

SPI SheetMetalWorks 2021

Getting Started

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Introduction

This Tutorial addresses to SolidWorks users who intensively deal with sheet metal construction or -production.

About this tutorial

This tutorial is meant to give you an understanding of the handling and the power of **SPI SheetMetalWorks**. It is divided into two sections:

Section 1

Section 1 gives a review of the installation of **SPI SheetMetalWorks**.

Section 2

Section 2 will give you an idea of the practical usage of **SPI SheetMetalWorks** by the explanation of some simple examples. All mentioned examples are located in the SPI installation subfolder `Documentation\Examples`.

The first example shows some possibilities of the processing of a sheet metal part which has been designed with SOLIDWORKS commands.

The second example shows the handling of a sharp-edged body using the unique possibilities of **SPI SheetMetalWorks**.

In the third and fourth example it will be shown how to create and unfold transitional parts with freeform surfaces.

Prerequisites

This document is designed for users with the following prerequisites:

- Experience in the use of SOLIDWORKS
- Knowledge about the design and manufacturing of sheet metal products.
- Knowledge of the Windows™ operating system

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Section 1
Installation

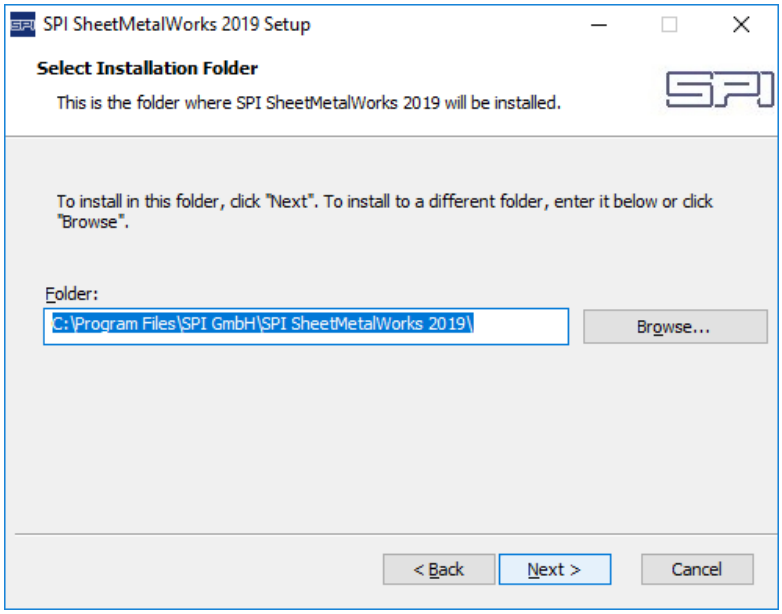
You get some indications for installation of **SPI SheetMetalWorks** in this chapter. It makes no difference, whether you carry out a test installation for getting to know the product, or whether you would like to install the software for the permanent usage.

Please extract the downloaded ZIP file and execute the EXE file.



Installation folder

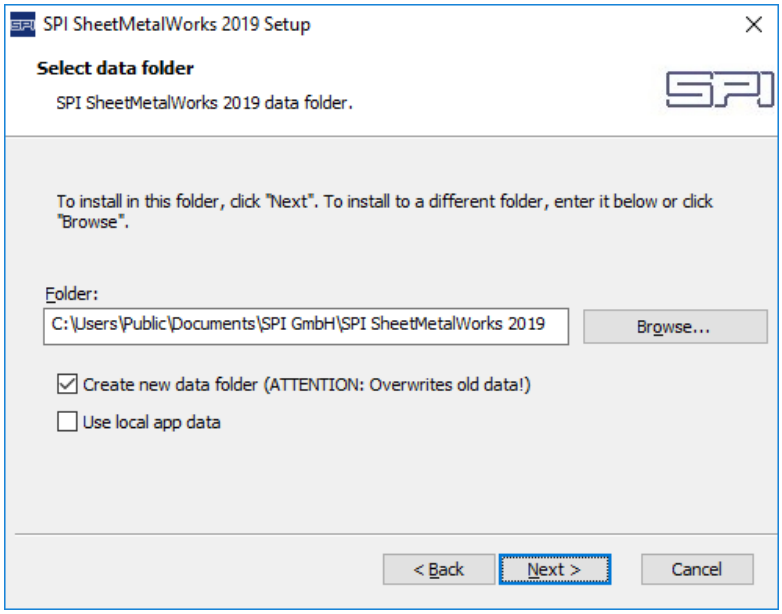
If desired, you may change the proposed installation folder.



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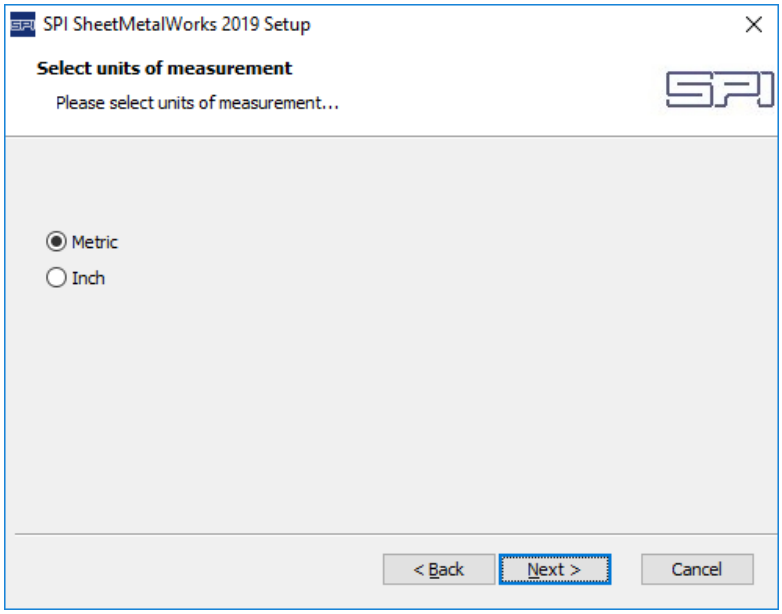
SPI data folder

This dialog will prompt you to set the data folder where the general SPI data will be located, for example the SPI material data. You can create a new data folder or use a folder of an older installation.



Units of measurement

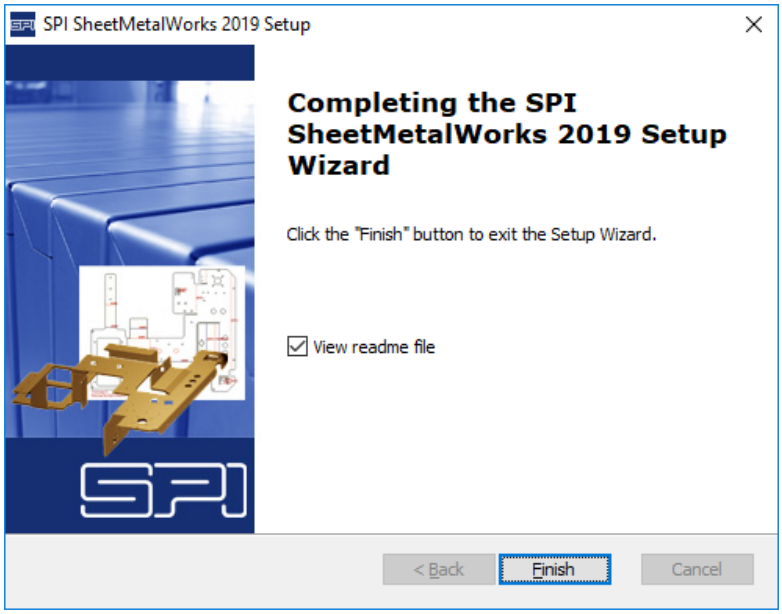
Select the units of measurement you want to use for your construction.



