for the corner spacing and for the adjacent Corner Split Attributes, to receive a clean unfolding.

17 Verify your design with an unfolding.
Please use the command **Create Unfolding.**

18 Set up the following values for the command:

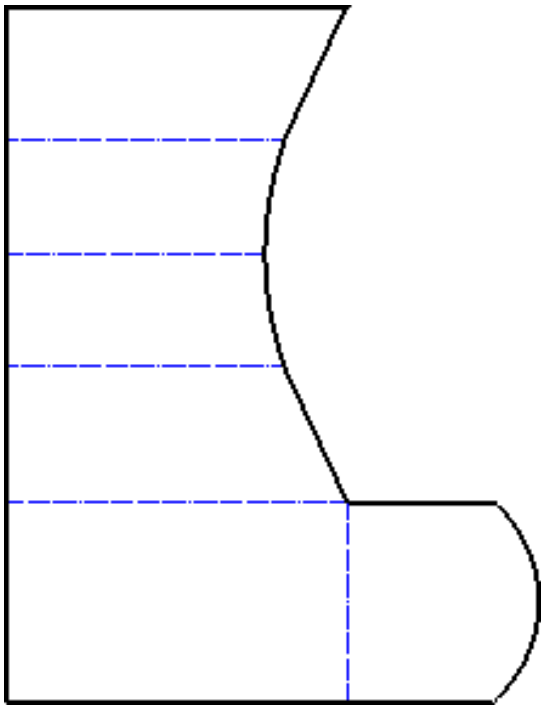
- Unfold a Body = activated
- Unfold Destination = New Drawing

19 Select this Start Edge:



20 Please hit the **OK** button to launch the calculation of the unfolding.

Compare your result with the picture below:



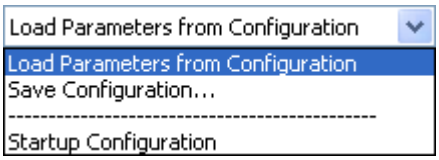
© SPI GmbH 2020

Unfold Parameters

The current unfold parameters controls the unfolding process of a sheet metal part. Some of the parameters also influence design commands. Having assigned material data to the part the unfold parameters are stored in the SOLIDWORKS part file. The parameters are bundled in groups which are displayed on the tabs of the unfold parameters dialog.

Unfold Configurations

Different types of sheet metal parts require different settings of unfold parameters. You can save the current settings as a configuration. You can reuse such a configuration for other parts. The standard settings are called **Startup Configuration**. This configuration is used for new part documents.



To load a setting scheme, open up the list **Load Parameters from Configuration** and select the desired configuration below the dashed line.

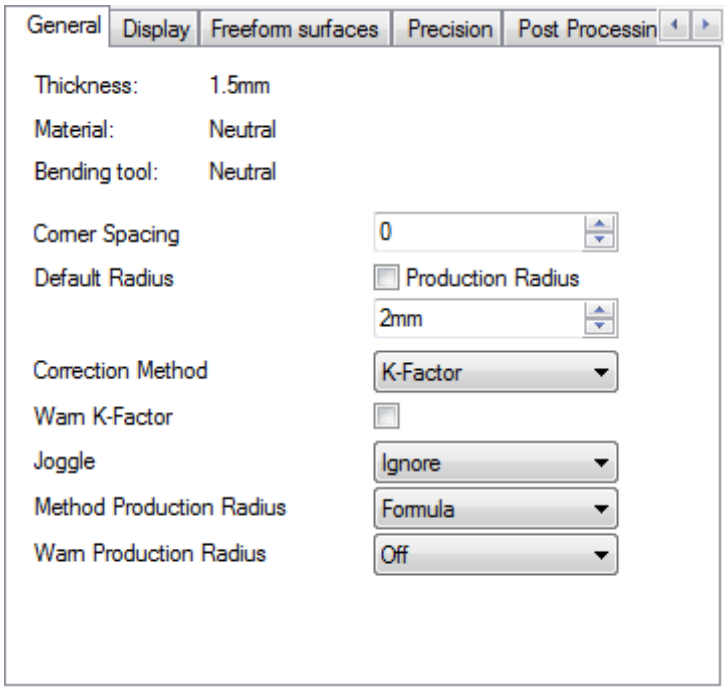
The entry **Save Configuration...** stores the settings in a configuration.

This can also be used to redefine the Startup Configuration.

To restore all unfold parameters to the Startup Configuration press the icon **Reset**.

© SPI GmbH 2020

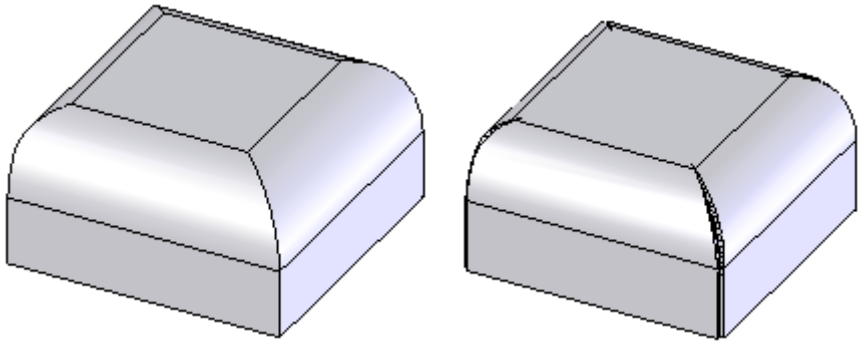
General



The upper part of the tab **General** shows the current material data (material, bending tool, thickness) of the current part. Use the SPI Sheet Metal Wizard to modify these settings.

Corner Spacing

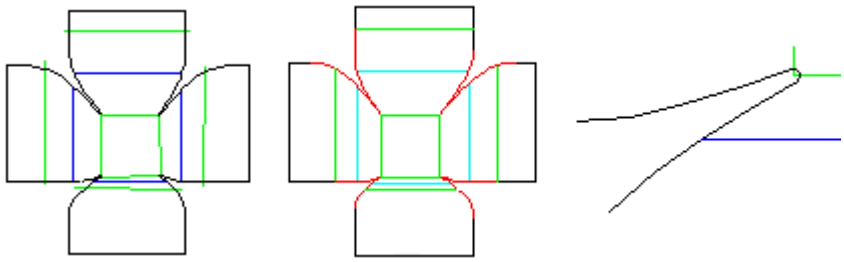
If this parameter is set to a value bigger than 0, all curved edges will be ripped symmetrically with a gap of this size during unfolding. Otherwise these edges were marked in error color (usually red).



The option **Corner Spacing** will only influence curved edges. Straight edges need a **Corner Split Attribute** that must be assigned manually. Corners that are ripped in this way will not be smoothed, even if the option **Smooth Filleted** is activated. This means there will be no relief in a corner like this.

© SPI GmbH 2020

SPI SheetMetalWorks smoothes the contour by inserting circular arcs, where two edges intersect in one point.



Picture 5: The left picture shows the unfolding with a Corner Spacing value bigger than 0, the second without Corner Spacing. The right image shows the smoothed corner

Default Radius

The value Default Radius will be assigned automatically to all sharp cornered bend zones when calculating an unfolding to determine the shortening. It is possible to override this general radius setting by assigning a dedicated **Radius Attribute** to the bend zone. Alternatively, the **Production Radius** can be evaluated and used as bend radius.

Correction Method

The Correction Method controls, how the SPI unfolding engine calculates the bend zone width when unfolding the part. To use a correction method, the required data must be defined in the SPI material database. You can choose between the following options:

- **K-Factor**: Calculation according the DIN correction formula.
- **Formula**: Calculation with a user defined formula that can be maintained in the SPI Material Editor.
- **Table**: Calculation with empiric shortening values which are stored in tables.
- **TruTops Table**: Calculation using correction values which were imported from a TruTops Bend (ToPs 600) database.
- Please refer to the documentation of the SPI Material Editor for further details.

Freeform surfaces and conical bend zones can not be unfolded with a correction method described above. These faces are always unfolded according to the neutral fiber.

Warn K-Factor

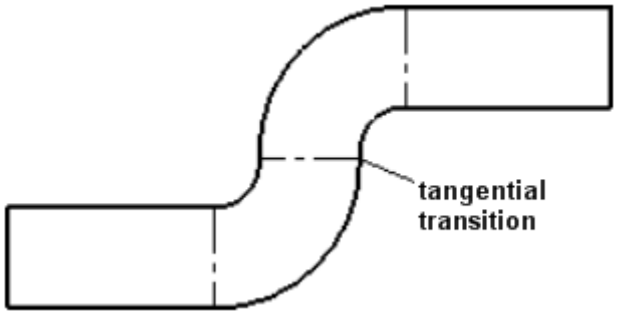
If this option is activated, the SPI unfolding engine will check the resulting value for the K-Factor when using a user defined formula. A warning will appear if the resulting K-Factor is bigger than 1 using ISO, or 0.5 using ANSI, since this means the definition of the K-Factor is no longer reasonable.

© SPI GmbH 2020

Joggle

Usually a Joggle, also known as Z-Bend, requires a separated shortening correction value since this type of deforming is not a typical bending but manufactured with a special tool. However, sometimes this geometry is made by two ordinary bends, too. To reflect these different situations there are three options available to calculate the Z-bend-deduction:

- **Ignore**: Uses the standard shortening calculation method, assuming two ordinary bends.
- **Formula**: Calculation with a dedicated Z-deduction formula.
- **Table**: Using a table with empiric Z-deduction values.



Method Production Radius

This option controls how the Production Radius will be evaluated. It can only be used if appropriate data are maintained in the SPI material database. You have the choice between these options:

- **Formula**: Calculate the production radius with a formula.
- **Table**: Determine the radius with a table of empiric values.
- **TruTops Table**: Use the imported data from a TruTops database.

