Sheet metal design

Publication Number
mt01419-s-1040

Sheet metal design

Publication Number
mt01419-s-1040

Proprietary and restricted rights notice

This software and related documentation are proprietary to Siemens Product Lifecycle Management Software Inc.

Contents

Course overview

The **Sheet Metal Design** course was developed to demonstrate how to utilize the sheet metal feature and design functions within Solid Edge to create sheet metal parts which maintain the design intent even when changes are made to the material thickness or bend radius.

Course objectives

There are two objectives for this course. The first objective is to learn the fundamental tools required to get started and understand the use of Solid Edge. The second objective is to learn the tools and process of creating and editing sheet metal parts.

If you have already taken the Solid Edge Fundamentals Course or have mastered the use of the Solid Edge fundamental tools, then you can go directly to the Sheet Metal specific lessons.

After successfully completing the fundamentals portion of this course, the student should be able to:

- Understand the Solid Edge user interface.
- Understand how to use reference planes.
- Create profiles and sketches.

After successfully completing sheet metal portion of this course, the student should be able to:

- Create sheet metal parts using both modeling and sheet metal features.
- Effectively create and edit sheet metal features.
- Create sheet metal deformation features.
- Convert a solid model to a sheet metal model.
- Create flat patterns.
- Create sheet metal drawings.

Why do I need sheet metal features?

You may be wondering why specific sheet metal features were created independent from modeling features. Sheet metal features are unique from modeling features in that they contain application-specific data such as bend allowance formula and other material property information. Sheet metal features also have the ability to change their geometric representation with respect to the formed state model.

Because of these characteristics, it is essential to the creation of a valid, usable sheet metal part that you know when and how to use sheet metal features during the construction of your part. Incorrect implementation or non-usage of sheet metal features may result in inaccurate or unproducible flat pattern definitions.

Lesson

1 Introduction to Solid Edge

Solid Edge overview

Solid Edge is an industry-leading mechanical design system with exceptional tools for creating and managing 3D digital prototypes. With superior core modeling and process workflows, a unique focus on the needs of specific industries, and fully integrated design management, Solid Edge guides projects toward an error free, accurate design solution. Solid Edge modeling and assembly tools enable your engineering team to easily develop a full range of products, from single parts to assemblies containing thousands of components. Tailored commands and structured workflows accelerate the design of features common in specific industries and you ensure accurate fit and function of parts by designing, analyzing and modifying them within the assembly model. With Solid Edge, your products come together right first time, every time.

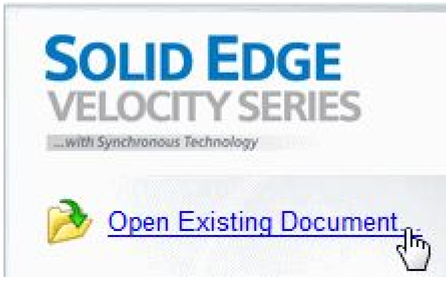
Solid Edge environments

To make the commands you need more accessible, Solid Edge has separate environments for creating parts, constructing assemblies, and producing drawings. Each environment is self-contained. For example, all the commands you need to create a sheet metal part are in the Sheet Metal environment. The environments are tightly integrated, making it easy to move among them to complete your designs.

Design a sheet metal part

Before covering any sheet metal topics, let's observe the simplicity of designing a sheet metal part.

- From the Start menu, click All Programs ® Solid Edge ST3® Solid Edge ST4.
- On the start up screen, click Open Existing Document.

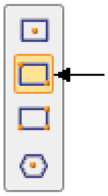


- Open *EZ design.psm* located in the sheet metal training files folder.

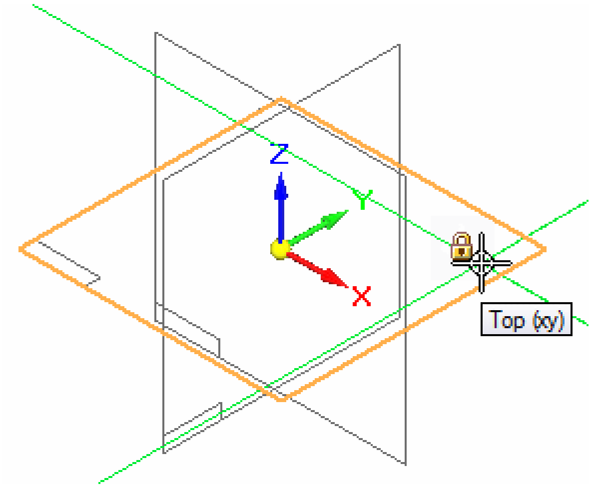
Create the first sheet metal feature

The first feature will be a tab that will be used as a base for building the sheet metal part.

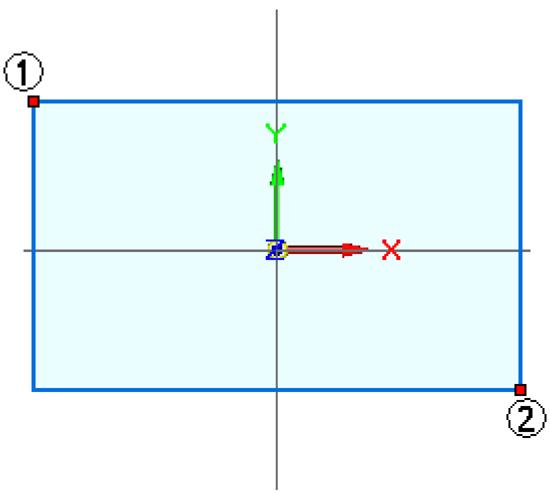
- On the Home tab® Draw group, click the Rectangle by 2 Points command.



- Move the cursor over the plane shown and then click the lock. This locks the sketch plane.

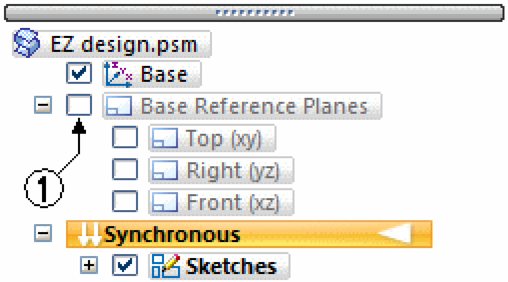


- On the View tab® Views group, choose the Sketch View command .
- Draw the rectangle approximately as shown.

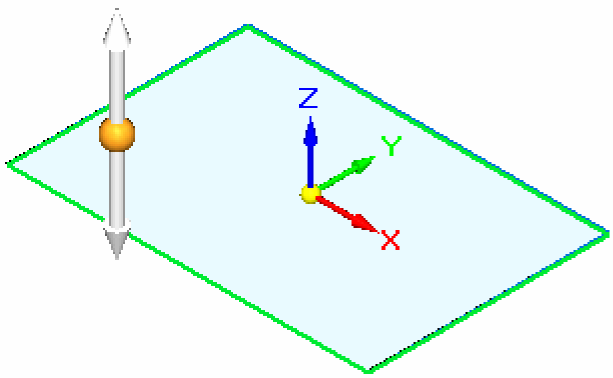


- Press **Ctrl +I** to return to an isometric view.

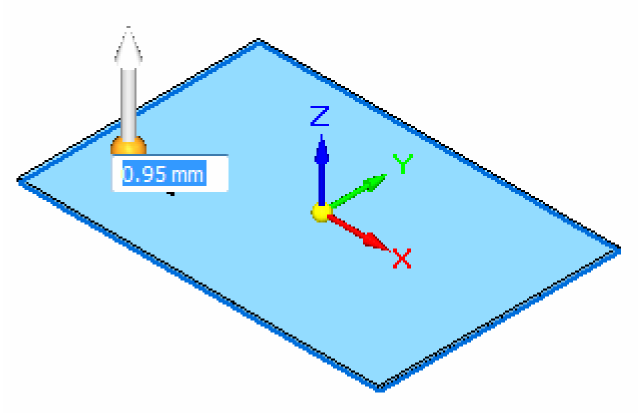
- Turn off the display of the Base reference Planes.

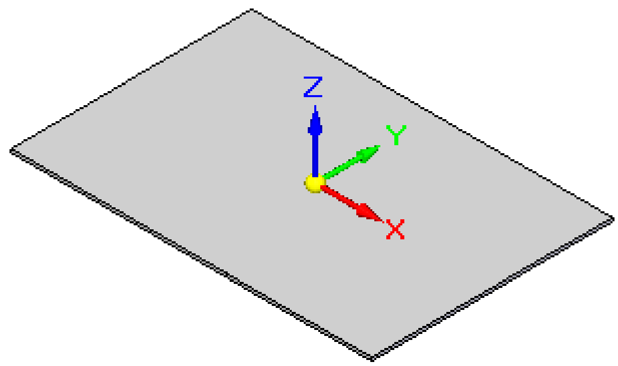


- Click on the region formed by the rectangle and then click on the extrude handle.



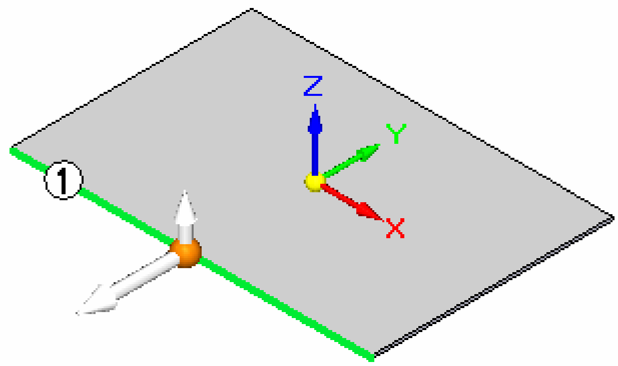
- Click in an open area of the window to accept the material direction and material thickness. If you click the handle, the material direction changes.



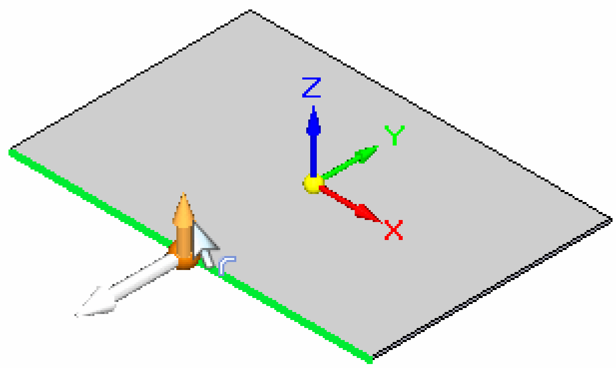


Construct flange features

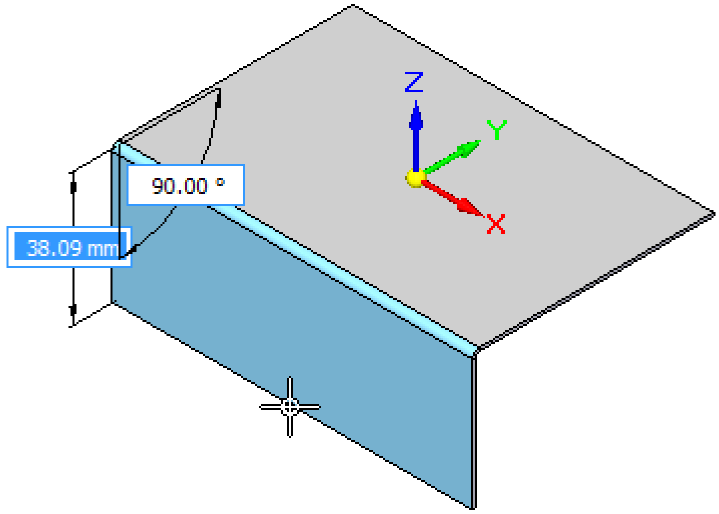
- Select edge (1) and a flange handle appears.

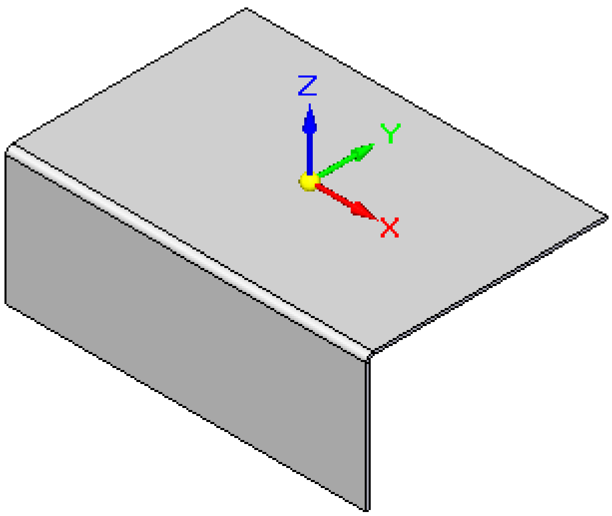


- Click the flange handle axis shown to start the flange creation command.

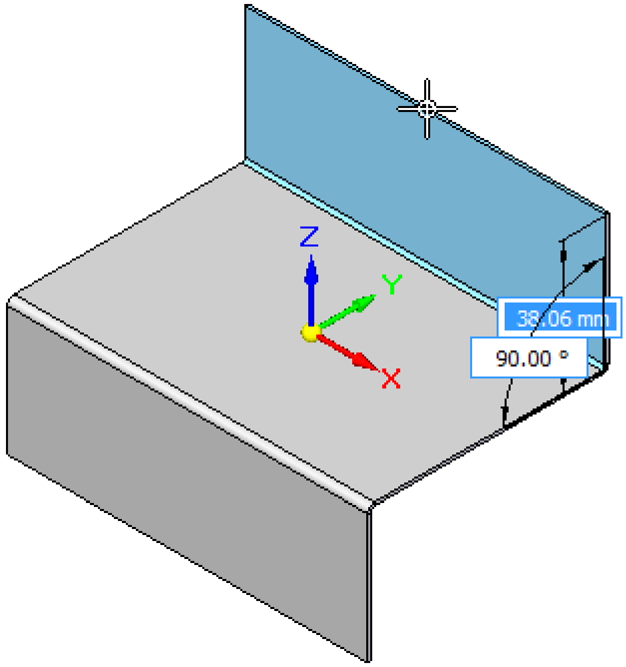


- Move the cursor to the approximate location shown and then click.



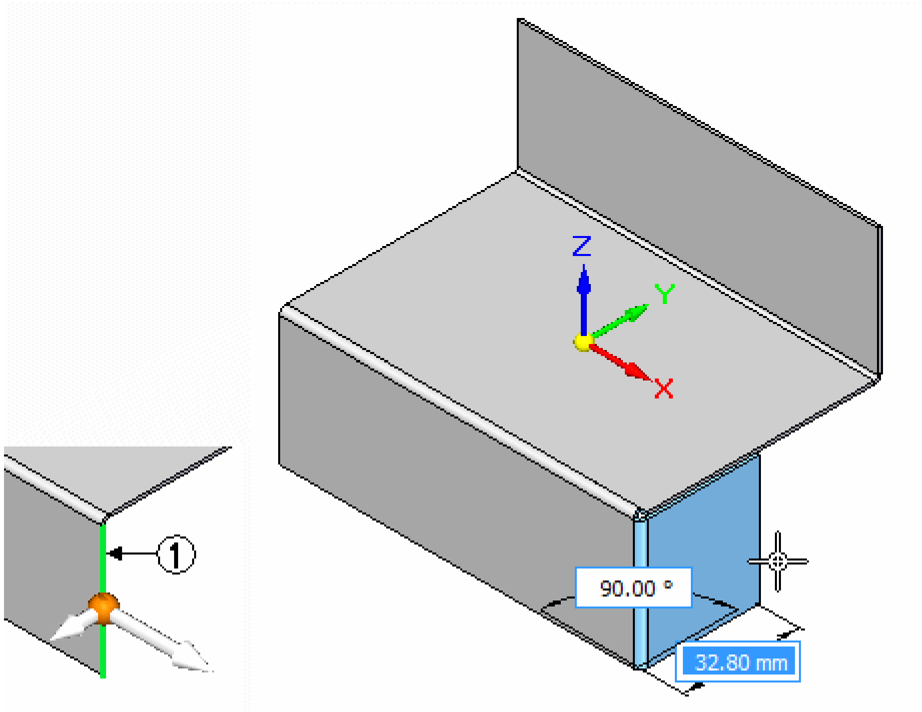


- Create another flange as shown.

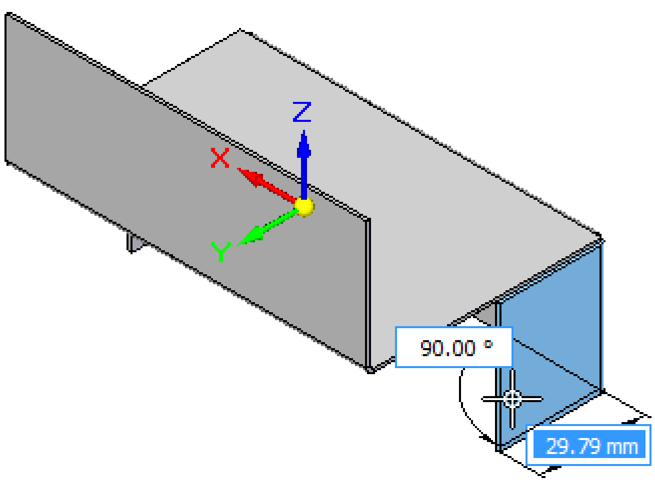


Create additional flanges

- Select edge (1) and create the flange shown.



- Rotate the view and create a similar flange as shown.

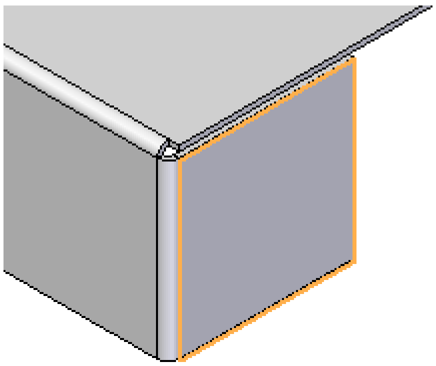


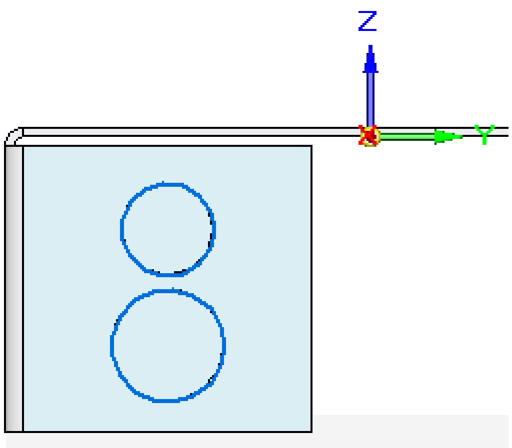
- Press **Ctrl+I** to return to an isometric view.

Insert two holes on the side flanges

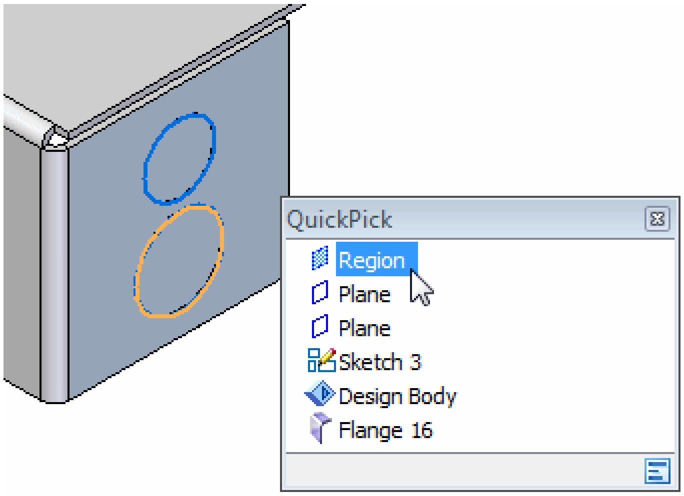
- On the Home tab® Draw group, select the Circle command.

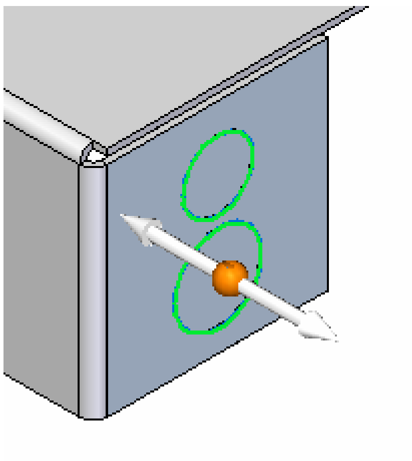
- Draw two circles on the face shown.



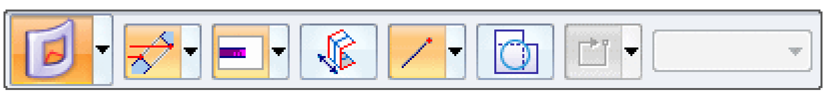


- Select the two regions formed by the circles. Use QuickPick to make sure the first circular region is selected. Press the **spacebar** key to add the second circular region to the select set.

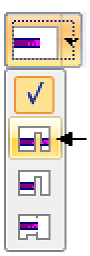




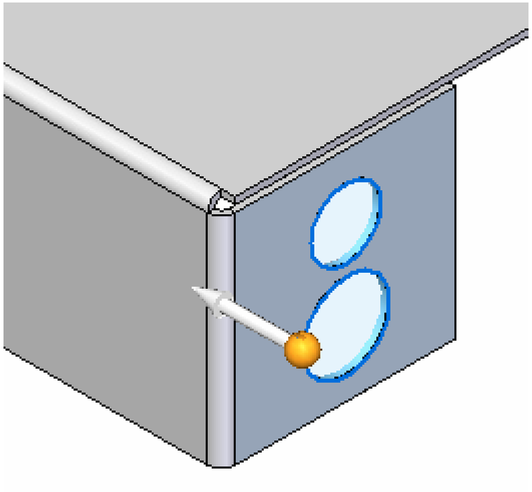
- The system detects that you want to remove material from the sheet metal flange. The Cut command bar appears.

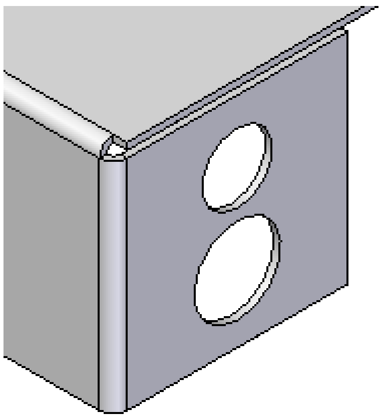


Click the extrude handle axis and then select the **Through All** option on the command bar.



- Click when the direction arrow is as shown.

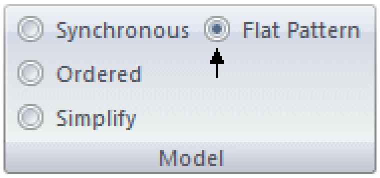




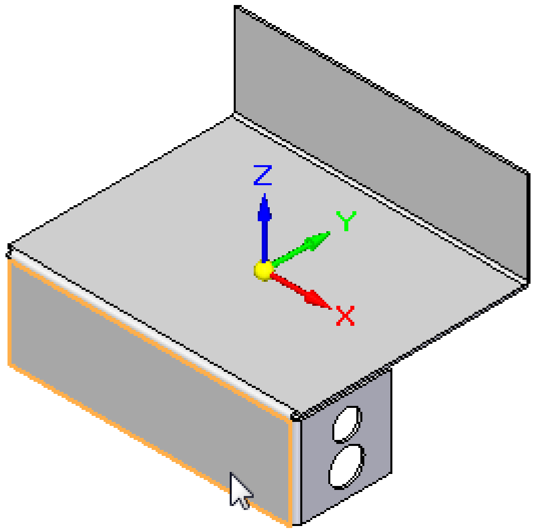
Create a flat pattern of the part

The sheet metal part is complete. Create a flat pattern of the part.

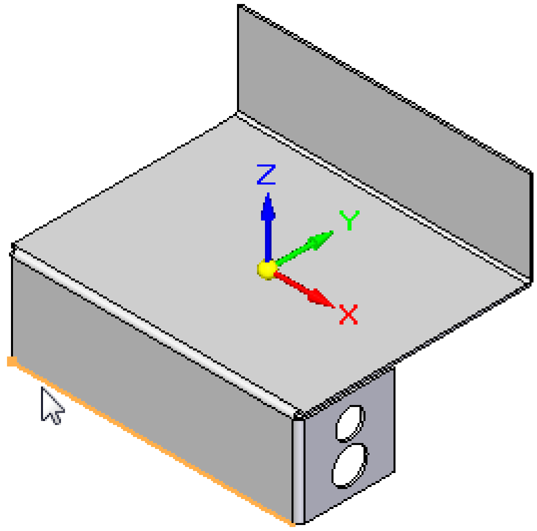
- On the Tools tab® Model group, select Flat Pattern.



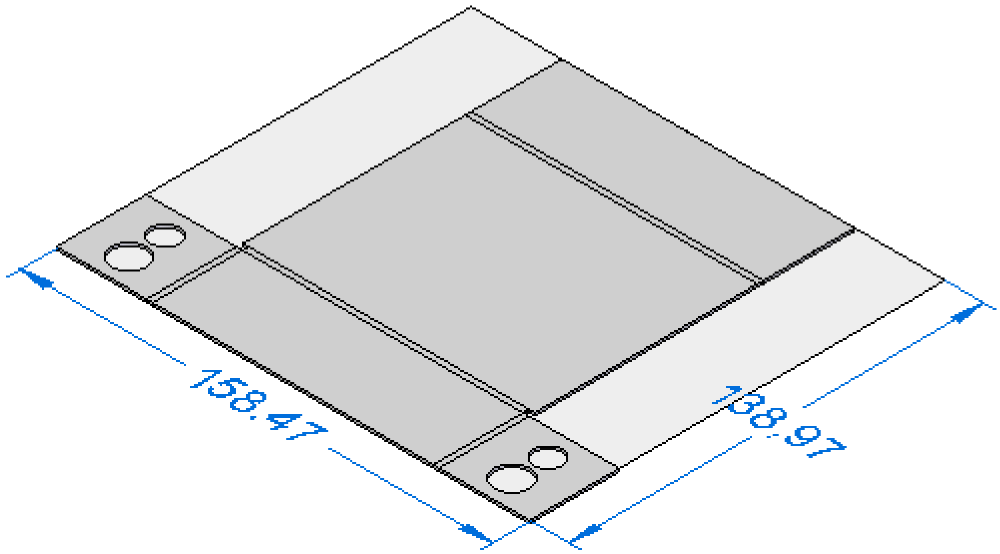
- Select the face shown.



- Select the edge shown.



- Select Finish.



- Save the file. Design is complete.

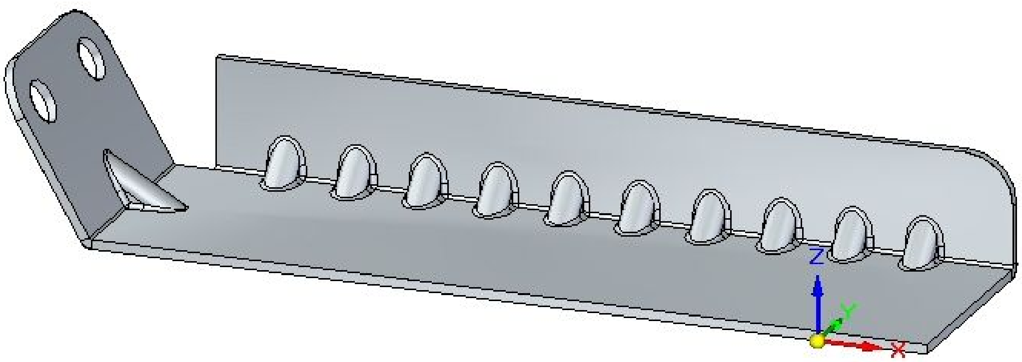
Lesson

2 Sheet metal overview and definitions

Sheet metal overview

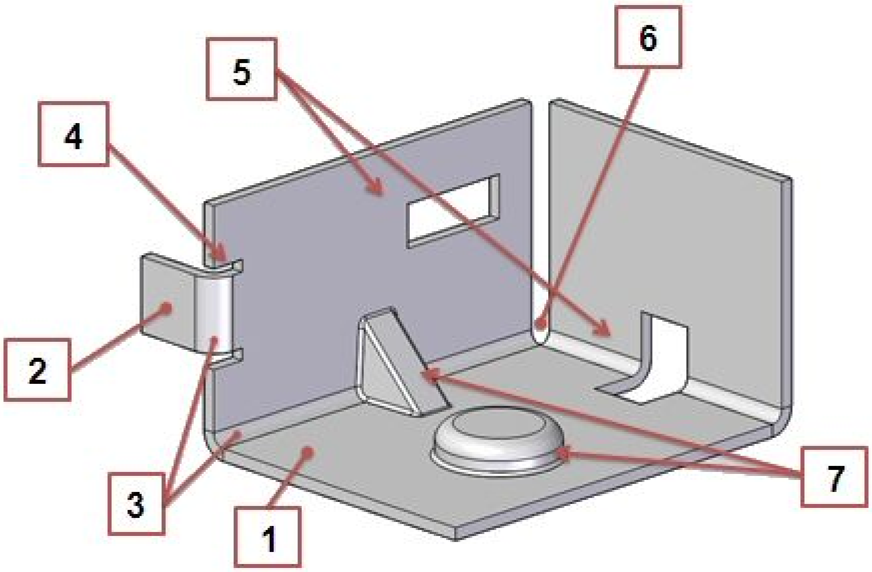
Sheet metal design is governed by the premise that the raw material used to form a sheet metal part is of common stock and of uniform thickness. The sheet metal part is designed in the formed state, but in the manufacturing process, many of the features of the part will be applied to the part before 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

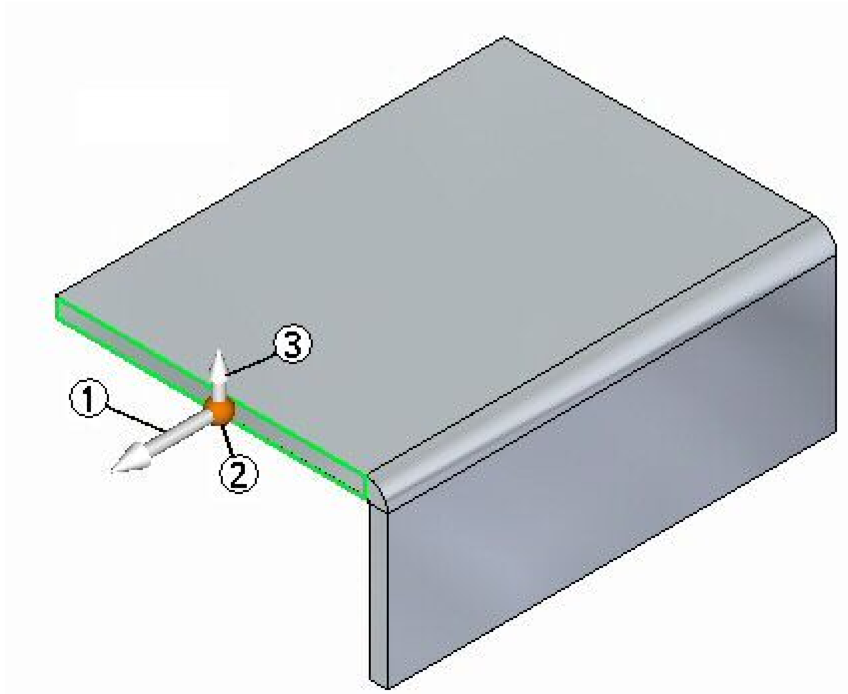
Sheet metal features



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

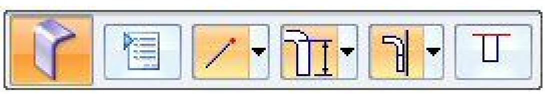
When you select a thickness face, Solid Edge displays a steering wheel unique to the sheet metal application. You can create flanges by selecting the flange start handle. You can use the primary axis, parallel to the layer face, to manipulate the size of the plate.



(1) Primary Axis: Used to move or rotate the thickness face.

(2) Origin

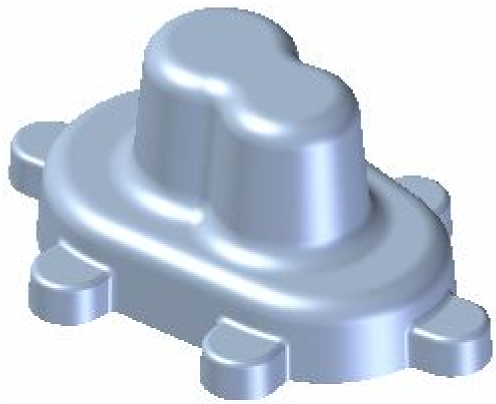
(3) Flange start handle: This opens flange creation options on Quickbar.



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 ST3 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 ST3 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.
- The validity of values entered for parameters like Material Thickness or Bend Radius.

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 ST3/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 = π * (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 Factors from Excel File	Specifies the neutral factor information is extracted from the Excel 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

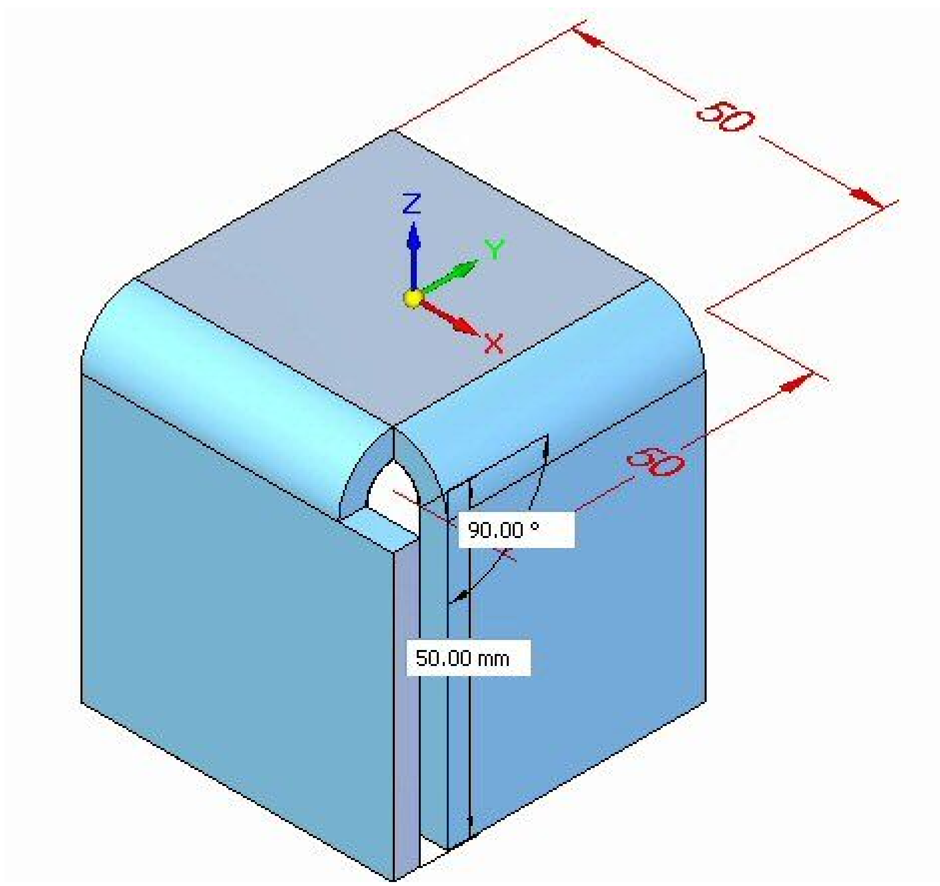
Deletes the existing material or gage in the library file. This button is available when you have selected an existing material or gage.

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.



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.

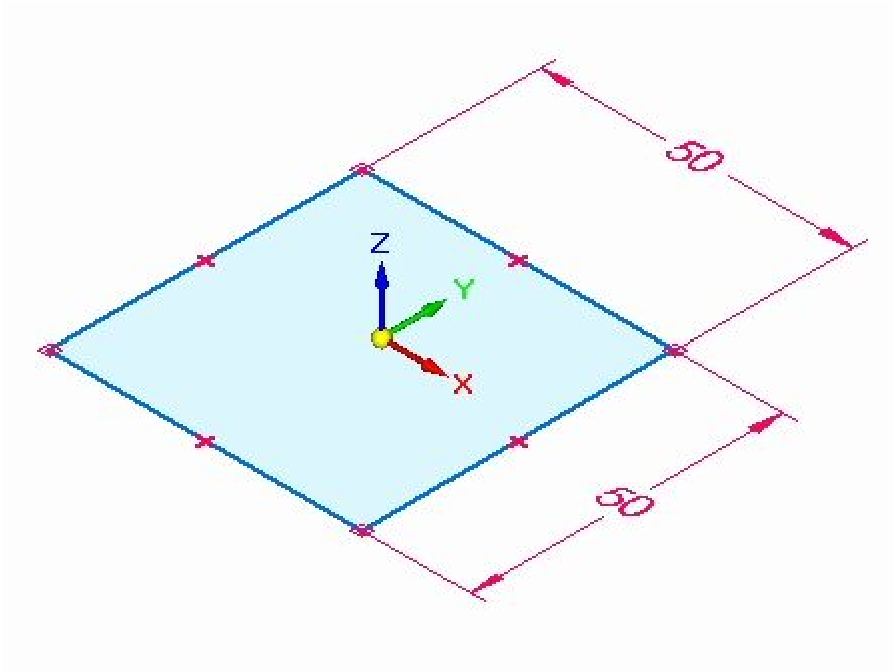
  Note

  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.

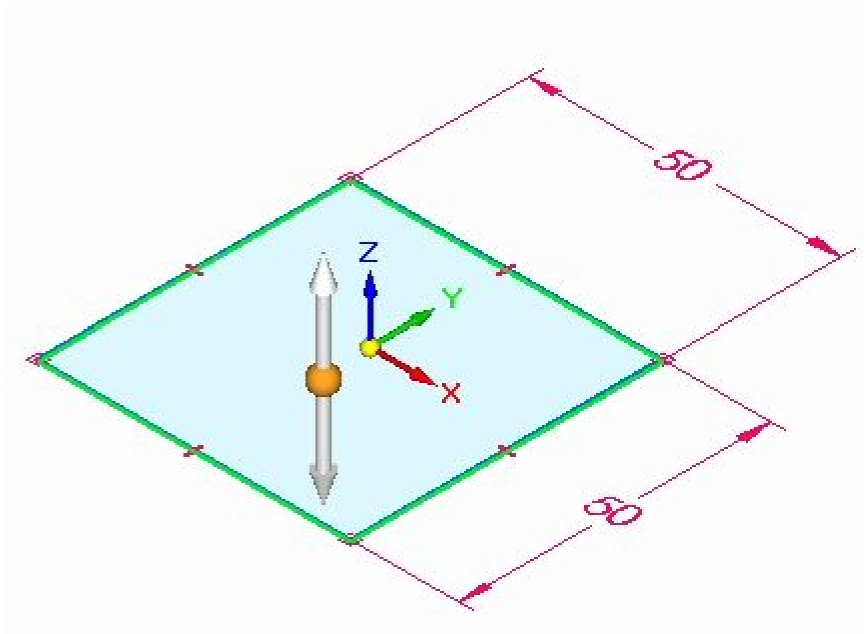
- Proceed to the next step.

Create a tab

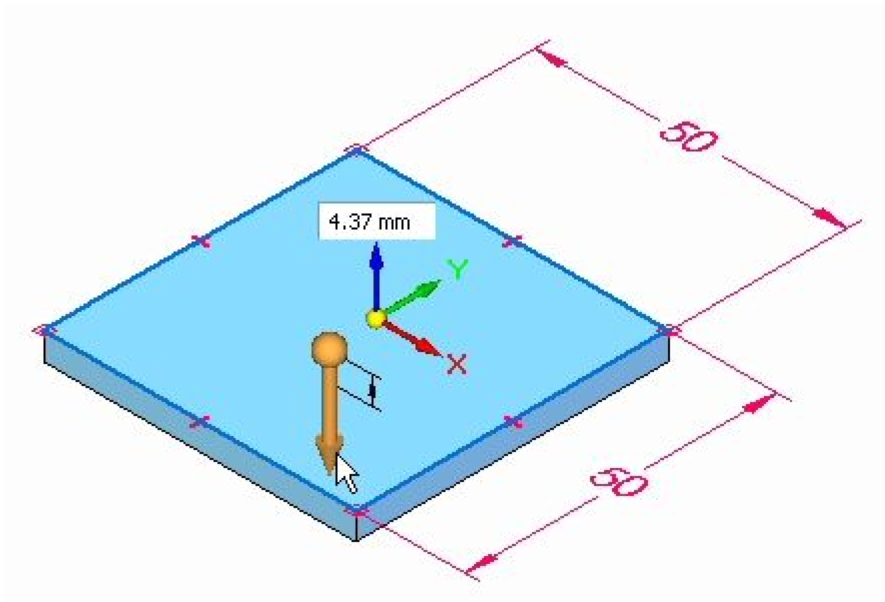
- Sketch a 50 mm square in the X-Y plane.



- Select the region shown.



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



Note

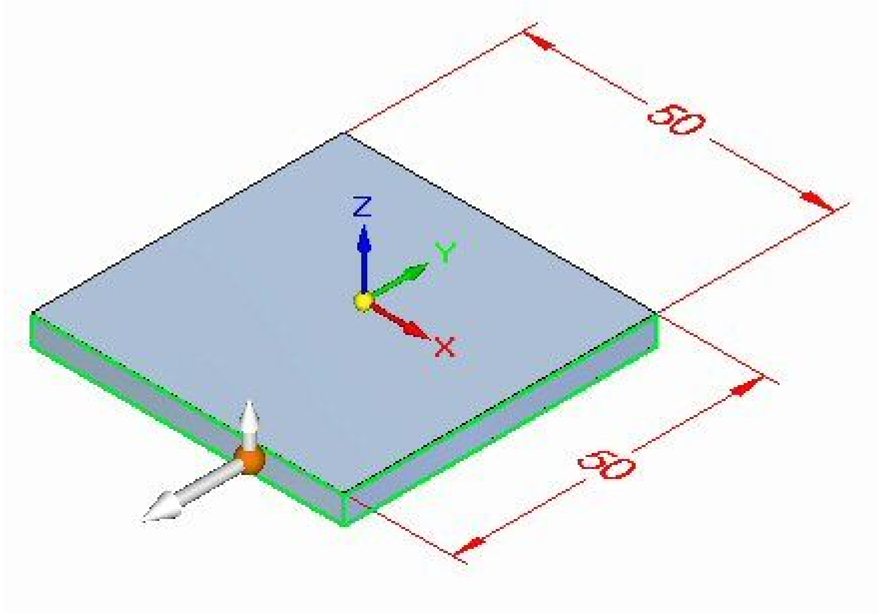
Notice the material thickness corresponds to the gage set in the previous step.

The tab is created.

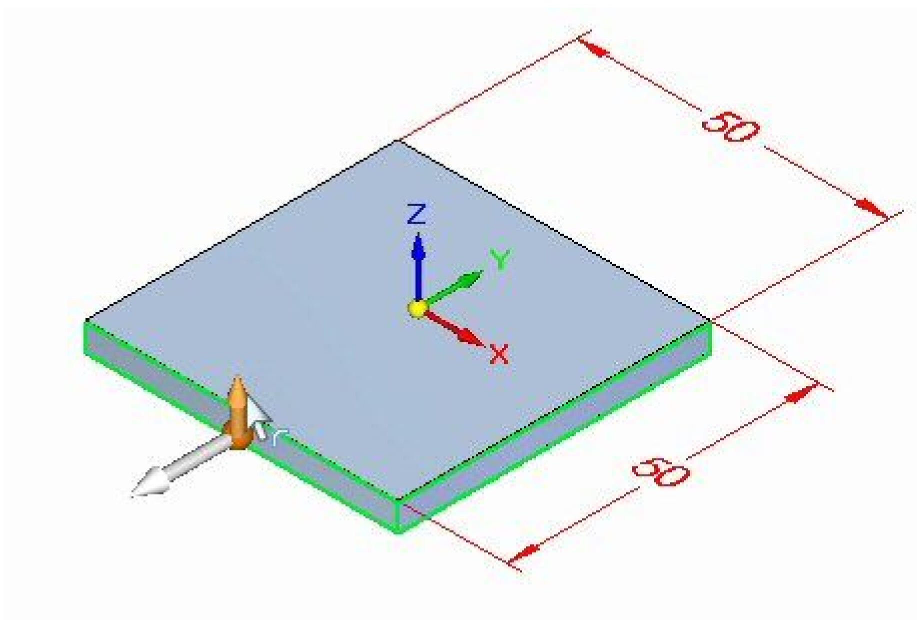
- Proceed to the next step.

Create flanges from the tab

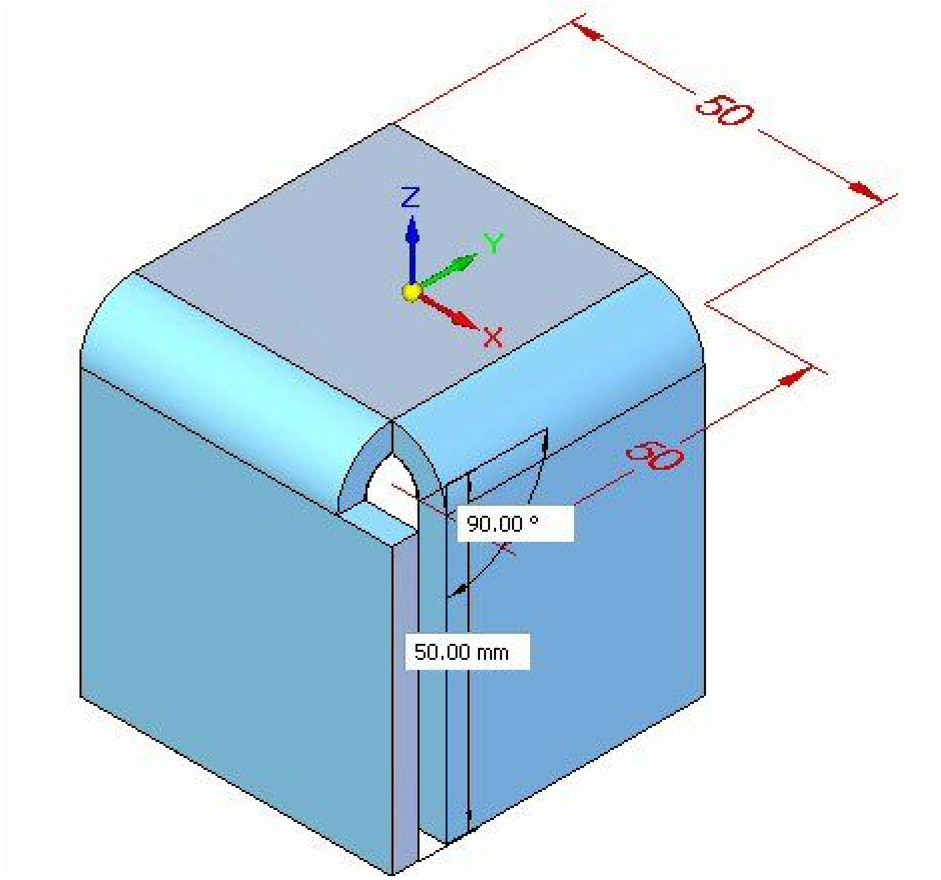
- Select the two thickness faces shown.



- 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.

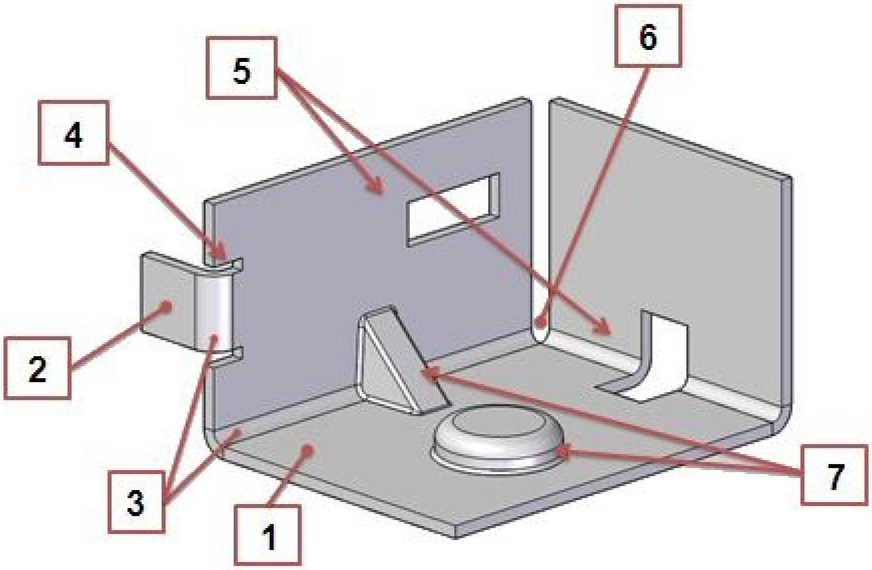
Activity summary

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.

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

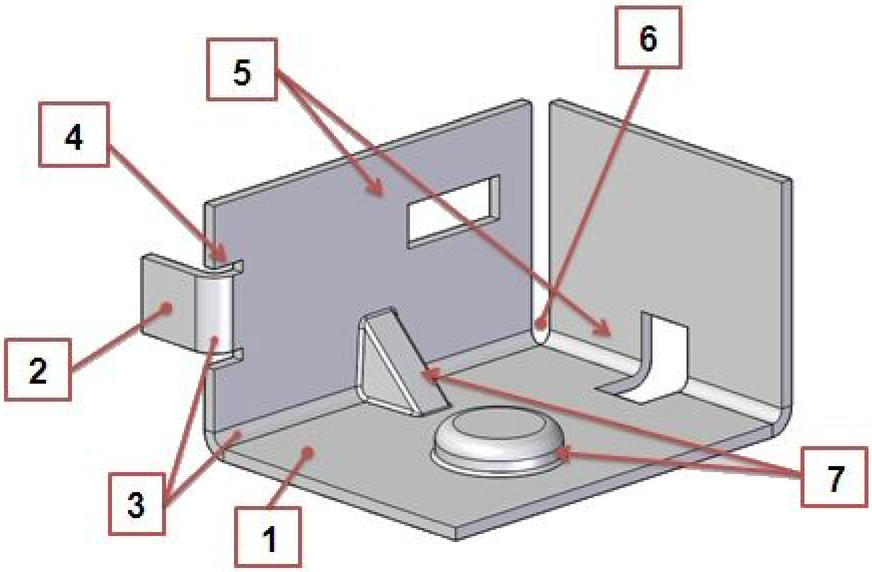
Bend

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.

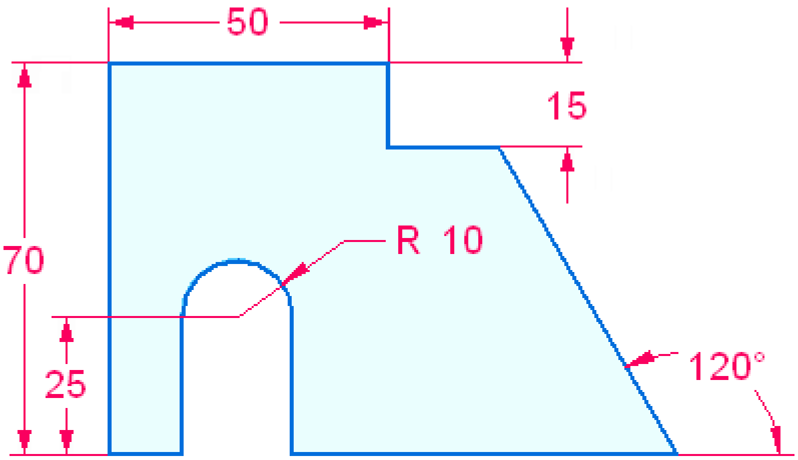
Lesson summary

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

3 *Sketching*

3D sketching overview



2D sketch geometry defines the cross-sectional shape used to create a base solid body or the shape used to create a feature on an existing solid body. Sketches are drawn in 3D on either a planar face or a reference plane. You lock onto a planar face or reference plane to draw sketch geometry.

Both open and closed sketches can be used to create a model feature. A sketch that forms a closed area (from sketch elements or a combination of sketch elements and model edges on the sketch plane) produces a selectable region. When a region is selected, the protrusion feature command is started. To use an open sketch, choose a protrusion command (Extrude or Revolve) in the Solids group which requires a step to define the material side of the open sketch.

Sketches do not drive features. Geometric relationships applied to the sketch geometry do not migrate to the feature created. The system can detect, on the resulting feature, tangent, parallel, coplanar and concentric faces. Dimensional relationships do migrate from the sketch geometry to the edges of the body as a feature is created.

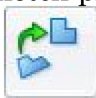
Sketch geometry used in creating a feature is consumed and placed in a “Used Sketches” collector in PathFinder. Any remaining sketch geometry not consumed remains in the “Sketches” collector.

By default, all sketch geometry placed on a sketch plane merge into a single sketch. This is controlled by the sketch option “Merge with Coplanar Sketches”. If separate sketches are required on a sketch plane, the “Merge with Coplanar Sketches” option can be turned off. This sketch option is primarily used in an Assembly Layout design workflow.

Sketch workflow

1. On the Sketching tab® Draw group, choose a sketching command.

2. Start drawing or lock to a sketch plane (reference plane or planar face) to draw sketch geometry on.

3. (Optional) Draw a sketch in the active view orientation or rotate the view normal to the sketch plane by choosing the View tab® Views group® Sketch View command .

4. Draw sketch geometry or perform any sketch related operation (for example: placing relationships, dimensions).

5. Finish or draw another sketch. If the sketch plane is locked and you need another sketch plane, unlock the plane. Repeat steps 2–4.

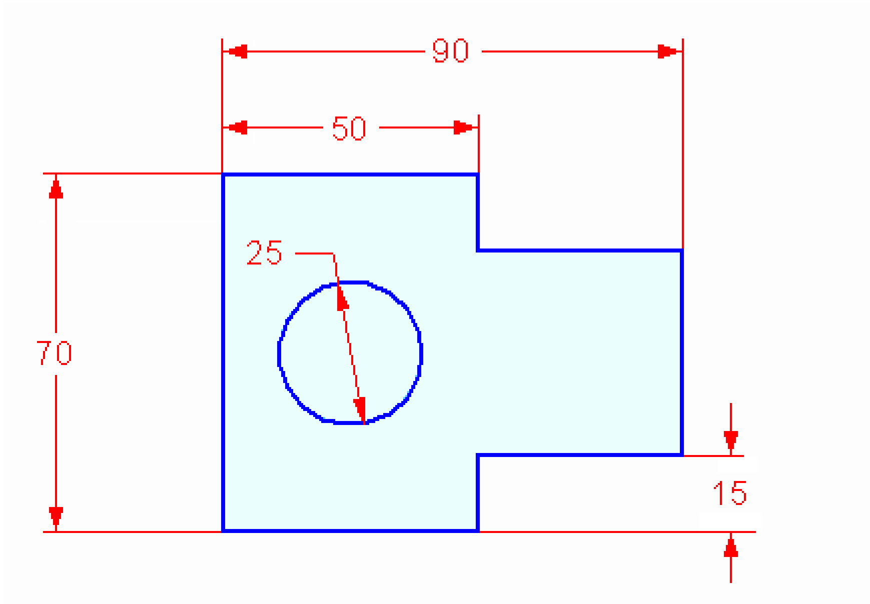
   If the new sketch area is on the same plane, continue sketching geometry.

   Note

   You can only have one sketch on a plane, but the sketch may contain as many regions and separate elements as you need. If separate sketches are required on a sketch plane, turn off the “Merge with Coplanar Sketches” option.

Activity: Draw a simple sketch

This activity guides you through the process of drawing a simple a sketch. You will add relationships and dimensions.



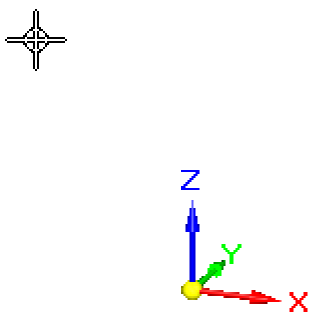
Activity: Draw a simple sketch

Open a part file

- Start Solid Edge.
- Click the  Application button® New® ISO Part.

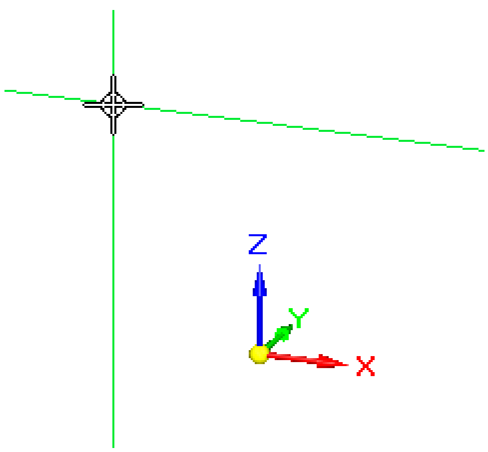
Choose a sketch command

- On the Sketching tab® Draw group, choose the Line command .
- Position the cursor as shown to place first point of line.

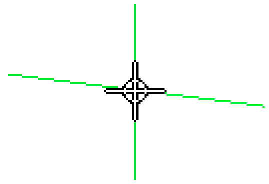


Draw the sketch shape with line segments

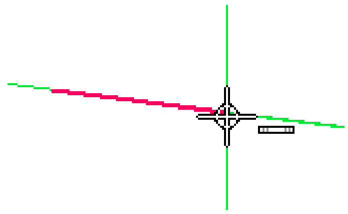
- The line command requires two points to create a line. Click to place the first point of a line.



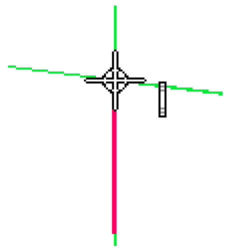
Notice the alignment lines connected to the cursor. These lines assist you in aligning sketch geometry.



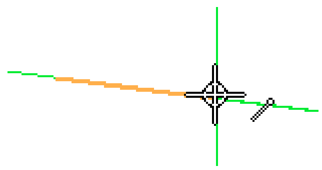
When a line alignment is horizontal, you see the horizontal indicator.



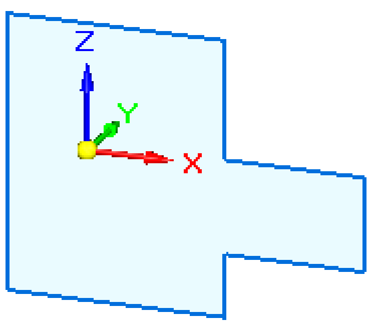
When a line alignment is vertical, you see the vertical indicator.



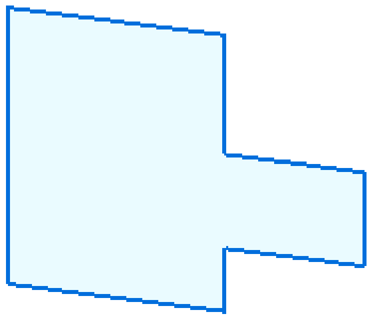
When you are at the endpoint of another line you see the endpoint indicator.



- Draw eight lines to form the basic shape shown. Be sure to make all of the lines horizontal or vertical, but do not worry about the line lengths at this time.

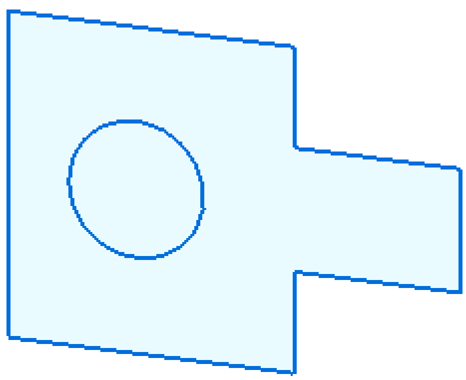


- In PathFinder, click the check box for Base to turn off the display of coordinate systems.

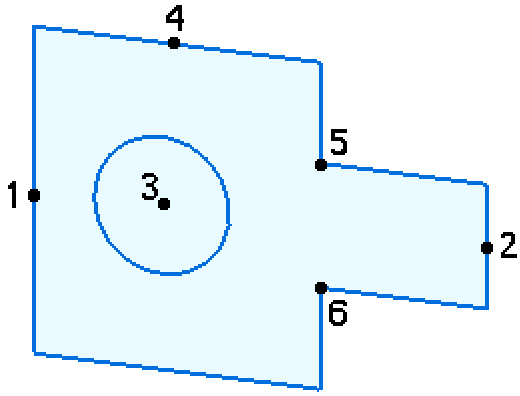


Add a circle to the sketch

- Place a circle  as shown.



Place sketch geometry relationships



- On the Sketching tab® Relate group, choose the Horizontal/Vertical command .

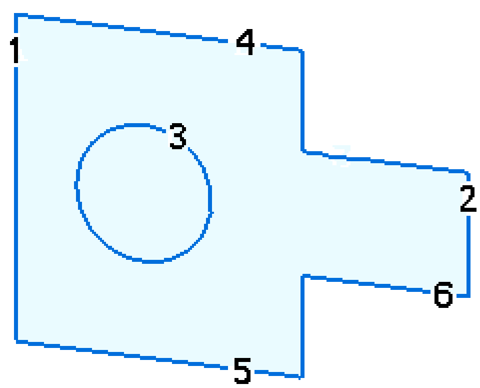
- Align midpoint (2) with midpoint (1). Make sure you get the midpoint indicator before clicking.

- Align circle center (3) with midpoint (1). Make sure you get the center point indicator  before clicking.

- Align circle center (3) with midpoint (4). Make sure you get the center point indicator  before clicking.

- Align point (5) with point (6).

Place sketch dimensions



Numbers denote the select location for dimensioning the sketch elements.

- On the Sketching tab® Dimension group, choose the Smart Dimension command .

- Dimension the circle by clicking at (3).

  In the dimension value edit box, type 25.

- Dimension the length of line (4) by clicking at (4).

  In the dimension value edit box, type 50.

- Dimension the length of line (1) by clicking at (1).

  In the dimension value edit box, type 70.

- On the Sketching tab® Dimension group, choose the Distance Between command .

- Dimension the distance between line (1) and line (2) by clicking line (1) and then line (2).

  In the dimension value edit box, type 90.

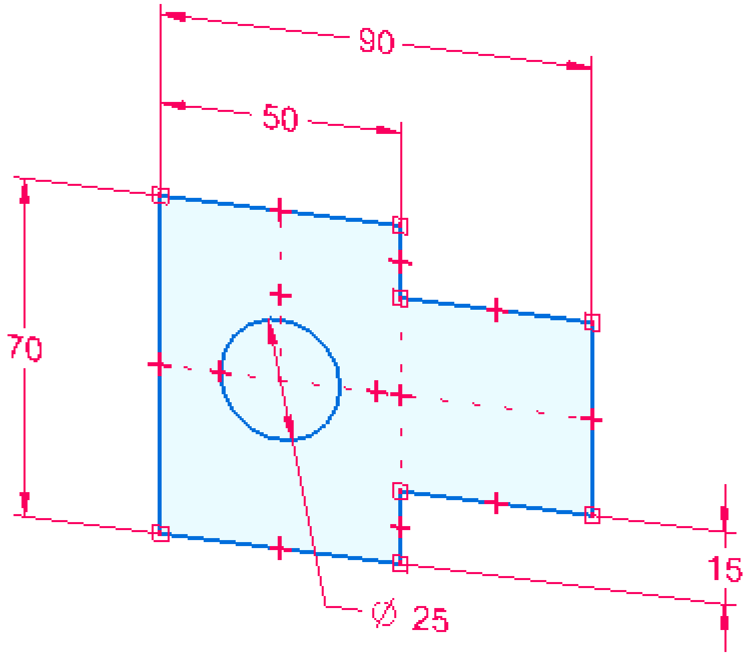
- Dimension the distance between line (5) and line (6) by clicking line (5) and then line (6).

  In the dimension value edit box, type 15.

Sketch complete

The sketch is complete. Turn on the relationships handle display to see the sketch relationships.

- On the Sketching tab® Relate group, choose the Relationship Handles command.



- Turn off the relationship handles.

Summary

In this activity you learned how to create a sketch. Dimensional and geometric relationships can be added at any time during the sketch creation. Extruded or revolved features in Solid Edge require sketches for creation.

Practice

- Try changing dimensions and adding sketch geometry for practice. Otherwise, close the file and do not save.

Sketch plane locking

Many commands in Solid Edge use a 2D plane for placement of geometry in 3D model space. For example, when drawing 2D sketch elements, such as lines, arcs, and circles, the 2D elements reside on a coordinate system plane, reference plane, or planar face on the model. This 2D plane is called the sketch plane. Only one sketch plane is available at a time.

There are two methods for locking input to the sketch plane:

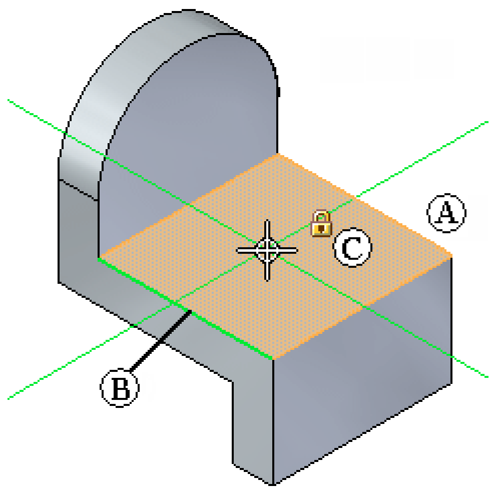
- Automatic locking, where the active command locks the sketch plane for you, and unlocks the sketch plane when you restart the command, or you start another command.

- Manual locking, where you lock the sketch plane, and unlock it later yourself.

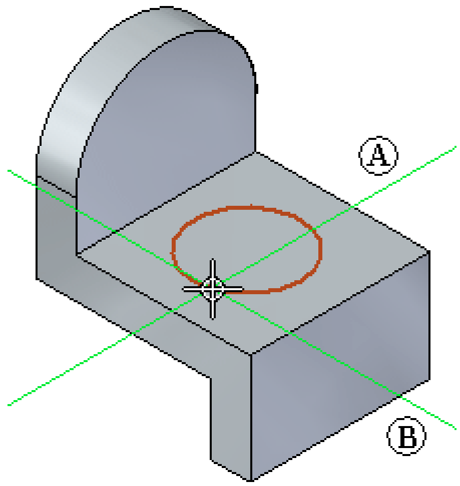
Sketch plane locking makes it easy to draw on several reference planes or planar faces quickly.

Automatic sketch plane locking

When you start a command that uses a sketch plane, and then position the cursor over a reference plane or planar face, the plane or face highlights (A), and an edge on the plane (B) is highlighted to indicate x-axis of the current sketch plane. The alignment lines, which extend outward from the cursor, also align themselves to the plane under the cursor. A lock symbol (C) is also displayed if you want to manually lock the sketch plane, which is discussed later.

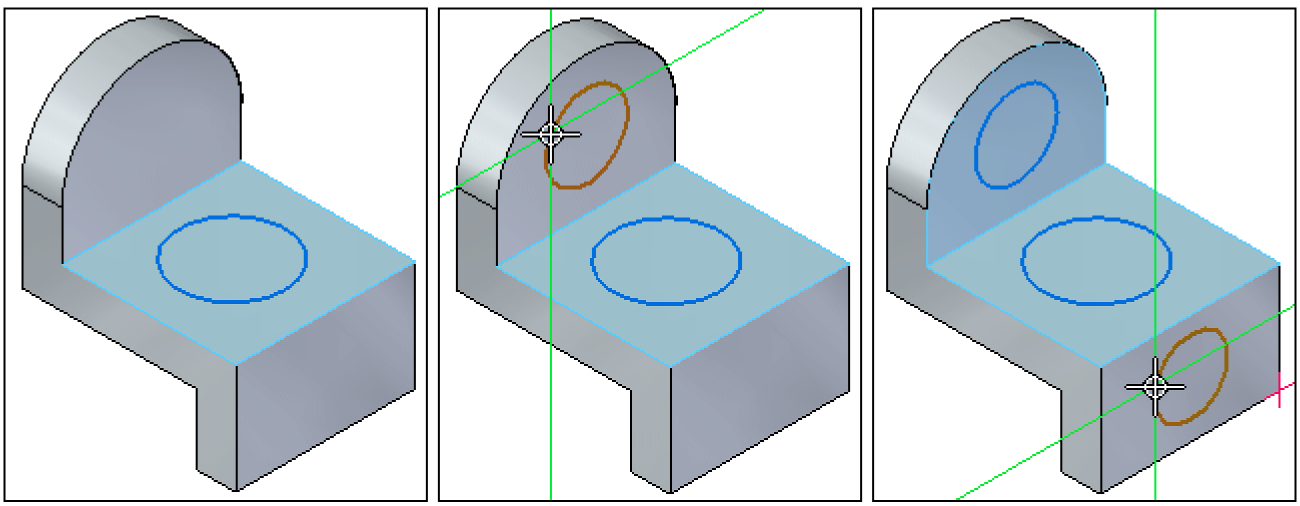


When you click to position the starting point for the sketch element, the sketch plane is automatically locked to the highlighted plane or face. The alignment lines (A) (B) remain displayed as you draw to indicate the current sketch plane's X and Y axes.



The sketch plane remains locked until you right-click to restart the current command, or start another command. This ensures all sketch input lies on the current sketch plane.

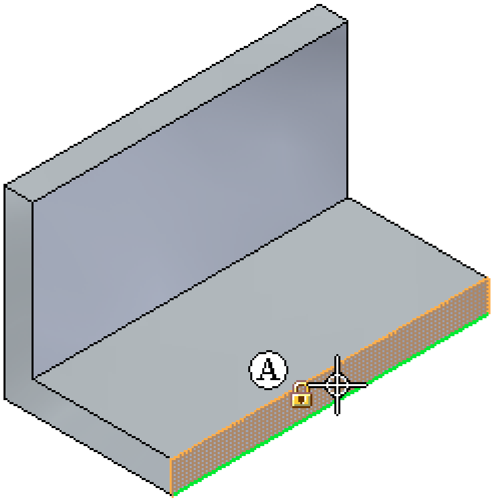
Sketch plane locking makes it easy to draw on several faces of the model quickly. For example, after drawing the first circle, you can right-click to restart the command, then draw a circle on a second face, right click again, and draw a circle on a third face.



Manual sketch plane locking

You can also manually lock the sketch plane. This is useful when the sketch geometry is complex or will extend beyond the outer edges of the planar face or reference plane on which you want to draw.

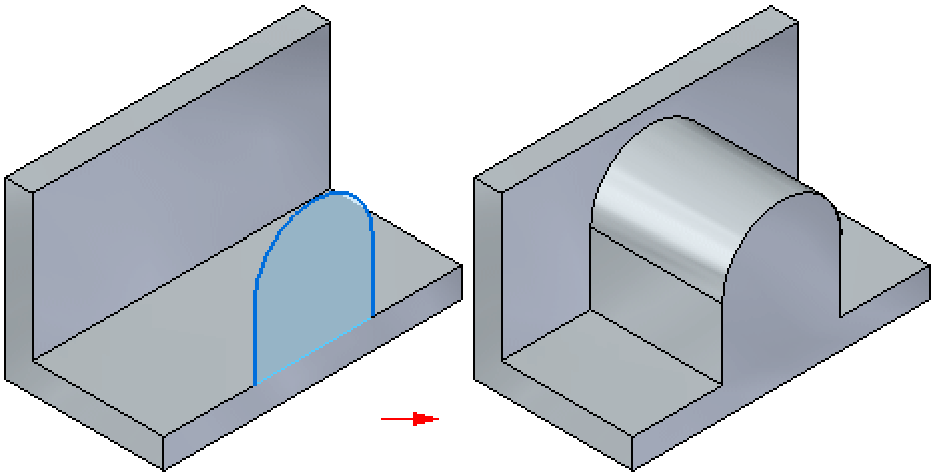
When you are in a command that supports manual sketch plane locking, a lock symbol is displayed near the cursor (A) when you are over a planar face or reference plane. You can click this symbol to manually lock the plane.



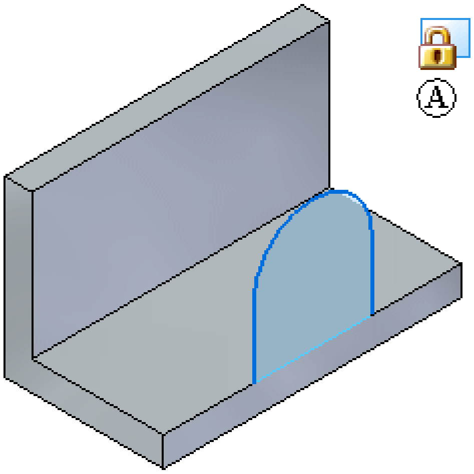
Tip

You can also lock and unlock the sketch plane by pressing the F3 key when you are in any command that supports sketch plane locking.

The sketch plane remains locked regardless of the cursor position until you manually unlock the plane. This makes it easy to draw beyond the outer edges of the planar face.



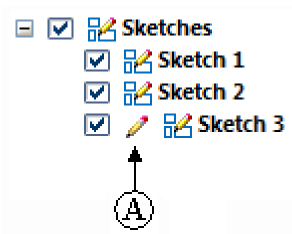
When the sketch plane has been locked manually, a locked plane indicator symbol (A) is displayed in the top-right corner of the graphics window.



When you want to unlock the sketch plane, you can click the locked plane indicator symbol in the graphics window to unlock the plane, or you can press the F3 key.

Plane locking and PathFinder

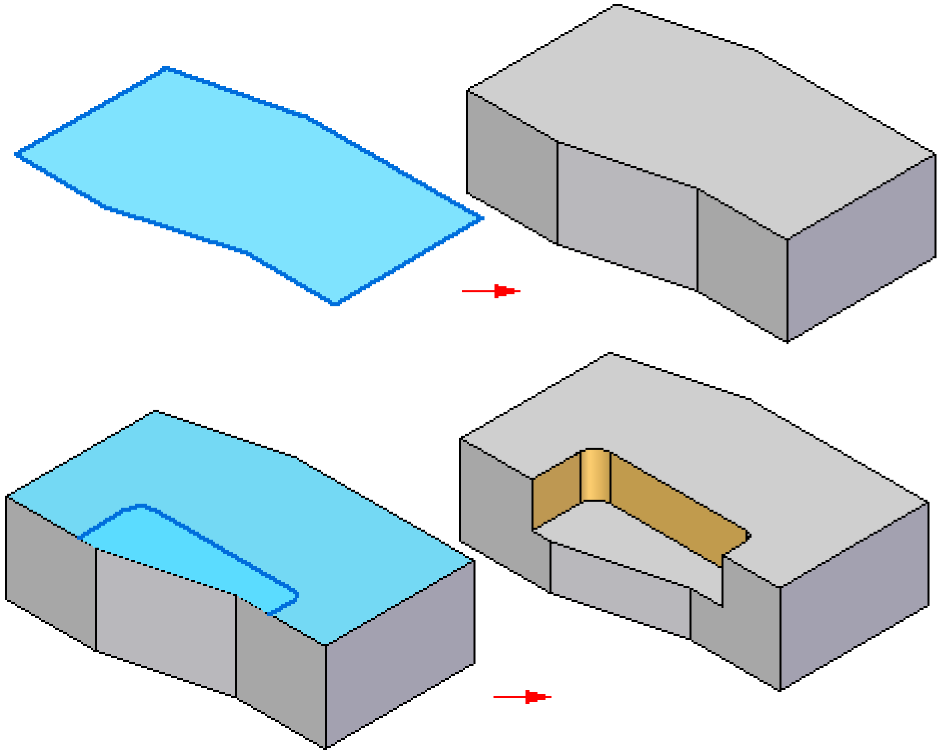
Whether you lock the sketch plane automatically or manually, a locked plane indicator (A) appears in PathFinder adjacent to the sketch which is locked.



If there are existing sketches in the model, you can lock and unlock the sketch plane using the Lock Sketch Plane command on the PathFinder shortcut menu when your cursor is over a sketch entry.

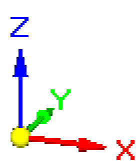
Drawing synchronous sketches of parts

You draw synchronous sketches to establish the basic shape requirements of a part before you construct any features. You can draw a synchronous sketch on a principal plane of the base coordinate system, a planar face on the model, or a reference plane. You can then use these sketches to create sketch-based features, such as extruded features which add or remove material.

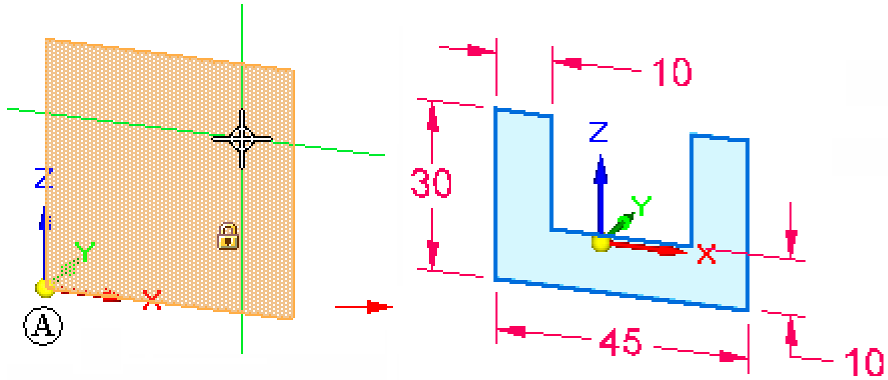


Visual sketching aids

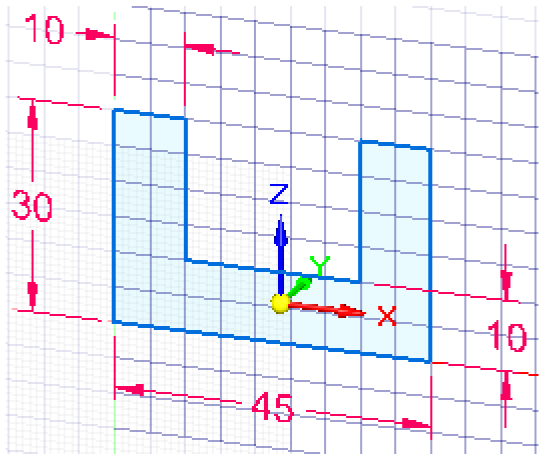
There are a variety of visual sketching aids available to you. The triad in the center of the graphics window is the base coordinate system.



The principal planes on the base coordinate system are typically used to draw the first sketch for the base feature on a new part.



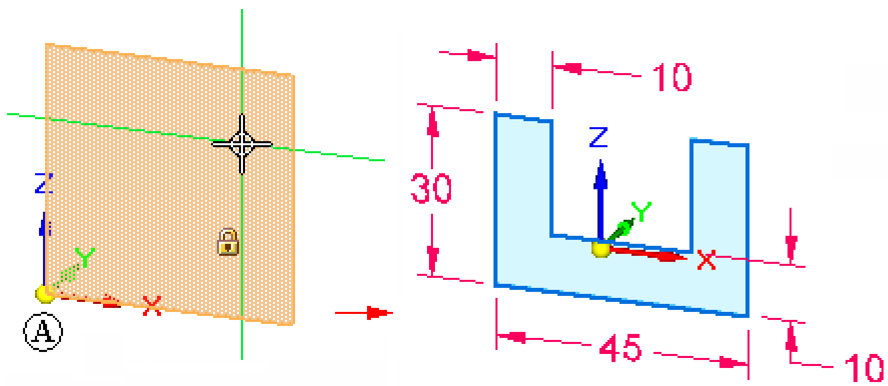
You can also independently display the sketching grid, alignment lines, and coordinate readouts using the Grid Options command.



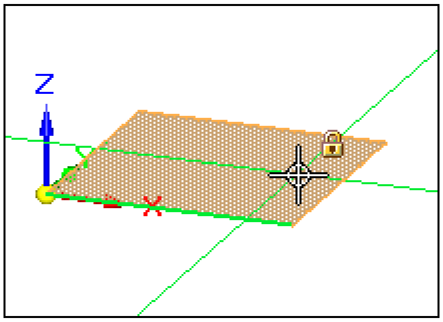
Getting started with sketching

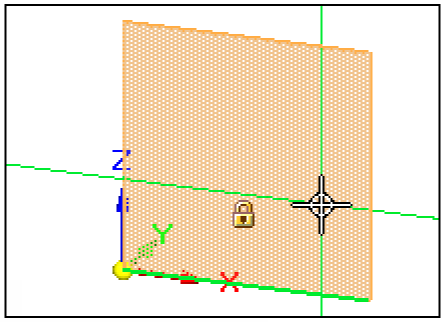
Getting started with sketching is easy. When you sketch elements, they will go on the coordinate system plane, planar face, or reference plane that is directly under your cursor when you start placing the element.

When starting a new part, you would typically draw a sketch on one of the three principal planes of the base coordinate system. For example, you can draw the first sketch for a new part on the XZ principal plane of the base coordinate system (A).



You can see which plane of the coordinate system you will draw on because the plane under the cursor highlights, and the alignment lines, which extend out from the cursor, adjust dynamically depending on what plane your cursor is over.





When you click to define the first endpoint of an element, such as a line, sketch input is locked to the current plane.

Note

- If there is not a coordinate system plane, model face, or reference plane under your cursor, the element will fall on one of the three principal planes of the document. The system will automatically choose the one that is flattest to the view.
- See the Help topic, Start a sketch, to learn how to get started.

Sketch plane locking

Many of the sketching commands require a locked sketch plane for placement of 2D geometry in 3D model space.

There are two methods for locking the sketch plane:

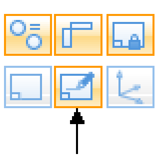
- Automatic locking, where the active command locks the sketch plane for you, and unlocks the sketch plane when you start another command. This makes it easy to get started.
- Manual locking, where you lock the sketch plane, and unlock it later yourself. This is useful for complex sketches or for sketches where the sketch geometry extends beyond the boundary of the sketch plane.

Note

To learn more, see: Sketch plane locking.

Synchronous sketches locked to faces

A synchronous sketch drawn on a model face is automatically locked to the face. As the face moves, the sketch moves with the face. By default, the Live Rules option “Maintain Sketch Planes” is on.

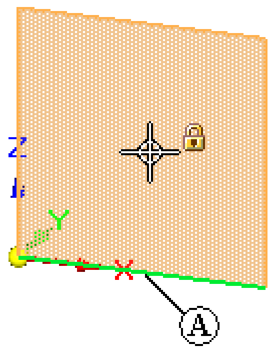


To unlock the sketch from the model face, turn off the “Maintain Sketch Planes” option in Live Rules.

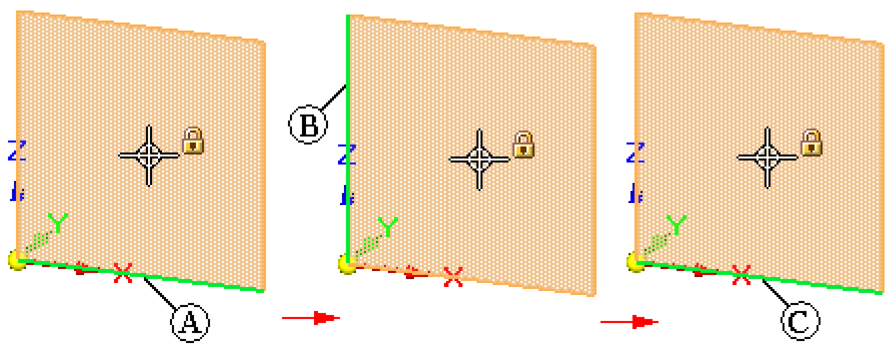
If a sketch is drawn on a model face that is coplanar to a base reference plane, the sketch is not locked to the model face.

Sketch plane X-axis orientation

When you highlight a coordinate system plane, planar face, or reference plane on which you want to draw a sketch, a default X-axis orientation is displayed automatically (A).

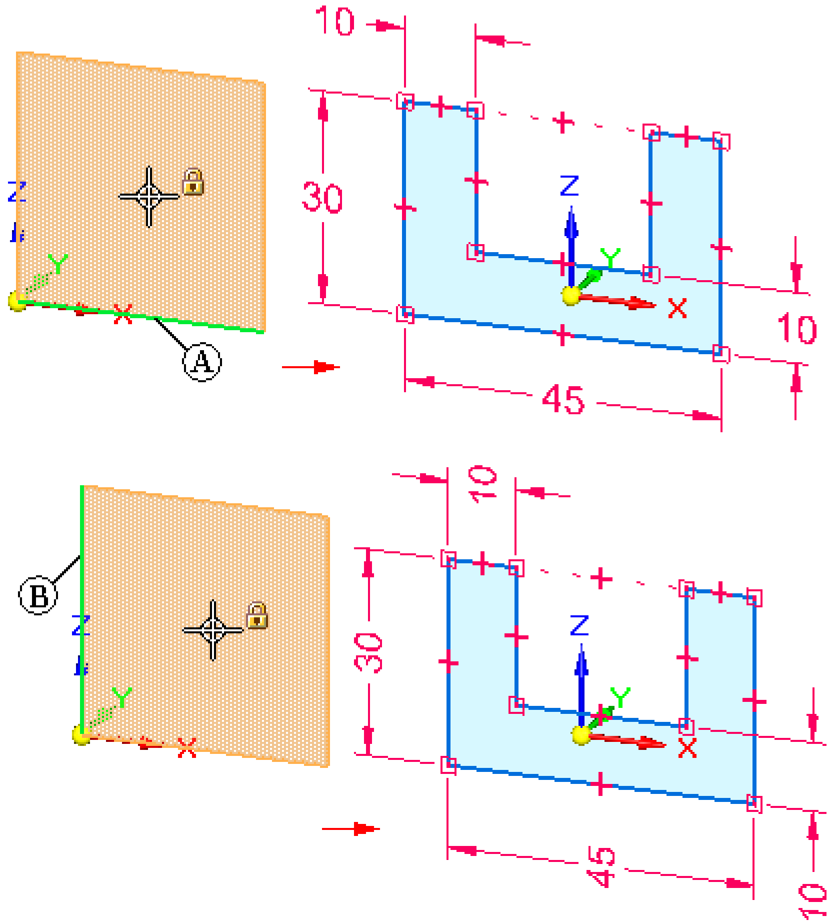


While you are defining the sketch plane and the default X-axis is highlighted (A), you can use the shortcut keys to change the X-axis orientation. For example, you can press the N key to select the next linear edge (B), or the B key to select the previous linear edge (C).



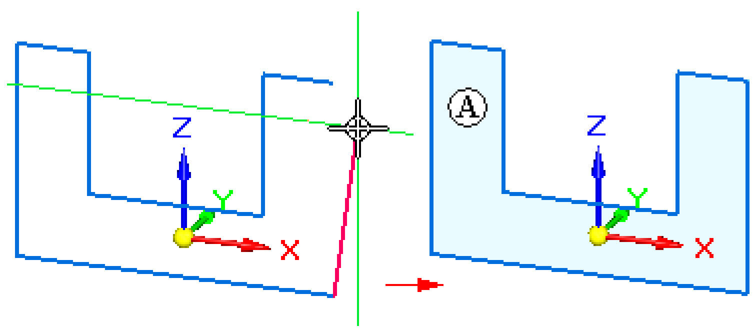
The valid shortcut keys for defining the X-axis orientation of a sketch plane are displayed in PromptBar when you are defining the sketch plane.

The X-axis orientation (A) (B) of a sketch controls the dimension text alignment for dimensions, and determines the horizontal and vertical axes for horizontal and vertical relationships.

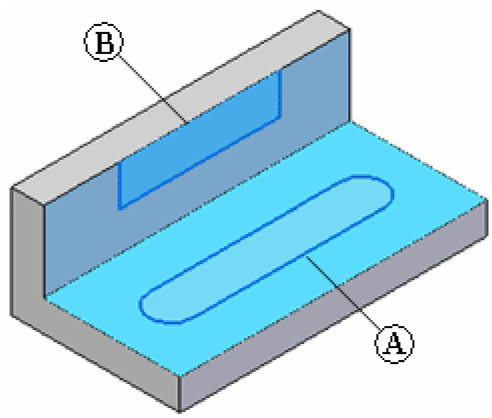


Sketch regions

In a part or sheet metal document, when you draw 2D sketch elements that form a closed area, the closed area is automatically displayed as a sketch region (A). When working in a shaded view, the closed region also displays as shaded.



In a part or sheet metal document, you can use sketch regions to construct features using the Select tool. Sketch regions are formed automatically when a series of sketch elements close on themselves (A), or when sketch elements and one or more model edges form a closed area (B).



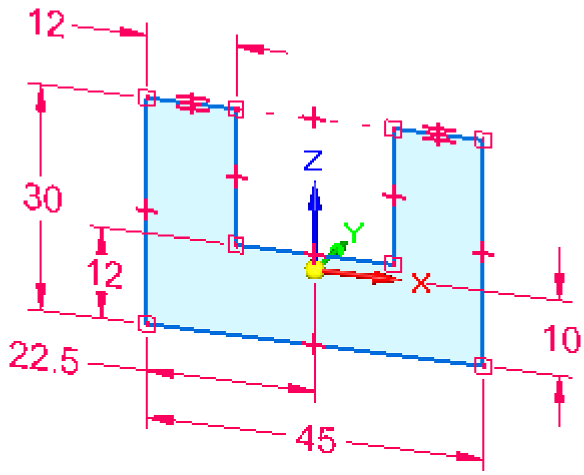
As you draw, you may want to disable sketch regions. You can do this by clearing the Enable Regions command, which is located on the shortcut menu when you select a sketch in PathFinder.

You can use the Enable Regions command to turn region selection on again.

The Enable Regions command is not available in an assembly document.

Adding dimensions and geometric relationships

You can add dimensions and geometric relationships to control the size, shape, and position of the sketch elements. You can also place dimensions and geometric relationships relative to the primary axes of the coordinate system. This can be especially useful for symmetric parts during later design modifications. For example, the 10 mm and 22.5 mm dimensions were placed relative to the X and Z axes of the base coordinate system.



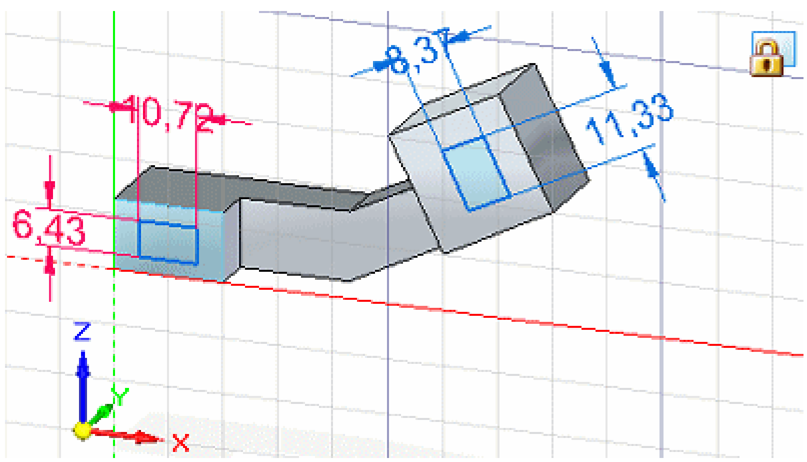
Note

You can display and hide geometric relationships using the Relationship Handles command.

You can also define functional relationships using the Variables command.

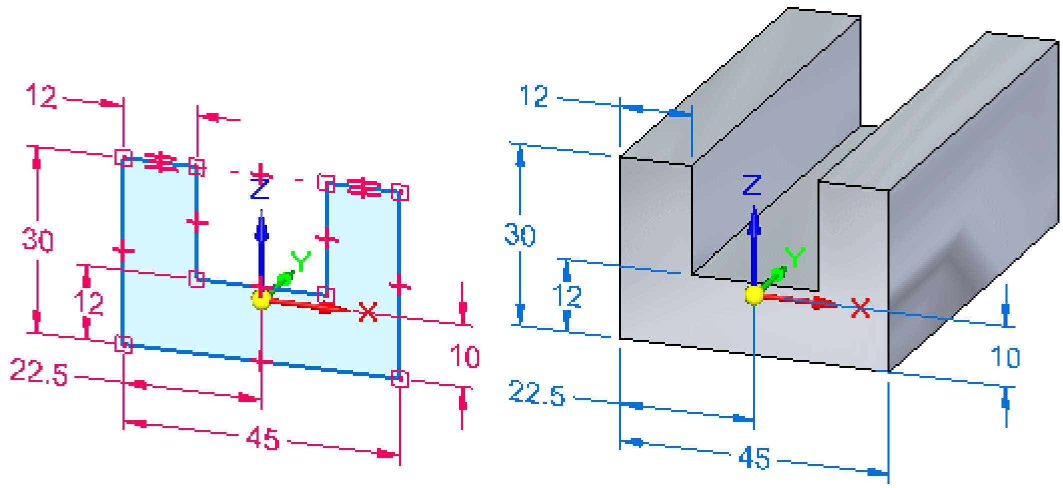
Keeping dimensions horizontal and vertical to the sketch geometry

To keep dimensions horizontal and vertical to the sketch geometry, you can move the sketch plane origin and reorient the sketch plane X-axis using the Reposition Origin command on the Sketching tab. This makes it possible to draw and dimension on different coplanar faces in the same sketch, yet keep dimension text and relationships oriented to an edge on the face, as shown.



Using sketches to construct features

When you use a sketch to construct a feature in a part or sheet metal document, by default, the sketch elements are automatically consumed and transferred to the Used Sketches collection in PathFinder and the dimensions on the sketch are automatically migrated to the appropriate model edges when possible.



Note

After you construct a feature in a synchronous model, the original sketch geometry does not drive the feature.

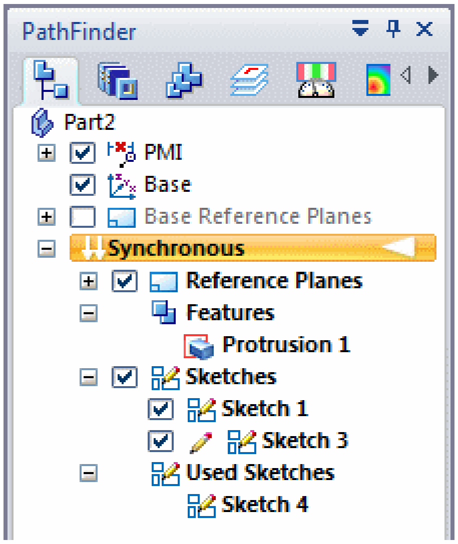
You can use the Migrate Geometry and Dimensions command on the shortcut menu when a sketch is selected in PathFinder to control whether sketch elements are consumed and dimensions are migrated when you construct features using the sketch.

Editing sketches

You can move and resize sketch elements using the Select tool. You also can edit sketch elements using commands such as Extend To Next, Trim, Mirror, Scale, Rotate, Stretch, and so forth. With these commands, you select the command first, then follow the prompts to edit the sketch elements you want.

Sketching and PathFinder

The sketches you draw are listed in PathFinder. PathFinder also lists the base coordinate system, PMI dimensions, the base reference planes, features you construct, used sketches, and so forth.



You can display or hide individual sketches or all the sketches in the document using the check box options in PathFinder and commands on the PathFinder shortcut menu.

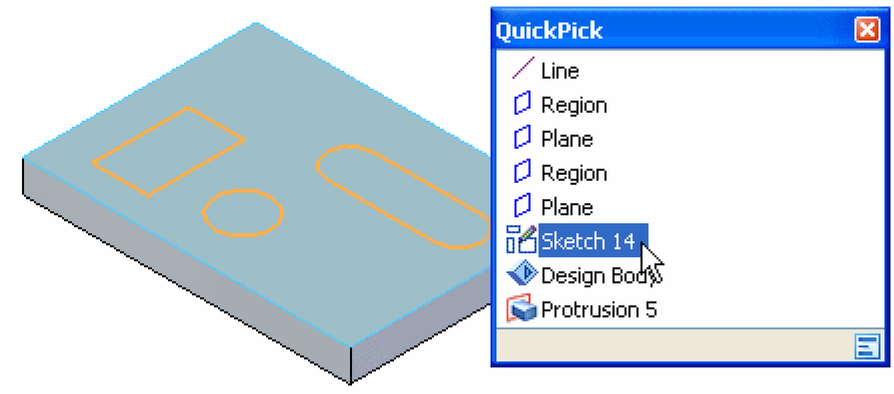
When a sketch name is selected in PathFinder, you can use shortcut commands to:

- Delete a sketch.
- Cut, copy, and paste a sketches.
- Rename a sketch.

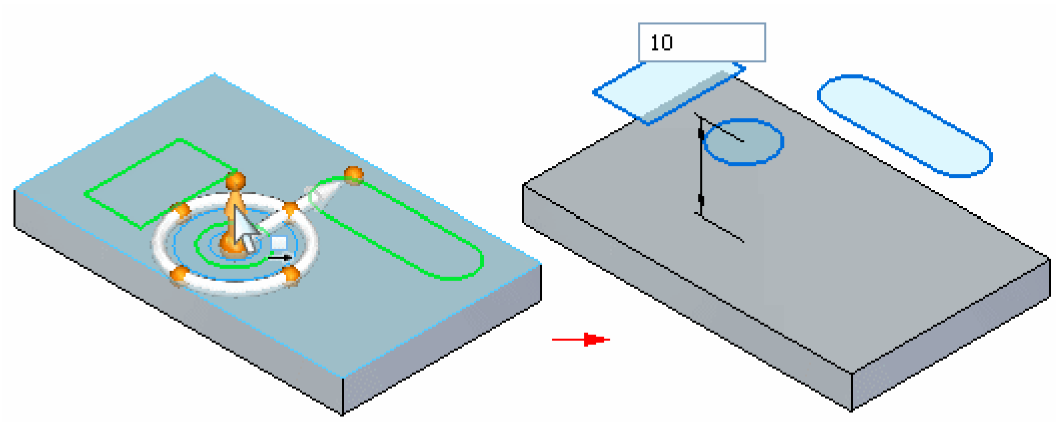
Moving sketches

Sometimes you may want to move or rotate an entire sketch to a new position in space. By default, when you use the Select tool to select sketch elements in the graphics window, only a sketch region or the selected sketch element is selectable.

To select an entire sketch, you can select the sketch entry in PathFinder, or you can use QuickPick to select the sketch in the graphics window.



You can then use the steering wheel to move or rotate the sketch to a new position in space.



If the sketch in moved such that it becomes coplanar to another sketch, the two sketches are combined into one sketch, unless the Merge Coplanar sketches option has been cleared for one of the sketches.

Sketches and associativity

Sketch geometry is not directly associative to the plane or face on which it is drawn. If you move the plane or face on which the sketch is drawn, the sketch geometry does not move unless it is also in the select set. This does not apply to sketches drawn on the principal planes of the base coordinate system or the base reference planes, as these planes are fixed in space.

You can apply 2D geometric relationships between sketch elements and model edges. If the model edges move, the sketch elements and geometric relationships update.

Restoring sketches

To restore a sketch to its original location on the model, use the Restore command on the shortcut menu when a used sketch is selected. This can be useful if you want to use the sketch to construct another feature elsewhere on the model or if you deleted the feature that the used sketch described.

Projecting elements onto a sketch

You can use the Project to Sketch command on the Sketching page to project model edges or sketch elements onto the current sketch plane. The sketch elements you project are associative to the parent element. If the parent element is modified, the projected element updates.

> **Note**
>
> The associative link between the parent element and the projected element is discarded when you construct a feature using the projected elements.

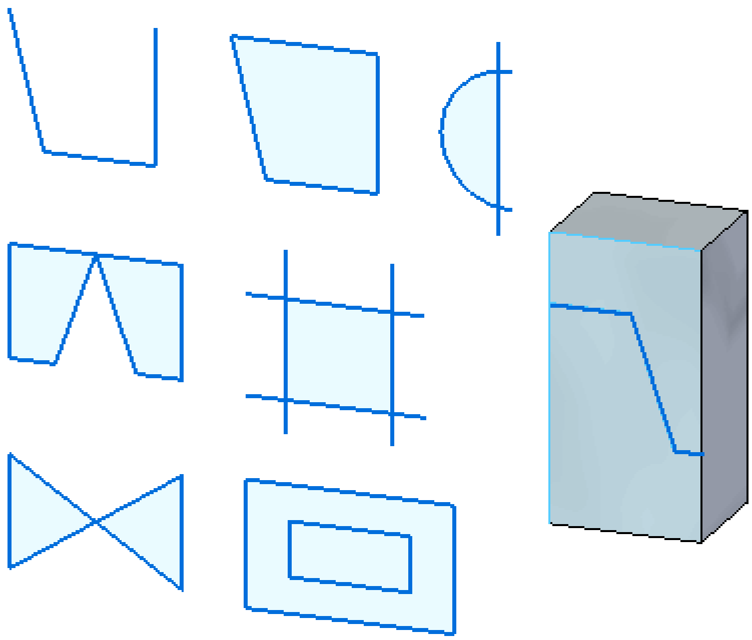
Regions

Definition

A helper object used to create a solid feature consisting of planar and non-planar faces. A region is a closed area formed by sketch elements or a combination of sketch elements and part edges.

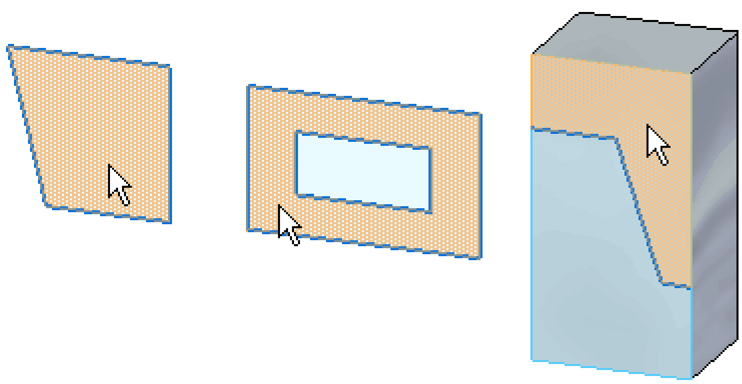
Regions are formed by the placement of 2D sketch geometry on sketch planes or part faces. Regions are created when a series of sketch elements or model edges form a closed area. Regions are a by-product of a closed sketch. Unselected regions appear with a shaded light blue color.

Region examples

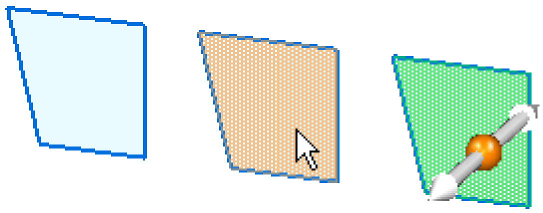


Selecting a region

As the cursor moves over a region, the region appears with a shaded tan color.



When the region is selected, the region appears with a shaded green color.



Regions can be selected in both object-action and action-object workflows.

Activity: Create regions

Draw a sketch and observe when regions are formed and how to select them.

Activity: Create regions

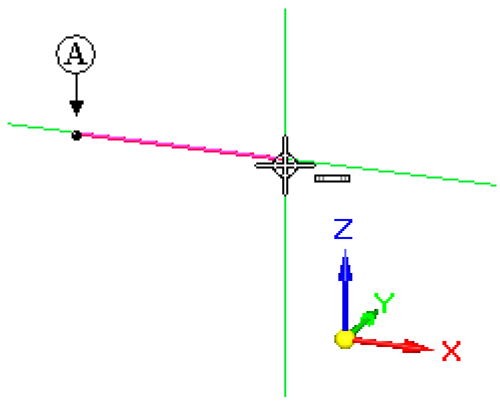
This activity guides you through the process of drawing a sketch to observe when regions are formed. You will also learn how to select regions.

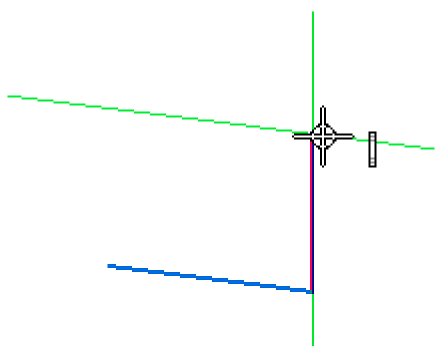
<u>Open a part file</u>

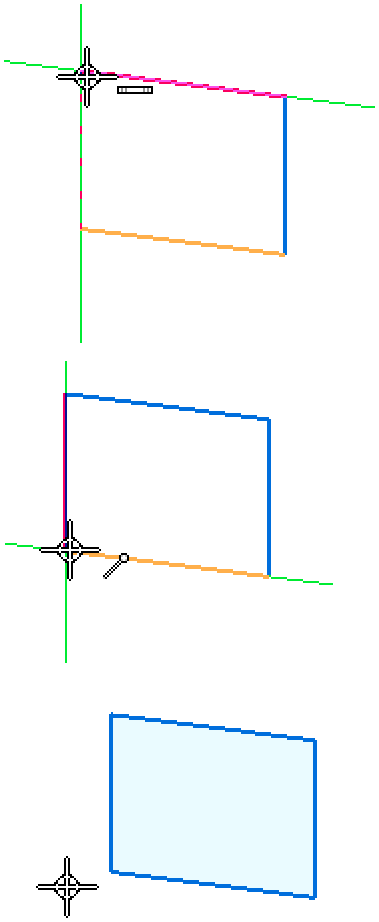
- Start Solid Edge.
- Click the  Application button® New® ISO Part.

<u>Draw a rectangle</u>

- On the Sketching tab® Draw group, choose the Line command .
- Draw a rectangle. Notice that as soon as the last line connects to the first line, a region forms. (A) denotes the first point.

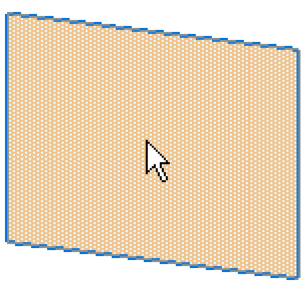




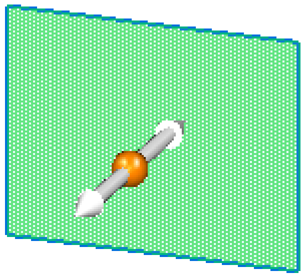


Select the region

- On the Home tab® Select group, choose the Select tool command .
- Move cursor over rectangle and notice the color change. Closed sketches (regions) and faces highlight as the cursor moves over them.

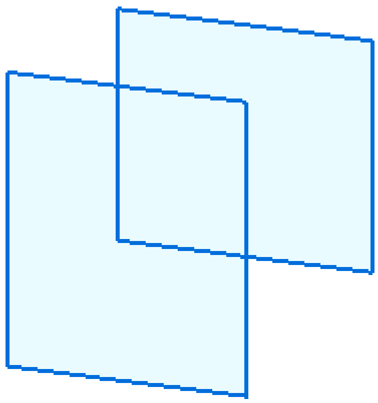


- Select the region and notice the color change. The region can extrude or revolve. This is covered in the Base Feature Creation course. Press the Esc key to end the select command.



Create nested regions

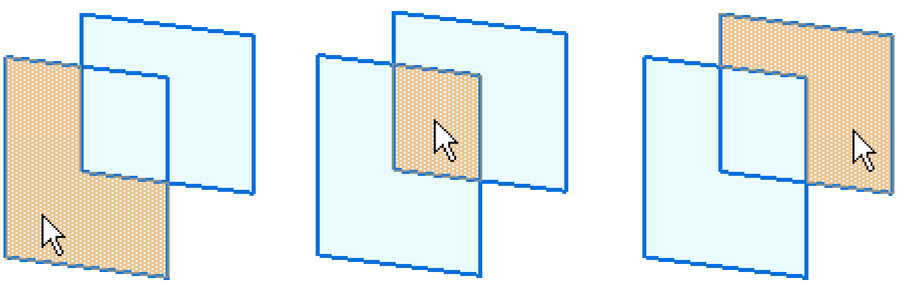
- On the Sketching tab® Draw group, choose the Rectangle command .
- Draw two rectangles that overlap as shown.



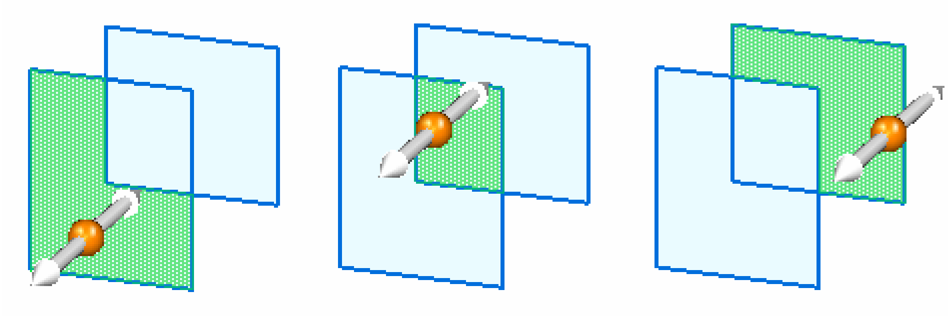
Select multiple regions

The two overlapping rectangles forms three regions.

- Move the cursor over the overlapping rectangles and notice the regions formed.



- Select each region and notice that the previously selected region is deselected.

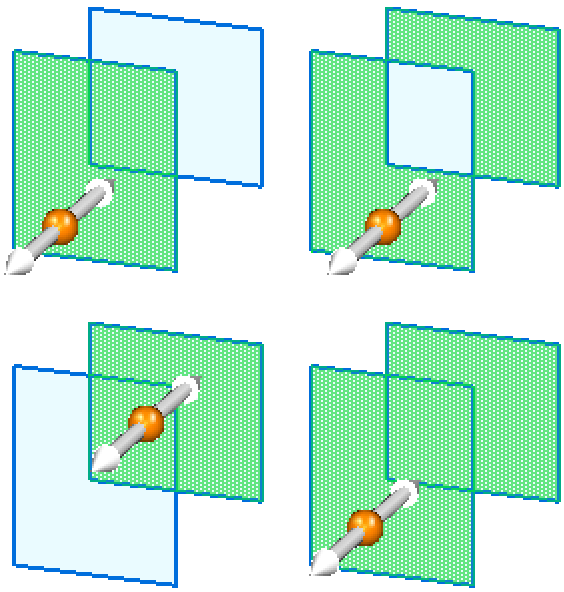


- To select multiple regions, select a region and then press the Spacebar.

  Note

  The Spacebar sets the select mode to add/remove . If you select an element already selected, it is deselected. If you select an element not already selected, it is selected.

- Create the following select sets. Press Esc after each select set is created.



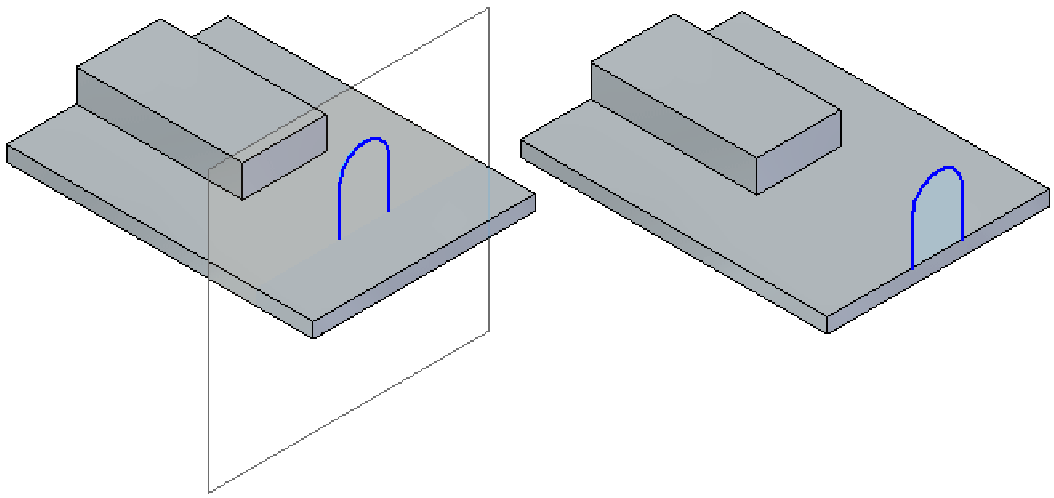
Summary

In this activity you learned how to create and select regions. Creating a synchronous feature in Solid Edge requires a region.

Practice

- Try creating other regions for practice. Otherwise, close the file and do not save.

Open sketches



An open sketch that is not coplanar with a body face or is coplanar with a body face but does not touch or cross a face edge does not create a region. A region is created if an open sketch is connected to or crosses a coplanar face edge.

Note

Open sketches use the extrude command to create a body feature. Define the side of the open sketch to add material to and the sketch automatically extends to next face to create a body feature.

Synchronous sketch behavior in the ordered environment

Synchronous sketches are used to create both synchronous and ordered features. Ordered sketches cannot be used to create a synchronous feature because while in the synchronous environment, ordered elements are not available for selection.

Synchronous sketches can only be selected when creating an ordered feature by using the “Select from Sketch” option in the Profile step.

Editing an ordered feature created with a synchronous sketch

Ordered features are driven by sketches. To edit the cross section definition of an ordered feature, edit the driving sketch.

The following are the methods available for editing a synchronous sketch which drives an ordered feature.

Directly edit the synchronous sketch

Step 1: Turn on the display of the driving synchronous sketch.

Step 2: Select a sketch element to edit.

You can move the selected sketch element and/or change the element properties on command bar.

Step 3: Edit sketch dimensions.

Note

You cannot edit or add synchronous sketch relationships using this method.

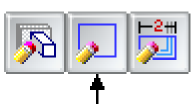
Note

As the synchronous sketch is edited, the ordered feature dynamically updates.

Feature edit (Edit Profile)

Step 1: Select the ordered feature to edit.

Step 2: Choose the Edit Profile command on the Feature Edit box.



Step 3: The modeling environment switches to synchronous. You can now fully edit the synchronous sketch.

Step 4: When the synchronous sketch edits are complete, switch to the ordered environment to observe the feature edits.

Feature edit (Dynamic Edit)

Step 1: Select the ordered feature to edit.

Step 2: Choose the Dynamic Edit command on the Feature Edit box.

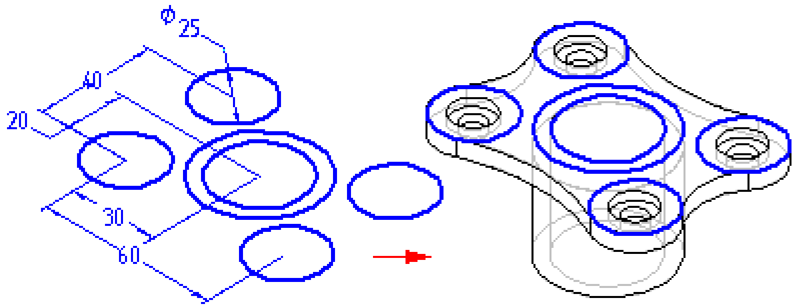
Step 3: The driving synchronous sketch appears. Make edits to synchronous sketch.

Synchronous sketch behavior in ordered modeling

- Synchronous sketch dimensions are not migrated to ordered features.
- Synchronous sketches are not consumed when creating an ordered feature.
- Synchronous sketches can drive ordered features.
- Synchronous sketches appear while in the ordered environment.
- Regions are disabled.
- Synchronous sketches appear in the synchronous sketch style and colors.
- When using the Select Tool in the ordered environment, synchronous sketch elements locate as individual elements.
- Synchronous sketches can be moved using the steering wheel handle. The entire sketch moves (not single elements).
- In the ordered environment, synchronous sketch geometry or relationships commands are not available.
- Synchronous and ordered sketches cannot be copied while in ordered environment.

Drawing ordered sketches of parts

Drawing ordered sketches allows you to establish the basic functional requirements of a part before you construct any features. You can draw a sketch on any reference plane using the Sketch command in the Part and Sheet Metal environments. Then you can use these sketches to create profile-based features.



Sketching a part before modeling it gives you several advantages:

- Allows you to draw multiple profiles on one reference plane.
- Allows you to define relationships, such as tangency or equality, between profiles on different reference planes.
- Allows you to draw the profiles you want without creating the subsequent features until later.

Drawing ordered sketches

When you click the Sketch button and then select a reference plane or planar face, a profile view is displayed. You can then use the drawing commands to draw 2D geometry.

The sketch elements you draw are assigned to the active layer. For example, when working with a complex sketch that will be used to construct a lofted feature, you may want to arrange the elements on multiple layers.

Note

For more information about 2D drawing in Solid Edge, see the following related topics: Drawing in Solid Edge and Drawing Profiles.

You can add dimensions and relationships to control the positions and sizes of the profiles. You can also define functional relationships using the Variables command. You can use the Save and Save All commands to save the sketch while you create them. When you have finished drawing, close the profile view using the Return button on the command bar.

For more information on drawing sketches, see the Drawing 2D elements Help topic.

Sketches and PathFinder

Sketches are represented in the PathFinder tab just like features are. You can display or hide them from the feature tree with the PathFinder Display: Sketches command on the shortcut menu. You can use PathFinder to reorder or rename a sketch just as you would any feature.

Displaying sketches

You can control the display of all the sketches in a document or individual sketches. To display or hide all sketches, use the Show All: Sketches and Hide All: Sketches commands. To display or hide individual sketches, select a sketch in the application window or PathFinder, then use the Show and Hide commands on the shortcut menu.

You can also control the display of elements in a sketch by assigning the sketch elements to a logical set of layers, and then display or hide the layers to control the display of the sketch elements.

When a sketch is active, it is displayed using the Profile color. When a sketch is not active, it is displayed using the Construction color. You can set the colors you want using the Options command.

Using sketches to construct features

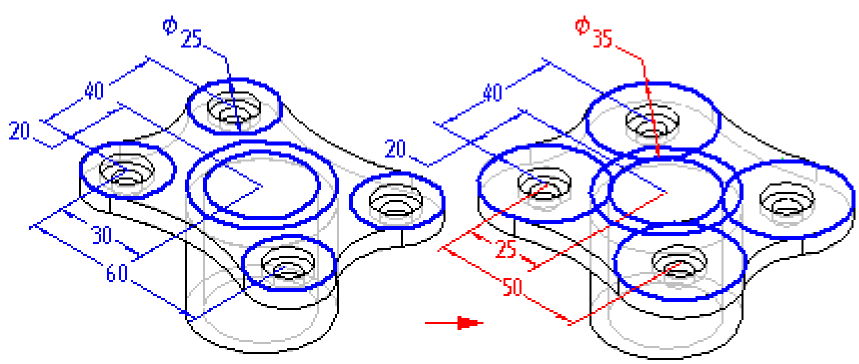
You can use sketches to construct features in the following ways:

- Directly, by clicking the Select From Sketch button on the feature command bar.
- Indirectly, by clicking the Draw button on the feature command bar and then associatively copying sketch geometry onto the active profile plane using the Include command.

Using sketches directly

You can use sketch profiles directly if no modifications to the profile are required. When constructing an ordered feature, click the Select From Sketch button on the feature command bar. You can then select one or more sketch profiles. When you click the Accept button on the command bar, the profiles you selected are checked to make sure they are valid for the type of feature you are constructing. For example, if you are constructing an ordered base feature, the profile you select must be closed. If you select an open profile or more than one profile, an error message is displayed. You can then select the Deselect (x) button on the command bar to clear the selected profiles.

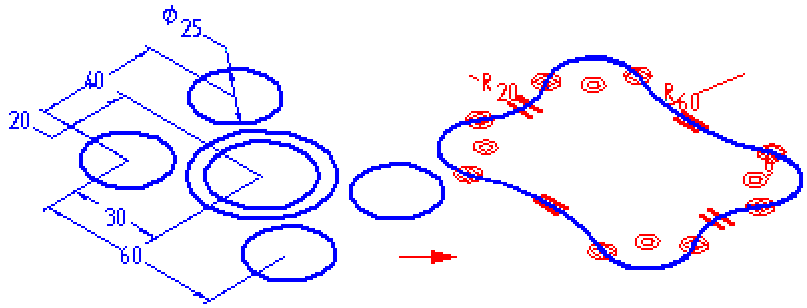
Ordered features constructed using sketched profiles are associative to the sketch and will update when the sketch is edited.



Using sketches indirectly

If the sketch profile requires modification before using it to construct a feature, you must first copy it to the active profile plane using the Include command. When you click the Draw Profile button on the feature command bar, and define the profile plane you want, a profile view is displayed. You can then use the Include command to copy elements from sketch profiles to the active profile plane.

After you have copied sketch elements, you can use the drawing commands to modify them. For example, you may need to add elements to the profile not contained in the sketch. You can also add dimensions and relationships between the elements on the active profile plane and the sketch.



The sketched elements you copy are associative to the sketch and will update if the sketch dimensions are edited.

Editing and modifying sketches

You can modify sketch elements using the command bar or the element's handles. When you modify an element, other elements may also change.

Selecting Elements

You can use the Select Tool to select elements in several ways:

- To select an individual element, position the cursor over the element and click when the element highlights.
- To select multiple elements, press the Ctrl or the Shift key while you select the elements.
- To select all 2D elements, press Ctrl+A. The Select Tool command does not need to be active for this to work.
- To deselect an element, press the Shift or Ctrl key and click the element.
- To select multiple elements using a fence, drag the cursor to define a rectangular fence. You can use the Selection Options button on the Select Tool command bar to specify the selection criteria you want.

Command bars

After you select an element, you can modify it by changing its values on a command bar. For example, you can change the length of a line by typing a new value in the Length box on the command bar.

Element handles

You can use an element's handles to modify an element. An element handle is represented by a solid square on the element, such as the end of a line or the center of an arc. You can dynamically drag a handle to modify an element. First, select the element, then drag the handle to modify it.

- Lines - Drag a handle to modify the length or angle of a line.
- Arcs - Drag an endpoint, midpoint, or center point handle to modify an arc.
- Fillets and Chamfers - Drag the handle to modify the size of a fillet or chamfer.

Sketches and revolved features

Sketches that are used for constructing revolved ordered features must have an axis defined in the sketch. If you select a sketch profile that does not have an axis, an error message is displayed. You will have to cancel the revolved feature you are constructing, then open the sketch to define the axis.

Sketches and the swept and loft commands

Drawing sketches can be especially useful when constructing swept and lofted features. Because the Sketch command allows you to define relationships between profiles on separate planes, you can more easily define the relationships you need to control these features properly. Additionally, the ability to exit a sketch profile window without creating a feature can be especially useful when drawing the profiles for swept and lofted features.

Converting 2D drawing view data to a 3D sketch

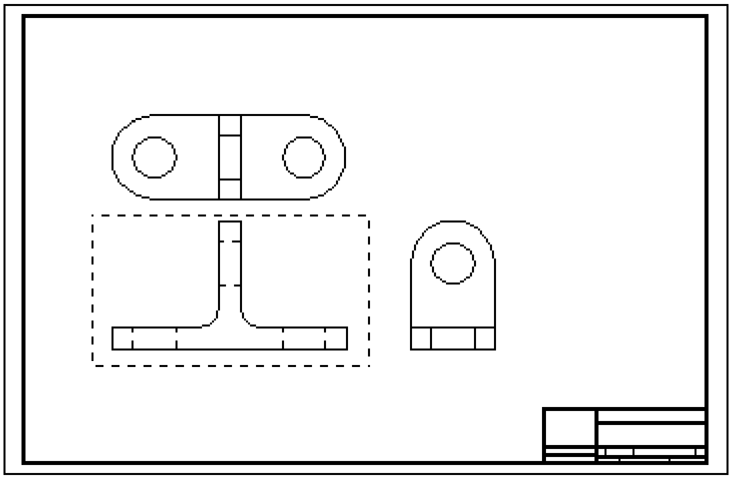
You can use the Create 3D command to convert two-dimensional drawing view data into a three-dimensional sketch.

The command displays the Create 3D dialog box that prompts you for the drawing view elements you want to include in the sketch.

Before selecting the elements that you want to include in the sketches, you need to select a template to create a part, assembly, or sheet metal file. After you select a template file, specify the projection angle that you want to use when the sketches are created in the new document. After you specify the projection angle, select the view type of the elements you want to include in the sketch:

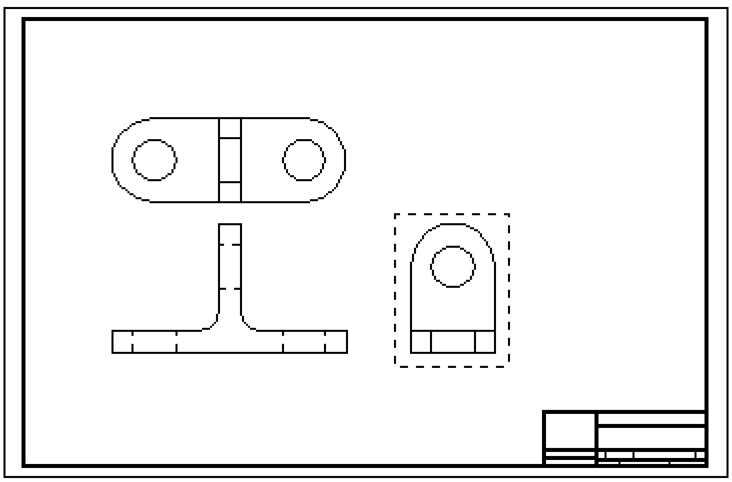
- Folded principal views are orthogonal or aligned with the primary view. You can select this view type to define the primary view.
- Folded auxiliary views are true auxiliary views that are generally derived from principal views and require a fold line to determine the edge or axis around which you want to fold the view.
- Copy views are not orthogonal and they may not actually align with the primary view. These views are placed as sketches on the same plane as the last principal view defined in the draft file.

After you define this information, you are ready to select the geometry to create the sketches. You can include lines, arcs, circles, curves, and polylines and line strings created with imported data. You can drag the mouse to fence elements or press the Shift key and click each element to select more than one element.