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Finish

The installation is done. Please register the software as described below.



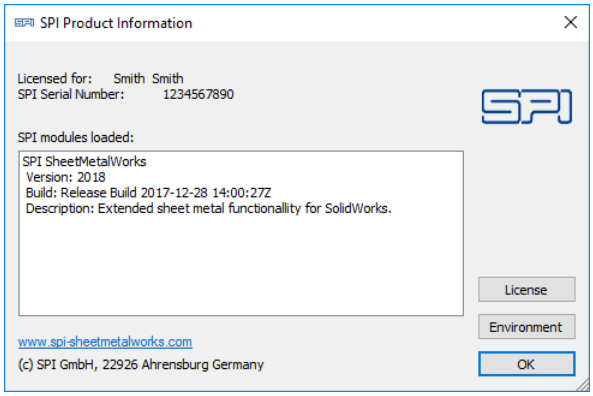
Please view the README file when finishing the installation.

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Registration

You can launch the registration process by using the **About SPI SheetMetalWorks** command in the SOLIDWORKS **Help** menu. Please click the button **License**:

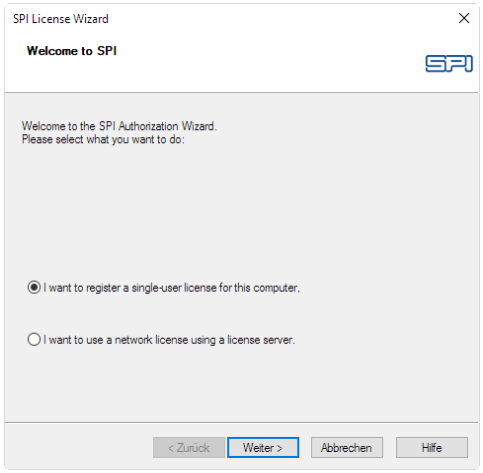
About SPI SheetMetalWorks



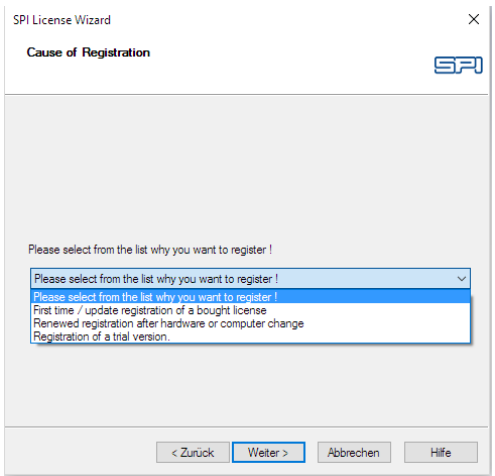
This will bring up the **SPI License Wizard**. Alternatively, the License Wizard will start automatically when executing any SPI command and the registration has not yet been done.

SPI License Wizard

Using the option *register a single-user license* enables you to attach you license to a computer or a dongle. Furthermore, it is possible to obtain a license from a SPI License Server. For this you have to install the SPI License Server first. View the README file for further information.



Please, process this registration in any case, either if it is a trial installation for getting to know the SPI software, or if you have already purchased the software!

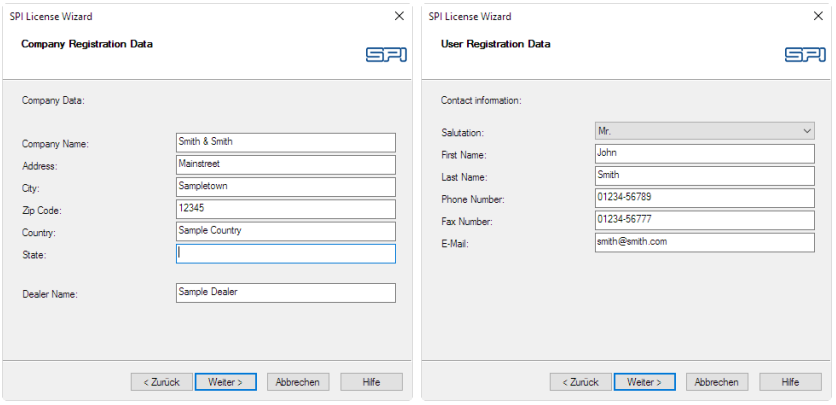


In case of a *single-user license* please make a choice in the following dialog:

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Company's data

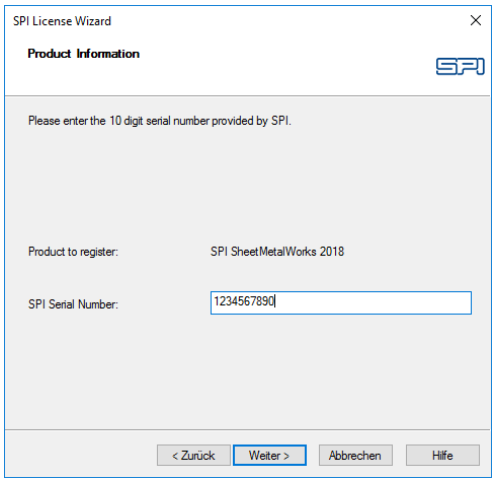
Please, fill out the two form pages with details on your person and company:



Your Email address will be needed for the automatic password processing.

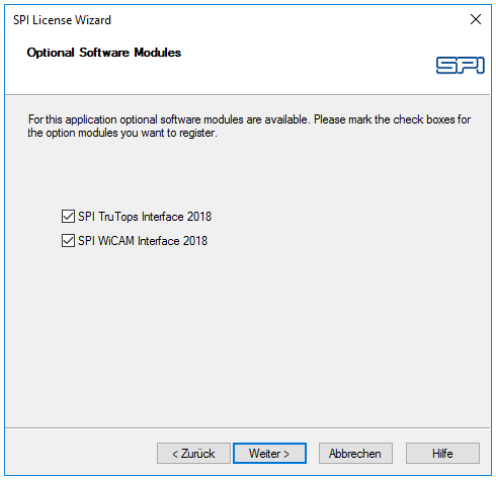
Serial number

If you have already purchased the software, please use your personal serial number. You find it on the delivery note. If necessary, your SOLIDWORKS reseller can provide the serial number.



Optional Software Modules

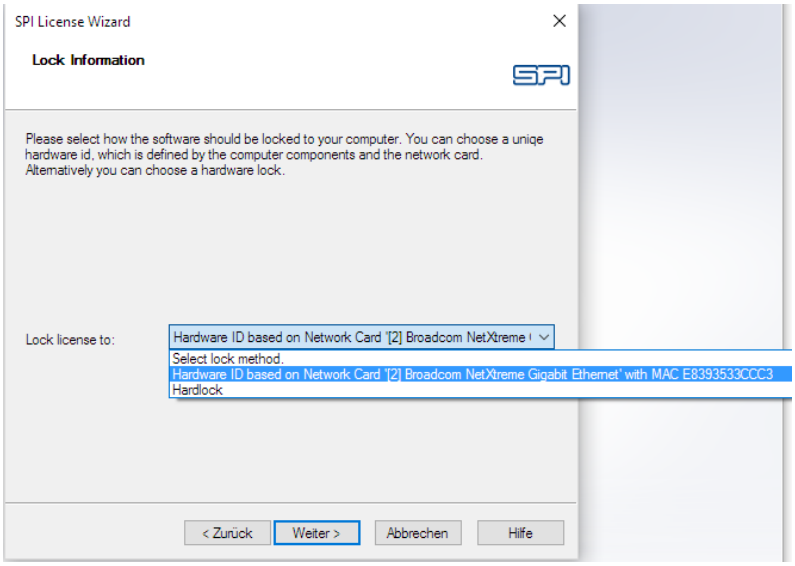
If you have purchased additional software modules, please mark here your modules to get your passwords from SPI.