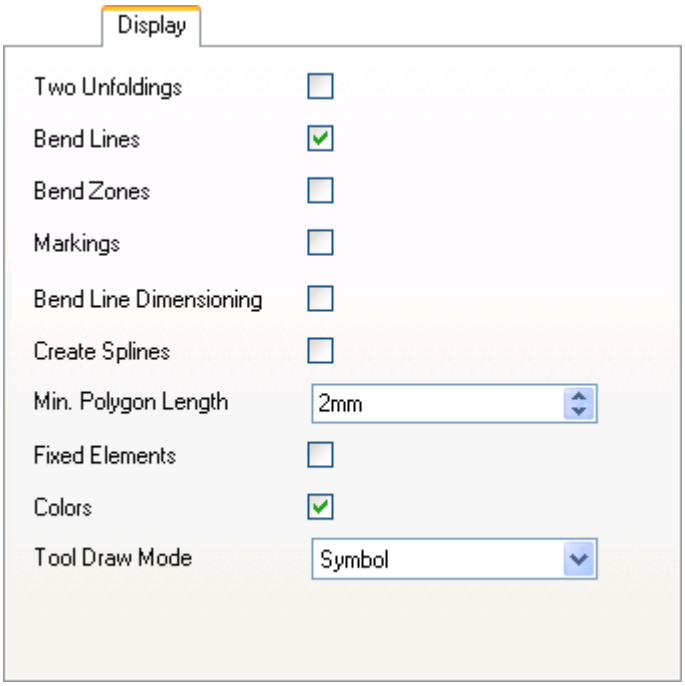
Warn Production Radius

This option controls the behavior when unfolding a sheet metal part in cases radii are detected which differ from the production radius.

- Off: No warnings.
- Fall below: A warning is displayed if a bend radius is smaller than the evaluated production radius.
- Differ: A warning is displayed if a bend radius differs from the evaluated production radius.

© SPI GmbH 2020

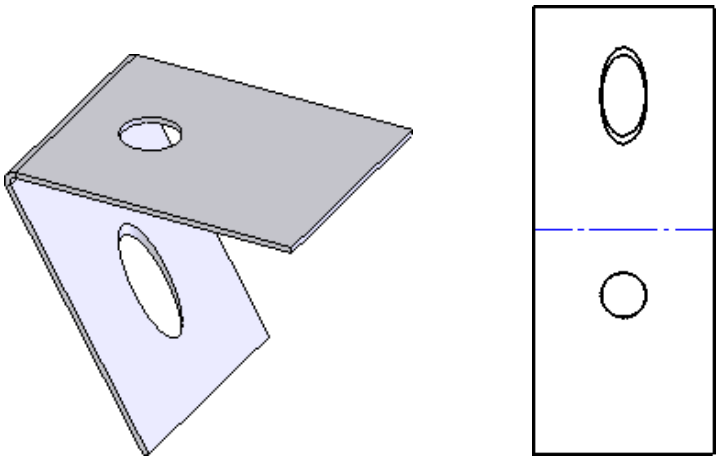
Display



Several options are attached to the solid sheet metal body that controls the details of presentation.

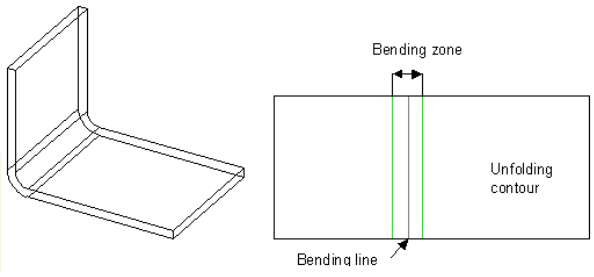
Two Unfoldings

You can create two unfoldings of the 3D body by activating this option to see the difference between the inside and outside of the part. If deactivated only the side will be displayed that is selected by the choice of the start edge.



Bend Lines

This option controls, if Bend Lines will be displayed in the unfolding or not. Normally the Bend Lines end at the outer contour. On the tab **Post Processing Parameters,** you can also set up a gap between the bend lines and the outer contour. This can help the NC software in order to distinguish between outer contours and bend lines.



© SPI GmbH 2020

Bend Zones

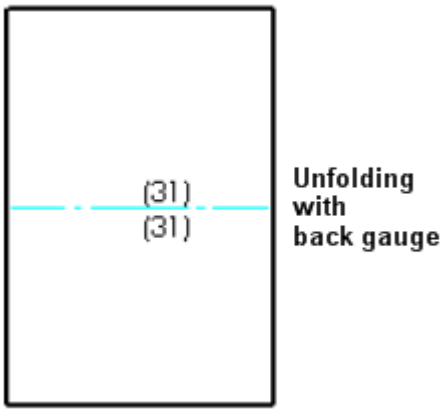
Activate this option to display the borders where the bend zones meet flat areas.

Markings

Activate this option in order to include unused sketches on planar faces in the unfolding as a marking.

Bend Line Dimensioning

If this option is activated, **SPI SheetMetalWorks** will add informations into the unfolding that will be serviceable for the manufacturing: At all bend lines back gauge value and the bend angle if not equal to 90 degrees.



Create Splines

If this option is activated complex curves in the unfolding will be drafted as splines. If this option is deactivated the splines will be transformed into polylines (chains of lines).

Min. Polygon Length

This parameter controls the smallest length of the polyline elements that approximate complex curves (for example ellipses) in the unfolding. The value must be greater than 0.

Fix Elements

Setting this option will fix the sketch elements in the unfolding sketch. This prevents undesired modifications of the unfolding. However, it might take longer to calculate the unfolding.

Colors

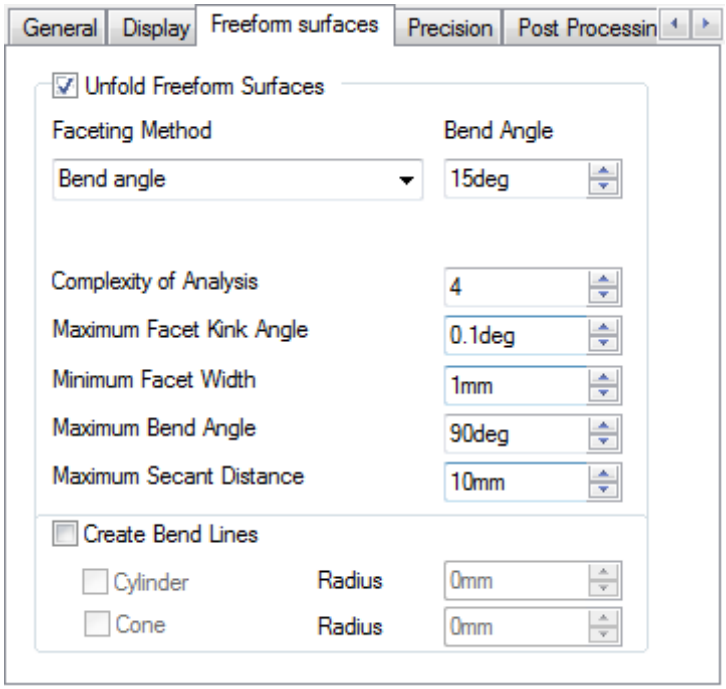
This option makes the sketch elements colored. If deactivated no color data will be assigned (see line settings).

Tool Draw Mode

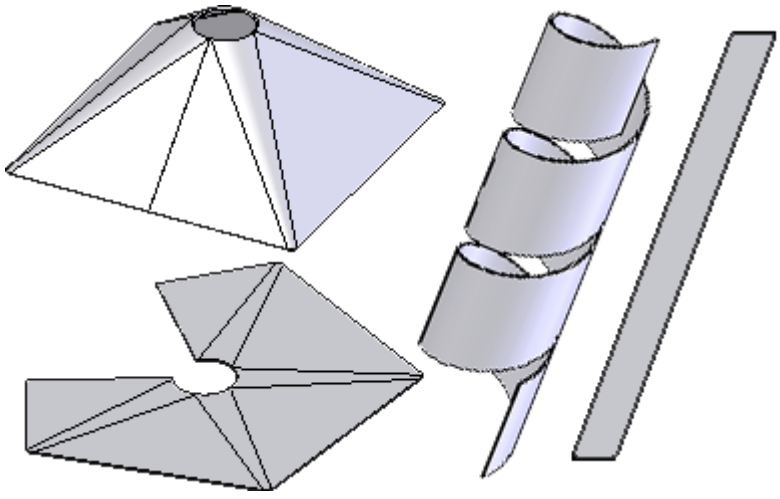
This option defines the kind of representation the identified forming tools will be added to the unfolding.

© SPI GmbH 2020

Freeform surfaces

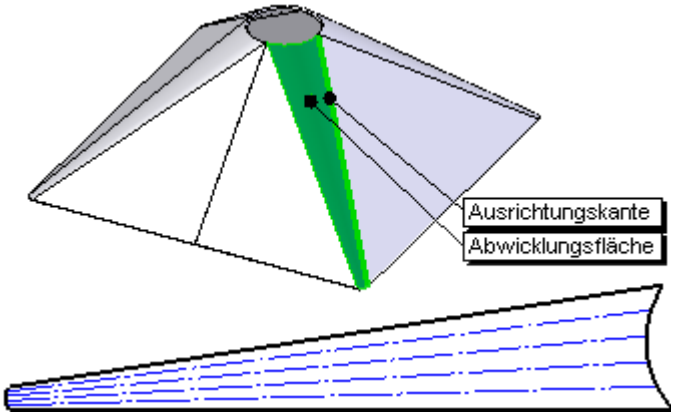


If the option **Unfold Freeform Surfaces** is activated, the SPI unfolding engine will unfold freeform surfaces, too. Such surfaces could be generated by Loft or Fillet features.



© SPI GmbH 2020

It is also possible to unfold a single face of a 3D body, even if this is not a sheet metal part. This can be applied to cylindrical, conical or freeform faces.



Prerequisites to unfold freeform surfaces

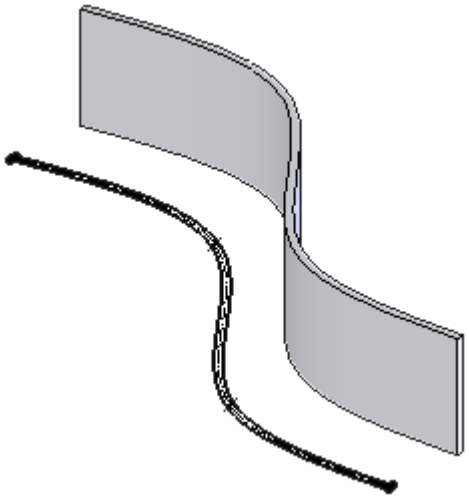
Freeform surfaces must fulfill some conditions to be unfoldable. The face must be unfoldable from a mathematical point of view, i.e. locally it must be curved in only one direction. For example, spheres are curved in two directions so they are not unfoldable.