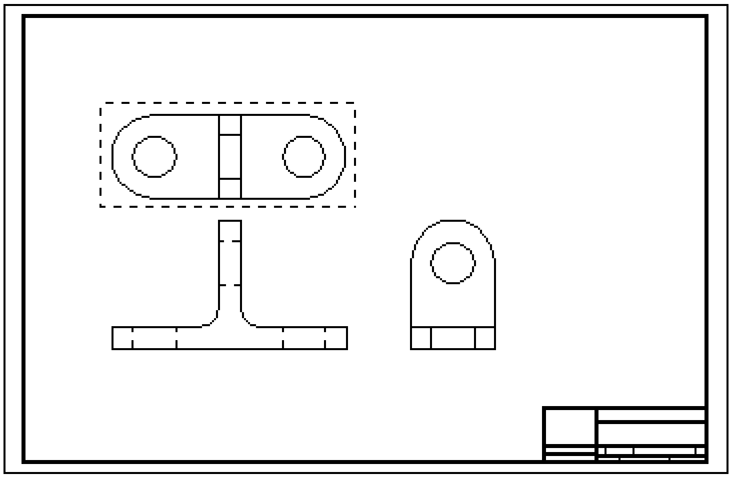


If you select the Fold Principal Views option or Fold Auxiliary Views option and it is not the primary view, you can click the Fold Line button after you select all of the elements for the view. The Fold Line button allows you to define a line or point in an orthogonal or auxiliary view on which to fold the primary view.

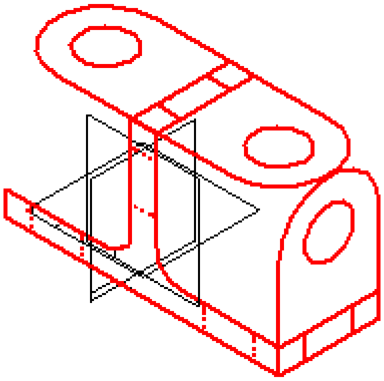
If you want to define another view, click the New View button and select the next view.



Continue this process to define any additional views.

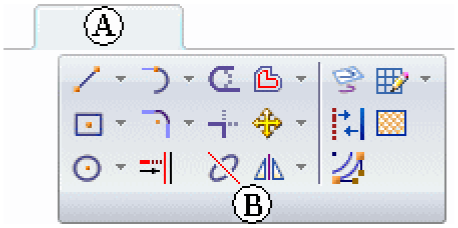


After you define all views, click the Finish button to launch the Part or Sheet Metal environment to create the model file in which the views are placed as sketches.



Drawing commands

The commands for creating and manipulating sketch elements are located on the Sketching tab (A) in the Draw group (B).



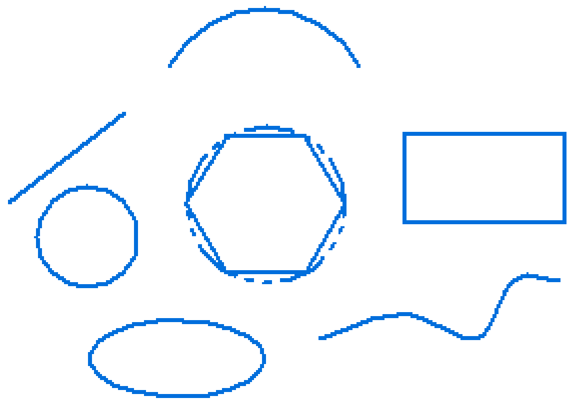
Drawing 2D elements

In Solid Edge, you can draw 2D elements to help you complete a variety of tasks. For example, you can use 2D elements to construct features in the Part environment and to draw layouts in the Assembly environment.

In the Draft environment, you can use 2D drawing tools to complete a variety of tasks such as drawing sketches from scratch on the 2D Model sheet or in 2D views, creating background sheet graphics, and defining cutting planes for section views. The drawing commands, relationships, and dimensions work similarly in all environments.

Drawing commands and tools

You can draw any type of 2D geometric element in Solid Edge, such as lines, arcs, circles, B-spline curves, rectangles, and polygons.



You can also use Solid Edge to do the following:

- Move, rotate, scale, and mirror elements
- Trim and extend elements
- Add chamfers and fillets
- Create precision graphics from a freehand sketch
- Change the color of elements

Tools that work with the drawing commands—IntelliSketch, Intent Zones, and Grid—allow you to easily relate elements to each other, define your drawing intentions as you sketch, and provide precise coordinate input relative to any key position in the drawing.

Drawing command input

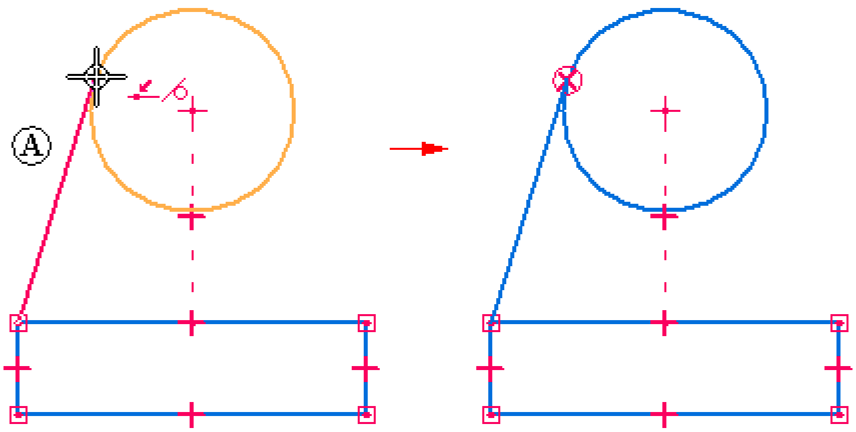
Use Solid Edge drawing commands to provide input by clicking in the graphics window, or by typing values in command bar boxes. No strict input order is required.

It is often productive to use a combination of graphics window and command bar input. For example, you can type a line length in the command bar, press the Enter or Tab key to lock the value, then set the orientation angle of the line in the graphics window. Or you can use the drawing command dynamics to get a graphic idea of the size and orientation you want, then type values in the command bar boxes to provide more precise input.

You can use the Line Color option on the element command bar to apply colors to 2D elements. You can click the More option on the Colors dialog box to define custom colors.

Drawing dynamics

As you draw, the software shows a temporary, dynamic display of the element you are drawing (A). This temporary display shows what the elements will look like if you click at the current cursor position.



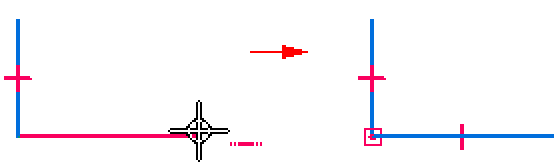
Until you click the point that completely defines the element that you are drawing, values in the command bar boxes update as you move the cursor. This gives you constant feedback on the size, shape, position, and other characteristics of the elements you draw.

When you lock a value by typing it into a command bar box, the dynamic display of the element you are drawing shows that the value is locked. For example, if you lock the length of a line, the length of the dynamic line does not change as you move the cursor to set the angle. If you want to free the dynamics for a value, you can clear the value box by double-clicking in the box and pressing the Backspace or Delete key.

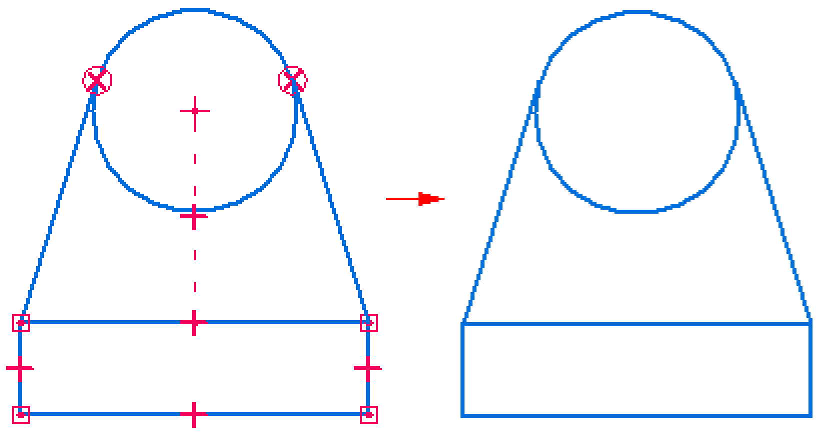
Applying and displaying relationships

As you draw, IntelliSketch recognizes and applies 2D relationships that control element size, shape, and position. When you make changes, relationships help the drawing retain the characteristics you do not want altered.

When a relationship indicator is displayed at the cursor, you can click to apply that relationship. For example, if the horizontal relationship indicator is displayed when you click to place the end point of a line, the line will be drawn exactly horizontal. You can also apply relationships to elements after you draw them.



Relationship handles displayed on the 2D geometry show you how elements are related. You can remove any relationship by deleting its handle. You can display or hide the relationship handles with the Relationship Handles command.



Maintaining relationships

You can draw and modify 2D elements in the way that best suits your design needs. You can make your assembly layouts and drawings associative by applying relationships, or you can draw them freely, without relationships. When you draw 2D elements in a part document, 2D relationships are maintained.

Maintaining relationships between 2D elements makes the elements associative (or related) to each other. When you modify a 2D element that is related to another 2D element, the other element updates automatically. For example, if you move a circle that has a tangent relationship with a line, the line also moves so that the elements remain tangent.

You can draw elements freely, or non-associatively. When you modify a non-associative portion of an assembly sketch or drawing, the changed elements move freely, without changing other portions of the design. For example, if you move a circle that is tangent to a line (but does not have a tangent relationship with the line) the line does not move with the circle.

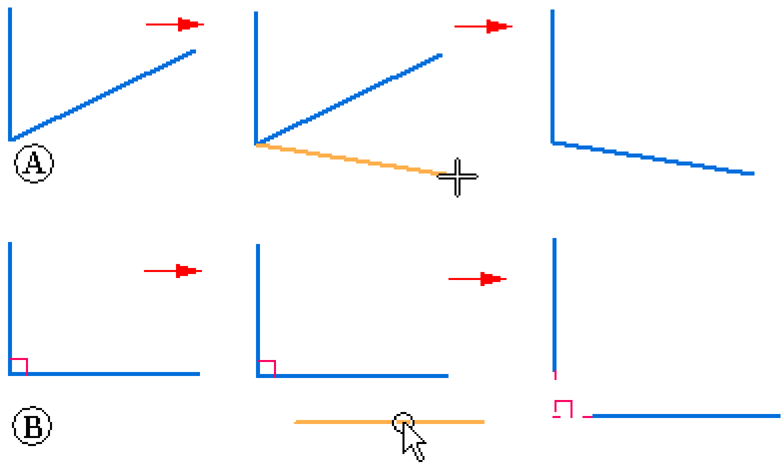
To control whether you draw and modify 2D elements freely or associatively in layouts and drawings, use the Maintain Relationships command in the Assembly and Draft environments.

Note

When you construct a synchronous feature using the 2D elements, the sketch elements are moved to the Used Sketches collector in PathFinder.

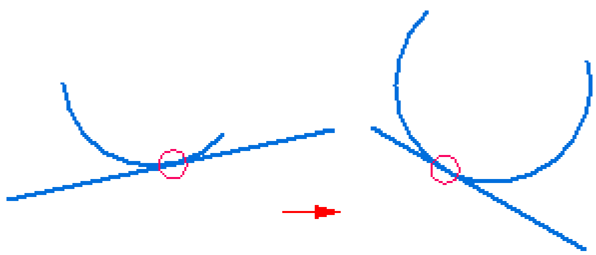
How 2D relationships work

An element that has no relationships applied can be moved and changed in various ways. For example, when there are no relationships between two lines (A), the lines can be moved and changed without affecting each other. If you apply a perpendicular relationship between the two lines (B), and move one line, the lines remain perpendicular.

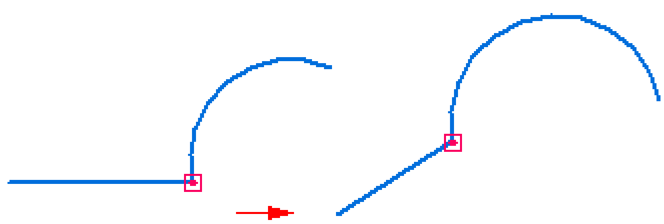


When you apply a relationship between elements, the relationship is maintained when you modify either element. For example:

- If a line and an arc share a tangent relationship, they remain tangent when either is modified.



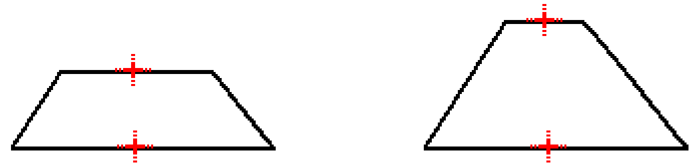
- If a line and arc share a connect relationship, they remain connected when either is modified.



Relationships also maintain physical characteristics such as size, orientation, and position.

- You can make the size of two circles equal with an equal relationship.
- You can make the orientation of two lines parallel with a parallel relationship.
- You can connect a line and an arc with a connect relationship.

A relationship can also maintain a physical characteristic of an individual element. For example, you can make a line horizontal. The line remains horizontal even if you change its position and length.



Construction elements

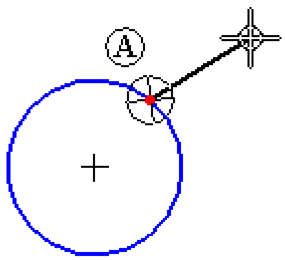
For 2D elements you draw in a part or assembly document, you can specify that the element is a considered a construction element. The Construction command on the Sketching tab allows you to specify that an element is a construction element. Construction elements are not used to construct features—they are used only as drawing aids. The line style for a construction element is dashed.

Intent Zones

Solid Edge uses intent zones to interpret your intentions as you draw and modify elements. Intent zones allow you to draw and modify elements many ways using few commands. You do not need to select a different command for every type of element.

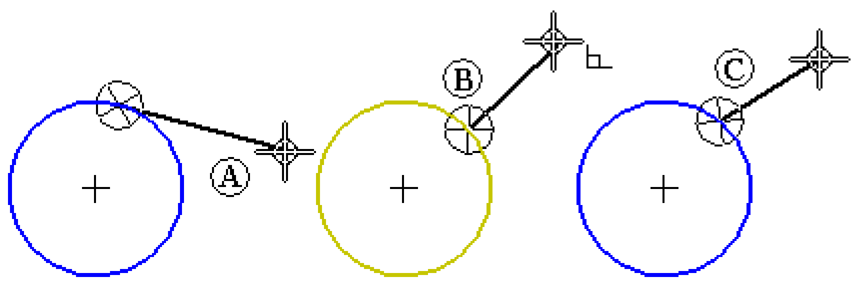
How intent zones work

When you click to begin drawing certain elements, the software divides the region around the clicked position into four intent zone quadrants. For example, when drawing a line that is connected to a circle, four intent zones are displayed around the point you clicked (A).



Two of these intent zones allow you to draw the line tangent to the circle. The other two intent zones allow you to draw the line perpendicular to, or at some other orientation relative to the circle.

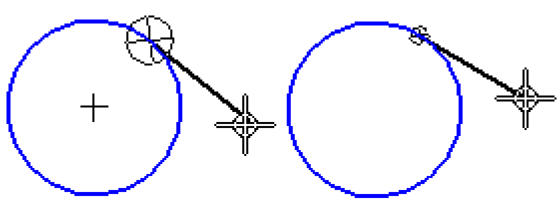
By moving the cursor through one of these intent zones on the way to your next click location, you can tell the software what you want to do next. This allows you to control whether the line is tangent to the circle (A), perpendicular to the circle (B), or at some other orientation (C).



The last intent zone you move the cursor into is the active zone. To change the active intent zone, move the cursor back into the zone circle, and then move the cursor out through the intent zone quadrant to the position where you want to click next.

Intent zone size

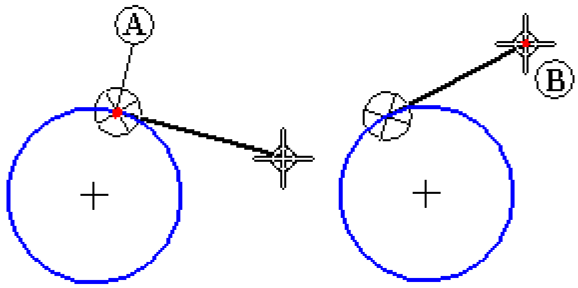
You can change the size of the intent zones with the IntelliSketch command. The Intent Zone option on the Cursor tab on the IntelliSketch dialog box allows you to set the intent zone size.



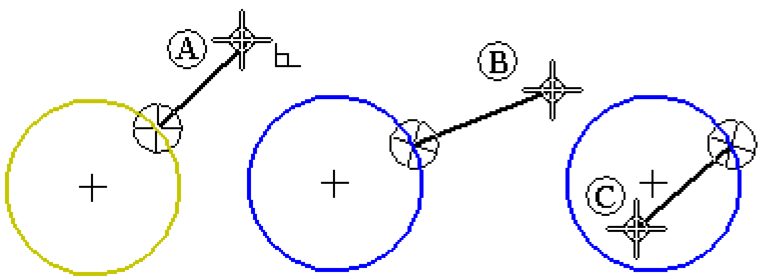
Drawing lines tangent or connected to curved elements

Using intent zones with the Line command, you can draw a line tangent to a circle or arc. Or you can draw a line that is connected to the circle or arc, but not tangent to it.

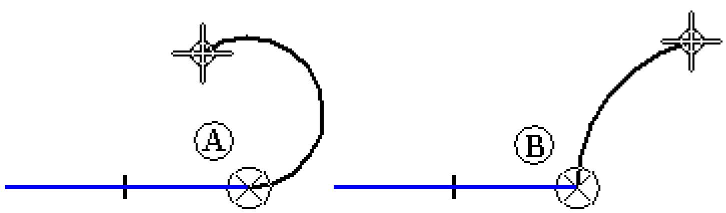
To draw an line tangent to a circle, first click a point on the circle (A) to place the first end point of the line. Then move the cursor through the tangent intent zone. As you move the cursor, the line remains tangent to the circle. Position the cursor where you want the second end point of the line (B), then click to place the second end point.



If you do not want the line to be tangent to the circle, you can move the cursor back into the intent zone region and out through one of the perpendicular zones (A) before clicking to place the second end point of the line. When you move the cursor through the perpendicular zones, you can also draw the line such that it is not perpendicular to the circle (B) and (C).



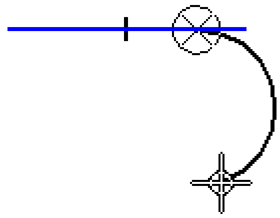
The Line command also allows you to draw a connected series of lines and arcs. You can use the L and A keys on the keyboard to switch from line mode to arc mode. When you switch modes, intent zones (A) and (B) are displayed at the last click point.



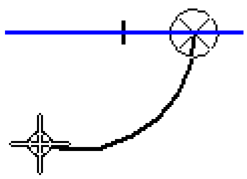
The intent zones allow you to control whether the new element is tangent to, perpendicular to, or at some other orientation to the previous element.

Drawing tangent or perpendicular arcs

You can use intent zones to change the result of the Tangent Arc command. To draw an arc tangent to a line, first click a point on the line to place the first end point of the arc. Then move the cursor through the tangent intent zone and click to place the second end point of the arc.

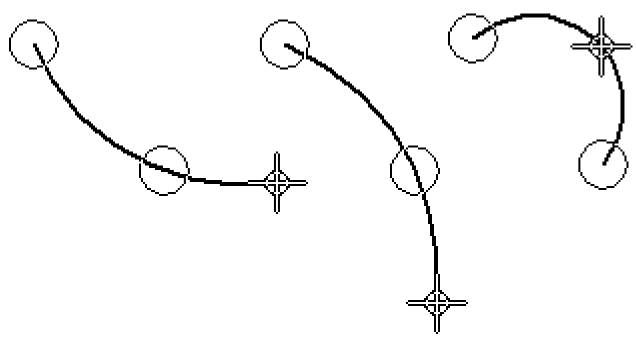


If you do not want the arc to be tangent to the line, you can move the cursor back into the intent zone region and out through the perpendicular zone before clicking to place the second end point of the arc.



Drawing arcs by three points

When you use the Arc By 3 Points command, intent zones allow you to input the three points in any order. You can also use intent zones to change the arc direction. The intent zone used with the Arc By 3 Points command is not divided into quadrants.

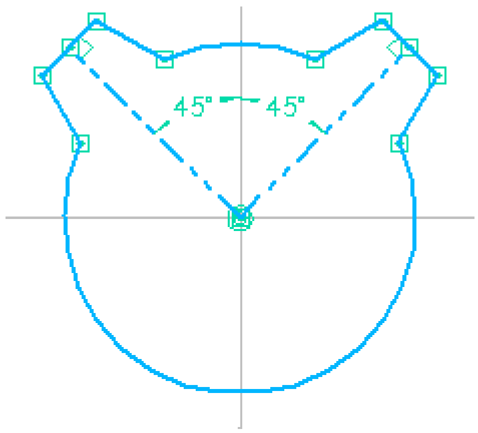


Construction Geometry

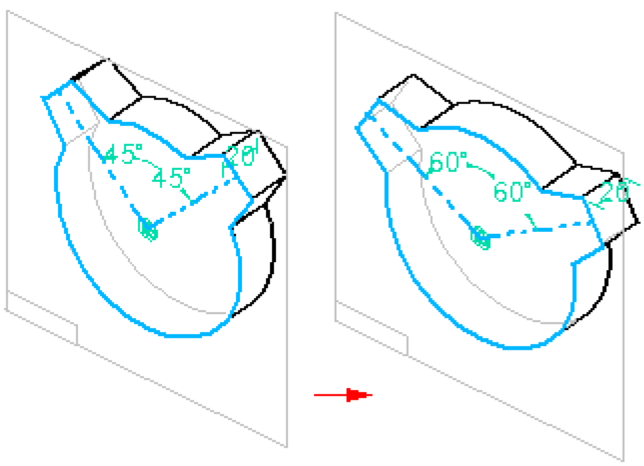
You can use construction geometry to help you draw and constrain a profile, but the construction geometry is not used to construct the surfaces for the feature. When the feature is created, the construction geometry is ignored. The Construction command is used to change a profile element or sketch element into a construction element.

- Construction elements use the double-chain line style so you can distinguish them from other elements.

- For example, you can use 45 degree construction lines to control the location of the tabs on the profile or sketch.



- The construction lines make it easier to edit the location of the tabs, but the construction lines are not used to produce the solid model.

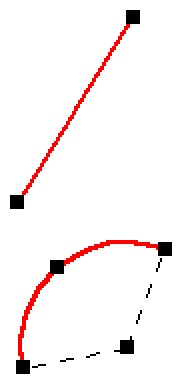


Modifying 2D elements

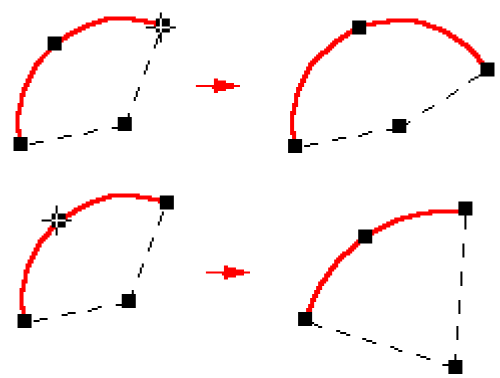
Solid Edge provides a wide range of tools for modifying 2D elements. 2D drawing and modification tools work together smoothly, so that you can modify your profiles, sketches, and 2D drawings as you work.

Using element handles

You can change the size, position, or orientation of an element with the cursor. When you select an element with the Select tool, its handles are displayed at key positions.

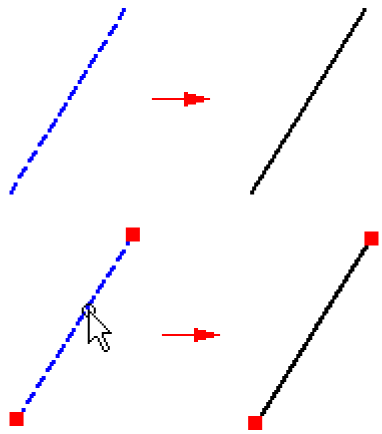


You can change the shape of a selected element by dragging one of its handles. The first figure shows the effect of dragging an end point handle. The second figure shows the effect of dragging the midpoint handle.

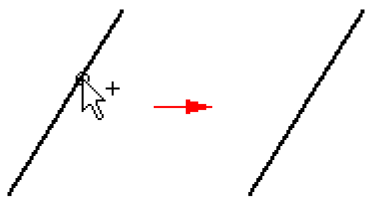


Moving and copying elements with the mouse

You can also drag a selected element to move it without changing its shape. Position the cursor so it is not over a handle, then drag the element to another location.

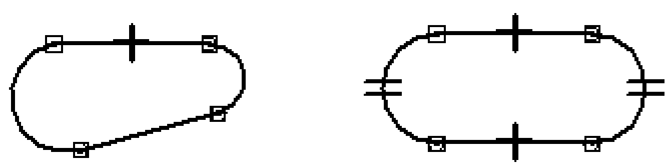


To copy an element, hold the Ctrl key while you drag.

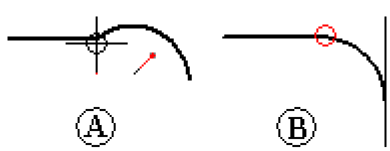


Applying relationships between elements

You can apply geometric relationships as you draw or after you draw. To apply a geometric relationship onto an existing element, select a relationship command and then select the element to which you want to add the relationship. When you apply a relationship to an element, the element is modified to reflect the new relationship.



If a line and arc are not tangent (A), applying a tangent relationship modifies one or both elements to make them tangent (B).



When you use relationship commands, the software allows you to select only elements that are valid input for that command. For example, when you use the Concentric command, the command allows you to select only circles, arcs, and ellipses.

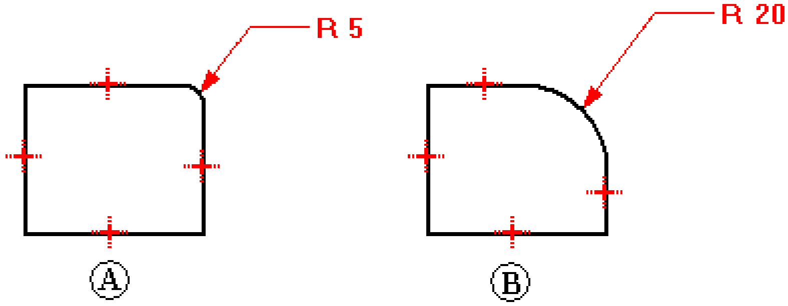
Changing relationships

You can delete a relationship as you would delete any other element by selecting a relationship handle, then press the Delete key on the keyboard.

Dimensions as relationships

Driving dimensions are relationships that allow you to maintain characteristics such as the size, orientation, and position of elements. When you place a driving dimension on or between elements, you can change the measured elements by editing the dimensional value. You do not have to delete or redraw elements at different sizes.

For example, you can dimension the radius of an arc to maintain its size (A), and then edit the value of the radius dimension to change its size (B).



To create dimensional relationships, select a dimension command and click the elements you want to control.

Changing elements with relationships

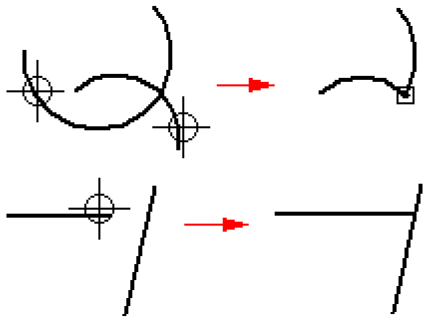
When you modify 2D elements, elements with maintained relationships automatically update to honor the relationship. For example, if you move an element that shares a parallel relationship with another element, the other element moves as needed to remain parallel. If a line and an arc share a tangent relationship, they remain tangent when either is modified.

If you want to change an element by adding or removing a relationship, and the element does not change the way you expect, it may be controlled by a driving dimension. You can toggle the dimension from driving to driven, then make the change.

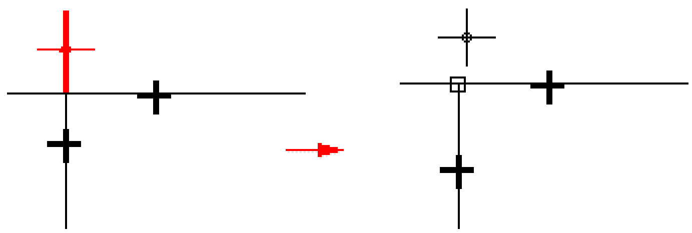
Element modification: trimming, extending, splitting, filleting, chamfering, offsetting, and stretching

Whether your sketching technique is to start big and whittle away or to start small and build up, relationships make it possible to sketch and evolve, rather than draw every element to its exact measurements. Solid Edge modification tools allow you to change a sketch and still maintain applied relationships.

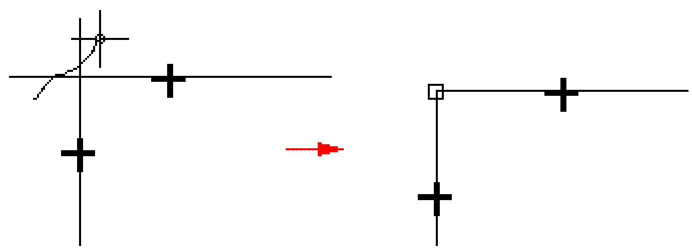
Solid Edge provides commands to trim, extend, or split elements.



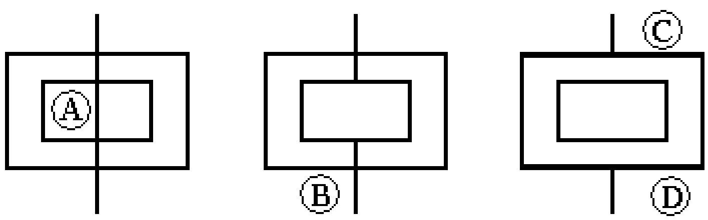
The Trim command trims an element back to the intersection with another element. To use the command, click on the part to trim.



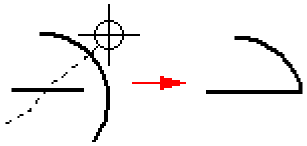
You can trim one or more elements by dragging the cursor across the part to trim.



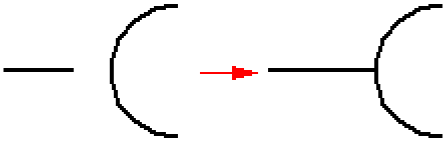
You can also select the elements you want to trim to. This selection overrides the default option of trimming to the next element only. To select an element to trim to, press the Ctrl key while selecting the element to trim to. For example, in normal operations, if you selected line (A) as the element to be trimmed, it would be trimmed at the intersection of the next element (B). However, you can select the edges (C) and (D) as the elements to trim to and the element will be trimmed at the intersection of those edges.



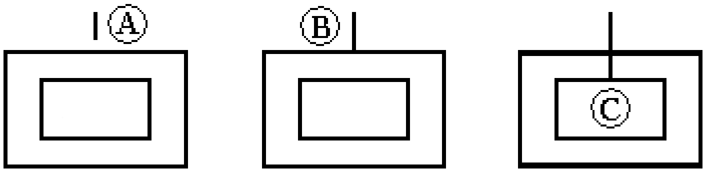
The Trim Corner command creates a corner by extending two open elements to their intersection.



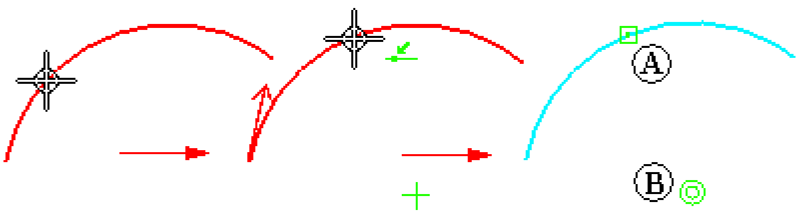
The Extend to Next command extends an open element to the next element. To do this, select the element and then click the mouse near the end to extend.



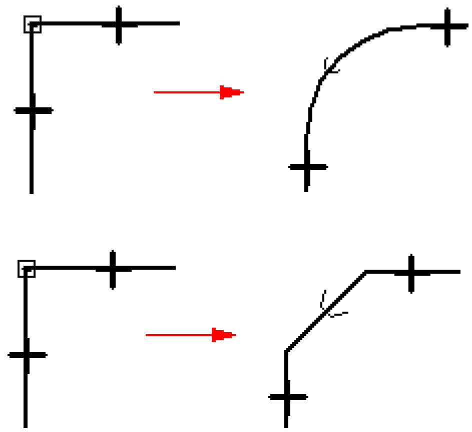
You can also select an element to extend to. This selection overrides the default option of extending to the next element only. To select an element to extend to, press the Ctrl key while selecting the element to extend to. For example, in normal operations, if you selected line (A) as the element to be extended, it would be extended to the intersection of the next element (B). However, you can select edge (C) to extend the line to that edge.



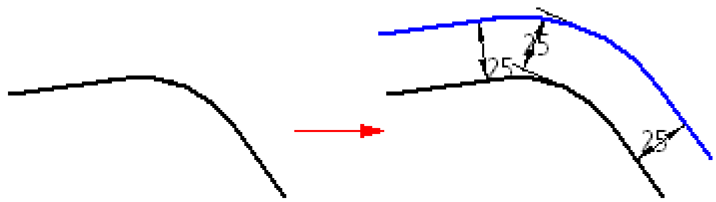
The Split command splits an open or closed element at the location you specify. When splitting elements, appropriate geometric relationships are applied automatically. For example, when splitting an arc, a connect relationship (A) is applied at the split point, and a concentric relationship (B) is applied at the center point of the arcs.



Fillet and Chamfer commands combine drawing and trimming operations.

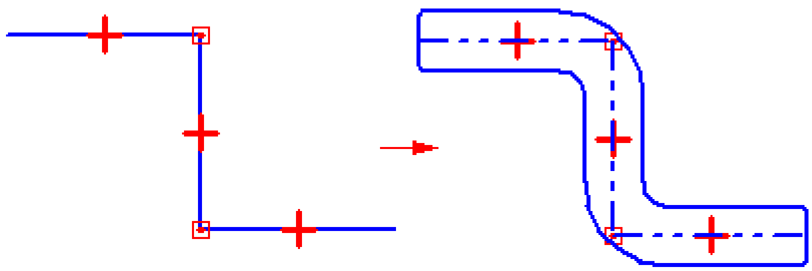


The Offset command draws a uniform-offset copy of selected elements.

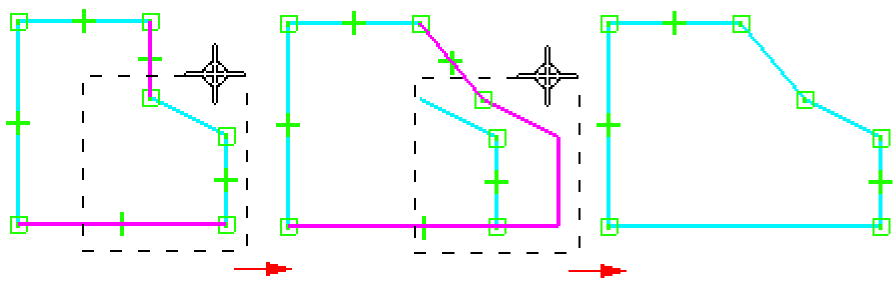


You cannot select model edges with this command. If you want to offset model edges, use the Include command.

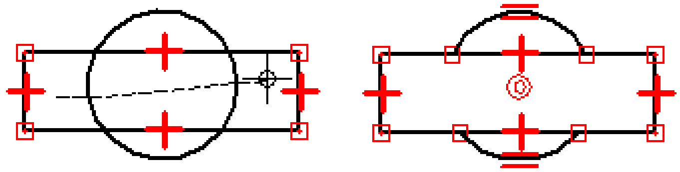
The Symmetric Offset command draws a symmetrically offset copy of a selected center line.



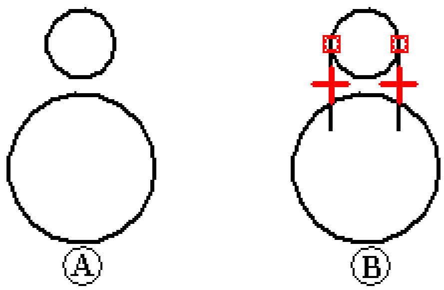
The Stretch command moves elements within the fence and stretches elements that overlap the fence.



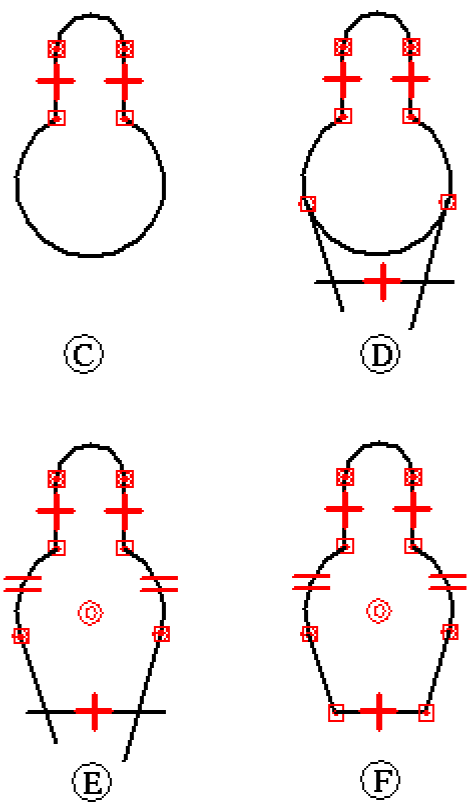
Relationships are added or removed as necessary during element modification. If you trim part of a circle and more than one arc remains, concentric and equal relationships are applied between the remaining arcs.



For example, you typically begin designing with key design parameters. You would draw known design elements in proper relation to one another (A) and then draw additional elements to fill in the blanks (B).

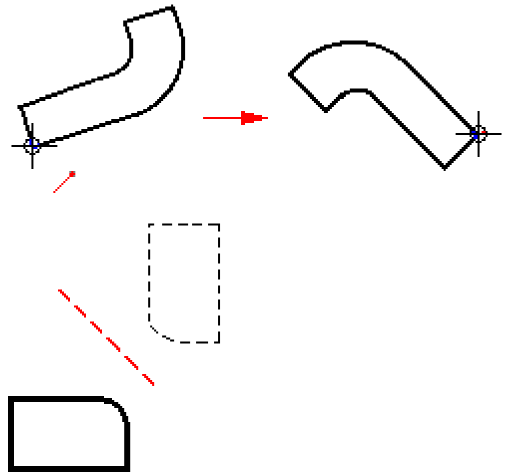


As you draw, you may need to modify elements to create a valid profile, or to make a drawing look the way you want it to (C-F). You can use modification commands such as Trim and Extend to modify the elements. The relationships are maintained and additional relationships are applied.



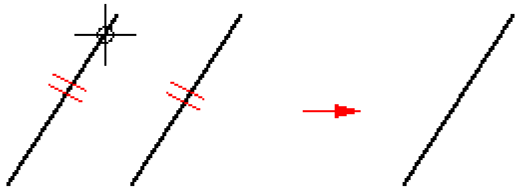
Element manipulation: rotating, scaling, mirroring, copying, and deleting

Tools are provided for moving, rotating, scaling, and mirroring elements. These tools can also be used for copying. For example, you can make a mirror copy, or you can cut or copy 2D elements from another application and paste them into the profile window, the assembly sketch window, or a drawing.

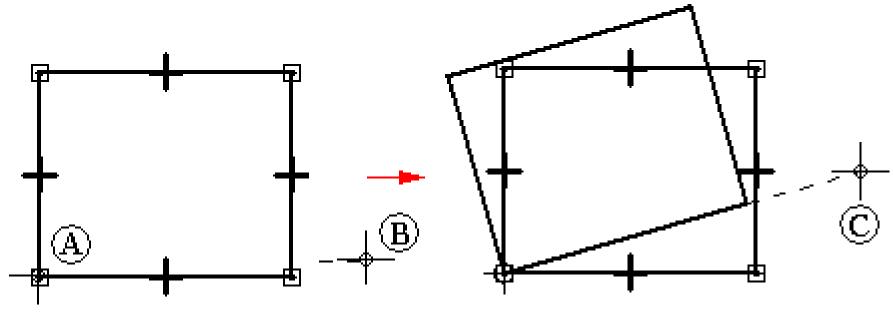


When you manipulate elements that have relationships, the relationships are retained when possible. For example, if you make a copy of two related elements, the relationship is also copied. However, if you copy one of two elements that are related to each other, the relationship is not copied.

Relationships that are no longer applicable after a manipulation are automatically deleted. For example, if you delete one of a pair of parallel lines, the parallel relationship is deleted from the remaining line.

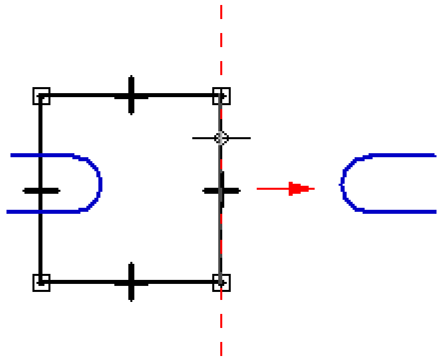


The Rotate command turns or turns and copies 2D elements about an axis. The command requires you to specify a center point for the rotation (A), a point to rotate from (B), and a point to rotate to (C).

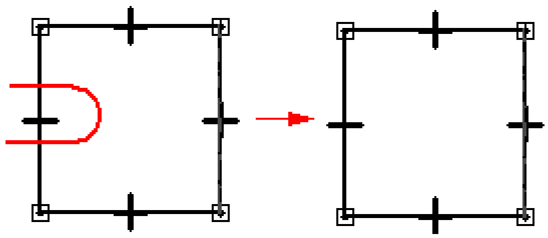


The Scale command uses a scale factor to proportionally scale or scale and copy 2D elements.

The Mirror command mirrors or mirror copies 2D elements about a line or two points.

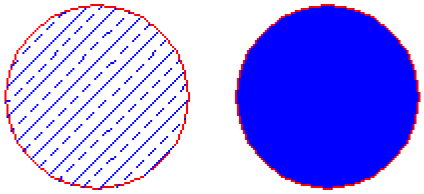


The Delete command removes 2D elements from the profile or sketch window.



Applying colors and patterns to closed boundaries

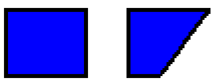
A boundary in a Solid Edge drawing, sketch, or profile can be filled with a pattern or solid color.



A fill is like other elements in that you can format it and move it around, but the fill is always associated with a boundary. The boundary can be made up of more than one element.

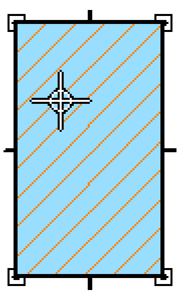
Modifying fills

A fill can exist only inside a closed boundary. A fill is associative, which means it maintains its original orientation to an element regardless of the way you manipulate the element. For example, if you move the boundary, the fill moves with it. If you change the boundary, the fill changes to conform to the new boundary area. You can delete a fill the same way you would delete an element.

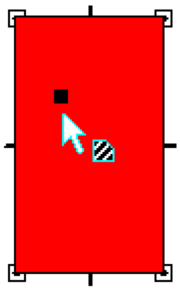


Fill insertion point

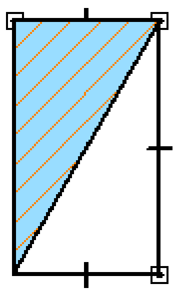
- When you click inside an object to fill it, the cursor location designates the fill insertion point.



- The fill insertion point is also the fill handle. You can select the fill handle and drag the fill to another object.



- If you use the Redo Fill option to refill the area based on a new boundary, the insertion point designates which side of the object will be refilled.



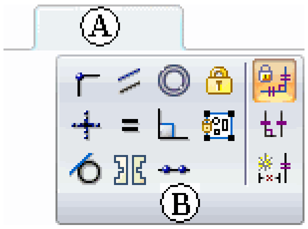
Formatting fills

Formatting a fill is similar to applying formats to an element. You can apply unique formats to fills with the Properties command or by setting options on the Fill command bar. To make several fills look the same, you can apply a fill style by selecting the style on the command bar.

The software provides fill styles for various engineering standards, such as ANSI, ISO, and AIA. You can modify an existing fill style or create a new one with the Style command.

Sketch geometric relationships

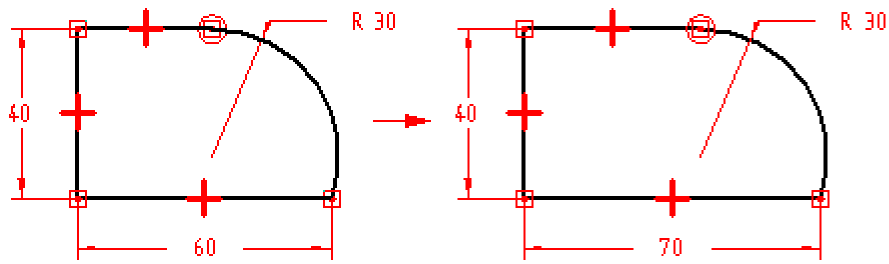
The sketch relationship commands are located on the Sketching tab (A) in the Relate group (B).



Sketch relationships do not migrate to the feature created from them.

Geometric Relationships

Geometric relationships control the orientation of an element with respect to another element or reference plane. For example, you can define a tangent relationship between a line and an arc. If the adjoining elements change, the tangent relationship is maintained between the elements.



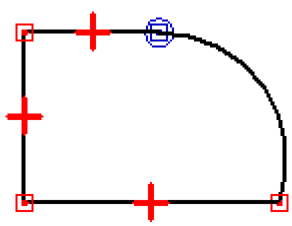
Geometric relationships control how a sketch changes when edits are made. IntelliSketch displays and places geometric relationships as you draw. After you complete the sketch, you can use the various relationship commands and the Relationship Assistant to apply additional geometric relationships.

Relationship Handles

Relationship handles are symbols used to represent a geometric relationship between elements, keypoints, and dimensions, or between keypoints and elements. The relationship handle shows that the designated relationship is being maintained.

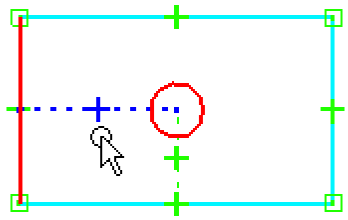
Relationship	Handle
Collinear	
Connect (1 degree of freedom)	
Connect (2 degrees of freedom)	
Concentric	
Equal	
Horizontal/Vertical	
Tangent	
Tangent (Tangent + Equal Curvature)	
Tangent (Parallel Tangent Vectors)	
Tangent (Parallel Tangent Vectors + Equal Curvature)	
Symmetric	
Parallel	
Perpendicular	
Fillet	
Chamfer	
Link (local)	
Link (peer-to-peer)	
Link (sketch to sketch)	
Rigid Set (2-D elements)	

In some cases, more than one relationship may be required and displayed at the same location on the profile. For example, a connect relationship and a tangent relationship can be used where an arc meets a line.

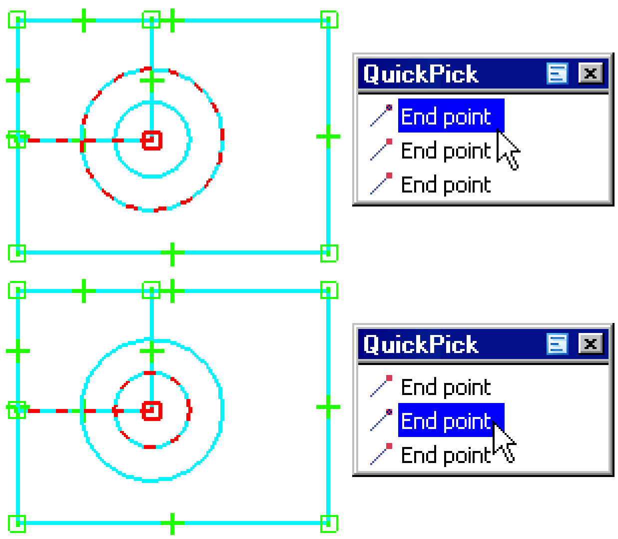


Displaying Parents for a Relationship

When modifying a profile or sketch, it can be useful to determine the parent elements for a relationship. When you select a geometric relationship, the parents highlight. For example, when you select the horizontal relationship shown in the first illustration, the left vertical line and the circle are highlighted as the parent elements.

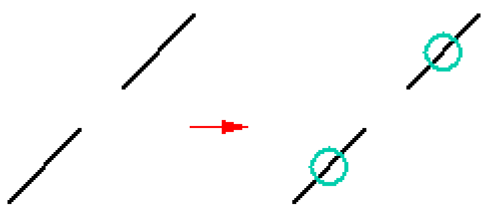


This can be useful when multiple relationships are in the same location and you need to delete one relationship. In this situation, you can use QuickPick to highlight the relationship, and the parent elements are displayed using a dashed line style.



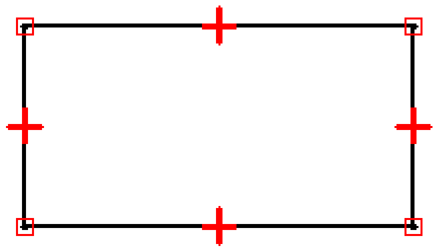
Collinear

The Collinear command forces two lines to be collinear. If the angle of one of the lines changes, the second line changes its angle and position to remain collinear with the first.

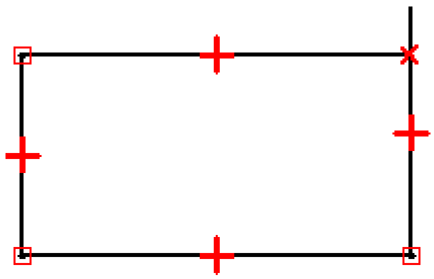


Connect

The Connect command joins a keypoint on one element to another element, or element keypoint. For example, you can apply a connect relationship between the endpoints of two elements. Establishing a connect relationship between element endpoints helps you draw a closed sketch. The symbol for connected endpoints displays a dot at the center of a rectangle.



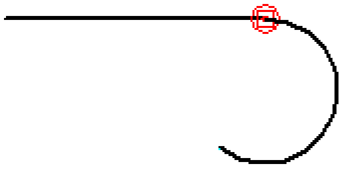
You can also use the Connect command to connect the endpoint of an element to any point on another element, not necessarily an endpoint or keypoint. This is called a point-on-element connection, and the symbol resembles an X. For example, the endpoint of the top horizontal line on the right side of the profile is connected to the vertical line, but not at an endpoint.



When drawing profiles, pay close attention to the relationship indicator symbols that IntelliSketch displays, and try to draw the elements as accurately as possible. Otherwise, you may accidentally apply a connect relationship in the wrong location, which can result in an invalid profile. For example, for a base feature you may accidentally create an open profile, rather than the required closed profile.

Tangent

The Tangent command maintains tangency between two elements or element groups.



When you apply a tangent relationship, you can use the Tangent command bar to specify the type of tangent relationship you want:

- Tangent
- Tangent + Equal Curvature
- Parallel Tangent Vectors

- Parallel Tangent Vectors + Equal Curvature

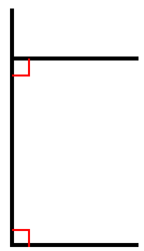
A simple tangent relationship is useful when you want a line and an arc, or two arcs to remain tangent. The other options are useful in situations where a b-spline curve must blend smoothly with other elements. The Tangent + Equal Curvature, Parallel Tangent Vectors, and Parallel Tangent Vectors + Equal Curvature options require that the first element you select is a b-spline curve.

Note

You can also apply a tangent or connect relationship to an end-point connected series of elements to define a profile group. For more information on profile groups, see the Working With Profile Groups topic.

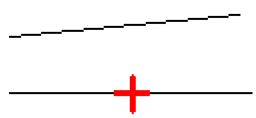
Perpendicular

The Perpendicular command maintains a 90-degree angle between two elements.

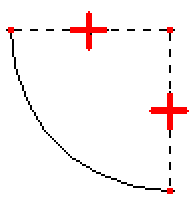


Horizontal/Vertical

The Horizontal/Vertical command works in two modes. In one mode, you can fix the orientation of a line as either horizontal or vertical by selecting any point on the line that is not an endpoint or a midpoint.

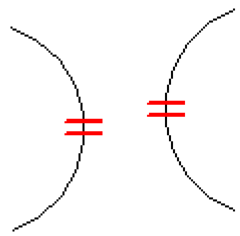


In the second mode, you can apply vertical/horizontal relationships between graphic elements by aligning their midpoints, center points, or endpoints so that their positions remain aligned with respect to each other.



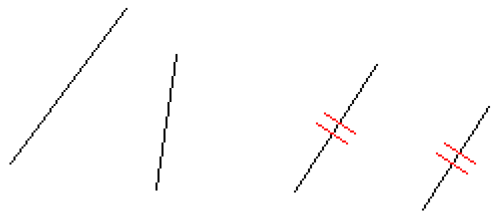
Equal

The Equal command maintains size equality between similar elements. When this relationship is applied between two lines, their lengths become equal. When applied between two arcs, their radii become equal.



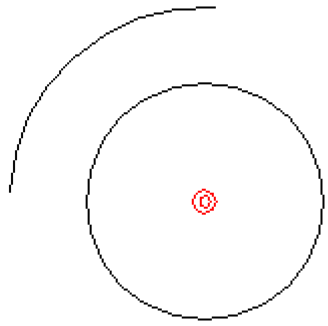
Parallel

The Parallel command makes two lines share the same angled orientation.



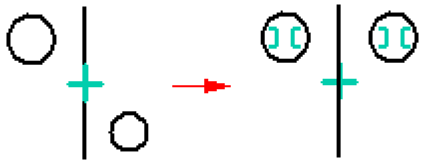
Concentric

The Concentric command maintains coincident centers for arcs and circles.



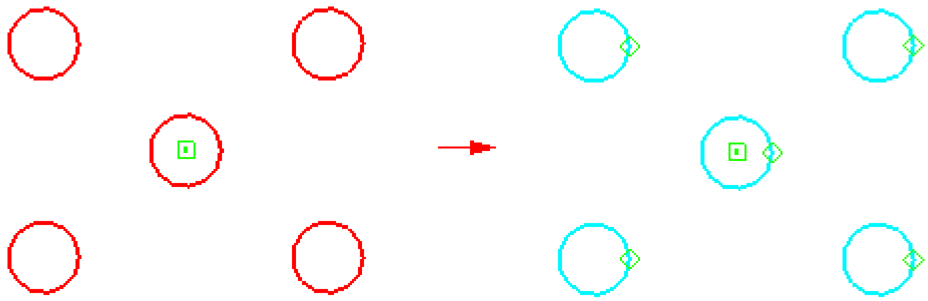
Symmetric

You can use the Symmetric command to make elements symmetric about a line or reference plane. The Symmetric command captures both the location and size of the elements.



Rigid Set

You can use the Rigid Set command to add a rigid set relationship to a group of 2-D elements.



Drawing Tools

Solid Edge provides tools to help you draw quickly and precisely in a variety of situations.

Grid

Grids help you draw with precision when the endpoints of elements you are drawing fit within regular intervals.

IntelliSketch

IntelliSketch helps you create, and optionally maintain, geometric relationships between elements. As you draw, IntelliSketch recognizes the opportunity to relate new elements to existing elements and displays visual cues that help make elements connected, tangent, collinear, perpendicular, parallel, and so forth.

Based on your preference, Solid Edge will either maintain the relationships that IntelliSketch creates or only use IntelliSketch to create new elements with precision, without maintaining relationships as you add and change geometry.

Projection Lines

Projection lines help you maintain alignment of key points, for example between related 2D Drawing Views of a model. Projection lines fulfill the function of the squares, triangles, and parallel rules used in classical drafting.

Working with grids

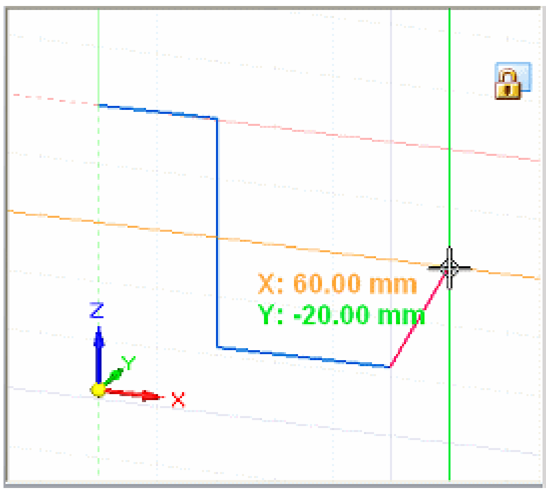
The grid helps you draw and modify elements relative to known positions in the working window. It displays a series of intersecting lines or points, and X and Y coordinates, which enable you to draw 2D elements with precision. You can use the grid with all sketching, dimensioning, and annotation functions. It also works with IntelliSketch and the Select command.

For example, you can use the grid to:

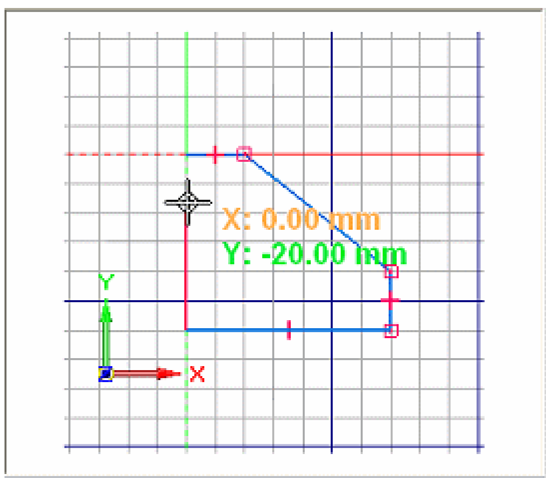
- Draw elements at known locations, draw elements known distances apart, and so forth. For an example, see Help topic Draw a line with a grid.

- Align dimensions and annotations by snapping them to grid points or lines. Only bolt hole circles and center marks cannot be snapped to a grid. For an example, see Help topic Place a dimension or annotation using a grid.

synchronous environment



ordered environment



Displaying the grid and setting options

Use the Grid Options command to open the Grid Options dialog box, where you can turn the grid on and off. When the Show Grid option is set, the grid is displayed whenever you create or modify 2D elements.

You also can use the Grid Options dialog box to:

- Turn alignment lines on and off.
- Turn snap-to-grid on and off.
- Turn coordinate display on and off.
- Change grid spacing.
- Change grid line color.

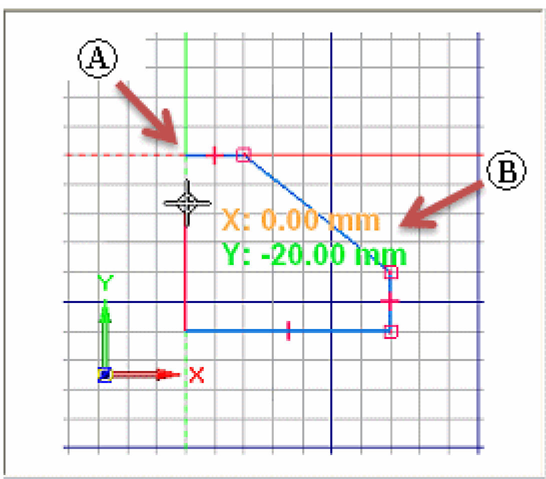
To change the grid origin line colors, you must change the Select and Highlight colors on the Colors page in the Solid Edge Options dialog box.

How grids work in the ordered environment

The grid is displayed in Draft and in profile and sketch mode as you draw, dimension, and annotate 2D elements. The X and Y coordinates it displays are relative to an origin point (A), which you can position anywhere in the window. The origin point is marked by the intersection of the X and Y origin lines.

As you move the cursor, the horizontal and vertical distance between the cursor position and the origin point is dynamically displayed (B).

If the Snap To Grid option is on when you add dimensions and annotations, they will snap to grid lines and points.



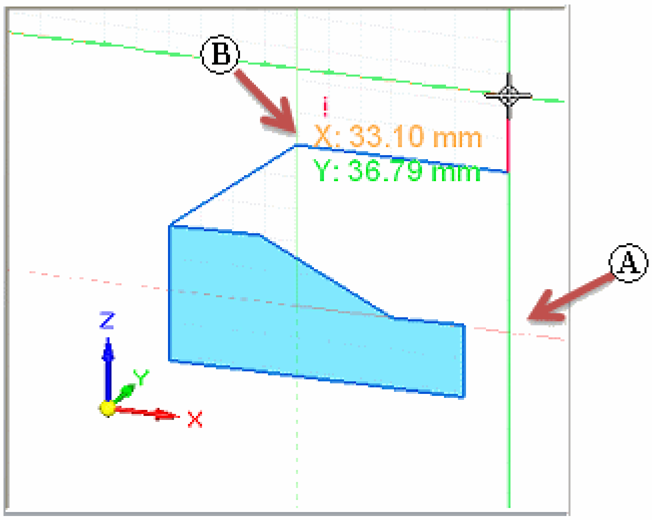
How grids work in the synchronous environment

The grid is available for drawing and editing 2D elements, and for adding 2D dimensions and annotations.

Grid visibility is somewhat different in Draft than in synchronous modeling environments. In Draft, when the grid is turned on, it is always visible. In synchronous modeling, the grid is visible only when a sketch plane is locked.

In 3D environments, the grid helps you draw horizontally and vertically with respect to part edges and model faces by displaying a series of intersecting lines or points, and by displaying alignment lines. The grid also helps you draw with precision by displaying X and Y coordinates that are relative to an origin point (A), which you can position anywhere in the window.

As you move the cursor, the horizontal and vertical distance (B) and orientation between the cursor position and the origin point is displayed and updated.

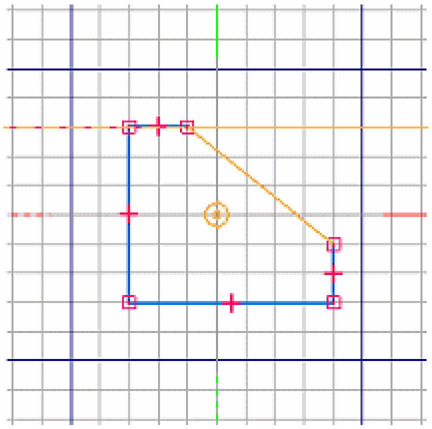


If the Snap To Grid option is on when you add dimensions and annotations, they will snap to grid lines and points.

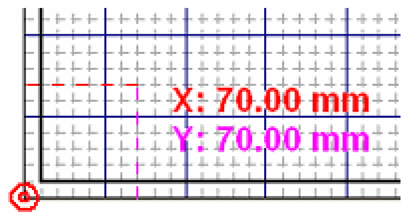
Recognizing the grid origin

The grid origin is marked by the intersection of the X and Y origin lines.

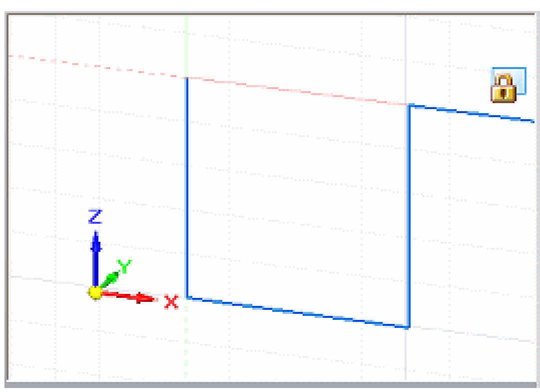
- In ordered profile and sketch, the default display mode is a red dashed line for the X axis and a green dashed line for the Y axis. The user-defined grid origin point is marked by a circle and dot. The default origin is at the center of the profile or sketch reference plane.



- In Draft, the default display mode is a red dashed line for the X axis and a magenta dashed line for the Y axis. The user-defined grid origin point is marked by a concentric circle and dot. The default origin is the (0,0) location of the drawing sheet.



- In the synchronous modeling environment, the default display color scheme matches that of the user-defined origin triad in the center of the graphics window. The X axis is a red line, and the Y axis is green. These lines are solid in the positive direction and dashed in the negative direction. There is no marker at the user-defined origin point. The default origin is the 0,0,0 center of the currently locked sketch plane.



Moving the grid origin

You can move the grid origin point using either of these commands:

- Use the Reposition Origin command to move the origin to a user-defined location. This is helpful when you want to do any of the following:
  - Add dimensions or constraints that are horizontal or vertical to a model edge.
  - Draw lines and other elements at a precise distance from another element at a known location.
  - Offset a series of elements by the same distance from a known location.
- To automatically reset the origin point to match the origin of the drawing sheet or working plane, use the Zero Origin command .

  Note

  The Reposition Origin and Zero Origin commands are available in synchronous modeling environments only when a sketch plane is locked.

See the Help topic, Reposition the grid origin point.

Changing the grid orientation

In ordered profile and sketch, the default orientation for the x-axis of the grid is horizontal to the profile or sketch reference plane. You can reorient the x-axis to any angle using the Angle option on the Grid Options dialog box.

In the synchronous modeling environment, the orientation of the grid axes matches the origin axes of the currently locked sketch plane. When you lock onto a different sketch plane, the origin axes reorient to the new plane. You can use the Reposition Origin command to do the following:

- Change the grid angle. See the Help topic, Reposition the sketch plane origin.
- Ensure that dimensions placed on coplanar geometry remain horizontal and vertical. See the Help topic, Set sketch plane horizontal and vertical for dimensioning.

In Draft, the default orientation for the X-axis of the grid is horizontal. You can reorient the X-axis to any angle using the Angle option on the Grid Options dialog box.

IntelliSketch

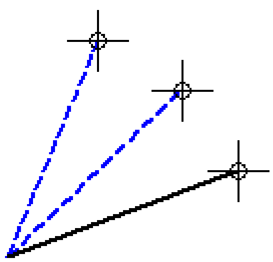
IntelliSketch is a dynamic drawing tool used for sketching and modifying elements. IntelliSketch allows you to sketch with precision by specifying characteristics of the design as you sketch.

For instance, IntelliSketch allows you to sketch a line that is horizontal or vertical, or a line that is parallel or perpendicular to another line or tangent to a circle. You can also draw an arc connected to the end point of an existing line, draw a circle concentric with another circle, draw a line tangent to a circle—the possibilities are too numerous to list.

IntelliSketch places dimensions and geometric relationships on any new 2D elements as you draw them. You can use another tool, the Relationship Assistant, to place dimensions and relationships automatically on existing profile elements.

How IntelliSketch works

As you draw, IntelliSketch tracks the movement of the cursor and shows a temporary, dynamic display of the element you are drawing. This temporary display shows what the new element will look like if you click at the current position.



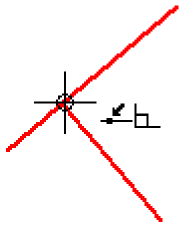
IntelliSketch gives you more information about the element you are drawing by displaying relationships between the temporary, dynamic element and the following:

- Other elements in the drawing
- Horizontal and vertical orientations
- The origin of the element you are drawing

When IntelliSketch recognizes a relationship, it displays a relationship indicator at the cursor. As you move the cursor, IntelliSketch updates the indicator to show new relationships. If a relationship indicator is displayed at the cursor when you click to draw the element, the software applies that relationship to the element. For example, if the Horizontal relationship indicator appears when you click to place the second end point of a line, then the line will be horizontal.

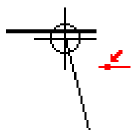
IntelliSketch relationships

You can set the types of relationships you want IntelliSketch to recognize on the Relationships page on the IntelliSketch dialog box. IntelliSketch can recognize one or two relationships at a time. When IntelliSketch recognizes two relationships, it displays both relationship indicators at the cursor.



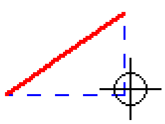
IntelliSketch locate zone

You do not have to move the cursor to an exact position for IntelliSketch to recognize a relationship. IntelliSketch recognizes relationships for any element within the locate zone of the cursor. The circle around the cursor crosshair or at the end of the cursor arrow indicates the locate zone. You can change the size of the locate zone with the IntelliSketch command on the Tools menu.



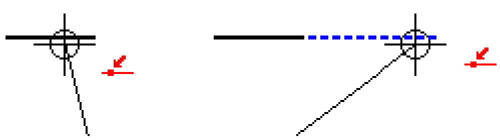
Alignment indicators

IntelliSketch displays a temporary dashed line to indicate when the cursor position is horizontally or vertically aligned with a key point on an element.



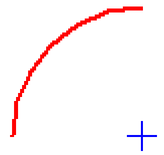
Infinite elements

IntelliSketch recognizes the Point On Element relationship for lines and arcs as if these elements were infinite. In the following example, IntelliSketch recognizes a Point On Element relationship when the cursor is positioned directly over an element and also when the cursor is moved off the element.



Center points

IntelliSketch displays an indicator at the center point of an arc or circle to make this keypoint easy to locate.



Snapping to points

When drawing and manipulating 2D elements, you can use shortcut keys with QuickPick to snap to keypoints and intersection points. This also applies the point coordinates as input to the command in progress.

Once you have highlighted the element you want to snap to with the cursor, you can use these shortcut keys to snap to points:

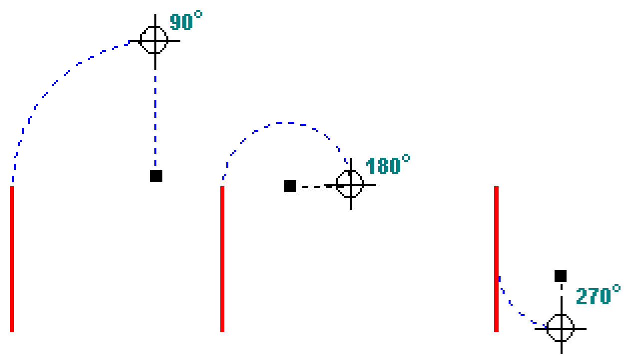
- Midpoint - press M.
- Intersection point - press I.
- Center point - press C.
- Endpoint - press E.

To learn more, see Selecting and snapping to points.

Sweep angle lock at quadrants

When you draw tangent or perpendicular arcs, the arc sweep angle locks at quadrant points of 0, 90, 180, and 270 degrees. This allows you to draw common arcs without typing the sweep value on the command bar.

A temporary dashed line appears from the arc endpoint to the center line of the arc to notify you that the arc is at a quadrant.



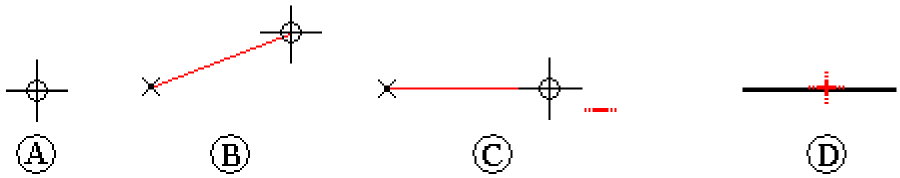
Automatic dimensioning

You can use options on the Auto-Dimension page in the IntelliSketch dialog box to automatically create dimensions for new geometry. The page provides several options to control when the dimensions are drawn as well as whether to use dimension style mapping or not.

You can use the Auto-Dimension command as a quick way to turn automatic dimensioning on and off.

Example: Draw a horizontal line

You can use IntelliSketch to draw a line that is exactly horizontal. You can apply a horizontal relationship as you draw the line, or draw the line without a horizontal relationship.



1. Choose the IntelliSketch command  on the Home tab or the Sketching tab.

2. In the IntelliSketch dialog box, on the Relationships tab, set the Horizontal Or Vertical option, and then click OK.

3. Choose the Line command.

4. Click where you want to place the first end point of the line, anywhere in the application window (A).

5. Move the cursor around in the window (B). Notice that the dynamic line display always extends from the end point you just placed to the current cursor position. You may also see IntelliSketch relationship indicators displayed at the cursor.

6. Move the cursor to make the dynamic line approximately horizontal.

7. When the IntelliSketch Horizontal relationship indicator is displayed at the cursor (C), click to place the second end point.

IntelliSketch places a horizontal relationship handle on the new line (D).

Tip

Relationship handles can be displayed or hidden with the Relationship Handles command.

Tip

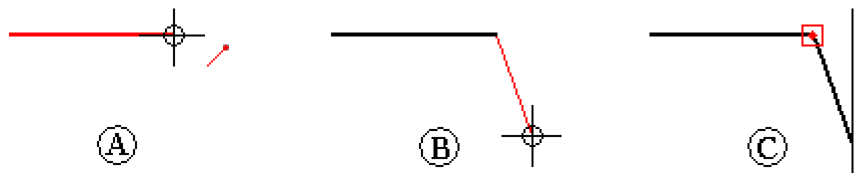
To snap to an intersection point or a keypoint, locate the element(s) with the cursor and then press one of these shortcut keys.

- Midpoint of a line or arc: press M.
- Intersection point of lines, circle, curves, and arcs: press I.
- Center point of a circle or arc: press C.
- Endpoint of a line, arc, or curve: press E.

For intersection points—If there are multiple eligible points located, then QuickPick opens and lists them. In QuickPick, click to select the point you want.

Example: Draw a line connected to another line

You can use IntelliSketch to connect an element you are drawing with an existing element. You can apply a connect relationship as you draw the lines, or draw the line without a connect relationship.



1. Choose the IntelliSketch command .
2. In the IntelliSketch dialog box, on the Relationships tab, set the End Point option, and then click OK.
3. Choose the Line command.
4. Move the cursor to the end of a line in the application window. As you move the cursor over it, the line is highlighted and IntelliSketch displays the End Point relationship indicator at the cursor.
5. While IntelliSketch displays the relationship indicator, click to place the first end point of the new line (A). This end point is connected to the end point of the previous line.

   Tip

   Rather than clicking, you can snap to the line end point nearest the cursor by pressing the E key.

6. Click where you want to place the second end point of the new line.
7. The new line and the previous line have connected end points (B).

IntelliSketch places a connect relationship handle at the point where the two lines connect (C).

Tip

Relationship handles can be displayed or hidden with the Relationship Handles command.

Tip

Relationships are maintained only if the Maintain Relationships command is set.

Tip

To snap to a keypoint or intersection point, locate the element(s) with the cursor and then press one of these shortcut keys.

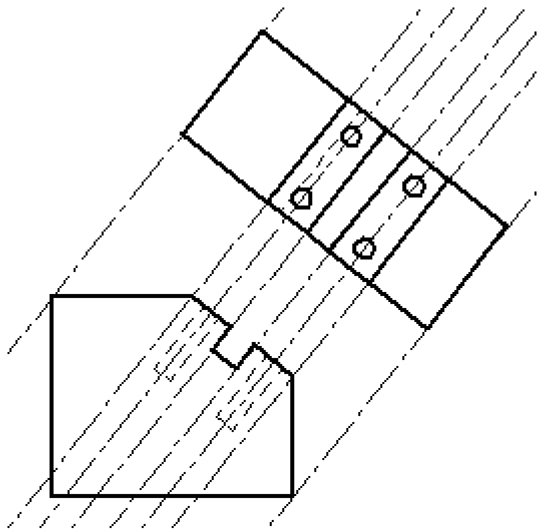
- Midpoint of a line or arc: press M.
- Intersection point of lines, circle, curves, and arcs: press I.
- Center point of a circle or arc: press C.
- Endpoint of a line, arc, or curve: press E.

For intersection points—If there are multiple eligible points located, then QuickPick opens and lists them. In QuickPick, click to select the point you want.

Projection lines

Projection lines are extensions of lines that assist in 2D drawing.

- You can use projection lines to help you create new geometry, and any constraints you create with them remain active even after you turn projection lines off. For example, in a drawing, you can use projection lines on an auxiliary view to enable creation of additional views with proper alignment and size.



- You can create a line with the projection line option set, or you can edit an existing line and set the projection line property later.

- You can place dimensions and annotations to projection lines. Dimensions and annotations connect to the defining segment of the projection line (the original 2D line on which the projection line is based).

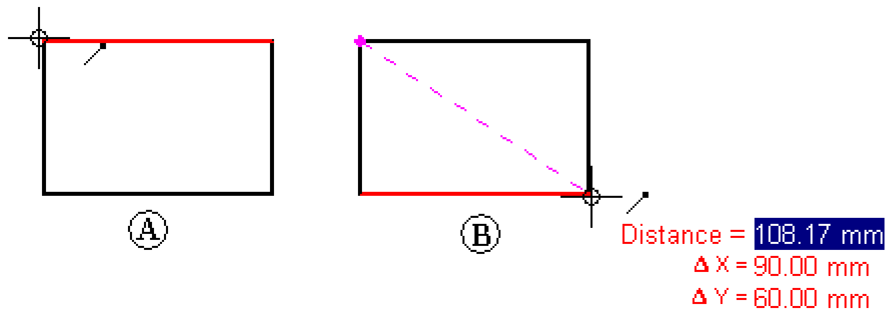
Projection lines are available as a line property on the Line command bar and on the Format page of the Element Properties dialog box.

Distance and area measurement

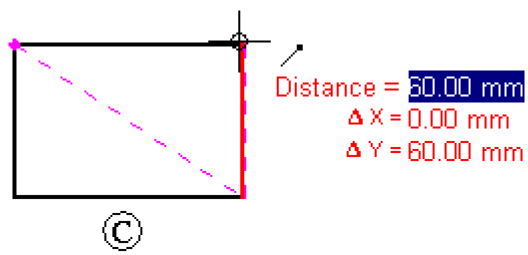
You can measure distances or areas, even when you are in the middle of another task. To set the units for measuring distances or areas, use the Properties command on the Application menu.

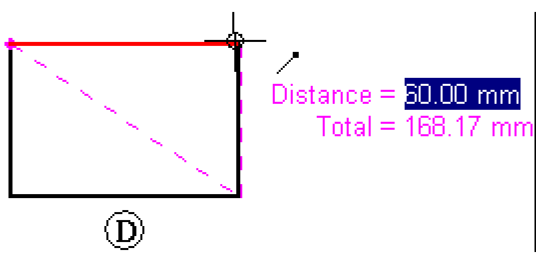
Measuring distances in 2D

In the Draft environment, you can measure distance using the Measure Distance command. These commands measure linear distances or measure the cumulative linear distance along a series of points. The first point you click establishes the origin of the measurement (A). After that, you can select any keypoint to see the distance between it and the origin, as well as the delta distance along each principal axis (B).



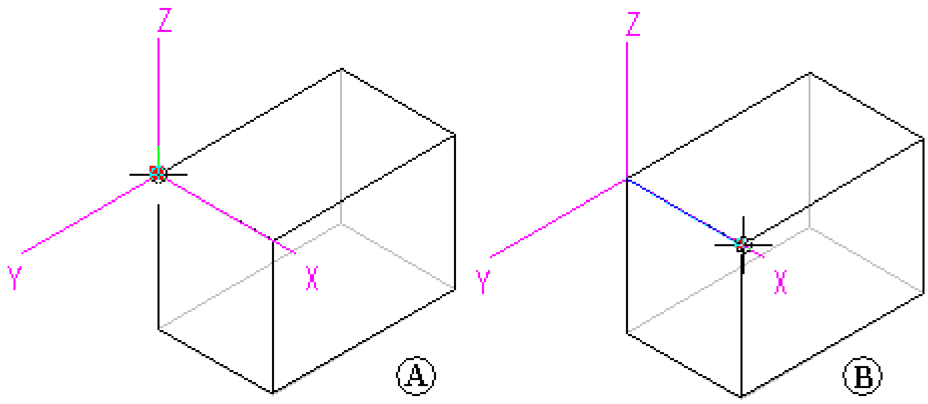
Clicking the keypoint adds it to a series of measurement points. Then you can select another point to see the new linear distance and deltas (C), or click it to see the distance between the last two points and the total cumulative distance from the origin to the last point (D). Click the right button to reset the command.



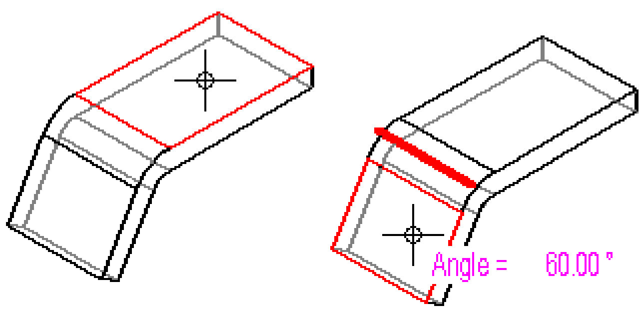


Measuring distances and angles in 3D

In the Part, Sheet Metal, and Assembly environments, the Measure Distance command measures linear distances. The first point you click establishes the origin of the measurement (A). After that, you can select any keypoint (B) to display the Measure Distance dialog box which displays the keypoint select type, the true distance, the apparent screen view distance, and the delta distance along each principal axis.



In the Part, Sheet Metal, and Assembly environments, the Measure Angle command measures angles. You can measure between any two faces or between any three points.



Measuring minimum distances

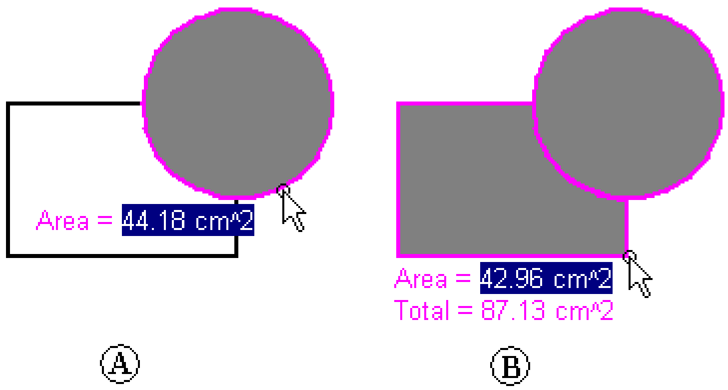
In the Part, Sheet Metal, and Assembly environments, you can use the Measure Minimum Distance command to measure the minimum distance between any two elements or keypoints. You can use the Select Type option on the Minimum Distance command bar to filter which type of elements you want to select. When working in the context of an assembly, you can also use the Activate Part option to activate the parts you want to measure.

Measuring normal distances

In the Part, Sheet Metal, and Assembly environments, the Measure Normal Distance command measures normal distances between a planar element or line and a keypoint. You can use the Element Types option on the Measure Normal Distance command bar to filter which type of elements you want to select. You can use the Key Point option to specify the type of keypoint you want to identify when measuring the distance. You can use the Coordinate System option to select a user-defined coordinate system to define one of the points. If you use a coordinate system, the returned values will be relative to the specified coordinate system. When working in the context of an assembly, you can also use the Activate Part option to activate the parts you want to measure.

Measuring areas

The Measure Area command, available only in the Draft environment and in 2D profiles and sketches, measures the area inside a closed boundary (A). You can also measure the cumulative area inside more than one closed boundary by holding the Shift key as you click elements (B). Each time you click, the area of the last element is displayed, along with the total area. Click another element without holding the Shift key to reset the command.



Measuring lengths

The Measure Total Length command measures the cumulative length of a select set of 2D geometry.

Measuring automatically

In addition to the individual distance, area, length, and angle commands, you can use the Smart Measure command in 2D and 3D environments to measure automatically based on what you select:

- Select a single 2D element or 3D object to measure its length or its angle or radius.

- Select two or more 2D elements or 3D objects to measure the distance or angle between them.

The Smart Measure command works like the Smart Dimension command, except that it does not place a dimension as a result.

Copying measurement values

You can copy the highlighted measurement value to the Clipboard by pressing Ctrl+C. You can then use the copied value as input for another command. For example, you can paste the copied value into the Line command bar to define the length of a line. Use the Tab key if you want to highlight a different value.

Measuring drawing view geometry

When you measure model geometry within a drawing view, or when you measure distances between model edges in two drawing views, you can select the Use Drawing View Scale check box on the command bar to specify that the measured value is displayed using the equivalent of the model distance.

Alternatively, you can apply a user-defined scale value by selecting it from the Scale list on the command bar.

Note

- You can show the scale of a drawing view using the General page (Drawing View Properties dialog box).

- User-defined scale values are defined in the Drawing View Scales section of the Custom.xml file, in the Solid Edge Program folder. See the Help topic, Add custom drawing view scales to Solid Edge.

Example: Measuring the length of a line

Even when you are in the middle of a task, you can measure distances with the Measure Distance command. For example, consider the following workflow.

1. Use the Line command to draw a line (A).

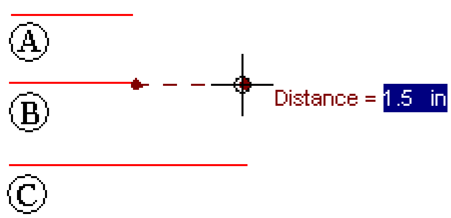
2. On the Inspect tab, click the Measure Distance command and measure a distance (B).

   Note

   You do not need to exit the Line command before measuring a distance.

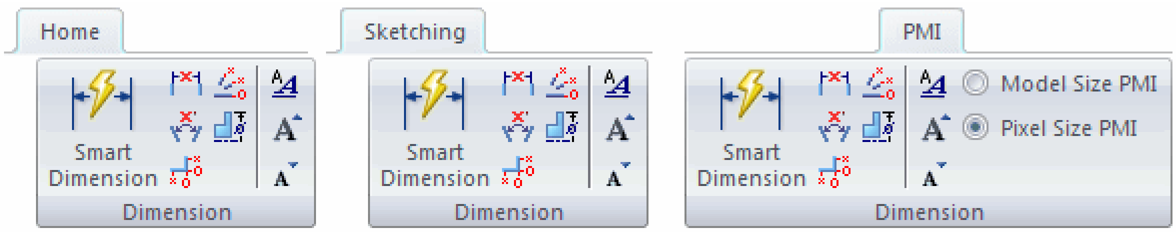
3. To exit the Measure Distance command, right-click. The Line command is still active-you can pick up where you left off.

4. Continue using the Line command (C).



Dimensioning sketches

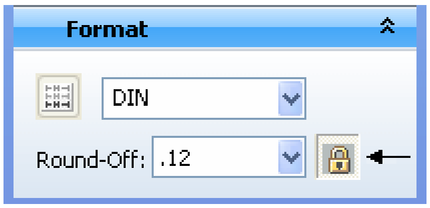
Dimensioning commands are located in three locations. They are located in the Dimension group on the Home, Sketching, and PMI tabs.

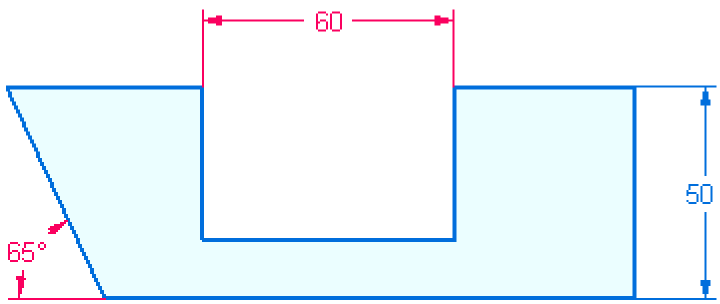


Locked dimensions

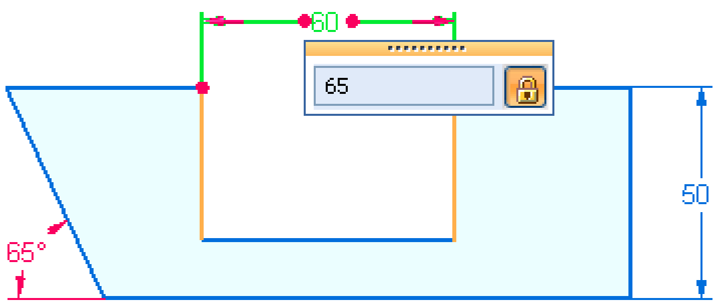
Sketch dimensions are placed as driving. A driving dimension is colored red. A driving dimension is also referred to as a locked dimension. A locked dimension cannot change unless it is edited directly. As sketch geometry is modified, a locked dimension does not change.

Change a dimension to driven (or unlocked) by selecting the dimension and then clicking the lock on the Dimension Value Edit QuickBar. A driven dimension is colored blue. A driven dimension value cannot be selected for editing. It must be changed to a locked dimension to change its value directly.



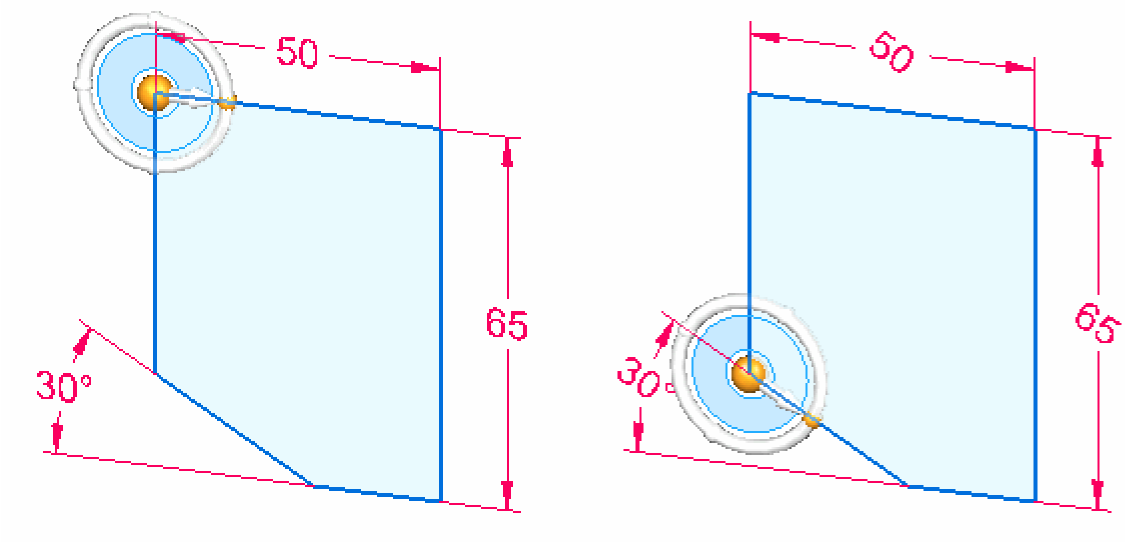


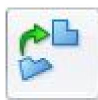
To change a dimension value of a locked dimension, click the dimension value and enter a new value.

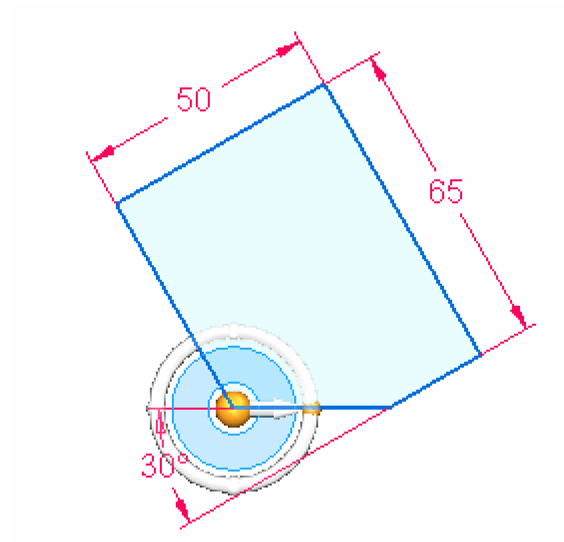


Dimension orientation

The orientation of a sketch dimension is controlled by the sketch plane origin. The sketch plane origin defines the horizontal/vertical direction.

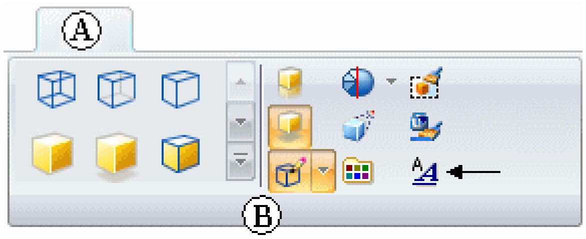


The Sketch View command  orients the view to where the dimension text is horizontal.

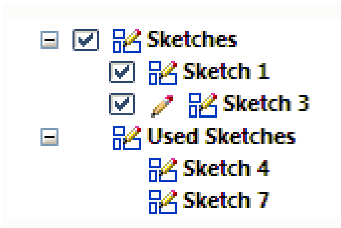


Dimension style

Modify the dimension style settings in the Style dialog box. The Style command is located on the View tab (A) in the Style group (B).

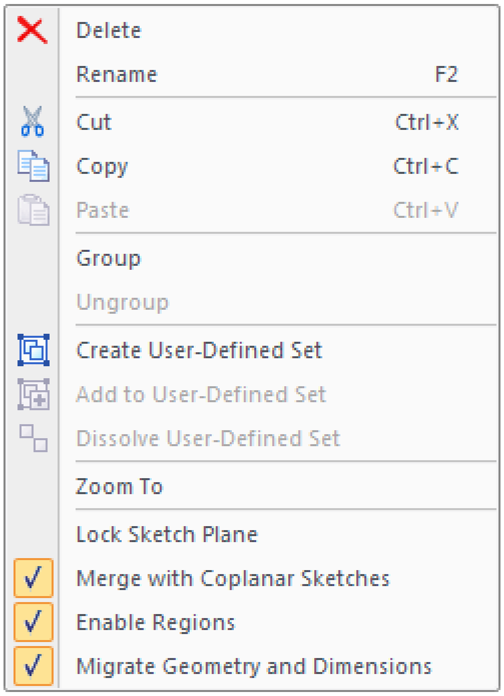


Sketches in PathFinder



- In PathFinder, there are two sketch collectors (Sketches and Used Sketches).
- Sketches are stored in a Sketches collector until they are consumed by body creation or deleted.
- The pencil symbol in front of a sketch denotes that its sketch plane is locked.
- Unconsumed sketches can be displayed or hidden with a check mark. All sketches or specific sketches can be hidden or shown.
- Sketch elements used to create a feature are removed from the Sketches collector and placed in the Used Sketches collector.
- Used sketches can be highlighted, deleted, renamed or restored.

Sketches context menu in PathFinder



The sketches context menu includes options on how a sketch responds to creation of regions and feature creation. These options are on a per sketch basis.

Merge with Coplanar Sketches

- If a sketch exists on the locked sketch plane, then any new sketch geometry merges with the existing sketch.

Enable Regions

- Locate regions as sketch geometry forms closed area(s).

Migrate Geometry and Dimensions

- Consume the sketch geometry used to create features and move into the geometry into the Used Sketches collector.
- Create the PMI dimensions on the body as the sketch geometry is used to create features.

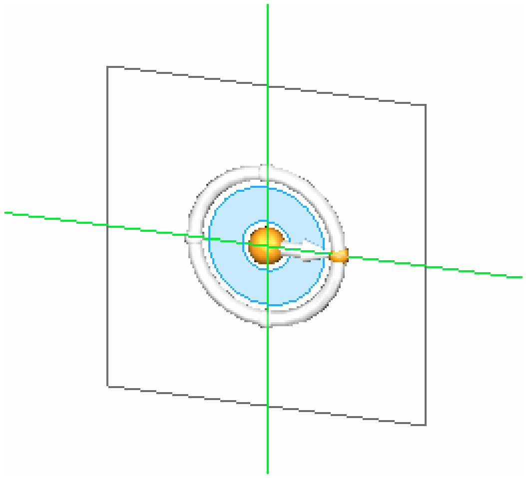
Used Sketches context menu in PathFinder

Right-click on a used sketch to bring up the context menu. The Restore command restores a consumed sketch to the Sketches collector. If a sketch plane exists that is the same as the used sketch plane, the restored sketch will merge with the existing sketch.

Sketch plane origin

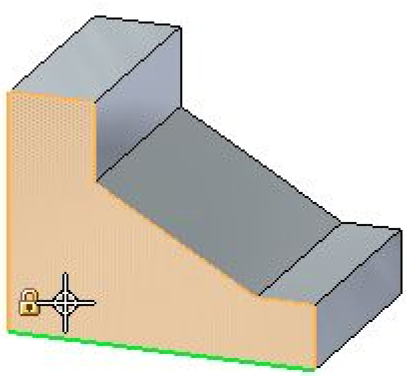
reference planes

The origin of a reference plane is system defined. The horizontal direction and the origin are positioned at the center of the reference plane. To change the reference origin, use the Reposition Origin command to define the new origin and horizontal direction.



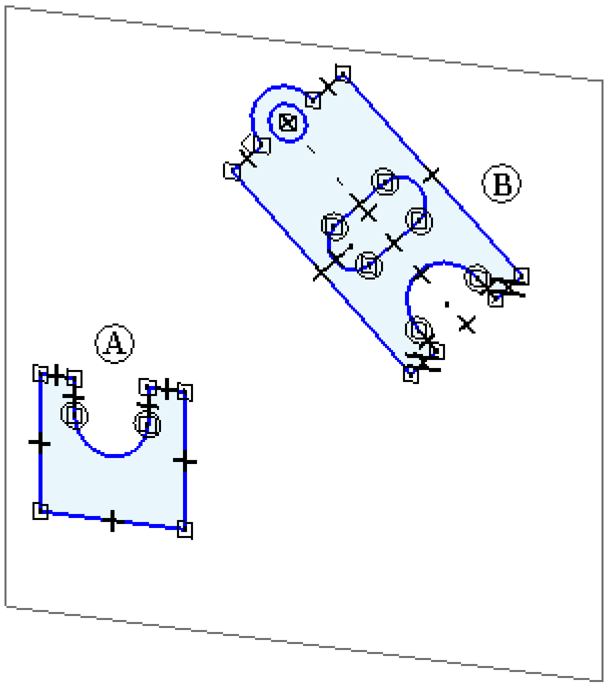
planar faces

The origin of a sketch on a planar face can be defined before locking the plane. The system determines a horizontal direction and origin. To change the system defined origin, cycle through the linear edges on the planar face. The edge displays green. Press (N) for next edge, (B) to go back to previous edge, (F) to flip the Y direction and (T) to toggle which end of the edge is used. Once the desired origin is displayed, click the lock to lock the sketch plane.



Reposition origin command

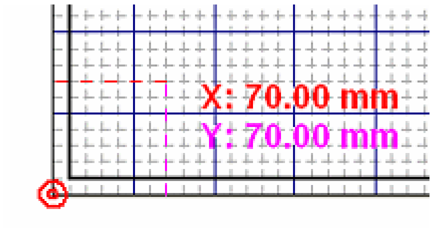
A sketch plane can only contain a single collection of sketch elements. However, there can be more than one sketch area per sketch plane. In the example, there are two sketch areas (A and B). Sketch area (A) orizontal/vertical directions are not the same as sketch area (B. The sketch plane origin can be repositioned to redefine the horizontal/vertical directions for a particular sketch area.



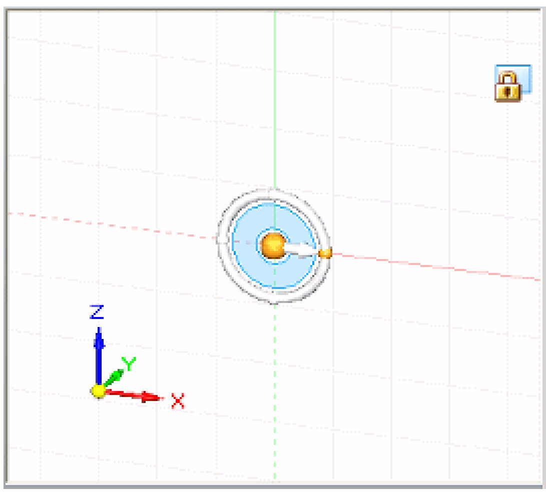
Zero Origin command

The Zero Origin command automatically resets the origin as follows:

- In Draft, the drawing grid origin is reset to the drawing sheet (0,0) coordinate.



- In the synchronous environment, both the drawing grid and the sketch plane origin are reset to the (0,0,0) coordinate and orientation at the center of the currently locked sketch plane.



Note

This command is available only when you have locked a sketch plane.

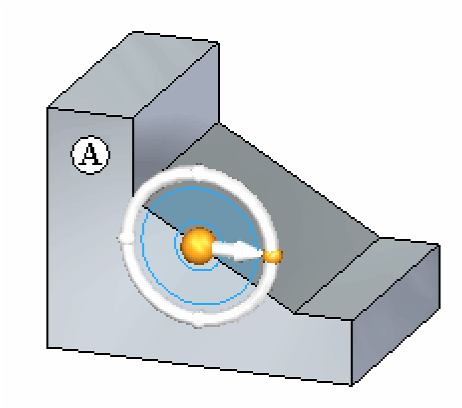
Reposition origin workflow

Workflow based on using a planar face for a sketch plane.

1. Lock the sketch plane.

2. In the Draw group, choose the **Reposition Origin** command.

   The reposition origin handle displays at the sketch plane origin on the locked plane (A.

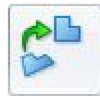
   

3. Click and drag the handle origin to a new vertex or edge.

   This defines the new origin.

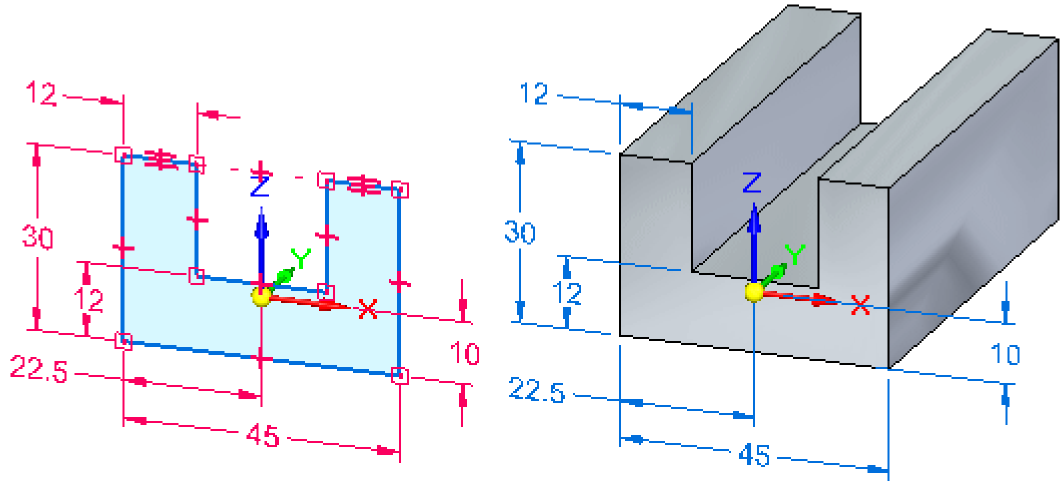
4. Click and drag the torus to position the horizontal direction. Select a keypoint or type in angle to lock the direction.

Sketch view

On the View tab® Views group, the Sketch View command  orients the active view normal to the horizontal/vertical direction of the locked sketch plane.

Sketch consumption and dimension migration

In synchronous part and sheet metal environments, you typically draw 2D sketch geometry for the purpose of constructing features on a solid model. In the synchronous environment, when you use sketch elements to construct a feature, the sketch elements are consumed and the 2D dimensions you placed on the sketch migrate to the appropriate edges on the solid body, whenever possible.



You can use the Migrate Geometry and Dimensions command command on the shortcut menu when a sketch is selected in PathFinder to control whether sketch elements are consumed and dimensions are migrated.

Automatic sketch consumption and dimension migration

By default, the Migrate Geometry and Dimensions command is set for a new document. The sketch elements are automatically consumed and 2D dimensions are automatically migrated when you use them to construct features. After you construct a feature, the 2D sketch geometry is moved to the Used Sketches collector in PathFinder, and the 2D dimensions are migrated as 3D PMI model dimensions.

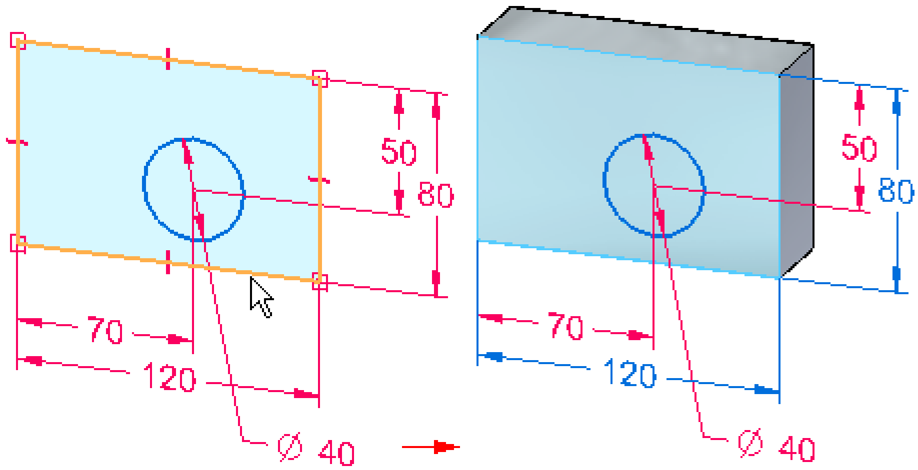
You can disable the automatic consumption of sketch elements and migration of 2D dimensions on a sketch-by-sketch basis by clearing the Migrate Geometry and Dimensions command on the shortcut menu when a sketch is selected in PathFinder.

All model dimensions, whether migrated from sketches or added to edges on the 3D model directly, are PMI dimensions. PMI dimensions are displayed on PathFinder in the PMI collection, Dimensions sub-collection.

To learn more about creating and using PMI, see the Help topic, PMI dimensions and annotations.

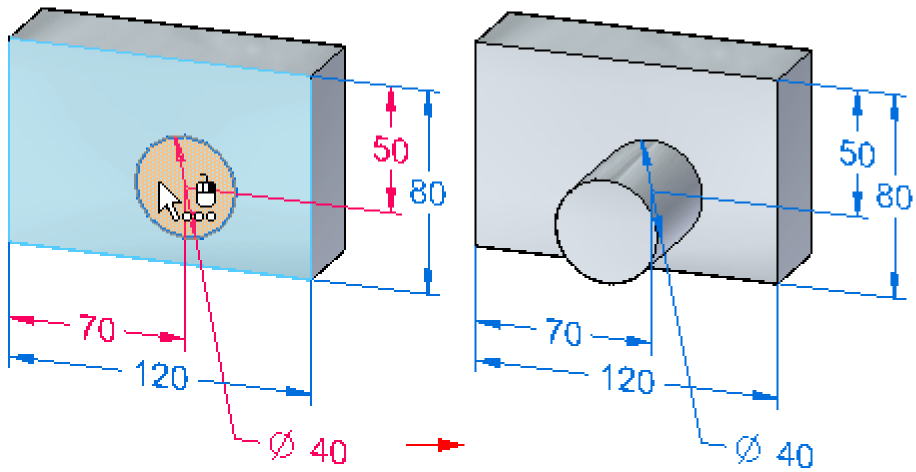
Partially migrated sketches and dimensions

In many cases, only some of the sketch elements on a single sketch are used to construct a feature. If this is the case, only the selected sketch elements and the associated 2D dimensions are consumed and migrated.



During this process, dimensions and constraints may be connected to both body edges and to remaining sketch geometry. If the sketch contains stacked dimensions, then some dimensions in the stack may migrate individually. Other dimensions, such as coordinate dimensions, do not migrate until all of the 2D geometry they are attached to has been used to construct a feature.

As you continue to construct features using the remaining sketch elements, sketch elements are consumed and dimensions are migrated.



Dimension locking status after migration

2D dimensions are locked by default. When they migrate to the 3D model, they remain locked.

Note

Dimension colors are determined by settings on the Colors page of the Options dialog box.

Dimension variable and formula migration

Sketch dimensions that use variables retain the variables after migration to PMI dimensions. If a sketch dimension is driven by a formula, the formula is maintained when the dimension is migrated to a PMI dimension. The PMI dimension is still driven by the formula, but must be driving for the formula to solve properly.

Working with combinable sketches

You can use the Merge Coplanar Sketches command on the shortcut menu to control whether a sketch is combined with another coplanar sketch in a synchronous part or assembly.

Although this command is available in synchronous part, sheet metal, and assembly documents, the merge property is most useful when working with assembly sketches, and it also plays a role when converting traditional parts and assemblies into synchronous documents.

When you set the Merge Coplanar Sketches option for a sketch, the following rules and conditions apply:

- New sketch elements that are drawn coplanar to the sketch in free space are added to the existing sketch. To create the new coplanar sketch elements as a separate sketch, you can select the existing sketch and clear the Merge with Coplanar Sketches command before you draw the new, coplanar sketch elements.

- The sketch will combine with another sketch if the two sketches become coplanar during a move operation.

- In part and sheet metal documents, sketch regions are automatically enabled for a combinable sketch. When sketch regions are enabled, you can use the Select tool to construct features using the sketch. You can clear the Enable Sketch Regions command on the shortcut menu to disable sketch regions.

- In part and sheet metal documents, sketch consumption is automatically enabled for the combinable sketch. When sketch consumption is enabled, sketch elements are consumed when you construct features from the sketch. You can clear the Migrate Geometry and Dimensions command on the shortcut menu to disable sketch consumption.

Unique symbols are used in PathFinder to indicate whether a sketch is a combinable sketch, noncombinable sketch, or the active sketch.

Legend

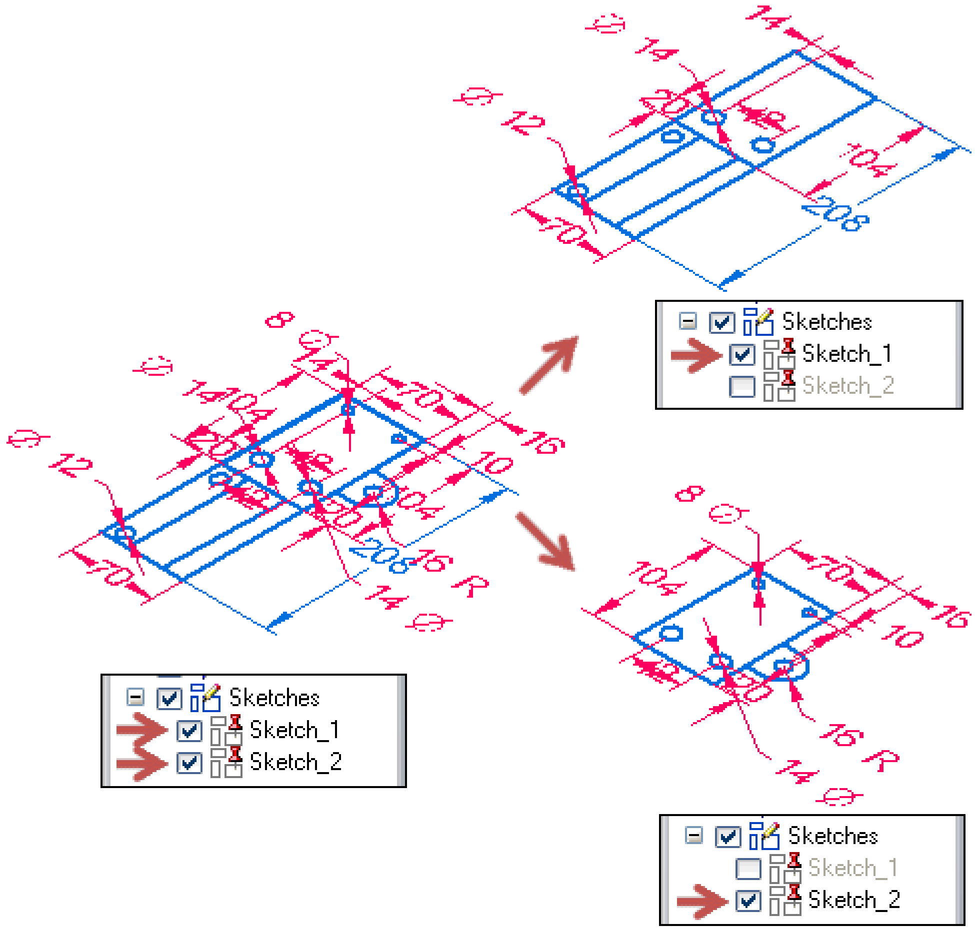
 Combinable sketch

 Noncombinable sketch

 Active sketch (combinable active sketch shown)

Assembly sketches

A noncombinable sketch is most useful when creating layout sketches in an assembly. Noncombinable sketches make it possible to draw multiple sketches which are coplanar. This can be useful if you want to create separate, coplanar sketches that represent individual parts or subassemblies for a new assembly. Noncombinable sketches makes it possible to show, hide, or move a set of sketch elements easily.



For more information, see HELP topic Drawing sketches in assemblies.

Moving sketches

Sketches can be moved/copied in 2D or 3D. Sketches can be deleted or cut for pasting in 2D or 3D. This section covers three methods available for manipulating sketch geometry.

- Manipulating sketch elements in 2D
- Moving and copying sketch elements in 3D
- Copy, cut and paste sketch elements

Manipulating sketch elements in 2D

Planar sketch elements can be moved or copied with the 2D move command  found on the Sketching tab, in the Draw group.

2D sketch element manipulation is confined to the selected sketch element's plane. If sketch elements in the select set are on different sketch planes, an error box appears when choosing any sketch manipulation command.

Error message: Selected sketch geometry must lie in the same plane.

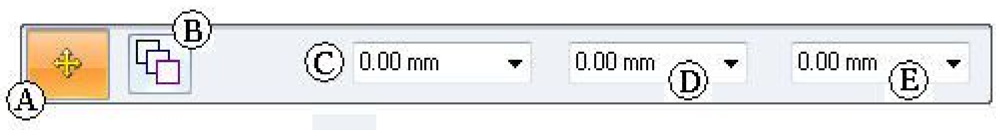
Workflow for moving or copying a sketch

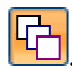
1. Select sketch elements to copy or move in the part window individually or with a select box. If the sketch elements form a region, disable regions for the sketch before using the select box.

2. Choose the Move command .

3. Select point (on any of the selected sketch elements) to move or copy from. You can use keypoints to define the move or copy from point.

4. Notice that there are options available in the Move (A) command bar.



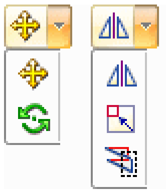
If you want to move a copy, select the **copy** option (B) . You can also enter the X (D), Y (E) distances to move or copy to. You can also enter a step distance Step field (C).

5. Click a 'to' point to move or copy to. If you are moving, then the command ends once a to point is clicked. The select set is still active. If you are copying, each click places a copy. A right-click during a copy will end the command with the select set still active.

6. Press Esc to clear select set.

2D sketch manipulation commands

Two lists are available that contain 2D commands for manipulating sketch geometry.



Manipulation commands are move, rotate, mirror, scale and stretch.

These commands each have a set of options in command bar. Each command also can manipulate a copy of the selected sketch elements.

Moving and copying sketch elements in 3D

Sketch elements can be moved/copied in 3D. The select set of sketch elements are not required to lie all on the same plane. The 3D move requires the use of the graphic handle. See the Graphic handle section in the “Moving and rotating faces” course to learn how to use the graphic handle.

The selected sketch elements can be:

- moved or copied in the sketch plane
- moved or copied to a parallel plane
- rotated or copy rotated to another plane

Workflow for a synchronous 3D move or rotate of sketch elements

1. Select sketch geometry.

   Selection methods

   - Select entire sketch(s) in PathFinder
   - Select sketch elements individually in the part window.
   - Select sketch elements in the part window with a select box

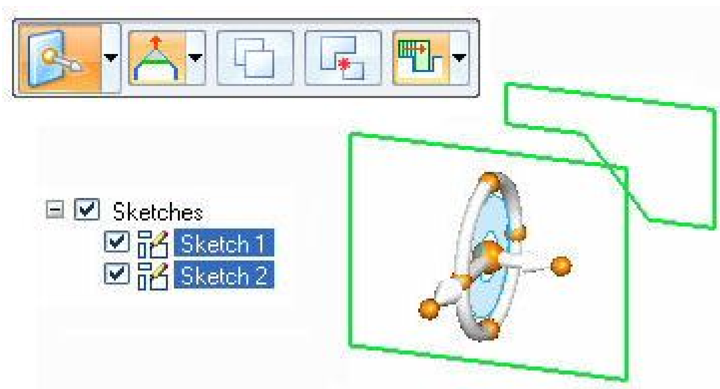
     Note

     If the sketch elements form a region, disable regions before using the select box.

     Note

     Select set can contain sketch elements on different planes.

2. If entire synchronous sketches are selected in PathFinder, the Move command starts.

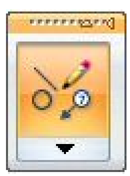
   

   Use the secondary axis or handle plane to move sketch elements in a plane.

   To rotate, drag the handle origin to an edge that will be the axis of rotation. Then click the torus to define the angle of rotation.

   Click the Copy option  on the command bar to move a copy of the selected sketch elements.

3. If sketch elements are selected in the part window, on the Modify command bar, choose the Move command from the drop list.

   Use the graphic handle as described in the previous step to move or rotate the selected sketch elements.

4. After sketches are manipulated and regions were disabled, you will need to remember to enable regions in order to create features from the sketches.

Copy, cut, and paste sketch elements

Sketch elements can be manipulated using the Microsoft clipboard behavior.

- Ctrl+C copies the selected sketch elements to the clipboard.
- Ctrl+X deletes the selected sketch elements from the model and adds them to the clipboard.
- Ctrl+V pastes the selected sketch elements in the model.

Paste behavior

A paste operation places the sketch elements (clipboard) onto the locked sketch plane at the location clicked. At this point, the paste elements are attached to the cursor and each click places another copy paste elements on the locked plane.

If there is no locked sketch plane, the sketch elements are placed onto the plane highlighted under the cursor at the location clicked. At this point, the paste elements are attached to the cursor and each click places another copy of the paste elements on the locked plane.

To select another plane to paste to, end the paste operation with the Esc key. Ctrl+V starts the paste operation again and then select the new plane.

Projecting elements onto a sketch plane

Face edges, sketch elements and base eference plane edges can be projected onto a locked sketch plane.

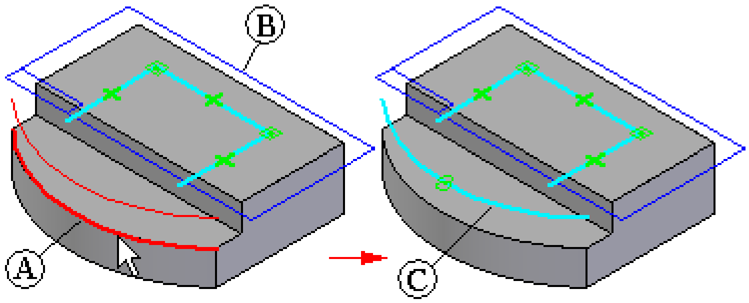
- The Project to Sketch command is located on the Sketching tab® Draw group.

- The Project to Sketch command requires a locked sketch plane.

- Use the Project to Sketch command bar to refine the selection of elements to project and to set the project options.

Project to sketch command

Copies part edges or sketch elements onto the current sketch plane. For example, select a part edge (A) to project onto the current sketch plane (B). The projected edge (C) can then be used in the current sketch.



You can project elements associatively or non-associatively.

Projected elements and associativity

Special relationship symbols are used to indicate that an element is associatively linked to another element. The symbols indicate whether the element is associatively linked to an element in the same document (local) or another document (peer-to-peer).

Link (local)	
Link (peer-to-peer)	

A projected element from the current part is always projected associatively. A projected element from another document can be associative or non-associative.

You can break the associative link on projected elements by deleting the link relationship symbols.

You can trim and modify associative or non-associatively projected elements, and incorporate associatively projected elements into a sketch that contains newly created non-associative elements.

Note

When you use sketch elements to construct a feature in a part document, the sketch elements are transferred to the Used Sketches collection in PathFinder. For projected elements, the associative link between the parent element and the projected element is discarded.

You can also add relationships or dimensions to associatively projected elements, but if the relationship or dimension conflicts with the associative relationship to the parent element, a warning message is displayed.

You can project edges between part documents, but it is not associative. It is a copy only.

Projecting elements from other documents

When working in an assembly, you can use the Project to Sketch command to project part elements, such as part edges, into an assembly sketch. You cannot project elements between two parts in an assembly.

Sketching instructional activities

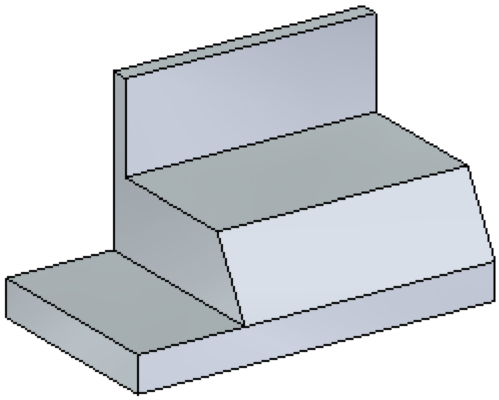
Part1

Activity covers plane locking, drawing sketch elements, placing dimensions, applying geometric relationships, showing relationship handles, reposition sketch plane origin and controlling sketch display.

Activity: Sketching (Part 1)

Open a part file

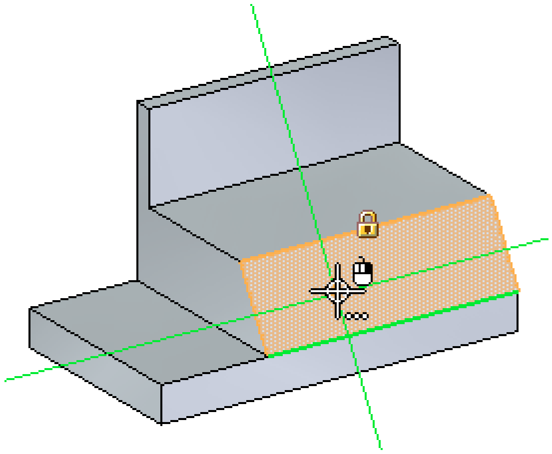
- Start Solid Edge.
- Click the  Application button® Open.
- In the Open File dialog box, set the Look in: field to the folder where the training files reside.
- Click *sketch_A* and then click Open.



Start the sketching process

- Choose the Line command.

- Define the sketch plane. Pause the cursor over the angled sketch plane. Press the N key until the green edge highlights as shown. This defines the horizontal direction for the sketch plane.



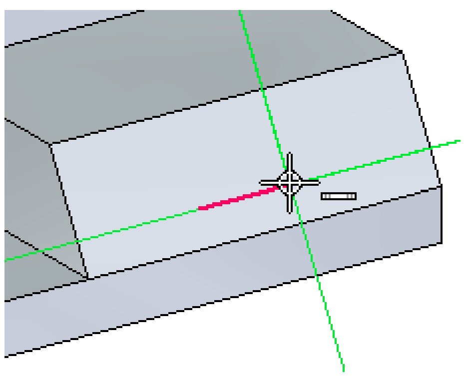
Note

While the plane highlights, you can begin sketching and you lock to the plane. If you move the cursor away from the plane before placing any geometry, you have to highlight the plane again. You could also click the lock on the highlighted plane to lock the plane. If you manually lock the plane, it remains locked until you unlock it.

Draw sketch geometry

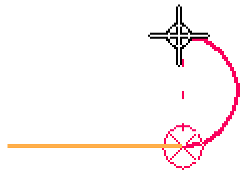
- Draw a slot shaped sketch consisting of two lines and two arcs. While the angled plane highlights, click to place the first point of the line.

- For the second point of the line, make sure the horizontal indicator displays and then click.

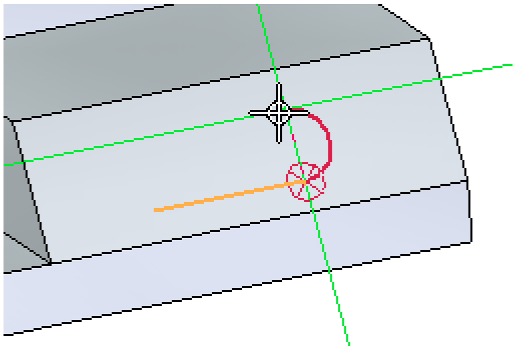


- Place a tangent arc. Press the A key to enter the place arc command.

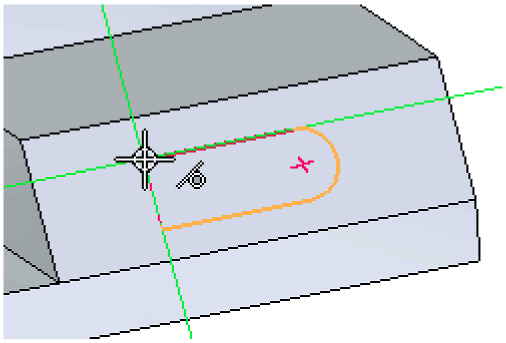
  Position the intent zone as shown.



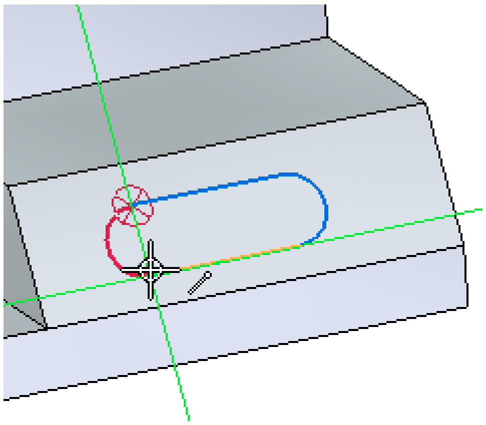
  Place the arc end point vertical from arc start point.



- Place the second line as shown. Make sure you get the tangent alignment symbol and the vertical alignment from the first point of the start line.

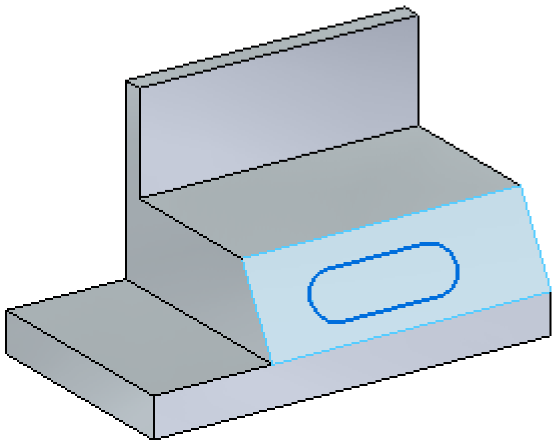


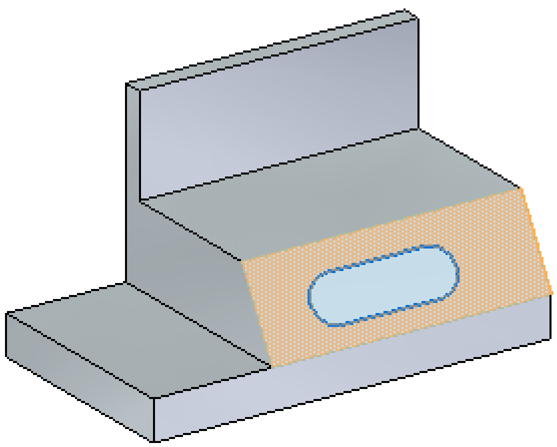
- Place the second tangent arc. Press A and then end the arc at the endpoint of the first line.

