

Sheet metal design

Sheet metal design

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SOLID EDGE
VELOCITY SERIES

...with Synchronous Technology

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Lesson

1 *Introduction*

Welcome to self paced training for Solid Edge. This course is designed to educate you in the use of Solid Edge. The course is self-paced and contains instruction followed by activities.

Solid Edge self-paced courses

- **spse01510**—Sketching
- **spse01515**—Constructing base features
- **spse01520**—Moving and rotating faces
- **spse01525**—Working with face relationships
- **spse01530**—Constructing treatment features
- **spse01535**—Constructing procedural features
- **spse01536**—Modeling synchronous and ordered features
- **spse01540**—Modeling assemblies
- **spse01545**—Creating detailed drawings
- **spse01546**—Sheet metal design
- **spse01550**—Practicing your skills with projects
- **spse01560**—Modeling a Part Using Surfaces
- **spse01610**—Solid Edge frame design
- **spse01640**—Assembly patterning
- **spse01645**—Assembly systems libraries
- **spse01650**—Working with large assemblies
- **spse01655**—Revising assemblies
- **spse01660**—Assembly reports
- **spse01665**—Replacing parts in an assembly
- **spse01670**—Designing in the context of an assembly

- **spse01675**—Assembly features
- **spse01680**—Inspecting assemblies
- **spse01685**—Alternate assemblies
- **spse01686**—Adjustable parts and assemblies
- **spse01690**—Virtual components in assemblies
- **spse01691**—Exploding assemblies
- **spse01692**—Rendering assemblies
- **spse01693**—Animating assemblies
- **spse01695**—XpresRoute (tubing)
- **spse01696**—Creating a Wire Harness with Harness Design
- **spse01424**—Working with Solid Edge Embedded Client

Start with the tutorials

Self-paced training begins where tutorials end. Tutorials are the quickest way for you to become familiar with the basics of using Solid Edge. If you do not have any experience with Solid Edge, please start by working through the tutorials for basic part modeling and editing before starting this self-paced training.

Lesson

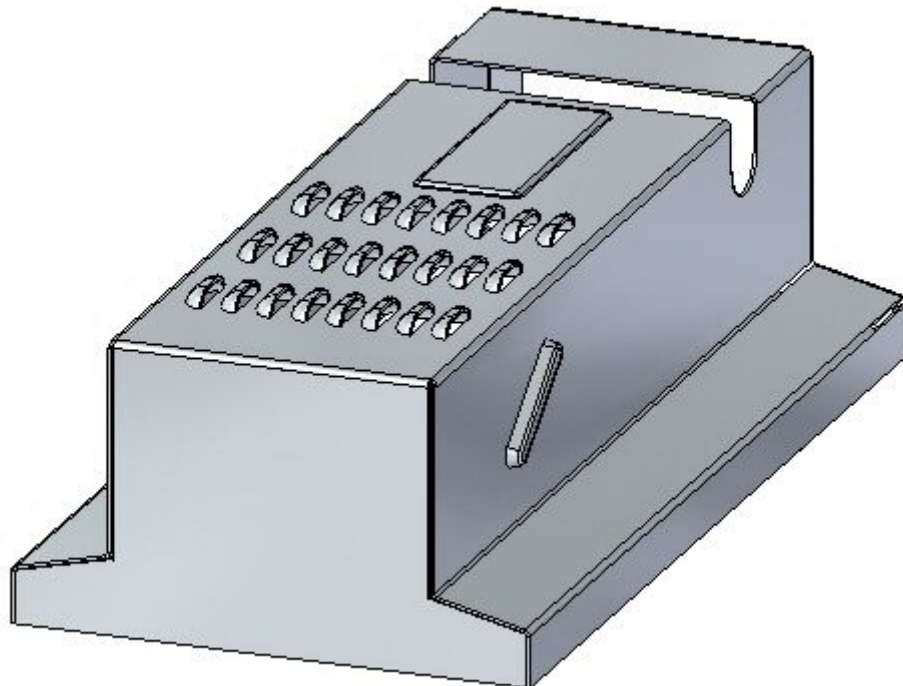
2 *Course Overview*

Course overview

The Solid Edge Sheet Metal application is specialized for modeling straight brake sheet metal parts.

Once you complete the activities in this course, you will be able to:

- Set parameters for sheet metal, such as bend radius and material thickness.
- Place and manipulate flanges and bends.
- Specify sheet metal treatments and corner parameters.
- Place holes, cutouts, dimples, louvers, beads, gussets.
- Modify geometry using live rules for synchronous design.
- Prepare the sheet metal geometry for downstream manufacturing processes such as creating a flat pattern.



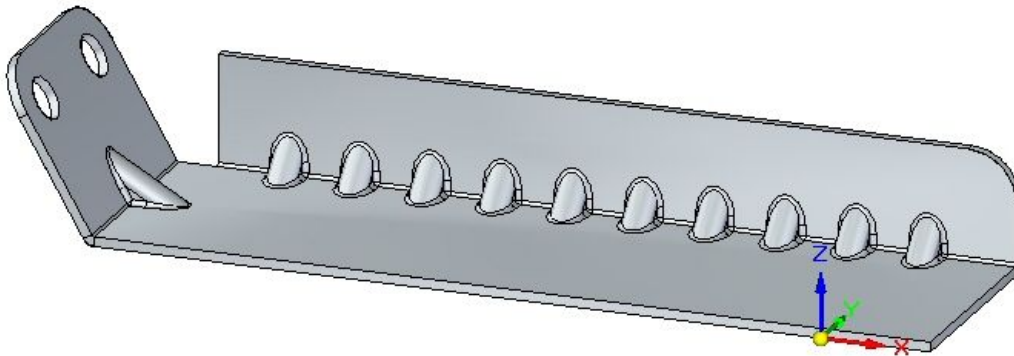
Lesson

3 *Sheet metal overview and definitions*

Sheet metal overview

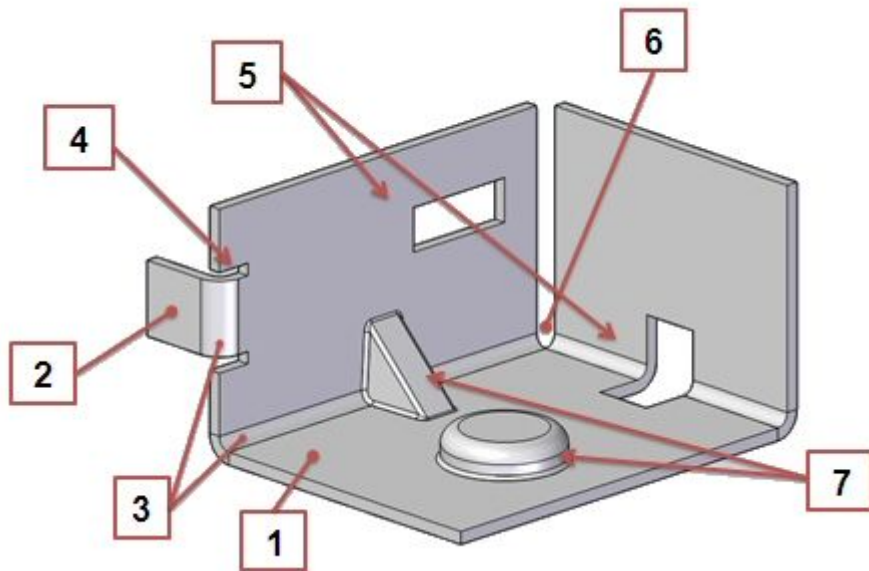
Sheet metal design is governed by the premise that the raw material used to form a sheet metal part is of common stock and of uniform thickness. The sheet metal part is designed in the formed state, but in the manufacturing process, many of the features of the part will be applied to the part before bending. The final locations of these features on the formed part is dependant on how the material behaves during the bending process. Material may stretch as the elastic limit is exceeded during bending and, while this stretching may be negligible in the final positioning of the feature, it may also make the target position after bending be incorrectly located.

The stretching of material during bending varies based on the material used and the thickness of the material. To correctly accommodate the stretching of material, calculations are made using a standard bend formula, which is provided. This bend formula can be customized for each stock material and by doing so, better accuracy is achieved in the resulting parts.



Terminology

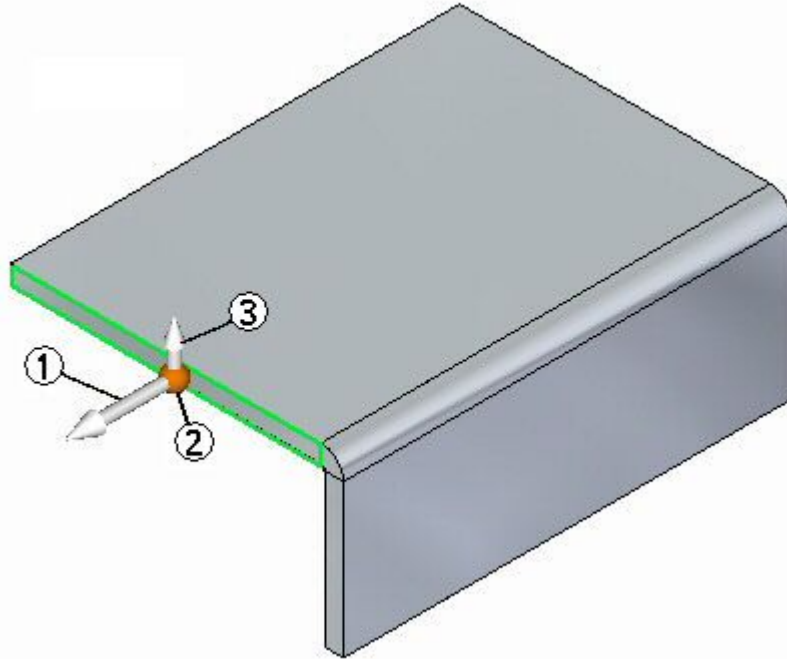
Sheet metal features



1. Plate: Consists of a layer face and a thickness face.
2. Tab-Flange: Two plates connected by a bend.
3. Bend: Connects two tab-flanges.
4. Bend Relief: Option to prevent tearing during bending.
5. Cutouts: Openings in the part.
6. Corner: Where 2 or 3 bends meet.
7. Procedural Feature: Deformation features such as dimples, drawn cutouts, louvers, beads, gussets, and so forth.

Steering wheel behavior in sheet metal

When you select a thickness face, Solid Edge displays a steering wheel unique to the sheet metal application. You can create flanges by selecting the flange start handle. You can use the primary axis, parallel to the layer face, to manipulate the size of the plate.



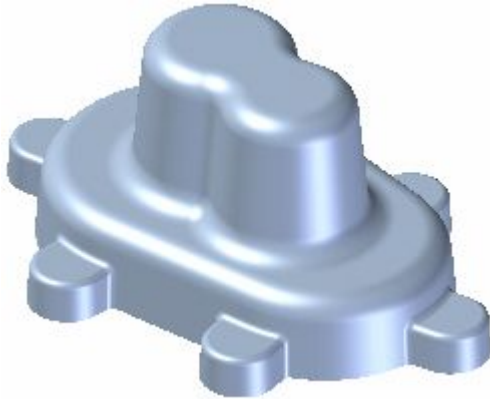
- (1) Primary Axis: Used to move or rotate the thickness face.
- (2) Origin
- (3) Flange start handle: This opens flange creation options on Quickbar.



When you move the steering wheel origin, all of the steering wheel capabilities become accessible.

Material Table command

Defines the material and mechanical properties for a part. When you select a material from the list, material and mechanical properties for the material such as face style, fill style, density, coefficient of thermal expansion, and so forth are assigned.



You can use the Solid Edge Material Table dialog box to do the following:

- Create, edit, and delete material property sets which are stored in the material library property file, *material.mtl*.
- Assign an existing material to the current document.
- Create a local material for use only in the current document.

The material and mechanical properties are used when you calculate the physical properties for a part or assembly, place the part in an assembly, render the assembly with Advanced Rendering, create a parts list on a drawing, define a bill of materials, and so forth.

When working with a sheet metal part, you also use the material table to define the properties for the sheet metal stock you are using, such as material thickness, bend radius, and so forth.

Material Library Property file

The material names and property sets are stored in an external material database file, *material.mtl*. The *material.mtl* file is used to populate the property set for each material on the Solid Edge Material Table dialog box. You can use these materials to define a material for any document on your computer, or other computers on your network.

You can use the Add To Library, Update In Library, and Delete From Library buttons to create, edit, and delete a material from the *material.mtl* file.

By default, the file is located in the Solid Edge ST4 Program folder. You can instruct Solid Edge to look for the *material.mtl* file in a different folder, including a folder on another machine on the network. This makes it easy for all your users to work with a consistent set of materials and properties while providing the capability to customize the materials list.

To define a new location for the *material.mtl* file, from the Application menu, choose Solid Edge Options® File Locations, select the Material Table entry, then click Modify. On the Browse dialog box, specify the drive and folder containing the *material.mtl* file. After specifying the location, click Update.

Note

You can use the Prompt For Material In New Model Documents option on the General tab of the Options dialog box to control whether you are prompted to assign a material when creating a new document.

Defining a local material

You can create one local material name and property set for a document. This can be useful when you need a variation of a common material to be displayed in the Material column in a parts list, bill of materials, or in Property Manager. For example, when using steel or aluminum structural shapes, you may want the shape information as part of the material name for the current part, but you do not want to add the shape information to the *material.mtl* file.

Type the new name and properties you want in the Solid Edge Material Table dialog box, then click the Apply To Model button. The material is then applied to only the current part, and the *material.mtl* file, located in the Solid Edge Program folder, is not updated.

Sheet metal gages

The sheet metal gage indicates the standard thickness of the sheet metal for a specific material. In Solid Edge, you can store sheet metal gage information in the material library or in a Microsoft Excel file.

Sheet metal gage information stored in an Excel file is stored in a *gage table*, which is simply a sheet within the Excel file that contains information, such as the gage name, material thickness, and bend radius. Solid Edge delivers a default gage file, *Gagetable.xls*, to the Solid Edge ST4 Program folder.

Gage tables can be very useful when defining the sheet metal gage. For example, suppose you have different materials with the same gage, but different material thickness or perhaps you have the same material with the same material thickness, but different bend attributes. Since the gage table is simply a sheet in the Excel file, to create additional gage tables to accommodate your different gage attribute combinations, all you have to do is insert a new sheet that contains the appropriate gage information. If you want to add a new gage to the gage table, you can copy and paste an existing gage, and then modify the information in the new gage.

Note

To ensure all changes are saved, ensure that you close the Excel file before closing the Material Table dialog box.

After you have created the gage table and gages, you can use the Associate Gage Table option on the Material Table dialog box to map the gage table and gage to a material or document. Once you select a material and the gage information you want to map, you can click the Update in Library button to add the gage information to the material library. For more information, see Map a gage file and gage table to a material or document.

When working with Gage tables, you should be careful when specifying the information about the table. If problems do occur, Solid Edge provides a number of error messages to help you resolve the issue. For example, Solid Edge verifies information such as:

- The path for the gage table file.
- The availability of the gage table worksheet.
- The gage parameters found in the gage table.
- Changes made to the Excel file since the last time the sheet metal file was accessed.

Gage tab

Use Excel File

Specifies that you want to get the gage information from an Excel file. You can use the Browse button to specify the Excel file you want to use.

Use Gage Table

Specifies the name of the gage table. You can use the Edit button to open the Excel file for edit.

Sheet Metal Gage

Displays the name of the current gage. When you select a name from the list, a set of associated material and mechanical properties is displayed. You can use the tabs on the dialog box to review or modify the properties. You can also define the material thickness using the Material Thickness option on the Gage tab.

Material Thickness

Specifies the material thickness for the part.

Bend Radius

Specifies the bend radius value for the part.

To facilitate creation of flat patterns, Solid Edge always creates a minimum bend radius for features that can be flattened, even if you specify a bend radius value of zero (0.00). For metric documents, a zero bend radius will actually be set to a value of approximately 0.002 millimeters. For English documents, a zero bend radius will actually be set to a value of approximately 0.0000788 inches. If you need the bend radius to be exactly zero, you will have to create the features in the Part environment.

If you would like to delete the minimum bend radius and minimum bend-relief surfaces that Solid Edge creates, you can use the Delete Relief Faces command.

Relief Depth

Specifies the relief depth value for the part.

Relief Width

Specifies the relief width value for the part.

Bend Equation

Defines the bend equation formula you want to use. The bend equation formula is used to calculate the flat pattern of a sheet metal part when using the Part Copy command. You can use the standard formula delivered with Solid Edge, one of the example formulas in the Solid Edge ST4/Custom/sheetmetal folder or a custom formula you develop.

Standard Formula

Specifies that the formula delivered with Solid Edge is used to calculate the flat pattern size. The standard formula is:

$$PZL = \pi * (BR + (NF * THK)) * BA / 180$$

Where:

PZL = Plastic Zone Length

BR = Bend Radius

NF = Neutral Factor

THK = Material Thickness

BA = Bend Angle

Neutral Factor

Specifies the neutral factor to use for the bend(s).

Neutral Factor Specifies the default neutral factor for the bend(s).

Use Neutral Specifies the neutral factor information is extracted
Factors from Excel from the Excel file.
File

Custom Formula

Specifies that a custom formula you define is used to calculate the flat pattern size.

ProgramID.ClassName:

Defines the custom bend formula you want to use. Type the program ID and class name using the following syntax:

ProgramID.ClassName

Add To Library

Adds the new material or gage to the library file. This button is available when you have defined a new material or gage.

Update In Library

Updates the existing material or gage in the library file. This button is available when you have changed the properties for an existing material or gage.

Delete From Library

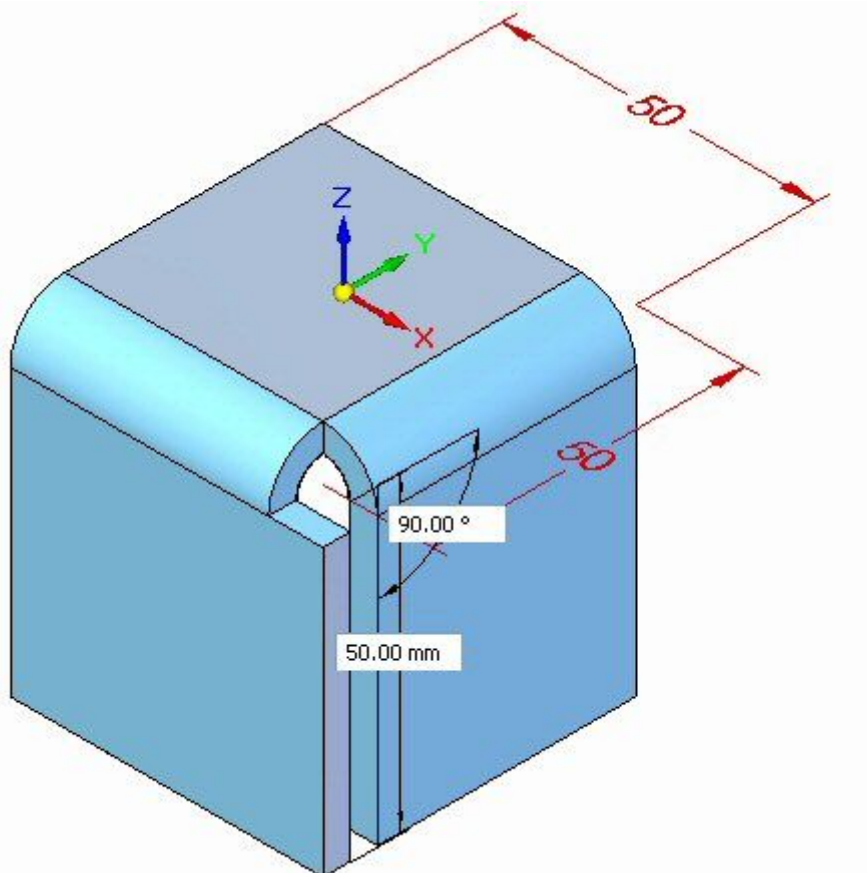
Deletes the existing material or gage in the library file. This button is available when you have selected an existing material or gage.

Activity: Starting sheet metal design

Activity objectives

This activity demonstrates how to begin working in sheet metal. The activity explores some of the settings used to create the part with the desired material and material properties. In this activity you will accomplish the following:

- Create a new sheet metal part.
- Create the material to be used for the part.
- Modify the thickness of the material.
- Examine the bend formula and change the neutral factor.
- Create a base geometry consisting of a tab and then create flanges around the tab.

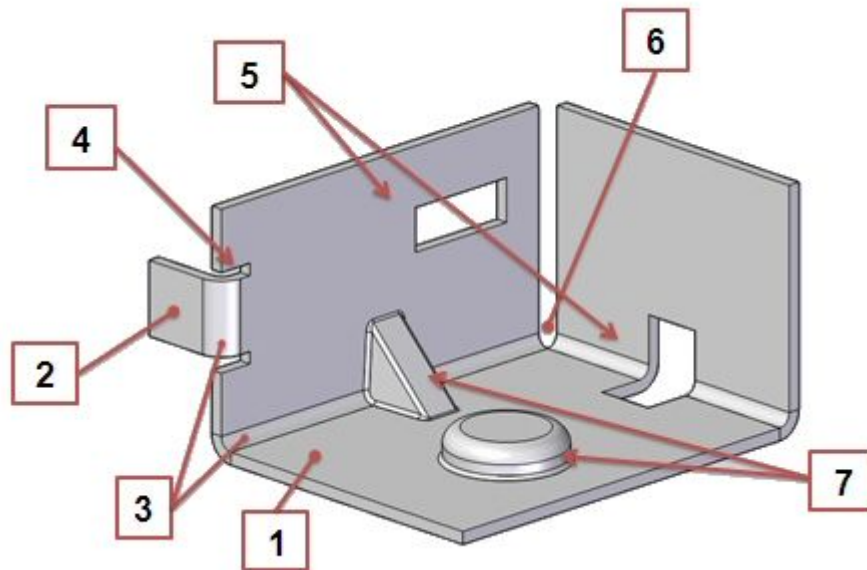


Turn to Appendix A for the activity: Starting sheet metal design.

Lesson review

Answer the following questions:

1. Assign a term from the list below to each number.



Corner

Bend Relief

Cutouts

Procedural Feature

Tab-Flange

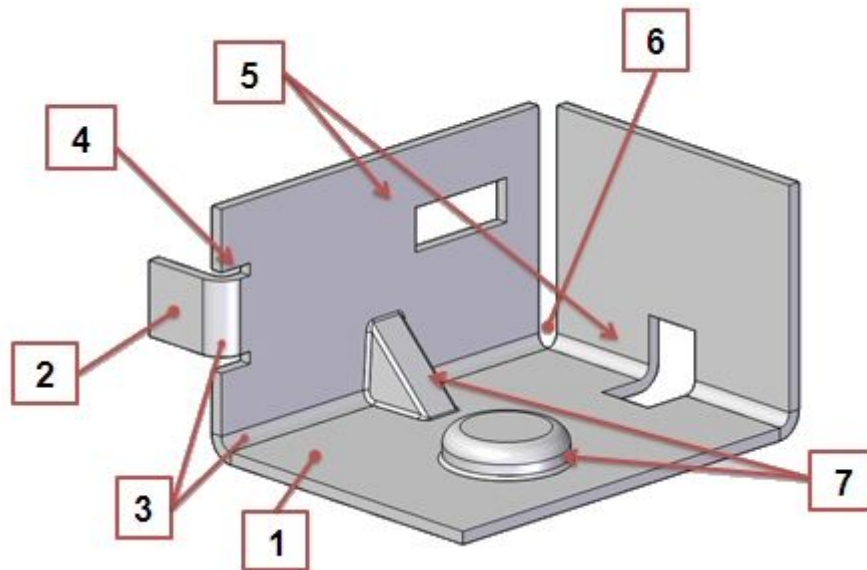
Bend

Plate

2. Name three methods of setting material thickness in a sheet metal document.

Answers

1. Assign a term from the list below to each number.



Corner

Bend Relief

Cutouts

Procedural Feature

Tab-Flange

Bend

Plate

1. Plate: Consists of a layer face and a thickness face.
 2. Tab-Flange: Two plates connected by a bend.
 3. Bend: Connects two tab-flanges.
 4. Bend Relief: Option to prevent tearing during bending.
 5. Cutouts: Openings in the part.
 6. Corner: Where 2 or 3 bends meet.
 7. Procedural Feature: Deformation features such as dimples, drawn cutouts, louvers, beads, gussets, and so forth.
2. Name three methods of setting material thickness in a sheet metal document.
- a. The material thickness can be set by entering the thickness when creating the first feature in a sheet metal document.
 - b. The material thickness can also be set through the material table by entering the thickness there, or using the gage tab to set a standard material thickness assigned to a gage of material.

- c. The material thickness can also be set through the material table by getting the gage information from an Excel file. You can use the Browse button to specify the Excel file you want to use.

Lesson summary

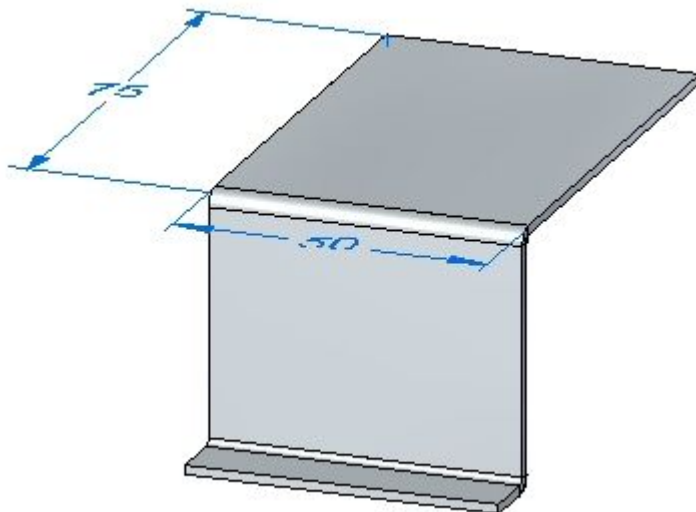
In this lesson you set the material thickness using the gage tab on the material table. A tab was placed and flanges were created from the thickness faces of the tab.

Lesson

4 *Base Features*

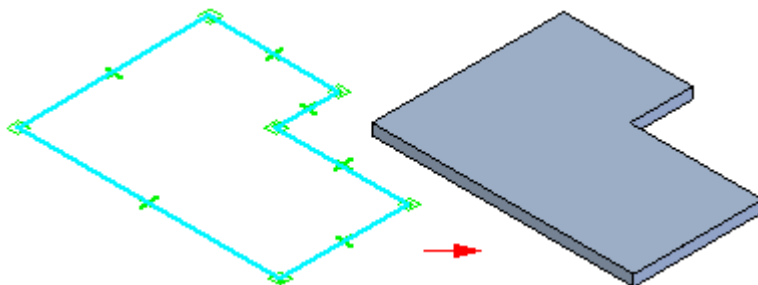
Base features in sheet metal

A base feature in sheet metal is the first thickness plate placed in a sheet metal file. You can create the base feature by placing a tab, which is a single thickness plate, or a contour flange, which can consist of additional flanges and bends.

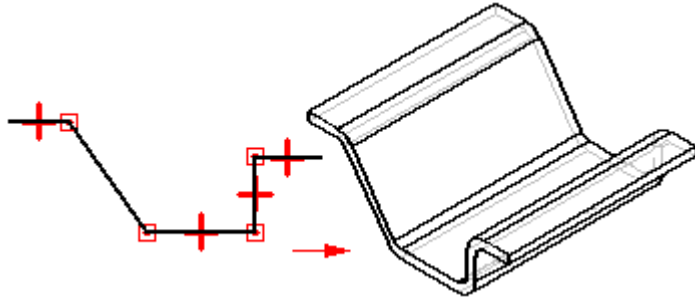


Construct the base feature

You can construct a base feature with the Tab and Contour Flange commands. The Tab command constructs a flat feature of any shape using a closed profile.



The **Contour Flange command** constructs a feature comprised of one or more bends and flats using an open profile.

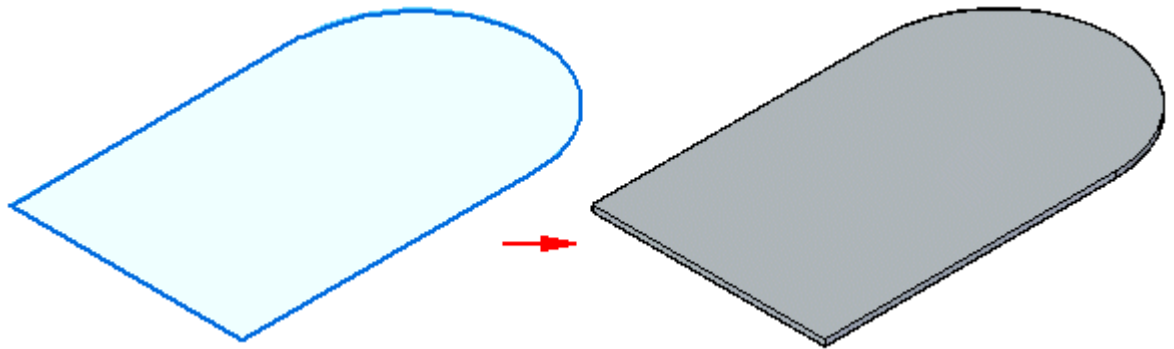


If you want to use a different bend radius value, you can do this by drawing arcs in the profiles.

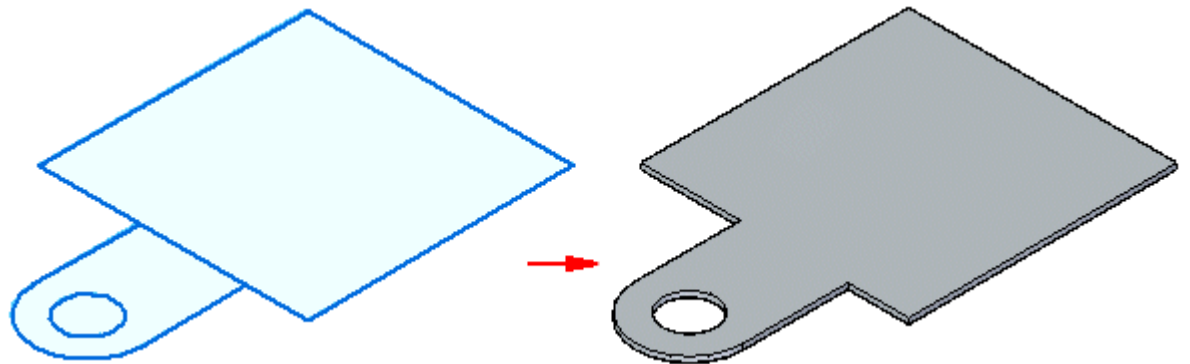
Tab command

Constructs a tab feature on a sheet metal part. You can use this command to construct a base feature or add a feature to an existing sheet metal part.

In the synchronous environment, you can construct a tab with a single sketch region,



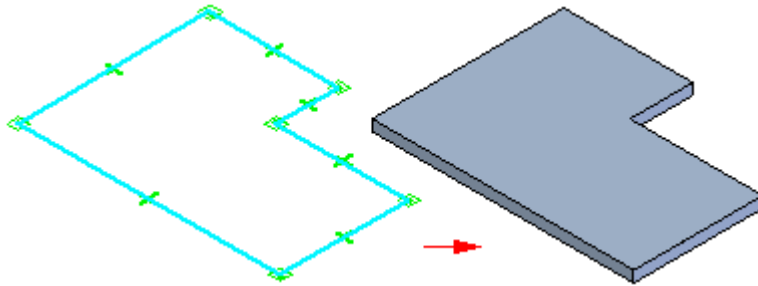
or with multiple sketch regions.



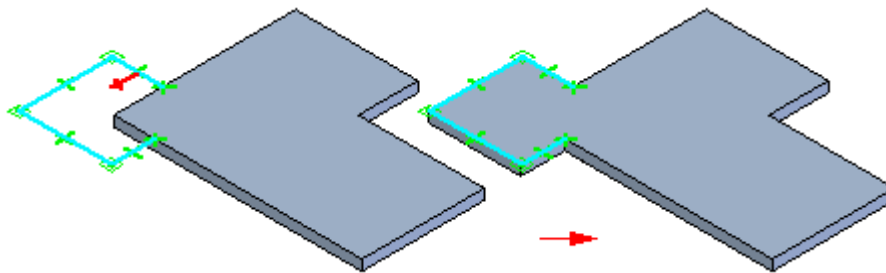
Creating tabs in the ordered environment

In the ordered environment, you can only have one profile per tab feature.

When selecting multiple regions, the regions must be contiguous and in the same plane. When constructing a base feature in the ordered environment, the profile must be closed, and you must also define the material direction and material thickness you want.

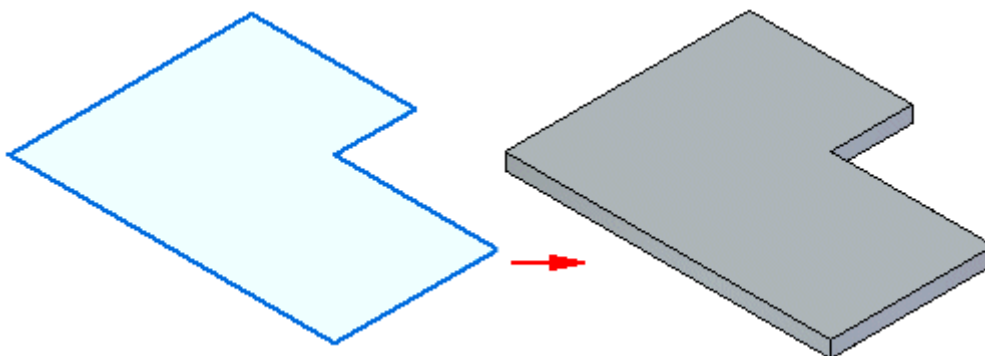


For subsequent features in the ordered environment, the profile can be open or closed. When using an open profile, you must define the side of the profile to which you want to add material.

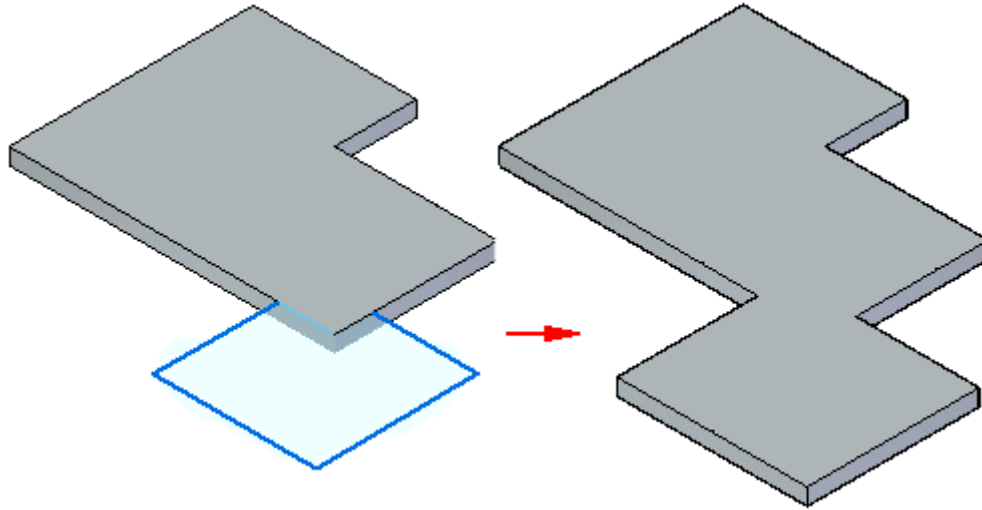


Creating tabs in the synchronous environment

When constructing a base feature in the synchronous environment, the sketch region must be closed, and you must also define the material direction and material thickness you want.



For subsequent features in the synchronous environment, the sketch can be open or closed. If the sketch is open the edge of the tab must close the sketch to form a sketch region. Subsequent features are automatically added when you select the extrude handle.




Editing tabs

Once you create a tab, you cannot change the thickness or offset direction for the tab. You can use the Material Table to change things such as global thickness, bend relief, and relief depth.

Construct a tab

You can construct a tab as a base feature or add a tab to an existing sheet metal part.

Construct a tab in the ordered environment

1. Choose Home tab@ Sheet Metal group@ Tab .
2. Define the profile plane.
3. Draw an open profile in any 2D shape or copy a profile into the profile window. The ends of an open profile are extended to the edges of the part plane. An arc with open ends is extended to form a circle.

Note

If you are using the Tab command to construct a base feature, the profile must be closed.

4. Choose Home tab@ Close group@ Close.

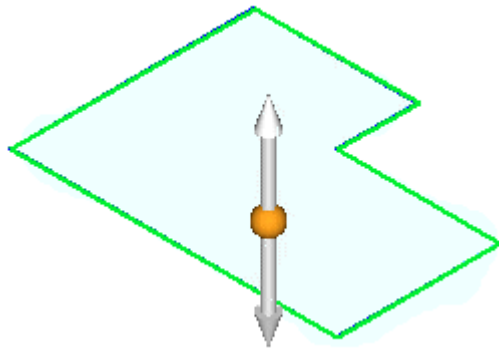


5. Finish the feature.

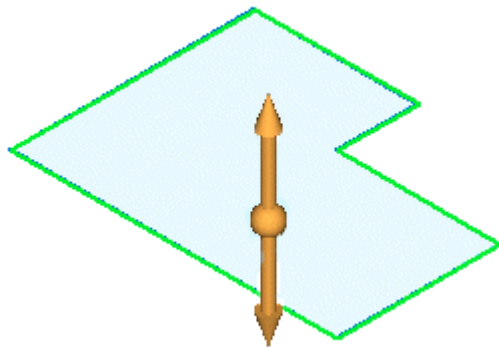
Construct a tab as a base feature in the synchronous environment

1. Position the cursor over a sketch region, then click to select it.

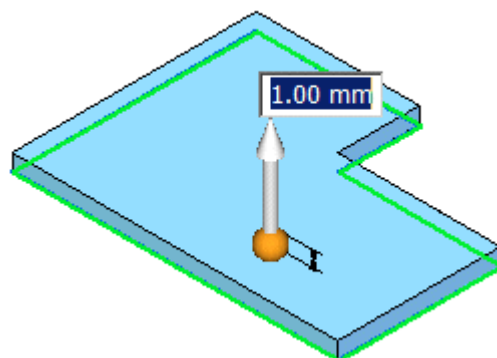
The extrude handle is displayed.



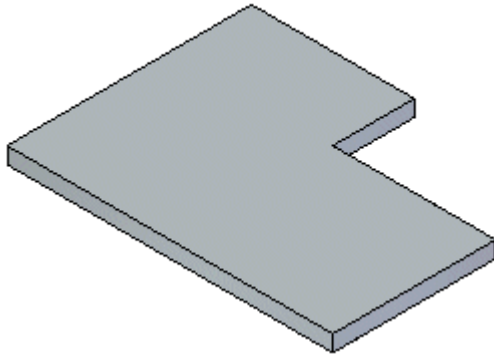
2. Click the Extrude handle.



3. Type a thickness value for the part.



4. Right-click to create the tab.

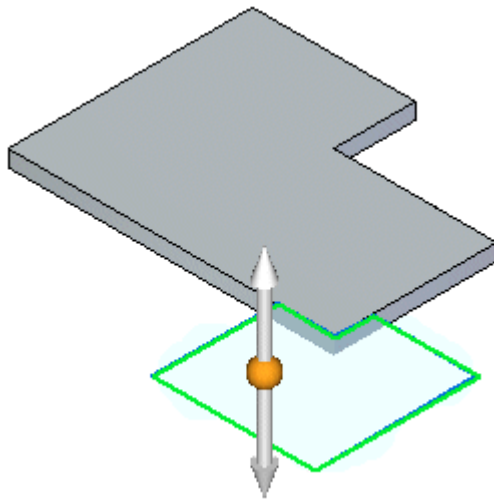
**Tip**

- You can choose the Material Table button on the QuickBar to display the Solid Edge Material Table dialog box to make changes to things such as global thickness, bend relief, and relief depth.
- You can click the direction indicator handle to change the offset direction.

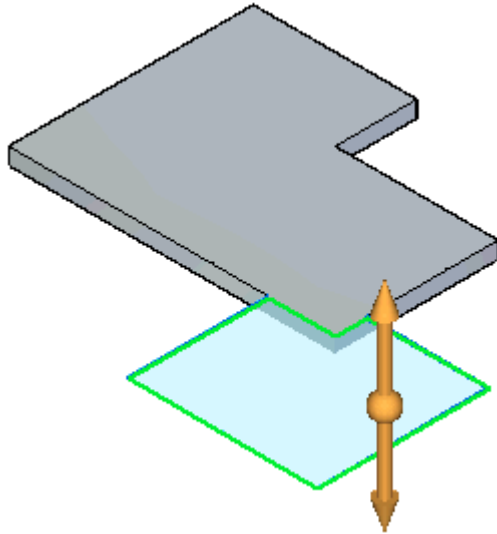
Add a tab to an existing sheet metal part in the synchronous environment

1. Position the cursor over a sketch region, then click to select it.

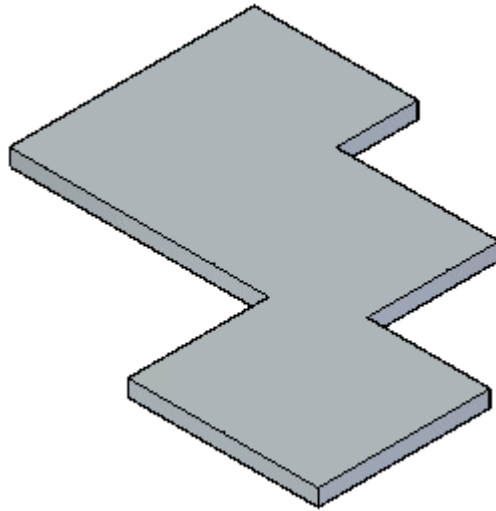
The extrude handle is displayed.



2. Click the Extrude handle.



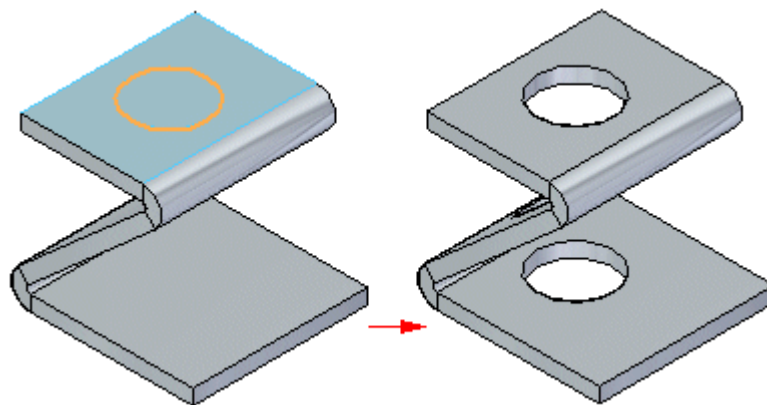
The tab is automatically added.



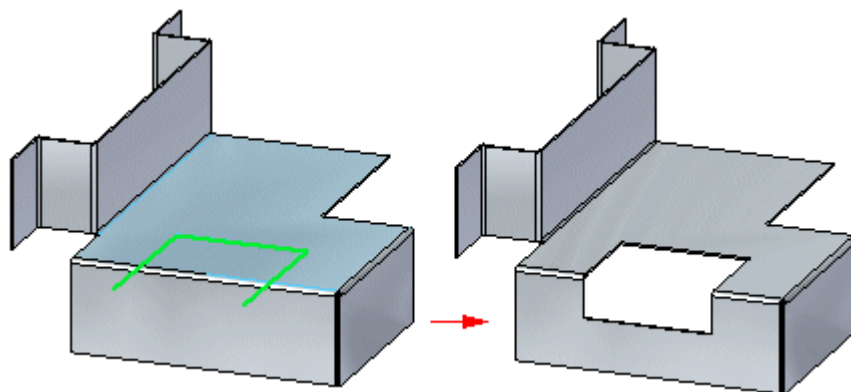


Cut command

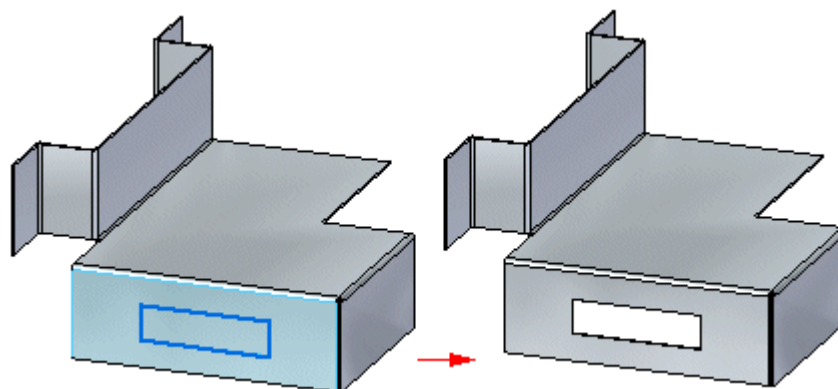
Creates a cut through a defined portion of a part.



You can create a sheet metal cutout with an open profile



or a closed profile.

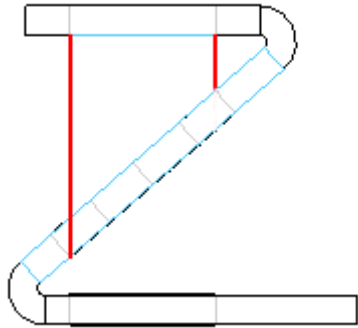


Face Normal cut types

Face Normal cut types include:

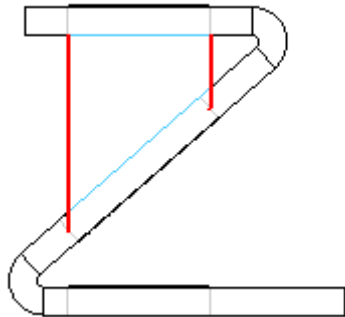
Thickness cut

This option creates a cutout that compensates for the material thickness of the part.



The Thickness cut option is useful when creating parts in which a shaft must pass through aligned circular cutouts.

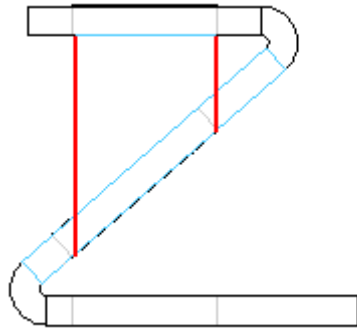
Mid-plane cut



This option creates a cutout based on the mid-plane of the part.

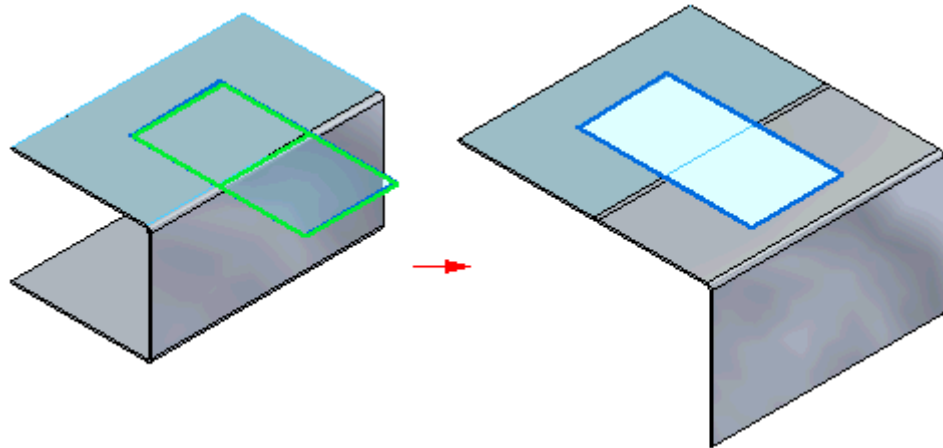
Nearest Face cut

This option creates a cutout based on the nearest face of the part.

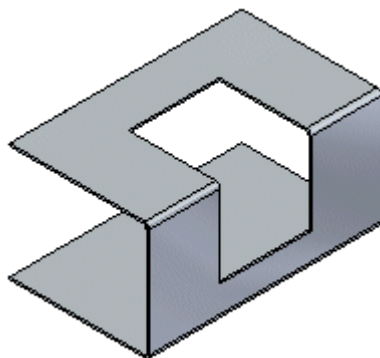


Cuts across bends

The Wrapped Cut option unfolds the bend to create a cut,



and then rebends when the cut is complete.

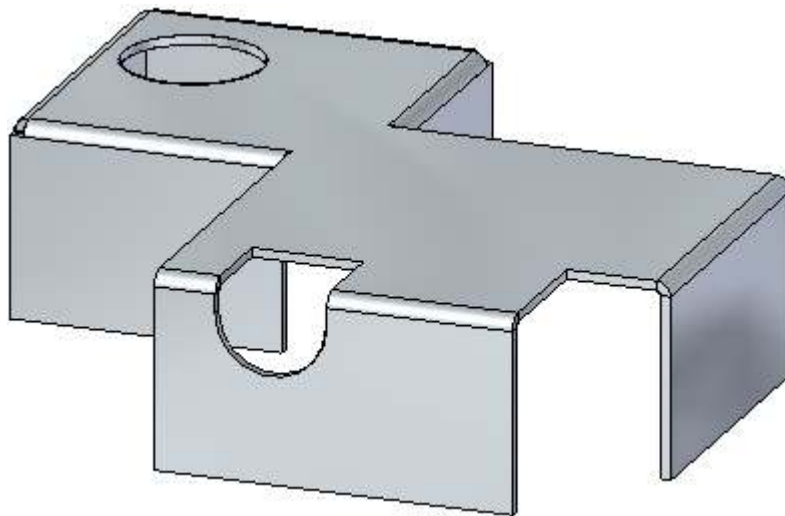


Activity: Using regions to create tabs and cuts

Activity objectives

This activity demonstrates how to create various tabs in sheet metal and how to use regions to make cuts. In this activity you will:

- Create a tab base feature from a sketch.
- Add additional tabs to the base feature.
- Create flanges.
- Explore the different options available when cutting a sheet metal part.



Turn to Appendix B for the activity: Using regions to create tabs and cuts.

Lesson review

Answer the following questions:

1. Name two commands which can generate a base feature in a sheet metal document.
2. Are open cutouts valid in a sheet metal document?
3. What is a wrapped cut in sheet metal?

Answers

1. Name two commands which can generate a base feature in a sheet metal document.

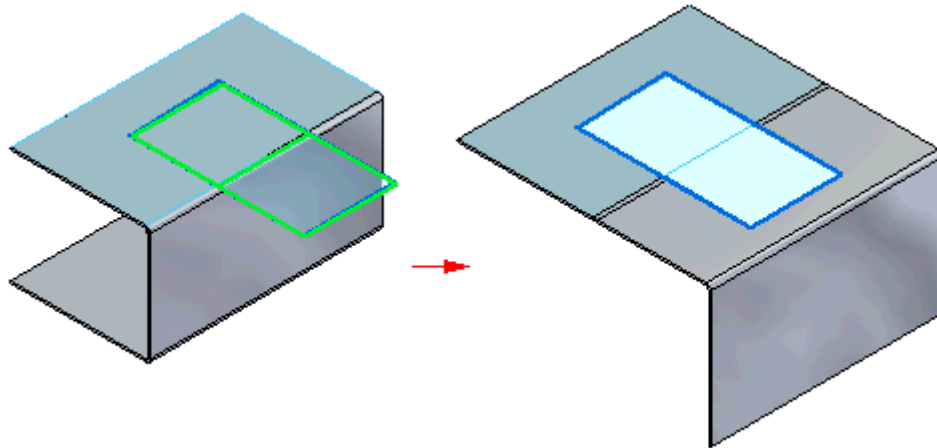
A tab or a contour flange can be used to generate a base feature in a sheet metal document.

2. Are open cutouts valid in a sheet metal document?

Open cutouts are valid in a sheet metal document.

3. What is a wrapped cut in sheet metal?

A wrapped is closed and continues across a bend. The bend is flattened to make the cut and bent back after the cut is made.



Lesson summary

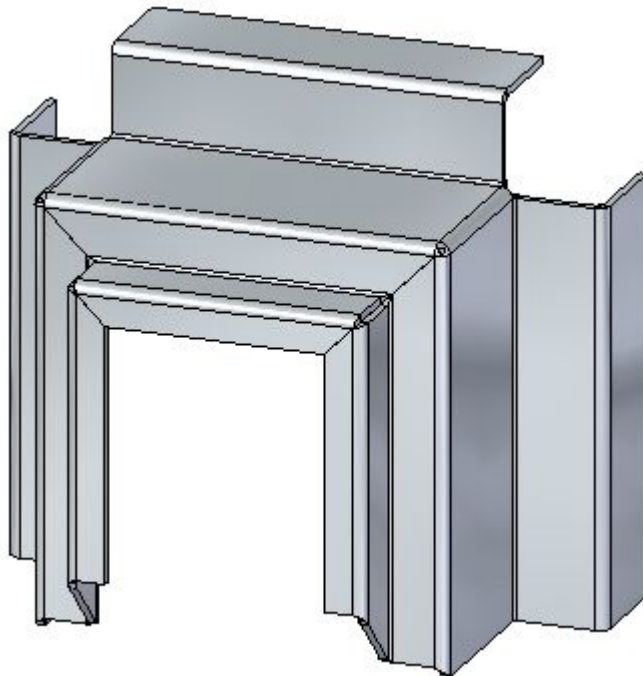
In this lesson you created a sheet metal base feature using a tab, and added additional material creating a tab from a sketch. Regions were used to create a cut and a wrapped cut. You learned the different options for moving a thickness face.

Lesson

5 *Contour Flange*

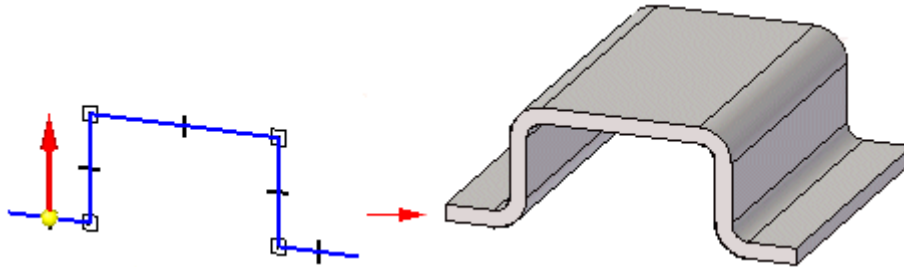
Contour Flange

A contour flange can be used to create a base feature from a sketch, or it can be used to quickly construct flange geometry along existing thickness edges of a sheet metal part. Parameters for mitering corners around bends can be set.



Contour Flange command

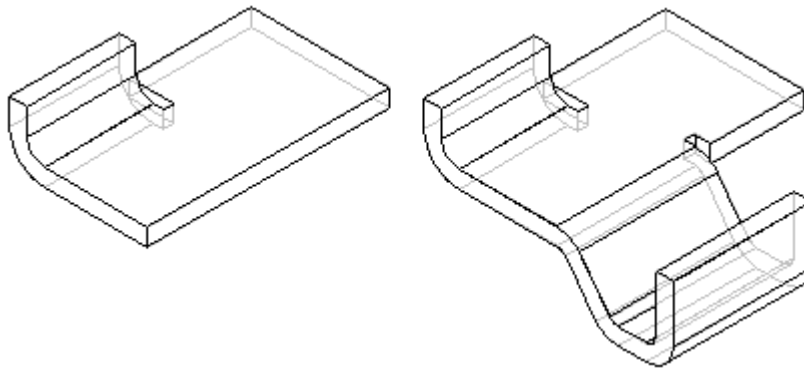
Constructs a contour flange by extruding a profile that represents the edge of the contour flange.



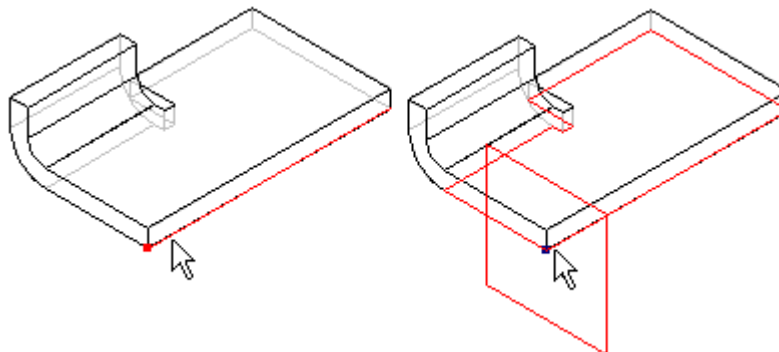
Examples: Defining Reference Plane Orientation to Construct a Contour Flange

When constructing a contour flange, you must define the orientation of the profile plane relative to an existing edge on the part. Doing this defines both the reference plane orientation and the path along which the contour flange will be constructed.

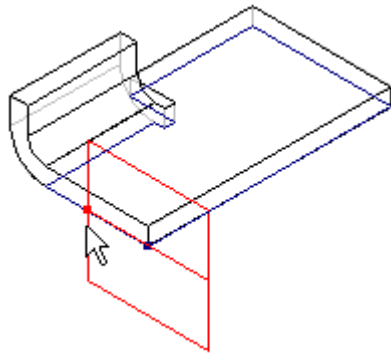
For example, suppose you wanted to construct a contour flange as shown in the figure.



You could do this by selecting the endpoint of the edge shown to locate the new reference plane, and then clicking the face shown on the right side of the figure to define the base of the reference plane.



You could then click near the end shown to define the x axis orientation.

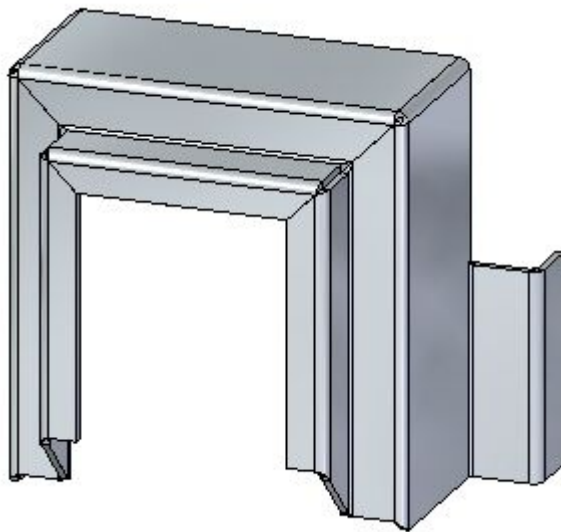


Activity: Constructing a base feature using contour flange

Activity objectives

This activity demonstrates how a contour flange can be used to create a base feature. In this activity you will accomplish the following:

- Create a new sheet metal part.
- Create the material to be used for the part.
- Modify the thickness of the material.
- Create a sketch that will be the basis for the contour flange.
- Examine the PathFinder and understand how a contour flange is defined.



Turn to Appendix C for the activity: Constructing a base feature using contour flange.

Lesson review

Answer the following questions:

1. How can you make a base feature using the contour flange command?
2. Can a part edge be used to define the extent of a contour flange and if so, can an adjacent edge be used to continue the extent?
3. What does the mitre option do when used in the creation of a contour flange?

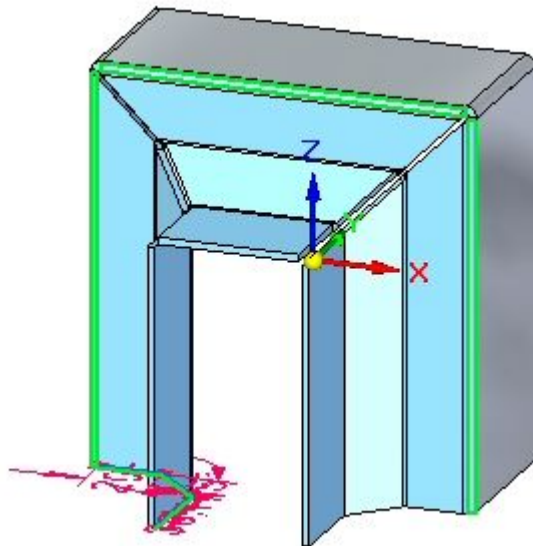
Answers

1. How can you make a base feature using the contour flange command?

To use the contour flange command to make a base feature, the profile is first defined. This can exist in a sketch. The thickness is set, either from a key in, or from the material table. The contour flange command then needs the extent and side to be defined before creating the contour flange.

2. Apart edge be used to define the extent of a contour flange and an adjacent edge be used to continue the extent.
3. What does the mitre option do when used in the creation of a contour flange?

Mitring a corner of a contour flange creates a corner condition where the material on each edge match and can be flattened. The image shown was a part that was created by using the contour flange with the corners mitred.



Lesson summary

In this lesson you set the material thickness and extent to create a base feature using a contour flange. The components of the contour flange were examined and

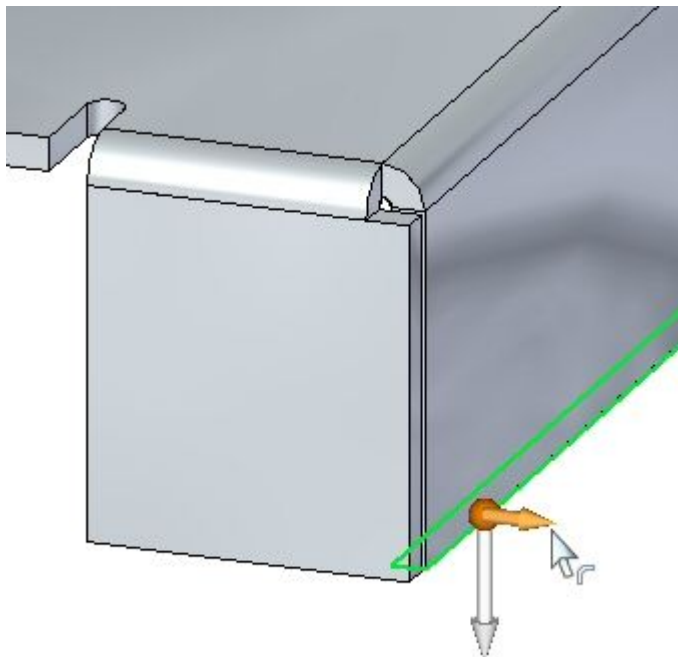
manipulated. Options for the construction of end conditions were explored, and a partial contour flange was placed.

Lesson

6 *Flanges, corners and bend relief*

Flanges, corners and bend relief

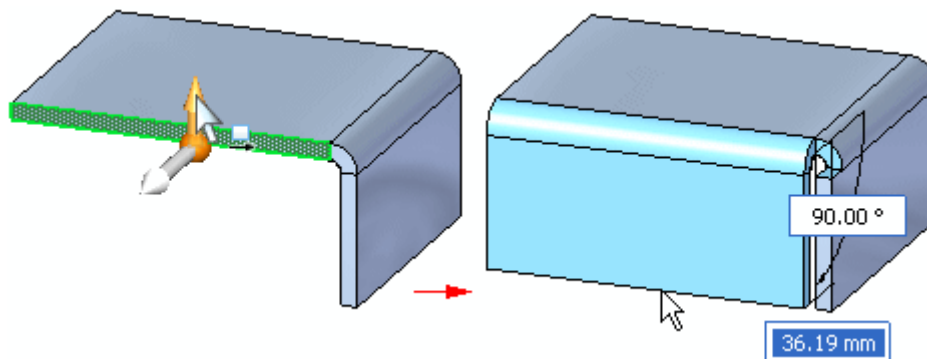
You create flanges using flange handles. As you create them, you can control end conditions such as bend relief and corner conditions. You can insert bends across layer faces.



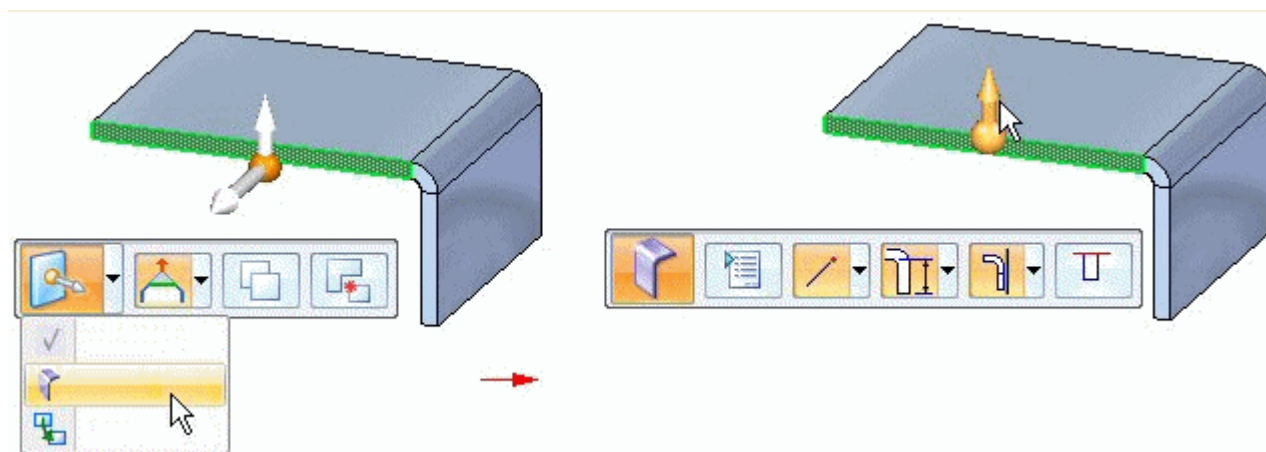
Creating flanges

Creating flanges

When you select a planar thickness face on a sheet metal model, the flange start handle is displayed.



The flange start handle is also displayed without the 2D steering wheel when you click the Flange command on QuickBar when a planar thickness face is selected.

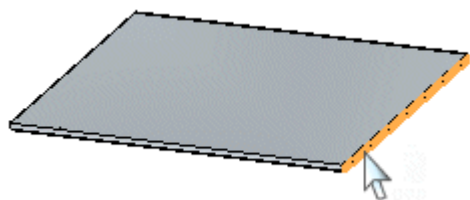


Flange command

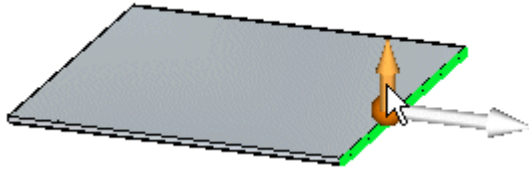
Constructs a flange by extruding material that represents the face of the flange.

Flanges in the synchronous environment

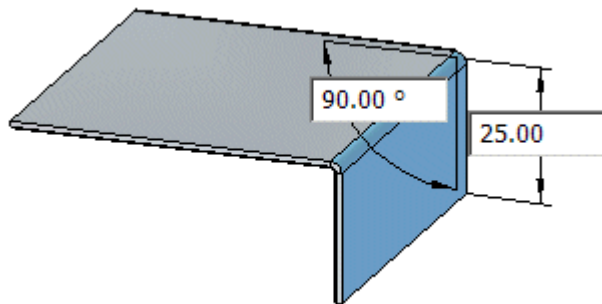
In the synchronous environment, you can construct a flange by selecting a linear thickness edge to display the flange start handle,



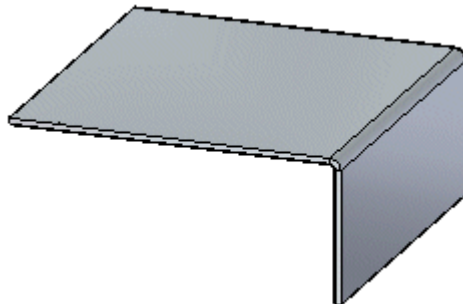
clicking the flange start handle,



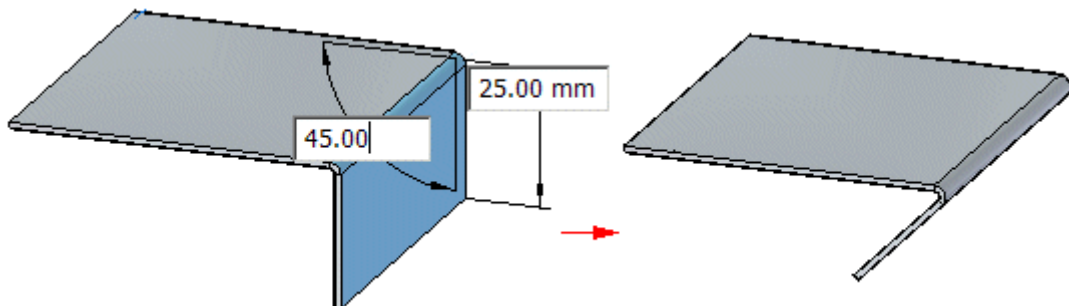
specifying a flange distance,



and clicking to place the flange.



When you click, a 90° flange is drawn automatically. However, when specifying the distance for the flange, you can also specify an angle.

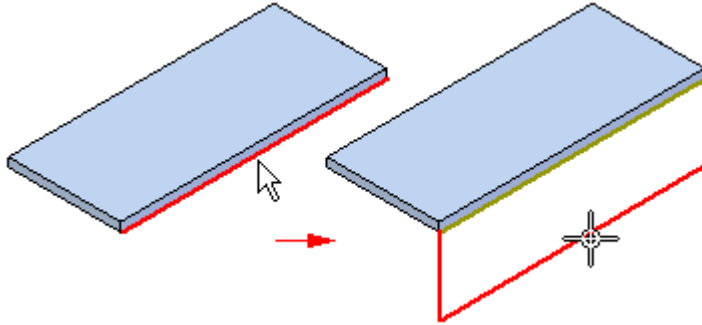


Note

Use the Tab button to switch between the distance and angular value controls.

Flanges in the ordered environment

In the ordered environment, you construct a flange by selecting a linear thickness edge, and then reposition the cursor to define the flange direction and length.



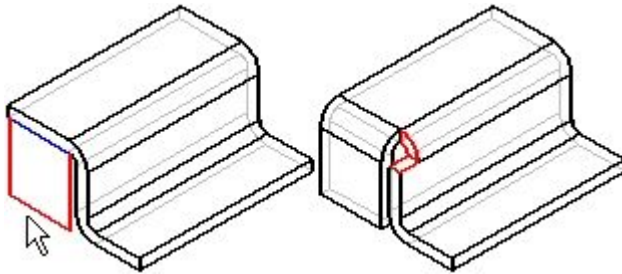
Corner Relief

Corner Relief

Specifies that you want to apply corner relief to flanges that are adjacent to the flange you are constructing. When you set this option, you can also specify how you want the corner relief applied.

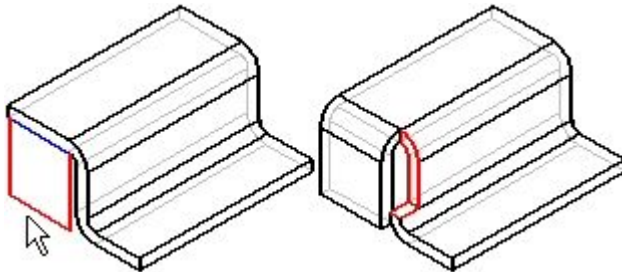
Bend Only

Specifies that corner relief is only applied to the bend portion of the adjacent flanges.



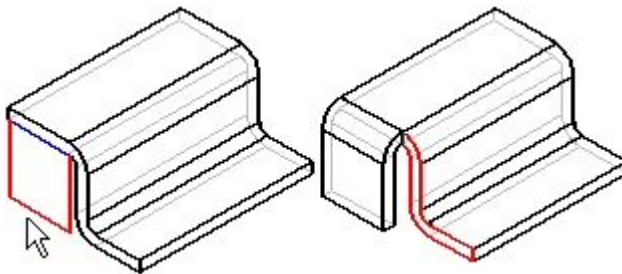
Bend and Face

Specifies that corner relief is applied to both the bend and face portions of the adjacent flanges.



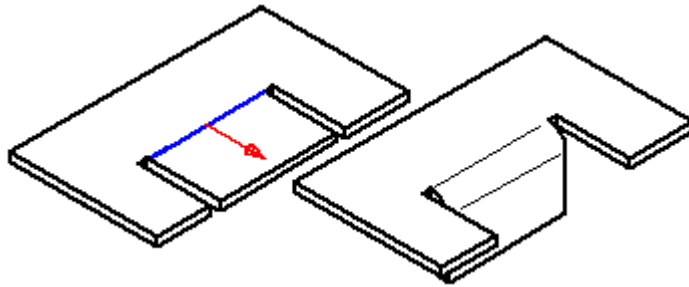
Bend and Face Chain

Specifies that corner relief is applied to the entire chain of bends and faces of the adjacent flanges.