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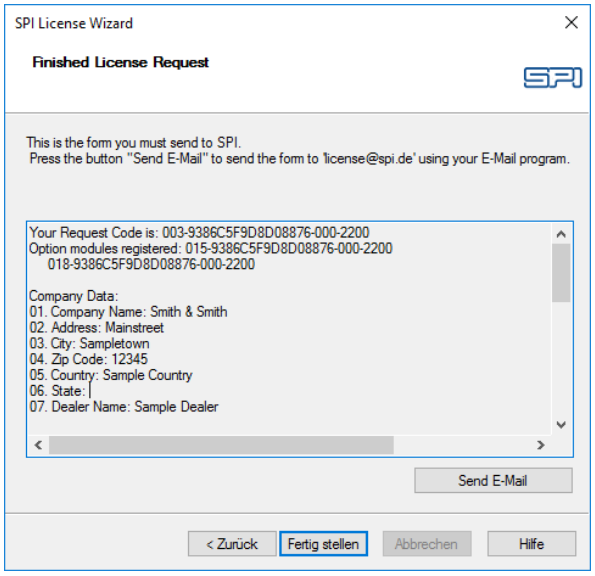
Lock method

The next step is your choice regarding the lock method:



Besides attaching the license to the unique *Hardware ID* of the computer it is also possible to use a *Hardlock*, also known as a dongle. An USB dongle can be ordered with costs at SPI. This will allow using one license on more than one computer, for example in the office and at home.

The registration is prepared completely, so the license request can be sent:



If the computer that has been used during the installation allows sending an email, please press the button “Send E-Mail” (see above). Alternatively, you can copy and paste the text into a text file and send the mail from another seat.

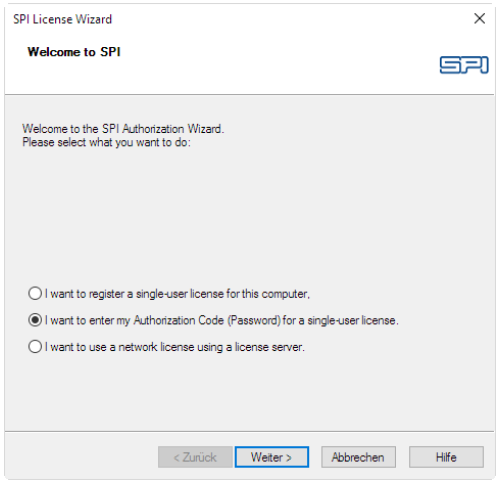
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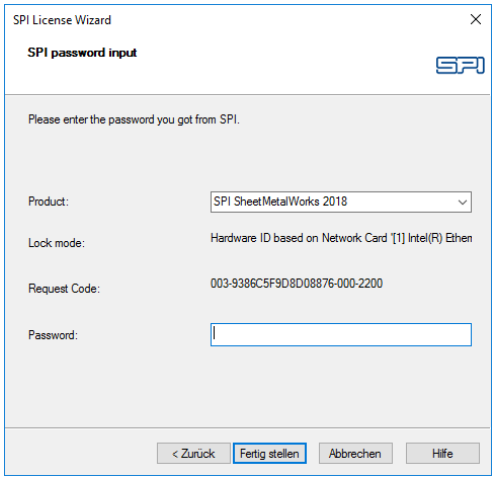
license@spi.de

Please send this text as content of an email, not as an attached text file, to the address *license@spi.de*". This will allow automatic processing of the request.

Enter Authorization Code

The next time, you execute any SPI command the SPI License Wizard offers to enter your Authorization Code:





The password can be either an unlimited code, consisting of 8 digits, or a code for a limited period of time. In this case please copy and paste the code including the date, for example 2020/03/15:838DE123. During the trial period the software will work without any restrictions.

When you receive the unlimited password, you can launch the SPI License Wizard at any time within the **About SPI SheetMetalWorks** dialog and enter the password.

Now you have registered and activated the software, so you are ready for *Getting Started* with **SPI SheetMetalWorks**!

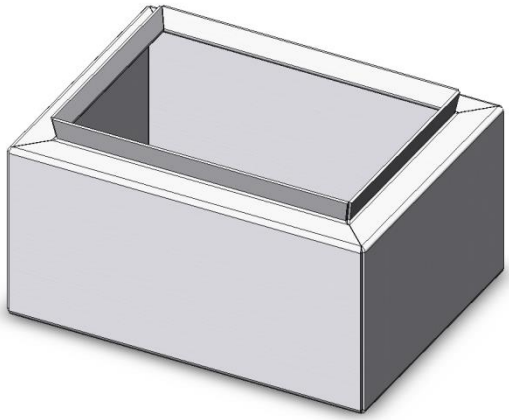
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Section 2

Sample 1: Enclosure

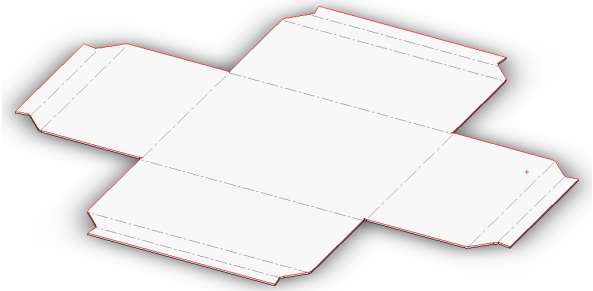
Please open the sample file **`enclosure.sldprt`**.

This is an ordinary sheet metal part. It has been made with common SOLIDWORKS commands, like Base Flange … Miter Flange, etc.:



SOLIDWORKS unfolding

Of course SOLIDWORKS is able to unfold the part:



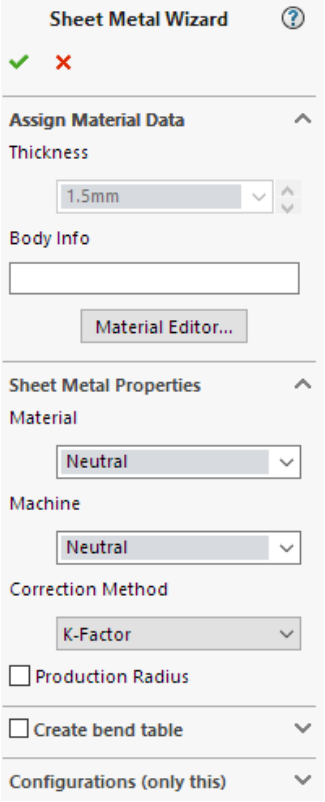
Sheet Metal Wizard

Please start the **SPI Sheet Metal Wizard**. This can be done within the **SPI SheetMetalWorks** tab in the SOLIDWORKS Command Manager.

Please note that all commands that are not suitable in this situation are deactivated to guide the user.

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Assign Material Data



The part already contains SOLIDWORKS sheet metal data. In this case the **Thickness** is displayed in the Property Manager and connot be modified. A modification is only possible by editing the SOLIDWORKS Sheet Metal Feature.