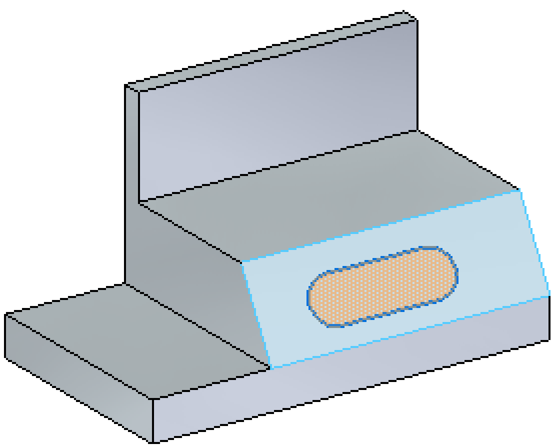


Regions formed

Notice the face changes to a blue color. This denotes the presence of regions. The sketch drawn on the face creates two regions.





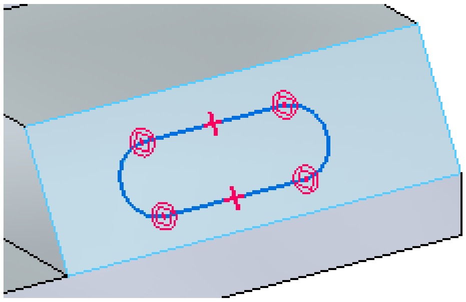


Place geometric relationships

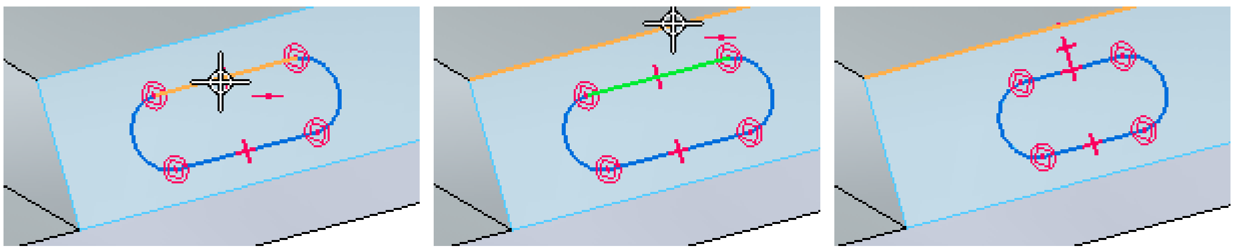
Center the slot sketch on the face using geometric relationships.

- Turn on the display of relationship handles. On the Sketching tab® Relate group, choose the Relationship Handles command.

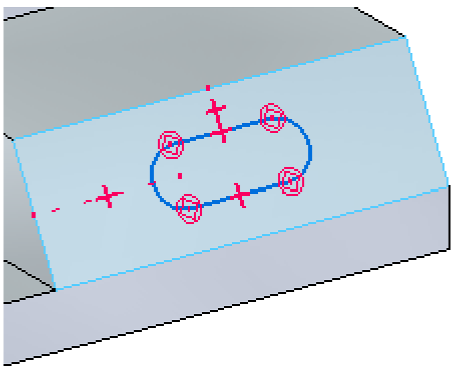
The handles show that the lines are horizontal and the arcs are tangent connected to the endpoints of the lines.



- Align the midpoint of one line to the midpoint of a face edge. In the Relate group, choose the Horizontal/Vertical command. Click the midpoint of the line and then click the midpoint of the face edge.



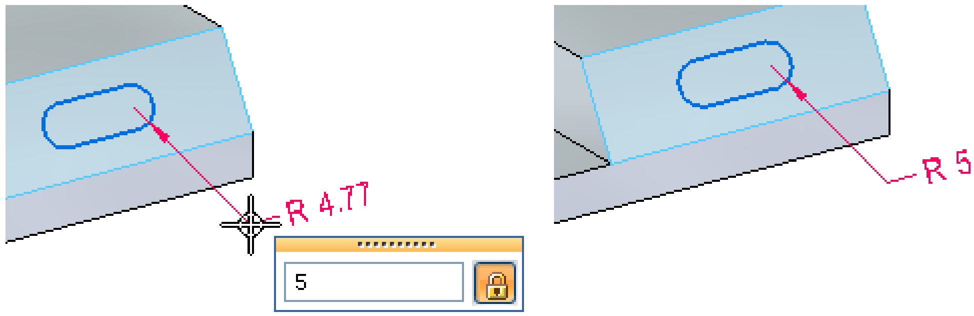
- Align the center of the arc to the midpoint of a face edge. Using the horizontal/vertical command, click the arc center and then the midpoint of the face edge. The slot is centered on the face.



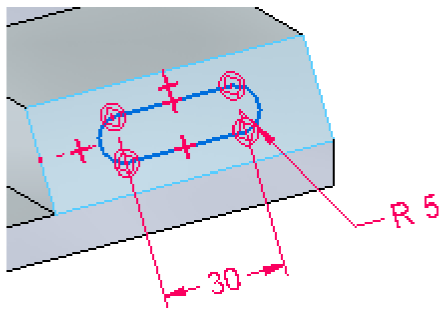
Add dimensions

Dimension the slot radius and distance between centers.

- On the Sketching tab® Dimension group, choose the Smart Dimension command. Click on one of the arcs and type 5 in the Dimension Value Edit dialog box.



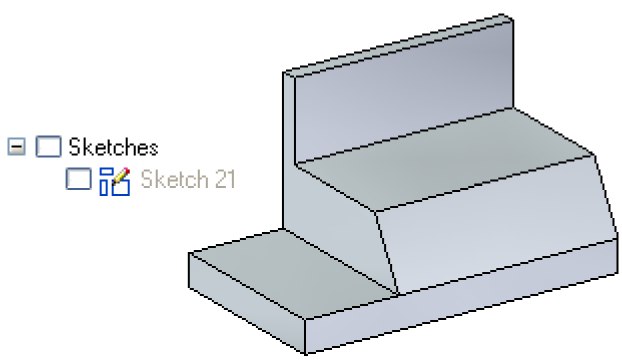
- On the Sketching tab® Dimension group, choose the Distance Between command. Select the center of each arc and type 30 in the edit box.



Turn off sketch

- If the sketch plane was manually locked, in PathFinder, right-click the on the sketch. On the short-cut menu, choose Lock Sketch Plane.

- Click the check box to turn off the sketch display.



- Activity is complete. Exit the file and do not save.

Summary

In this activity you learned how to create a sketch on a part face. You learned how to apply relationships and dimensions to a sketch.

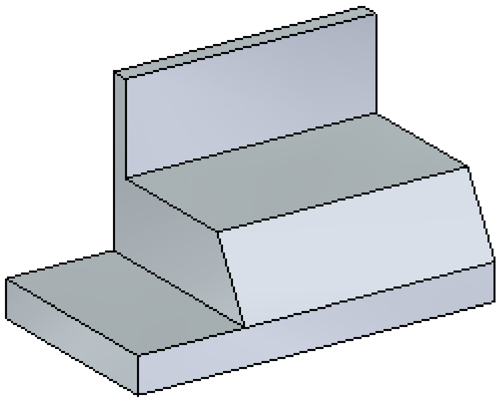
Part 2

Activity covers drawing a sketch on a reference plane, including edges from part faces, sketch associativity to part model edges and the sketch view command.

Activity: Sketching (Part 2)

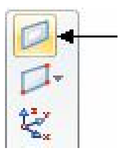
Open a part file

- Start Solid Edge.
- Click the  Application button® Open.
- In the Open File dialog box, set the Look in: field to the folder where the training files reside.
- Click *sketch_B* and then click Open.

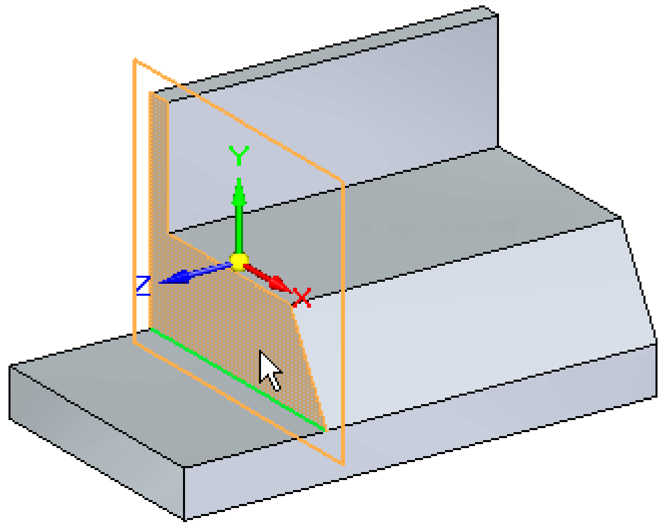


Create a sketch plane

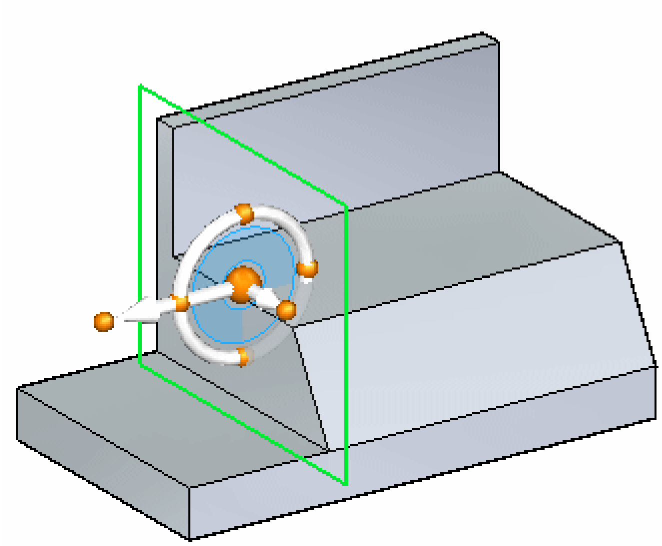
- On the Home tab® Planes group, choose the Coincident Plane command.



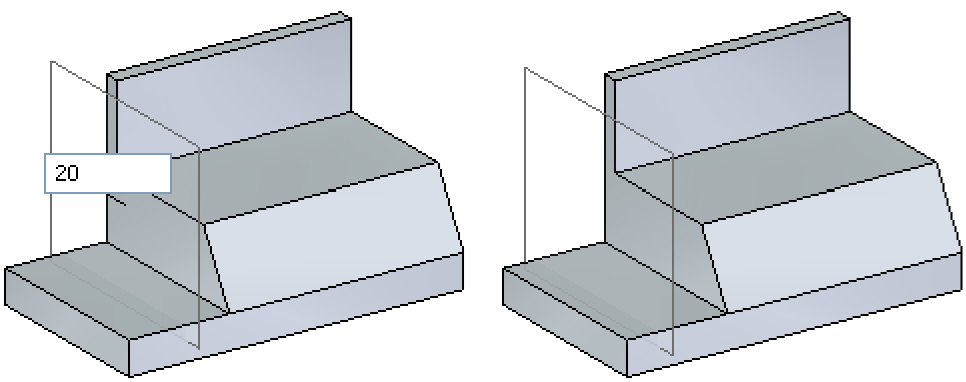
- Select the part face shown.



- Click the primary axis on the graphic move handle.



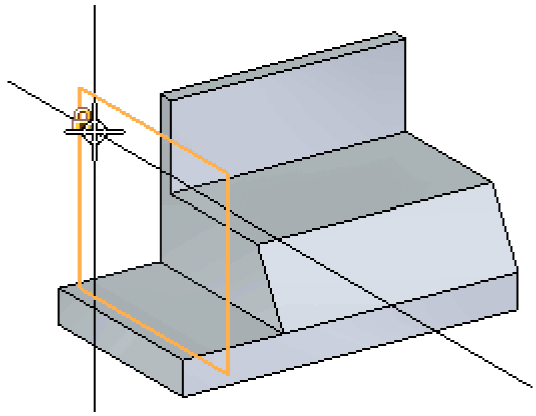
- In the distance edit box, type 20.



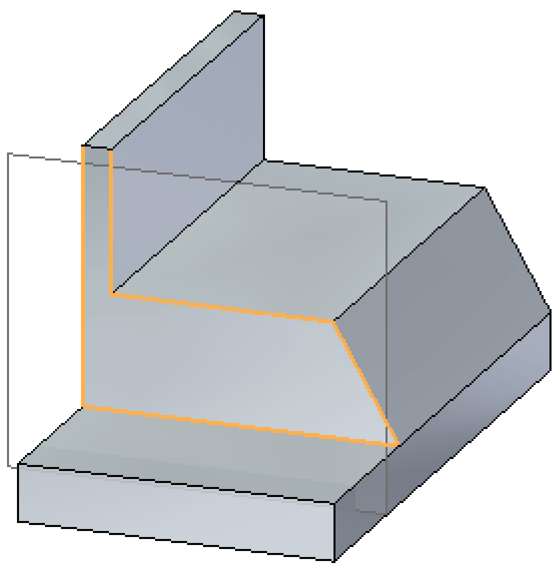
Start the sketching process

- You will use edges of the part in the sketch. On the Sketching tab® Draw group, choose the Project to Sketch command. The command requires a locked plane.

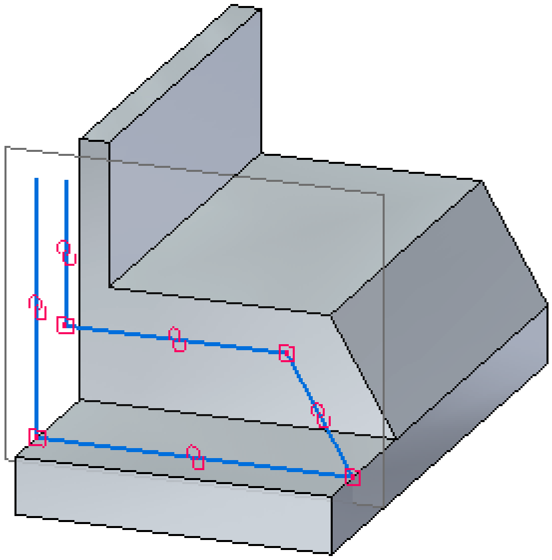
- Lock the sketch plane. Pause over the sketch plane created earlier and then click the lock.



- Select the edges shown.

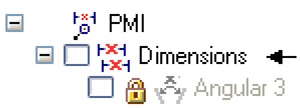


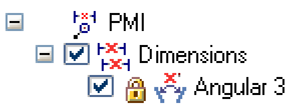
- Notice how these edges project to the locked sketch plane.

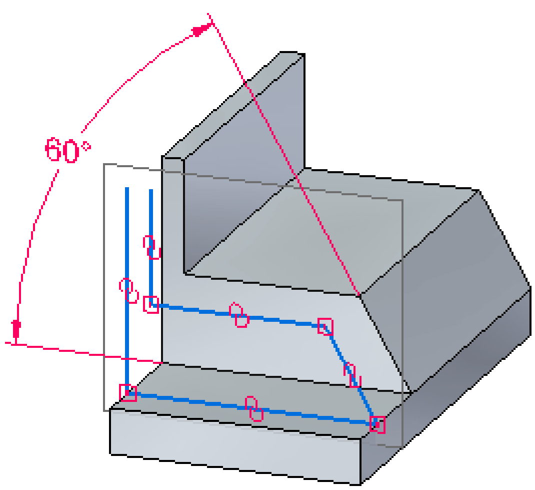


Observe sketch associativity

- Display PMI dimensions. In PathFinder, click the Dimensions check box.

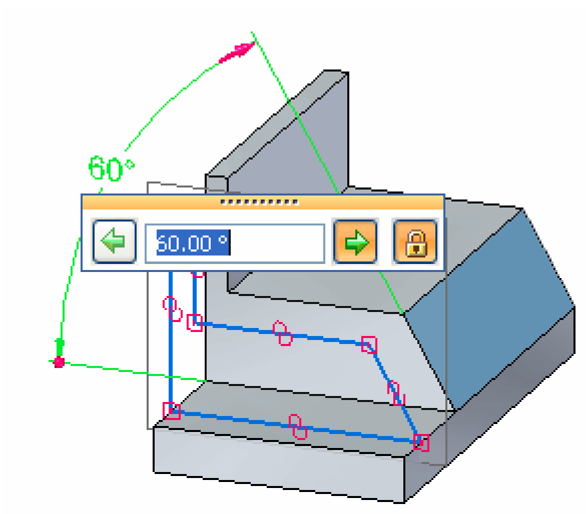






- Click the 60° value on the dimension.

- Change the dimension (any value between 45° and 75°) and notice how the edge that was projected to the sketch plane follows the angle of the face. Make sure the direction arrow on the dimension matches the illustration. You can change the direction by clicking the arrow buttons in the dynamic edit box.

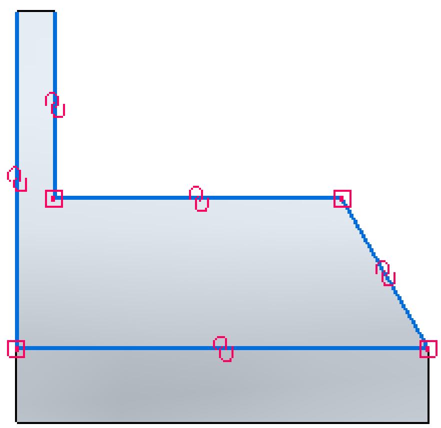


- Set dimension to 60° and turn off the PMI dimension display.

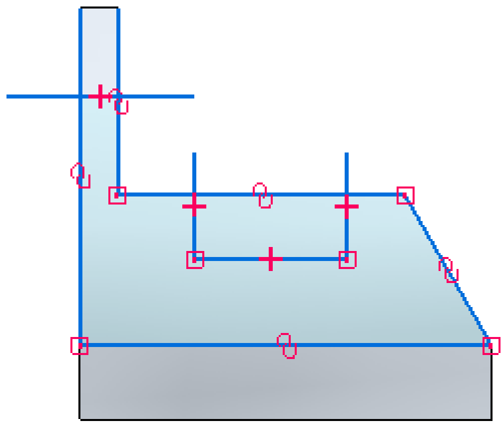
Draw sketch geometry

Add and modify sketch geometry.

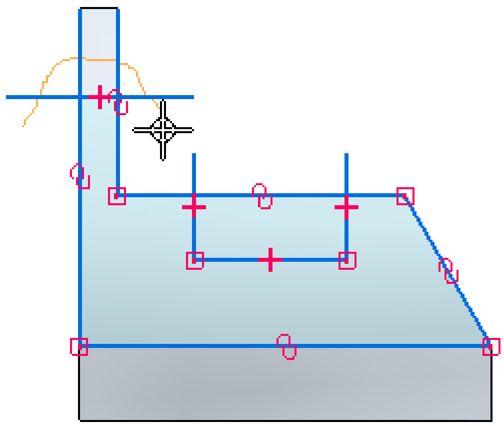
- Orient the sketch plane normal to the view. On the View tab® Views group, choose the Sketch View command.

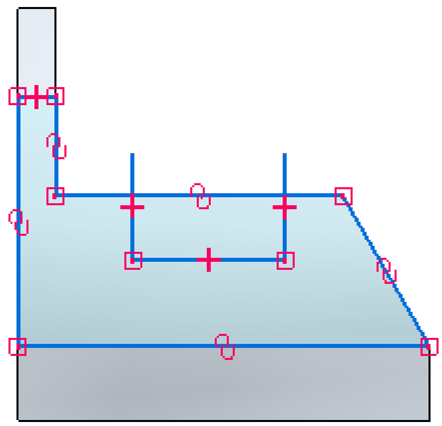


- Draw the sketch geometry as shown. Segment lengths and location are not important.

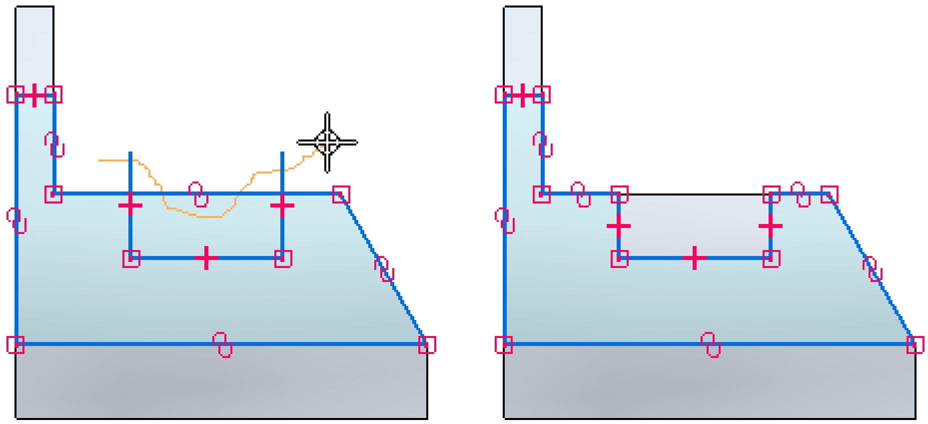


- Trim line segments. On the Sketching tab® Draw group, choose the Trim command.
- Click and drag the cursor over the line segments shown.



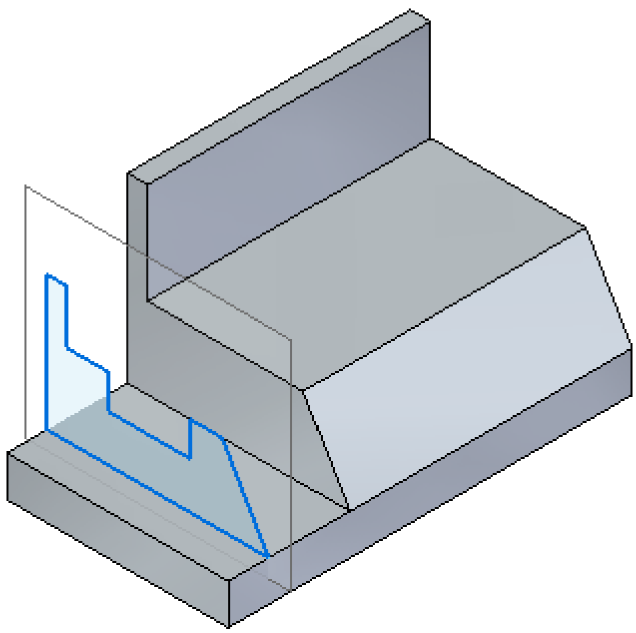


- Click and drag the cursor over the three line segments shown.



Edit display

- Turn off the Relationship Handles display.
- Switch to an isometric view. Type Ctrl+I.



- Close the file and do not save.

Summary

In this activity you learned how to draw a sketch on a reference plane and how to include edges from part faces. You observed sketch associativity to part model edges and used the Sketch View command.

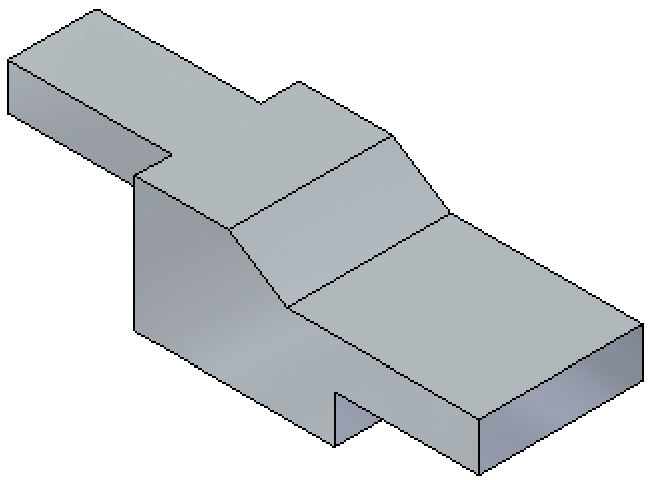
Part 3

Activity covers drawing a sketch on a face, copying the sketch to another face, rotating and moving the copied sketch.

Activity: Sketching (Part 3)

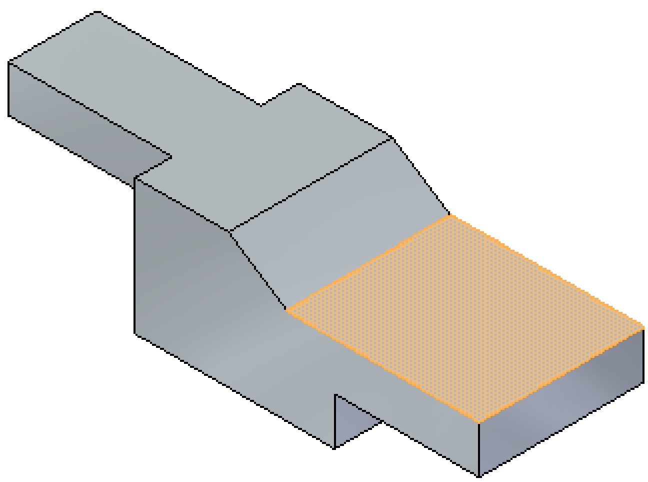
Open a part file

- Start Solid Edge.
- Click the  Application button® Open.
- In the Open File dialog box, set the Look in: field to the folder where the training files reside.
- Click *sketch_C* and then click Open.

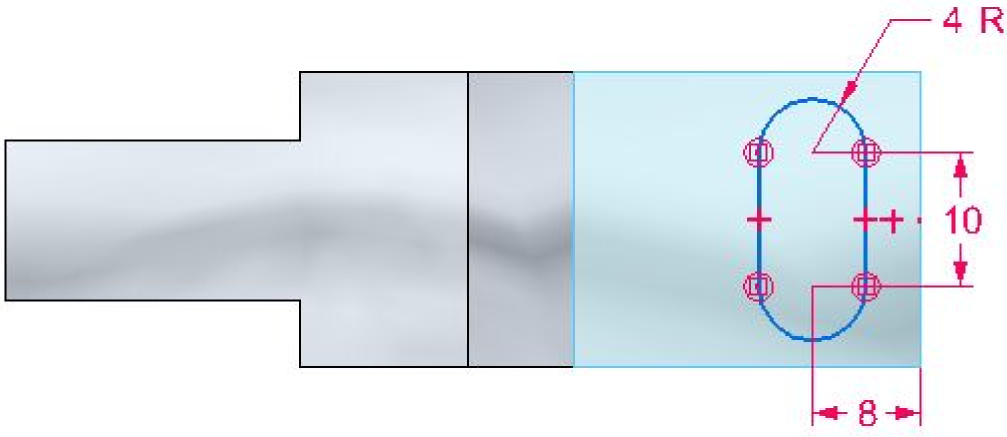


Draw a sketch on a face

- Lock to the face shown.



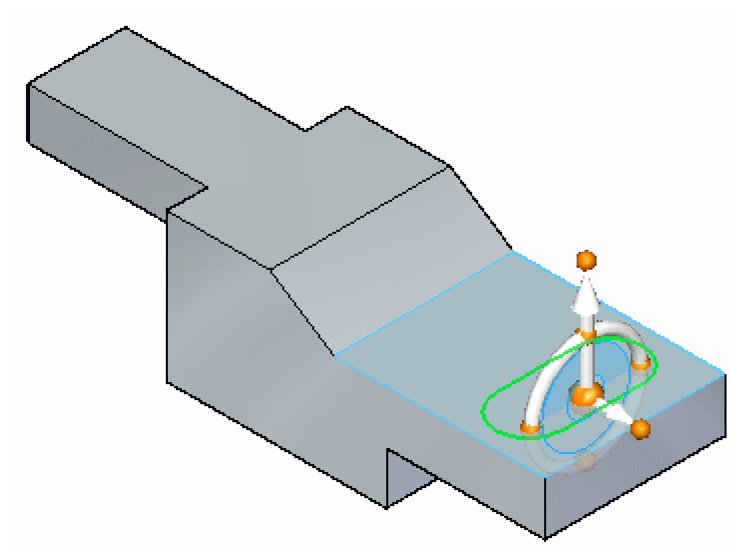
- Draw the following sketch.



- Delete the sketch dimensions. The dimensions were placed to only define the size.
- Change the view to an isometric view. Press Ctrl+I.

Copy the sketch

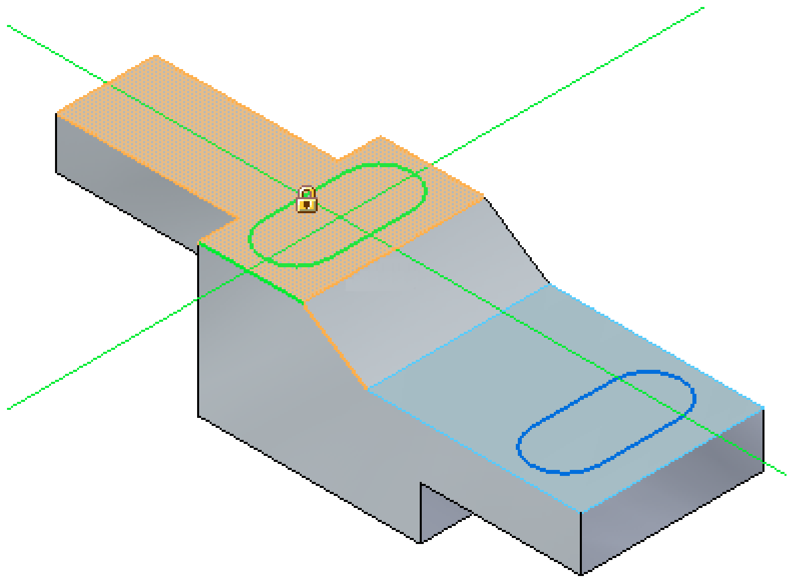
- Unlock the sketch plane.
- Select the sketch in PathFinder.



- Press Ctrl+C to copy the selected sketch. The sketch is added to the clipboard.

Paste the sketch

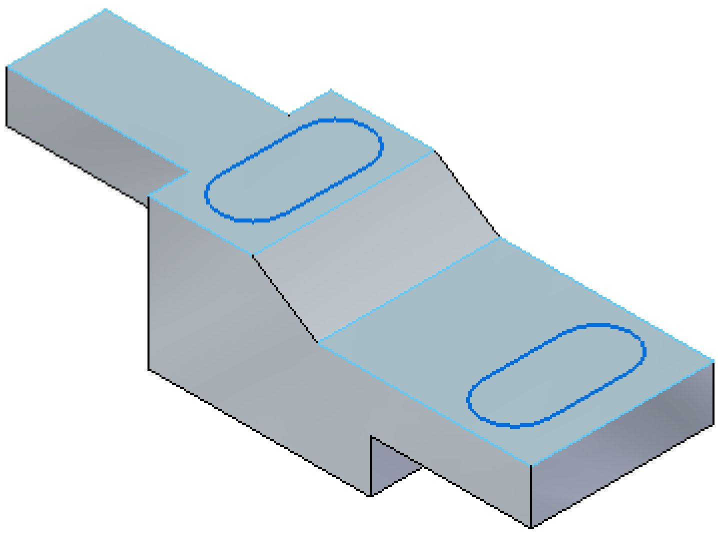
- Press Ctrl+V. The copied sketch attaches to the cursor. Pause the cursor over the face and then click to place the sketch as shown. You will position the sketch next.



Note

You can press the N or B keys to control the copied sketch orientation. However, in this activity, use the rotate command to position the sketch.

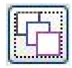
- Press the Esc key to end the paste operation.

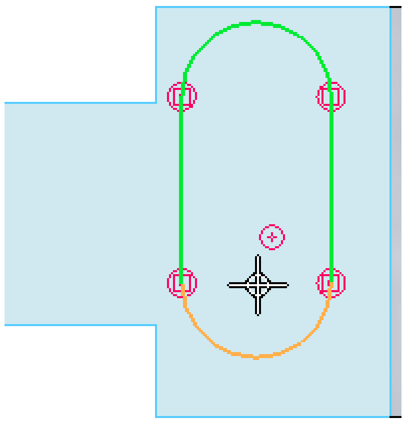


Rotate the copied sketch

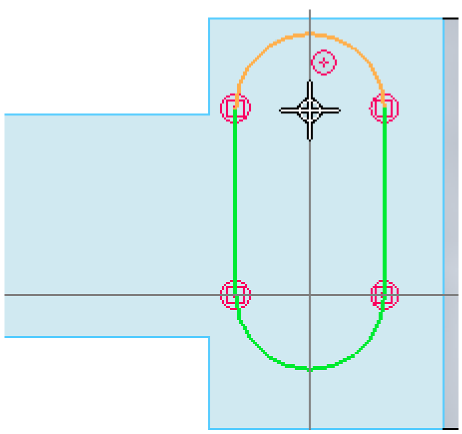
- In PathFinder, right-click the copied sketch and choose Lock Sketch Plane.

- Choose the Sketch View command.
- On the Move command drop list, choose the Rotate command.

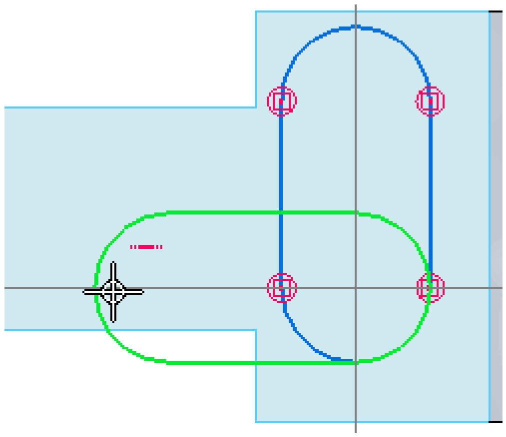
- On command bar, make sure the copy option  is not on.
- While holding down the Ctrl key, click the two lines and two arcs. The elements turn green as they are selected.
- Select the arc center as center of rotation.

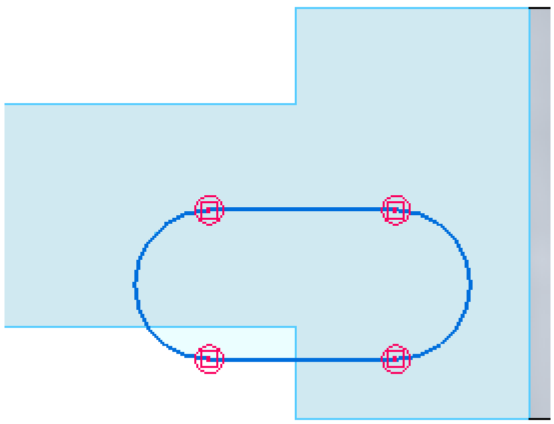


- Select the other arc center as the start point for rotation.



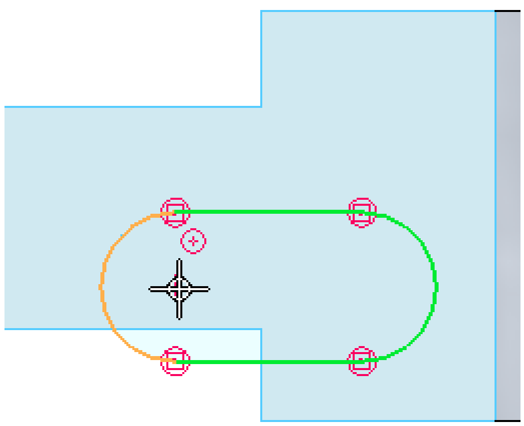
- Click when the horizontal indicator appears. This rotates the sketch 90°.



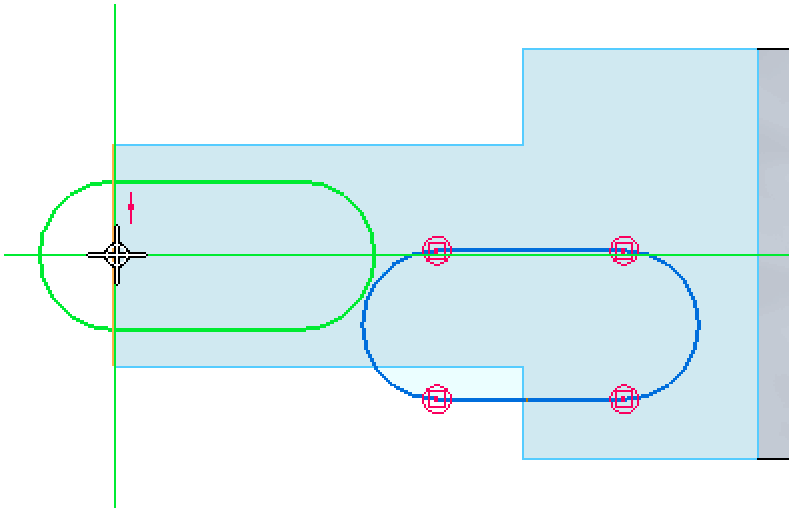


Move the copied sketch

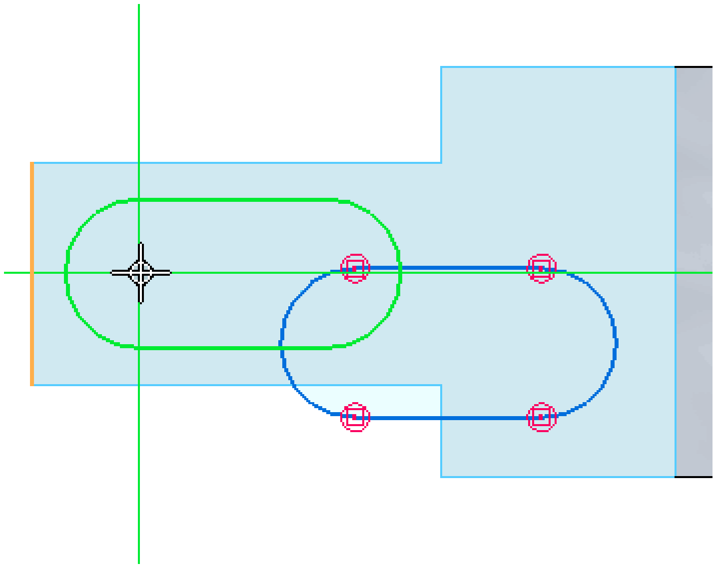
- Choose the Move command.
- Select the four elements again. For the move from point, select the center of an arc.

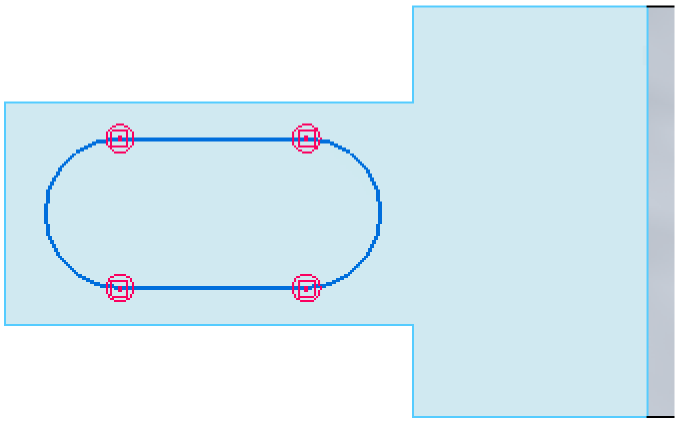


- For the to point, move the cursor over the midpoint of the top edge. The sketch will be centered to this point. Do not click.

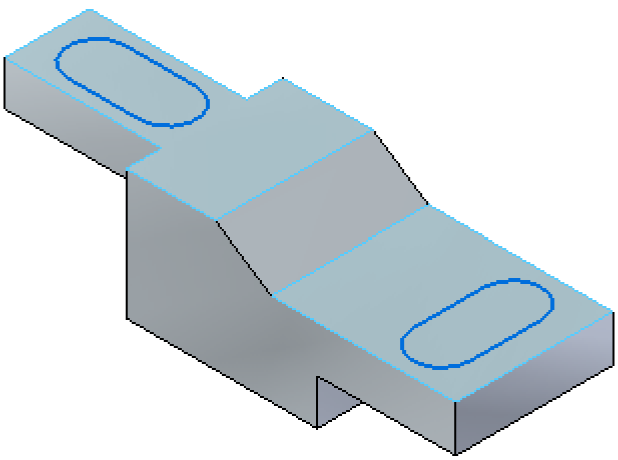


- While maintaining midpoint alignment display, move the cursor down to the location shown and click.





- Press Ctrl+I.



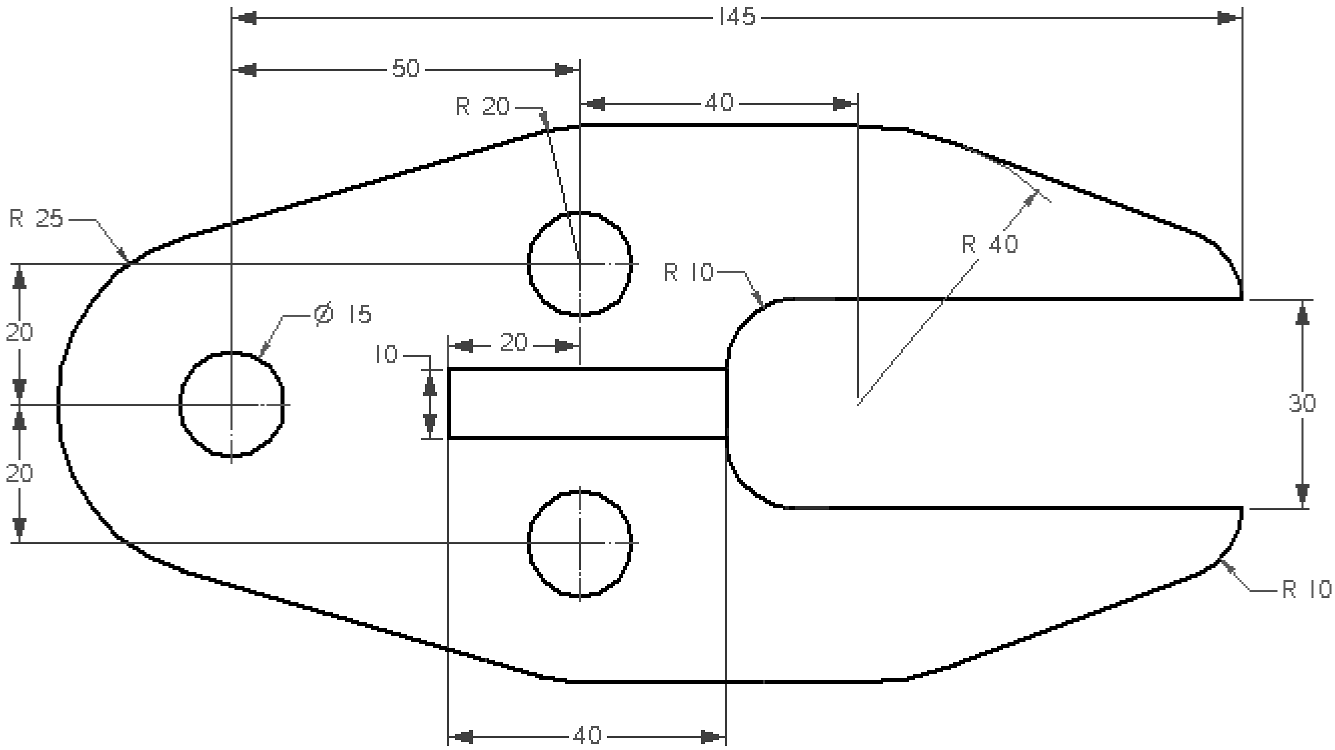
- Activity complete. Close the file and do not save.

Summary

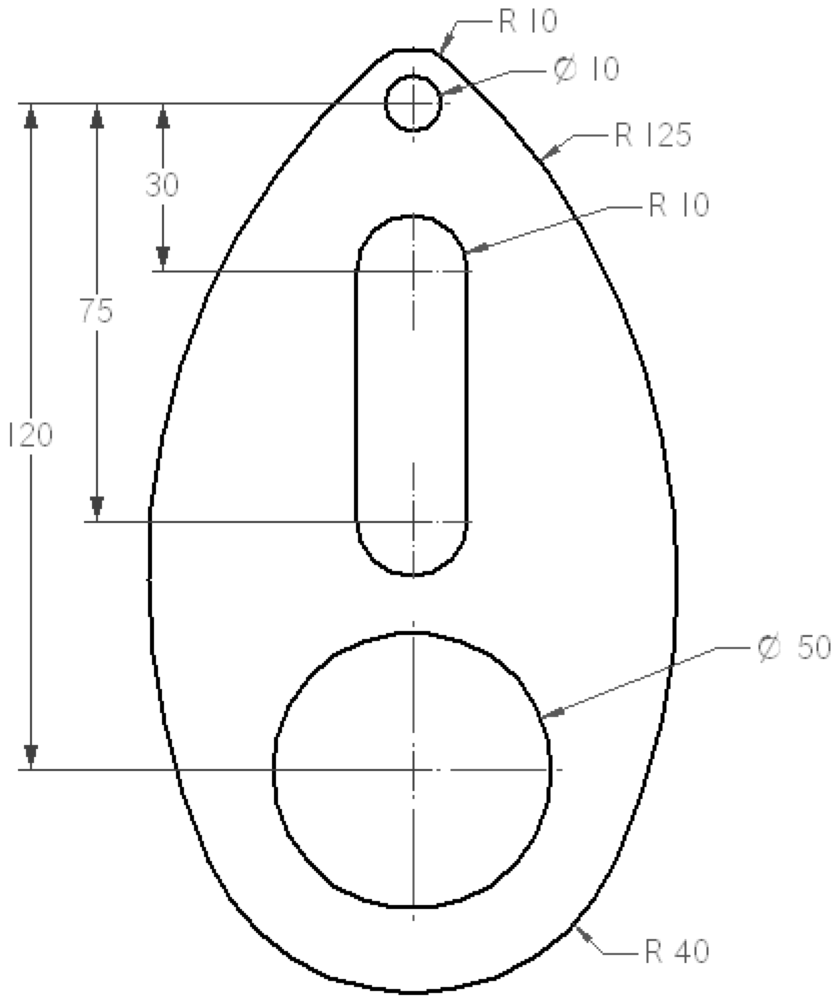
In this activity you drew a sketch on face and learned how to copy the sketch to another face. You also learned how to rotate and move a sketch.

Sketch projects

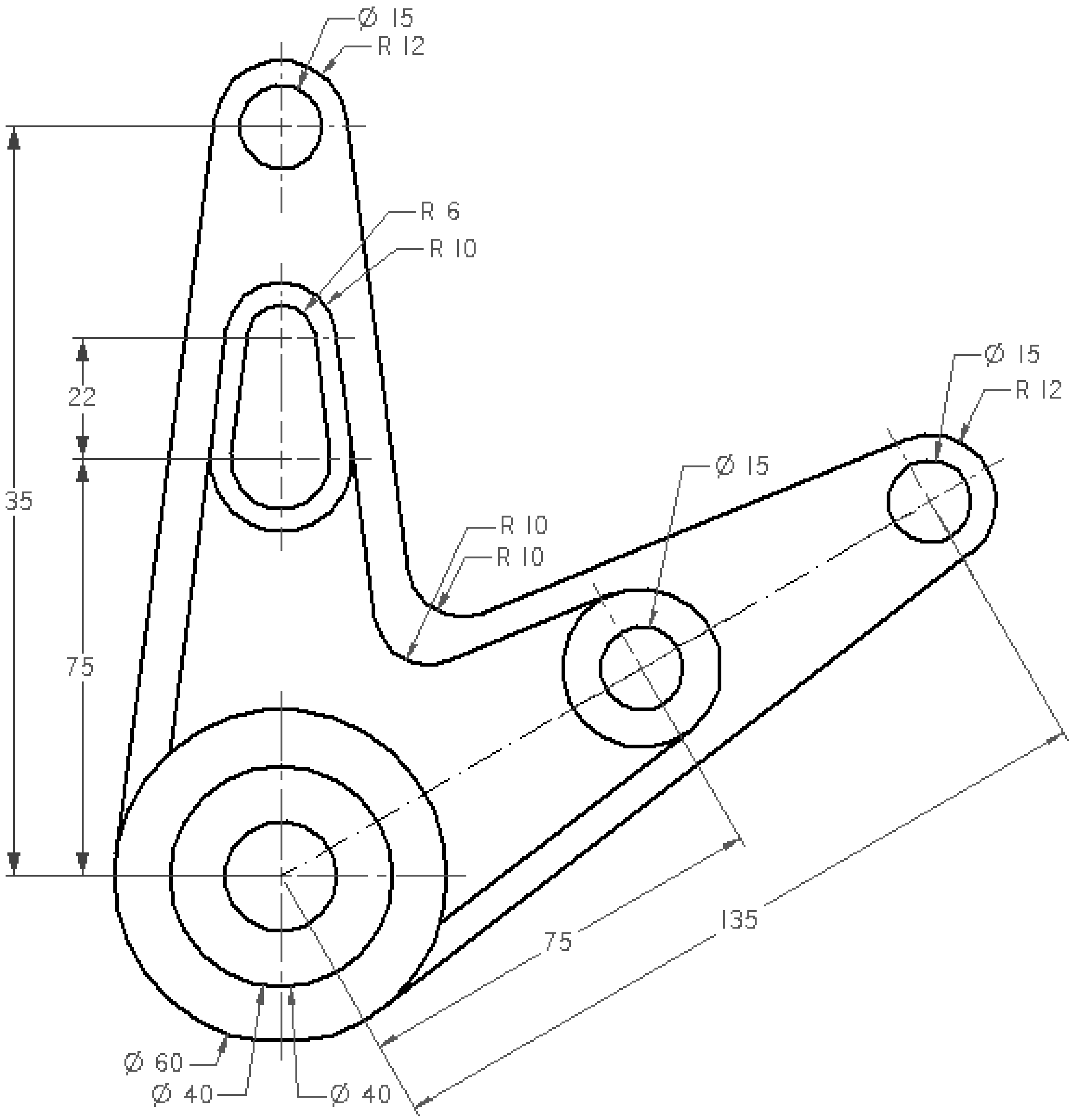
Drawing A



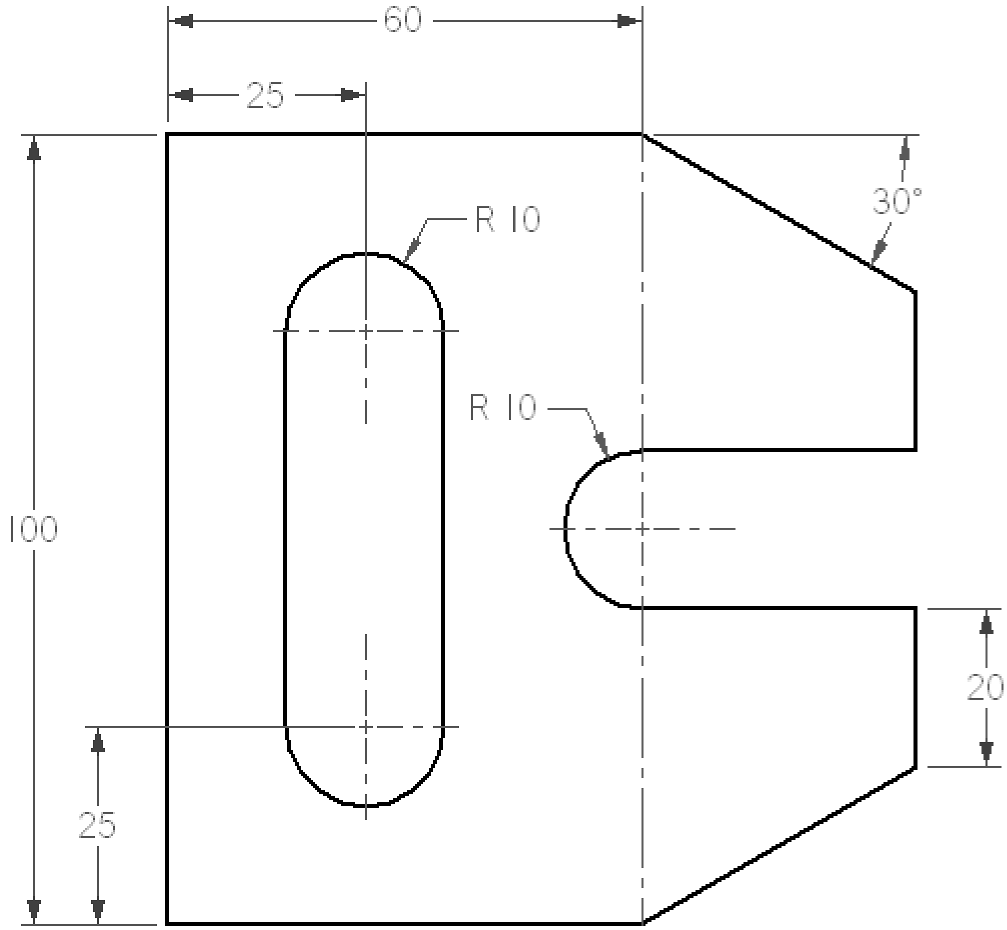
Drawing B



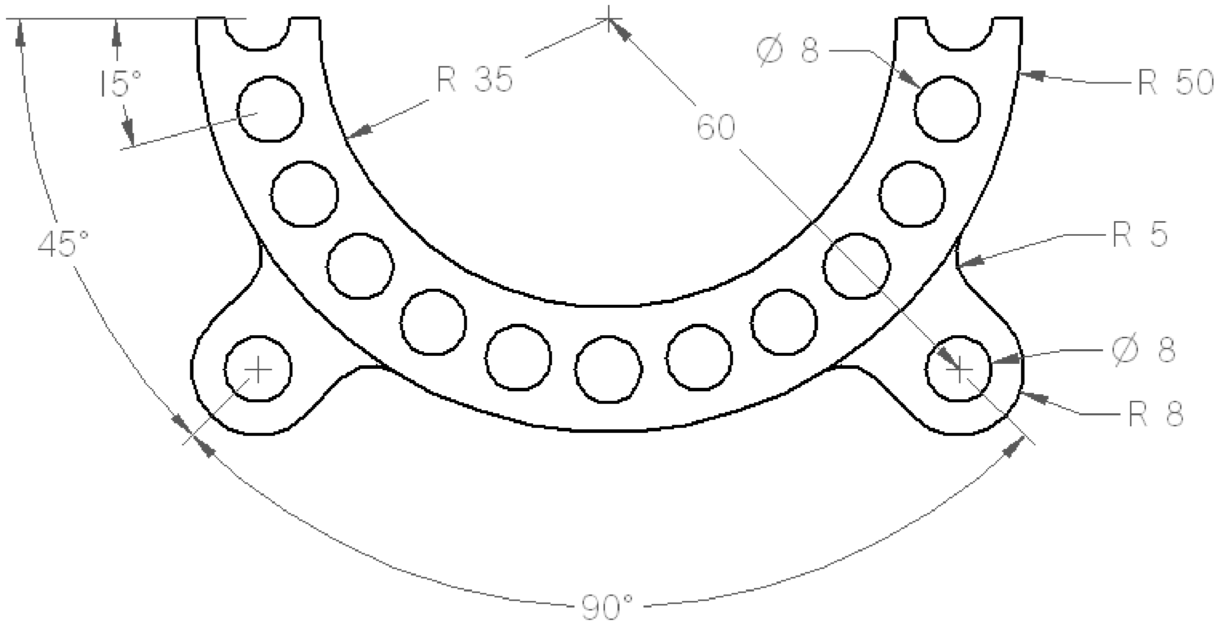
Drawing C



Drawing D



Drawing E



Course review

Answer the following questions:

1. What is the first step in creating a sketch?
    - Choose a command from the Draw group.
    - Select a sketch plane.
    - Switch to a sketch view.
    - Select the base coordinate system.
2. How do you lock a sketch plane?
    - Move cursor over a planar face or reference plane and select the lock icon.
    - Move cursor over a planar face or reference plane and press the F3 key.
    - Right-click on an existing sketch in PathFinder and choose Lock Sketch Plane.
    - None of the above.
    - All of the above.
3. How do you unlock a sketch plane?
    - Click the Lock icon in upper right corner of screen.
    - Press the F3 key.
    - Right-click on a locked sketch in PathFinder and choose Lock Sketch Plane.
    - None of the above.
    - All of the above.
4. What controls the horizontal/vertical direction of a sketch plane?
5. How do you know what sketch relationships exist?
6. How do you use the maintain relationships command?
7. What is a region?
8. Can an open sketch create a region?
9. What is the Used Sketches collector used for?
10. How do you reposition the sketch plane origin?
11. Explain what the Enable Regions command is used for.
12. Explain what the Merge with Coplanar Sketches command is used for.

Answers

1. What is the first step in creating a sketch?

   Choose a command from the Draw group.

2. How do you lock a sketch plane?

   All of the above.

3. How do you unlock a sketch plane?

   All of the above.

4. What controls the horizontal/vertical direction of a sketch plane?

   The green highlighted edge of a face defines the horizontal direction of a sketch plane. Use the N key to cycle through the linear edges of a face to define the horizontal direction. If a face has no linear edges, then the horizontal/vertical directions align with the base reference planes.

5. How do you know what sketch relationships exist?

   On the Sketching tab® Relate group, choose the Relationships Handles command to turn on the display all sketch relationships.

6. How do you use the maintain relationships command?

   On the Sketching tab® Relate group, choose the Maintain Relationships. When this is turned on, all relationships applied are remembered. If this is off, all relationships applied are not remembered.

7. What is a region?

   When sketch elements create a closed area, a region forms. The sketch elements do not have to be endpoint connected.

8. Can an open sketch create a region?

   Yes. If an open sketch is coplanar with a face and the open sketch touches or crosses a part edge on the face, then a region forms.

9. What is the Used Sketches collector used for?

   When a sketch is used to create a feature, all sketch elements that create feature edges are consumed and moved to the Used Sketch collector. You can turn off the sketch option *Migrate Geometry and Dimensions* to not consume the sketch during feature creation.

10. How do you reposition the sketch plane origin?

    On the Sketching tab® Draw group, on the Grid drop list, choose the Reposition Origin command. The 2D steering wheel appears and you can redefine the horizontal/vertical direction with the torus or by typing an angular value. Choose the Zero Origin command to reset the origin direction.

11. Explain what the Enable Regions command is used for.

When the Enable Regions options is on, a region highlights when the cursor moves over it. Select the region to create an extrusion or revolved extrusion. When the option is off, you cannot select a region.

12. Explain what the Merge with Coplanar Sketches command is used for.

    When this option is on, if you create a new sketch and there is an existing coplanar sketch plane, the new sketch merges with the existing sketch. When the option is off, the new sketch does not merge with the existing coplanar sketch.

Course summary

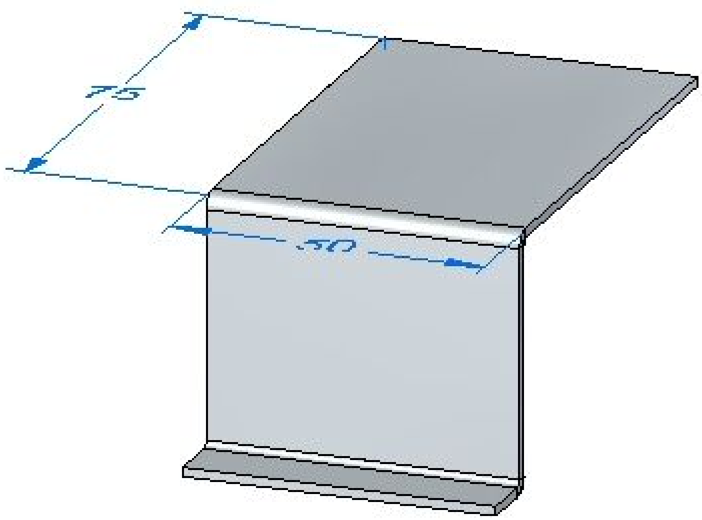
- Sketches that form closed areas are called regions.
- Use regions to define the cross section of a synchronous feature.
- Sketches do not drive the feature.
- Sketches move to the Used Sketch collector when used to create a feature.
- Lock to a sketch plane to create a sketch.
- Sketches relationships do not migrate to the feature. However Live Rules can detect if feature faces are coplanar. parallel, perpendicular, etc.

Lesson

4 Base Features

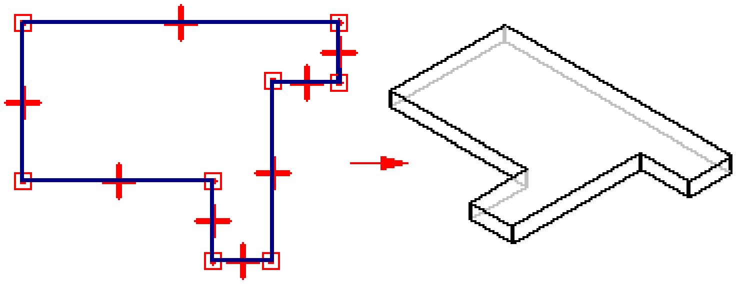
Base features in sheet metal

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.

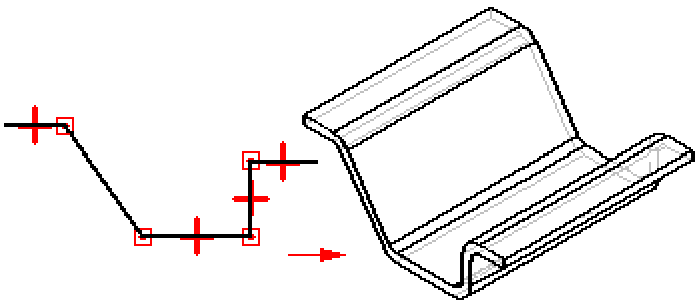


Construct the base feature

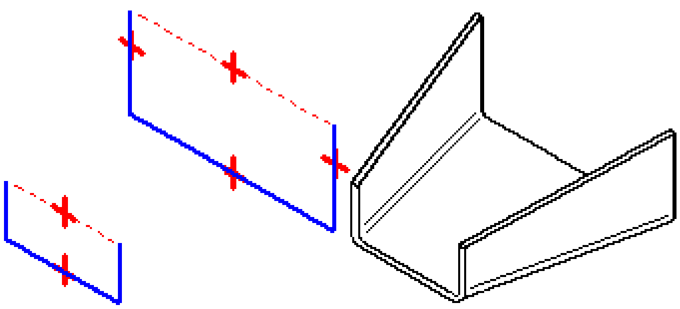
You can construct a base feature with the Tab, Contour Flange, and Lofted Flange commands. The Tab command constructs a flat feature of any shape using a closed profile.



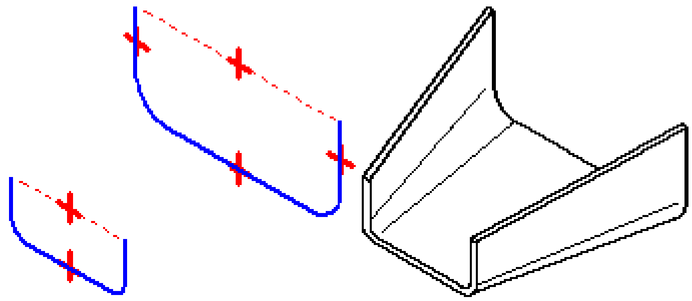
The Contour Flange command constructs a feature comprised of one or more bends and flats using an open profile.



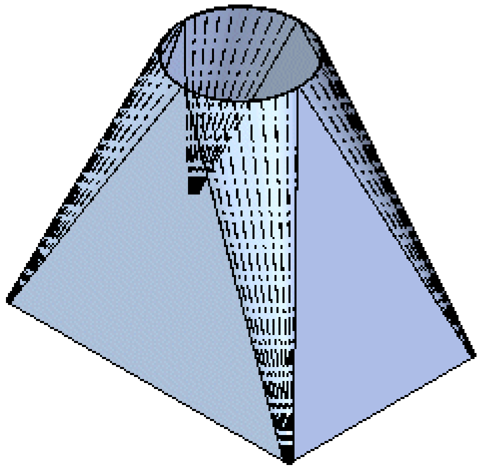
The Lofted Flange command quickly constructs a flange using two open profiles on parallel reference planes. Like the Contour Flange command, the Lofted Flange command automatically adds bends using the bend radius property. You do not have to draw an arc at each bend location.



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



The Bending Method tab on the Lofted Flange dialog box creates incremental bends for all bends in the flange.

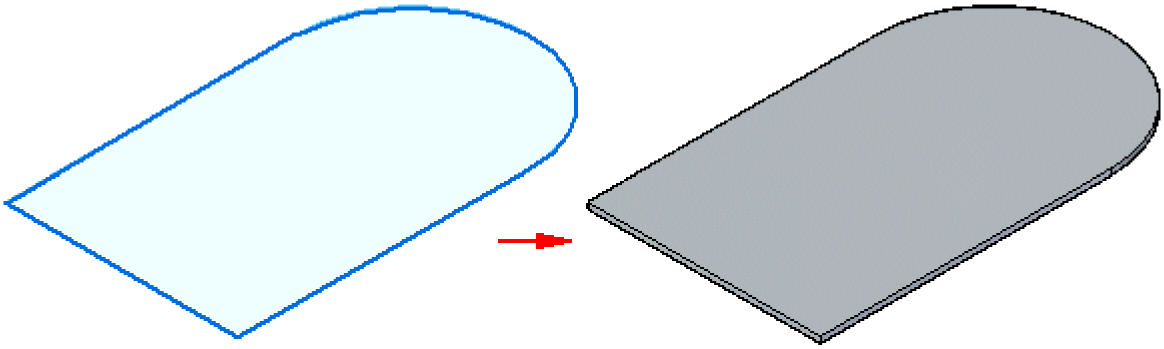


You can set the number of bends. For the lofted flange to flatten, the arc angle must match between the two cross sections.

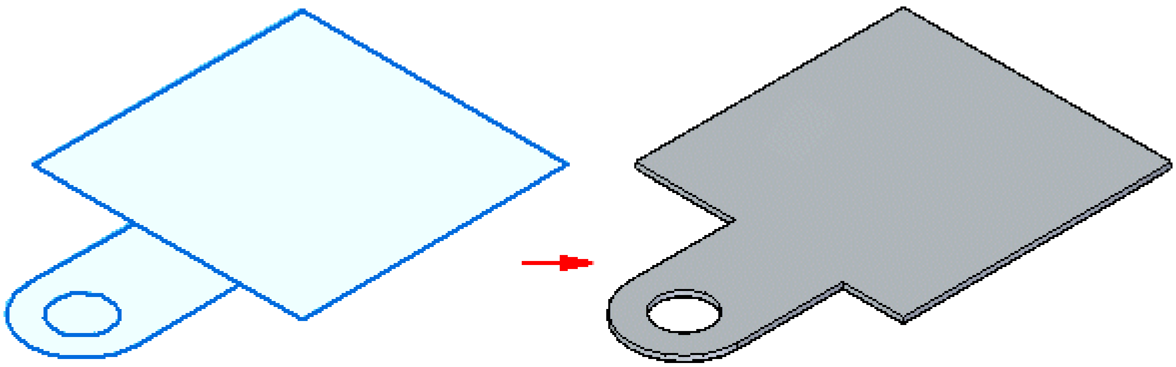
Tab command

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,



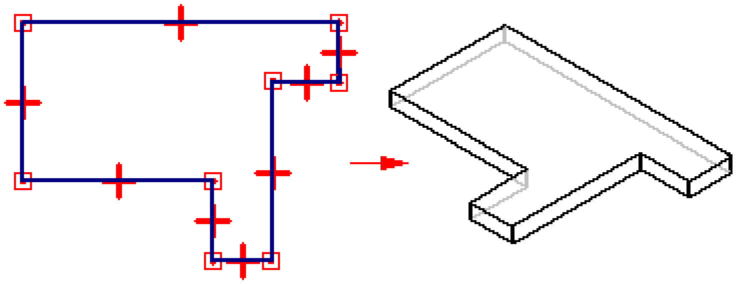
or with multiple sketch regions.



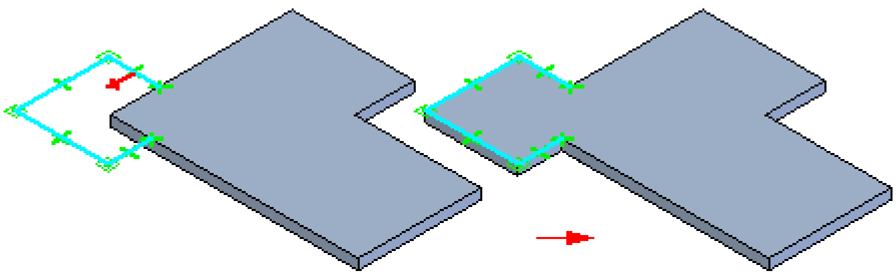
Creating tabs in the ordered environment

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.

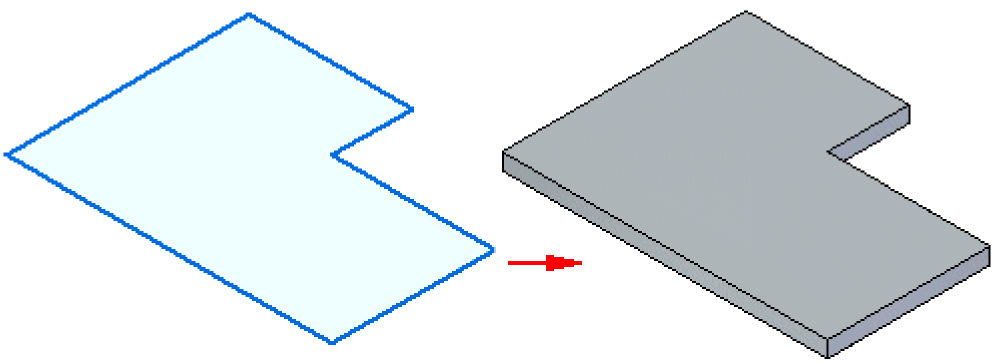


For subsequent features in the ordered environment, the profile can be open or closed. When using an open profile, you must define the side of the profile to which you want to add material.

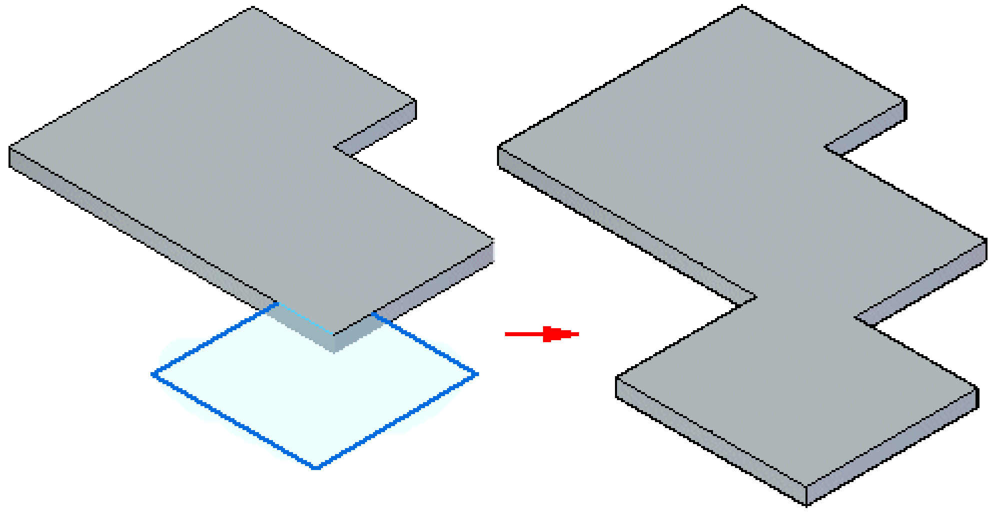


Creating tabs in the synchronous environment

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.

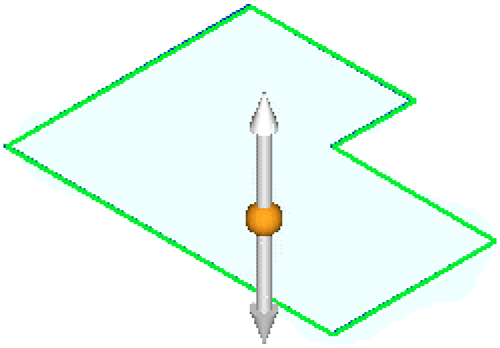
   

5. Finish the feature.

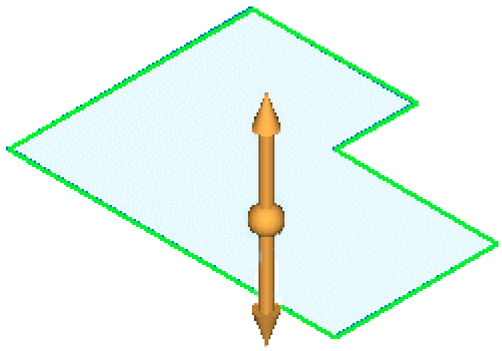
Construct a tab as a base feature in the synchronous environment

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

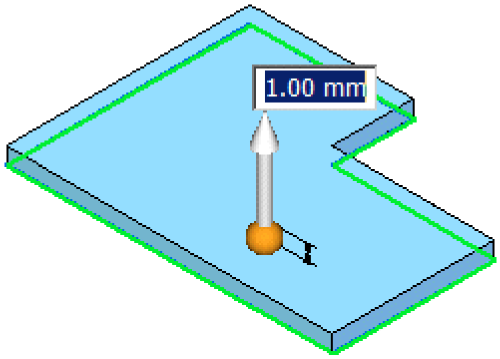
   The extrude handle is displayed.



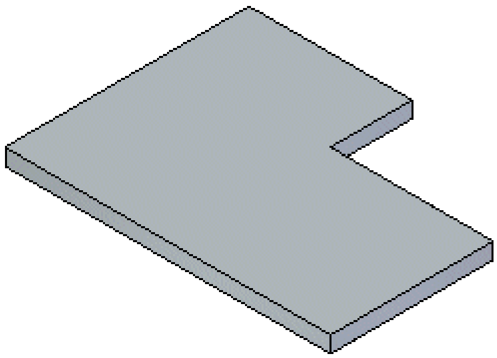
2. Click the Extrude handle.



3. Type a thickness value for the part.



4. Right-click to create the tab.



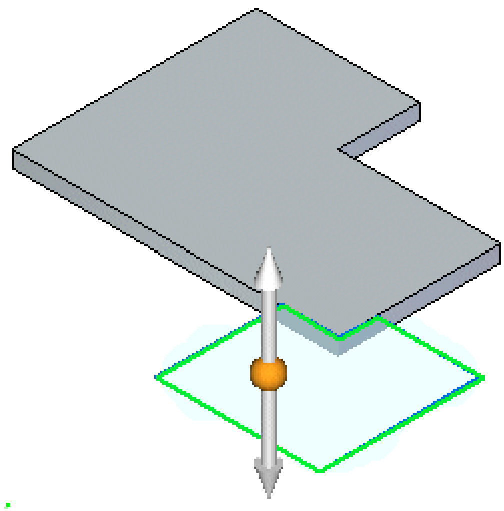
Tip

- 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.

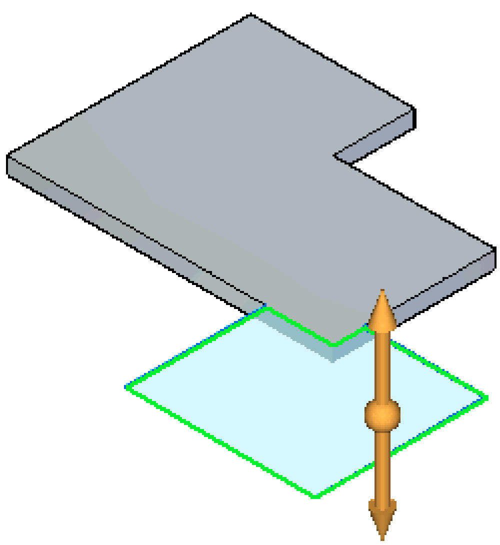
Add a tab to an existing sheet metal part in the synchronous environment

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

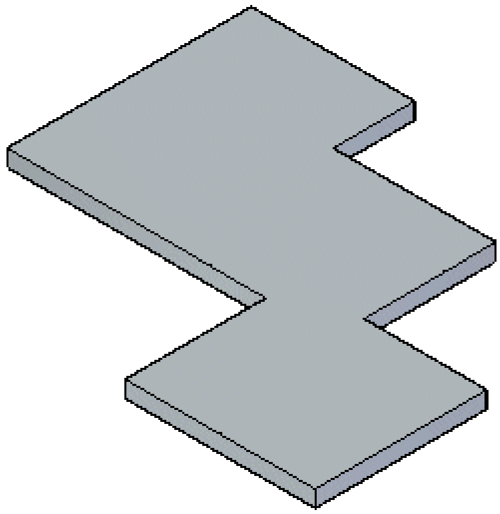
   The extrude handle is displayed.



2. Click the Extrude handle.

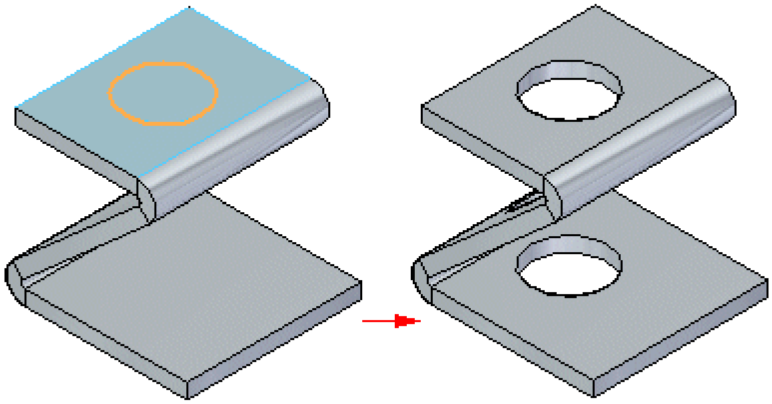


The tab is automatically added.

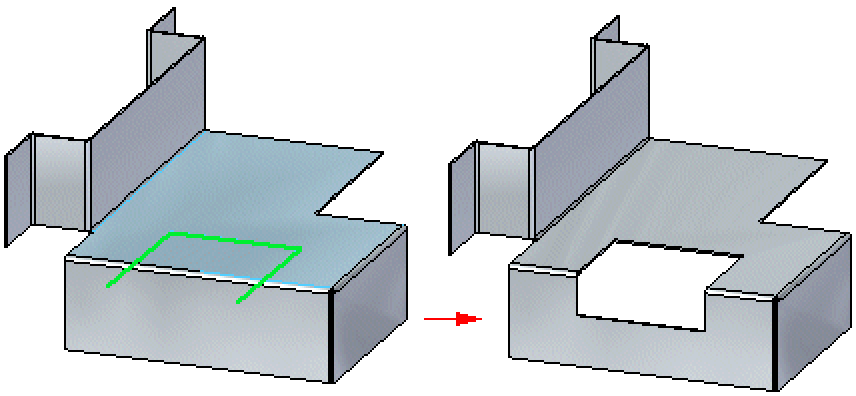


Cut command

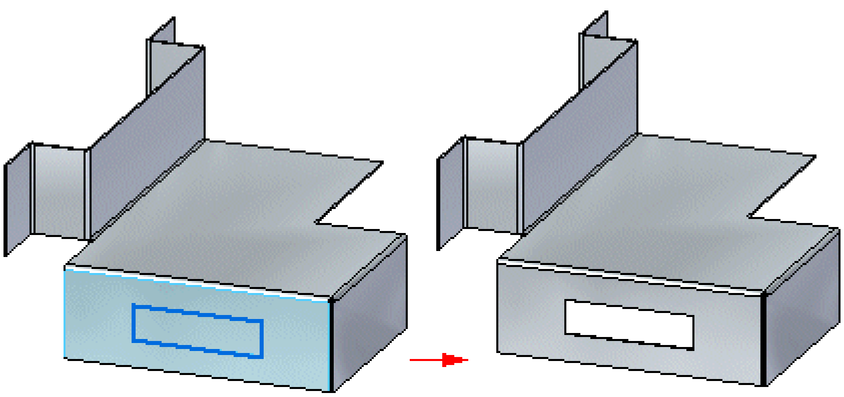
Creates a cut through a defined portion of a part.



You can create a sheet metal cutout with an open profile



or a closed profile.

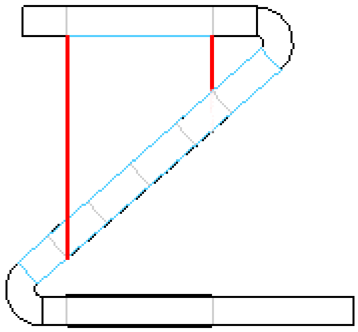


Face Normal cut types

Face Normal cut types include:

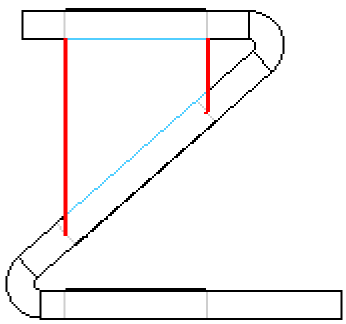
Thickness cut

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.

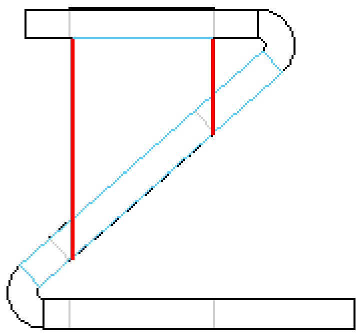
Mid–plane cut



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

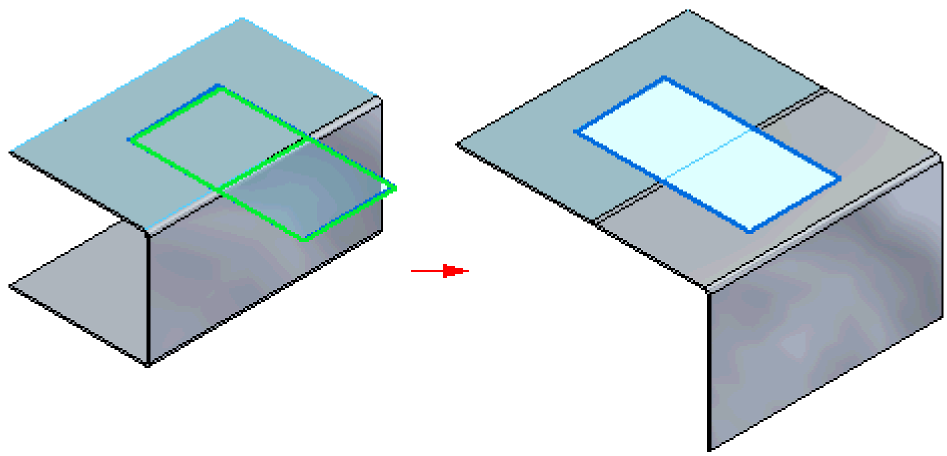
Nearest Face cut

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

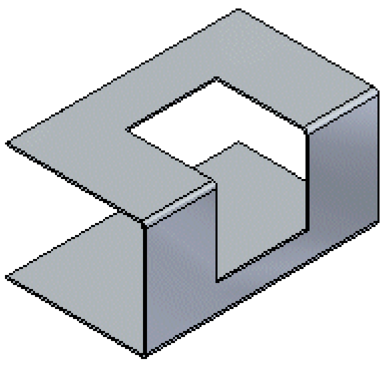


Cuts across bends

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



and then rebends when the cut is complete.

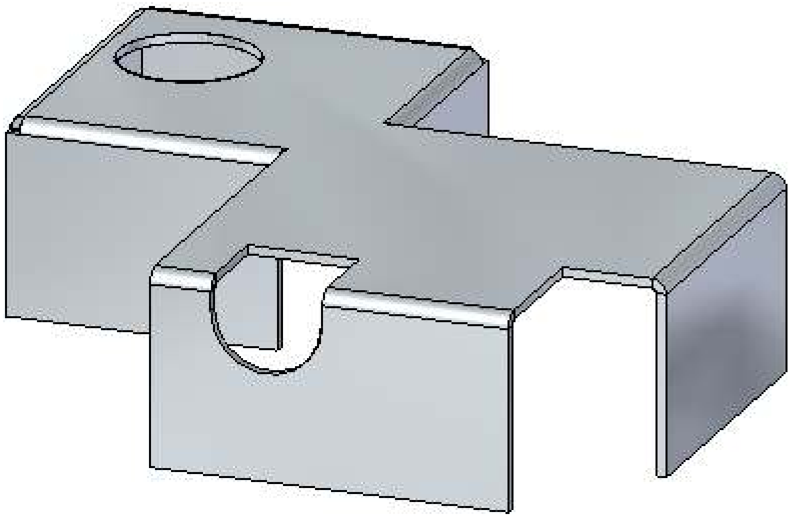


Activity: Using regions to create tabs and cuts

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.



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*.

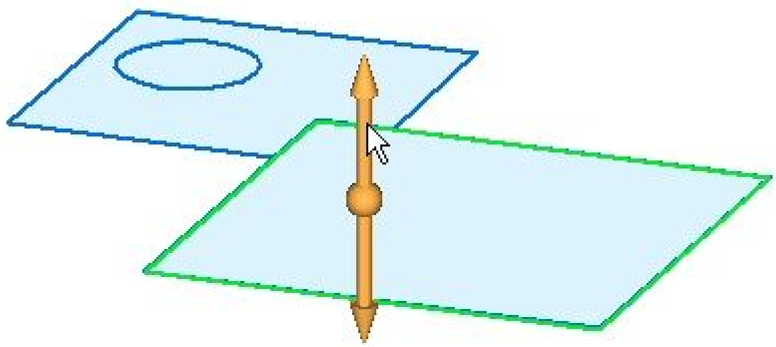
  Note

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

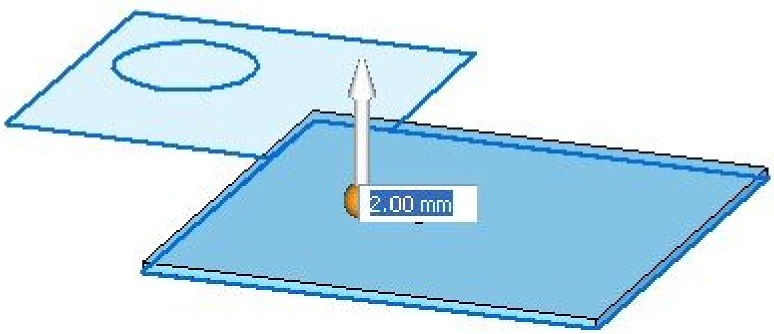
- Proceed to the next step.

Use the sketch to create the base feature

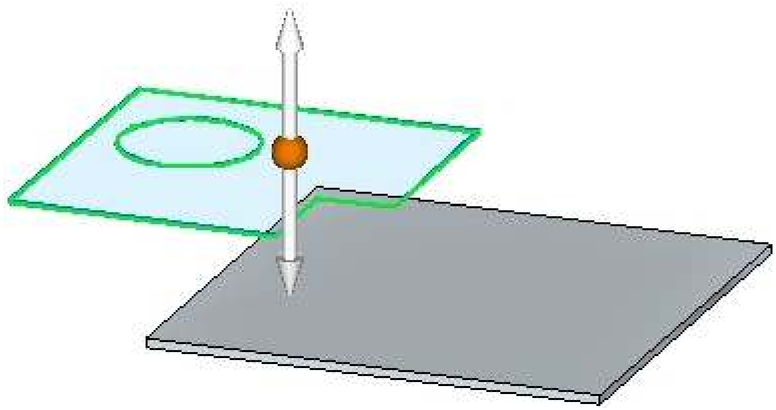
- Use the region shown to create the base feature from the sketch geometry. Select the handle pointing up.



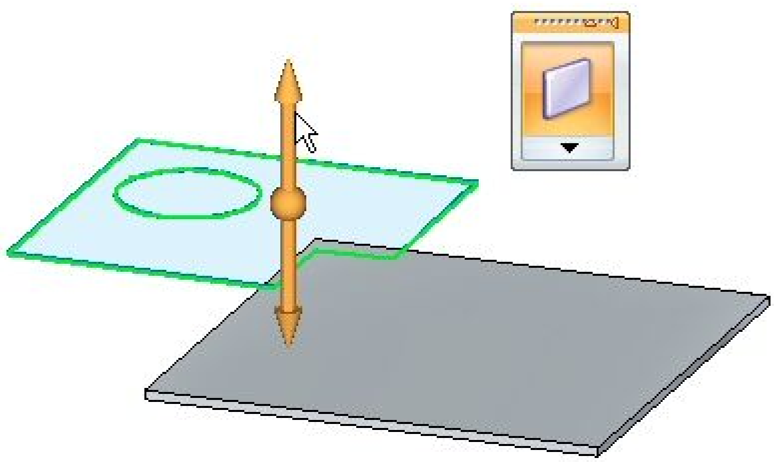
- Click to place the base feature, a tab, above the sketch as shown. Press Enter to accept the material thickness of 2.00 mm.



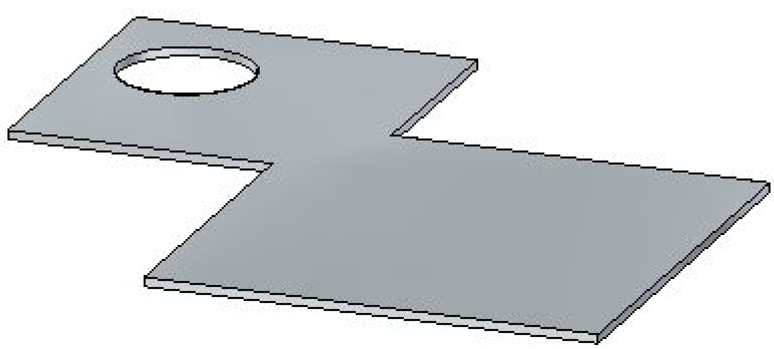
- To place the next tab, select the region shown.



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



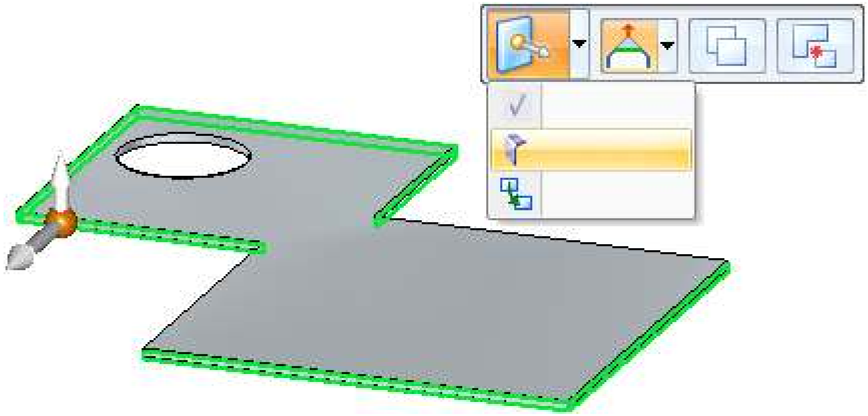
The base feature appears as shown.



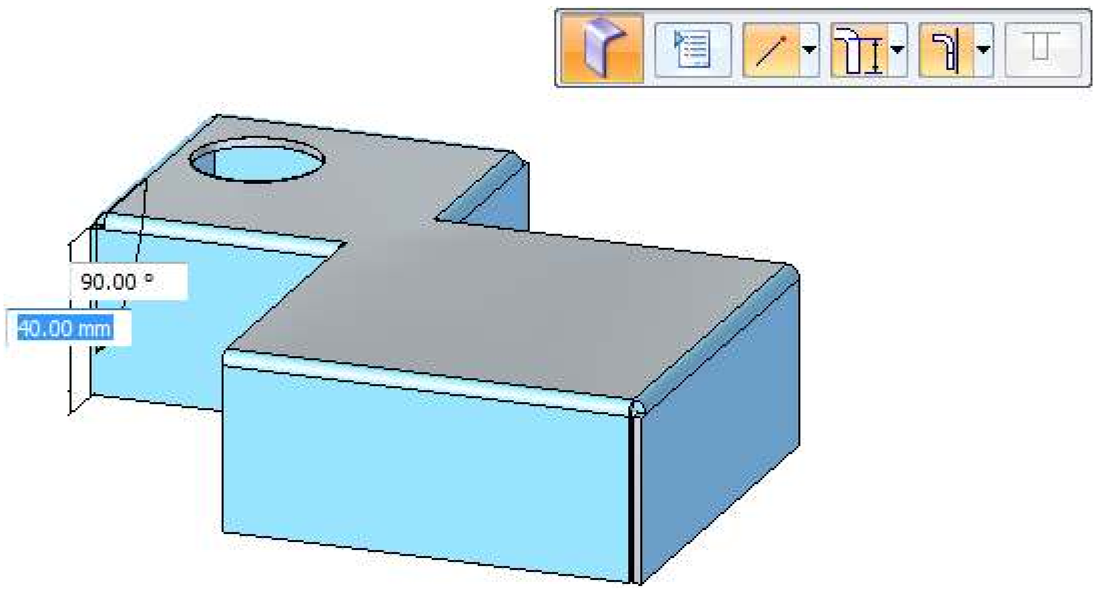
- Proceed to the next step.

Create the flanges

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



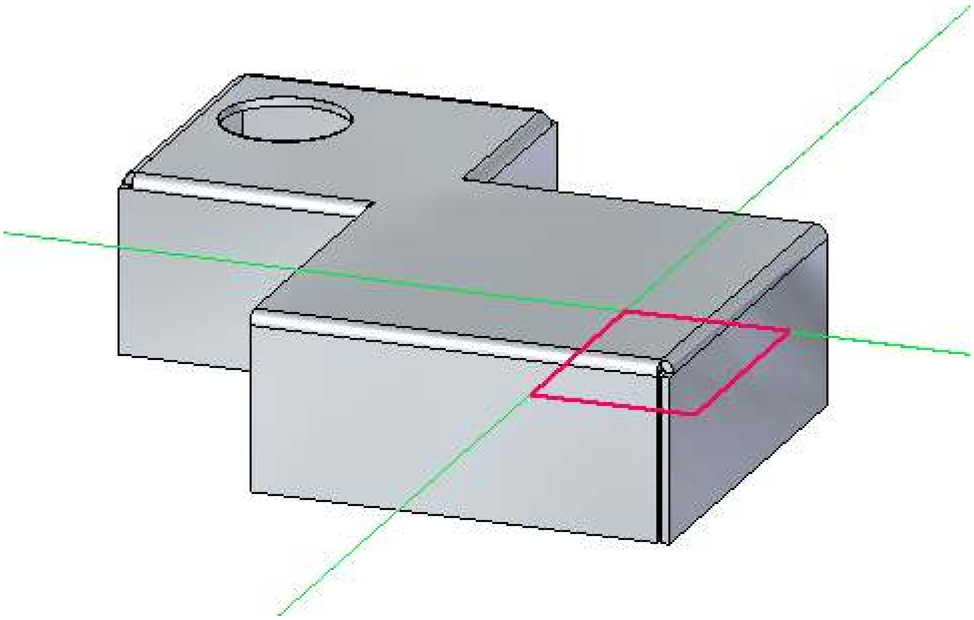
- Pull the flange down a distance of 40.00 mm.



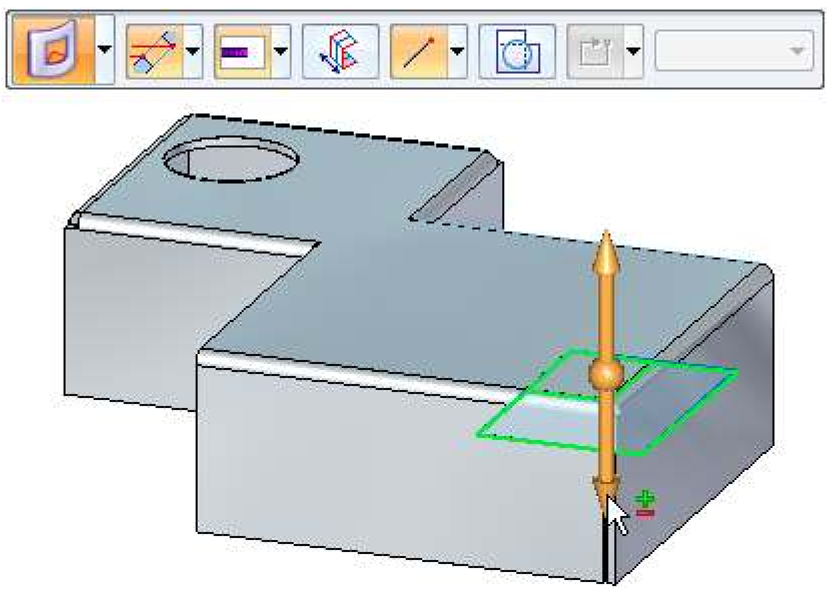
- Proceed to the next step.

Creating a cut

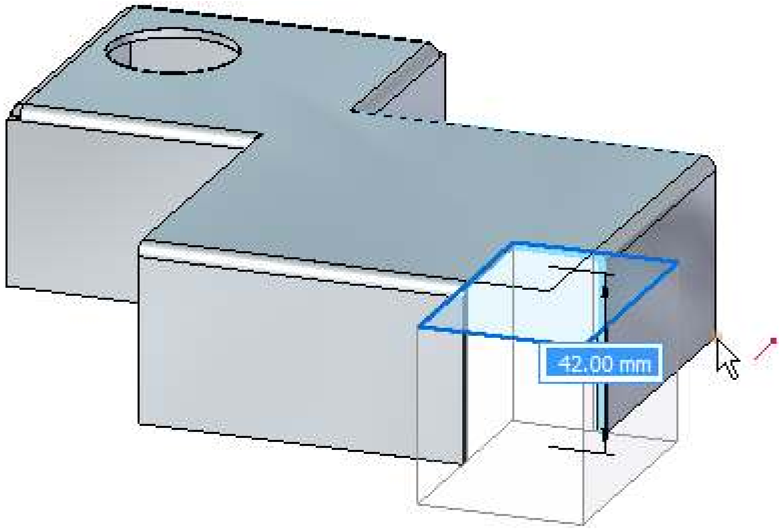
- Lock the sketch plane to the top face and place a rectangle approximately as the one shown below.



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

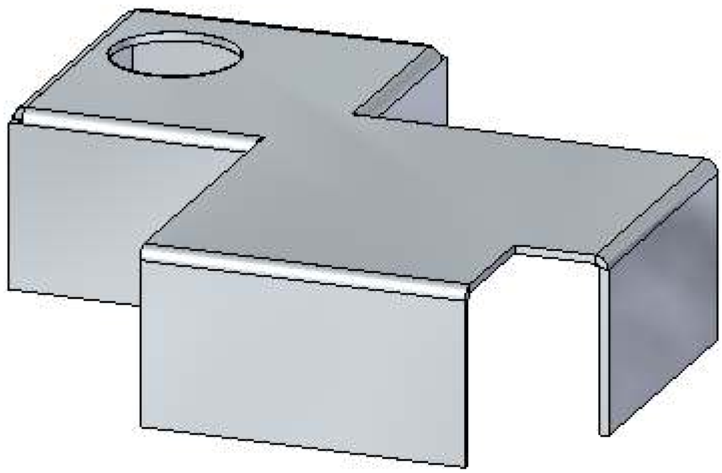


- Click the endpoint on the edge shown to create the cut.



Note

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

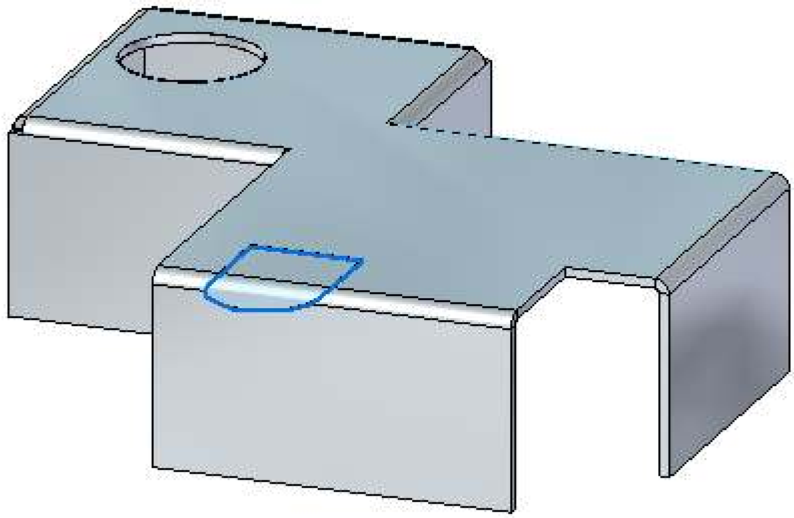


- Proceed to the next step.

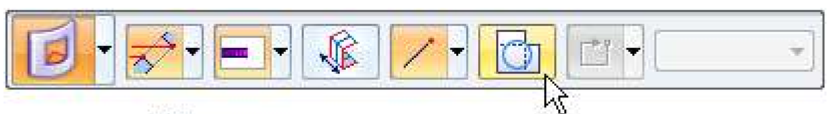
Creating a wrapped cut

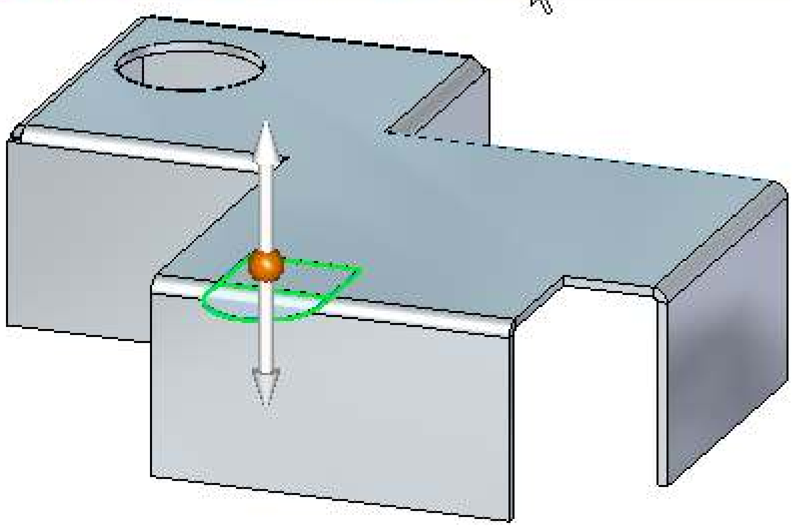
The wrapped cut command temporarily flattens the bend to place the cut.

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

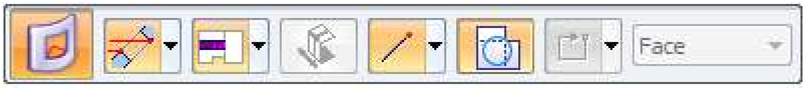


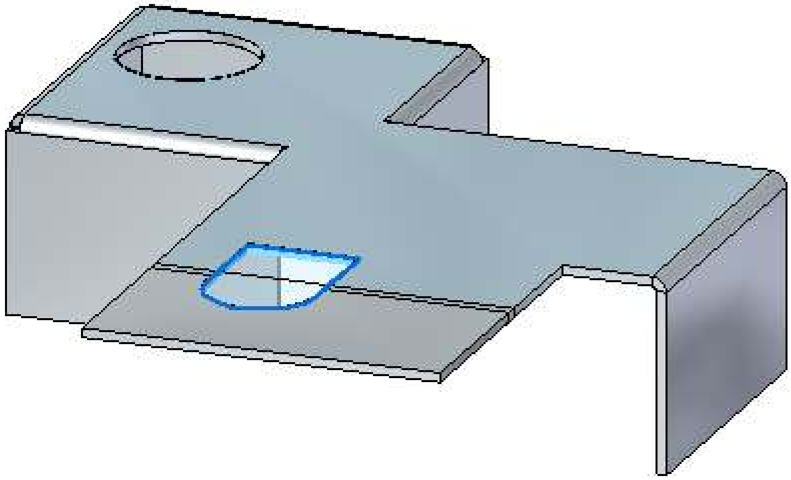
- Select the two regions shown and click the Wrapped Cut option as shown



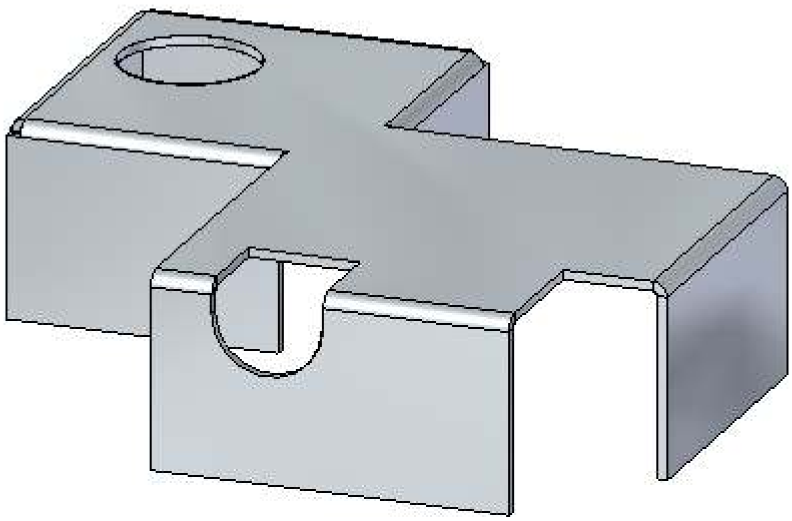


- Select the downward pointing handle. The part is unfolded showing a preview of the wrapped cut. right-click to accept.





- The results are shown.

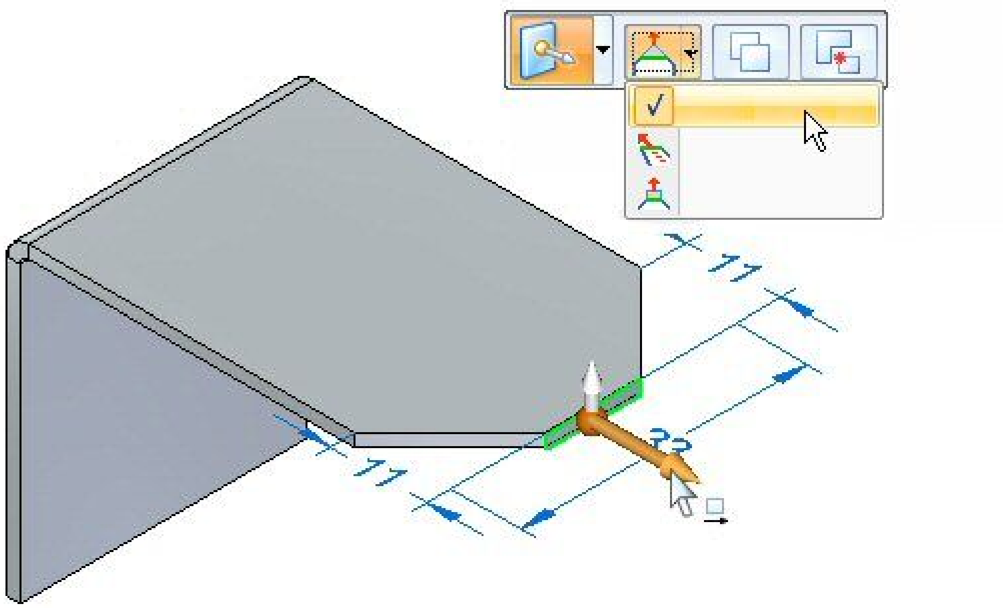


- Close the sheet metal document without saving.

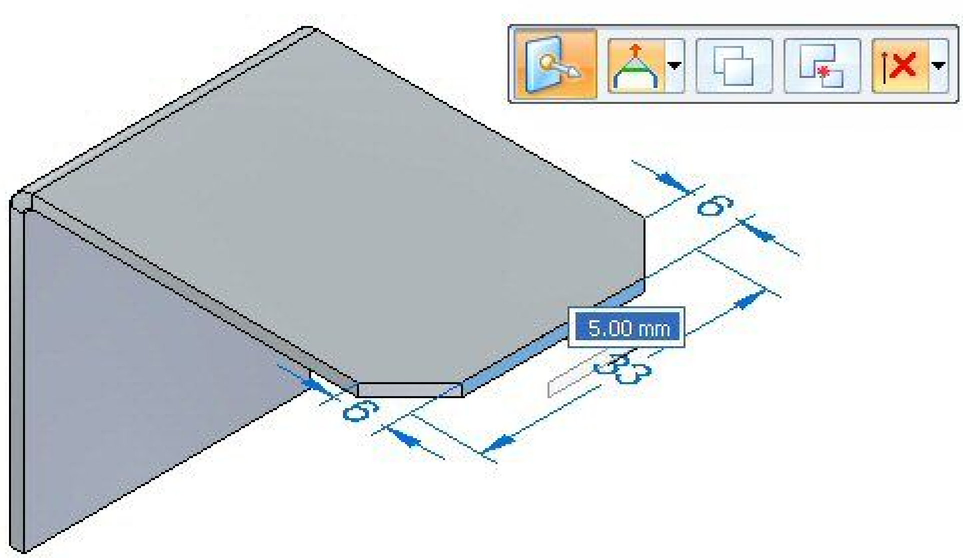
- Proceed to the next step.

Moving thickness faces

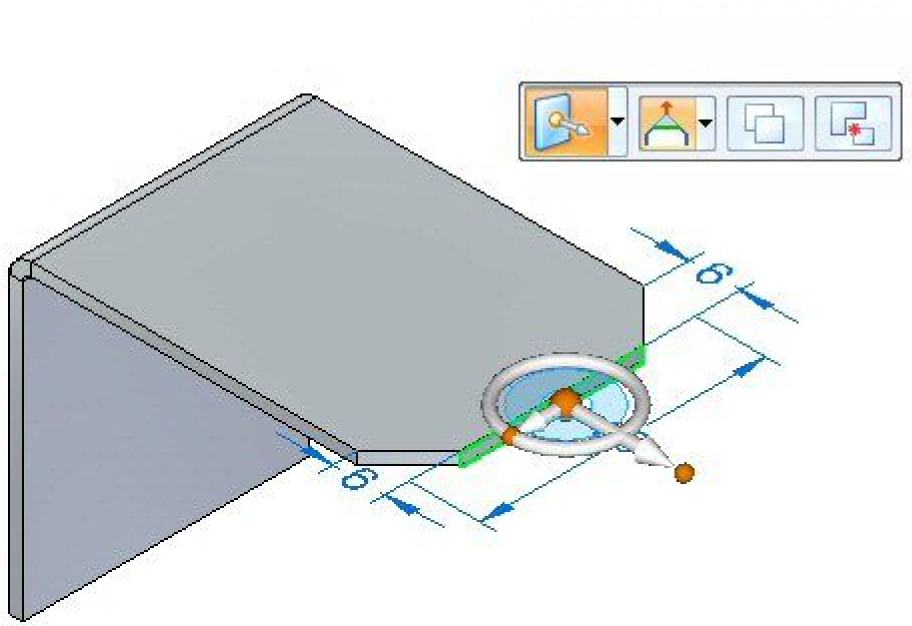
- Click the Application button ® **Open** ® *tab_move_activity.psm*.
- Select the primary axis of the thickness face shown. The Extend/Trim option is the default.



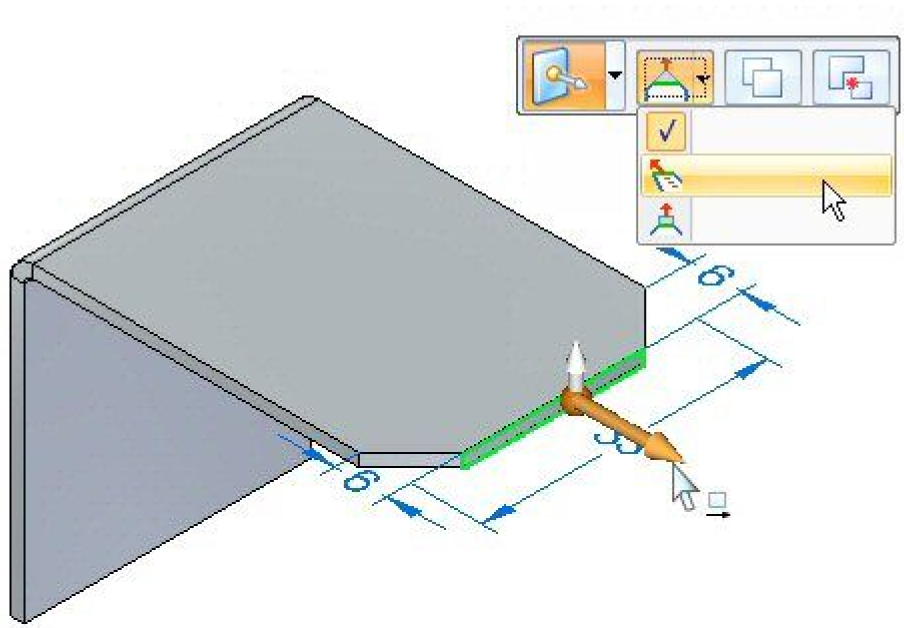
- Move the primary axis and move in the direction toward the bend. Enter a distance of 5.00 mm.



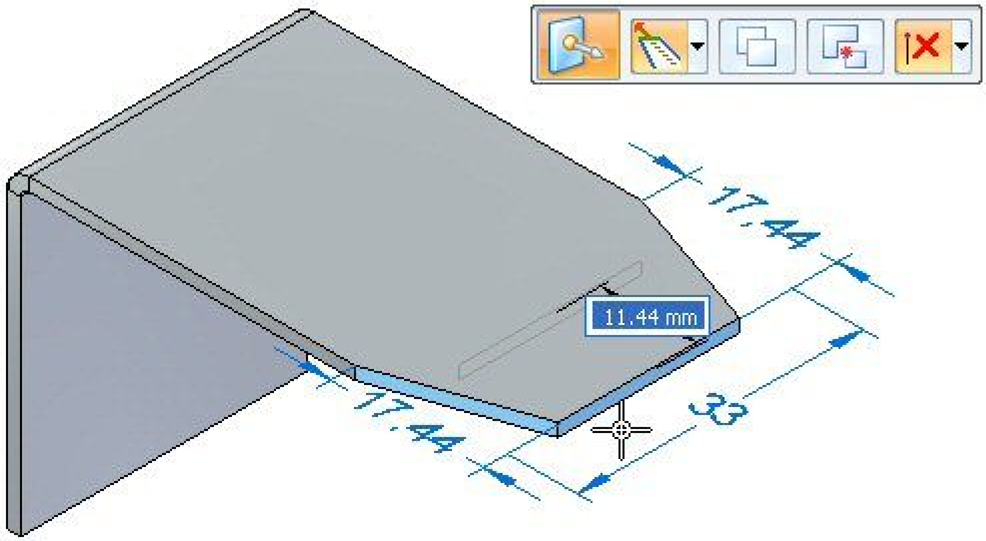
- Observe the behavior. The length of the thickness face changes and the orientation of the adjacent faces remains constant. The result is as shown below.



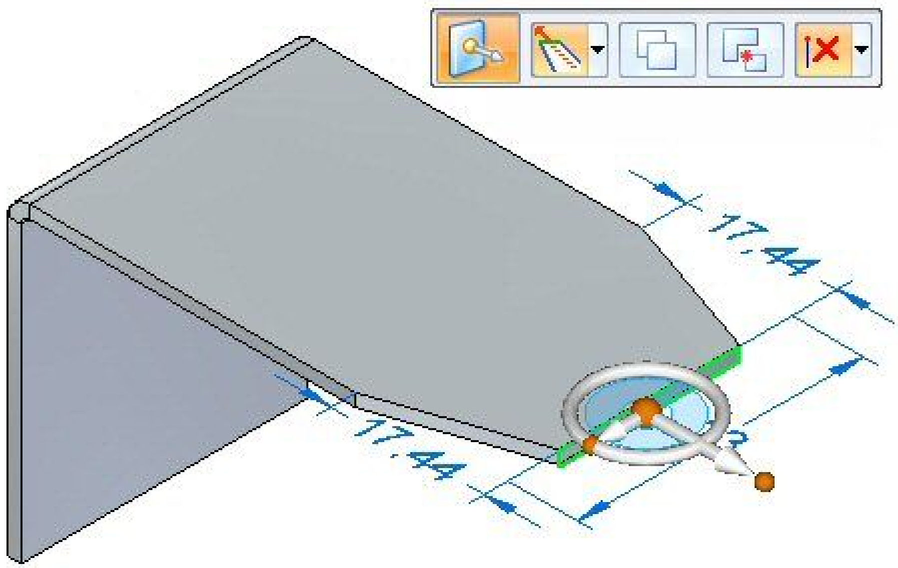
- Select the primary axis and the Tip option as shown.



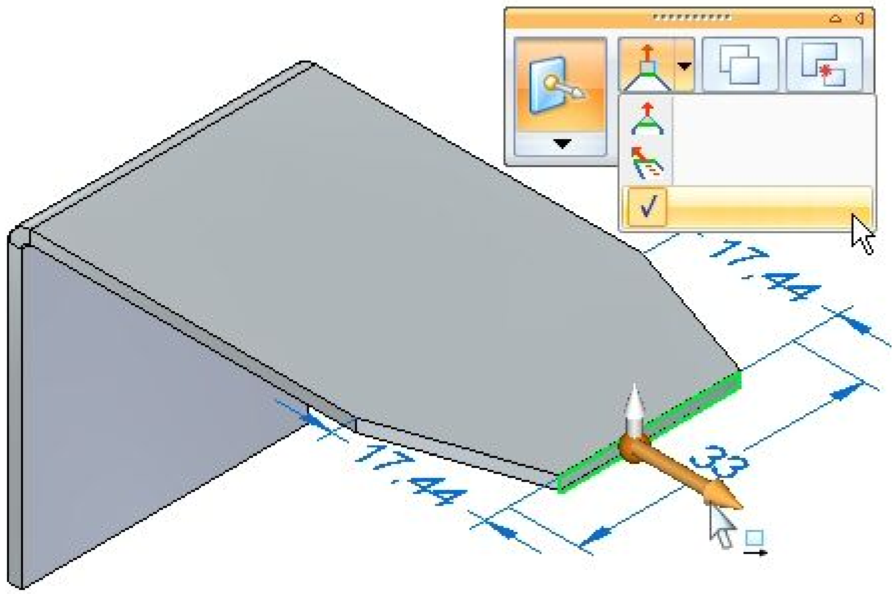
- Enter 11.44 mm for the distance to move the thickness face.



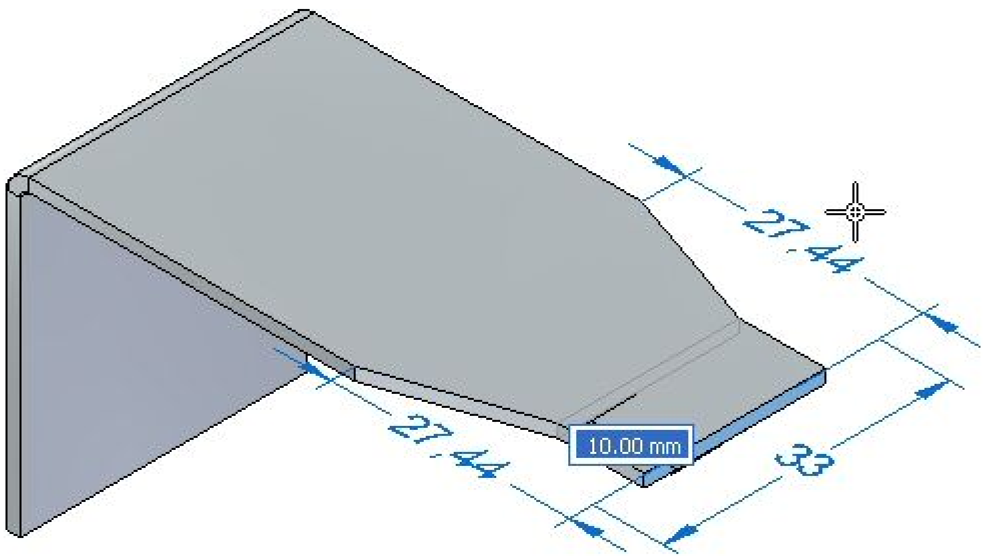
- Observe the behavior. The length of the thickness face is constant and the orientation of the adjacent faces changes. The result is as shown below.



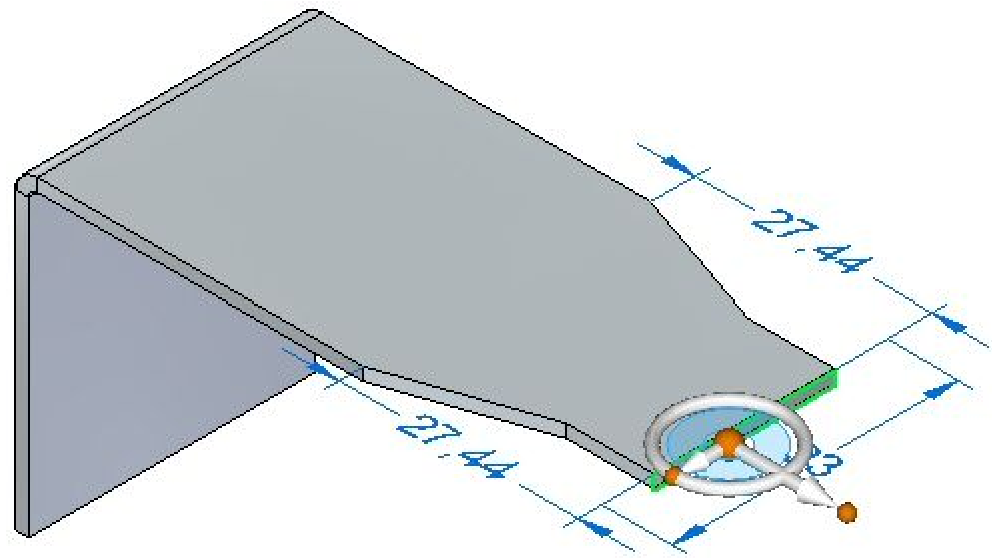
- Select the primary axis and the Lift option as shown.



- 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.

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.

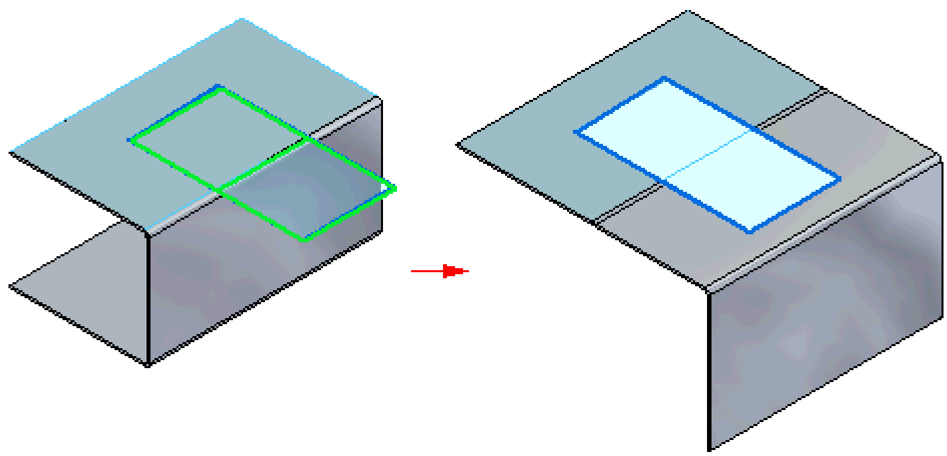
   A tab or a contour flange can be used to generate a base feature in a sheet metal document.

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

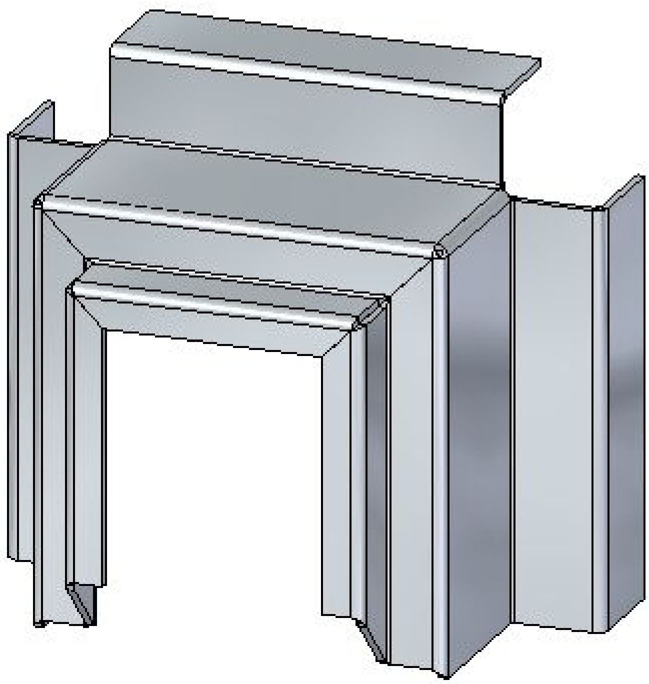
In this lesson you created a sheet metal base feature using a tab, and added additional material creating a tab from a sketch. Regions were used to create a cut and a wrapped cut. You learned the different options for moving a thickness face.

Lesson

5 Contour Flange

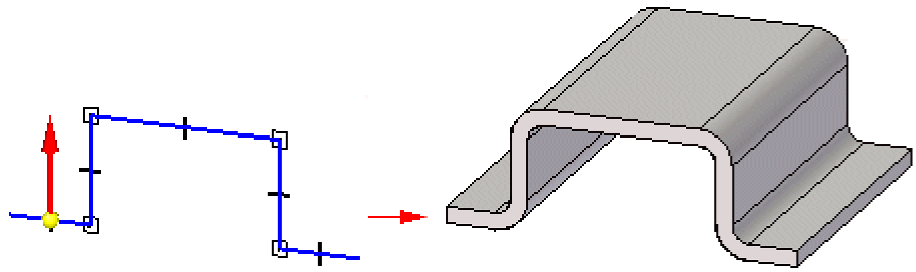
Contour Flange

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

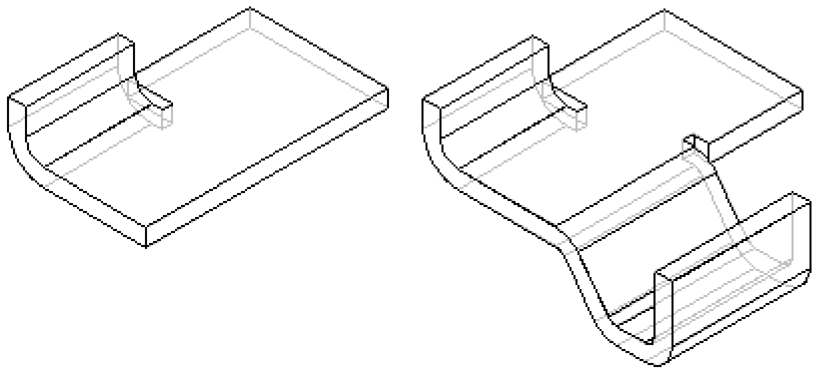
Constructs a contour flange by extruding a profile that represents the edge of the contour flange.