Bend command

Inserts a bend across a planar face. You can use the command to add a bend in the middle of a part. The bend profile must be a single linear element. You cannot insert a bend across an existing flange.



Insert a bend

In the ordered environment, you can insert a bend with the Bend command.

In the synchronous environment, you can [insert a bend with the Select tool](#) or [insert a bend with the Bend command](#). Both workflows are explained in this topic.

Insert a bend in the ordered environment

1. Choose Home tab® Sheet Metal group® Bends list® Bend.



2. Define the profile plane.
3. Draw a profile. The profile, which must be a single linear element, represents the approximate location of the bend.
4. Choose Home tab® Close group® Close.




5. Define the bend location with respect to the profile.
6. Define which side of the part will move.
7. Define the bend direction.
8. Finish the feature.

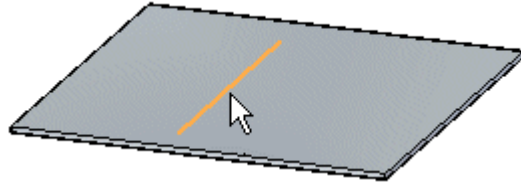
Tip

- You can automatically flatten the bend by setting the Flatten Bend option on the Bend Options dialog box.

Insert a bend in the synchronous environment with the Select tool

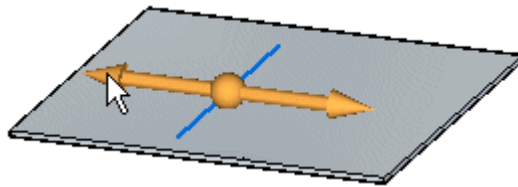


1. Choose Home tab® Select group® Select .
2. Select the sketch element to create the bend.

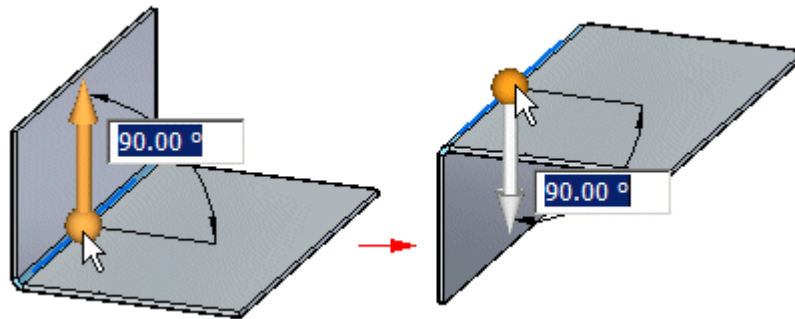


3. Choose Home tab® Sheet Metal group® Bends list® Bend .

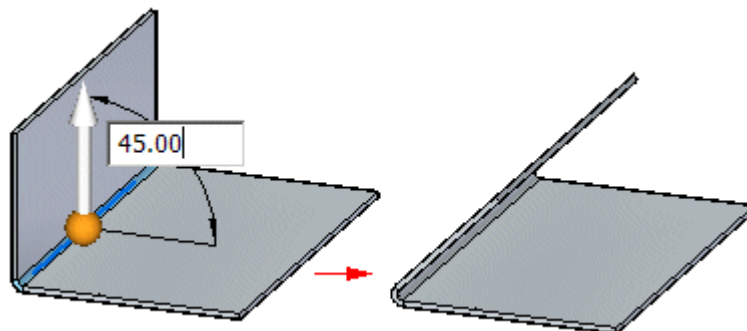
4. Click the side of the sketch to move.



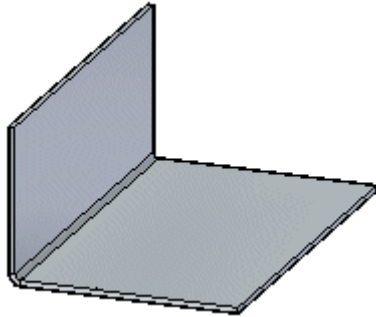
5. (Optional) Click to the direction arrow to change the direction of the bend.




6. (Optional) Type a value to change the bend angle.

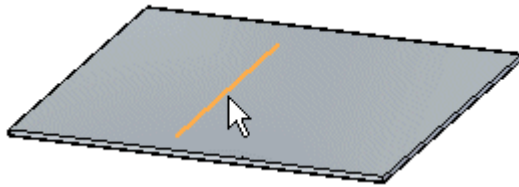


7. Click to create the bend.

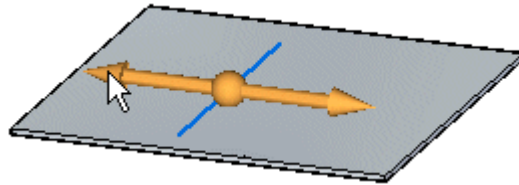


Insert a bend in the synchronous environment with the Bend command

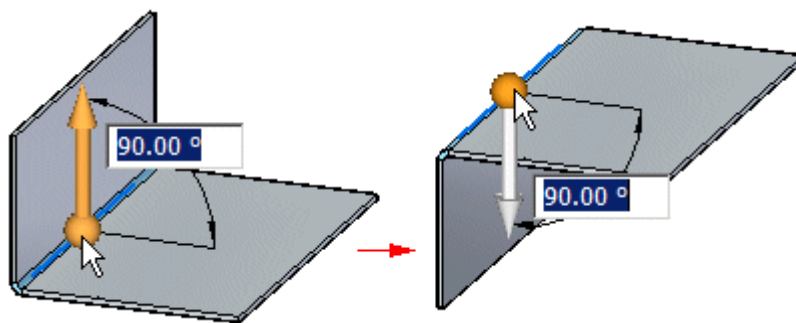
1. Choose Home tab® Sheet Metal group® Bends list® Bend .
2. Select the sketch element to create the bend.



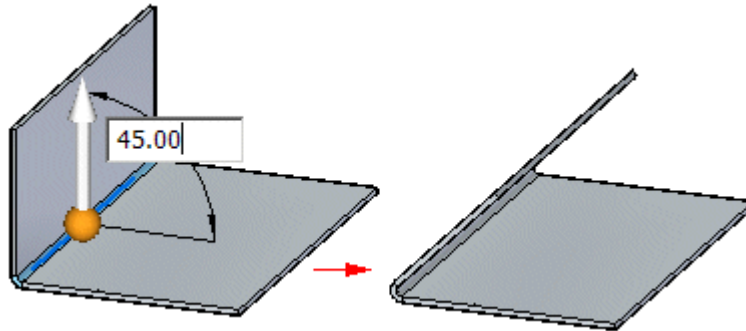
3. Click the side of the sketch to move.



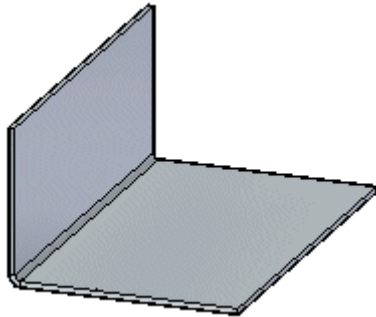
4. (Optional) Click to the direction arrow to change the direction of the bend.



- (Optional) Type a value to change the bend angle.

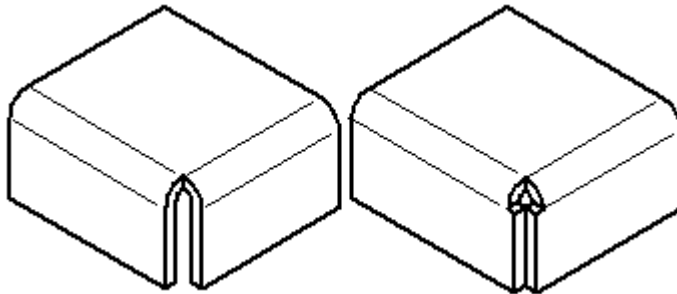


- Click to create the bend.

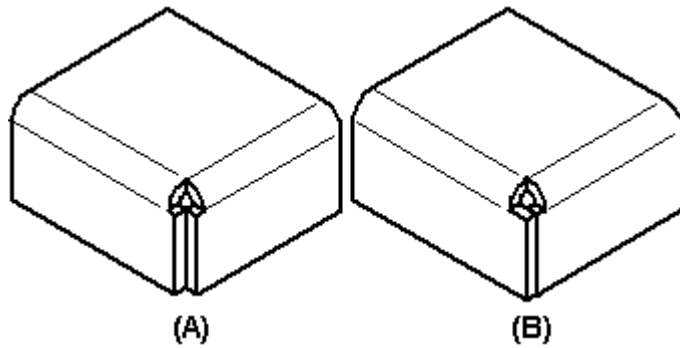


Close 2-Bend Corner command

Closes the corner where two flanges meet and creates the smallest gap permissible without joining the corner. Flange edges can equally meet, overlap, totally intersect, or intersect with circular corner relief.



You can specify whether you want to close (A) or overlap (B) the corners.



You cannot directly move or rotate a bend corner. However, you can move or rotate the bend corner by repositioning the adjacent flanges that form the corner. If a plate that contributes to the closed corner is deleted, the bend faces created by the closed corner are deleted and the closed corner definition is removed from the model.

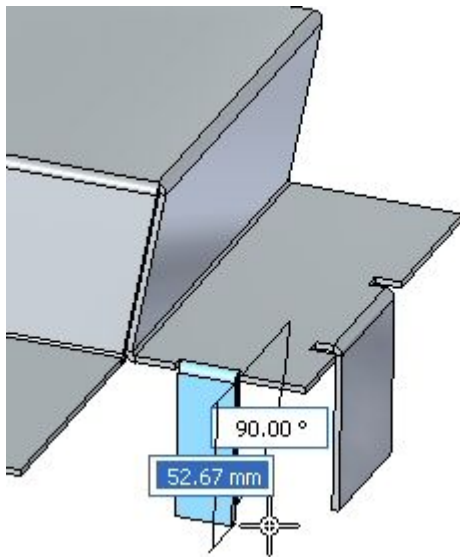
You can select a closed corner for deletion, either in PathFinder or in the graphics window. When you delete a closed corner, the corner definition is removed from the model and bends return to the default bend state.

Activity: Flange and corner conditions

Activity objectives

This activity demonstrates control flange geometry and end conditions within a sheet metal part. In this activity you will:

- Place flanges.
- Place partial flanges.
- Define and edit bend relief for bends.
- Defining corner conditions.
- Inserting a bend across a layer face.
- Rotating faces.



Turn to Appendix D for the activity: Flange and corner conditions.

Lesson review

Answer the following questions:

1. What is the purpose of bend relief in a sheet metal part?
2. How do you insert a synchronous bend into a sheet metal part?
3. How can you use the steering wheel to change the angle of a bend?
4. Can change and customize the values of the bend formula?
5. List three types of corner relief and describe each type.

Answers

1. What is the purpose of bend relief in a sheet metal part?

Bend relief eliminates tearing of the material when the material is being bent.

2. How do you insert a synchronous bend into a sheet metal part?

Sketch a line to define the bend. With the bend command, select the line define the bend angle.

3. How can you use the steering wheel to change the angle of a bend?

Select the face and move the origin of the steering wheel onto the bend. Click the torus and rotate it to change the angle of the bend.

4. Can change and customize the values of the bend formula?

Yes, the bend formula can be customized.

The standard sheet metal bend formula delivered with Solid Edge is:

$$PZL = \pi * (BR + (NF * THK)) * BA / 180$$

Where:

PZL = Plastic Zone Length

BR = Bend Radius

NF = Neutral Factor

THK = Material Thickness

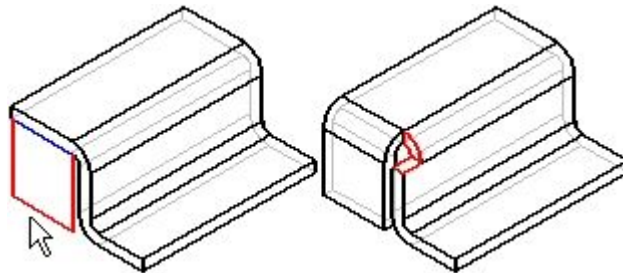
BA = Bend Angle

See the Help Topic entitled Sheet Metal Bend Formulas.

5. List three types of corner relief and describe each type.

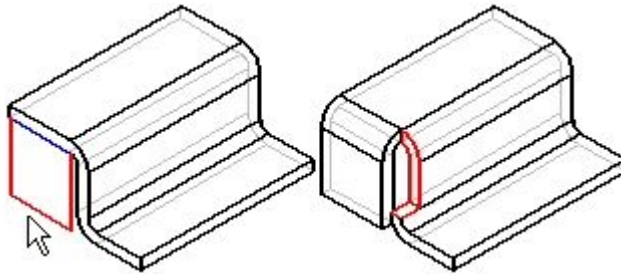
Bend Only

Specifies that corner relief is only applied to the bend portion of the adjacent flanges.



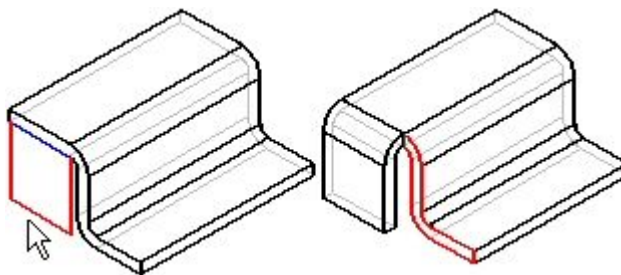
Bend and Face

Specifies that corner relief is applied to both the bend and face portions of the adjacent flanges.



Bend and Face Chain

Specifies that corner relief is applied to the entire chain of bends and faces of the adjacent flanges.



Lesson summary

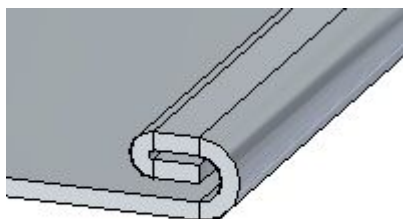
In this lesson you placed flanges and partial flanges. You edited end and corner treatments for bends. You used the closed the corners of adjacent thickness faces at the intersection of two bends. Bends were placed on the layer face and flanges were created and edited from these bends.

Lesson

7 *Hem*

Constructing a hem in a sheet metal part

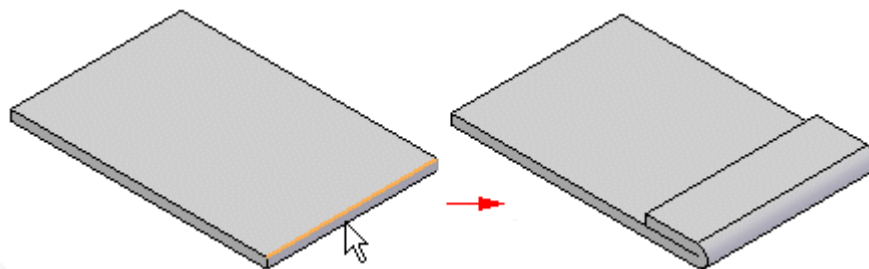
A hem feature creates a rigid edge for a sheet metal part. Modeling hems can be as easy as selecting the edges where you want to place them.



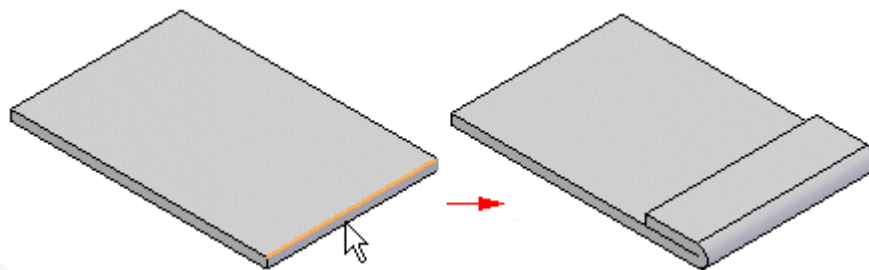
Hem command

Constructs a hem, where the material folds back.

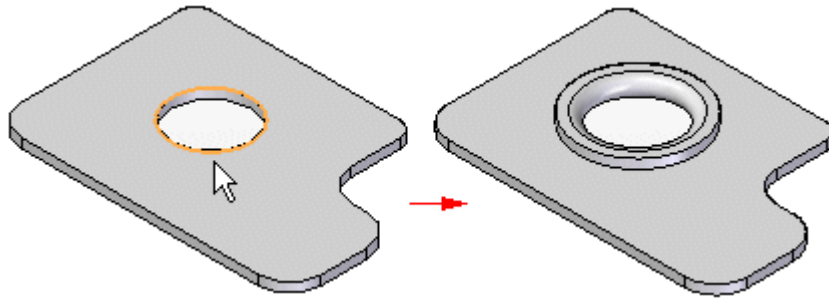
In the synchronous environment, you can construct a hem along a linear edge.



In the ordered environment, you can construct a hem along any edge on a sheet metal part. For example, you can construct a hem along a liner edge



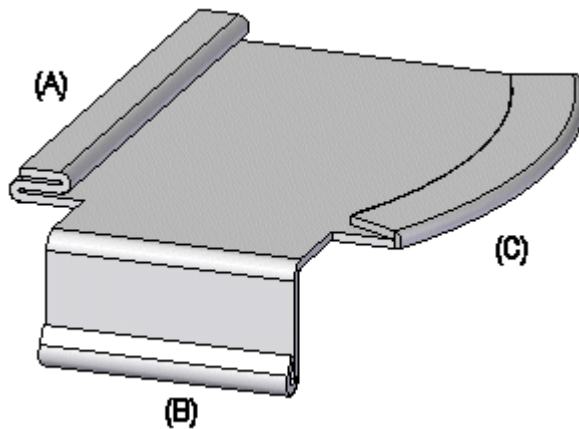
or, along the curved edge of a circular cutout.



Note


Bends created with the command are included in bend table.

You can use the [Hem Options dialog box](#) to specify the type of hem to be created. The Hem Type list contains several types of hems from which to choose. For example, you can define s-flange (A), loop (B), and closed (C) hems.



You can use the [Hem Options dialog box](#) to specify the type of hem to be created. The Hem Type list contains several types of hems from which to choose.

Construct a hem

1. Choose Home tab® Sheet Metal group® Contour Flange list® Hem .
2. Select the edge(s) for the hem.
3. Click to complete the hem.

Tip

- You can use the [Hem Options dialog box](#) to specify the type of hem to be created, along with bend radius, and flange length for the hem. The options that are available depend on the type of hem being created.

Hem Options dialog box

Saved Settings

Lists saved hem settings. You can access the saved settings by selecting them from the list. The settings on the dialog box display the characteristics of the hem you select. You can type a name in the box to name a group of settings.

You can then use the Saved Settings list on the Hem Options dialog box or the Hem command bar to select a saved setting later in any Solid Edge document that allows you to construct hem features. The saved settings are added to the Custom.xml file in the Program folder. You can also use the File Locations tab on the Options dialog box to specify a different folder for the Custom.xml file.

Save

Saves the current settings with the name you type.

Delete

Deletes the saved settings selected in the Save Settings box.

Hem Profile

Specifies the type of hem being created along with information such as bend radius, flange length, and sweep angle for the hem. The options that are available depend on the type of hem being created. A graphic displays an example of the selected hem type along with the location of the options available for the hem type.

Hem Type

Specifies the type of hem to be created.

Bend Radius 1

Specifies the bend radius for the first bend in the hem.

Flange Length 1

Specifies the length for the first flange in the hem.

Bend Radius 2

Specifies the bend radius for the second bend in the hem.

Flange Length 2

Specifies the length for the second flange in the hem.

Sweep Angle

Specifies the sweep angle for Open loop and Centered Loop hems.

Miter Hem

Miters the end of the hem when checked.

Bend Relief

Specifies that you want to apply bend relief to the source face from which the hem is constructed. When you set this option, you can also specify whether the bend relief is round or square, and whether the bend relief applies to only the material adjacent to the bend or to the entire face.

Square

Specifies that the internal corners of the bend relief are to be square.

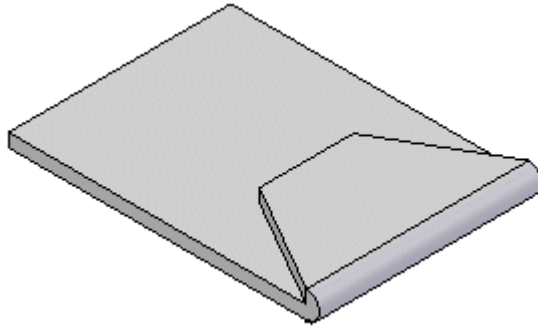
Round

Specifies that the internal corners of the bend relief are to be round.

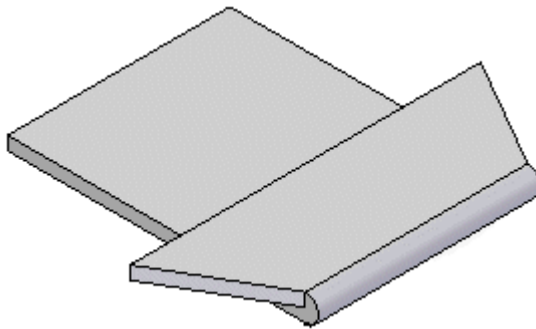
Angle

Sets the miter angle for the specified end of the hem.

A negative value will miter the flange inward and will typically remove material.



A positive value will miter the flange outward and will typically add material.

**Depth**

Specifies the depth of the bend relief.

Use Default Value

Uses the default value specified on the Options dialog box.

Width

Specifies the width of the bend relief.

Neutral Factor

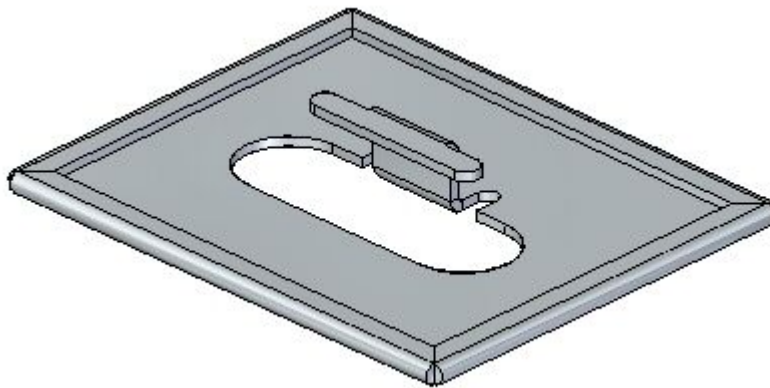
Specifies the neutral factor for the bend.

Activity: Using the hem command in sheet metal design

Activity objectives

This activity demonstrates how to create a hem on the edge of a sheet metal part. In this activity you will:

- Create a simple hem on a single edge of a sheet metal part.
- Vary the options for creating hems.
- Control the extent and end treatments of hems placed along adjacent thickness faces.



Turn to Appendix E for the activity: Using the hem command in sheet metal design.

Lesson review

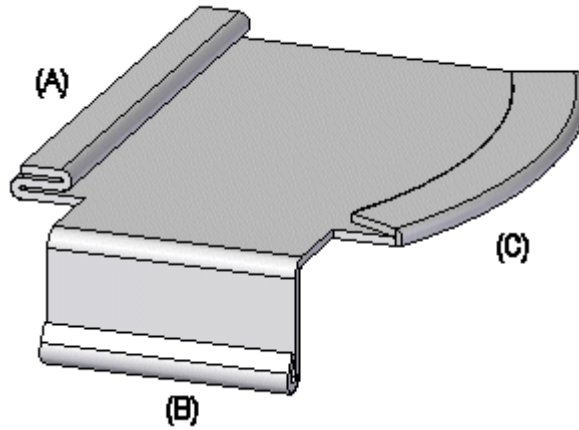
Answer the following questions:

1. Using hem options, define the three types of hems that can be created.
2. What is the difference between using a positive value versus a negative value in defining a hem mitre?
3. When creating a sheet metal hem feature, list at least three options needed to create the hem.

Answers

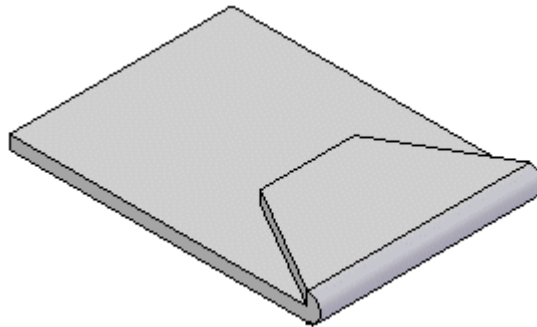
1. Using hem options, define the three types of hems that can be created.

You can use the Hem Options dialog box to specify the type of hem to be created. The Hem Type list contains several types of hems from which to choose. For example, you can define s-flange (A), loop (B), and closed (C) hems.

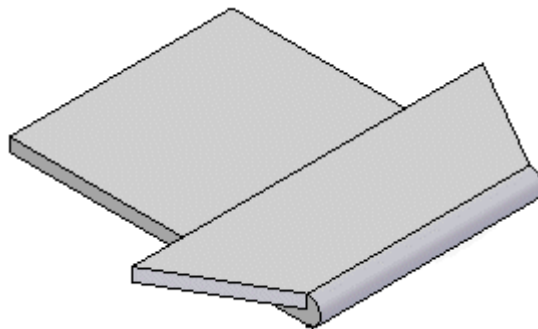


2. What is the difference between using a positive value versus a negative value in defining a hem mitre?

In hem mitre options, a negative value will miter the flange inward and will typically remove material.



In hem mitre options, a positive value will miter the flange outward and will typically add material.



3. When creating a sheet metal hem feature, list at least three options needed to create the hem.

Hem Type

Specifies the type of hem to be created.

Bend Radius 1

Specifies the bend radius for the first bend in the hem.

Flange Length 1

Specifies the length for the first flange in the hem.

Bend Radius 2

Specifies the bend radius for the second bend in the hem.

Flange Length 2

Specifies the length for the second flange in the hem.

Sweep Angle

Specifies the sweep angle for Open loop and Centered Loop hems.

Lesson summary

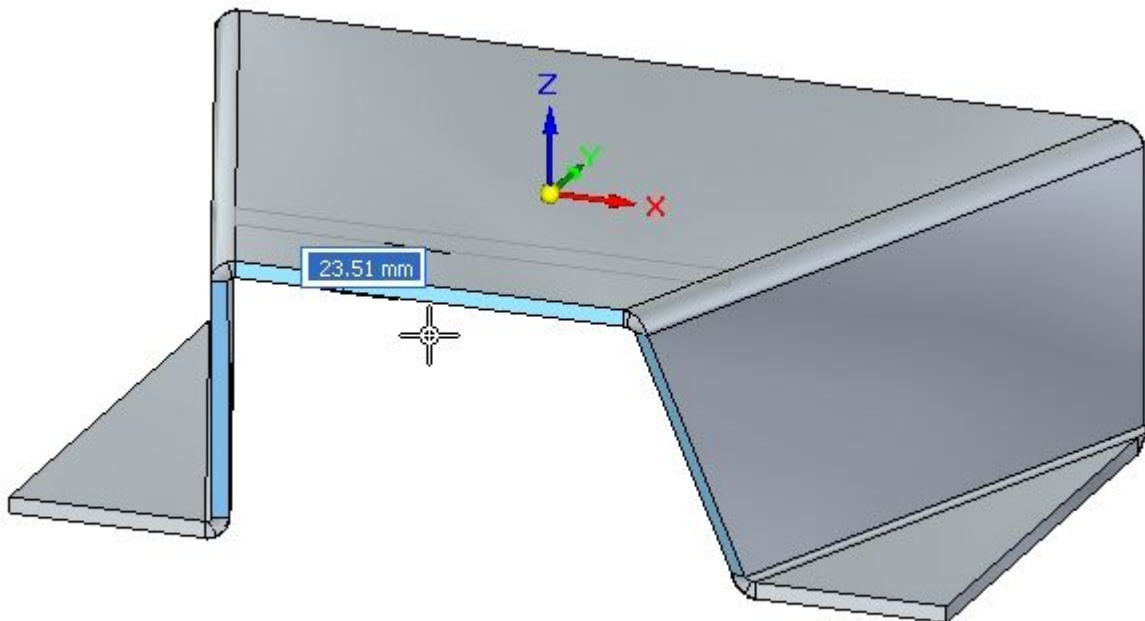
In this lesson you created a variety of hems in sheet metal parts. You learned how to set the parameters to create the hems, and how to edit the values when needed.

Lesson

8 *Using live rules in sheet metal*

Live rules in sheet metal

When you use the steering wheel to modify a portion of a model, Live Rules and relationships control how the rest of the model responds.



Working with Live Rules

Use the Live Rules options to locate and display the inferred geometric relationships between faces in the current select set and the rest of the model. Use this information to control how synchronous modifications are performed.

For example, when moving a planar face, use Live Rules to locate and display all the faces in the model that are coplanar to the face you are moving. Then use Live Rules to specify whether any, some, or all of these coplanar faces are moved when the selected face moves.

Live Rules is available for the following types of synchronous modeling modifications:

- Moving or rotating model faces or features in a synchronous part or assembly document.

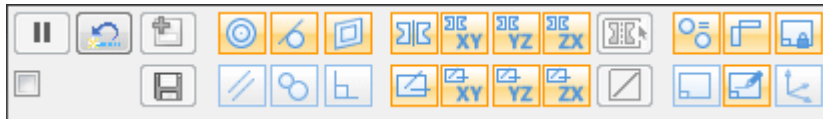
- Defining 3D geometric relationships between model faces using the Relate command in a synchronous part document.
- Editing the dimensional value of a 3D PMI dimension in a synchronous part or assembly document.
- Editing the dimensional value of a locked 3D PMI dimension using the Variable Table.

Note


Live Rules are not used when editing hole and round features using the Edit Definition handle.

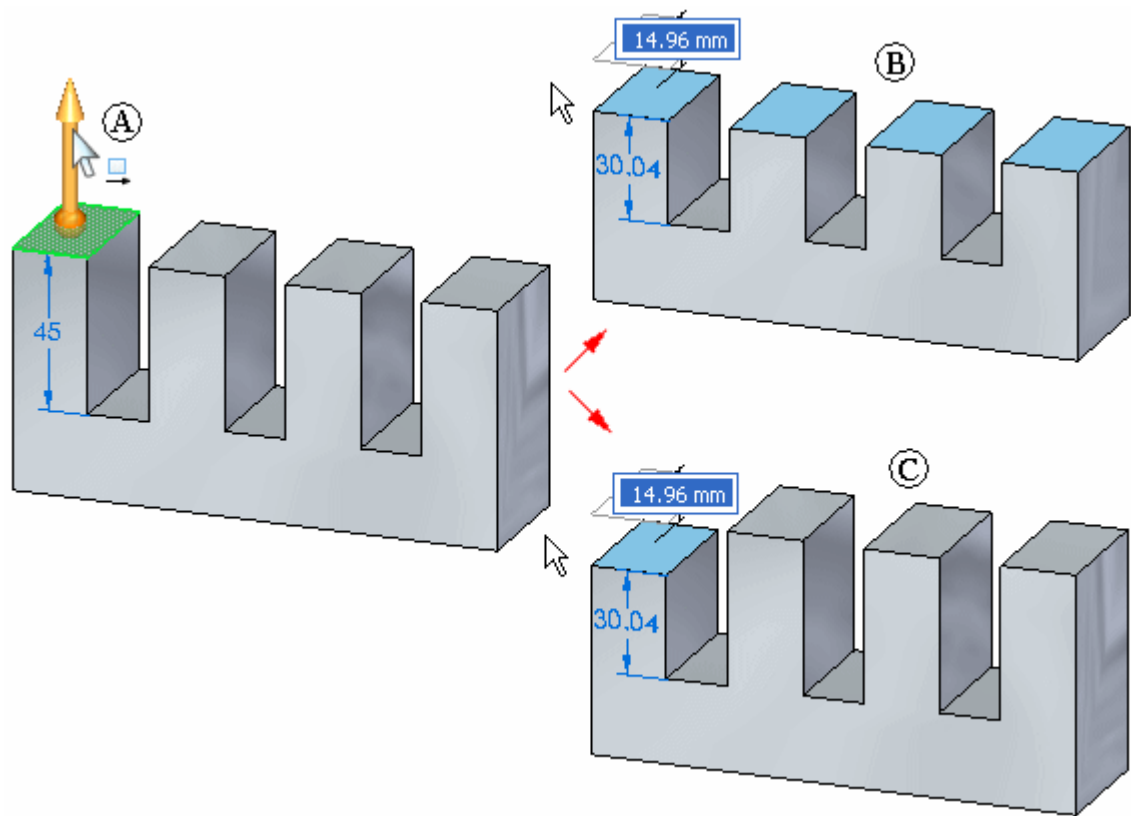
Live Rules options

Live Rules automatically appears when moving faces, defining 3D relationships, or editing dimensions. The active options in Live Rules determine how the rest of the model reacts to the edit you are performing.



For example, when moving a single planar face with the steering wheel (A), you can use Live Rules to specify whether other coplanar faces, which are not in the select set, stay coplanar during the move operation.

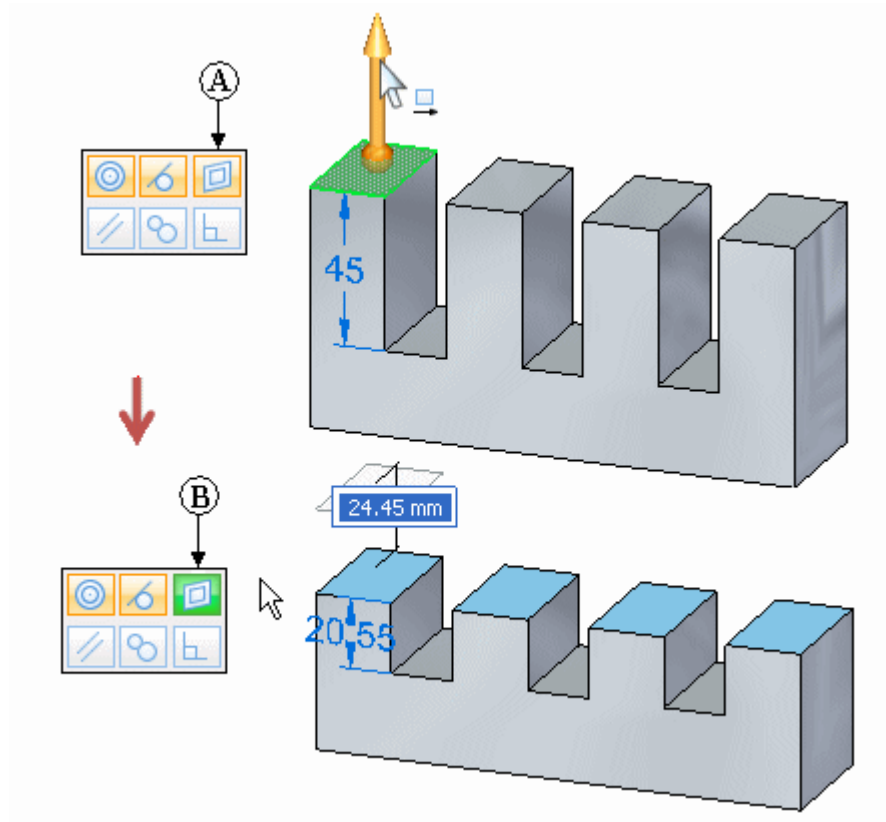
In this example, when the Coplanar option  in Live Rules is set, the deselected coplanar faces stay coplanar (B) when moving the selected face. When the Coplanar option in Live Rules is off, the deselected coplanar faces remain stationary (C) when moving the selected face.



Relationship detection indicators in Live Rules

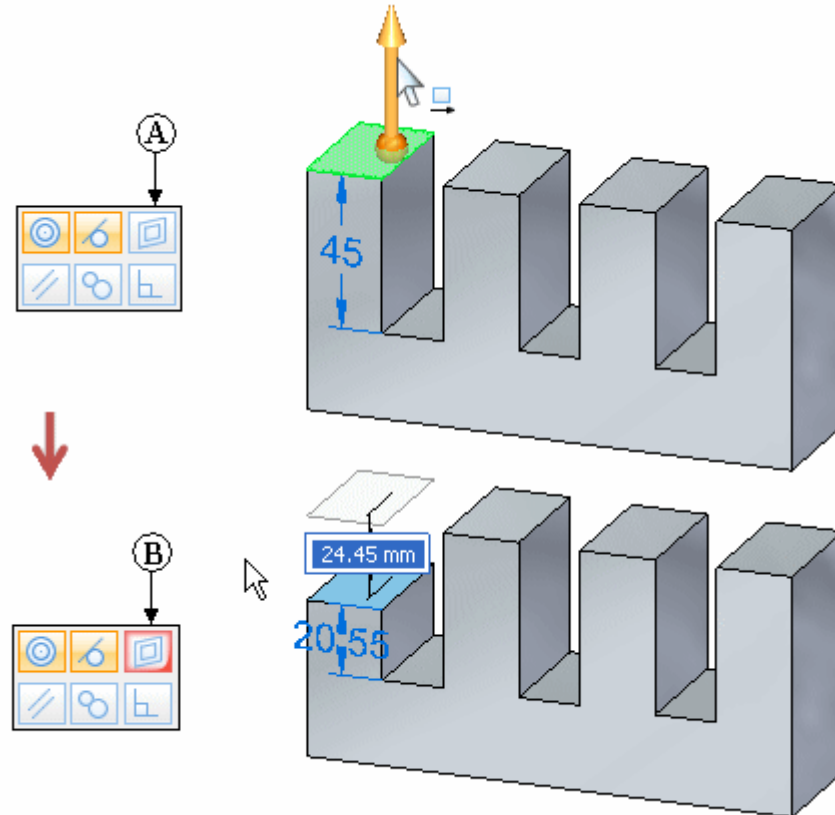
Detected and active

When Live Rules detects model geometry that matches an active setting (A) in Live Rules, the setting display in Live Rules appears in green (B).



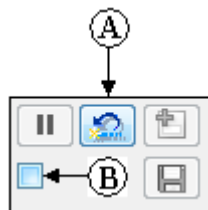
Detected and inactive

When Live Rules detects model geometry that matches an inactive setting (A) in Live Rules, the setting display in Live Rules appears in red (B).



The options you select or clear in the Maintain list for the current edit operation are saved for future edit operations in the current design session. When you exit Solid Edge, the current Live Rules settings are maintained for the next design session.


You can click the Restore Defaults button (A) to restore the default Live Rules options. You can also set the Suspend Live Rules option (B) to disable Live Rules for the current edit operation.



Note

Live Rules will not indicate recognized relationships for faces that have a persisted relationship applied.

Pausing model changes


When editing a synchronous part or assembly, you can use the Pause/Play button  in Live Rules to temporarily freeze the edit process. This makes it possible to closely examine the faces which are moving, dimensions values that are changing, and so forth. For more information, see [Examining synchronous model changes](#).

Advanced Live Rules dialog

The Advanced Live Rules dialog displays the selected geometry and any related, unselected geometry, based on the current Live Rules settings, in a tree structure. You can use the options on the Advanced Live Rules dialog to specify whether any, some or all related faces respond to the current settings in Live Rules.

Displaying the Advanced page

The Advanced Live Rules dialog is only available during a synchronous modification. For example, you can display the Advanced dialog after you have selected the primary axis on the steering wheel when moving a face.

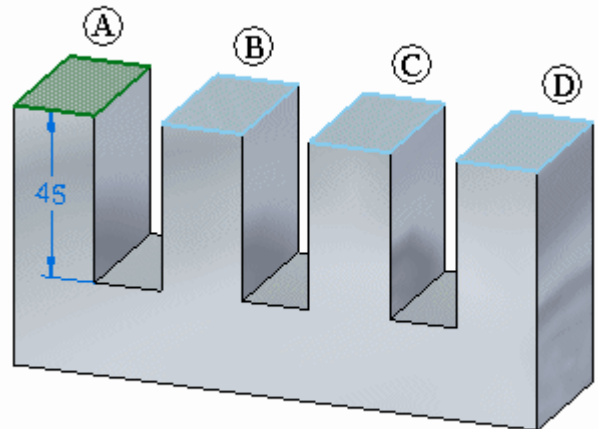
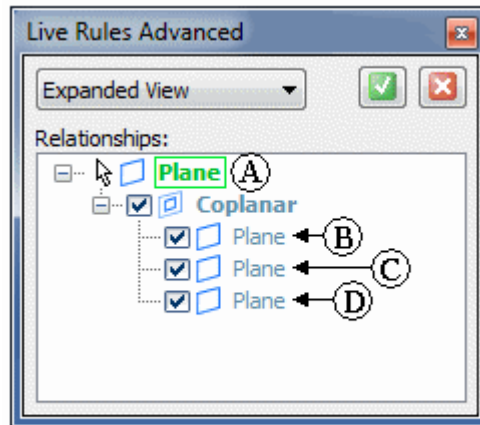
To show the Advanced Live Rules dialog, click the Advanced button , or press Ctrl+E.

Advanced Live Rules dialog example

When you display the Advanced dialog, the faces in the select set, and any deselected faces which match the active relationship criteria in Live Rules appear in a tree structure. For example, when moving a single planar face with the steering wheel, and the Coplanar option is set in Live Rules, the selected face (A), and the three deselected coplanar faces (B) (C) (D) appear in the tree structure on the Advanced dialog.

Notice that the color of the face entries on the Advanced dialog and the color of the faces on the model indicate which face(s) were selected, and which faces are deselected, but match the Live Rules relationship criteria. In this example, green is used for selected faces, and gray-blue is used for deselected faces.

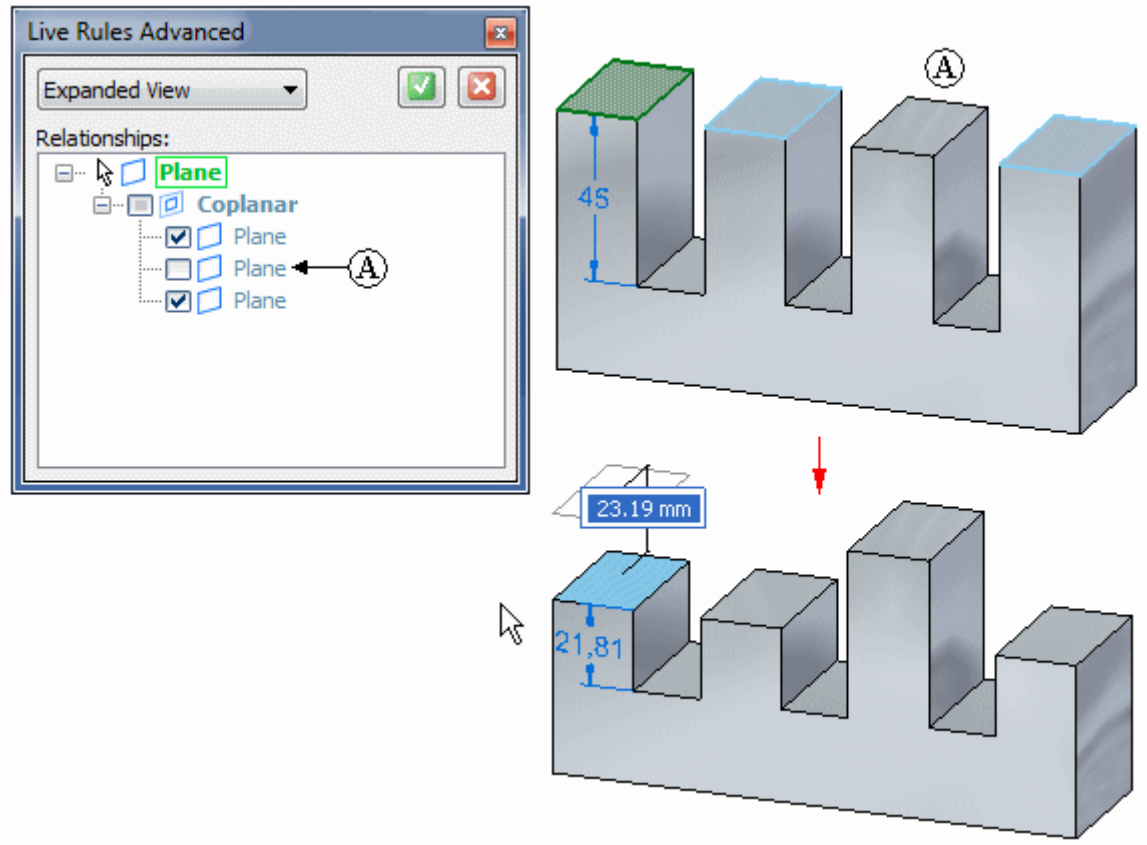
Also notice that the deselected faces are indented underneath the selected face and are grouped under a relationship heading, in this example Coplanar.



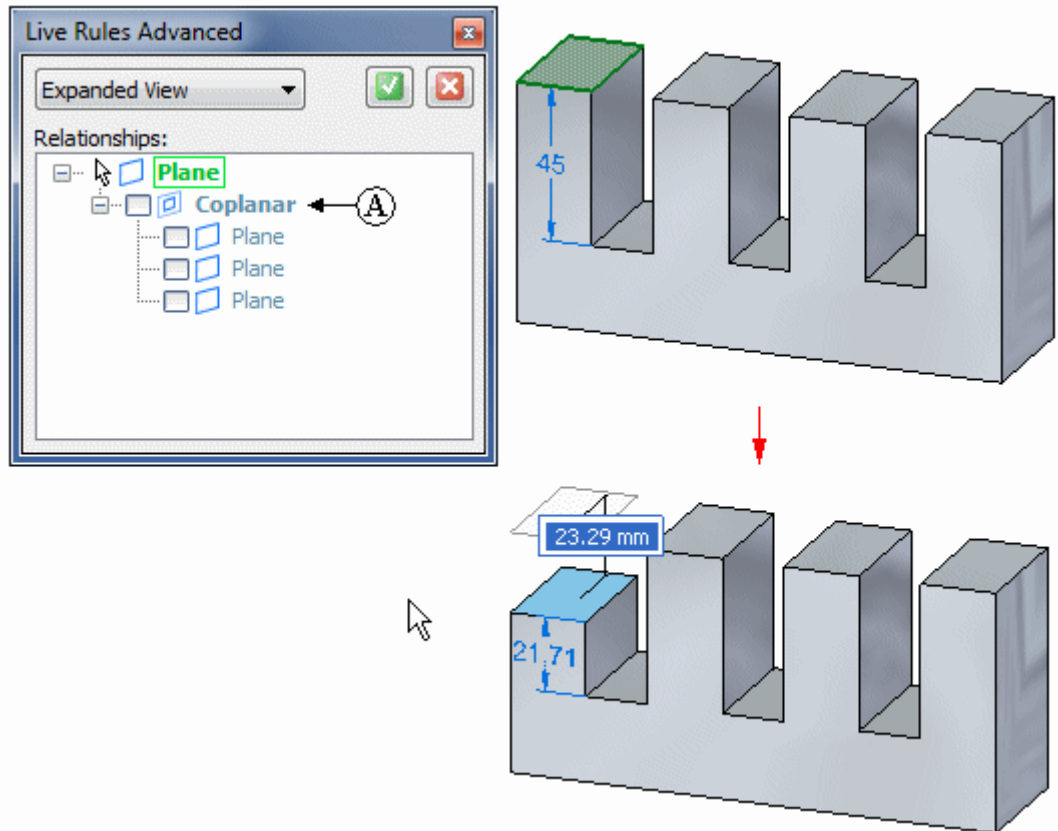
Using the Advanced Live Rules dialog

You can use the check box options on the Advanced dialog to control which faces are modified during a synchronous modification. For example, you can clear the check box for one of the deselected coplanar faces (A) to specify that you do not want that face to remain coplanar when you move the selected face. Notice that when you clear the check box for a face, the color for the face also changes in the graphics window to indicate that the face will not be modified.

After you have set the check box options you want, click the Accept (check mark) button to restart the synchronous modification with the changes you made. To cancel the changes to the Advanced dialog and restart the synchronous modification, you can click the Cancel (x) button.



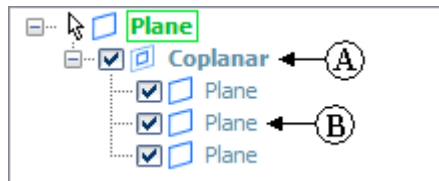
You can also clear the check box adjacent to the relationship heading (A) to specify you do not want any of the deselected coplanar faces to remain coplanar when you move the selected face.



Advanced Live Rules dialog shortcut menus

Two shortcut menus are available on the Advanced dialog:

- A Relationship shortcut menu is available when you right-click a relationship heading entry (A).
- An Element shortcut menu is available when you right-click a related element entry on the Advanced page, such as a face or reference plane (B).



Element shortcut menu

You can use the options on the Element shortcut menu to perform a variety of tasks. You can add the related element to the current select set, search the model for additional elements based on a geometric condition to the related element, and so forth. For example, you can specify that you want to search the model and add related elements to the Advanced dialog that

are coplanar to an element that is already in the related element list on the Advanced dialog.


Relationship shortcut menu

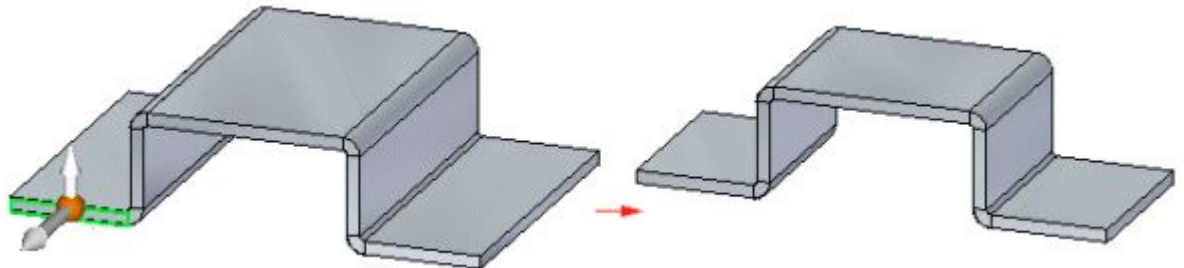
You can use the Select option on the Relationships shortcut menu to add the list of related elements under the selected relationship heading to the current select set.

You can use the Save option on the Relationships shortcut menu to define a persisted relationship between the elements under the selected relationship heading and the selected element.

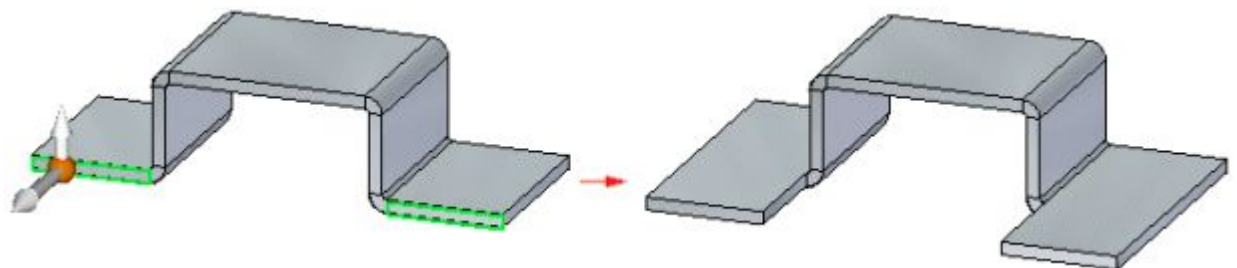
Live Rules in sheet metal modeling

Live Rules works the same in synchronous sheet metal modeling as it does in synchronous part modeling. An additional Live Rules option is available in the synchronous sheet metal modeling environment. The option is called Maintain Thickness Chain.

The Maintain Thickness Chain option  maintains the position of a thickness chain, made up of thickness faces connected by bends, during a move operation. When the Thickness Chain option is set, if you move one thickness face, the other connected faces move also.

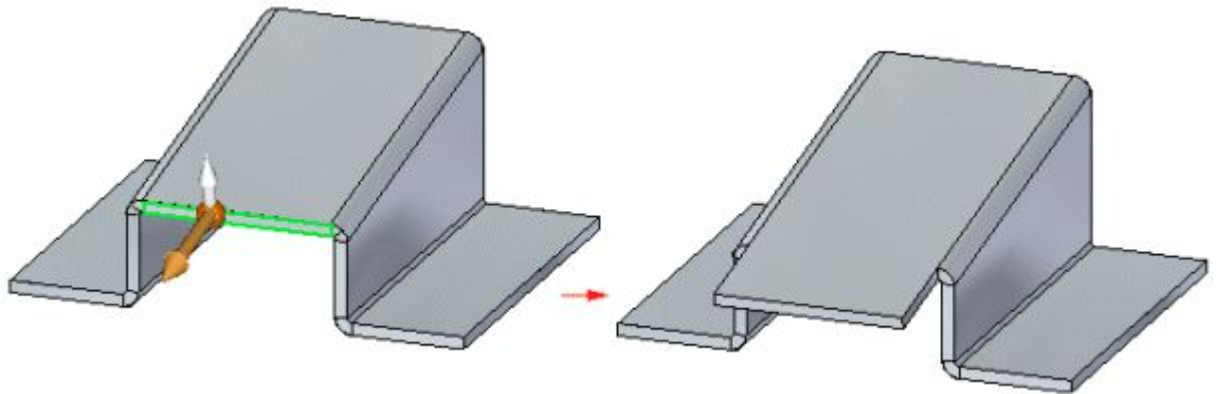


When the Thickness Chain option is not set, only the selected face or faces move.

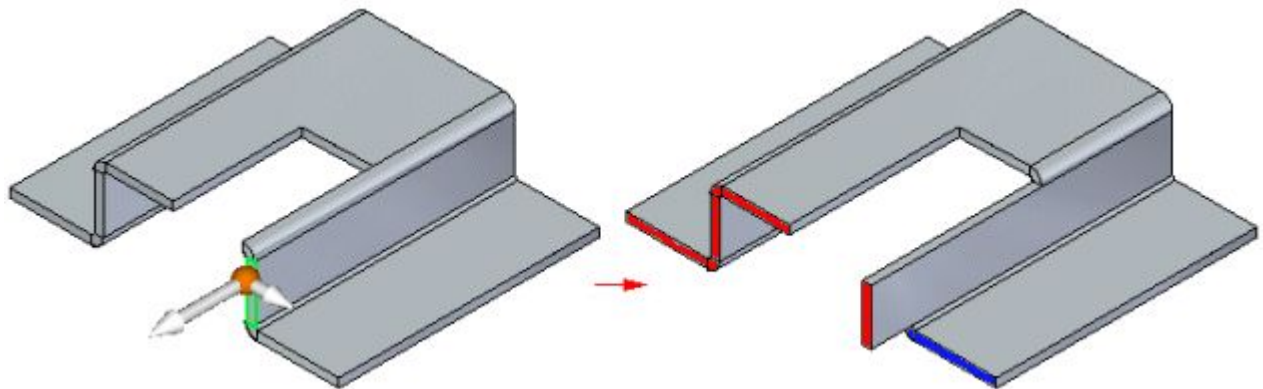


Selecting the Suspend Live Rules option does not affect the setting of the Thickness Chain option. In other words, if the Thickness Chain option is set and you select the Suspend Live Rules option, the Thickness Chain options remains set.

The Thickness Chain option ignores the Coplanar rule within the thickness chain so the thickness chain does not have to be coplanar to work.



Relationships are not detected between members of the same thickness chain, but are detected between members of separate chains. So even though the Coplanar rule is not detected within one thickness chain, it is detected from one thickness chain to another. In the following example, Symmetry and Thickness Chain are disabled. When the selected face is moved, the faces in red move also because they are coplanar and are part of a separate thickness chain. Since Thickness Chain is disabled and the Coplanar rule is not detected within the thickness chain containing the face selected to move, the blue face does not move.

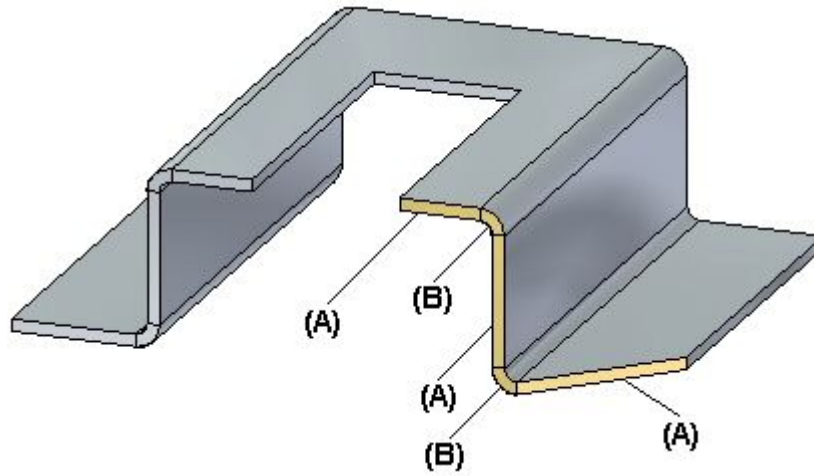


You may need to set the Orthogonal to Base if Possible option if you want to move or rotate a face that will cause a plate or thickness face to tip at an angle not orthogonal to the base reference plane.

Thickness chain

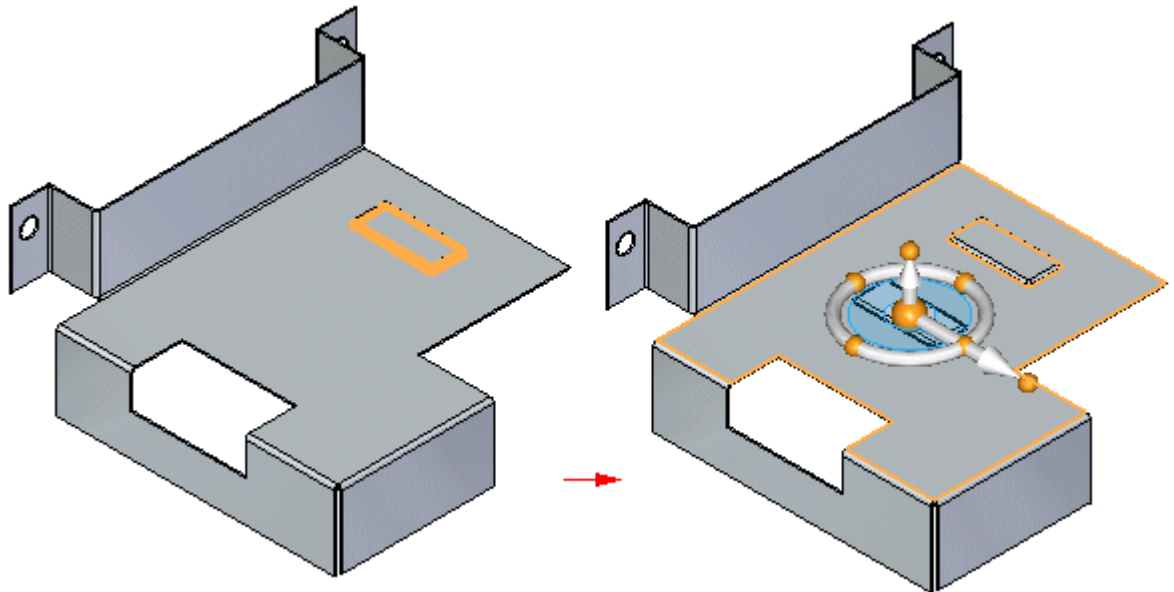
Thickness chain on a sheet metal part

A contiguous series of thickness faces (A) and bend end caps (B) in a sheet metal part.

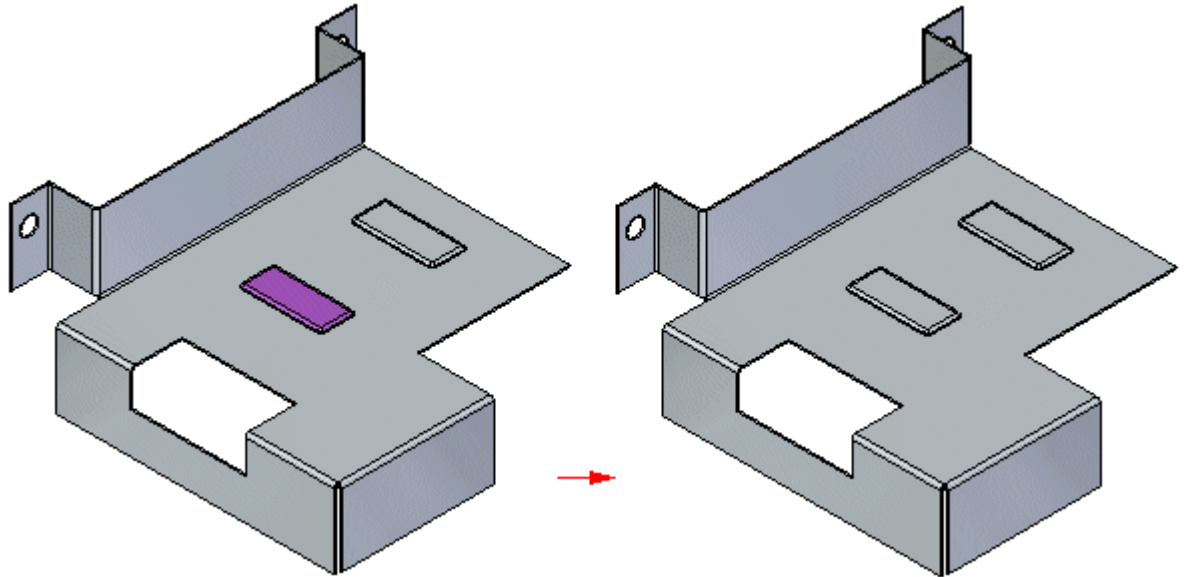


Copying, pasting, and attaching sheet metal features

Many times, you may find it useful to copy and paste an existing sheet metal feature rather than create a new feature.



Once you have pasted the feature, you can use the Attach command to attach the feature to the face.



Copying sheet metal features

You can copy for pasting or attaching:

- flanges, along with the associated bend
- procedural features
- sketches

You cannot copy for pasting or attaching:

- bends
- complex contour flanges
- hems
- dimensions
- closed corners
- gussets, unless the associated bend and flange are selected

When you copy a sheet metal element, the following is copied to the clipboard:

- steering wheel location
- attributes
- profile
- feature origin for procedural features

You can copy and paste multiple elements at once.

Copying elements in feature libraries

You can add eligible sheet metal features to a feature library for copy and paste. You can copy sketches between Synchronous Sheet Metal and Synchronous Part when creating feature libraries. However, the feature library cannot be a mixture of files. In other words, you cannot drag a Synchronous Part document from a feature library into a Synchronous Sheet Metal document.

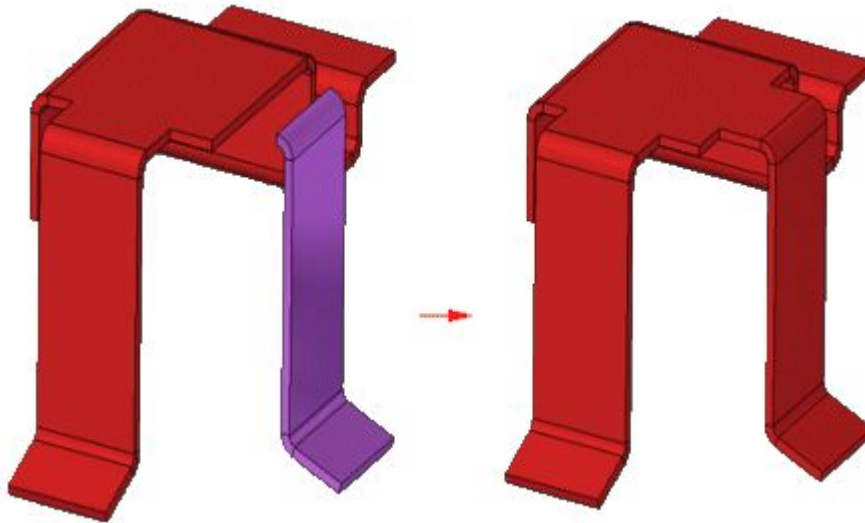
Pasting and attaching sheet metal features

The Paste command pastes features from the clipboard to the sheet metal model. When you paste a feature to a model, it is not actively added to the solid model. Once pasted, you use the Attach command to add the faces to the solid model.

When sheet metal elements are added to the model, they are added to PathFinder as distinct face sets. For example, each procedural feature has its own face set. Bend and flange combinations are added together as a flange entry.

As previously stated, you can use the Attach command to add faces that have been pasted from the clipboard to the solid model.

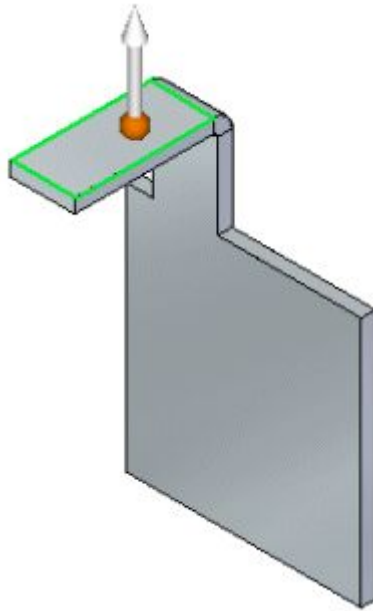
When using the Attach command you can use the extend to next and capping behavior to join the construction to the body for thickness faces.



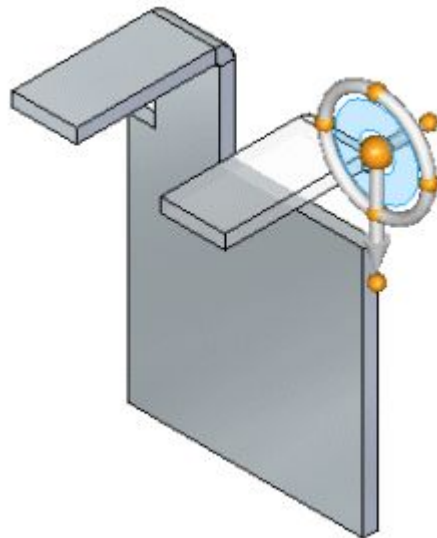
You cannot extend or trim manufactured features with the Attach command.

Copying, pasting, and attaching flanges

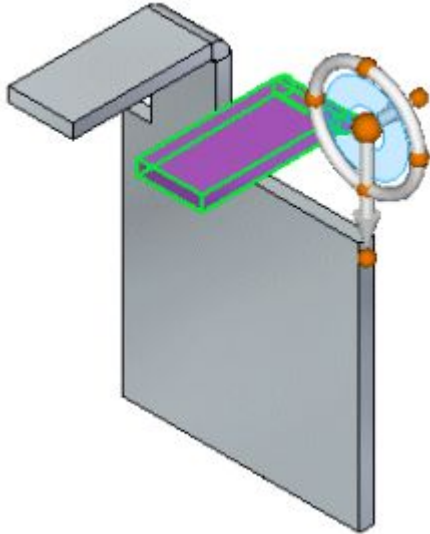
When you select a face of a flange to copy, all other faces in the flange are selected for copy.



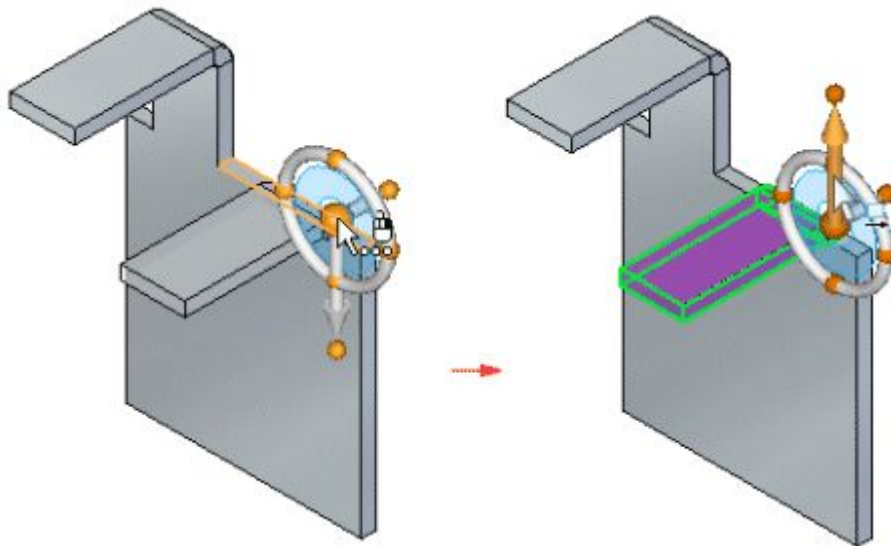
Once you copy the flange and then select the Paste command, a glass image of the flange attaches to the steering wheel.



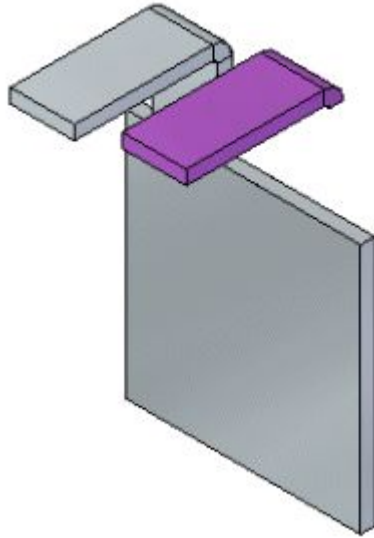
After you paste the flange, it appears as a detached flange.



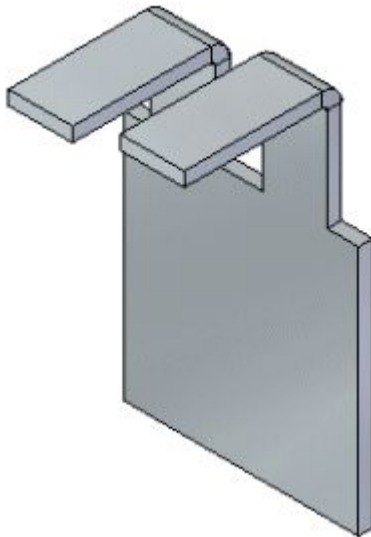
You can locate and lock to a plane and then use the steering wheel to adjust the position of the detached flange.



Once the position of the flange is valid so that it can be attached, you can use the Attach command to attach the flange to the main body of the model.

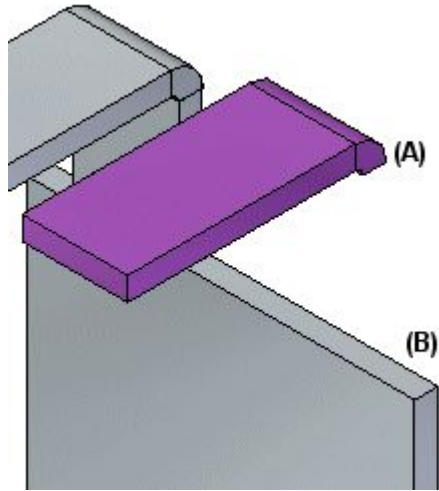


When the flange is attached to the main body, all other required faces are automatically created.

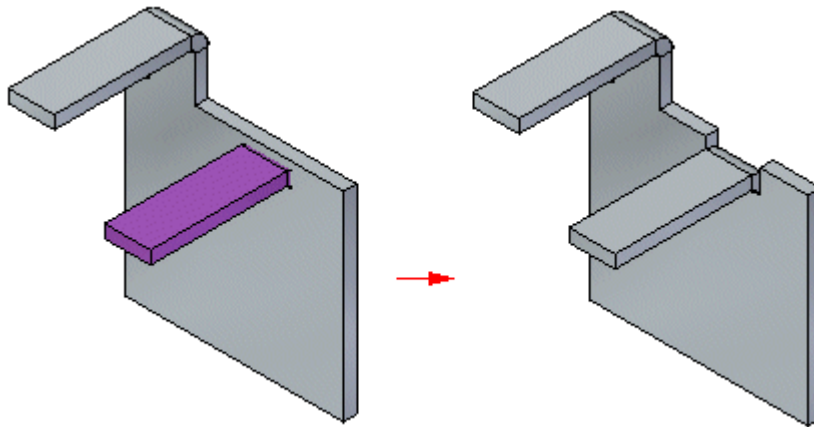


Note

When pasting a flange, the open side of the bend (A) must face and be tangent to the target plate (B).