If you want to unfold more than a single surface, the freeform surface must fulfill these conditions:

- It must be an element of a SOLIDWORKS part,
- there must be tangent situations where the freeform surface borders other faces,
- another parallel face must exist, with an offset of sheet metal thickness.

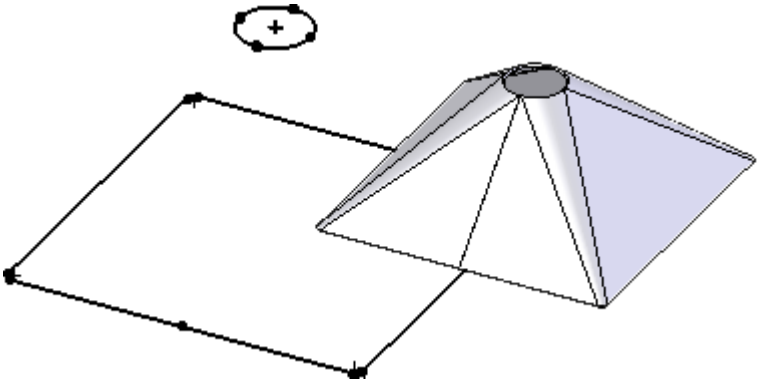
Typical freeform surfaces are created with the following commands:

- Extrusion of a spline curve or an ellipse



© SPI GmbH 2020

- Lofting (without twisting) plus shelling

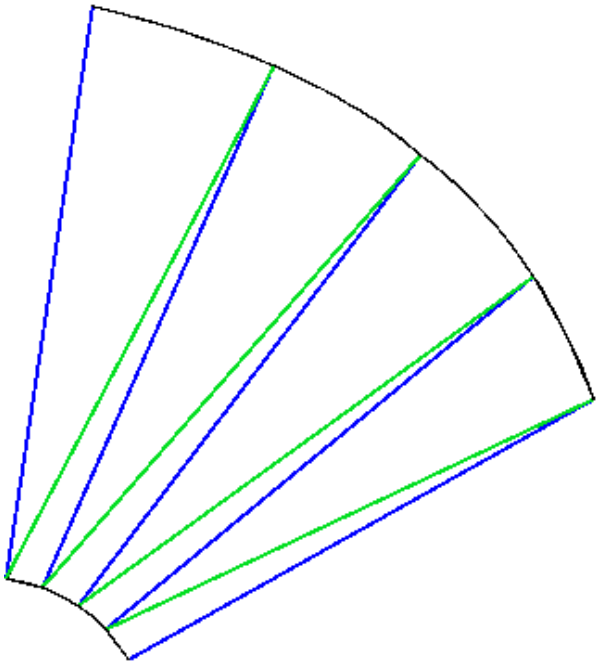


- Filleting with a variable radius of an edge plus shelling

Surfaces that are curved in two directions can not be unfolded.

How SPI SheetMetalWorks unfolds freeform surfaces

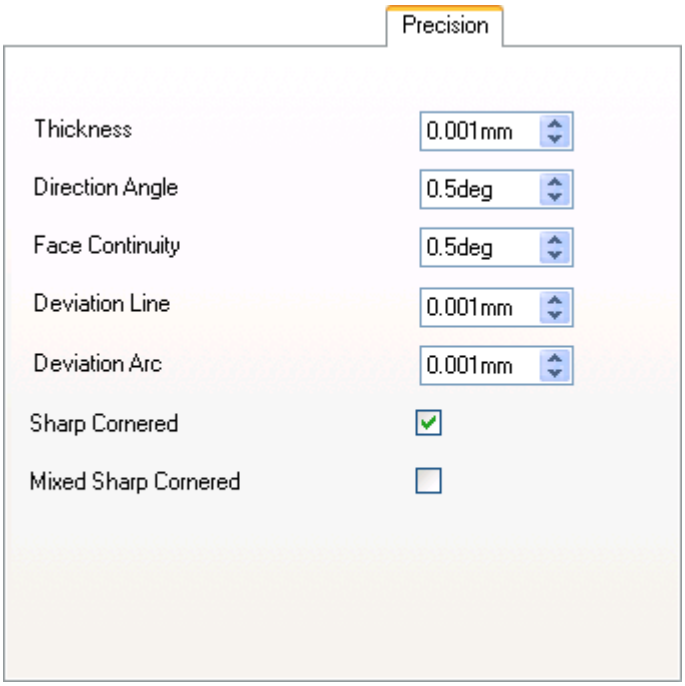
The SPI unfolding engine uses the triangulation method. This approximates the freeform surface with a mesh of plane triangles and four-sided figures. Four-sided figures are made of the neighbor's triangles are in the same plane. By unfolding this approximation, a net is made as shown in the figure. The amount of triangle areas can be adjusted by a parameter. The amount of triangle areas has influence on the accuracy of the unfolding. However, the calculation time increases with a larger amount of triangles.



The blue straight lines lie on the freeform surface. The green ones arise from the triangle approximation. It must be possible to place a straight line on the face at any point.

© SPI GmbH 2020

Precision



The precision settings give additional control of the unfolding. Using rough values, it is possible to unfold parts containing inaccuracies. This is especially helpful when dealing with imported data.

Modify these settings with caution!

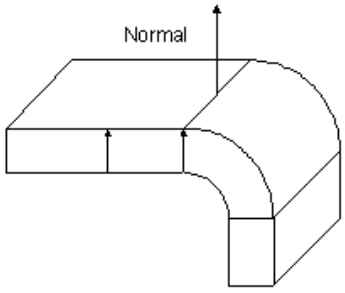
Thickness

With the parameter thickness you set the maximum thickness deviation which the 3D sheet metal body may have so that the sheet metal faces are recognized as parallel surfaces. The value must be greater than 0 and less or equal than 1.

Direction Angle

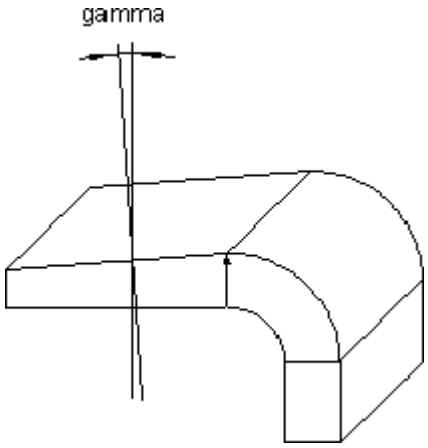
With the parameter **Direction Angle** you decide which direction deviation is permitted by parallel surfaces so that these surfaces are still recognized as parallel in the unfolding process. The value is indicated as an angle and must be greater than 0 and smaller than 180 degrees.

The following illustration shows a proper bend zone transition. The matching surfaces and the cylindrical surfaces are parallel. The flat surfaces change steadily into the cylindrical surfaces (without "crease"). The normal vectors of the two adjacent faces are equal onto the edge.



© SPI GmbH 2020

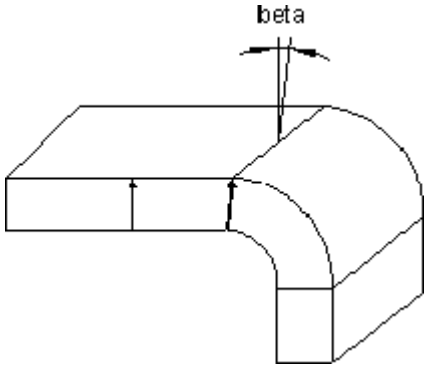
The following illustration shows that the sheet metal side faces are not parallel to each other. The normal vectors are in the angle gamma to each other. This deviation can be adjusted with the parameter **direction angle**.



Face Continuity

With the parameter **Face Continuity,** you set up the maximum "crease" between neighboring surfaces which is authorized so that the unfolding recognizes that these surfaces change into each other. The value is indicated as an angle and must be greater than 0 and smaller than 180 degrees.

It can be seen in the following illustration that the two cylindrical surfaces do not change into the upper plane tangentially. The normal vectors differ about the angle *beta*. This deviation can be adjusted with the parameter Face Continuity.



Deviation Line and Deviation Arc

These deviations are used to produce simpler elements in the unfolding. The unfolding engine tries to make straight lines and circular arcs instead of polylines and spline curves.

Sharp Cornered

If this option is switched on, the unfolding accepts sharp edged bend zones. Otherwise a corresponding error message is written into the Unfolding Log.

© SPI GmbH 2020

Mixed Sharp Cornered

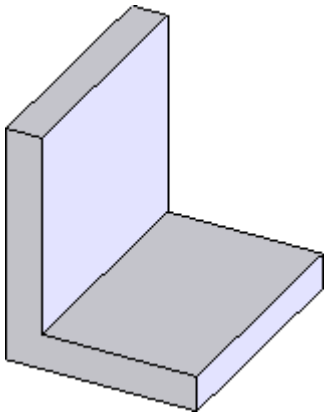
If this option is activated, then mixed sharp-cornered bend zones are unfolded. A mixed sharp cornered means that an inner radius of 0 and an outer radius with the value of the sheet metal thickness are at the bend zone. Reports otherwise are written in the **Unfolding Log** if bends of this type were recognized.

The shortening for this type of bending is calculated under use of the bend radius 0. So the correction value is used for the radius 0 from the column from the correction table. At the use of a formula the radius 0 is submitted to these for the calculation.

Please, take notice of the documentation of the **SPI Material Editor** in which the shortening calculation is also described.

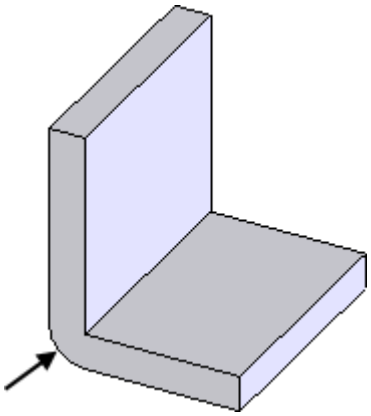
The following illustrations show the different kinds of bend zones which can be unfolded with **SPI SheetMetalWorks** if the corresponding option is switched on.