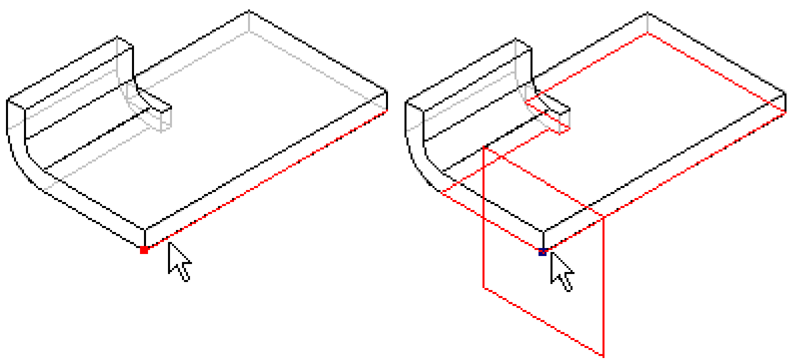
Examples: Defining Reference Plane Orientation to Construct a Contour Flange

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.

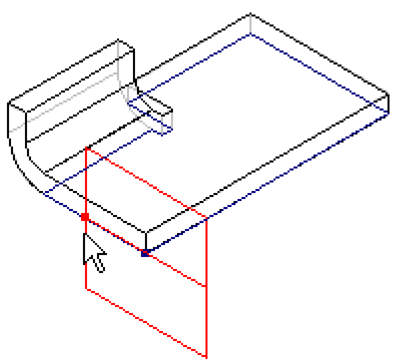
For example, suppose you wanted to construct a contour flange as shown in the figure.



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.

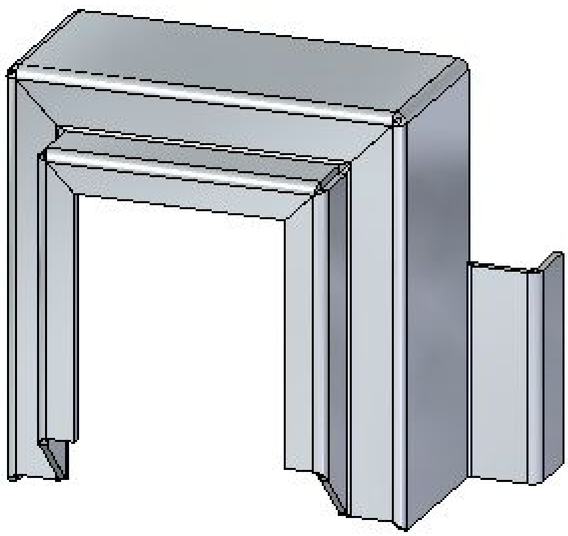


Activity: Constructing a base feature using contour flange

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.



Activity: Constructing a base feature using contour flange

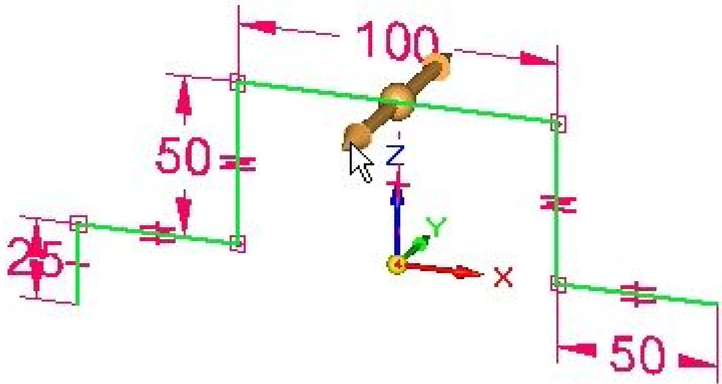
Open a sheet metal file

- Click the **Application** button ® **Open** ® *contour_actvity_1.psm*.
- Proceed to the next step.

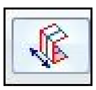
Create a base feature using the Contour Flange command

- Click the Contour Flange command.

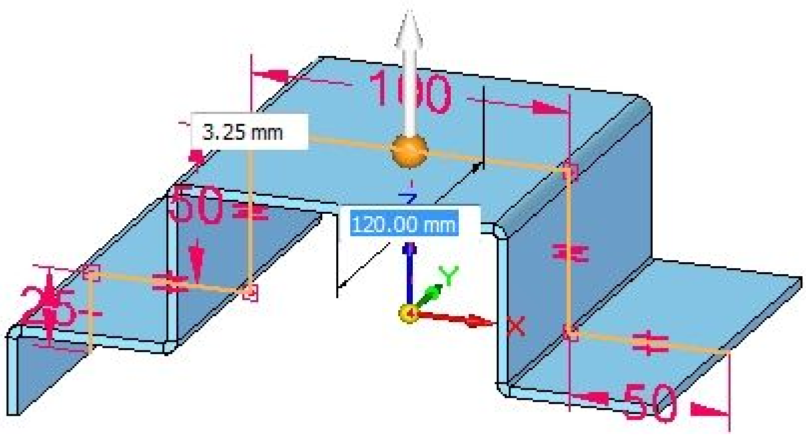
- Select the sketch shown, then click the flange handle.



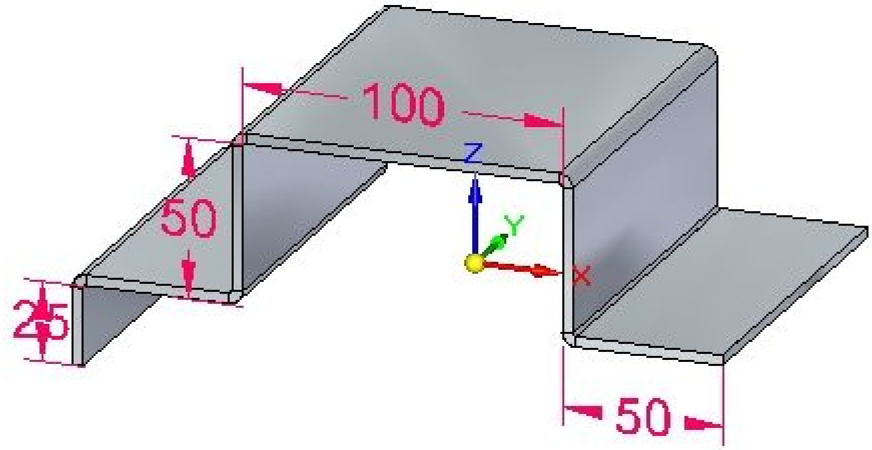
- Click the symmetric extent option.



- 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.



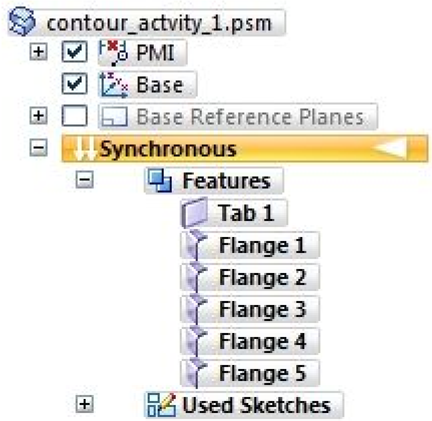
- The results are shown.



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.



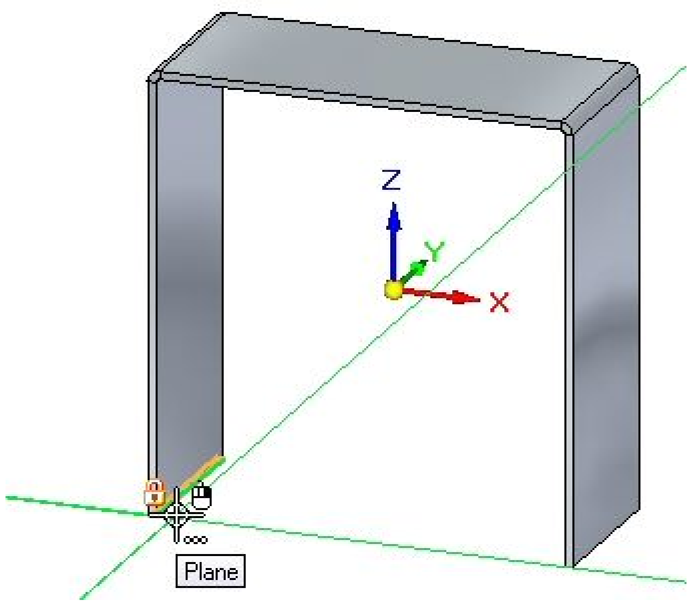
Note

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

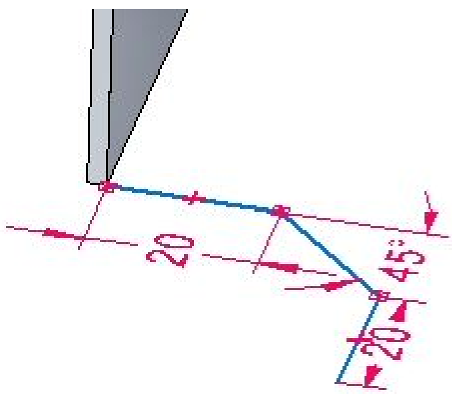
- Save and close the sheet metal document. Proceed to the next step.

Create a contour flange

- Click the  **Application** button ® **Open** ® *contour_actvity_2.psm*.
- Lock the sketch plane to the plane shown.

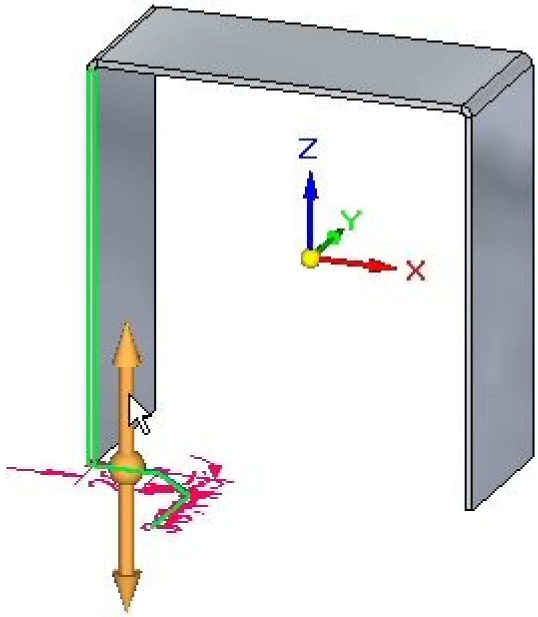


- Create the sketch shown. All segments are 20.00 mm.

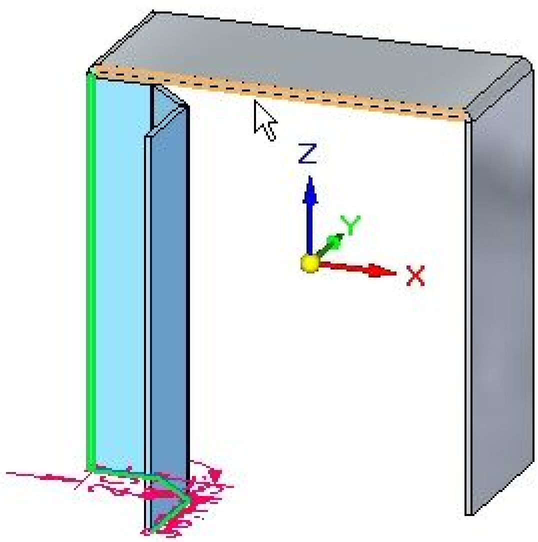


- Select the Contour Flange command.

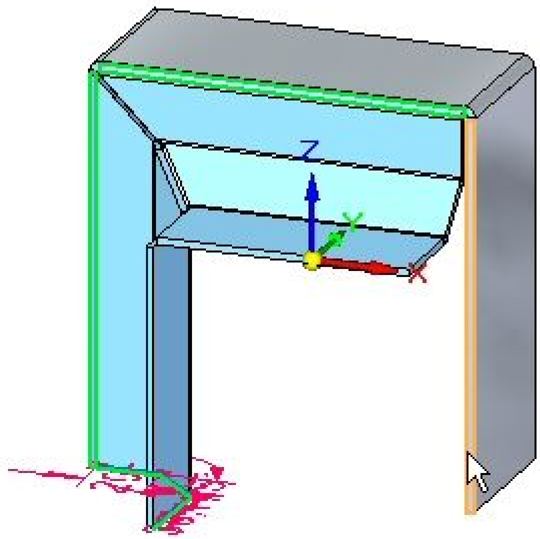
- Select the handle shown.



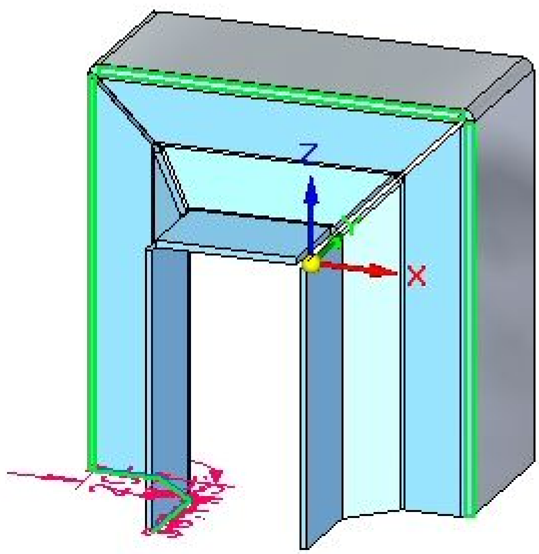
- Continue by selecting the adjacent edge shown.



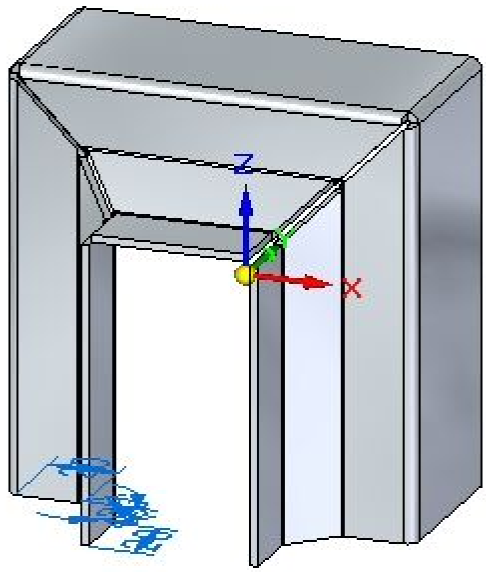
- Continue by selecting the adjacent edge shown.



- The preview is shown.



- Right-click to complete the contour flange.

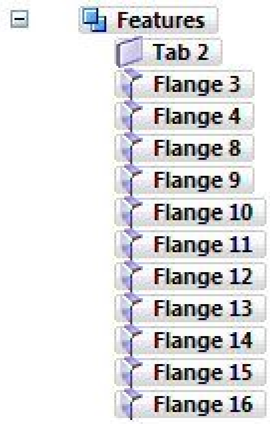


- Observe PathFinder by moving the cursor over the features.

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

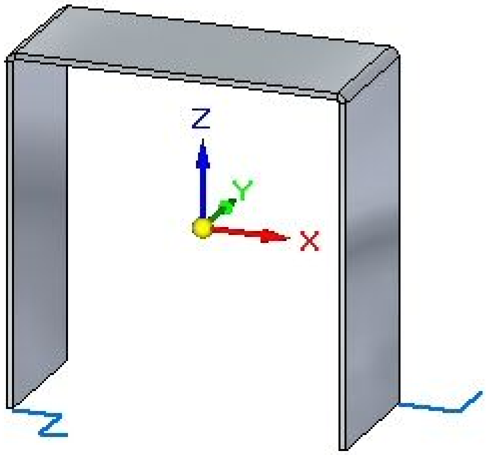
The flange numbers in PathFinder may not match the numbers in the image above. This is not a problem.

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_actvity_3.psm*.

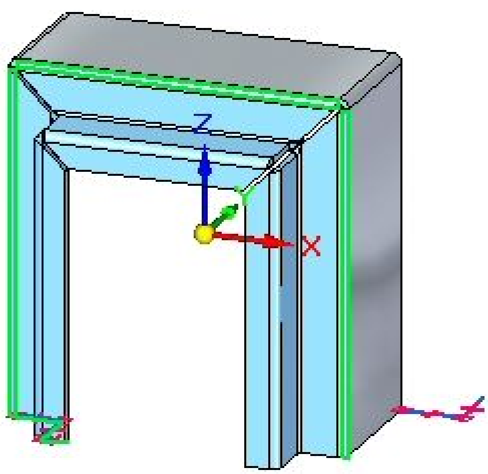


- Select the Contour Flange command.

Note

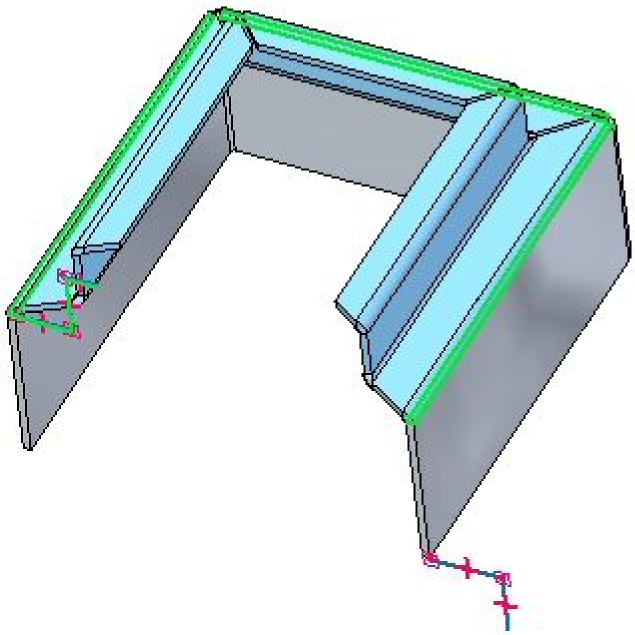
In the following steps you will change the options for end conditions on the contour flange and view the end conditions in the preview without accepting until the last step.

- Begin the contour flange shown using the default parameters.



- 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 –30º, then click OK. Observe the results.

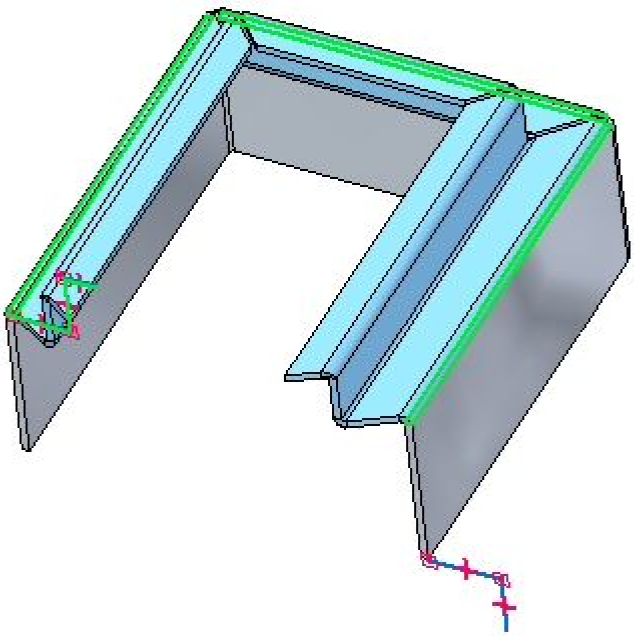


Note

The view has been rotated for clarity.

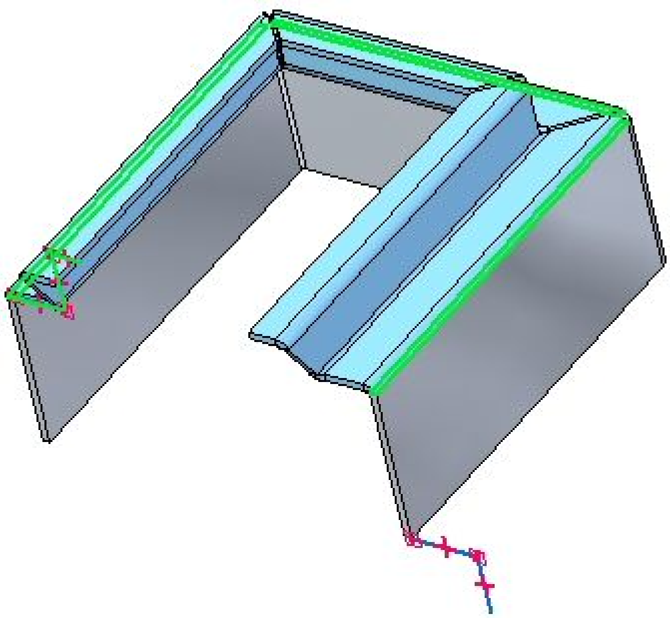
- Click the Options button.

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



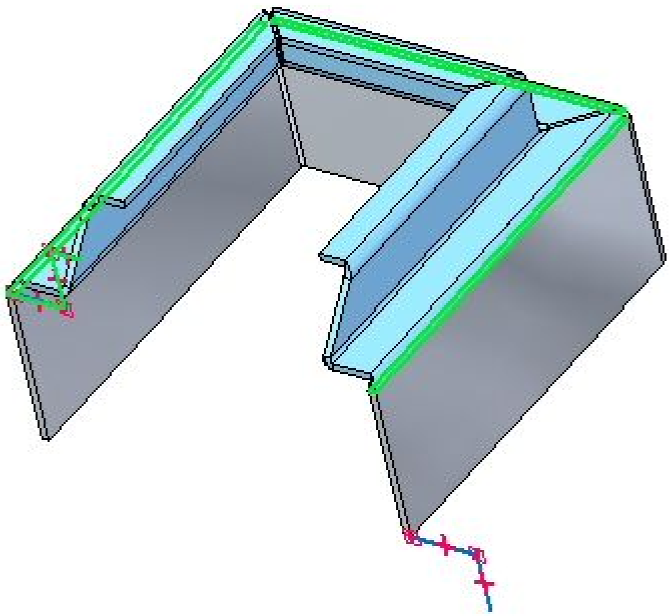
- Click the Options button.

- On the Miters 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.



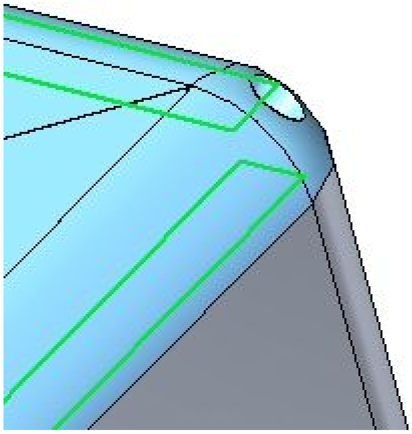
- Click the Options button.

- 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 –45º. Then click OK. Observe the results.

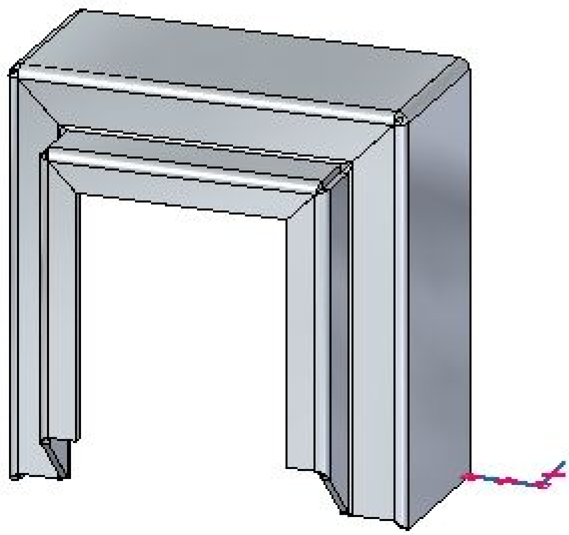


- Click the Options button.

- On the Miters 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.

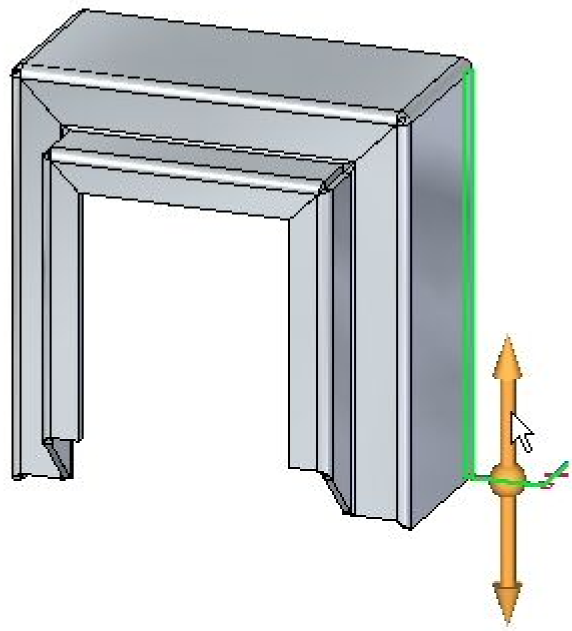


- Proceed to the next step.

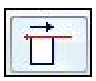
Create a partial contour flange

- Select the Contour Flange command.

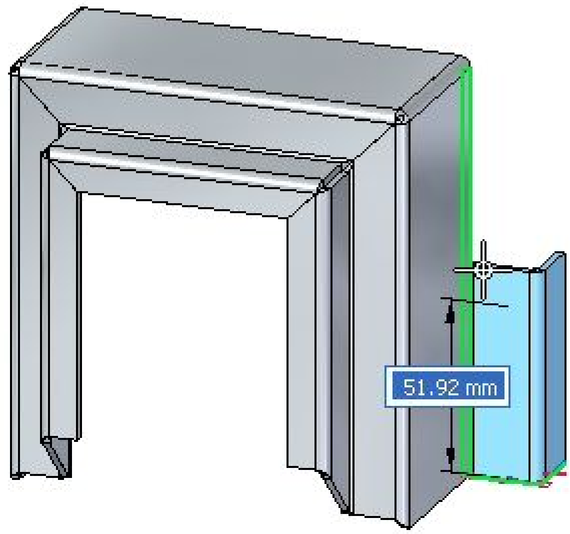
- Select the sketch as shown.



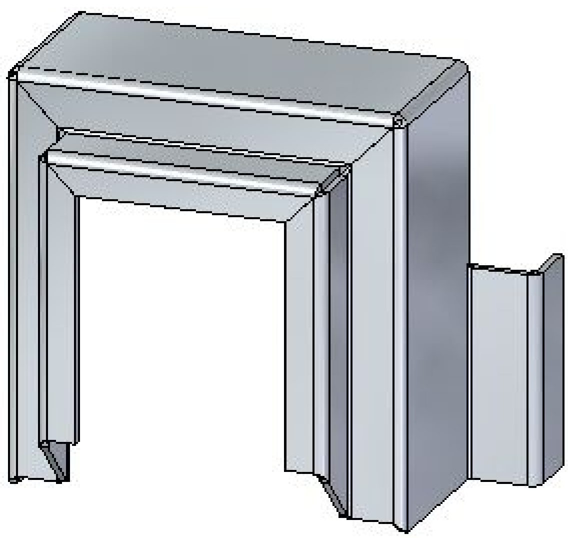
- Click the Partial Contour Flange option.



- 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.

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

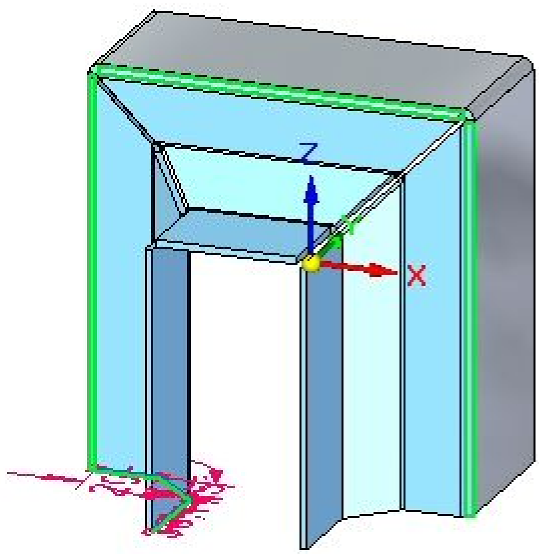
1. How can you make a base feature using the contour flange command?

   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

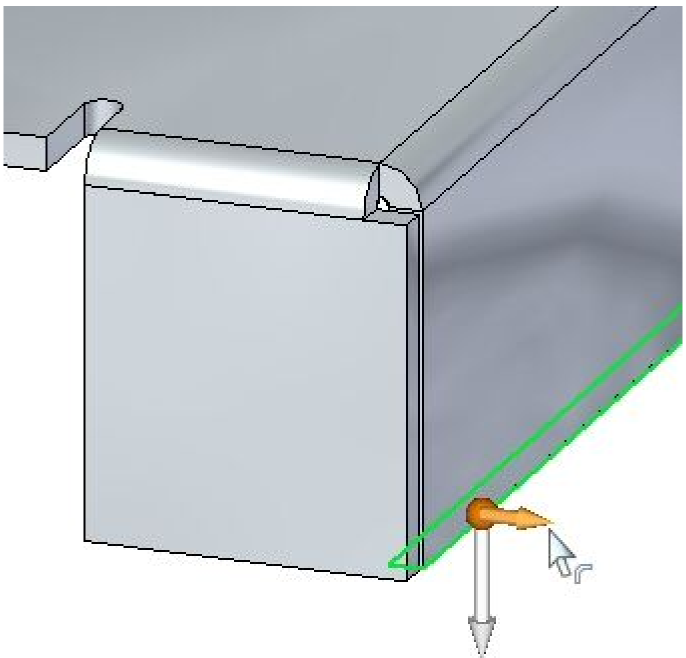
In this lesson you set the material thickness and extent to create a base feature using a contour flange. The components of the contour flange were examined and manipulated. Options for the construction of end conditions were explored, and a partial contour flange was placed.

Lesson

6 Flanges, corners and bend relief

Flanges, corners and bend relief

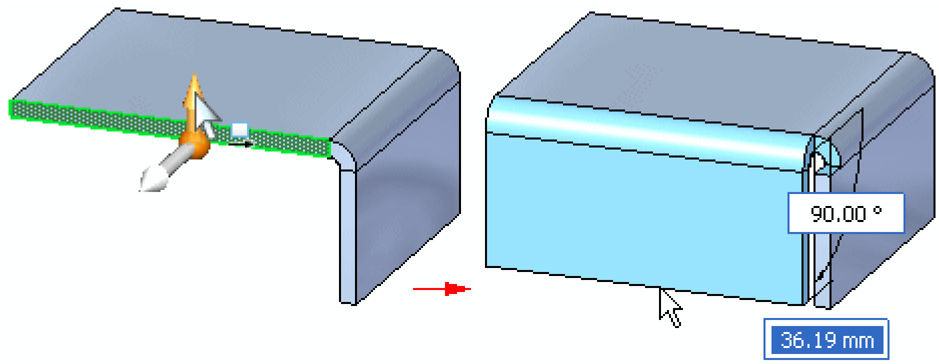
You create flanges using flange handles. As you create them, you can control end conditions such as bend relief and corner conditions. You can insert bends across layer faces.



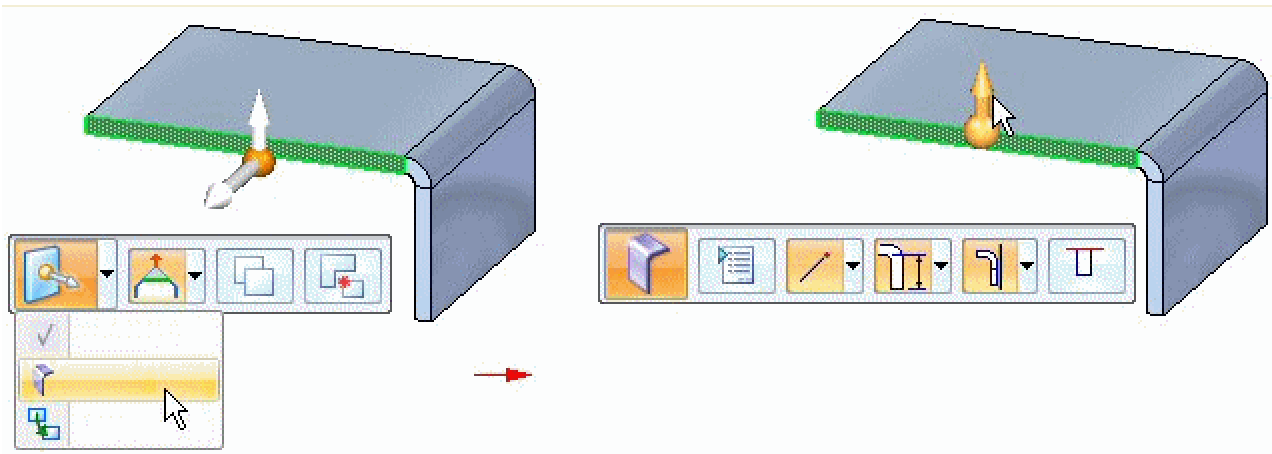
Creating flanges

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.

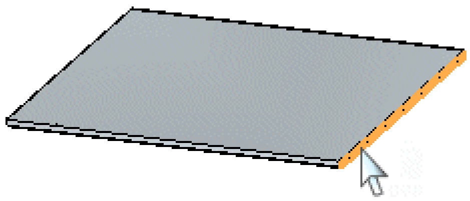


Flange command

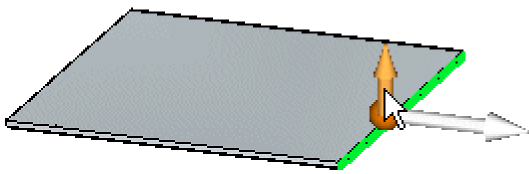
Constructs a flange by extruding material that represents the face of the flange.

Flanges in the synchronous environment

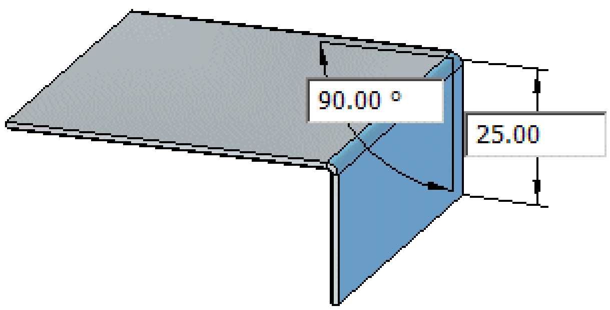
In the synchronous environment, you can construct a flange by selecting a linear thickness edge to display the flange start handle,



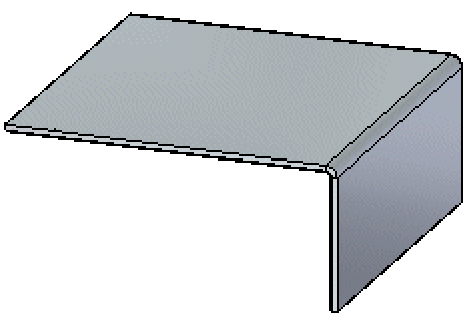
clicking the flange start handle,



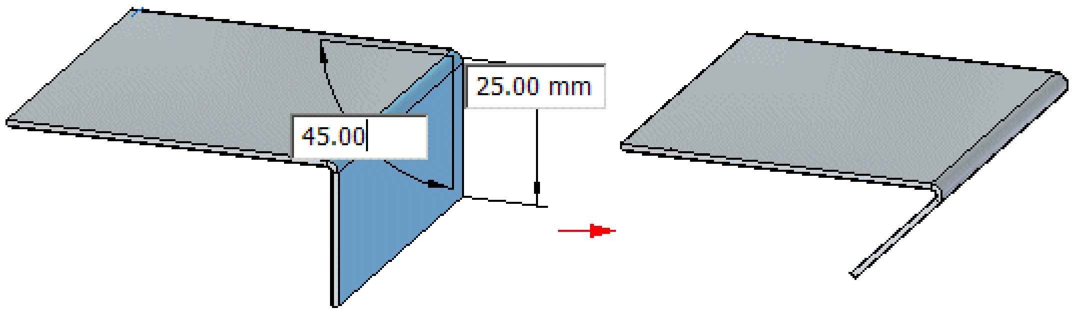
specifying a flange distance,



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.

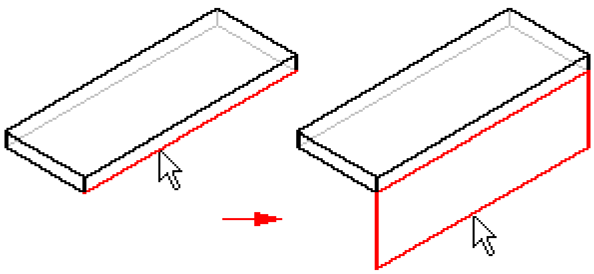


Note

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.



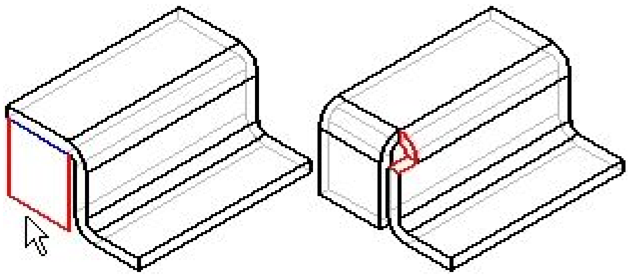
Corner Relief

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.

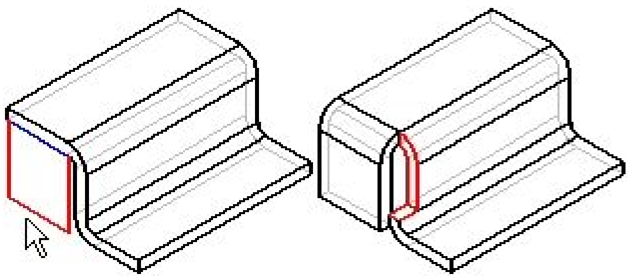
Bend Only

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



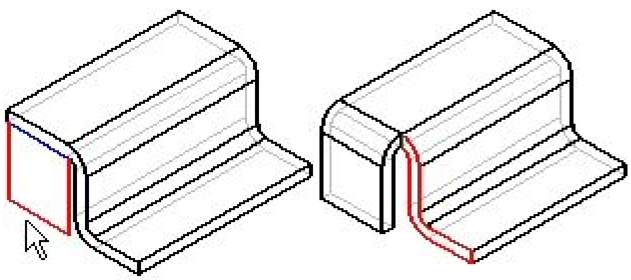
Bend and Face

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



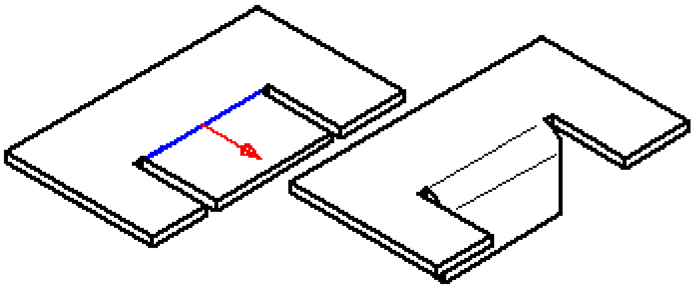
Bend and Face Chain

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.

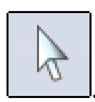
7. Define the bend direction.

8. Finish the feature.

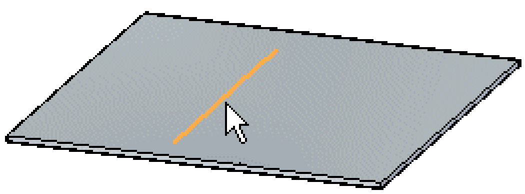
   Tip

   - 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

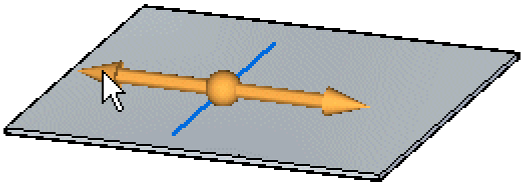
1. Choose Home tab® Select group® Select .

2. Select the sketch element to create the bend.

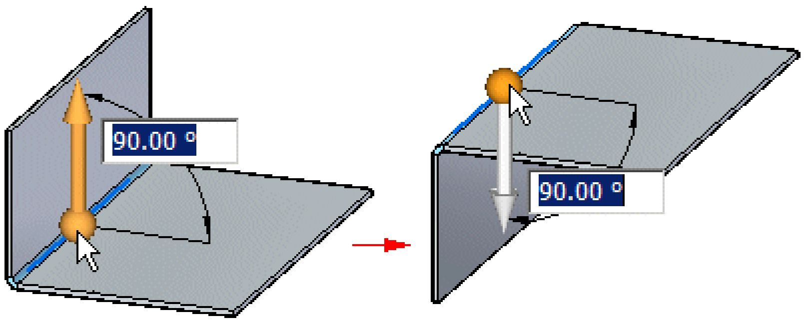
   

3. Choose Home tab® Sheet Metal group® Bends list® Bend .

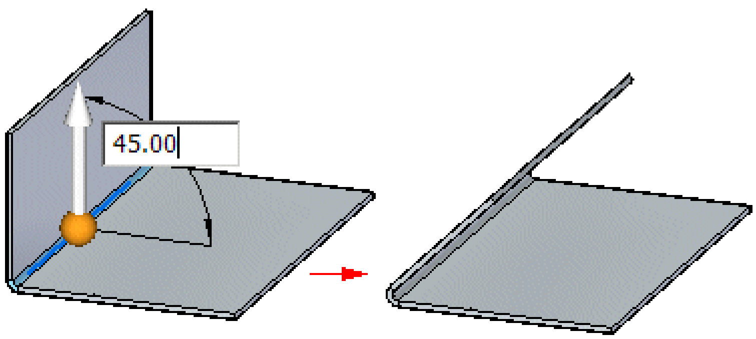
4. Click the side of the sketch to move.



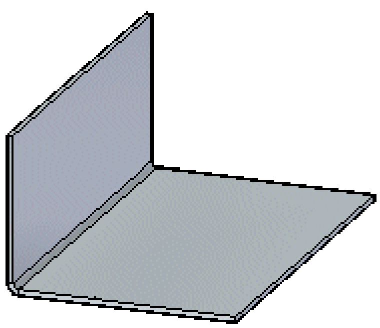
5. (Optional) Click to the direction arrow to change the direction of the bend.



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



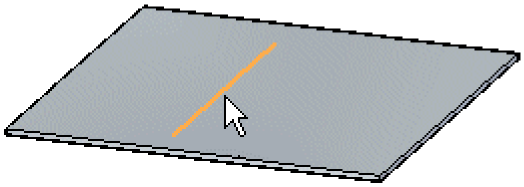
7. Click to create the bend.



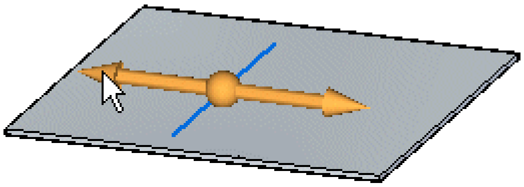
Insert a bend in the synchronous environment with the Bend command

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

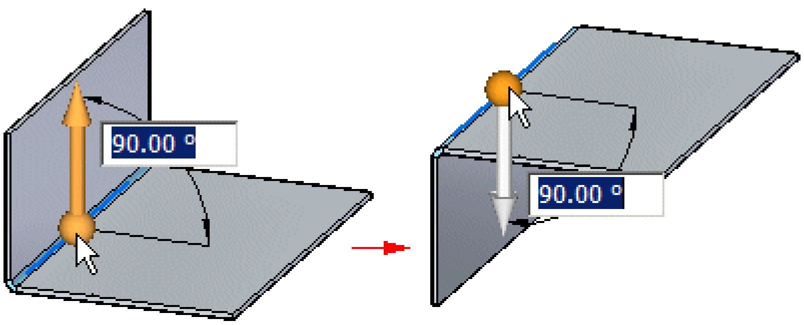
2. Select the sketch element to create the bend.



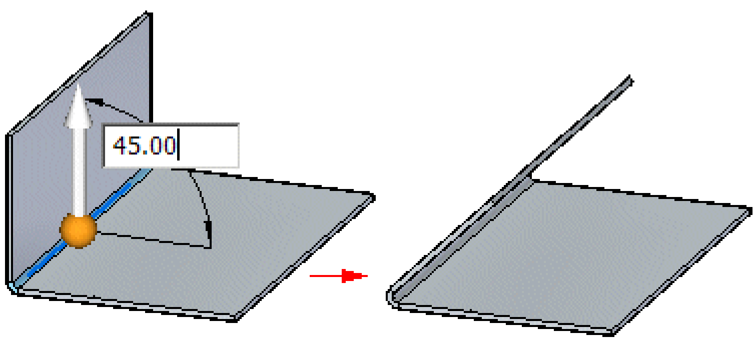
3. Click the side of the sketch to move.



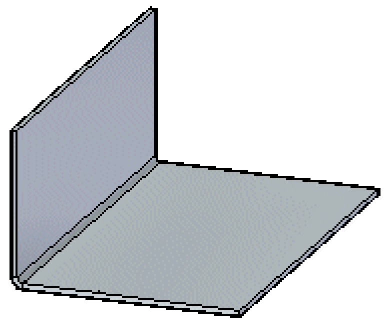
4. (Optional) Click to the direction arrow to change the direction of the bend.



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

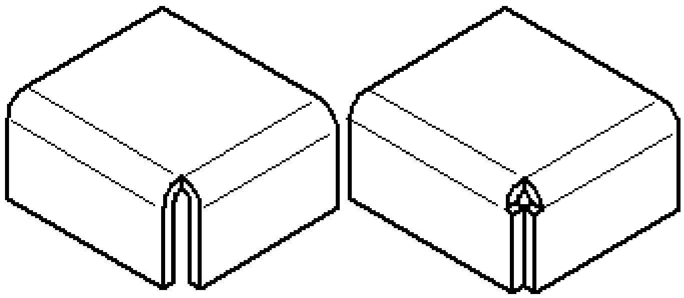


6. Click to create the bend.

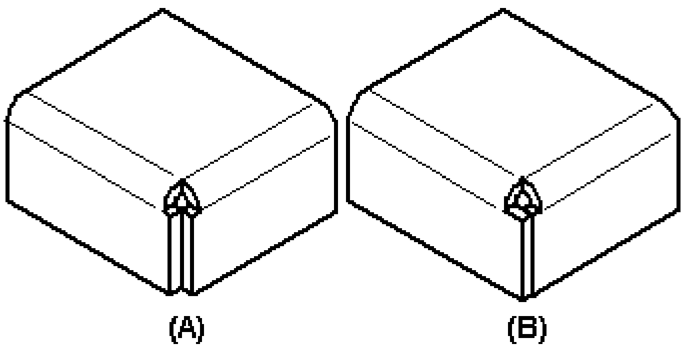


Close 2-Bend Corner command

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 of 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.

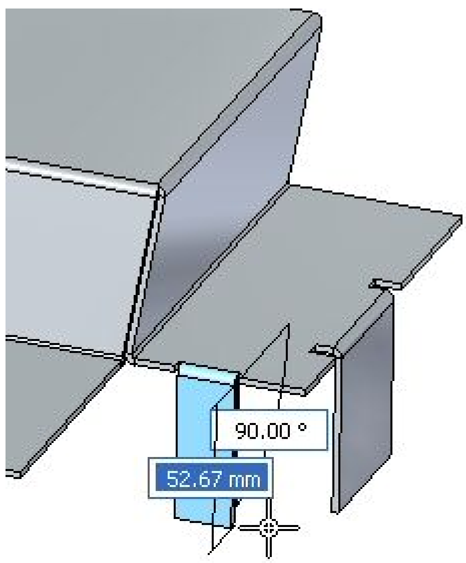
You can select a closed corner for deletion, either in PathFinder or in the graphics window. When you delete a closed corner, the corner definition is removed from the model and bends return to the default bend state.

Activity: Flange and corner conditions

Activity objectives

This activity demonstrates control flange geometry and end contditions 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.



Activity: Flange and corner conditions

Open a sheet metal file

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

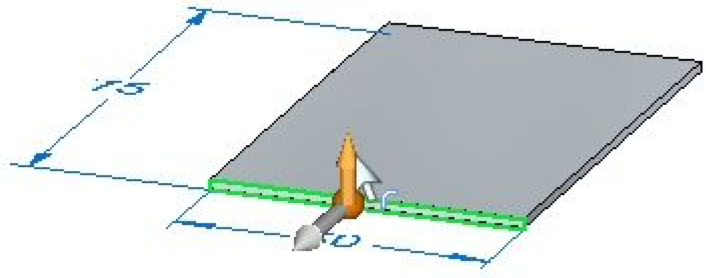
  Note

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

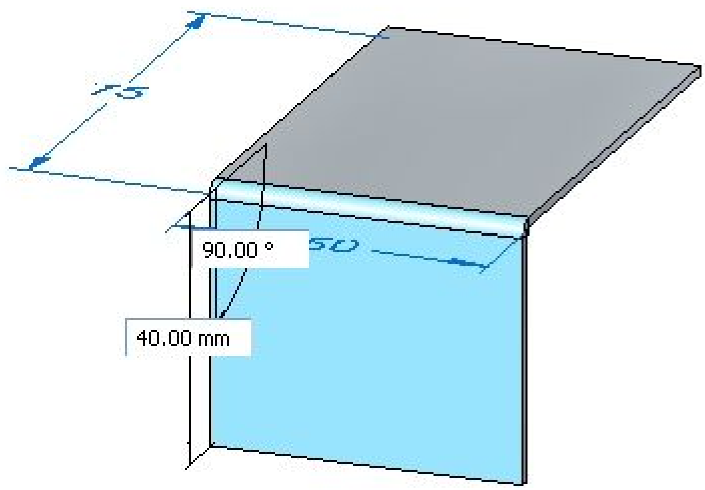
- Proceed to the next step.

Flange creation options

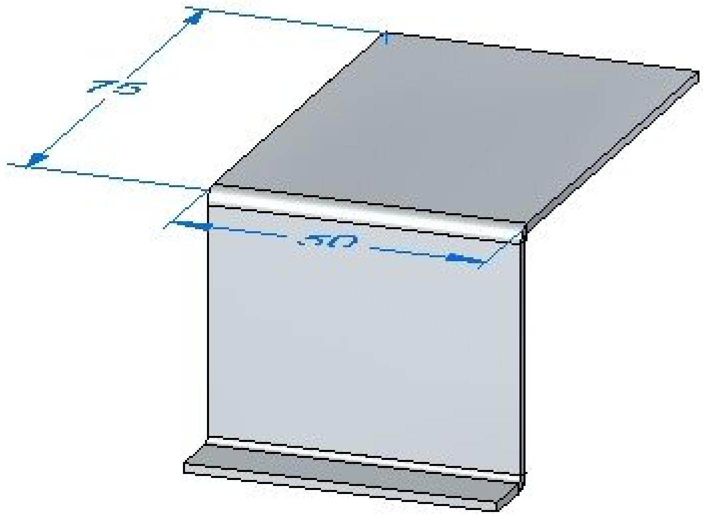
- Select the face shown and click the flange start handle.



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



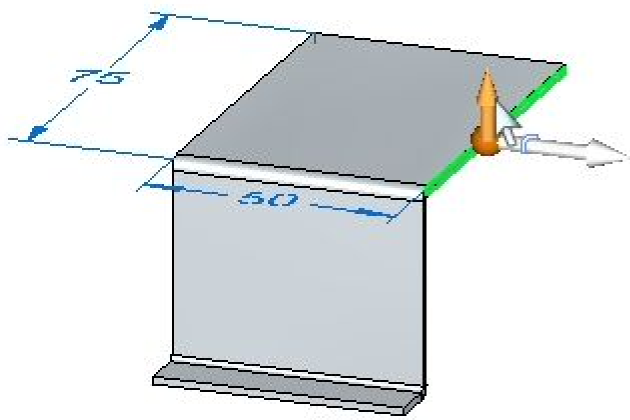
- Create the flange shown below with a length of 10.00 mm.



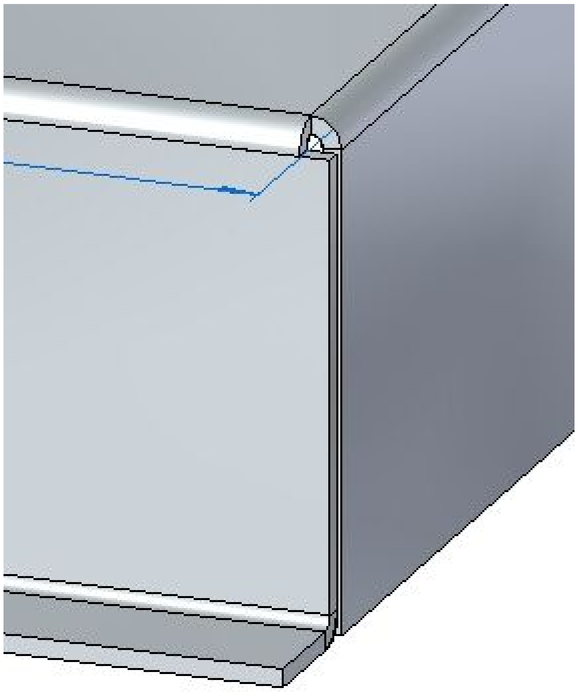
Note

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**.

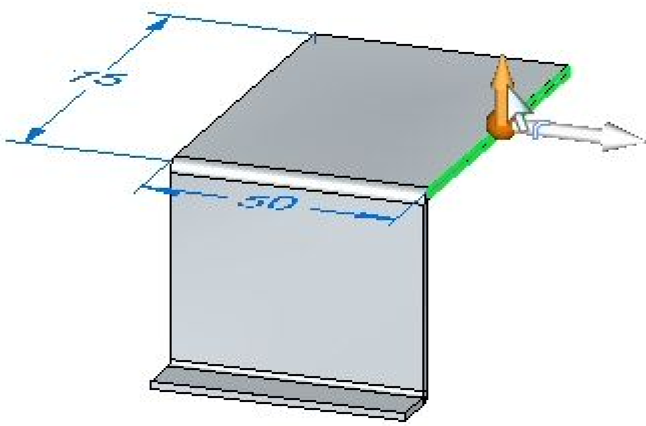


- Pull the flange to the end of the bottom of the flange just created. Observe the corner relief.

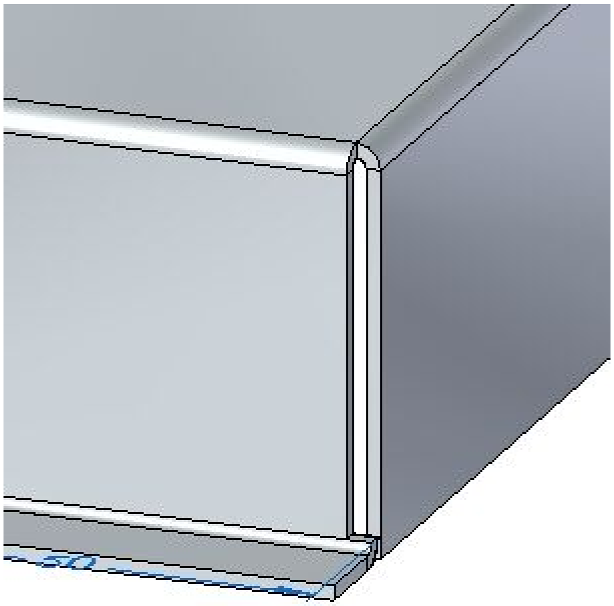


- Click the Undo command to remove the flange you just created.

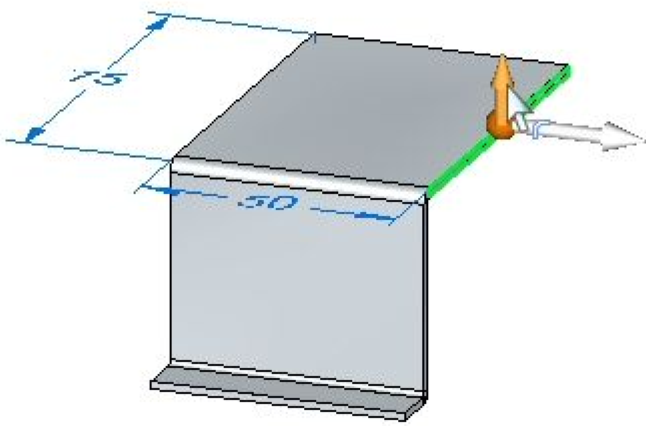
- Select the face shown and click the flange handle. Click the Options button and set the Corner Relief to **Bend and Face**.



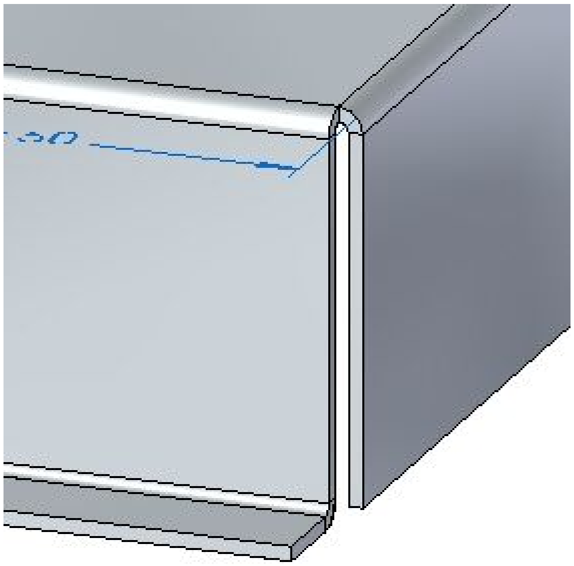
- Pull the flange to the same distance as in the previous step. Observe the corner relief.



- 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.

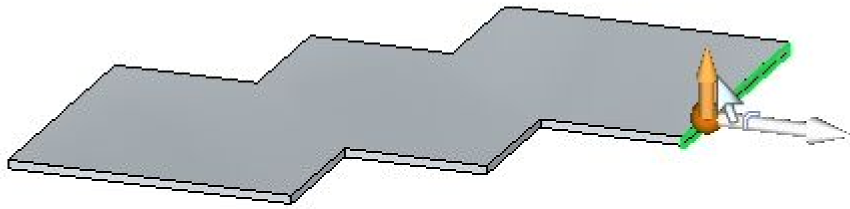
Partial flanges

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

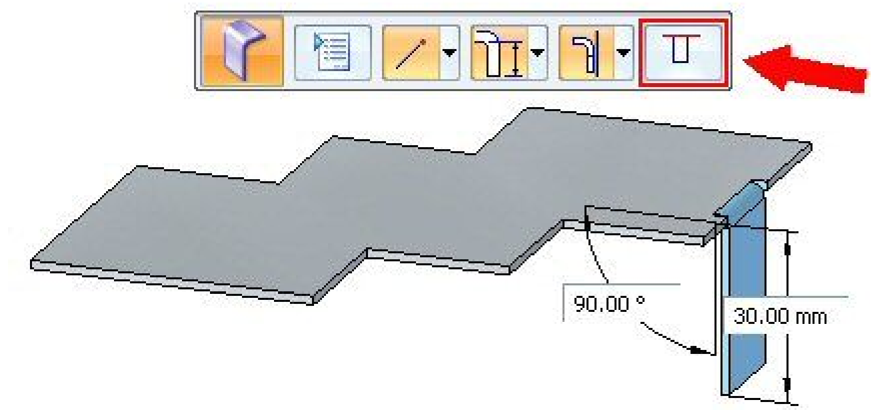
  Note

  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.

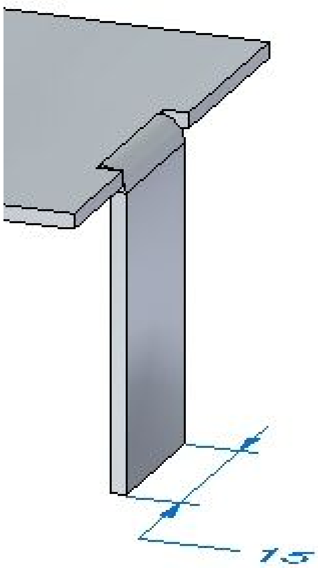


  Note

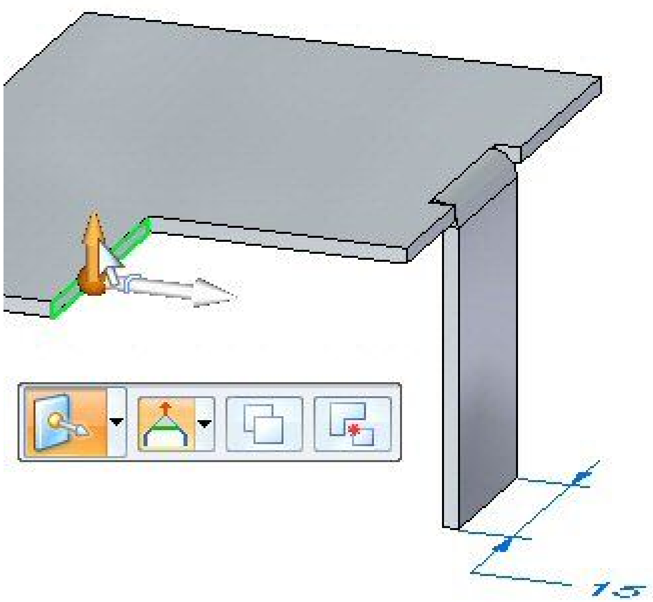
  Partial flanges are created with a width equal to 1/3 of the thickness face chosen, and the selection point is defines the edge of the partial flange. The flange can be modified to the desired width using dimensions to control the width.

- Click the smart dimension command.

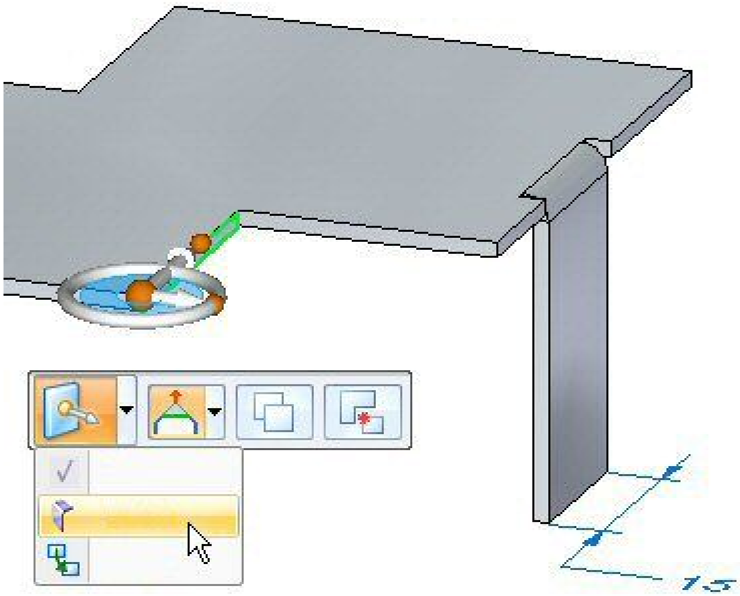
- Place a dimension on the bottom edge of the flange just created. Change the width of the flange to 15.00 mm by editing the dimension.



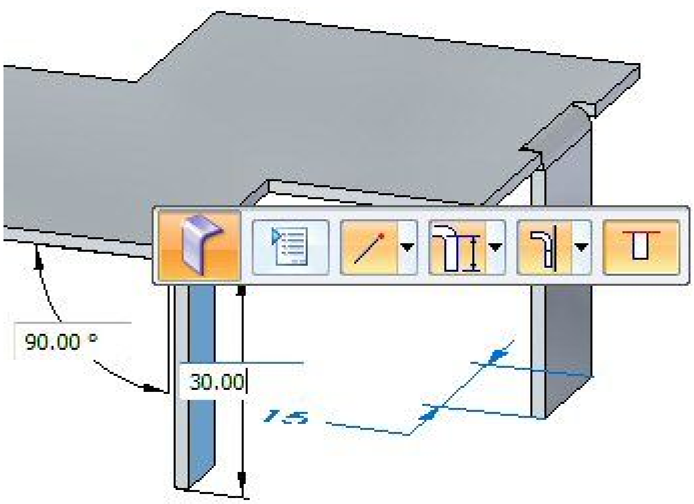
- Select the face shown.



- 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.

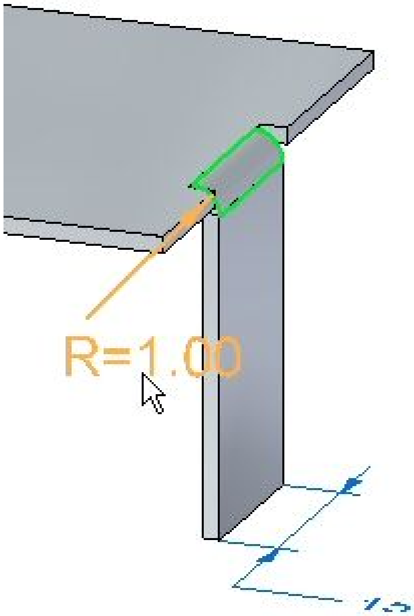
- Proceed to the next step.

Bend relief

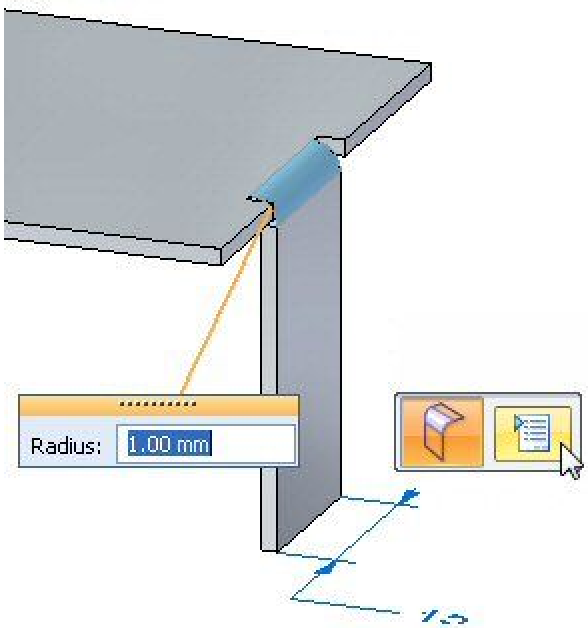
Note

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

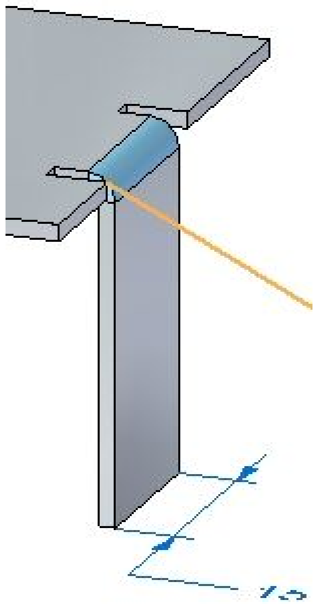
- Select the bend shown, then click the edit feature handle.



- 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.

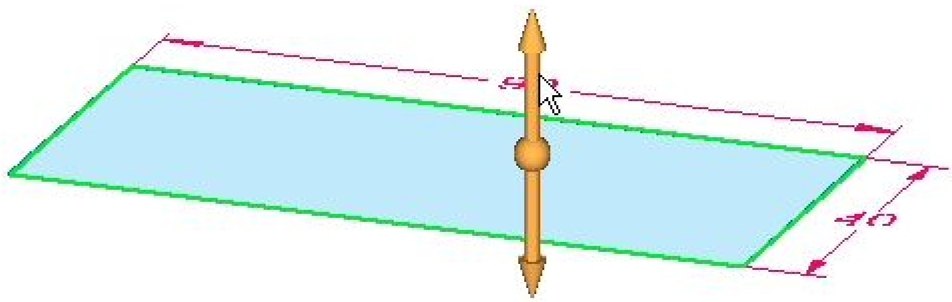
Corners

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

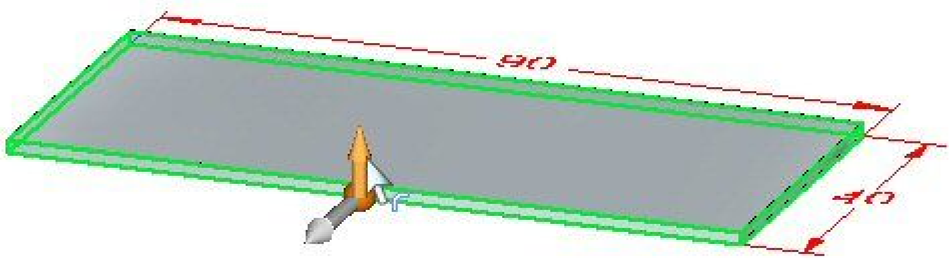
  Note

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

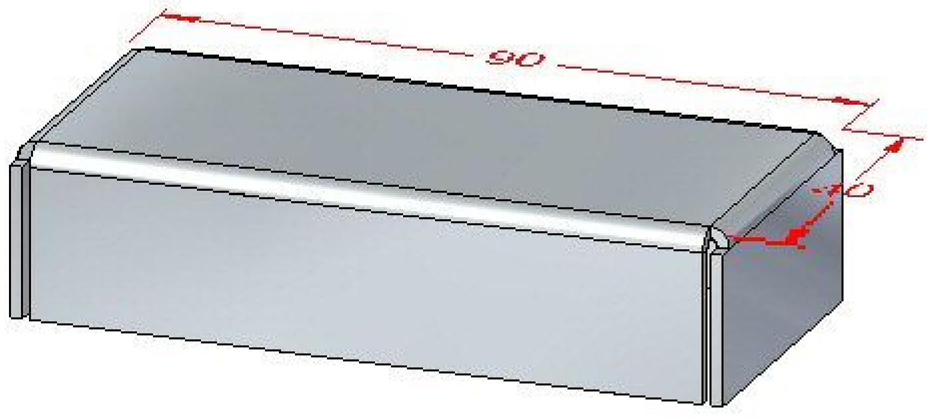
- Select the region shown and create a tab by pulling the handle up.



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



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

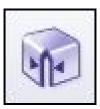


- When more than one thickness edge is used to create flanges, observe the following:
  - Shorter sides are bent first.
  - Relief cuts, when required are made to the longer sides.

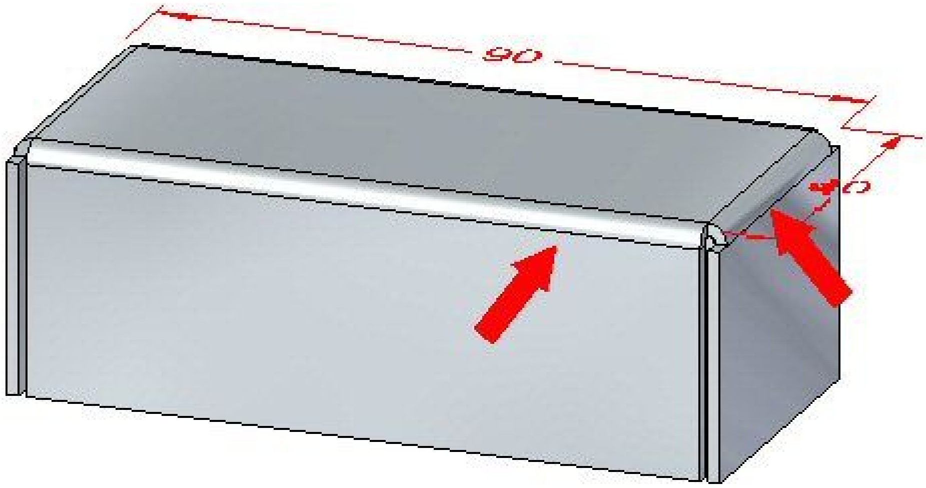
    Note

    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 the2–Bend Corner command.



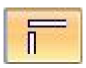
- Select the two bends shown below.



Note

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

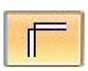
- Click the Overlapping Corner option with the Corner Treatment set to Open.



- 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.



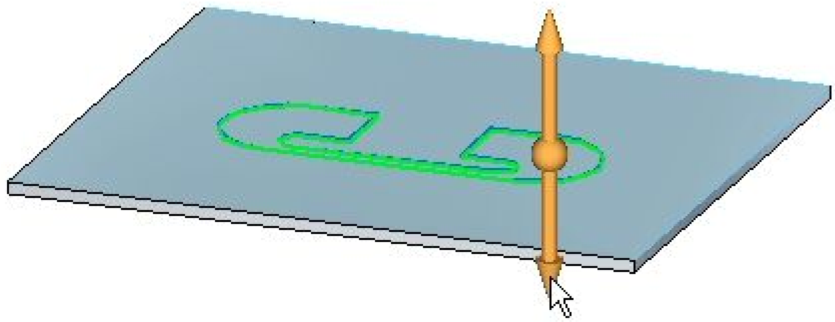
- Change the Corner Treatment to Closed, and set the gap value to 0.30 mm. Observe the change.

- 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.

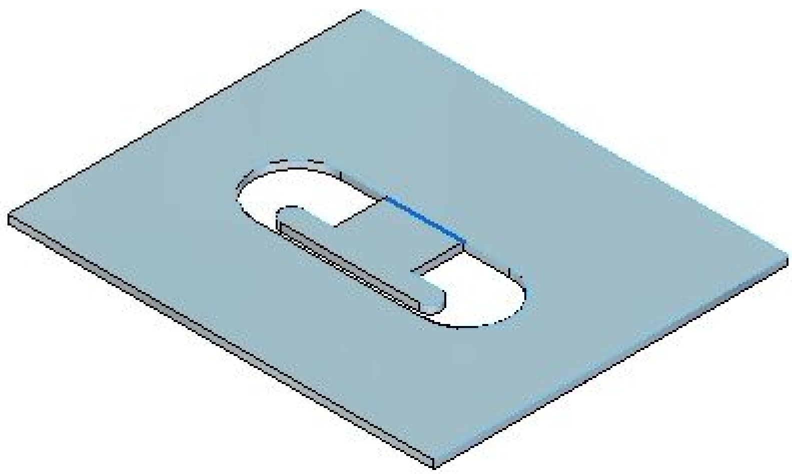
- Close the sheet metal document without saving. Proceed to the next step.

Inserting a bend

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

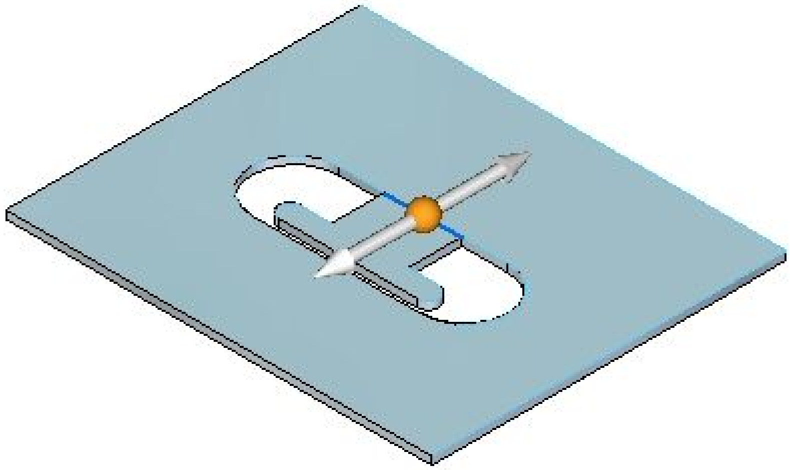


- Sketch a line as shown across the tab.

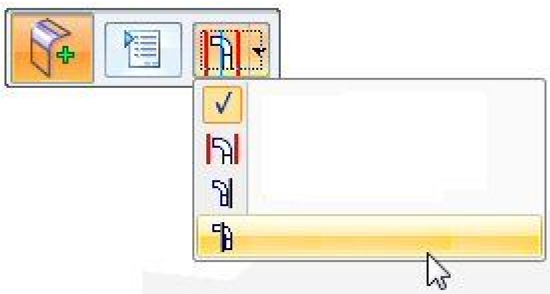


- Click the Bend command.

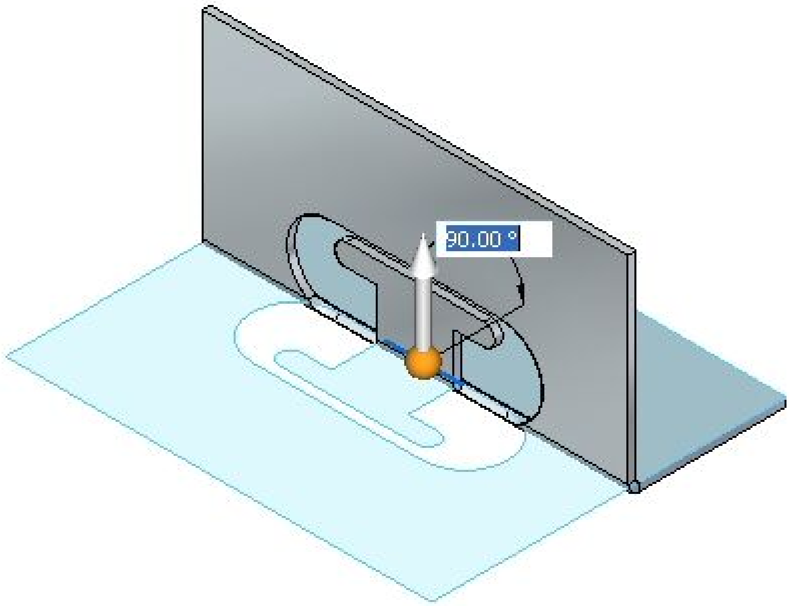
- Select the line as shown.



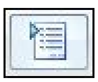
- Select the Material Outside option.



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

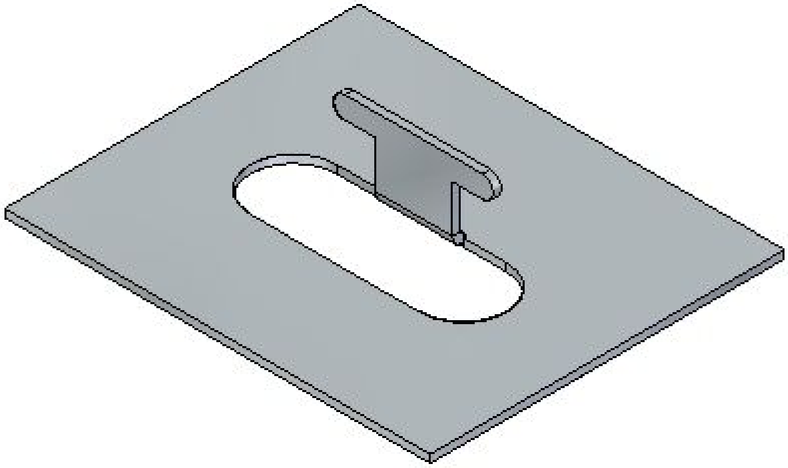


- Click the Bend Options button



- Deselect Extend Profile and click OK.

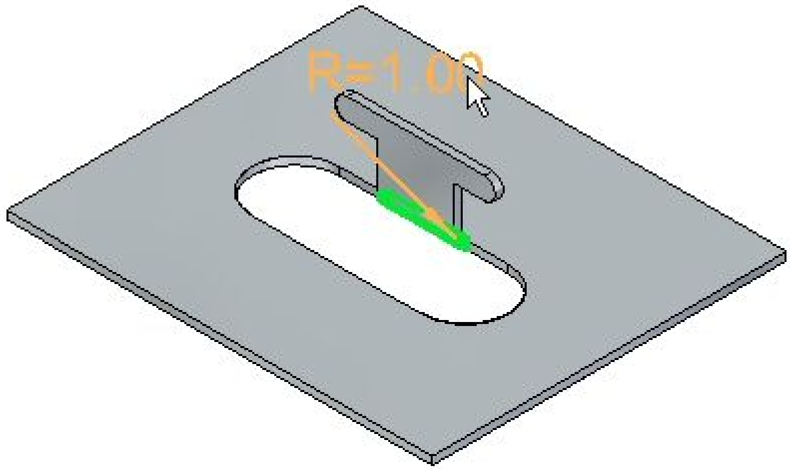
- Right—click to complete the bend and add the flange. The results are shown.



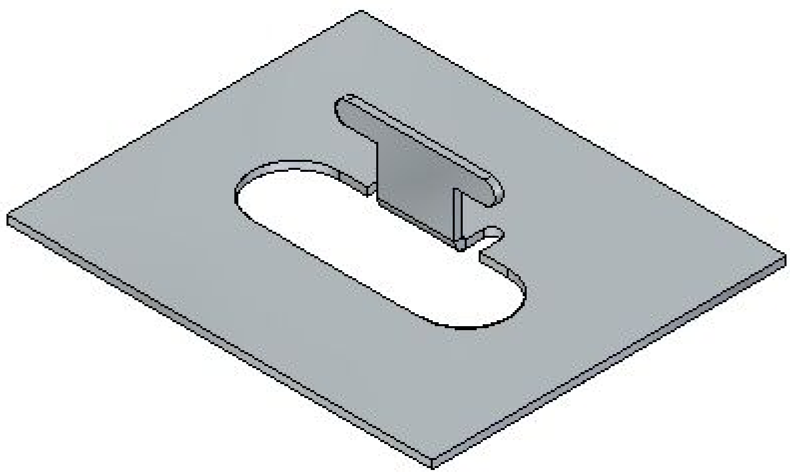
- Proceed to the next step.

Editing a bend

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



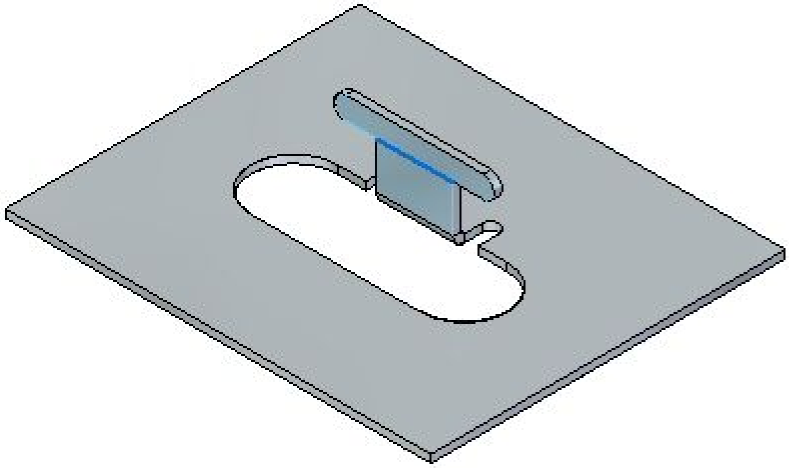
- Click the Options button and set the relief to be round with a width of 5.00 mm and a depth of 5.00 mm.



Note

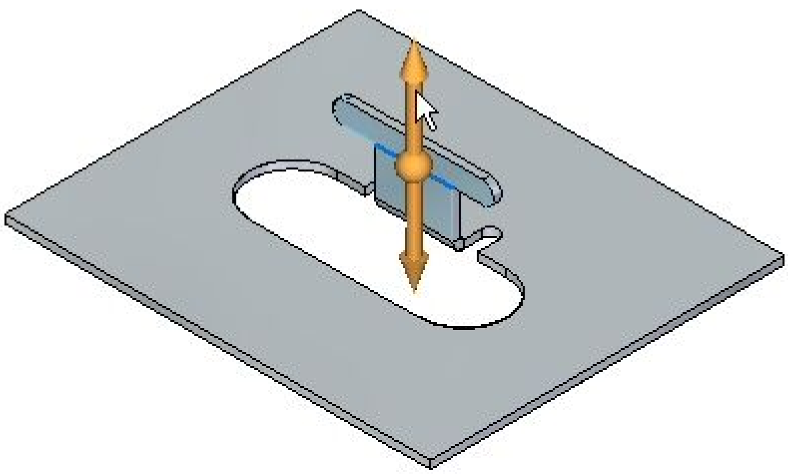
The relief could have been set during the creation of the bend in the previous step. The purpose of changing the relief at this point is to demonstrate the ability to edit a previously placed feature.

- Sketch a line as shown across the tab.

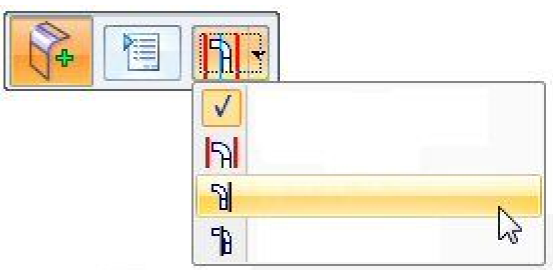


- Click the Bend command.

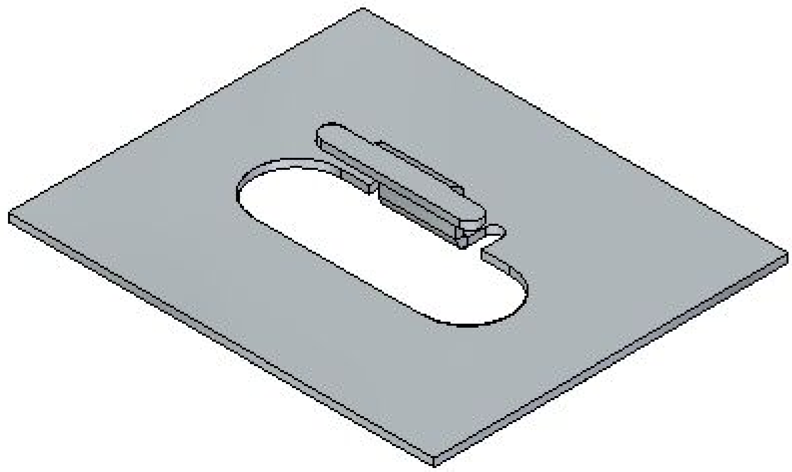
- Select the line as shown.



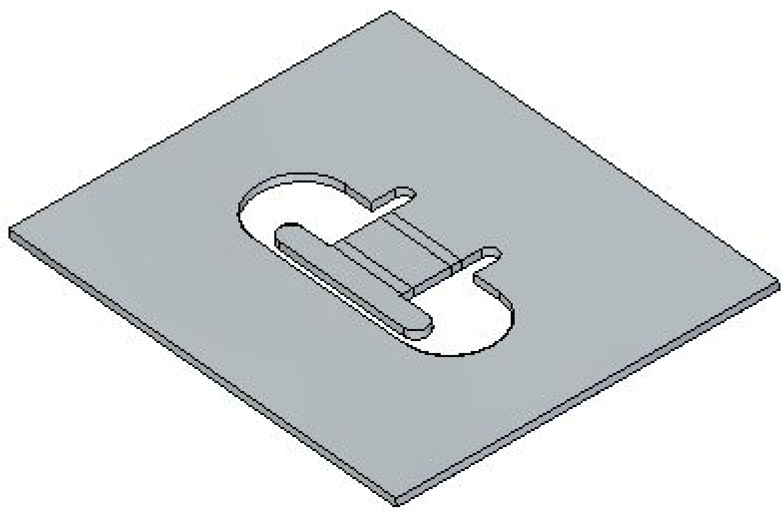
- Select the Material Inside option.



- 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.

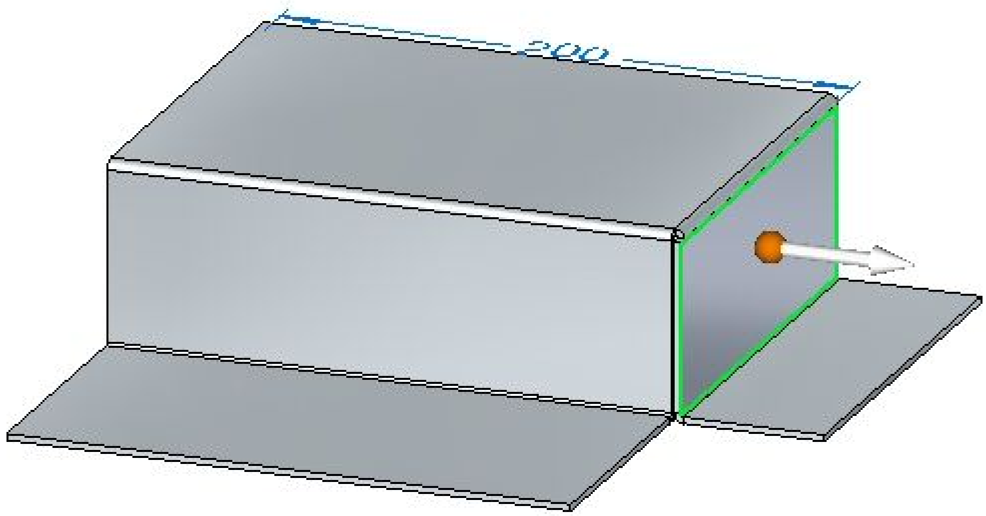
Note

Creating a flat pattern will be covered in another activity.

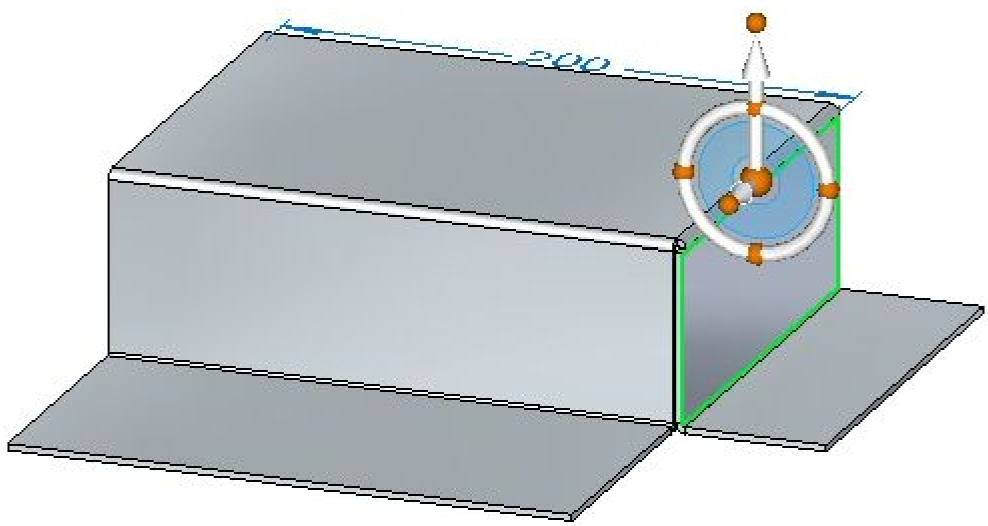
- Close the file without saving. Proceed to the next step.

Moving faces

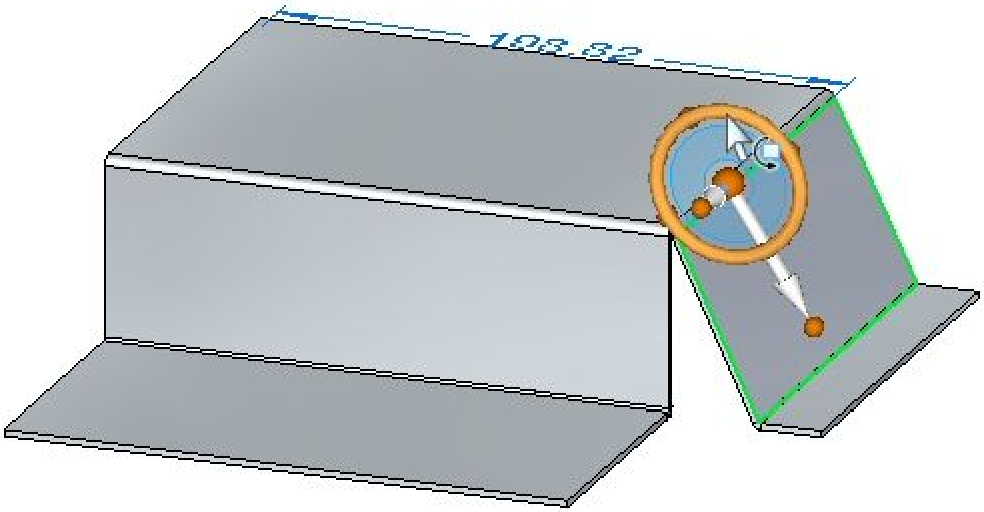
- Click the  **Application** button ® **Open** ® *move_activity.psm*.
- Select the face shown.



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



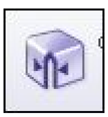
- Select the steering wheel torus and rotate the flange by an angle of 25º as shown.



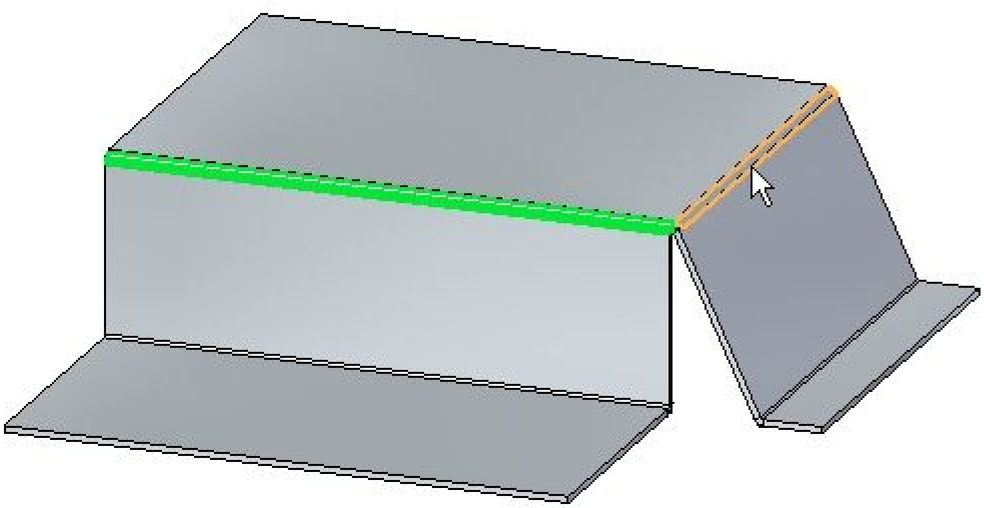
Note

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

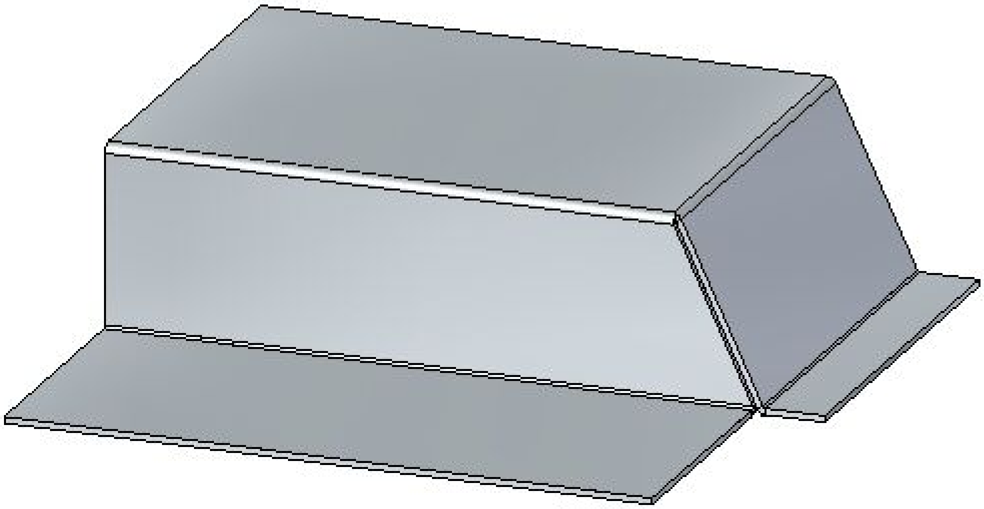
- Click the 2–Bend corner command.



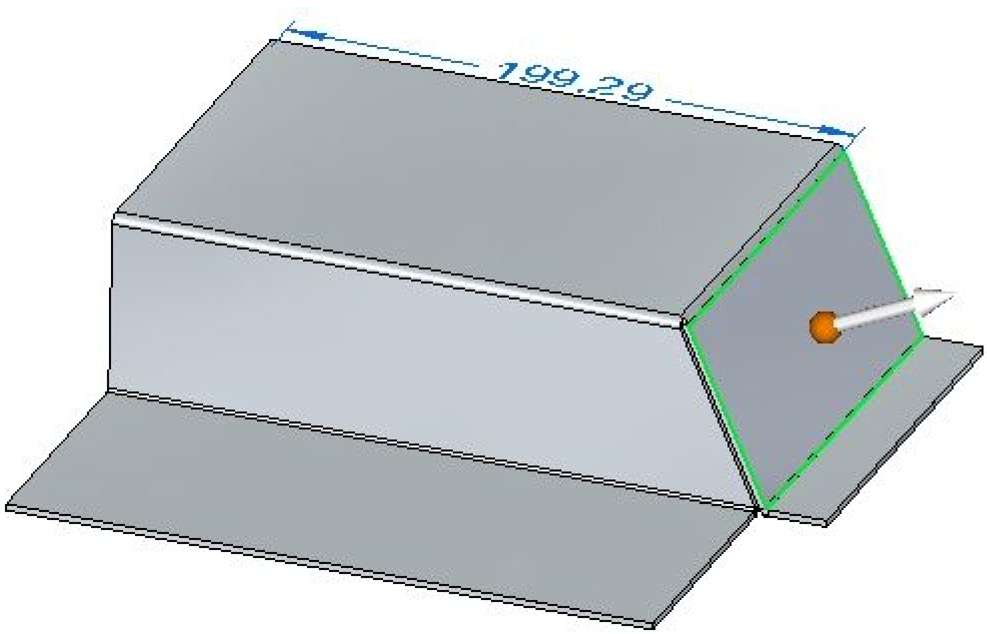
- Select the two bends shown.



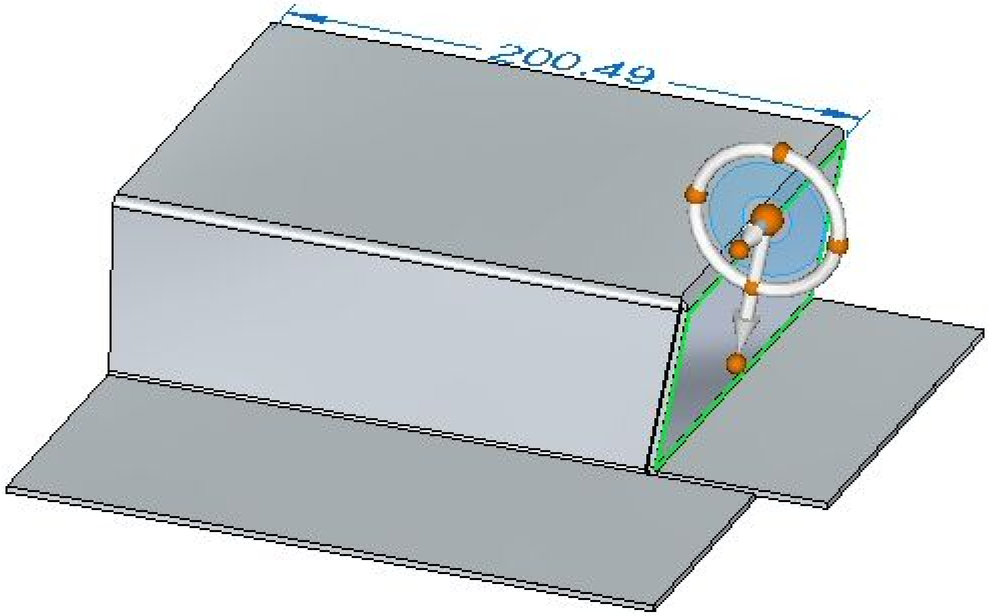
- The corner is closed.



- Select the face shown.



- Select the steering wheel torus and rotate the flange by an angle of –35º 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.

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 =π * (BR + (NF * THK)) * BA / 180

   Where:

   PZL = Plastic Zone Length

   BR = Bend Radius

   NF = Neutral Factor

   THK = Material Thickness

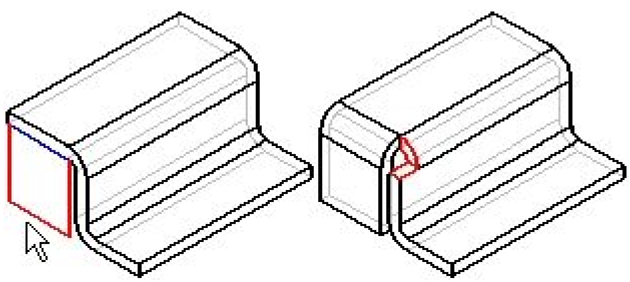
   BA = Bend Angle

   See the Help Topic entitled Sheet Metal Bend Formulas.

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

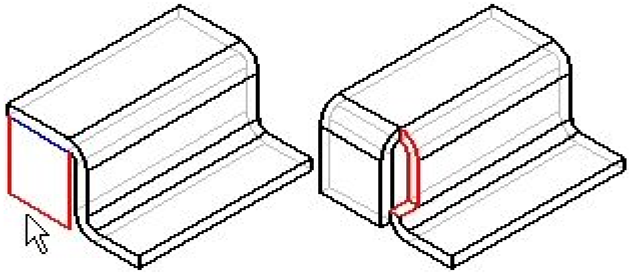
   Bend Only

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

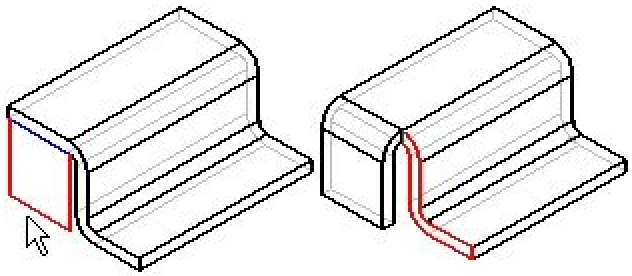
   Bend and Face

   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

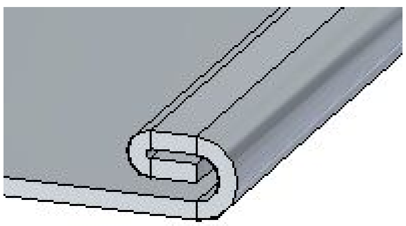
In this lesson you placed flanges and partial flanges. You edited end and corner treatments for bends. You used the closed the corners of adjacent thickness faces at the intersection of two bends. Bends were placed on the layer face and flanges were created and edited from these bends.

Lesson

7 Hem

Constructing a hem in a sheet metal part

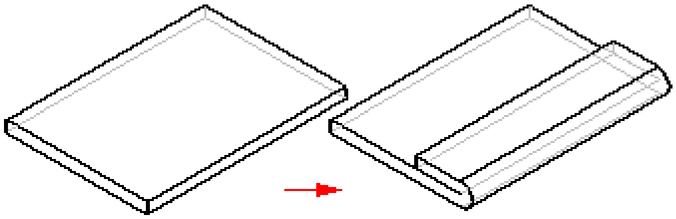
A hem feature creates a rigid edge for a sheet metal part. Modeling hems can be as easy as selecting the edges where you want to place them.



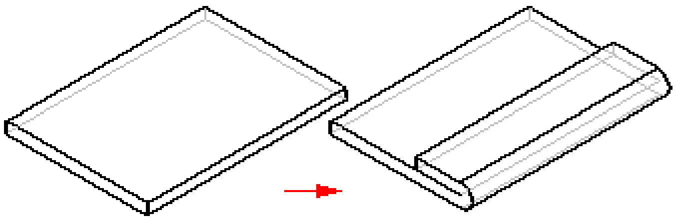
Hem command

Constructs a hem, where the material folds back.

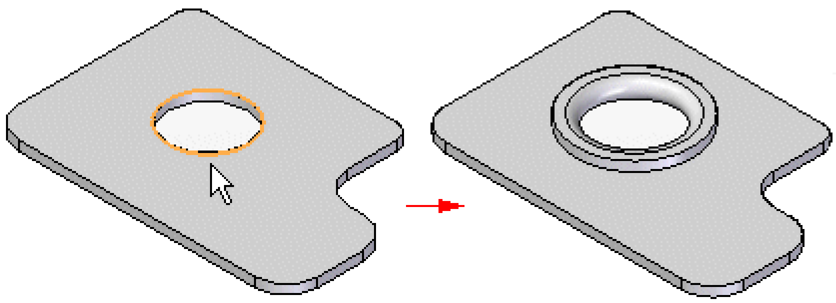
In the synchronous environment, you can construct a hem along a linear edge.



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



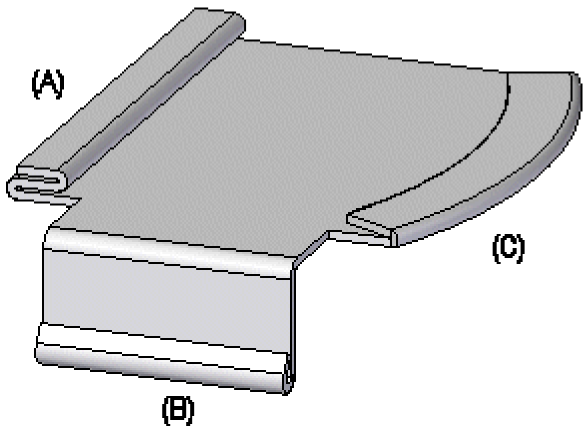
or, along the curved edge of a circular cutout.



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.

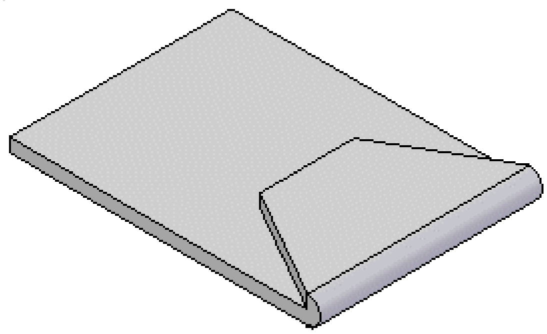
Round

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

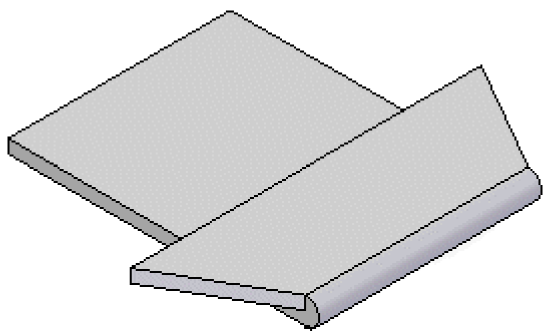
Angle

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

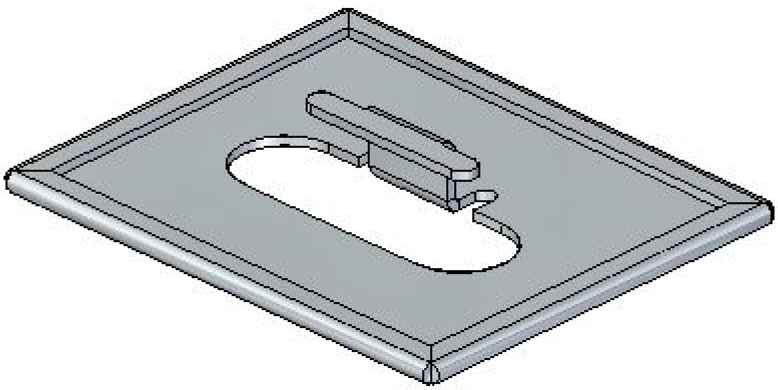
Specifies the neutral factor for the bend.

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.

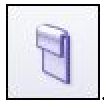


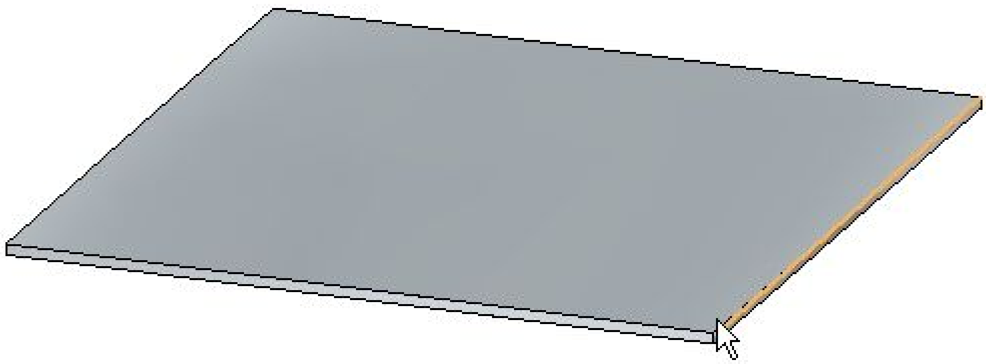
Activity: Using the hem command in sheet metal design

Open a sheet metal file

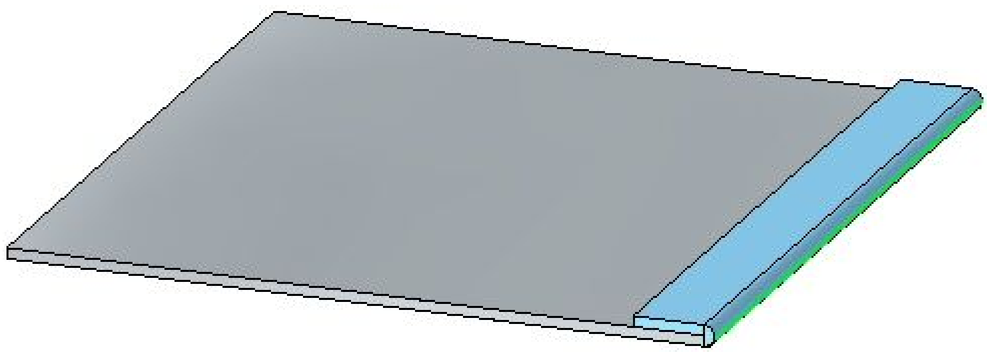
- Start Solid Edge ST4.
- Click the **Application** button ® **Open** ® *hem_activity_1.psm*.
- Proceed to the next step.

Create a hem on a single edge

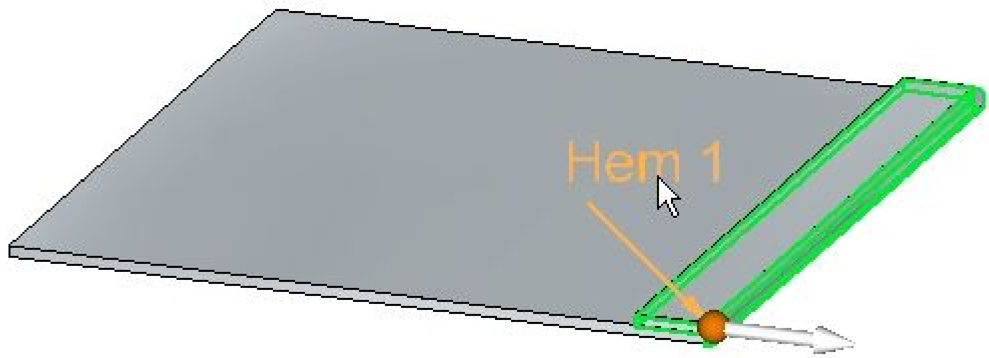
- Click the Hem command .
- Click the Hem Options button .
- Set the Hem Type to be Closed and Flange Length 1 to be 15.00 mm.
- Select the edge shown.



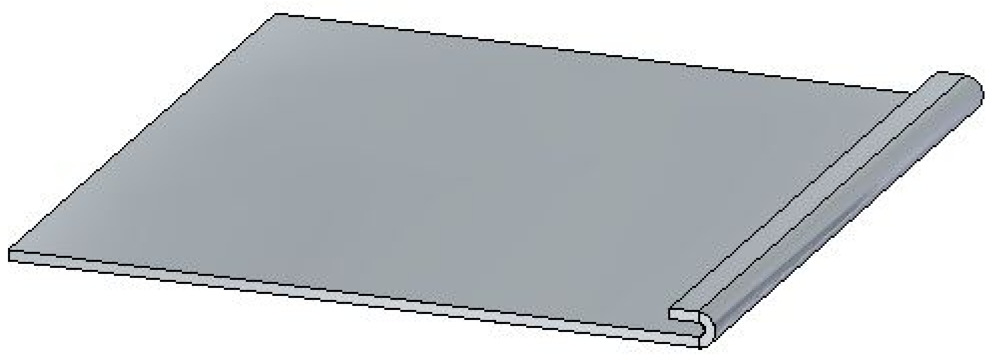
- Right-click to complete the hem shown.



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

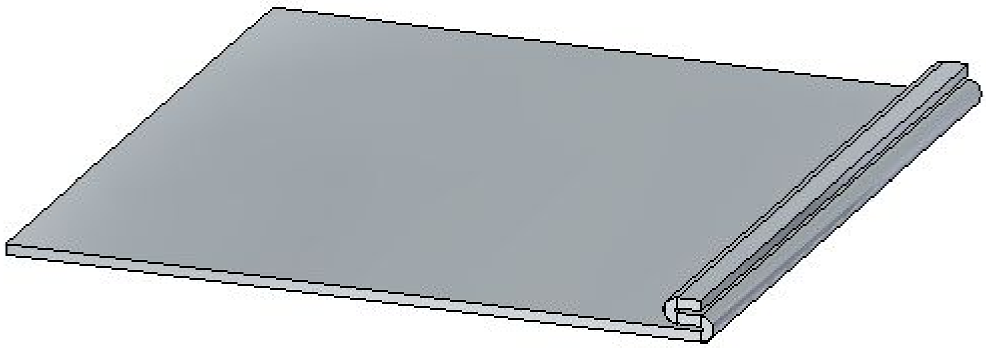


- Click the Hem Options button and set the Type to Open. Set Bend Radius 1 to 1.50 mm. Set Flange Length 1 to 6.00 mm and then click OK. The hem is modified as shown.

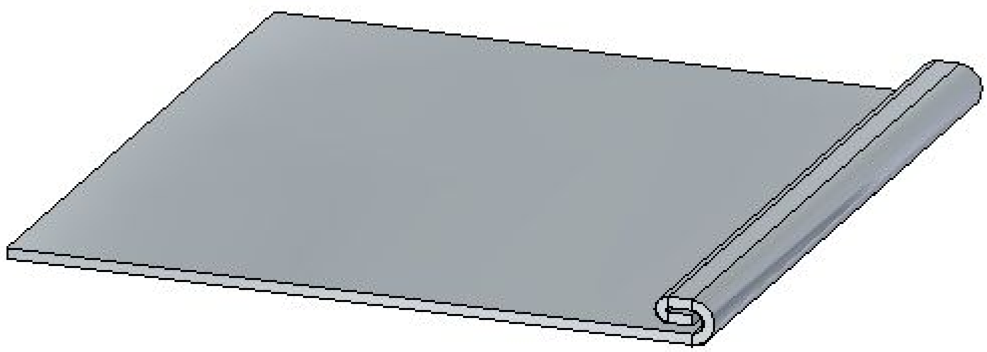


- Click the Select tool and then click the hem feature in PathFinder. Click the edit handle to change the settings on the hem.

- 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.

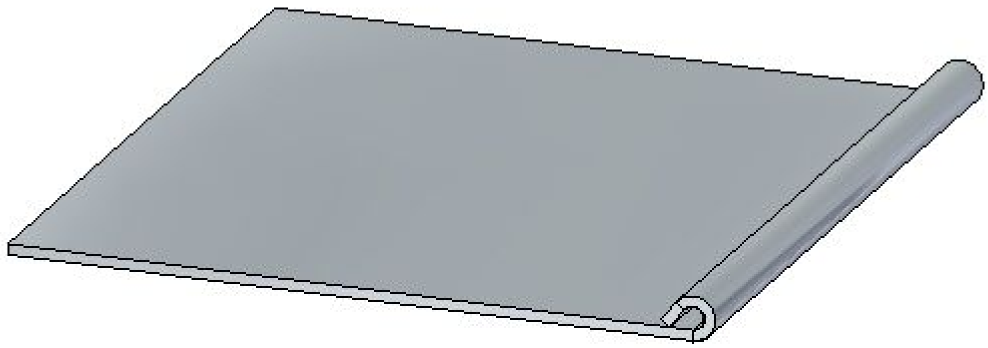


- Click the Select tool and then click the hem feature in PathFinder. Click the edit handle to change the settings on the hem.
- Click the Hem Options button and set the Type to Curl. Set Flange Length 1 to 11.25 mm and keep the existing values for the remaining lengths and radii, then click OK. The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



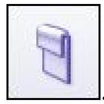
- Click the Select tool and then click the hem feature in PathFinder. Click the edit handle to change the settings on the hem.

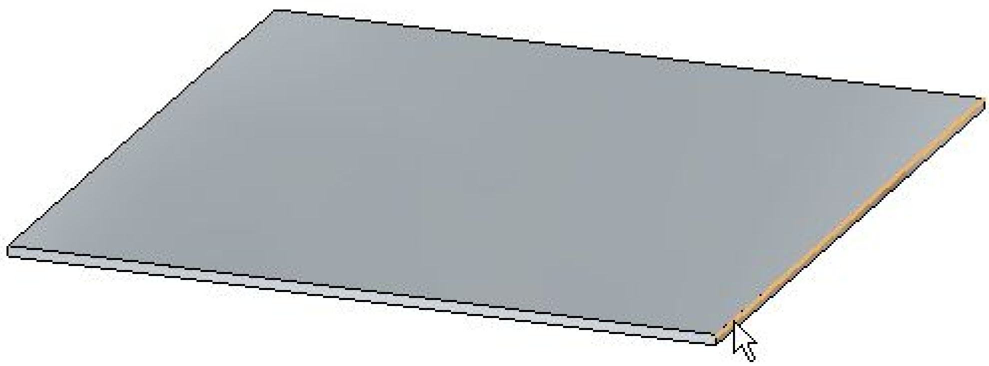
- Click the Hem Options button and set the Type to Closed Loop. Keep the default values for bend radii and flange lengths, then click OK. The hem is modified as shown.



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

Create a hem on multiple adjacent edges

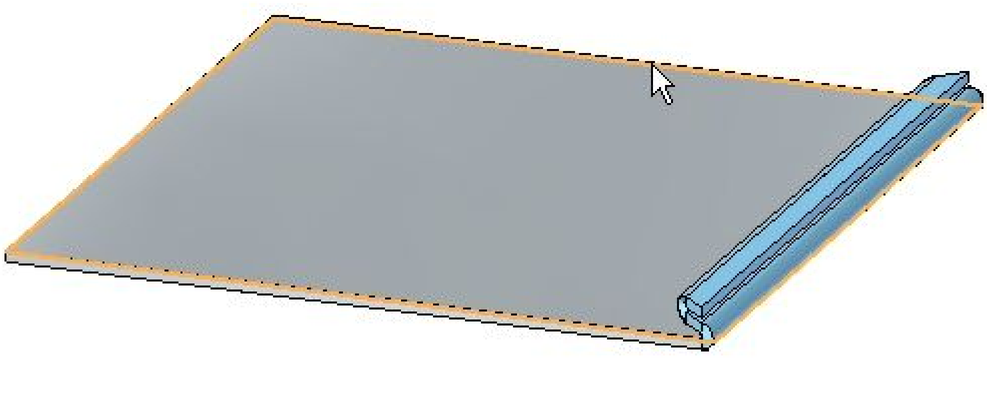
- Click the  **Application** button ® **Open** ® *hem_activity_2.psm*.
- Click the Hem command .
- Click the Hem Options button .
- Set Type to S-Flange.
- Check the Miter Hem option and ensure the Miter Angle is –45º.
- Select the edge shown.



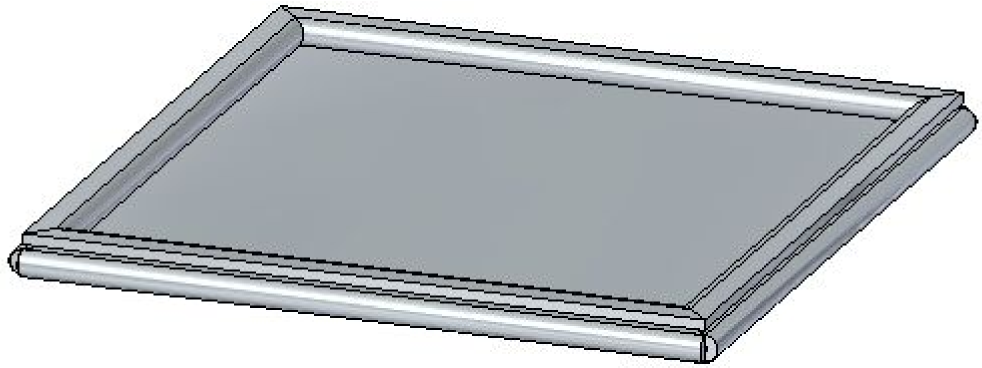
- Select the edge set shown.

  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.



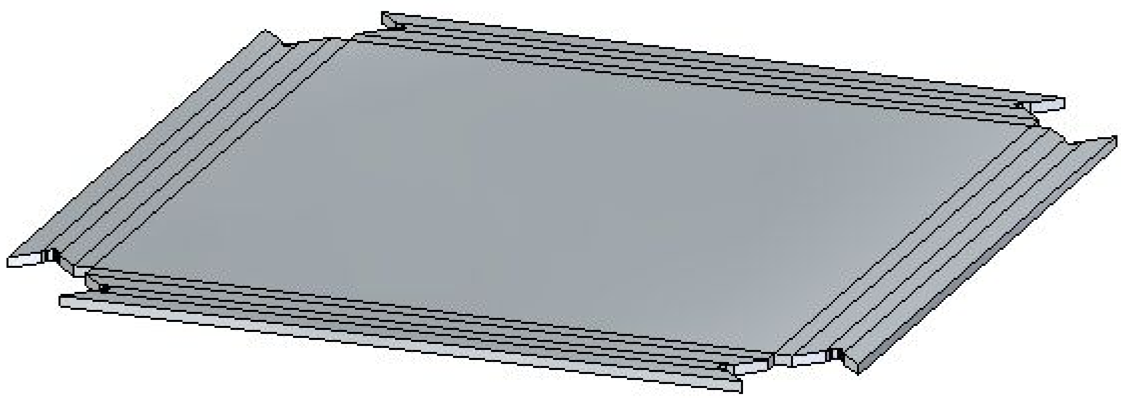
- Right-click to accept the hem.



- 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.

Lesson review

Answer the following questions:

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

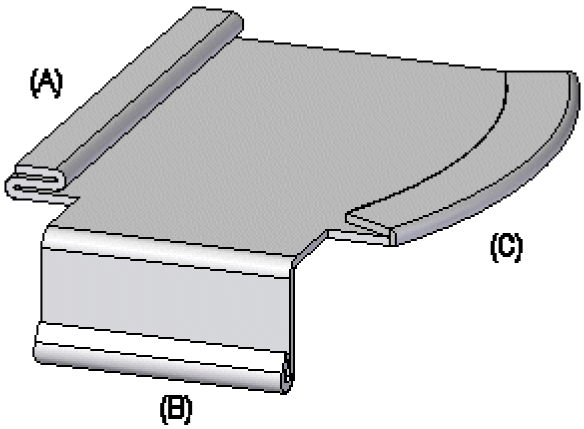
2. What is the difference between using a positive value versus a negative value in defining a hem mitre?

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

Answers

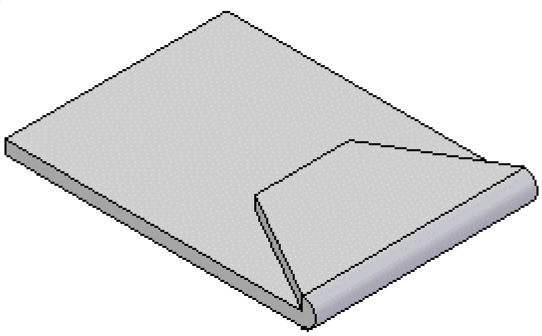
1. Using hem options, define the three types of hems that can be created.

   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.

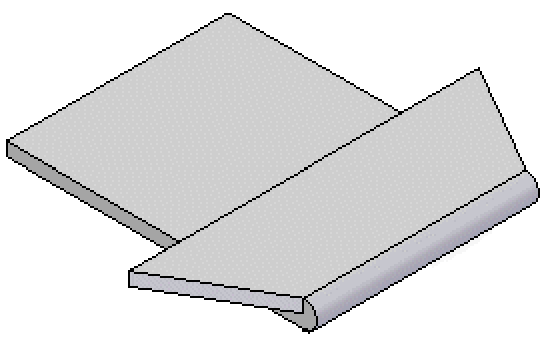


2. What is the difference between using a positive value versus a negative value in defining a hem mitre?

   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

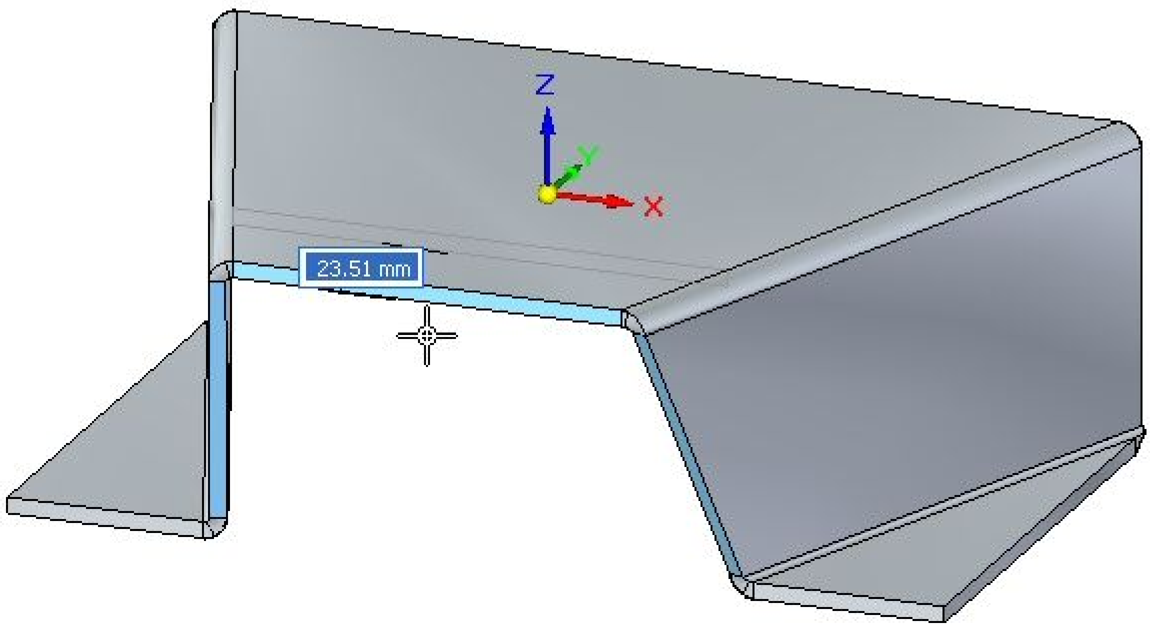
In this lesson you created a variety of hems in sheet metal parts. You learned how to set the parameters to create the hems, and how to edit the values when needed.