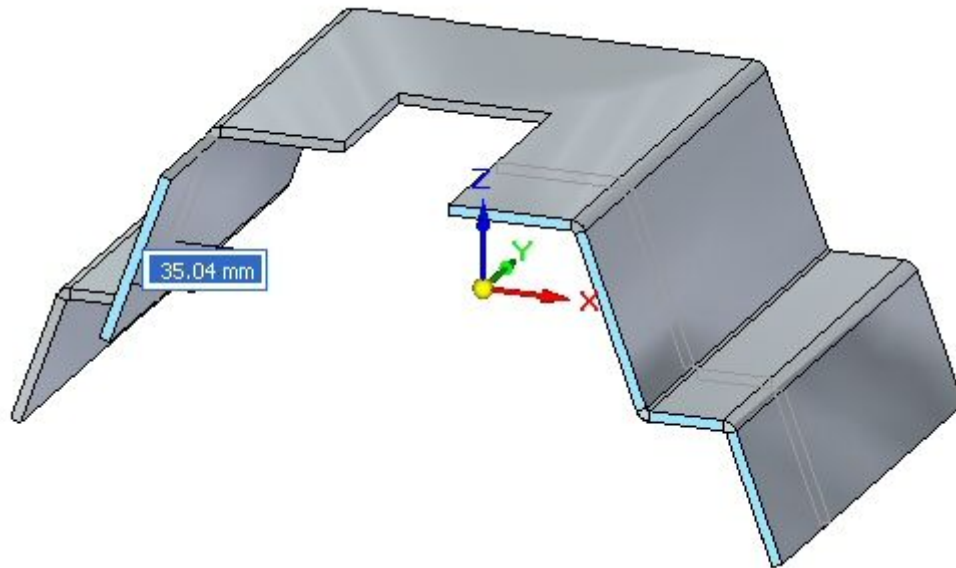


The flange bend can also be embedded into the boundary of the plate.

**Activity: Using live rules in sheet metal****Activity objectives**

This activity demonstrates how to control behavior when modifying sheet metal parts. In this activity you will:

- Explore Live rules.
- Establish relationships to control behavior of faces.
- Mirror, copy, cut and paste sheet metal features.



Turn to Appendix F for the activity: Using live rules in sheet metal.

Lesson review

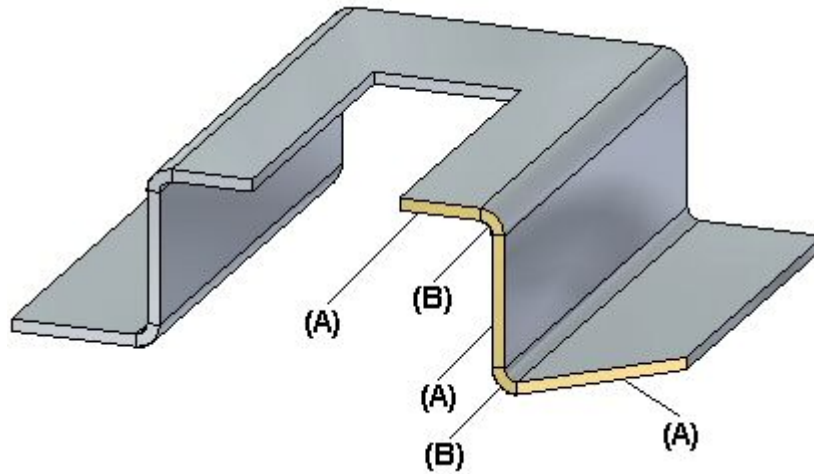
Answer the following questions:

1. Define sheet metal thickness chain.
2. Describe how the maintain thickness chain option affects the movement of a face contained in the thickness chain.
3. Describe how live rules affects faces of the same thickness chain as opposed to another thickness chain.

Answers

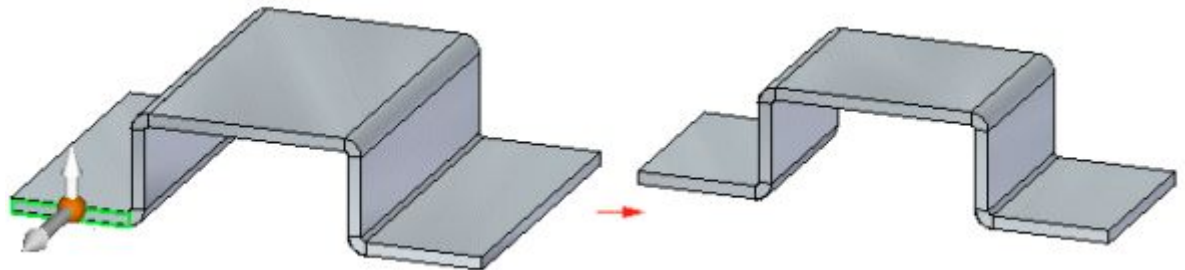
1. Define sheet metal thickness chain.

A thickness chain is made up of continuous thickness faces (A) connected by bend end caps (B).

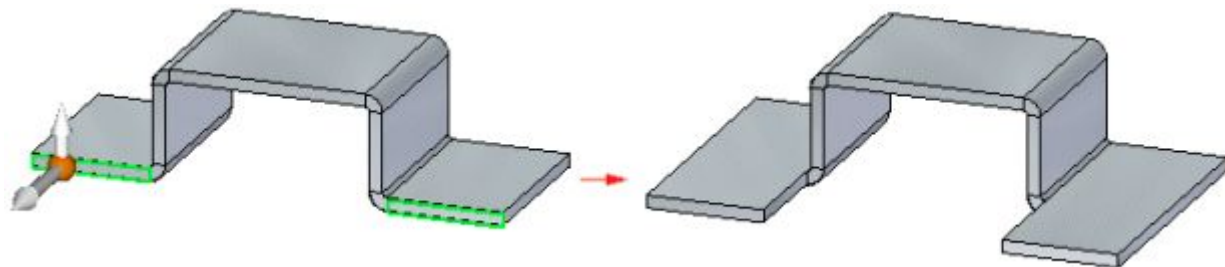


- Describe how the maintain thickness chain option affects the movement of a face contained in the thickness chain.

When the Thickness Chain option is set, if you move one thickness face, the other connected faces move also.



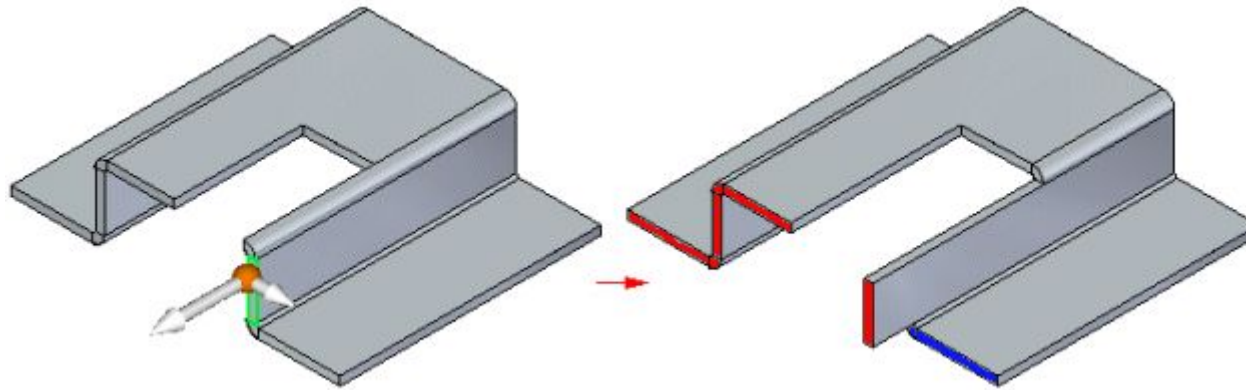
When the Thickness Chain option is not set, only the selected face or faces move.



- Describe how live rules affects faces of the same thickness chain as opposed to another thickness chain.

Relationships are not detected between members of the same thickness chain, but are detected between members of separate chains. So even though the Coplanar rule is not detected within one thickness chain, it is detected from one thickness chain to another. In the following example, Symmetry and Thickness Chain are disabled. When the selected face is moved, the faces in red move also because they are coplanar and are part of a separate thickness chain. Since

Thickness Chain is disabled and the Coplanar rule is not detected within the thickness chain containing the face selected to move, the blue face does not move.



Lesson summary

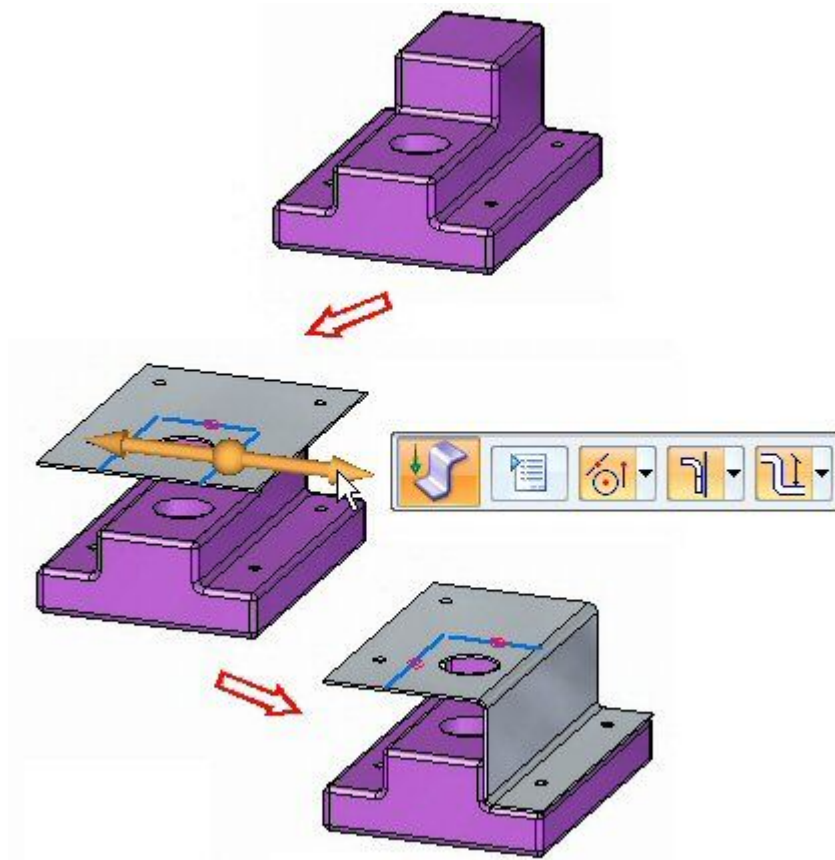
In this lesson you explored the behavior of sheet metal geometry by creating relationships and changing live rules.

Lesson

9 *Jog*

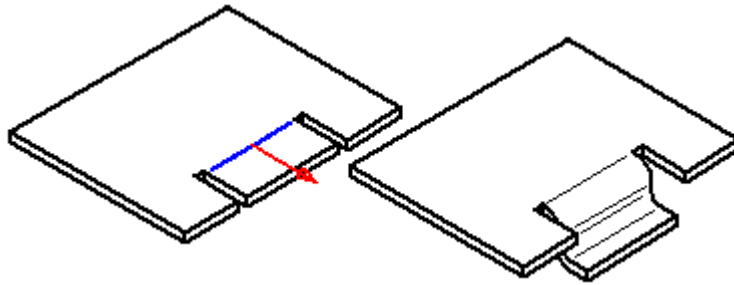
Constructing a Jog in a sheet metal part

A jog constructs an offset face with a connecting flange and maintains the positions of any features contained on the face, such as holes and deformation features.



Jog command

Constructs two bends to add a jog to a planar face of a sheet metal part. In the ordered environment, the profile for a jog feature must be a single linear element. In the synchronous environment, the sketch element used to construct the jog must be a single line that is coplanar with the face being bent. The jog can be minimal: for example, a slight offset or step to provide clearance or rigidity to a part.



Jog QuickBar

Bend Options

Displays the Bend Options dialog box so you can set the bend construction options.

Material Side

Positions the bend with respect to the material side.

Material Inside Positions the portion of the feature that is perpendicular to the profile plane such that it is inside of the profile plane.

Material Outside Positions the portion of the feature that is perpendicular to the profile plane such that it is outside of the profile plane.

Bend Outside Positions both the portion of the feature that is perpendicular to the profile plane and the bend such that they are outside of the profile plane.

Measurement Point

Measures from the selected face to the inside or outside of the new feature.

Measurement Inside Measures from the selected face to the near side of the feature.

Measurement Outside Measures from the selected face to the far side of the feature.

Keypoints

Sets the type of keypoint you can select to define a feature extent or to position a new reference plane. You can then define the feature extent or the location of the reference plane using a keypoint on other existing geometry. The available keypoint options are specific to the command and workflow you use.



Selects any keypoint.



Selects an end point.



Selects a midpoint.





Selects the center point of a circle or arc.



Selects a tangency point on an analytic curved face such as a cylinder, sphere, torus, or cone.

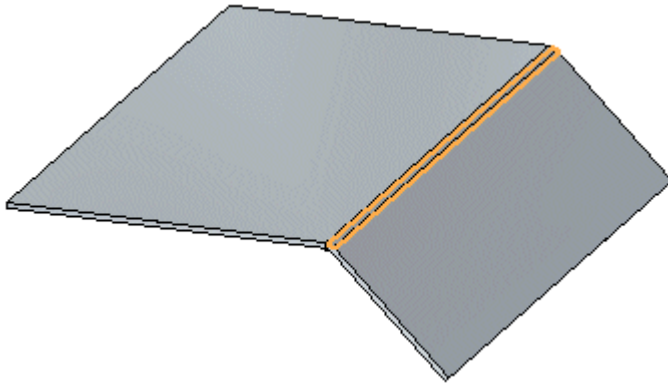


Selects a silhouette point.

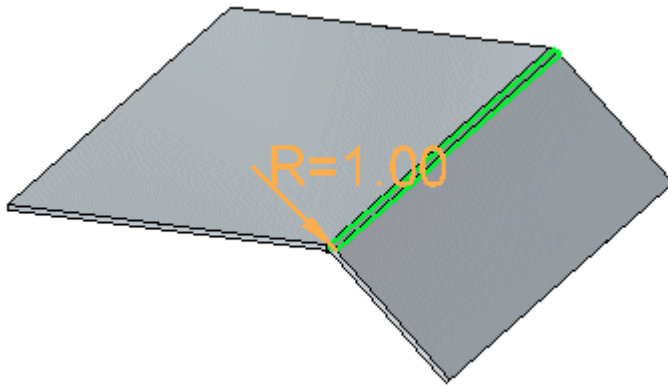
-  Selects an edit point on a curve.
-  Sets the no keypoint option.

Editing the bend radius

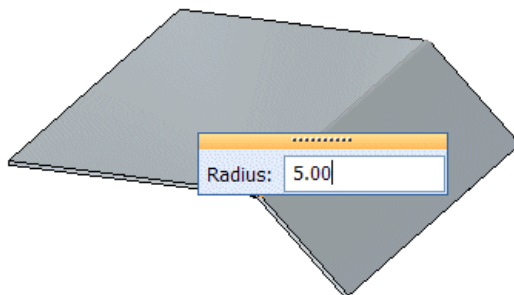
When you create a bend, the bend radius is defined from the default global parameters. Once a bend is created, it can be edited by selecting the bend,



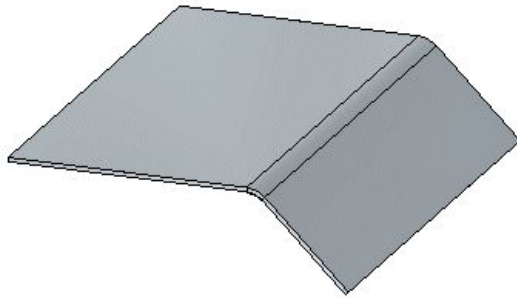
then selecting the bend radius handle,



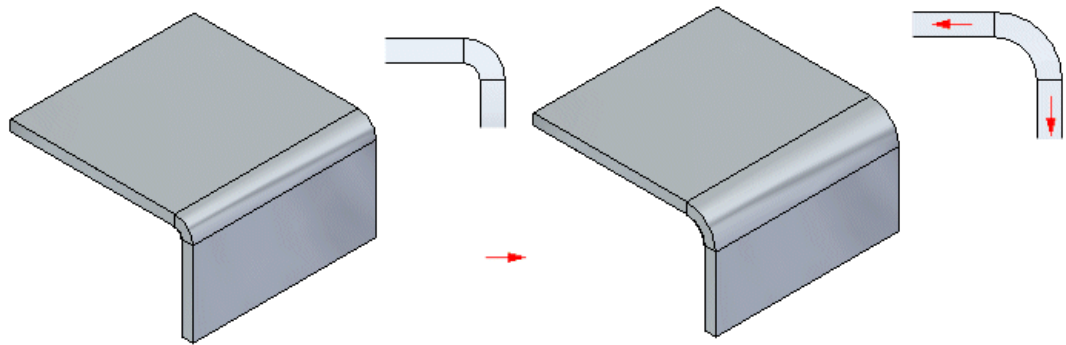
then typing a new value in the dynamic edit control,



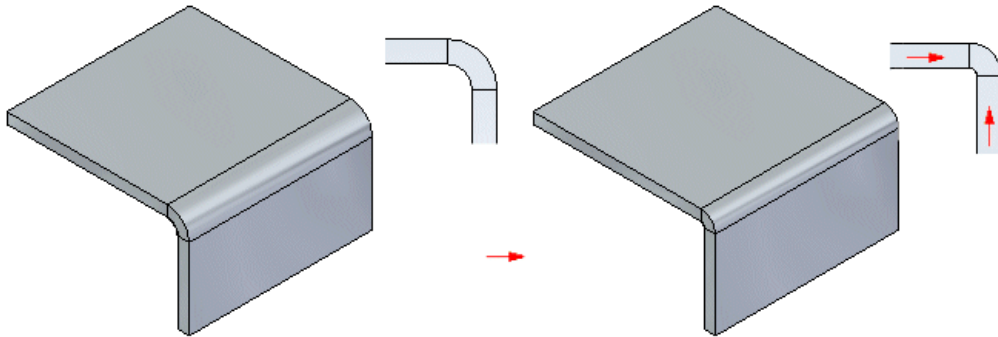
and then clicking to finalize editing the radius.



If you increase the bend radius, bend relief is applied to the model and the bend extends into the flange.

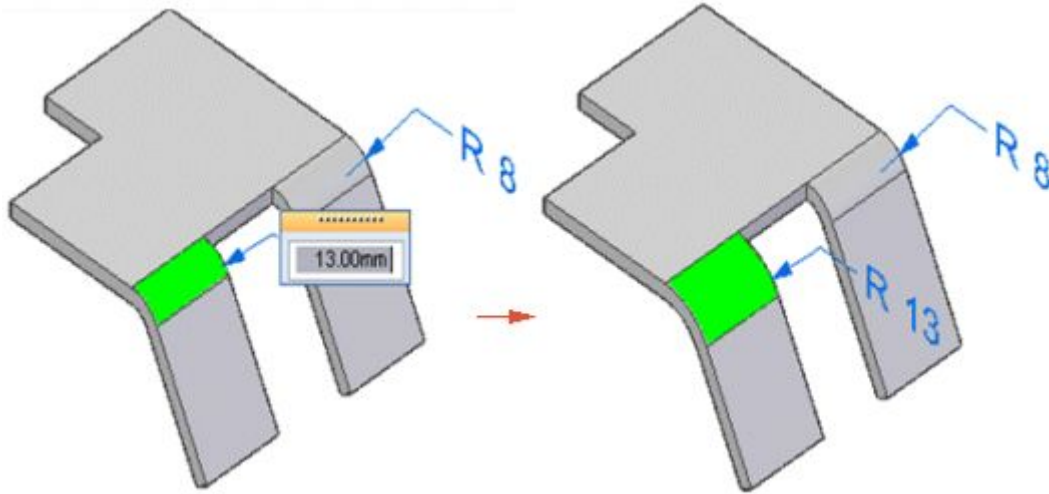


If you decrease the bend radius, the tab and flange extend into the bend to decrease the bend radius.

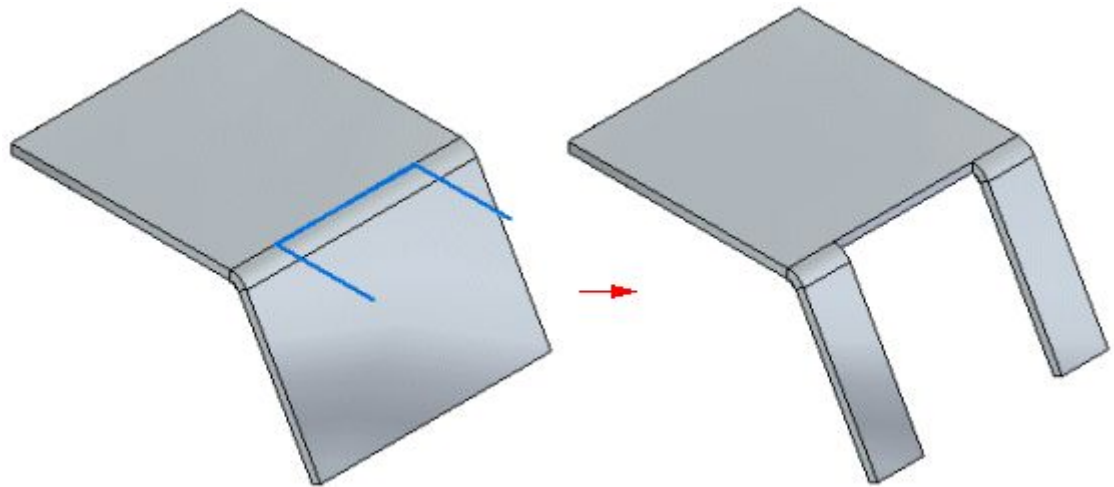


Editing the bend radius in split bends

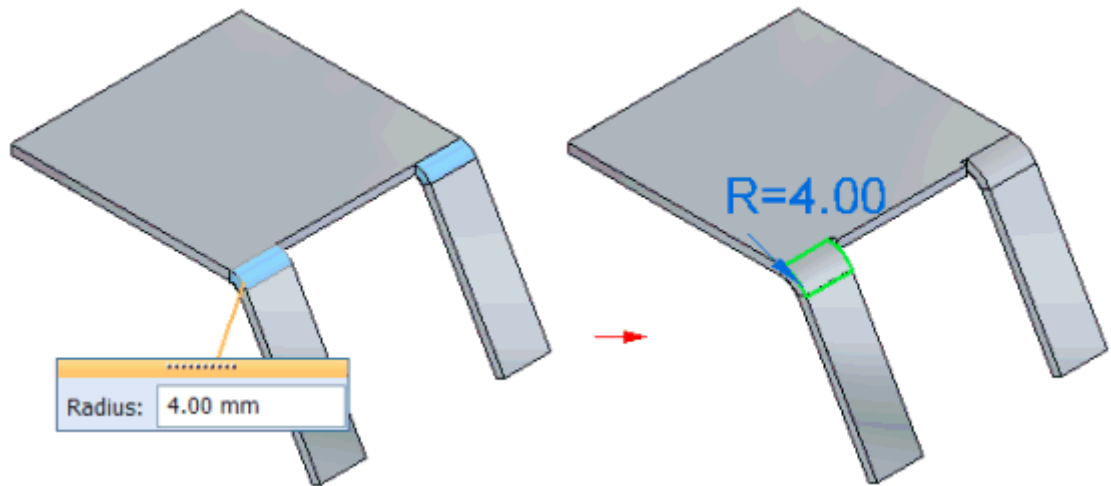
Depending on how split bends are created, the bend radius for split bends may be edited separately or together. If the split bends are created when two flanges are created independently as partial flanges, the bend radius can be edited separately.



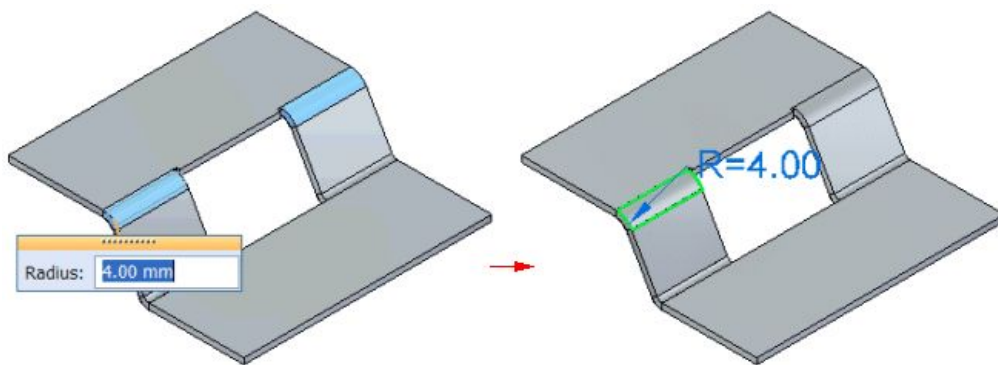
If the split bends were originally a single bend in a flange, and then cut to make two separate flanges,



the bend radius is edited together.



If the flanges on both sides of the bend are bound by the model, the bend radius for each bend row is edited together. For example, if you edit the radius for the top bend, the radius is updated for the top bend in both flanges.

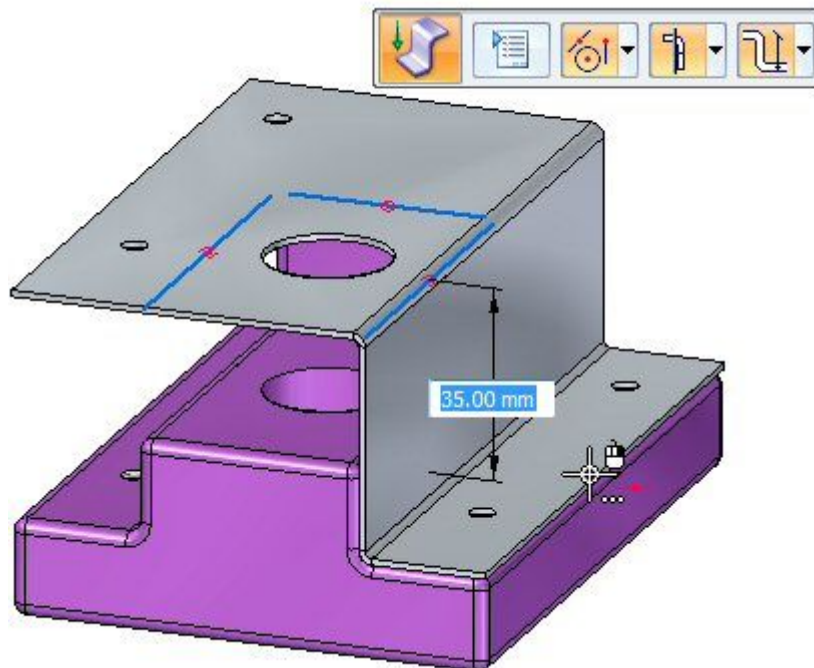


Activity: Using the jog and break corner command in sheet metal design

Activity objectives

This activity demonstrates how to create a jog in a sheet metal tab, and place flanges and trim away unwanted material from the part. In this activity you will:

- Create a tab based on reference geometry.
- Create the sketches needed by the jog command.
- Set the parameters for the jog.
- Modify the bend radius when and where appropriate.
- Use the break corner command to get rid of sharp corners.



Turn to Appendix G for the activity: Using the jog and break corner command in sheet metal design.

Lesson review

Answer the following questions:

1. What does the jog command do in a sheet metal document?
2. Explain the material side options when placing a jog?

Answers

1. What does the jog command do in a sheet metal document?

A jog constructs an offset face with a connecting flange and maintains the positions of any features contained on the face, such as holes and deformation features.

2. Explain the material side options when placing a jog?

The material side positions the bend with respect to the material side. The options are:

Material Inside	Positions the portion of the feature that is perpendicular to the profile plane such that it is inside of the profile plane.
Material Outside	Positions the portion of the feature that is perpendicular to the profile plane such that it is outside of the profile plane.
Bend Outside	Positions both the portion of the feature that is perpendicular to the profile plane and the bend such that they are outside of the profile plane.

Lesson summary

In this lesson you created a sheet metal base feature and used the jog command to form the sheet metal around an existing part. You modified the bend radius where needed and then used the cutout command and the break corner command to finish the model.

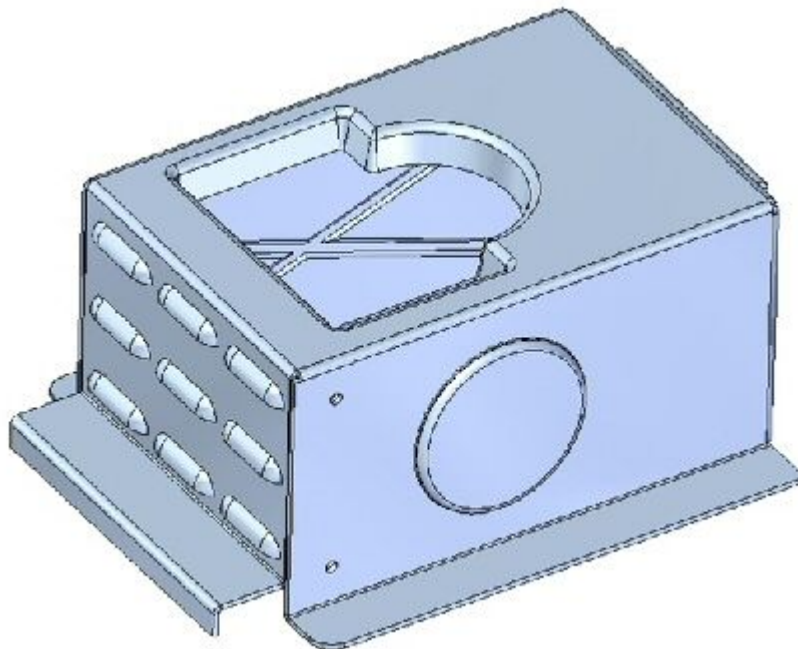
Lesson

10 *Deformation features*

Deformation features in a sheet metal part

Deformation features model features on the thickness faces of sheet metal parts, such as louvers, beads, dimples, drawn cutouts, and gussets, that can be manufactured by striking the stock with a tool. The values you use to define deformation features as you create them are stored with the features, and you can edit them later. Also the feature origin, or strike point, of the feature is positioned on the face such that if the face is later rotated or a jog is added, the feature will remain positioned. The feature can be relocated by modifying the position of the feature origin.

Deformation features consist of louvers, beads, dimples, drawn cutouts, and gussets.

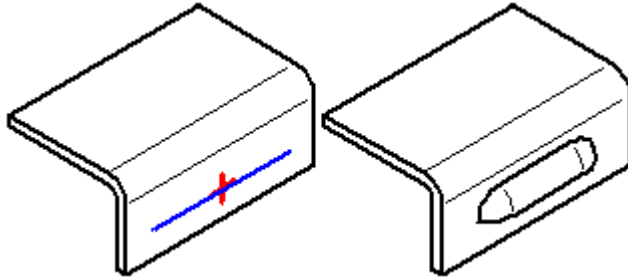


Adding sheet metal deformation features

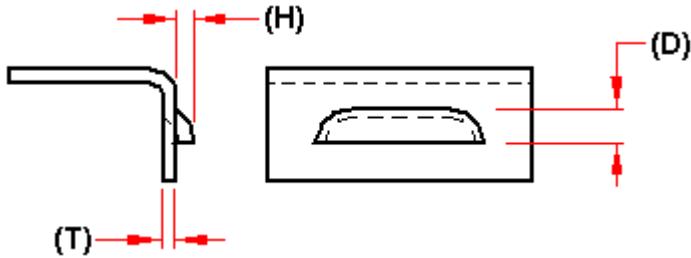
You can model features in the Sheet Metal environment that are manufactured with metal deformation techniques, such as deep drawing and coining. When parts are manufactured using deformation techniques, material thinning typically occurs. In Solid Edge, this material thinning is ignored and deformation features are constructed using the same material thickness specified for the model.

Constructing louvers

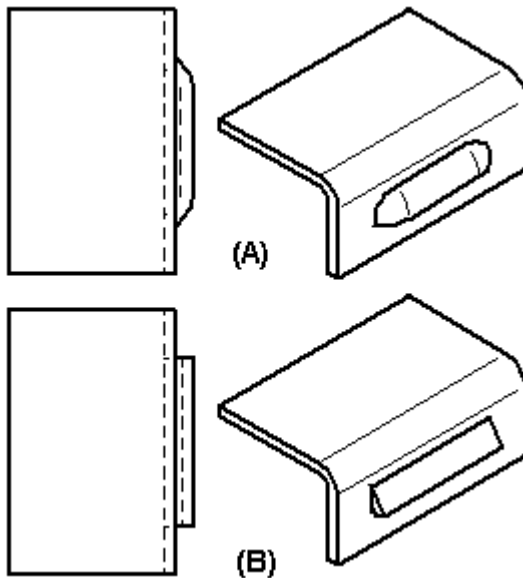
Like a jog feature, a louver feature is constructed using a single, linear element.



When constructing a louver, the louver height (H) must be equal to or less than the louver depth (D) minus the material thickness (T).



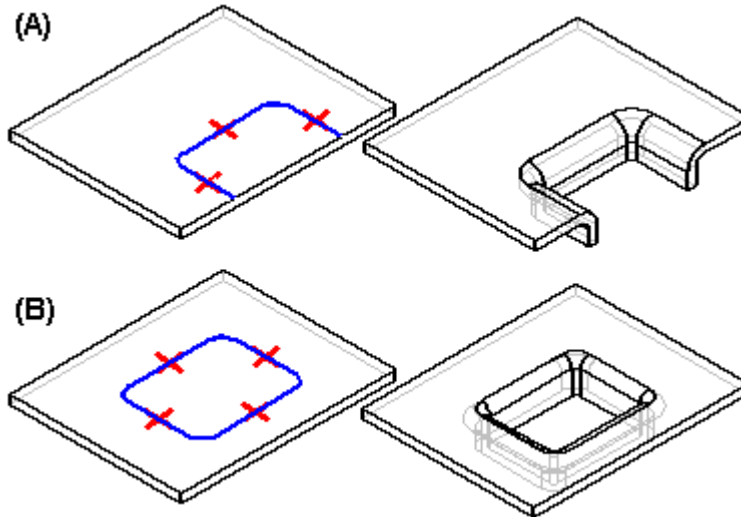
You can also specify whether you want the louver ends formed (A) or lanced (B) using the Louver Options dialog box.



Louver features cannot be flattened.

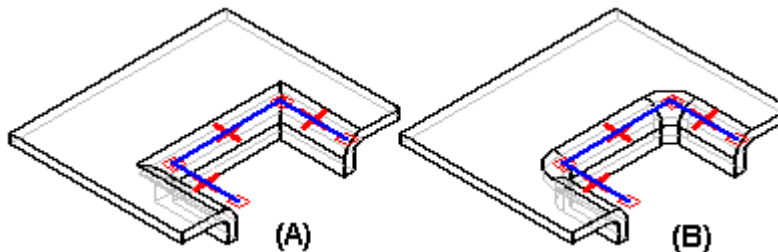
Constructing drawn cutouts

You can construct a drawn cutout using an open profile (A) or a closed profile (B).

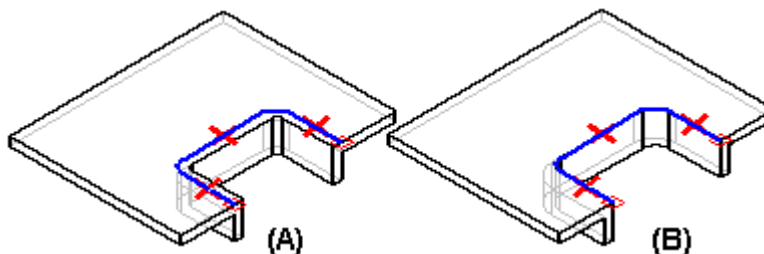


The ends of an open profile must theoretically intersect a part edge. A closed profile cannot touch any part edges. A drawn cutout can be constructed only on a planar face. You can use the Drawn Cutout Options dialog box to specify punch radius, die radius, and taper options.

When you draw the profile for a drawn cutout without arcs, you also can specify whether the corners are mitered (A), or rounded (B) using the Automatically Round Profile Corners option on the Drawn Cutout Options dialog box.



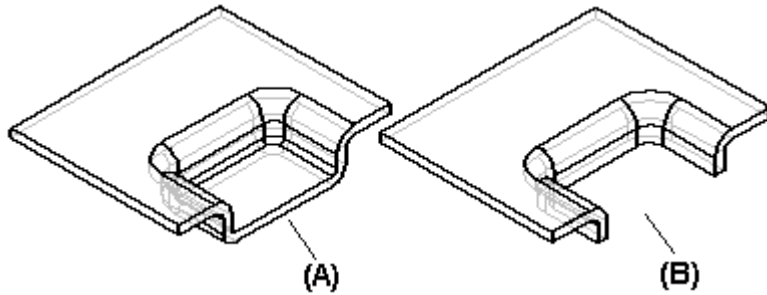
When you construct a drawn cutout, the sidewalls are constructed such that they lie inside the profile (A). After the feature is constructed, you can use the options to specify that the sidewalls lie outside the profile (B).



Drawn cutouts cannot be flattened.

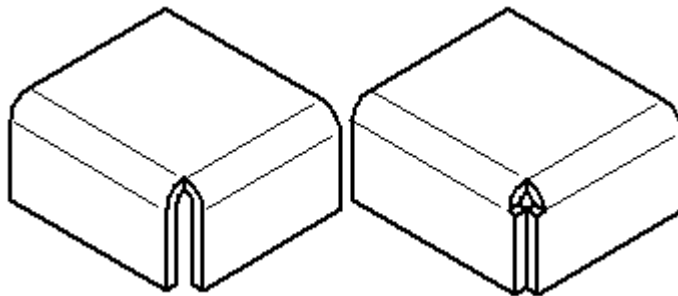
Constructing dimples

Constructing a dimple is just like constructing a drawn cutout. The principal difference between the two features is that a dimple has a "bottom" (A), and a drawn cutout (B) does not.

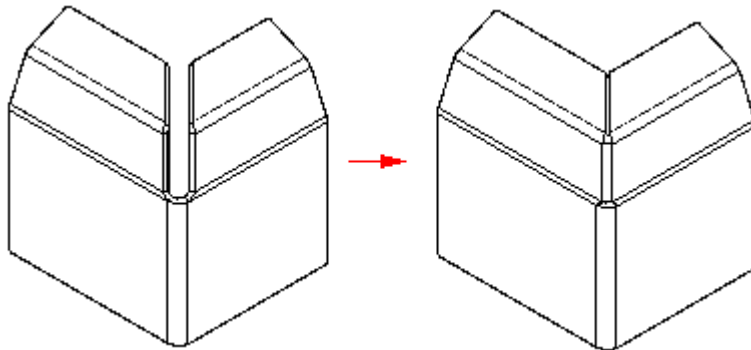


Closing corners

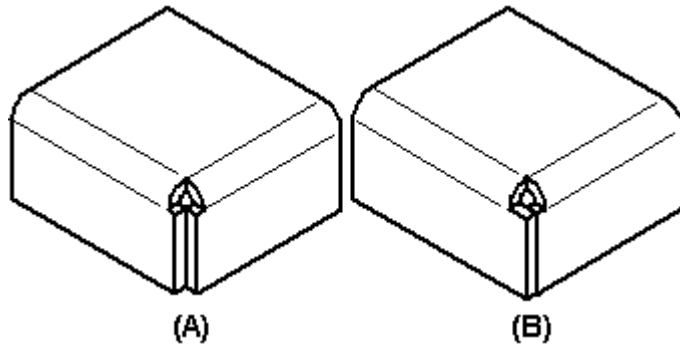
The [Close 2-Bend Corner command](#) modifies two flanges in one operation to close a corner where two flanges meet.



In the ordered environment, the [Close 3-Bend Corner command](#) closes corners that contain three bends.



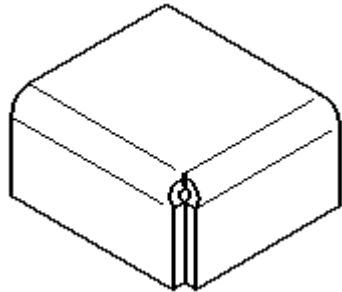
A closed corner is a treatment feature. You do not have to draw a profile, just select the edges you want to modify. With 2-bend corners, you can specify whether to close the corner (A), or overlap the corner (B).



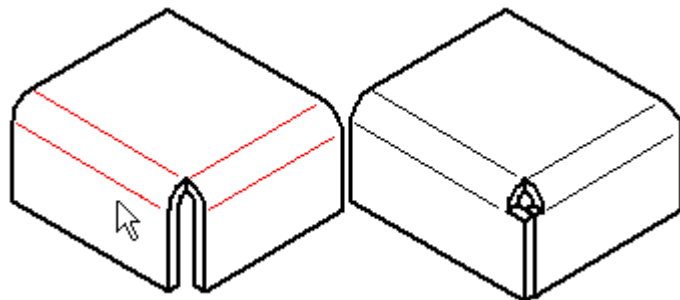
Note

The Overlap option is not available for 3-bend corners.

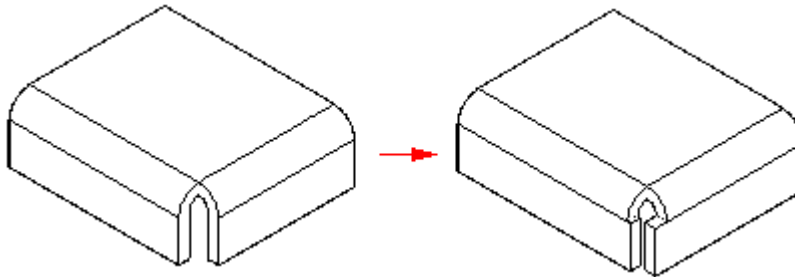
When you close the corner, you can also specify what type of bend treatment you want. For example, you can specify that you want a circular cutout applied to the bent faces.



When you overlap a corner, select the bend to be overlapped.



In the ordered environment, when you overlap a corner, you can use the Overlap Ratio option to compute the overlap as a percentage of the global material thickness.

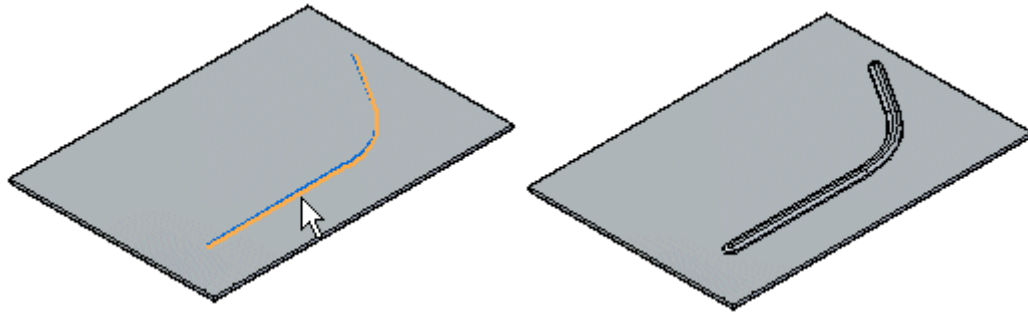


Note

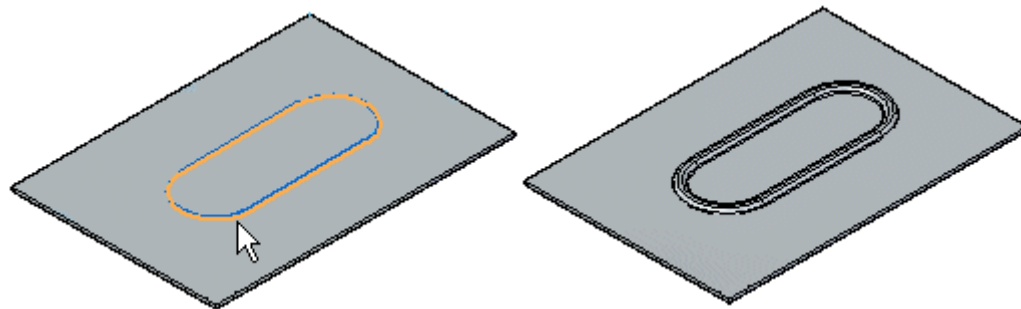
It is best to apply bend and corner relief before using the Close 2-Bends Corner command, so there is a clean corner to close. The corner should be symmetric, with equal bend radii and bend angles on the adjacent flanges. If there is more than one way to close the corner, edit the flanges themselves to close the corner the way you want.

Constructing beads

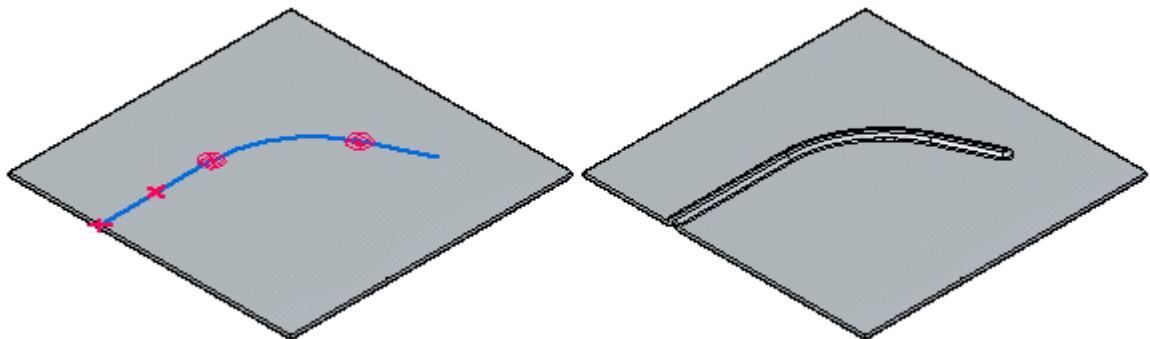
You can construct a bead with an open sketch element,



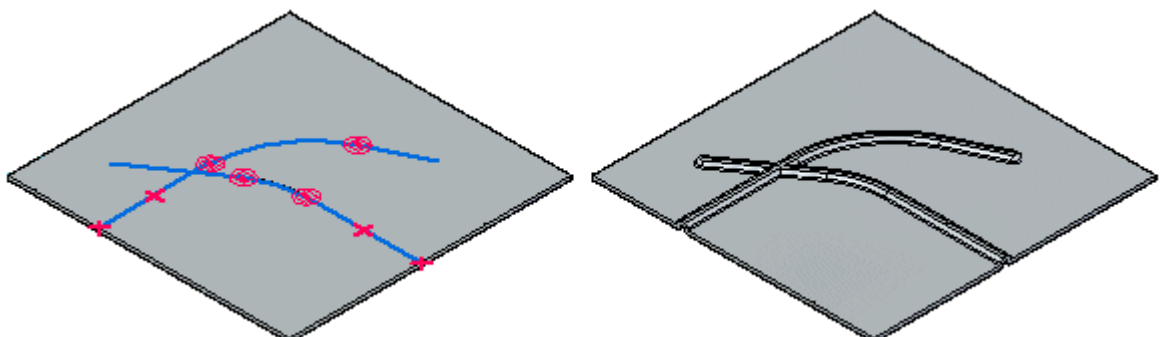
or closed sketch region.



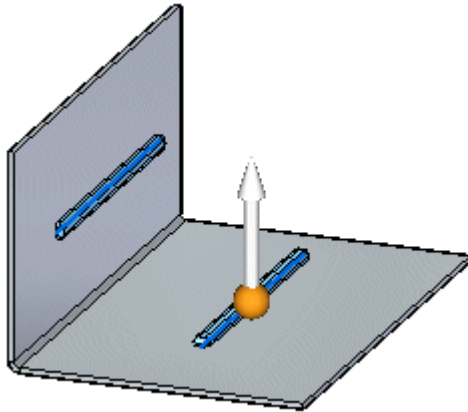
When constructing a bead profile using multiple elements, the element must be a continuous set of tangent elements.



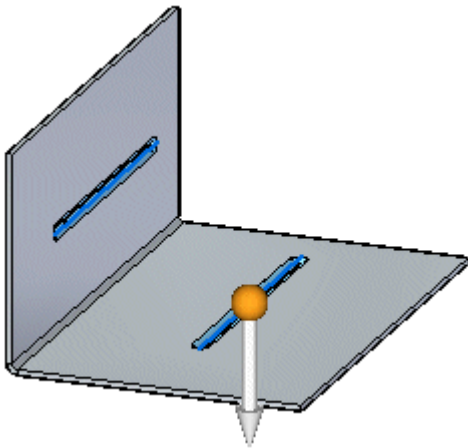
You can also construct a bead feature using multiple, separate sketch elements. Each element must be a continuous set of tangent elements, but the profiles can cross each other.



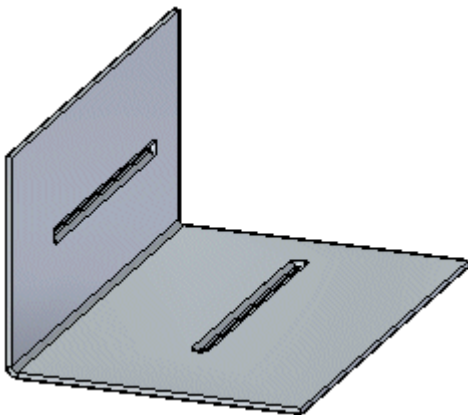
You can select more than one sketch element to construct multiple beads in a single operation.



You can use the direction arrow to change the direction of the beads.



All disjoint beads created in a single operation offset to the same side.

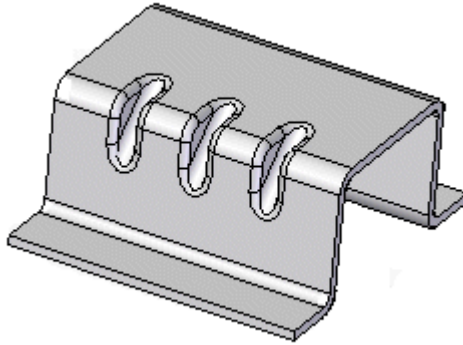


When constructing multiple disjoint beads, an entry in PathFinder, a feature profile, and a nail is created for each disjoint bead. Beads cannot be flattened and they cannot cross a bend.

You can specify the shape of the bead cross section and the type of end condition treatment you want using the Bead Options dialog box. For example, you can specify whether the bead shape is circular, U-shaped, or V-shaped. You can also specify whether the ends of the bead are formed, lanced, or punched.

Constructing gussets

You can use the [Gusset command](#) to add support across a bend. In the synchronous environment, you can construct gussets automatically across a bend. In the ordered environment, you can either construct gussets automatically or you can construct them from a drawn profile.



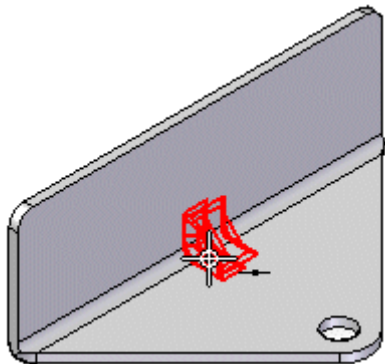
Note

Gussets are not displayed in the flat pattern representation or in drawing views of the flat pattern in the Draft environment.

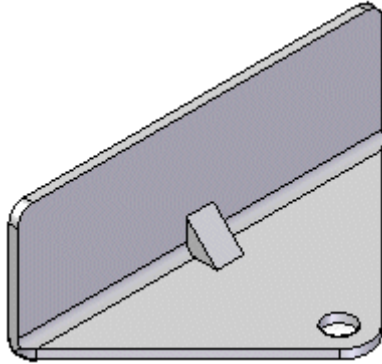
You can use the Gusset Options dialog box to specify the definition of the gusset. You can control such things as the shape of the gusset, the width and taper angle of the gusset, and the punch and die radius if the gusset is rounded. You can also use the dialog box to specify if the gusset is created automatically or from a user-drawn profile.

Constructing gussets automatically in the ordered environment

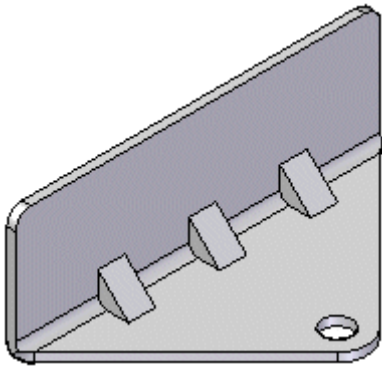
Select the Automatic Profile option on the Gusset Options dialog box to automatically create a gusset. Once you select a bend, the gusset profile is automatically displayed along the bend.



You can then click a keypoint to place the gusset,



or, use the Pattern Type option to specify whether you want to place one gusset or a pattern of gussets. For example, you can use the Fit option to place three gussets that are equally spaced along the selected edge.

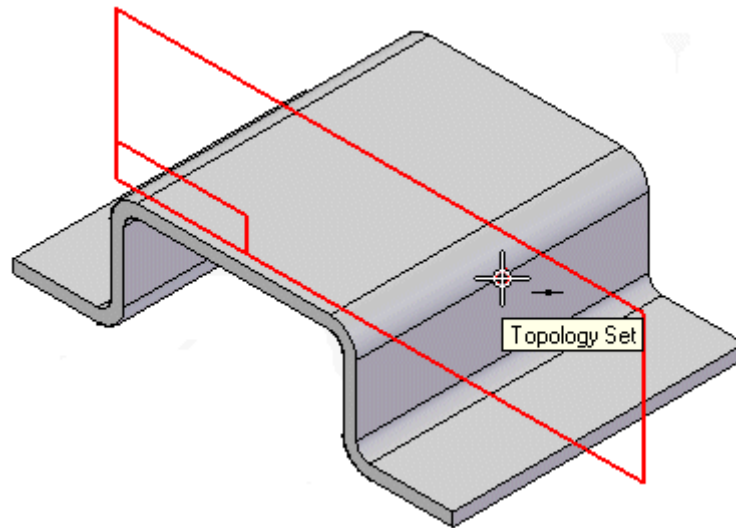


Constructing gussets from a user-drawn profile in the ordered environment

Select the User—Drawn Profile option on the Gusset Options dialog box to use a drawn profile to create a gusset. The profile can be an existing sketch or you can draw the profile while in the Draw Profile step.

To create a gusset from a user-drawn profile:

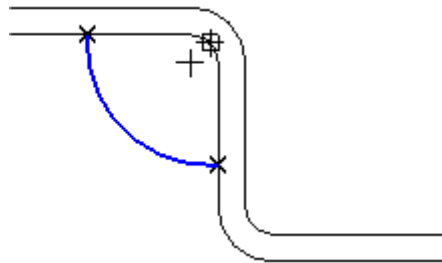
1. Click a keypoint to create a plane on which you want to draw the profile.



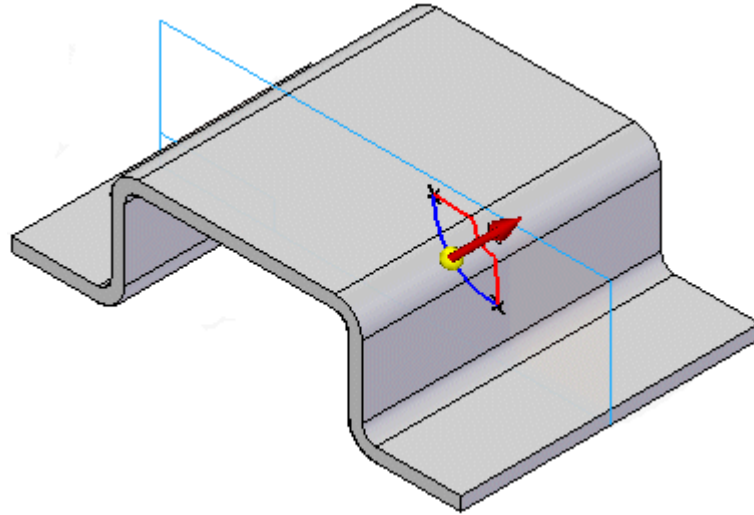
Note

You may select an existing sketch to define the gusset profile and skip to step 3.

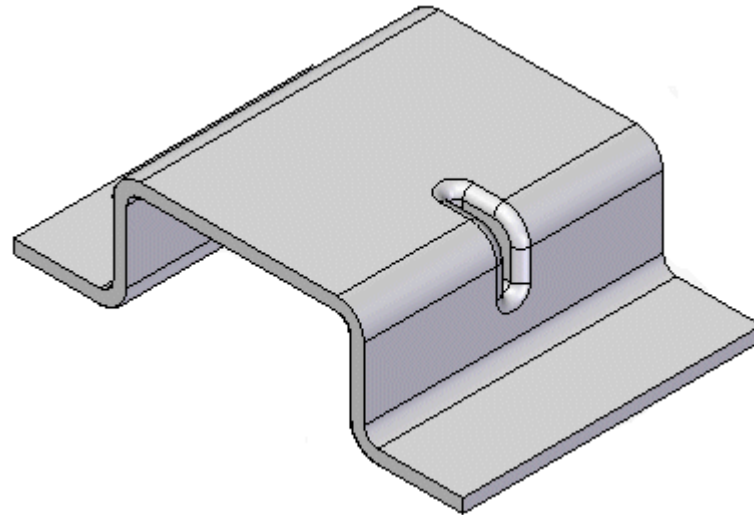
2. Draw the profile.



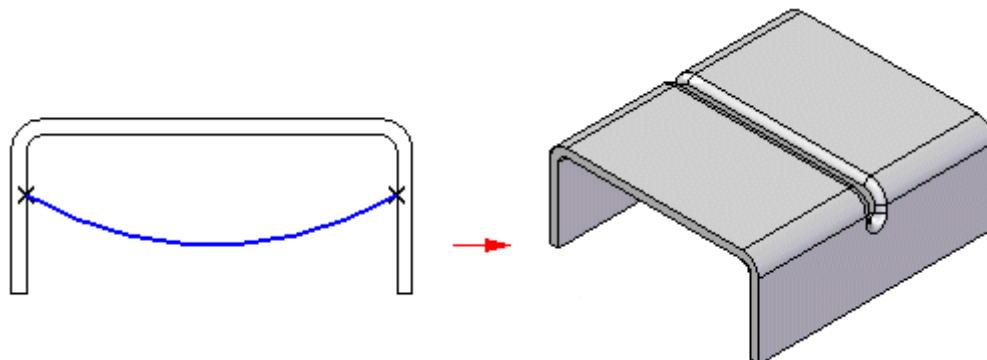
3. Click to define the direction of the gusset.



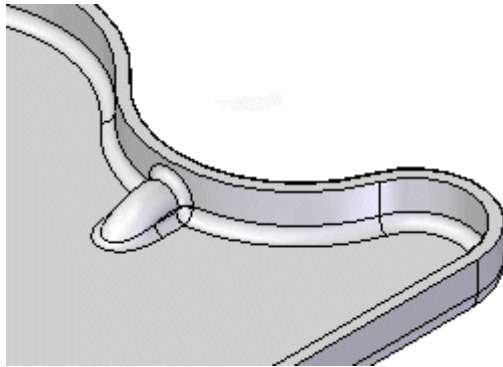
4. Click Finish to place the gusset.



When using the User-Drawn Profile option, you can also you can draw a profile that constructs a gusset across two bends,

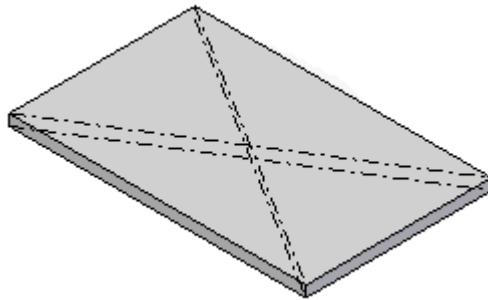


or, across a non-linear bend.



Constructing cross brakes

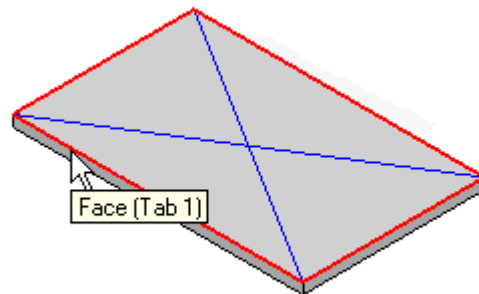
In the ordered environment, you can use the Cross Brake command to stiffen a sheet metal panel. The command creates a set of bends from a sketch that is coincident to the sheet metal part face.



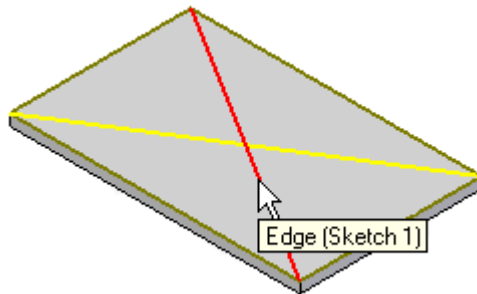
A cross brake feature does not deform the 3D model. It adds attributes containing information about the bends. This attribute information is used when creating a flat pattern or drawing of the sheet metal part.

To create a cross brake feature:

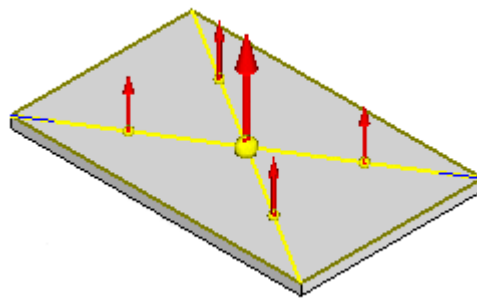
1. Select the face on which you want to construct the cross brake.



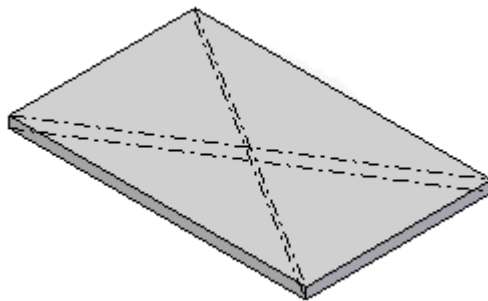
2. Select the sketch(es) you want to use to construct the cross brake.



3. Specify the bend angle and direction for the cross brake.

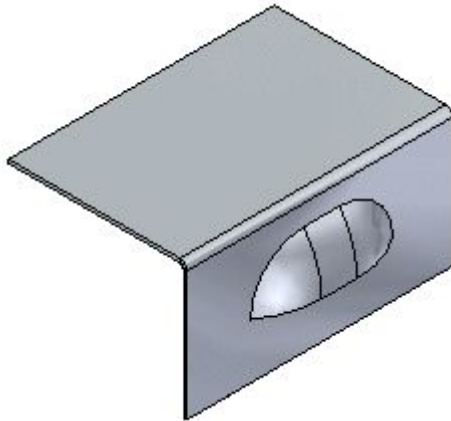


4. Click Finish to construct the cross brake.

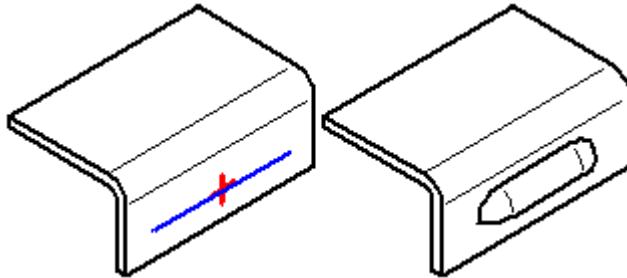


 **Louver command**

Constructs a louver with lanced or formed ends.

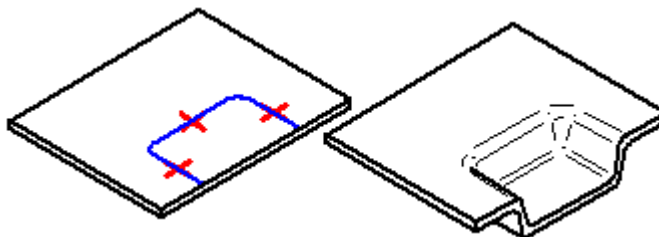


In the ordered environment, the profile for a louver feature must be a single linear element. Louvers cannot be flattened.



Dimple command

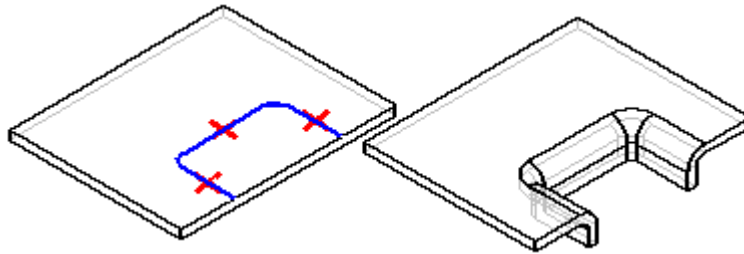
Constructs a sheet metal dimple from a selected region. If you use an open profile, the open ends of the profile must theoretically intersect part edges. Dimples are special die-formed features in which material deformation occurs. Dimples cannot be flattened.



Drawn Cutout command

Constructs a drawn cutout.

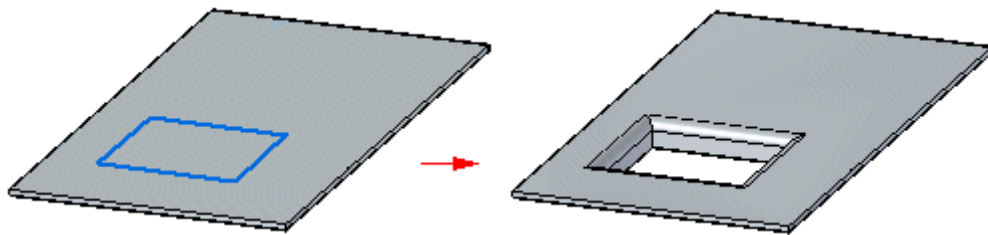
In the ordered environment, if you use an open profile, the open ends of the profile must theoretically intersect part edges. A closed profile cannot touch any part edges. Drawn cutouts cannot be flattened.



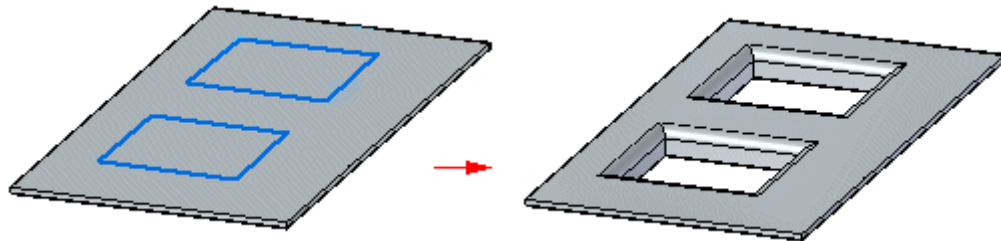
In the synchronous environment, the geometry used to create the cutout can be a closed internal profile that creates a region or an open profile extended to a part edge to create a closed region.

In the synchronous environment, regions that are valid for drawn cutouts are:

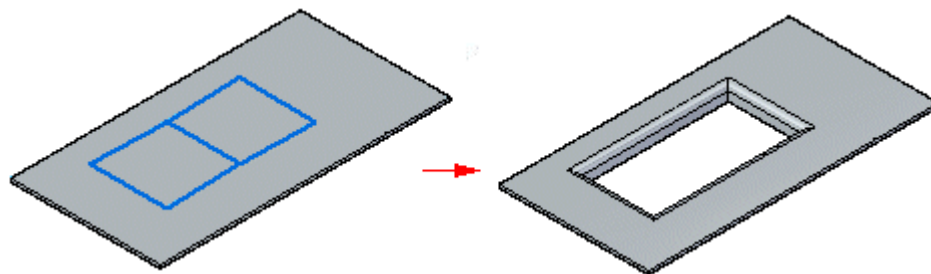
- Single



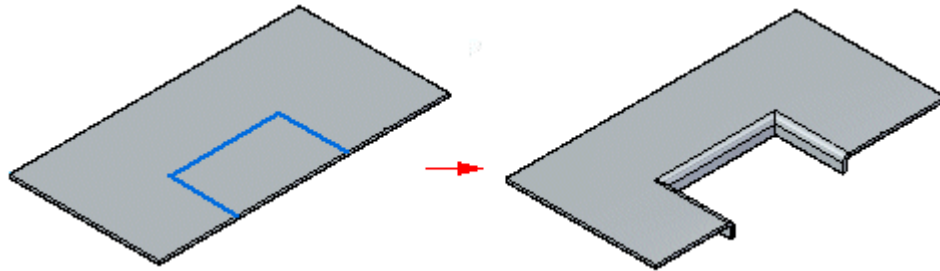
- Disjoint



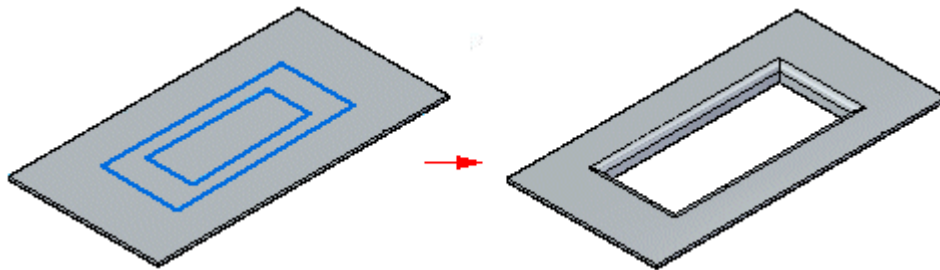
- Contiguous



- Coincident to a sketch or edge



- Nested contiguous

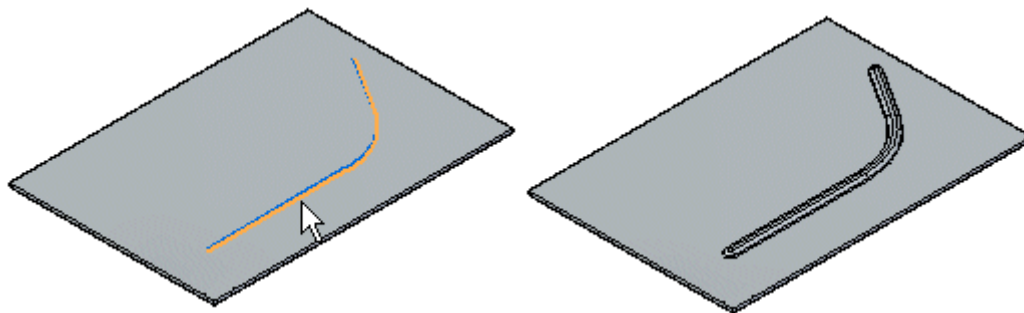


You can select multiple regions at one time and the regions must all lie on the same plane.

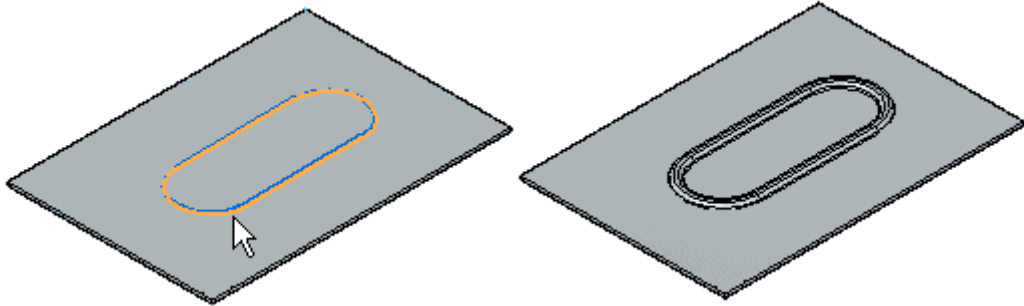
Bead command

Constructs a bead feature on a sheet metal part. A bead feature is often used to stiffen a sheet metal part.

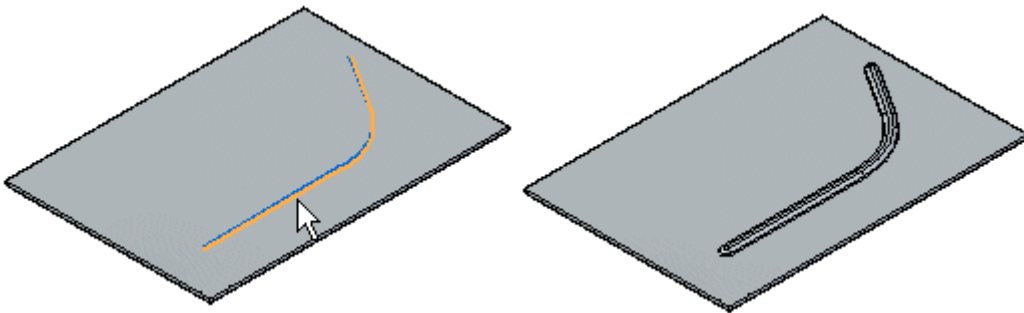
In the ordered environment, you can construct a bead with an open



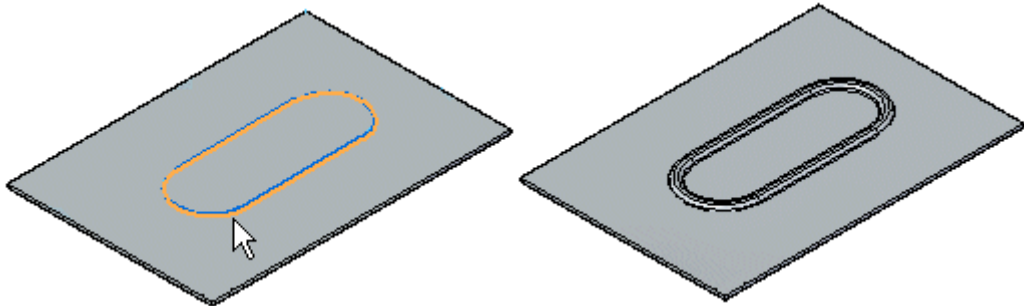
or closed profile.