Sharp Cornered



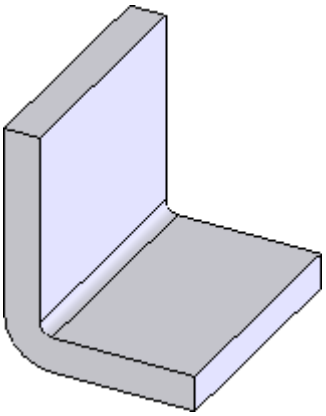
Mixed sharp cornered

Mixed sharp-cornered bend with inside radius 0 and outer radius equal to thickness (arrow)



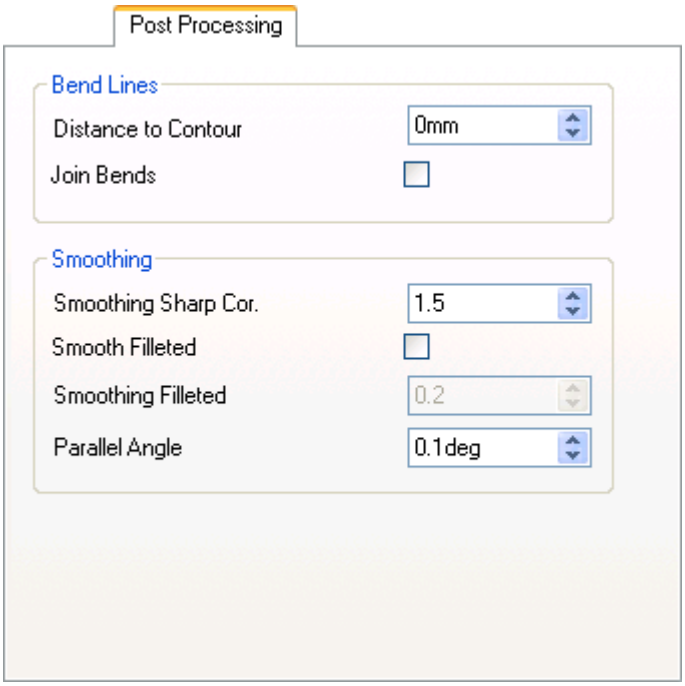
Filleted bend zone

Rounded bend zone with inside radius greater than 0 and on the outside radius equals (inside radius+sheet metal thickness



© SPI GmbH 2020

Post Processing

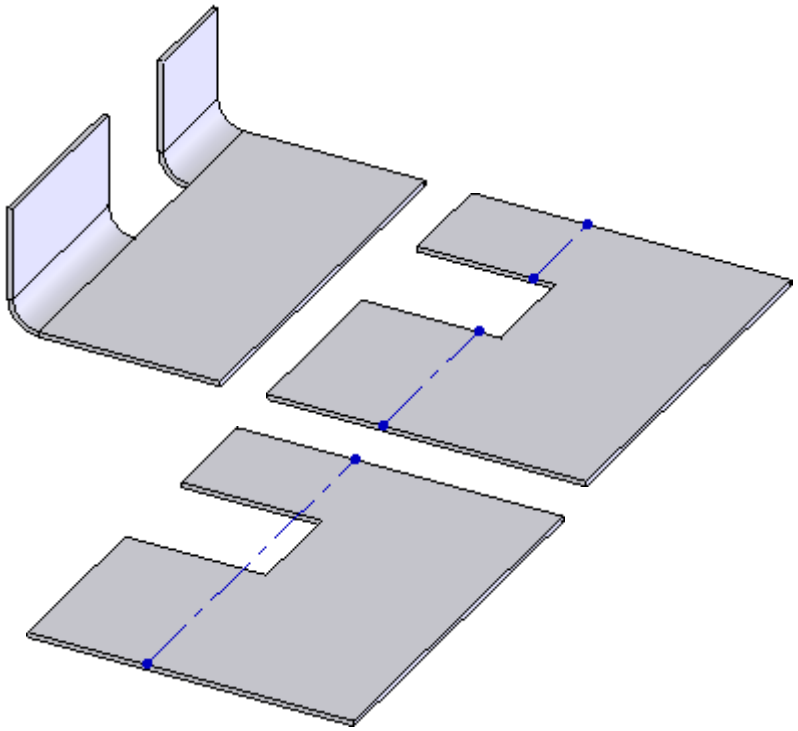


Distance to Contour

Normally the bend lines end at the outer contour. The parameter **Distance to Contour** controls the gap between the outer contour and the end points of the bend lines. This is needed for some NC software packages to avoid confusion between bend lines and outer contour.

Join Bends

If this option is activated, bend lines, which can be manufactured in one step, are merged. This is displayed in the figure below.



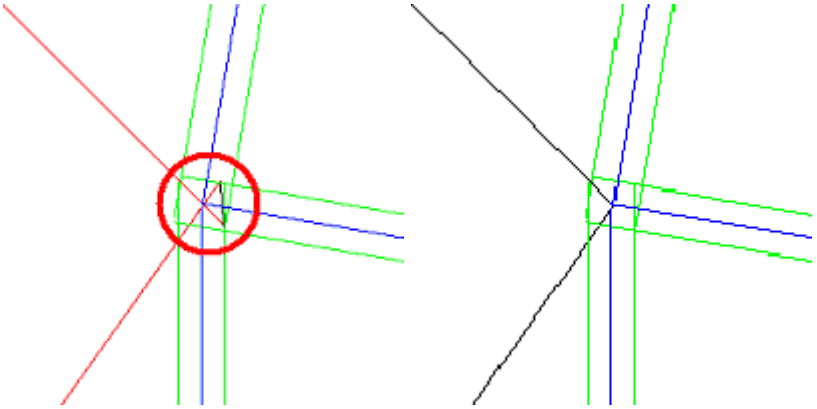
© SPI GmbH 2020

Smoothing

When bending flat patterns, which are from an unfolding with sharp-cornered flanges, the sheet metal will normally tear at the not relieved flange edges. An analogous phenomenon also shows the sheet metal unfolding with **SPI SheetMetalWorks**. To get a closed contour after the unfolding again, short help edges must be retracted at the bend zones. On the whole these are combined possibly, steered by smoothing factors, with neighbor edges of them. The value of these factors must more greatly or at once be worth zero.

Smoothing Sharp Cornered

This parameter controls the size of the area around a corner situation in which small edges are deleted to receive a clean corner design. All unfolding lines are deleted within the area with a diameter of "**smoothing factor * bend zone width**". The default of 1.5 is usually sufficient. If you, however, should find a corner as represented in the left lower illustration, then you can increase the value.



The above illustrations show a part of an unfolding, these with a smoothing factor of 0 (no smoothing) or from 1.5 on the right side. The red circle marks the area in which all lines are deleted. This circle has the diameter d = smoothing factor * bend zone width. If this parameter is too big, then perhaps also important lines are deleted.

Smooth Filleted

The contours of filleted bend zones usually do not need to be smoothed. Therefore, this option is turned off in the default settings. This option must be switched on, if the unfolding engine is supposed to insert reliefs at filleted bend zones (see also **Unfolding Parameters, Relief p.96**).

© SPI GmbH 2020

Smoothing Filleted

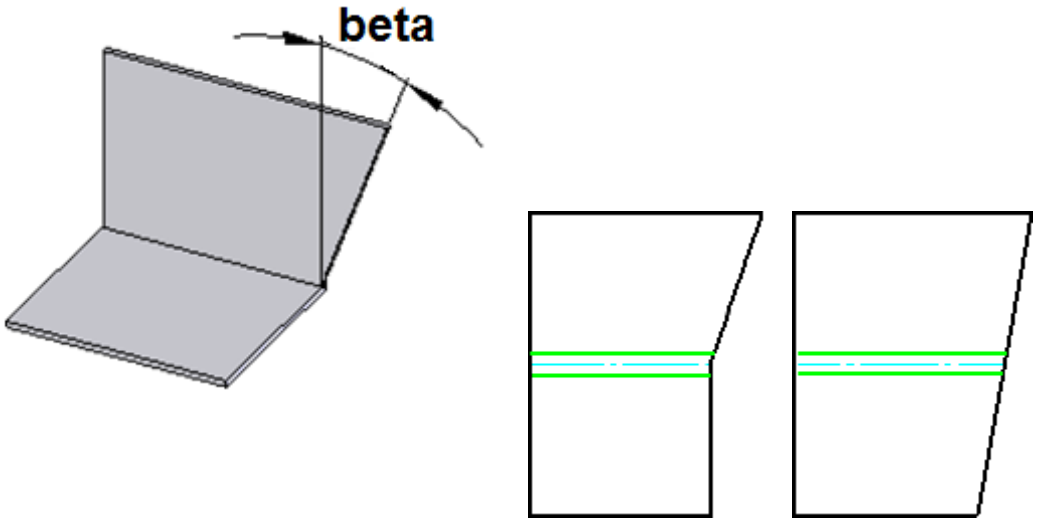
You also can combine edges nearby a bend line over the option "**Smoothing Filleted**" with the neighbor edges which are part of a rounded bend zone, however, if they lie within a circle around the end-points of the bend lines. The diameter of this circle calculates itself from "**2 * smoothing factor * bend zone width**". Short additional edges in the bend zone rounded originally appear only at inaccurate tangential transitions.

You take into account an appropriate corner is then not smoothed and no automatic relief brought in there either, if the **general parameter "Corner Spacing"** is greater than 0.

Parallel Angle

With this parameter you define the maximum angle which lines of the unfolding contour succeeding one another may have to each other to be recognized as equal directional. These elements are then summarized to a line. The value must be greater than 0 and smaller than 180 degrees.