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:

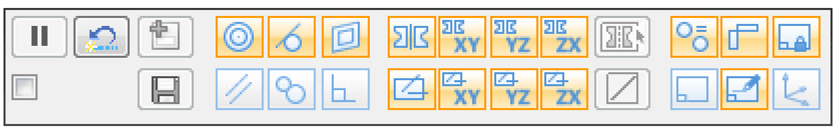
- Moving or rotating model faces or features in a synchronous part or assembly document.
- Defining 3D geometric relationships between model faces using the Relate command in a synchronous part document.
- Editing the dimensional value of a 3D PMI dimension in a synchronous part or assembly document.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

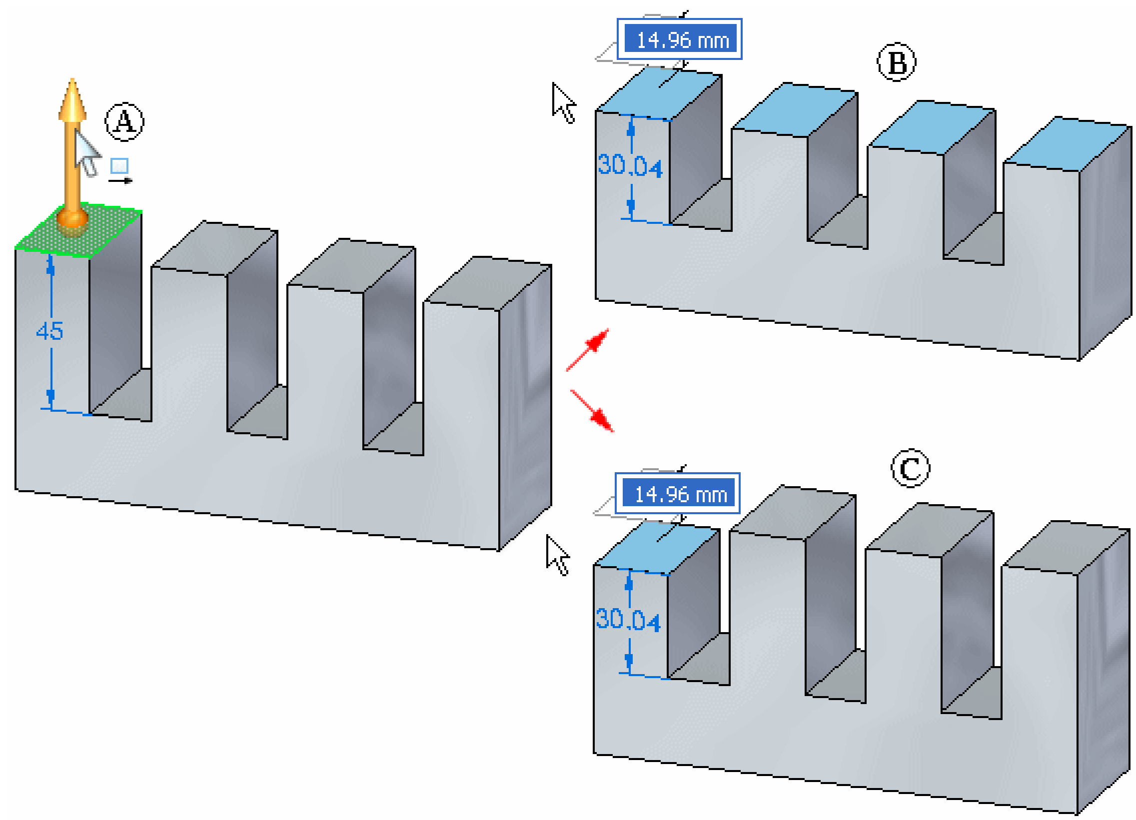
Live Rules options

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.

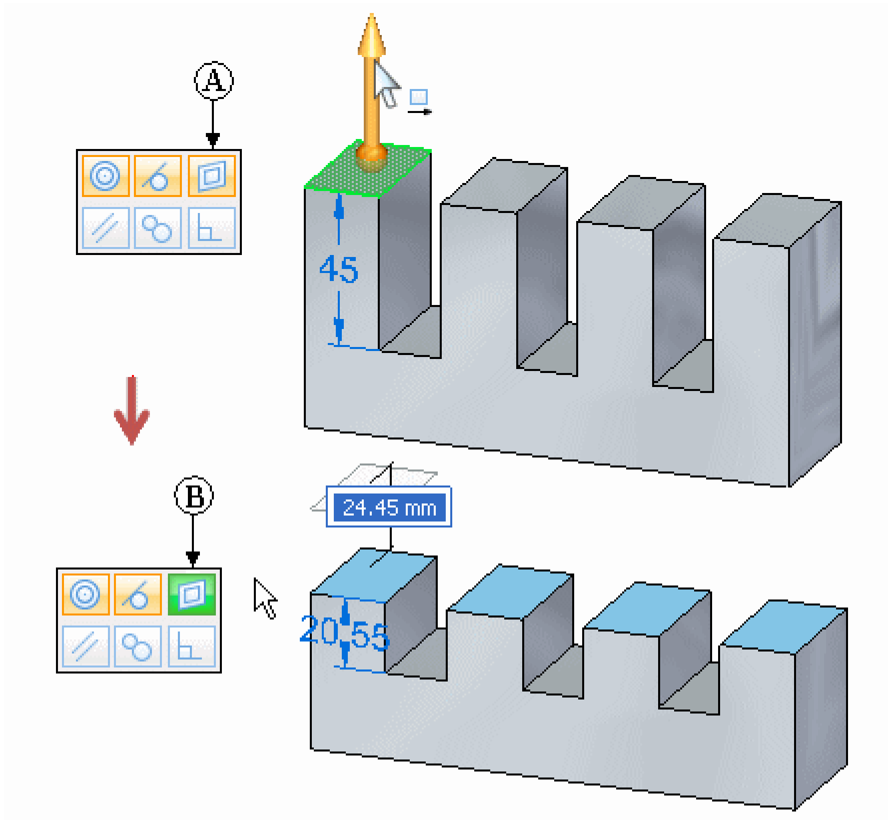
In this example, when the Coplanar option in Live Rules is set, the deselected coplanar faces stay coplanar (B) when moving the selected face. When the Coplanar option in Live Rules is off, the deselected coplanar faces remain stationary (C) when moving the selected face.



Relationship detection indicators in Live Rules

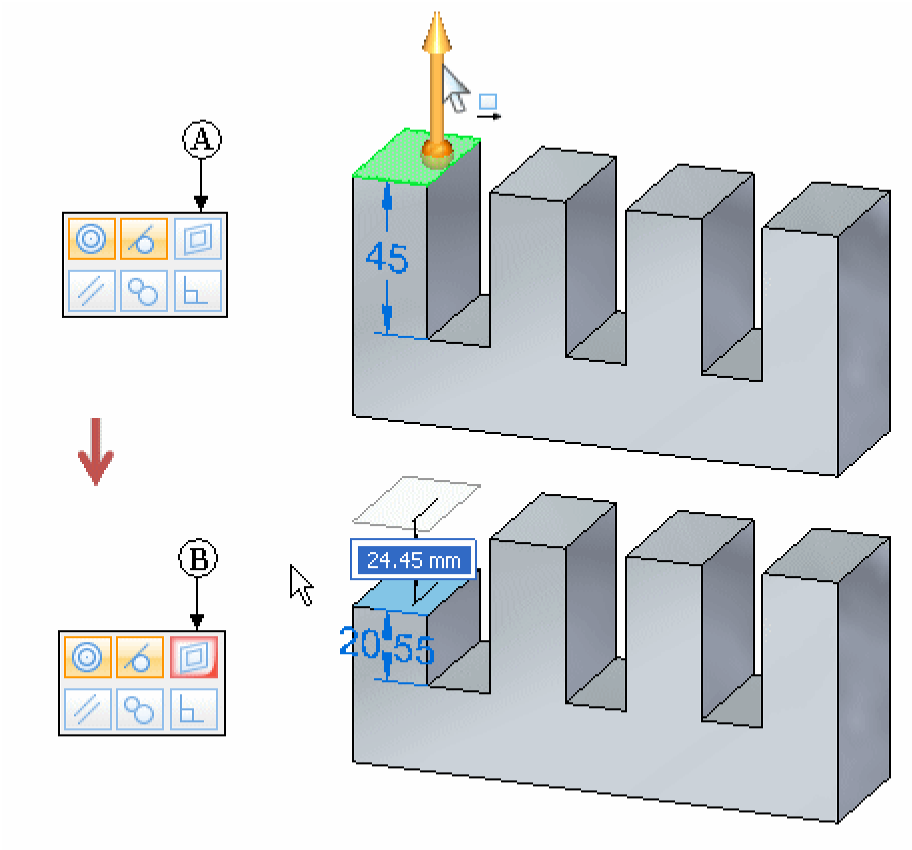
Detected and active

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).



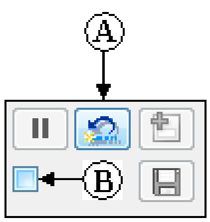
Detected and inactive

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.

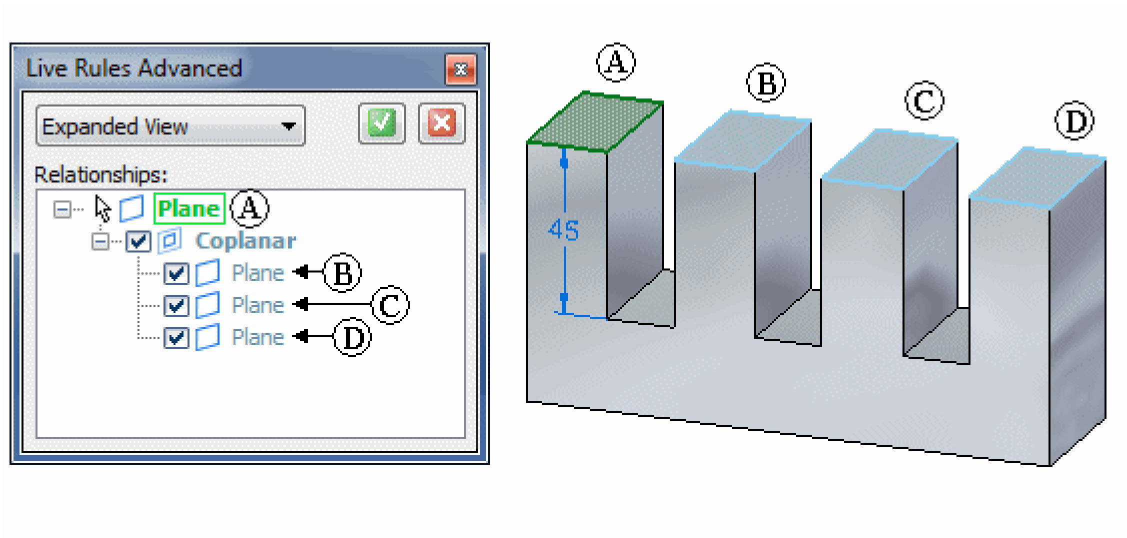
To show the Advanced Live Rules dialog, click the Advanced button , or press Ctrl+E.

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.

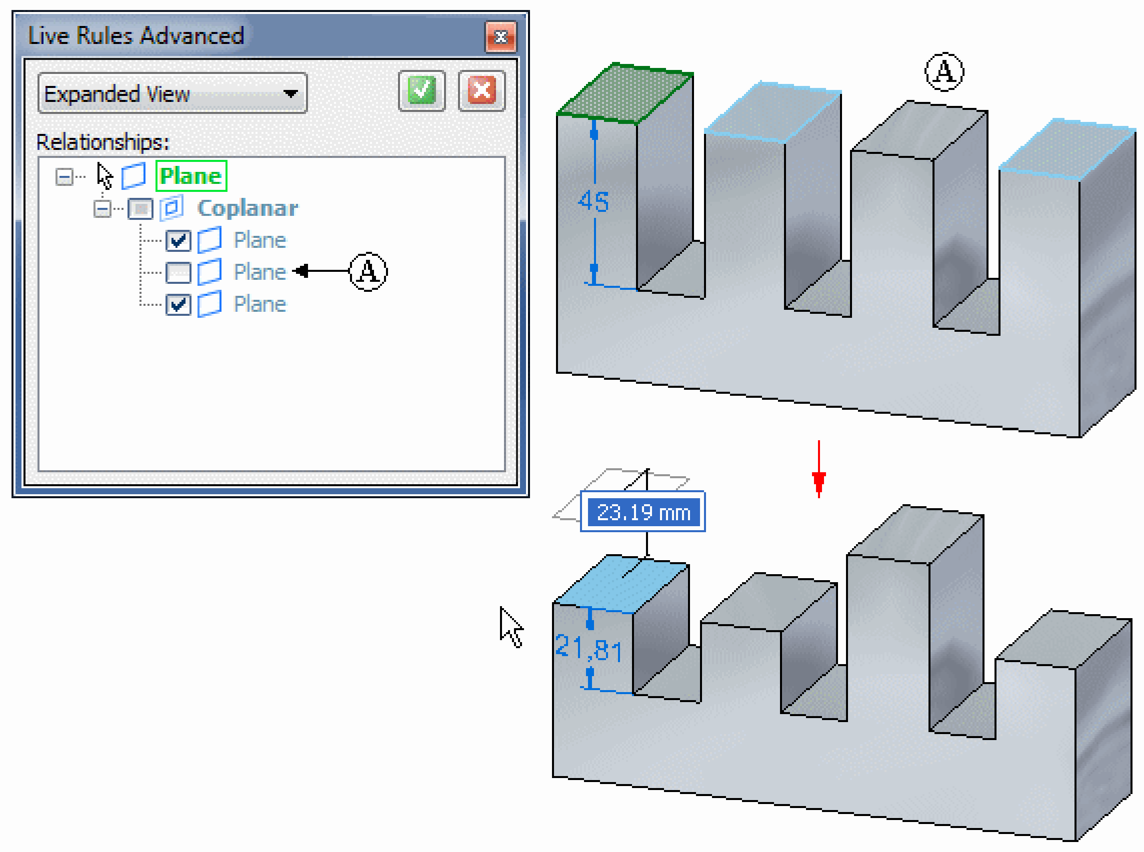
Also notice that the deselected faces are indented underneath the selected face and are grouped under a relationship heading, in this example Coplanar.



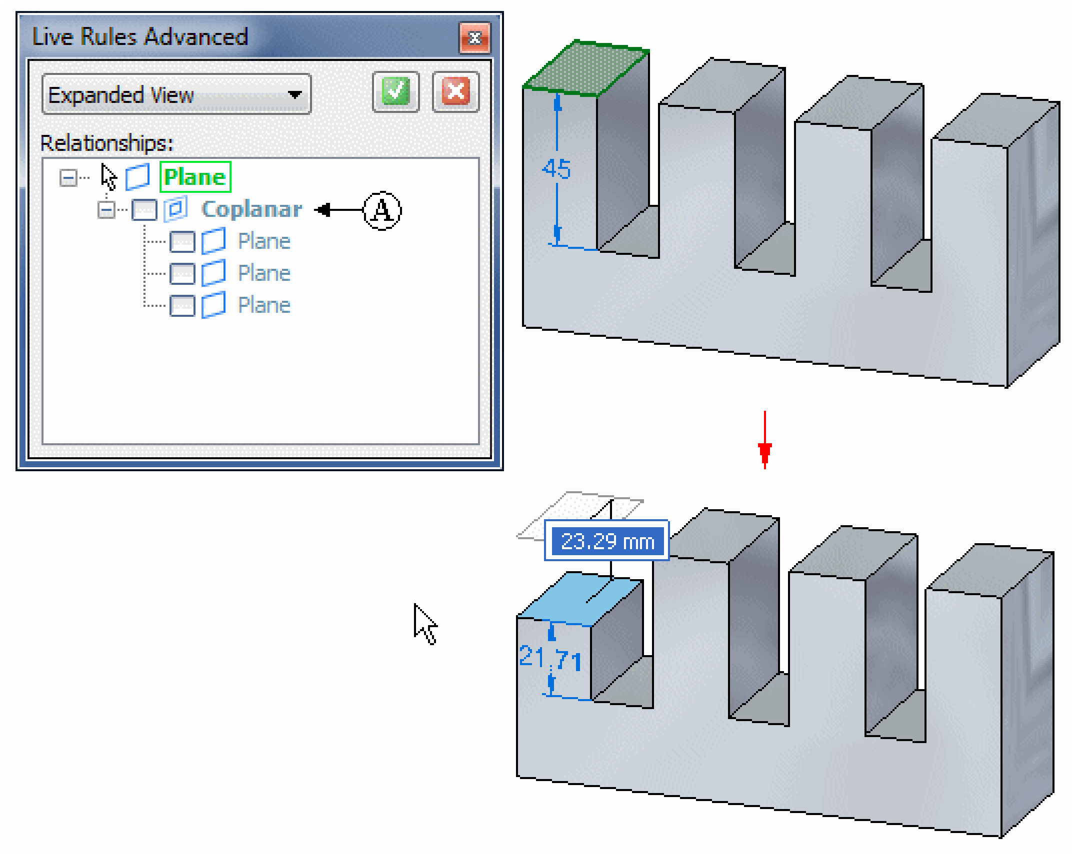
Using the Advanced Live Rules dialog

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.



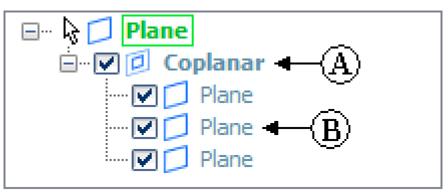
You can also clear the check box adjacent to the relationship heading (A) to specify you do not want any of the deselected coplanar faces to remain coplanar when you move the selected face.



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

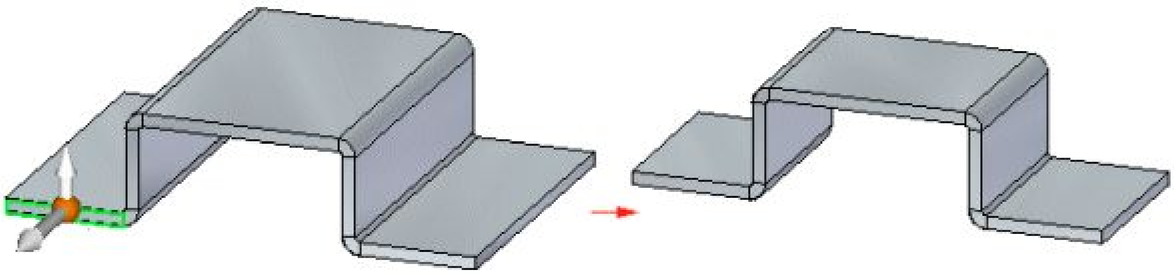
You can use the Select option on the Relationships shortcut menu to add the list of related elements under the selected relationship heading to the current select set.

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.

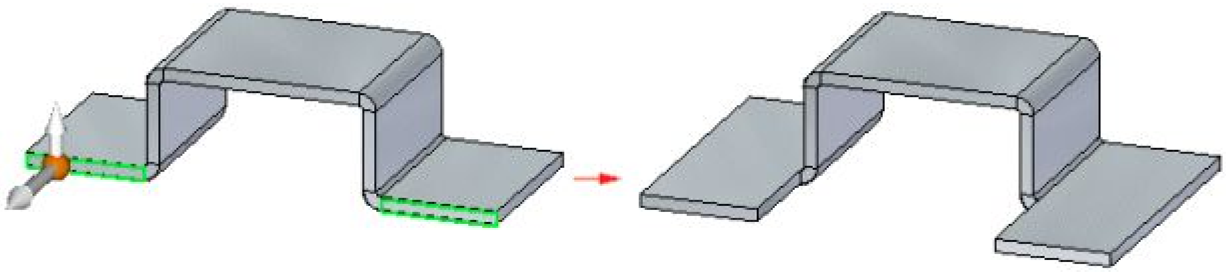
Live Rules in sheet metal modeling

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.

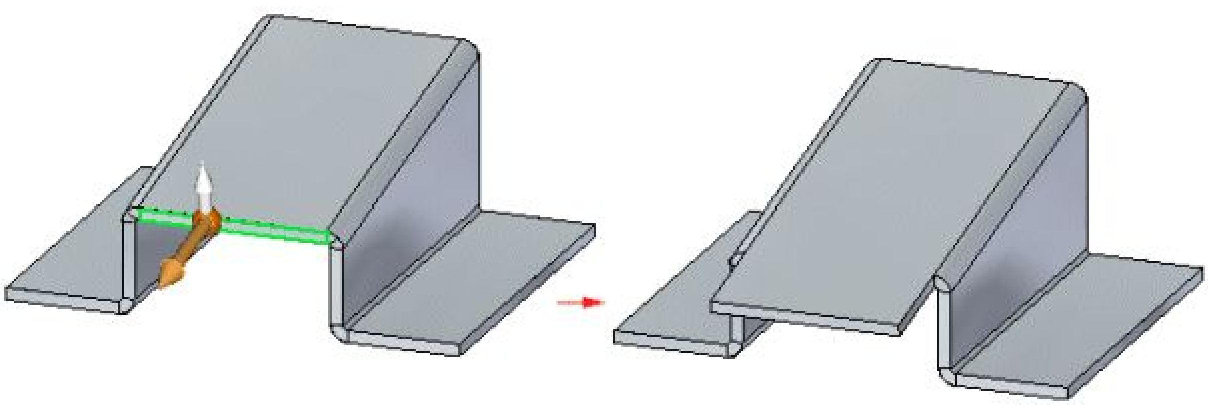


When the Thickness Chain option is not set, only the selected face or faces move.

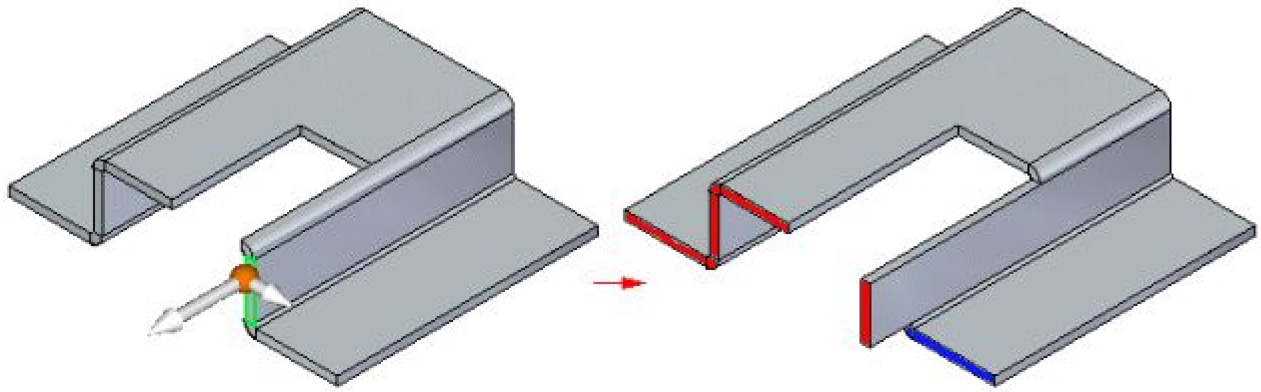


Selecting the Suspend Live Rules option does not affect the setting of the Thickness Chain option. In other words, if the Thickness Chain option is set and you select the Suspend Live Rules option, the Thickness Chain options remains set.

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.

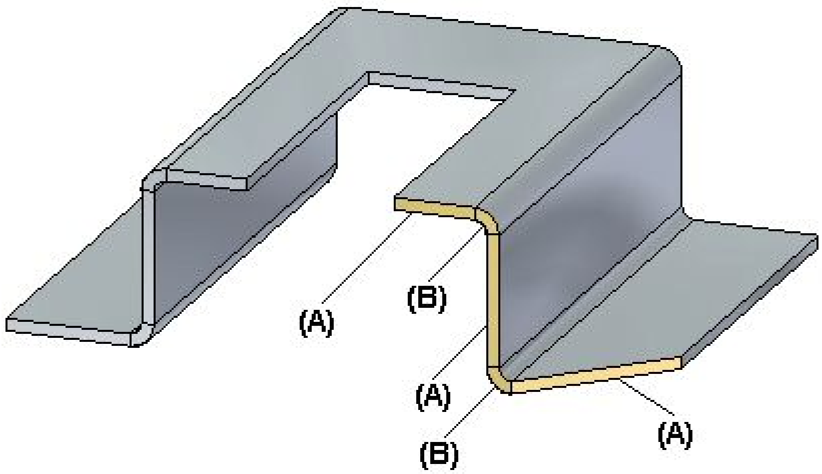


You may need to set the Orthogonal to Base if Possible option if you want to move or rotate a face that will cause a plate or thickness face to tip at an angle not orthogonal to the base reference plane.

Thickness chain

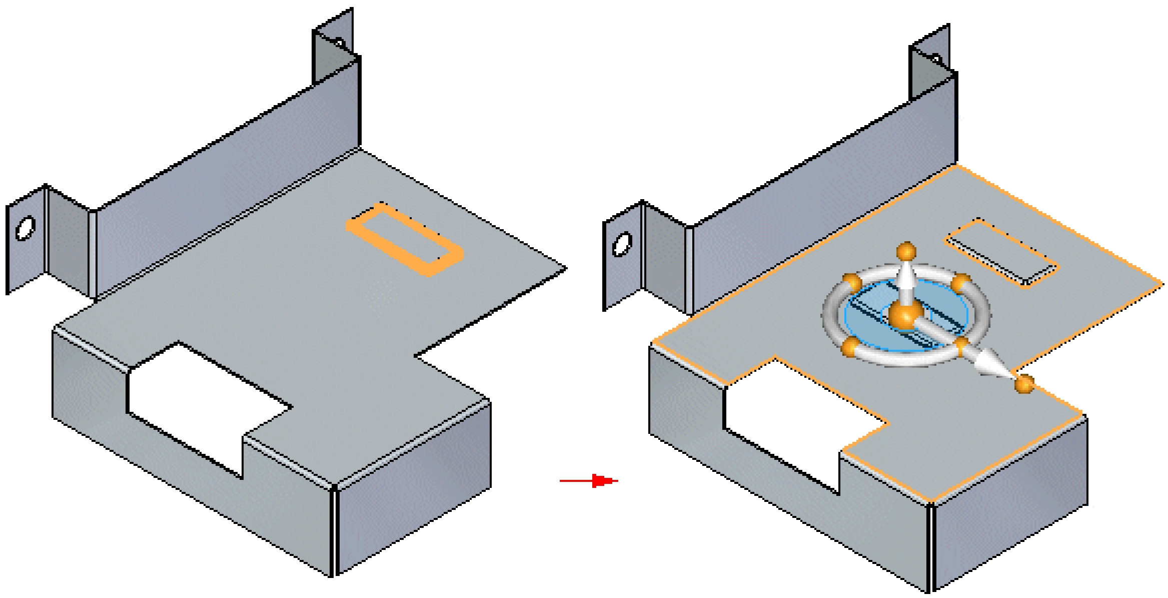
Thickness chain on a sheet metal part

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

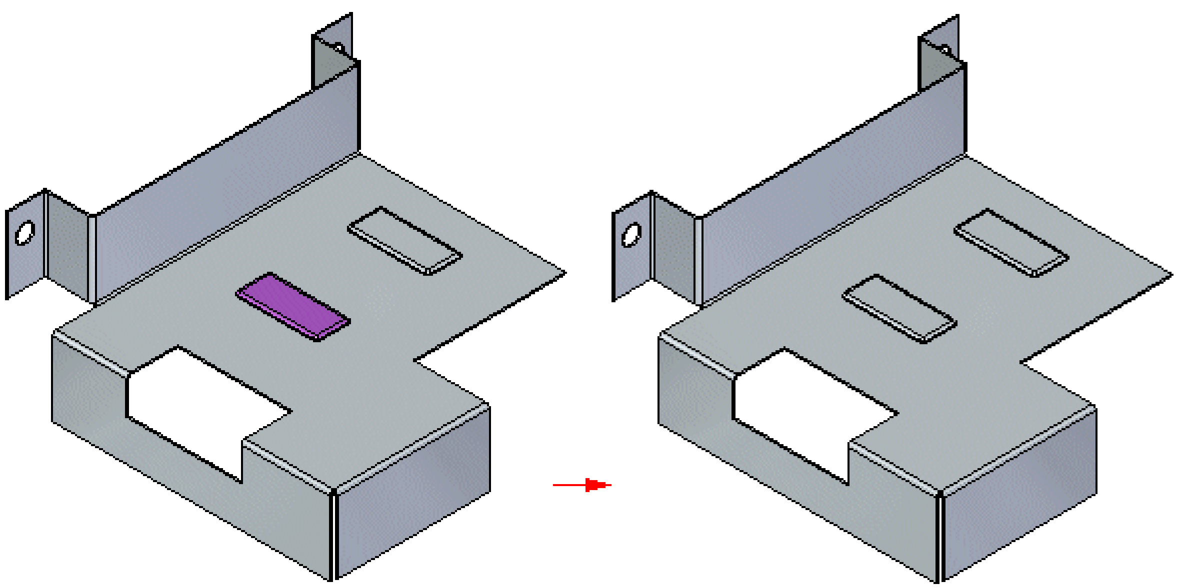


Copying, pasting, and attaching sheet metal features

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.

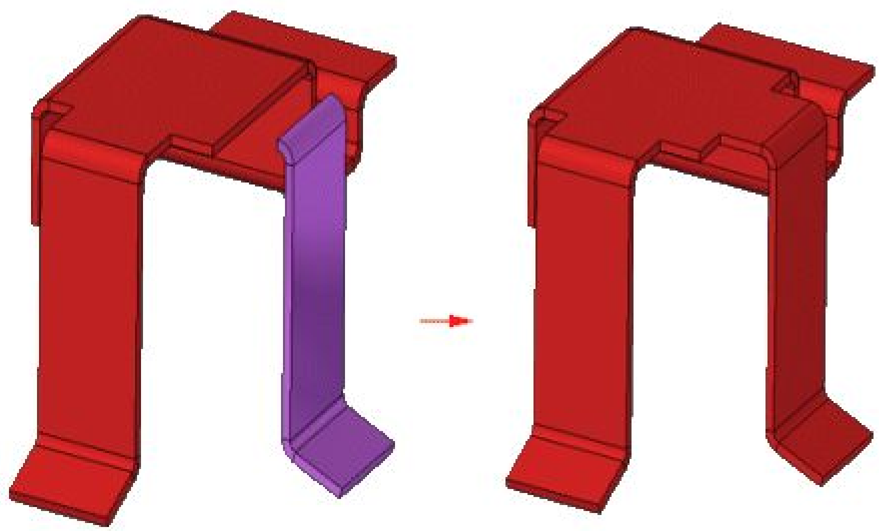
Pasting and attaching sheet metal features

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.

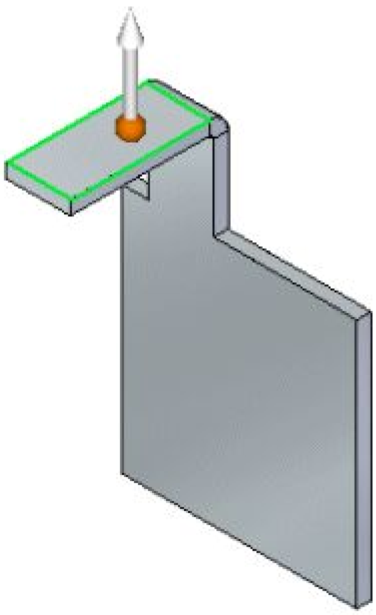
When using the Attach command you can use the extend to next and capping behavior to join the construction to the body for thickness faces.



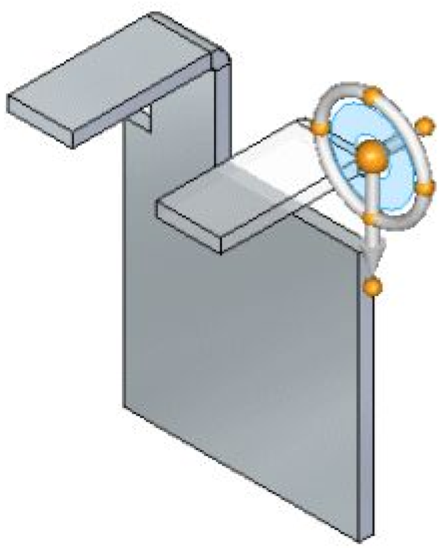
You cannot extend or trim manufactured features with the Attach command.

Copying, pasting, and attaching flanges

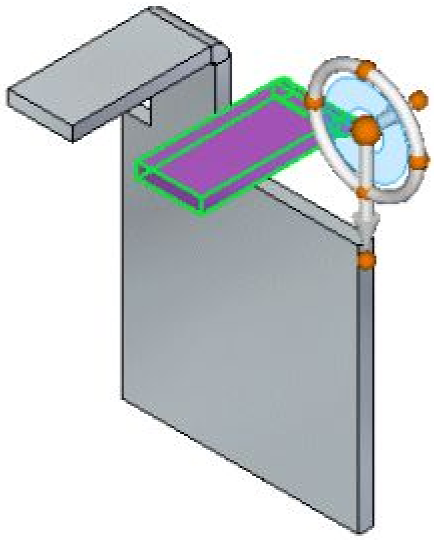
When you select a face of a flange to copy, all other faces in the flange are selected for copy.



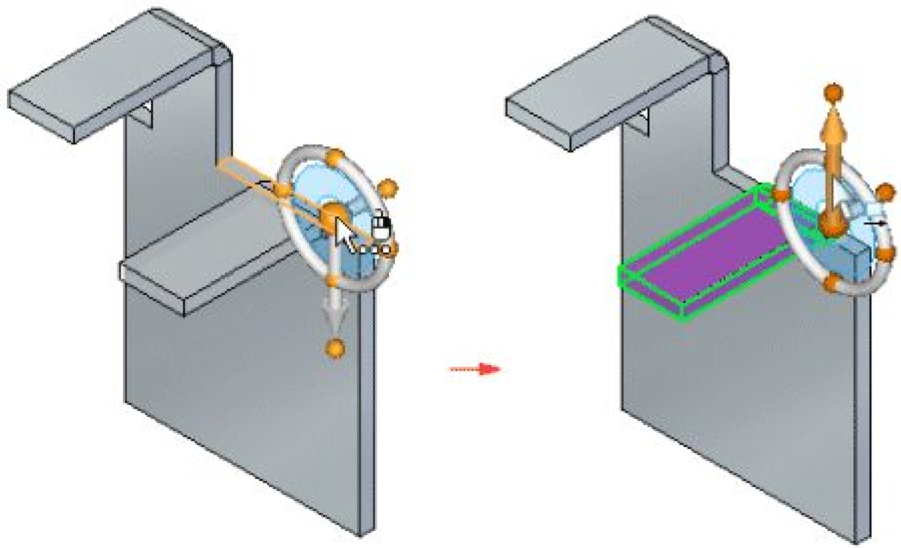
Once you copy the flange and then select the Paste command, a glass image of the flange attaches to the steering wheel.



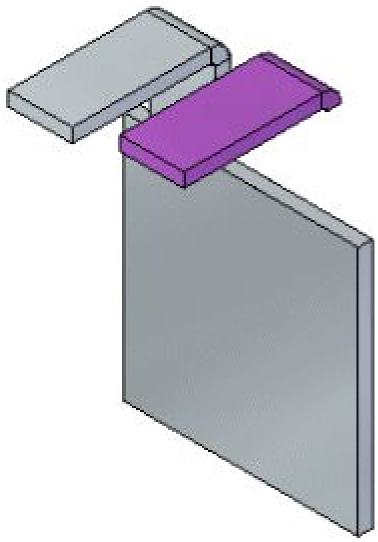
After you paste the flange, it appears as a detached flange.



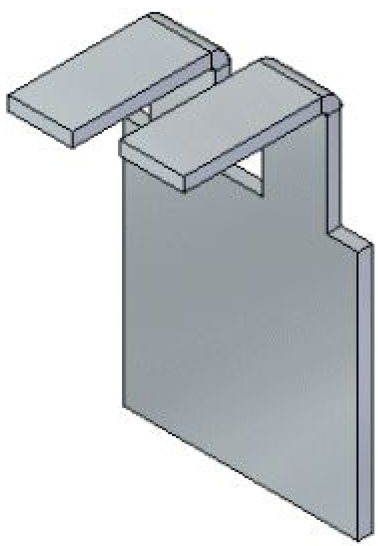
You can locate and lock to a plane and then use the steering wheel to adjust the position of the detached flange.



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.

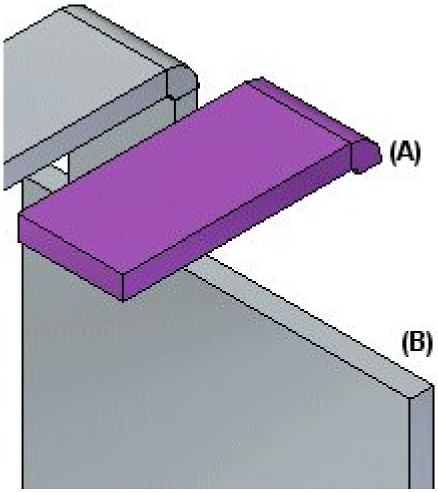


When the flange is attached to the main body, all other required faces are automatically created.

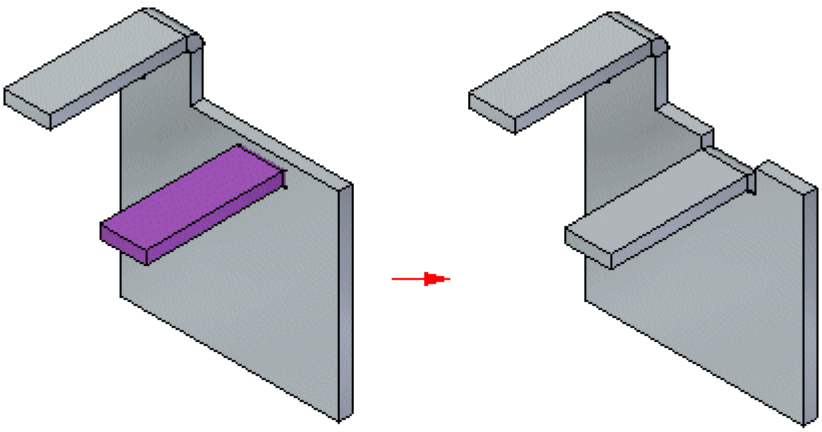


Note

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.

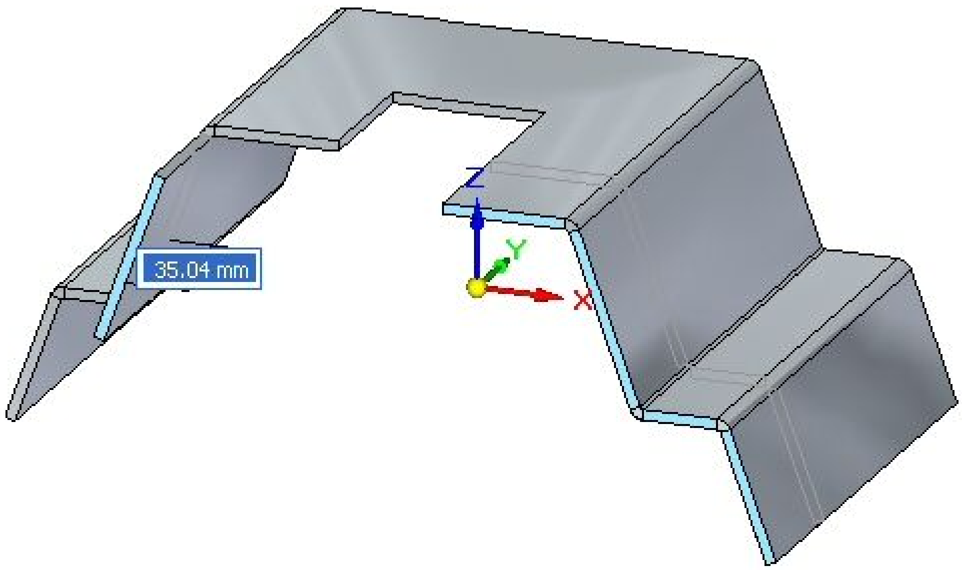


Activity: Using live rules in sheet metal

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.

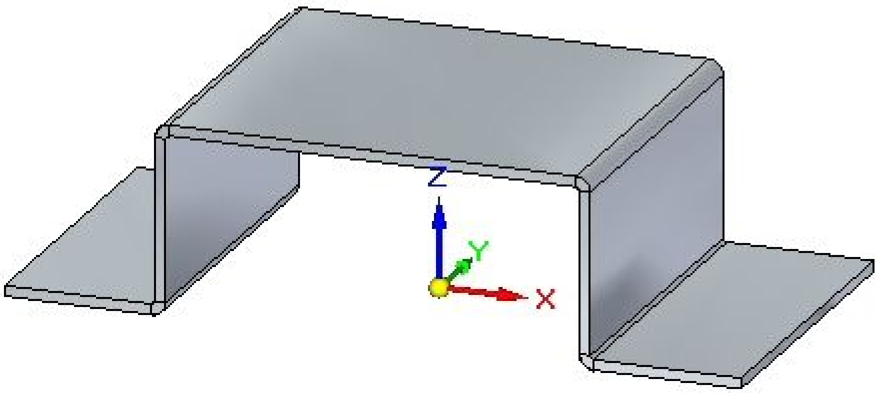


Activity: Using live rules in sheet metal

Open a sheet metal file

- Start Solid Edge ST4.

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



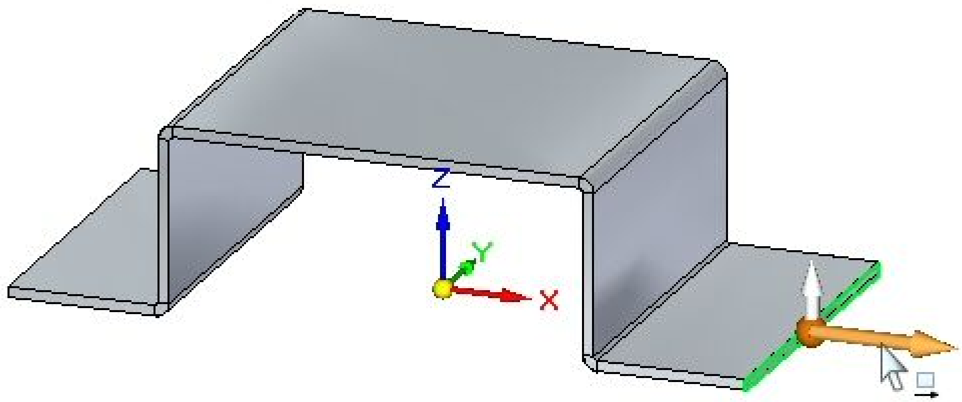
- Proceed to the next step.

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.

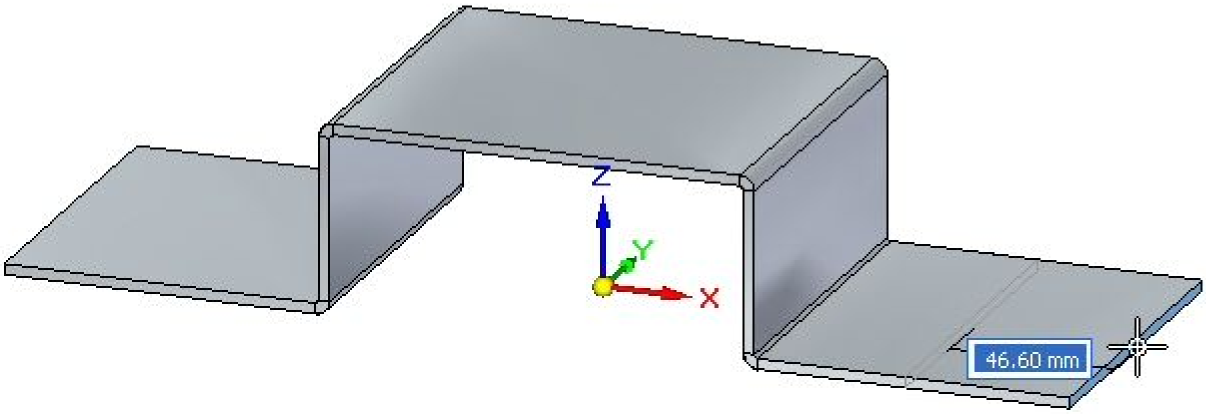


- Click the Live Rules Restore button.

Note

This resets the live rules to the default values.

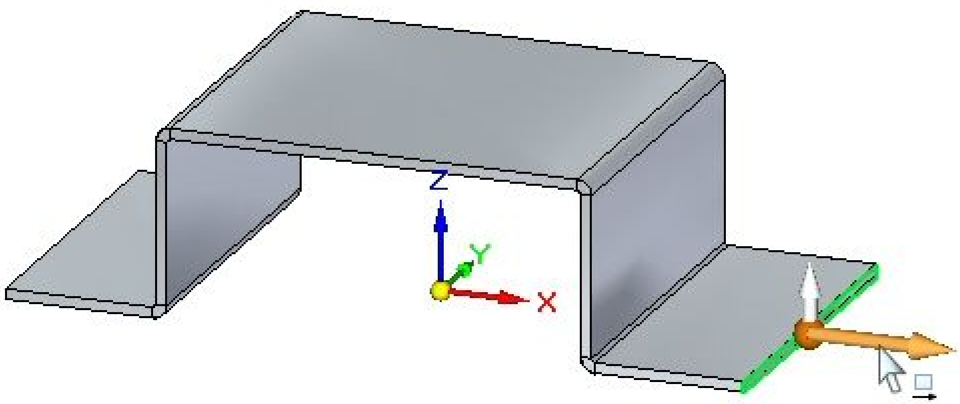
- Drag the handle as shown and observe the behavior, then press the **Esc** key.



Observations:

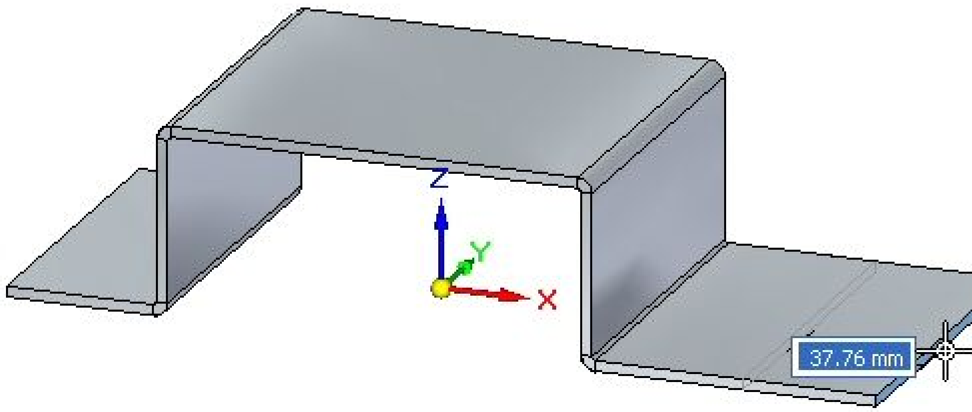
- The flange chosen is symmetrical to the opposite flange about the right YZ base reference plane. The live rule for symmetry is controlling the behavior of the opposite flange.

- 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:

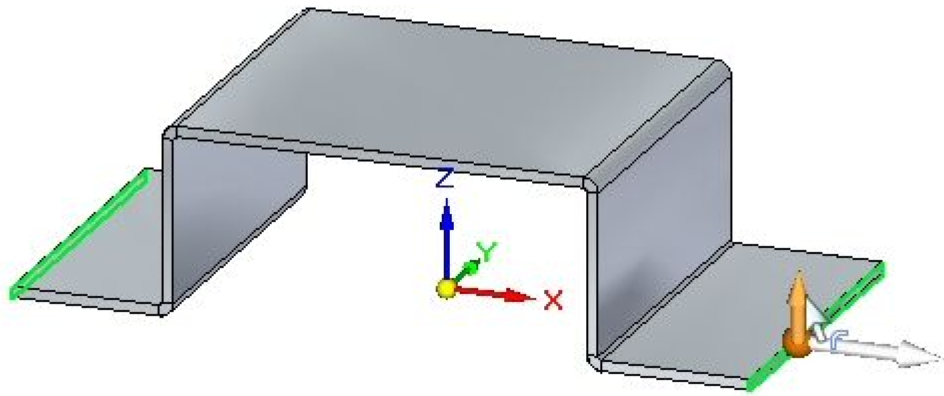
- The live rule controlling symmetry about the YZ plane is turned off and the moving the thickness face causes the flange to be modified independently.

- 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.

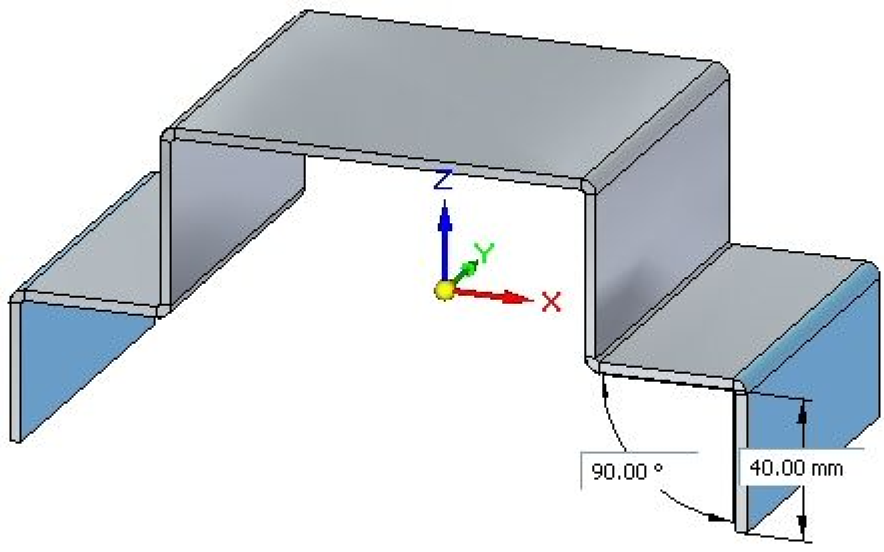


- Click the Live Rules Restore button.

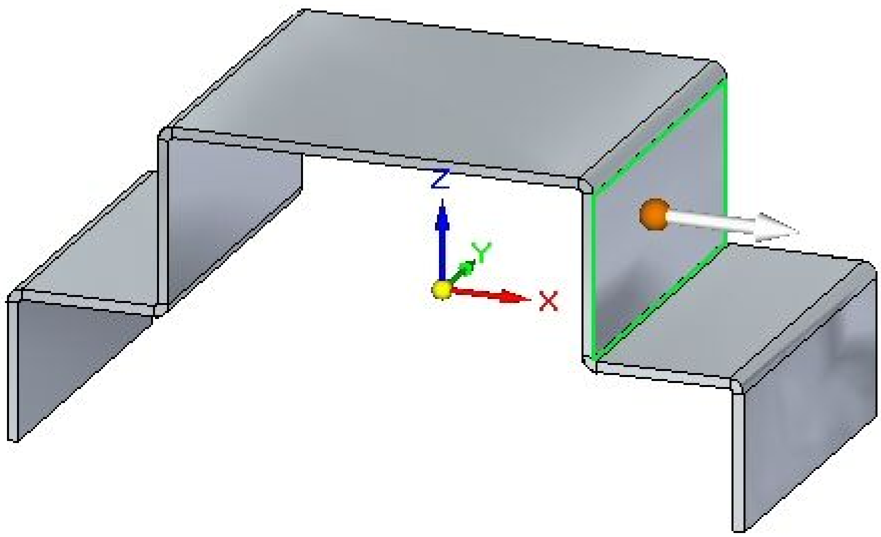
Note

This resets the live rules to the default values.

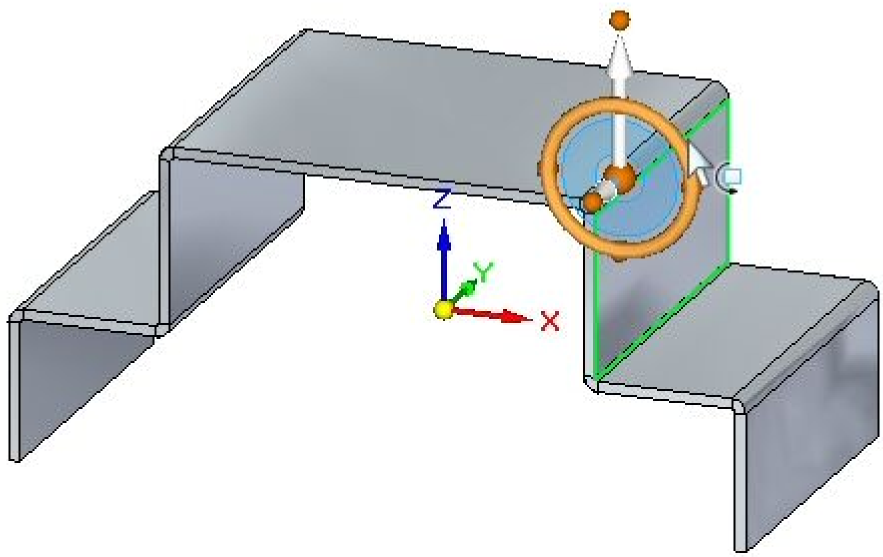
- Drag the flange start handle a distance of 40.00 mm as shown.



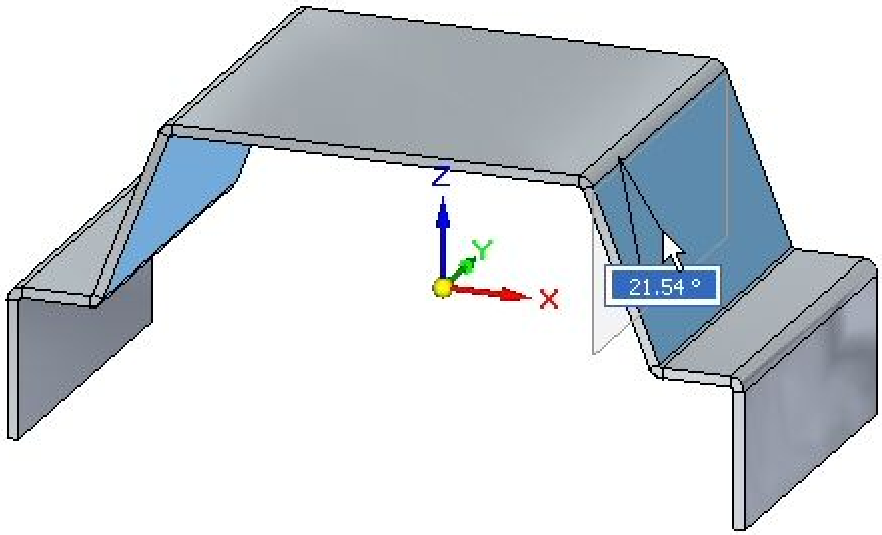
- Select the face shown.



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



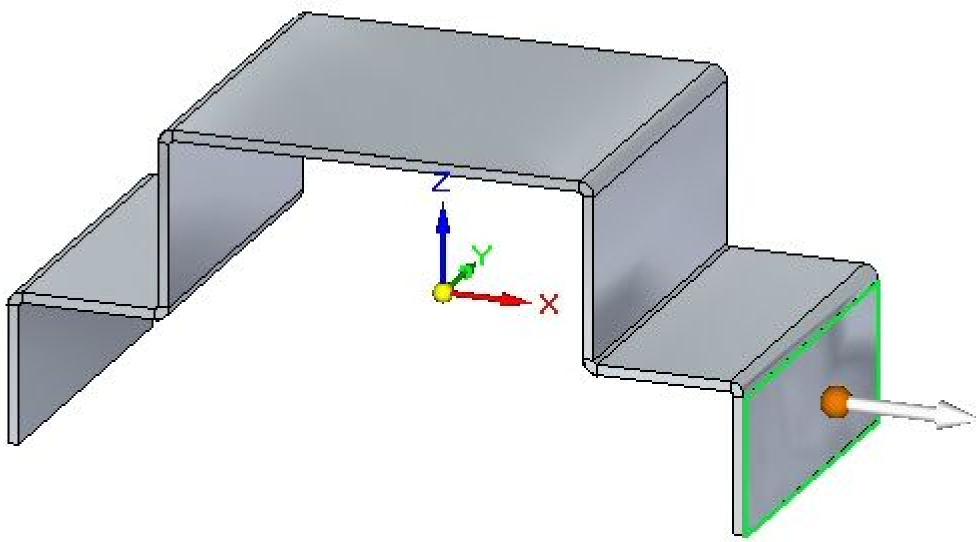
- Rotate the face and observe the behavior, then press the **Esc** key.



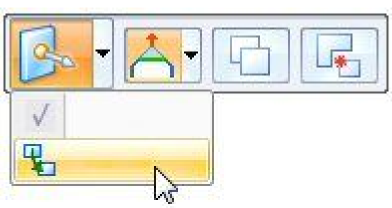
Observations:

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

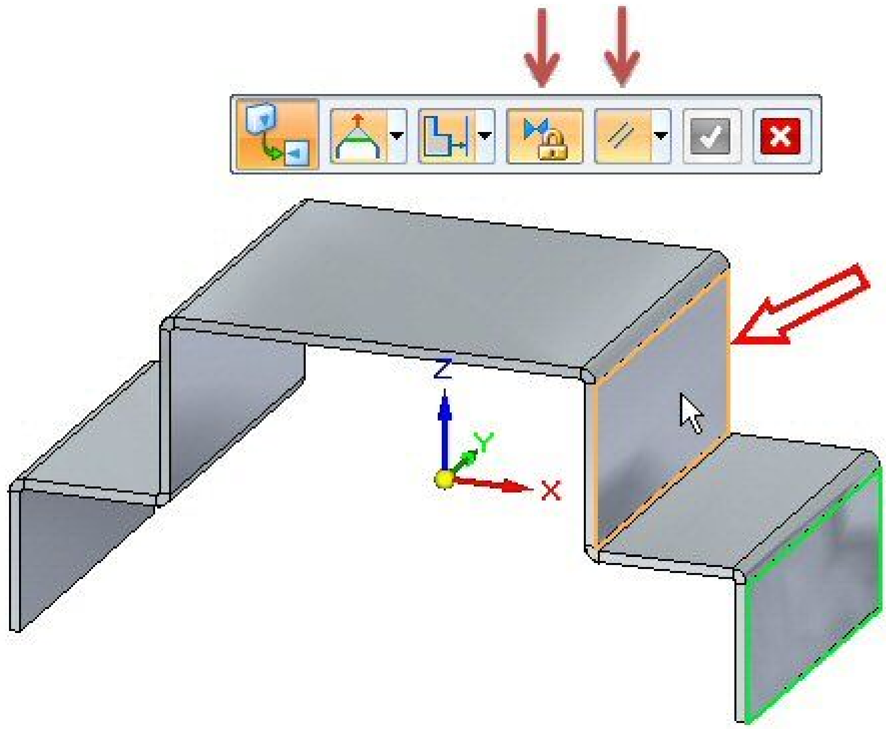
- Select the face shown.



- Click the Relate command on the command bar.



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

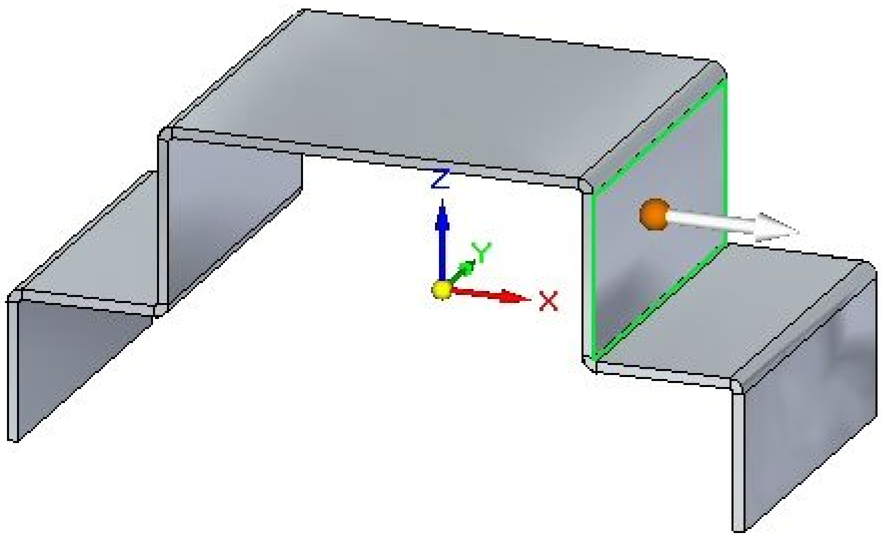


- Click the green check to accept.

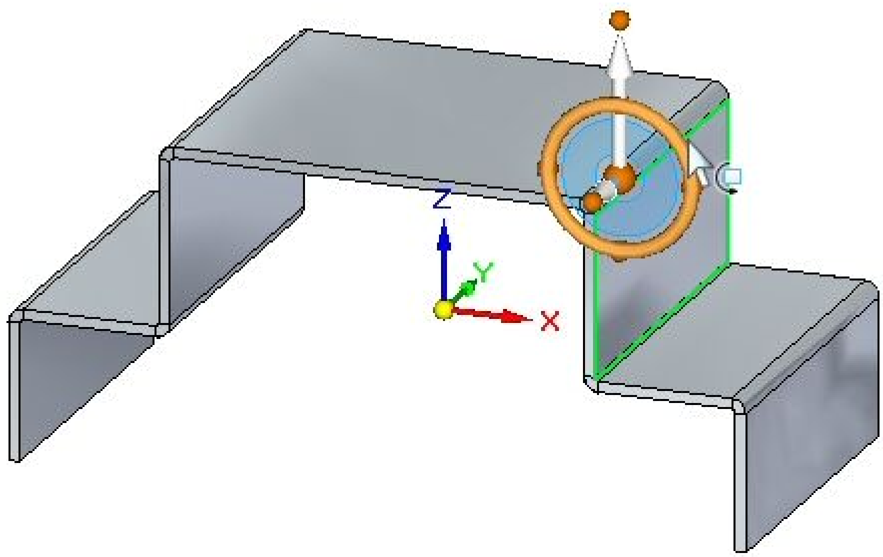
  Note

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

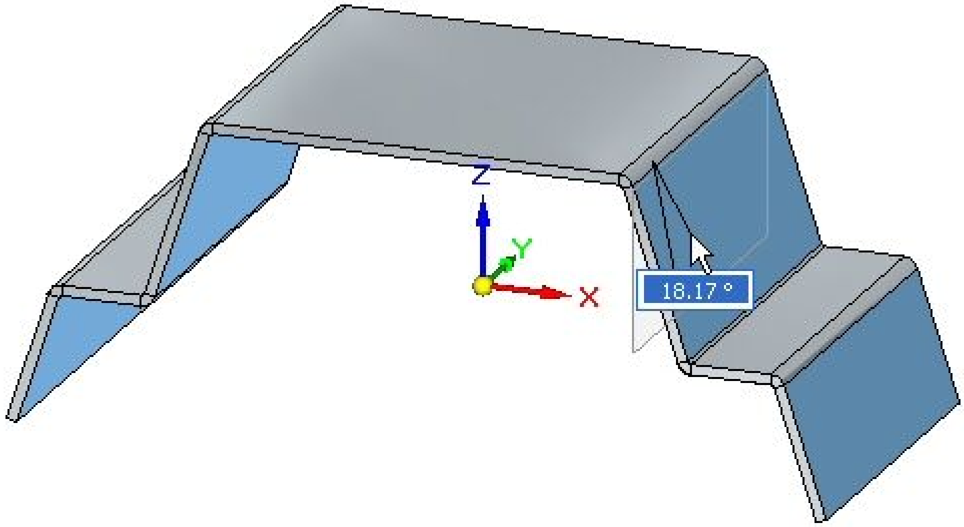
- Select the face shown.



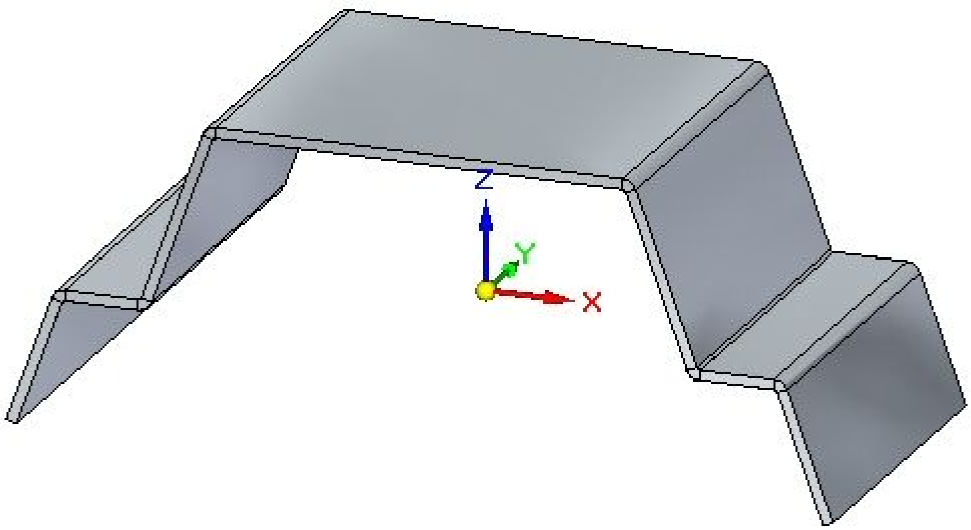
- Position the steering wheel on the bend and click the torus to rotate the face as shown.



- Rotate the face and observe the behavior.



- Rotate the face 20º.



Observations:

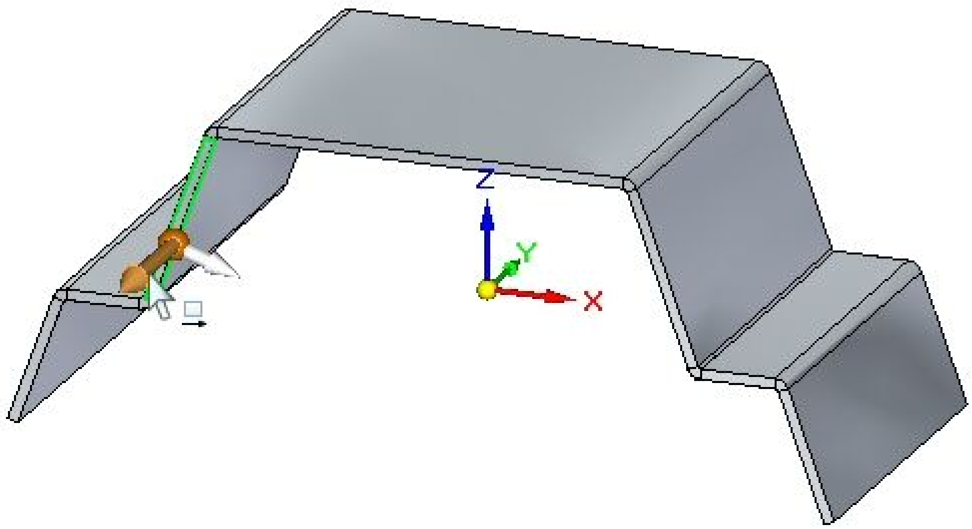
- Symmetry about the base reference planes is causing the opposite flanges to remain symmetrical. The vertical flanges remain parallel due to the persistent relationship established previously.

- 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.

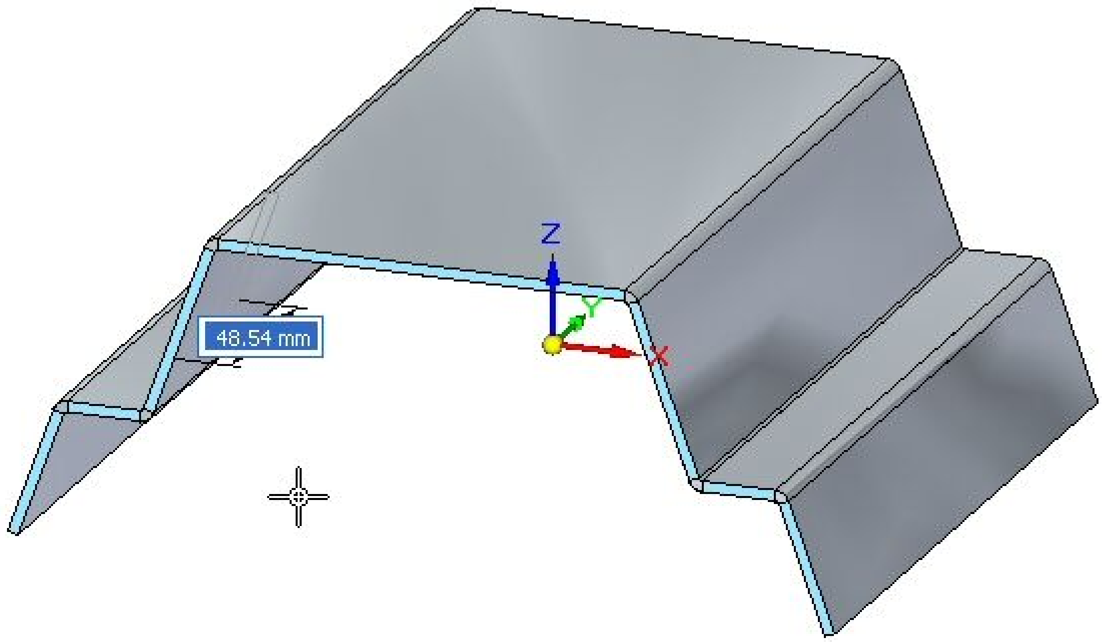


- Click the Live Rules Restore button.

Note

This resets the live rules to the default values.

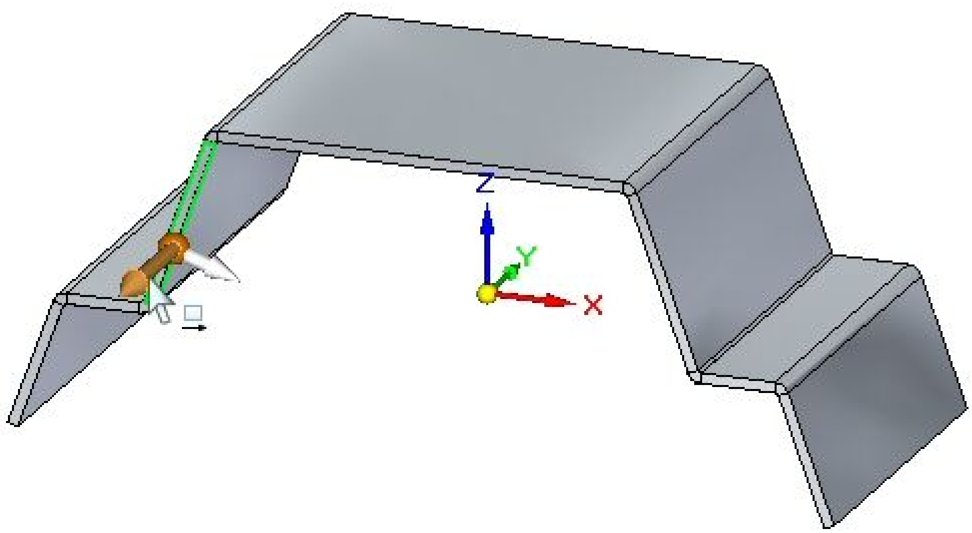
‣ Drag the handle as shown and observe the behavior, then press the **Esc** key.



Observations:

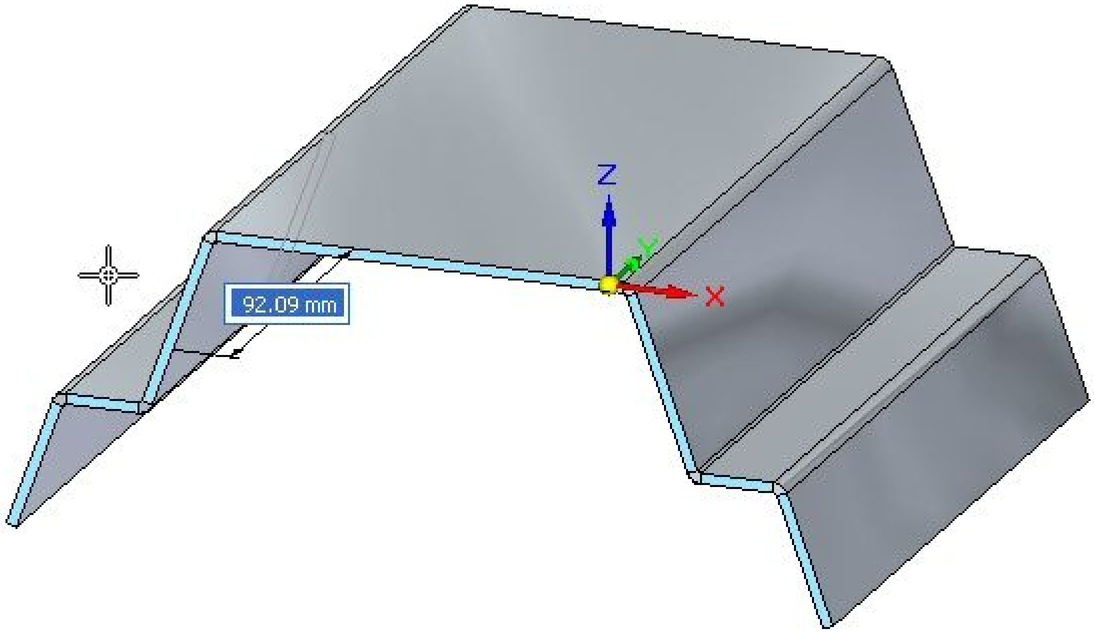
- The live rules affecting the behavior are thickness chain, maintain coplanar faces and symmetry about the base reference planes.

‣ Select the thickness face shown and select the move handle.



‣ Turn off symmetry in Live Rules.

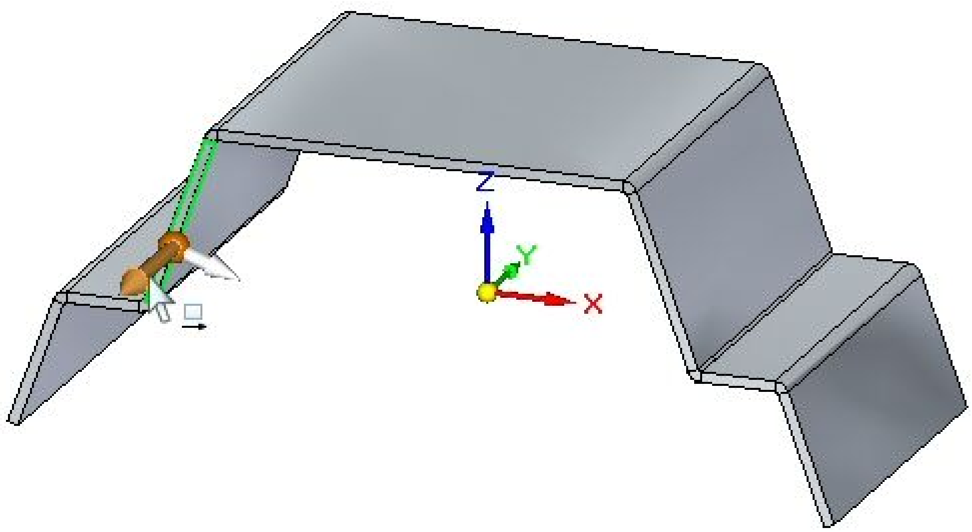
- Drag the handle as shown and observe the behavior, then press the **Esc** key.



Observations:

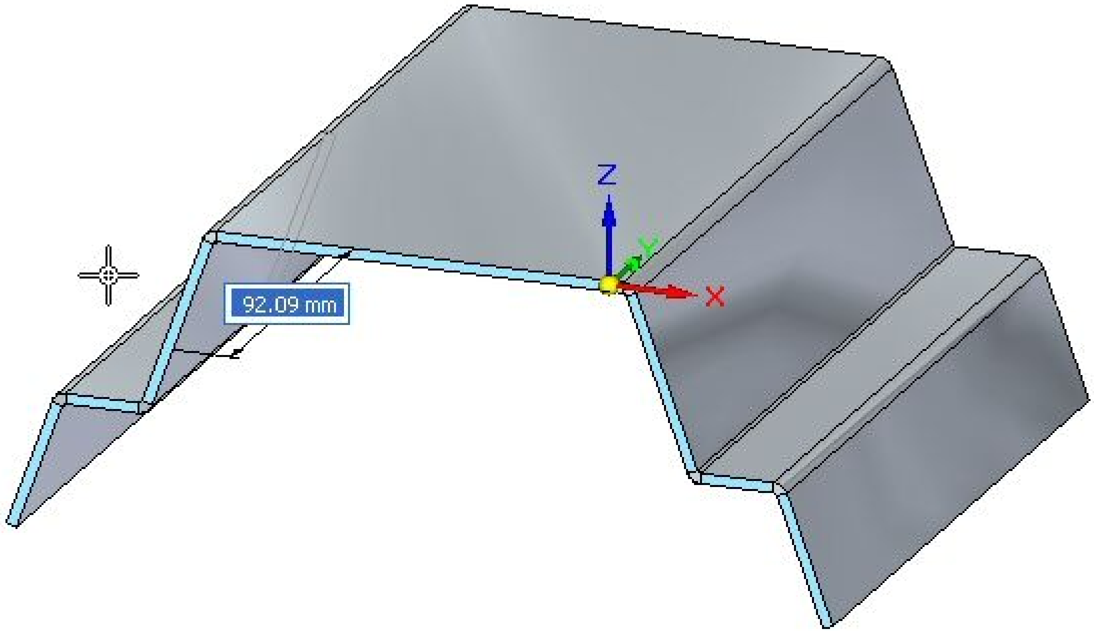
- The live rules affecting the behavior are thickness chain, and maintain coplanar faces.

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



- Turn off Maintain Coplanar Faces in Live Rules.

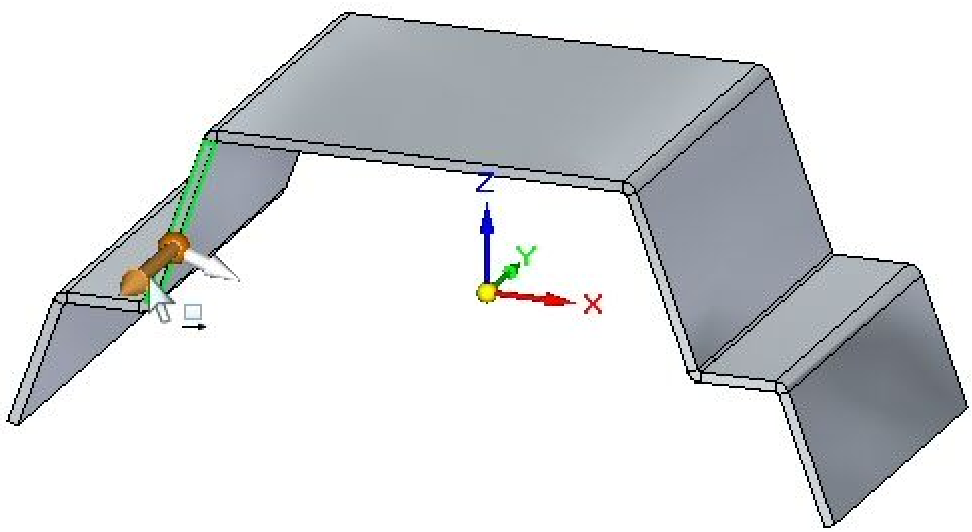
- Drag the handle as shown and observe the behavior, then press the **Esc** key.



Observations:

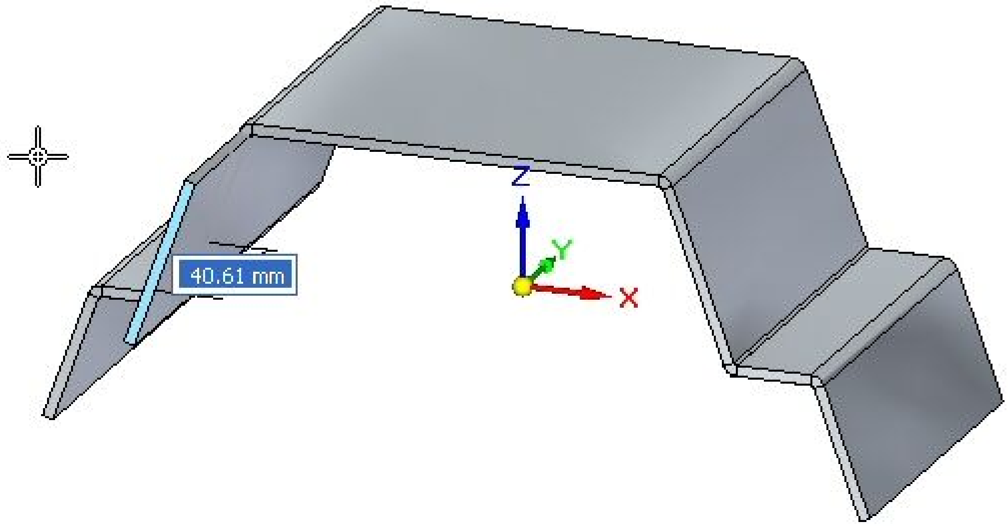
- When a face is selected and it is part of a thickness chain, the maintain coplanar faces for that thickness chain is overridden. This will become apparent in the next step.

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



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

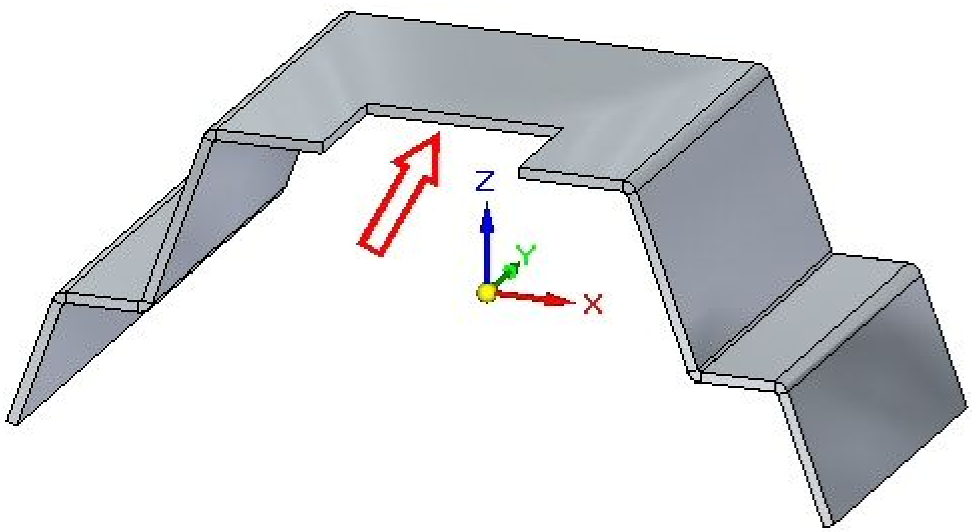
‣ Drag the handle as shown and observe the behavior, then press the **Esc** key.



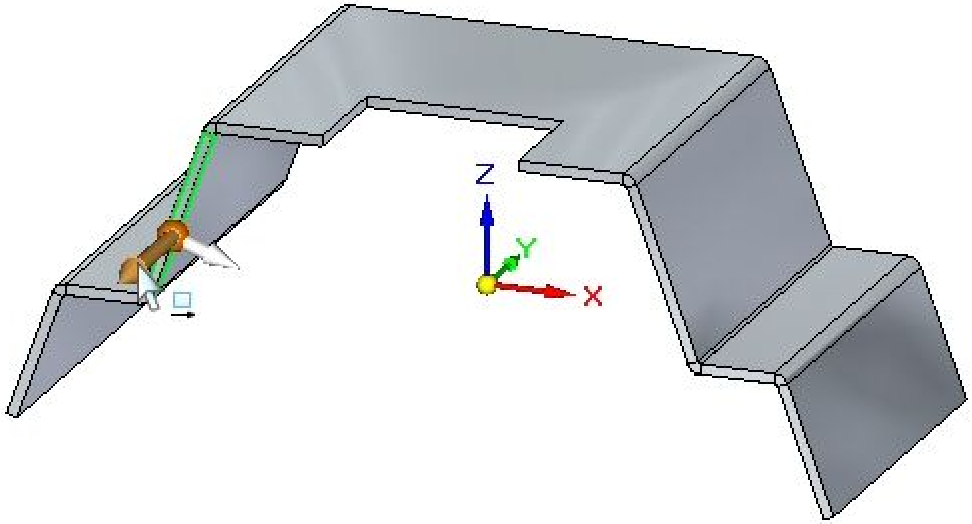
Observations:

- Even though Maintain Coplanar Faces is turned on, because Thickness Chain is off, only the face selected is moved.

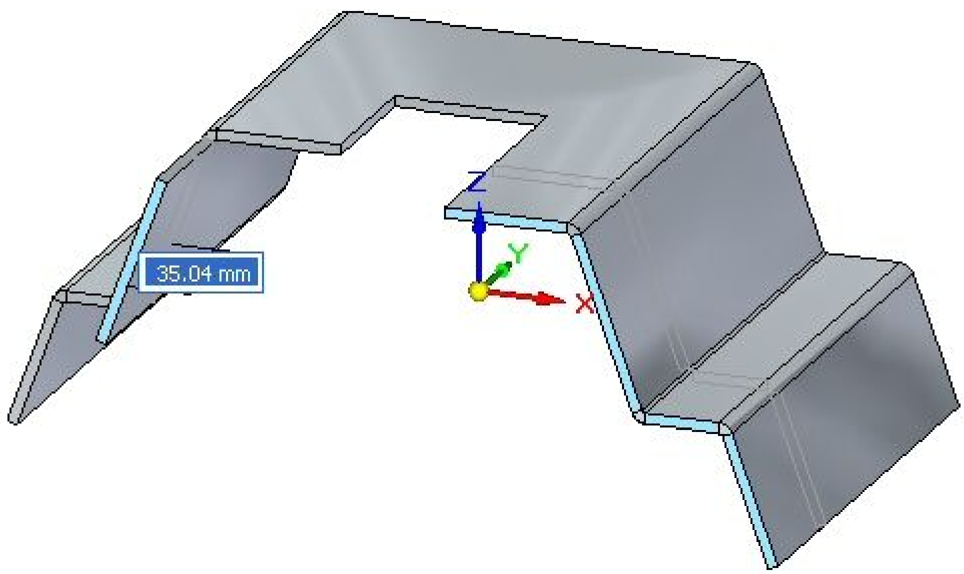
‣ Create the cutout approximately as shown.



- 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.

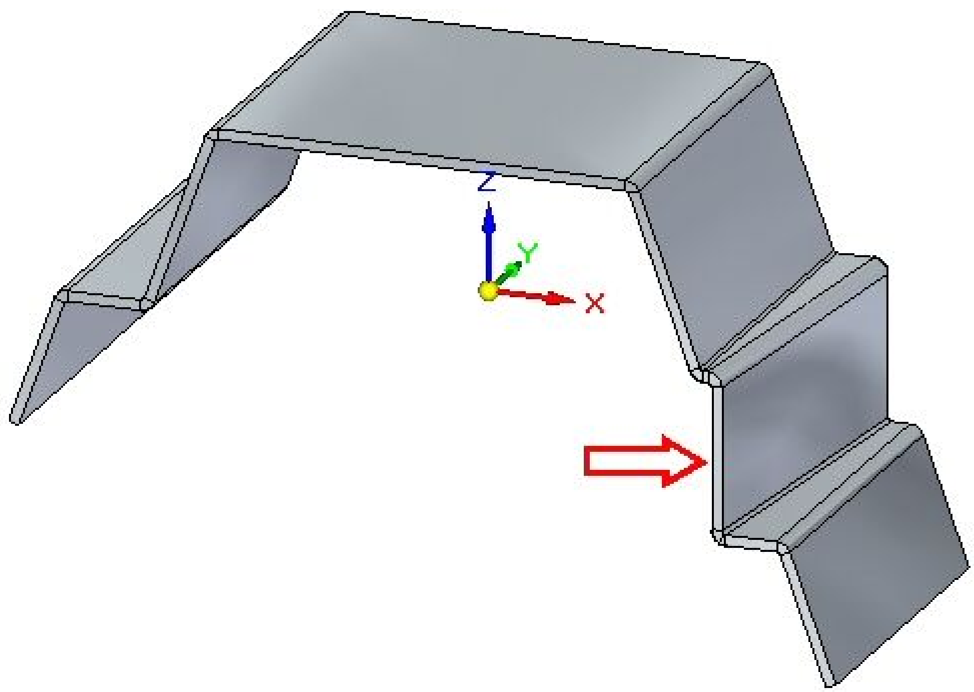


Observations:

- 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.

Lesson review

Answer the following questions:

1. Define sheet metal thickness chain.

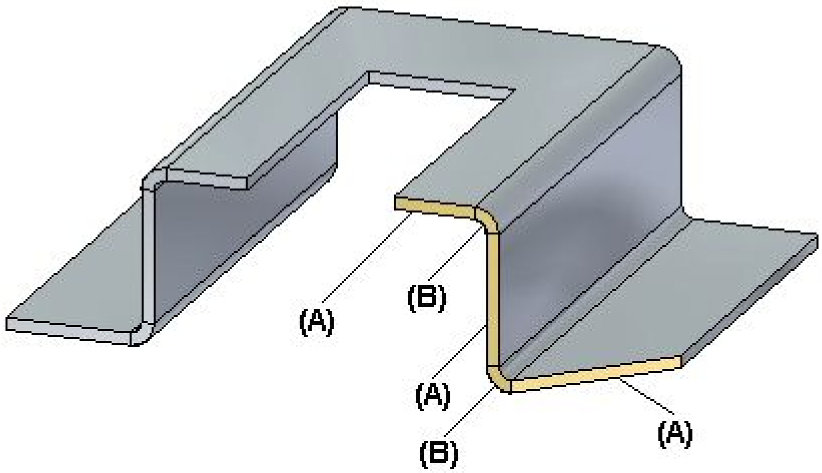
2. Describe how the maintain thickness chain option affects the movement of a face contained in the thickness chain.

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

Answers

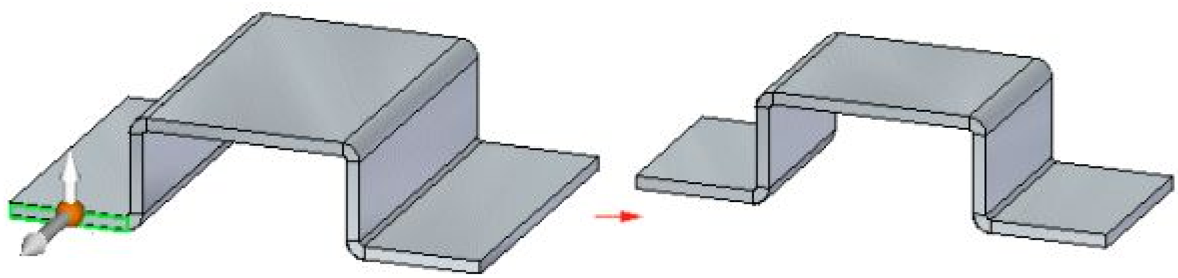
1. Define sheet metal thickness chain.

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

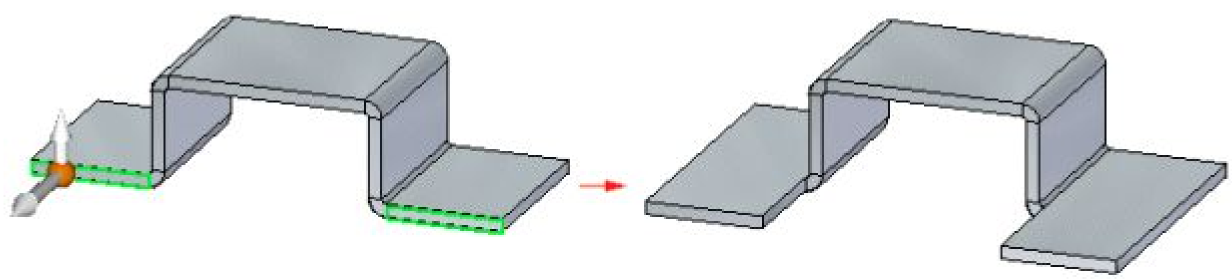


2. Describe how the maintain thickness chain option affects the movement of a face contained in the thickness chain.

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

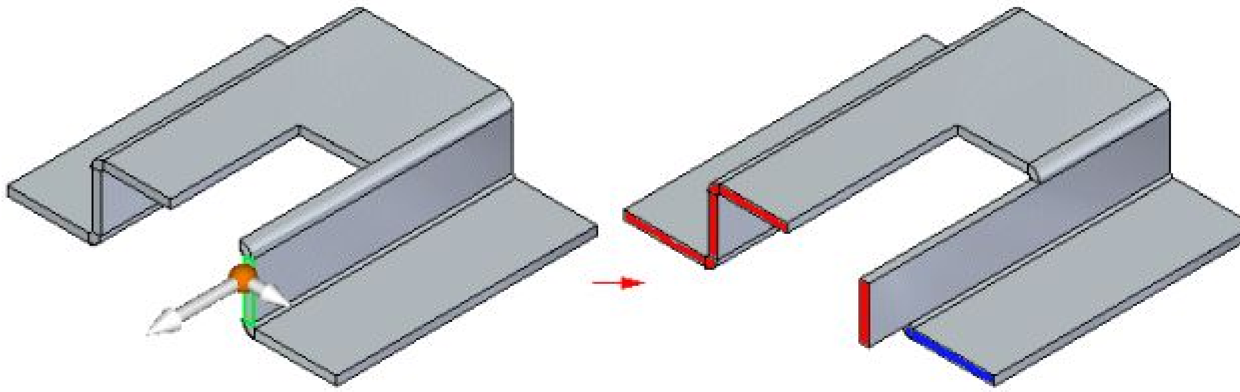


   When the Thickness Chain option is not set, only the selected face or faces move.



3. 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

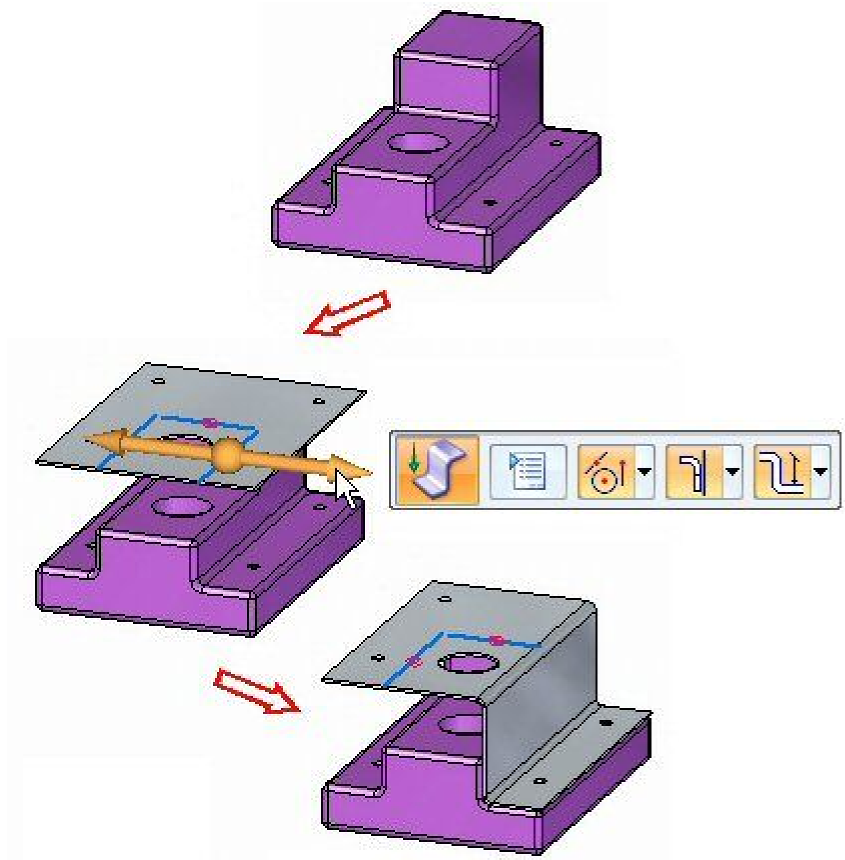
In this lesson you explored the behavior of sheet metal geometry by creating relationships and changing live rules.

Lesson

9 Jog

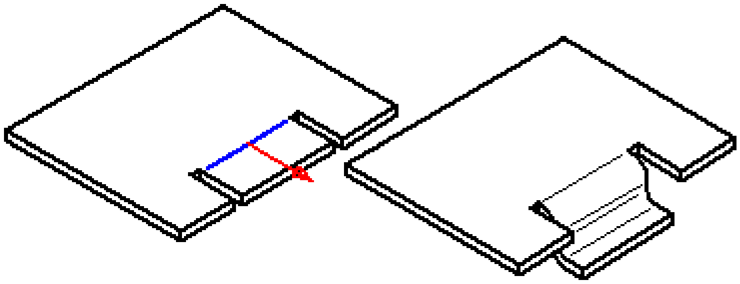
Constructing a Jog in a sheet metal part

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.

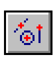
Measurement Point

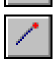
Measures from the selected face to the inside or outside of the new feature.

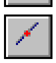
Measurement Inside	Measures from the selected face to the near side of the feature.
Measurement Outside	Measures from the selected face to the far side of the feature.

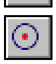
Keypoints

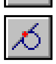
Sets the type of keypoint you can select to define a feature extent or to position a new reference plane. You can then define the feature extent or the location of the reference plane using a keypoint on other existing geometry. The available keypoint options are specific to the command and workflow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

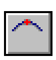
 Selects any keypoint.

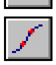
 Selects an end point.

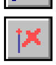
 Selects a midpoint.

 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.

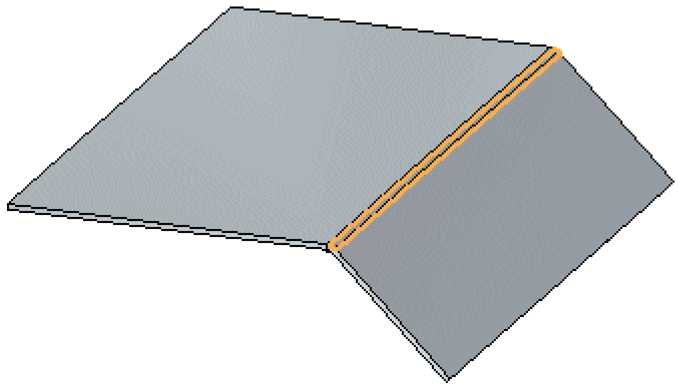
 Selects a silhouette point.

 Selects an edit point on a curve.

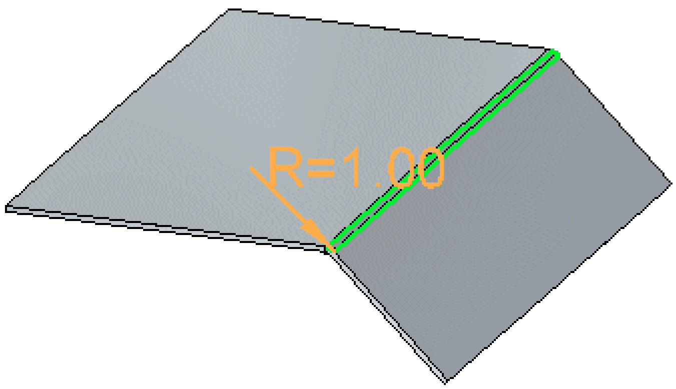
 Sets the no keypoint option.

Editing the bend radius

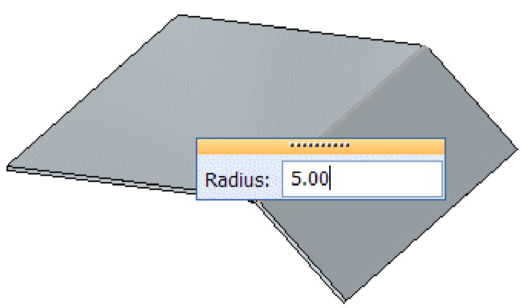
When you create a bend, the bend radius is defined from the default global parameters. Once a bend is created, it can be edited by selecting the bend,



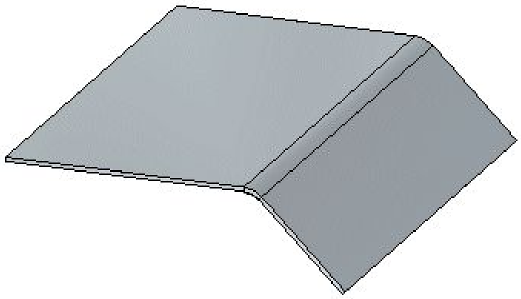
then selecting the bend radius handle,



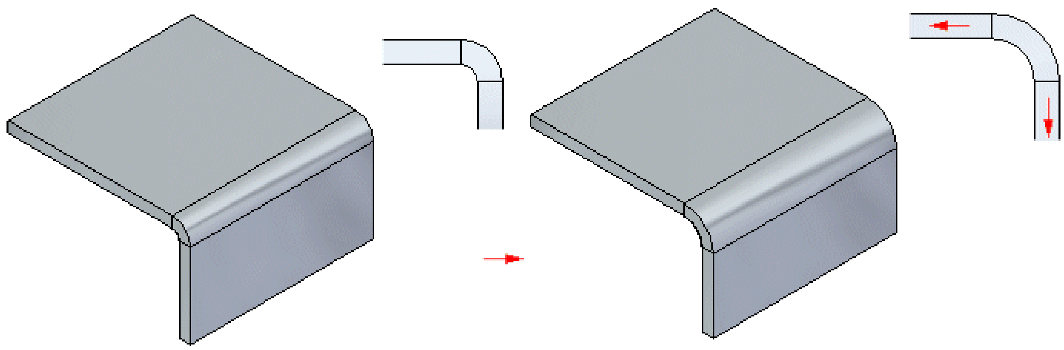
then typing a new value in the dynamic edit control,



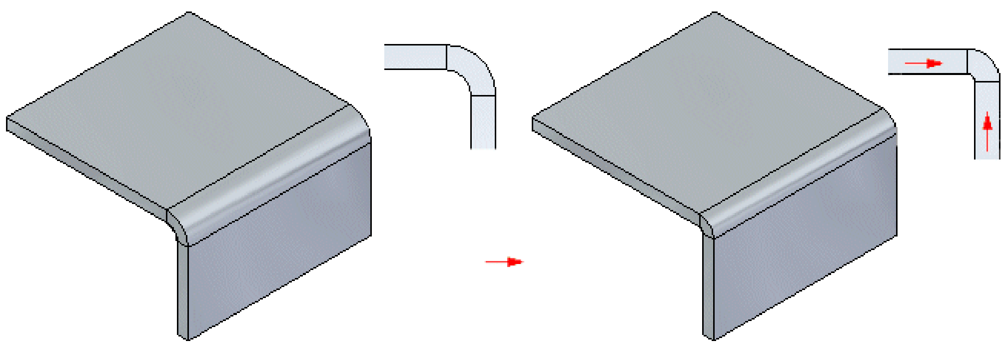
and then clicking to finalize editing the radius.



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

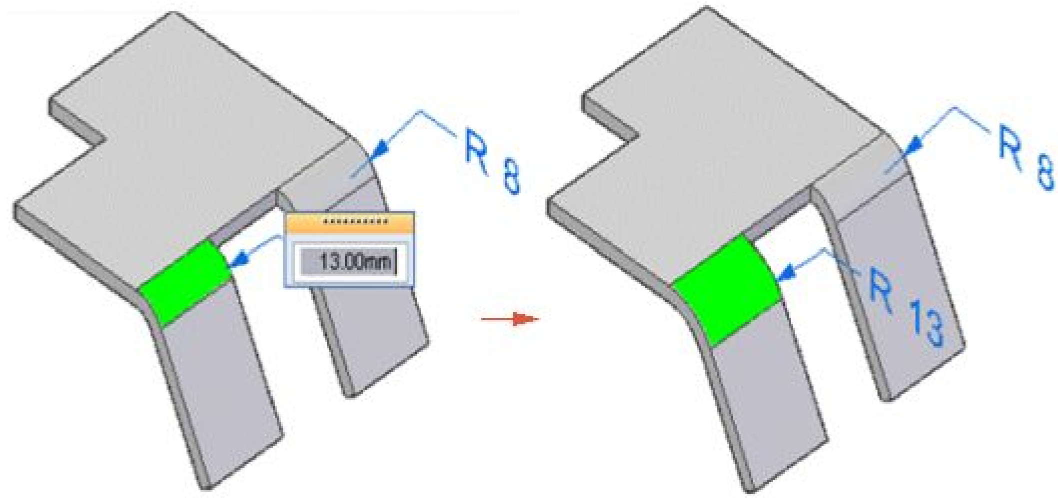


If you decrease the bend radius, the tab and flange extend into the bend to decrease the bend radius.

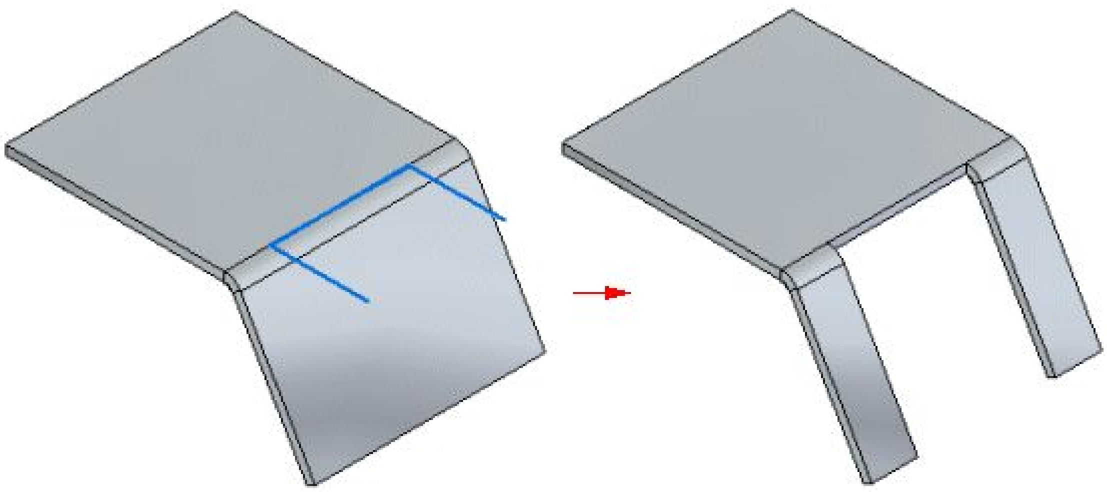


Editing the bend radius in split bends

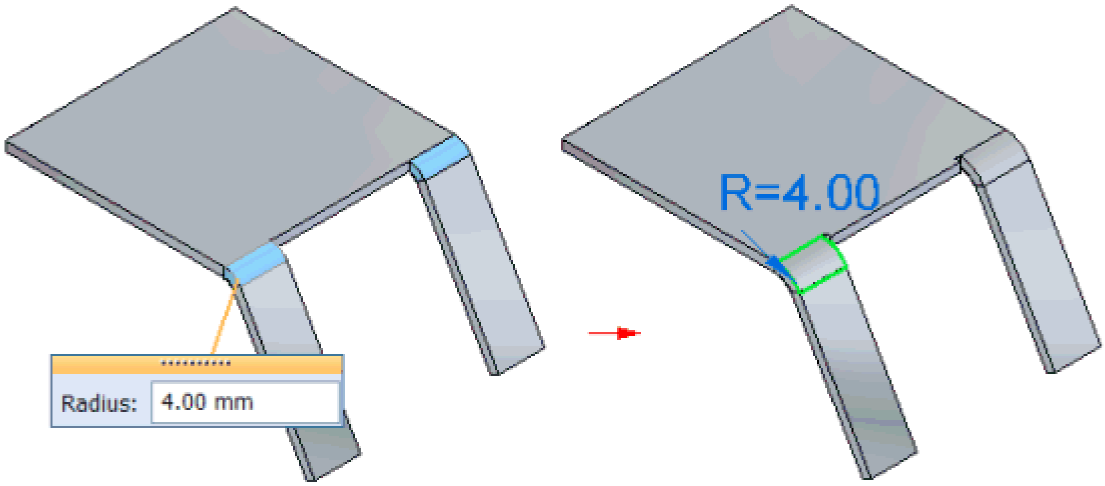
Depending on how split bends are created, the bend radius for split bends may be edited separately or together. If the split bends are created when two flanges are created independently as partial flanges, the bend radius can be edited separately.



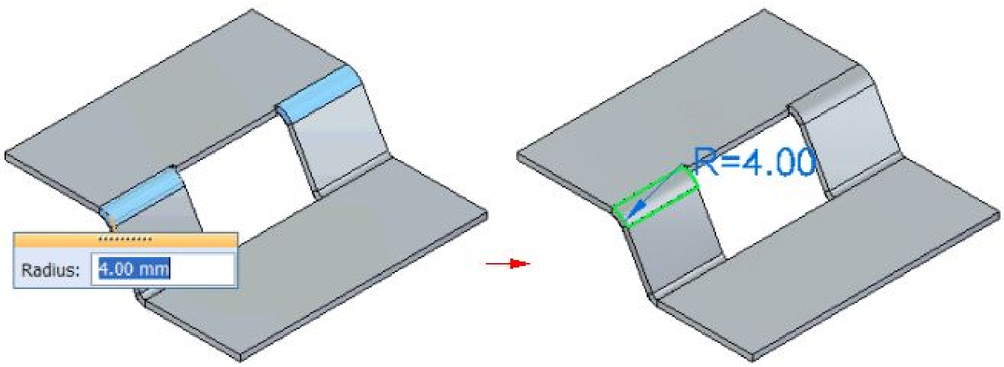
If the split bends were originally a single bend in a flange, and then cut to make two separate flanges,



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.

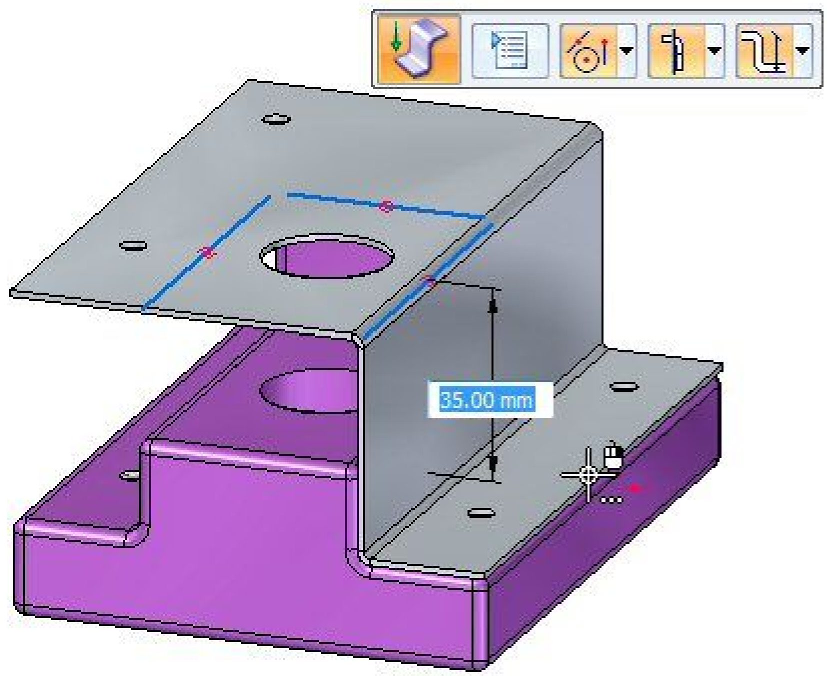


Activity: Using the jog and break corner command in sheet metal design

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.



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.

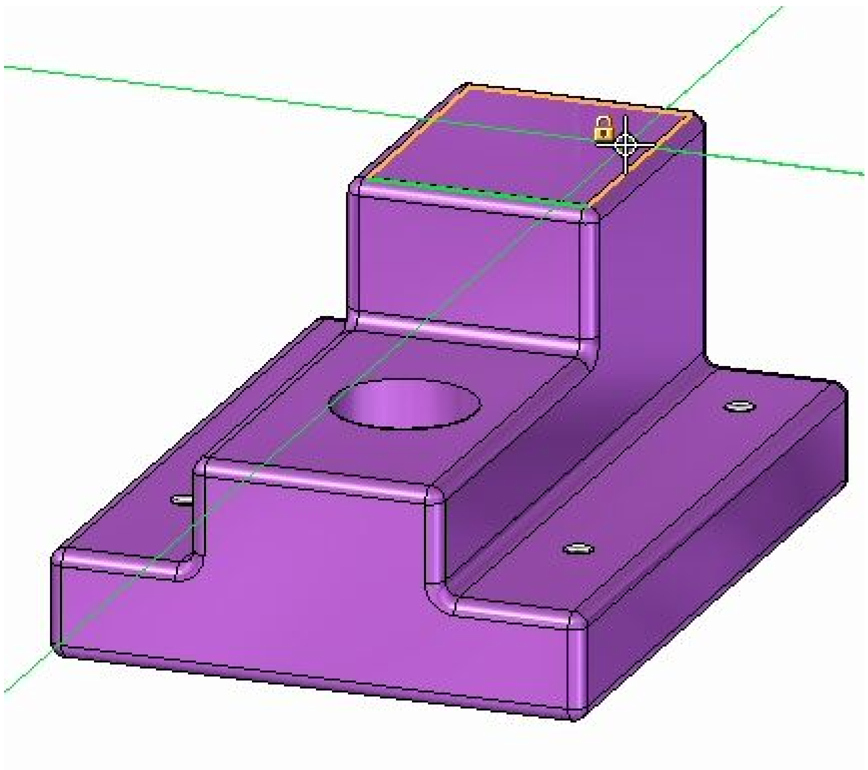
- Proceed to the next step.

Draw the sketch and create the base feature

Note

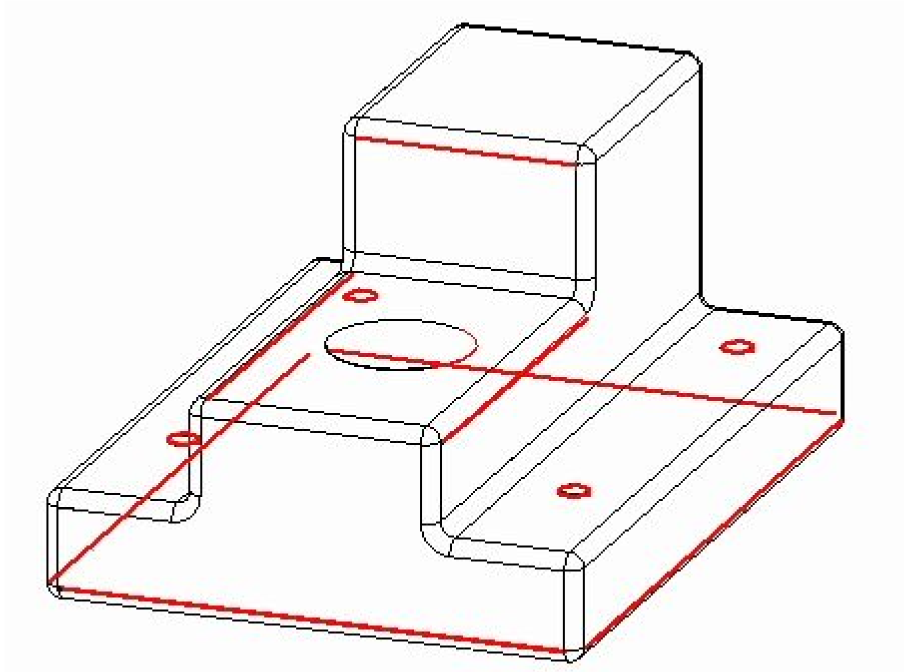
This sheet metal file was created in the context of an assembly and inter-part geometry from a part file was imported into the file. This geometry will be used to define the extents of the sheet metal that will cover the part. It is visible in PathFinder and the display of the geometry can be toggled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

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



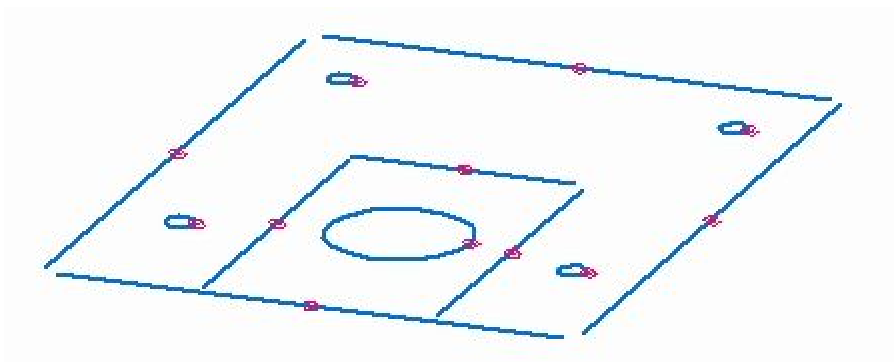
- Use the command to include the following geometry in the sketch:

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



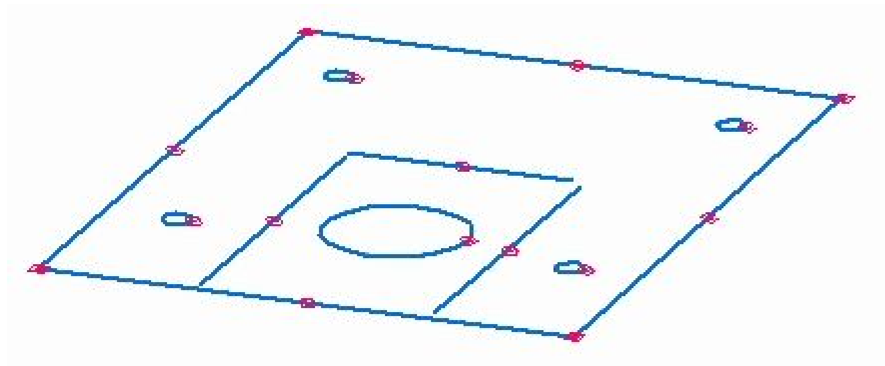
- The 5 holes.

The sketch appears as shown.

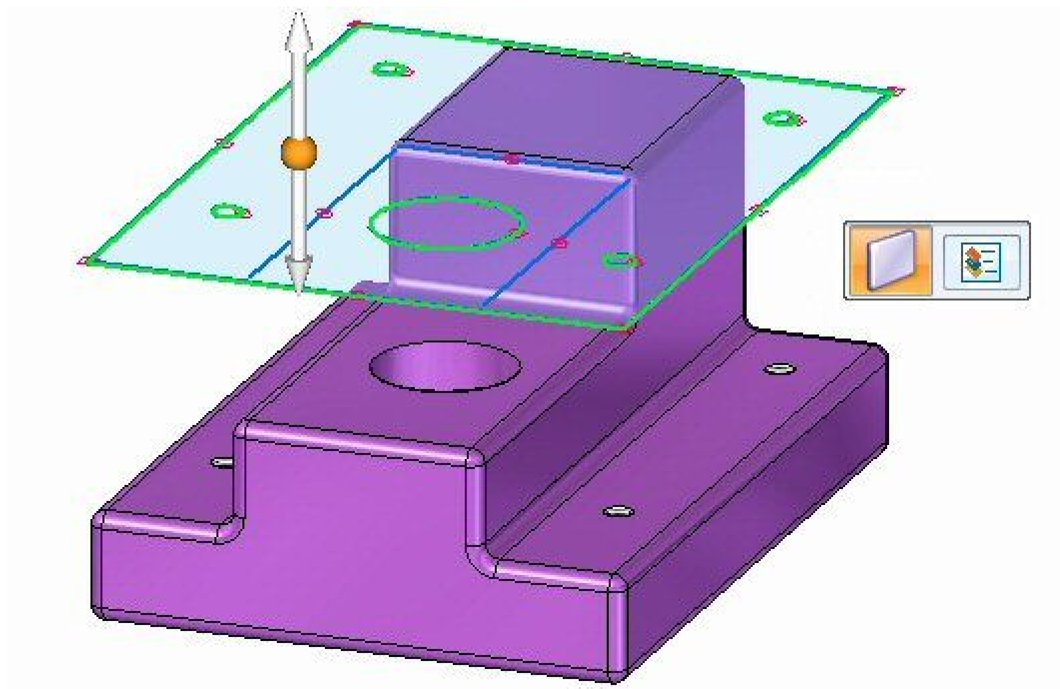


- Click the Trim Corner command  and trim the outer lines so that they intersect and form a closed region.

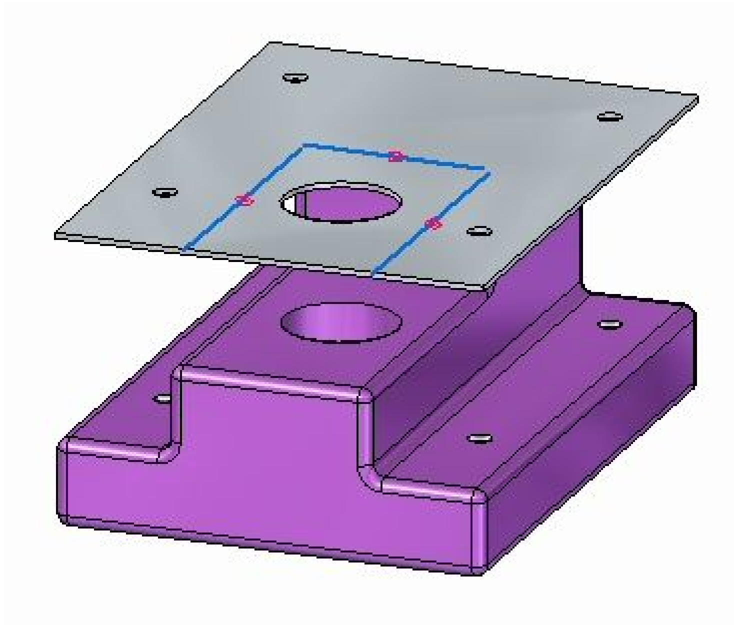
  The sketch appears as shown.



- Select the region shown.

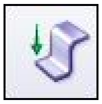


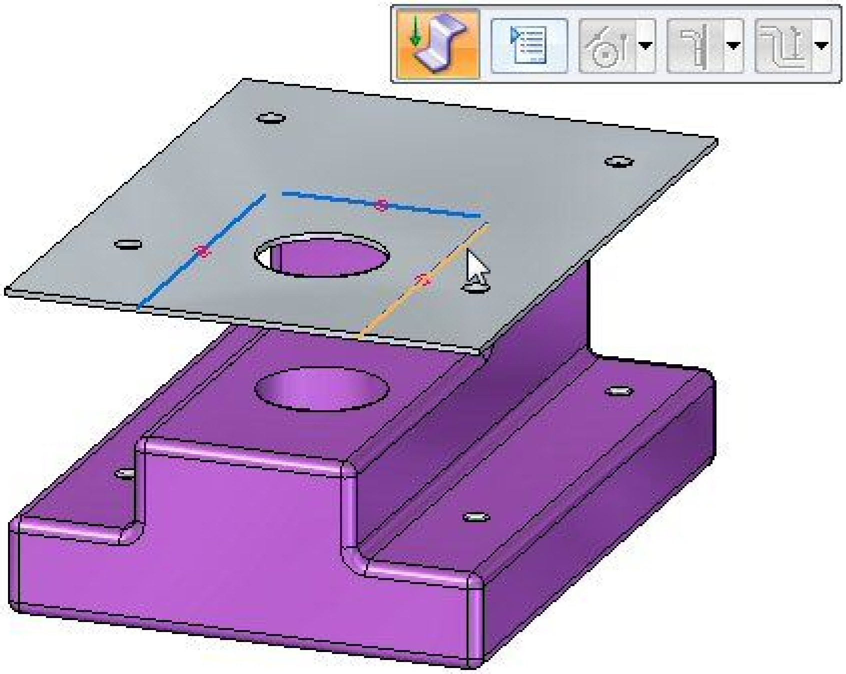
- Click the top arrow to create the base feature above the part geometry as shown.



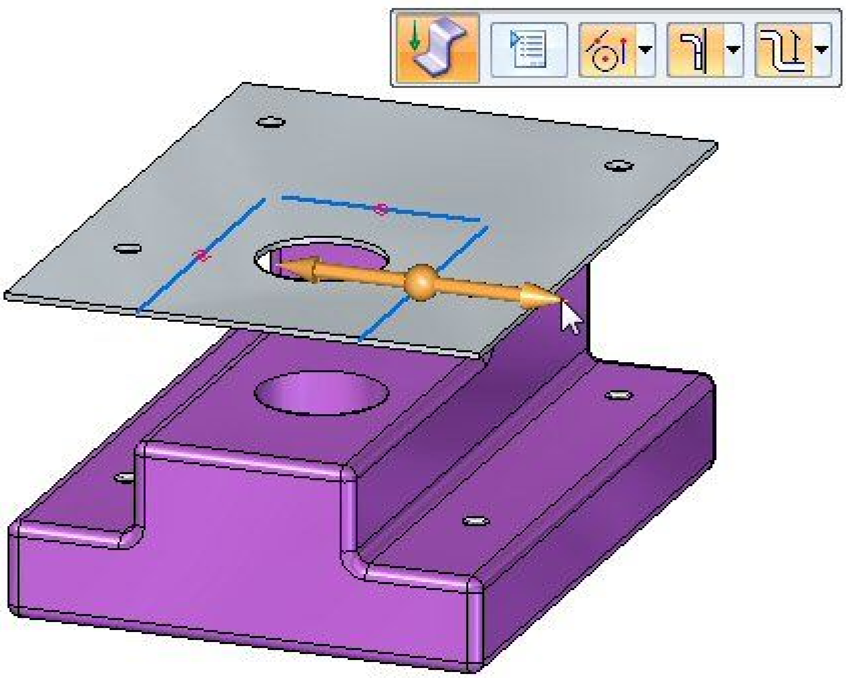
- Proceed to the next step.