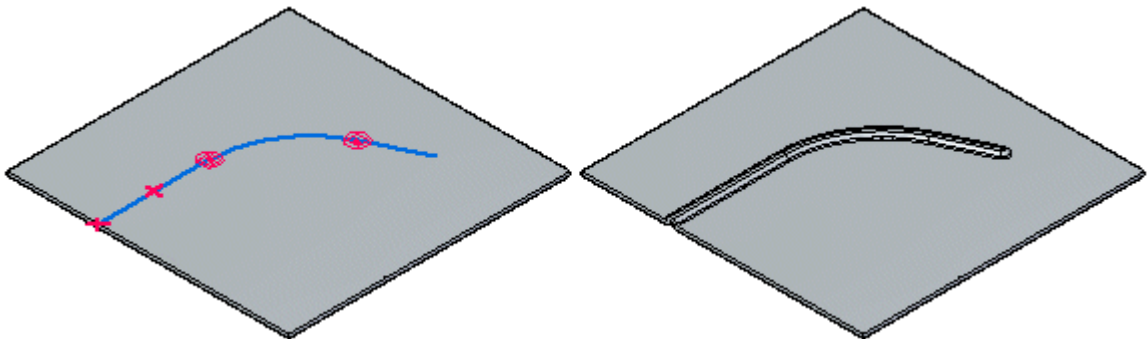
In the synchronous environment, you can construct a bead with an open sketch element,



or closed sketch region.

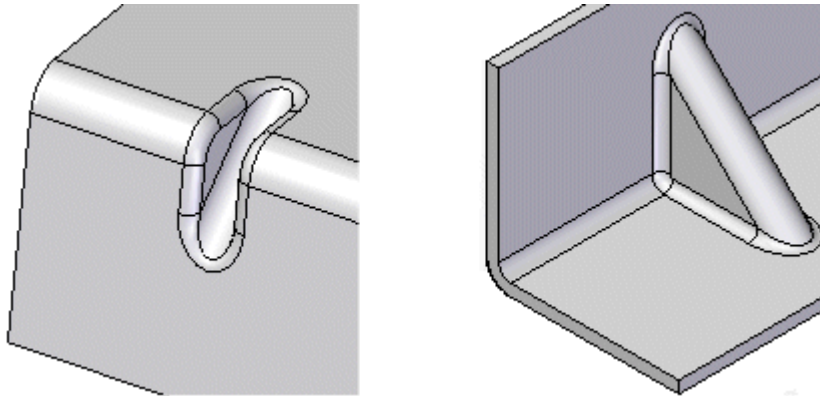


When constructing a bead profile using multiple elements, the element must be a continuous set of tangent elements.



Gusset command

Constructs a stiffening gusset across a bend to provide reinforcement in a sheet metal part.



You can create a gusset automatically or from a user-drawn profile. You can use the Gusset Options dialog box to specify the method to use when constructing the gusset. The steps required to construct the gusset are different depending on the method you use.

Note

Gussets are not displayed in the flat pattern representation or in drawing views of the flat pattern in the Draft environment. If the resident bend is removed, the gusset is also removed.

For more information on constructing gussets, see [Adding sheet metal deformation features](#).

Working with feature origins

You can use the feature origin handle to move or rotate manufactured features that contain a feature origin. The feature origin provides a reference point that can be used to move a feature without changing its shape.

The feature origin is used primarily in sheet metal models (.psm) for features such as dimples, drawn cutouts, and louvers.

Note

A feature origin is also used for hole features in part and sheet metal documents. The feature origin for a hole feature does not have XY fins.

You can dimension to a feature origin, and then edit the dimensional value to move the entire feature.

Note

When using Smart Dimension to dimension to a feature origin you cannot select the feature origin first.

Show and Hide commands are available to display and hide the feature origin when you select a feature that contains a feature origin. You can also display and hide all the features origins in a document.

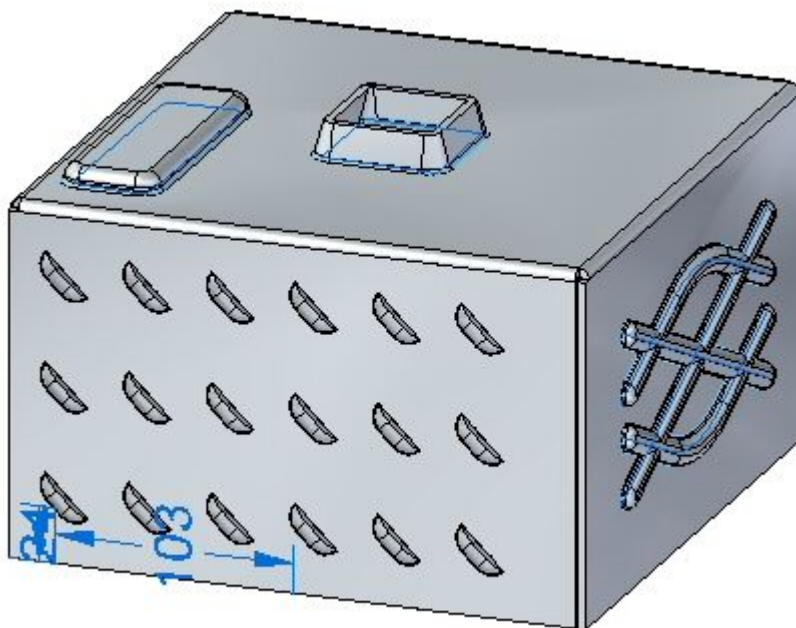
Shortcut menu commands are available to reposition the feature origin for a feature.

Activity: Working with deformation features in sheet metal.

Activity objectives

This activity demonstrates how to place and manipulate and edit deformation features and feature origins within a sheet metal part. In this activity you will:

- Place deformation features such as louvers, beads, dimples, drawn cutouts, beads and gussets.
- Pattern a deformation feature.
- Show, hide and move the feature origin of a deformation feature.
- Edit the values of a deformation feature.



Turn to Appendix H for the activity: Working with deformation features in sheet metal.

Lesson review

Answer the following questions:

1. What is the definition of a deformation feature?

2. What is the difference between a drawn cutout and a dimple?

Answers

1. What is the definition of a deformation feature?

A deformation feature is a feature that is created by stamping, punching or some similar process. These features cannot be flattened, but can be located to control the machine used to create the feature.

2. What is the difference between a drawn cutout and a dimple?

Both commands behave the same with the exception of the drawn cutout having removal of material at the extent of the profile. A dimple does not cut the material.

Lesson summary

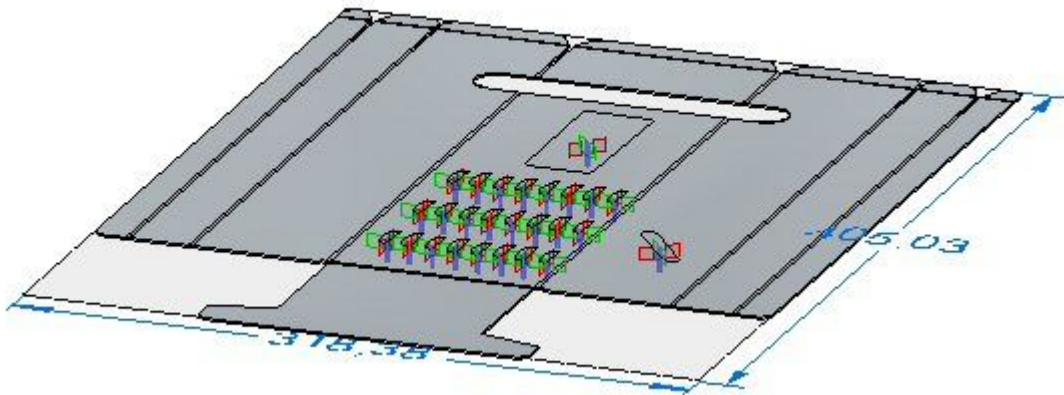
In this lesson you created a variety of deformation features. The feature origin for a rigid procedural feature was displayed and moved to a new location. The feature was rotated. Multiple occurrences were created with the pattern command.

Lesson

11 *Creating flat patterns*

Manipulating sheet metal geometry

After constructing a sheet metal part, you may need to create a flat pattern of the part for manufacturing.



Flattening sheet metal parts

After constructing a sheet metal part, you can use the Flatten and Save As Flat commands to create a flat pattern of a sheet metal part.

Using the Flatten command

Use the Tools tab @ Flat group @ [Flatten command](#) in the Sheet Metal environment to create a flat pattern in the same file as the formed sheet metal part.

When you flatten a sheet metal part with the Flatten command, a Flat Pattern feature is added to the PathFinder tab.

If the sheet metal model changes, the flat pattern becomes outdated. This is indicated by a symbol adjacent to the Flat Pattern feature in PathFinder. To update the flat pattern, select the Flat Pattern feature in PathFinder, then on the shortcut menu click Update.

Using the Save As Flat command

The [Save As Flat command](#) flattens a sheet metal part and saves the part as one of the following document types:

Part document (.par)

Sheet Metal document (.psm)

AutoCAD document (.dxf)

Note

When you use the Save As Flat command, the flattened document is not associative to the folded document.

You can create the flat pattern definition based on:

- An existing flat pattern
- The folded model state

Select the Use Existing Flat Pattern (Use Folded Model if Not Defined) option on the Flat Pattern Treatments page of the Solid Edge Options dialog box to create the flat pattern based on an existing flat pattern. Any material you add or remove in the flat pattern environment is included when the flat is saved. If no flat pattern exists, the folded model is used to define the pattern.

Select the Use Folded Model option on the Flat Pattern Treatments page of the Solid Edge Options dialog box to create the flat pattern definition based on the folded model state, even if a flat pattern already exists. Any material you add or remove in the flat pattern environment is excluded when the flat is saved.

Minimum bend radius

To facilitate creation of flat patterns, Solid Edge always creates a minimum bend radius for flanges, contour flanges, and lofted flanges, even if you specify a bend radius value of zero (0.00). For metric documents, a zero bend radius is set to a value of approximately 0.002 millimeters. For English documents, a zero bend radius is set to a value of approximately 0.0000788 inches. If you need the bend radius to be exactly zero, you have to create the features in the Part environment.

Cleaning up flat patterns

When flattening sheet metal parts, the system adds bend relief to the flat pattern. This system-generated bend relief can cause problems to downstream manufacturing processes such as punching and nesting. While working in the Sheet Metal environment, you can set options on the [Flat Pattern Treatments page](#) of the Options dialog box to automatically clean up the flat pattern.

The options on the Flat Pattern Treatments tab control corner treatments, simplify B-splines in the model to arcs and lines, and remove the system-generated bend relief.

If you change the options on this tab after a flat pattern is generated it recomputes the flat pattern, or updates the flat pattern.

Managing flat pattern size

Use the Flat Pattern Options dialog box to set the maximum flat pattern size and issue a warning if that size is violated. This is useful if a part cannot be manufactured because of sheet size limitations.

The Current section of the dialog box displays the length and width of the current flat pattern. These values are read-only and cannot be changed manually. They are updated when the values change in the flat model and the model is updated. The Alarm section allows you to specify the maximum length and width values for the flat pattern. You can specify a maximum length, maximum width, or both. You can either key in these values or use default values that are specified on the [Flat Pattern Treatments page](#) on the Options dialog box. If the flat pattern violates these size limitations, an alarm icon is displayed adjacent to the flat pattern entry in PathFinder. If you pause the cursor over the flat pattern entry, a tool tip displays the current flat pattern size along with the maximum size limitations.

You can use the Show Cut Size Range and Dimensions option to display a range box for the flat pattern along with the dimensions for the current length and width of the flat pattern. The size of the pattern is determined when the flat pattern is created and is recalculated when the flat model is updated.

Flattening deformation features

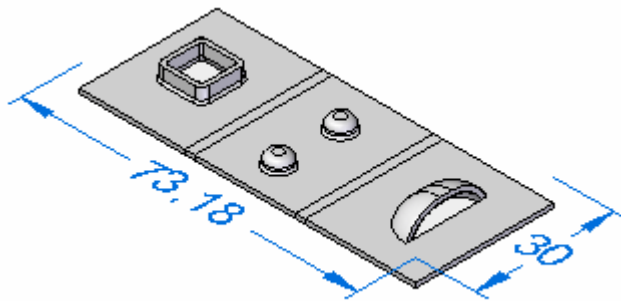
If you want to remove a deformation feature after you flatten a part using the Flatten and Part Copy commands, you can construct a cutout feature that is sized according to the area the deformation feature occupied. In many cases, you can use the Include command to create a cutout profile that is associatively linked to the edges of the deformation feature. Later, if the deformation feature changes, the cutouts also update. This approach maintains the true position for the deformation feature, which can be useful for creating downstream manufacturing documentation.

Alternatively, you can use the commands on the command bar to remove the deformation features prior to or after you flatten the part. For example, you can use the Delete Faces command to delete a deformation feature. The deformation feature is not physically deleted from the part, it is still available when working in the Sheet Metal environment. With this approach, the location of the deformation feature is lost in the flattened version of the part.

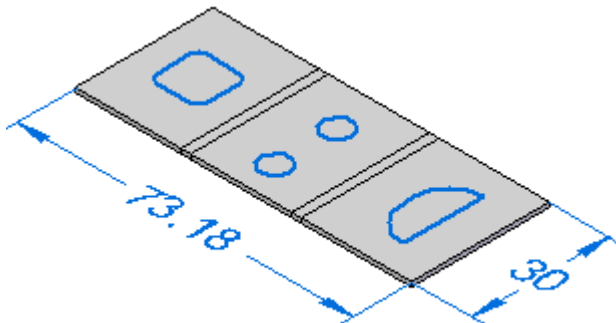
Displaying deformation features in the flat pattern

You can use the options in the Formed Feature Display section of the [Flat Pattern Treatments](#) page to specify how deformation features are displayed in the flat pattern.

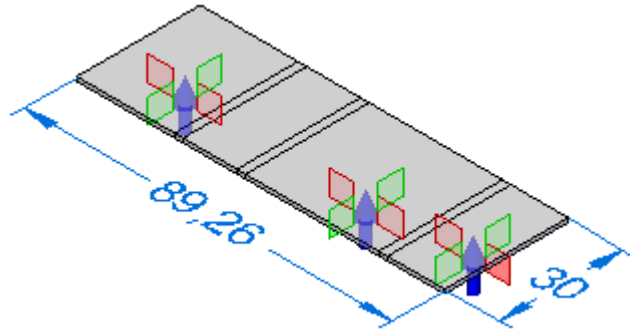
You can display the deformation features as a formed feature,



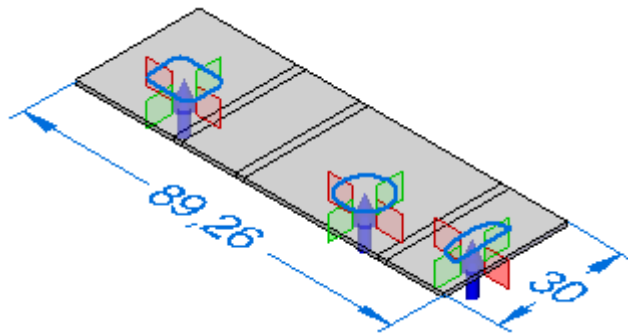
as a feature loop,



as a feature origin,



or as a feature loop and feature origin.



Saving deformation features to other files

The Formed Feature Display section of the [Flat Pattern Treatments](#) page specifies how deformation features are exported when you use the Save as Flat to flatten the sheet metal model and save it to another document.

When saving the document to .prn or .psm format:

- As Formed Feature replaces the deformed feature with a cutout the size of the area consumed by the feature.
- As Feature Loops replaces the loops representing the deformation feature with a curve.
- As Feature Origin does not export the deformation feature or the feature origin.
- As Feature Loops and Feature Origin replaces the loops representing the deformation feature with a curve and feature origins are not exported.

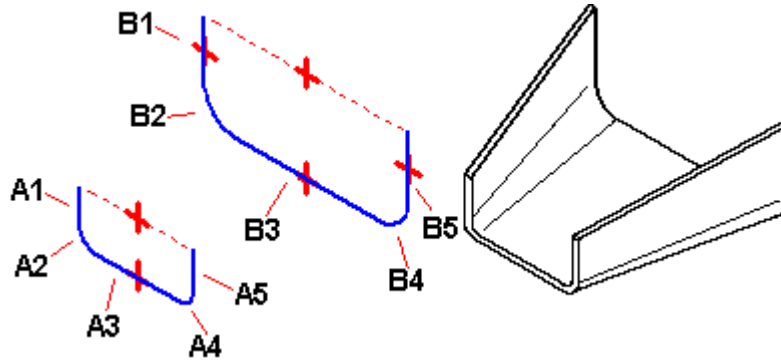
When saving the document to .dxf format:

- As Formed Feature replaces the deformed feature with a 2D wireframe representation as they would appear in the formed condition.
- As Feature Loops replaces the loops representing the deformation feature with a curve and are specified as either an up or down feature.
- As Feature Origin does not export the deformation feature or the feature origin.
- As Feature Loops and Feature Origin replaces the loops representing the deformation feature with a curve and are specified as either an up or down feature. Feature origins are not exported.

Flattening lofted flanges

Only lofted flanges that consist of planes, partial cylinders, and partial cones can be flattened. Lofted flanges that contain ruled surfaces cannot be flattened. The type of geometry constructed depends upon how you draw the profiles.

A lofted flange is constructed by mapping the faces between corresponding profile elements. For example, profile lines A1 and B1 are mapped to construct planar faces. Profile arcs A2 and B2 are mapped to construct conical faces.



If the two profiles have the same number and type of elements, and each element on the first profile maps to the same element type on the second profile (line to line, or arc to arc), in most cases, you can flatten it.

Ruled surface examples

Any lofted flange that contains a ruled surface cannot be flattened. The following examples describe when a ruled surface is constructed:

- A face constructed where line A1 has a different angle relative to line B1.
- A face constructed where arc A2 has a different start angle or included angle relative to arc B2.
- A face constructed using an arc and a line.

Note

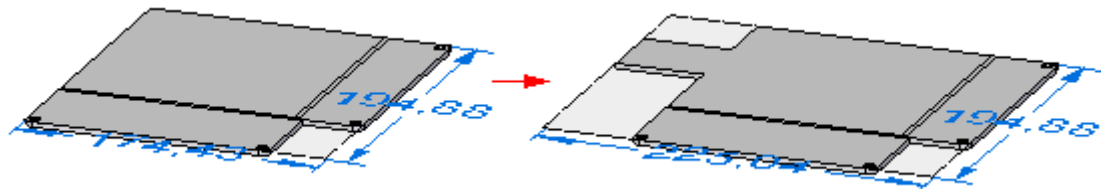
If the lofted flange contains faces that would prevent it from being flattened, a gray arrow is displayed adjacent to the feature on the PathFinder tab. If you pause your cursor over the feature in PathFinder, a message is displayed in the status bar describing the problem.

PMI dimensions in the flat pattern

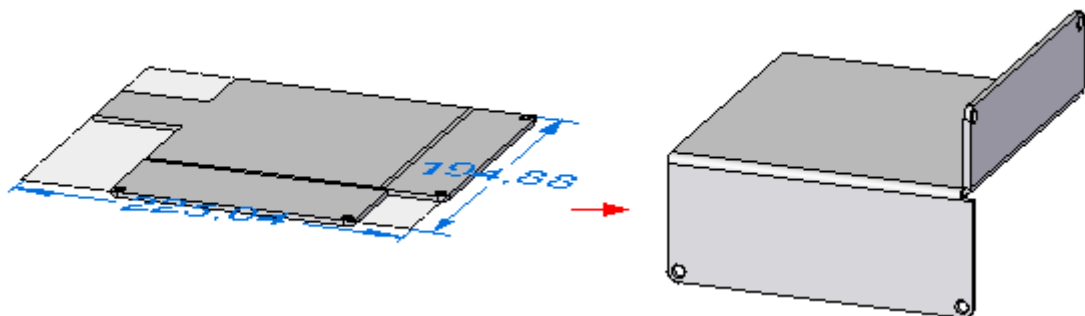
When a flat pattern is created, PMI dimensions are placed as driven dimensions. In other words, these dimensions are for reference only and cannot be changed while in the flat pattern. If you select a PMI dimension in the flat pattern, all fields on the dimension edit control are disabled. If you make changes to the model, the PMI dimensions are updated when the flat pattern updates.

Adding material to the flat pattern

You can use the [Tab command](#) to add material to a flat pattern.



Any tabs created in the flat pattern are placed in the flat pattern node of PathFinder. Any material added to the flat pattern appears only in the flat pattern state. The folded model will not reflect the material addition.



Saving sheet metal files as AutoCAD documents (.dxf)

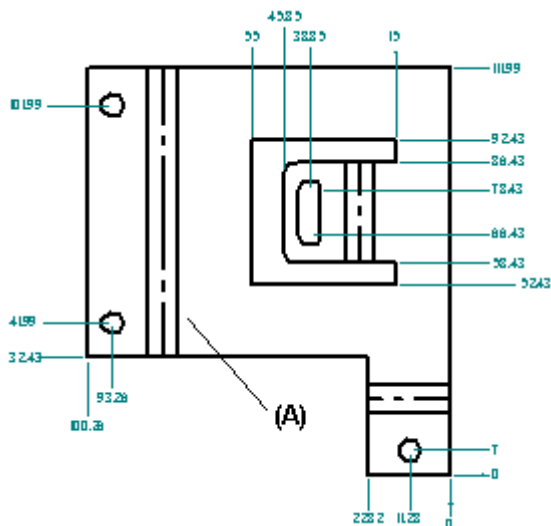
When you save a sheet metal part as an AutoCAD document (.dxf), it is saved as 2D. Collinear and concentric arcs are merged into single elements. Bend lines are added as wireframe bodies.

Layers are used to separate the various types of information such as bends, deformation features, and edges. A layer scheme defines the information that is stored on the different layers.

- Default or Normal edges are saved to layers named Outer_Loop and Interior_Loops. All edges from flanges, contour flanges, lofted flanges, tabs, cutouts, and sheet metal cutouts are placed on these layers. The layers can contain visible and hidden edges.
- Bend down centerlines are saved to a layer named DownCenterlines. This layer contains the bend centerlines of all linear and conical bends that are in the down direction relative to the selected output face. These lines are generated in the flatten process and do not exist in the model. The line style associated with bend down centerlines can be saved to this layer.
- Bend up centerlines are saved to a layer named UpCenterlines. This layer contains the bend centerlines of all linear and conical bends that are in the up direction relative to the selected output face. These lines are generated in the flatten process and do not exist in the model. The line style associated with bend up centerlines can be saved to this layer.
- Deformation features found on down bends are output to a layer named DownFeatures. This layer contains the edges of all deformation features that are in the down direction relative to the selected output face. This layer can contain visible and hidden edges.
- Deformation features found on the up bends are output to a layer named UpFeatures. This layer contains the edges of all deformation features that are in the up direction relative to the selected output face. This layer can contain visible and hidden edges.

Creating flat pattern drawings

You can create drawings of flattened sheet metal parts in the Draft environment. A special template can be applied when a flat pattern drawing is created. This template has tangent edges displayed so that the lines that represent the edges of the bends (A) are shown in the drawing.

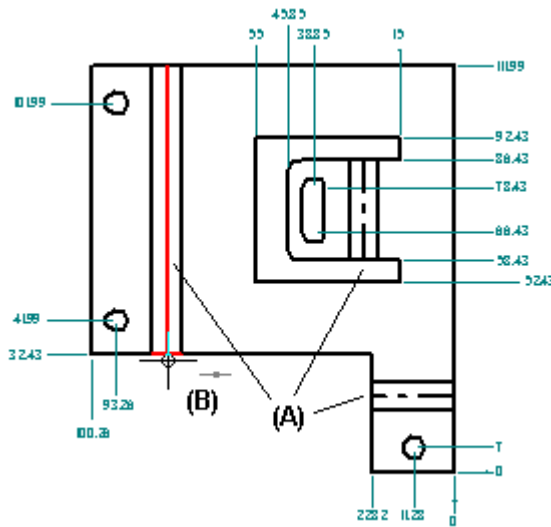


To apply the template, you must run the Drawing View Wizard and set the Part and Sheet Metal Drawing View Options on the first page of the Drawing View Creation Wizard to *Flat Pattern*.

You also can display tangent edges in a drawing that was created with a different template. Use the Edge Display tab on the Solid Edge Options dialog box.

The flat pattern created with the **Flatten command** contains all bend centerline information used to create bend centerlines in drawing views.

In draft, you also can add the centerline to a bend (A) using the By Two Lines option with the Center Line command.



Specifying bend options

In part, sheet metal, and draft, you can use the options on the Annotation page (Solid Edge Options dialog box) to:

- Customize bend direction strings for Up, Down, and Undefined bends.

- Create and assign independent styles to bend up centerlines and to bend down centerlines.
- Specify which part face is the *top* face in the flat pattern drawing view. By default, bend direction is derived from the face that is designated the *top* face when a sheet metal part is flattened. In draft, you can keep the model bend direction properly aligned with the flattened drawing view using the Derive Bend Direction from Drawing View option.

Updating flat pattern drawings

When you make design changes to a folded sheet metal part, you need to update the associatively flattened part first, and then update the flattened drawing to see the changes. When you open the flattened part document, an out of date symbol is displayed adjacent to the base feature in the PathFinder tab. To update the flattened part, select the Flat Pattern entry in PathFinder, then use the Update command on the shortcut menu.

When you open the drawing of the flattened part, a box is displayed around each drawing view to indicate that they are out of date. To update the drawing views, use the Update Views command.

Placing bend tables on drawings

Once you create a drawing of a flattened sheet metal part in the Draft environment, an associated Bend Table can be added to the drawing sheet. Use the Bend Table command in the Draft environment. To learn how to do this, see Save bend data with flat patterns.

Flat Pattern Treatments page (Solid Edge Options dialog box)

Controls flat pattern treatment settings for the active sheet metal document. Changing these options once a flat pattern is generated causes a recompute of the flat pattern feature, or an update to the associative flat pattern if it was placed with the Insert Copy command. These tab options are available in the Solid Edge Sheet Metal environment.

Outside Corner Treatments

Applies the treatment to outside corners. You can specify no corner treatment, a chamfer corner treatment, or a tangent arc corner treatment.

Outside Value

Specifies the value for the outside corner treatment. This option is only available if the Outside Corner Treatments option is set to Chamfer or Radius.

Inside Corner Treatments

Applies the treatment to inside corners. You can specify no corner treatment, a chamfer corner treatment, or a tangent arc corner treatment.

Inside Value

Specifies the value for the inside corner treatment. This option is only available if the Outside Corner Treatments option is set to Chamfer or Radius.

Simplify B-splines

Specifies that any B-spline curves on the formed part are simplified to lines and arcs. B-spline curves can be created when creating cutouts across bends and when using stencil font characters.

Minimum Arc

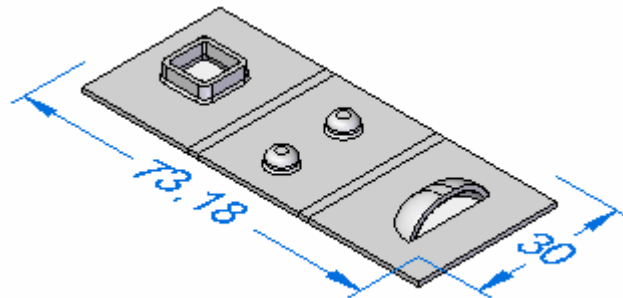
Specifies the minimum value for the arc created from the B-spline curve.

Deviational Tolerance

Specifies the deviation tolerance for the B-spline curve.

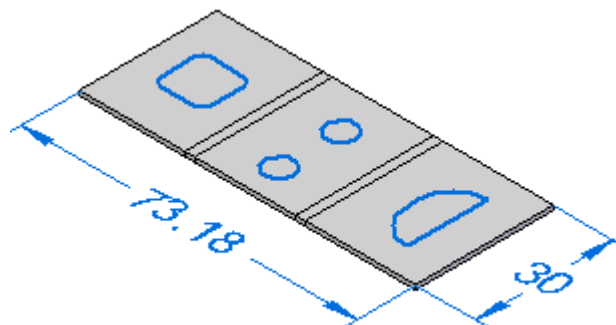
Formed Feature Display

As Formed Feature Displays the sheet metal features as formed features in the flat pattern.



As Feature Loops

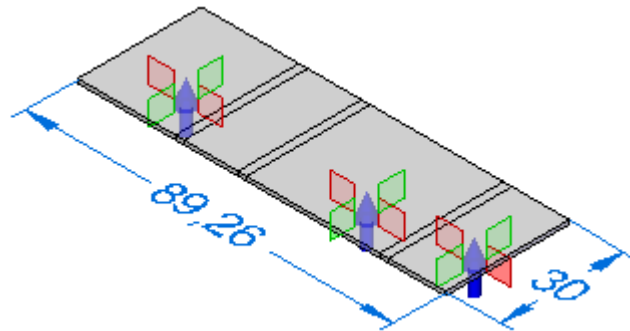
Displays the sheet metal features as feature loops in the flat pattern.



Note

The location of the feature loops is based on the punch side of the deformation feature.

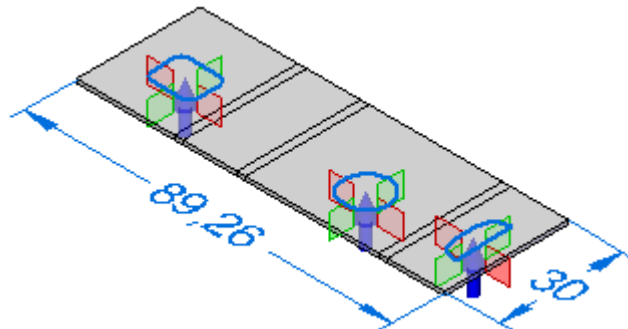
As Feature Origin Displays the sheet metal features as feature origins in the flat pattern.



Note

The location of the feature origins is based on the punch side of the deformation feature.

As Feature Loops and Feature Origin Displays the sheet metal features as feature loops with feature origins in the flat pattern.



Cut Size Default Values

Specifies the default maximum length (X) and maximum width (Y) for the flat pattern. These values appear in the Alarm section of the Flat Pattern Options dialog box.

Removed System-Generated Bend Reliefs

When you create a close corner with no relief, the system creates a very small bend relief in the 3D model. Specifies that you want to remove system-generated bend reliefs when the flat pattern is created.

Maintain Holes in the Flat Pattern


When a hole is placed on a non-orthogonal surface or angle, the resulting hole is elliptical when flattened. Creates the flat pattern with actual round holes.

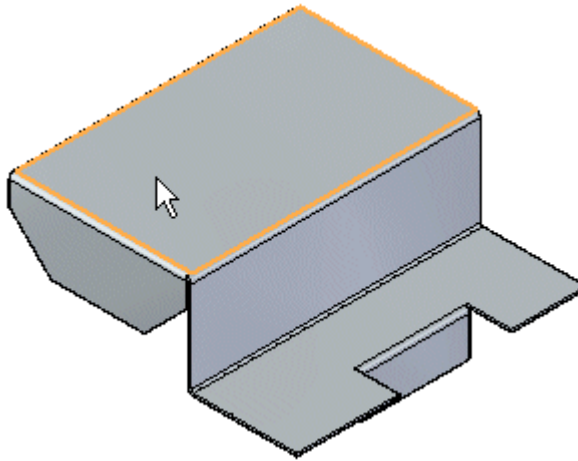
Save as Flat Command

Use Existing Flat Pattern (Use Folded Model if Not Defined) Creates the flat pattern definition based on an existing flat pattern. Any material added or removed in the flat pattern environment is included when the flat is saved. If no flat pattern exists, the folded model is used to define the pattern.

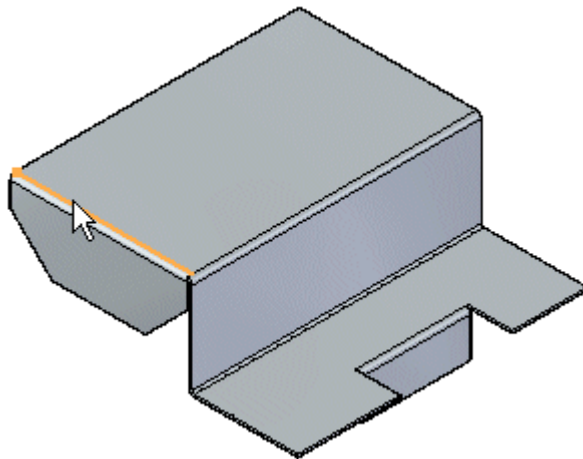
Use Folded Mold Creates the flat pattern definition based on the folded model state, even if a flat pattern already exists. Any material added or removed in the flat pattern environment is excluded when the flat is saved.

Construct a flat pattern in the sheet metal part document

1. Select Tools tab® Model ® Pattern.
2. Select Tools tab® Flat group® Flatten .
3. Click a face to be oriented upward in the flat.



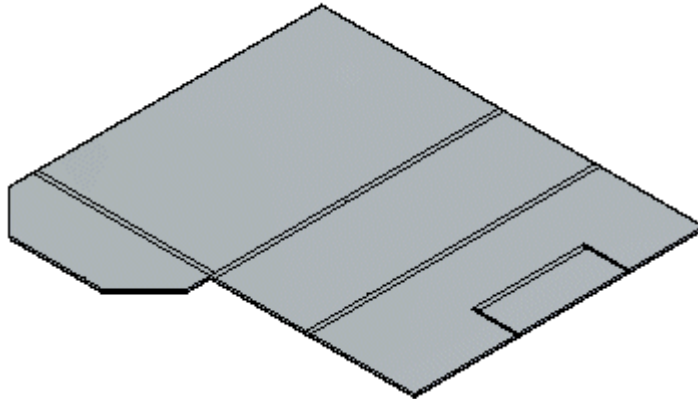
4. Click an edge to define the X axis and origin.



Note

The definition of the X axis is aligned or orientated with the Global X-axis of the sheet metal file.

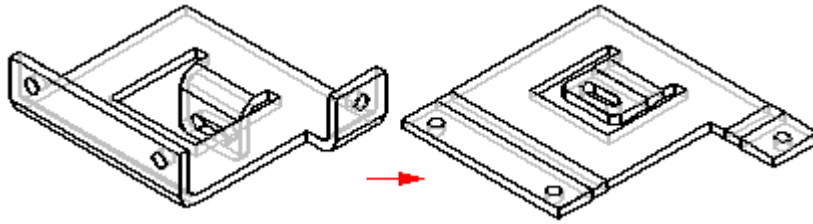
5. Click to complete the flat pattern.

**Tip**

- After flattening, you can use this command multiple times to adjust the orientation by selecting a new edge for alignment.
- When you create a flat pattern, a Flat Pattern tab is added to PathFinder. You can delete the flat pattern by deleting the Flat Pattern entry in the Flat Pattern tab of PathFinder.
- If the Sheet Metal model changes, the flat pattern will go out-of-date. This is indicated with a clock symbol overlapping the Flat Pattern tab in PathFinder. To update the flat pattern, click the Flat Pattern tab in PathFinder.
- You can add PMI dimensions to the flat pattern.
- The flat pattern created with the Flatten command contains all bend centerline information used to create bend centerlines in Draft drawing views. It also contains the same information used by the Save As Flat command.
- You can use the Flat Pattern Options dialog box to set a maximum flat pattern size. If the flat pattern violates the maximum size a warning icon is displayed adjacent to the flat pattern entry in PathFinder. This can help you determine whether or not the part can be manufactured due to sheet size limitations.
- You can use the Simplify B-Splines option on the [Flat Pattern Treatments tab](#) on the Options dialog box to specify that any b-spline curves in the part are simplified to lines and arcs when creating the flat pattern. B-spline curves can be created when creating cutouts across bends and when using stencil font characters.

**Flatten command**

Flattens a sheet metal part in the same document as the design model. The flattened version of the part is associative to the formed version of the part.



You can use the [Flat Pattern Treatments tab](#) on the Options dialog box to control output parameters for the flat pattern. For example, you can specify that any B-spline curves in the formed part are simplified to lines and arcs when creating the flat pattern. B-spline curves can be created when creating cutouts across bends and when using stencil font characters.

Note

When you use this command to construct the flat pattern, Solid Edge places a Flat Pattern entry in PathFinder.

Note

Note: When you use this command to construct the flat pattern, Solid Edge places a Flat Pattern tab and entry in PathFinder.

Save As Flat command

Flattens the sheet metal part and saves it to a document type you define. You can specify that the part is saved as a .par, .psm, or .dxf file.

You can create the flat pattern definition based on:

- An existing flat pattern
- The folded model state

You can use the options on the [Flat Pattern Treatments page](#) of the Solid Edge Options dialog box to specify how to define the flat pattern. You can also use options to specify how deformation features are displayed in the flat pattern.

Use Save As Flat when a flat pattern of a Solid Edge sheet metal file is needed in .dxf format. The resulting 3D planar geometry is merged wherever possible to provide a more efficient tool path for CNC programming. A drawing can be produced from the geometry generated by this process. However, it will neither be associative nor linked to the 3D sheet metal file from which it derives.

Note

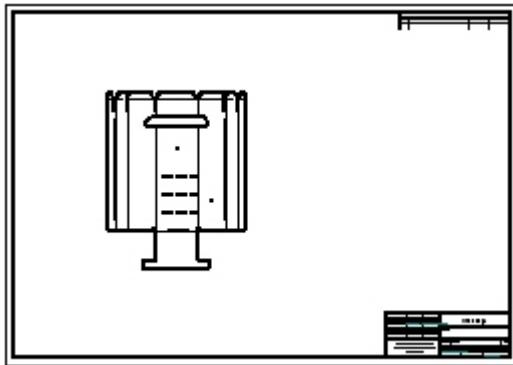
Bend lines are not automatically created with this command. If you want to automatically create bend lines, you must use the Insert Part Copy command to create a flat .psm file. You can then place a part view for the flat .psm file in a draft file.

Activity: Creating a flat pattern from a sheet metal part

Activity objectives

This activity demonstrates how to create a flat pattern from a sheet metal part, and the various options available. In this activity you will:

- Create a flat pattern from a sheet metal part.
- Control the orientation of the flat pattern.
- Understand the options available for using the flat pattern with downstream manufacturing applications.



Turn to Appendix I for the activity: Creating a flat pattern from a sheet metal part.

Lesson review

Answer the following questions:

1. Describe the steps needed to create a flat pattern.
2. When saving a flat pattern using the save as flat command, what file types are available?
3. What is a feature origin on a flat pattern?

Answers

1. Describe the steps needed to create a flat pattern.
A flat pattern is created using the following steps:
 - a. The flat pattern is initiated from Tools>Model and then clicking Flat Pattern.
 - b. The desired treatment options are set.
 - c. The face to be oriented upward in the flat is selected.
 - d. The horizontal axis and origin is defined.

2. When saving a flat pattern using the save as flat command, what file types are available?

Available file types for saving a flat pattern using the save as flat command are:

- AutoCAD (*.dxf)
- Sheet metal document (*.psm)
- Part document (*.par)

3. What is a feature origin on a flat pattern?


A feature origin is a defined point that represents the physical location that a tool will strike a sheet metal part to create a deformation feature, such as a louver, hole, drawn cutout and so forth.

Lesson summary


In this lesson you created a flat pattern and changed the display options. You also placed the flat pattern on a drawing sheet.

A Activity: Starting sheet metal design

Open a sheet metal file

- ▶ Start Solid Edge ST4.
- ▶ Click the  Application button ® **New** ® **ISO Sheet Metal**.
- ▶ Proceed to the next step.

Set the material properties

- ▶ To set the file properties, click the  Application button ® **Properties** ® **Material Table**.
- ▶ Click the Gage tab.
- ▶ Notice the default value of the neutral factor.

Note

This value will not be changed for this exercise. This step is to show where the neutral factor can be modified if the need exists.

- ▶ Set the Sheet Metal Gage to 8 gage.
- ▶ Click Apply to Model.

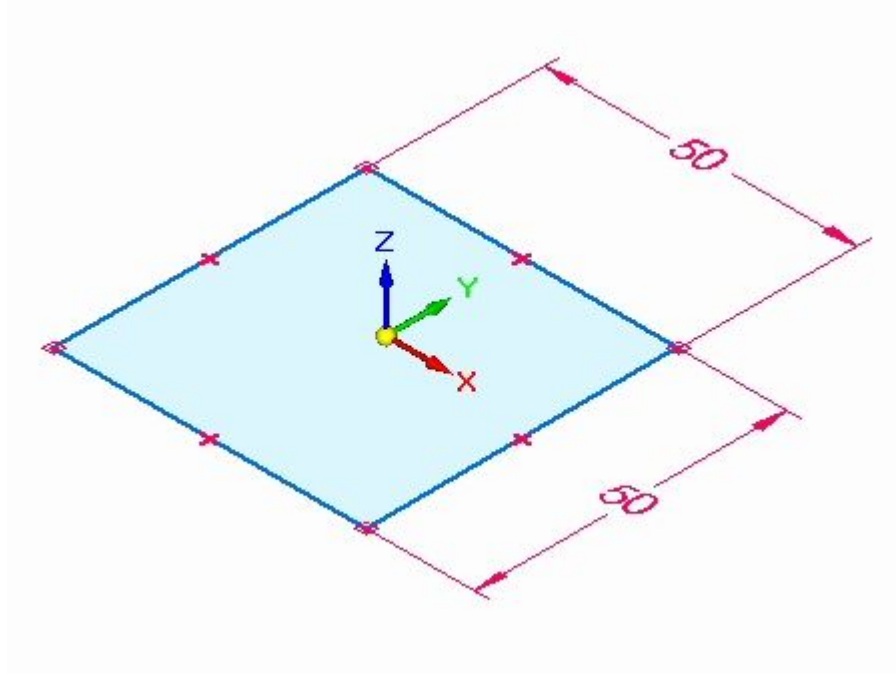
Note

Changes to material properties are applied to the current open document. If you need to customize these values and use the new values across your enterprise, then the values can be edited in the material library property file *material.mtl*, and that file put in a location accessible to all that need it.

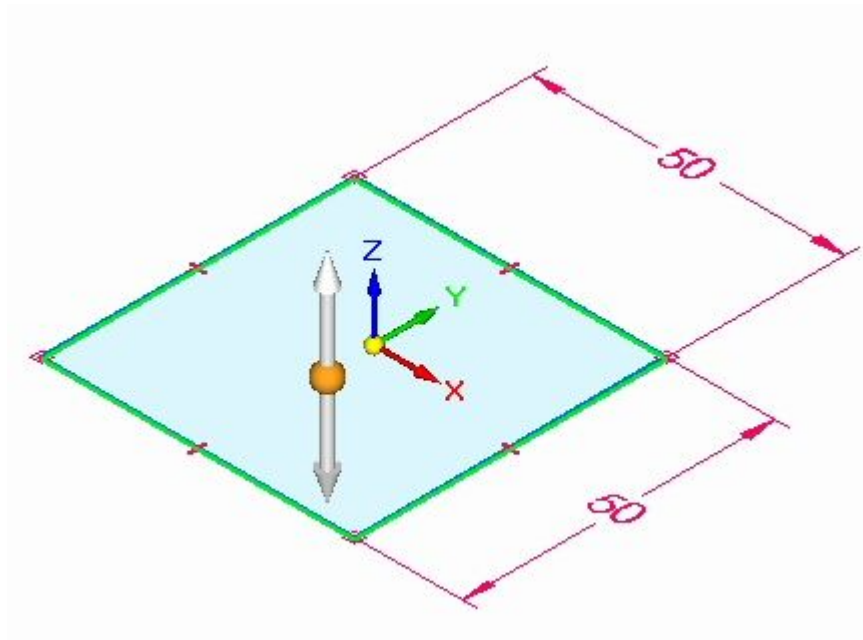
- ▶ Proceed to the next step.

Create a tab

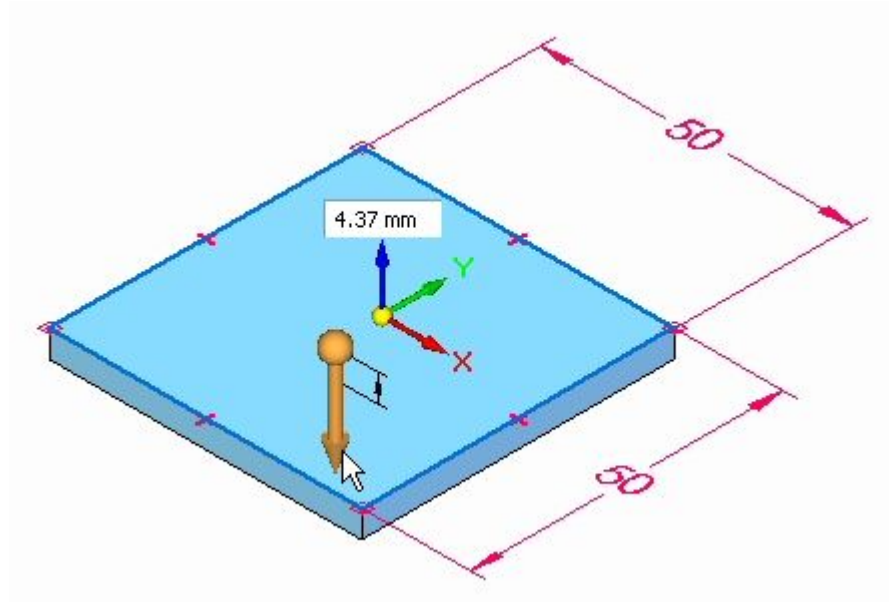
- Sketch a 50 mm square in the X-Y plane.



- Select the region shown.



- ▶ Create a tab by selecting the downward pointing vertical handle down. Click to accept.



Note

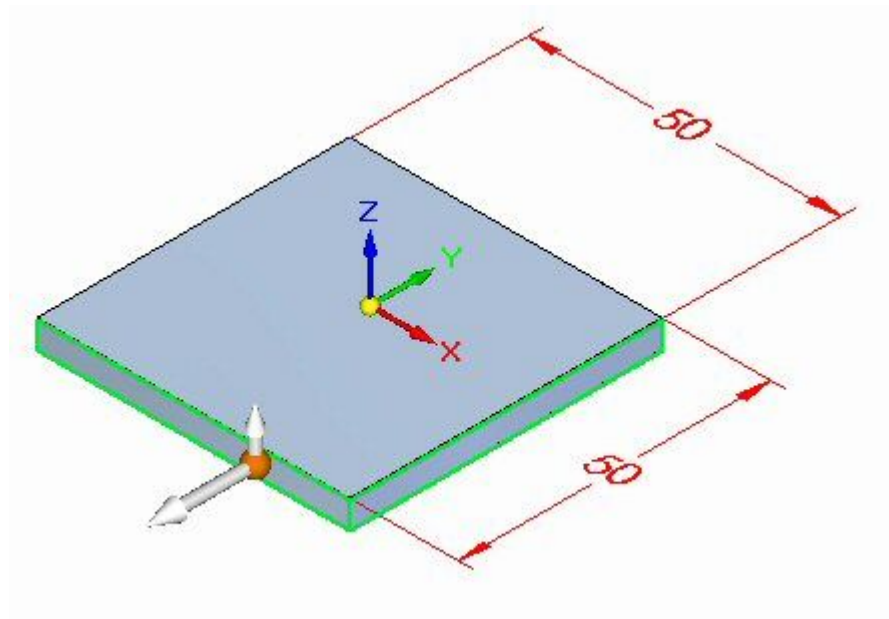
Notice the material thickness corresponds to the gage set in the previous step.

The tab is created.

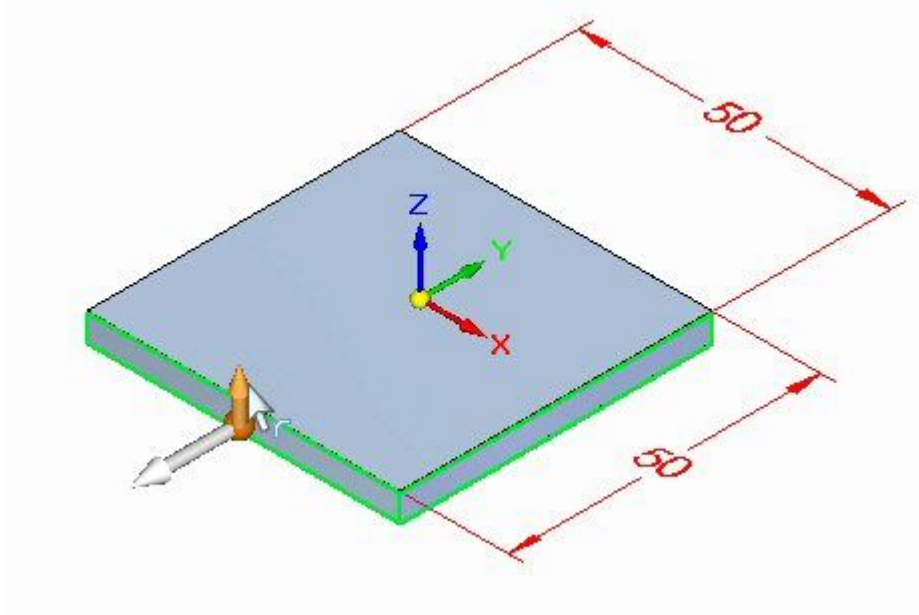
- ▶ Proceed to the next step.

Create flanges from the tab

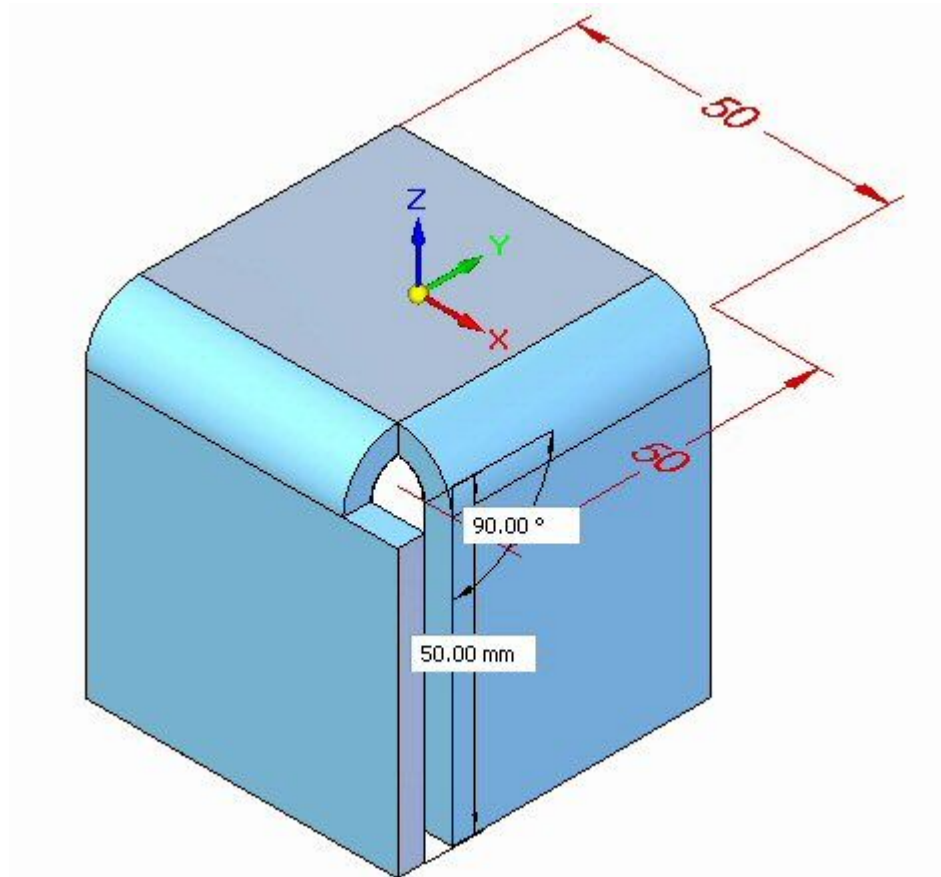
- ▶ Select the two thickness faces shown.



- Select the flange start handle



- Enter a distance of 50 mm for the new flanges.



Note

Two flanges were created from the thickness faces on the tab. Notice the new entries in PathFinder.



- Close the sheet metal document without saving. This completes the activity. Proceed to the activity summary.

Activity summary

In this activity you set the material thickness using the gage tab on the material table. A tab was placed and flanges were created from the thickness faces of the tab.

B Activity: Using regions to create tabs and cuts

Open a sheet metal file

- Start Solid Edge ST4.
- Click the  **Application** button ® **Open** ® *tab_cut_activity.psm*.

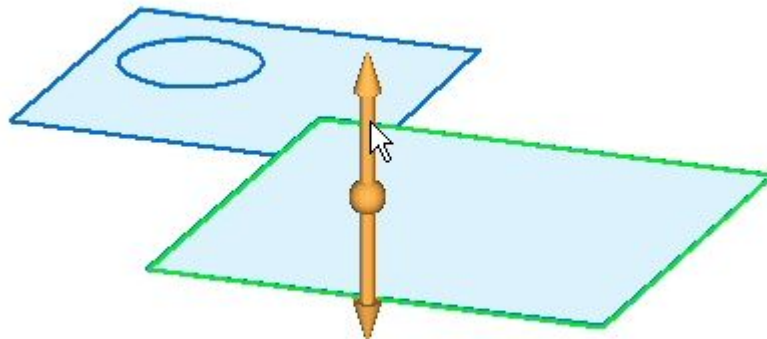
Note

In this activity, the material thickness has been set to 2.0 mm and the bend radius has been set to 1.0 mm.

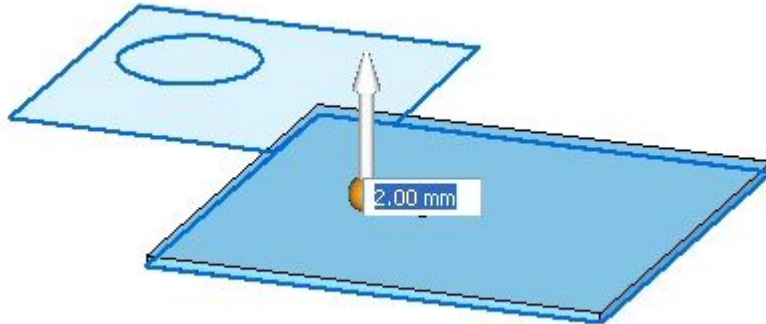
- Proceed to the next step.

Use the sketch to create the base feature

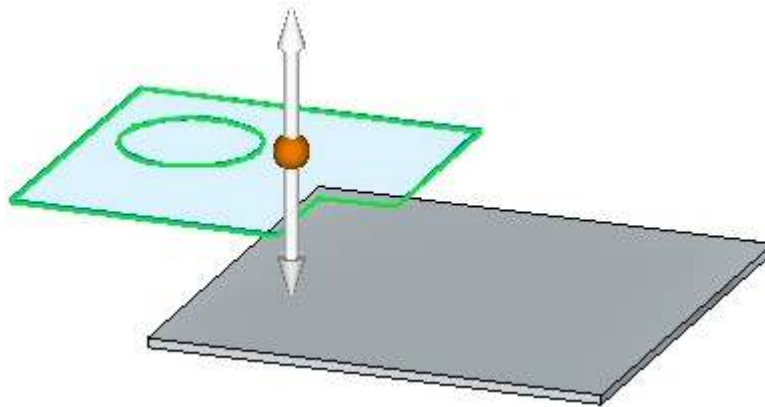
- Use the region shown to create the base feature from the sketch geometry. Select the handle pointing up.



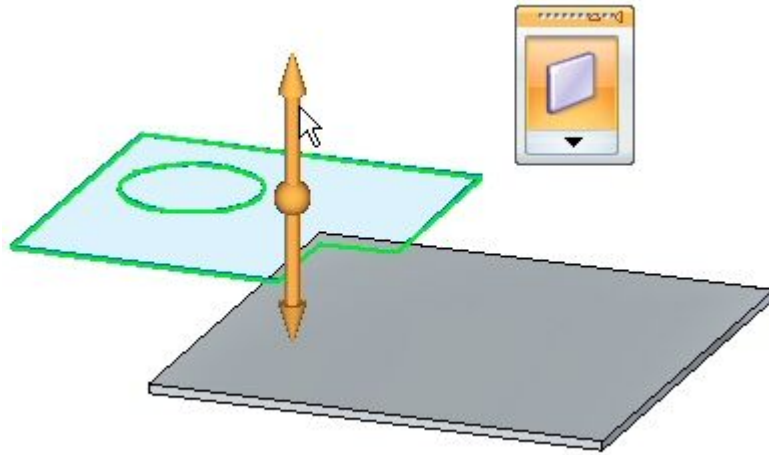
- ▶ Click to place the base feature, a tab, above the sketch as shown. Press Enter to accept the material thickness of 2.00 mm.



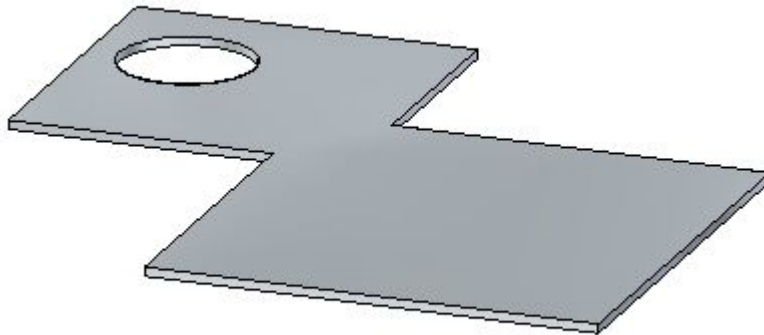
- ▶ To place the next tab, select the region shown.



- ▶ Select the handle pointing up as shown. Since the material thickness is defined, the tab will be placed when the handle is selected.



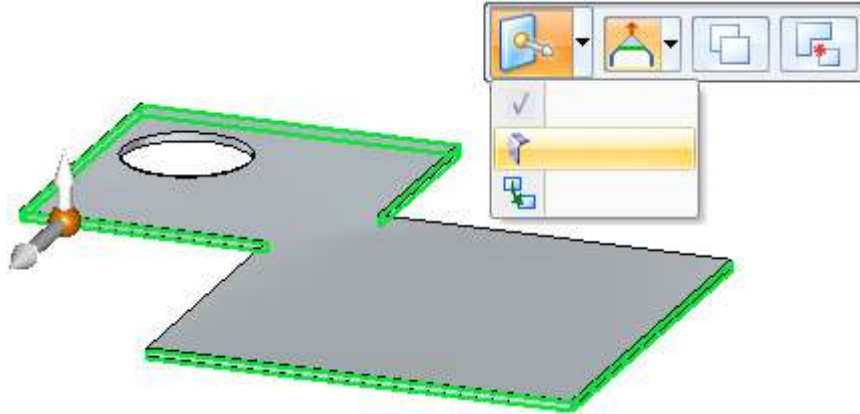
The base feature appears as shown.



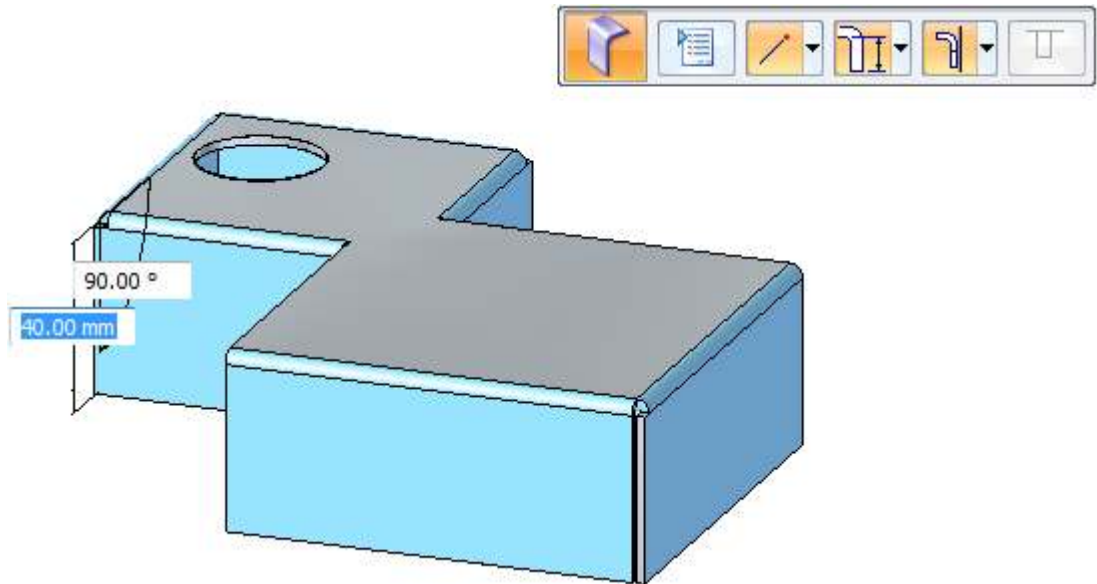
- ▶ Proceed to the next step.

Create the flanges

- ▶ Select the edges as shown and click the flange command.



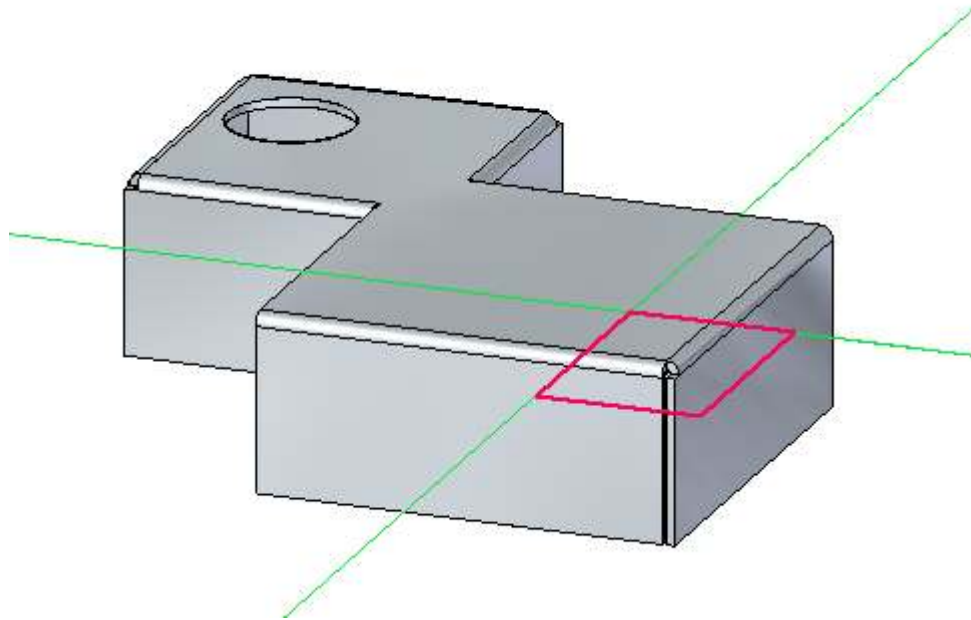
- ▶ Pull the flange down a distance of 40.00 mm.



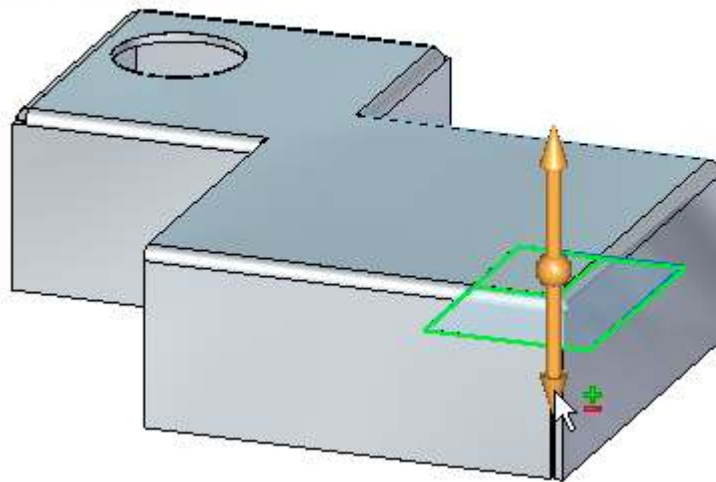
- ▶ Proceed to the next step.

Creating a cut

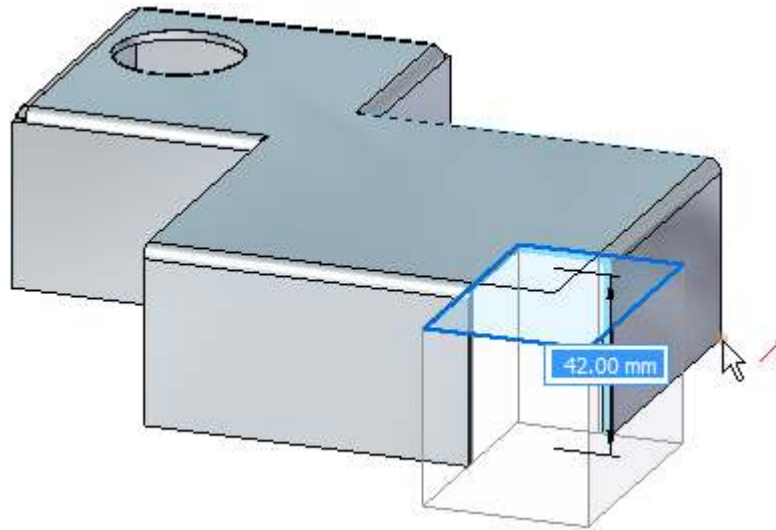
- Lock the sketch plane to the top face and place a rectangle approximately as the one shown below.



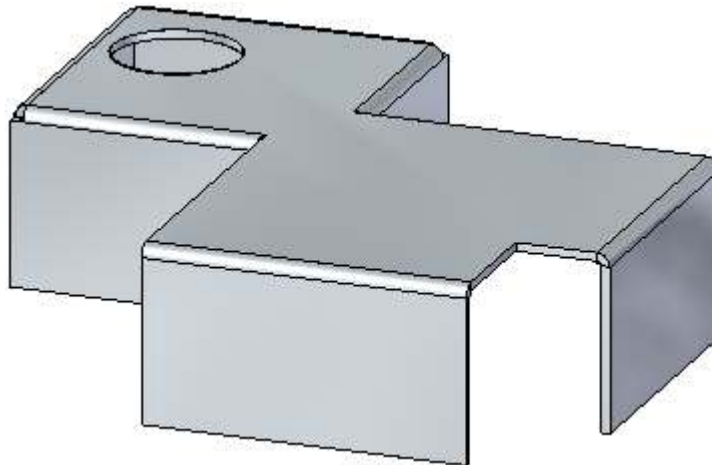
- Select the two regions shown. Notice the Cut command is selected. Click the downward pointing handle as shown.



- ▶ Click the endpoint on the edge shown to create the cut.

**Note**

Notice the depth of the cut is defined by the vertical distance below the regions.

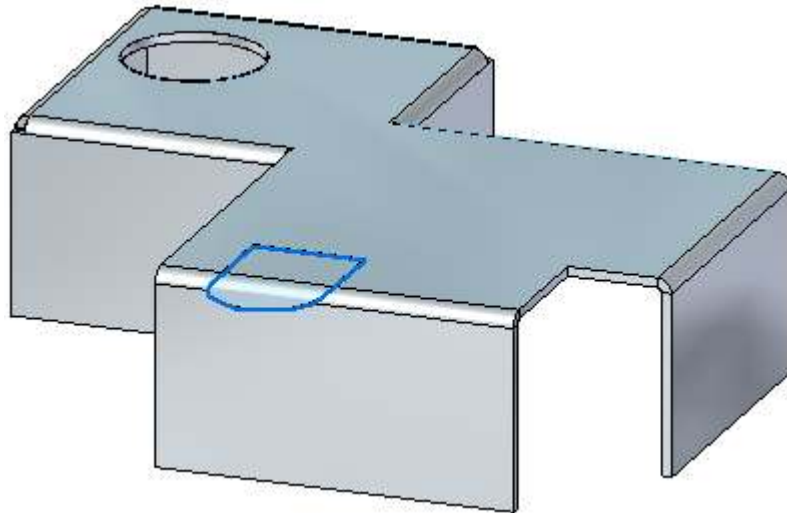


- ▶ Proceed to the next step.

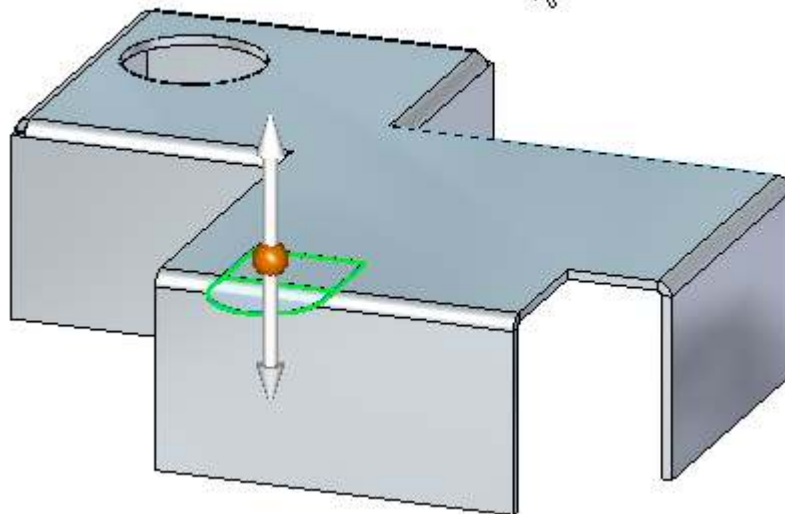
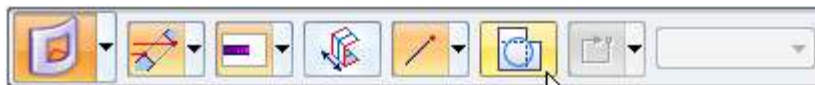
Creating a wrapped cut

The wrapped cut command temporarily flattens the bend to place the cut.

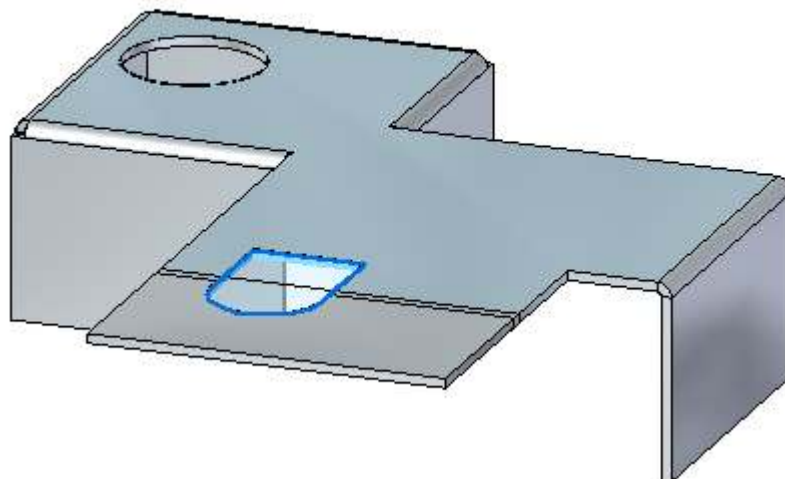
- ▶ Create an approximate sketch as shown below. Do not extend the sketch more than 30.00 mm from the edge of the flange.