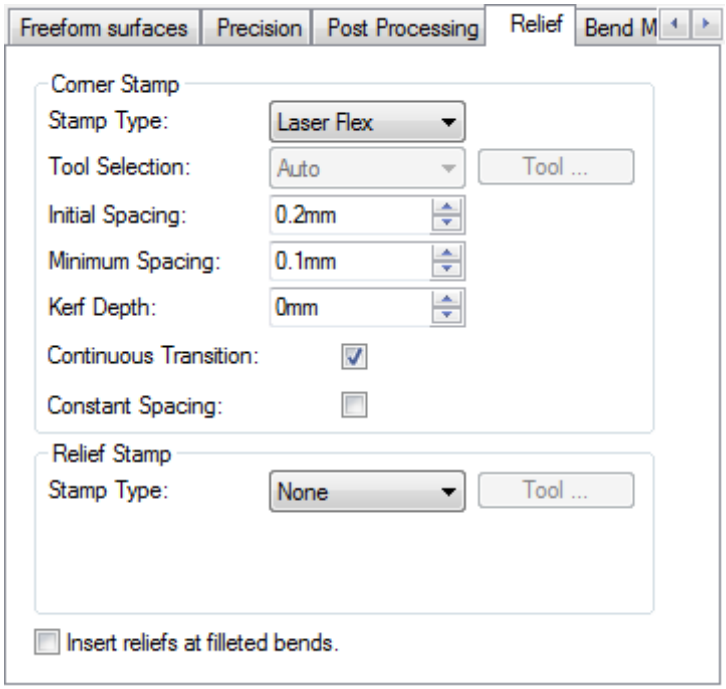
The lines are not combined to a line in the following illustration at the Unfolding 1 if the value for the parallel angle is smaller than the angle *beta* of the lines to each other. If the value is, however, greater than *beta*, a result is provided like into Unfolding 2.



Unfolding 1 **Unfolding 2**

© SPI GmbH 2020

Relief



The unfolding contour can be enhanced with reliefs at the ends of the bend lines automatically. This is given at the unfolding of sharp cornered bend zones generally. It is distinguished between **Corner Stamps** and **Relief Stamps**.

A corner situation is defined as follows: The bend zones of at least two bend zones conflict. The intersection point of the bend lines more exactly lies within a circle around the end-points of the bend lines with the diameter 2* width of the bend zone or 2* width of the relief stamp depending on which of the two values is bigger.

You can define a divergent relief with the **Relief Attributes** for every end of a bend. The following parameters steer producing the relief automatically.

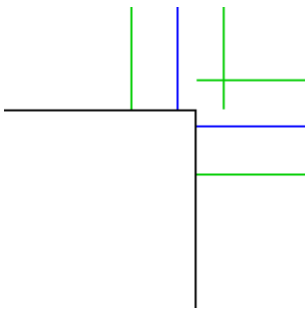
Corner Stamp

Definition that will used for corner situations:

Stamp Type

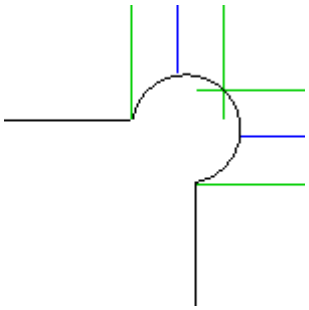
None

No stamp – the corner is simplified

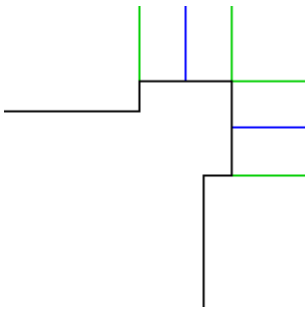


© SPI GmbH 2020

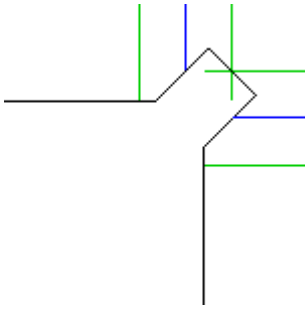
Circle Round stamp (circle)



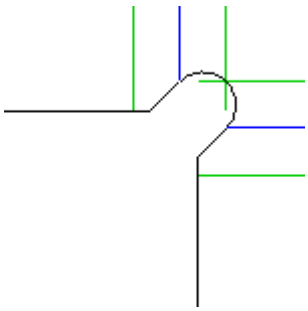
Square Square stamp, horizontally aligned



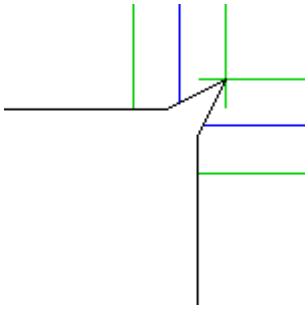
Diag Rectangle stamp, oriented diagonally



Oblong Oblong stamp



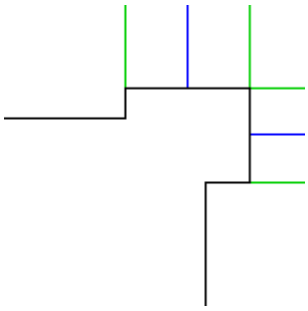
Rhomb Rhomb resp. triangle



© SPI GmbH 2020

Generic

Stamp cuts the bend zone perpendicularly.
Exactly the area in which the bend zones intersect



(LaserEd)

Laser stamp also see laser relief, aligned at the flange edge.

This laser relief type is outdated: The new Type **Laser Flex** makes more accurate results.

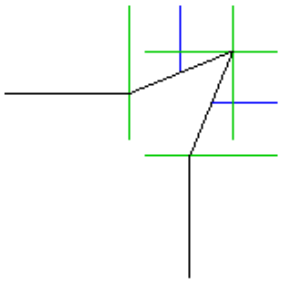
(LaserSym)

Symmetric Laser stamp

This laser relief type is outdated: The new Type **Laser Flex** makes more accurate results.

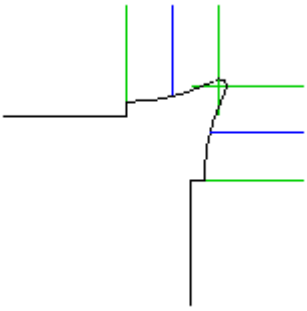
Laser Lin

Laser stamp as linear cut from the intersection point of the inner bendzone lines.



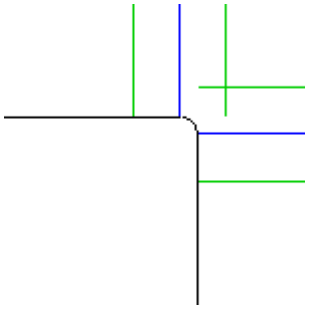
Laser Flex

Flexible laser stamp for arbitrary corner and bend angles.
For an exact description of this type please refer to the online documentation of **SPI SheetMetalWorks**.



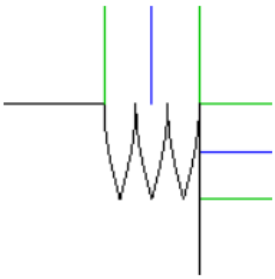
Smoothing

Smoothes the corner with a circular arc.



Corner Blend

This type should be used for the unfolding of a ball-shaped corner. For an exact description of this type please refer to the Online-Help of **SPI SheetMetalWorks**.



© SPI GmbH 2020

Tool Selection

You can set up for some types of corner tools how the dimensions of the stamp are calculated.

Tool

Press the button **Tool...** to choose a dedicated tool from the tool database or enter the desired size values.

Auto

Automatic calculated parameters for an optimal relief. You can define an offset for the optimal size.

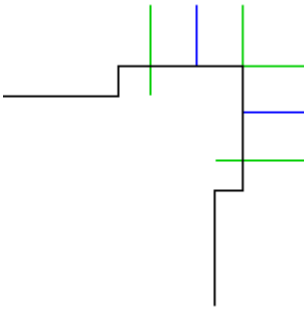
AutoTool

The next available tool from the **tool database** for automatic calculated parameters

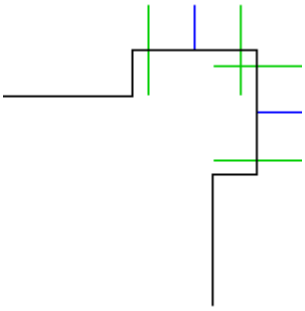
Reference Type

The position of the corner stamps also can be precised. There are two possible ways to reference the positioning point.

Corner

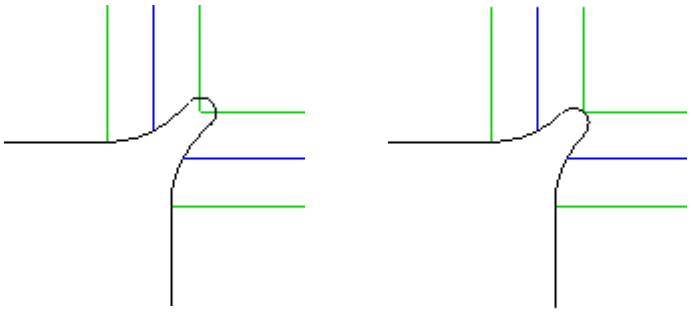


Middle



For the (outdated) laser stamps Laser Sym and Laser Ka the definition of the way of reference differs since laser stamp geometry does not have a tool centre. The centre of the circular arc is located at the placing option corner on the intersection point of the bend zones. If the option **middle** is chosen, then the circular arc touches this intersection point.

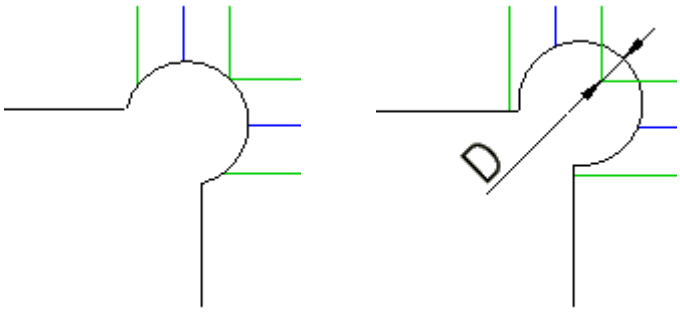
© SPI GmbH 2020



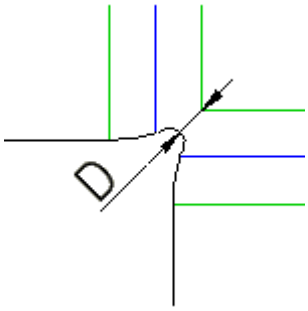
The left picture shows a laser relief with the Reference type **corner**. The right picture represents the same laser relief, though with the reference kind of M**iddle**.

Displacement

Additionally, you can define an offset value. This value allows shifting the reference point along the imaginary diagonal line. A positive value removes more material from the corner, a negative value less.