Create a jog

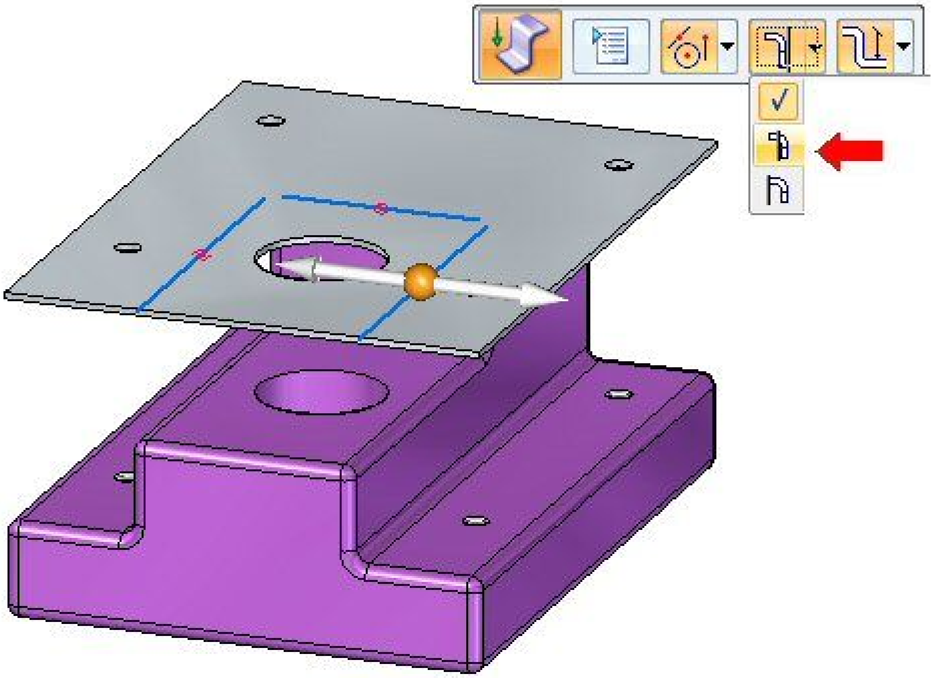
- Click the Jog command  and select the line shown.



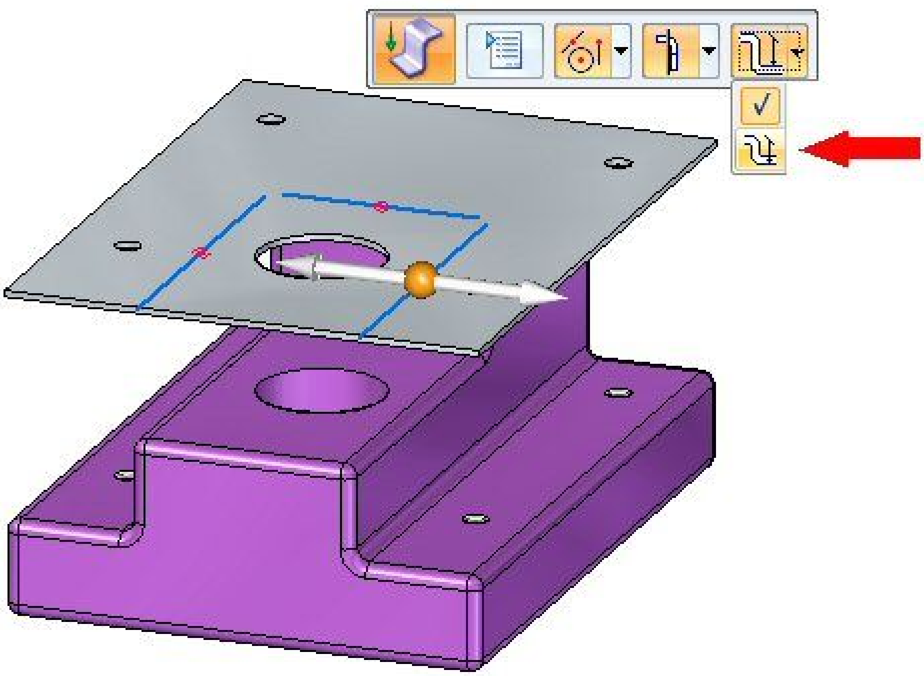
- Select the direction arrow as shown.



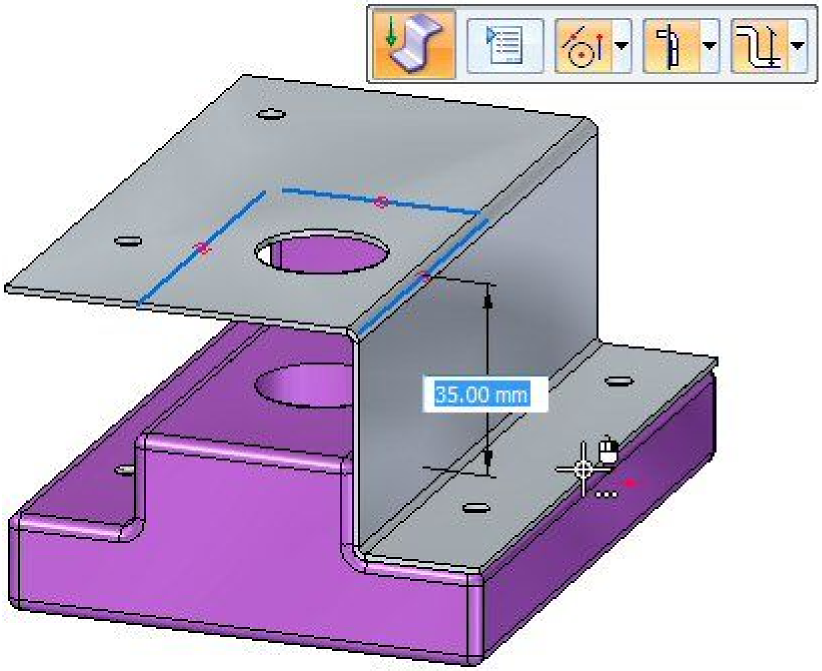
- Select the Material Outside option on command bar.



- 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.

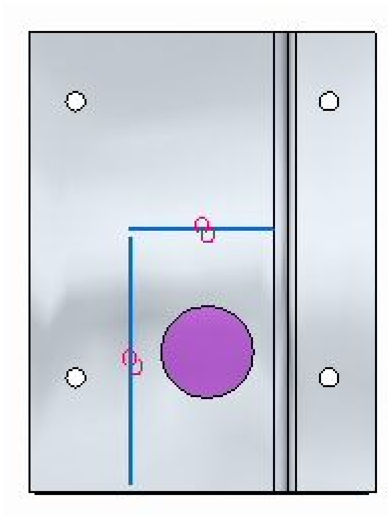


The jog is created.

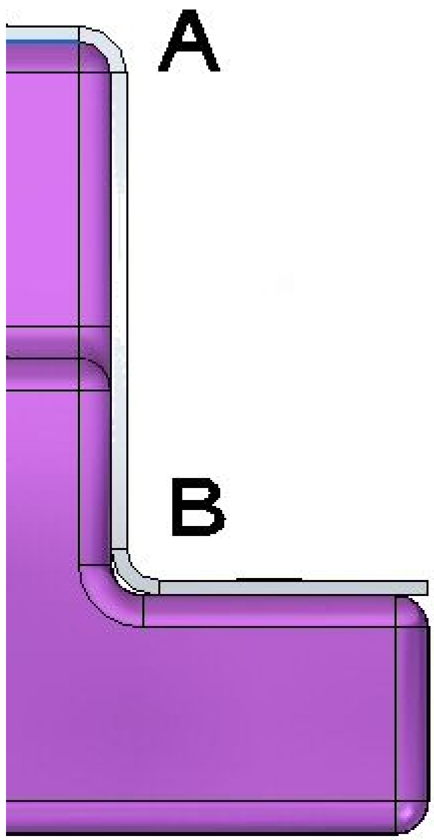
- Proceed to the next step.

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.

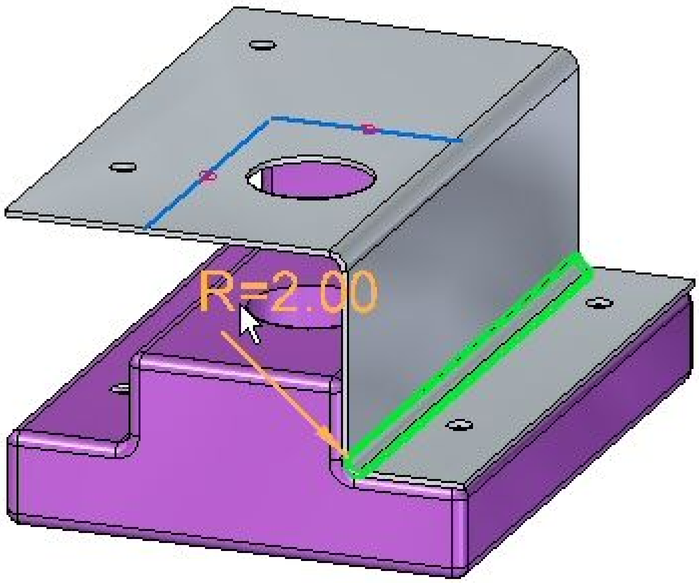


- Press **Ctrl+F** to rotate the view to a front view.

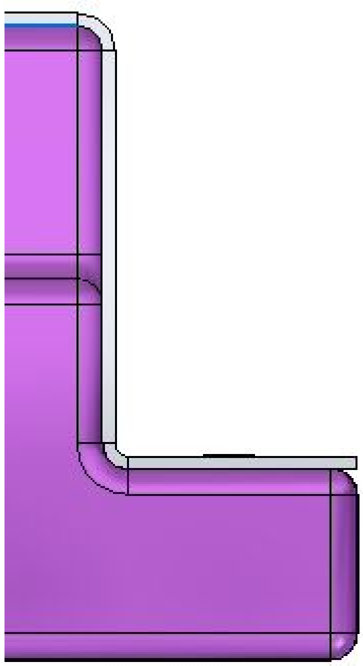


At (A) in the figure above, the sheet metal fits exactly over the 2.0 mm round on the part because the bend radius is 2.0 mm. The material thickness is 1.0 mm. The outer radius is the sum of these two values and is 3.0 mm. On the bottom bend, (B) the sheet metal does not fit exactly because of this.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

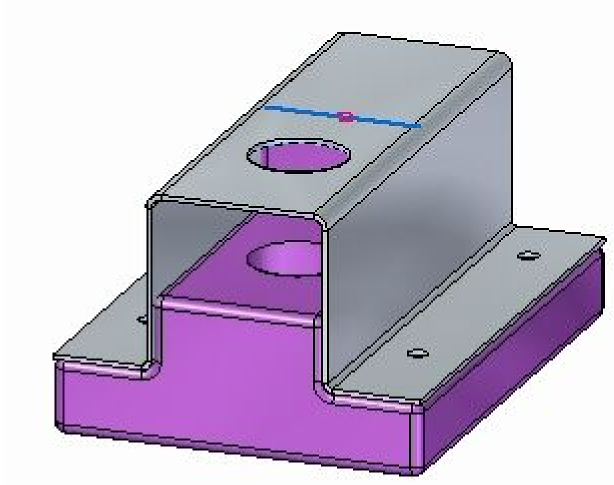
- Press **Ctrl+J** to rotate the view. Select the bend shown, then click the text. This will allow edits to the bend radius for that bend.



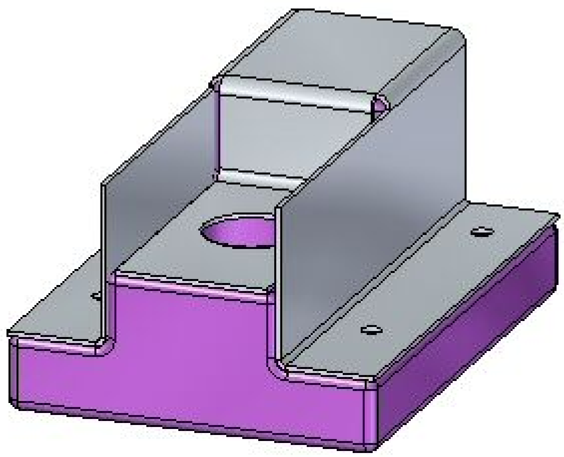
- Change the Bend Radius to 1.0 mm.
- Press **Ctrl+F** to rotate the view to a front view. Observe the bend.



- 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.



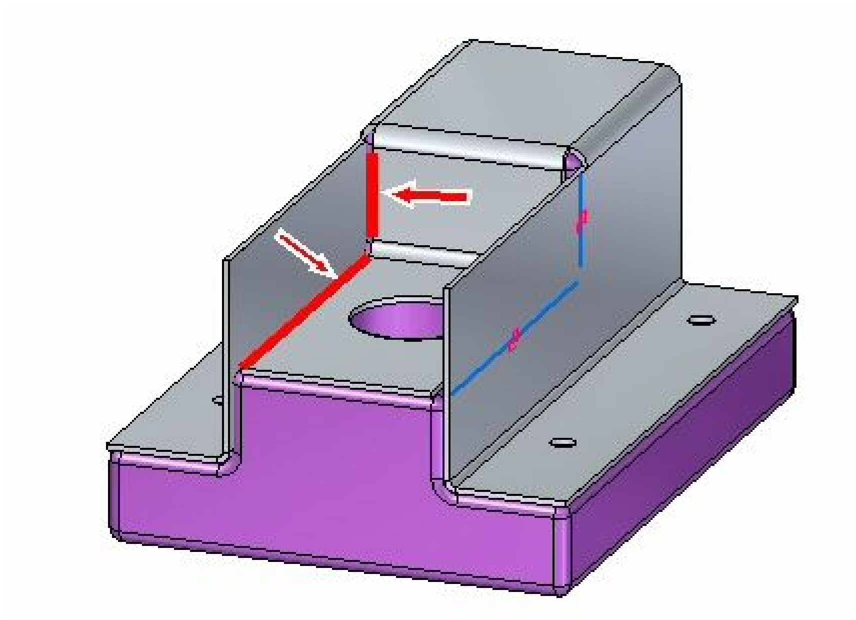
- Proceed to the next step.

Using the cut command to trim unwanted edges.

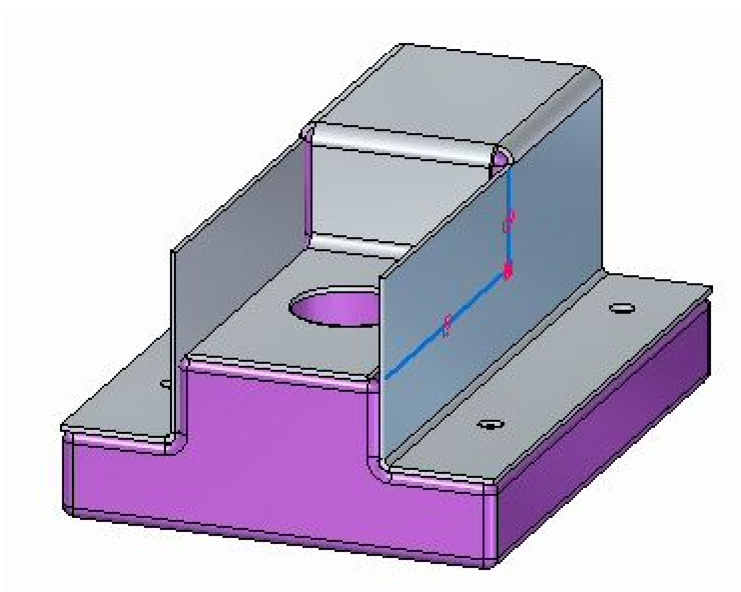
In this step the cut command will be used to trim away unwanted parts of the vertical flanges.

- Select the Project to Sketch command .

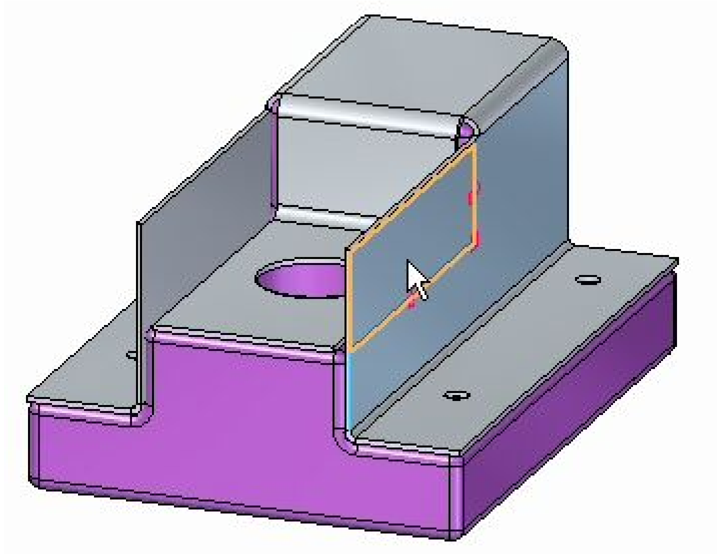
- Lock the sketch plane to the outer face of the vertical flange and include the following geometry in the sketch:



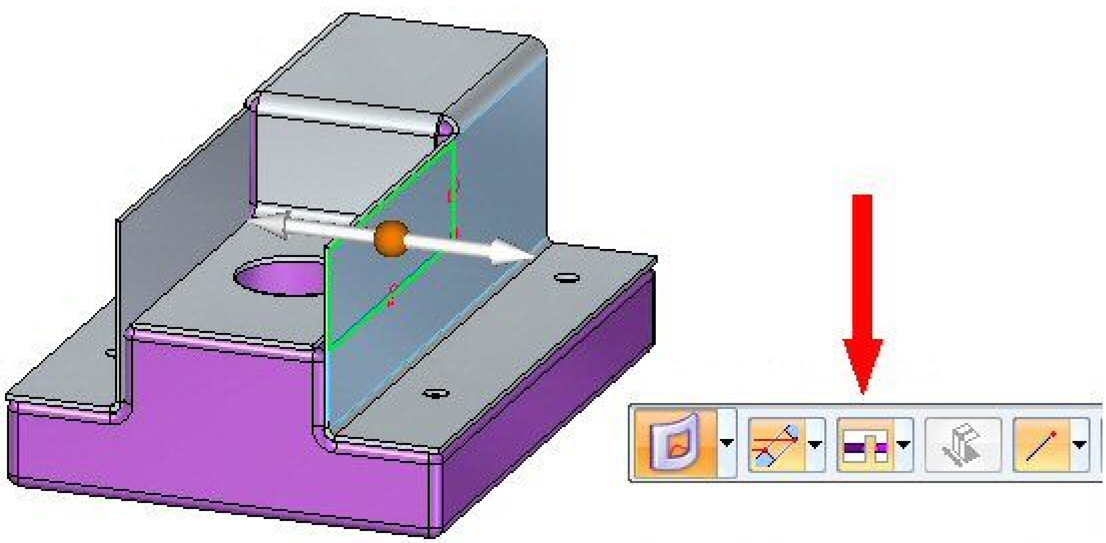
- Click the Trim Corner command and trim the lines so that they intersect and form a region.



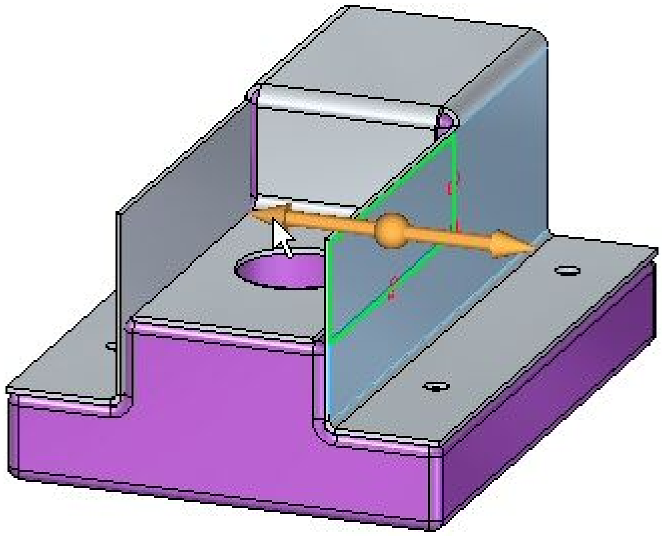
- Select the region shown.



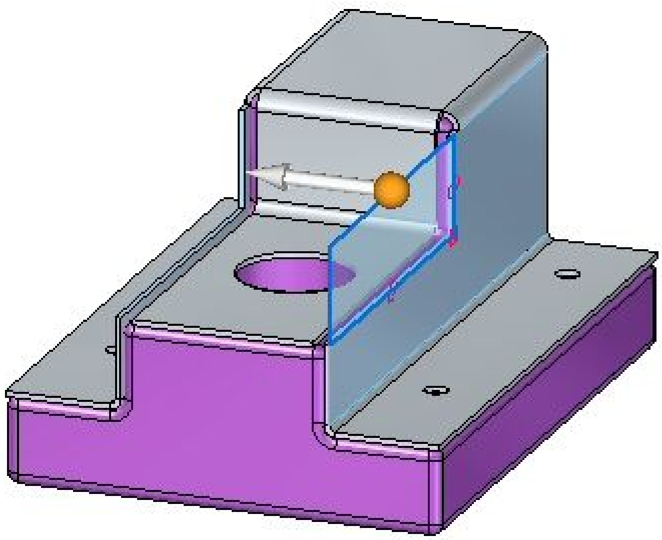
- Selecting the region initiates the cut command. Set the extent parameter to through all.



- Click the left arrow.



- The flanges are trimmed.



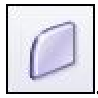
- Proceed to the next step.

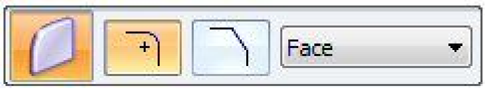
Using the cut command to trim unwanted edges.

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

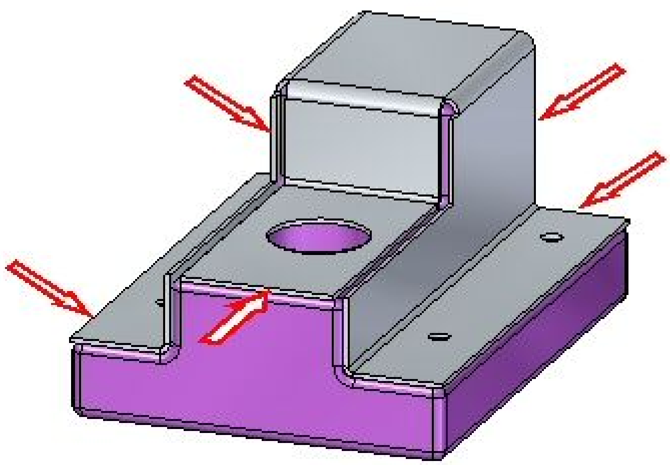
Note

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.

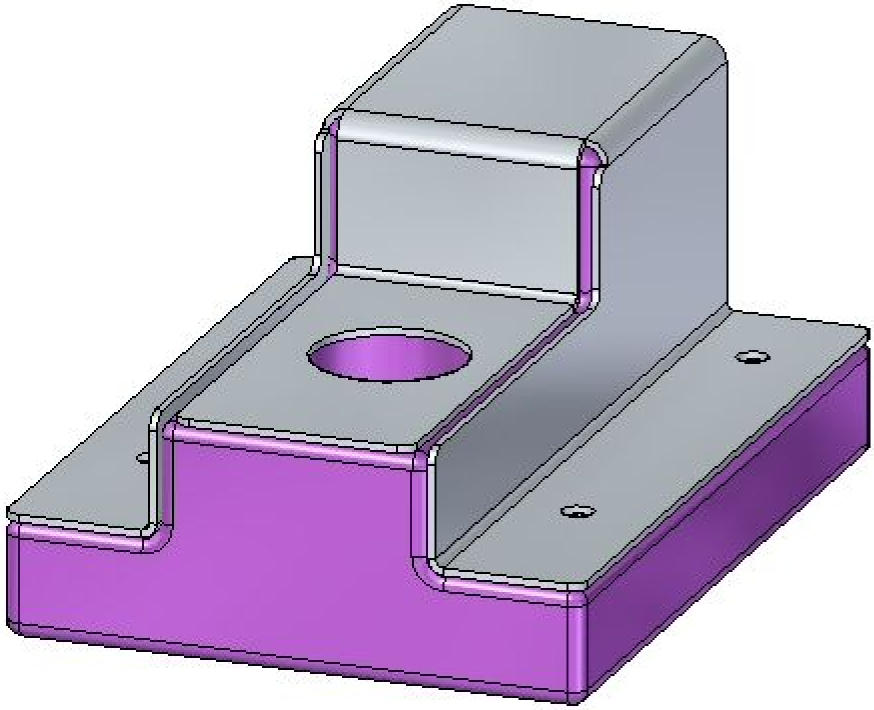
- Click the Break Corner command .
- On the command bar, set the Corner Type to Radius and the Selection Type to Face.



- 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 and 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 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 and existing part. You modified the bend radius where needed and then used the cutout command and the break corner command to finish the model.

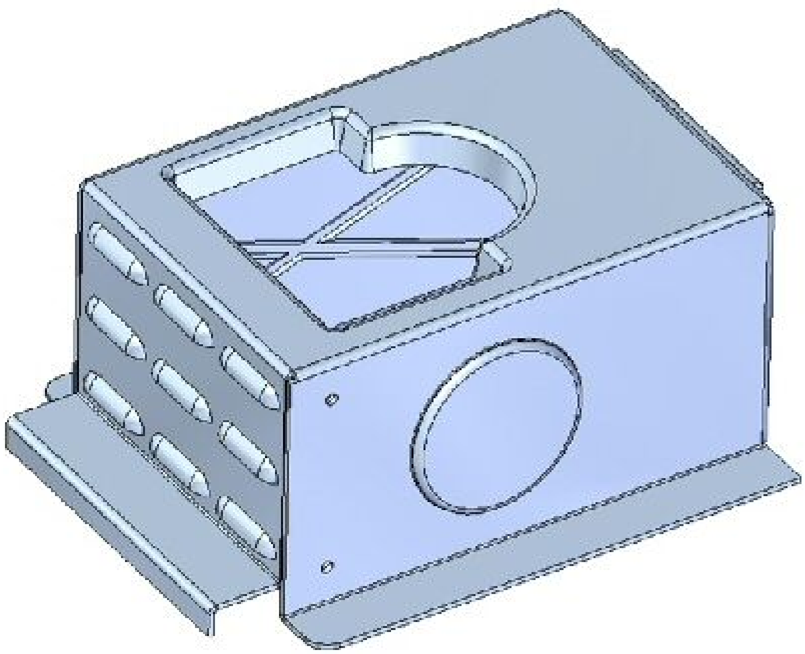
Lesson

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 remained 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.

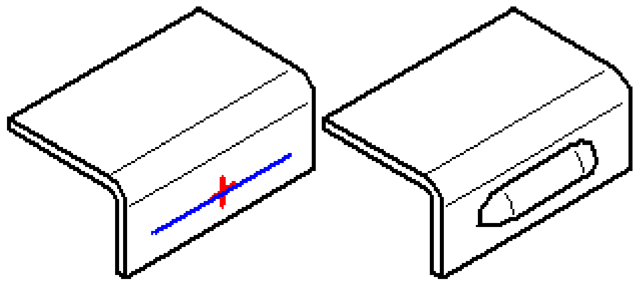


Adding sheet metal deformation features

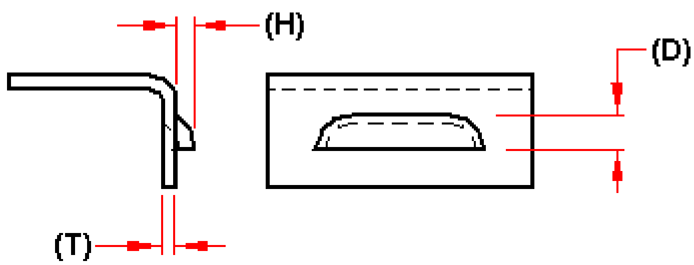
You can model features in the Sheet Metal environment that are manufactured with metal deformation techniques, such as deep drawing and coining. When parts are manufactured using deformation techniques, material thinning typically occurs. In Solid Edge, this material thinning is ignored and deformation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

Constructing louvers

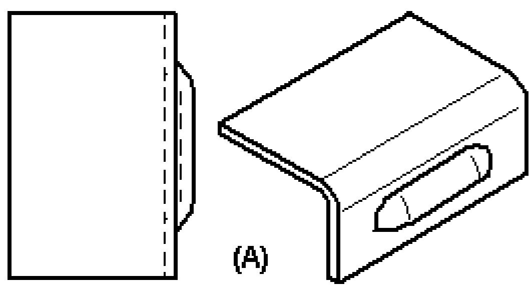
Like a jog feature, a louver feature is constructed using a single, linear element.

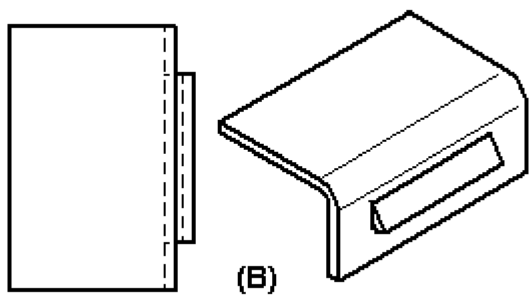


When constructing a louver, the louver height (H) must be equal to or less than the louver depth (D) minus the material thickness (T).



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

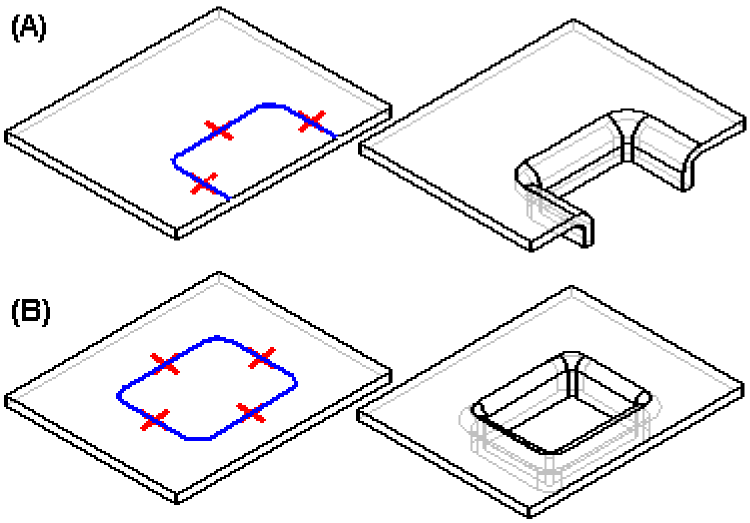




Louver features cannot be flattened.

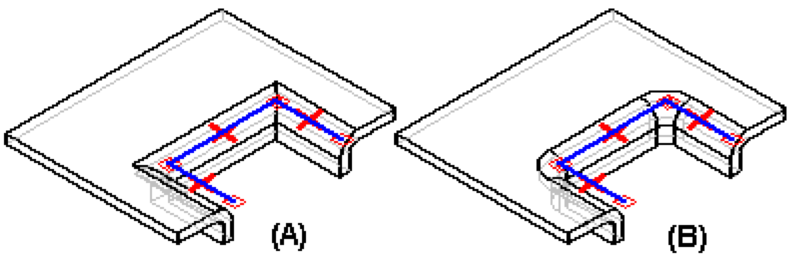
Constructing drawn cutouts

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

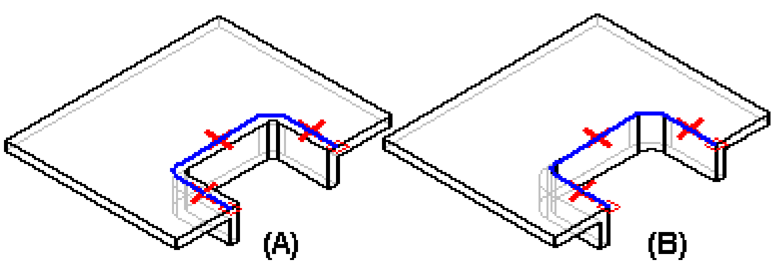


The ends of an open profile must theoretically intersect a part edge. A closed profile cannot touch any part edges. A drawn cutout can be constructed only on a planar face. You can use the Drawn Cutout Options dialog box to specify punch radius, die radius, and taper options.

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.



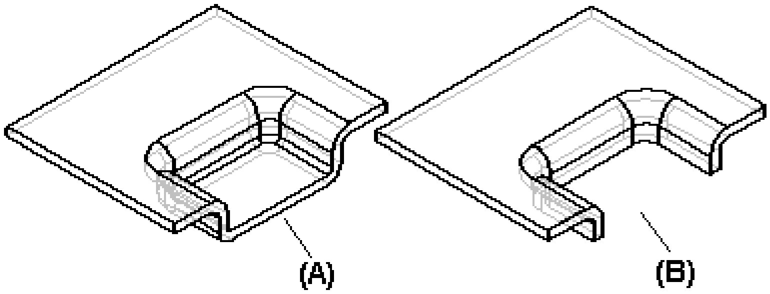
When you construct a drawn cutout, the sidewalls are constructed such that they lie inside the profile (A). After the feature is constructed, you can use the options to specify that the sidewalls lie outside the profile (B).



Drawn cutouts cannot be flattened.

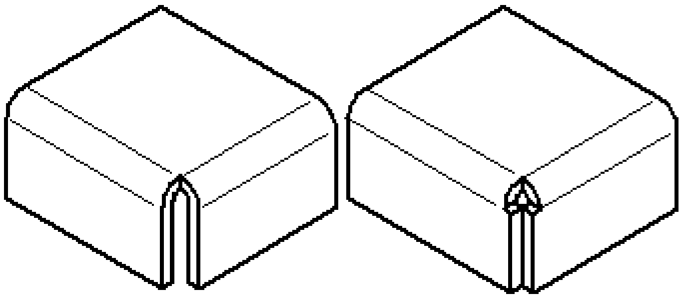
Constructing dimples

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.

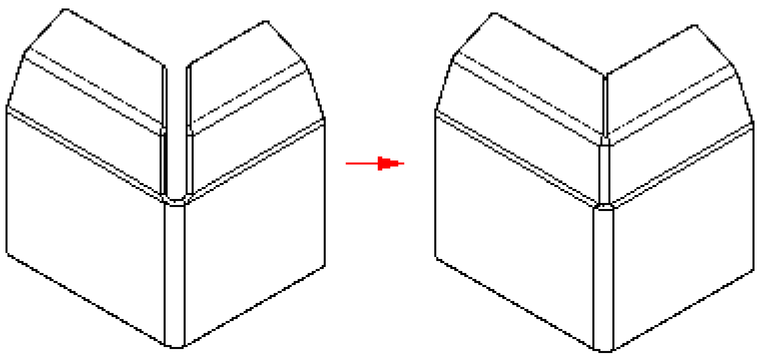


Closing corners

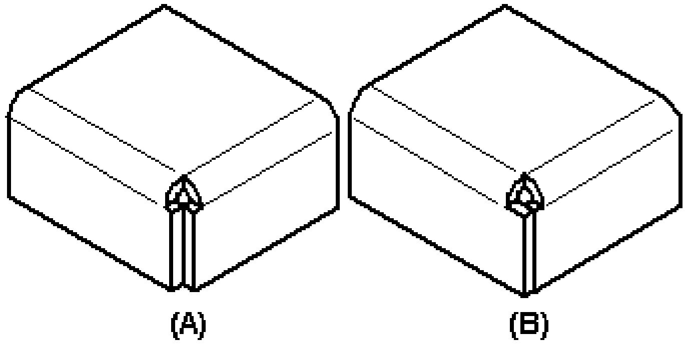
The Close 2–Bend Corner command modifies two flanges in one operation to close a corner where two flanges meet.



In the ordered environment, the Close 3-Bend Corner command closes corners that contain three bends.



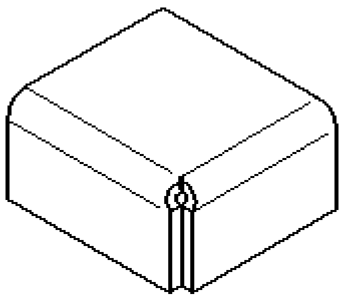
A closed corner is a treatment feature. You do not have to draw a profile, just select the edges you want to modify. With 2-bend corners, you can specify whether to close the corner (A), or overlap the corner (B).



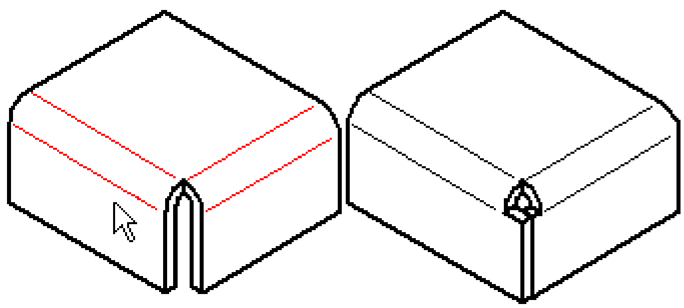
Note

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

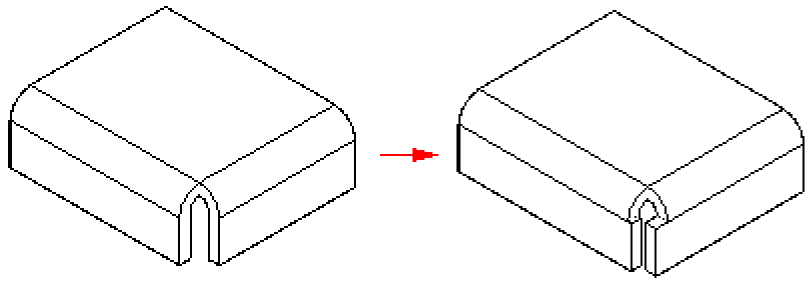
When you close the corner, you can also specify what type of bend treatment you want. For example, you can specify that you want a circular cutout applied to the bent faces.



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.

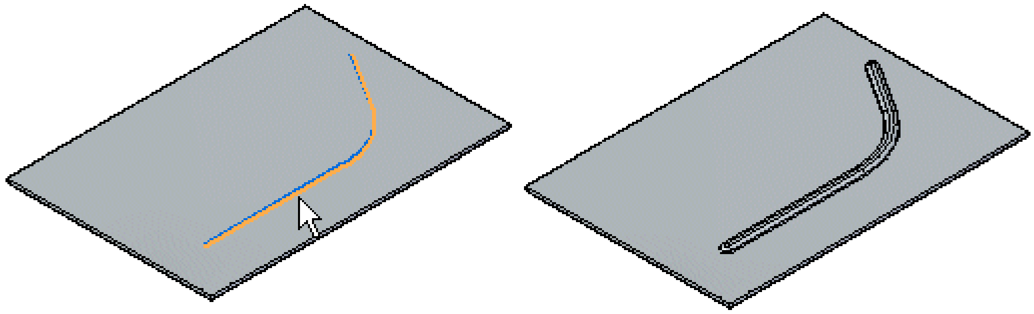


Note

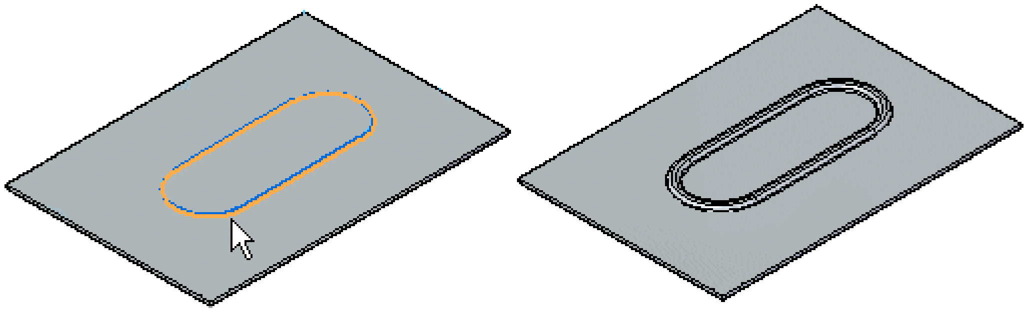
It is best to apply bend and corner relief before using the Close 2–Bends Corner command, so there is a clean corner to close. The corner should be symmetric, with equal bend radii and bend angles on the adjacent flanges. If there is more than one way to close the corner, edit the flanges themselves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

Constructing beads

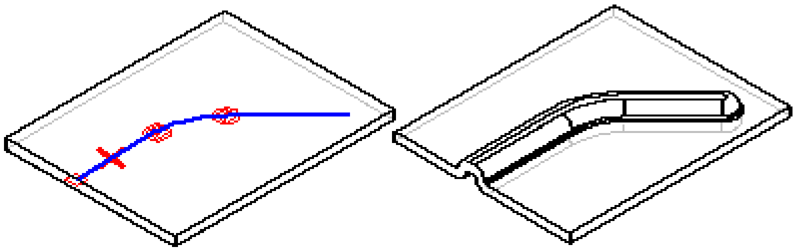
You can construct a bead with an open sketch element,



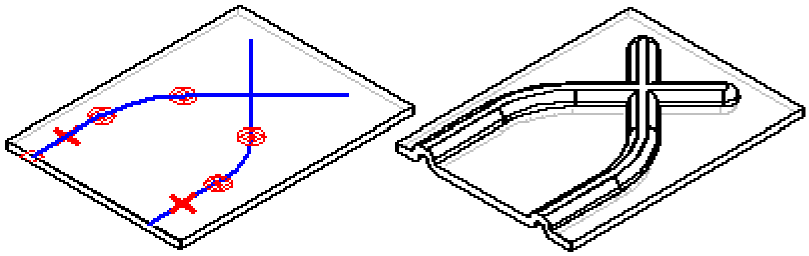
or closed sketch region.



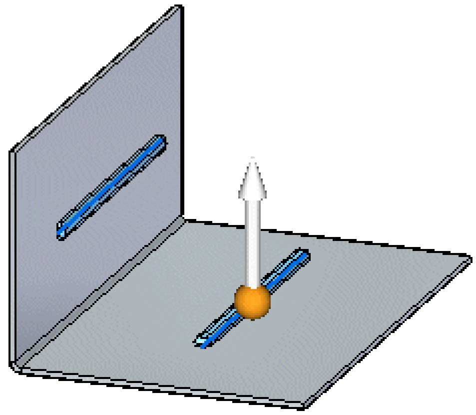
When constructing a bead profile using multiple elements, the element must be a continuous set of tangent elements.



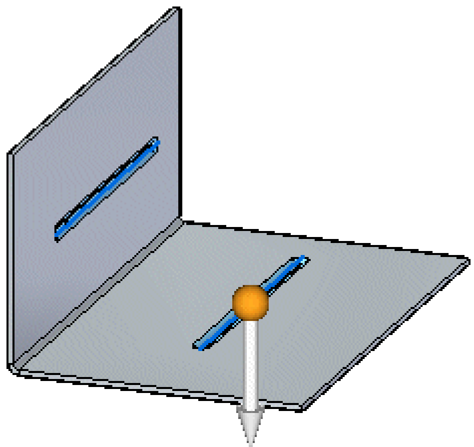
You can also construct a bead feature using multiple, separate sketch elements. Each element must be a continuous set of tangent elements, but the profiles can cross each other.



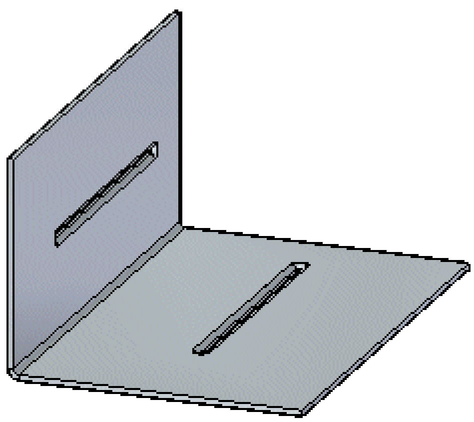
You can select more than one sketch element to construct multiple beads in a single operation.



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.

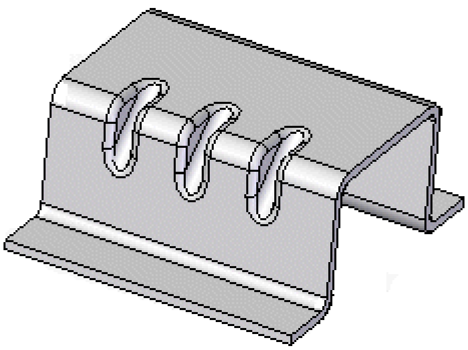


When constructing multiple disjoint beads, an entry in PathFinder, a feature profile, and a nail is created for each disjoint bead. Beads cannot be flattened and they cannot cross a bend.

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.



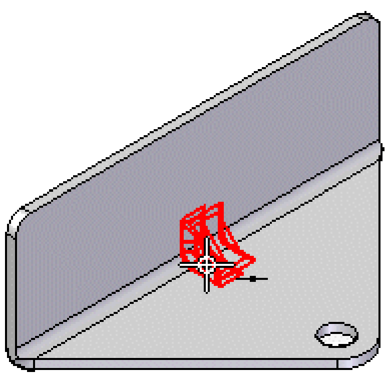
Note

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

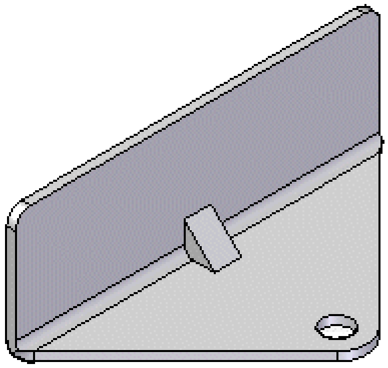
You can use the Gusset Options dialog box to specify the definition of the gusset. You can control such things as the shape of the gusset, the width and taper angle of the gusset, and the punch and die radius if the gusset is rounded. You can also use the dialog box to specify if the gusset is created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

Constructing gussets automatically in the ordered environment

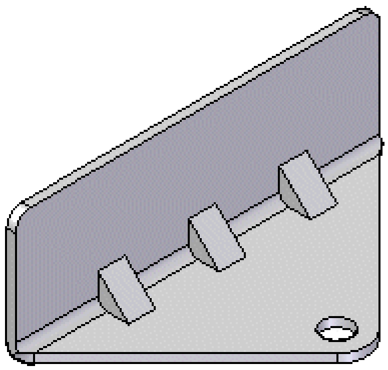
Select the Automatic Profile option on the Gusset Options dialog box to automatically create a gusset. Once you select a bend, the gusset profile is automatically displayed along the bend.



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.

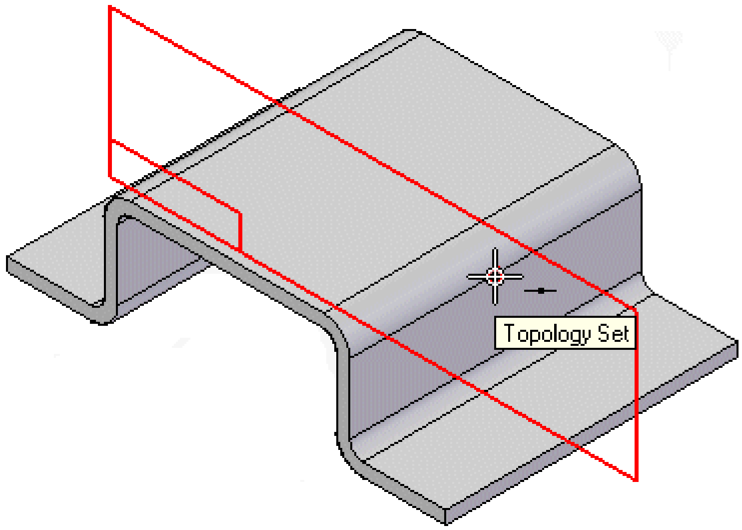


Constructing gussets from a user-drawn profile in the ordered environment

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:

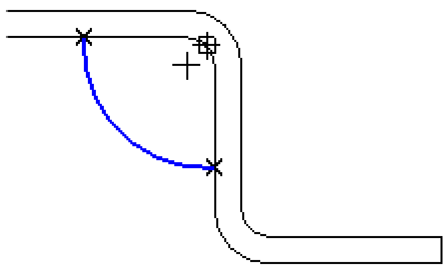
1. Click a keypoint to create a plane on which you want to draw the profile.



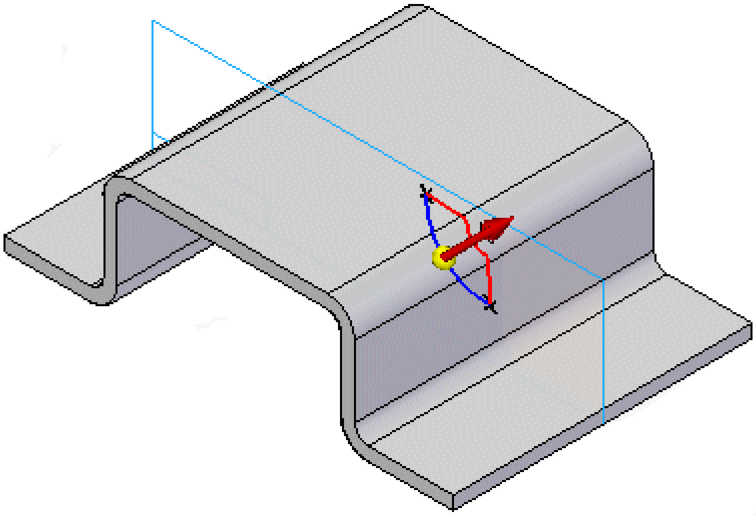
Note

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

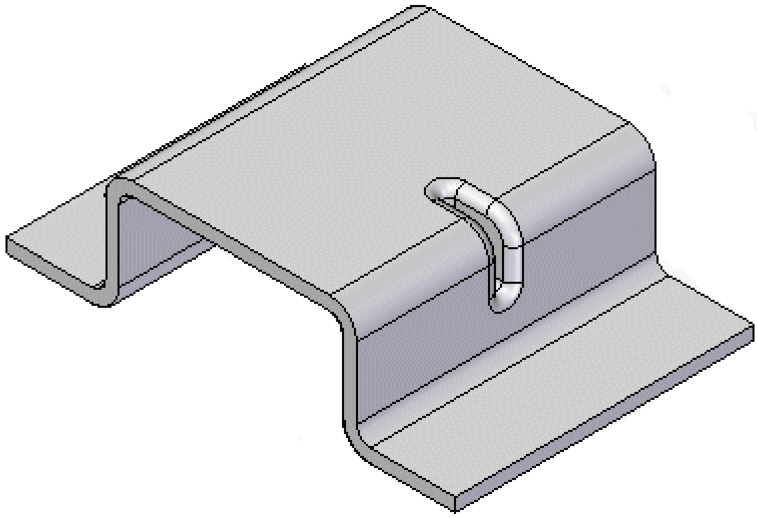
2. Draw the profile.



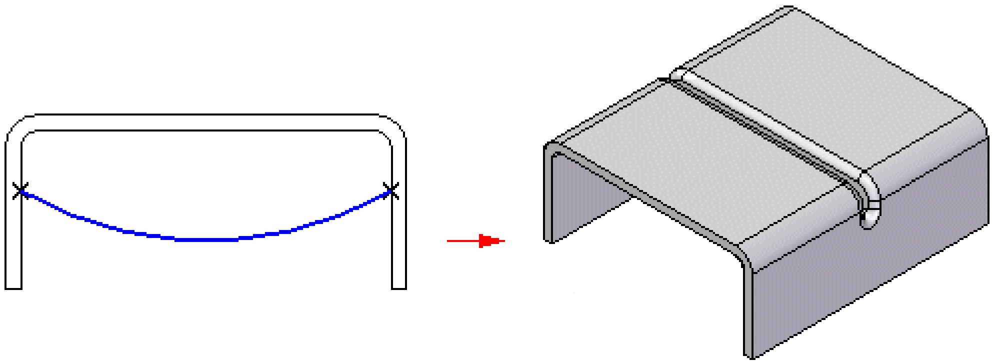
3. Click to define the direction of the gusset.



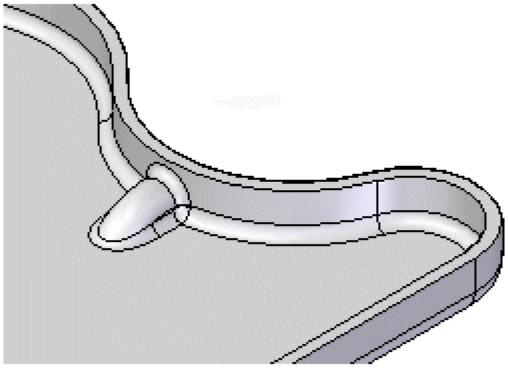
4. Click Finish to place the gusset.



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

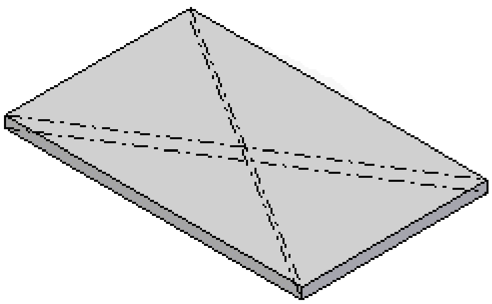


or, across a non-linear bend.



Constructing cross brakes

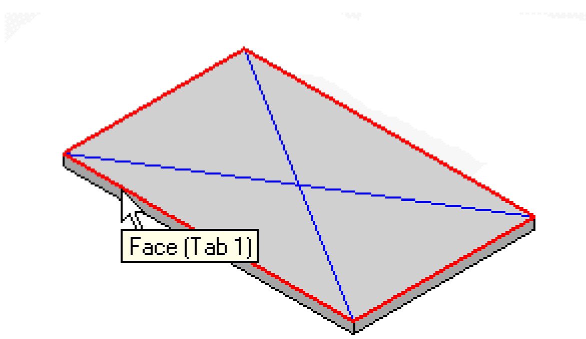
In the ordered environment, you can use the Cross Brake command to stiffen a sheet metal panel. The command creates a set of bends from a sketch that is coincident to the sheet metal part face.



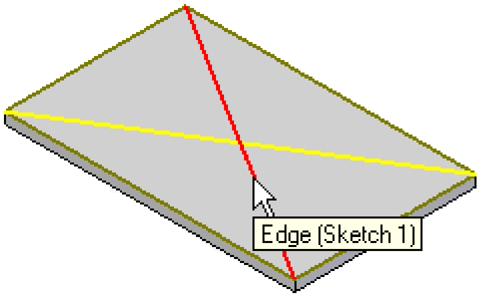
A cross brake feature does not deform the 3D model. It adds attributes containing information about the bends. This attribute information is used when creating a flat pattern or drawing of the sheet metal part.

To create a cross brake feature:

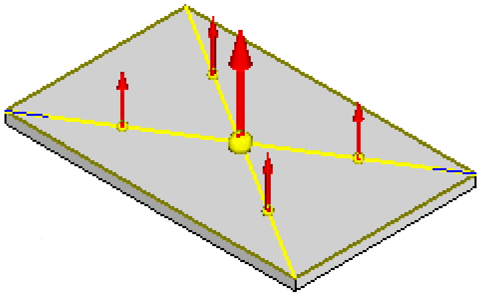
1. Select the face on which you want to construct the cross brake.



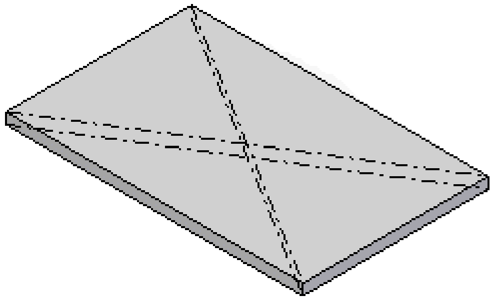
2. Select the sketch(es) you want to use to construct the cross brake.

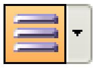


3. Specify the bend angle and direction for the cross brake.



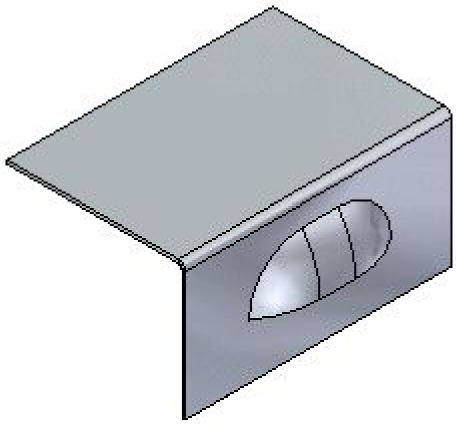
4. Click Finish to construct the cross brake.



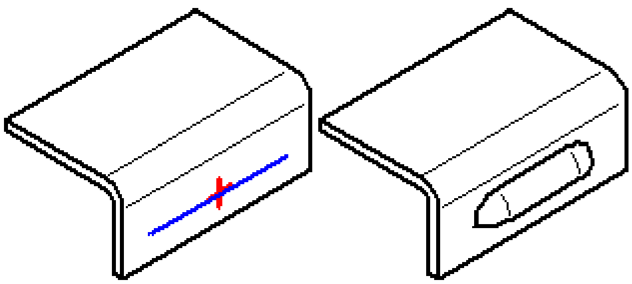


Louver command

Constructs a louver with lanced or formed ends.

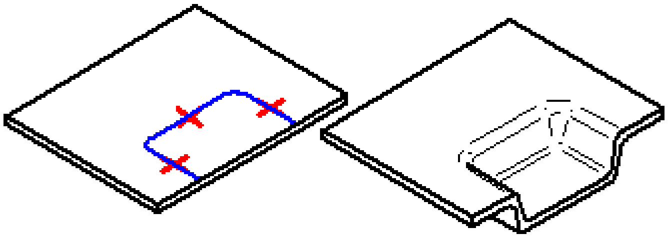


In the ordered environment, the profile for a louver feature must be a single linear element. Louvers cannot be flattened.



Dimple command

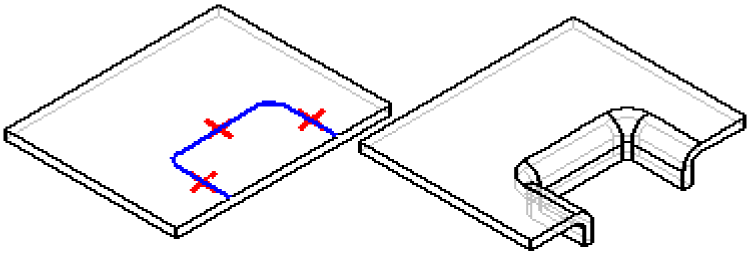
Constructs a sheet metal dimple from a selected region. If you use an open profile, the open ends of the profile must theoretically intersect part edges. Dimples are special die-formed features in which material deformation occurs. Dimples cannot be flattened.



Drawn Cutout command

Constructs a drawn cutout.

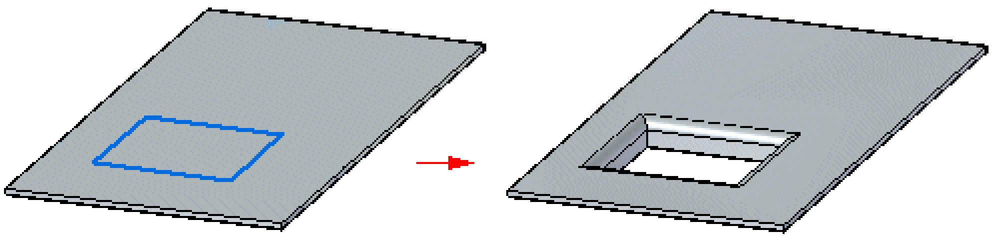
In the ordered environment, if you use an open profile, the open ends of the profile must theoretically intersect part edges. A closed profile cannot touch any part edges. Drawn cutouts cannot be flattened.



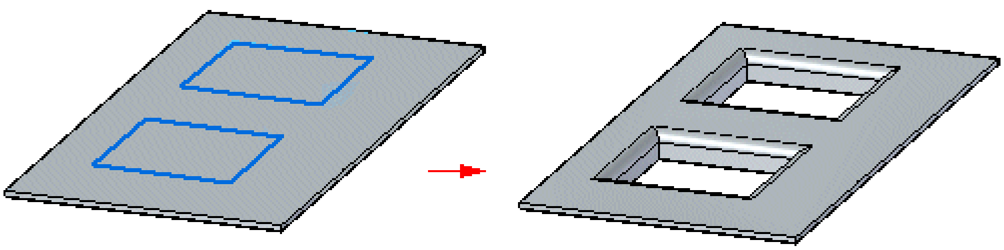
In the synchronous environment, the geometry used to create the cutout can be a closed internal profile that creates a region or an open profile extended to a part edge to create a closed region.

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

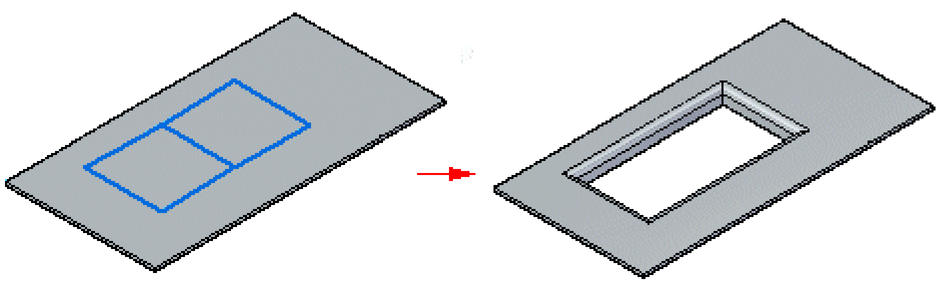
- Single



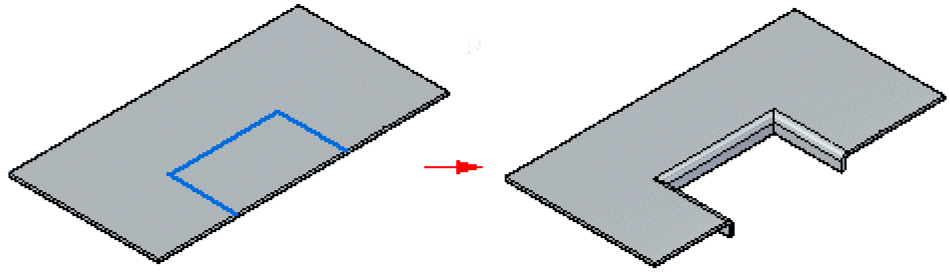
- Disjoint



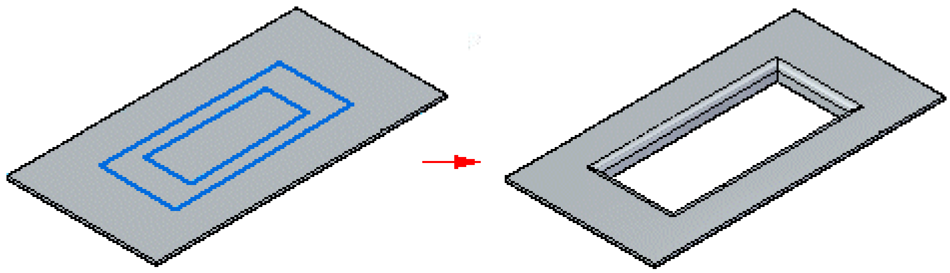
- Contiguous



- Coincident to a sketch or edge



- Nested contiguous

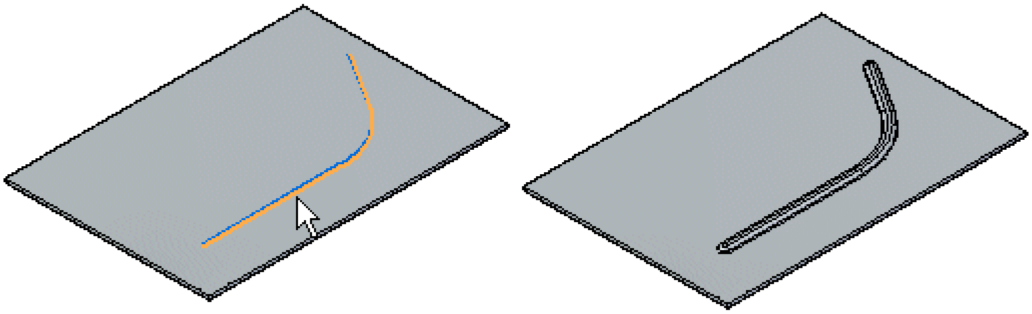


You can select multiple regions at one time and the regions must all lie on the same plane.

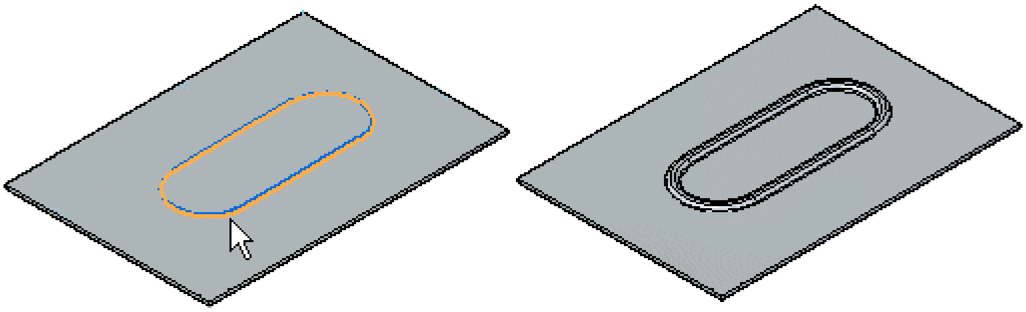
Bead command

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

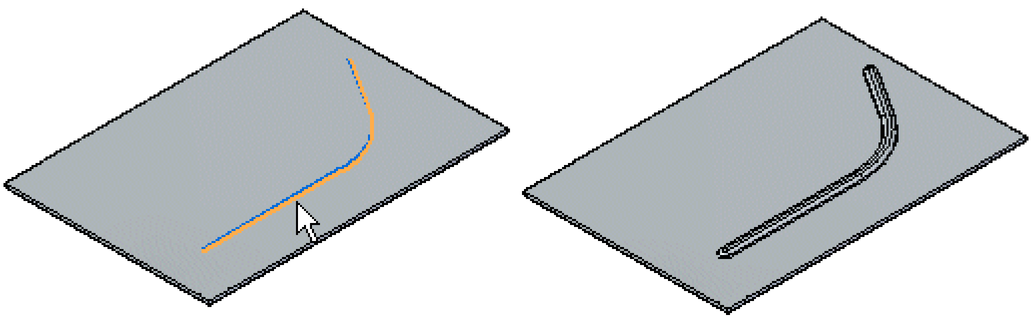
In the ordered environment, you can construct a bead with an open



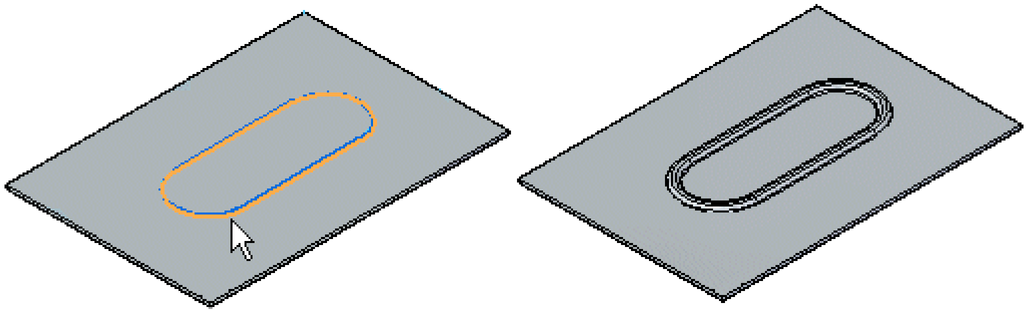
or closed profile.



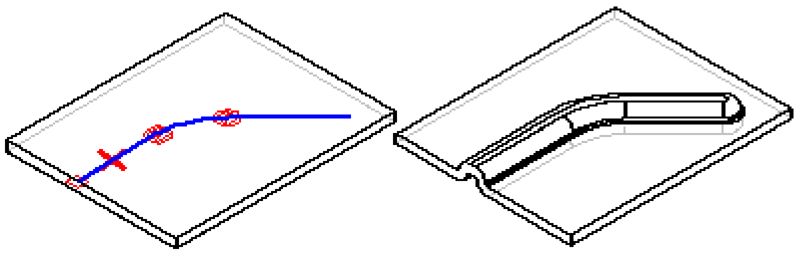
In the synchronous environment, you can construct a bead with an open sketch element,



or closed sketch region.

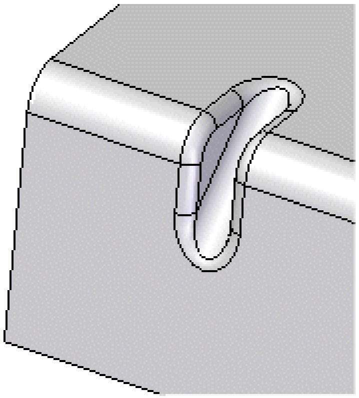


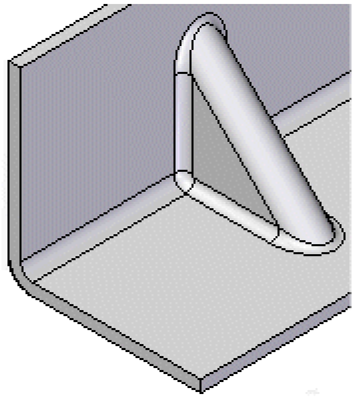
When constructing a bead profile using multiple elements, the element must be a continuous set of tangent elements.



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.

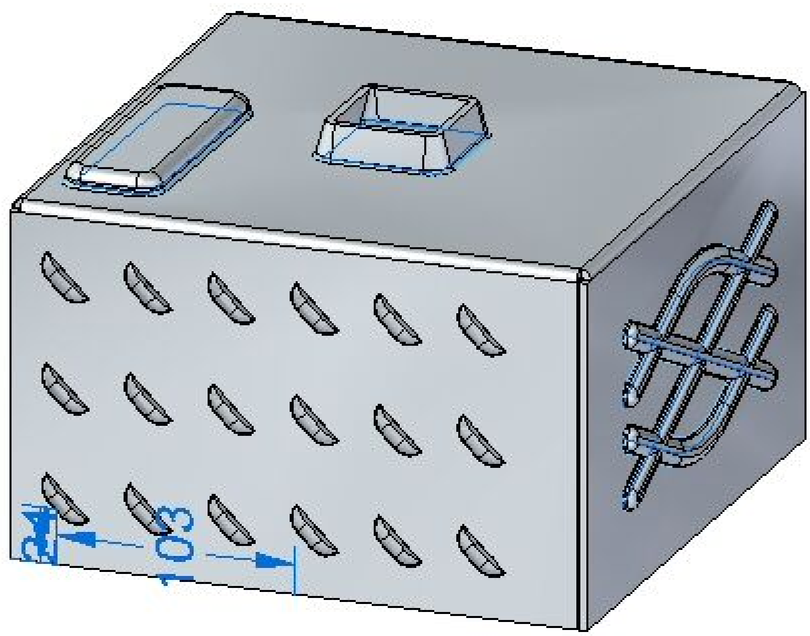
Shortcut menu commands are available to reposition the feature origin for a feature.

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.



Activity: Deformation Features

Open a sheet metal file

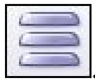
- Start Solid Edge ST4.
- Click the  **Application** button ® **Open** ® *deformation_activity.psm*.

  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.

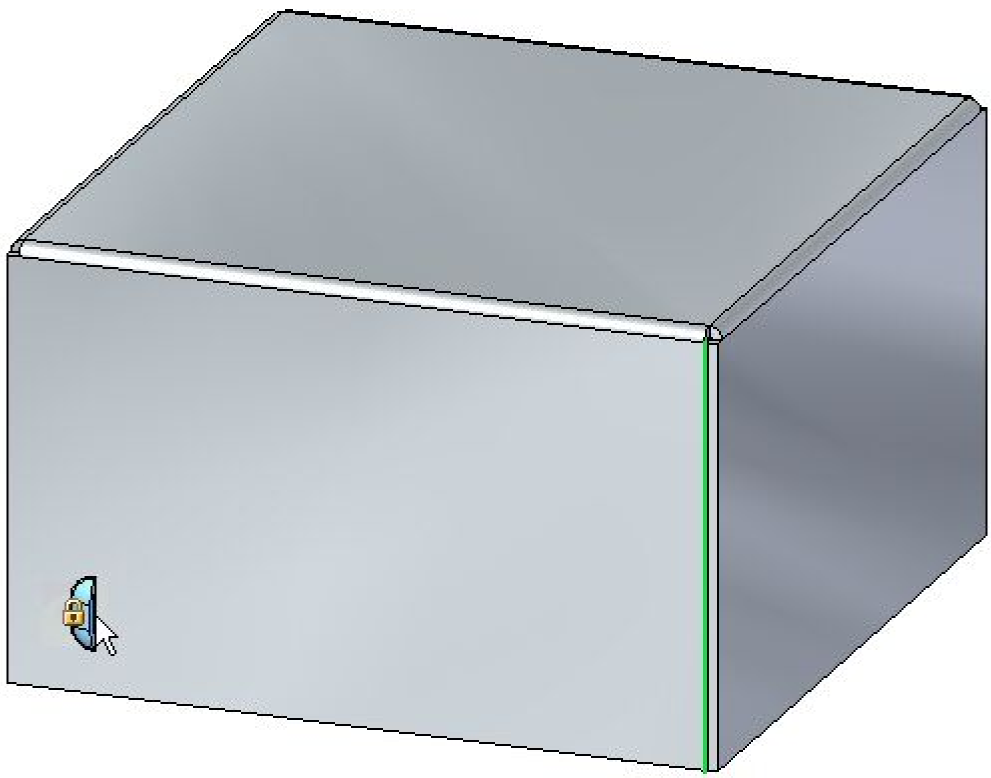
  Note

  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.

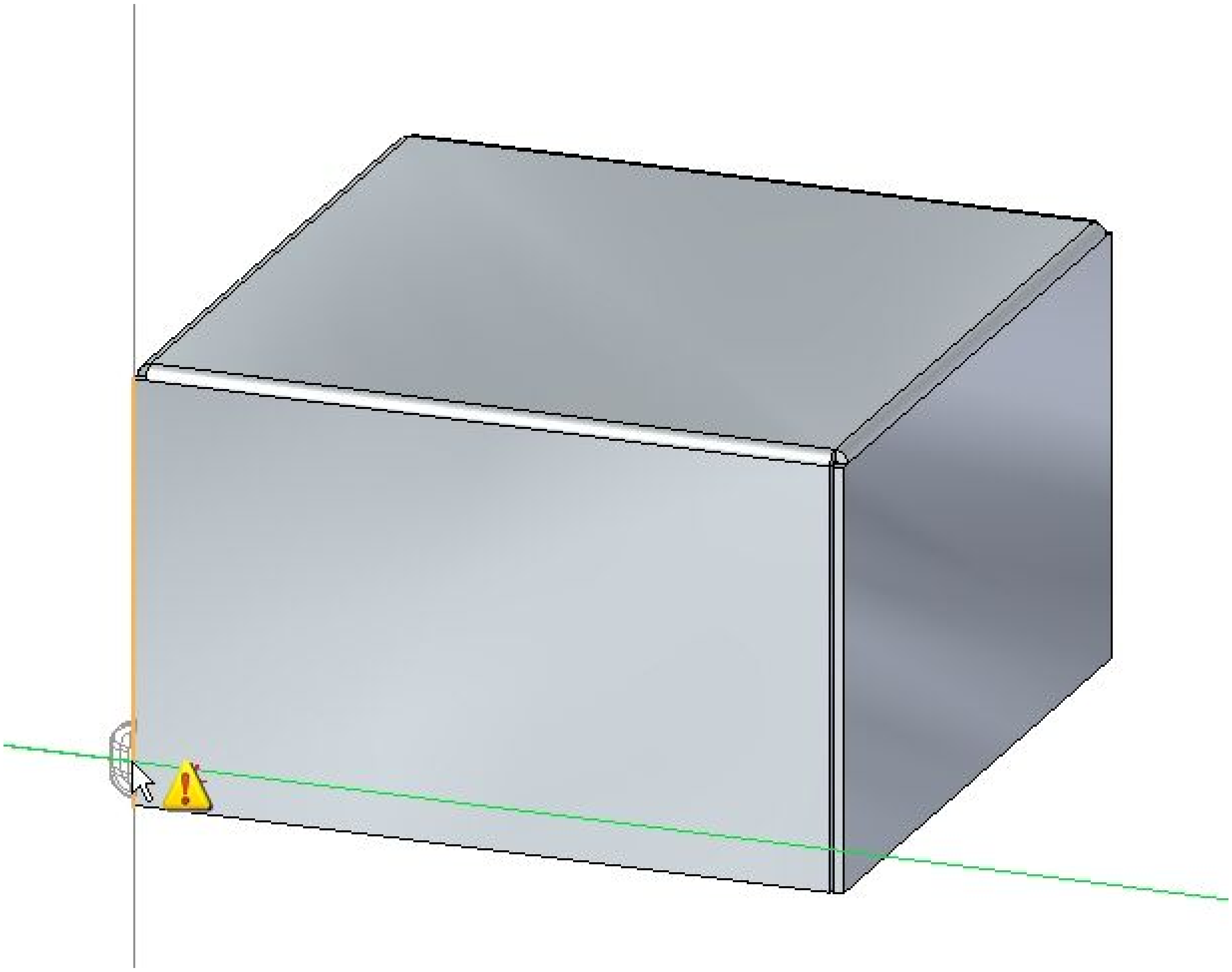
Orient the louver as shown by entering N as needed. Once the orientation is established, enter F3 to lock to the face and orientation.

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.



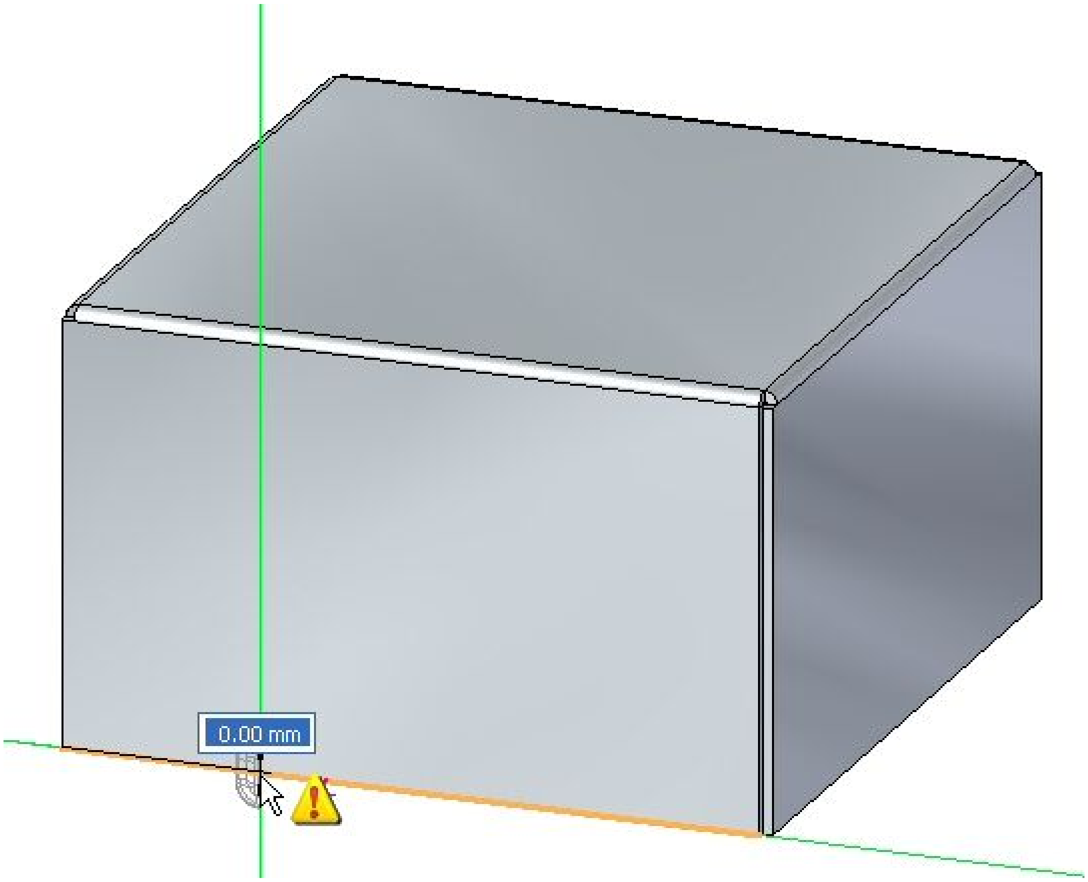
- Move the cursor over the edge shown below and press the **E** key.



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.

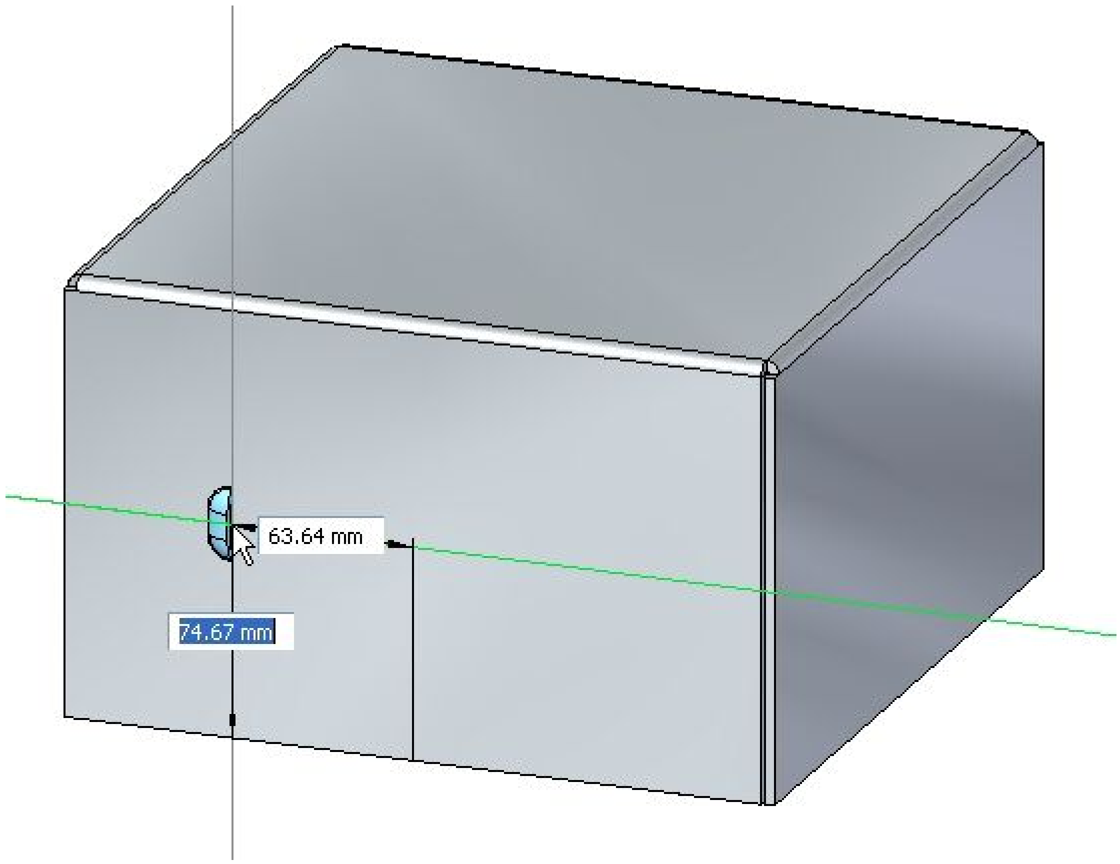
- Move the cursor over the edge shown below and enter M from the keyboard.



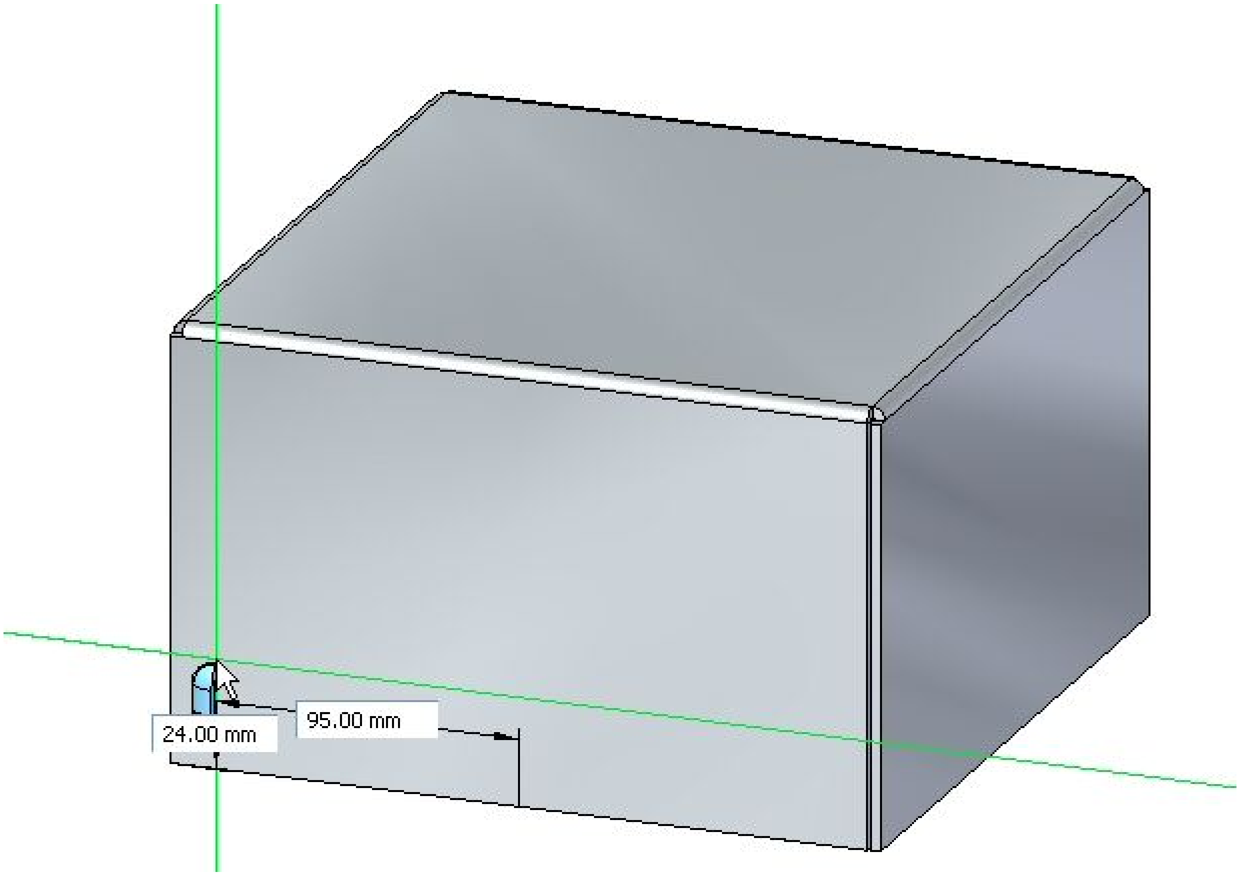
Note

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

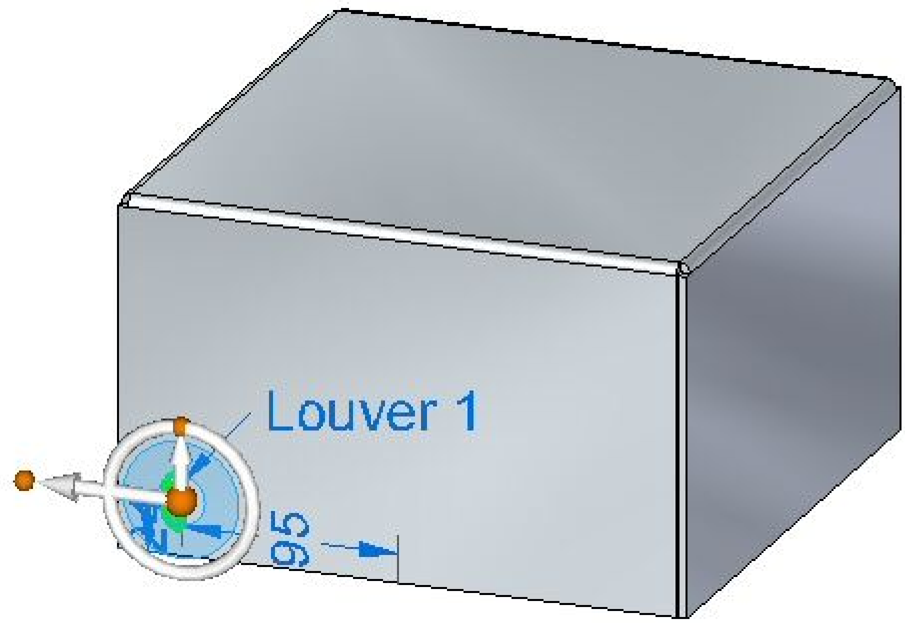
- Without clicking the mouse, move the cursor to the approximate position shown below.



- 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.

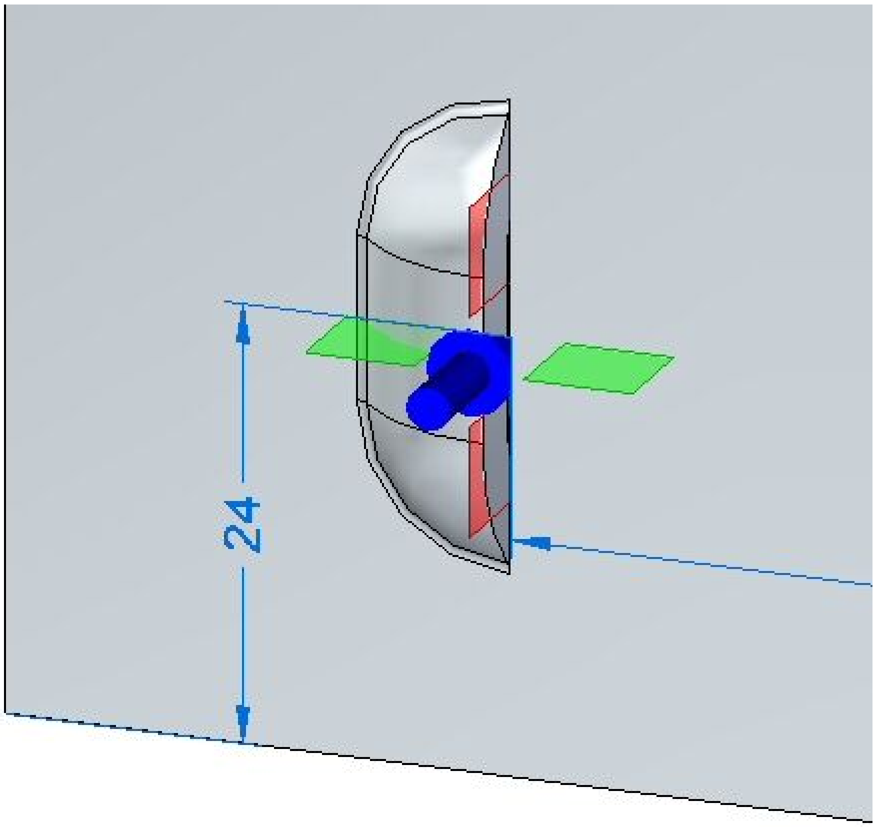


- Proceed to the next step.

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.

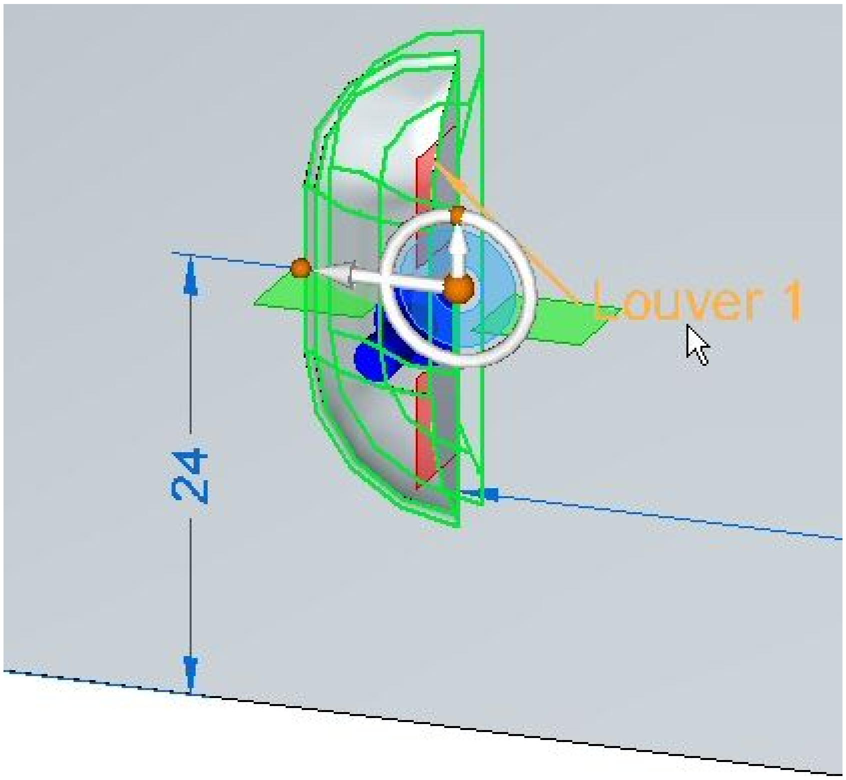
- In PathFinder right-click and select Show Feature Origin.



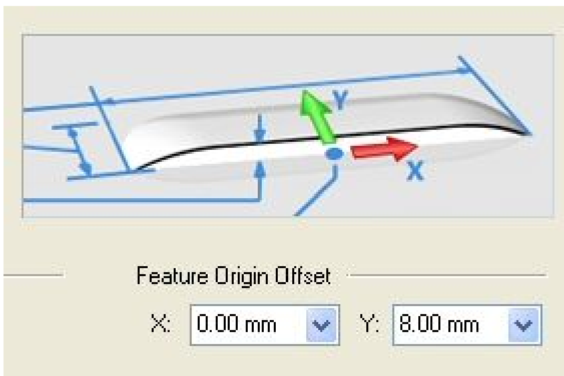
The feature origin appears.

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

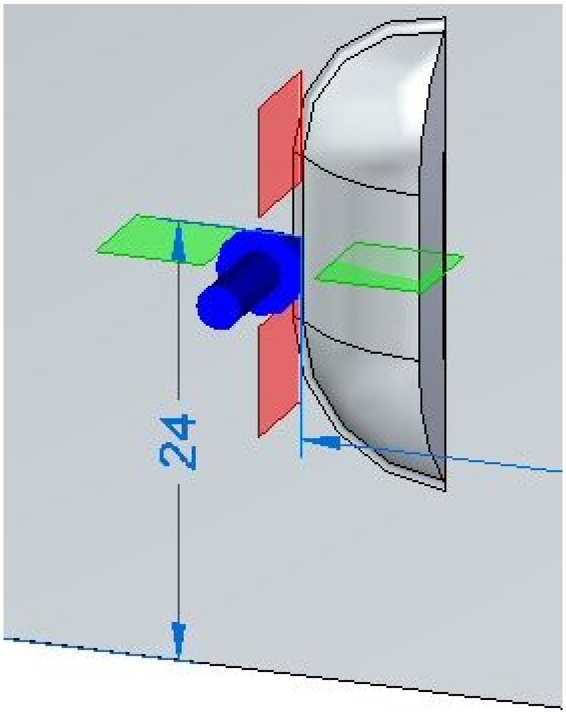
- Click the louver feature in PathFinder. When displayed, click the edit handle as shown.



- 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.

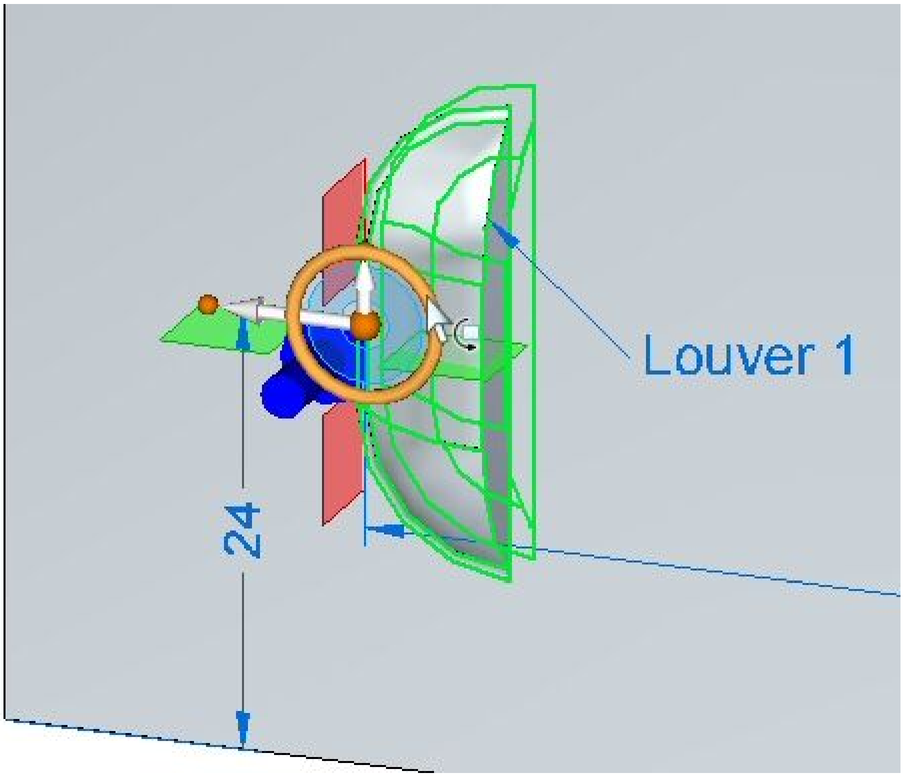


Note

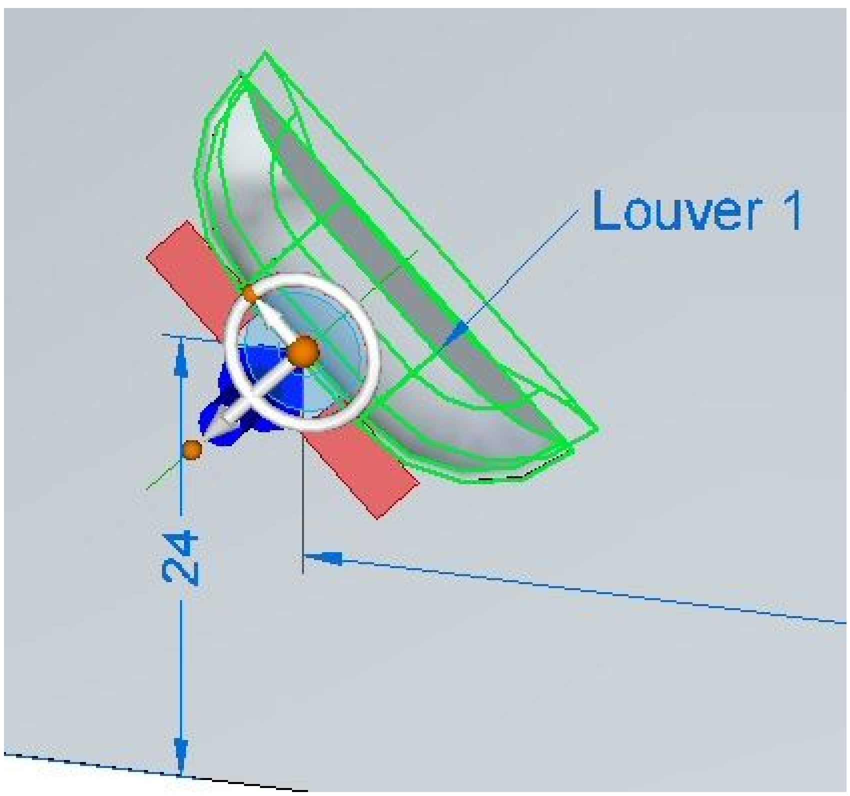
Notice that only the feature origin changed position. The louver is still located in the same position.

Now the louver will be rotated 45 degrees.

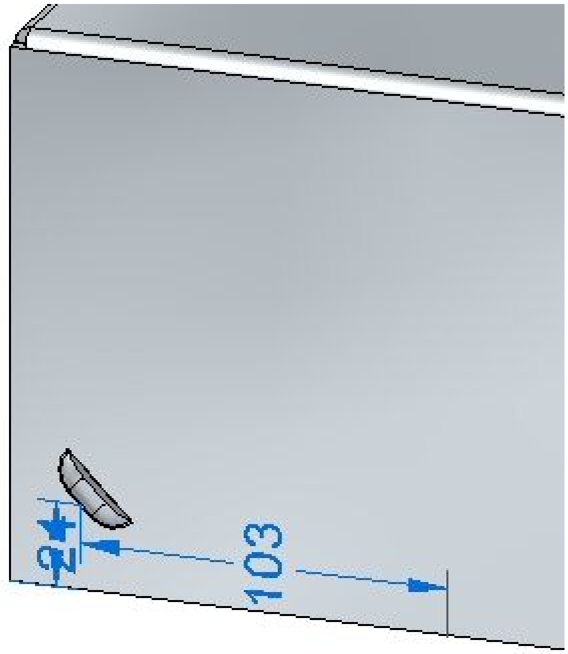
- Select the louver, then select the torus of the steering wheel.



- Rotate the louver 45º as shown.

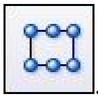


- The louver has been rotated about the feature origin. Right-click the louver in PathFinder and hide the feature origin.



- Proceed to the next step.

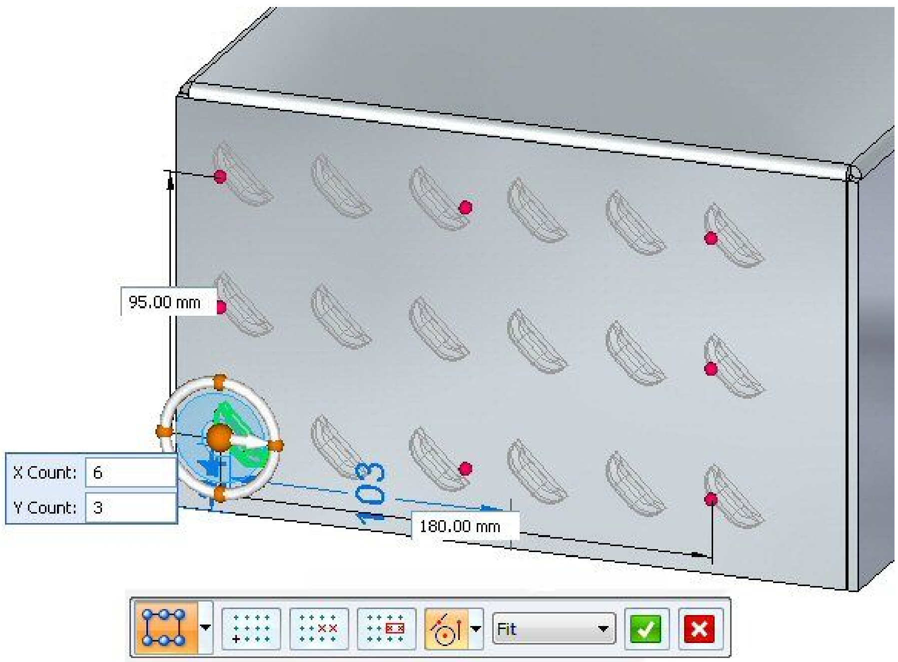
Patterning the deformation feature

- 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.

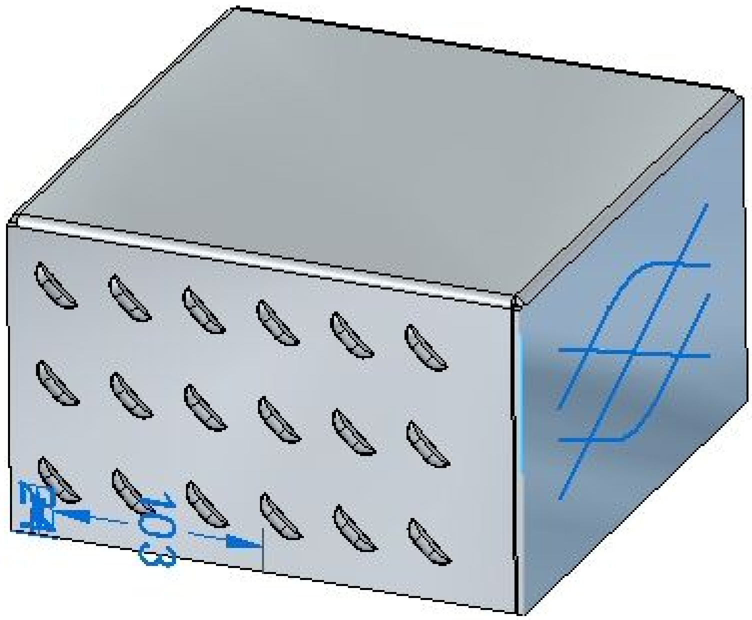


- Click the accept button to finish creating the rectangular pattern.
- Proceed to the next step.

Placing Beads

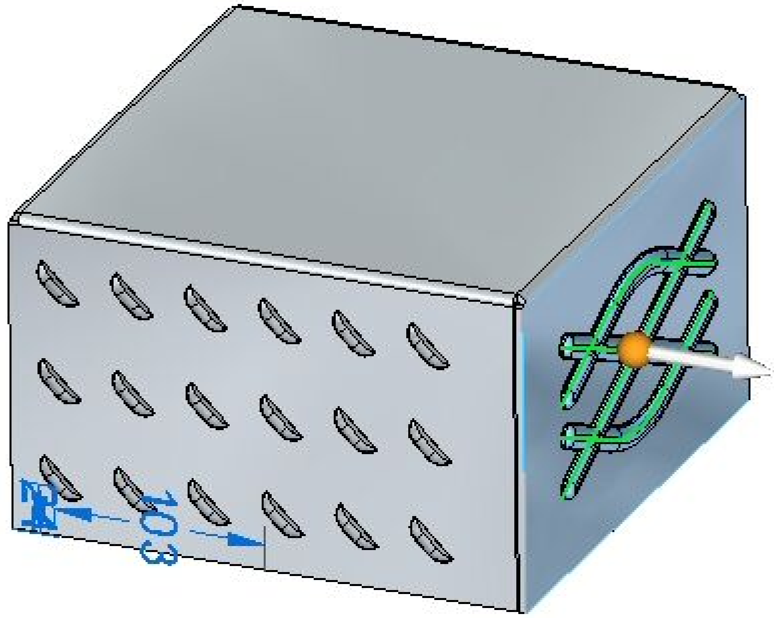
Beads are added to sheet metal parts as stiffeners.

- 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: 20º.
  - End Condition: Formed.
  - Include rounding with a punch radius of 0.50 mm, and a die radius of 0.50 mm.
  - Click OK.

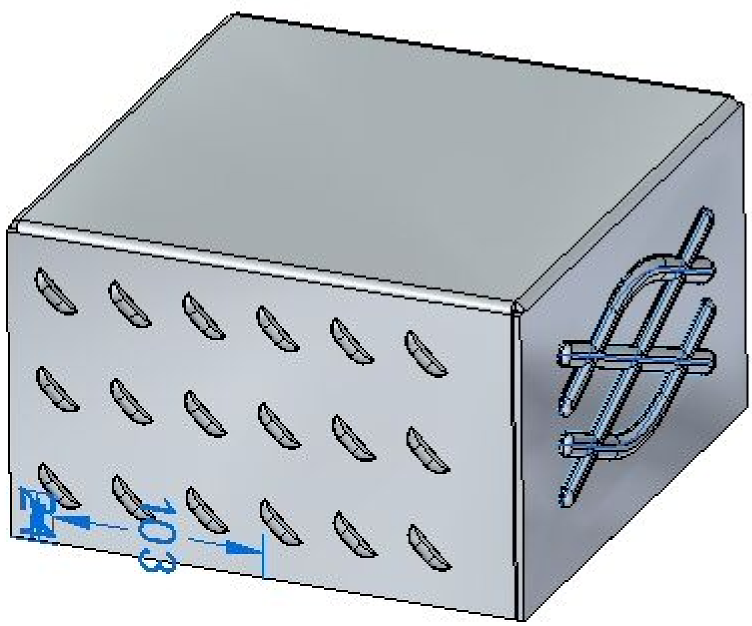
- Select all the elements in the sketch as shown.



Note

Clicking the arrow will reverse the direction of the beads.

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

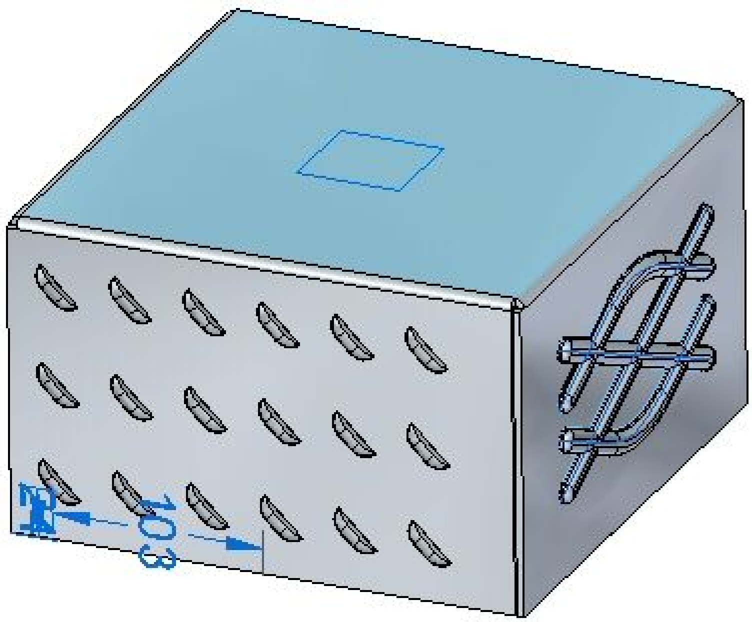


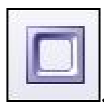
- Proceed to the next step.

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.



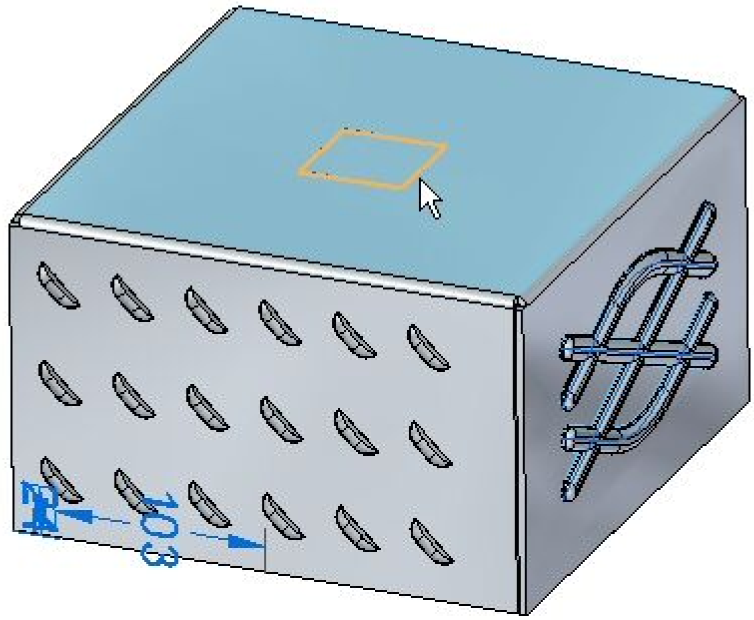
‣ Click the Drawn Cutout command .

‣ Click the Drawn Cutout Options command .

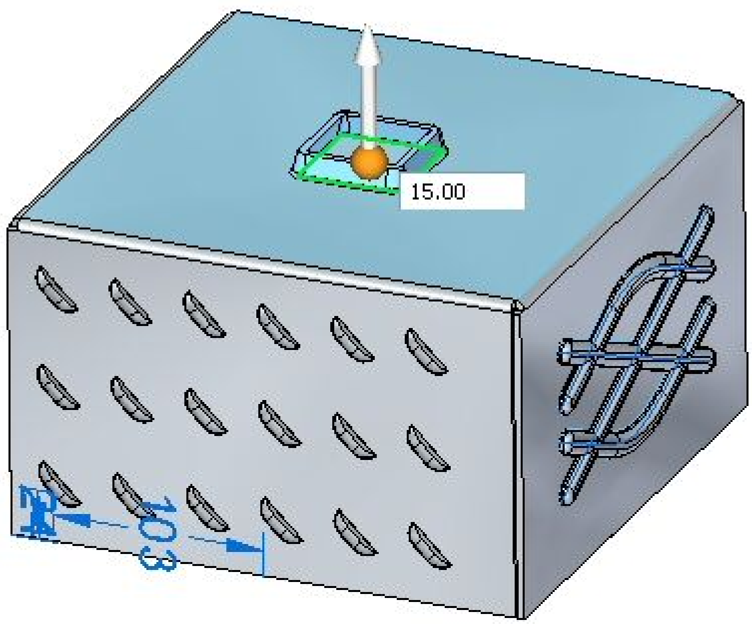
‣ Set the following options:

- Taper angle: 15º.
- Include rounding: die radius 1.75 mm.
- Include punch side corner radius: 1.75 mm..
- Click OK.

- Select the region shown.



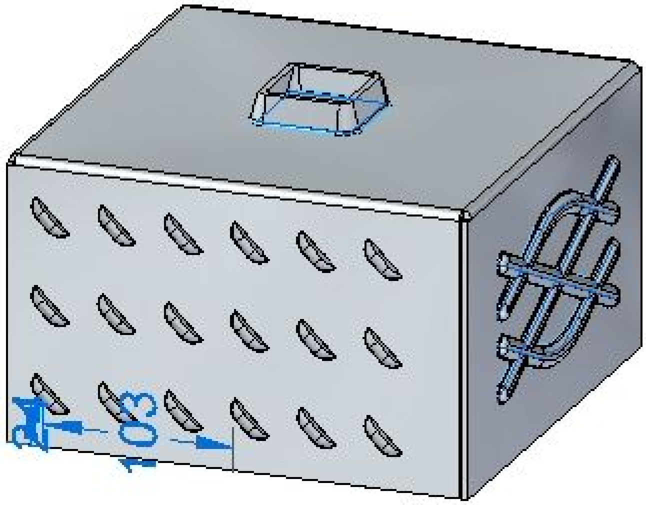
- Enter a distance of 15.00 mm.



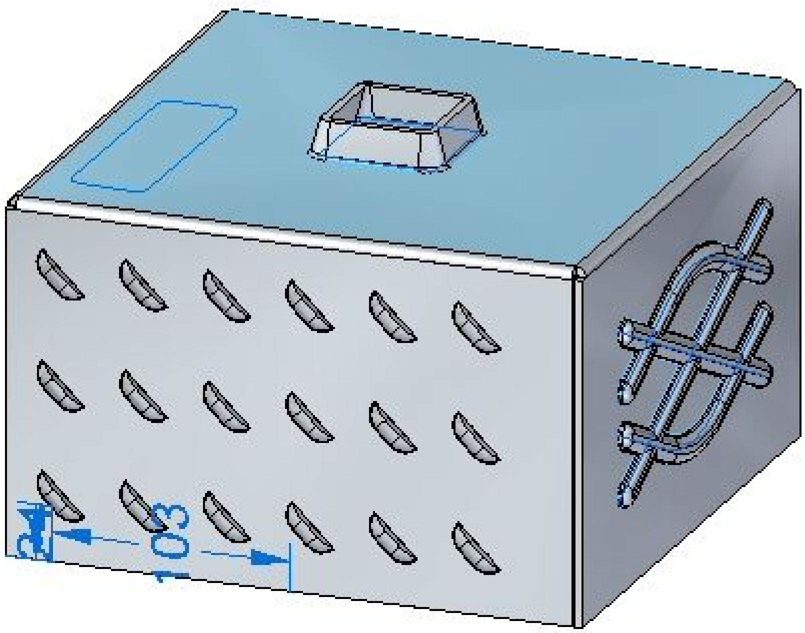
Note

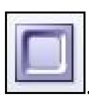
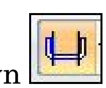
Clicking the arrow will reverse the direction of the drawn cutout.

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

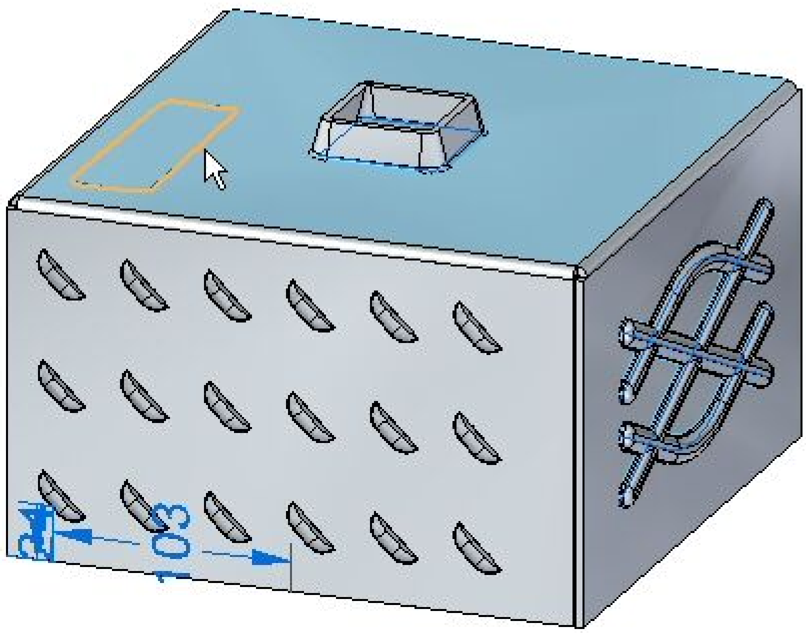


- In PathFinder, display the sketch named Dimple.

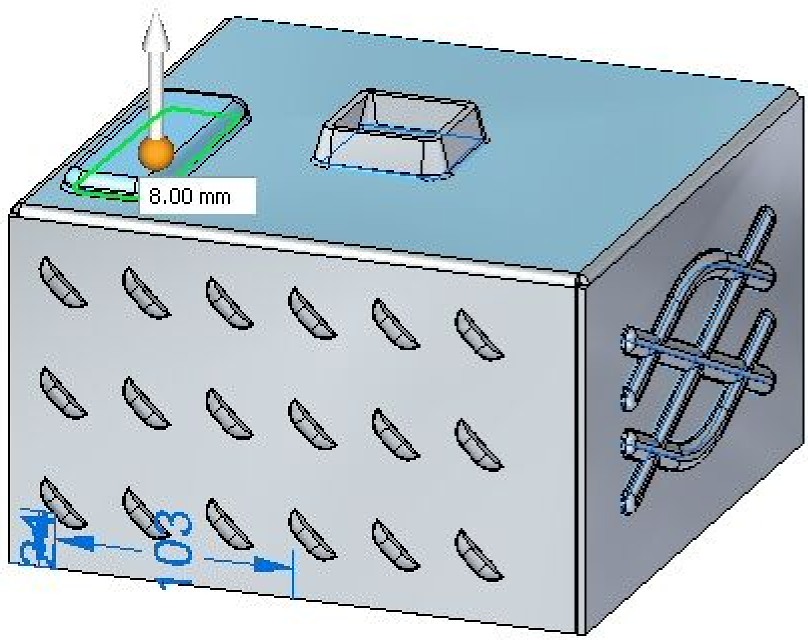


- Click the Dimple command .
- Click the Dimple Options button .
- Observe the options, but do not change any of the options.
- Change the sketch profile to represent punch as shown 

- Select the regions shown.



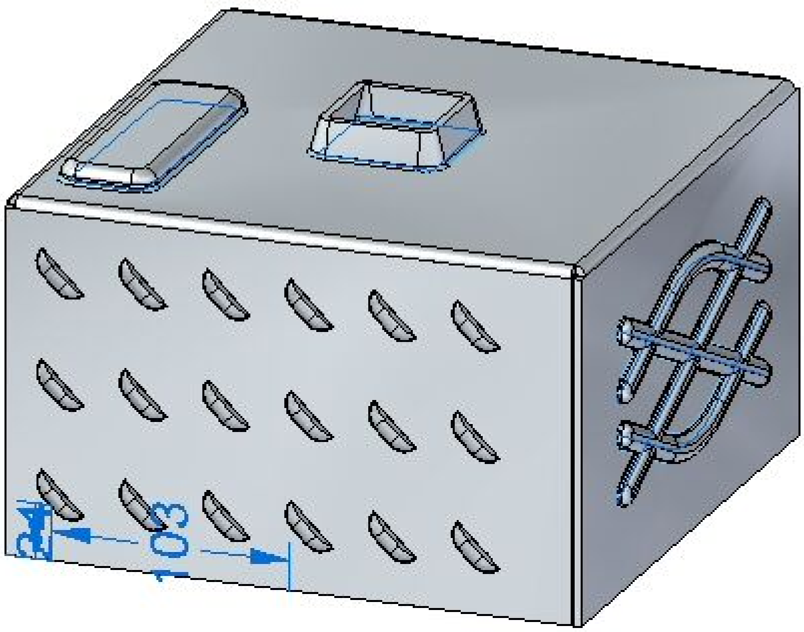
- Enter a distance of 8.00 mm.



Note

Clicking the arrow will reverse the direction of the dimple.

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

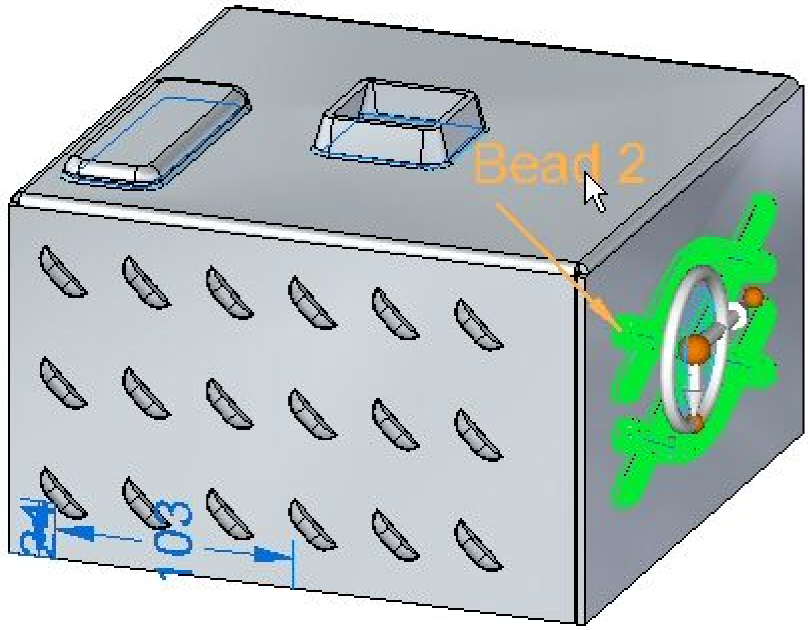


- Proceed to the next step.

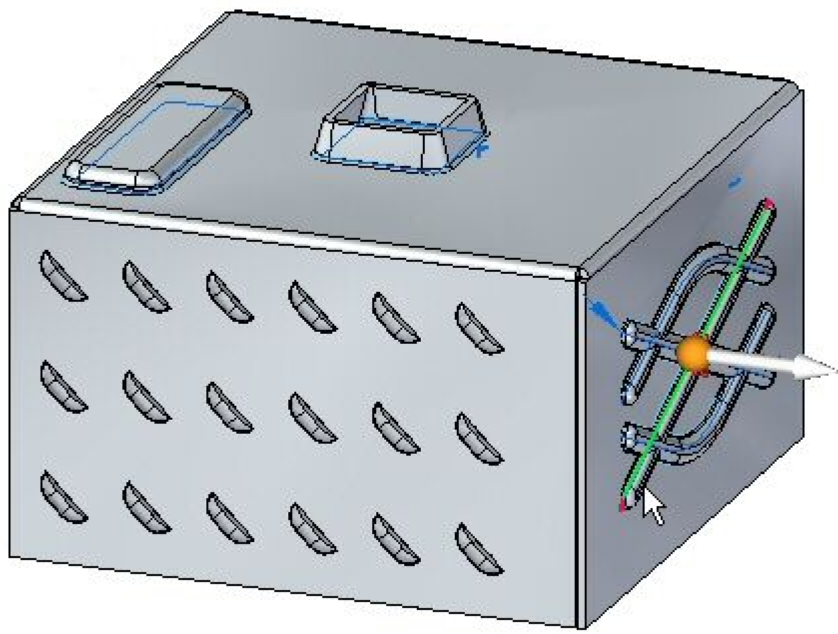
Editing deformation features

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

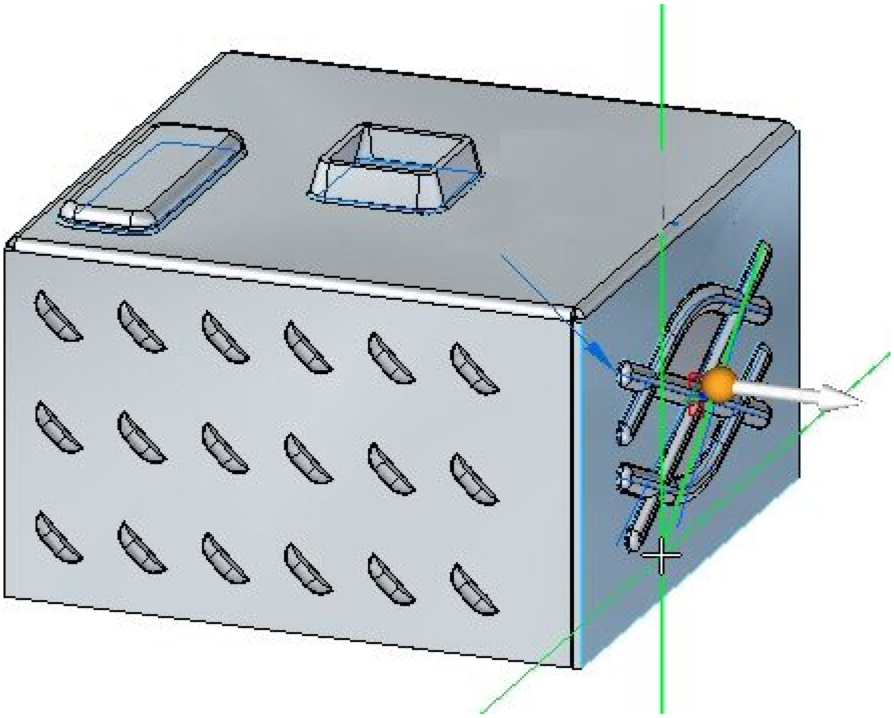
- In PathFinder, select the bead feature. Click the edit handle to edit the feature.