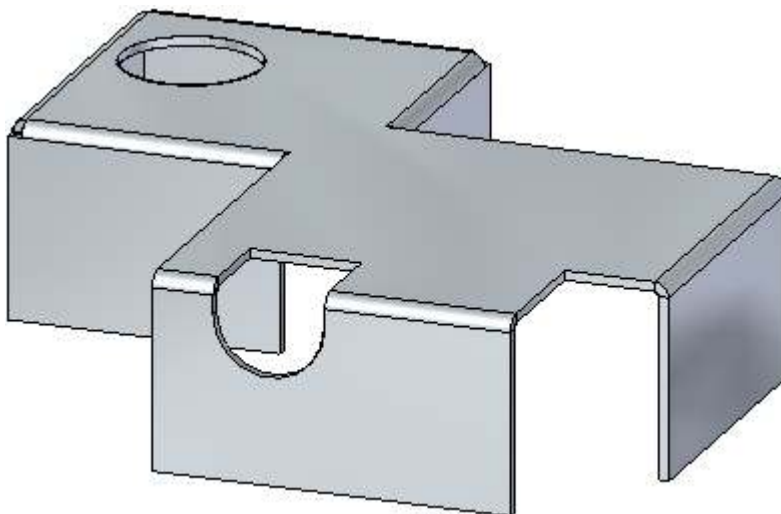
- ▶ Select the two regions shown and click the Wrapped Cut option as shown



- ▶ Select the downward pointing handle. The part is unfolded showing a preview of the wrapped cut. right-click to accept.



- ▶ The results are shown.

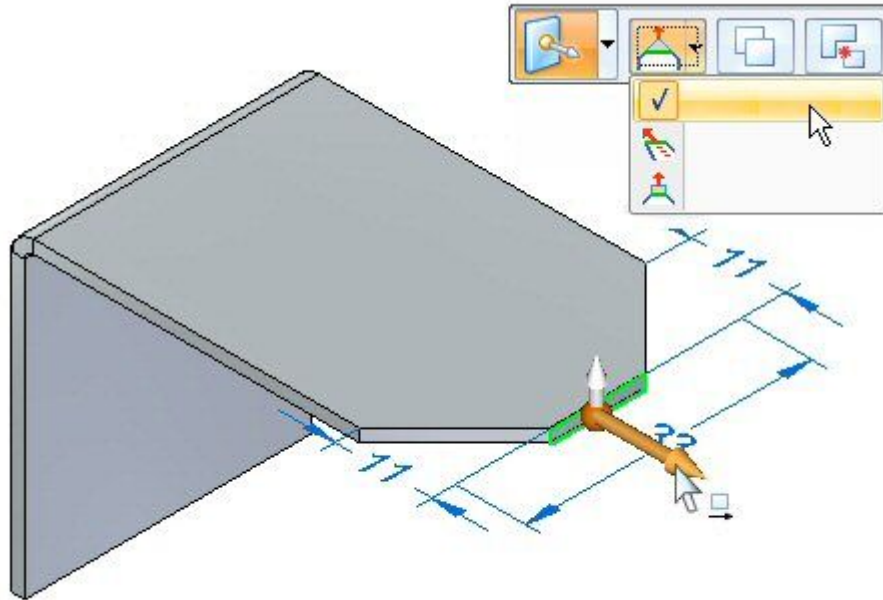


- ▶ Close the sheet metal document without saving.
- ▶ Proceed to the next step.

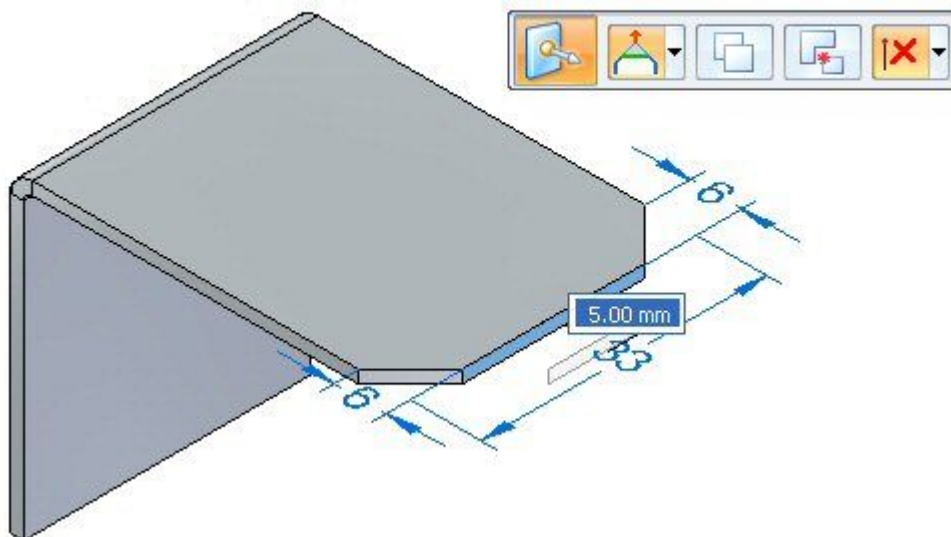
Moving thickness faces

- ▶ Click the  **Application** button ® **Open** ® *tab_move_activity.psm*.

- ▶ Select the primary axis of the thickness face shown. The Extend/Trim option is the default.

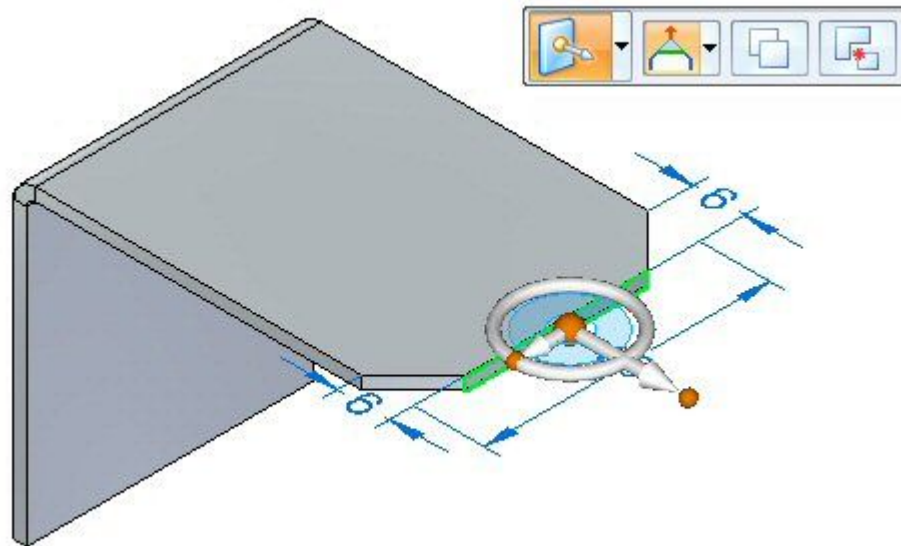


- ▶ Move the primary axis and move in the direction toward the bend. Enter a distance of 5.00 mm.

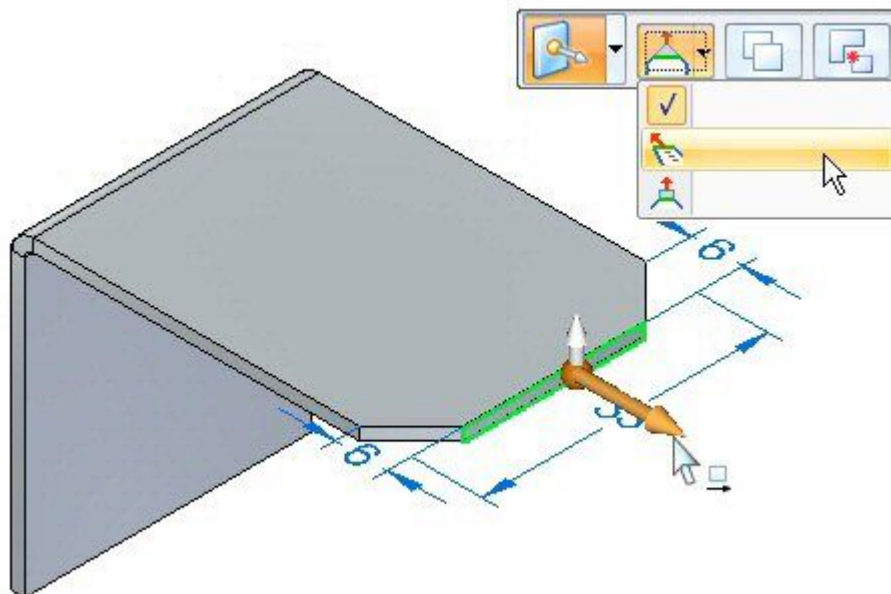


B*Activity: Using regions to create tabs and cuts*

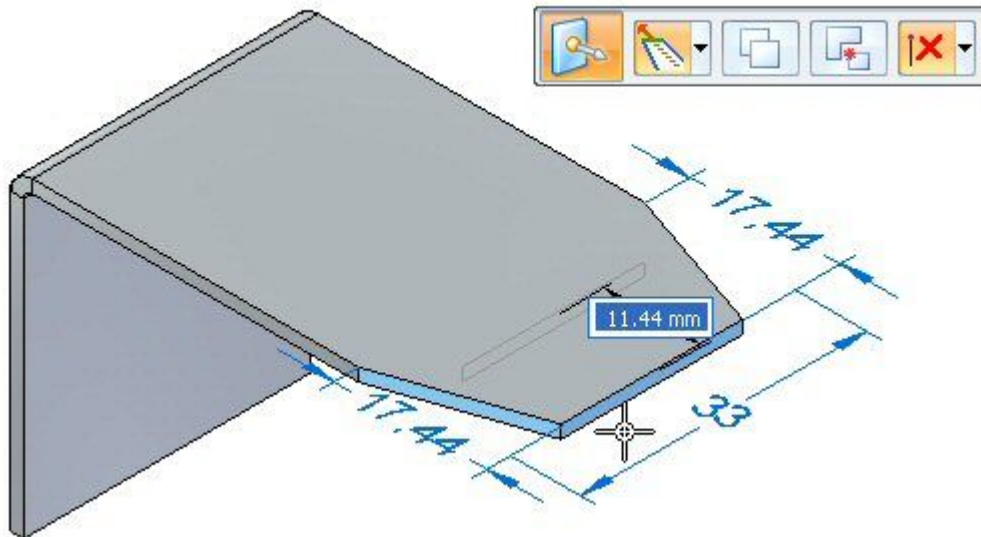
- ▶ Observe the behavior. The length of the thickness face changes and the orientation of the adjacent faces remains constant. The result is as shown below.



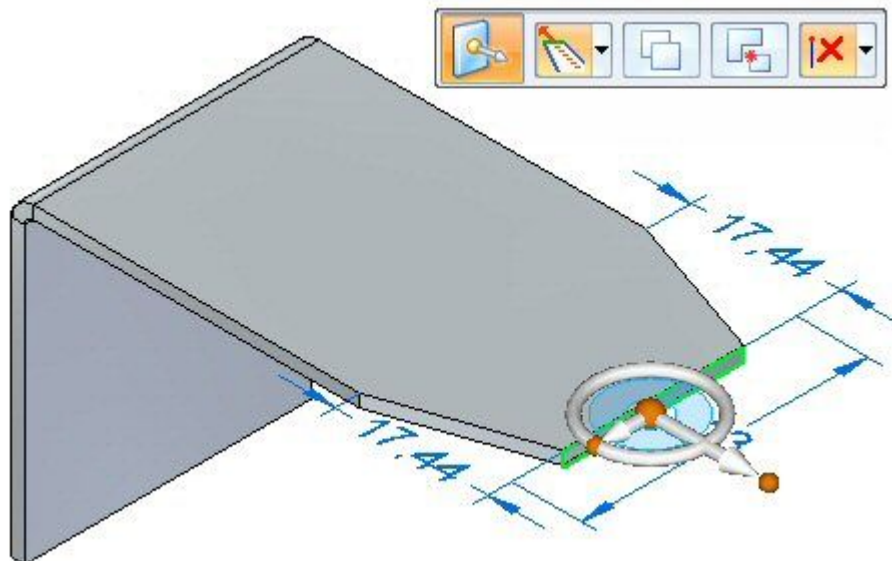
- ▶ Select the primary axis and the Tip option as shown.



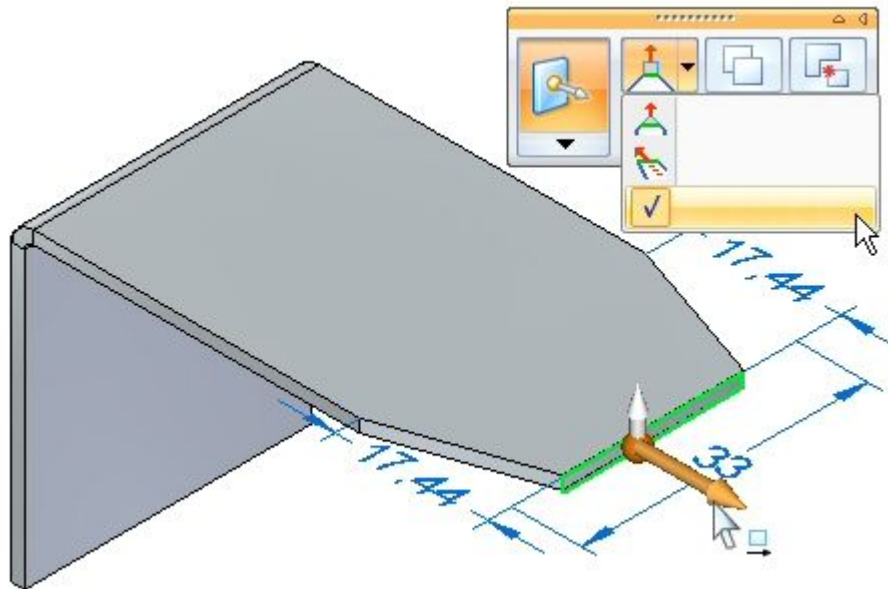
- ▶ Enter 11.44 mm for the distance to move the thickness face.



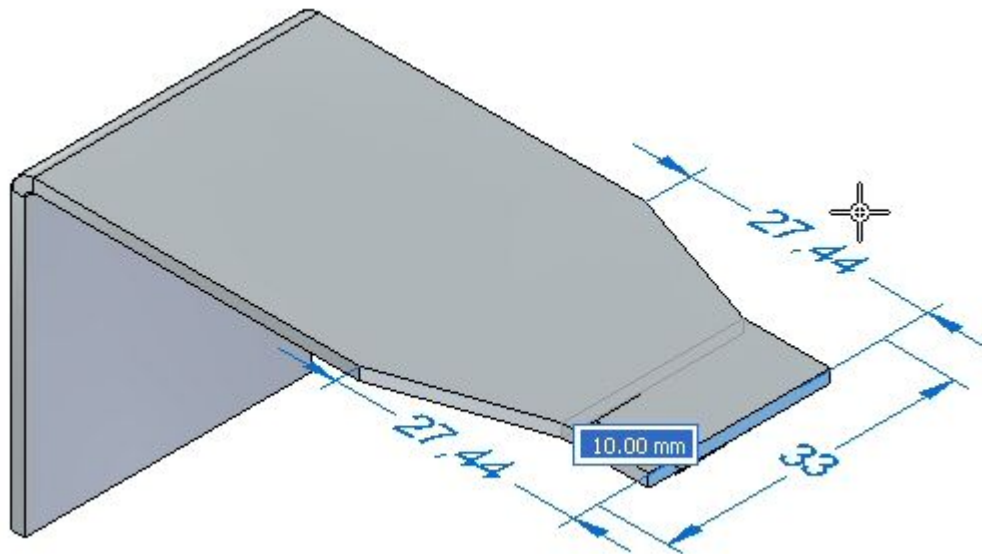
- ▶ Observe the behavior. The length of the thickness face is constant and the orientation of the adjacent faces changes. The result is as shown below.



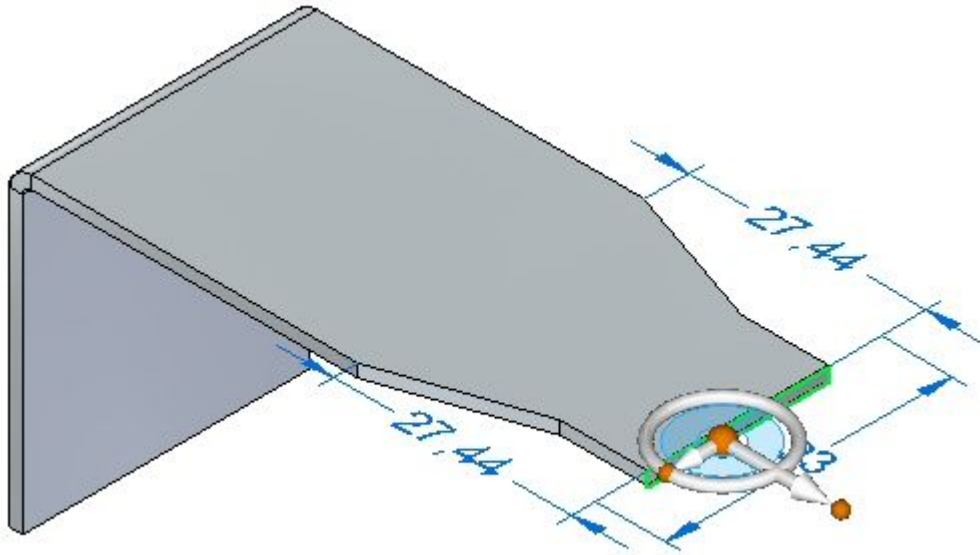
- ▶ Select the primary axis and the Lift option as shown.



- ▶ Enter a value of 10.00 mm.



- Observe the behavior. The length of the thickness face is constant and the orientation of the adjacent faces are constant. The tab extends perpendicular to the thickness face. The result is as shown below.




- This completes the activity. Close the sheet metal document without saving. Proceed to the activity summary.

Activity summary

In this activity you created a sheet metal base feature using a tab, and added additional material creating a tab from a sketch. Regions were used to create a cut and a wrapped cut. You learned the different options for moving a thickness face.

C Activity: Constructing a base feature using contour flange

Open a sheet metal file

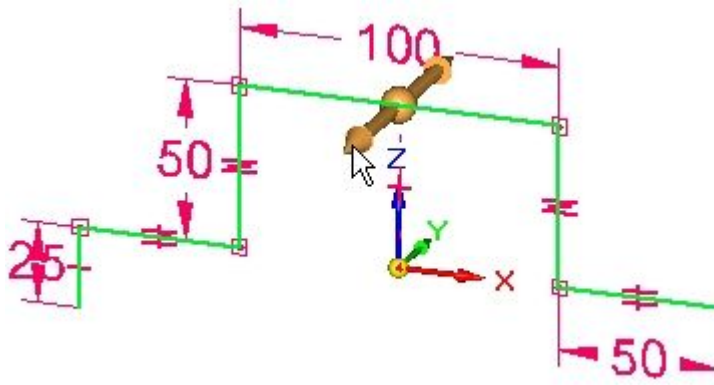
- ▶ Click the  **Application** button ® **Open** ® *contour_activity_1.psm*.
- ▶ Proceed to the next step.

Create a base feature using the Contour Flange command

- ▶ Click the Contour Flange command.



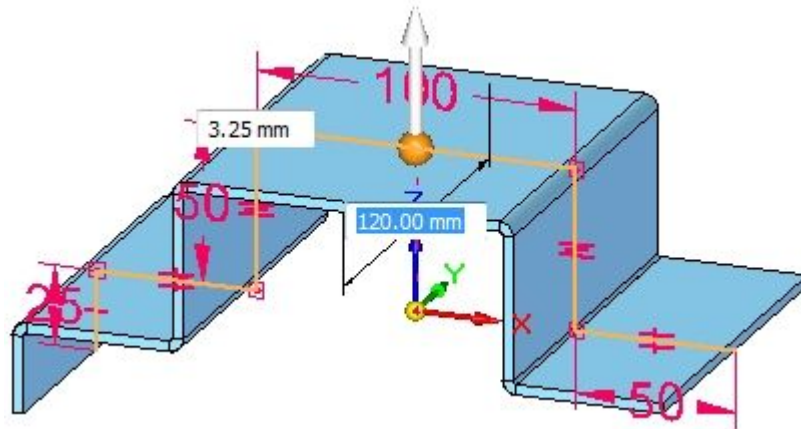
- ▶ Select the sketch shown, then click the flange handle.



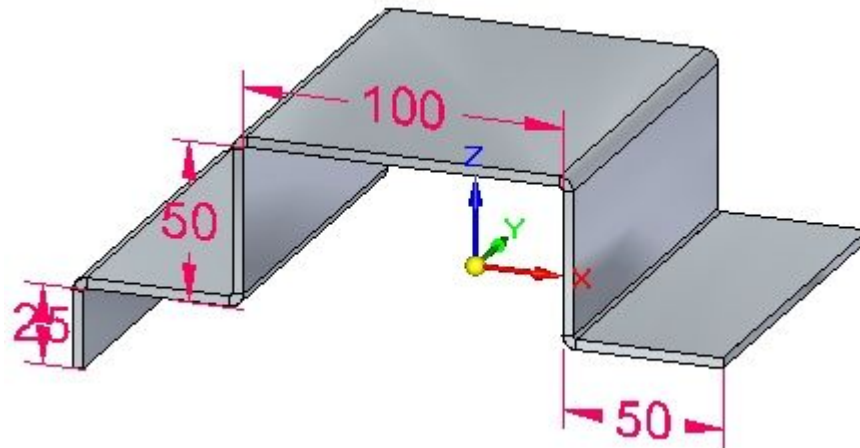
- ▶ Click the symmetric extent option.



- ▶ Use the tab key to change focus between the material thickness field and the extent field. Set the material thickness to be 3.25 mm and the extent to be 120.00 mm, and then press the enter key to complete the contour flange.



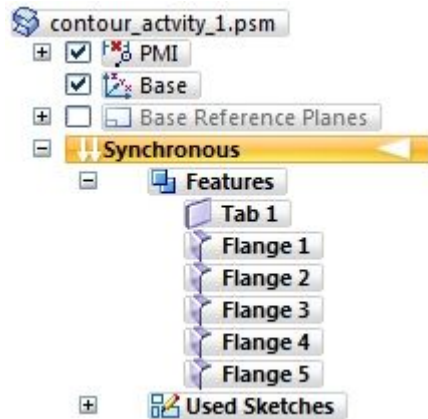
- ▶ The results are shown.



Note

The base feature can be created from a contour flange. Tangent curves in sketches are used to create bends.

- ▶ Observe PathFinder by moving the cursor over the features.




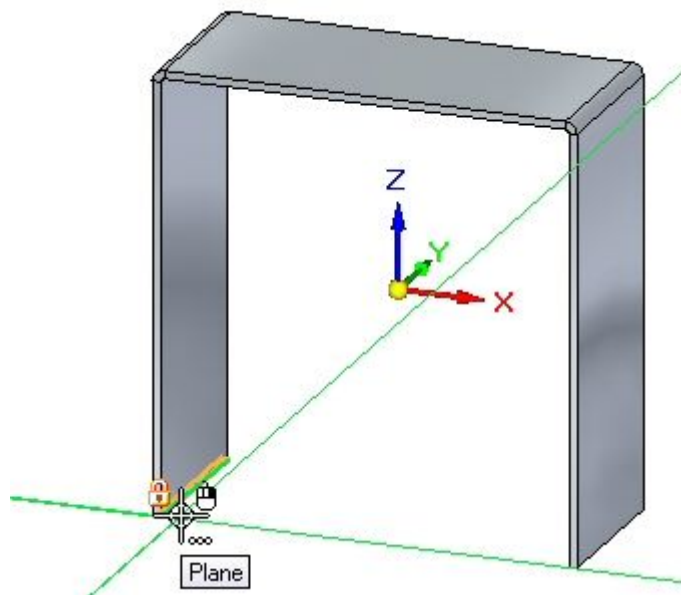
Note

The tab is created from the element chosen. Connected lines and tangent curves create flanges.

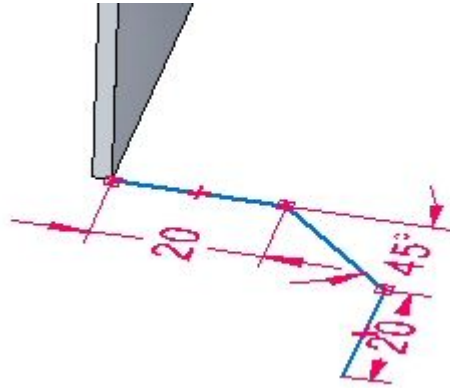
- ▶ Save and close the sheet metal document. Proceed to the next step.

Create a contour flange

- ▶ Click the  **Application** button ® **Open** ® *contour_activity_2.psm*.
- ▶ Lock the sketch plane to the plane shown.



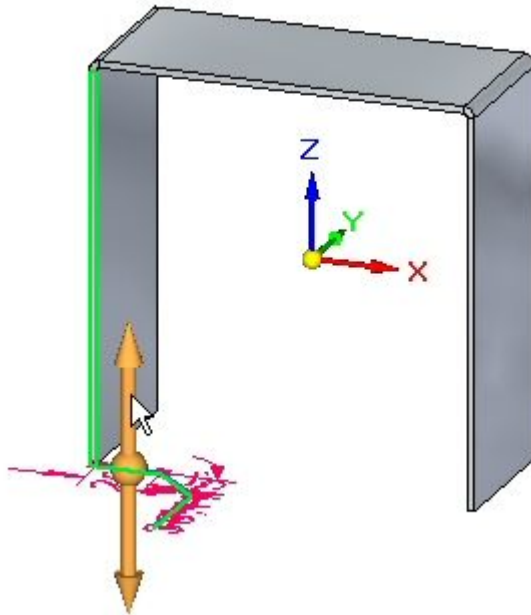
- ▶ Create the sketch shown. All segments are 20.00 mm.



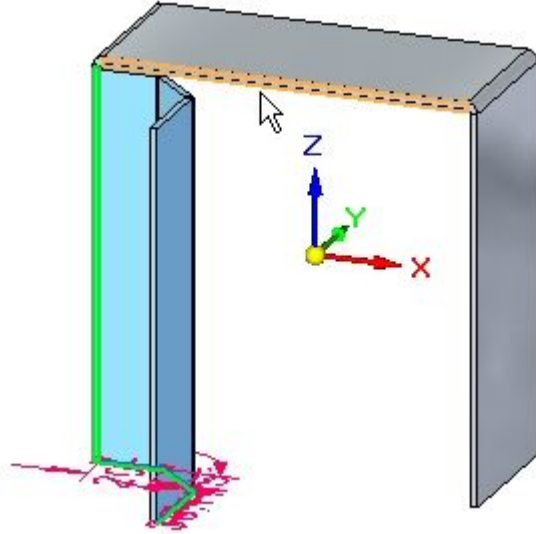
- ▶ Select the Contour Flange command.



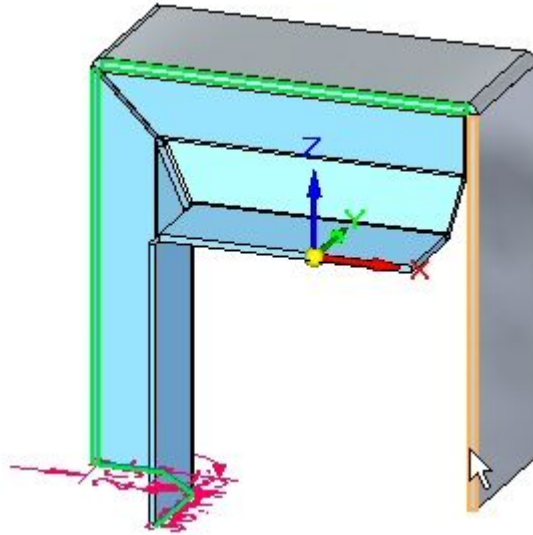
- ▶ Select the handle shown.



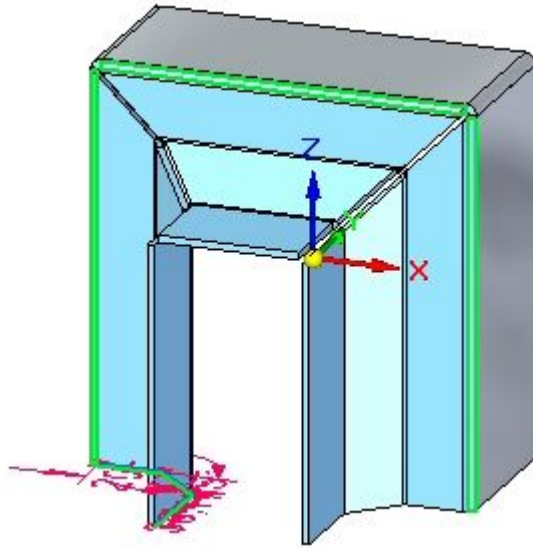
- ▶ Continue by selecting the adjacent edge shown.



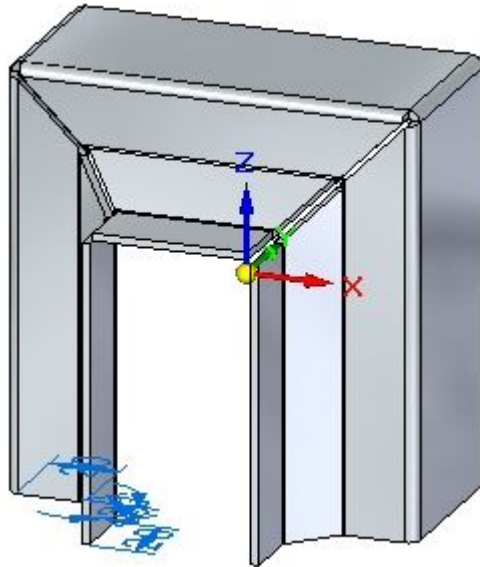
- ▶ Continue by selecting the adjacent edge shown.



- ▶ The preview is shown.



- ▶ Right-click to complete the contour flange.



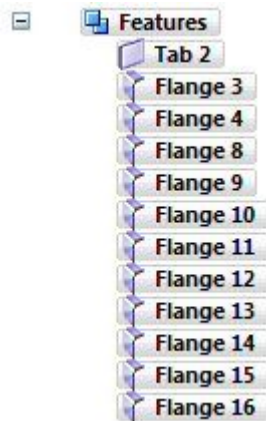
- ▶ Observe PathFinder by moving the cursor over the features.



Note

The contour flange is a single feature. The corner conditions can be edited.

- ▶ In PathFinder, right-click the contour flange feature, and then click separate. Observe the results.



Note

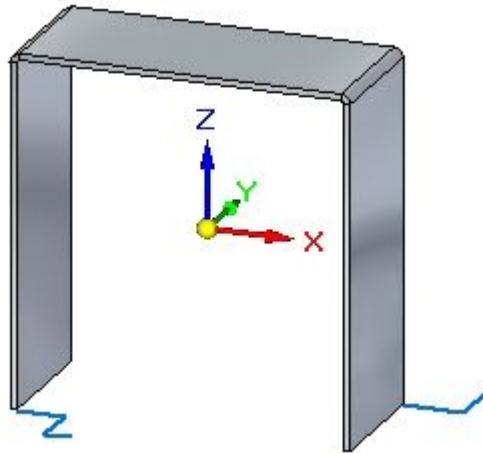
The flange numbers in PathFinder may not match the numbers in the image above. This is not a problem.

Notice that the contour flange feature was replaced by individual flanges. As a result, there is no associativity between the flanges, but you can edit the individual flanges independently of one another.

- ▶ Save and close the sheet metal document. Proceed to the next step.

Contour flange options

- ▶ Click the  **Application** button ® **Open** ® *contour_activity_3.psm*.



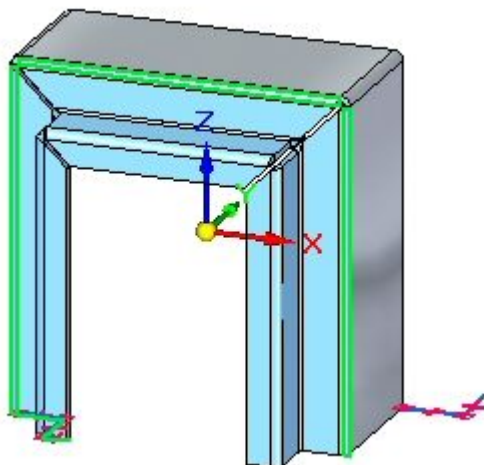
- ▶ Select the Contour Flange command.



Note

In the following steps you will change the options for end conditions on the contour flange and view the end conditions in the preview without accepting until the last step.

- ▶ Begin the contour flange shown using the default parameters.

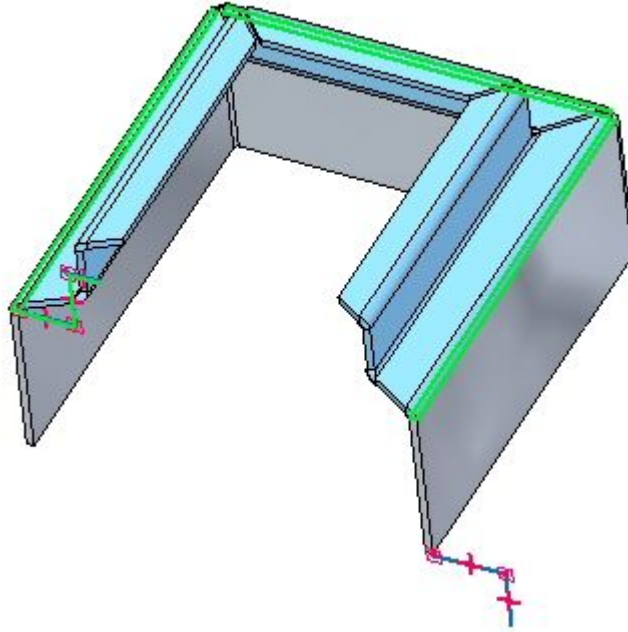


Activity: Constructing a base feature using contour flange

- ▶ Click the Options button.



- ▶ On the Miters and Corners tab, set the Miter option for the Start End and Finish End. Set each angle to -30° , then click OK. Observe the results.



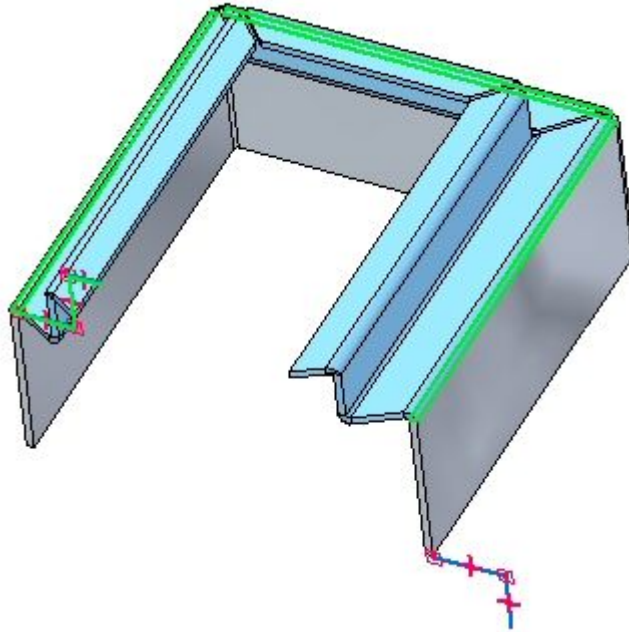
Note

The view has been rotated for clarity.

- ▶ Click the Options button.



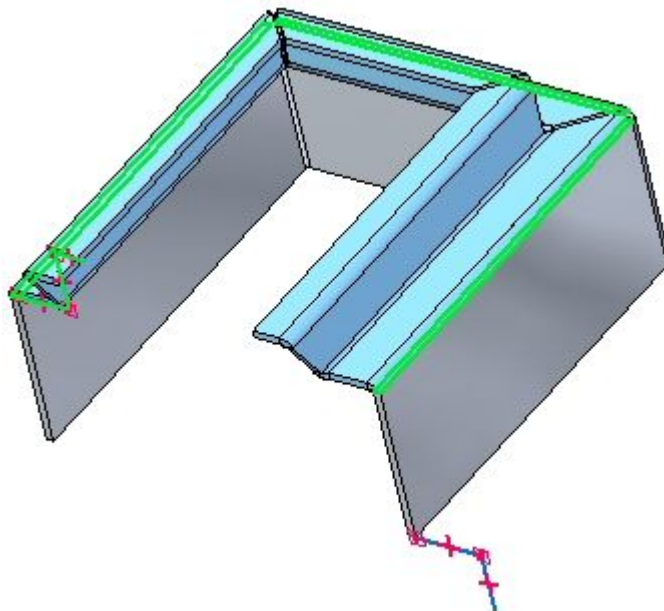
- ▶ On the Mitters and Corners tab, change the start end and finish end miter angles from a negative value to 30°, then click OK. Observe the results.



- ▶ Click the Options button.



- ▶ On the Mitters and Corners tab, for both the start end and finish end miter options, set the Normal to Source Face option, then click OK. Observe the results.

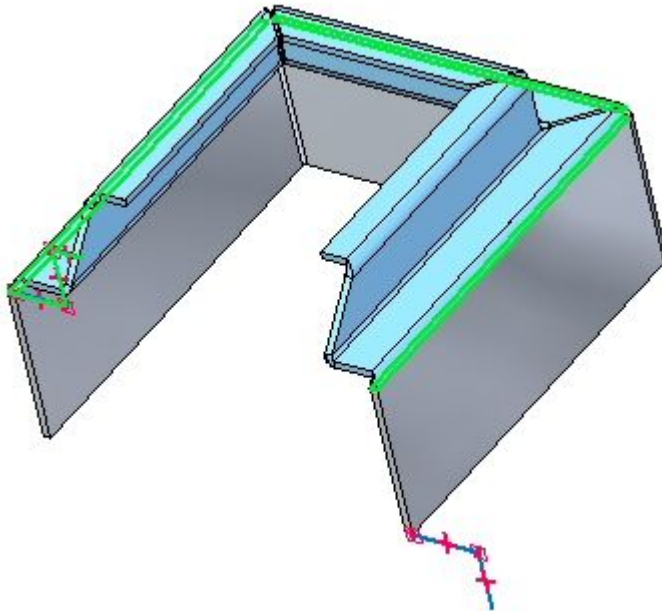


Activity: Constructing a base feature using contour flange

- ▶ Click the Options button.



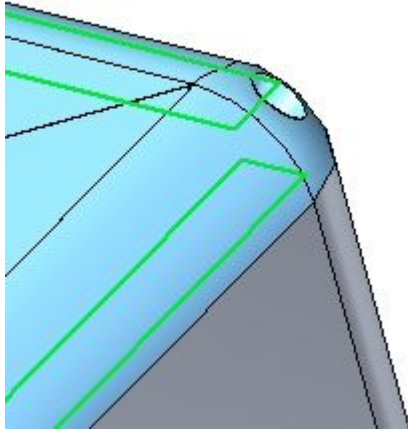
- ▶ Click the Miters and Corners tab.
- ▶ On the Miters and Corners tab, for both the start end and finish end miter options, set the miter angle to -45° . Then click OK. Observe the results.



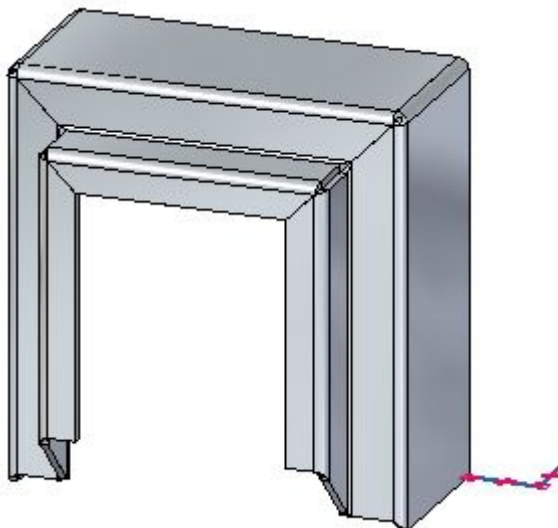
- ▶ Click the Options button.



- ▶ On the Mitters and Corners tab, in the Interior Corners section, set the Close Corner option. Set the Treatment option to Circular Cutout and click OK. Observe the result.



- ▶ Right-click to complete the contour flange. The result is shown.



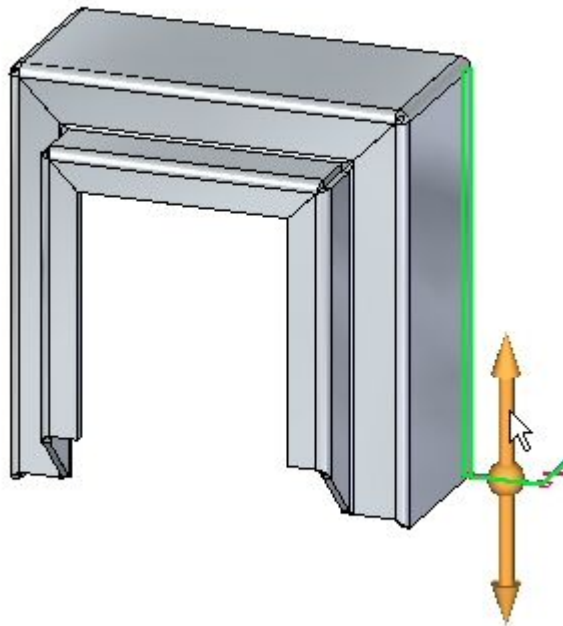
- ▶ Proceed to the next step.

Create a partial contour flange

- ▶ Select the Contour Flange command.



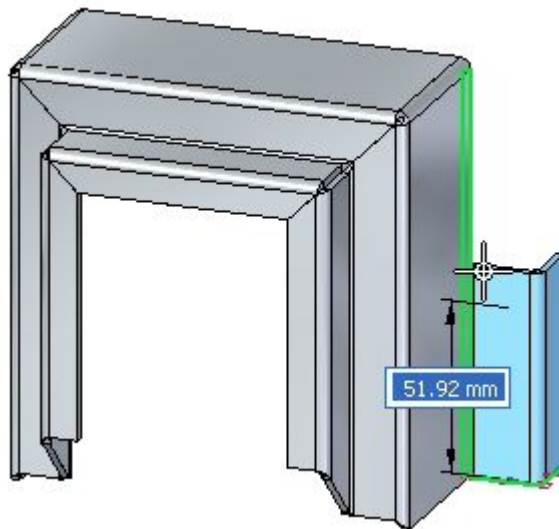
- ▶ Select the sketch as shown.



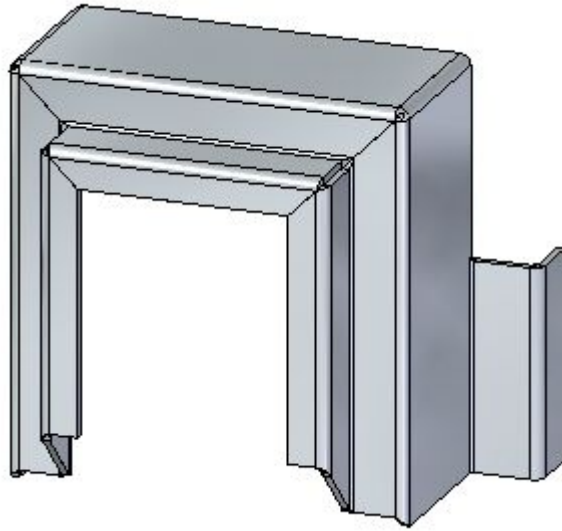
- ▶ Click the Partial Contour Flange option.



- ▶ Position the cursor approximately as shown, then click.



- ▶ The result is shown.

**Note**

Partial flanges can be further positioned by moving thickness faces or with dimensions.

- This completes the activity. Proceed to the activity summary.


Activity summary

In this activity you set the material thickness and extent to create a base feature using a contour flange. The components of the contour flange were examined and manipulated. Options for the construction of end conditions were explored, and a partial contour flange was placed.

D Activity: Flange and corner conditions

Open a sheet metal file

- Start Solid Edge ST4.

- Click the  **Application** button ® **Open** ® *flange_activity.psm*.

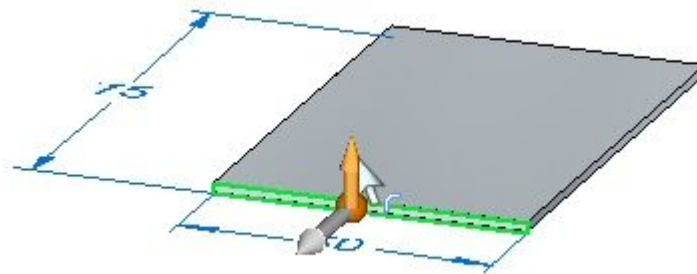
Note

This sheet metal part has a material thickness of 1.50 mm and a bend radius of 1.00 mm.

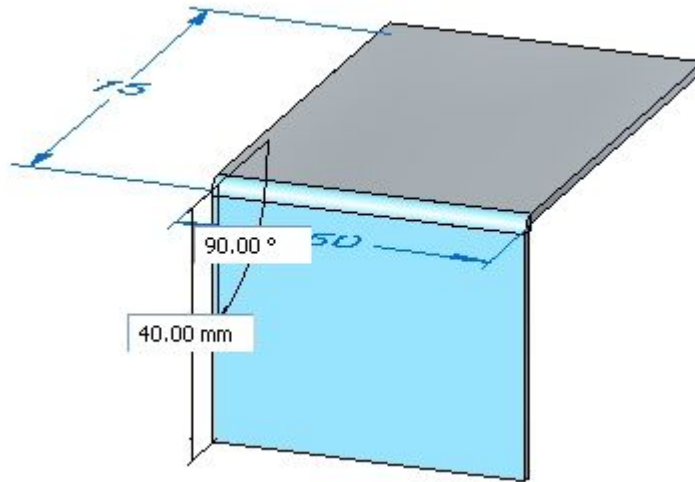
- Proceed to the next step.

Flange creation options

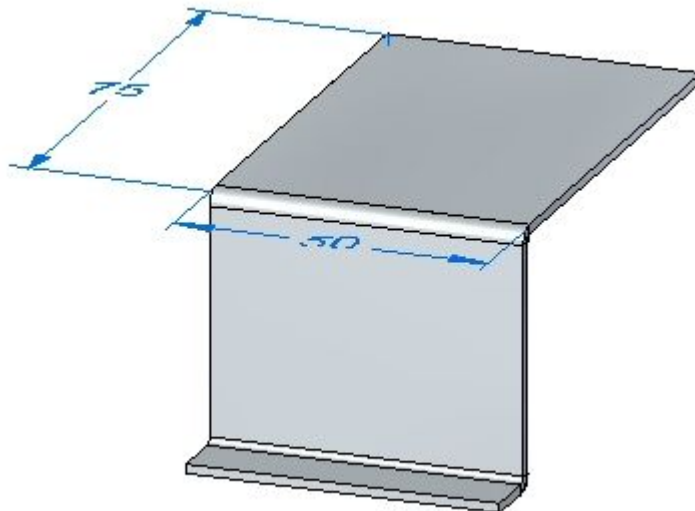
- Select the face shown and click the flange start handle.



- ▶ Create a flange with the default parameters that has the length of 40.00 mm.



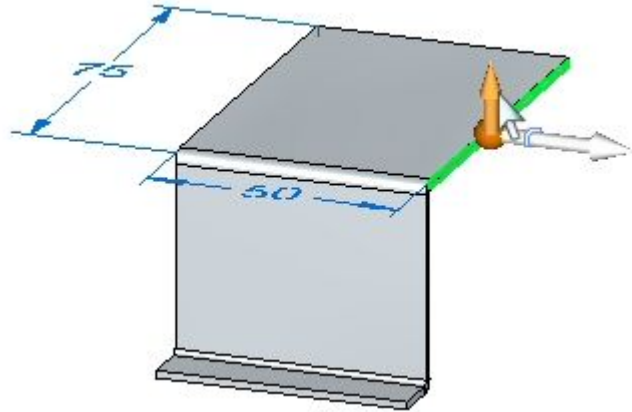
- ▶ Create the flange shown below with a length of 10.00 mm.



Note

The following steps will demonstrate the different options for corner relief.

- ▶ Select the face shown and click the flange handle. Click flange options and ensure the Corner Relief is set to **Bend Only**.

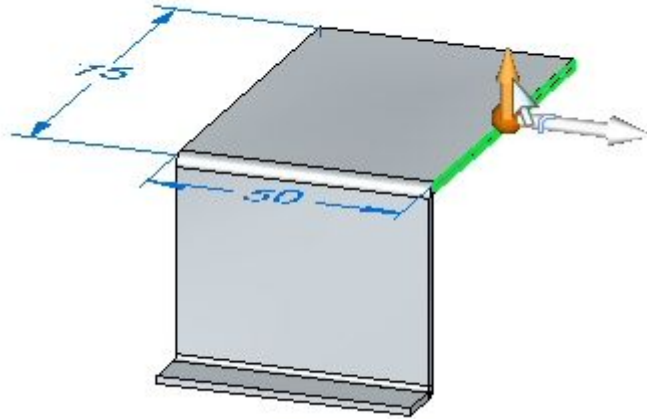


- ▶ Pull the flange to the end of the bottom of the flange just created. Observe the corner relief.

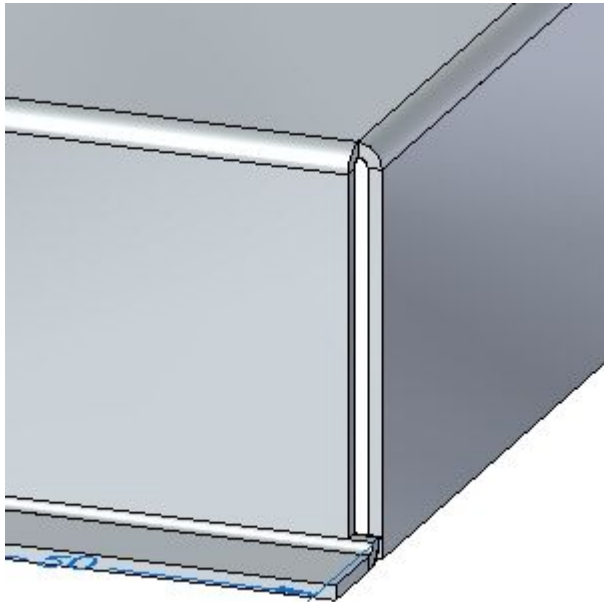


- ▶ Click the Undo command to remove the flange you just created.

- ▶ Select the face shown and click the flange handle. Click the Options button and set the Corner Relief to **Bend and Face**.

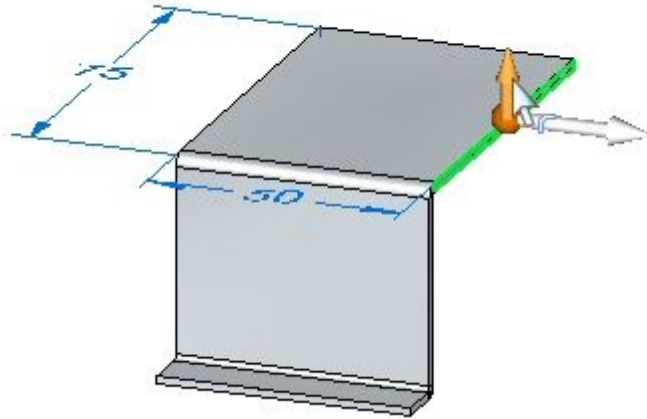


- ▶ Pull the flange to the same distance as in the previous step. Observe the corner relief.

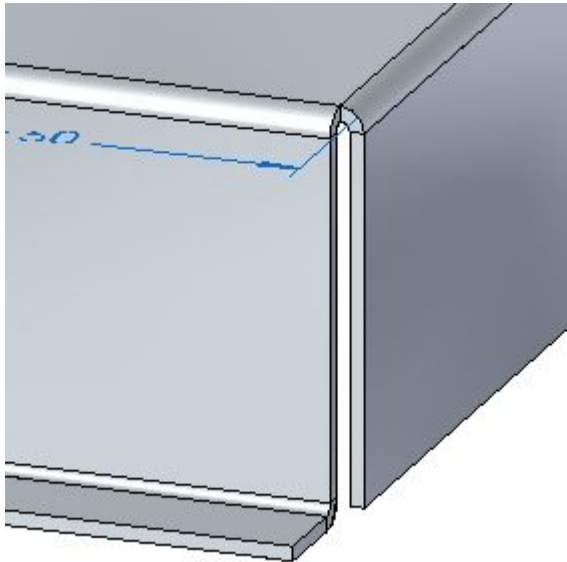


- ▶ Click the Undo command to remove the flange you just created.

- ▶ Select the face shown and click the flange handle. Click the Options button and set the Corner Relief to **Bend and Face Chain**.



- ▶ Pull the flange to the same distance as in the previous step. Observe the corner relief.



- ▶ Close the file without saving.
- ▶ Proceed to the next step.

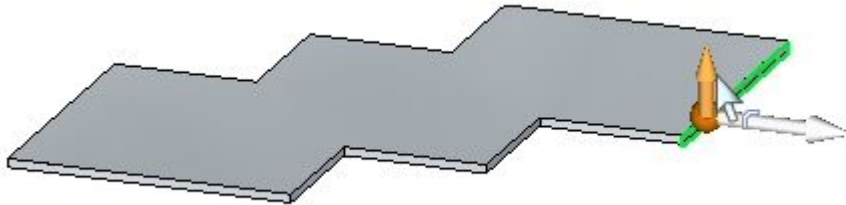
Partial flanges

- ▶ Click the  **Application** button ® **Open** ® *relief_activity.psm*.

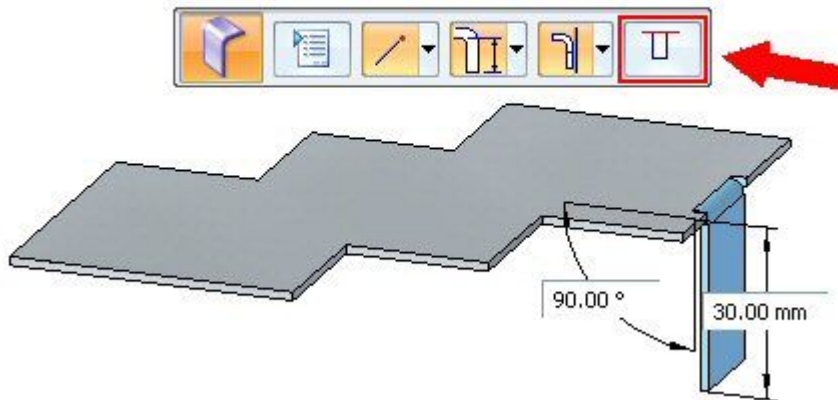
Note

This sheet metal part has a material thickness of 1.50 mm and a bend radius of 1.00 mm.

- ▶ Select the face shown and select the flange start handle.



- ▶ Click the Partial Flange option and create a flange with a length of 30.00 mm.



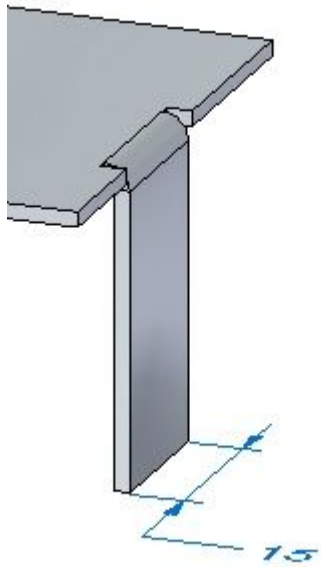
Note

Partial flanges are created with a width equal to 1/3 of the thickness face chosen, and the selection point is defines the edge of the partial flange. The flange can be modified to the desired width using dimensions to control the width.

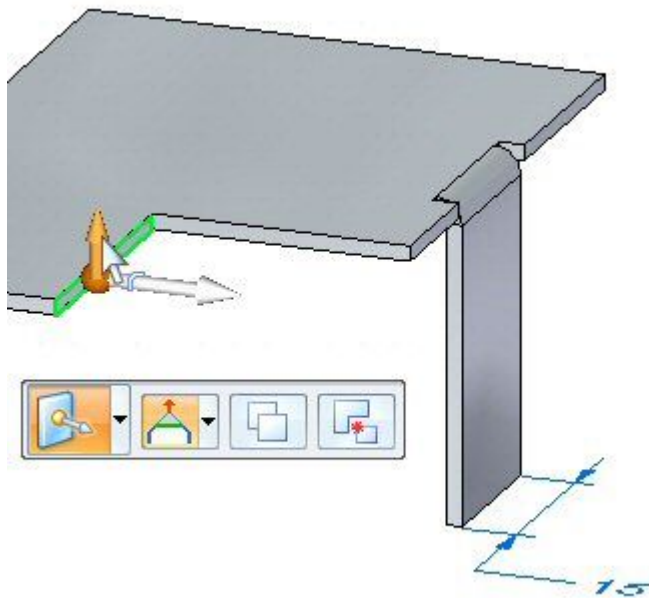
- ▶ Click the smart dimension command.



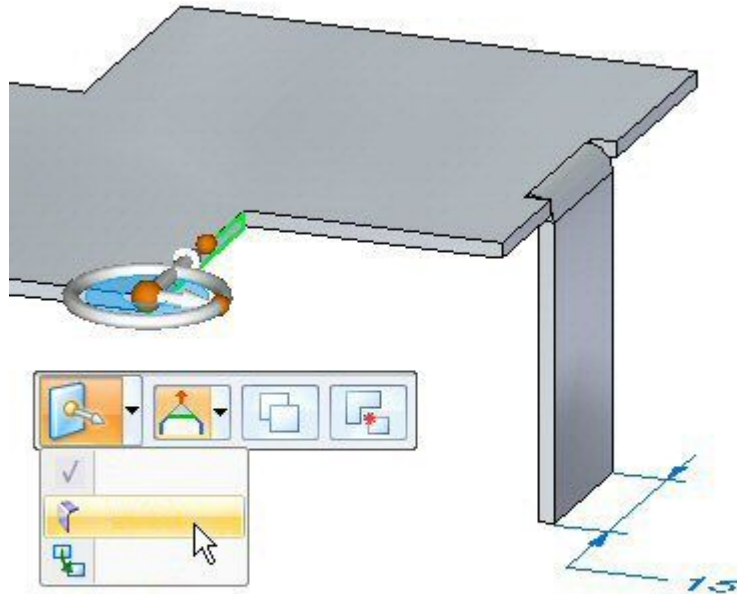
- ▶ Place a dimension on the bottom edge of the flange just created. Change the width of the flange to 15.00 mm by editing the dimension.



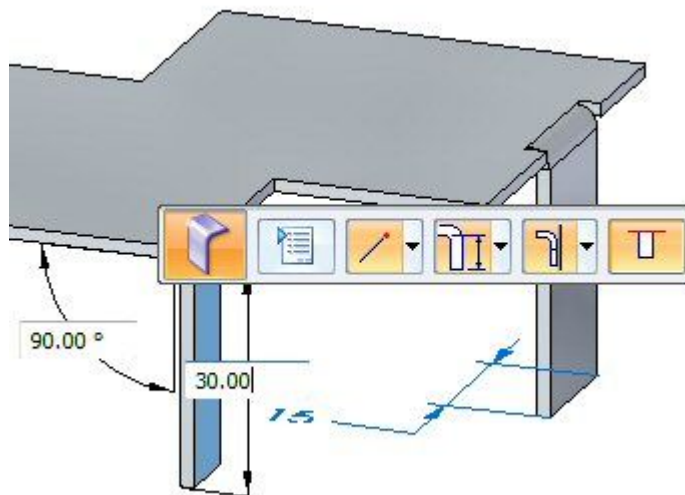
- ▶ Select the face shown.



- ▶ The origin of the start point will be changed by moving the steering wheel to the end of the thickness face. Move the steering wheel to the position shown, and select the flange command from the command bar.



- ▶ Click the partial flange option and create a flange with a length of 30.00 mm.

**Note**

The origin of this flange partial flange is at the end of the thickness face and is 1/3 the length of the thickness face.

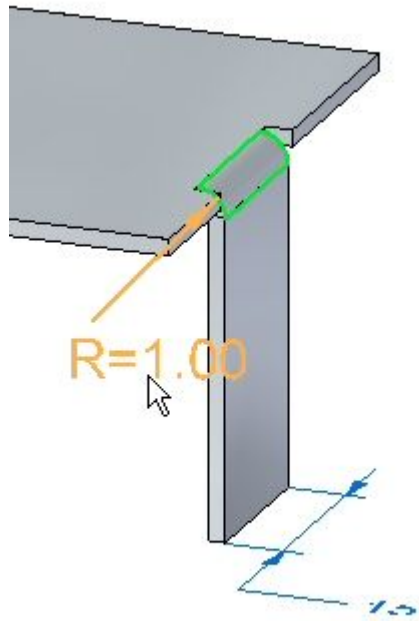
- ▶ Proceed to the next step.

Bend relief

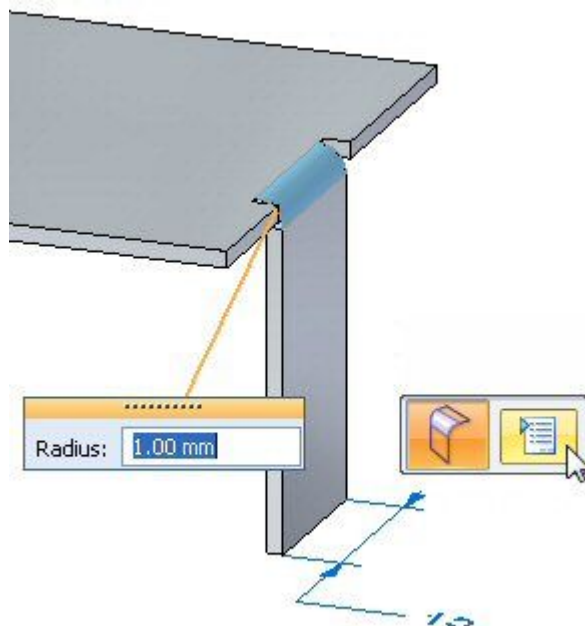
Note

The default bend relief can be overridden during placement, or after placement when editing a bend.

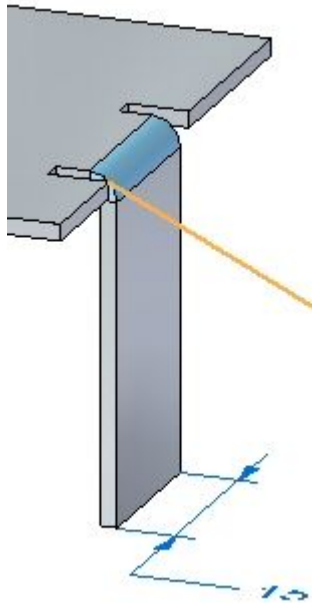
- ▶ Select the bend shown, then click the edit feature handle.



- ▶ Click the Bend Options button.




- ▶ Select the Override Global Value next to the Depth field and change the depth to 3.00 mm. Repeat the step to change the width to 2.00 mm.



- ▶ Experiment with different lengths, widths, and types of bend relief before dismissing the bend options dialog box and observe the results.
- ▶ Close the sheet metal document without saving. Proceed to the next step.

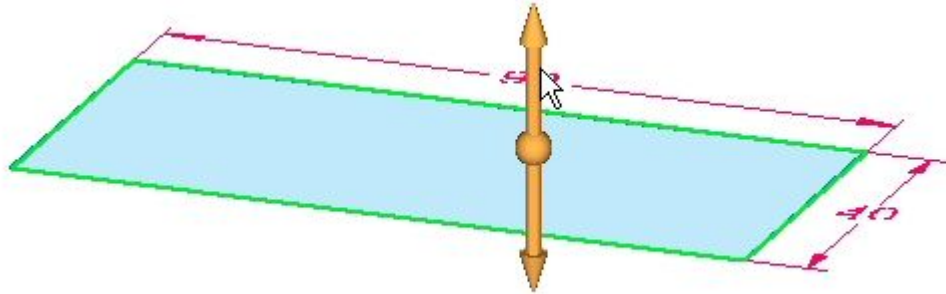
Corners

- ▶ Click the  **Application** button ® **Open** ® *corner_activity.psm*.

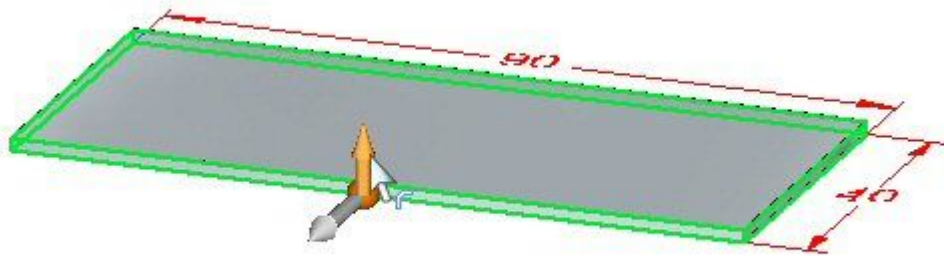
Note

This sheet metal part has a material thickness of 1.50 mm and a bend radius of 1.00 mm.

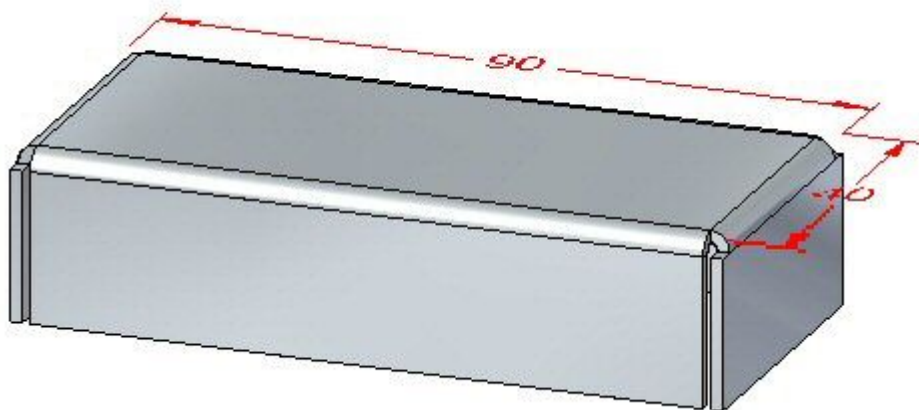
- ▶ Select the region shown and create a tab by pulling the handle up.



- ▶ Select all thickness faces and then click the flange start handle.



- ▶ Create flanges with the length of 20.00 mm as shown.



- ▶ When more than one thickness edge is used to create flanges, observe the following:
 - Shorter sides are bent first.
 - Relief cuts, when required are made to the longer sides.

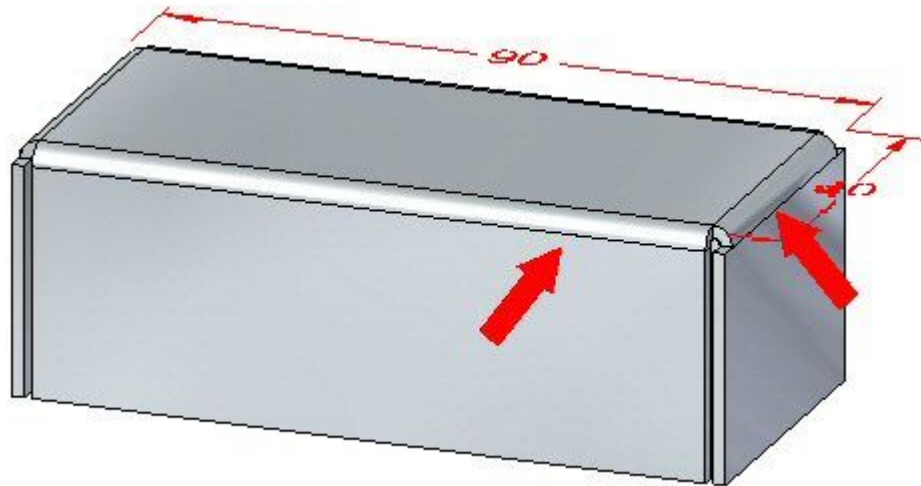
Note

When 3 or more thickness faces of the same length are encountered: Thickness faces are sorted according to length and parallelism. Parallel faces are bent first.

- ▶ Click the 2-Bend Corner command.



- ▶ Select the two bends shown below.

**Note**

The command closes the corner upon selection of the two bends.

- ▶ Click the Overlapping Corner option with the Corner Treatment set to Open.



- ▶ Change the gap value to 0.30 mm and the overlap ratio to 0.75. Observe the results.
- ▶ Click the Flip option and observe the results.




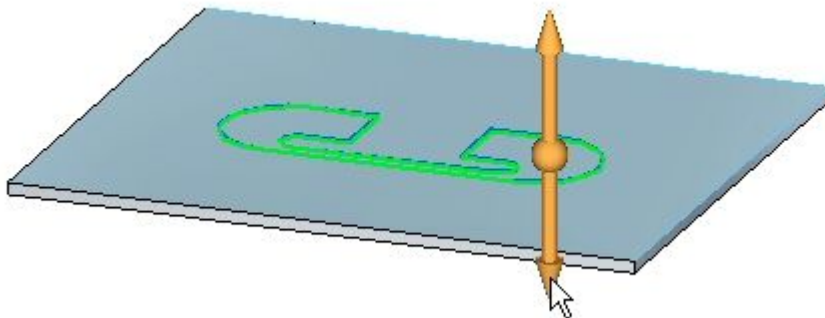
- ▶ Click the Closed Corner option. Observe the how the corner closes.



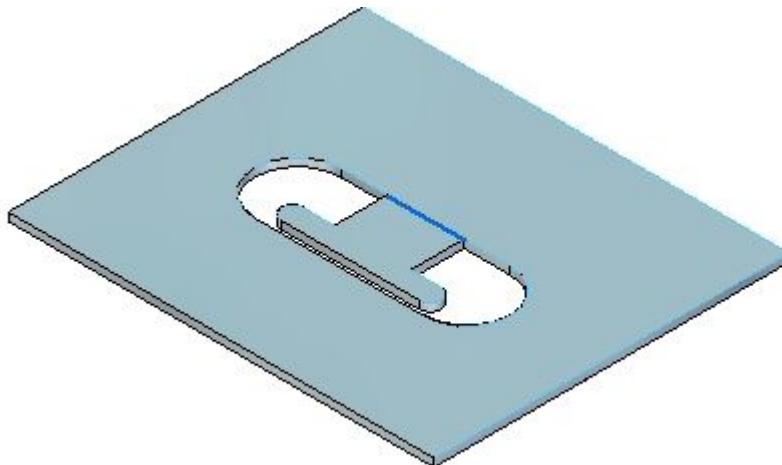
- ▶ Change the Corner Treatment to Closed, and set the gap value to 0.30 mm. Observe the change.
- ▶ Change the Corner Treatment to Circular Cutout, and set the gap value to 0.40 mm. Set the diameter to 1.50 mm. Observe the change.
- ▶ Close the sheet metal document without saving. Proceed to the next step.

Inserting a bend

- ▶ Click the  **Application** button ® **Open** ® *bend_activity.psm*.
- ▶ Select the region shown and create a cutout.



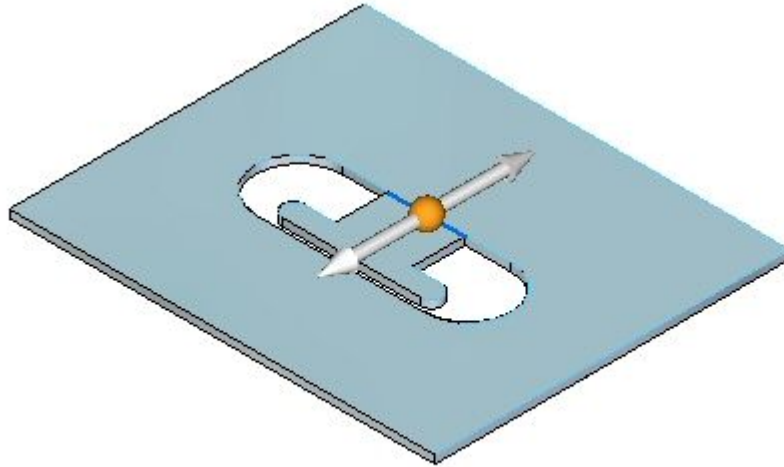
- ▶ Sketch a line as shown across the tab.



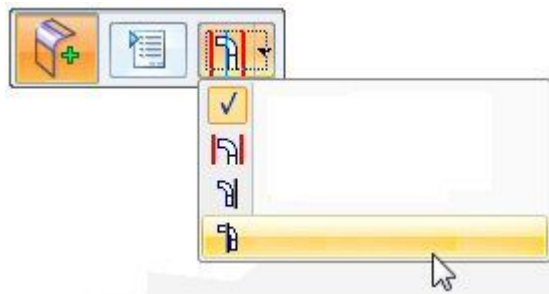
- ▶ Click the Bend command.