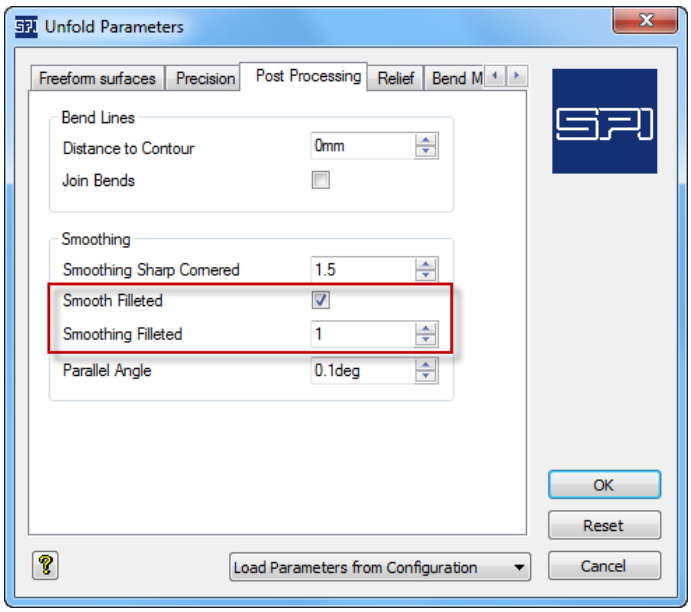
The Sheet Metal Wizard is especially meant to assign the desired **Material Data**. The bend deduction can be calculated using one of the following methods: *K-Factor, Formula, Table, TruTops Table or DIN 6935*.

You may **create and assign a bend table** containing the shortening values which will be used by the SOLIDWORKS Sheet Metal Feature.

Please notice, that more SPI commands are available after the material assignment.

Unfold Parameters

Start the SPI command **Unfold Parameters**. In the tab **Post Processing** you have to activate the parameter **Smooth Filleted** and set the value of **Smoothing Filleted** to 1:



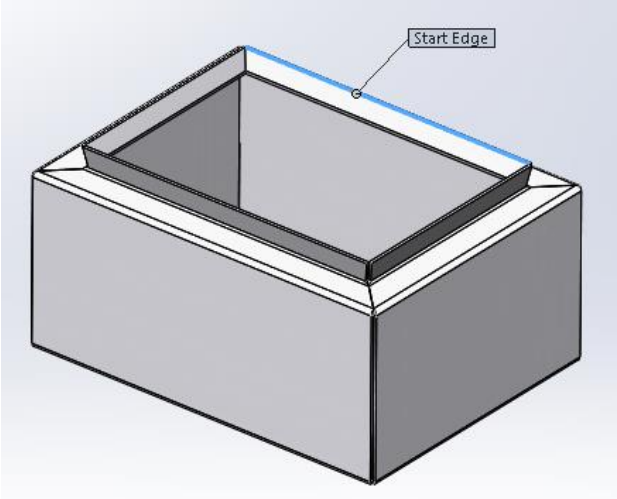
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Create Unfolding

Now **SPI SheetMetalWorks** can unfold the part. Please execute the command **Create Unfolding**.

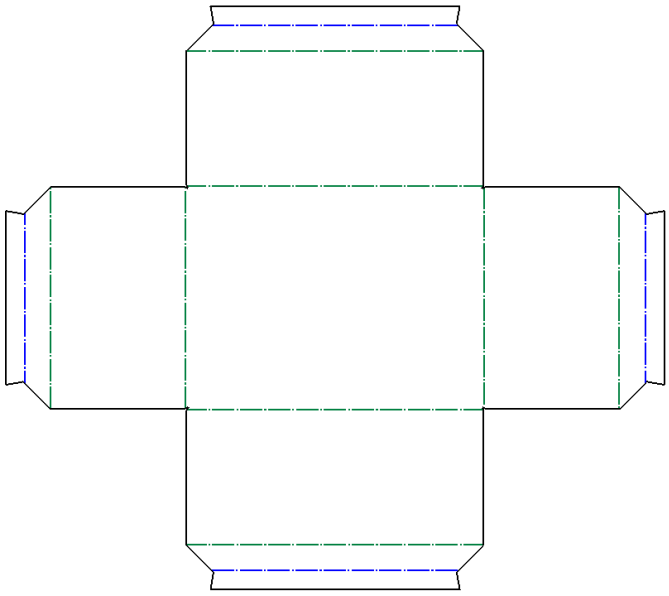
Start edge

Select one straight edge (so called Start Edge). This edge will be horizontal in the flat pattern:



Having chosen SOLIDWORKS Document - *New Drawing* as **Unfold Destination** you'll see the flat pattern in a SOLIDWORKS drawing document:

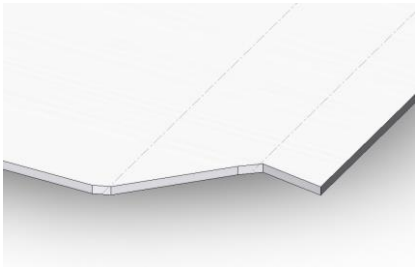
SPI Unfolding



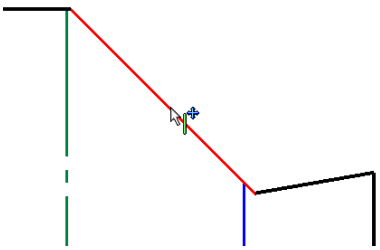
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Details of the flat pattern

Please compare the unfolding of SOLIDWORKS (left picture) and the SPI unfolding (right picture):



The ending of the flange and the bend zone can be seen as single plane faces.



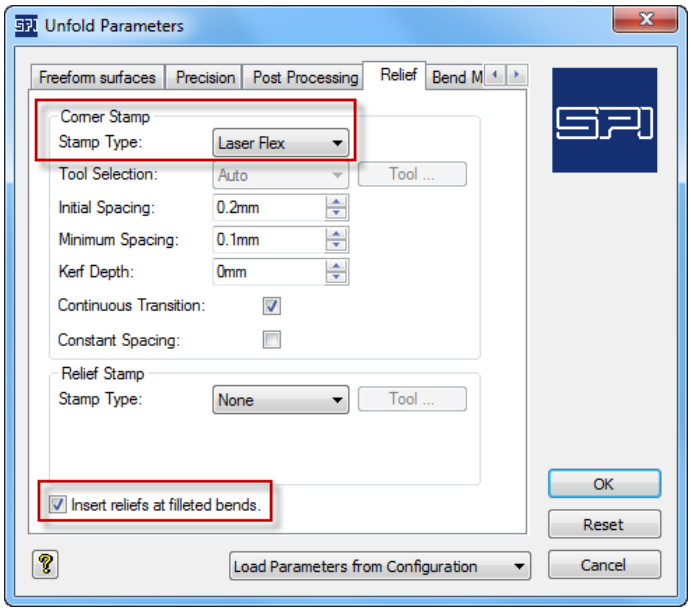
The contour is simplified to save tool operations when punching out the flat pattern.

Switch back to the 3D model and launch the SPI command **Unfold Parameters**:

Corner Relief

At first set **Stamp Type** of the **Corner Stamps** to *Laser Flex*.

To let **SPI SheetMetalWorks** insert reliefs also in our case of filleted bend zones, activate the option **Insert reliefs at filleted bends**:

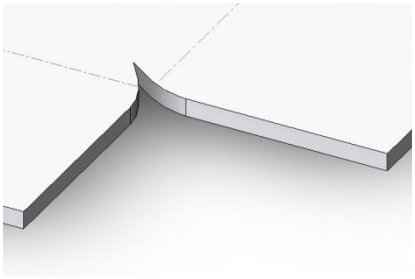


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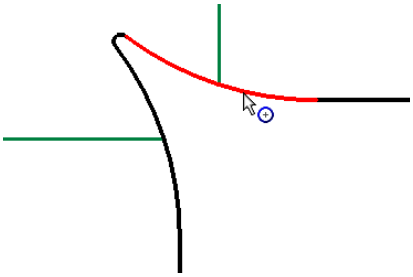
Update Unfolding

Execute the SPI command **Update Unfolding**.