A circle stamp is put with the reference kind **Corner** without offset on the left side. The right picture shows the same tool with a positive offset. The stamp removes more material from the corner.

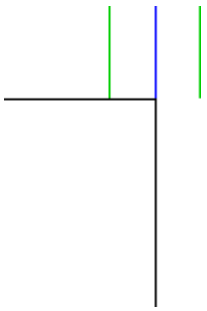


A laser relief is defined in the upper picture with a negative offset to remove less material from the corner.

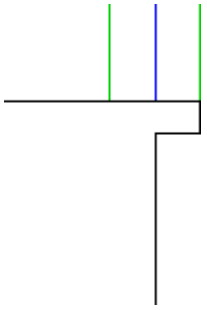
© SPI GmbH 2020

Relief Stamp **Define the stamp type for the flange relief, i.e. no other bend zone lies nearby the bend zone looked at:**

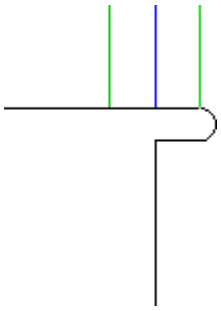
None No stamp for flange situations



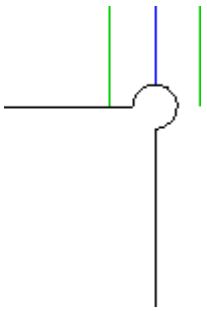
Rectangle Rectangular flange relief



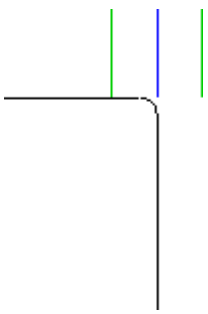
Oblong Oblong stamp



Circle Round stamp (Circle)



Smoothing Smoothes the transition from the flange to the contour with a circular arc



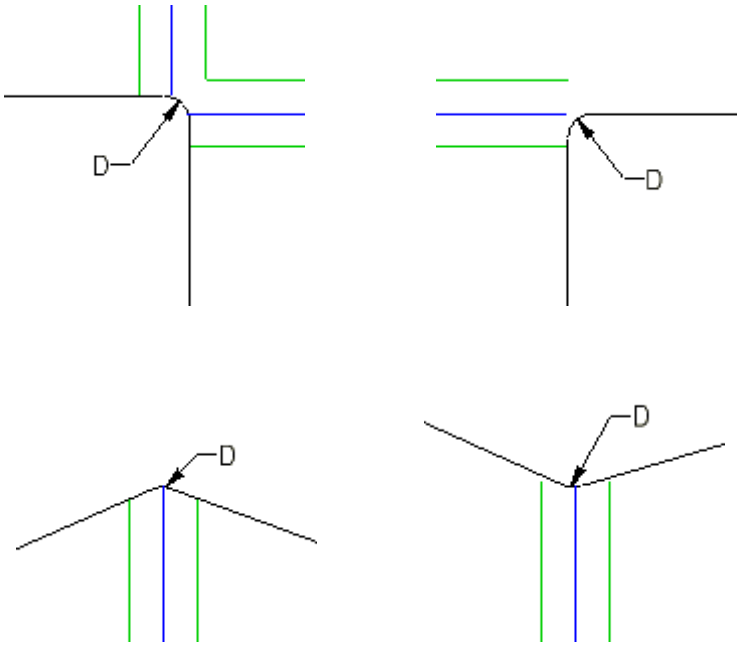
© SPI GmbH 2020

You can, depending on the relief stamp class, enter the parameters of the tool or select a tool from the **tool database**. In addition, you can indicate an additional relief depth which is added up to the value which is necessary for relieving the bend zone.

The tools are defined in the **tool database**. You can modify them with the **SPI Tool Editor**.

Smooth

If a relief stamp S**mooth** is chosen, then a circular arc is inserted in the unfolding contour with a predefined diameter instead of a relief. The option **Smooth** is available for both: corners and flanges. The illustrations below show the smoothing of one corner, a flanges and bend zone situation.



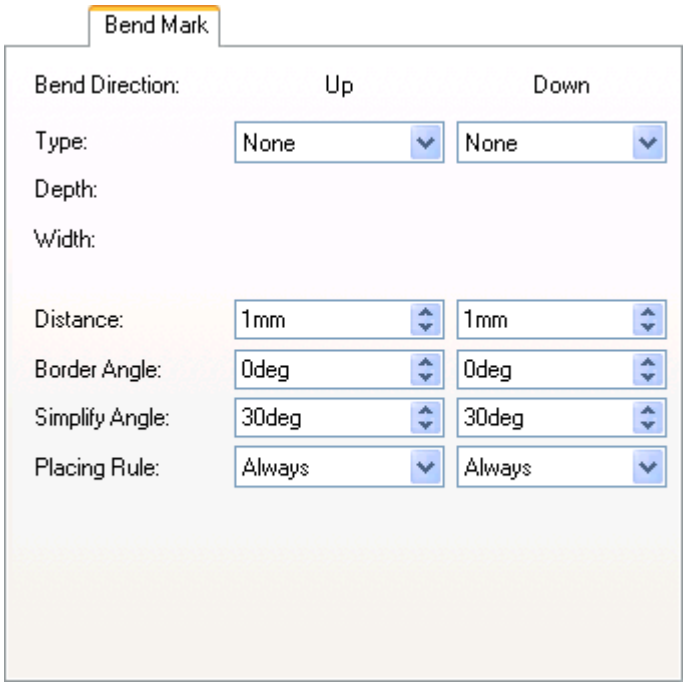
Insert reliefs at filleted bends

Rounded bend zones usually already have relief in the 3D model. If not, you can activate this option to also generate reliefs at filleted bend zones. So that the unfolding process can install the relief successfully the post processing option **Smooth filleted** must be switched on and the factor **Smoothing Filleted** must be greater than 1.

Take into account: If the general parameter **Corner Spacing** is greater than 0 an appropriate corner is not smoothed and no automatic relief will be brought in there either.

© SPI GmbH 2020

Bend Mark



Some productional processes need certain markings at the sheet metal flat pattern so that this can be bent. These markings, usually at the ends of a bend, are used to position the flat pattern on the die of the press brake. Such bend marks may be either existing reliefs or additional reliefs such as etch or engraving markings. With the bend mark parameters, you can insert such markings in the unfolding. Different types and control parameters can be used for the definition of the bend marks.

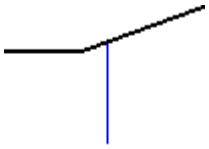
You can define a divergent bend mark for every end of a bend zone with the **Bend Mark Attributes**. The following parameters control producing the bend marks automatically. You can adjust different values for bends up and down**.**

Type of Bend Mark

You have to select one of the following bend mark types:

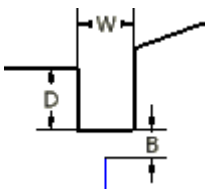
None

No bend mark



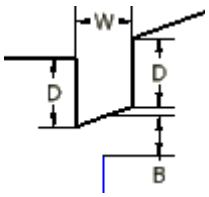
Rectangle

Inserts a rectangular cut

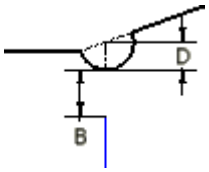


© SPI GmbH 2020

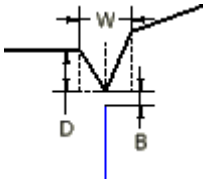
Parallel Inserts a parallel cut in which the cuts have the same length on the two sides.



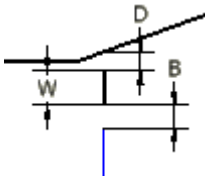
Circle Cuts a circular arc out of the contour.



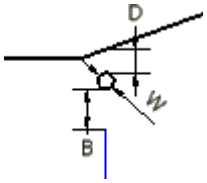
Rhomb Inserts a rhomboid (triangular) cut.



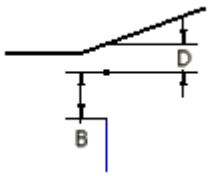
Etch Line Produces an etch line.



Etch Circle Creates an etch line to position a small hole punch, etc.



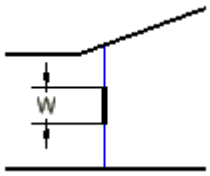
Center Mark Creates a center mark.



© SPI GmbH 2020

Centerline

Creates an etch centerline in the middle of both bendline endpoints.



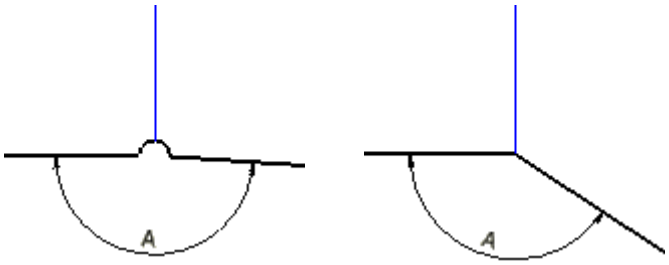
You can enter a value for a selected bend mark type for the depth **D** and for the width **W** if possible. The parameter distance **B** is described in the following section.

Distance

The parameter defines the distance of the end of the bend line to the bend mark. This parameter overwrites the parameter **Distance to Contour** which is defined on the tab **Post Processing** of the Unfold parameters dialog.

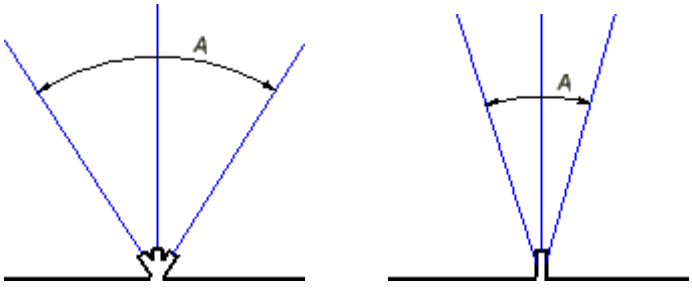
Border Angle

If there is a kink where the contour meets up the bend line, it often is not necessarily to create a bend mark there that way. The kink on the edge is already sufficient to mark the bend. The edge angle defines a threshold at which a bend mark shall be created. The two pictures below show these situations. In the left picture the kink angle is greater or equal to the border angle, thus a bend mark will be created. In the right picture the kink angle is smaller than the threshold angle and a bend mark is not put.