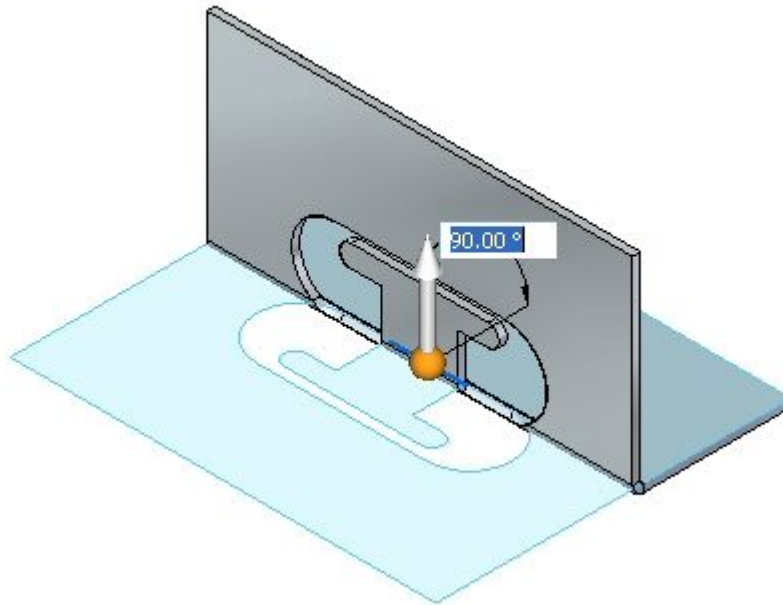
- ▶ Select the line as shown.



- ▶ Select the Material Outside option.



- ▶ Select the side shown. Notice the extent of the bend traverses the length of the layer face.



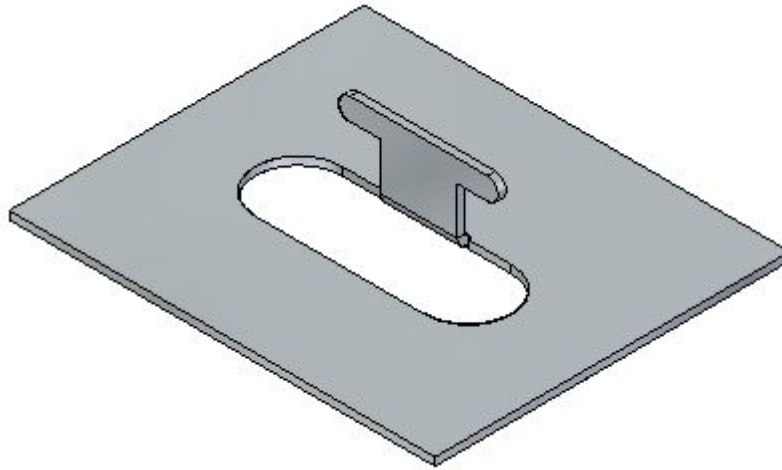
- ▶ Click the Bend Options button



- ▶ Deselect Extend Profile and click OK.



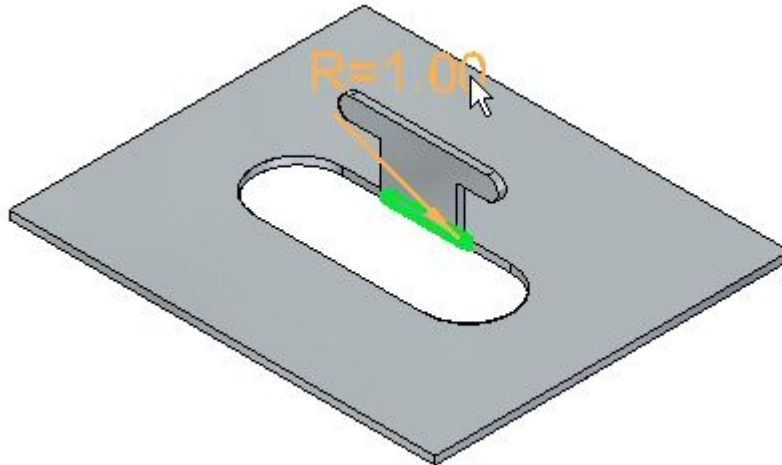
- ▶ Right-click to complete the bend and add the flange. The results are shown.



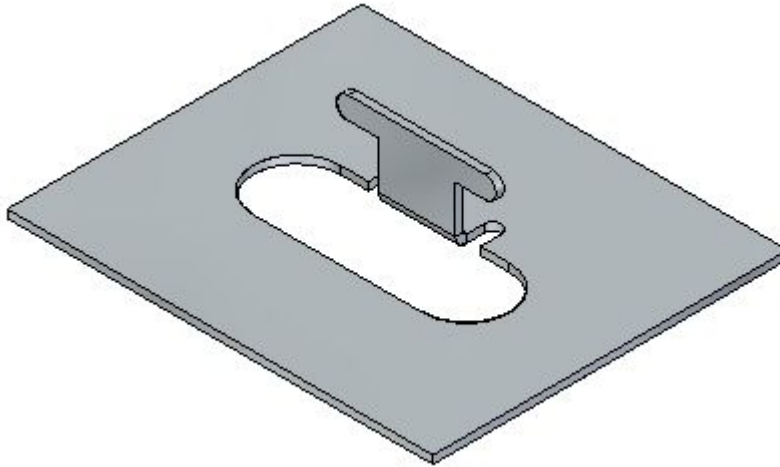
- Proceed to the next step.

Editing a bend

- Click the Select tool and select the bend. Click the edit handle as shown.



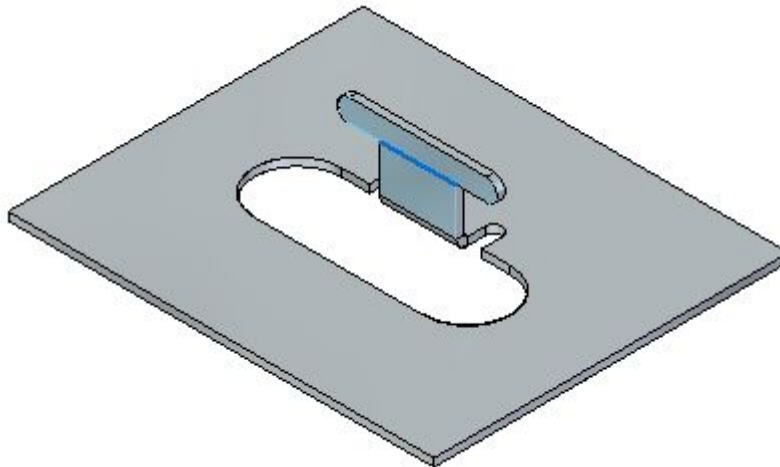
- ▶ Click the Options button and set the relief to be round with a width of 5.00 mm and a depth of 5.00 mm.



Note

The relief could have been set during the creation of the bend in the previous step. The purpose of changing the relief at this point is to demonstrate the ability to edit a previously placed feature.

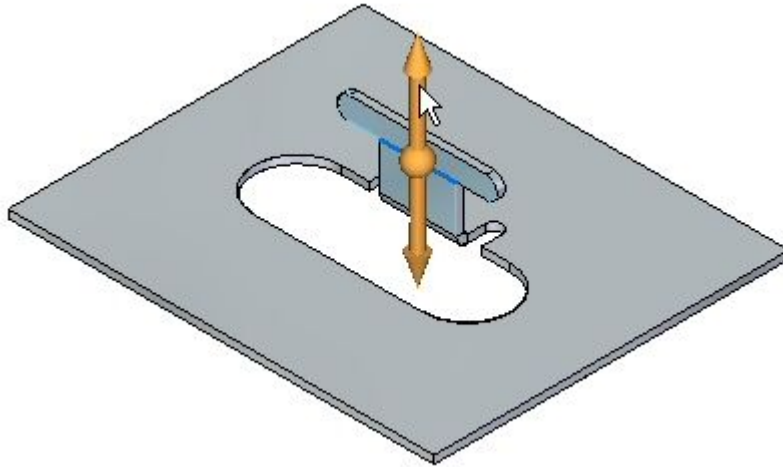
- ▶ Sketch a line as shown across the tab.



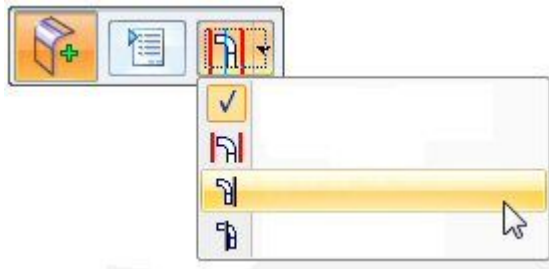
- ▶ Click the Bend command.



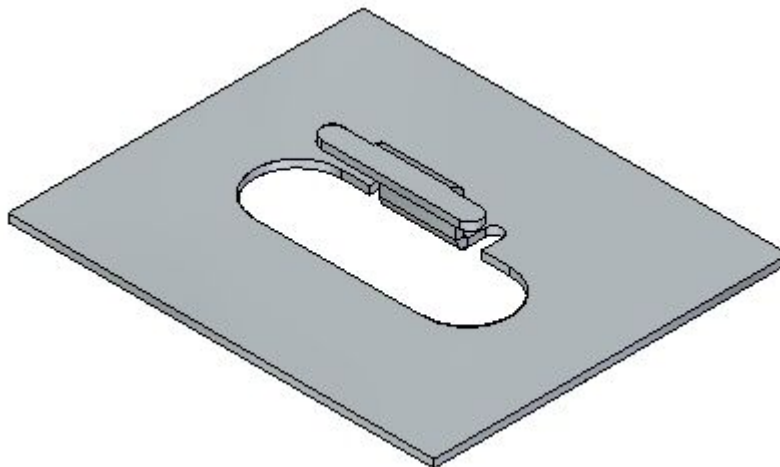
- ▶ Select the line as shown.



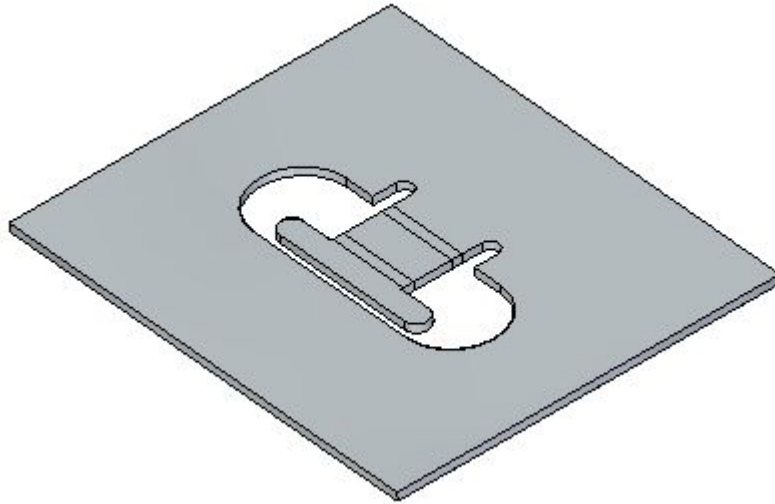
- ▶ Select the Material Inside option.



- ▶ Right— click to complete the bend.



- ▶ Rotate the view and examine the two bends just placed. The flat pattern is shown.



Note


The two bends placed used the length of existing material to create the flanges. Compare this workflow to the jog command in another activity.

Note

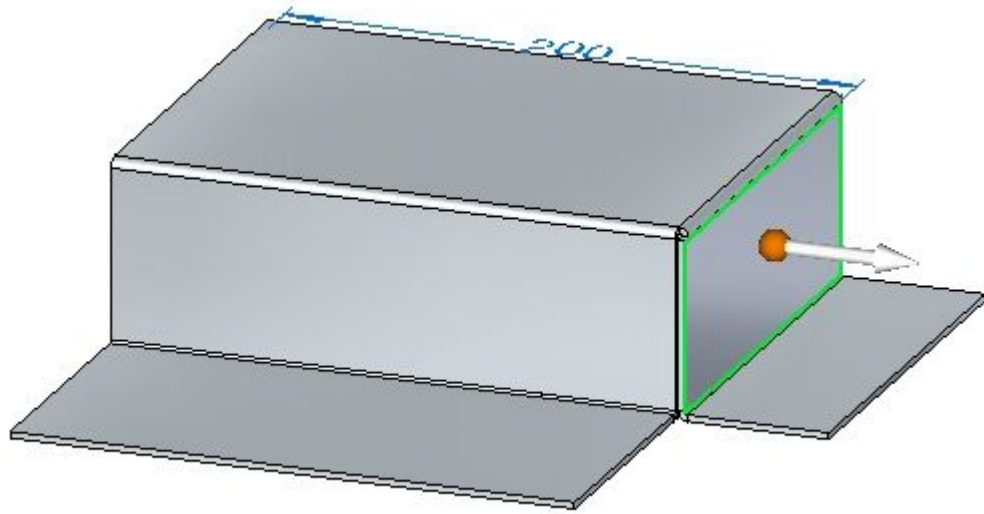
Creating a flat pattern will be covered in another activity.

- ▶ Close the file without saving. Proceed to the next step.

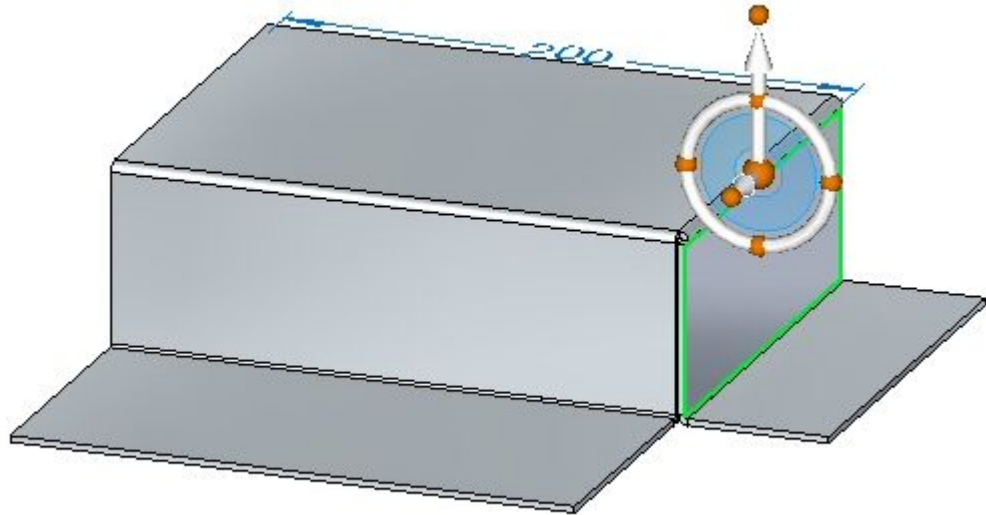
Moving faces

- ▶ Click the  **Application** button ® **Open** ® *move_activity.psm*.

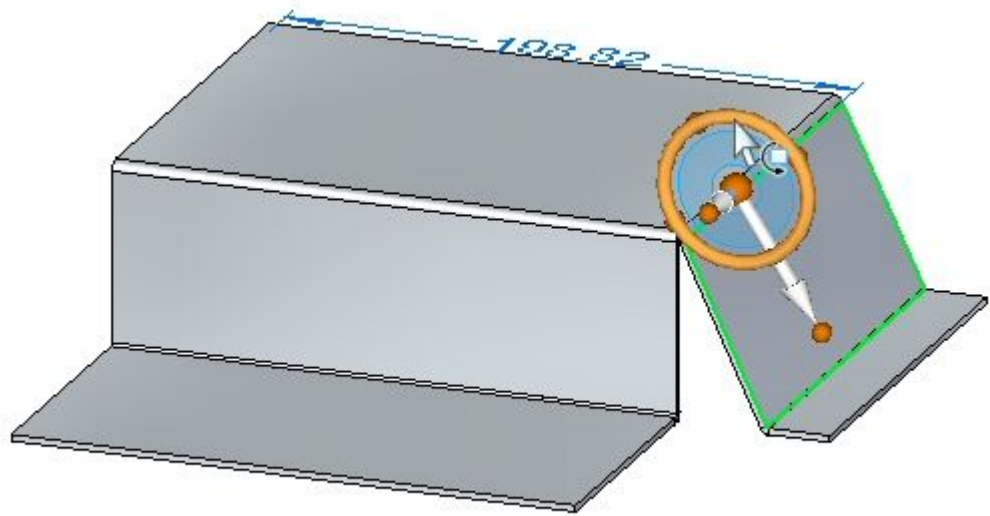
- ▶ Select the face shown.



- ▶ Select the origin of the steering wheel and position the steering wheel as shown.



- ▶ Select the steering wheel torus and rotate the flange by an angle of 25° as shown.



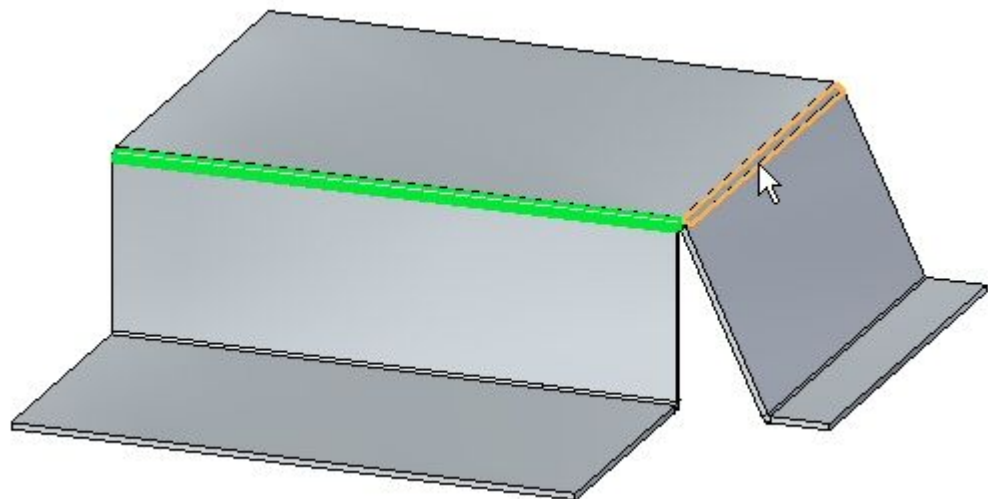
Note

The lower horizontal flange is shortened as the face is moved. The horizontal flange can be made to maintain the 90° bend angle by adjusting live rules and adding the components of the flange to the selection. Live rules are covered in another activity.

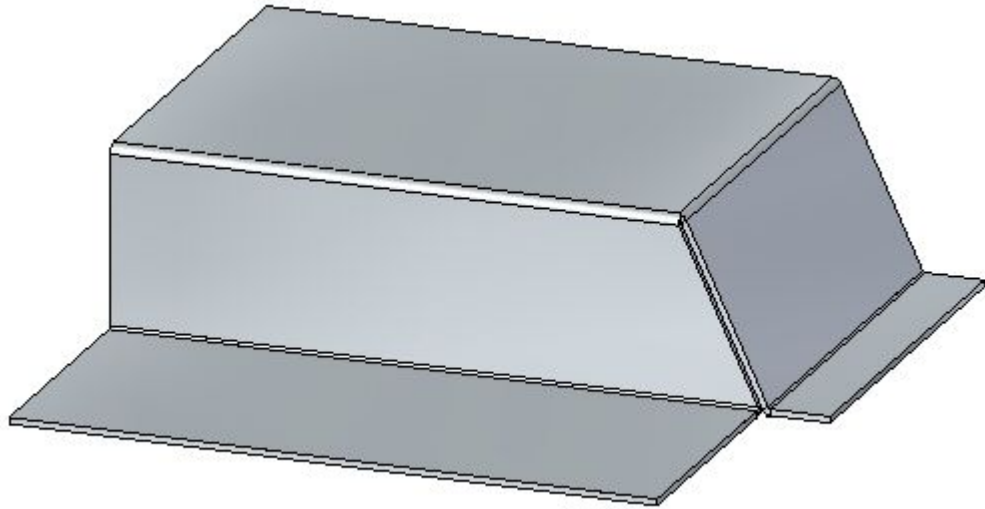
- ▶ Click the 2-Bend corner command.



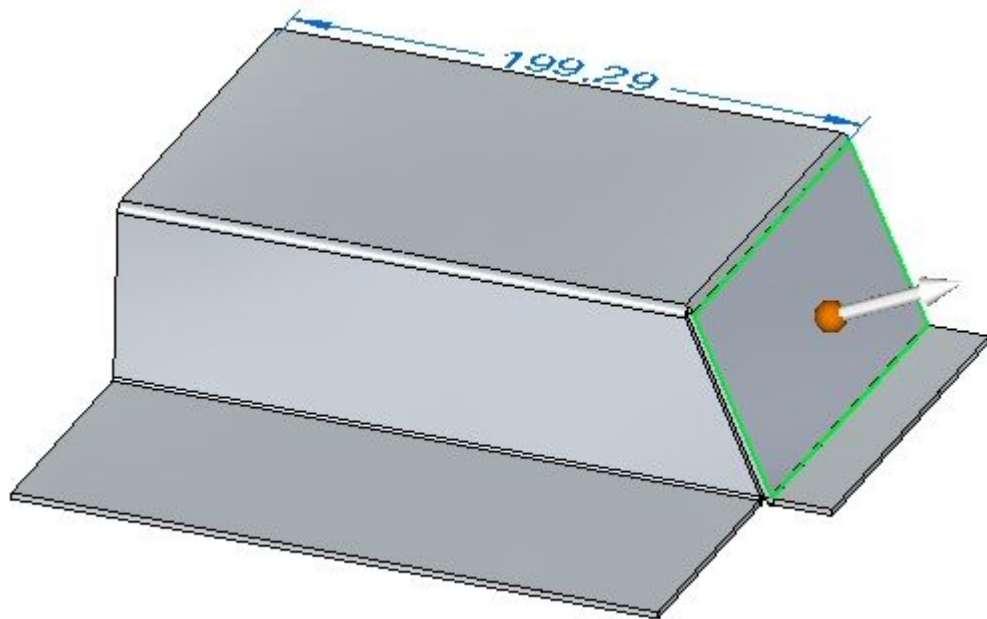
- ▶ Select the two bends shown.



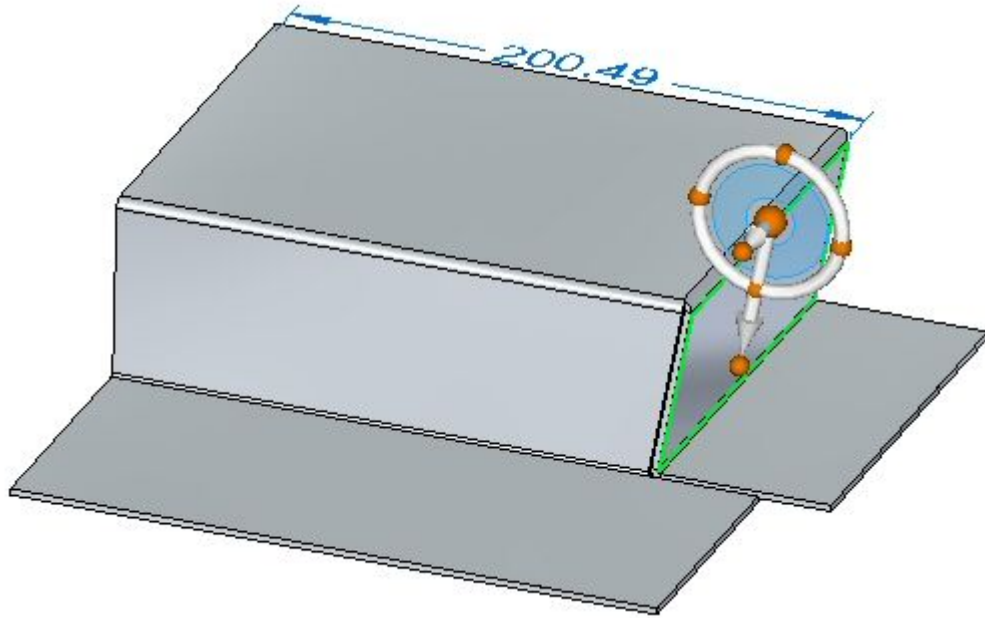
- ▶ The corner is closed.



- ▶ Select the face shown.



- Select the steering wheel torus and rotate the flange by an angle of -35° as shown.



Note

The closed corner remains closed and modifies both flanges associated with the corner.


- Close the sheet metal document without saving. Proceed to the activity summary.

Activity summary



In this activity you placed flanges and partial flanges. You edited end and corner treatments for bends. You used the closed the corners of adjacent thickness faces at the intersection of two bends. Bends were placed on the layer face and flanges were created and edited from these bends.

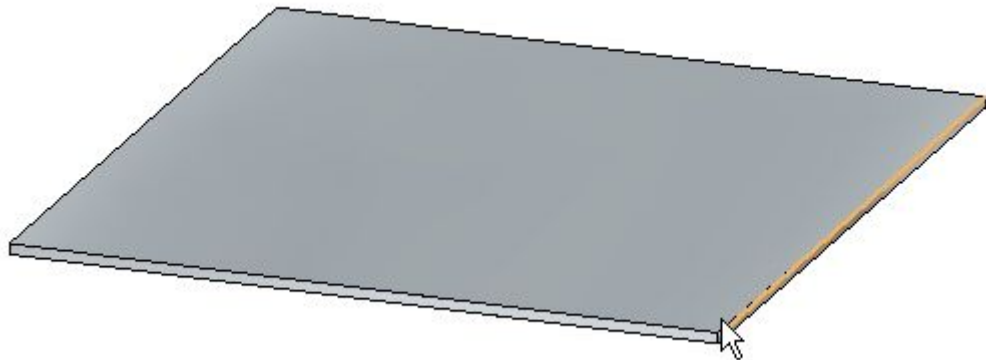
E Activity: Using the hem command in sheet metal design

Open a sheet metal file

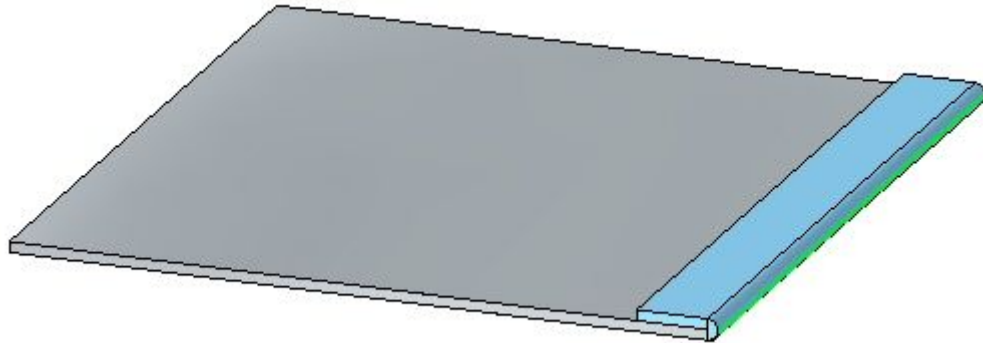
- ▶ Start Solid Edge ST4.
- ▶ Click the  **Application** button ® **Open** ® *hem_activity_1.psm*.
- ▶ Proceed to the next step.

Create a hem on a single edge

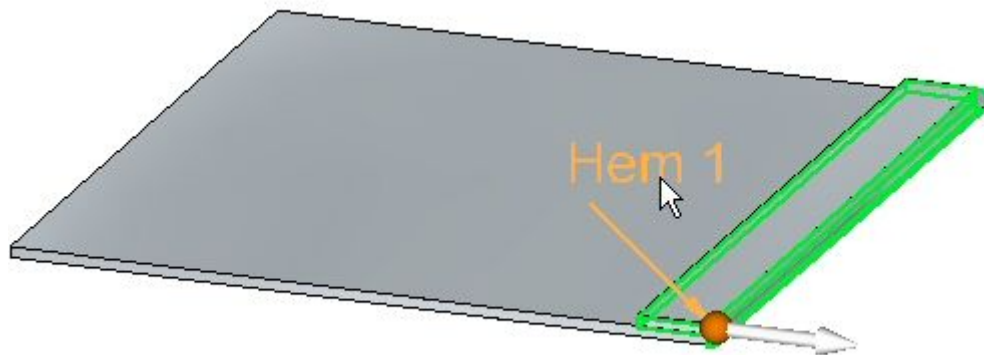
- ▶ Click the Hem command .
- ▶ Click the Hem Options button .
- ▶ Set the Hem Type to be Closed and Flange Length 1 to be 15.00 mm.
- ▶ Select the edge shown.



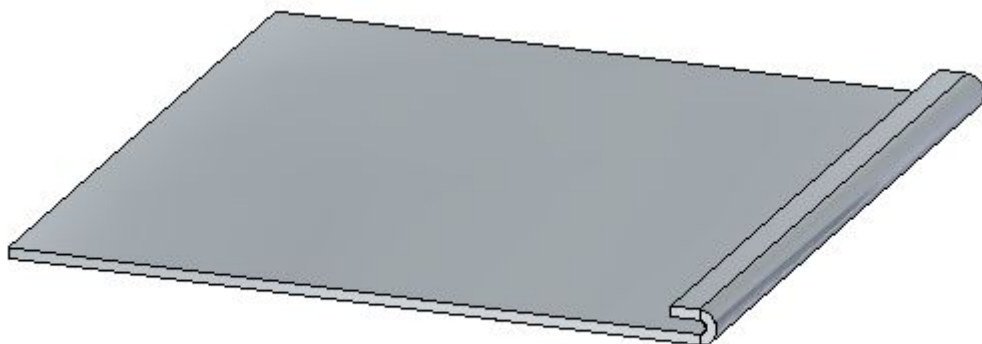
- ▶ Right-click to complete the hem shown.



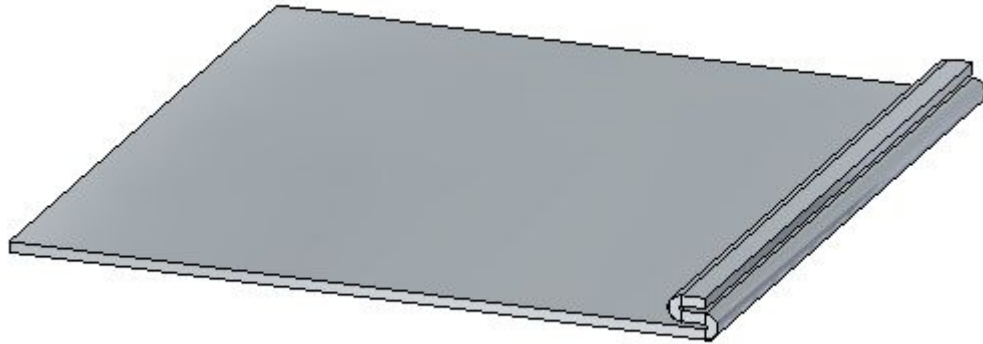
- ▶ Click the Select tool and then click the hem feature in PathFinder. Click the edit handle to change the settings on the hem.



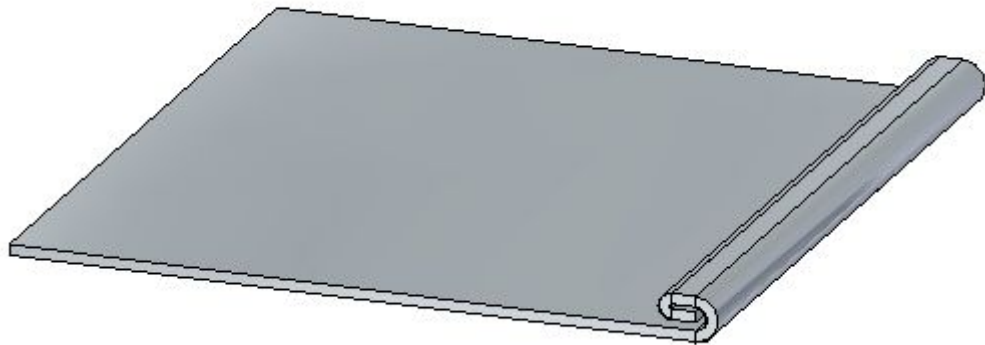
- ▶ Click the Hem Options button and set the Type to Open. Set Bend Radius 1 to 1.50 mm. Set Flange Length 1 to 6.00 mm and then click OK. The hem is modified as shown.



- ▶ Click the Select tool and then click the hem feature in PathFinder. Click the edit handle to change the settings on the hem.
- ▶ Click the Hem Options button and set the Type to S-Flange. Keep the default values for bend radii and flange lengths, then click OK. The hem is modified as shown.

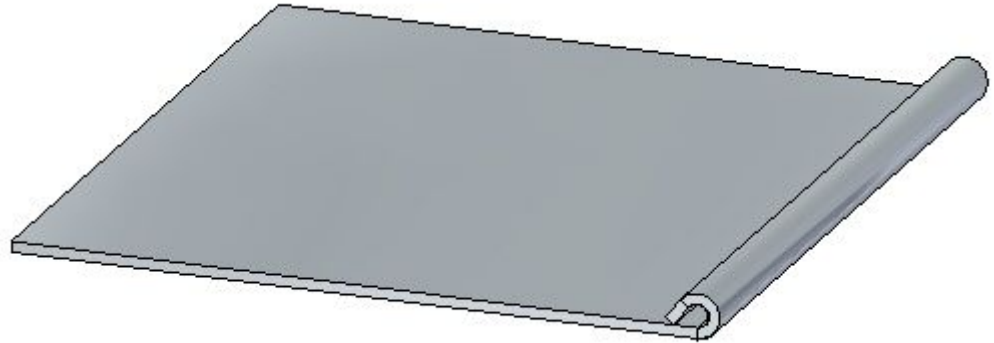


- ▶ Click the Select tool and then click the hem feature in PathFinder. Click the edit handle to change the settings on the hem.
- ▶ Click the Hem Options button and set the Type to Curl. Set Flange Length 1 to 11.25 mm and keep the existing values for the remaining lengths and radii, then click OK. The hem is modified as shown.



- ▶ Click the Select tool and then click the hem feature in PathFinder. Click the edit handle to change the settings on the hem.


- ▶ Click the Hem Options button and set the Type to Closed Loop. Keep the default values for bend radii and flange lengths, then click OK. The hem is modified as shown.



- ▶ Close the file without saving. Proceed to the next step.

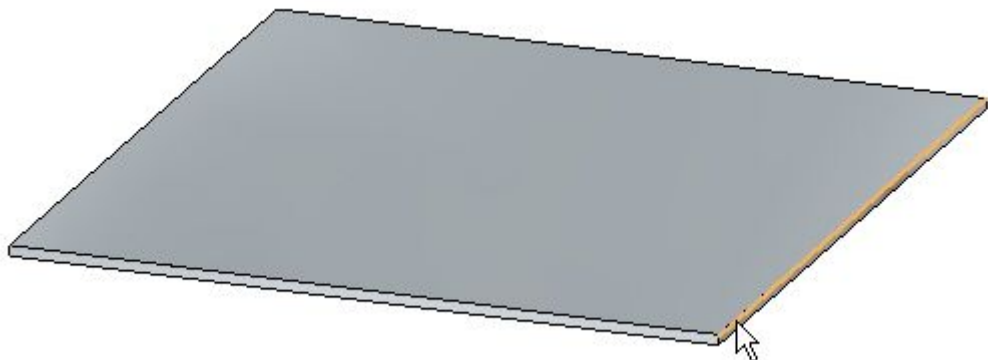
Create a hem on multiple adjacent edges

- ▶ Click the  **Application** button ® **Open** ® *hem_activity_2.psm*.

- ▶ Click the Hem command .

- ▶ Click the Hem Options button .

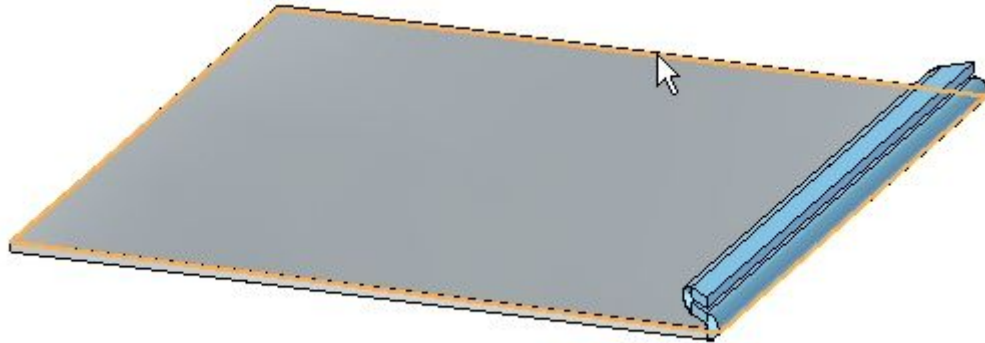
- ▶ Set Type to S-Flange.
- ▶ Check the Miter Hem option and ensure the Miter Angle is -45° .
- ▶ Select the edge shown.



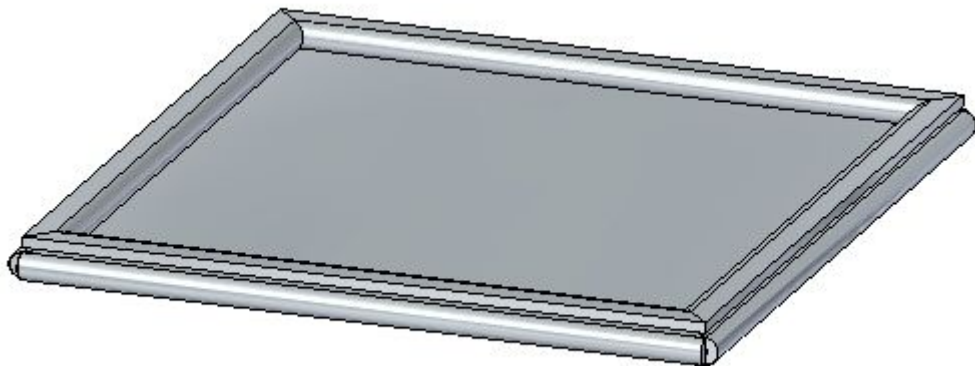
- ▶ Select the edge set shown.

Note

Since the selection type is set to chain, the edge set is defined by the perimeter of the part. If a single edge is desired, the selection type can be set to single rather than chain.



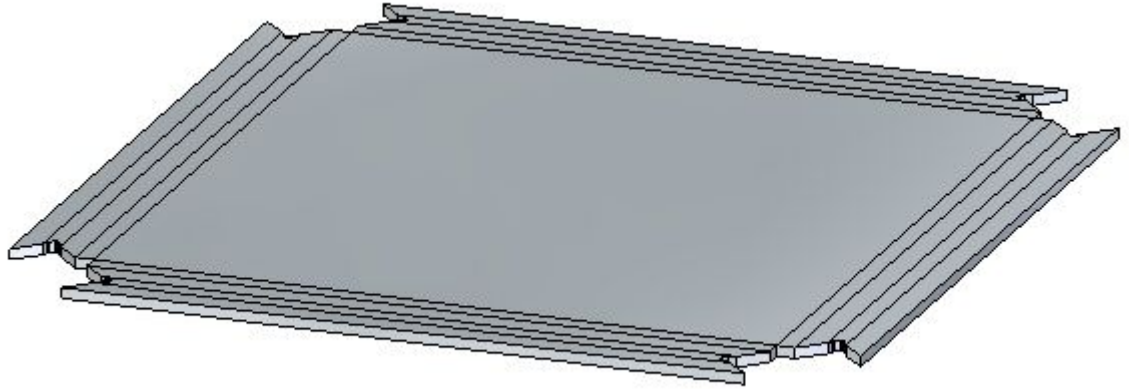
- ▶ Right-click to accept the hem.



- Shown below is a flat pattern of the sheet metal part is shown below.

Note

Flat pattern creation is covered in another activity. This is for information purposes only.


**Activity summary**

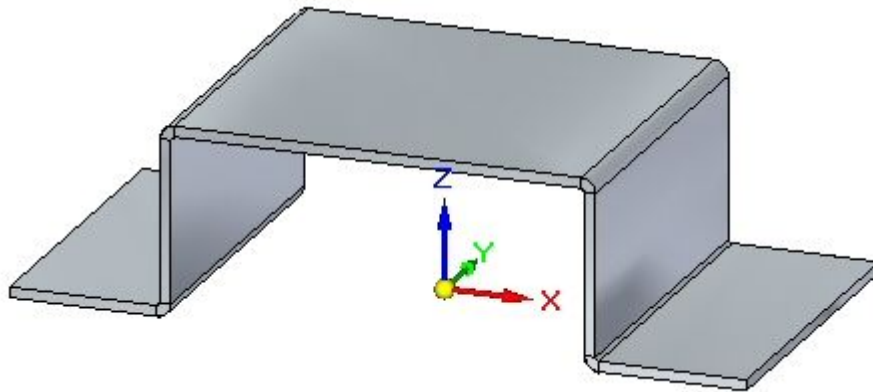
In this activity you created a variety of hems in sheet metal parts. You learned how to set the parameters to create the hems, and how to edit the values when needed.

F Activity: Using live rules in sheet metal

Open a sheet metal file

- Start Solid Edge ST4.

- Click the  **Application** button ® **Open** ® *live_rules_activity.psm*.



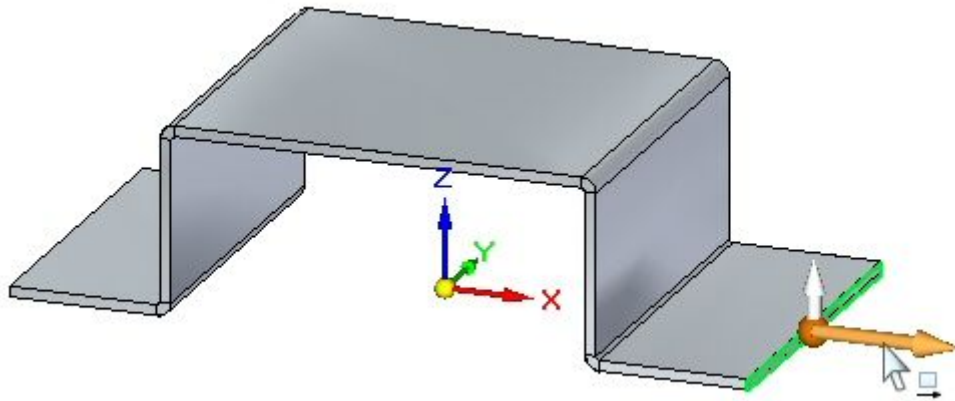
- Proceed to the next step.

Symmetry about the base reference planes

Note

In this activity, behavior of the live rules can be observed by watching the dynamic preview while pulling one of the handles. In many cases, just observing the behavior without actually making the change is what will be accomplished. Pressing the **Esc** key will exit a the command without making any modifications. If a flange is moved erroneously, use the undo command to restore it's position.

- ▶ Select the face shown and click the primary axis.



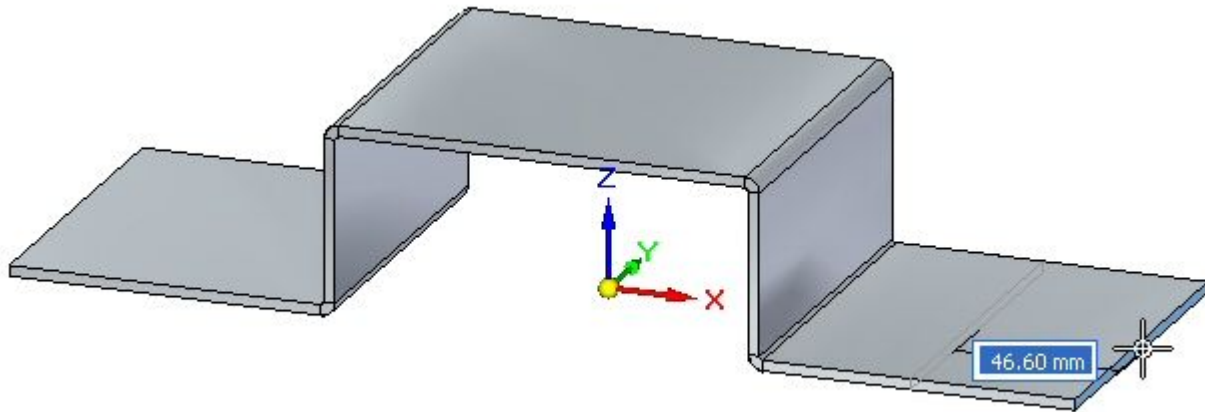
- ▶ Click the Live Rules Restore button.



Note

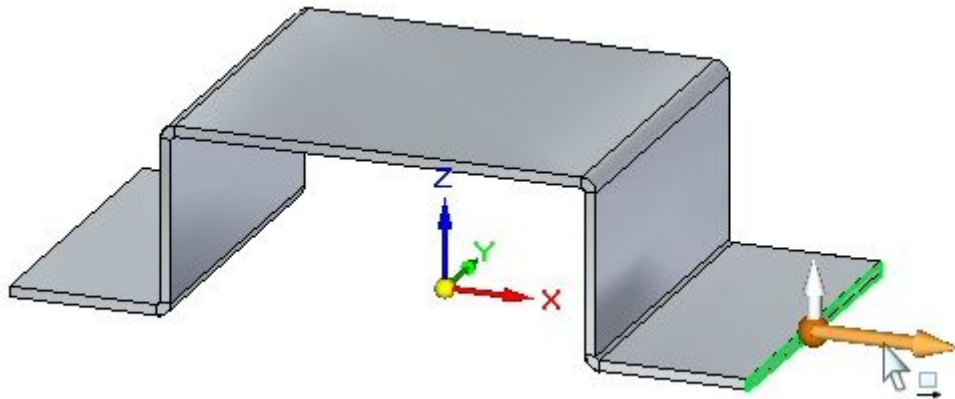
This resets the live rules to the default values.

- ▶ Drag the handle as shown and observe the behavior, then press the **Esc** key.



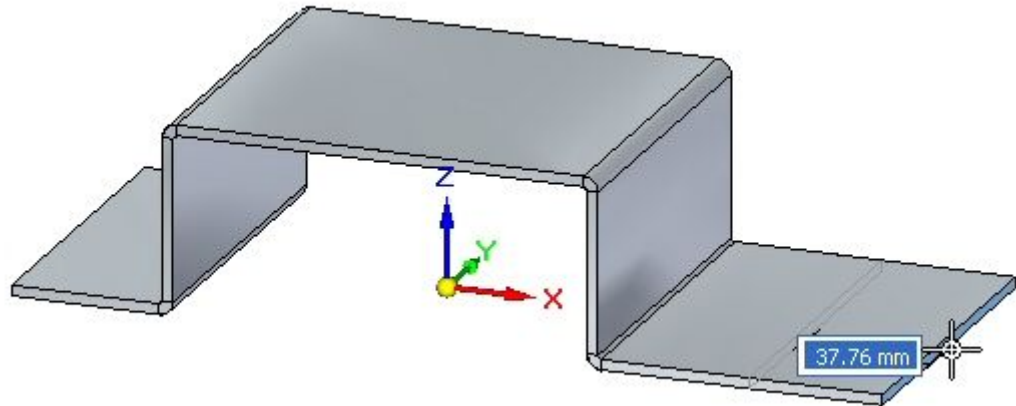
Observations:

- The flange chosen is symmetrical to the opposite flange about the right YZ base reference plane. The live rule for symmetry is controlling the behavior of the opposite flange.
- ▶ Select the face shown and click the primary axis.



- ▶ In Live Rules, turn off symmetry about the YZ base reference plane.

- ▶ Drag the handle as shown and observe the behavior, then press the **Esc** key.



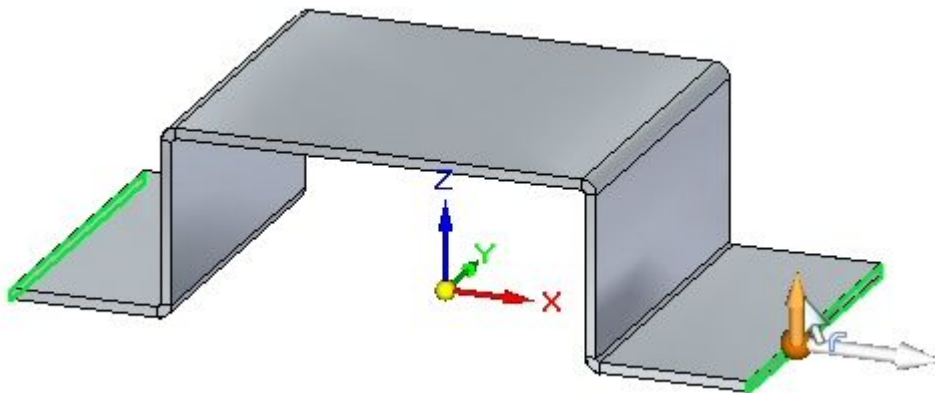
Observations:

- The live rule controlling symmetry about the YZ plane is turned off and the moving the thickness face causes the flange to be modified independently.
- ▶ Proceed to the next step.

Creating persistent rules

A permanent relationship between two faces will be created.

- ▶ Select the two faces shown and click the flange start handle.



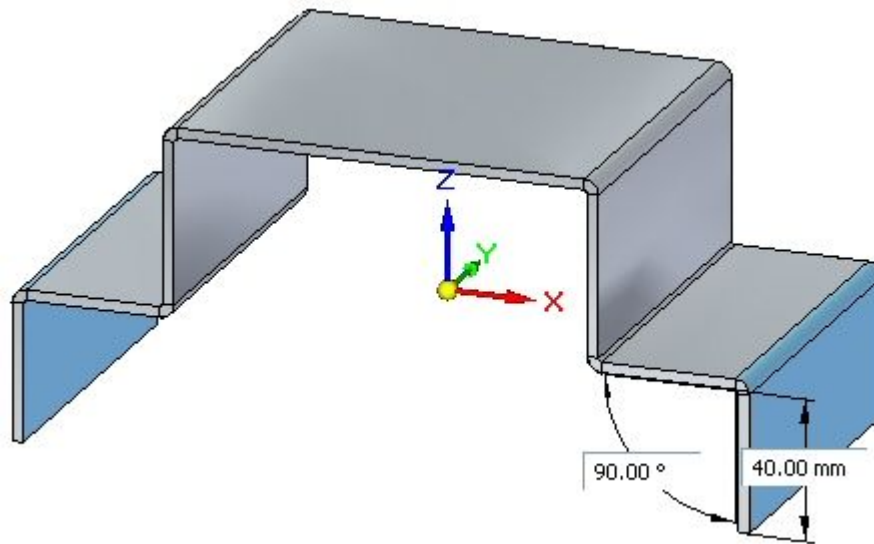
- ▶ Click the Live Rules Restore button.



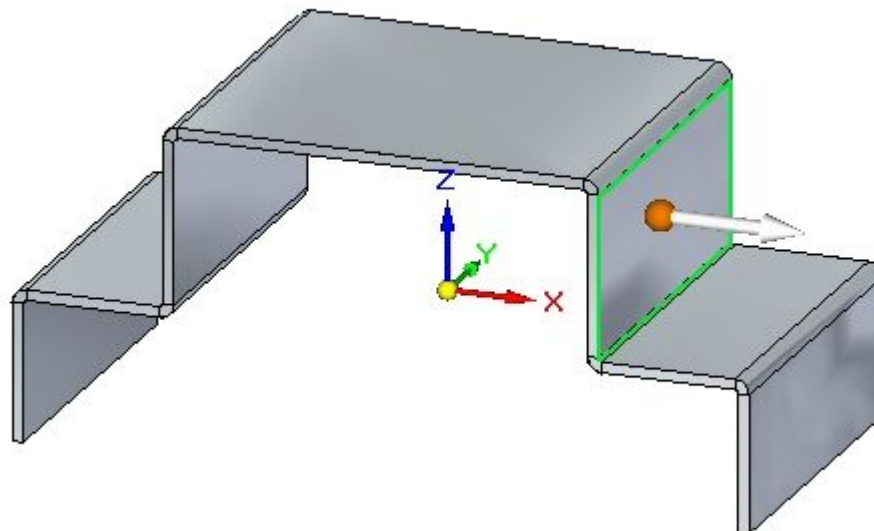
Note

This resets the live rules to the default values.

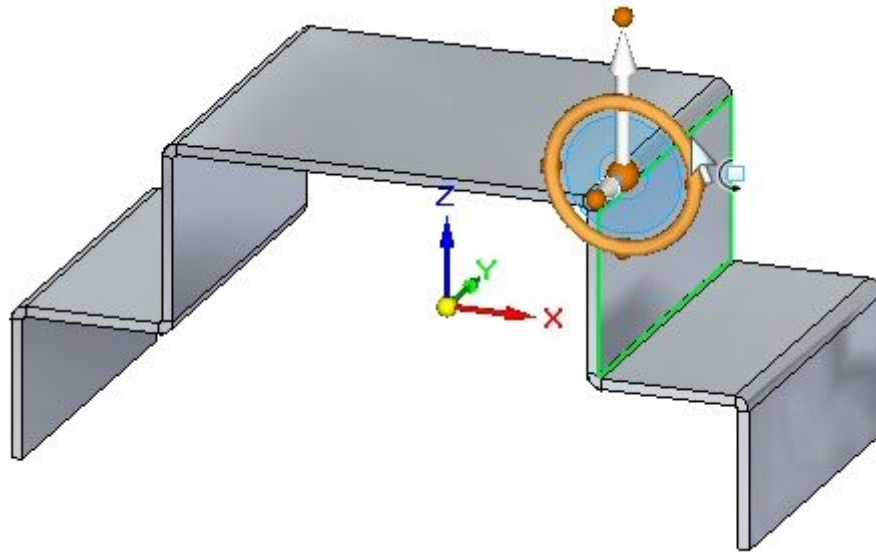
- ▶ Drag the flange start handle a distance of 40.00 mm as shown.



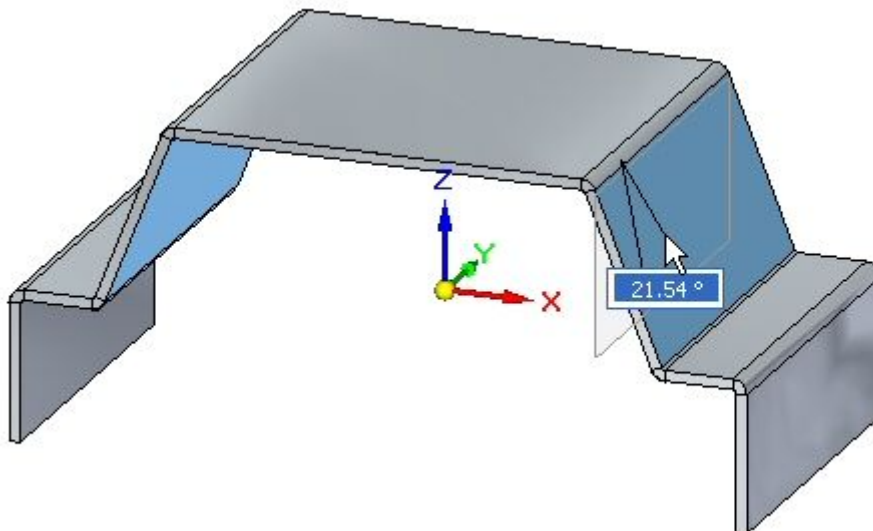
- ▶ Select the face shown.



- ▶ Position the steering wheel on the bend and click the torus to rotate the face as shown.



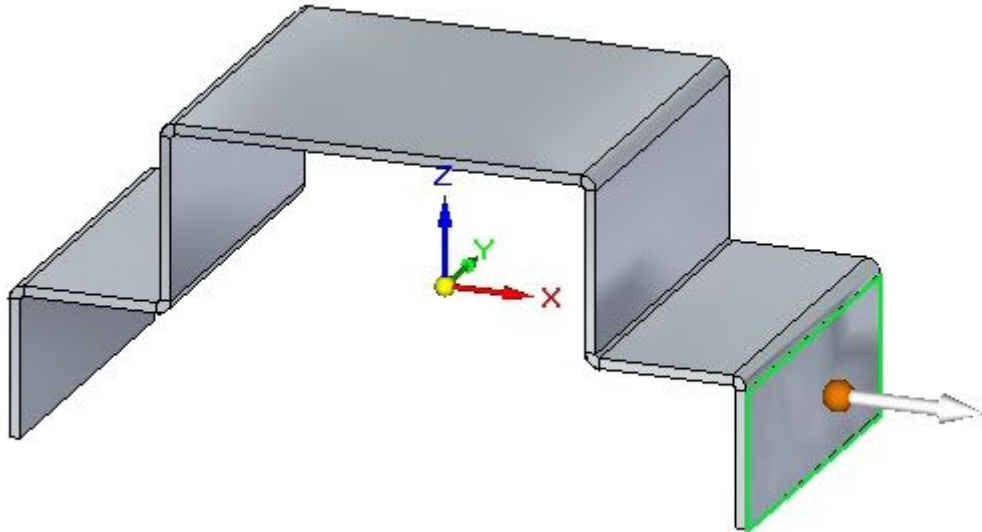
- ▶ Rotate the face and observe the behavior, then press the **Esc** key.



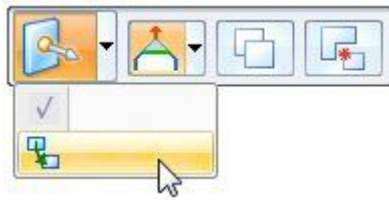
Observations:

- Symmetry about the base reference planes is causing the opposite flange to remain symmetrical.

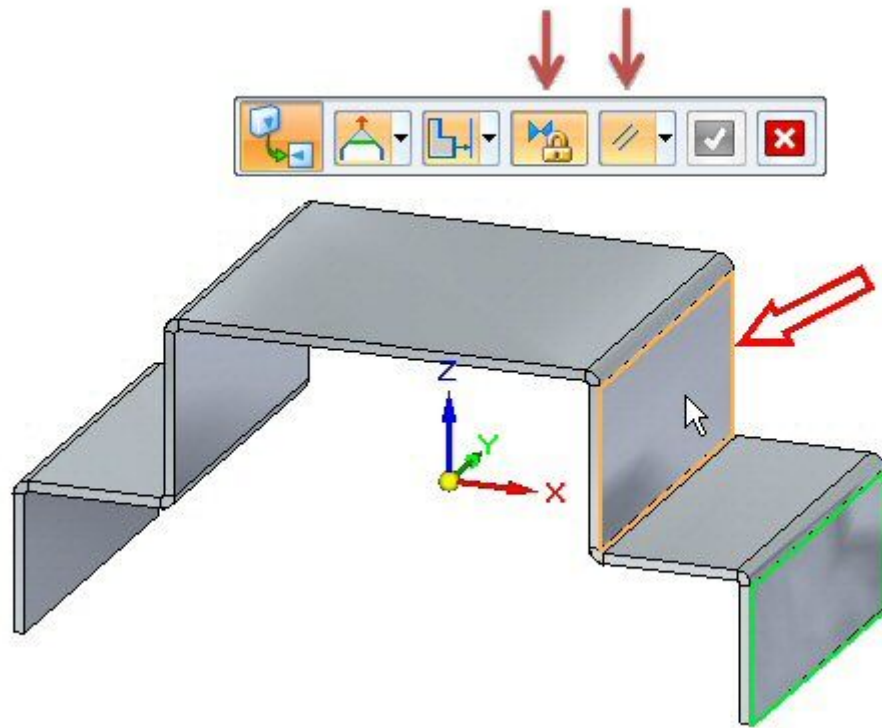
- ▶ Select the face shown.



- ▶ Click the Relate command on the command bar.



- ▶ Select Parallel Relationship and the Persist option. Select the face shown.

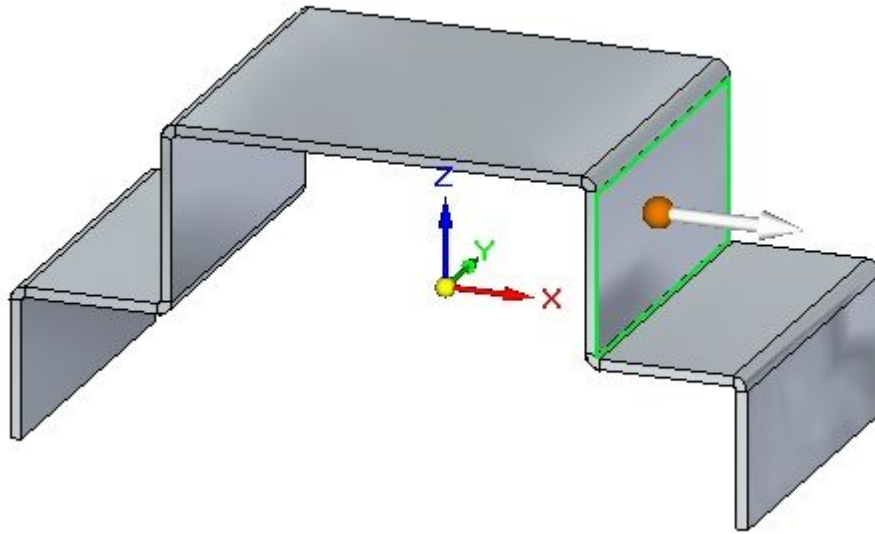


- ▶ Click the green check to accept.

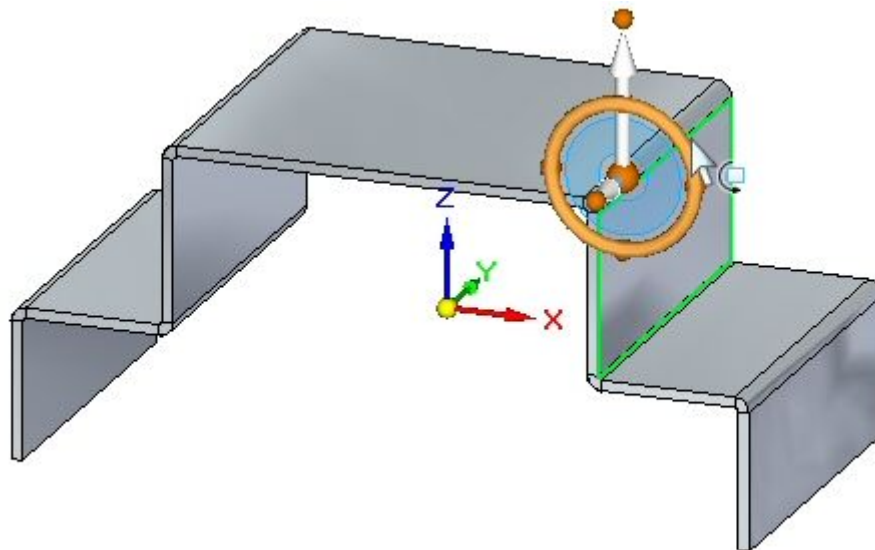
Note

A persistent relationship making the two faces parallel has been established and is displayed in PathFinder.

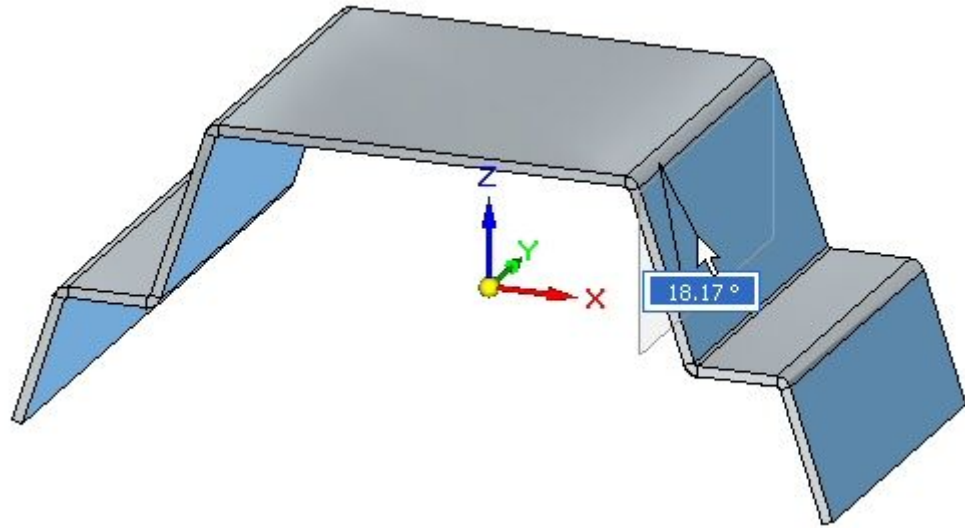
- ▶ Select the face shown.



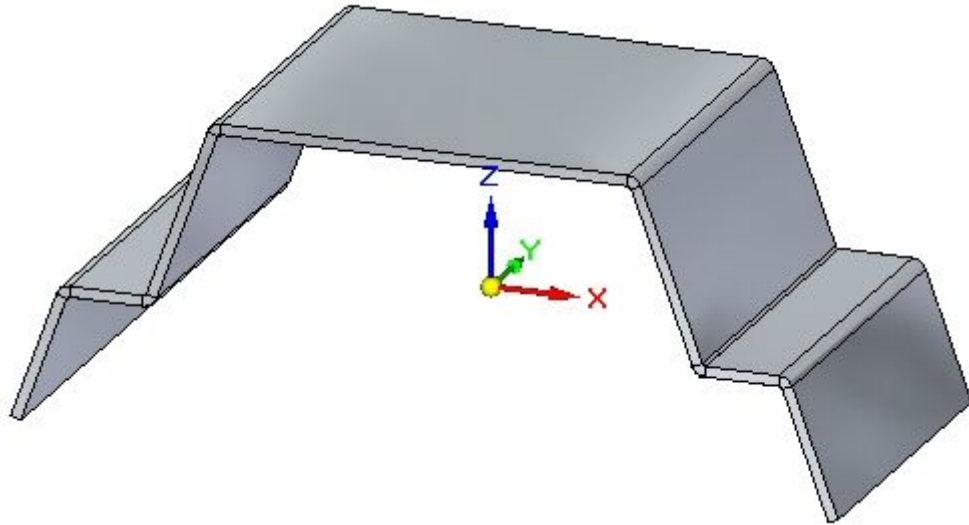
- ▶ Position the steering wheel on the bend and click the torus to rotate the face as shown.



- ▶ Rotate the face and observe the behavior.



- ▶ Rotate the face 20°.



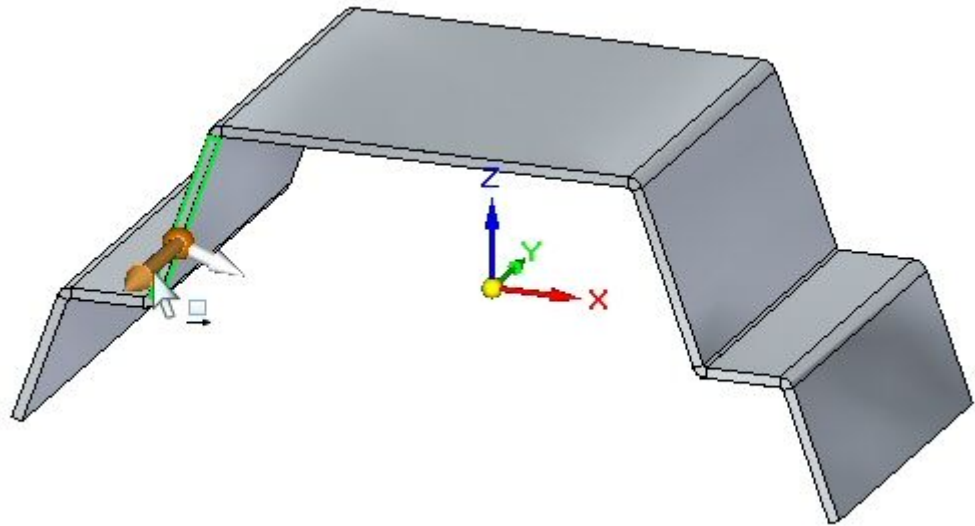
Observations:

- Symmetry about the base reference planes is causing the opposite flanges to remain symmetrical. The vertical flanges remain parallel due to the persistent relationship established previously.
- ▶ Proceed to the next step.

Thickness Chain

Thickness chain is a live rule unique to sheet metal.

- ▶ Select the thickness face shown and select the move handle.



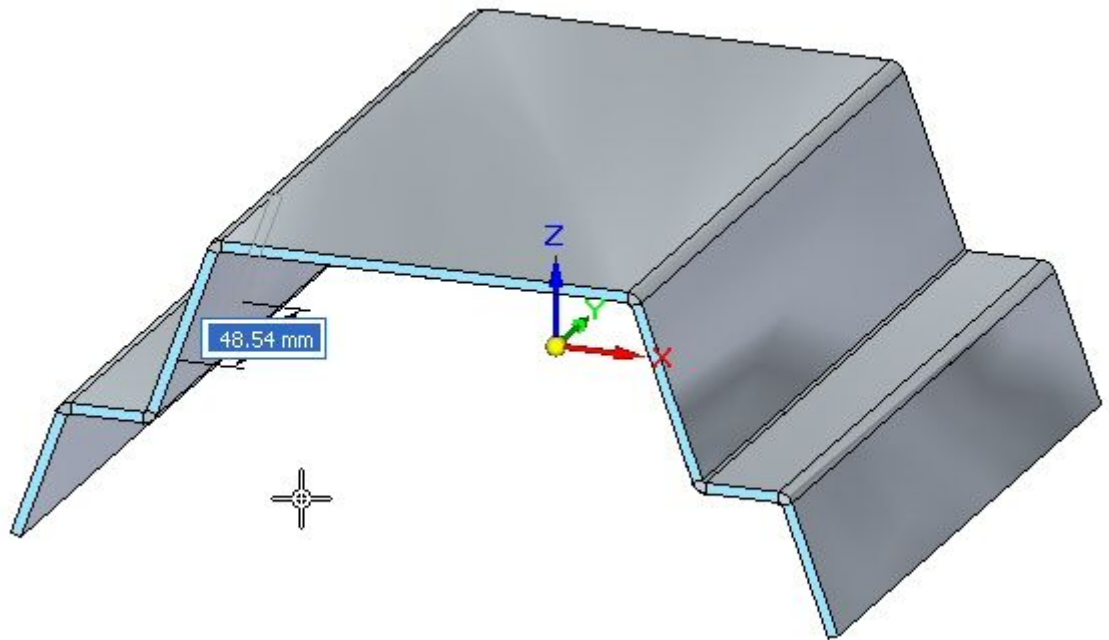
- ▶ Click the Live Rules Restore button.



Note

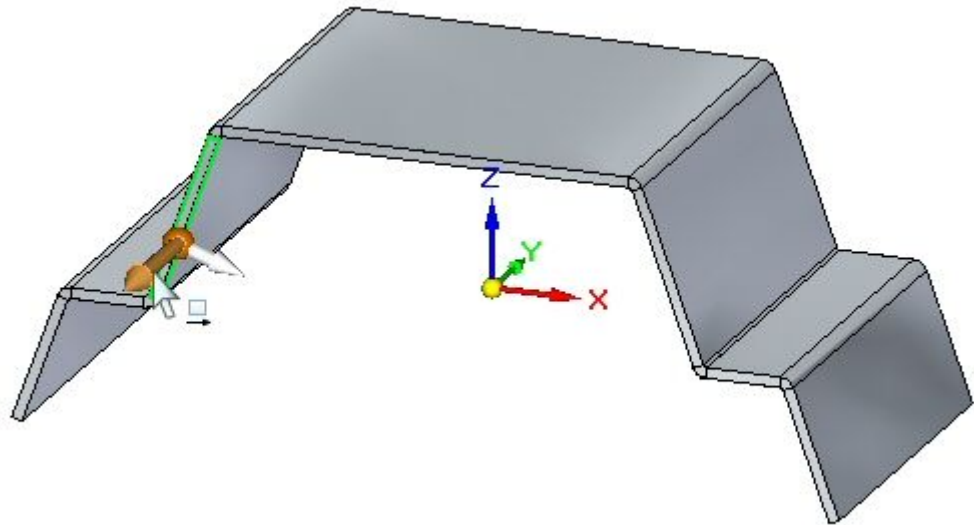
This resets the live rules to the default values.

- ▶ Drag the handle as shown and observe the behavior, then press the **Esc** key.



