

SHEET METAL IN SOLIDWORKS

Sheet metal parts are generally used as enclosures for components or to provide support to other components. When you create a sheet metal part in SolidWorks, you generally design the part in the folded state. This allows you to capture the design intent and the dimensions of the finished part.

Base Flange

A base flange is the first feature in a new sheet metal part. When you add a base flange feature to a SolidWorks part, the part is marked as a sheet metal part. Bends are added wherever appropriate, and sheet metal specific features are added to the FeatureManager design tree.

Note:

- The Base Flange feature is created from a sketch. The sketch can be a single open, a single closed, or multiple-enclosed profiles.
- There can be only one base flange feature in a SolidWorks part.
- The thickness and bend radius of the Base Flange feature become the default values for the other sheet metal features.

A Base Flange feature creates three new features in the FeatureManager design tree:



Sheet-Metal1. The Sheet-Metal feature contains the default bend parameters. To edit the default bend radius, bend allowance, bend deduction, or default relief type, right-click the Sheet-Metal feature and select Edit Definition.

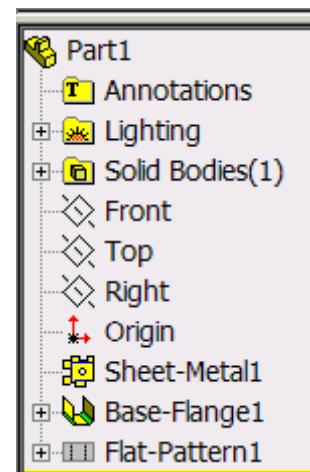


Base-Flange. The Base-Flange feature is the first solid feature of this sheet metal part.



Flat-Pattern1. The Flat-Pattern feature flattens the sheet metal part. Notice it is suppressed by default as the part is in its bent state. Unsuppress the feature to flatten the sheet metal part.

When the Flat-Pattern feature is suppressed, all new features that you add to the part are automatically inserted above the Flat-Pattern feature in the FeatureManager design tree. When the Flat-Pattern feature is unsuppressed, all new features go below it in the FeatureManager design tree and are not shown in the folded part.

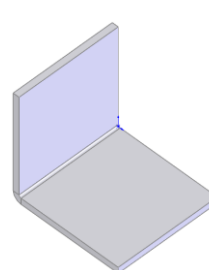


Edge Flange

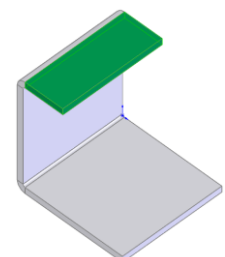
The Edge Flange feature adds a flange to your sheet metal part at an edge that you select.

Note:

- The selected edge must be linear.
- The thickness is automatically linked to the thickness of the sheet metal part.
- One sketch line of the profile must lie on the selected edge.



Initial Part



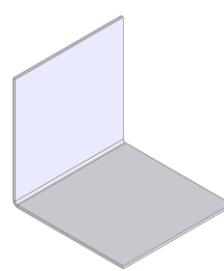
Edge Flange Added

Miter Flange

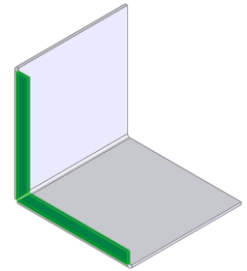
A miter flange feature adds a series of flanges to one or more edges of a sheet metal part.

Note:

- The sketch for a miter flange must adhere to the following requirements:
 - The sketch can contain lines or arcs (arc cannot be tangent to the thickness edge)
 - The Miter Flange profile can contain more than one continuous line. For example, it can be an L-shaped profile.
 - The sketch plane must be normal to the first edge where the Miter Flange is created.
- You can create a miter flange feature on a series of tangent or non-tangent edges



Initial Part



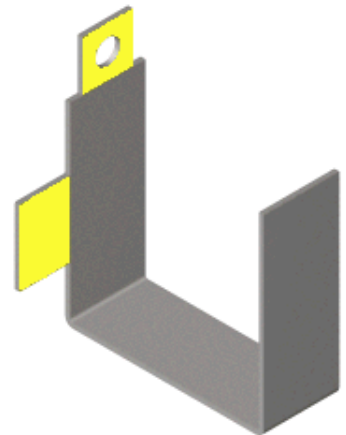
Miter Flange Added

Sheet Metal Tab

A Tab feature adds a tab to the sheet metal part. The depth of a Tab feature is automatically set to the thickness of the sheet metal part. The direction of the depth automatically coincides with the sheet metal part to prevent a disjoint body.