Now, let's compare the unfolding details in the corners:



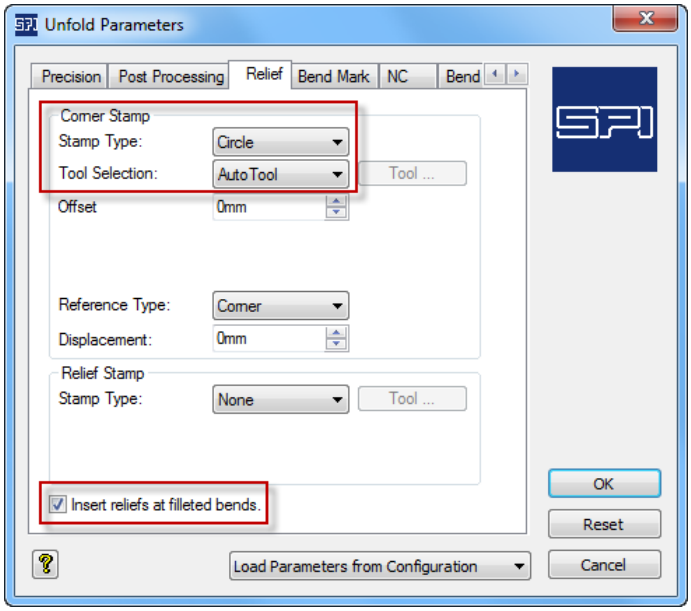
The corner is composed of two splines that meet in one point.



The corner contour contains circular arcs and tangent lines – this will yield a perfect NC program in the end!

Relief with circular stamp

Please have another look into the **Unfold Parameters** and set the **Stamp Type** to *Circle*:

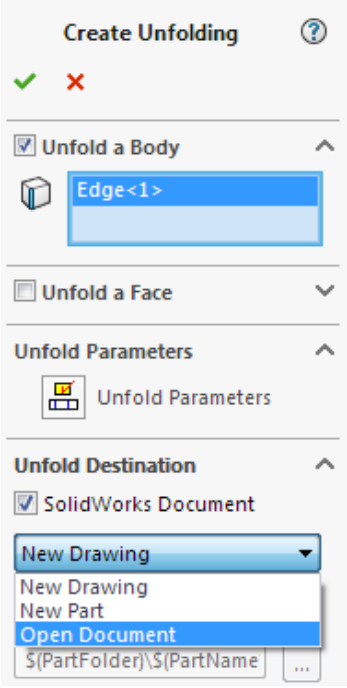


Now the unfolding can be updated using the **Update Unfolding** command. However we will store the unfolding into a drawing of the part.

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Unfold into open document

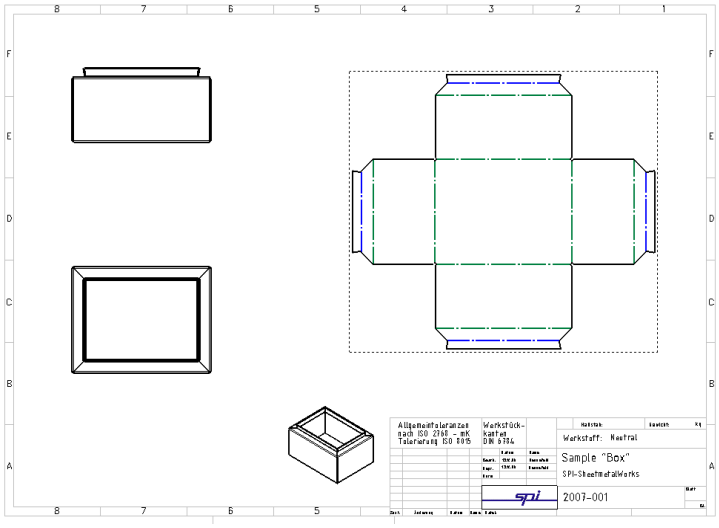
Please create an ordinary SOLIDWORKS drawing of the sheet metal part. Then toggle to the part and execute the command **Create Unfolding**:



Since the part has been unfolded before, the **Unfold Destination** is preset to *Update*. Toggle the **Unfold Destination** to *Open Document* and choose the drawing file.

Please take into consideration, that the unfolding can be put directly into a DXF or GEO file. Also an export of the unfolding into a WiCAM XML file is possible.

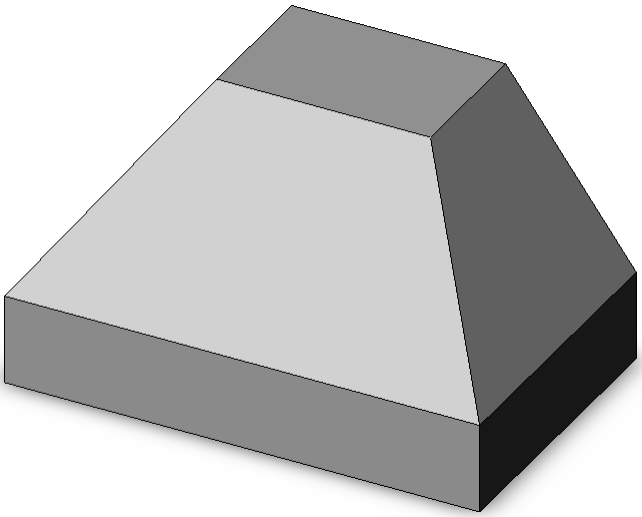
The unfolding is put into the SOLIDWORKS drawing in the drawing's scale:



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Sample 2: Fan Hood

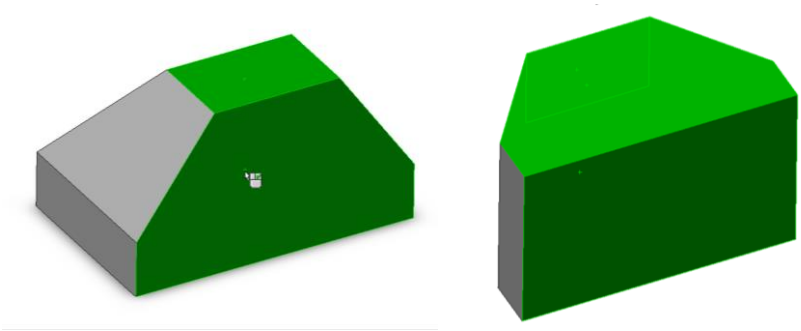
In this example you will transfer a solid body into a sheet metal part. We will use the shape of a typical fan hood:



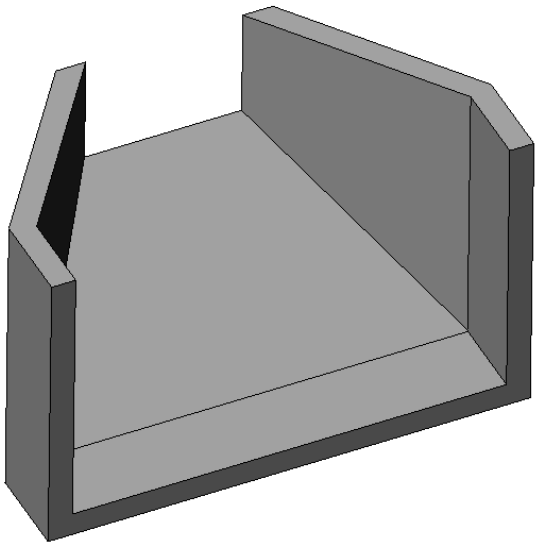
Please open the file **`hood (design body).sldprt`**. Now let's perform the shelling command twice to make a sharp edged sheet metal part:

1st Shelling

Execute the SOLIDWORKS command **Shell**, set the **Thickness** to *20mm* and select the faces to remove on these sides: Top, back and bottom:



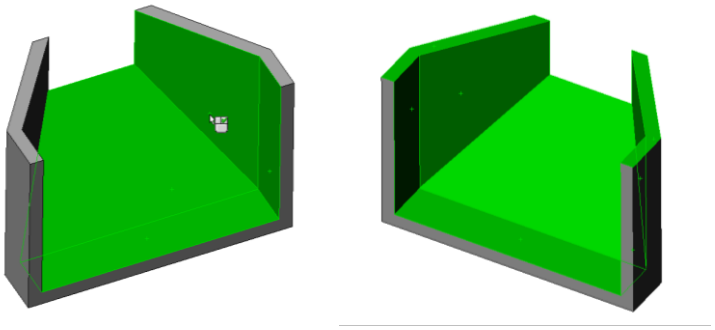
After that the part looks this way:



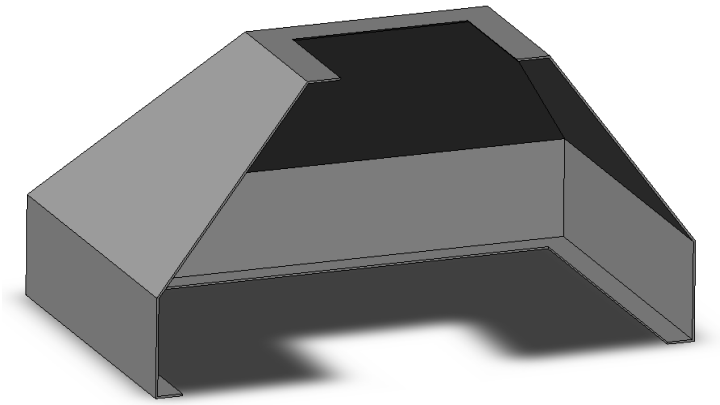
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2nd Shelling with desired thickness

Execute the **Shell** command once again and set the **Thickness** to the desired sheet metal thickness, for example *1mm*. Please remove the six faces on the inside and the two faces on the rear side:



Now the part has the desired shape and thickness:



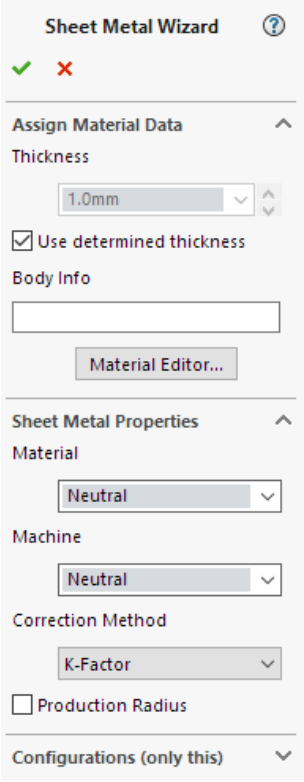
SPI Sheet Metal Wizard

Execute the SPI Sheet Metal Wizard to assign the desired material data:

All other SPI commands are disabled as long as no material data were assigned to the part.

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Material data for shelling body



The thickness is obtained from the geometry automatically. If this thickness is not the correct one you may enter the correct thickness or determine it by selecting a face. For this uncheck the option *Use determined thickness*.

However, it cannot be modified directly since the part has no SOLIDWORKS sheet metal feature.

All sharp edges will be treated later during the unfolding as bend with standard bend radius.

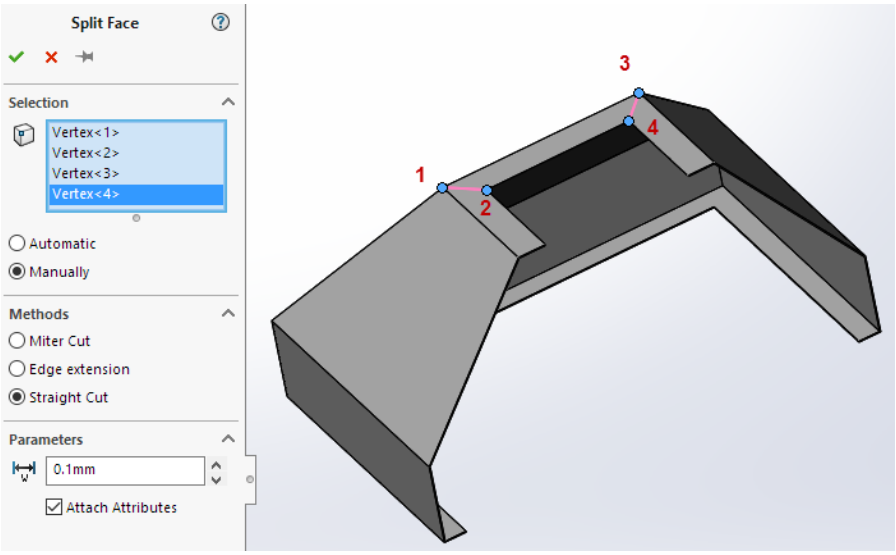
This applies to parts that have been designed within SOLIDWORKS without sheet metal features, as well as to imported bodies. The thickness is evaluated at the selected face.

Now only corner split data is needed to unfold the part.

Split Face

On the top and bottom sides the collar face is to be splitted into miter flanges. This can be achieved quickly with the SPI command **Split Face**.

Set the option *Manually* and *Straight Cut*. Then pick the points 1 to 4 at the face that is to be split:

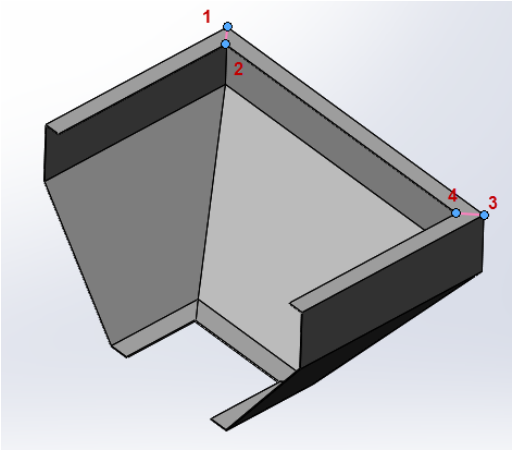


The preview indicates the way **SPI SheetMetalWorks** will split the face. It will insert a sketch, a split face feature and store the necessary corner split attributes.