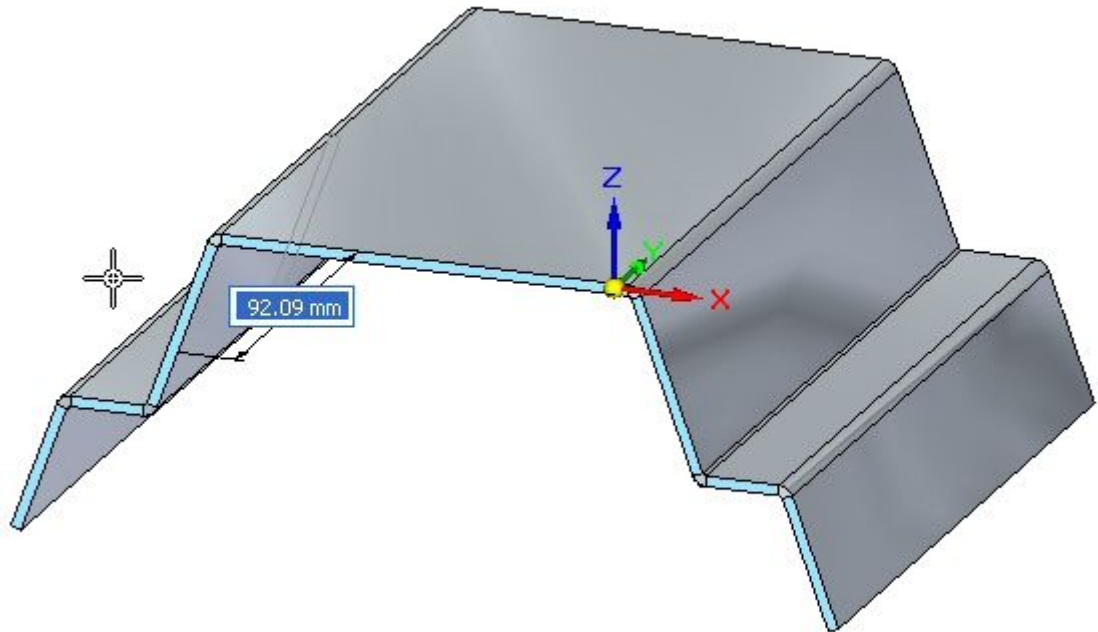
Observations:

- The live rules affecting the behavior are thickness chain, maintain coplanar faces and symmetry about the base reference planes.
- ▶ Select the thickness face shown and select the move handle.



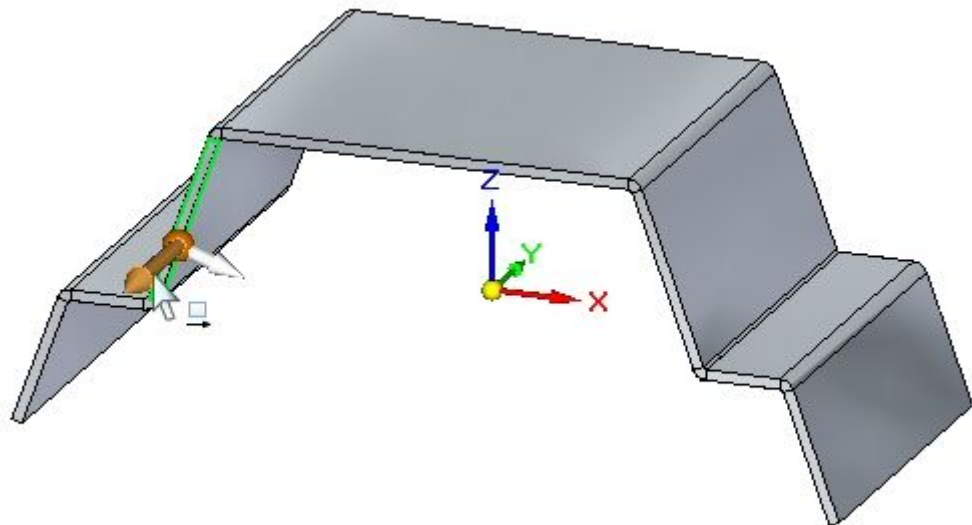
- ▶ Turn off symmetry in Live Rules.

- ▶ Drag the handle as shown and observe the behavior, then press the **Esc** key.



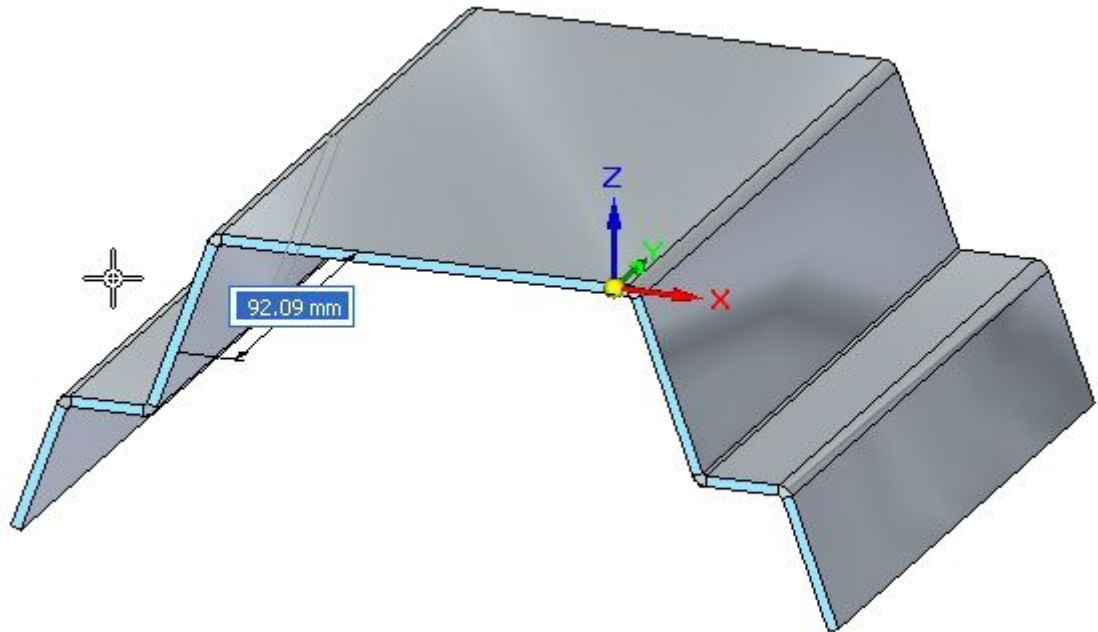
Observations:

- The live rules affecting the behavior are thickness chain, and maintain coplanar faces.
- ▶ Select the thickness face shown and select the move handle.



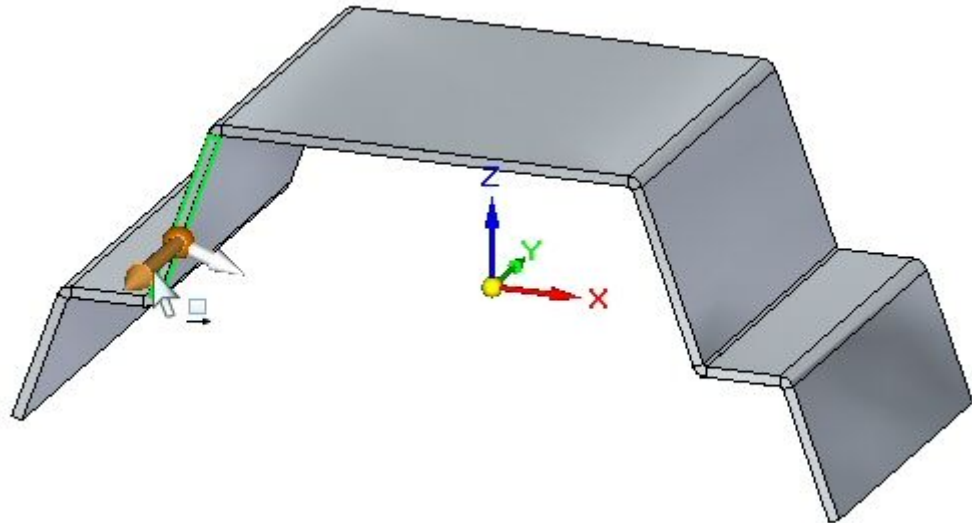
- ▶ Turn off Maintain Coplanar Faces in Live Rules.

- ▶ Drag the handle as shown and observe the behavior, then press the **Esc** key.



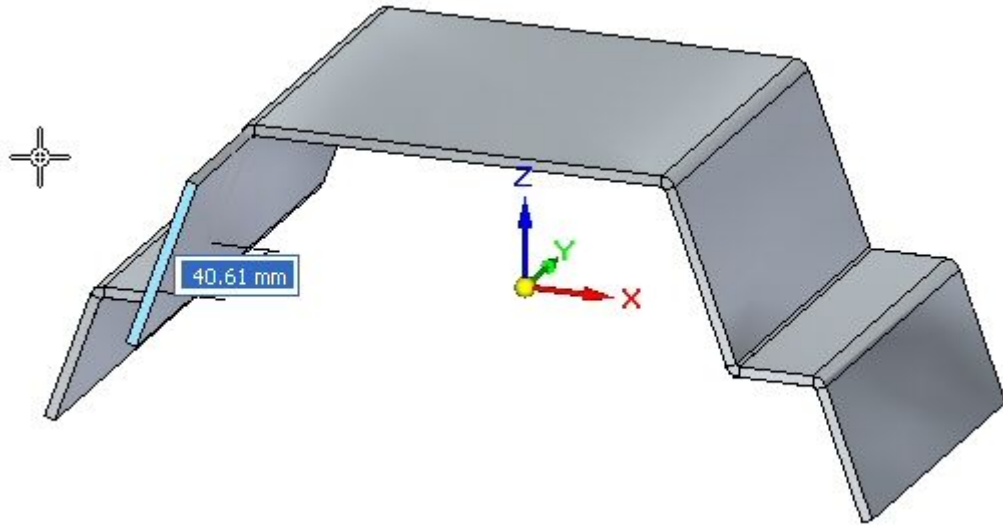
Observations:

- When a face is selected and it is part of a thickness chain, the maintain coplanar faces for that thickness chain is overridden. This will become apparent in the next step.
- ▶ Select the thickness face shown and select the move handle.



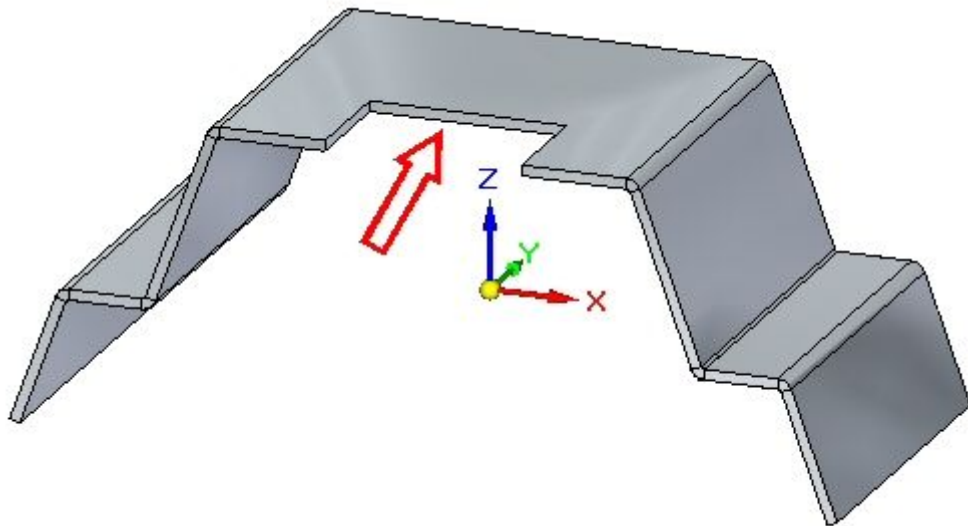
- ▶ Turn on Maintain Coplanar Faces, and turn off Thickness Chain in Live Rules.

- ▶ Drag the handle as shown and observe the behavior, then press the **Esc** key.

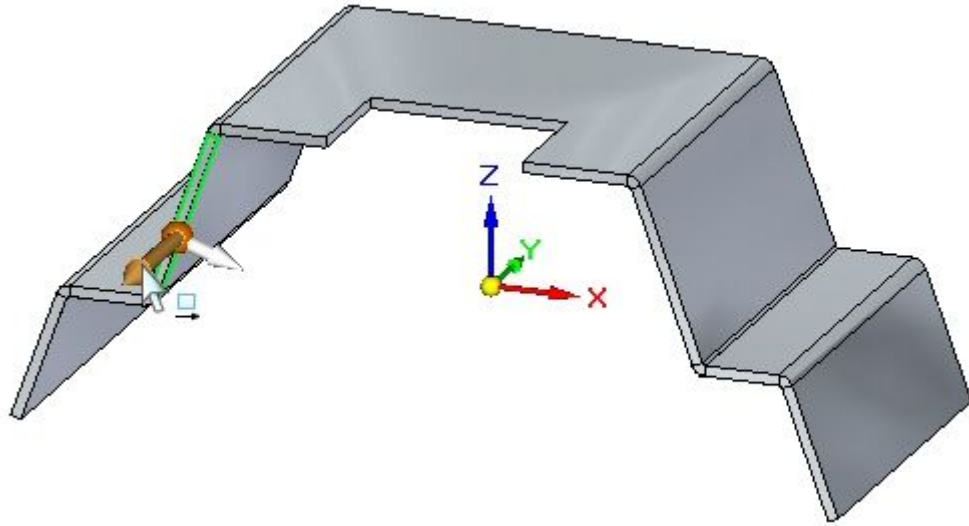


Observations:

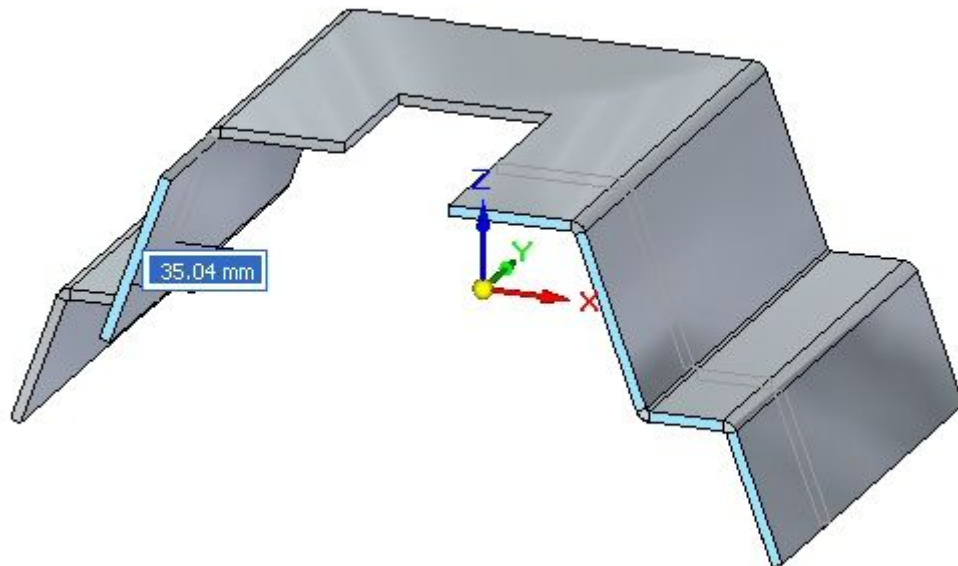
- Even though Maintain Coplanar Faces is turned on, because Thickness Chain is off, only the face selected is moved.
- ▶ Create the cutout approximately as shown.



- ▶ Select the thickness face shown and select the move handle.



- ▶ Turn on Maintain Coplanar Faces, and turn off thickness Chain in Live Rules.
- ▶ Drag the handle as shown and observe the behavior, then press the **Esc** key.

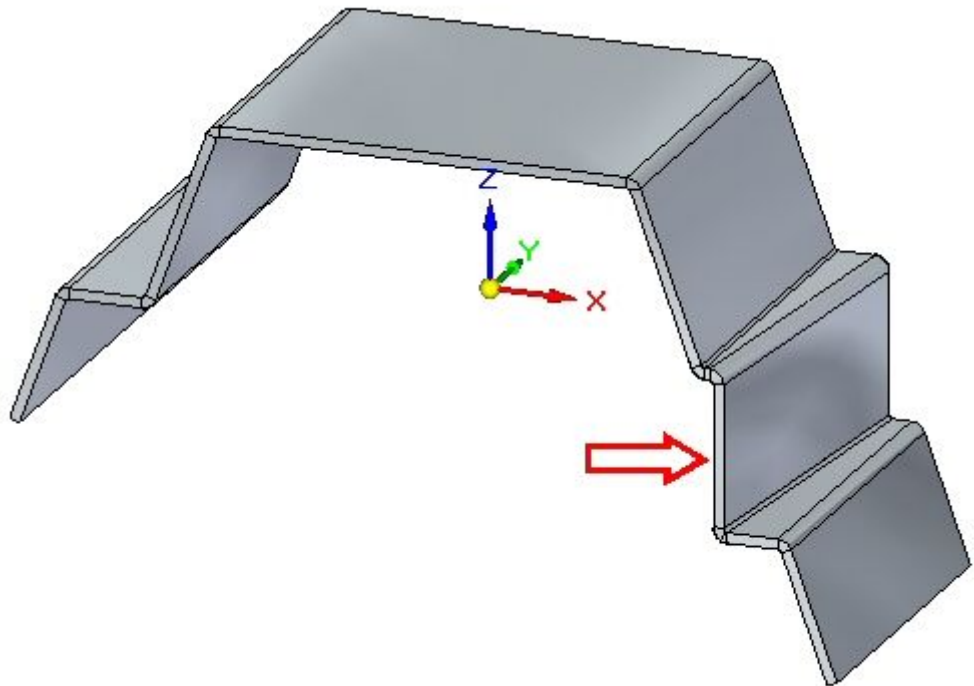


Observations:

- Even though Maintain Coplanar Faces is turned on, because the thickness chain is off, only the face selected is moved. Maintain coplanar faces does not apply to faces residing in a thickness chain.

Note

Thickness chain can also consist of non planar thickness faces connected by bends as shown in the example below. The arrow points to a face, created by a jog, which is not coplanar to the other thickness faces.



Activity summary

In this activity you explored the behavior of sheet metal geometry by creating relationships and changing live rules.

G Activity: Using the jog and break corner command in sheet metal design

Open a sheet metal file

- Start Solid Edge ST4.



- Click the **Application** button ® **Open** ® *jog_activity.psm*.

Note


This sheet metal file was created in the context of an assembly. An interpart-copy contains geometry from a part file that will be used to define the extents of the sheet metal part being created. The geometry has rounded edges with a 2.0 mm radius. Knowing this, the proper bend radius can be established.

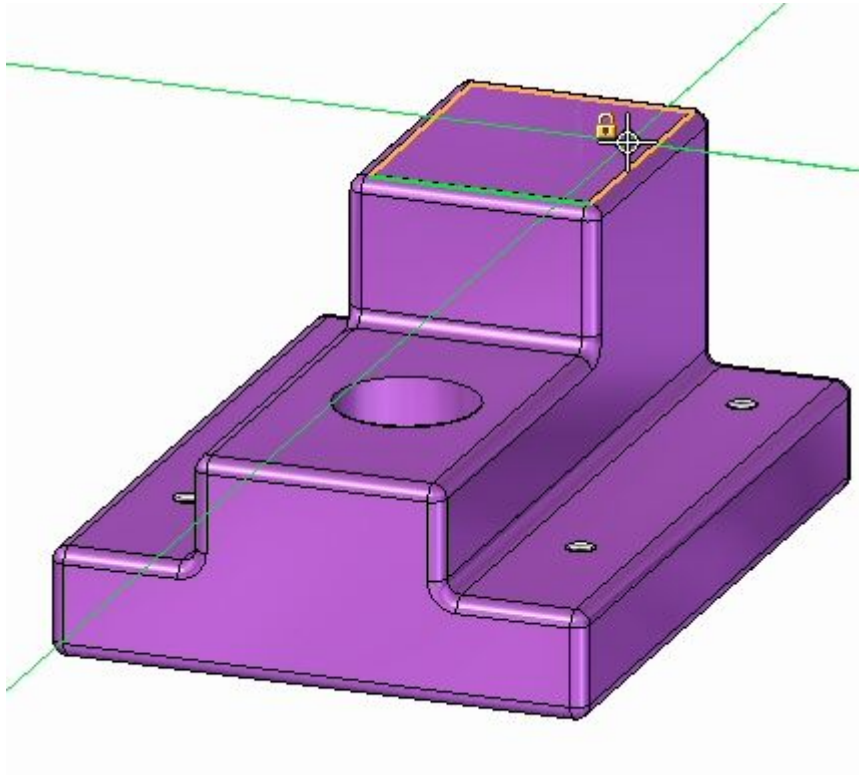
- Proceed to the next step.

Draw the sketch and create the base feature

Note

This sheet metal file was created in the context of an assembly and inter-part geometry from a part file was imported into the file. This geometry will be used to define the extents of the sheet metal that will cover the part. It is visible in PathFinder and the display of the geometry can be toggled on or off there.

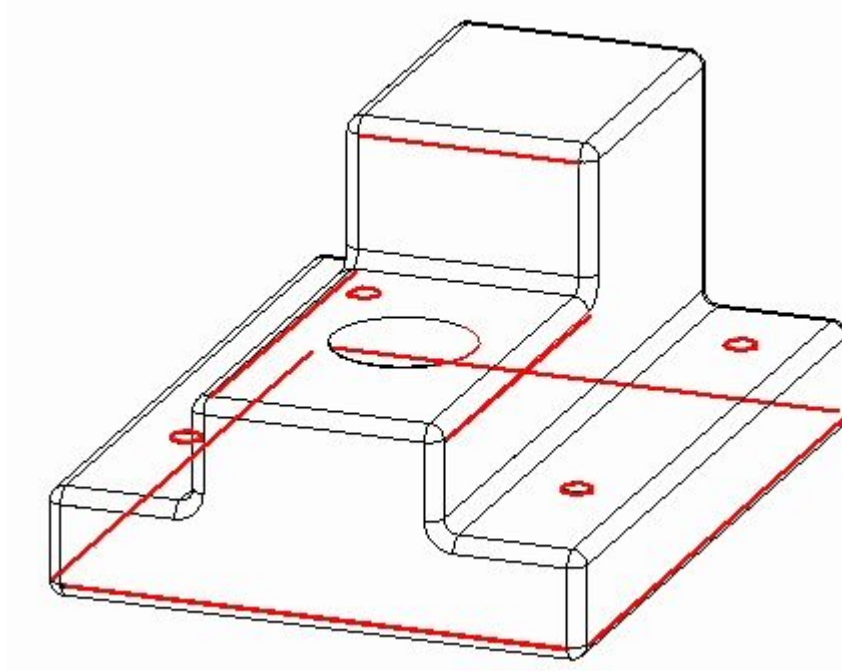
- ▶ Select the Project to Sketch command .
- ▶ Lock the sketch plane to the top most face on the part.



- ▶ Use the command to include the following geometry in the sketch:

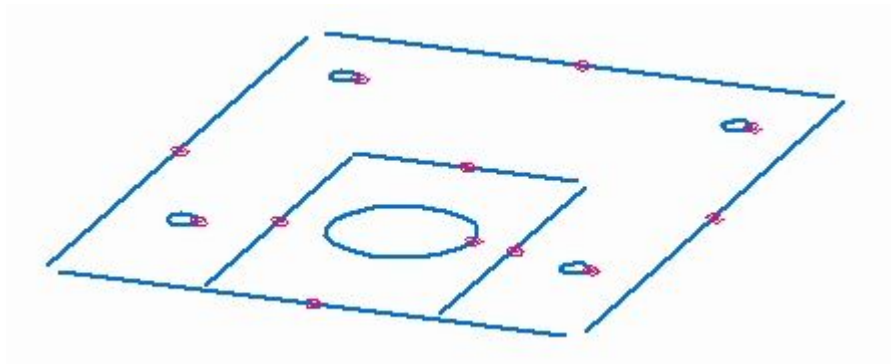
Activity: Using the jog and break corner command in sheet metal design


- The outer edges around the base. Only the straight edges need to be included.



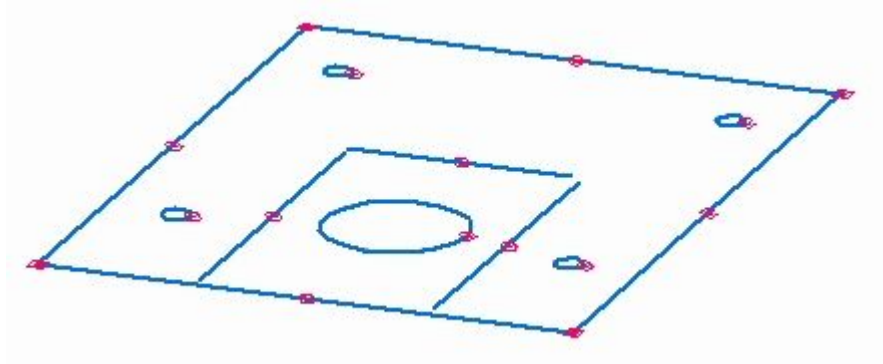
- The 5 holes.

The sketch appears as shown.

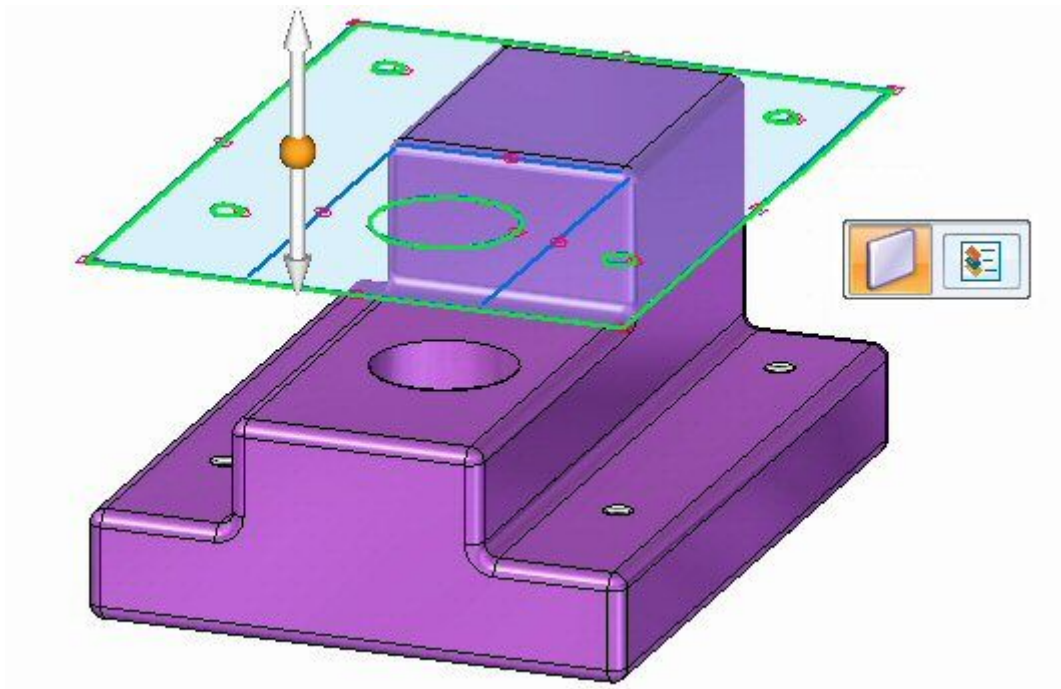


- ▶ Click the Trim Corner command  and trim the outer lines so that they intersect and form a closed region.

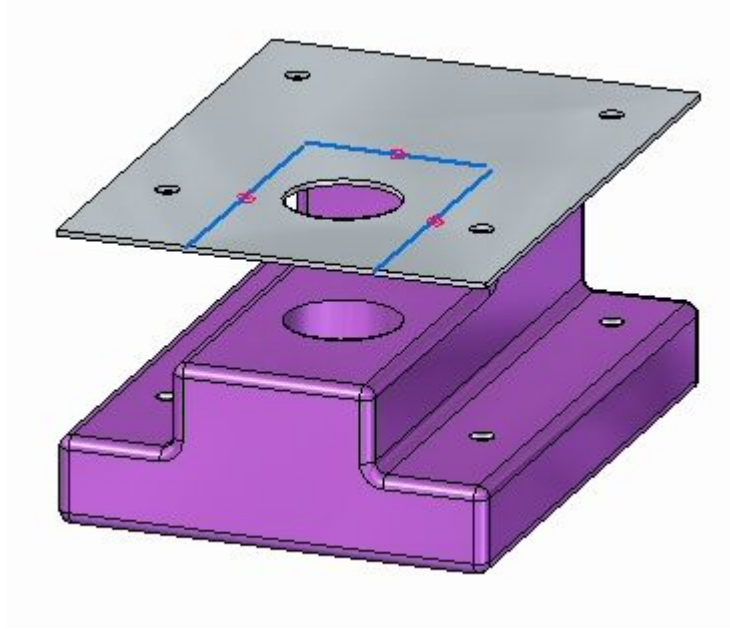
The sketch appears as shown.



- ▶ Select the region shown.



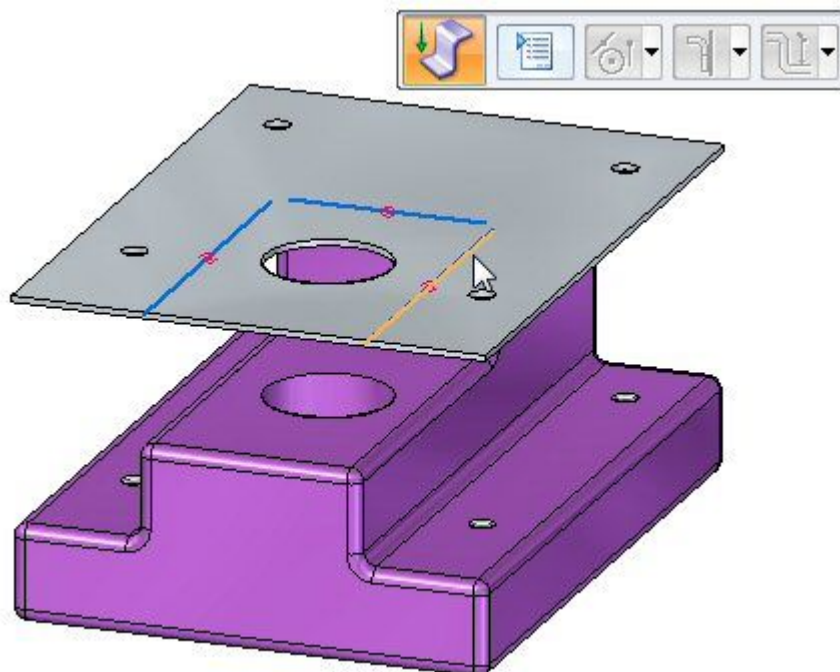
- ▶ Click the top arrow to create the base feature above the part geometry as shown.



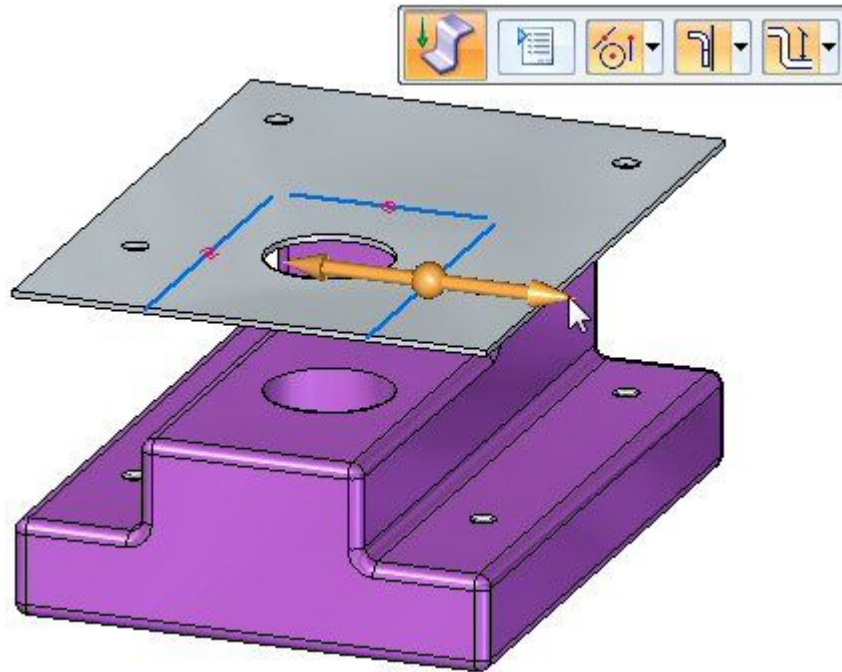
- ▶ Proceed to the next step.

Create a jog

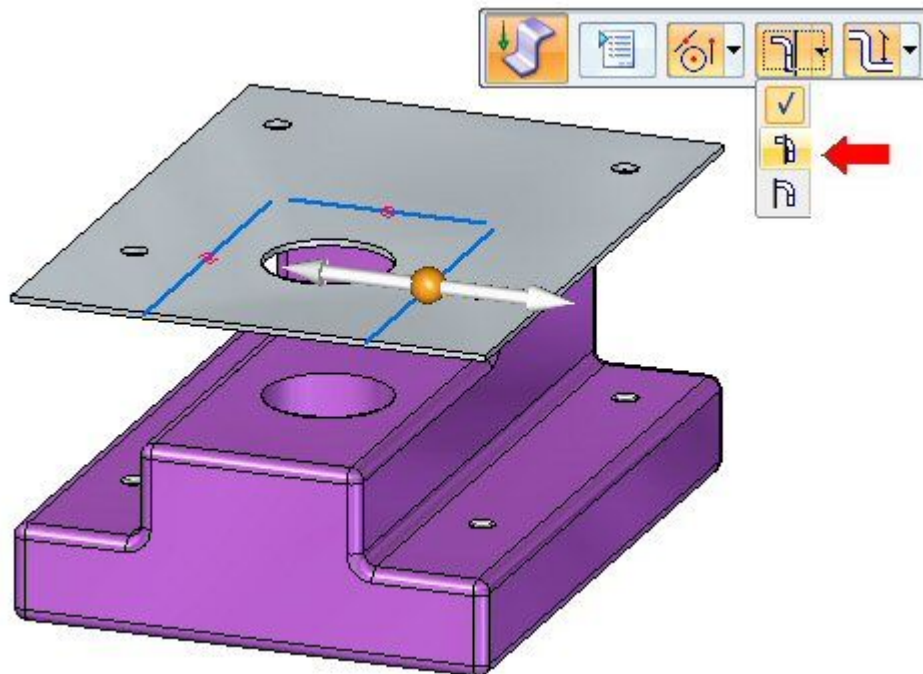
- ▶ Click the Jog command  and select the line shown.



- ▶ Select the direction arrow as shown.

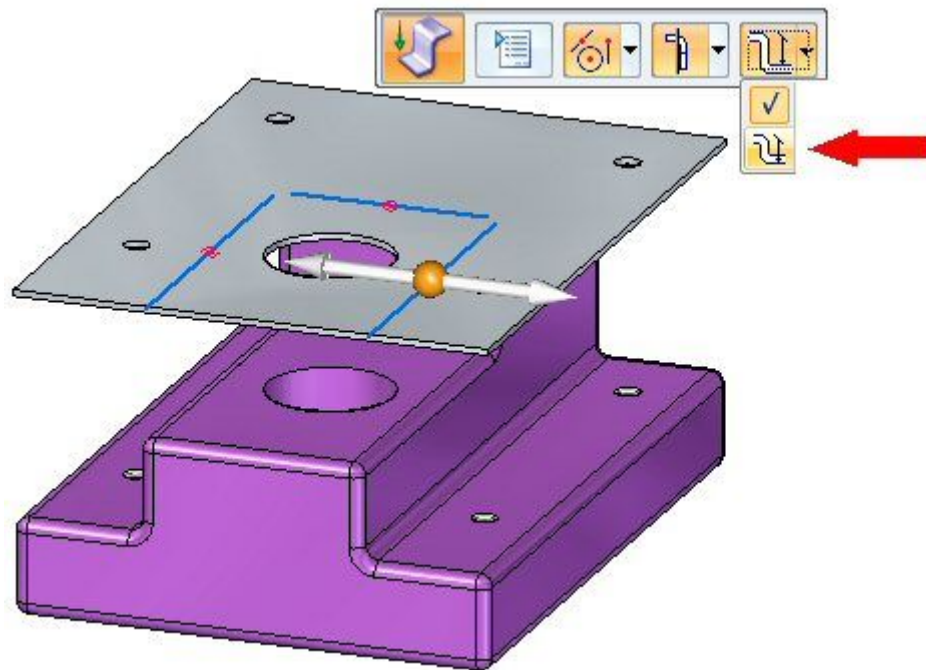


- ▶ Select the Material Outside option on command bar.

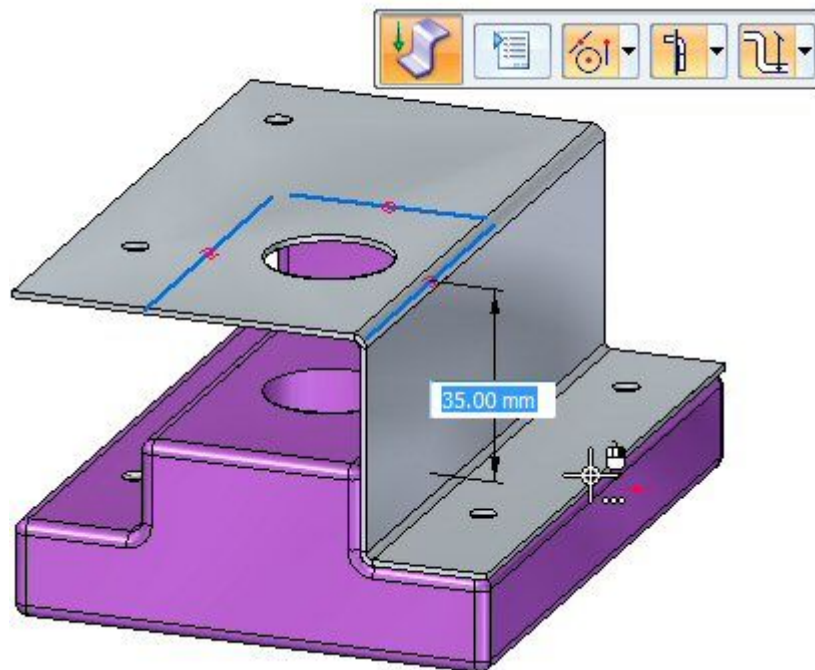


Activity: Using the jog and break corner command in sheet metal design

- ▶ Select the Dimension to Die option on command bar.



- ▶ Drag the jog down by clicking on a keypoint on the lower face of the part.

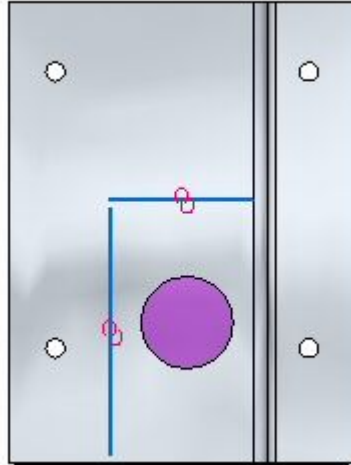


The jog is created.

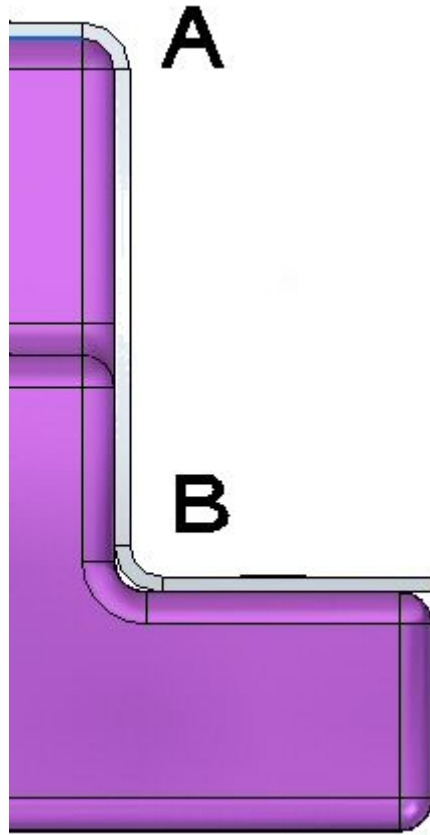
- ▶ Proceed to the next step.

Modifying the bend radius

- Observe the following about the jog just created:
 - Press **Ctrl+T** to rotate the view to a top view. Observe that the holes are in the part are still aligned with the holes in the sheet metal.

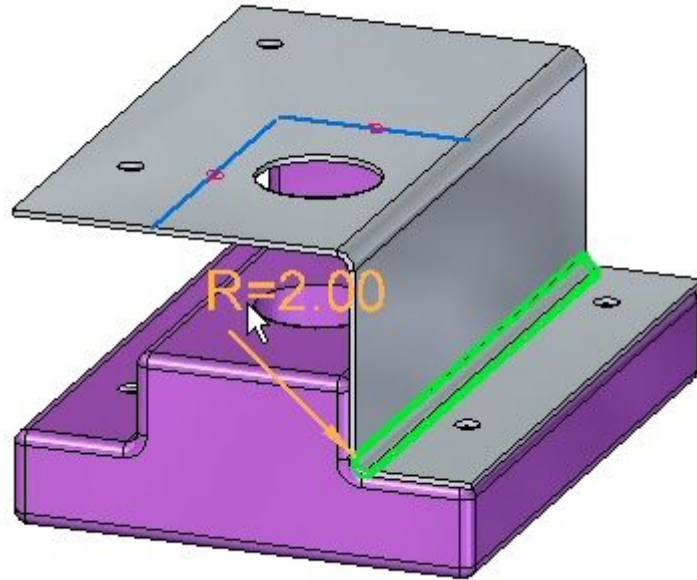


- Press **Ctrl+F** to rotate the view to a front view.

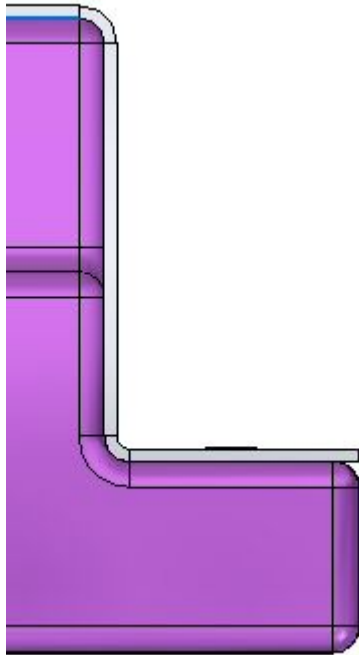


At (A) in the figure above, the sheet metal fits exactly over the 2.0 mm round on the part because the bend radius is 2.0 mm. The material thickness is 1.0 mm. The outer radius is the sum of these two values and is 3.0 mm. On the bottom bend, (B) the sheet metal does not fit exactly because of this. In the next step the bend radius will be modified so that the sheet metal part is correctly positioned at (B).

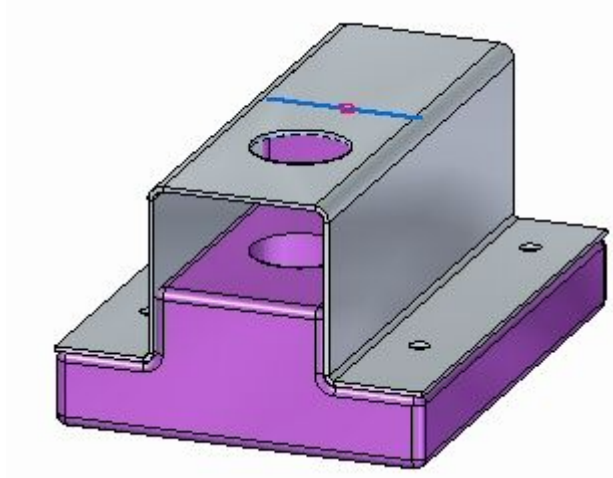
- ▶ Press **Ctrl+J** to rotate the view. Select the bend shown, then click the text. This will allow edits to the bend radius for that bend.



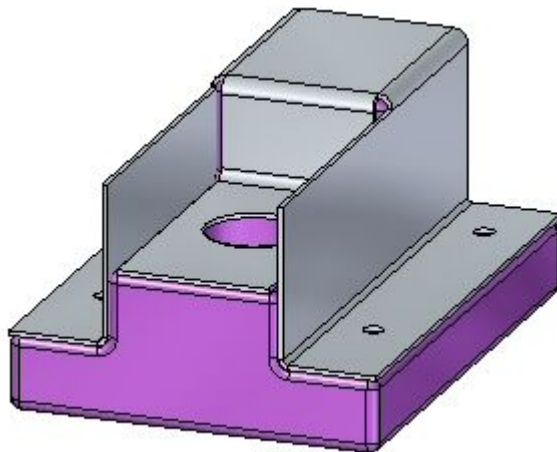
- ▶ Change the Bend Radius to 1.0 mm.
- ▶ Press **Ctrl+F** to rotate the view to a front view. Observe the bend.



- ▶ Use the same steps to place a jog on the opposite side of the part and modify the bend radius on the lower bend. The result is as shown below.




- ▶ Place the last jog using the remaining line in the sketch. Modify the bend radius on the lower bend. The result is as shown below.

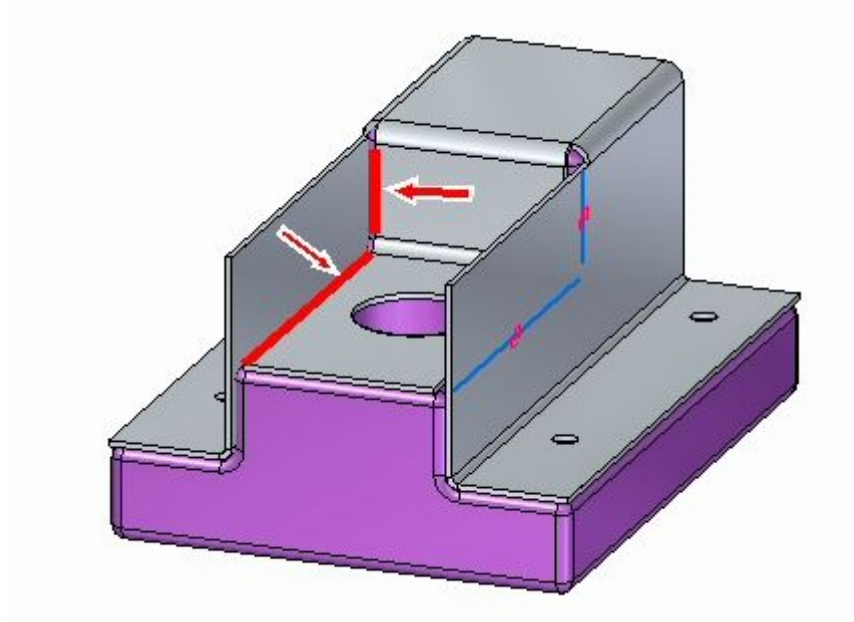


- ▶ Proceed to the next step.

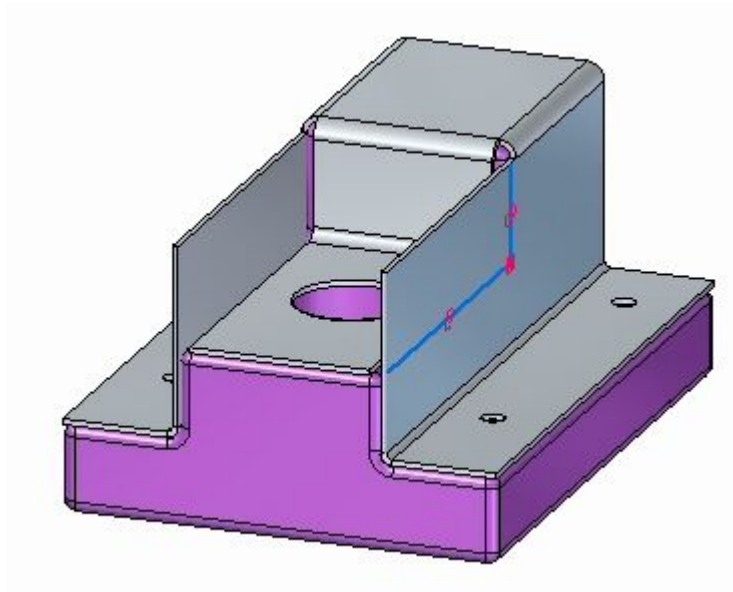
Using the cut command to trim unwanted edges.

In this step the cut command will be used to trim away unwanted parts of the vertical flanges.

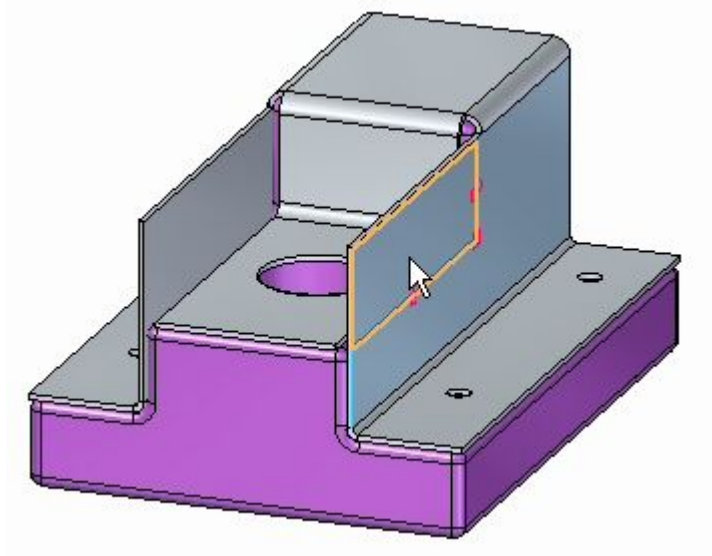
- ▶ Select the Project to Sketch command .
- ▶ Lock the sketch plane to the outer face of the vertical flange and include the following geometry in the sketch:



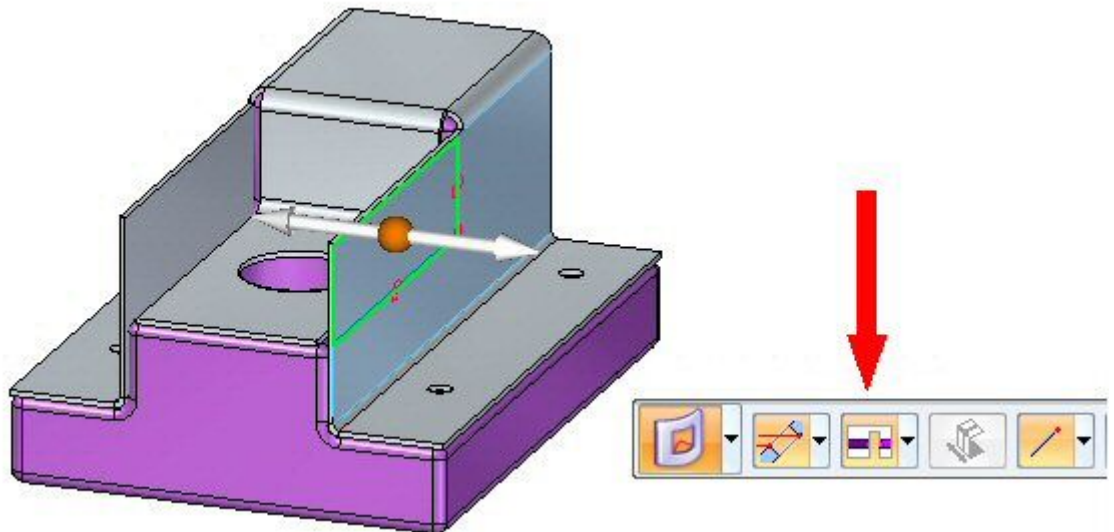
- ▶ Click the Trim Corner command  and trim the lines so that they intersect and form a region.



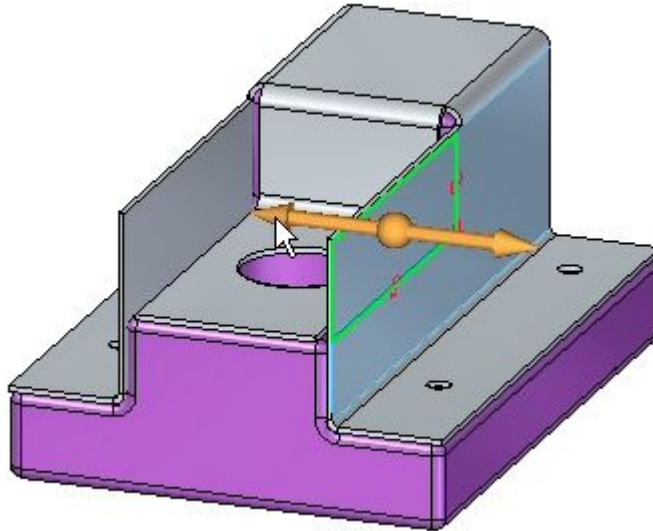
- ▶ Select the region shown.



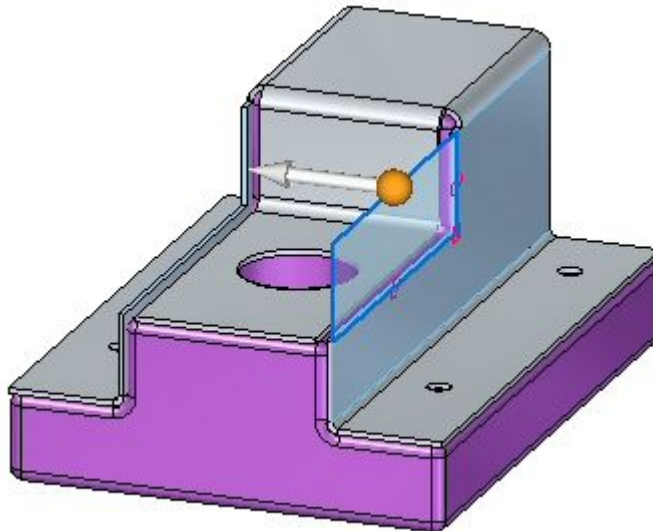
- ▶ Selecting the region initiates the cut command. Set the extent parameter to through all.



- ▶ Click the left arrow.



- ▶ The flanges are trimmed.




- ▶ Proceed to the next step.

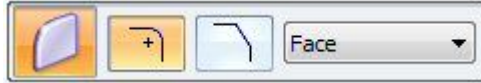
Using the cut command to trim unwanted edges.

In this step the break corner command will be used to round the sheet metal corners.

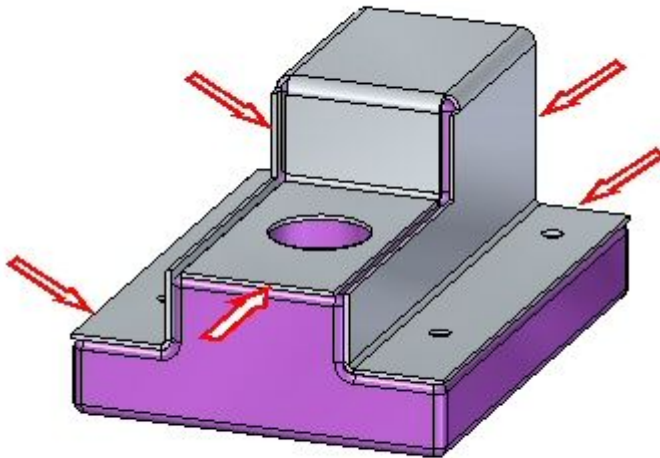
Note

The break corner command can be used to either round or chamfer a corner. In this activity a 2.0 mm round will be placed on each sheet metal corner.

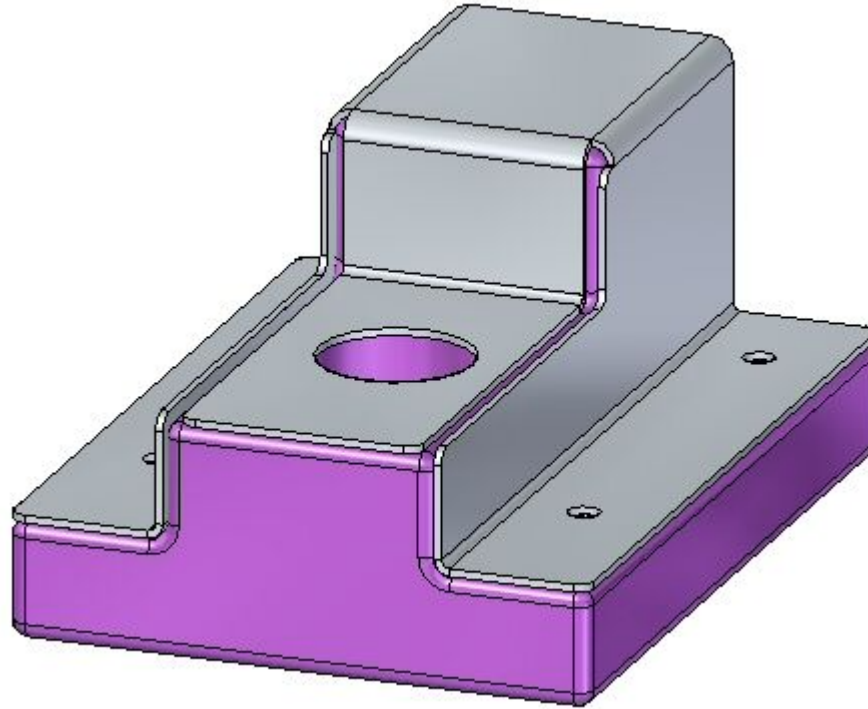
- ▶ Click the Break Corner command .
- ▶ On the command bar, set the Corner Type to Radius and the Selection Type to Face.



- ▶ Select the 5 faces shown and enter a radius of 2.0 mm.



- The results are shown. This completes the activity.




Activity summary

In this activity you created a sheet metal base feature and used the jog command to form the sheet metal around an existing part. You modified the bend radius where needed and then used the cutout command and the break corner command to finish the model.

H Activity: Deformation Features

Open a sheet metal file

- Start Solid Edge ST4.


- Click the  **Application** button ® **Open** ® *deformation_activity.psm*.

Note

This sheet metal part was created with a material thickness of 3.50 mm and a bend radius of 1.00 mm.

- Proceed to the next step.

Place a louver on the front face

- Select the Louver command .
- Click the Louver options button on the command bar.

Note

The louver depth cannot be greater than half the louver length. The louver height cannot be greater than the material thickness.

- Enter the following values:
 - Type: Formed-end louver
 - Length: 25.00 mm
 - Depth: 8.00 mm
 - Height: 4.00 mm
 - Turn on rounding and set the die radius to 0.88 mm.

Click OK.

- Move the cursor over the front face and observe the behavior.

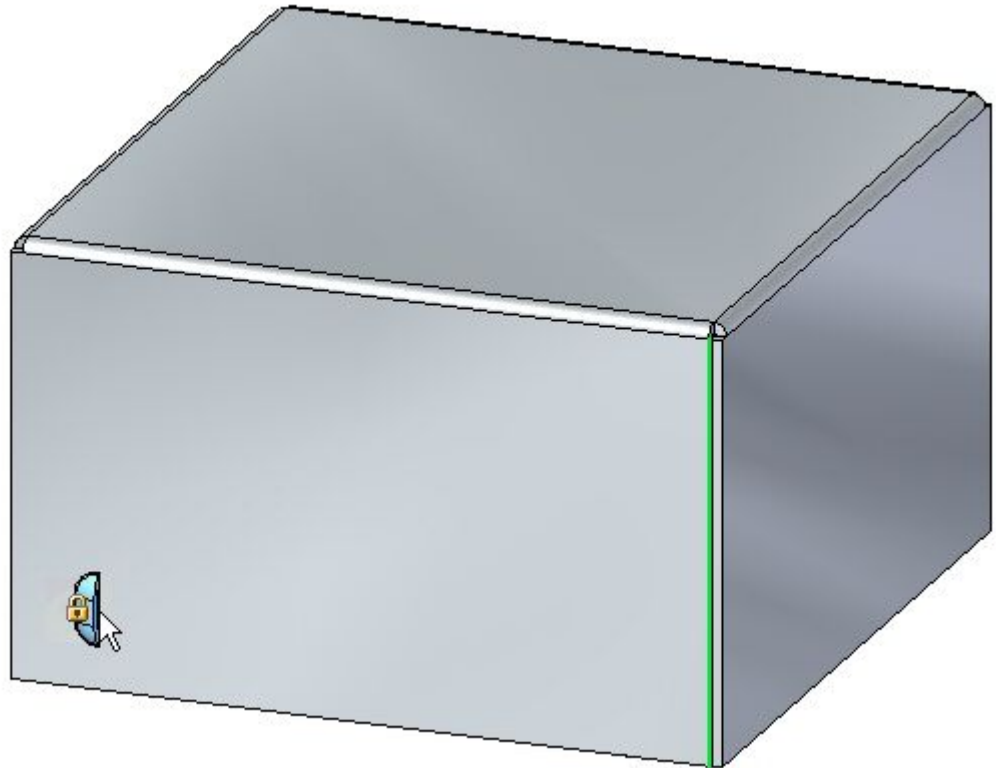
Note

The length of the louver is parallel with an edge the plane that the cursor is positioned over. The **N** (next) and **B** (back) keys can be used to cycle through the plane edges. The louver will orient itself to be parallel to the edge displayed. When the desired orientation is achieved, the **F3** key will lock the louver to that plane and in the orientation chosen.

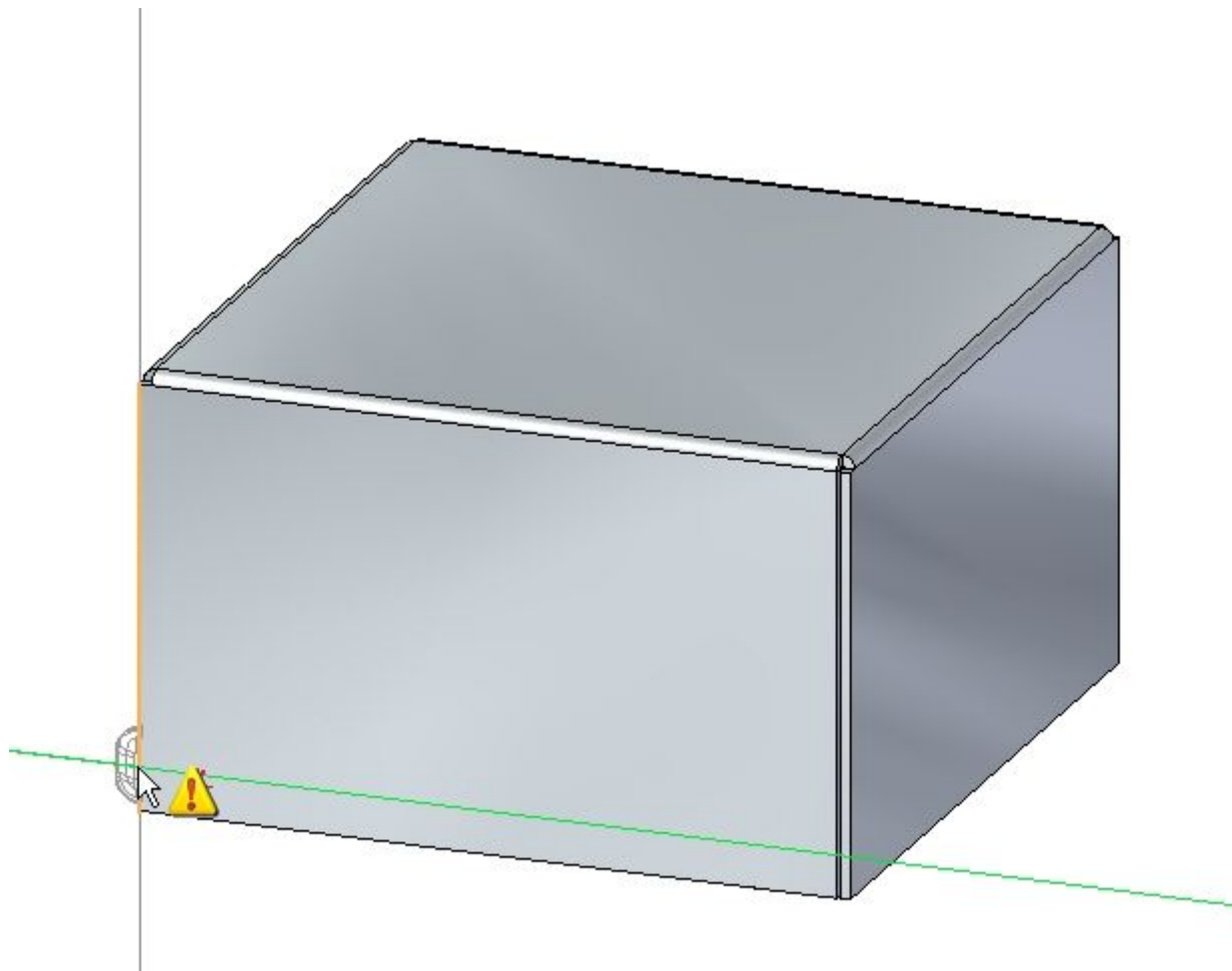
Orient the louver as shown by entering **N** as needed. Once the orientation is established, enter **F3** to lock to the face and orientation.

Note

Once the plane and orientation has been set can be positioned with a left mouse click, or my using dimensions to precisely locate the louver. In the next step, the louver will be located using dimensions.



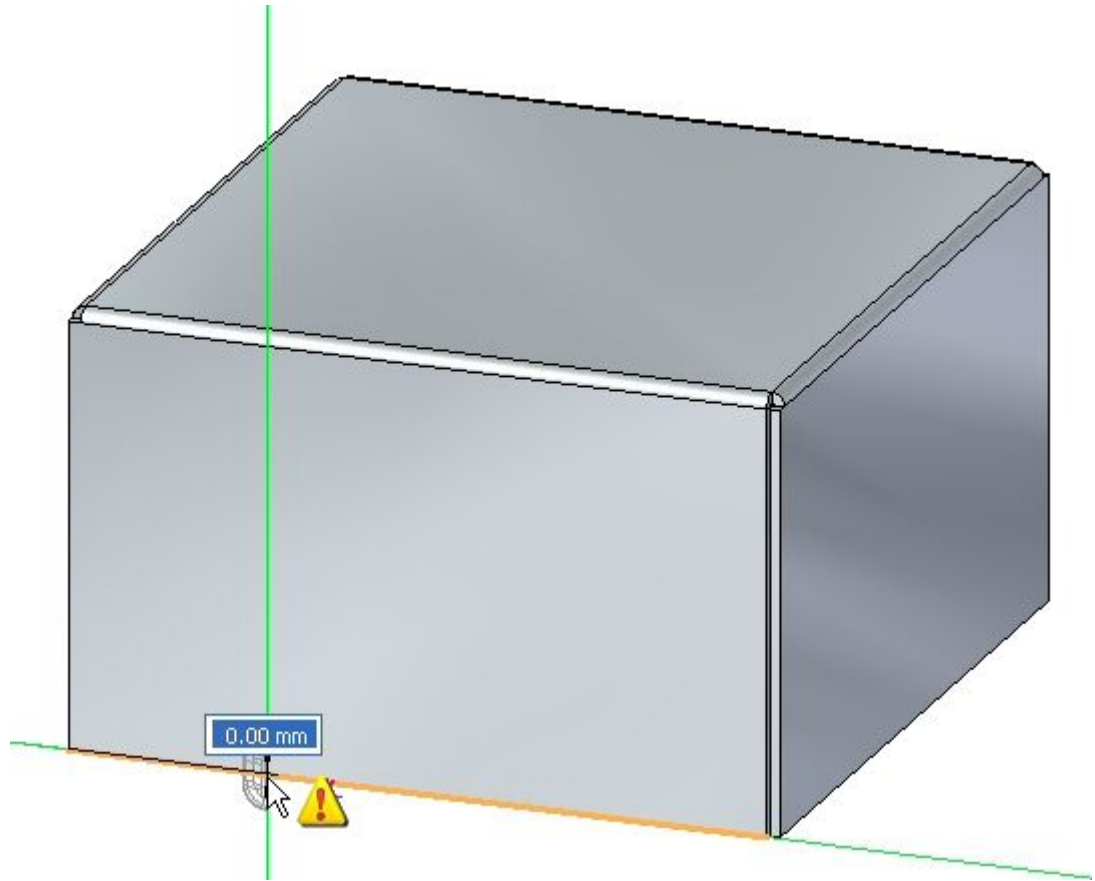
- ▶ Move the cursor over the edge shown below and press the **E** key.



Note

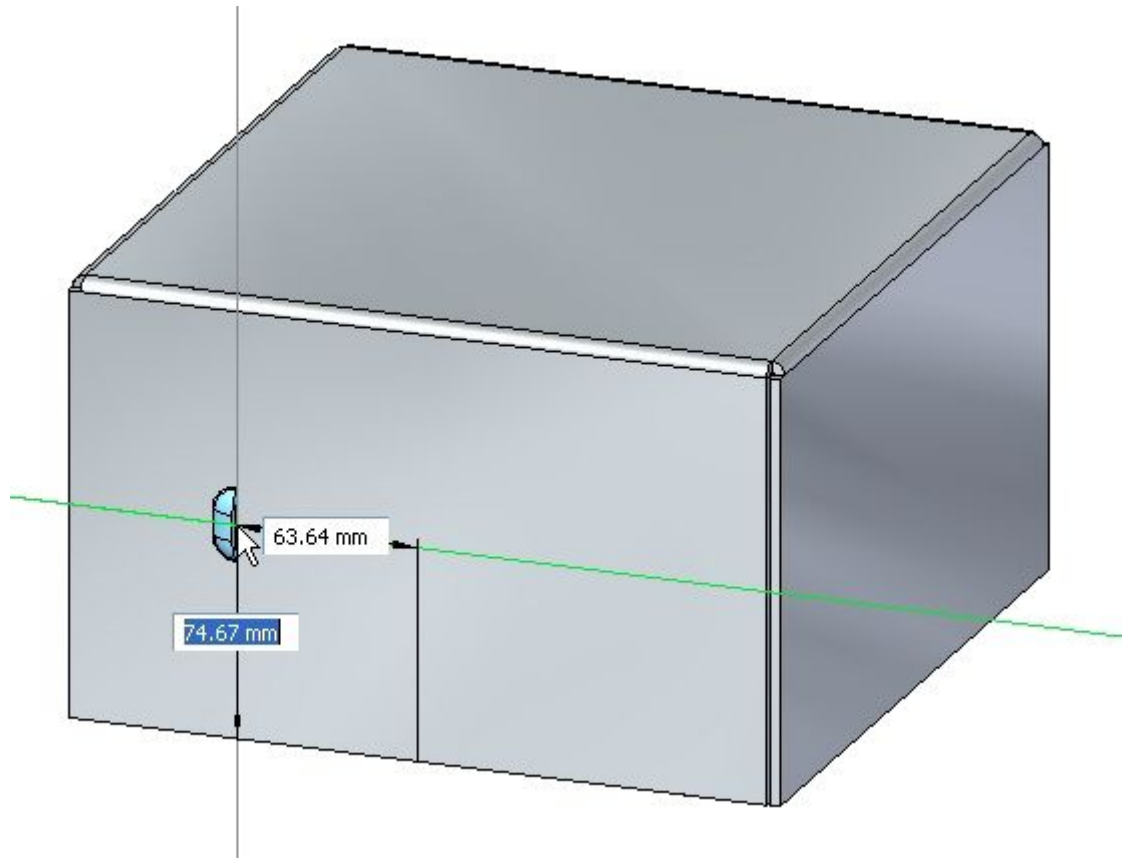
To position using dimensions, entering the character **E** dimensions from the endpoint of an edge and entering the character **M** dimensions from the midpoint of an edge.

- ▶ Move the cursor over the edge shown below and enter M from the keyboard.