Simplify Angle

The simplification angle defines a threshold to reduce the number of bend marks. If two or more bend lines meet the contour in a corner, then the angle A between the utmost lines will be compared with the threshold angle. If the angle A is less, then the simplification angle only one bend mark will be created (right picture). The following pictures show the difference.



© SPI GmbH 2020

Placing Rule

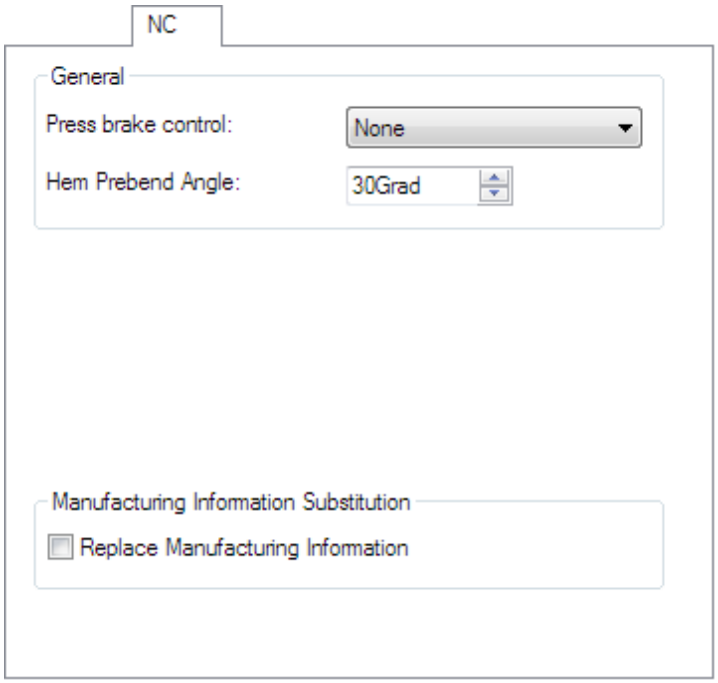
With the placing rule you have another possibility of controlling the bend mark.

The option **Always** will apply bend marks in all cases that are not overruled by other settings.

The option **Not near Reliefs** avoids creating a bend mark if a relief is already put there.

© SPI GmbH 2020

NC



Use these settings to create unfoldings for the NC control of your press brake. These data help the NC control to read the unfolding output file more efficiently.

The following press brakes NC programs are supported:

- Cybelec
- Delem
- TruTops with bend angle
- TruTops with included angle
- User defined

Having activated one of the NC option sets, **SPI SheetMetalWorks** verifies all unfolding settings. When running into a conflict the software will prompt the user if the settings shall be corrected automatically. The table **NC press brake control** (see p. 15) will show you the relation between NC program and related parameter settings.

Please read the documentation of your NC software, too.

© SPI GmbH 2020

Generated NC information

At the bend lines in the unfolding for Cybelec and Delem additional textual information like bend angle, radius etc. are placed. In case of **Delem** all hems will get a text with the value defined with the **Hem Prebend Angle** option.

DXF export of an unfolding

Most of the NC programs could import 2D files in the DXF-12 format. In SOLIDWORKS you can save the unfold drawing in this format. The drawing should contain only the unfolding and the additional NC information. Use the command **Save As...** and choose the DXF file type.

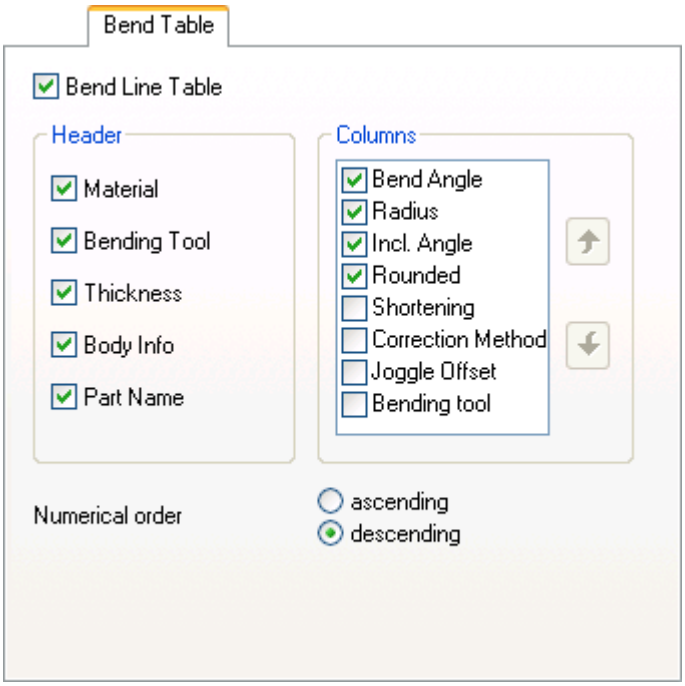


SOLIDWORKS collects all texts in one block. For this reason it can be that the NC-Programm cannot import the NC-Informations. In this case you have to explode the blocks by an external application.

Since SOLIDWORKS 2005 the DWG-Editor is very usefull to carry out this job: Import the DXF file into the editor, select all elements in the drawing, run "_explode" by the command line and finally save the drawing in the "AutoCAD Release 11/12 ASCII DXF" format.

© SPI GmbH 2020

Bend Table



SPI SheetMetalWorks also generates a bend table on request. This bend table is added to the unfolding on the drawing document. The option **Bend Line Table** must be activated.

Header

The following values can be displayed in the header:

- Material
- Bending Tool
- Thickness
- Body Info
- Part Name

Columns

The following columns can be selected:

- Bend Angle
- Radius
- Incl. Angle = Inclusive Angle
- Rounded
- Shortening
- Correction Method
- Joggle Offset
- Bending Tool
- Length

Use the arrow buttons to change the order of the columns.

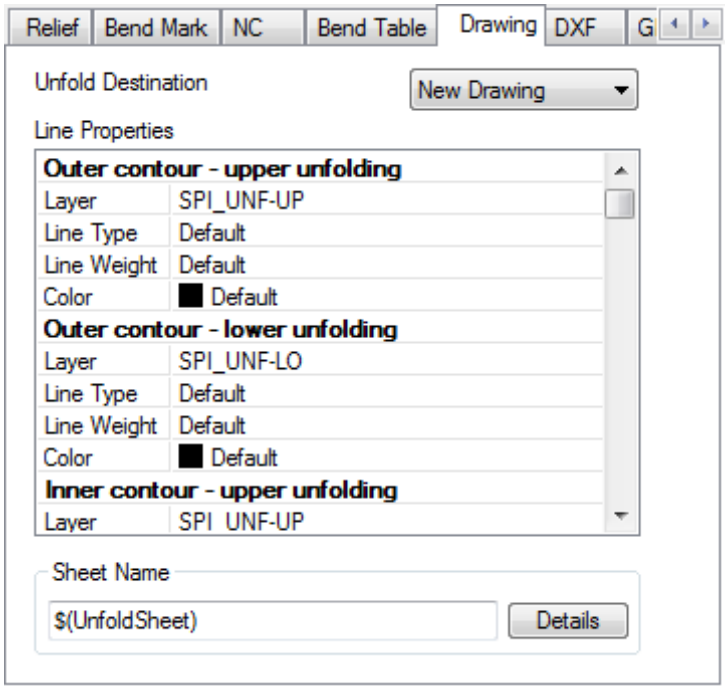
Numerical Order

The option **Numerical order** controls if the lines are ascending or descending.

Use the **Line Properties** in the tab **Drawing** to modify the color and layer of the bend table elements.

© SPI GmbH 2020

Drawing



Unfold Destination

This is the default unfold destination used in the command **Create Unfolding** when unfolding into a SOLIDWORKS document.

Line Properties

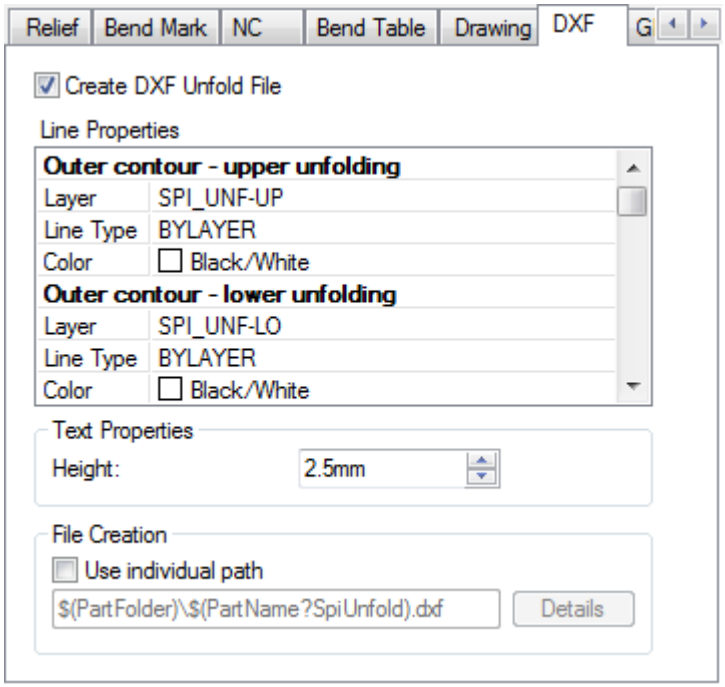
SPI SheetMetalWorks allows setting the line properties for several contour types in the unfold drawing.

Sheet Name

SPI SheetMetalWorks uses the **Sheet Name** parameter whenever you unfold in a drawing document. The name may contain variable fields, for example the name of the active SOLIDWORKS configuration. For this click on **Details** and edit the sheet name in the **Unfold Destination** dialog.

© SPI GmbH 2020

DXF



Create DXF Unfold File

If this option is set a DXF file with the unfolding will be created.

Line properties

SPI SheetMetalWorks allows setting the line properties for several contour types in the DXF file to create.