Note:

- The sketch can be a single closed, multiple closed, or multiple-enclosed profile. The illustration shows a single tab feature that adds two tabs to the sheet metal part.
- The sketch must be on a plane or planar face that is perpendicular to the direction of thickness of the sheet metal part.
- You can edit the sketch, but you cannot edit the definition. This is because the depth, direction, and other parameters are set to match the parameters of the sheet metal part.

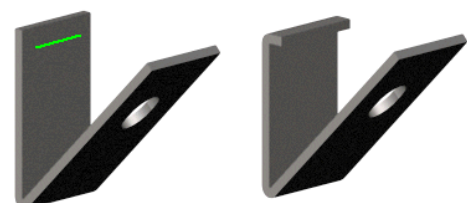


Sketched Bend

You can add bend lines to the sheet metal part while the part is in its folded state with a sketched bend feature. This allows you to dimension the bend line to other folded-up geometry.

Note:

- Only lines are allowed in the sketch. You can add more than one line per sketch.
- The bend line does not have to be the exact length of the faces you are bending.

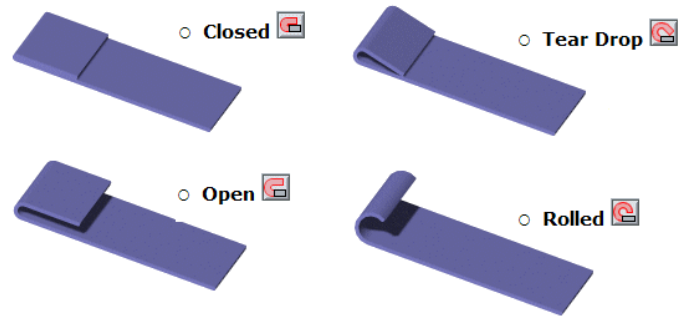


Hem

The Hem tool adds a hem to your sheet metal part at a selected edge.

Note:

- The selected edge must be linear.
- Mitered corners are automatically added to intersecting hems.
- If you select multiple edges to add a hem, the edges must lie on the same face.

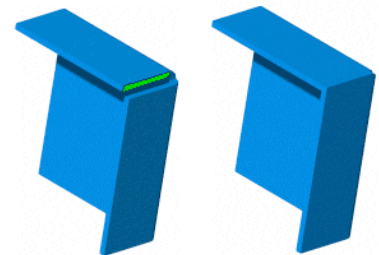


Closed Corner

You can create a Closed Corner feature to extend one face of a Butt rip so it overlaps the other face of the Butt rip.

Note:

- You can close more than one corner at a time. Select the faces for all of the rips that you want to close.
- You can select only planar faces to close. The planar faces must be perpendicular to each other.
- A closed corner cannot be applied in some cases where flange angles are not 90°.



Flattening Sheet Metal Bends

You can flatten the bends in a sheet metal part in the following ways:

- To flatten the entire part, if the **Flat-Pattern** feature is present, unsuppress **Flat-Pattern**, or click **Flattened**  on the Sheet Metal toolbar. The bend lines are shown by default when you unsuppress the **Flat-Pattern** feature. To hide the bend lines, expand **Flat-Pattern**, right-click **Bend-Lines**, and select **Hide Sketch**.
- To flatten the entire part, if the **Process-Bends** feature is present, suppress the **Process-Bends** feature, or click **Flattened**  on the Sheet Metal toolbar.
- To flatten one or more individual bends, add an **Unfold**  feature.

Approaches to Sheet Metal Part Design

There are two ways to create a sheet metal part:

1. Build a part, then convert it to sheet metal.
2. Create the part as a sheet metal part using sheet metal-specific features. This eliminates extra steps because you create a part as sheet metal from the initial design stage.

There are links in SolidWorks on-line help providing information on the following:

- **Design a Part from the Flattened State, then Convert it to Sheet Metal**
- **Design a Sheet Metal Part from the Flattened State**
- **Design a Part from a Solid, then Convert it to Sheet Metal**
- **Design a Sheet Metal Part from a Solid**
- **Reasons to Build a Part, then Convert the Part to Sheet Metal**
- **Combining the Different Sheet Metal Design Methods**

Sheet Metal Parameters

Once a Sheet-Metal feature is created default bend parameters are applied and used for other sheet metal features. These parameters contain the thickness, default bend radius, bend allowance, bend deduction, or relief type. For any particular bend, specifying custom radius or bend values can change these parameters.

Bend Allowance

You can specify the bend allowance or bend deduction values for your part in a variety of ways.