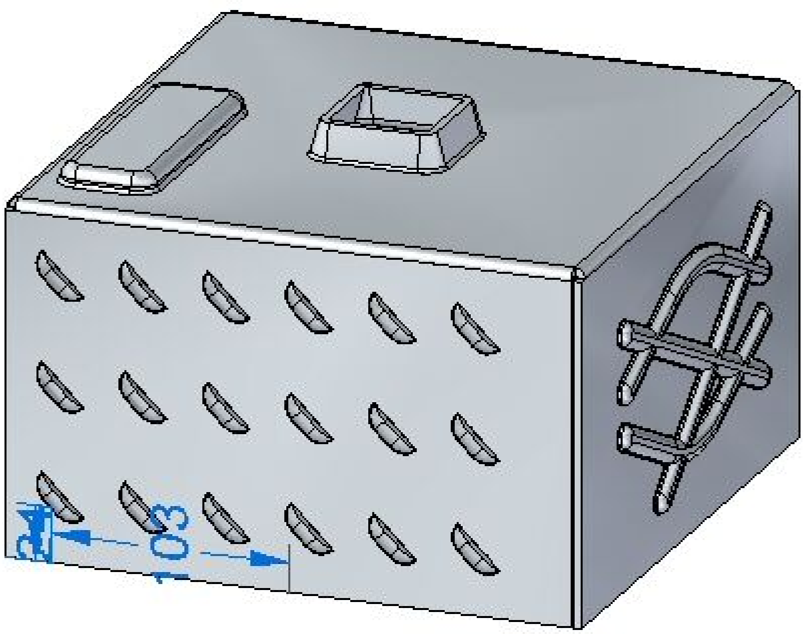
- Click the Edit Profile Handle.
- Select the line shown.



- 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

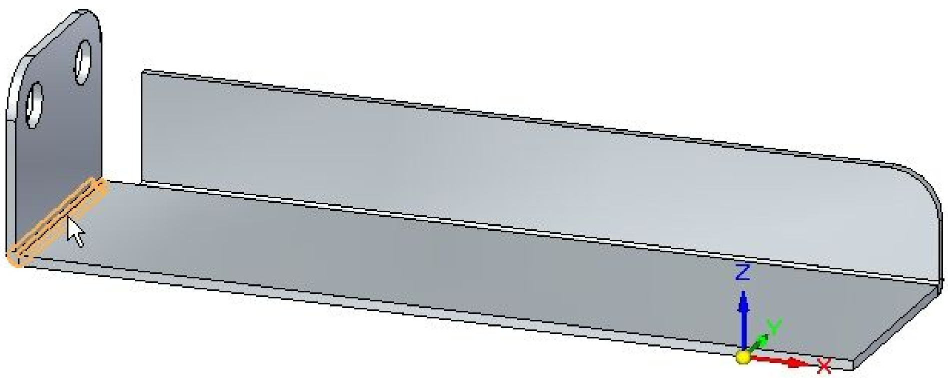
When editing a rigid procedural feature, the sketch used to create the feature remains a part of the feature and can be modified at a later time.

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

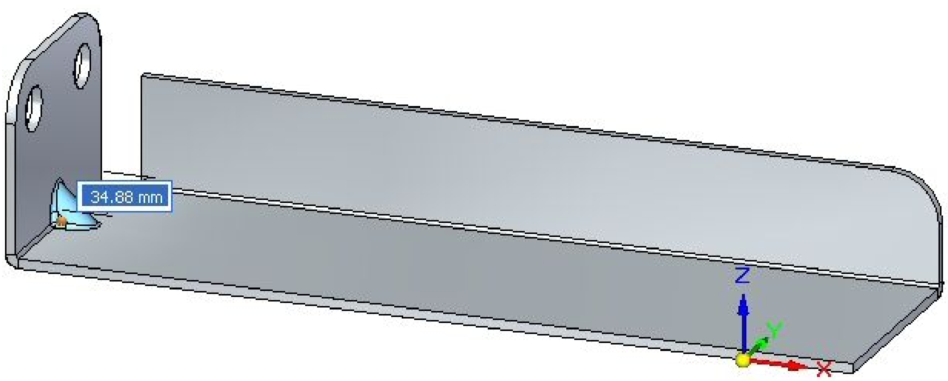
Placing a single gusset

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

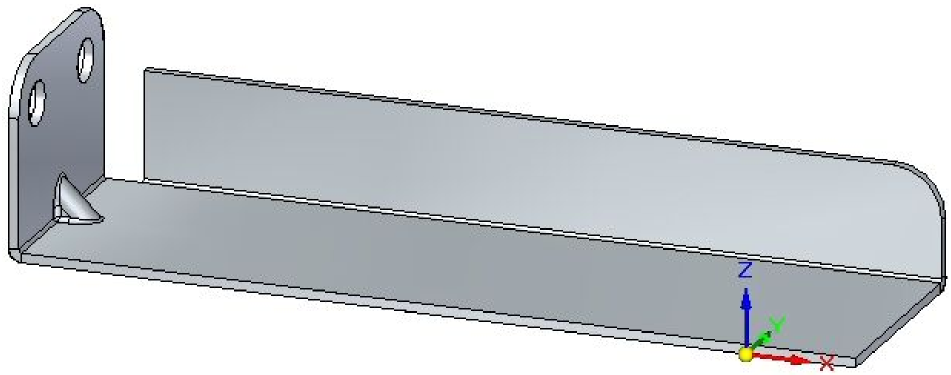
- Click the  **Application** button ® **Open** ® *gusset_activity.psm*.
- Click the Gusset command .
- Click the Gusset Options button .
- Set the following parameters:
  - Depth: 11.25 mm.
  - Include rounding with the both the punch and die radius being 1.50 mm.
  - Set the gusset shape to round.
  - 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.

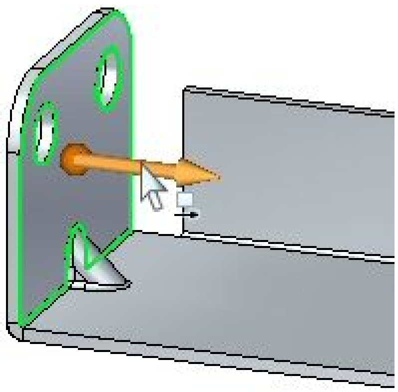


- Proceed to the next step

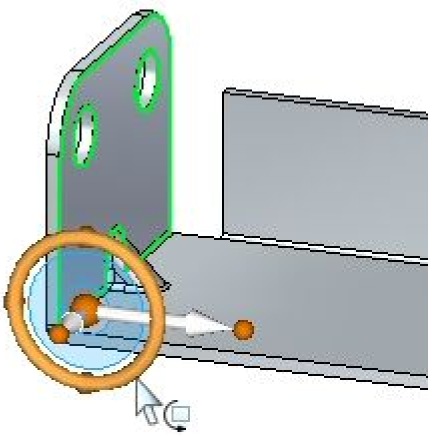
Rotating faces containing gussets

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

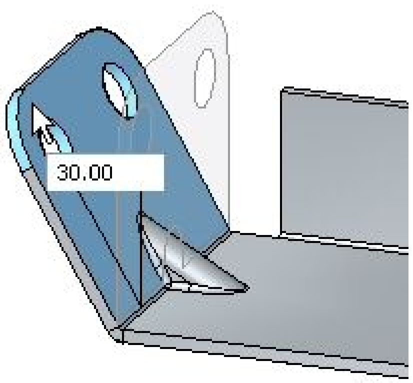
- Click the Select tool and select he face shown.



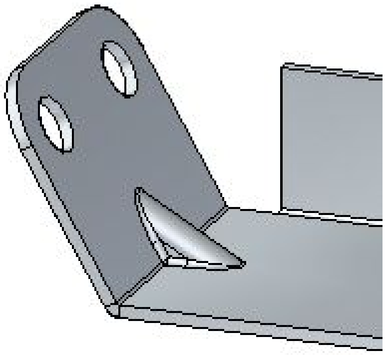
- Move the steering wheel to the bend as shown and select the torus so as to rotate the face.



- Enter an angle of 30º as shown.



- The result is as shown.



Note

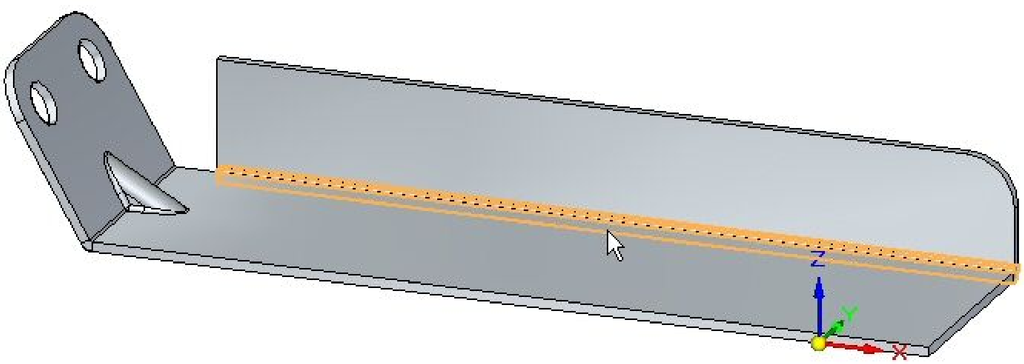
The gusset is an adaptive procedural feature that will change shape to as the angle between the faces changes.

- Proceed to the next step.

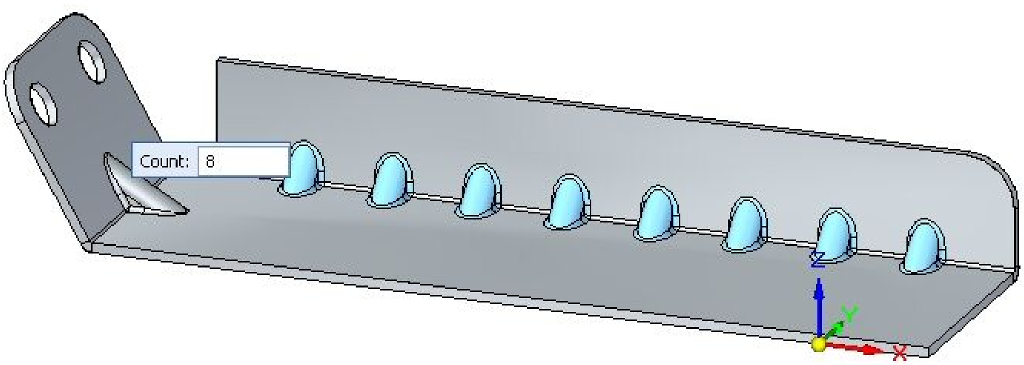
Placing a gusset pattern

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

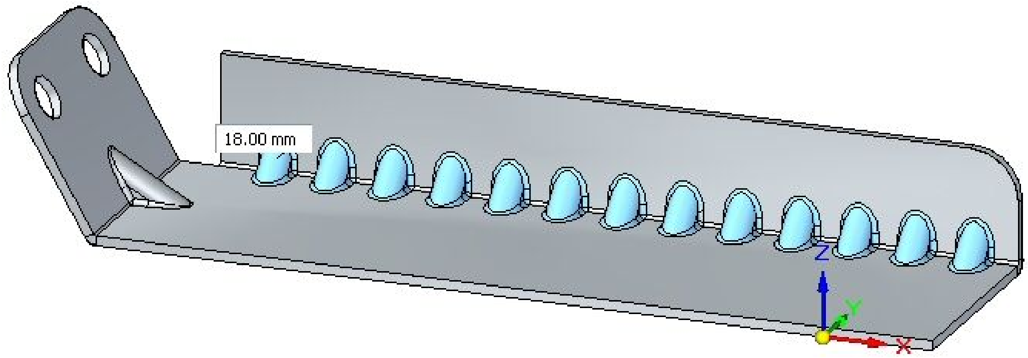
- Click the Gusset command .
- Click the Gusset Options button .
- Set the following parameters:
  - Depth: 11.25 mm.
  - Include rounding with the both the punch and die radius being 1.50 mm.
  - Set the gusset shape to round.
  - Set the width to 9.00 mm. Click OK.
- Set the gusset patterning parameter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



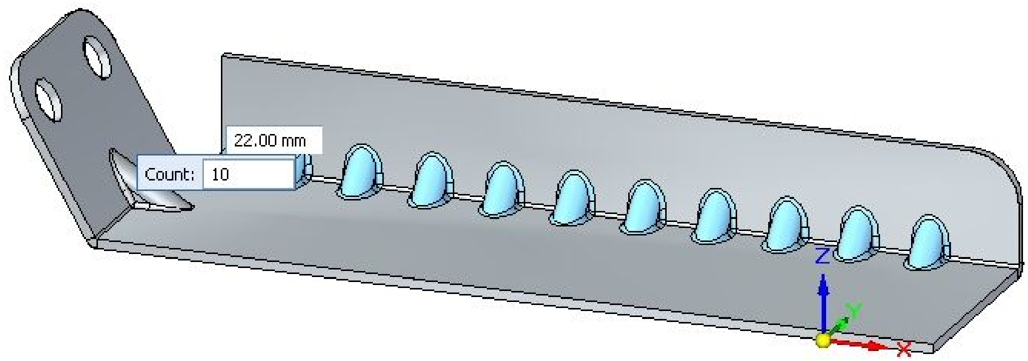
- set the count to 8. Observe the results.



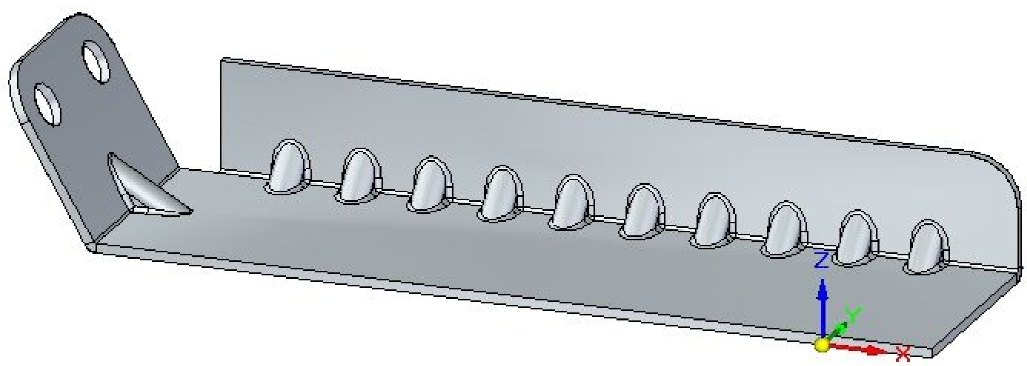
- Set the pattern type to fill. Observe the results.



- 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 where created with the pattern command.

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 where created with the pattern command.

Lesson

11 *Modeling synchronous and ordered features*

Modeling synchronous and ordered features

In a Solid Edge modeling document, two environments coexist for creating model features. The two environments are synchronous and ordered. You create synchronous features in the synchronous modeling environment. You create ordered features in the ordered modeling environment. A model can contain only synchronous features, only ordered features, or a combination of both feature types.

A synchronous feature is a collection of faces that define the feature shape. There is no history retained of how a synchronous feature was created. You can edit the faces of a synchronous feature.

An ordered feature is history based. You can edit an ordered feature by returning to any step used in the feature creation process. You do not edit faces of an ordered feature.

Opening a Solid Edge modeling document

- The Solid Edge Options® Helpers page provides a setting for the modeling environment to use when a new document opens. The default setting is Synchronous modeling.

- If an existing modeling document contains only synchronous elements, the document opens in the synchronous environment.

- If an existing modeling document contains only ordered elements or a combination of ordered and synchronous elements, the document opens in the ordered environment.

Moving between modeling environments

You can switch between environments at any time during the modeling process.

- Right-click in PathFinder or the graphics window to activate the shortcut menu, and then choose either *Transition to Synchronous* or *Transition to Ordered*, depending on the environment that is active.

- If a model contains both synchronous and ordered features, click the Ordered environment bar or the Synchronous environment bar in PathFinder.

- On the ribbon, from the Tools tab® Model group, choose the modeling environment to transition to.

  Note

  Each environment presents its own set of modeling commands.

Feature display

In the ordered modeling environment, ordered and synchronous features appear.

In the synchronous modeling environment, only synchronous features appear.

Editing features

In ordered modeling, selecting an ordered feature displays the Edit Feature command bar for ordered editing.

In ordered or synchronous modeling, selecting a synchronous body face displays the steering wheel for synchronous editing.

Moving ordered features to synchronous

You can convert ordered features to synchronous features while in a part or sheet metal modeling file. The conversion performs by moving ordered features into the synchronous portion of the PathFinder tree. This move results in the feature geometry consuming into the synchronous body and therefore available for synchronous editing.

The move to synchronous workflow occurs only when the file is in the ordered environment. Single features or any number of features can convert with the Move to Synchronous command.

The ordered conversion is one way only. Synchronous features cannot convert to ordered features.

> **Note**
>
> You can also convert ordered features to synchronous features at a file level with the Convert command. Multiple files can process simultaneously.

Feature conversion must start at the top of the ordered feature tree and be in a contiguous order. All features in the tree above the selected feature include in the conversion. Mirror and pattern features require both child and parent features for conversion to be successful. If any of the parents in the select set have a child relationship to either a mirror or pattern feature, all features above these children features are in the select set.

If a problem occurs in the conversion process, the Undo command is available.

A Move to Synchronous dialog displays to alert the users if additional dependency found, and to provide any warning message that may affect the outcome of the move. This dialog only displays when warnings exist and or there are additional dependencies found.

Warning message: Feature dependency found. It is recommended that all dependencies be moved with the selected feature.

You can click the “Selection only” button in the dialog to exclude the dependencies from the Move operation.

> **Note**
>
> It is recommended to recompute the ordered node, and resolve any possible warnings or failures before moving the ordered features to synchronous.

Moving local dimensions and sketches

When ordered local dimensions move to synchronous, Solid Edge attempts to locate and bind dimensions to a vertex. If no vertex is found, then the dimensions become dangling dimensions. Once moved, all ordered dimensions, except for dangling dimensions, display along with the synchronous dimensions in the Dimensions node of the Synchronous portion of PathFinder. All ordered dimensions that are driving or driven dimensions move as driven dimensions. When you do a Move to Sync for a feature at a time,Solid Edge creates a user defined set each time there are dangling dimensions.

Note

Synchronous does not support dimensions between a part edge and a reference plane. Therefore, dimensions placed between an ordered part edge and a reference plane move to synchronous as dangling dimensions.

Local profile sketches in ordered convert as used sketches when they move to synchronous. The profile sketch name in synchronous is the same as the ordered feature name.

Activity: Creating ordered features

This activity guides you through the process of creating ordered features. Learn how to switch between modeling environments.

Activity: Creating ordered features

Create a new part document

When creating a new part document, you can control the environment to begin modeling in. The Solid Edge Options dialog provides a setting to start in Synchronous or Ordered environment. The default setting is Synchronous environment.

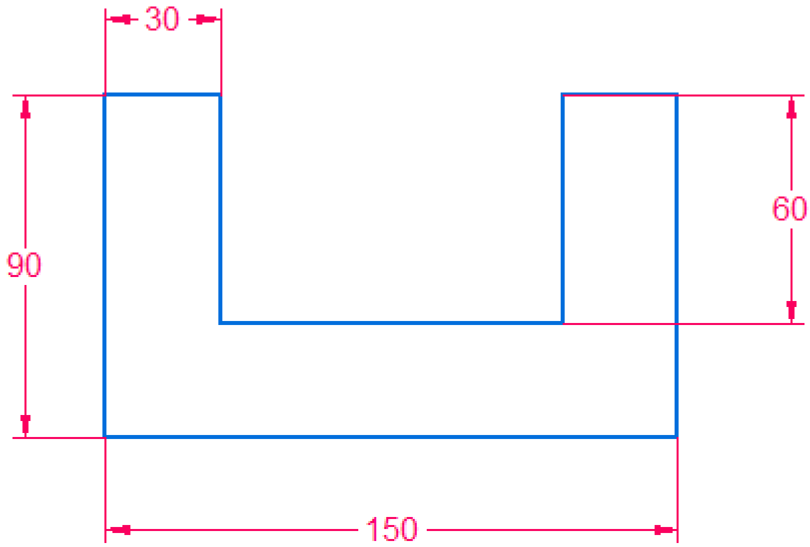
Note

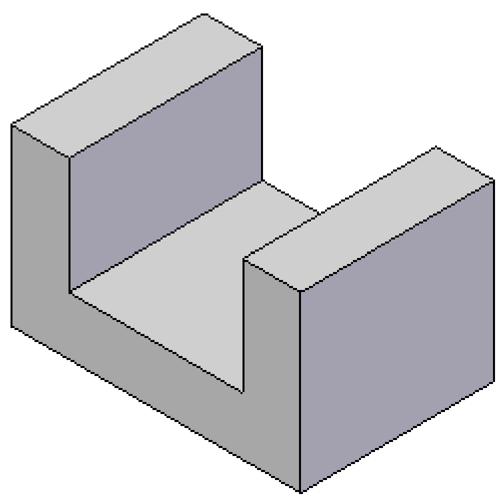
Existing files that contain only synchronous elements, open in the synchronous environment. Existing files that contain only ordered elements or a combination of ordered and synchronous elements, open in the ordered environment.

- Start Solid Edge ST4.
- On the Start-up page, click the  Application button.
- Click Solid Edge Options.
- On the Solid Edge Options dialog, click the Helpers page.
- On the Helpers page, under *Start Part and Sheet Metal documents using this environment:*, click the Ordered button. Click OK.
- On the Start-up page, under Create, click ISO Part.

Create an ordered base feature

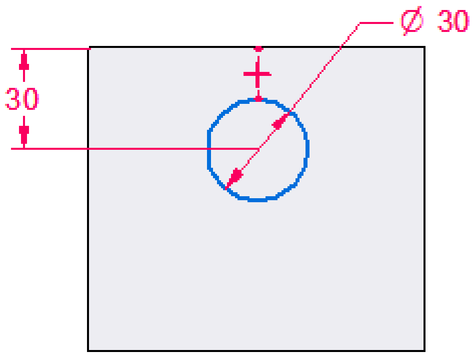
- Create an extrusion with the cross-section shown. Extend symmetrically at a distance of 100 mm.

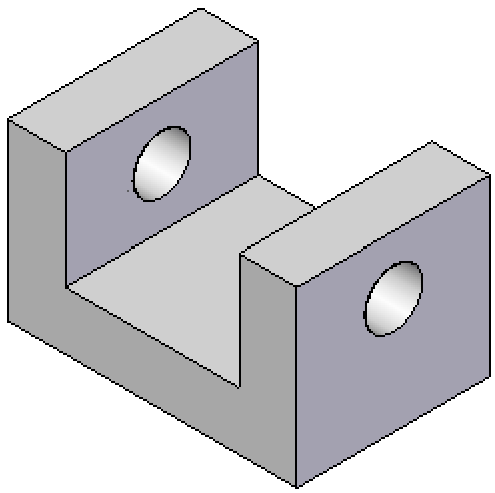




Create an ordered cut feature

- Create a cut with the cross-section shown. Extend through all.





Transition to the Synchronous environment

There are three ways to transition to the other environment.

1. Right-click in PathFinder or modeling window and choose Transition to Synchronous (or Transition to Ordered).

2. On the Tools tab® Model group, click the environment bar to transition to.

3. In PathFinder, click the environment bar to transition to.

Synchronous

Ordered

Note

An environment bar is only available for selection if features exist in that environment.

- Transition to the Synchronous environment using a method of your choice.

Note

Notice that the ordered features do not appear. Only synchronous features appear in the Synchronous environment. In the Ordered environment, both synchronous and ordered features appear.

Transition to the Ordered environment

- Click the Ordered environment bar to transition back to the Ordered environment.

  Ordered

- Save the file as *ordered.par*.

- Close the file.

Summary

In this activity you learned how to create ordered features. You also learned how to switch between modeling environments.

Activity: Creating both ordered and synchronous features in a model

This activity guides you through the process of creating both ordered and synchronous features in the a model. Learn how to edit both feature types and how to convert an ordered feature to a synchronous feature.

Activity: Creating ordered and synchronous features in a model

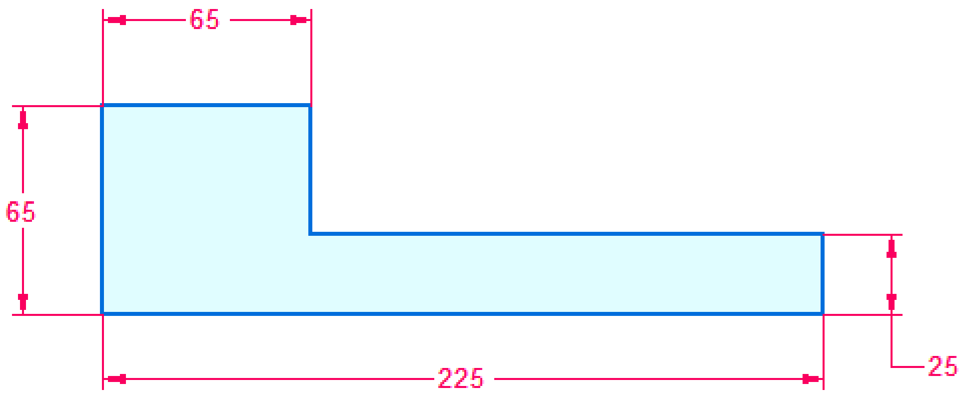
Create a new part document

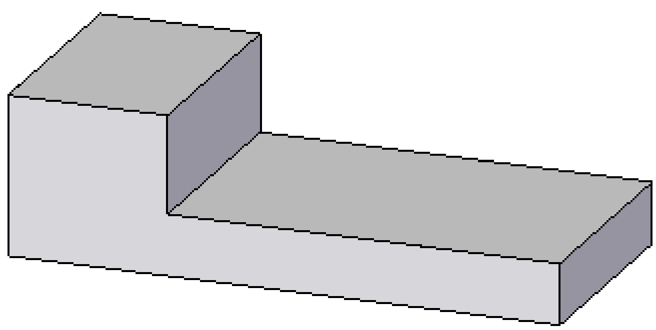
- Create a new part document.
- Switch to the Synchronous environment. See the previous activity (Creating ordered features) if you need help switching modeling environments.

  Synchronous

Create a synchronous base feature

- Create an extrusion with the cross-section shown. Extend symmetrically at a distance of 100 mm.

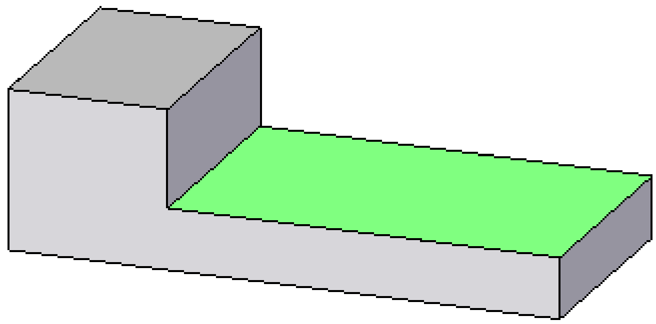


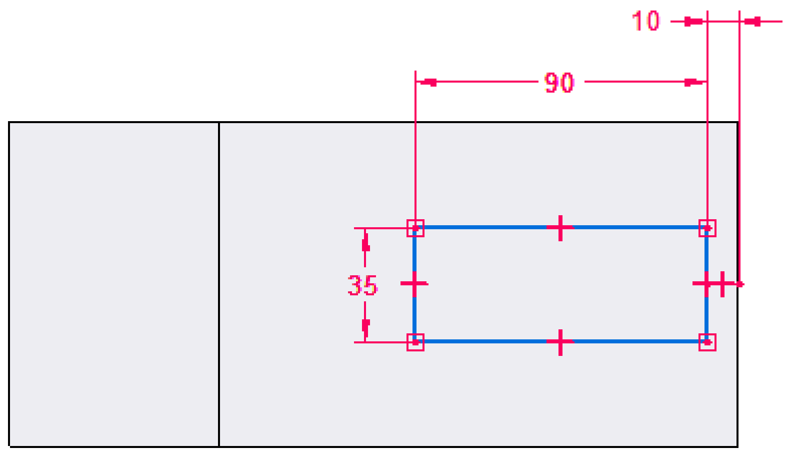


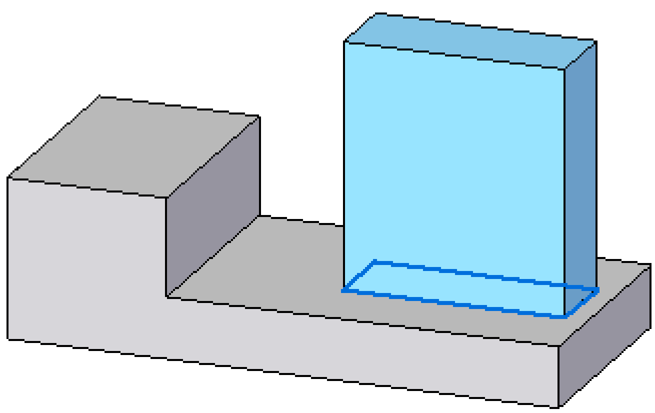
Create an ordered feature

- Transition to the ordered environment.

- Create an extrusion with the cross-section shown. Extend upward at a distance of 100 mm. Draw the cross-section on the green face.

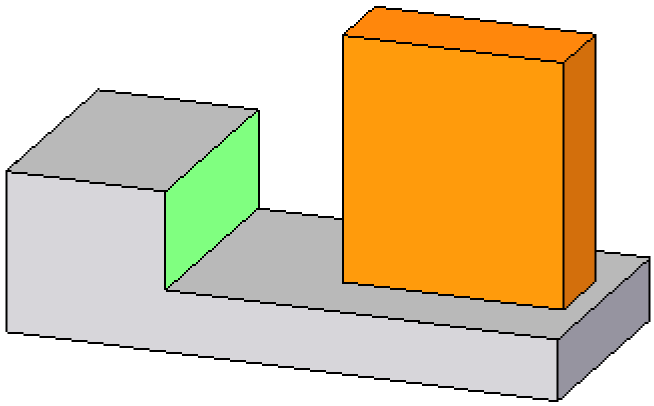




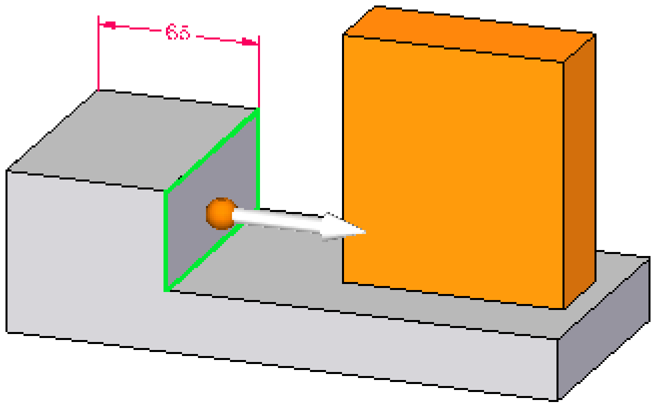


Edit a synchronous feature face while in the ordered environment

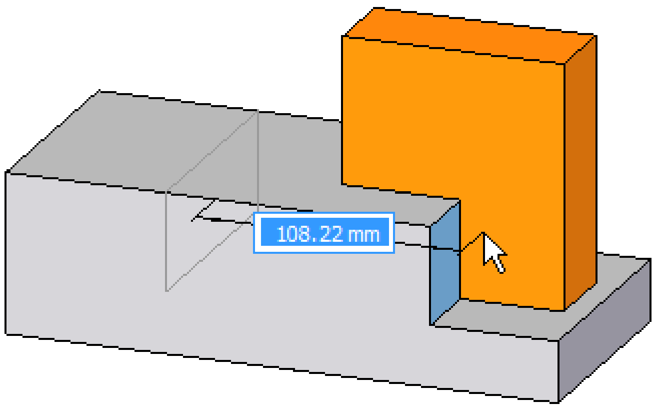
Move the green face on the synchronous feature. The ordered feature is colored orange for clarity only.



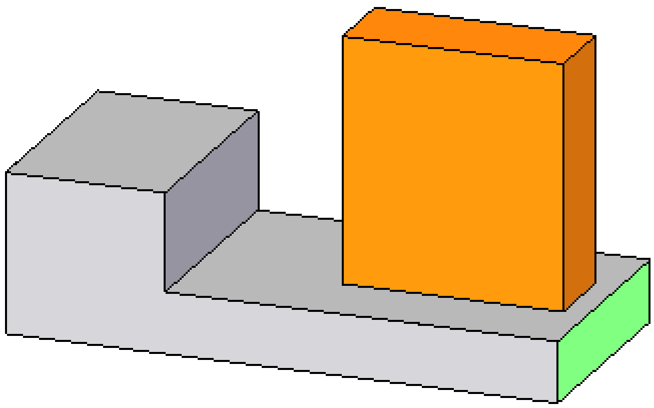
- Select the green face. Notice the face has a locked dimension on it. This dimension migrates from the sketch to the feature. Either delete the dimension or unlock the dimension.



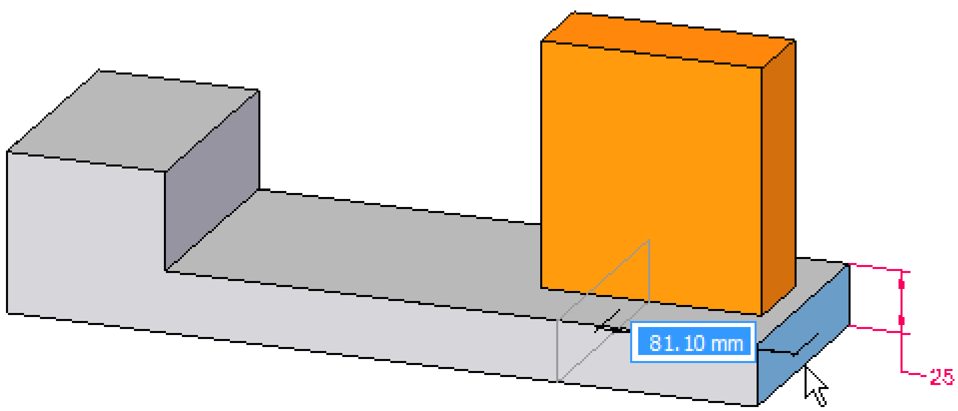
- Select the move handle and drag the face in an area around the ordered feature. Notice how the ordered feature is recognized during an edit. Press the Esc key to end the move operation.



- Select the green face. Notice the face has a locked dimension on it. This dimension migrates from the sketch to the feature. Either delete or unlock the (225 mm) dimension.



- Select the move handle and drag the face to the right. Notice how the ordered feature moves with face. This occurs because the ordered feature sketch was dimensionally locked to the synchronous feature edge. Press the Esc key to end the move operation.



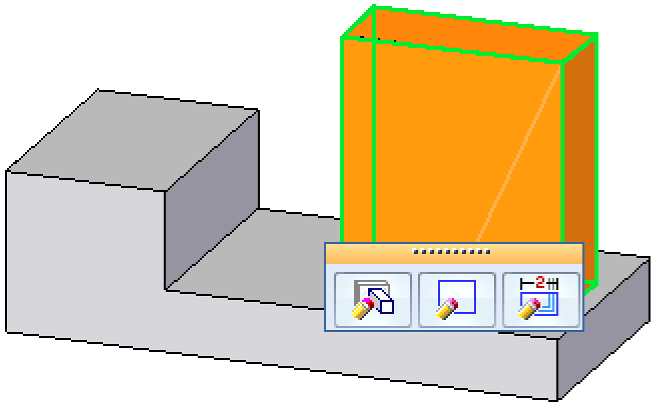
Transition to synchronous

- Switch to the synchronous environment. Notice the ordered feature does not display.

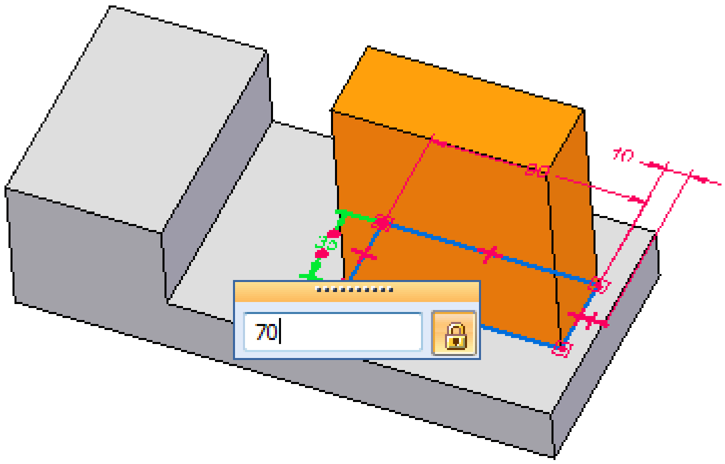
Edit the ordered feature

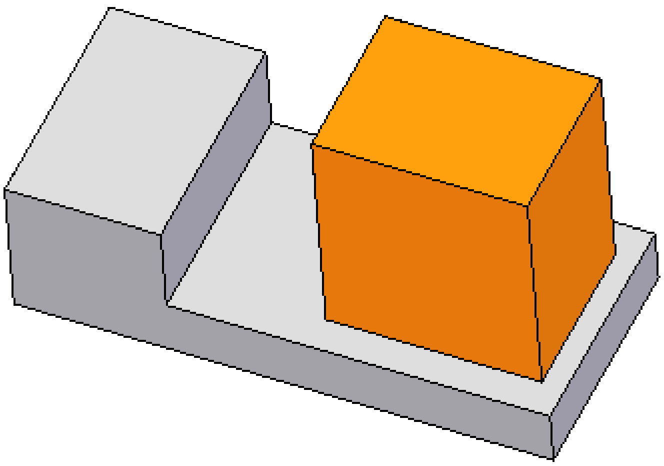
- Switch to the ordered environment.

- Select the ordered feature.



- Click the Dynamic Edit button. Change the 35 mm dimension to 70 mm.





Convert the ordered feature to a synchronous feature

An ordered feature converts to a synchronous feature by moving the ordered feature to the synchronous portion of the PathFinder. Once converted, all dimensions are dropped. The converted feature can be manipulated as an entire synchronous feature or have individual face(s) manipulated.

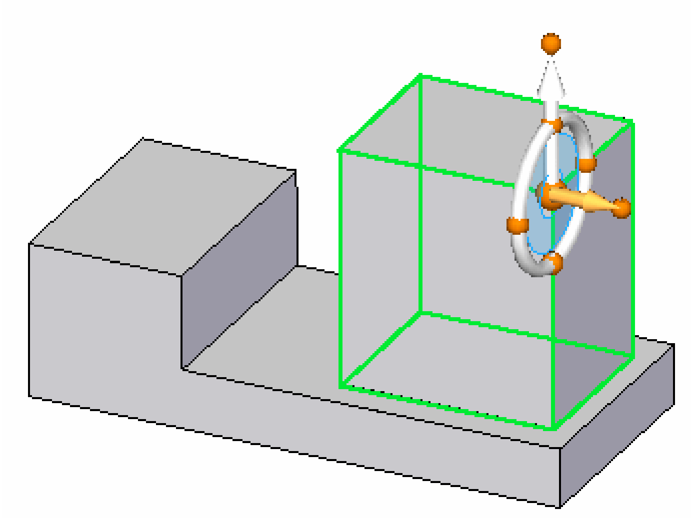
- You must be in the ordered environment to convert ordered features. In PathFinder, right-click on the ordered protrusion feature.

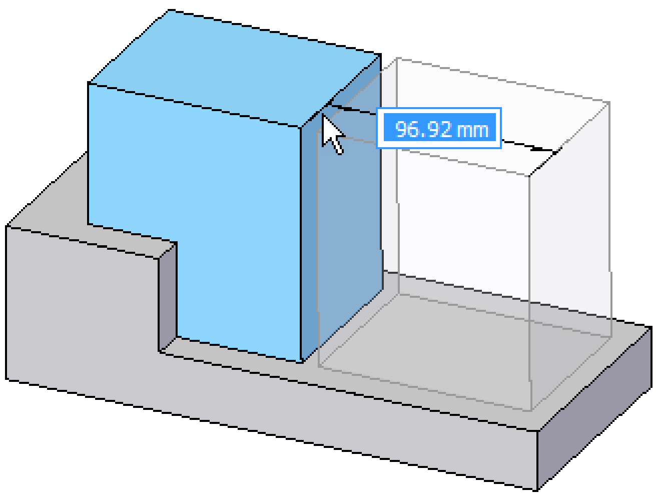
- On the shortcut menu, choose the Move to Synchronous command.

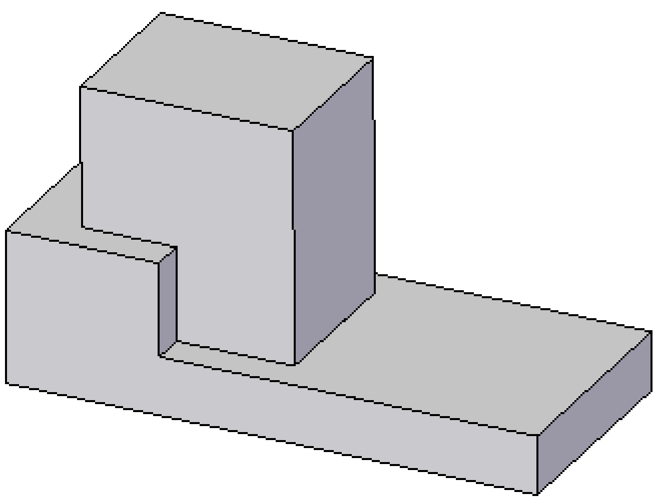
Move the converted feature

- In PathFinder, select the converted protrusion.

- Click the move handle and move feature to the approximate location shown and click.







This completes the activity.

Summary

In this activity you learned how to create both ordered and synchronous features in a single model. You also learned how to edit both feature types and how to convert an ordered feature to a synchronous feature.

Lesson review

Answer the following questions:

1. What is an ordered feature?

2. What is a synchronous feature?

3. What are the differences in ordered and synchronous environments?

4. How do you convert ordered features to synchronous features?

5. How do you convert synchronous features to ordered features?

Answers

1. What is an ordered feature?

   An ordered feature is history based. Sketches drive the feature definition. You can go back and edit the any step in the creation of the feature.

2. What is a synchronous feature?

   A synchronous feature has no history. Once you create a feature, you cannot go back and edit any step in the feature creation process. Model faces drive the model. You manipulate faces to edit a synchronous model. Some features have handles where you change a value that defines the feature.

3. What are the differences in ordered and synchronous environments?

   - In the synchronous environment, only synchronous features and sketches display.

   - In the ordered environment, both synchronous and ordered features and sketches display.

   - Each environment presents its own command ribbon bar.

4. How do you convert ordered features to synchronous features?

   Select the ordered feature in PathFinder, right-click and choose the Move to Synchronous command.

5. How do you convert synchronous features to ordered features?

   You cannot convert synchronous features to ordered features.

Lesson summary

Solid Edge provides environments for modeling either synchronous or ordered features. You work in a single model file with only synchronous features, only ordered features, or a combination of both features types. You can convert ordered features to synchronous features.

Modeling ordered features activities

This section is a collection of activities that focuses on modeling ordered features.

Sketching activities

Learn the tools for creating sketches that describe a feature cross-section.

Using IntelliSketch

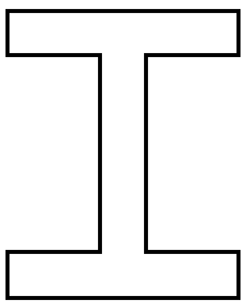
Create a sketch in this activity. Apply relationships, dimensions and variables to the geometry so that you can reliably and predictably change the shape of the profile by editing dimensions.

Activity: Using IntelliSketch

Objectives

Create an ordered sketch in this activity. You can also perform this activity in the synchronous environment with a slightly different interface. Apply relationships, dimensions and variables to the geometry so that you can reliably and predictably change the shape of the profile by editing dimensions.

- The sketch is in the shape of a cross-section of an I-beam.
- Relationships, dimensions and variables control the width of the web and flanges of the "I" shape.



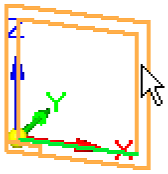
- ‣ Create a new part document.
- ‣ Make sure you are in the Ordered environment.

Ordered

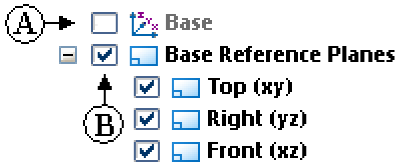
<u>Draw the sketch</u>

Draw an "I" shaped sketch.

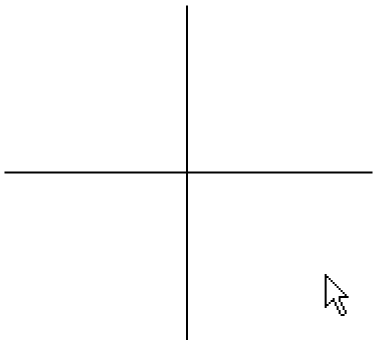
- ‣ On the Home tab® Sketch group, choose the Sketch command.
- ‣ Select the reference plane shown.



- In PathFinder, turn off the display of the base coordinate system (A) and turn on the display of the base reference planes (B).

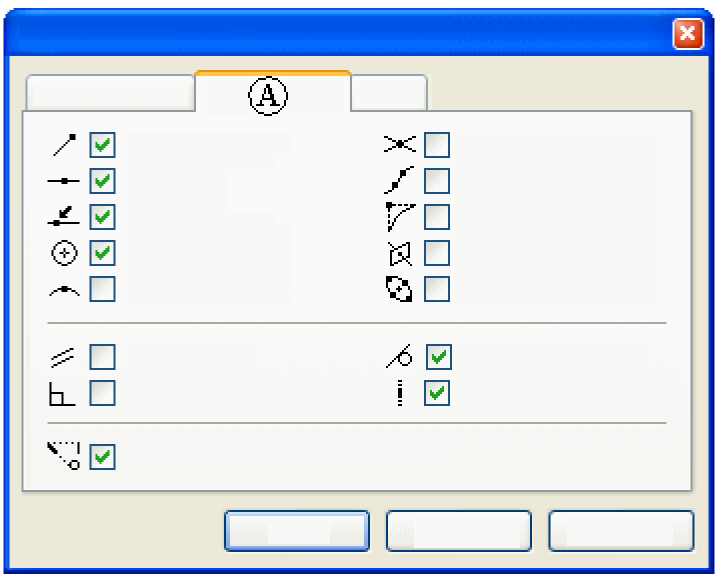


- Fit the window and zoom out until the base reference planes appear as shown.



- On the Home tab® IntelliSketch group, choose IntelliSketch options.

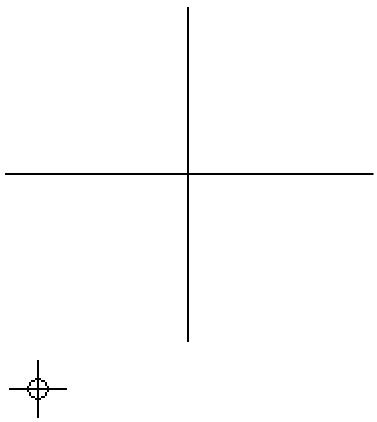
- On the Relationships page (A), set the options shown. Click OK.



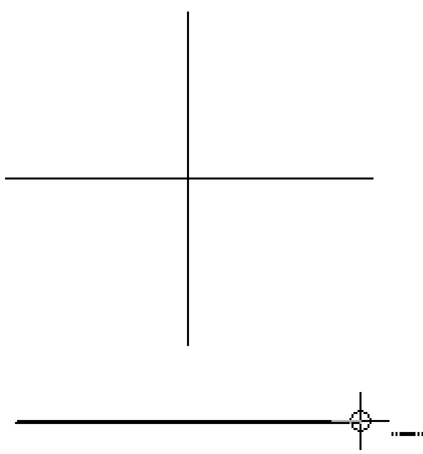
- In the IntelliSketch group, click the Horizontal or Vertical option to make it recognizable if a line is horizontal or vertical during placement.

- On the Home tab® Draw group, choose the Line command .

- Draw the first line by positioning the cursor below and left of the reference planes as shown and click to place the first point of the line.



- Place the second point by moving the cursor to the right. When the horizontal indicator is shown and the line is approximately in the same position as shown below, click to place the line.



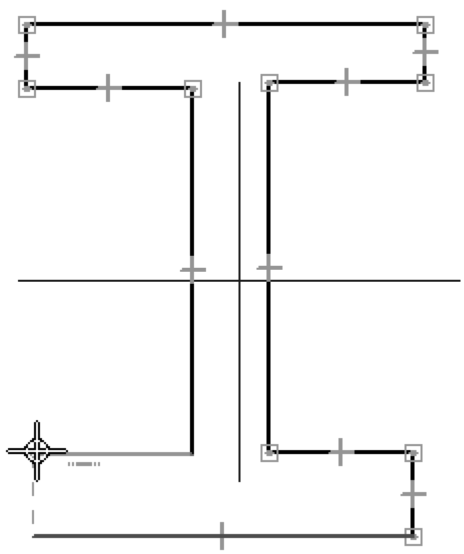
- Continue drawing the “I” shape with the following considerations. Draw each segment with the horizontal or vertical indicator displayed. Exact lengths of the lines are unimportant at this stage.

  Note

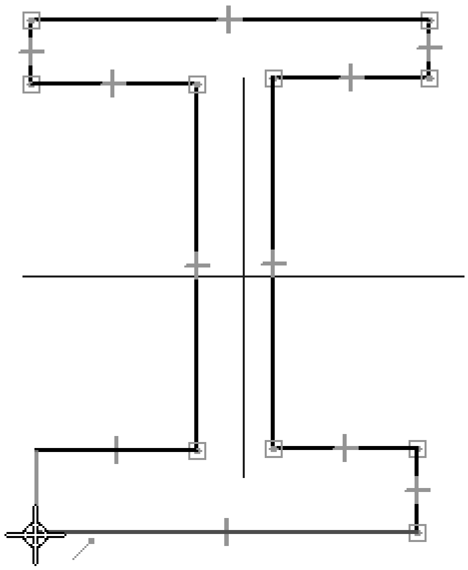
  If you make a mistake, you can delete a line by first clicking the Select tool , selecting the line, and pressing the Delete key on the keyboard.

  Also by choosing the Undo command , you can step back through the creation of the sketch.

- Draw the rough shape of the “I” in a counterclockwise order. Use the alignment indicator to position the endpoint of the next to the last line above the left endpoint of the first line as shown. To activate the alignment indicator for the last segment, brush (move the cursor over without clicking) the horizontal line.

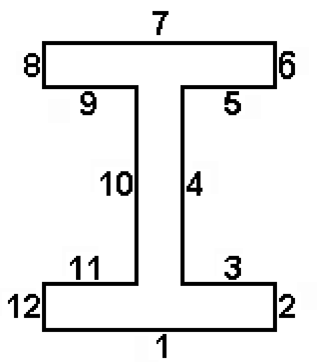


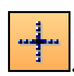
- To place the last segment, click on the endpoint of the first line when the endpoint indicator is displayed as shown.



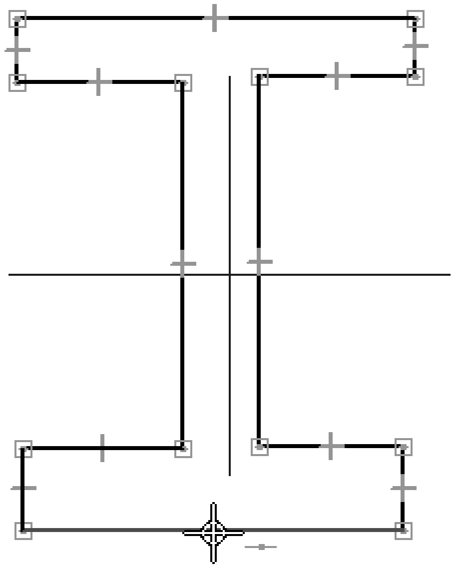
Add relationships

Add relationships to control the behavior of the shape. When you anticipate the need to make a shape symmetrical, it is useful to establish relationships between the geometry of the shape and reference planes. Reference the line segments by numbers as shown.

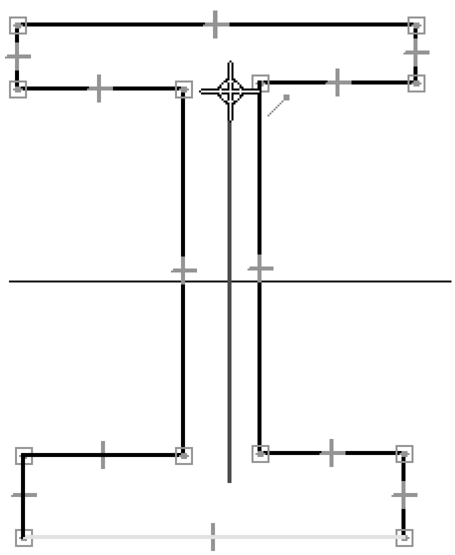


- In the Relate group, choose the Horizontal/Vertical command .

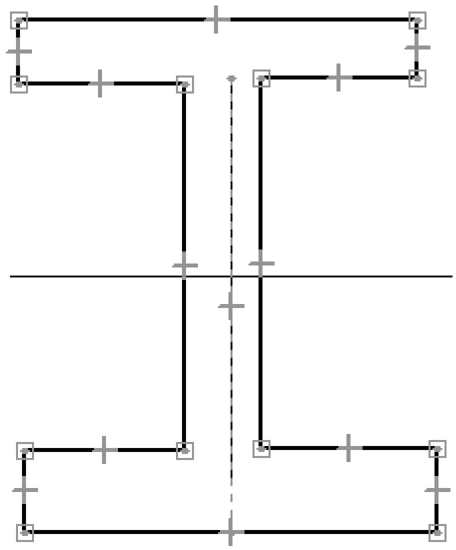
- Position the cursor over middle of segment 1. When the midpoint indicator displays, click.



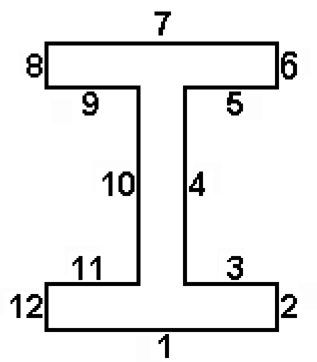
- Move the cursor to the top of the vertical reference plane, and when the endpoint indicator displays, click.



- A relationship applies represented by a dashed line that forces the midpoint of segment 1 to remain vertically aligned with the endpoint of the reference plane.



- In the Relate group, choose the Equal command.



- Select segment 1, then select segment 7. This applies an equal relationship to the lines, which keeps their lengths the same while other constraints alter the shape of the profile. Line segment 1 is equal to line segment 7.

- Continue applying the equal relationship between the following lines:

  2 and 12

  8 and 6

  8 and 12

  11 and 3

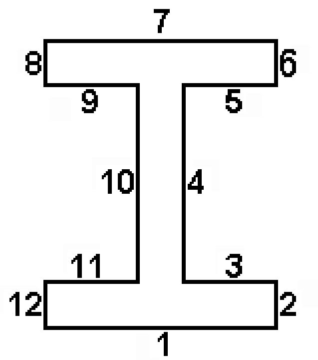
  9 and 5

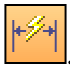
  9 and 11

  10 and 4

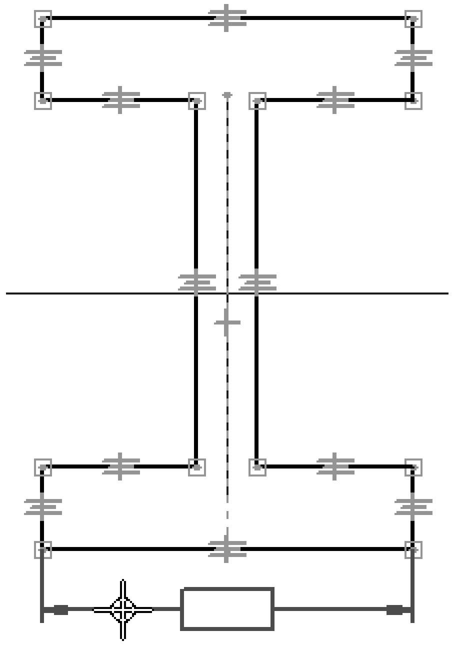
<u>Add dimensions</u>

Add dimensions to control the size of the shape.



- In the Dimension group, choose the SmartDimension command .

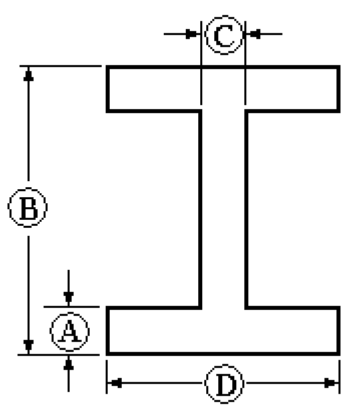
- Select segment 1, position the dimension below the line, and then click to place it.



- Dimension segment 12 in the same way.
- Choose the Distance Between command .
- Select segment 10, select 4, position the dimension above the "I" shape, and then click to place it. Right-click to restart the Distance Between command.
- Dimension the distance between segments 1 and 7 in the same way.

Edit dimension values

Edit the dimensions placed in the previous step. Because of the dimensions and relationships defined, the shape responds to dimensional changes predictably.



- Choose the Select Tool command.

- Select dimension (A). Type 15 and then press the Enter key.
- Select dimension (B) and change the value to 120.
- Select dimension (C) and change the value to 12.
- Select dimension (D) and change the value to 95.
- Practice altering the shape by editing the values of the dimensions (A, B, C, D) and observe how the shape responds. Return the dimension values to those shown above.

Using dimension variables

Dimensions and relationships make it easier to control the shape of a profile. Variables are used to make the shape of a profile parametric. Formulas are applied that define mathematical relationships between variables and dimensions. In this step, make the width of the web (dimension (C)) 2/3 the thickness of the flange (dimension (A)), and make the flange height (dimension (B)) 3/4 the flange width (dimension (D)).

Each time a dimension is placed, a randomly named variable is created to represent it. Rename the variables and assign mathematical expressions to further control the behavior of the shape.

- Right-click on the 95 mm dimension. Choose the Edit Formula command on the shortcut menu. The Edit Formula command bar displays to edit the dimension name and formula. In the Name: field, change the variable name to D and then press the Enter key. Click the Select tool to end the dimension edit.
- Repeat the previous step to make the following dimension edits:

15 mm dimension	Name=A
120 mm dimension	Name=B
12 mm dimension	Name=C

Note

To enter a formula, click the formula field, type the formula, and press the Enter key. Basic mathematical operators in formulas can be used:

+ to add

- to subtract

* to multiply

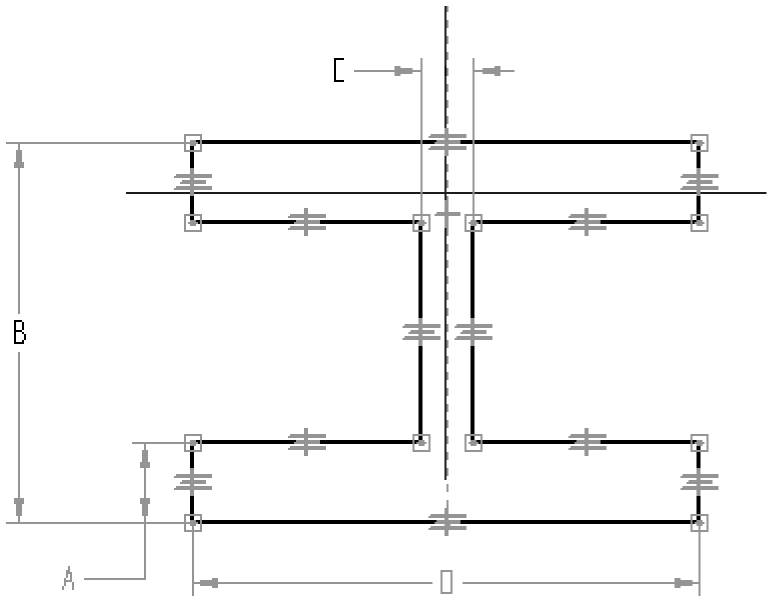
/ to divide

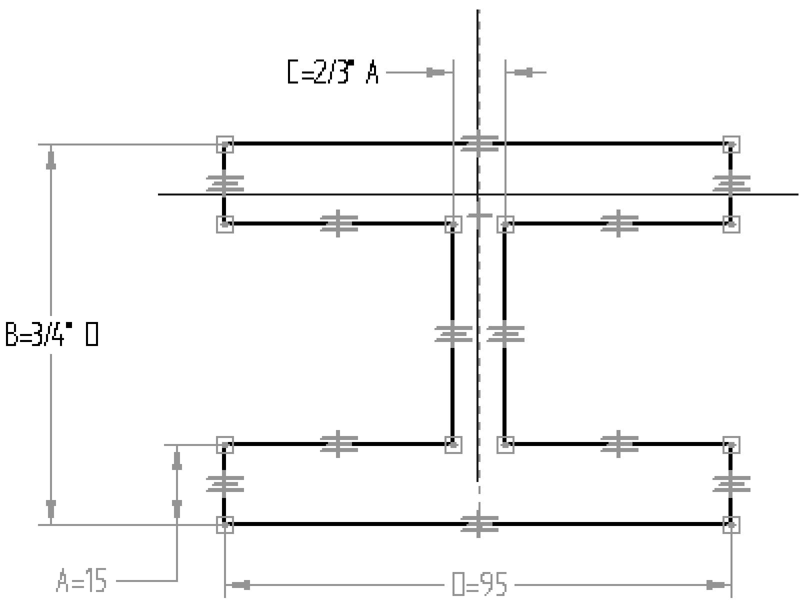
Mathematical functions can be grouped with parenthesis if necessary. Many functions are available. For more information, see the Variables Help topic.

- Assign a mathematical expression to dimensions named B and D. Right-click the 120 mm dimension and choose Edit Formula. In the Formula field, enter 3/4*D and press the Enter key.

- Edit the formula for the 12 mm dimension. In the Formula field, enter 2/3*A and press the Enter key.

- Notice the dimension display below. On the shortcut menu, the Show All Values option is on. All variable names or formulas can be shown.





Using the Variable Table

The same operations performed in the previous could also be done using the Variable Table.

- On the Tools menu® Variables group, choose the Variables command to display the Variable Table.

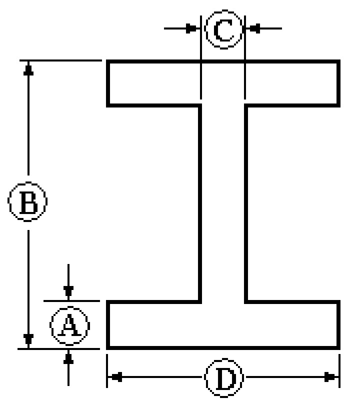
- Notice the same fields as in the Edit Formula command bar are available. Click the field to edit, type in the appropriate value and then press the Enter key.

  Note

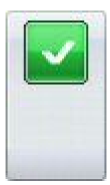
  The shadowed values represent values that cannot be directly changed because they are controlled by relationships, dimensions or formulas.

- Close the variable table by clicking the X in the upper right corner.

- On the sketch, modify the dimension values of (A) and (D) and observe how the sketch responds.



Save the sketch

- Choose Close Sketch  to complete the Sketch.

  You can also complete the sketch by clicking the checkmark  located in the upper left corner of the sketch window.

- On the command bar, click Finish.

- Close and save this file as *Ishape.par*. This completes this activity.

Summary

In this activity, you learned how to use dimensions and relationships to control the size and position of 2D geometry in a profile. You also learned how to use mathematical formulas within the variable table to establish relative behavior between geometry. This is useful in establishing design intent within a model. If a critical dimension changes, the profile adjusts itself predictably and accordingly.

Applying sketch relationships (collinear, parallel, equal)

In this activity, learn to use more relationships in the profile/sketch environment.

This activity covers the collinear, parallel, and equal relationships.

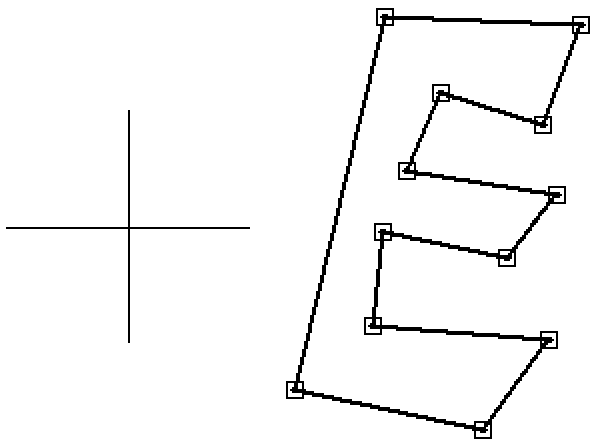
Activity: Applying sketch relationships (collinear, parallel, equal)

Objectives

In this activity, learn to use more relationships in the profile/sketch environment.

This activity covers the collinear, parallel and equal sketch relationships.

- Open *sketch_a1.par*.



Apply relationships

Apply relationships to control the E shape.

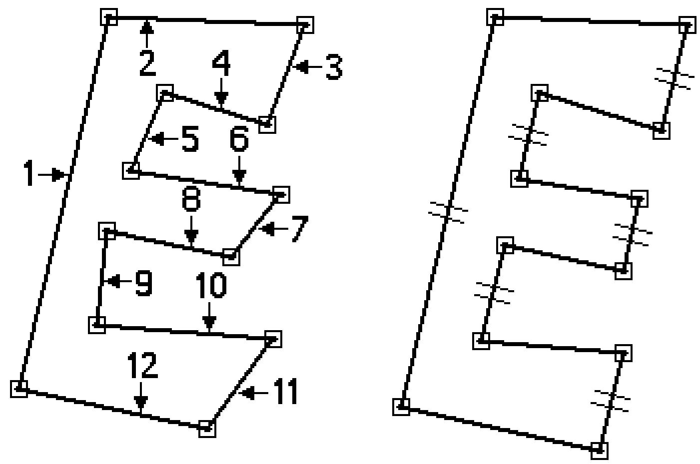
> **Note**
>
> No horizontal/vertical relationships are used. This allows the sketch to rotate at any angle and maintain the E shape.

- In Pathfinder, right-click on the sketch named *Sketch A*. On the short cut menu, choose the Edit Profile command.

- Define the shape by applying parallel relationships. The first element you select is made parallel to the second element selected. In the Relate group, choose the Parallel command .

‣ Select the line segments as described below.

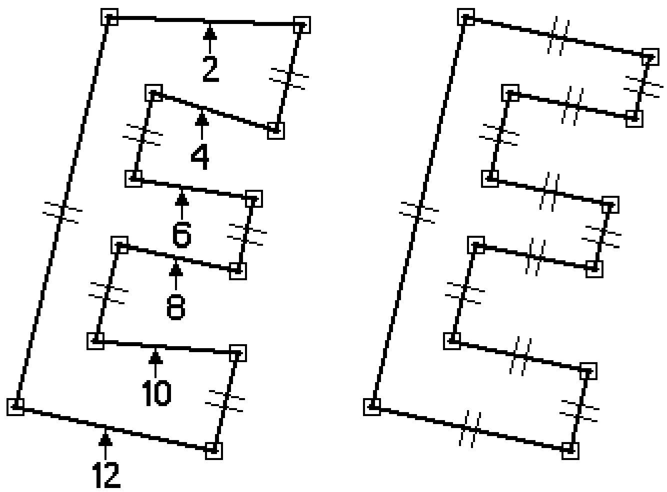
- Click (3), then click (1).
- Click (5), then click (1).
- Click (7), then click (1).
- Click (9), then click (1).
- Click (11), then click (1).



Continue adding parallel relationships

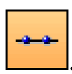
Continue to add parallel relationships to the remaining line segments.

- Apply the parallel relationships shown:
  - Click (10), then click (12).
  - Click (8), then click (12).
  - Click (6), then click (12).
  - Click (4), then click (12).
  - Click (2), then click (12).

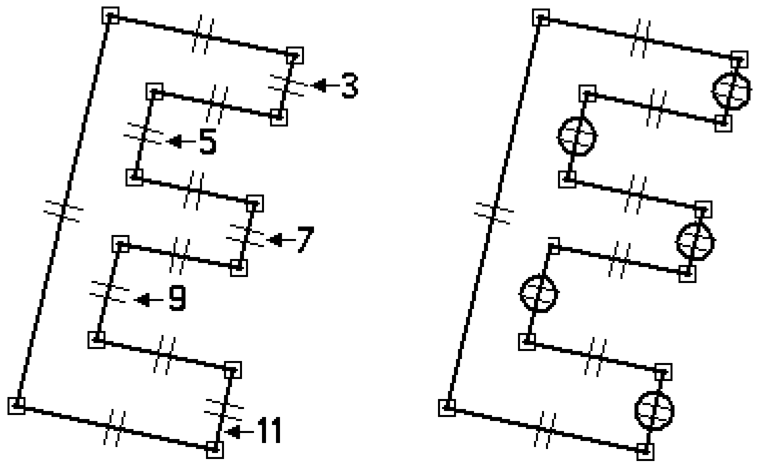


Apply collinear relationships

Apply collinear relationships to align line segments. The first line segment you select is made collinear to the second line segment selected.

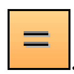
- Choose the Collinear command .

- Select the line segments as shown.
  - Click (7), then click (11).
  - Click (3), then click (11).
  - Click (5), then click (9).

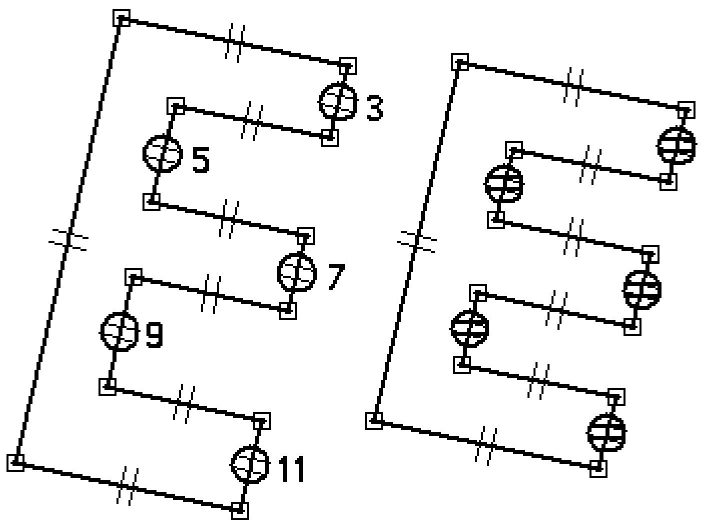


Apply equal relationships

Apply equal relationships to control the thickness of the E shape.

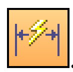
- Choose the Equal command .

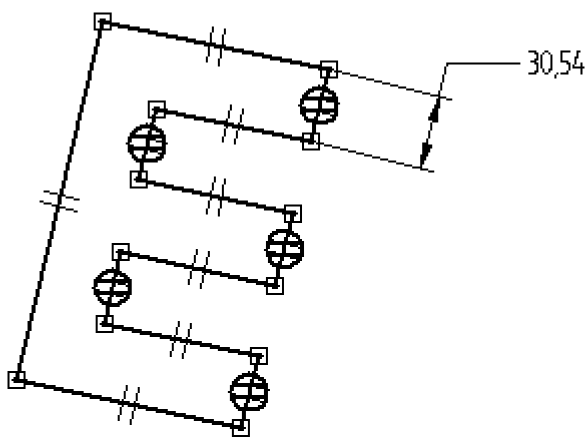
- The first line segment you select is made equal to the second line segment selected.
  - Click line segment (5), then click line segment (3).
  - Click line segment (7), then click line segment (3).
  - Click line segment (9), then click line segment (3).
  - Click line segment (11), then click line segment (3).

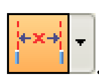


Add dimensional constraints

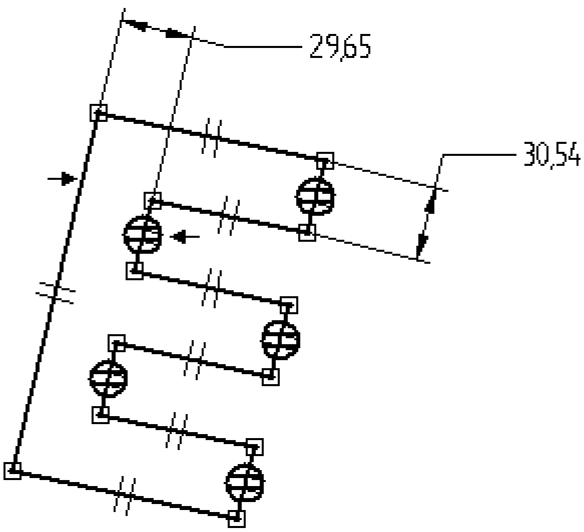
Add dimensional constraints to complete the E shape.

- Choose the SmartDimension command .
- Dimension the line as shown. The value is not important at this point.

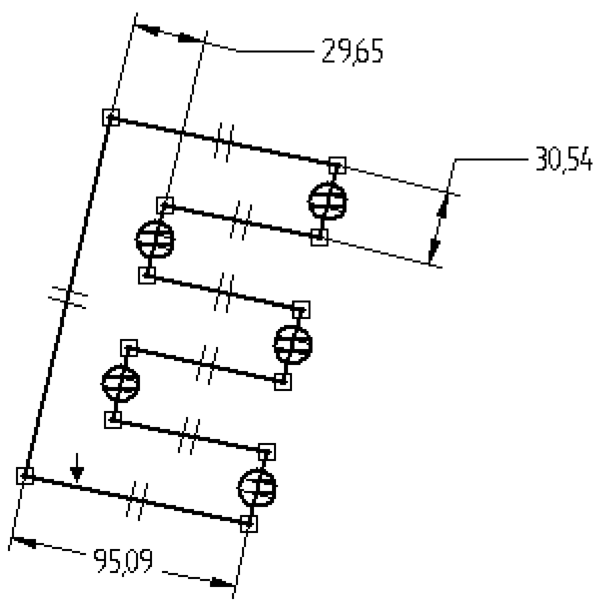


- Choose the Distance Between command .

- On the command bar, click the By 2 Points option.
- Dimension the two line segments as shown. Click on the lines (do not click the endpoints or midpoints).

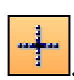


- Choose the SmartDimension command and dimension the line segment shown.

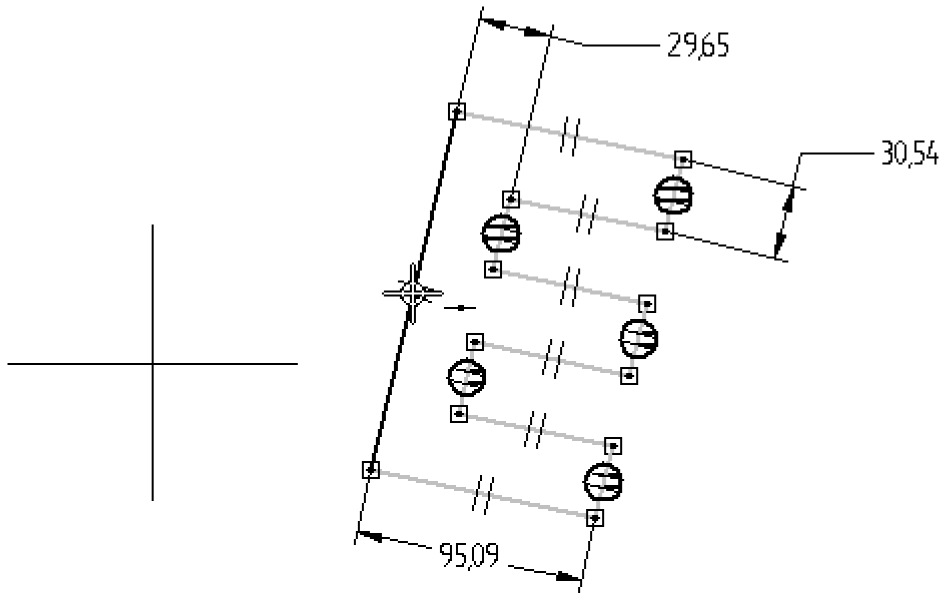


<u>Align the sketch</u>

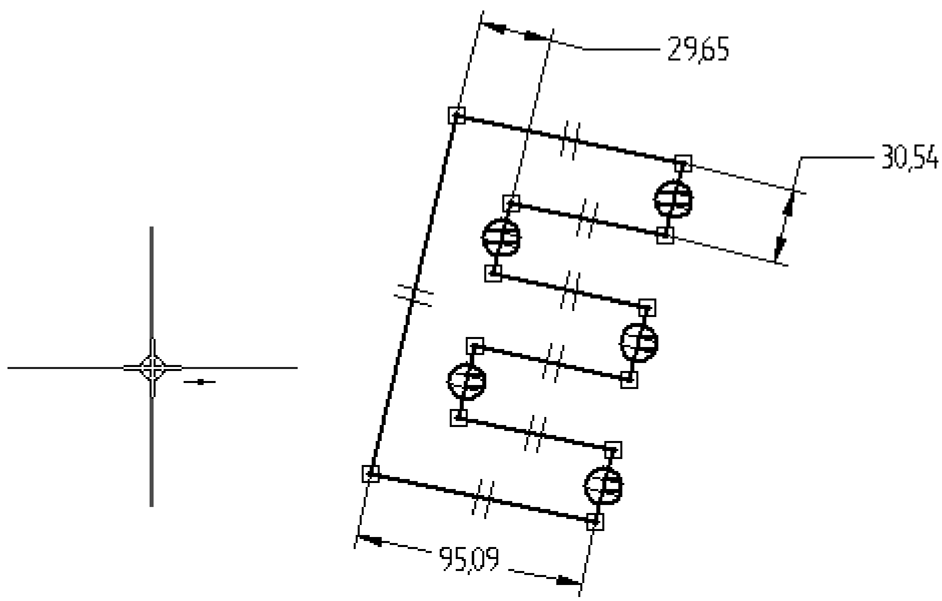
Align the midpoint of the left line segment to the center of the reference planes.

- Choose the Horizontal/Vertical command .

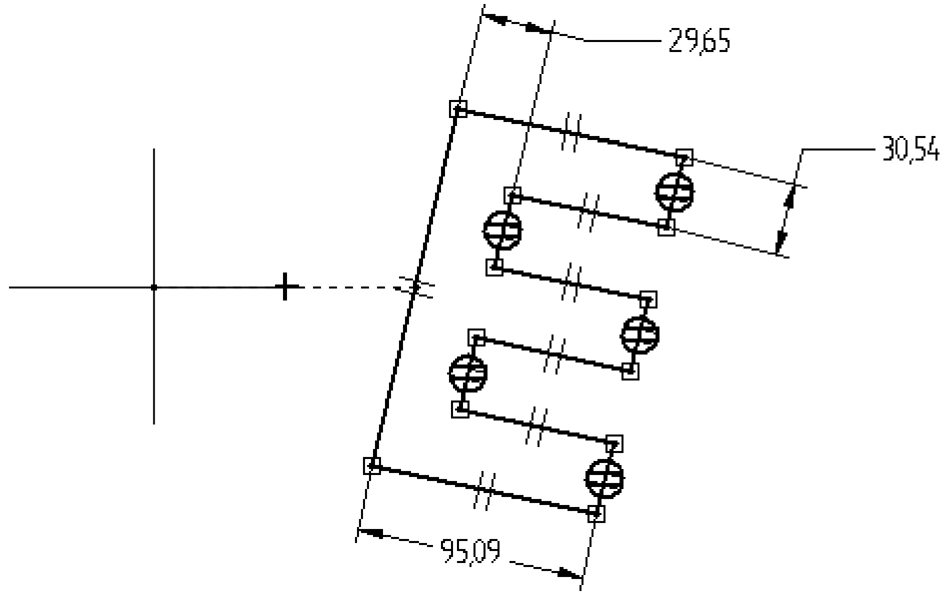
- Click on the midpoint of the left line segment as shown.



- Click on the midpoint of the reference plane edge as shown.

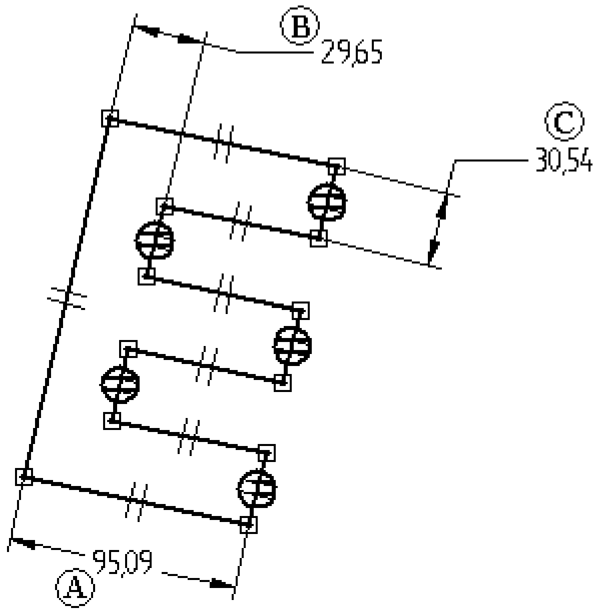


- The midpoint of the left line segment aligns with center of the reference planes.



Edit the dimensions

Edit the dimensions to complete the E shape.



- Edit the dimensions as shown.
  - Dimension (A) = 200
  - Dimension (B) = 50
  - Dimension (C) = Dimension (B)

  Note

  How to make two dimensions equal

  Step 1: Right-click on dimension (C).

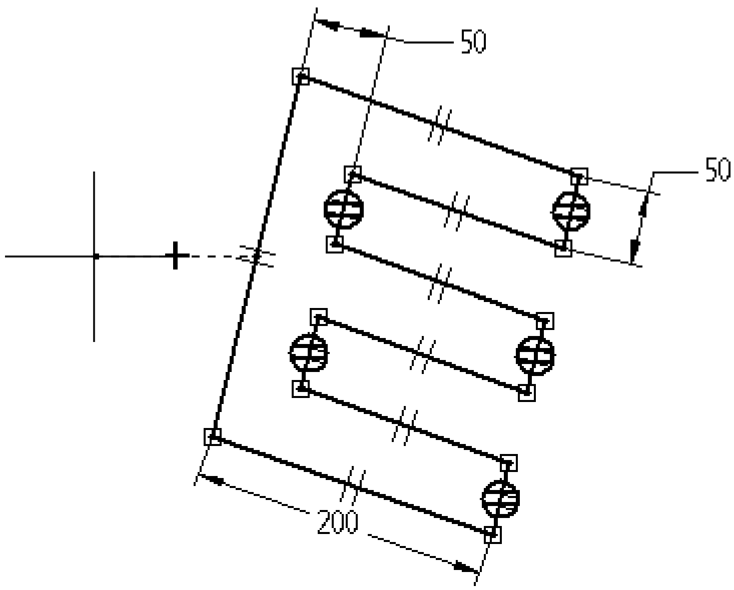
  Step 2: On the shortcut menu, choose the Edit Formula command.

  Step 3: On the Edit Formula command bar, in the Formula field, type = and then click on dimension (B).

  Step 4: Click the Accept button.

  Step 5: Click the Select tool to end Edit Formula.

- The result should be as shown.

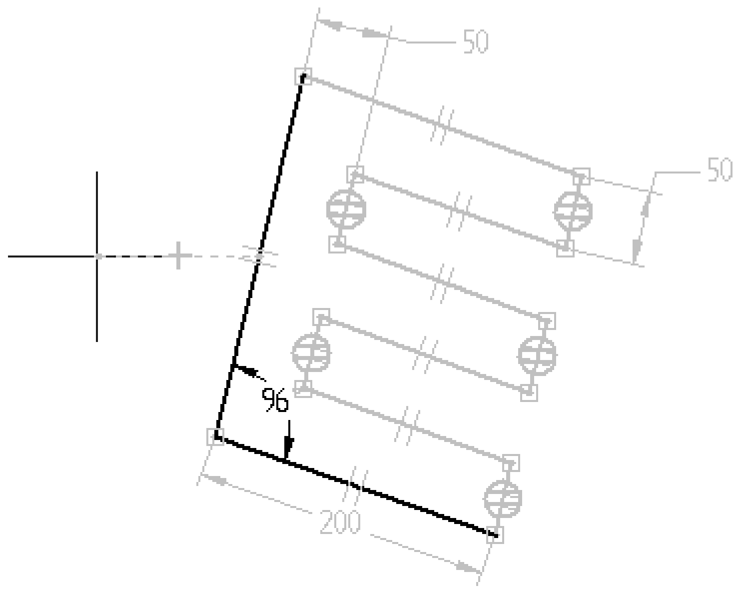


Add angular dimensions

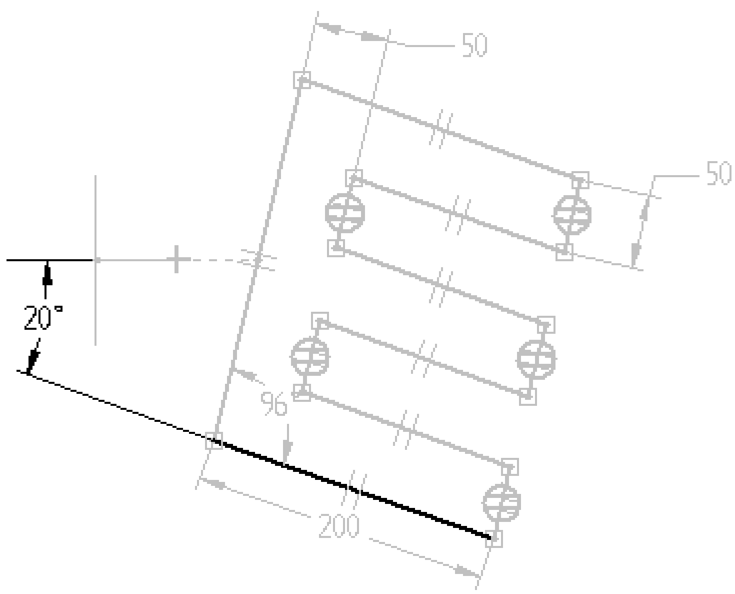
Add angular dimensions which controls the shape and orientation relative to the horizontal reference plane.

- Choose the Angle Between command .

- Place the dimension shown by clicking on the two lines (do not select any keypoints).



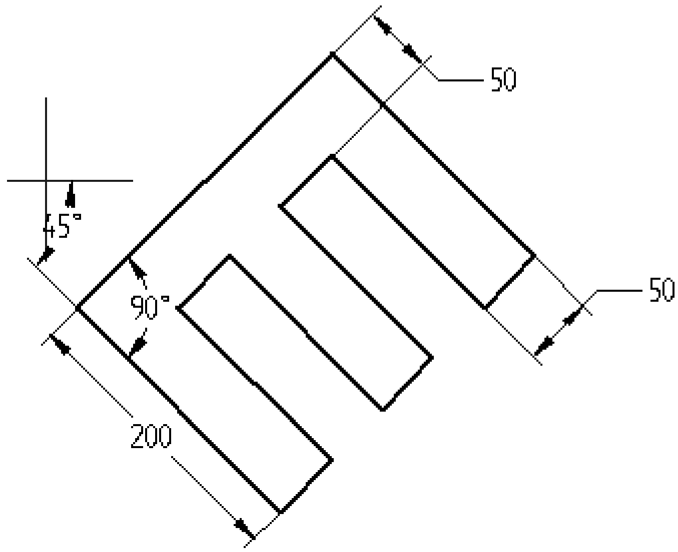
- Place an angular dimension between the horizontal reference plane and the bottom line segment to control the E shape orientation. First right-click to restart the Angle Between command. Click the horizontal reference plane and the bottom line segment as shown (again do not click any keypoints).



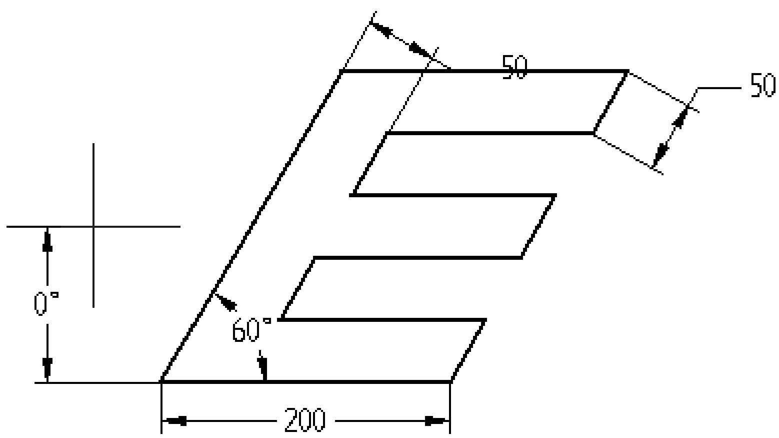
Edit the angular dimensions

Edit the angular dimensions to observe the control over the shape and orientation.

- Orientation angle = 45, shape angle = 90



- Orientation angle = 0, shape angle = 60



- Click Close Sketch. On the command bar, click Finish.

- This completes the activity.

<u>Summary</u>

In this activity, you learned how to use dimensions and relationships to position a profile containing interior features. Relationships were used to position various features relative to each other. By varying the dimensions, you are able to control the size and position of the interior features and maintain design intent.

Applying sketch relationships (symmetric)

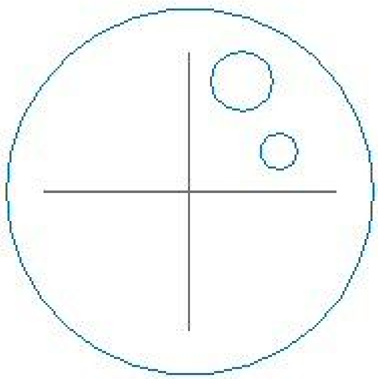
In this activity, learn to use symmetric relationships in the profile/sketch environment.

Activity: Applying sketch relationships (symmetric)

Objectives

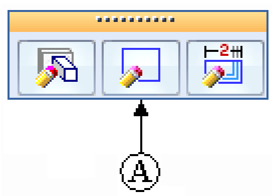
In this activity, use the symmetric relationships in the profile/sketch environment.

- Open *sketch_b1.par*.

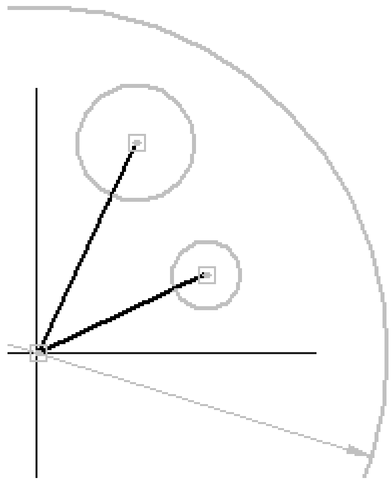


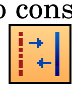
<u>Add construction elements</u>

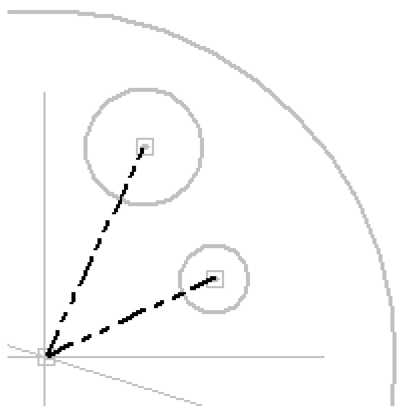
- Select the sketch in the window and then click the Edit Profile command (A).



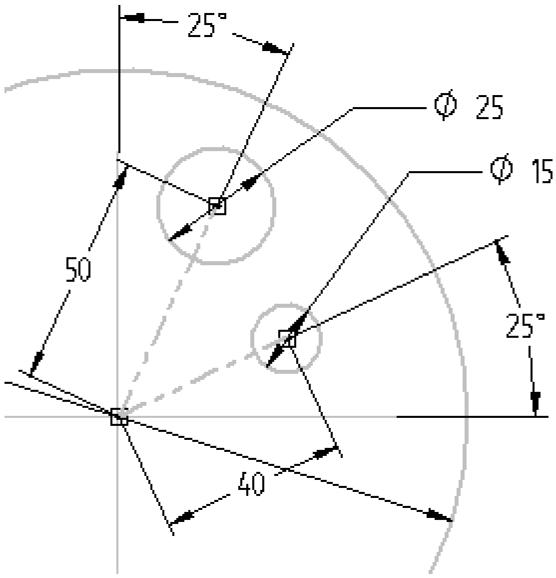
- Place the lines as shown. Lines connect to the centers of the circles and center of the reference planes.



- Change the two lines to construction elements. In the Draw group, choose the Construction command . Select the two lines just placed.



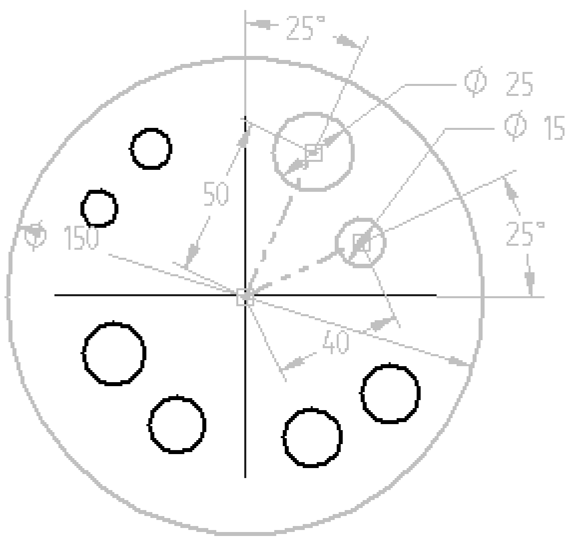
- Dimension the circles and lines as shown.



Place circle sketch elements

Place six circles in the remaining three quadrants of the main circle.

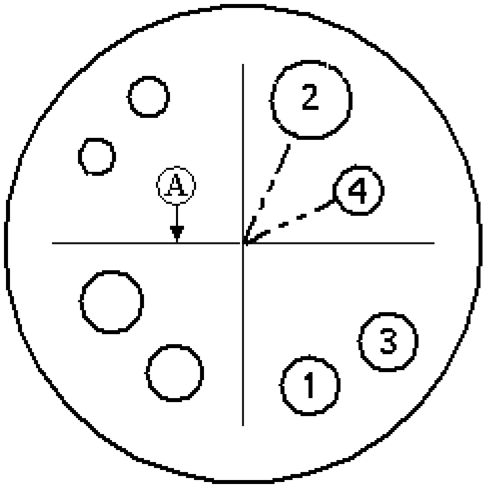
- Place the circles as shown. Position and size do not matter. Be sure not to pick up any relationships from other geometry while placing the circles. If you have problems doing this, place a circle outside the main circle and then drag it inside the main circle.

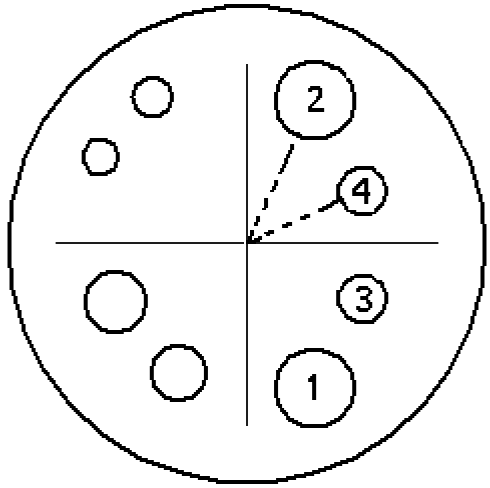


Apply symmetric relationships between the circles

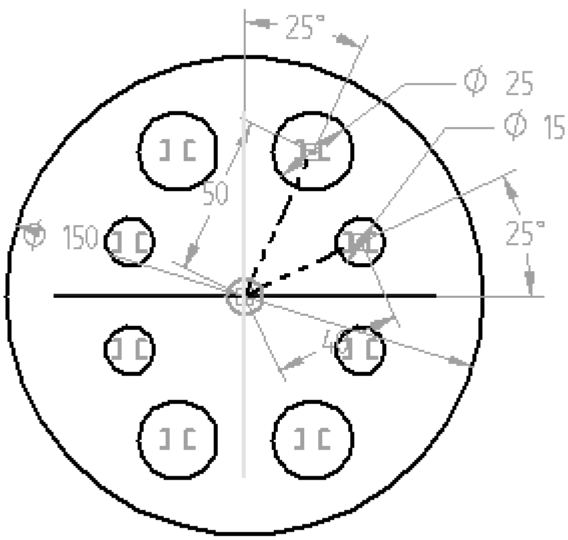
- In the Relate group, choose the Symmetric Relationship command .

- Click the horizontal reference plane (A). Click circle (1) and then click circle (2). Circle (1) is now symmetrical to circle (2). Click circle (3) and then click circle (4). Circle (3) is now symmetrical to circle (4).





- Apply symmetric relationships to the remaining circles using the vertical reference plane as the symmetry axis. In order to do this you must select a new symmetry axis. Choose the Set Symmetry Axis command .
- Click the vertical reference plane.
- Click the Symmetric Relationship command and then click the remaining circles to apply symmetry as shown.

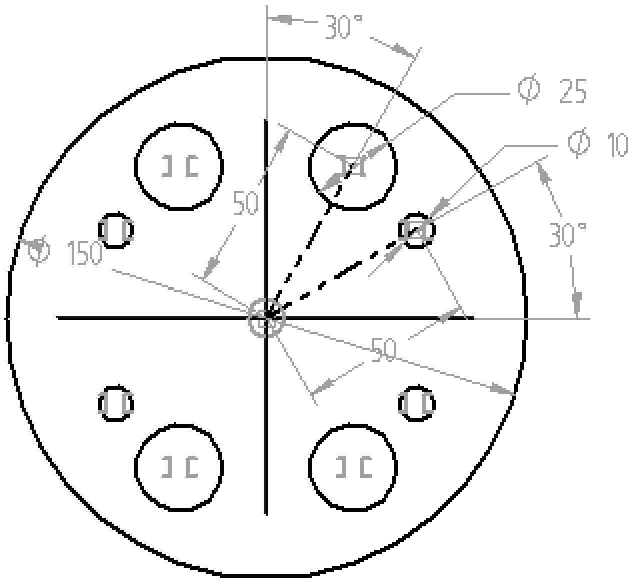


Edit dimensions

Edit the dimensions and observe the results.

- Edit the 40 dimension on the angled construction line to 50.

- Edit both 25° dimensions to 30°.
- Edit the 15 diameter to 10.



- Choose the Close Sketch command. On the command bar, click Finish.
- Close the file and do not save. This completes the activity.

Summary

In this activity, you learned how to use dimensions and relationships to position a profile containing interior features. Relationships were used to position various features relative to each other. By varying the dimensions, you are able to control the size and position of the interior features and maintain design intent.

Using construction elements in a profile

In this activity learn to use construction elements when drawing a profile or sketch in order to capture design intent.

Activity: Using construction elements in profiles

Overview

In this activity, learn to use construction elements when drawing a profile or sketch in order to capture design intent.

Objectives

After completing this activity, you are able to:

- Use construction elements to simplify profile or sketch construction.
- Use the construction elements to drive the resulting geometry (a cutout feature).

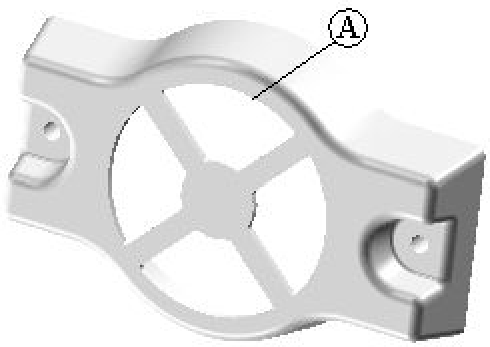
In this activity, examine a specific feature within a part. You do not construct the part in this activity, but you draw the profile for the feature. To simplify profile creation, use a construction element in the Sketch drawing environment. As previously mentioned, construction elements aid in profile creation but are ignored during profile validation checks.

Note

Construction elements serve as skeletal elements that helps drive the other elements in the profile.

Examining the Problem

Examine the patterned cutout feature (A).



Each of the four cutouts must sweep 90°. A narrow web of material must occupy space between each cutout to avoid breakout. To create this model, use construction elements to locate the cutout, provide the mechanism for the sweep angle, and provide the distance between each cutout.

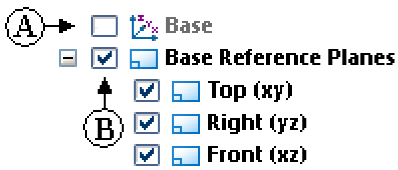
<u>Create part document</u>

- Create a new ISO part document.

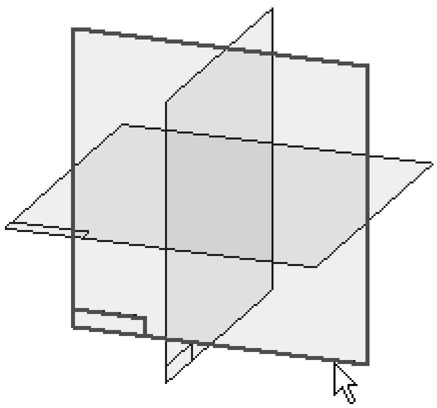
- Make sure you are in the ordered environment.

Define the sketch plane

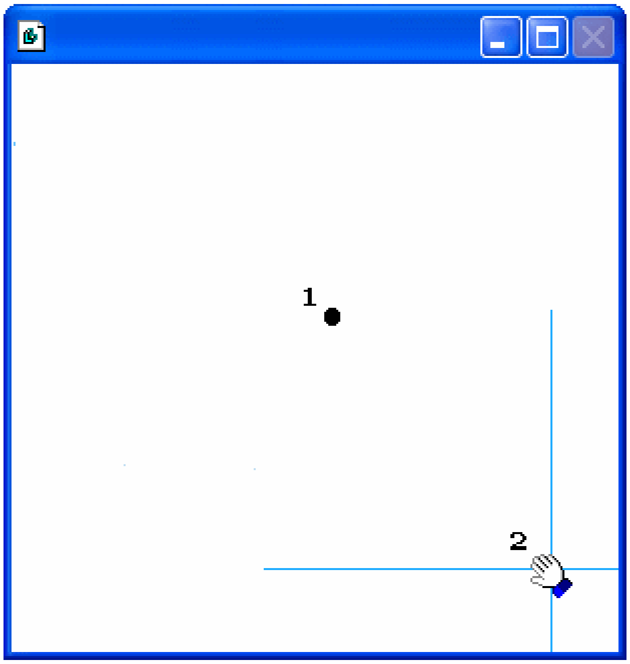
- Choose the Sketch command .
- In PathFinder, turn off the display of the base coordinate system (A) and turn on the display of the base reference planes (B).



- Select the reference plane shown.



- On the status bar, click the Pan command . Hold the left mouse button at the center or intersection of the reference planes. Move the cursor from position 1 to the lower right corner of the Sketch window (position 2). This moves the reference planes out of the way and prevents unwanted relationship placement between a profile element and a reference plane.

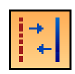


<u>Construct a sketch</u>

- Choose the Line command. Draw the three lines as shown in the illustration.

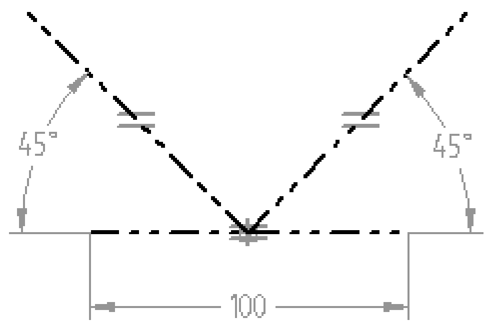
- Add the dimensions and edit their values as shown.

- Make each of the lines construction geometry.

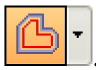
  In the Draw group, choose the Construction command  and select each of the three lines.

- The angled lines attach to the horizontal line at its midpoint.

- Using the Equal relationship, make each of the angled lines equal to the horizontal line.

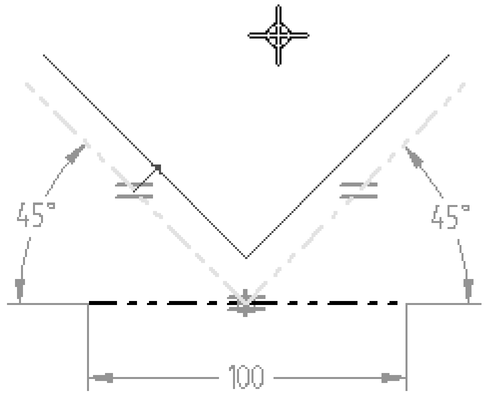


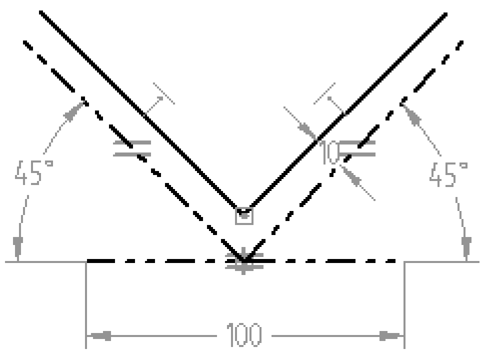
Add lines

Add lines using the Offset command .

- In the Draw group, choose the Offset command.
- Type a value of 10 for the offset distance.
- Set the Chain option in the Offset Select box.

- Offset the two angled lines as seen in the illustration below.
- Click the Accept button to confirm selection. Move the cursor to the interior of the “V” shape as shown and click.

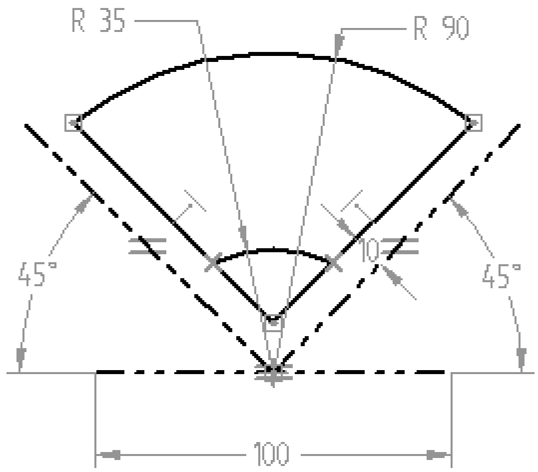




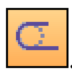
<u>Place two arcs</u>

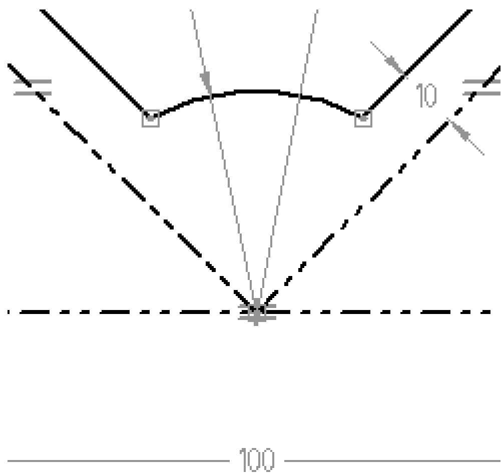
- Choose the Arc by Center command . Place two arcs as shown in the following illustration.
  - Both arc center point origins are the midpoint of the horizontal construction line.
  - Small arc Point-2 is on the left angled line, Point-3 on the right-angled line.
  - Large arc Point-2 connects to the end point of the left angled line, and Point-3 connects to the end point of the right-angled line.

- Use SmartDimension to dimension the two arcs and edit the values of the dimensions to those shown in the illustration.



Trim the sketch elements

- Choose the Trim command .
- Trim away the offset lines below the small arc. The result of the trim is shown.

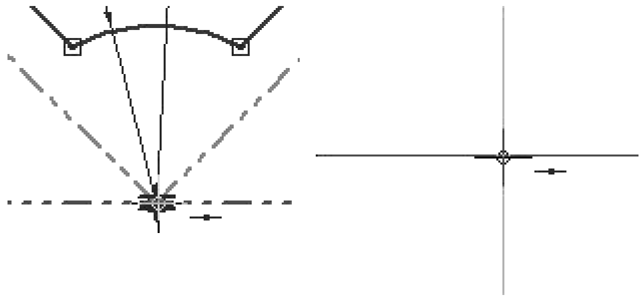


Relationship assistant

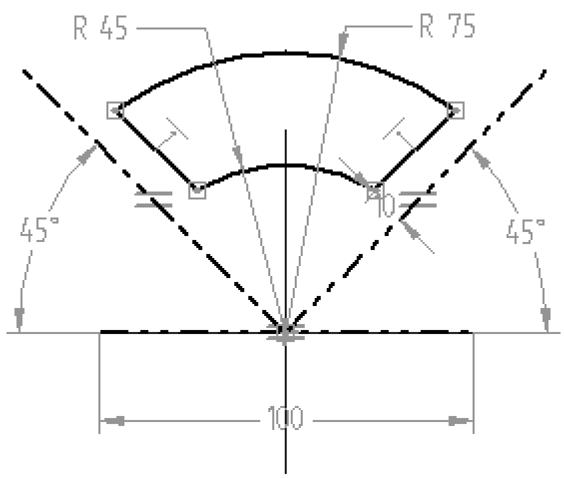
- On the Home tab® Relate group, choose Relationship Assistant. Use the Show Variability command to verify that the profile has only two degrees of freedom.

- Resolve the two remaining degrees of freedom.

  In the Relate group, choose the Connect command and place a Connect relationship between the midpoint of the horizontal construction line, as seen in the illustration to the left, and the midpoint of a reference plane, as seen in the illustration to the right. This anchors the profile and eliminates any remaining degrees of freedom.



- Edit the dimension values as shown in the illustration and then change them back to the original values. This sketch is ready to be used in a feature function such as cutout.



- This completes the activity. Close the file and save as *cutout.par*.

<u>Summary</u>

In this activity, you learned how to use construction elements, dimensions and relationships to position a profile. Design intent is maintained by positioning the construction elements. Construction elements do not become a part of the feature but are handy in controlling the position of the geometry.

Creating ordered base feature activities

Constructing ordered features from sketches

This activity demonstrates how to construct ordered features using sketches. The sketches to construct the individual features of the solid model are provided.

Activity: Constructing a model from sketches

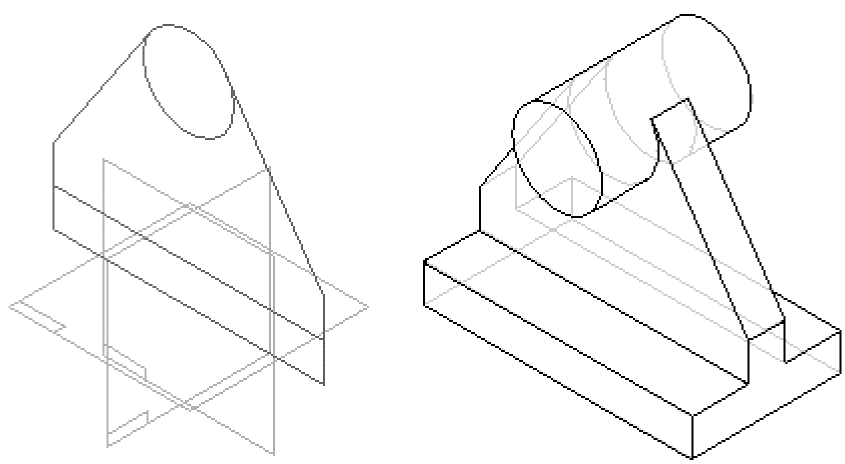
Overview

This activity demonstrates how to construct an ordered solid model using sketches. The sketches you will use to construct the individual features of the solid model are provided.

Objectives

In this activity you will:

- Use the Extrude command.
- Create multiple protrusions from a single sketch.
- Use the Include command.
- Use the Trim command.
- Apply profile relationships.

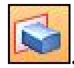


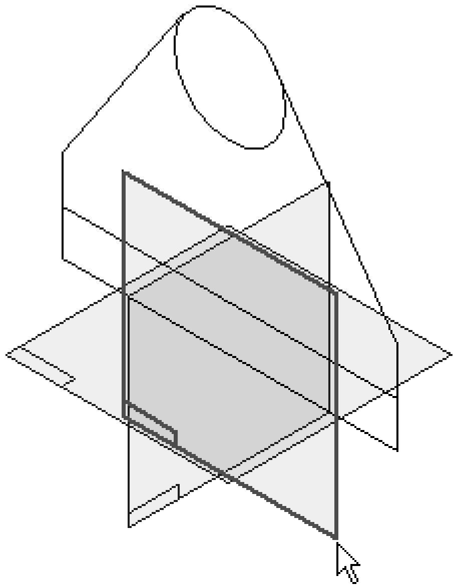
Open an existing part document

- ‣ Open *bracket02.par*. Notice that this file contains a sketch. Use this sketch to construct two protrusions. This will demonstrate how a single sketch can be used to create multiple features.

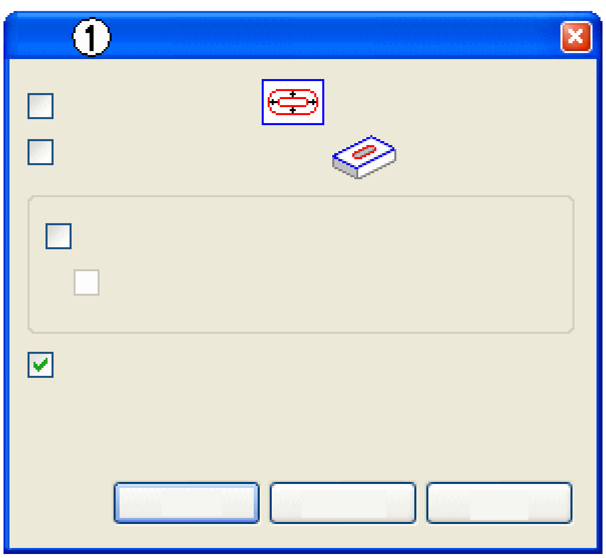
Construct the base feature

Use the Extrude command to construct the base feature of the part. Rather than draw the profile of the protrusion, include elements from an existing sketch.

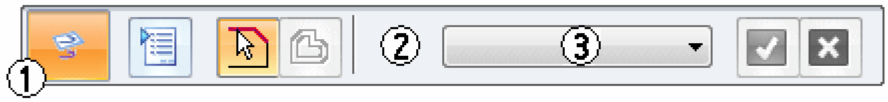
- In the Solids group, choose the Extrude command .
- On command bar, click the Coincident Plane option.
- Select the reference plane shown.



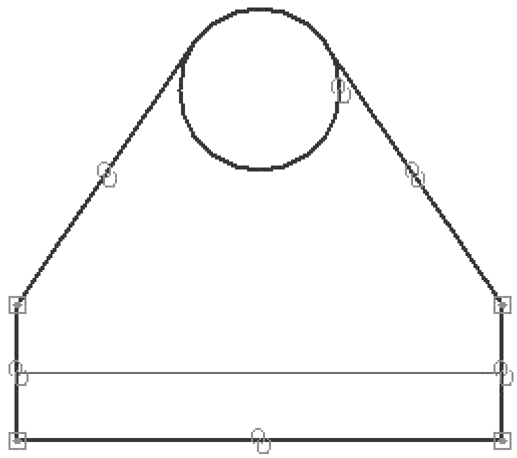
- Choose the Fit command .
- Right-click in PathFinder and choose Hide All ® Reference Planes.
- In the Draw group, choose the Include command .
- When the Include Options dialog box (1) is displayed, set the options shown in the illustration and click OK.



- The Include command bar (1) is appears as shown. Change the Select option (2) to Single Wireframe (3).

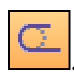
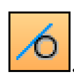


- Choose the Fit command.
- Select the elements shown in bold in the illustration to include them in the profile. Include all elements except for the top horizontal line. The link symbols indicate that the elements have been included.

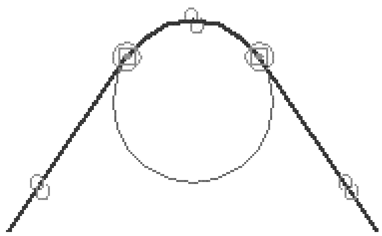


<u>Edit sketch elements</u>

The elements included from the sketch do not form a valid profile. The elements must be modified. Trim an element and apply relationships to create the final shape.

- In the Draw group, choose the Trim command .
- Select the bottom portion of the circle. The full circle is trimmed to an arc with its endpoints connected to the two angled lines. Notice that only the element included in the profile is trimmed. The original circle in the sketch remains intact.
- In the Relate group, choose the Tangent command .

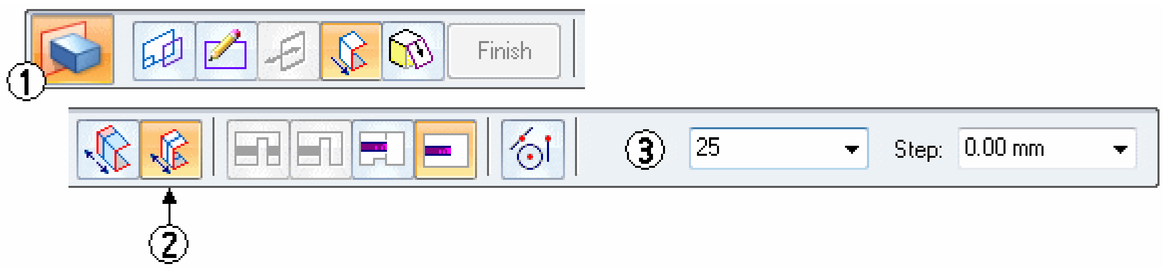
- Select the intersection between the line and the arc to place the Tangent relationship. Do this to both sides of the arc. Notice the tangent relationship handles.



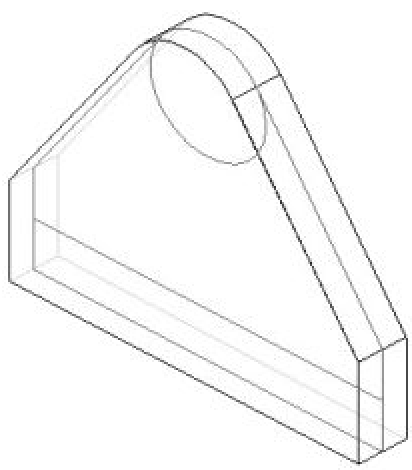
- The profile step is complete. Choose the Close Sketch command.

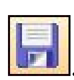
<u>Extent definition step</u>

- On the Extrude command bar (1), click the Symmetric Extent option (2). Type 25 in the Distance box (3) and press the **Enter** key.



- Click Finish to complete the extrusion. The sketch is still displayed with the new protrusion.

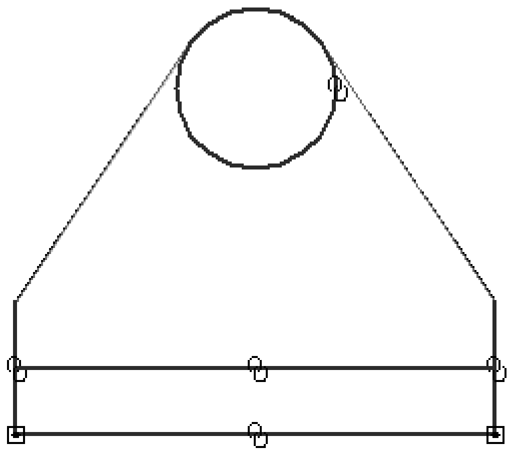


- Save the file. Choose the Save command .

Construct another profile using the same sketch

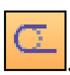
Construct another profile using the same sketch. Use the same profile plane used in the previous feature.

- Choose the Extrude command.
- In the Sketch Step, select Last Plane. This uses the same reference plane specified for the previous protrusion feature.
- Choose the Include command. Click OK on the Include Options dialog box and select the elements shown in bold. Use the Fit command to fit all of the profile elements into the profile window.

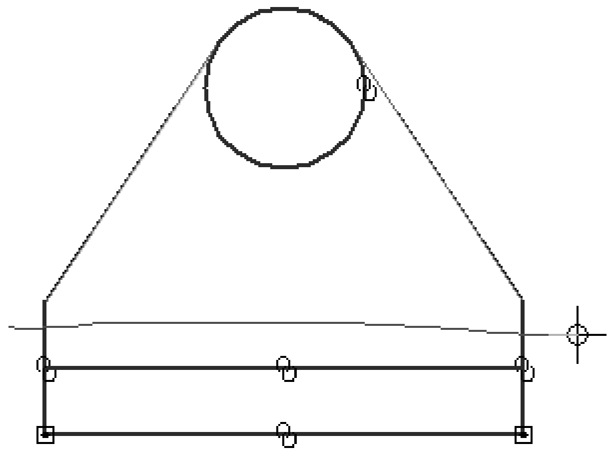


Trim sketch elements

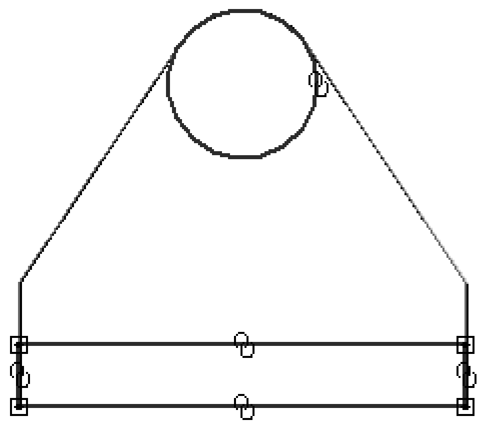
Trim the included sketch elements to complete the profile.

- Choose the Trim command .

- Click and drag the cursor across the lines shown.



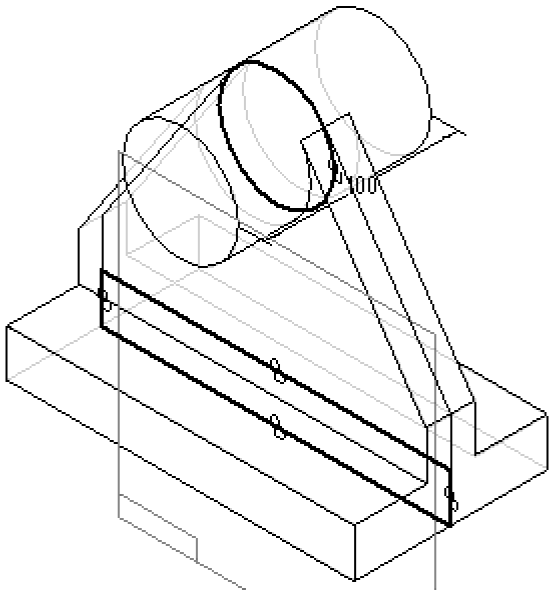
- When the mouse button is released, the trim completes. Notice that the Link symbol on the two vertical lines moves.



- The profile step is complete. Choose Close Sketch.

Extent definition step

- Click the Symmetric Extent option, type 100 in the Distance box, and then press the **Enter** key.



- To complete the protrusion, click Finish.

Turn off the sketch display

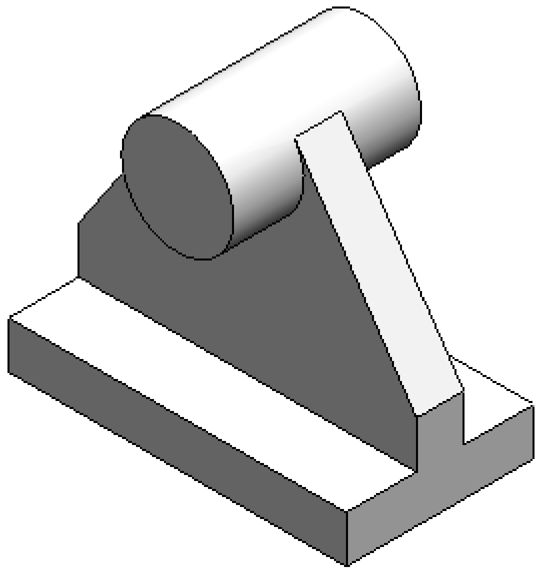
The two protrusions are complete. Turn off the display of the sketch to show only the finished part.

- Click the Select Tool, and then right-click in an open space of the model window..

- On the short-cut menu, choose the Hide All® Sketches command to turn off the display of the sketch you used to construct the protrusions.

- Choose the Fit command to view the entire part.

Save and close the file. This completes the activity.



Summary

In this activity you learned how to create features from sketch geometry. Because the geometry is associative, changes in the sketch will also change the features that are derived from the sketch.

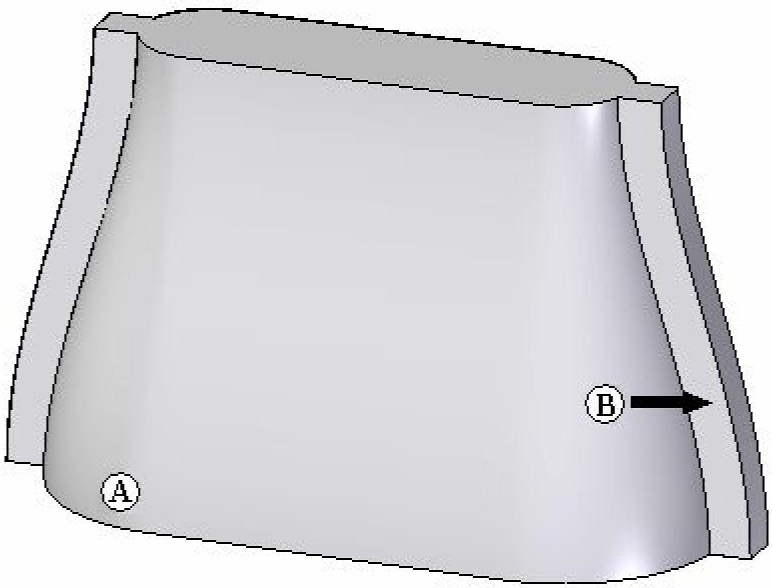
Creating a loft and swept protrusion

In this activity, construct a solid model using the Loft and Swept Protrusion commands. Edit end conditions and curves to adjust the overall shape of the model.