Text Properties

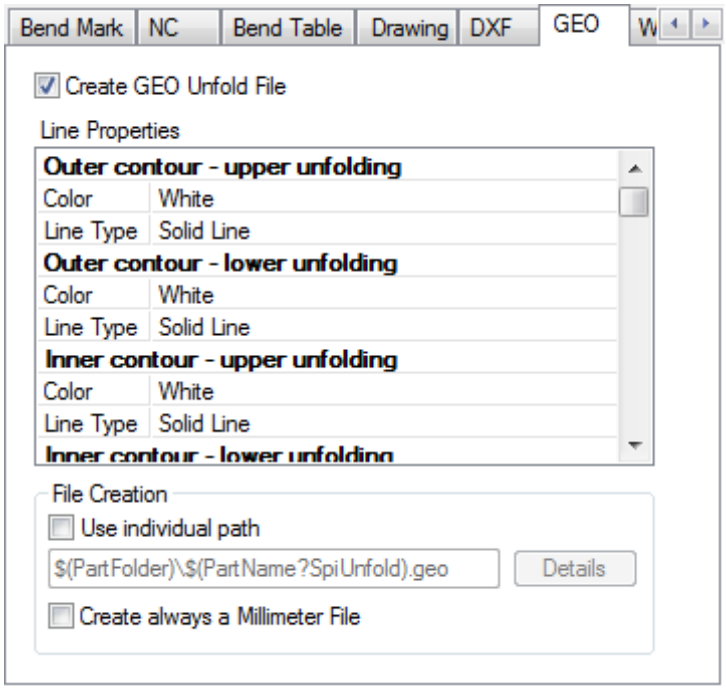
With the text property **Height** you can control the size of the texts, which will be created for the bend line information.

File creation

Activate the **Use individual path** option, if you like to define a special destination file path only for the active part. Otherwise SPI SheetMetalWorks uses the destination path defined in the global **Application Options**. The path can contain variable fields. For this click on **Details** and define the file path in the **Unfold Destination** dialog.

© SPI GmbH 2020

GEO



Create GEO Unfold File

If this option is set a GEO file with the unfolding will be created.

Line properties

SPI SheetMetalWorks allows setting the line properties for several contour types in the GEO file to create.

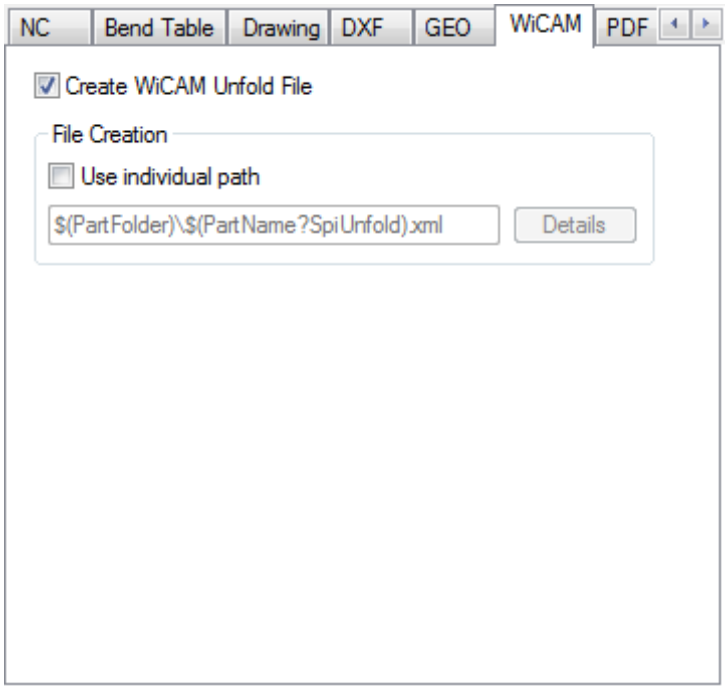
File creation

Activate the **Use individual path** option, if you like to define a special destination file path only for the active part. Otherwise SPI SheetMetalWorks uses the destination path from the global **Application Options**. The path can contain variable fields. For this click on **Details** and define the file path in the **Unfold Destination** dialog.

With the option **Create always a Millimeter File** you can control if the unfold GEO file to create has the **millimeter** units, even if the part has the **inch** units. The unfold file will normally created in the units of the part document. This option is useful if the unfolding should be imported into TruTops Bend.

© SPI GmbH 2020

WiCAM



Create WiCAM Unfold File

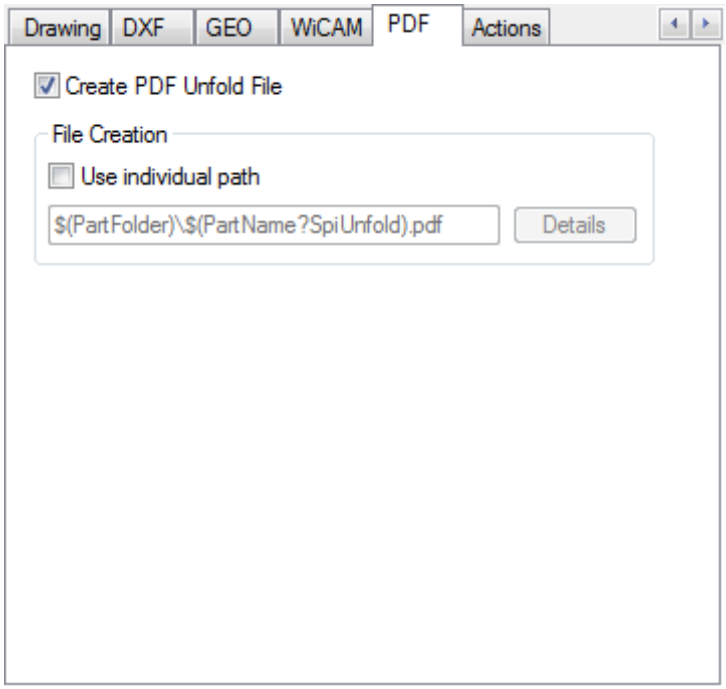
If this option is set a WiCAM XML file with the unfolding will be created.

File creation

Activate the **Use individual path** option, if you like to define a special destination file path only for the active part. Otherwise SPI SheetMetalWorks uses the destination path from the global **Application Options**. The path can contain variable fields. For this click on **Details** and define the file path in the **Unfold Destination** dialog.

© SPI GmbH 2020

PDF



Create PDF Unfold File

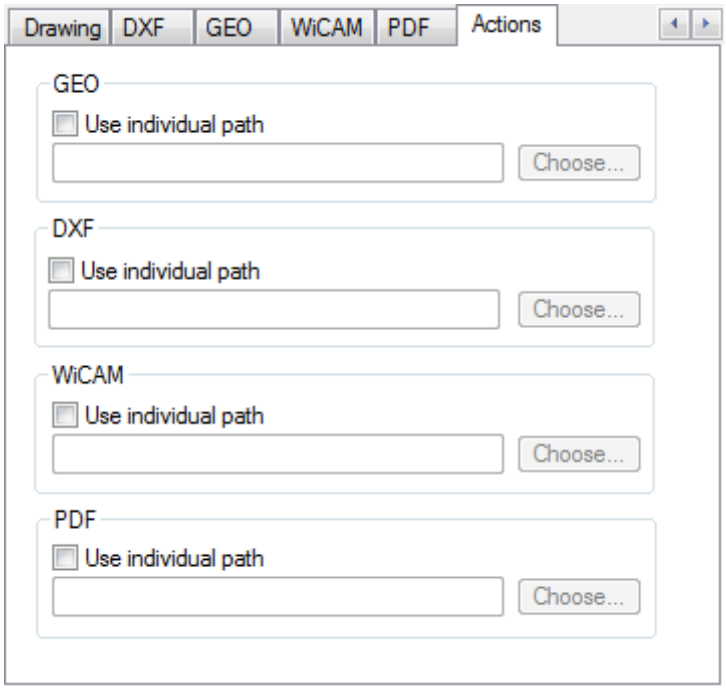
If this option is set a drawing with the unfolding will be created and saved as a PDF file.

File creation

Activate the **Use individual path** option, if you like to define a special destination file path only for the active part. Otherwise SPI SheetMetalWorks uses the destination path from the global **Application Options**. The path can contain variable fields. For this click on **Details** and define the file path in the **Unfold Destination** dialog.

© SPI GmbH 2020

Actions



For each file format, used as unfold destination, you may specify an external program which will be called automatically after the sheet metal part has been successfully unfolded. The name of the created unfold file will be supplied as argument to this program. These actions may be set globally in the **Application Options** or especially only for the actual part.

Activate the **Use individual path** option, if you like to define a special program name only for the actual part. Otherwise SPI SheetMetalWorks uses the program name specified in the **Application Options**.

© SPI GmbH 2020

Index

B

Base Sheet - New 34
Bend Line Dimen-sioning 86
Bend Lines 85
Bend Mark Attributes 43, 57
Bend Radius 34, 37, 52, 74
Bend Radius Attributes 43
Bend zone 68
Bend Zone 37
Bend Zone Lines 86
Bend Zone Width 68
Bending Tool 68

C

Configurations 81
Corner Relief 9
Corner Split Attributes 43, 48, 50
Correction Method 68
Cost Module 77
Cybelec 15

D

Delem 15
Drawing 21
DXF 15
DXF file 22

E

Error Protocol 24

F

Facet Attributes 43, 75
Fillet 65
Flange 36, 37
Flange parameters 36
Flange Relief 37
Freeform Surfaces 11, 74

G

GEO file 22

K

K-Factor 68

L

Laser Edge 9

M

Manufacturing Info Attributes 43
Material Data 34, 61, 82
Mixed Sharp Cornered 92

N

Neutral Fiber 7

O

Offset Attributes 43

P

PDF 22
Production Radius 34, 37, 52, 84
Profile– New 61
Profile Properties 61

R

Relief 8, 54
Relief Attributes 43

S

Sharp cornered 91
Shortening Attributes 43, 68
Sketch Attributes 43
Split Face 48

U

Unfold Destination 21
Unfold Parameters 81
- Actions 115
- Bend Mark 103
- Bend Table 109
- Display 85
- Drawing 110
- DXF 111
- Freeform surfaces 87
- General 82
- GEO 112
- NC 107
- PDF 114
- Post Processing 93
- Precision 90
- Relief 96
- WiCAM 113

Unfolding 6, 14, 15, 21
Unfolding Log 23, 24, 42
Update Unfolding 69
User Interface 16

W

WiCAM 22

© SPI GmbH 2020