Bend Table

K-Factor

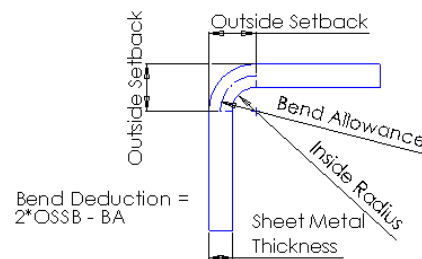
Bend Allowance

Bend Deduction

Bend Allowance Value

You can specify an explicit bend allowance for any sheet metal bend by entering the value when you create the bend.

NOTE: By definition, the bend allowance is the arc length of the bend as measured along the neutral axis of the material.



Auto Relief

Relief cuts are automatically added wherever needed when inserting bends if you select Auto Relief.

The following types of relief cuts are used:

- Rectangular
- Tear (minimum size required to insert the bend and flatten the part)
- Obround

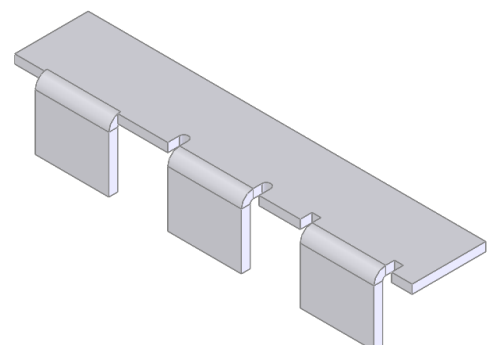
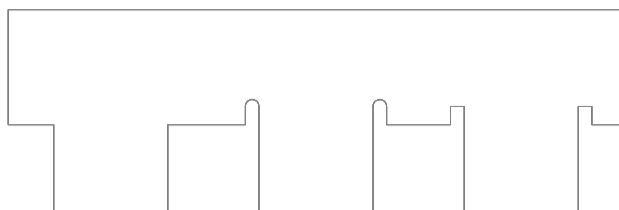
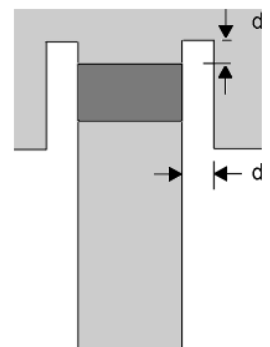
If you want to automatically add **Rectangular** or **Obround** reliefs, you must specify the Relief Ratio.

Relief Ratio

The distance **d** represents the width of the **Rectangular** or **Obround** relief cut and the depth by which the side of the **Rectangular** or **Obround** relief cut extends past the bend region. The distance **d** is determined by the following equation:

$$d = (\text{relief ratio}) * (\text{part thickness})$$

N.B. The value of the relief ratio must be between 0.05 and 2.0. The higher the value, the larger the size of the relief cut added during insertion of bends.



Examples of (i) Tear, (ii) Obround and (iii) Rectangle

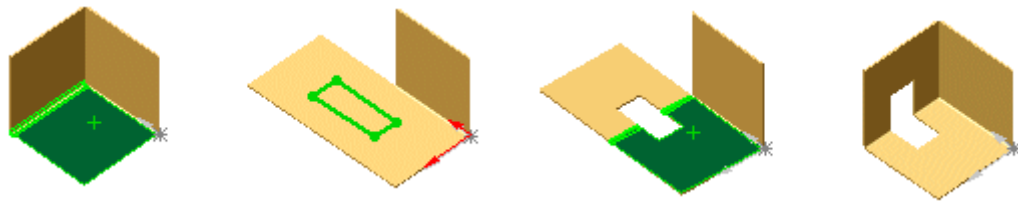
Standard SolidWorks Tools

Standard Solidworks tools, such as fillet, chamfer mirror, pattern etc. can be used. Cutouts can also be used to create holes and shaped cutouts in a sheet metal part.

Unfold and Fold

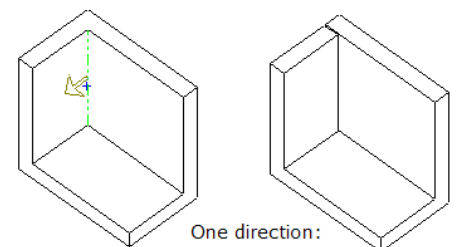
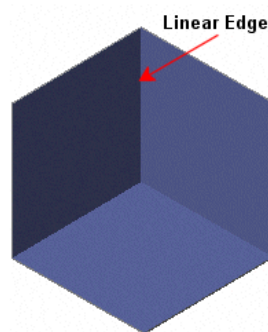
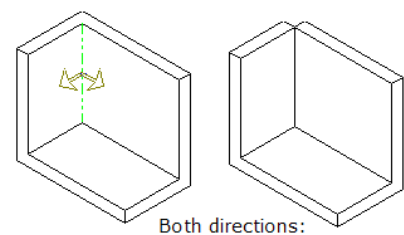
With the **Unfold** and **Fold** tools, you can flatten and bend one, more than one, or all of the bends in a sheet metal part.