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Split Face

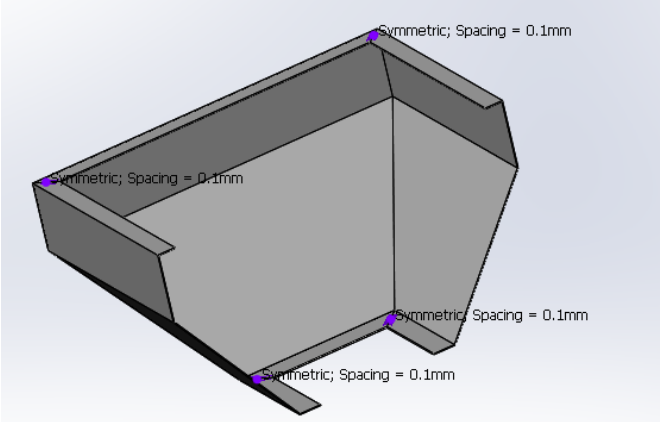
Please use the **Split Face** command again to modify the bottom side, too:



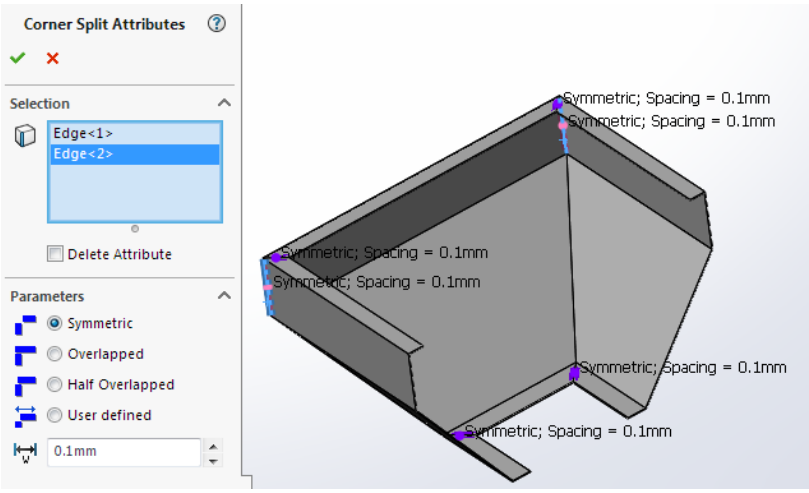
Corner Split Attributes

Use the SPI command **Corner Split Attribute**.

The beforehand used command **Split Face** has already made some **Corner Split Attributes**, which are displayed now:



Now pick the two vertical edges. It makes no difference whether the inner or outer edge is used:



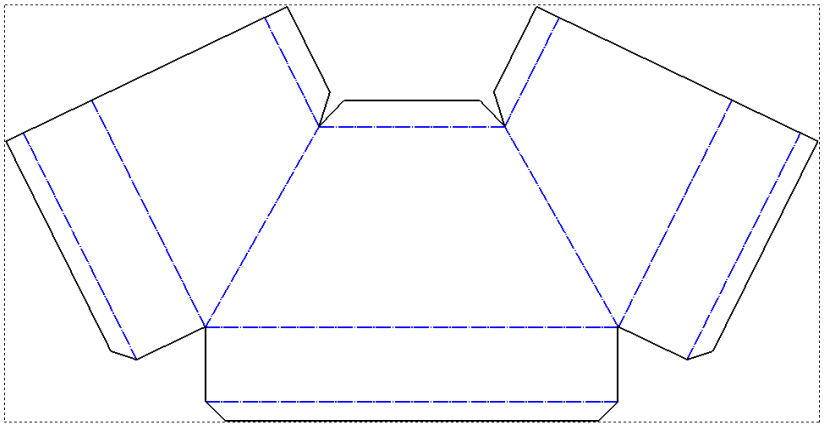
You can repeat this command at any time to add further or delete the existing Corner Split Attributes.

Please notice, that these elements are only attributes, so to say virtual rips that will be used when unfolding the part.

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Unfolding

Please run the SPI command **Create Unfolding** to obtain a perfect flat pattern of the fan hood:



Insert bend table

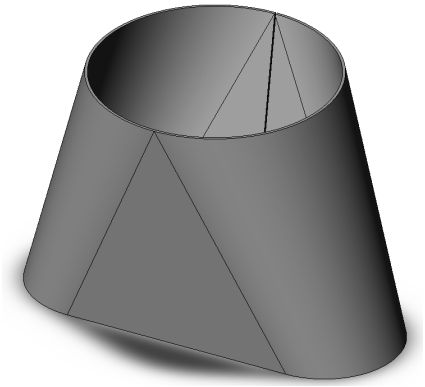
To modify details of the flat pattern, please switch back to the 3D part. Change the **Unfold Parameters** in the tab **Bend Table.** Activate the parameter **Bend Line Table** to create a bend line table in the unfolding. Create the unfolding again by invoking the command **Update Unfolding**. After this the bend lines in the unfolding will be numbered. In addition to the unfold contour the bend table will be created. There the bend line numbers with the corresponding bend information are listed:

Material:	Neutral		
Bending Tool:	Neutral		
Thickness:	1mm		
Part Name:	hood (design body)		
No.	Bend Angle	Radius	Shortening
11	-49.09deg	2mm	-0.6mm
10	-40.91deg	2mm	-0.45mm
9	-90deg	2mm	-2.07mm
8	-67.21deg	2mm	-1.05mm
7	-36.25deg	2mm	-0.38mm
6	-90deg	2mm	-2.07mm
5	-53.75deg	2mm	-0.7mm
4	-67.21deg	2mm	-1.05mm
3	-36.25deg	2mm	-0.38mm
2	-90deg	2mm	-2.07mm
1	-53.75deg	2mm	-0.7mm

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Sample 3: Unfolding of a transition part

Please open the part **`Nozzle.sldprt`**:



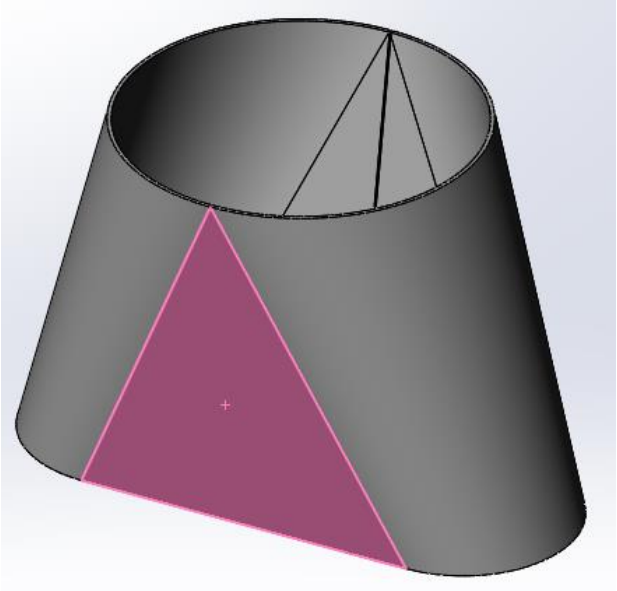
If you like to draft this part for practice yourself, please take care of these details:

The circle on the upper side is split into two halves. The lofting needs four guide lines to obtain plane side faces. The option **Merge tangent faces** within the **Loft** command must be deactivated.

A gap is needed to unfold the part completely in one row.