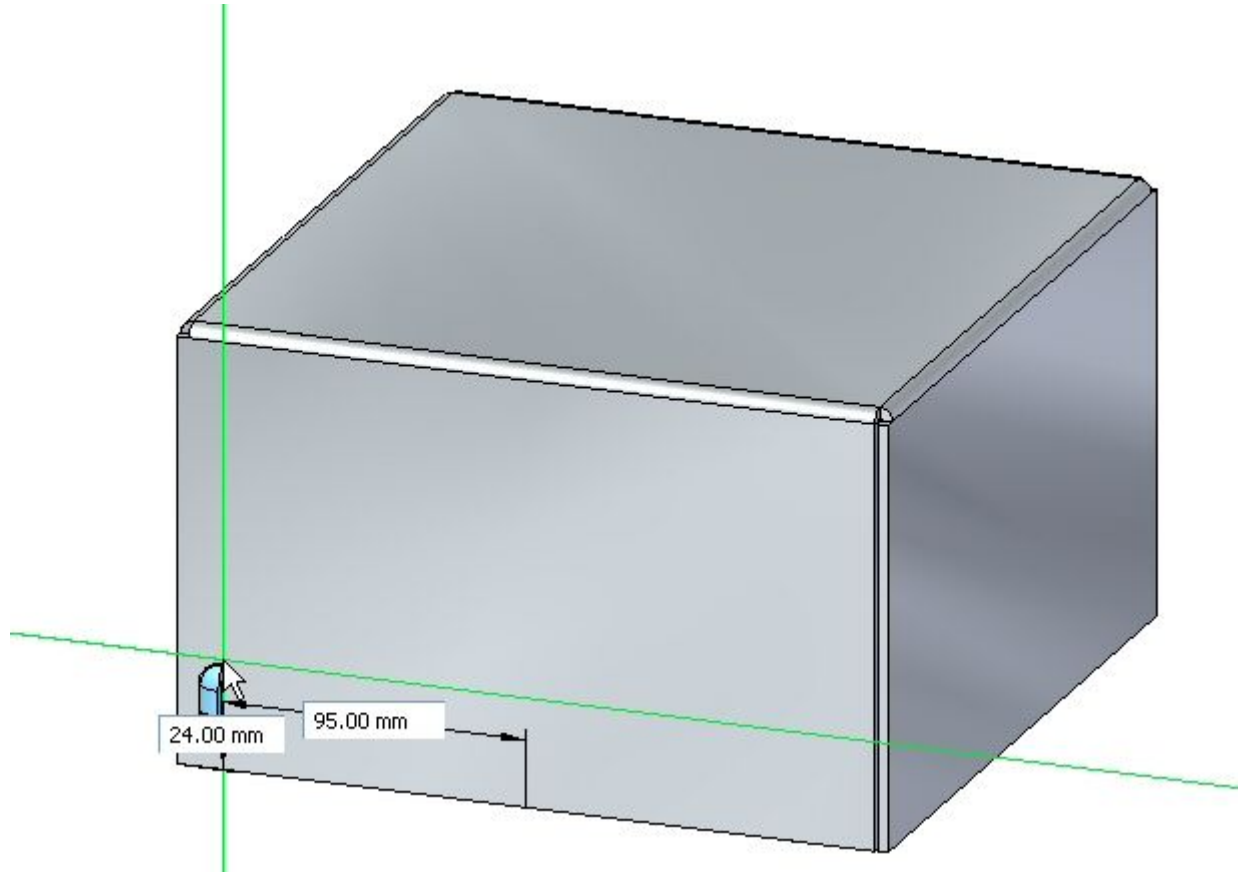
**Note**

Notice the dimension originating from the end of the previous edge chosen.

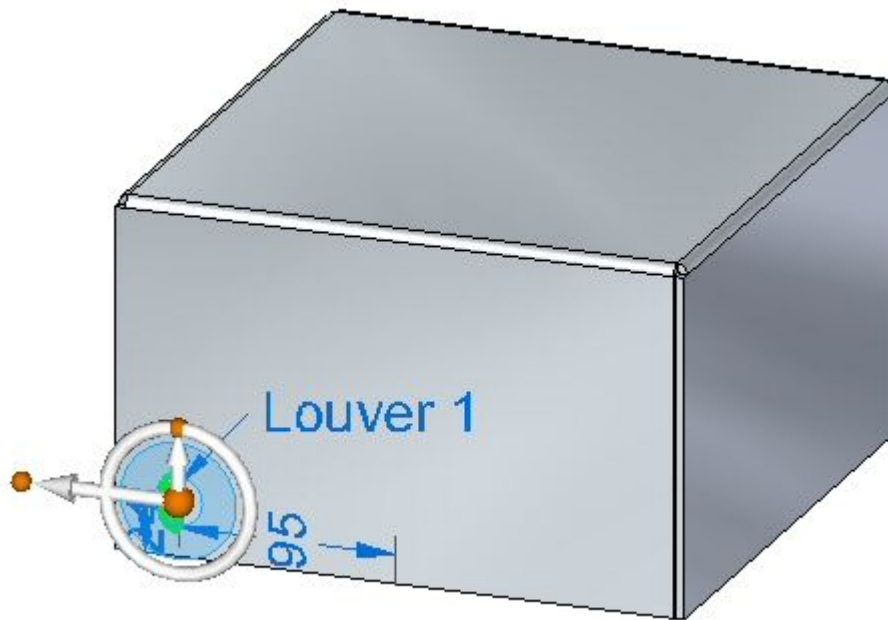
- Without clicking the mouse, move the cursor to the approximate position shown below.



- For the horizontal dimension value enter 95.00 mm, and for the vertical dimension value enter 24.00 mm as shown. Use the Tab key to toggle between fields, then press Enter.



- The louver is placed.

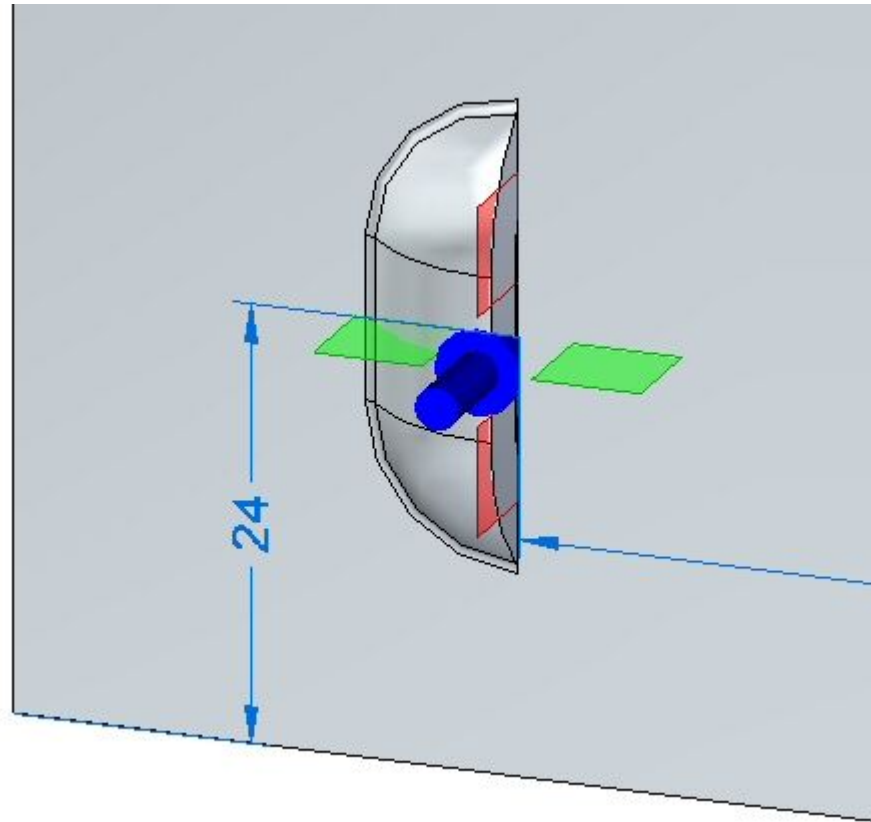


- Proceed to the next step.

Feature Origin

The origin of a feature is called the feature origin, and is also referred to as the strike point for manufacturing purposes and which can be shown and detailed in Solid Edge Draft. The feature origin can be offset upon creation, or offset after placement. The feature origin can also be used to apply a rotation angle to a rigid procedural feature such as a louver. In the following steps, the feature origin of the louver just created will be moved and rotated.

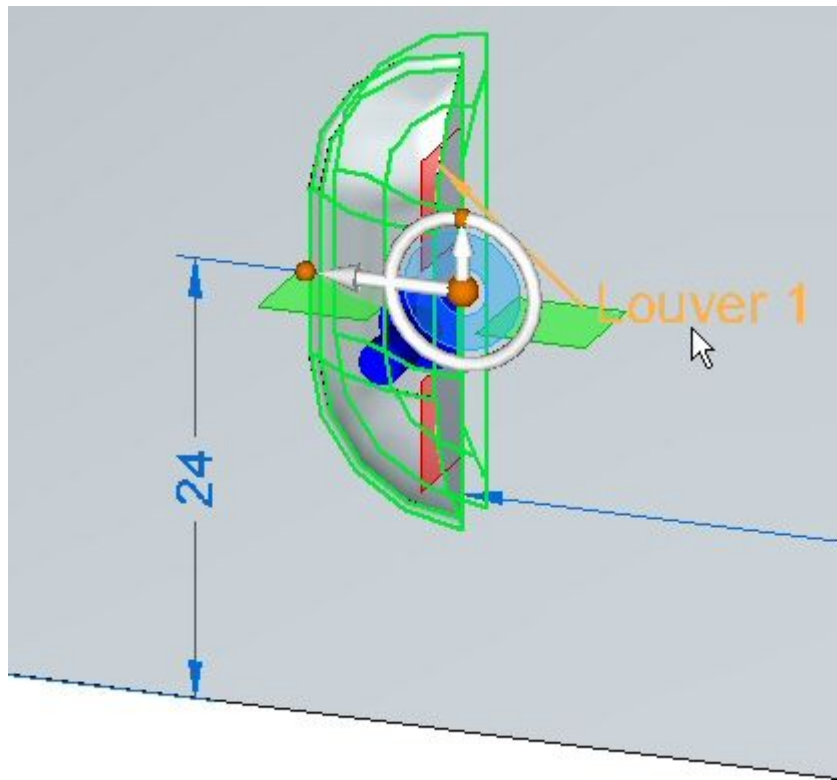
- ▶ In PathFinder right-click and select Show Feature Origin.



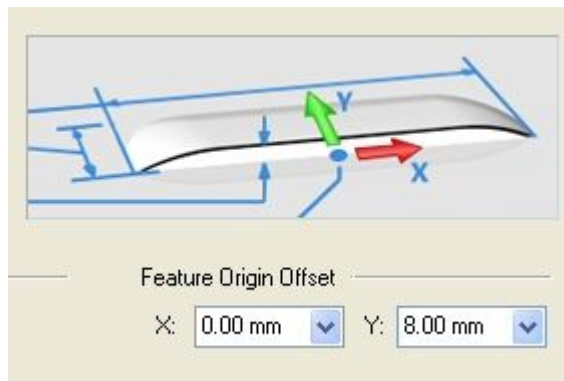
The feature origin appears.

The values for the louver will be edited in the following steps.

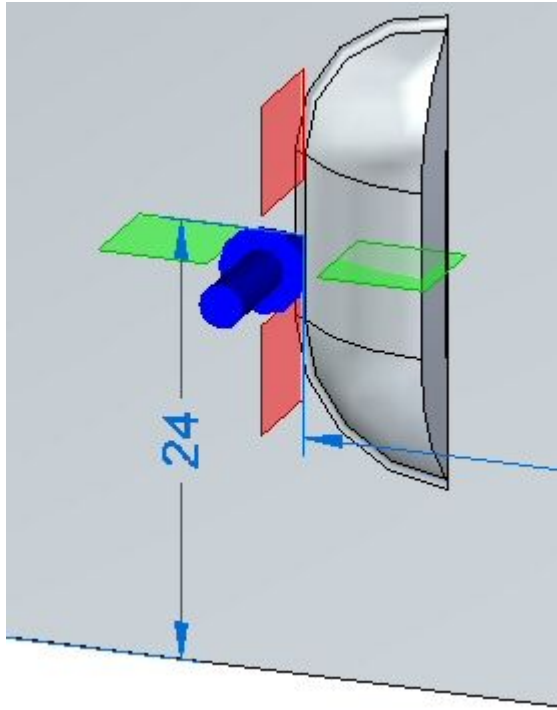
- ▶ Click the louver feature in PathFinder. When displayed, click the edit handle as shown.



- ▶ Click the Louver Options button. Change the Y value of the feature origin offset to 8.00 mm, then click OK.



The feature origin changes position.

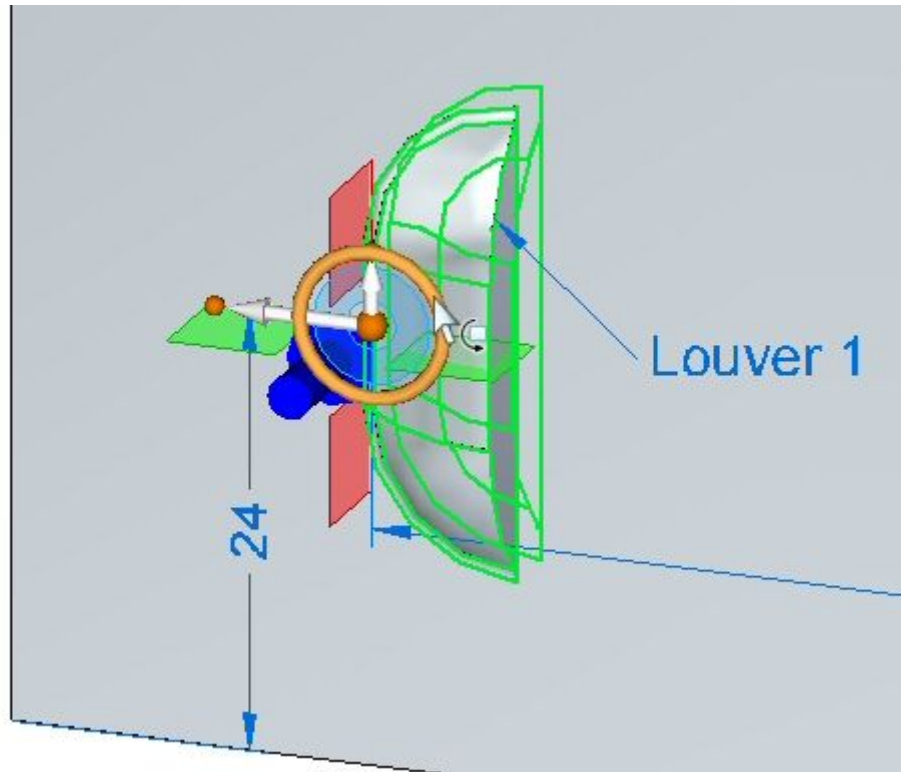


Note

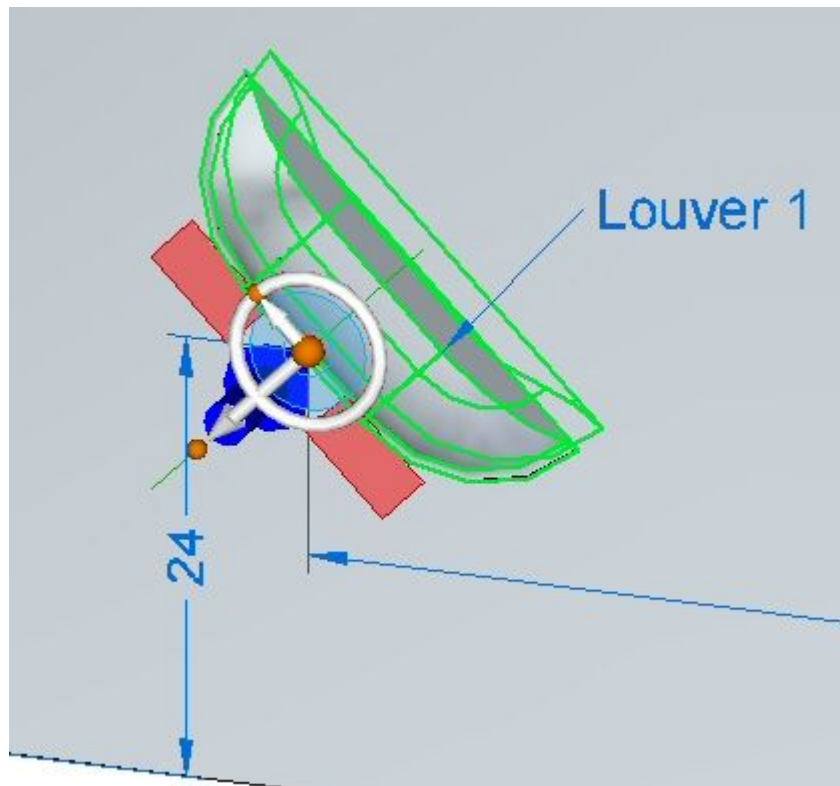
Notice that only the feature origin changed position. The louver is still located in the same position.

Now the louver will be rotated 45 degrees.

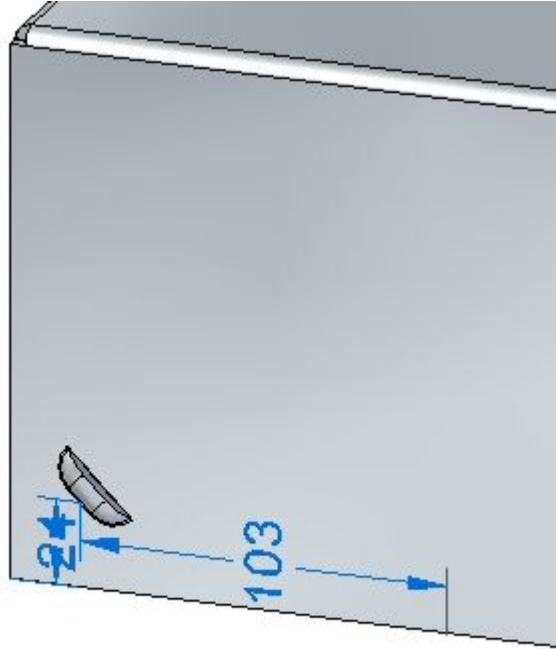
- ▶ Select the louver, then select the torus of the steering wheel.



- ▶ Rotate the louver 45° as shown.



- The louver has been rotated about the feature origin. Right-click the louver in PathFinder and hide the feature origin.



- Proceed to the next step.

Patterning the deformation feature

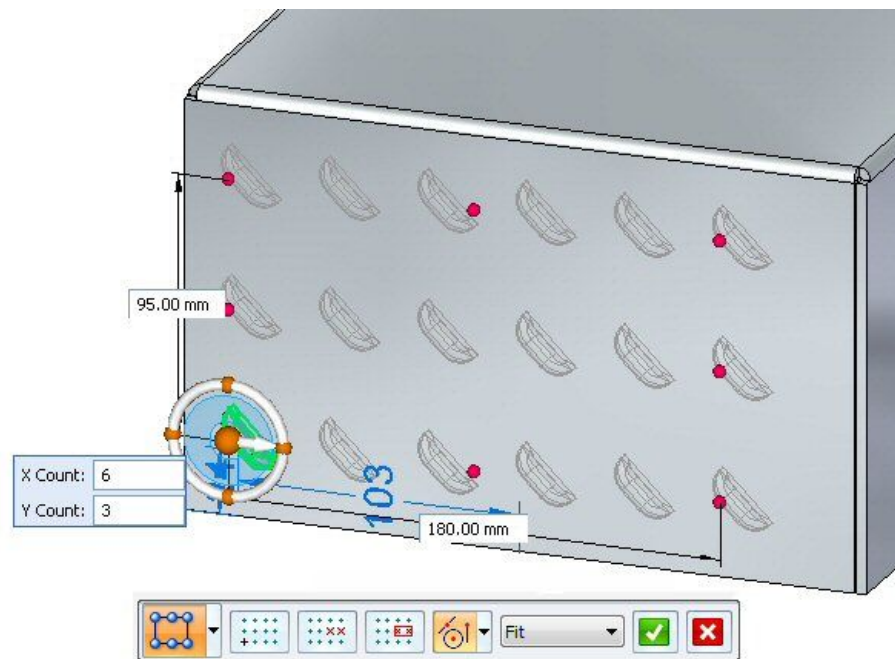
- Select the louver in PathFinder and then click the Rectangular Pattern command



Note

When selecting a reference plane for the pattern, press F3 to lock to the appropriate plane.

- Set the pattern parameters as follows:
 - Type: Fit
 - X count: 6
 - Y count: 3
 - Vertical distance: 95.00 mm.
 - Horizontal distance: 180.00 mm.

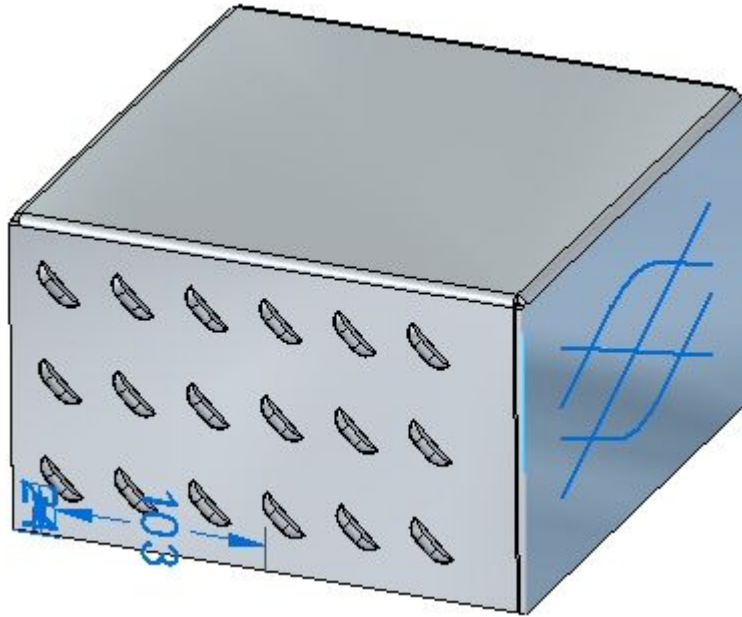


- Click the accept button to finish creating the rectangular pattern.
- Proceed to the next step.

Placing Beads

Beads are added to sheet metal parts as stiffeners.

- ▶ In PathFinder, display the sketch named Beads.



- ▶ Click the Bead command

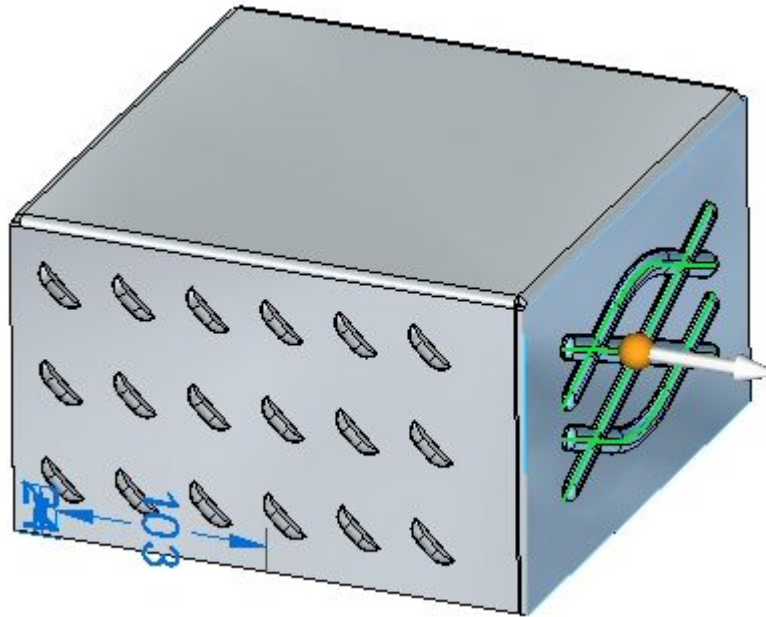


- ▶ Click the Bead Options command

- ▶ Set the following options:

- Cross section: U-Shaped.
- Height: 4.00 mm.
- Width: 3.50 mm.
- Angle: 20°.
- End Condition: Formed.
- Include rounding with a punch radius of 0.50 mm, and a die radius of 0.50 mm.
- Click OK.

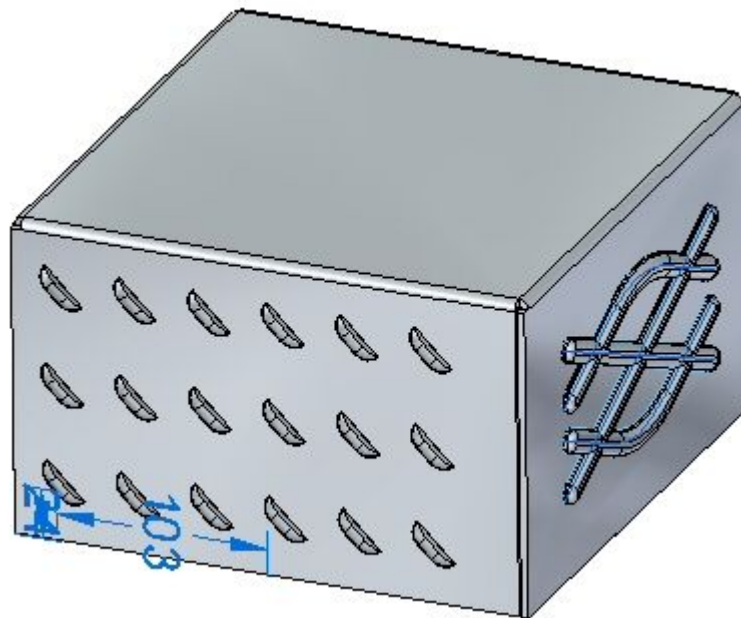
- ▶ Select all the elements in the sketch as shown.



Note

Clicking the arrow will reverse the direction of the beads.

- ▶ Right mouse click to accept the beads. The beads are created.

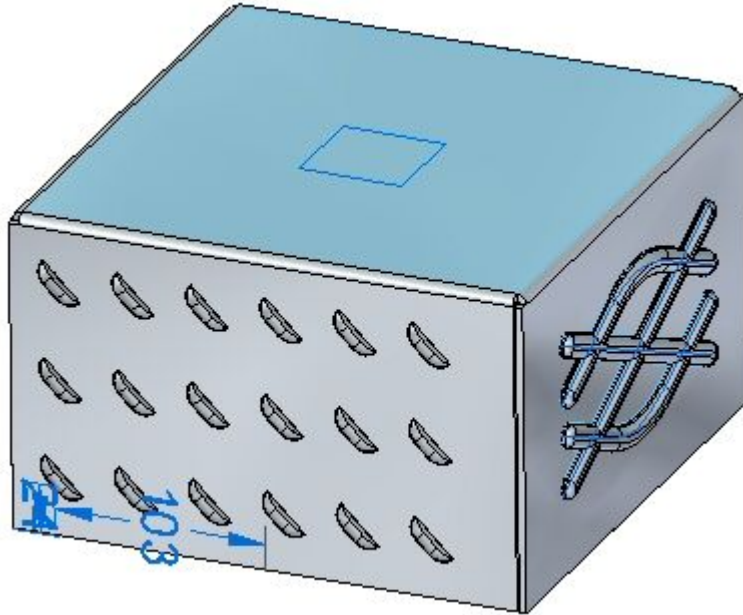


- ▶ Proceed to the next step.

Placing dimples and drawn cutouts

In this step you will place a dimple and a drawn cutout from a two different sketches.

- ▶ In PathFinder, display the sketch named Drawn.

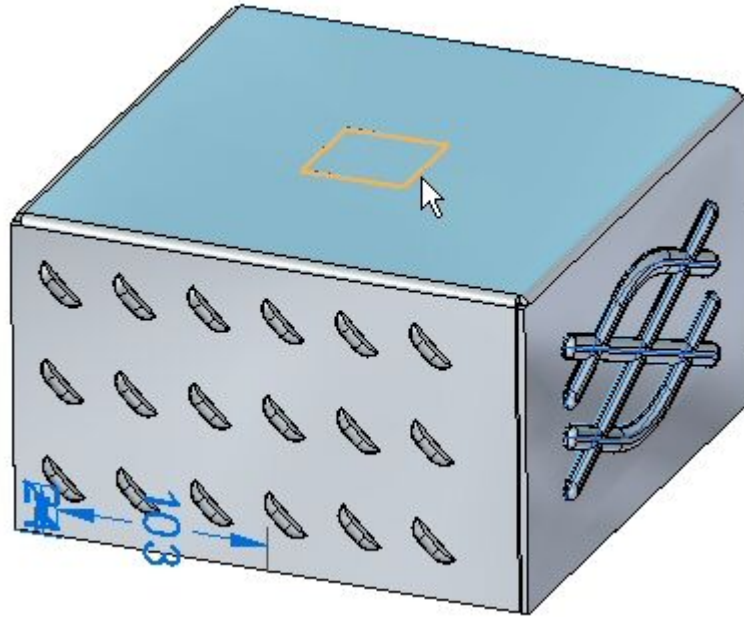


- ▶ Click the Drawn Cutout command

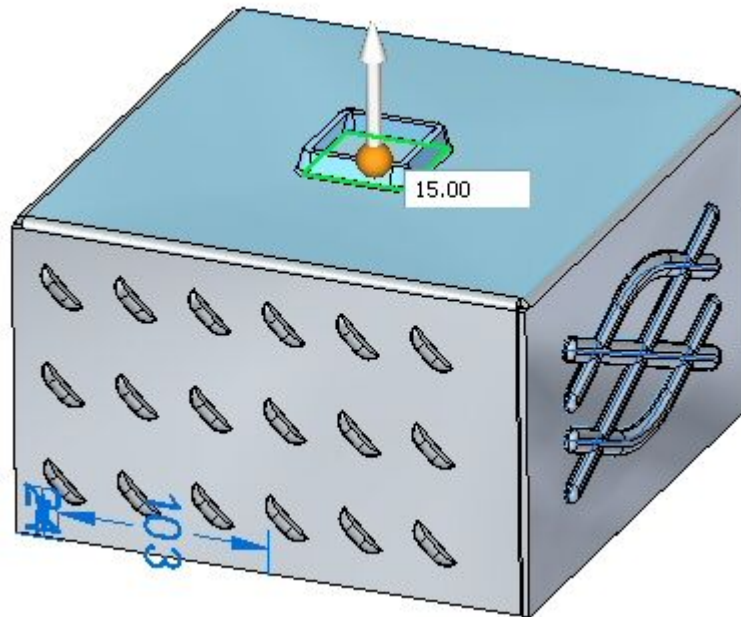


- ▶ Click the Drawn Cutout Options command
- ▶ Set the following options:
 - Taper angle: 15°.
 - Include rounding: die radius 1.75 mm.
 - Include punch side corner radius: 1.75 mm..
 - Click OK.

- ▶ Select the region shown.



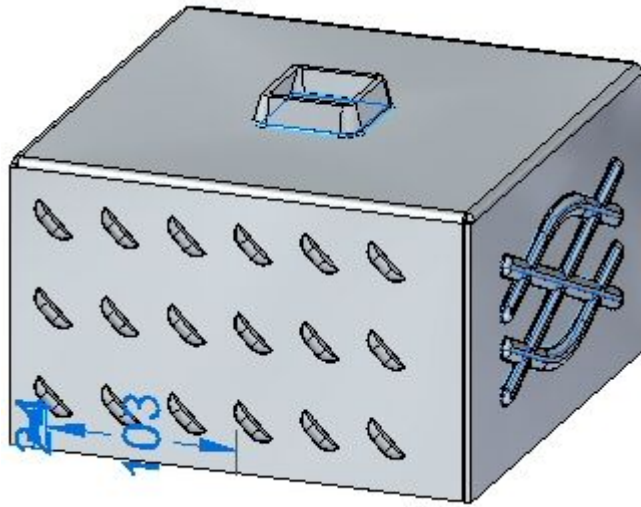
- ▶ Enter a distance of 15.00 mm.



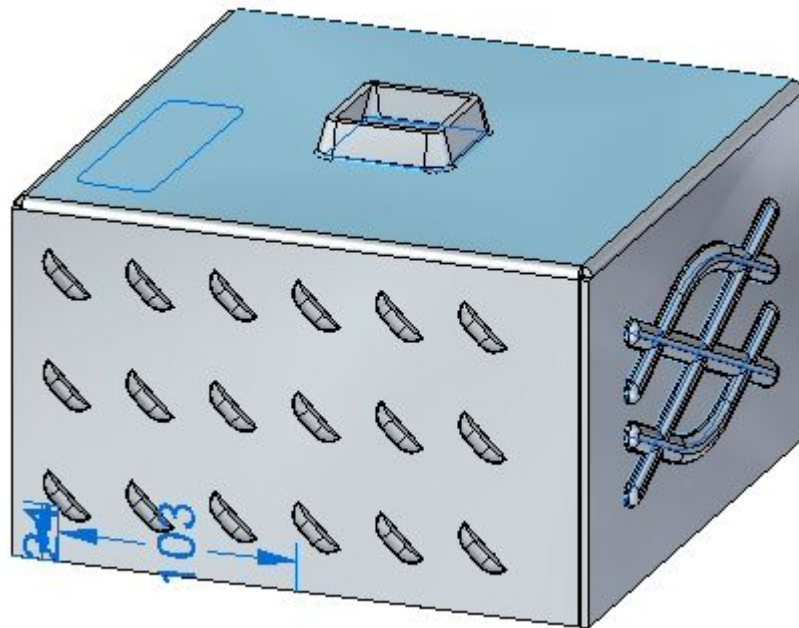
Note




Clicking the arrow will reverse the direction of the drawn cutout.

- ▶ Right mouse click to accept the drawn cutout. The drawn cutout is created.

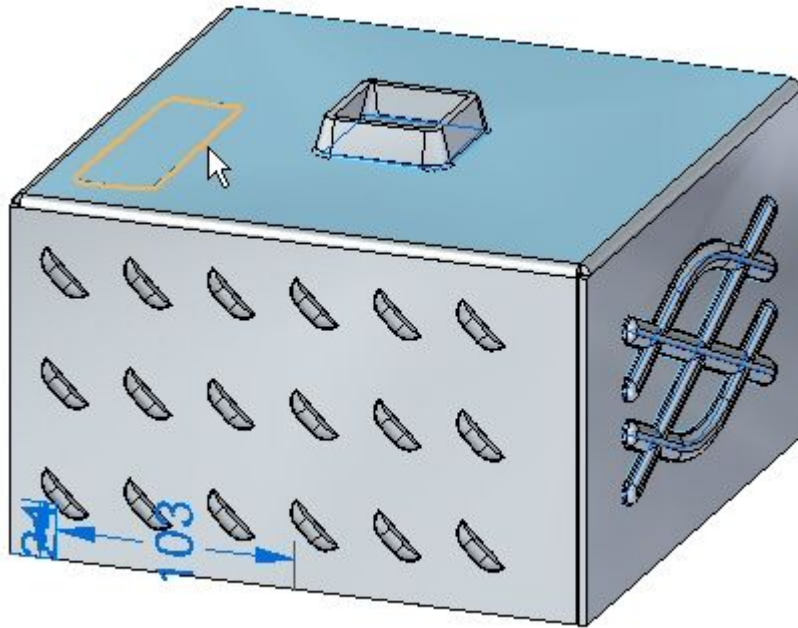


- ▶ In PathFinder, display the sketch named Dimple.

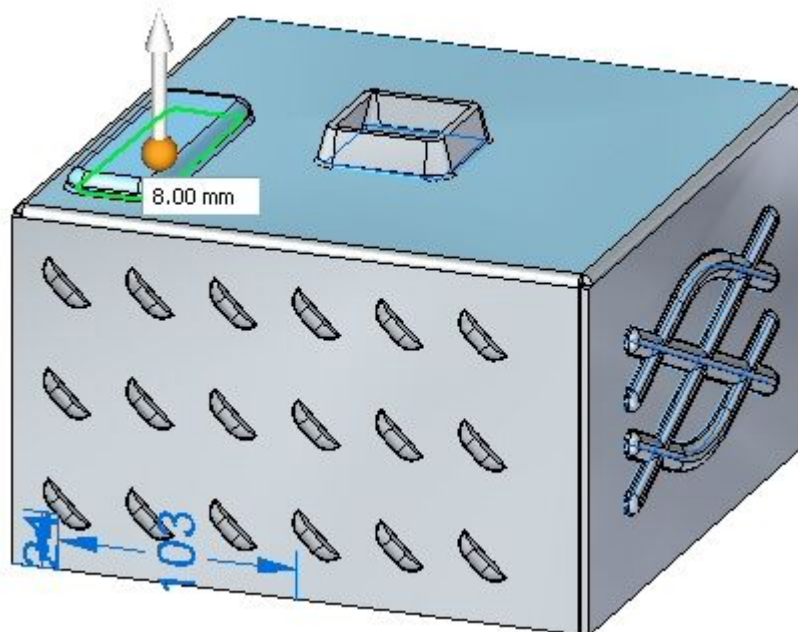


- ▶ Click the Dimple command .
- ▶ Click the Dimple Options button .
- ▶ Observe the options, but do not change any of the options.
- ▶ Change the sketch profile to represent punch as shown .

- ▶ Select the regions shown.



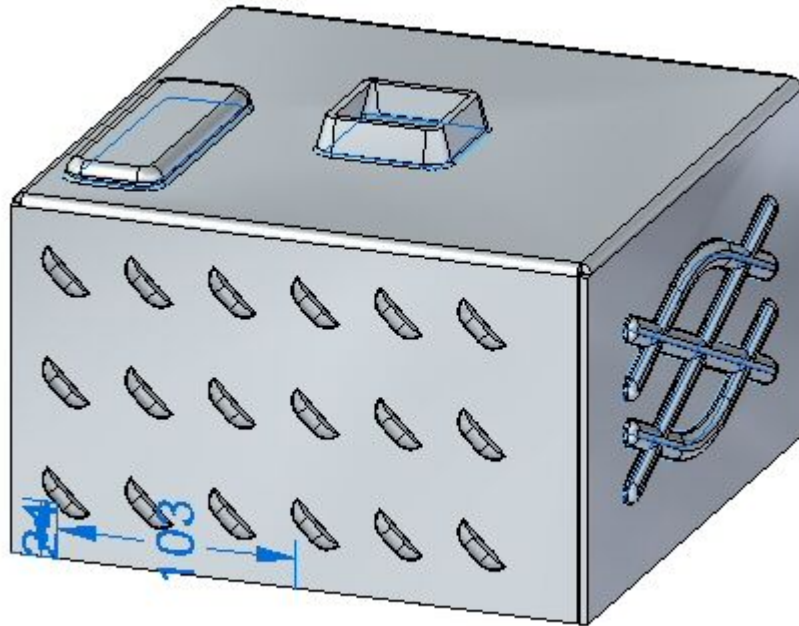
- ▶ Enter a distance of 8.00 mm.



Note

Clicking the arrow will reverse the direction of the dimple.

- ▶ Right-click to accept the dimple. The dimple is created.

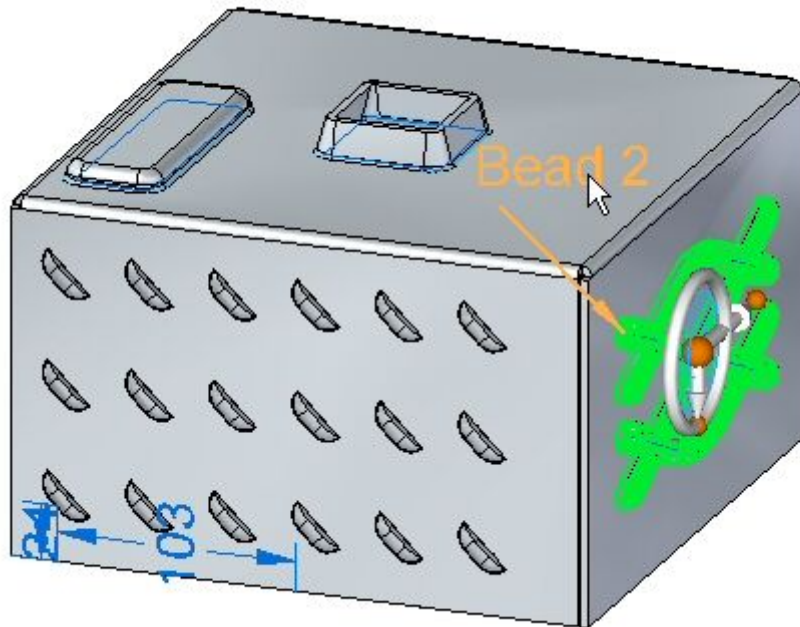


- ▶ Proceed to the next step.

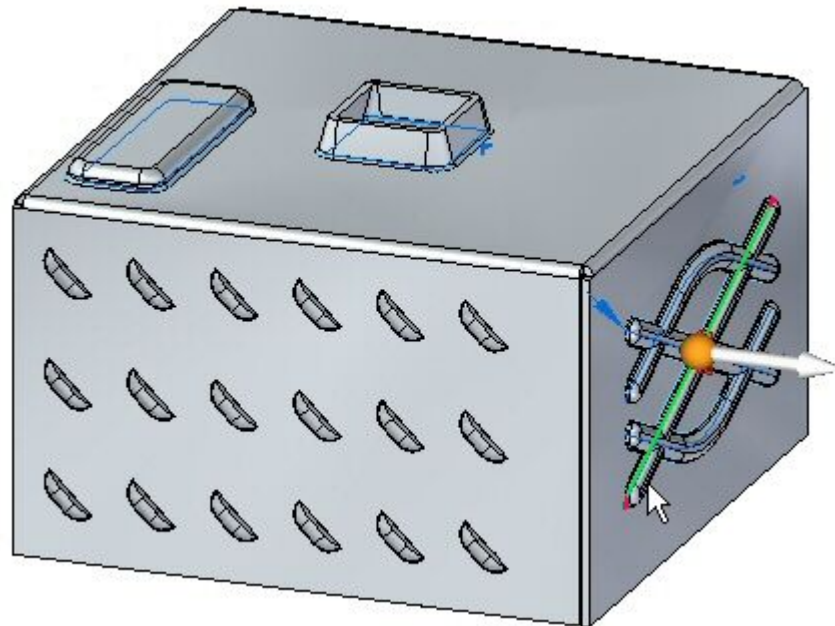
Editing deformation features

In this step you will edit the bead feature created in an earlier step.

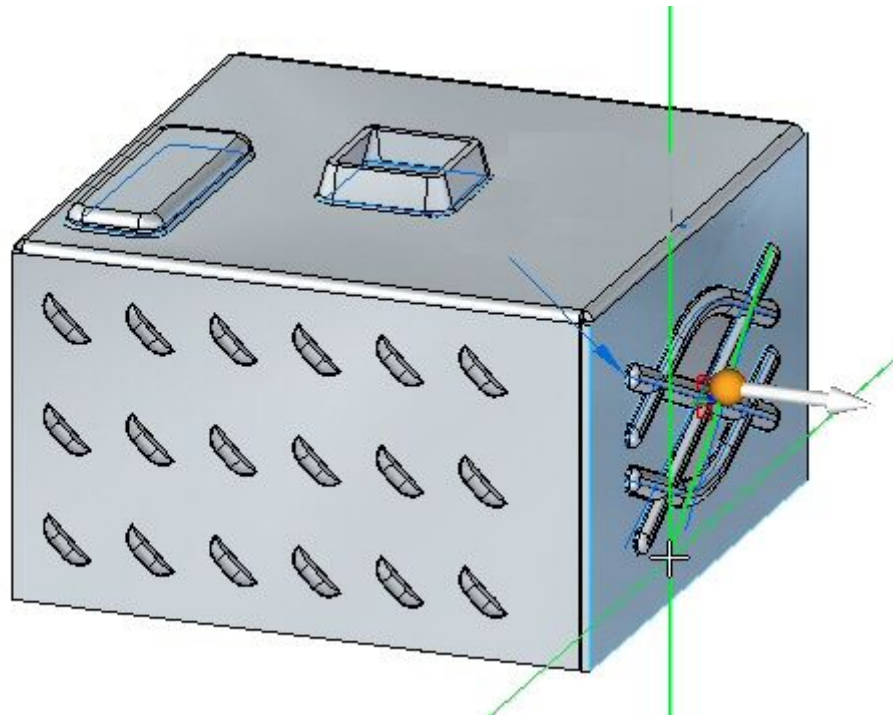
- ▶ In PathFinder, select the bead feature. Click the edit handle to edit the feature.



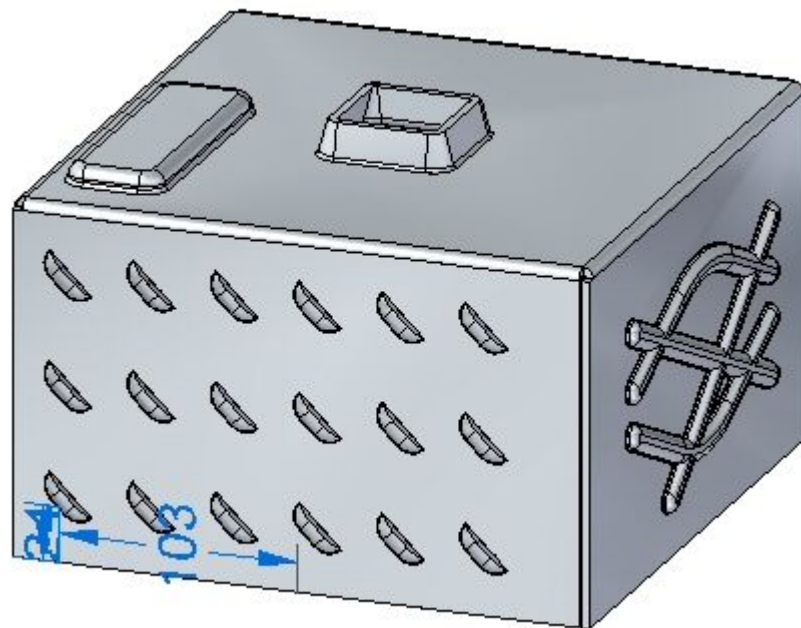
- ▶ Click the Edit Profile Handle.
- ▶ Select the line shown.



- ▶ Drag the endpoint of the line to a new position. Click the green check mark and then right-click.



- ▶ The deformation feature has been edited.




**Note**

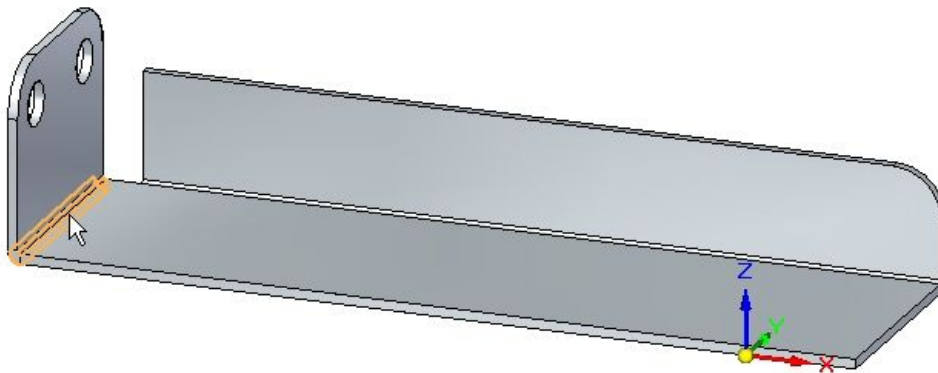
When editing a rigid procedural feature, the sketch used to create the feature remains a part of the feature and can be modified at a later time.

- ▶ Save and close the sheet metal document. Proceed to the next step.

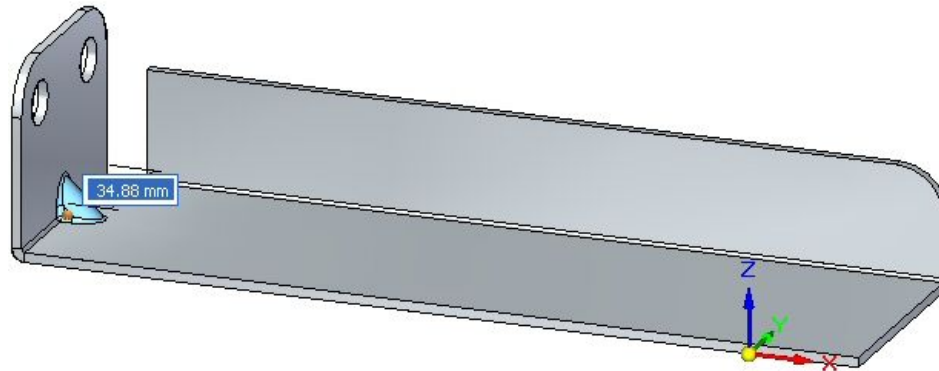
Placing a single gusset

In this step you will place gussets between two thickness faces.

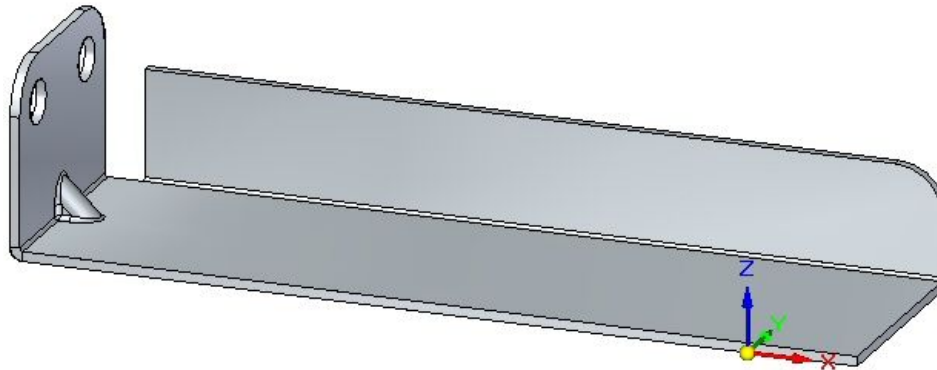
- ▶ Click the  **Application** button ® **Open** ® *gusset_activity.psm*.
- ▶ Click the Gusset command .
- ▶ Click the Gusset Options button .
- ▶ Set the following parameters:
 - Depth: 11.25 mm.
 - Include rounding with the both the punch and die radius being 1.50 mm.
 - Set the gusset shape to round.
 - Set the width to 9.00 mm. Click OK.
- ▶ Set the gusset patterning parameter to single.
- ▶ Select the bend shown.



- ▶ Click the midpoint shown to place the gusset.



- ▶ Right-click to complete the placement. The gusset is placed.

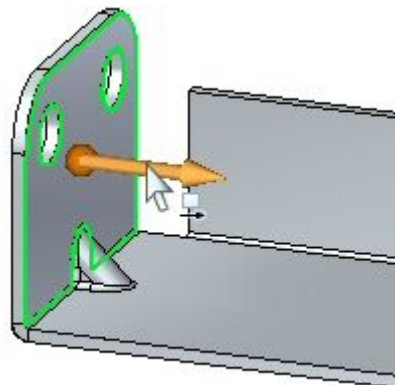


- ▶ Proceed to the next step

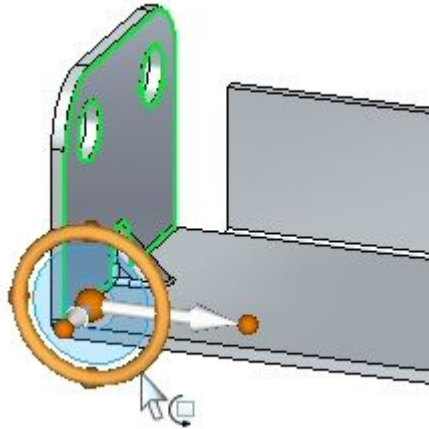
Rotating faces containing gussets

In this step you will rotate a face containing a gusset and observe how the gusset responds.

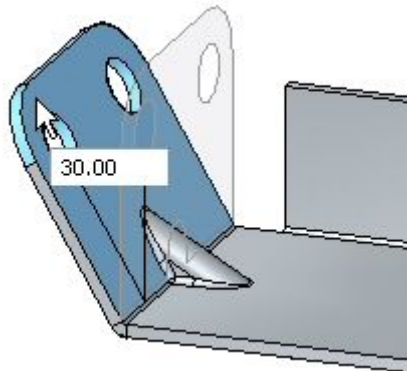
- ▶ Click the Select tool and select the face shown.



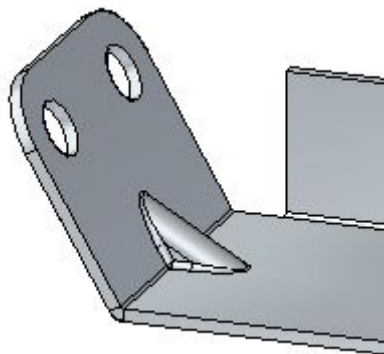
- ▶ Move the steering wheel to the bend as shown and select the torus so as to rotate the face.



- ▶ Enter an angle of 30° as shown.



- ▶ The result is as shown.





Note

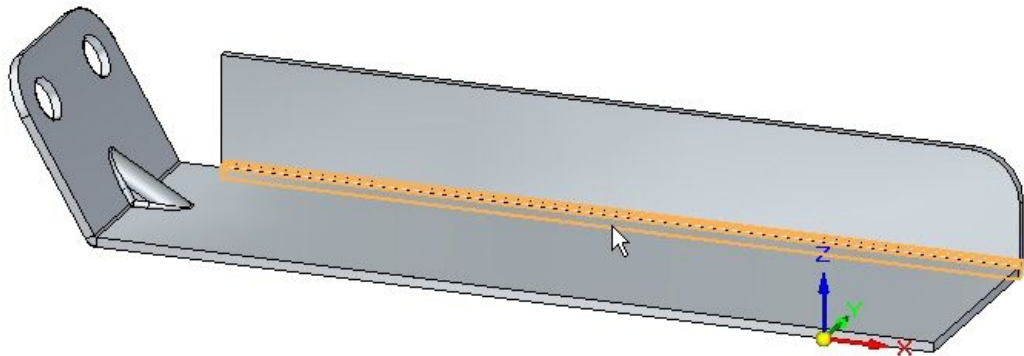
The gusset is an adaptive procedural feature that will change shape to as the angle between the faces changes.

- ▶ Proceed to the next step.

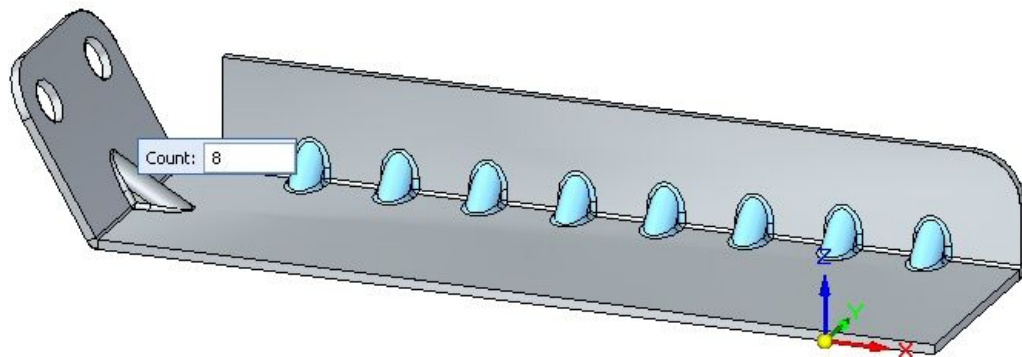
Placing a gusset pattern

In this step you will place multiple gussets along a bend.

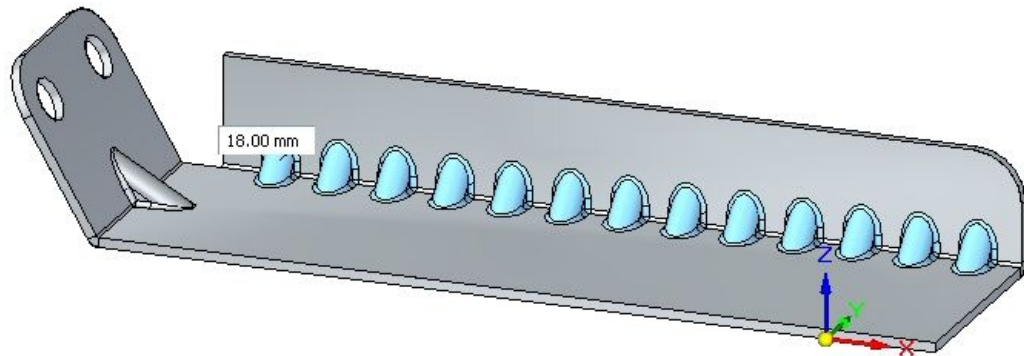
- ▶ Click the Gusset command .
- ▶ Click the Gusset Options button .
- ▶ Set the following parameters:
 - Depth: 11.25 mm.
 - Include rounding with the both the punch and die radius being 1.50 mm.
 - Set the gusset shape to round.
 - Set the width to 9.00 mm. Click OK.
- ▶ Set the gusset patterning parameter to fit.
- ▶ Select the bend shown.



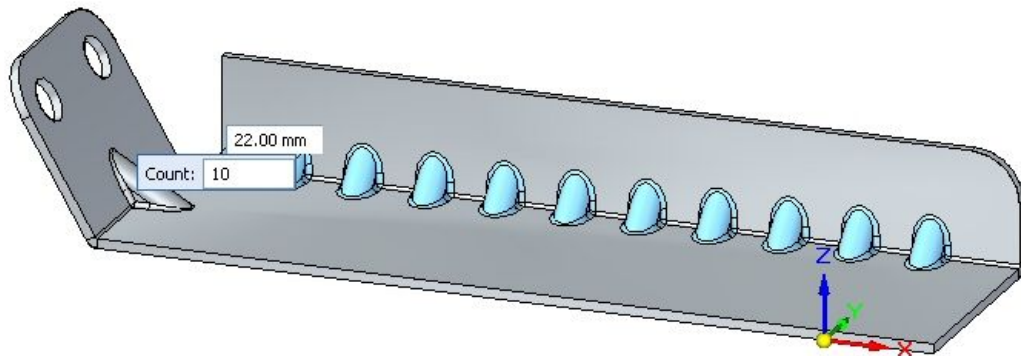
- ▶ set the count to 8. Observe the results.



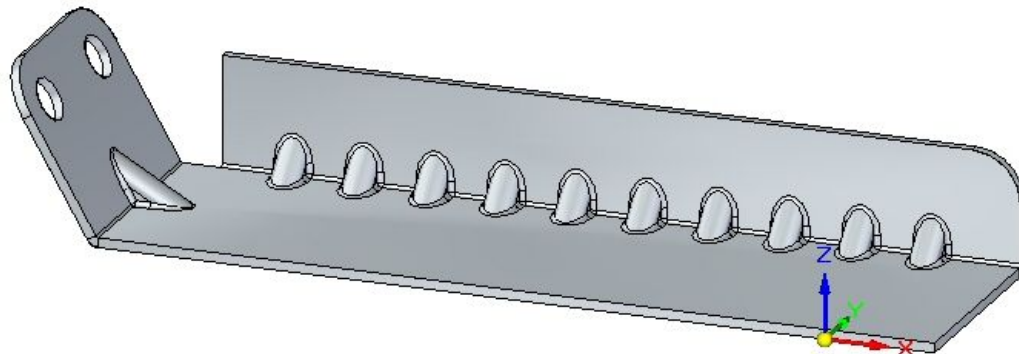
- ▶ Set the pattern type to fill. Observe the results.



- ▶ Set the pattern type to fixed. Set the count to 10 and the distance to 22.00 mm.



- ▶ Right-click to complete the gusset pattern. Observe the results.



- ▶ Save and close the sheet metal document. This concludes this activity.

Activity summary

In this activity you created a variety of deformation features. The feature origin for a rigid procedural feature was displayed and moved to a new location. The feature was rotated. Multiple occurrences were created with the pattern command.

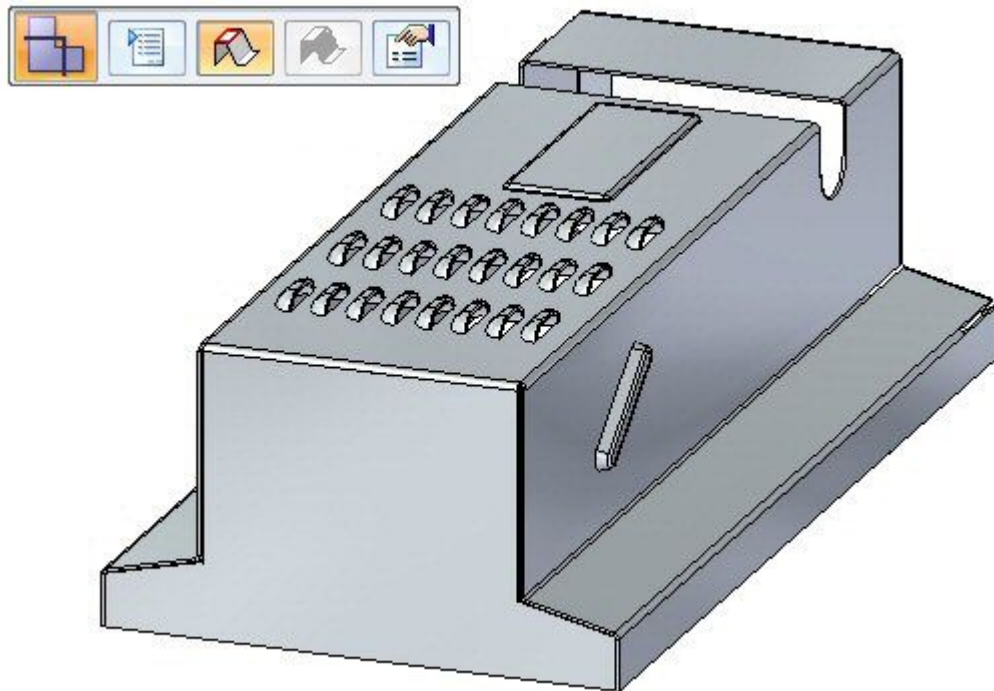
I Activity: Creating a flat pattern from a sheet metal part

Open a sheet metal file

- Start Solid Edge ST4.
- Click the  **Application** button ® **Open** ® *flatpattern_activity.psm*.
- Proceed to the next step.

Creating a flat pattern

- Click Tools® Model® Flat Pattern.



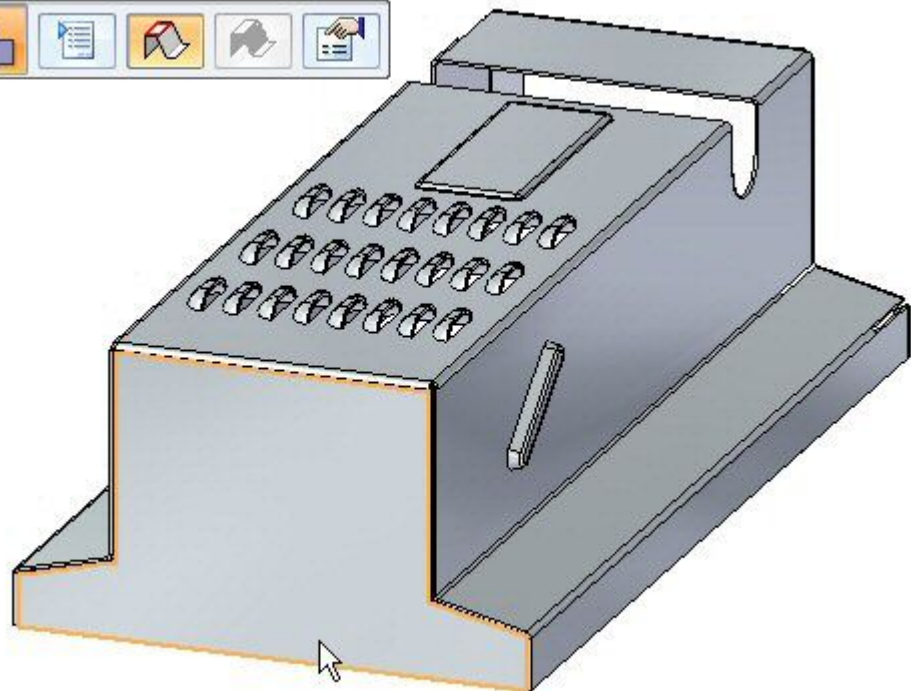
- ▶ Click Tools® Flat® Flatten.



The Flat Pattern command bar is displayed.



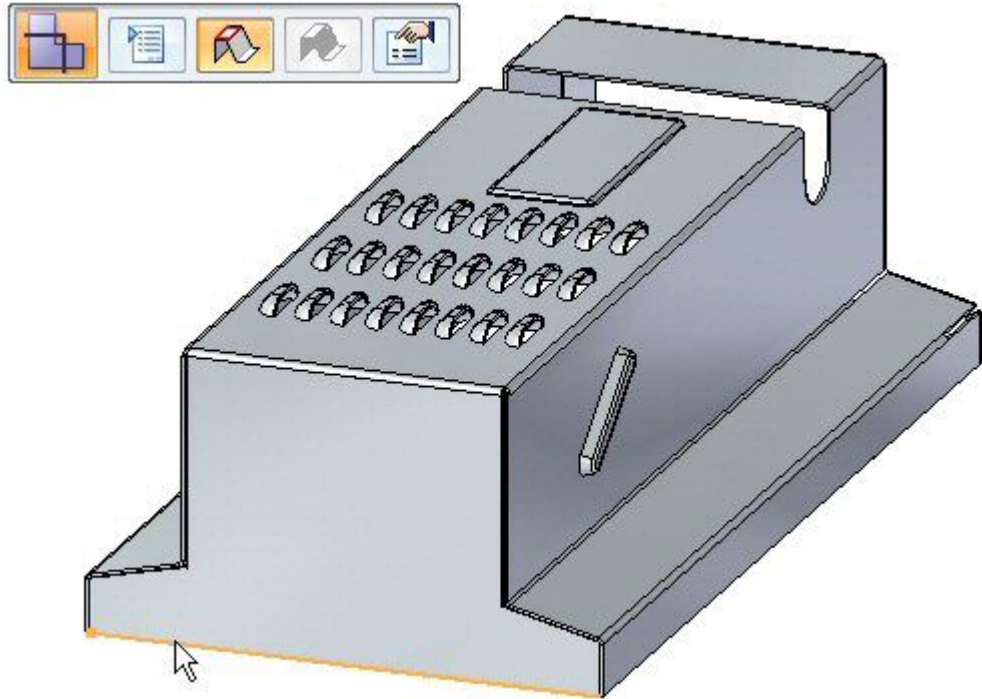
- ▶ Select the face shown.



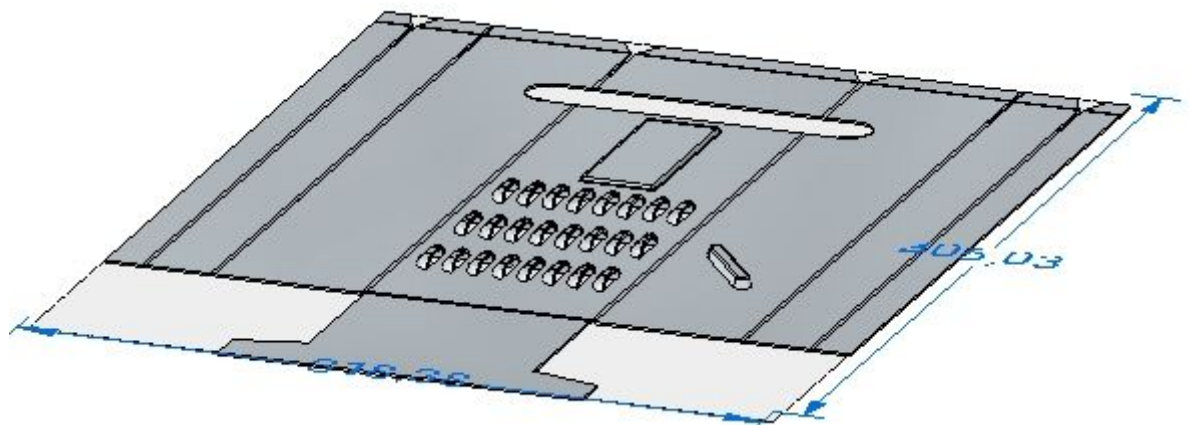
- ▶ Select the edge shown to orient the flat pattern.

Note

The edge selected defines the x axis of the flat pattern.



- ▶ The flat pattern is created.



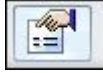
Note

Notice there is a new tab in PathFinder for the flat pattern.

- ▶ Proceed to the next step.

Flat pattern options

- ▶ Click the Flat Pattern Cut Size button.



- ▶ Observe the following on this dialog box.
 - The cut size of the flat pattern dimensions can be displayed.
 - Alarms can be set to notify if the cut size is larger than a desired size.
 - The current cut size is displayed if this command is chosen in the flattened state rather than the design state.

Close the dialog box.

- ▶ Click the Flat Pattern Treatment Options button.

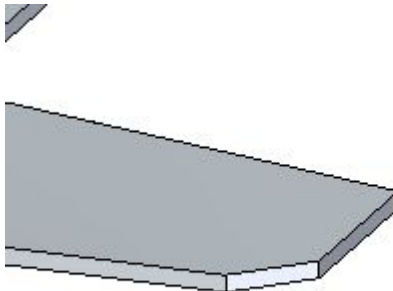


Note

The flat pattern treatment options can also be set by clicking the application button and then clicking Solid Edge Options.



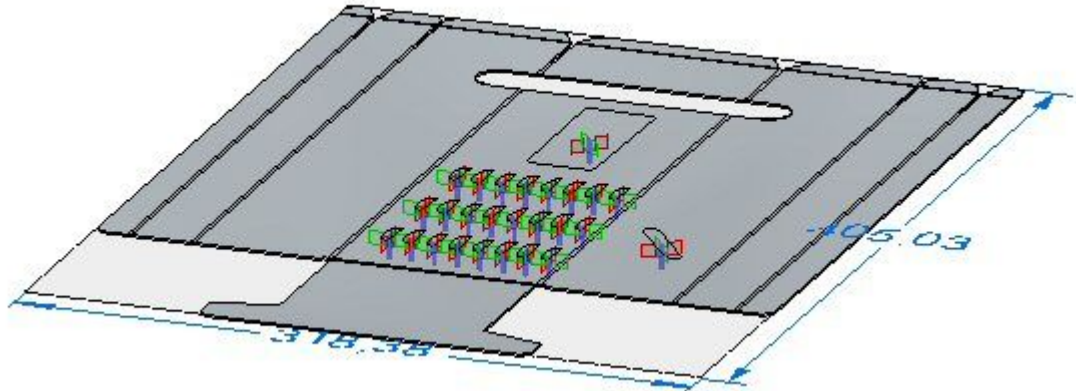
- ▶ Set Outside Corner Treatments to Chamfer and set the value to 4.00 mm. Click Apply. Observe the results. Chamfers are applied to the exterior corners that do not contain rounds.



Note

This corner treatment only shows up in the flattened state and not in the designed state.

- ▶ Set Formed Feature Display to As Feature Loops and Feature Origin and click Apply. Observe the results.



Note

When placing a flat pattern onto a drawing sheet in Solid Edge Draft, this display controls the what geometry is placed. If the sketches or 3D geometry is desired on the drawing sheet, the option will need to be set here.

- ▶ Set Formed Feature Display to As Feature Origin and click Apply. Observe the results.
- ▶ Save the file.
- ▶ Proceed to the next step.

Saving the flat pattern as a .dxf or .par file

- ▶ Click the Application menu.



- ▶ Click Save as® Save as Flat.



- ▶ Save the file as *my_flat.dxf*.

Note

Some numerical control machines can read a .dxf file directly. The flat pattern can also be saved as a Solid Edge part file.

- ▶ Proceed to the next step.

Placing a flat pattern on a drawing sheet

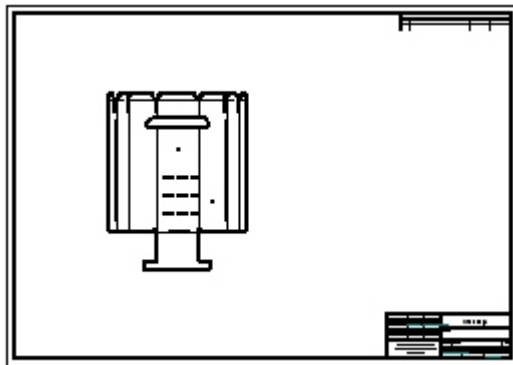
- ▶ Click the Application button.



- ▶ Click New@ Create Drawing.



- ▶ Use the default template and ensure the Run Drawing View Creation Wizard button is checked.
- ▶ In the Drawing View Creation Wizard, click the Flat Pattern option in Drawing view options. Click next.
- ▶ In Drawing View Orientation, use the top view and then click next.
- ▶ Click Finish and place the flat pattern on the drawing sheet.



Note

The feature origins, or strike points, for the deformation features are displayed. These can precisely located by dimensioning.

Note

The display of tangent edges on the drawing view represent the location of the edges of bends.

- ▶ Close the sheet metal documents without saving. Proceed to the activity summary.

Activity summary

In this activity you created a flat pattern and changed the display options. You also placed the flat pattern on a drawing sheet.