This combination is useful when adding a cut across a bend. First, add an Unfold feature to flatten the bend. Next, add your cut. Lastly, add a Fold feature to return the bend to its folded state.



Rip

Creates a rip feature along the selected model edges. A rip feature is commonly used in sheet metal parts, but you can add a rip feature to any part.

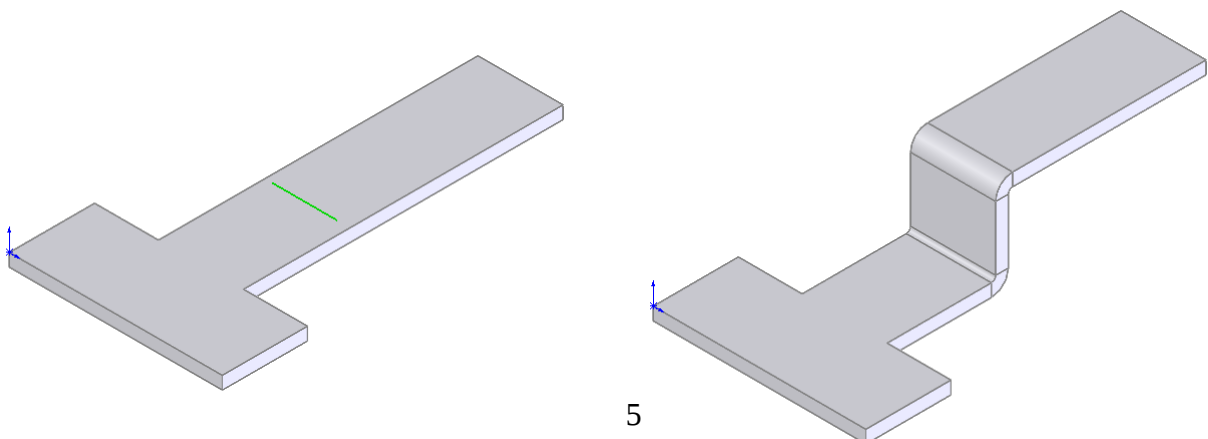


Jog

The Jog tool adds material to a sheet metal part by creating two bends from a sketched line.

Notes:

- The sketch must contain only one line.
- The line does not need to be horizontal or vertical.
- The bend line does not have to be the exact length of the faces you are bending.



Sheet Metal Exercise 1 - Sheet Metal Tutorial

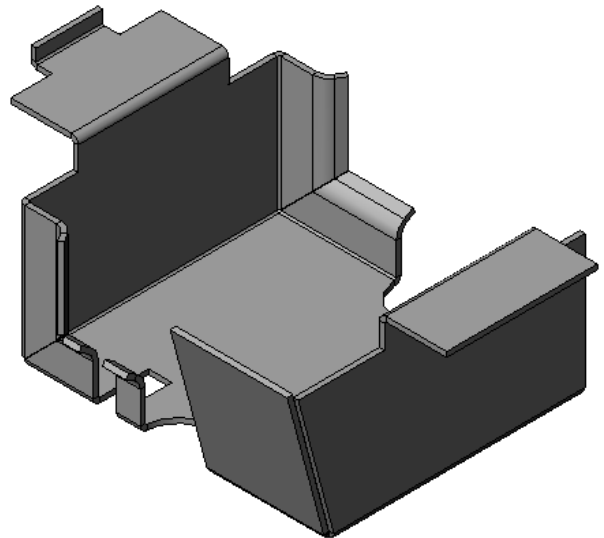
Using the SolidWorks Tutorials in the **Help** menu, complete the Sheet Metal assignment.

After selecting **SolidWorks Tutorials** in the **Help** menu, first choose the option “**All SolidWorks Tutorials**” shown in the **Tutorials by Category** box on the Home page and then pick the **Sheet Metal** assignment.

This involves creating the part shown below and demonstrates the following:

Creating a base flange

- Adding a miter flange
- Mirroring the part and creating new bends
- Adding an edge flange and editing its sketch profile
- Mirroring a feature
- Adding and bending a tab
- Adding a cut across a bend
- Folding and unfolding bends
- Creating a closed corner
- Creating a sheet metal drawing
- Adding bend line notes



Sheet Metal Exercise 2 - Bracket

Create a SolidWorks sheet metal model of the Bracket shown below and detailed in the supplied drawing.

On an A3 sheet create a drawing for the sheet metal part. Include a pictorial drawing and an undimensioned Flat pattern (blank development) view of the part.