SPI Sheet Metal Wizard

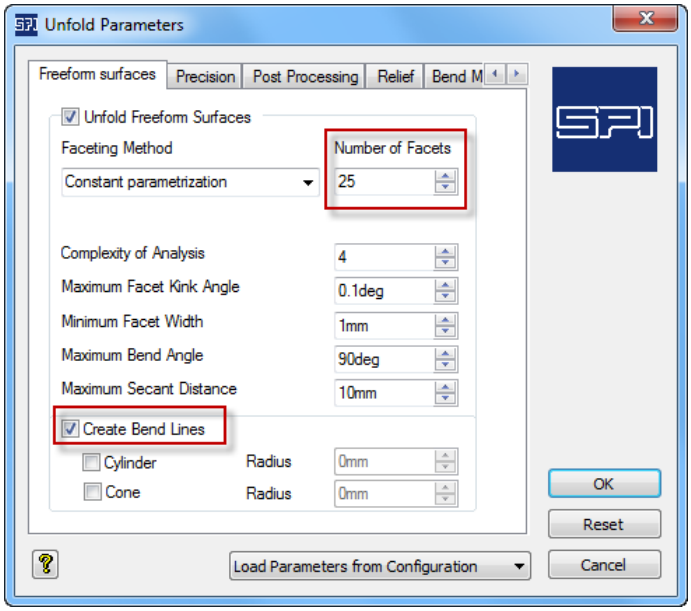
Execute the **SPI Sheet Metal Wizard** to assign the desired material data. Select a face to determine the thickness and assign material data:



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Freeform Surfaces

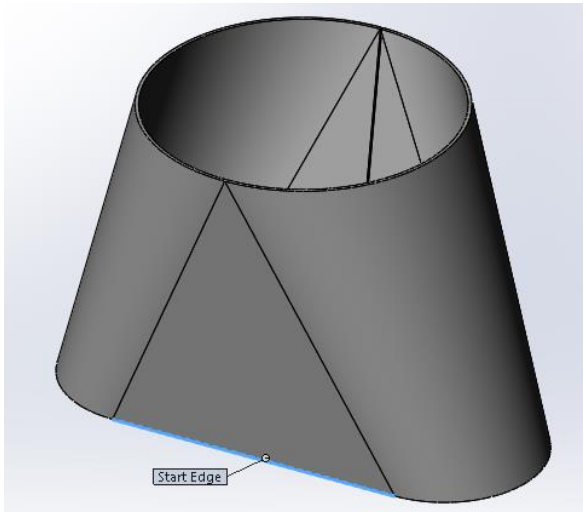
Please check the settings in the **Unfold Parameters** regarding the unfolding of **Freeform Surfaces**:



The **Number of Facets** represents the number of bends that will be used for all curved surfaces.

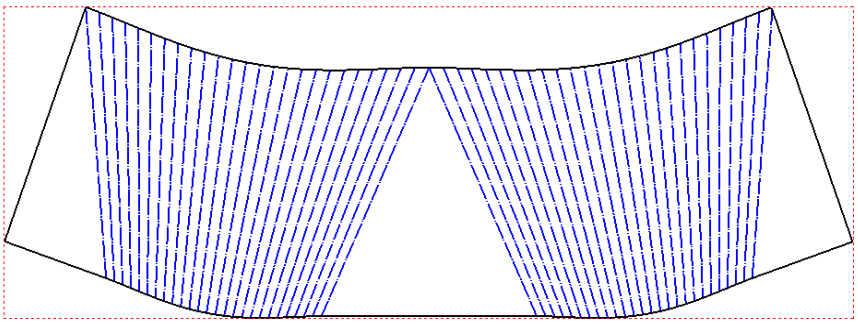
Create Unfolding

Now all prerequisites are fulfilled to unfold the part. The **Start Edge** needs to be straight:



The result is a flat pattern that is applicable for bending on a bend press:

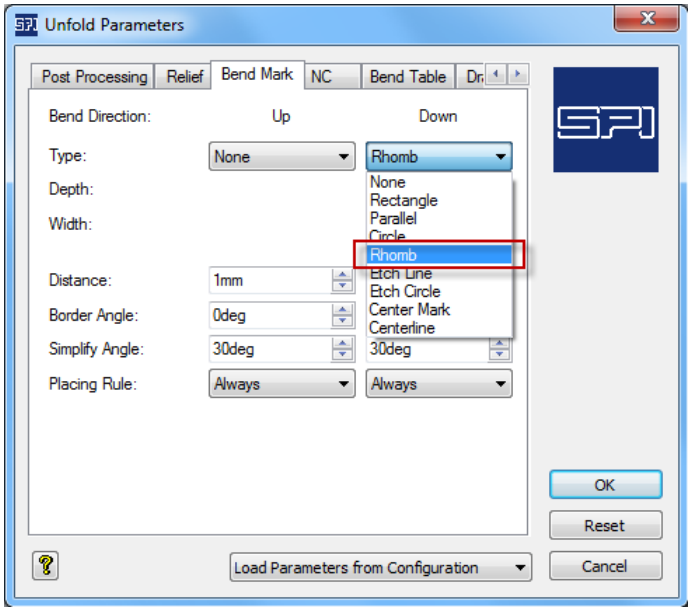
Unfolding of freeform surfaces



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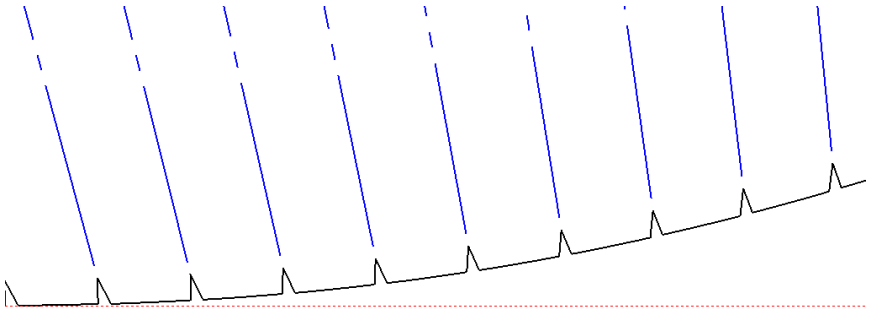
Bend Marks

To manufacture parts like this on a press brake, it is useful to have little notches in the contour that mark the bend line. Please switch back to the 3D part and modify the **Unfold Parameters**. The tab **Bend Mark** allows activating the bend marks. Select *Rhomb* as type for the bend mark:



When selecting *Etch Line* or *Etch Circle* as bend mark type, the bend marks will only be created as small sketch elements. *Rhomb* will put small notches into the contour at the end of the bend lines:

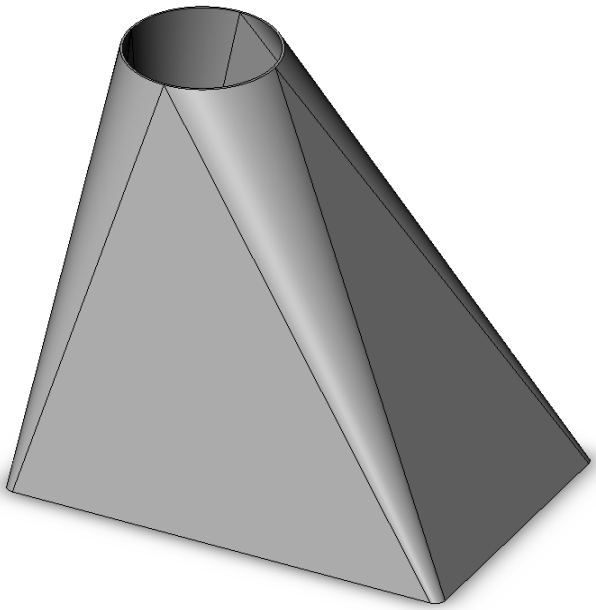
Unfolding with Bend Marks



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Sample 4: Round-Square-Transition

SPI SheetMetalWorks is able to unfold various Round-Square transitions. Please have a look at the sample file **`transition.sldprt`**. Assign material data. The centers of the two apertures are offset:



In some cases it is necessary to increase the values for the precision on the tab **Freeform Surface** within the **Unfold Parameters**.

Outlook

We hope that this document has provided you a first glance on the possibilities and concepts of **SPI SheetMetalWorks**.

To continue your orientation we recommend the comprehensive Tutorial for SPI SheetMetalWorks that can be accessed from the SOLIDWORKS Help menu. Additionally you will find the **Help for SPI SheetMetalWorks** in this menu.

All commands allow you to open up the related help file using the ? within the Property Manager.

We wish you a lot of fun and success with **SPI SheetMetalWorks**!

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