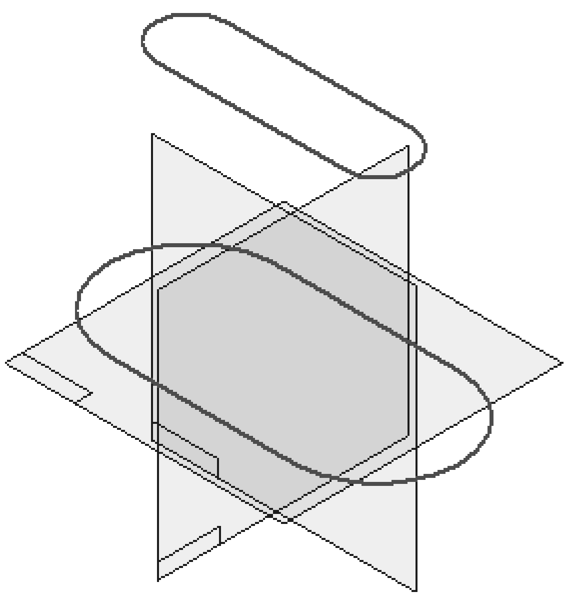
Activity: Creating a loft and swept protrusion

Objectives

In this activity, construct a solid model using the Loft (A) and Swept Protrusion (B) commands. Edit end conditions and curves to adjust the overall shape of the model.

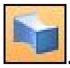


- Open *loft.par*. This file contains sketches and curves that will be used to model the part.

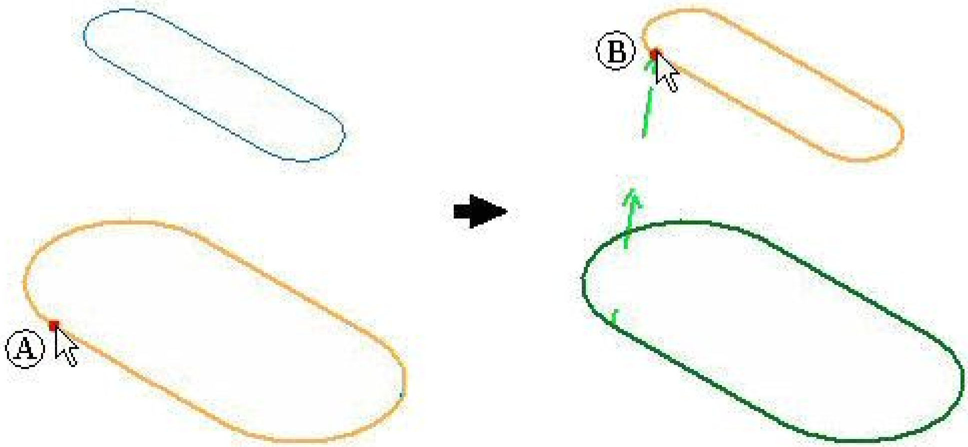


Create a loft protrusion

Create a loft protrusion using sketches provided in the file.

- On the Home tab® Solids group, choose the Loft command in the Add drop down list .
- Hide the reference planes.
- Select the sketch (labeled base sketch) at location (A) shown for the first cross-section.

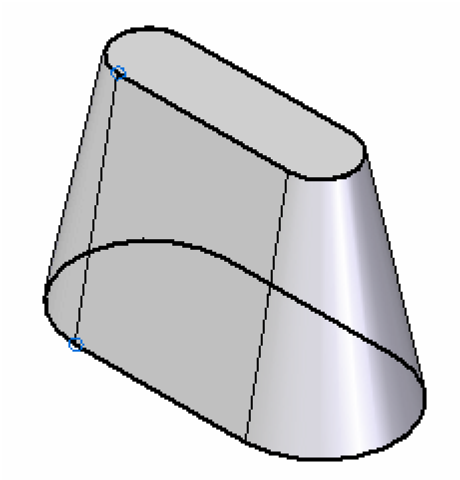
  Select the sketch (labeled top sketch) at location (B) shown for the second cross-section.



Note

It is important to select the cross-sections at start locations where a twist will not be introduced in the geometry (or a self-intersecting result). If this condition occurs, an error message displays.

- On the command bar, click Preview. Do not click Finish.



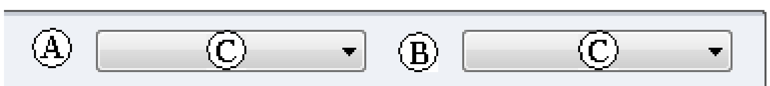
Note

The result shown above uses the default end-condition of "Natural". This is where the cross-sections connect using a linear vector.

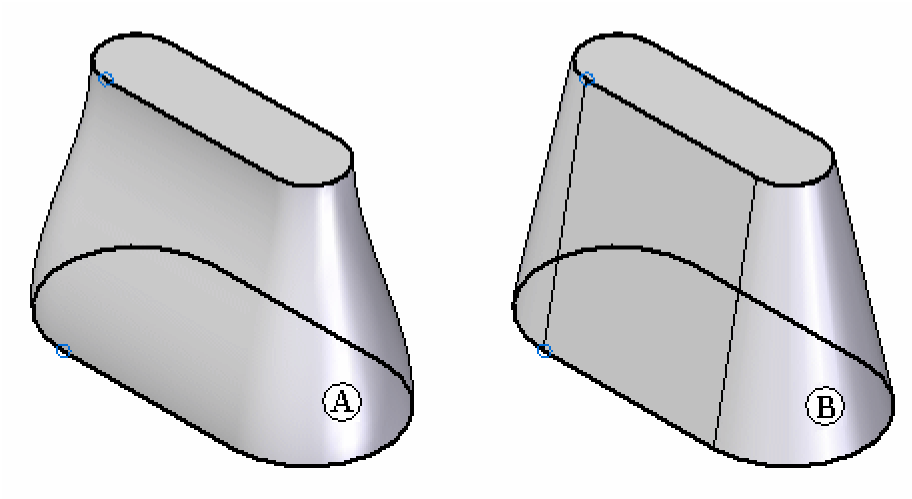
Edit end conditions

- On command bar, click the Extent Step.

- Change the end-conditions of both cross-sections. Set both End 1: (A) and End 2: (B) to"Normal to section" (C). This setting creates a lofted feature where the surface starts and ends with a normal vector to the cross-sections.



- Click Preview and then click Finish. Notice the results [(A) Normal to section, (B) Natural].



Add guide curves

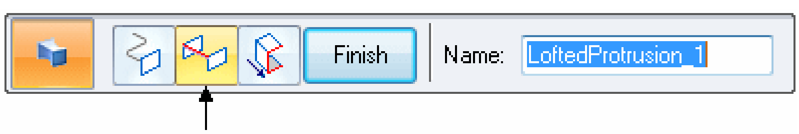
Add guide curves to further control the overall shape of the lofted protrusion feature. Edit the definition of the lofted protrusion completed in the previous step.

- Turn on the display of curves. In PathFinder, click the check box on the curves named *side curve 1*, *mirrored side curve 1*, *side curve 2* and *mirrored side curve 2*.

- Click the Select tool and then select the protrusion in the part window.

- Click Edit Definition.

- On command bar, click the Guide Curve step.



- Select each curve and then click the Accept button. Select and accept only one curve at a time. The right-mouse button or Enter on the keyboard can also be used to accept the guide curve.

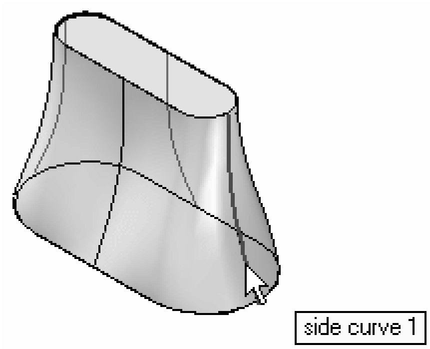
- After selecting all four curves, click the Preview button.

- Notice how the shape of the loft protrusion follows these guide curves. Dynamically rotate the model to better observe the shape. Click Finish.

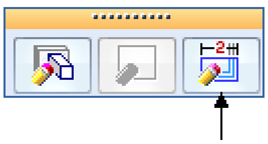
<u>Edit the guide curves</u>

Continue to refine the loft protrusion shape by editing the guide curves. When one curve is edited, the curve on the opposite side will adjust automatically because it is a mirrored element.

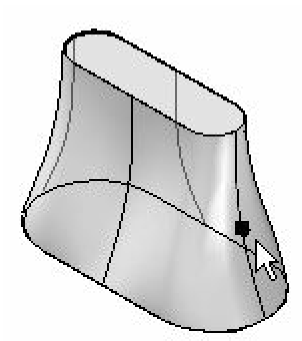
- Click the Select tool.
- Select curve named *side curve 1*.



- Click the Dynamic Edit button.

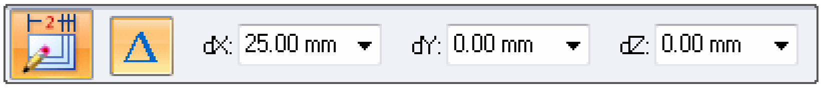


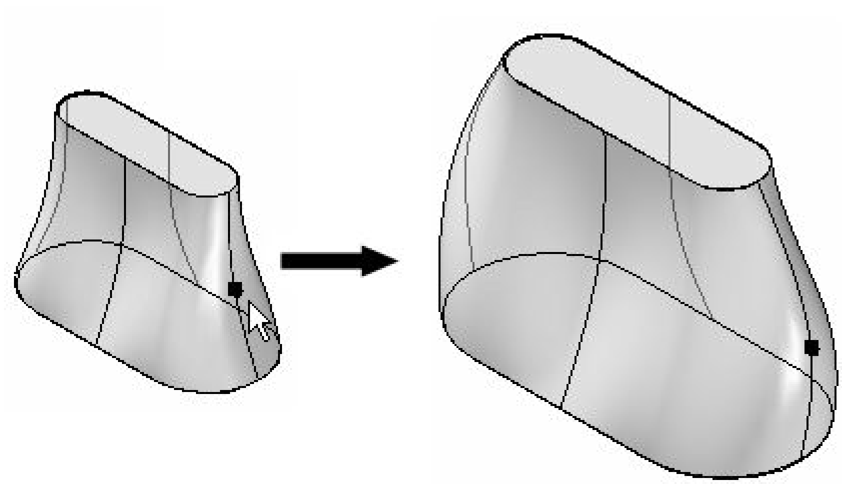
- Select the green dot on the curve. This will be the edit point on the curve.



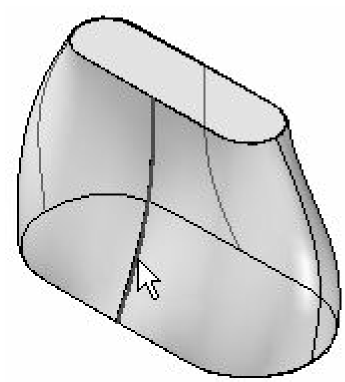
- Click the Relative/Absolute Position button . Absolute uses the actual X-Y-Z coordinates for positioning. Relative uses a delta distance for positioning. Use relative positioning.

- Type 25 in the dX: field and press the Enter key. This moves the edit point 25 units in the positive X direction and 0 units in the Y and Z direction. The edit is made when the Enter key is pressed. Each time the Enter key is pressed after this point will apply a move again of the values displayed in the ribbon bar delta fields.



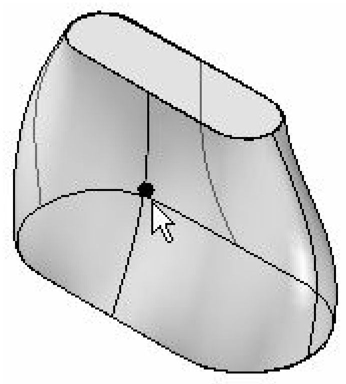


- Click the Select tool.
- Select curve named side curve 2.



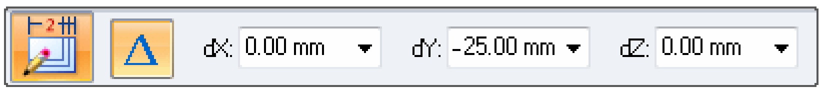
- Click the Dynamic Edit button.

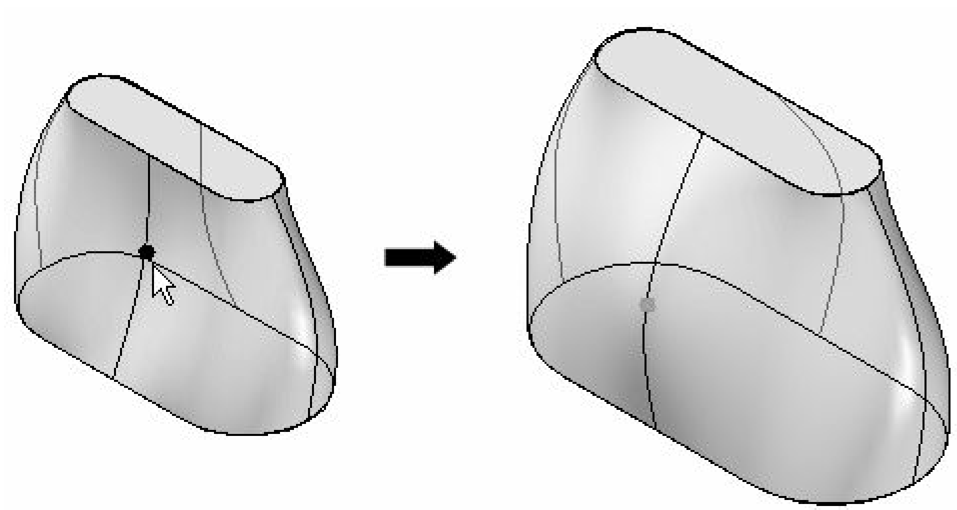
- Select the green dot on the curve. This will be the edit point on the curve.



- Click the Relative/Absolute Position button.

- Type -25 in the dY: field and press the Enter key. This moves the edit point 25 units in the negative Y direction and 0 units in the X and Z direction. The edit is made when the Enter key is pressed.





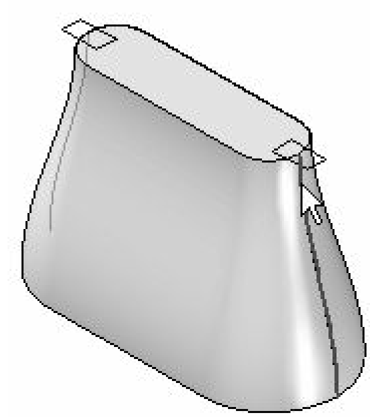
- Continue to modify the shape on your own. This completes this portion of the activity. Close and do not save the file.

<u>Create a swept protrusion</u>

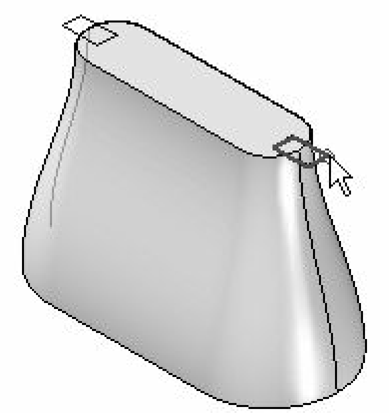
- Open *sweep.par*. This file contains sketches and curves to use to define swept protrusions.

- The curves provided were created using the project curve onto surface command. This command is not be covered in this course. These are the trace curves for the swept feature. Lines, arcs, curves, etc. can be used to define the path trace for the sweep.

- On the Home tab® Solids group, choose Swept Protrusion command on the Add drop down list .
- On the Sweep Options dialog box, click the Single path and cross section option. Click OK.
- Select the curve shown.



- Click the Accept button (or right-click) to accept the trace curve.
- The cross section select step is now active. Select the sketch as shown for the cross section.



- On the command bar, click Finish.
- Repeat the previous steps to create a swept protrusion on the opposite side.
- Hide the curves and sketch. Right-click in the part window and choose Hide All ® . Curves. Choose Hide All ® . Sketches.
- This completes the activity. Close the file.

Summary

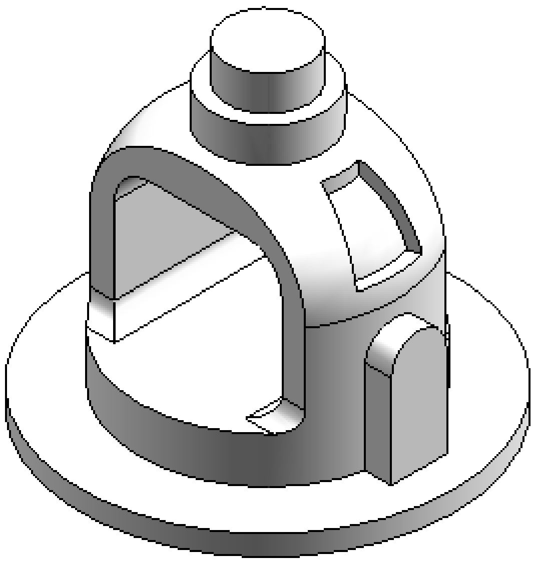
In this activity you learned how to create both a swept protrusion and a lofted protrusion. To better manage the geometry, sketches were used to define the cross sections to be swept and lofted. Guide paths were used to control the transition of geometry between cross sections.

Creating profile-based ordered features activity

This activity demonstrates the construction of profile-based features.

Construct a revolved protrusion and then add cutouts and secondary protrusions.

Use the following commands to create profile-based features: (revolve, extrude, cut, revolved cut, select from sketch, parallel plane, profile, mirror, fillet, include, trim).



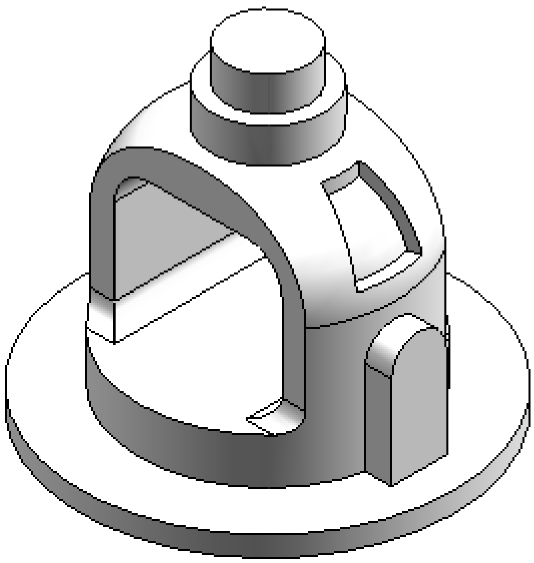
Activity: Creating profile-based features

Overview

This activity demonstrates the construction of profile-based features.

Objectives

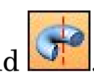
Construct a revolved protrusion and then add cutouts and secondary protrusions.



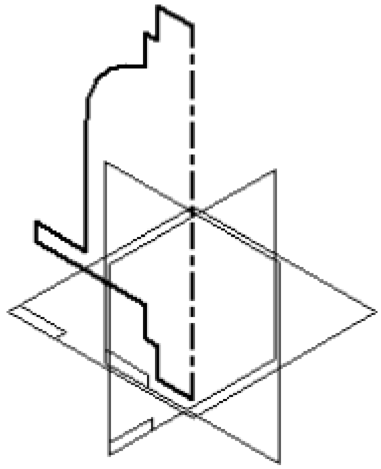
- Open *bell.par*.

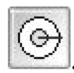
Create a revolved protrusion

Create a revolved protrusion using the sketch provided with the file.

- On the Home tab® Solids group, choose the Revolve command .
- In the Sketch step, click the Select from Sketch option.
- In the part window, select the sketch and then, click the Accept button.

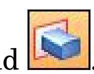
- For the axis of revolution, select the vertical dashed line.



- On the command bar, click Revolve 360° .
- Click Finish.
- The sketch and axis of revolution are no longer needed. Turn off their display. Right-click in the part window. Choose Hide All® Sketches and choose Hide All® Reference Axes.

Create an extrusion

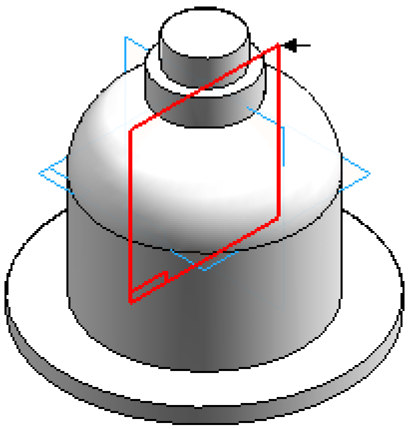
Create an extrusion. Draw the profile on a parallel plane.

- In the Solids group, choose the Extrude command .
- In the Sketch step, click the Parallel Plane option.

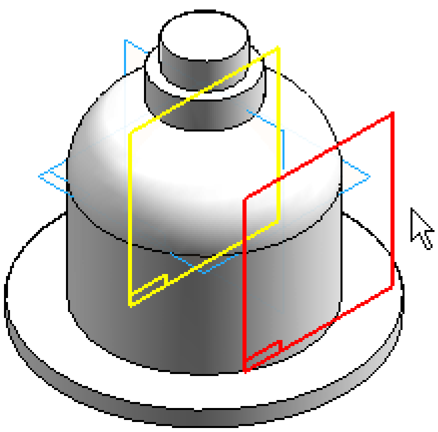
- Select the reference plane shown.

  Note

  Throughout this activity, hidden lines and reference planes are removed from illustrations for clarity.



- Type 82.5 in the Distance box and press the Enter key.
- Move the cursor to the bottom right of the window, and click to define the location of the new parallel reference plane.



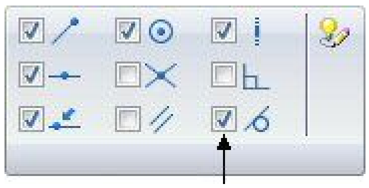
- Choose the Fit command .

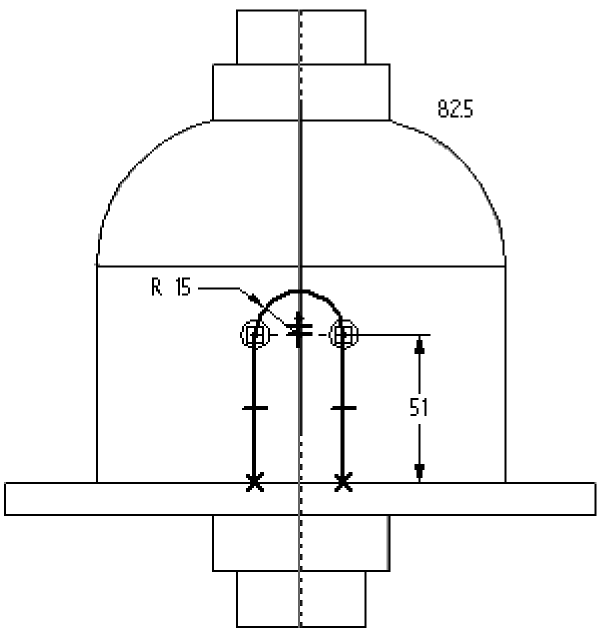
- In the Draw group, use the Line command to draw the profile shown. Draw the profile with the same dimensional values and relationships shown below. Notice the vertical relationship between the midpoint of the vertical reference plane and the center of the profile arc.

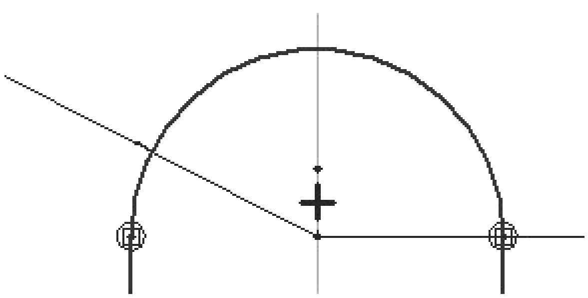
  Note

  Within the line command, press A on the keyboard or click the arc option on the ribbon bar to enter arc mode. Once you place the arc, the command reverts back to line mode. When in arc mode, notice the intent zones available for arc placement.

  Turn on the Tangent option in IntelliSketch. This applies a tangent relationship when placing the arc.





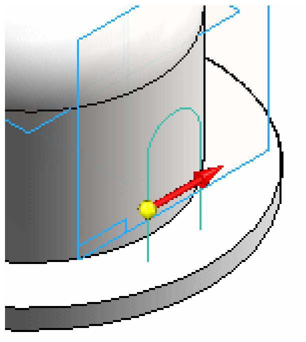


- Choose Close Sketch.

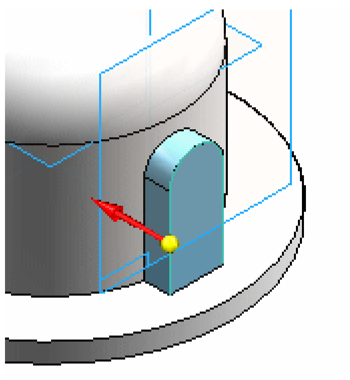
- Move the cursor so that the arrow points as shown and click. This will add material to the inside of the profile.

  Note

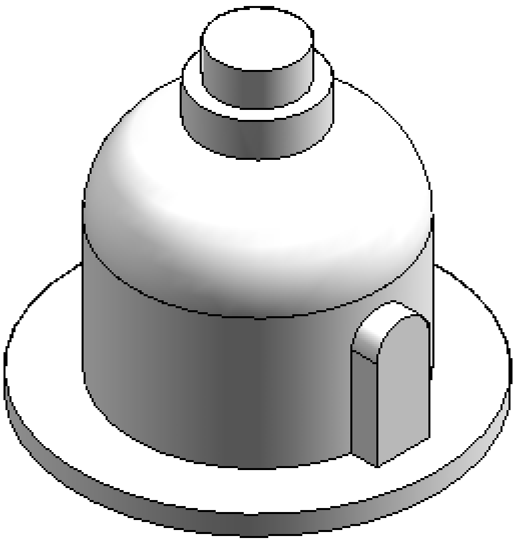
  Notice the side step on command bar. When you use an open profile, you must specify the side of the profile to add material to.



- On command bar, click Through Next.
- Move the cursor so that the arrow points as shown and click.

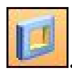


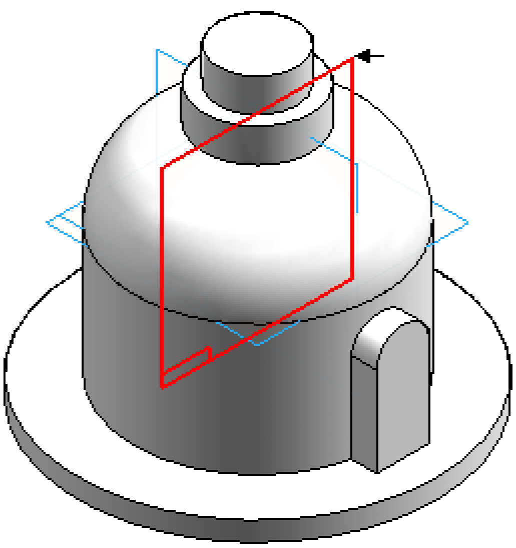
- Click Finish to complete the protrusion.



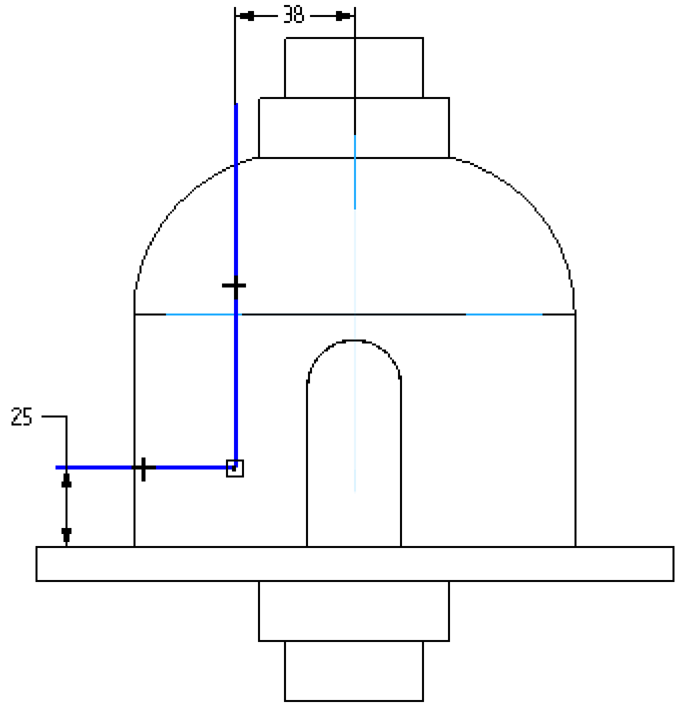
Remove material from the base feature

Remove material from the part using an open profile.

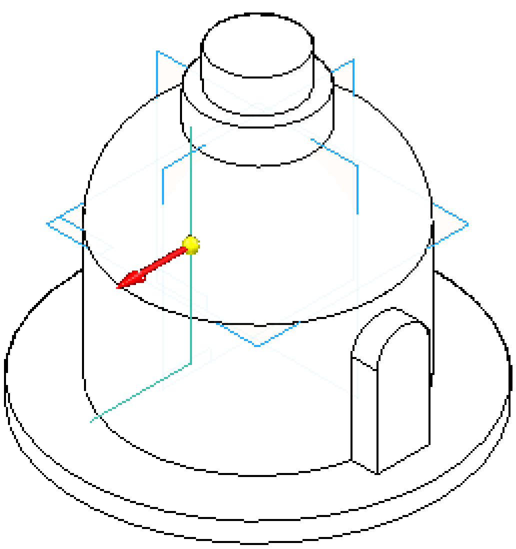
- Choose the Cut command .
- On command bar, click Coincident Plane from the plane type list. Select the reference plane shown.



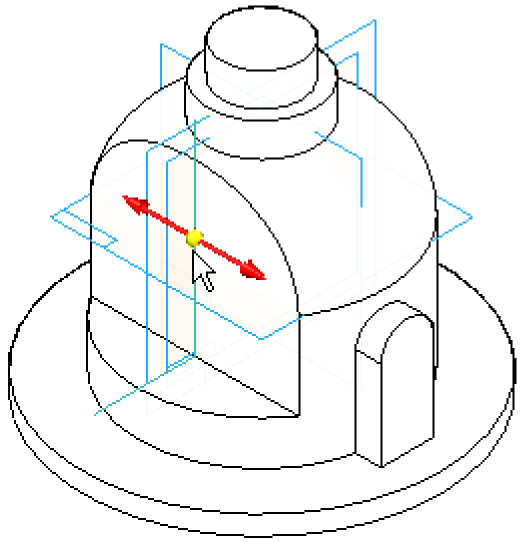
- Draw the open profile.



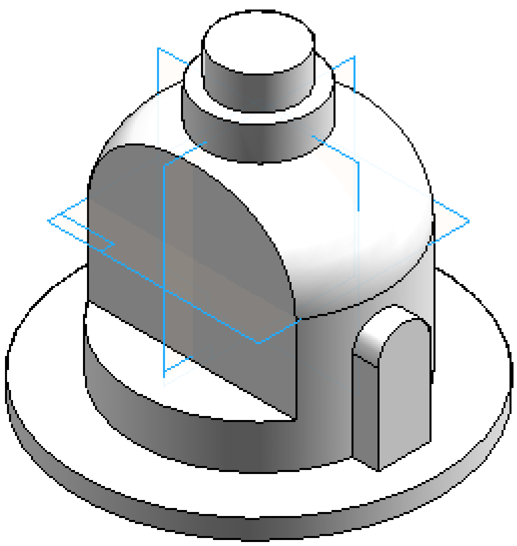
- Click Close Sketch.
- Position the direction arrow as shown to remove material outside the open profile.



- On command bar, click the Through All extent option. Position the arrow as shown to remove material in both directions.

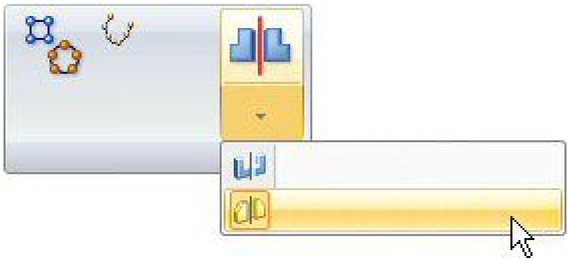


- Click Finish.

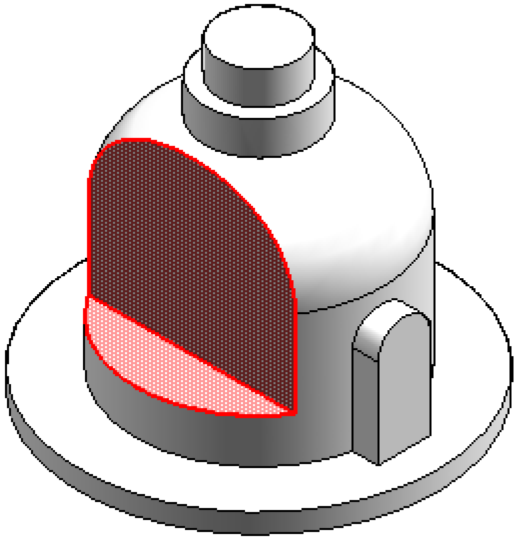


Mirror the cutout feature

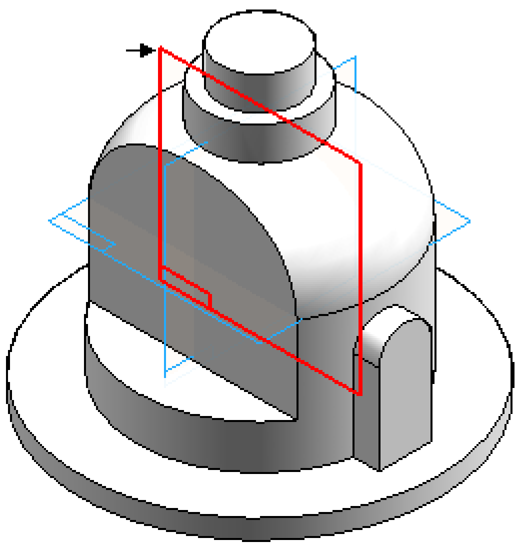
- In the Pattern group, choose the Mirror Copy Feature command on the Mirror drop down list.



- Select the cutout feature.



- On command bar, click the Smart option and then click the Accept button.
- Select the Front (xz) reference plane as the mirror plane.

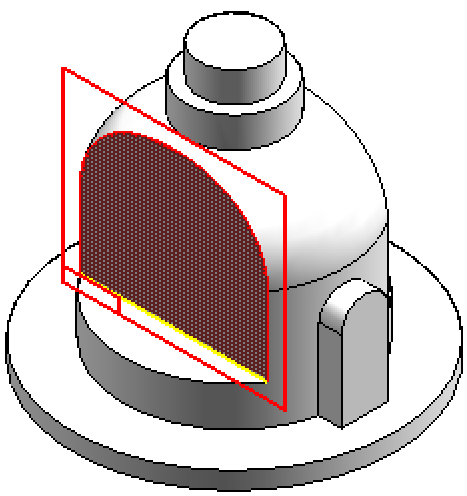


- Click Finish.

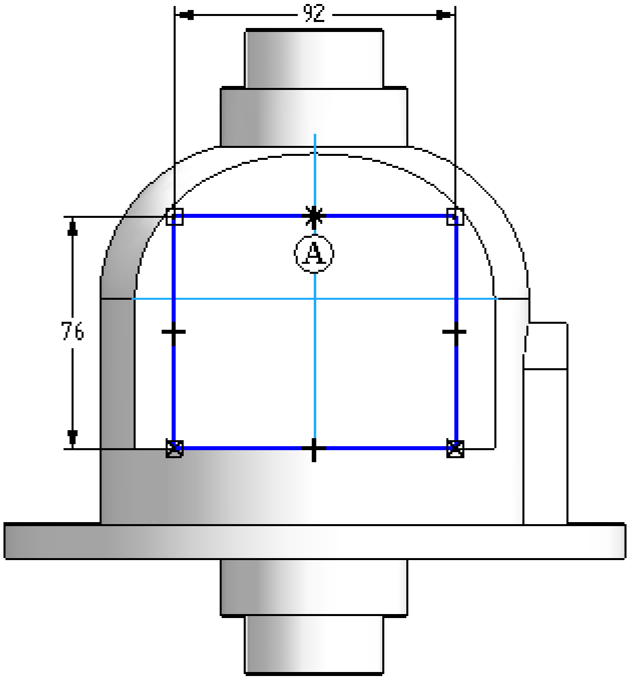
Remove material from part

Remove material from the middle of the part using a closed profile.

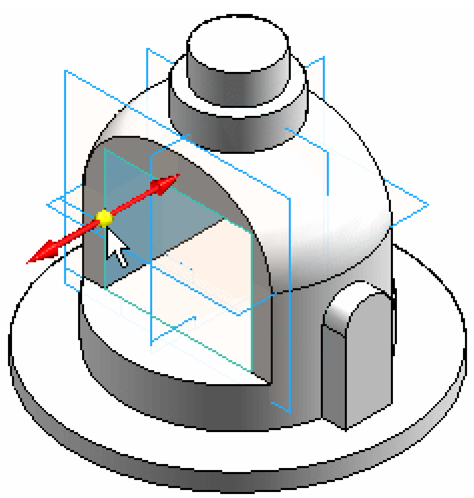
- Choose the Cut command.
- Select the reference plane shown.



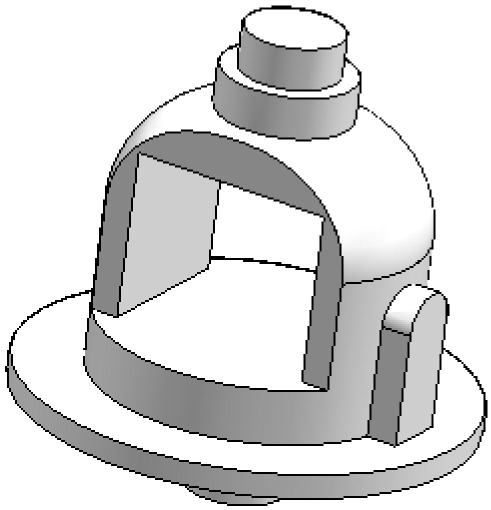
- Draw the profile. Connect the midpoint of the top line segment to the vertical reference plane (A).



- Click Close Sketch.
- Click the Through All extent option. Position the arrow as shown to remove material in both directions.



- Click Finish.



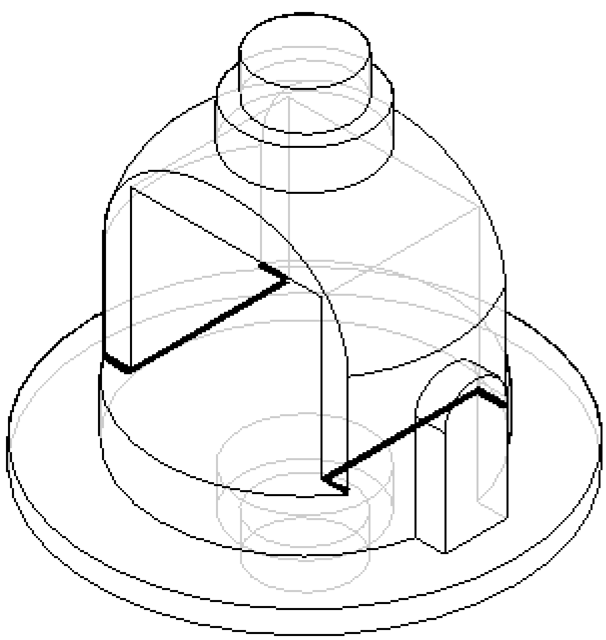
- Save the file.

Add rounds

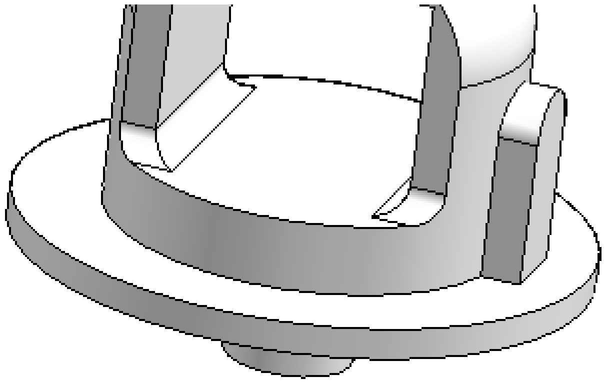
Round edges of the cutout features.

> **Note**
>
> The Constructing treatment features self-paced course (spse01530) covers rounding. It is appropriate at this point to add rounds to the part.

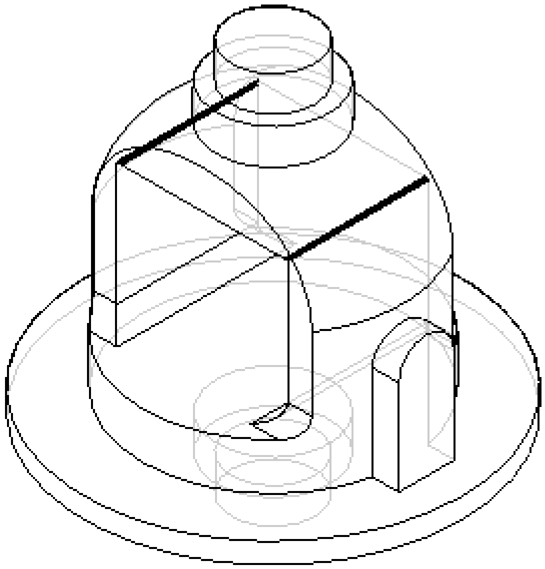
- In the Solids group, choose the Round command .
- Select the six edges as shown.

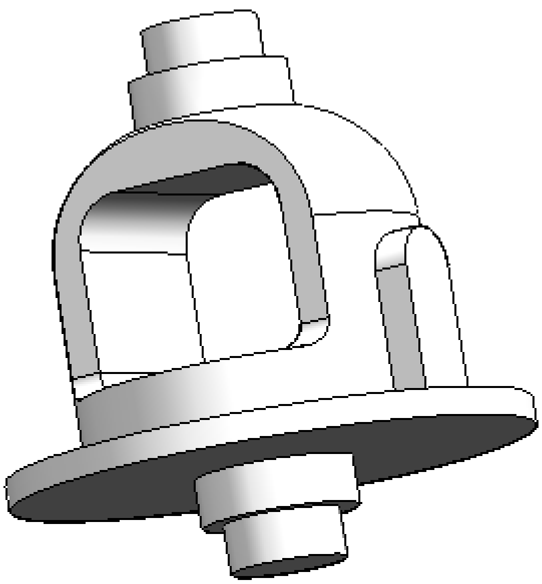


- Type 10 for the radius and then click the Accept button.
- Click Preview then click Finish.



- Place 19 mm rounds on the two edges shown.



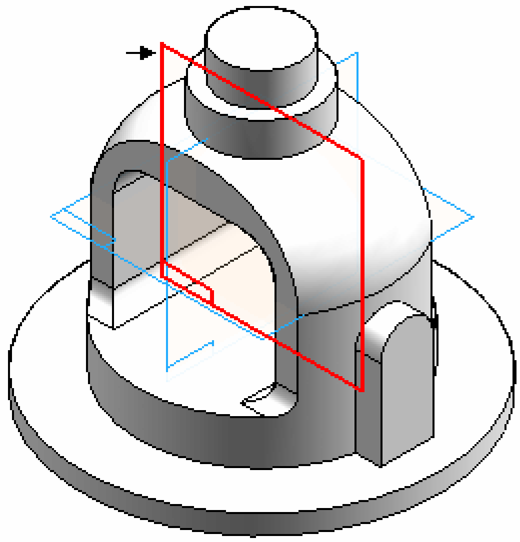


Add a revolved cutout

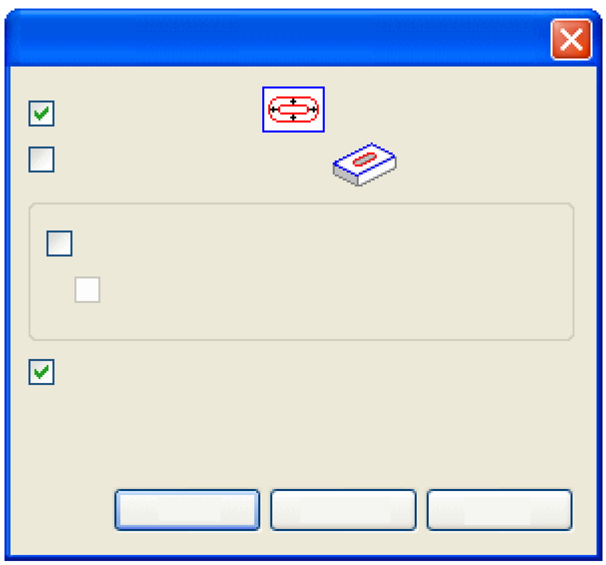
Add a revolved cutout to the part. To create this cutout, include and offset an existing part edge.

- Choose the Revolved Cut command .

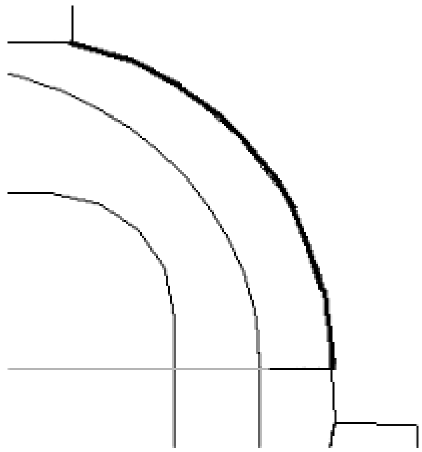
- Select the reference plane as shown.



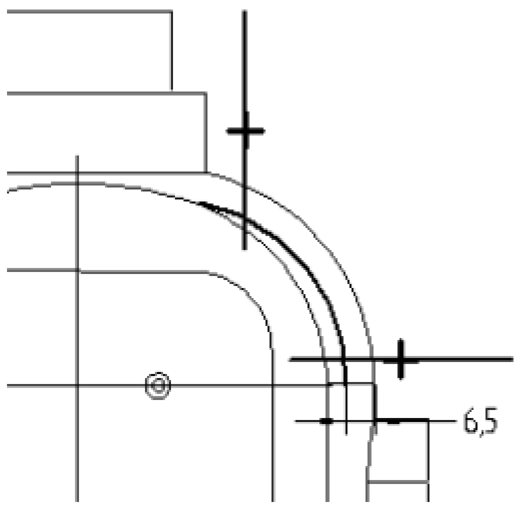
- In the Draw group, choose the Include command .
- On the Include Options dialog box, set the Include with offset option and click OK.

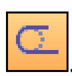


- Select the arc shown, and on the command bar click the Accept button.

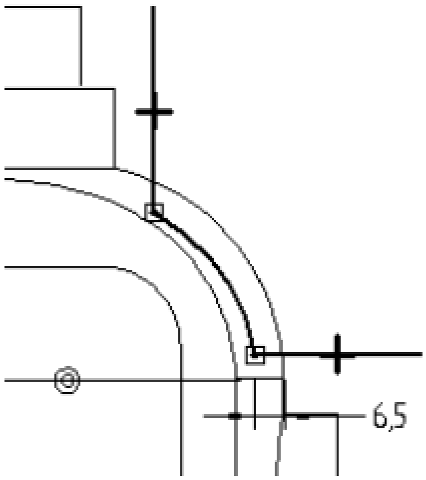


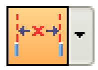
- Type 6.5 in the Distance field and press the Enter key.
- Click inside the arc to accept the offset. Notice that the system places a dimension between the offset element and the arc from which it is offset.
- Draw a horizontal line and a vertical line as shown.

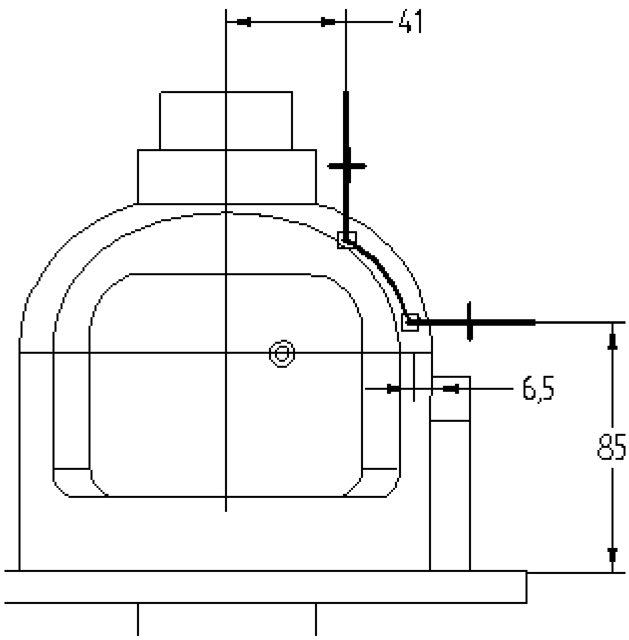


- Choose the Trim command .

- Trim away the lines and arc to produce the following profile shape. If a mistake is made, click Undo  and repeat the step.

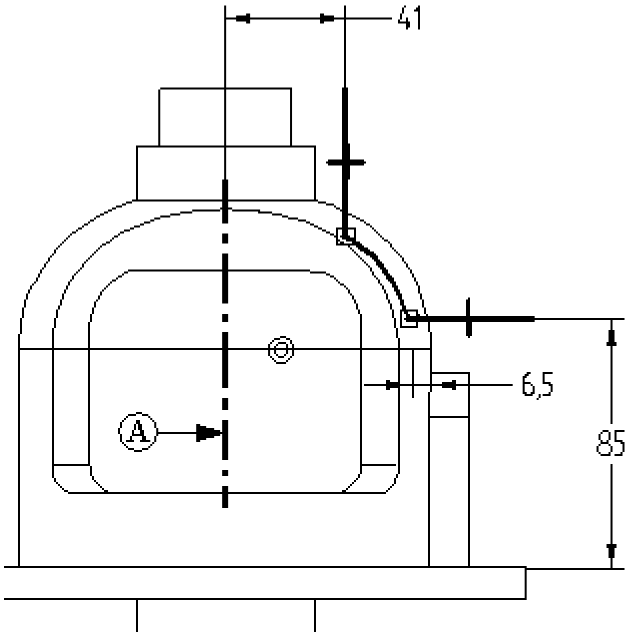


- Choose the Distance Between command , and place dimensions as shown. Edit the values of the dimensions to the values shown below.

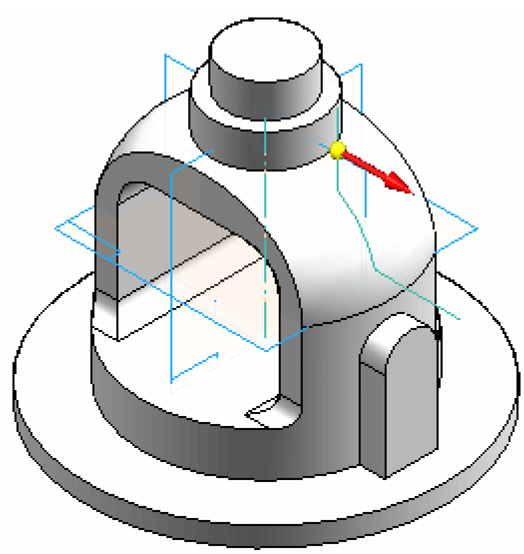


- Click the Axis of Revolution command .

  To define the axis of revolution, select the reference plane labeled (A).

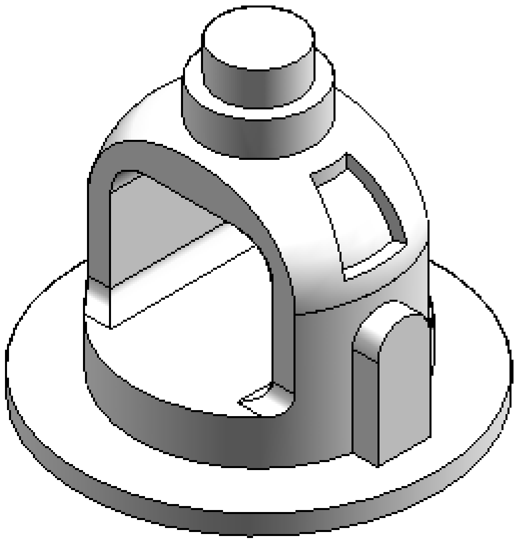


- Click Close Sketch.
- To define the direction of material removal, position the cursor so that the arrow points to the outside of the part, and click.



- On command bar, click the Symmetric Extent button. Type 30 in the Angle field and then press the Enter key.

- Click Finish to complete the revolved cutout.



- Save and close the file. This completes the activity.

Summary

In this activity you learned how to create a base feature and then construct additional features to complete the part. The include command used existing geometry which made the features associative. Because the geometry is associative, it will respond predictably to modifications. An open profile in the Revolved Cut command was used to show that the profile adjusts itself to intersect the face of the protrusion it is cutting.

Miscellaneous activities

Constructing a mouse base

In the following activity, construct a computer mouse base. This activity reinforces the feature construction techniques you have already learned, and it utilizes treatment features.

Activity: Constructing a mouse base

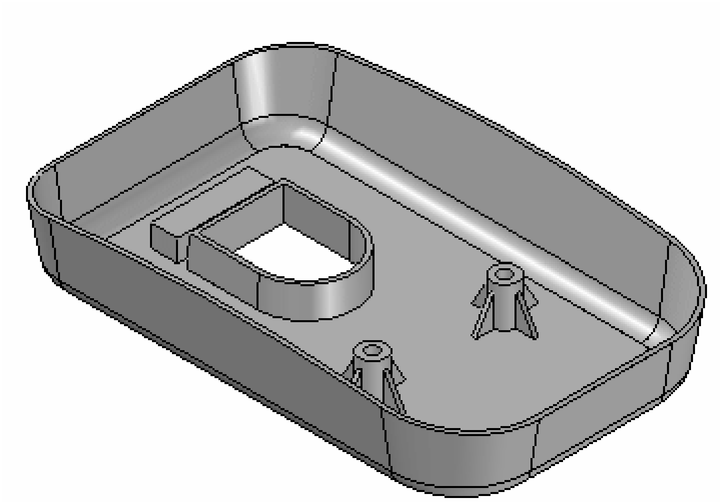
Overview

Construct the computer mouse base shown in the illustration. This activity reinforces the ordered feature construction techniques you have already learned, and it utilizes treatment features.

Objectives

In this activity, learn how to:

- Construct a solid model with holes, cutout, and draft.
- Use the Thin Wall command.
- Use the Mounting Boss command.
- Use PathFinder to select features.



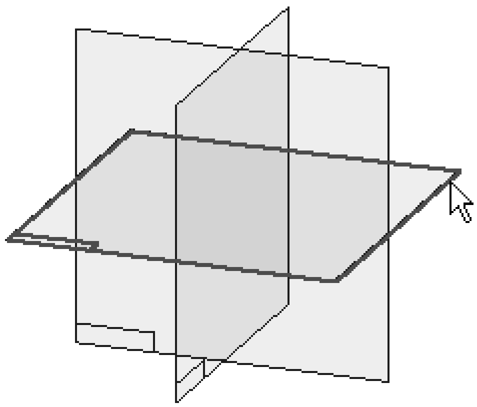
- Create a new ISO part file.
- Make sure you are in the ordered environment.

<u>Create the base feature</u>

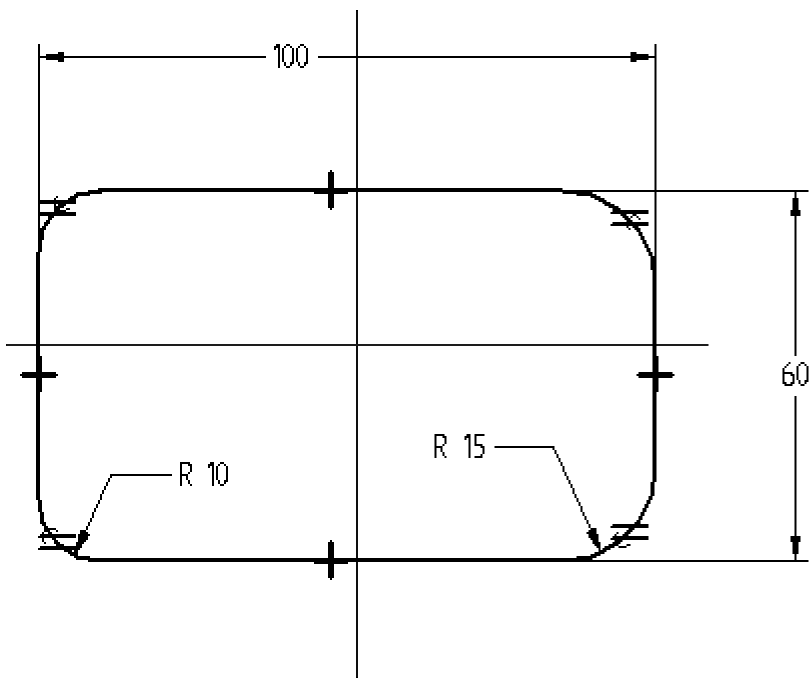
Create an extrusion as the base feature for the mouse.

- In PathFinder, turn off the base coordinate system display. Turn on the base reference planes display.

- Choose the Extrude command.
- On the command bar, click the Coincident Plane option, and select the reference plane shown.



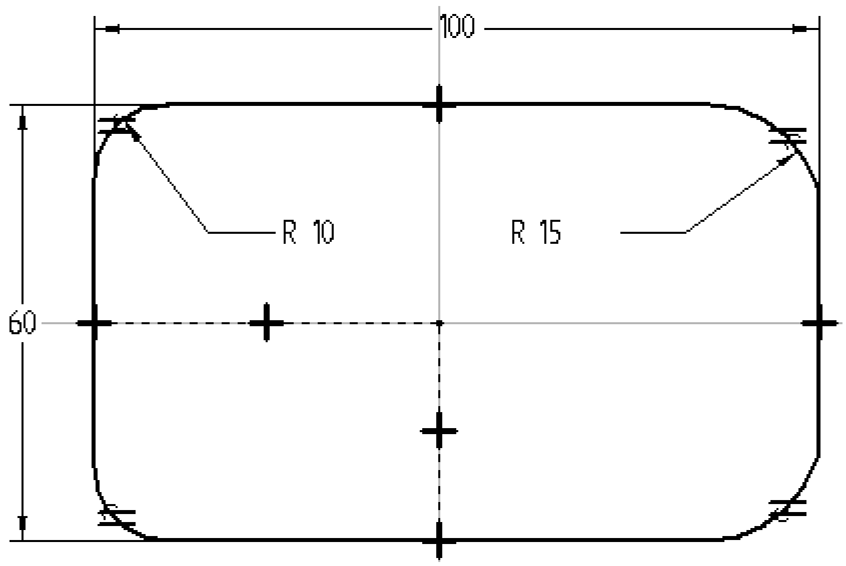
- Draw the profile.



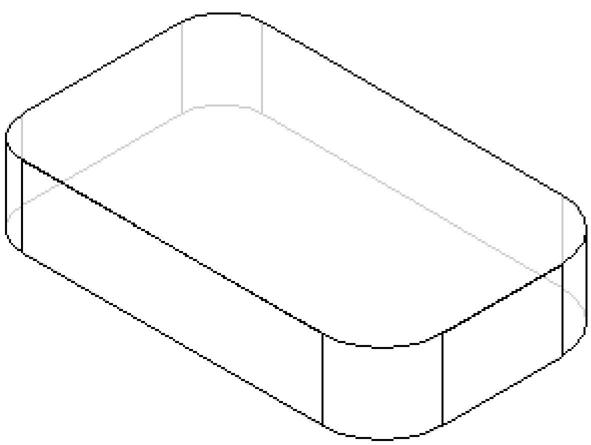
- Place Horizontal/Vertical relationships to center the profile on the midpoints of the reference planes.

  Note

  Fillets (R 10 and R 15) are in two places with equal relationships applied.

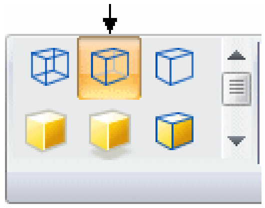


- Click Close Sketch.
- Extend the profile upward 20 and click Finish.



- Hide all reference planes.

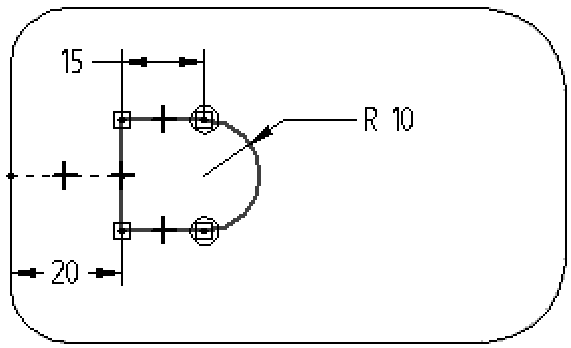
- Change the display of the part. In the Styles group, click the Visible and Hidden Edges display.



Create a cutout

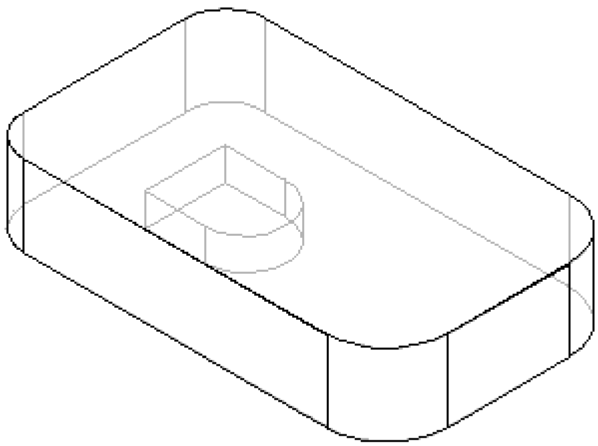
Create a cutout on the bottom side of the part.

- Choose the Cut command.
- Use the reference plane used to create the base feature. On command bar, select the Last Plane option.
- Draw the profile and apply the dimensional constraints.



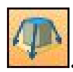
- Click Close Sketch.
- On command bar, click the Finite Extent option, and in the Distance box type 8.

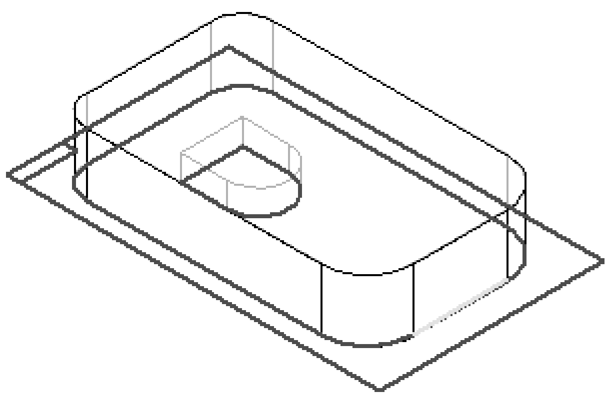
- Project the cutout upward, and finish the feature.



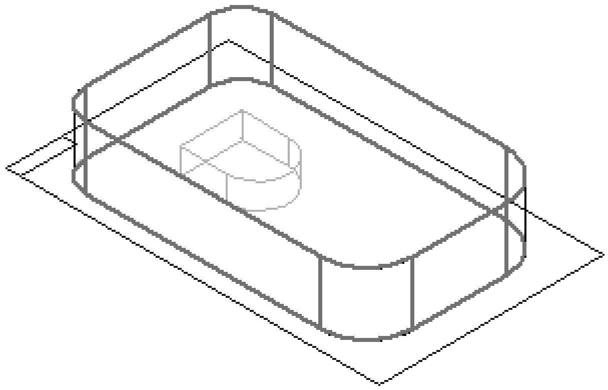
- Save the file as *mouse.par*.

Apply draft to the part

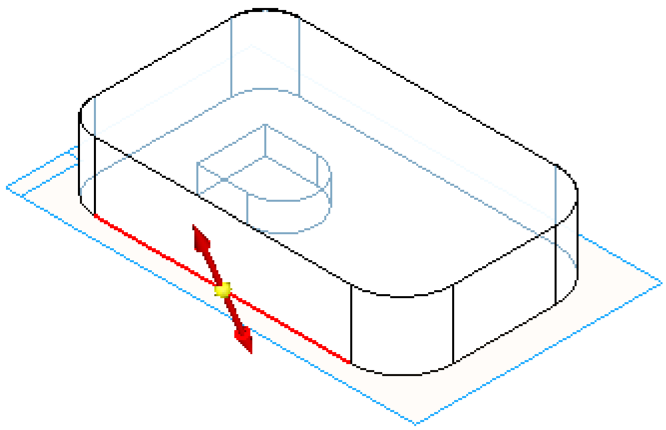
- In the Solids group, choose the Draft command .
- For the Draft Plane Step, select the bottom face as shown.

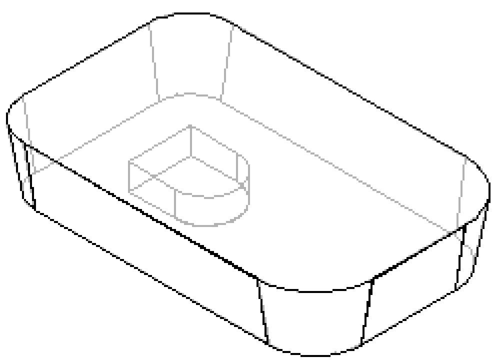


- For the Select Face Step, select one side face of the mouse base. All side faces of the mouse base should highlight. The default Select option is set to Chain which selects all chained faces not parallel to the draft plane.



- Type 10 in the Draft Angle field, and click the Accept button.
- You can specify different draft angles for multiple faces in the Select Face Step. If no other faces are to be drafted, click Next to leave the Select Face Step
- For the Draft Direction Step, orient the direction as shown so that the draft is applied outward, and then click.

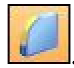


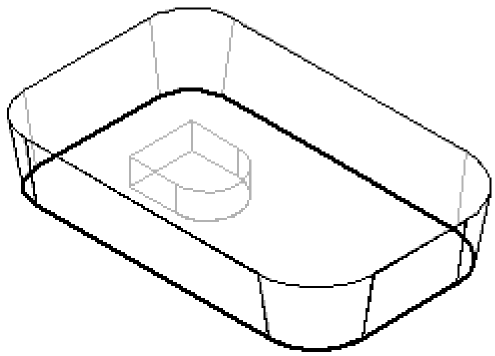


- Click Finish.

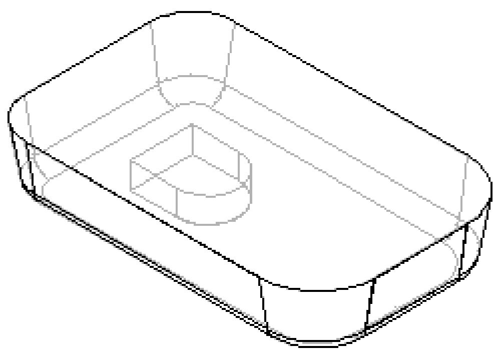
Add a round

Add a round feature to the bottom edge of the part.

- Choose the Round command .
- For the Select Step, identify the edges to round. On command bar, in the Select box, click the Chain option. This lets you select a connected chain of edges with one click.
- Select the chain of edges around the bottom face of the part as shown.



- Type 5 in the Radius field and click the Accept button.
- Use the default parameters. Skip the Round Parameters Step. Click Preview and then Finish.



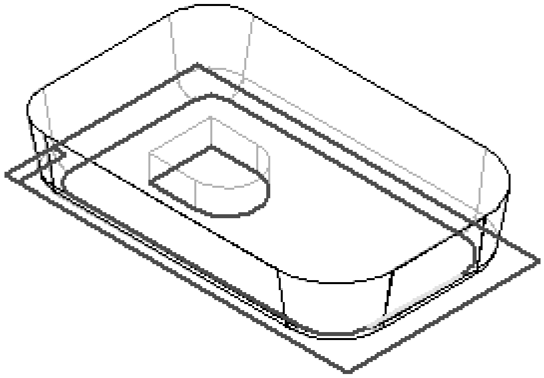
- Save the document.

Add draft

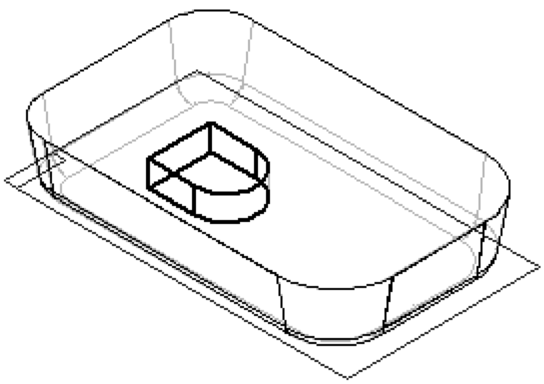
Add draft to the cutout feature in the part.

- Choose the Draft command.

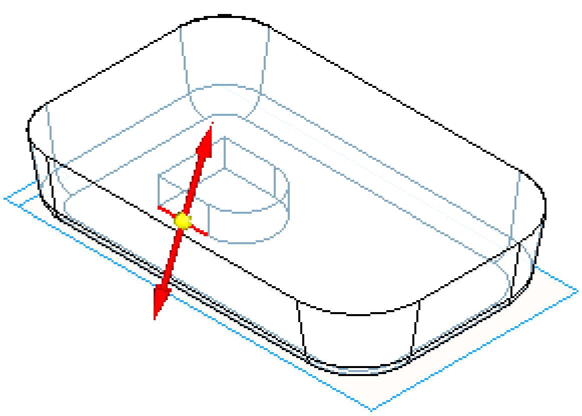
- Use QuickPick to select the bottom face to define the draft plane as shown.



- Select the chain of faces that form the sides of the cutout. Click once to select the three faces that are tangent to each other, and click once more to select the remaining face.



- Type 2 in the Draft Angle field and click the Accept button.
- Click Next.
- Orient the draft direction as shown, and then click to accept.



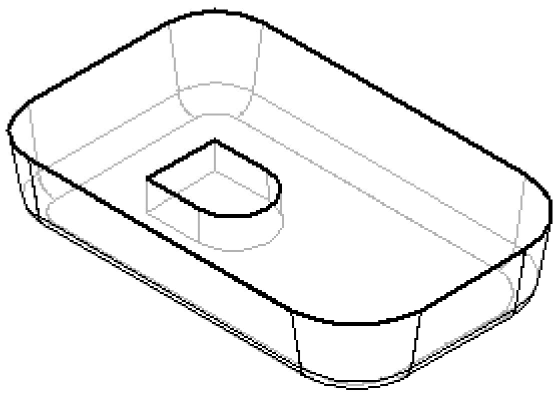
- Click Finish.

- Save the file.

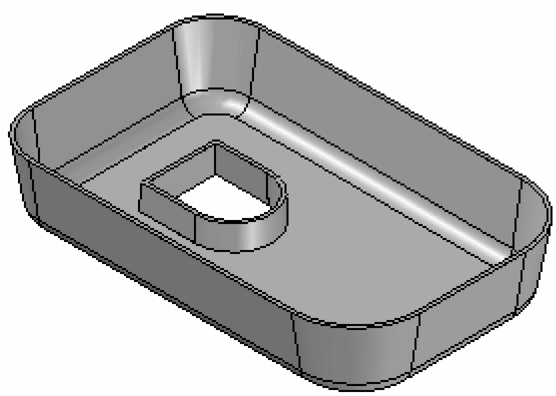
Apply a thin wall feature

Use the Thin Wall command to remove the interior material from the part.

- Choose the Thin Wall command.
- For the Common Thickness Step, specify the thickness to apply to all faces of the part. In the Common Thickness box, type 1 and press the Enter key.
- For the Open Faces Step, select the top face of the part and the top face of the cutout as the open surfaces.



- Click the Accept button to accept the faces.
- You can apply unique thickness to faces of the part. To skip this step, click Preview to process the thin wall. Click Finish to complete the feature placement.
- Click the Shaded with Visible Edges display.

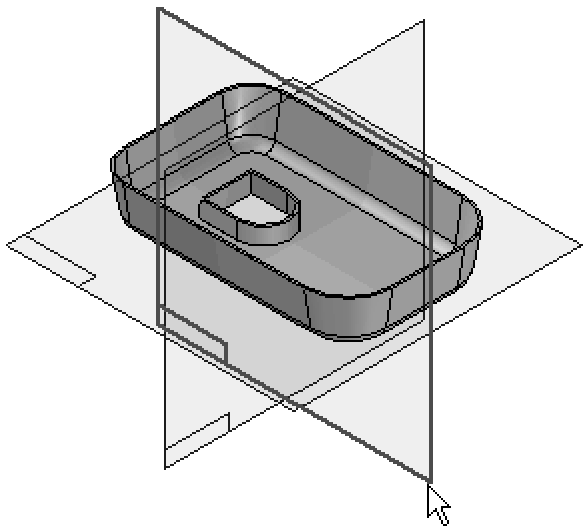


Add a cutout

Add a cutout to remove material from the top of the mouse base.

- Right-click in the part window and click Show All ® Reference Planes.

- Choose the Cut command, and select the reference plane shown.

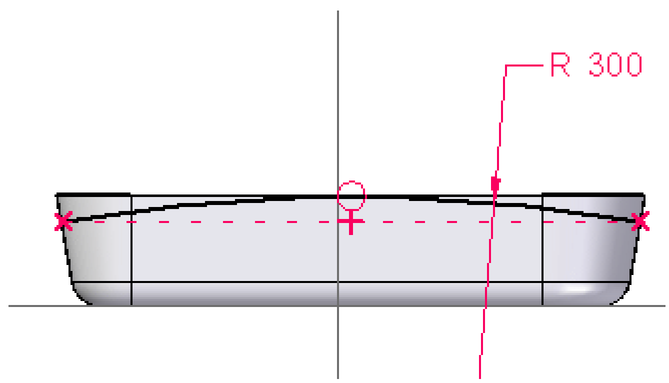


- Choose the Arc by 3 Points command, and place an arc that touches the two sides and is tangent to the top of the part. The command is in the Draw group on the Tangent Arc drop list.

  Note

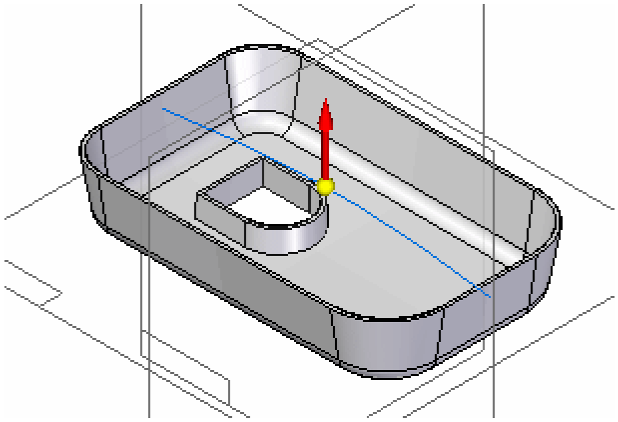
  The first and second points define the arc sweep. The third point defines the radius.

- Place and modify the dimension as shown. Add a Horizontal relationship to the two endpoints of the arc as shown.

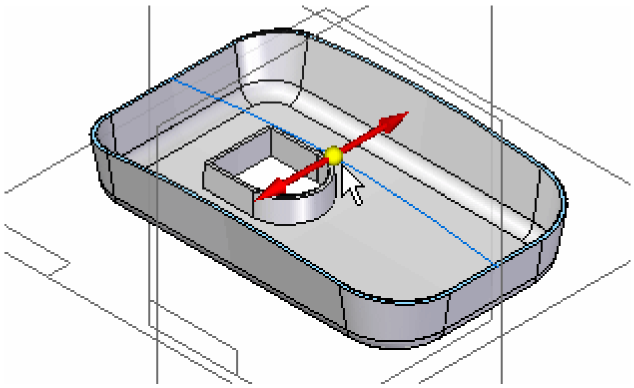


- Click Close Sketch.

- For the Side Step, position the cursor as shown in the illustration and click.

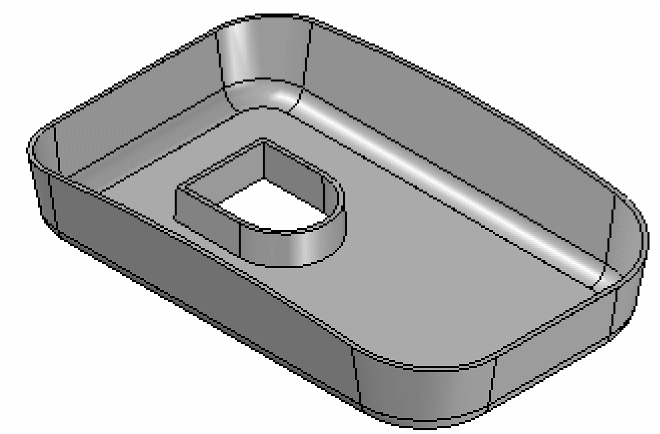


- On command bar, set the Extent to Through All. Position the cursor so that arrows point from both sides of the profile and click.



- Finish the cutout and save the file.

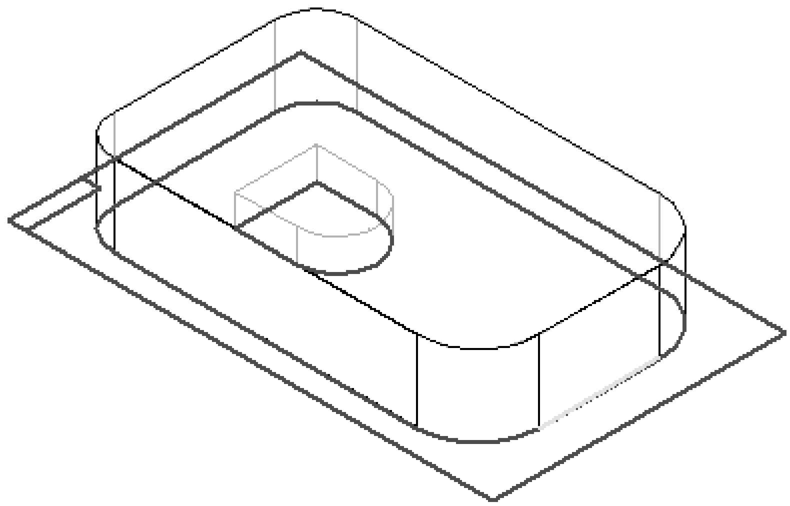
- Hide all reference planes.



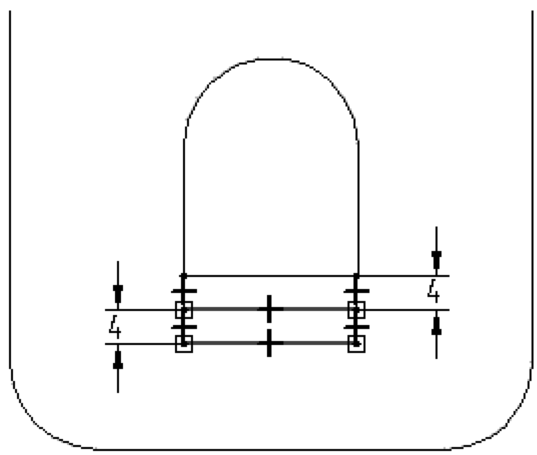
Add a cutout and use GoTo

Add another cutout. Since the part is thin walled, the additional cutout is not thin walled unless it is constructed before the thin wall step. The following steps demonstrate how to go back in the creation process to a point before the thin wall had been applied and place another cutout.

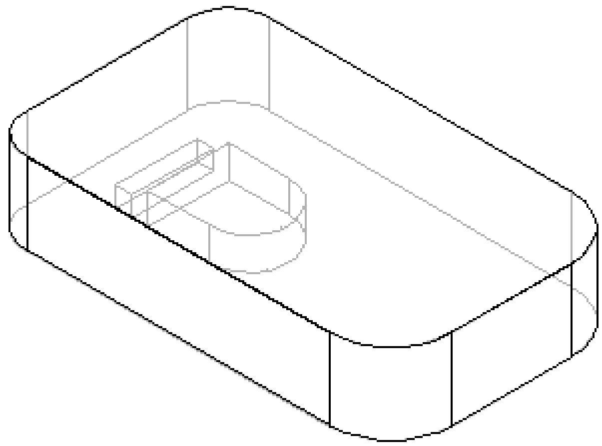
- Change display to Visible and Hidden Edges.
- Choose the Select Tool.
- In PathFinder, right-click on the feature named *Cutout 1*, and on the shortcut menu, select the GoTo command.
- Choose the Cut command and use QuickPick to select the reference plane shown.



- Draw the rectangular profile.



- Click Close Sketch and project the cutout upward 5 using the Finite Extent option.



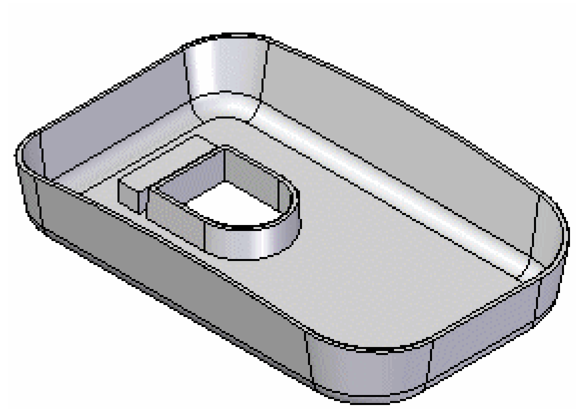
- Click Finish.

  Note

  Since this cutout was placed before the thin wall feature, use the GoTo command to apply the thin wall to the new cutout.

- Choose the Select Tool.

- Right-click on the last feature listed in Feature PathFinder, and select the GoTo option from the shortcut menu. The part returns to the thin wall state. The cutout just constructed has thin wall sides because it was placed before the thin wall feature.

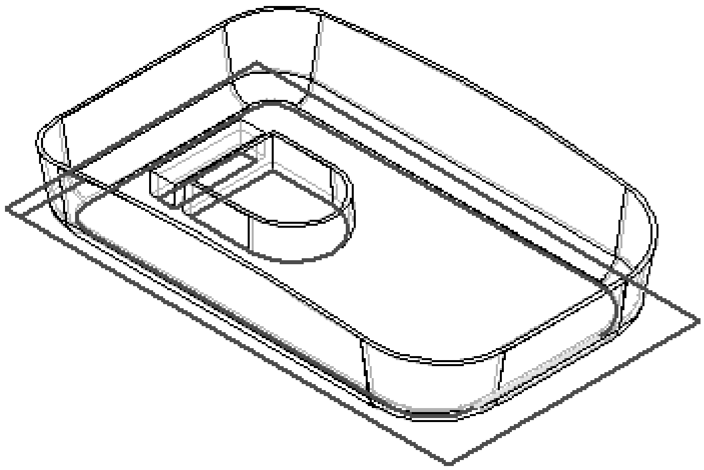


<u>Add mounting boss features</u>

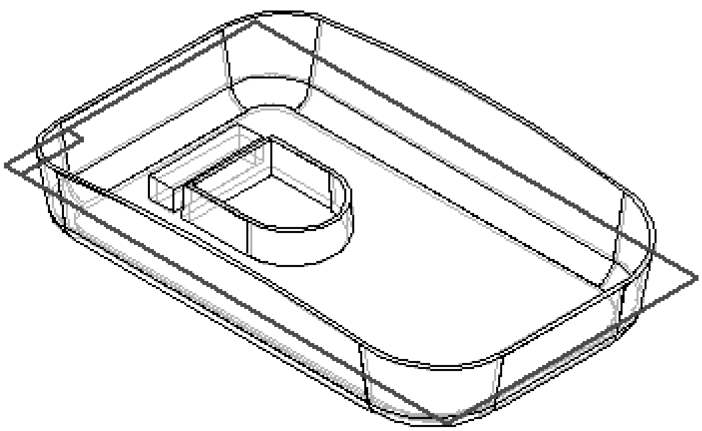
- In the Solids group, choose the Mounting Boss command  on the Thin Wall drop list.

- On the Mounting Boss command bar, click the Parallel Plane option.

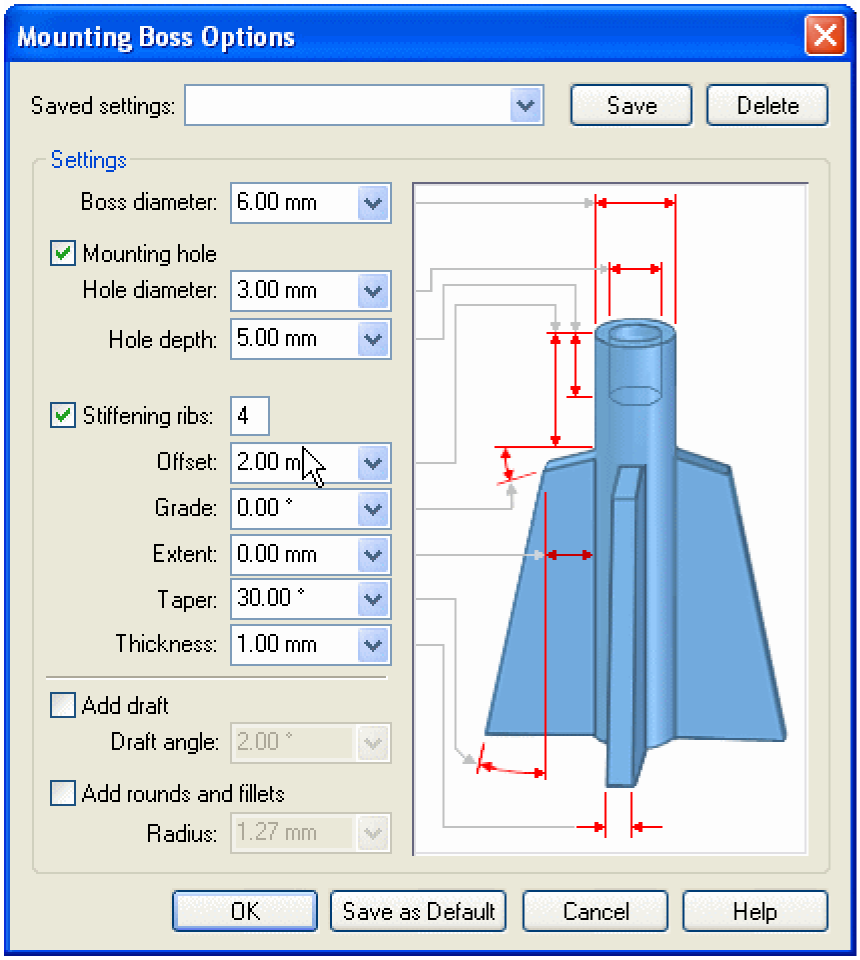
- Select the bottom plane as shown.



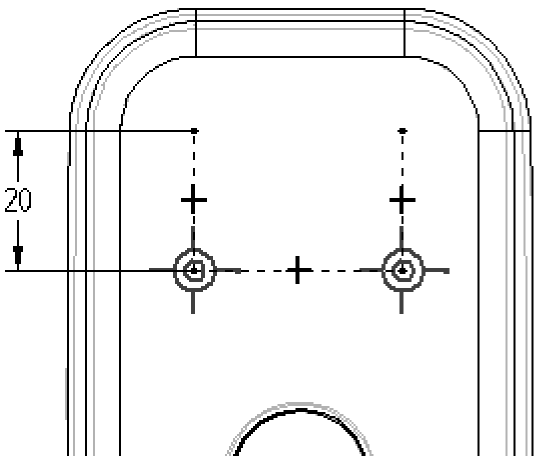
- On the command bar, type 10 in the Distance field. Position the parallel plane above the bottom plane as shown and click.



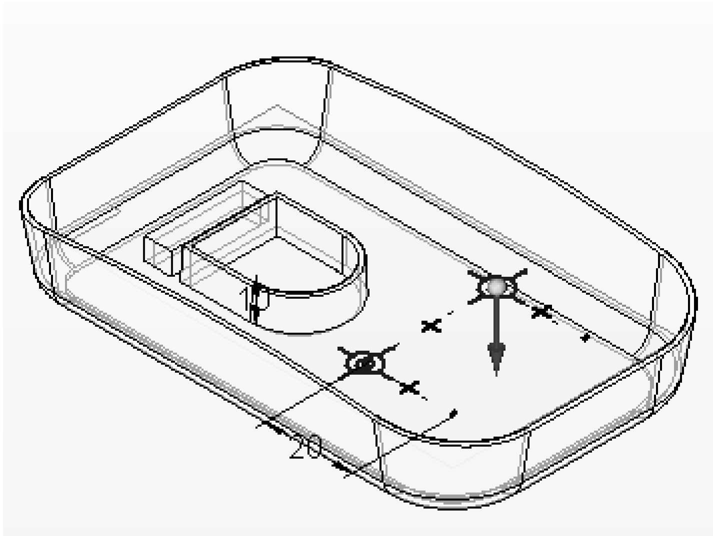
- On the command bar, click the Mounting Boss Options button and set the Mounting Boss Options as shown and click OK.



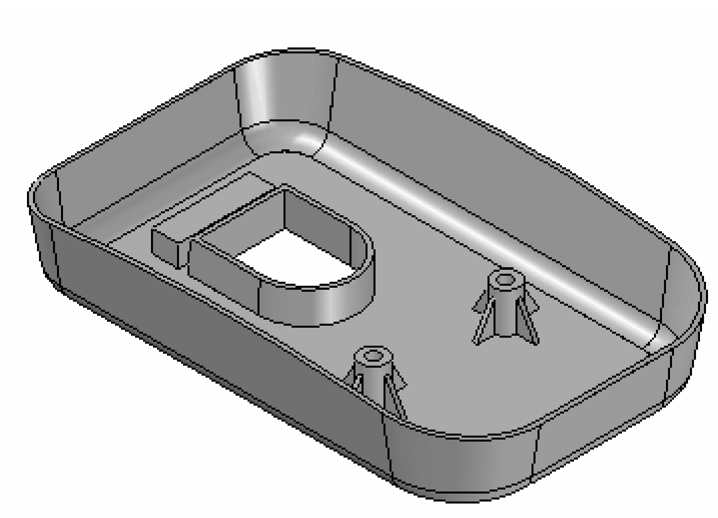
- Position the bosses as shown and then click Close Sketch.



- Define the extent direction as shown.



- Click Finish.



- Save the document and close the file. This completes the activity.

Summary

In this activity you learned how to add draft to some of the faces of a molded part. You learned how use the GoTo command to insert a feature at a desired location within Feature Pathfinder. You learned to place bosses using the Mounting Boss command.

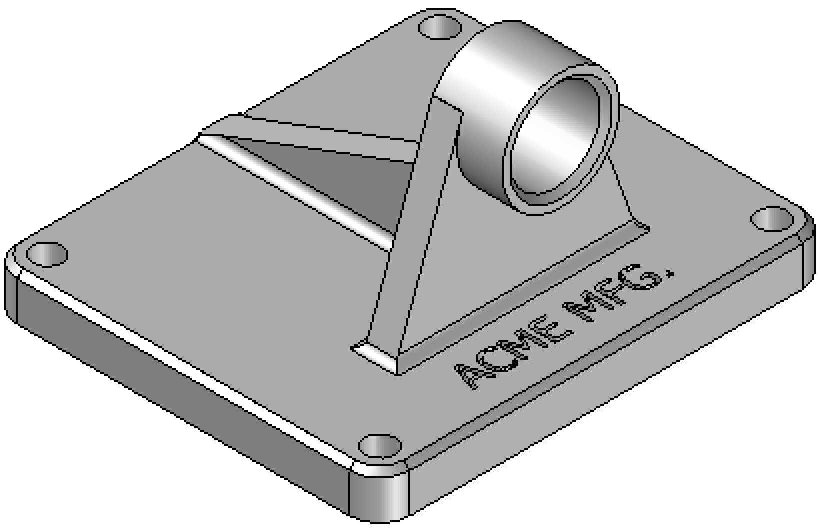
Embossing text on a part

This activity covers the procedure of embossing text characters onto a simple model of a casting.

Activity: Embossing text

Overview

This activity covers the procedure of embossing text characters onto a simple model of a casting.



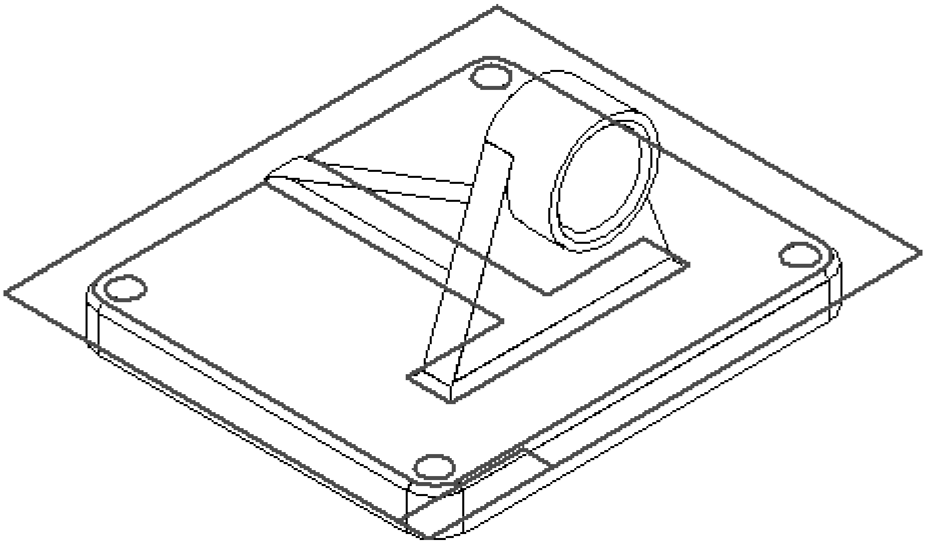
- Open *support.par*.

<u>Create a sketch containing the text profile</u>

To emboss text on a part, create a sketch containing the text profile.

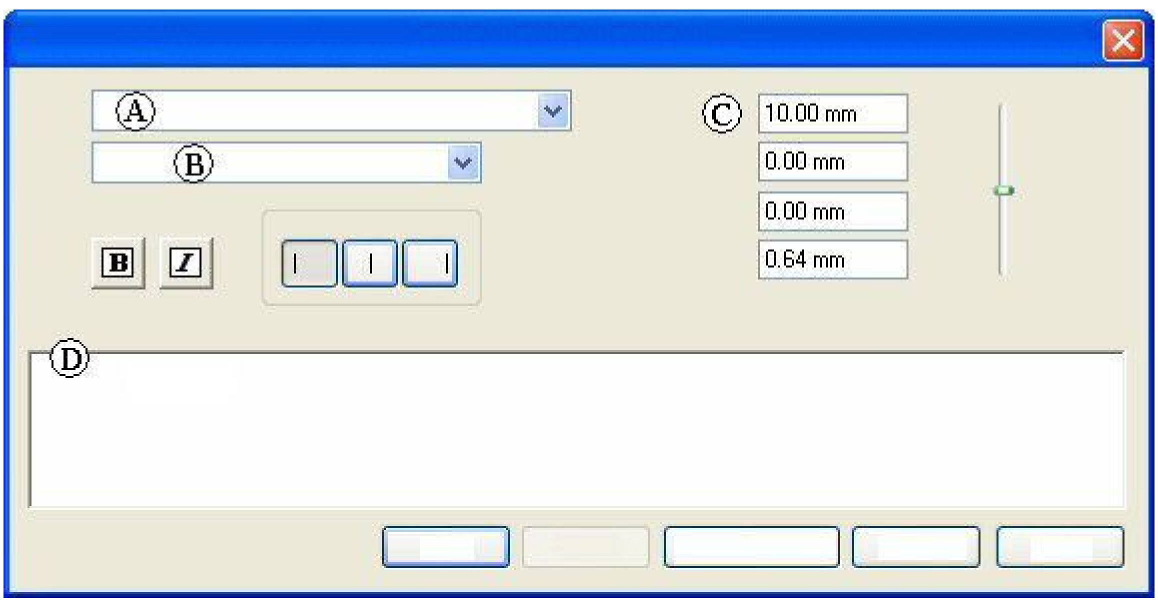
- Choose the Sketch command .

- Select the face shown for the sketch plane.

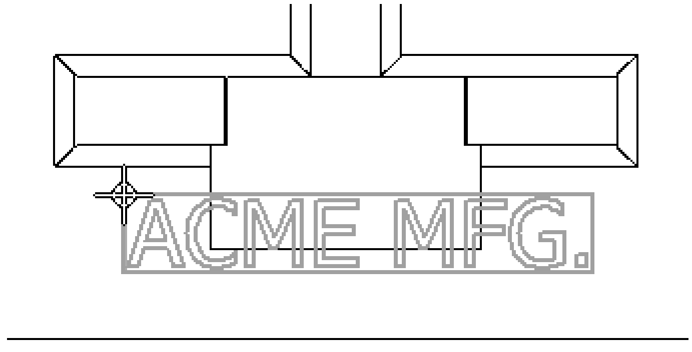


- On the Tools tab® Insert group, choose the Text Profile command .

- In the Text dialog box, set the values as shown. In the Font field (A), type Tahoma. In the Script field (B), type Arabic. In the Font size control fields (C), set the values shown. In the Text box (D), type *ACME MFG.* and click OK.



- Position the text in the approximate position shown, and click.

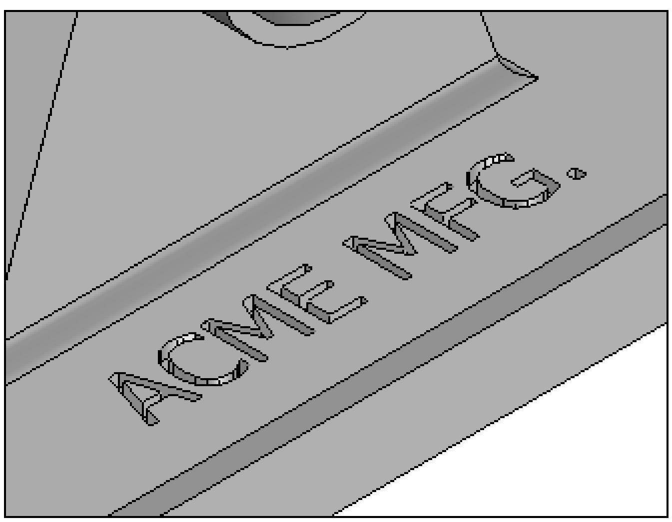


- Click Close Sketch to complete the profile.
- Click Finish.

<u>Cut the text profile from part</u>

Use the Cut command and the text sketch created in the previous step to remove material from the part.

- Choose the Cut command.
- On command bar, click the Select from Sketch option.
- Select the sketch (text) and click the Accept button.
- In the distance box, type 2 and press the Enter key.
- Click below the profile to extend the text into the part.
- Click Finish to complete the cutout.
- Hide all sketches.



- Save the file as *myblock.par*.

- Close the file. This completes the activity.

Summary

In this activity you learned how to create and add embossed text to a part.

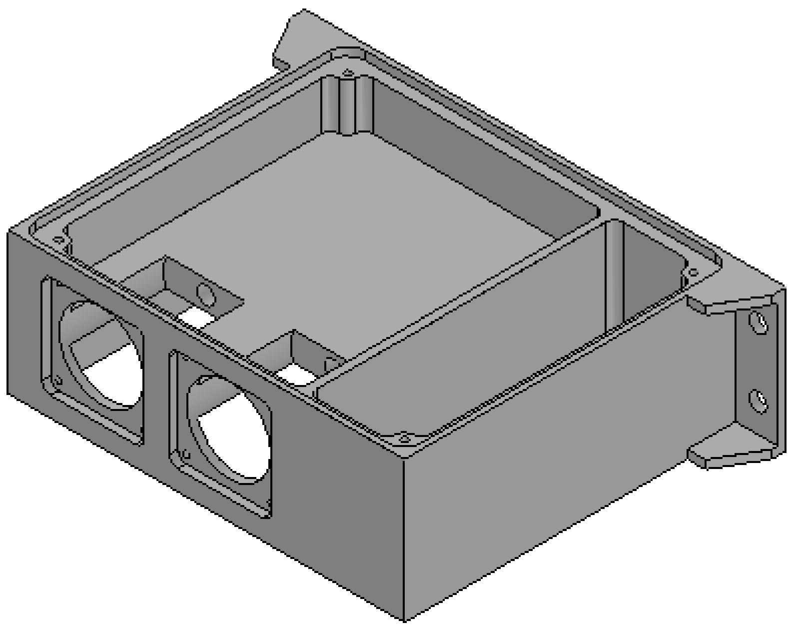
Modeling a machined part

This activity uses treatment feature commands, cutouts, rounds, patterns, mirror copied features, ribs, lip and hole. This activity is advanced and it might take a while to complete. There is a stopping point in the activity you can decide to continue or finish later. Pay careful attention to the instructions and illustrations.

Activity: Modeling a machined part

Overview

This activity uses ordered treatment feature commands, cutouts, rounds, patterns, mirror copied features, ribs, lip and hole. This activity is advanced and it might take a while to complete. There is a stopping point in the activity where the you can decide to continue or finish later. Pay careful attention to the instructions and illustrations.



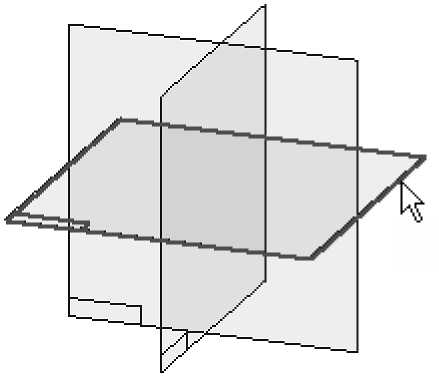
- Open a new ISO part file. Save the file as *machine01.par*.
- Make sure you are in the ordered environment.

<u>Create the base feature</u>

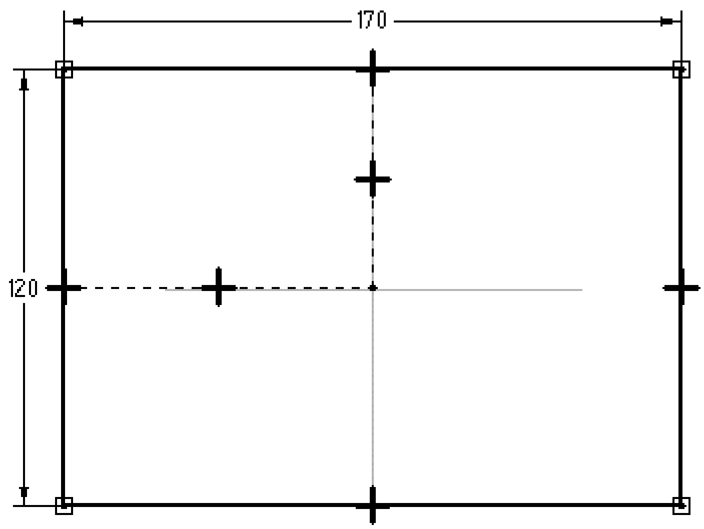
Begin the activity by creating a rectangular extrusion as the base feature for this part.

- In PathFinder, turn off the display of the base coordinate system. Turn on the display of the base reference planes.
- Choose the Extrude command.

- For the plane step, select the reference plane shown.

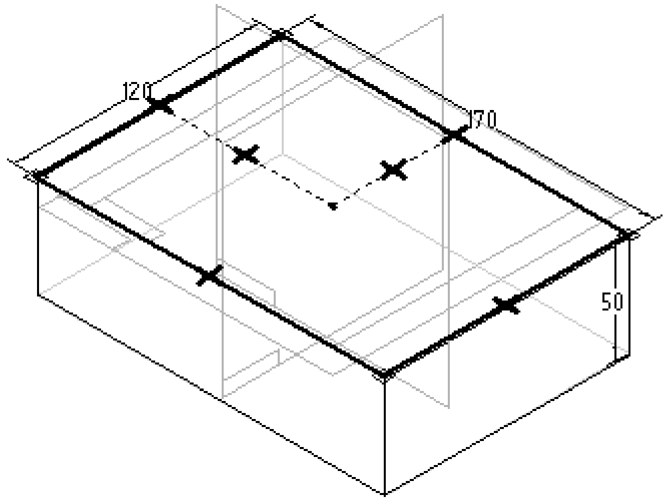


- Draw the profile and center the profile at the intersection of the default reference planes.



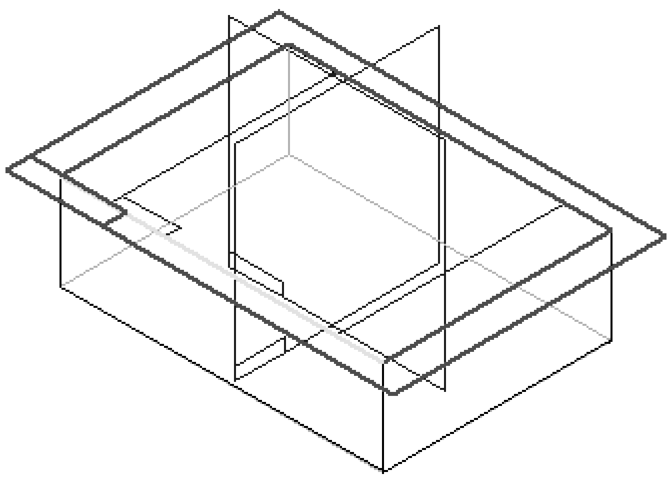
- Choose Close Sketch.

- Extrude the profile 50 mm below the reference plane and click Finish

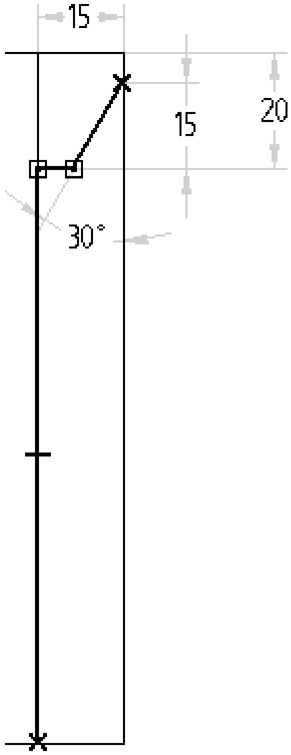


Add a cutout to the base feature

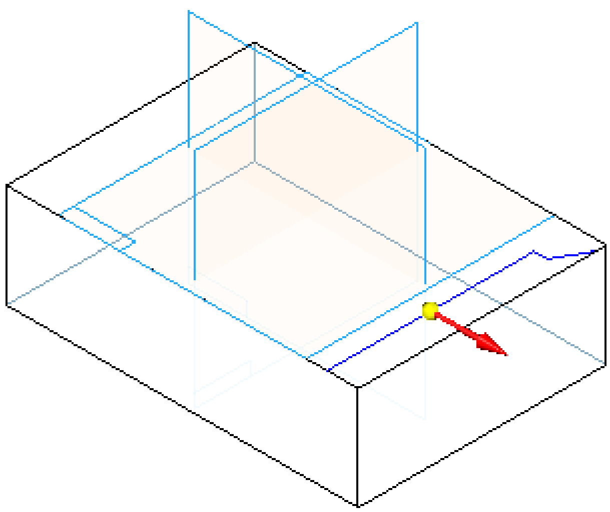
- Choose the Cut command.
- Select the Coincident Plane option and orient the plane as shown.



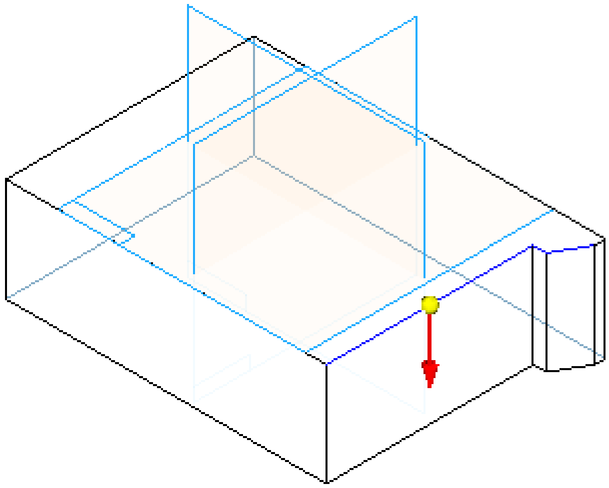
- On the right side of the part, draw the profile.



- Choose Close Sketch.
- Click as shown for direction to remove material.



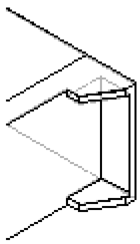
- For the Extent step, on the command bar, select the Through All option and click the direction as shown.



- Click Finish.

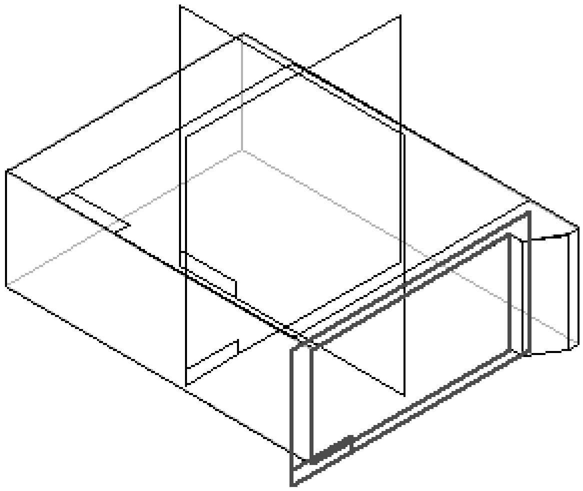
Create a cutout

Create a second cutout on a side face created by the cutout in the previous step. The cutout looks like the one shown.

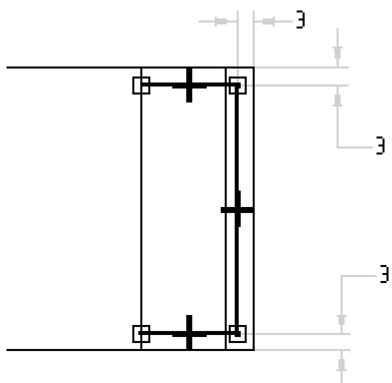


- Choose the Cut command.

- For the profile plane, select the right surface shown using the Coincident Plane option on the command bar.

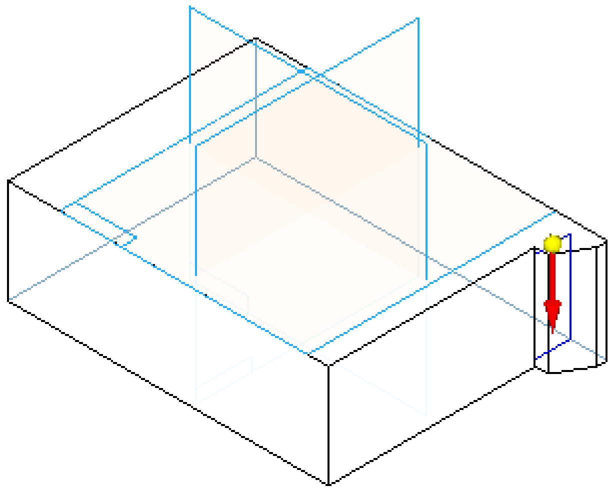


- Draw the open profile.

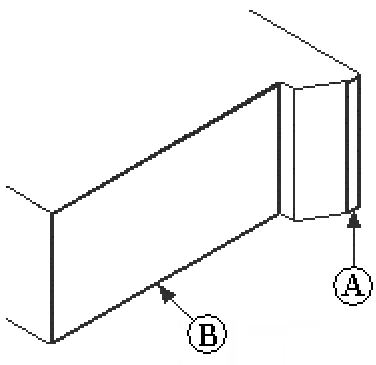


- Choose Close Sketch.

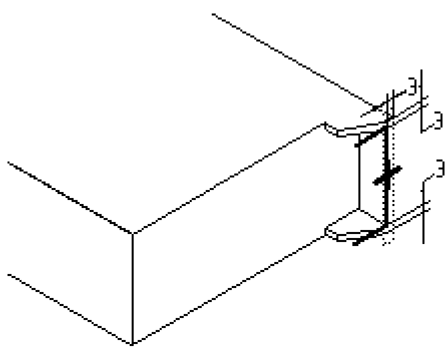
- For the Side step, position the cursor so the arrow points to the inside of the profile, as shown, and click.



- For the Extent Step, on the command bar, click the From/To Extent button. Make the depth of the cutout from surface (A) to surface (B).

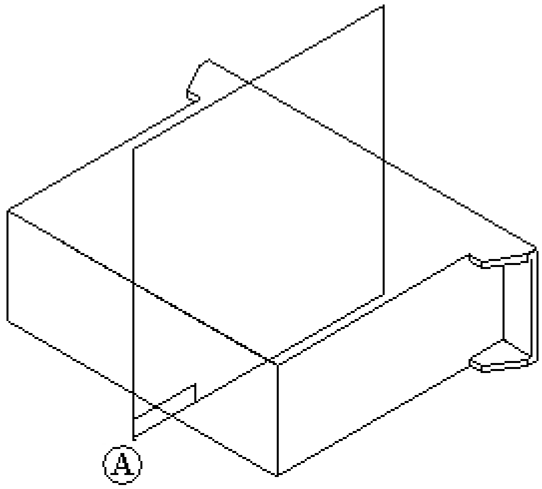


- Click Finish.

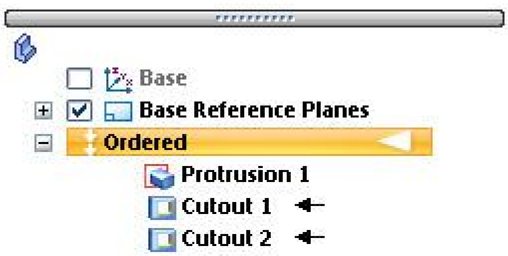


Mirror cutouts

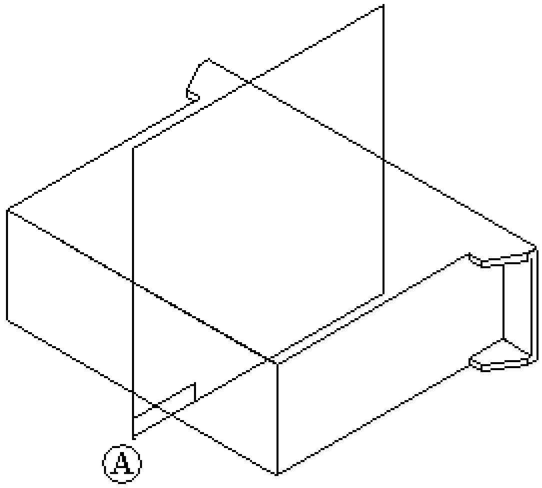
Mirror the cutouts created in the previous two steps about reference plane (A). Using this reference plane, which lies at the center of the part, ensures that the two cutouts are mirrored symmetrically on the opposite side of the part.



- In the Pattern group, on the Mirror drop list, choose the Mirror Copy Feature command .
- On command bar, click the Smart button.
- Select the two cutout features in PathFinder and click the Accept button.



- For the plane to mirror about, select reference plane (A).

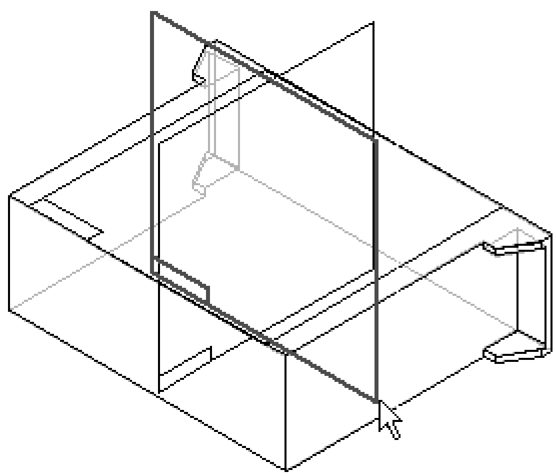


- Click Finish.

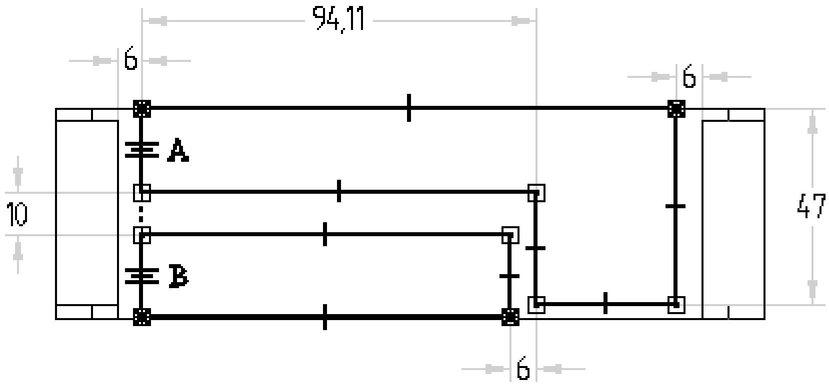
Create a cutout

Create a cutout using two profiles created in a single profile step. This allows removing or adding material of a complex shape in a single step.

- Choose the Cut command.
- Select the reference plane shown.

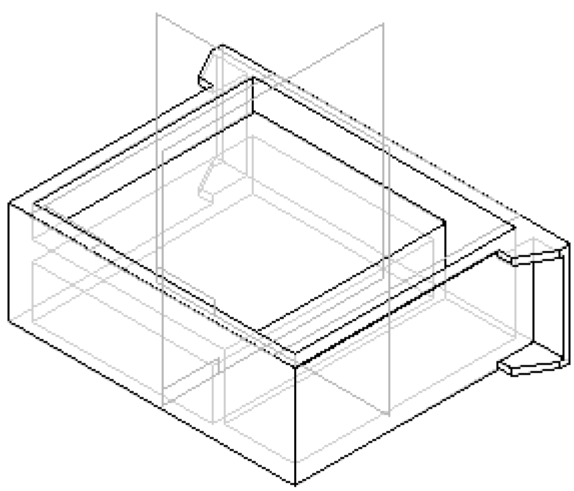


- Draw and dimension the two profiles as shown. The top and bottom lines are coincident with the part edges. Notice that lines A and B have equal relationships applied.



- Click Close Sketch.

- For the extent step, use the Through All extent and click the Symmetric Extent button. Type 108 in the Distance field and press Enter.



- Click Finish.

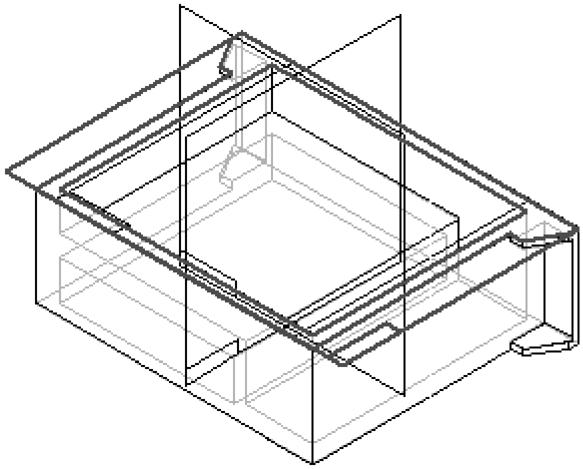
Construct a rib

Construct a rib to strengthen the interior of the part.

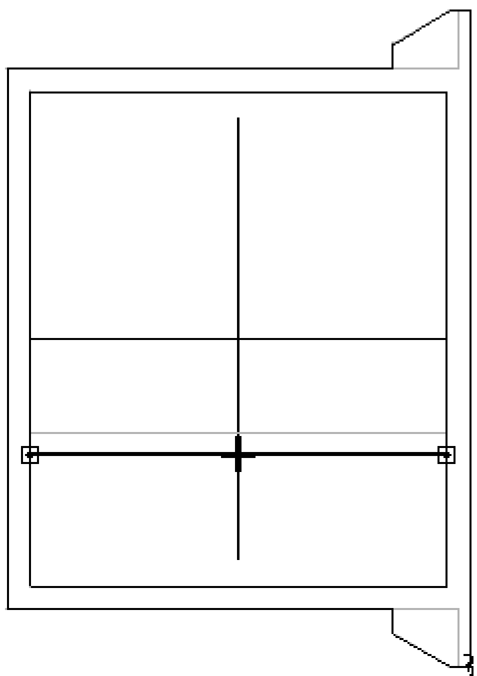
- In the Solids group, on the Thin Wall drop list, choose the Rib command .

- On command bar, click the Parallel Plane option.

- Select the top face as shown.

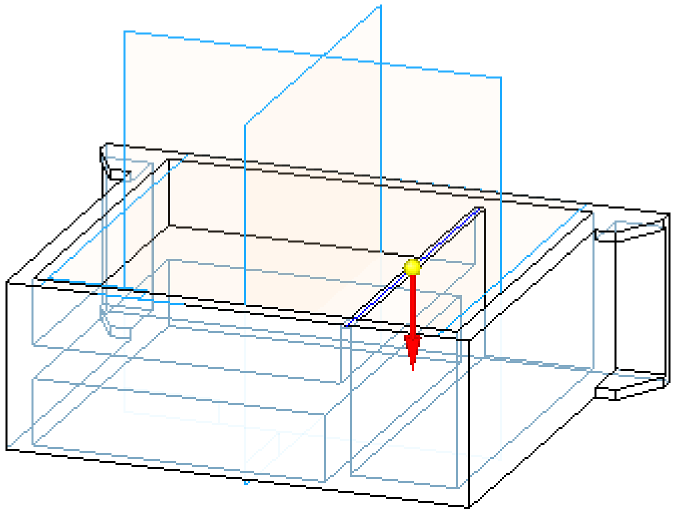


- On the command bar, type 3 and position the cursor so the parallel plane is placed below the top face and click.
- Draw the rib profile. Looking down from the top of the model, the profile endpoints are connected to the cutout edges.

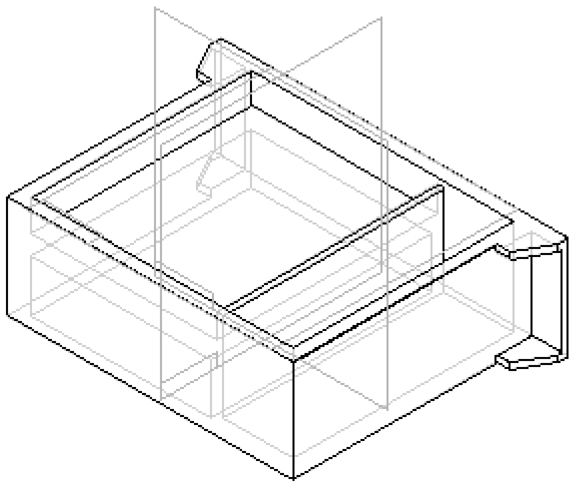


- Choose Close Sketch.
- On the command bar, type 3 for rib thickness.

- Select the direction shown.



- Click Finish.

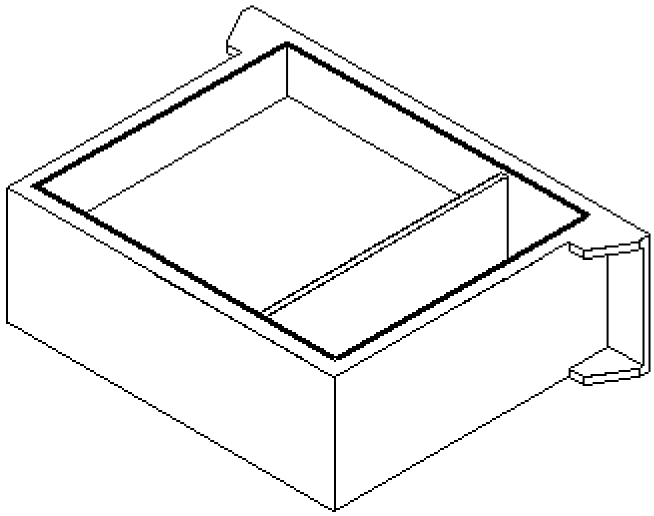


Create a groove

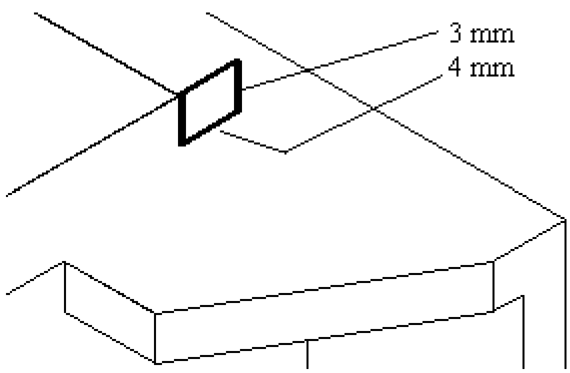
Create a groove around the top inside edge of the part. Use the Lip command. Use this command to add material to create lips or remove material to create grooves.

- On the Thin Wall drop list, choose the Lip command .

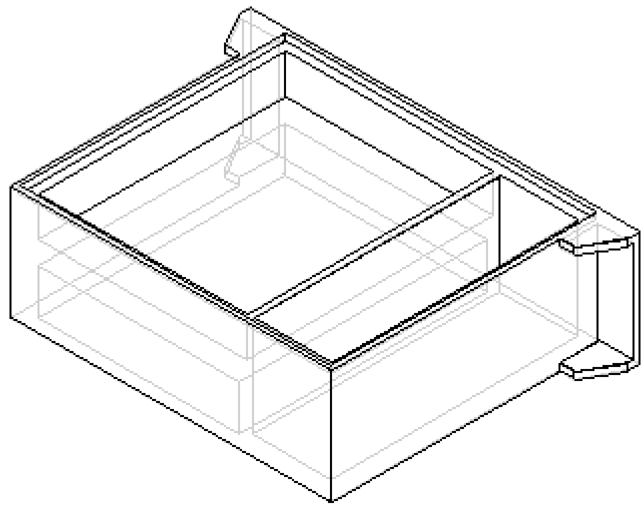
- Select the four edges shown and then click the Accept button.



- On the command bar, type 4 for the width and 3 for the height. Use the Zoom command to adequately see this rectangle. This rectangle defines whether material will be added to create a lip or removed to create a groove. Position the rectangle as shown to create the groove.



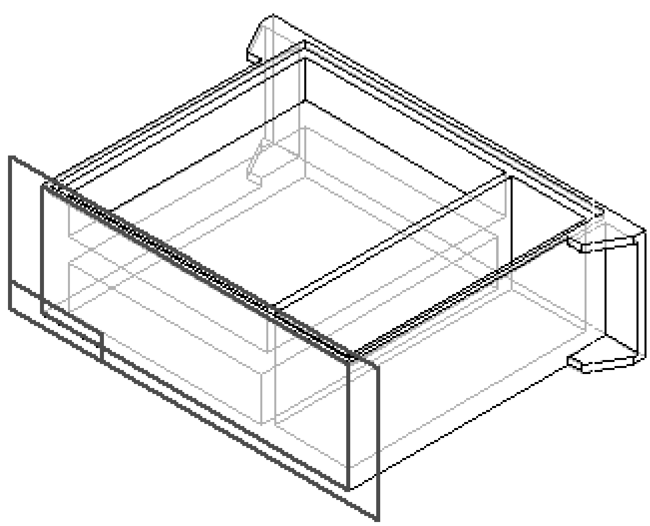
- Click Finish.



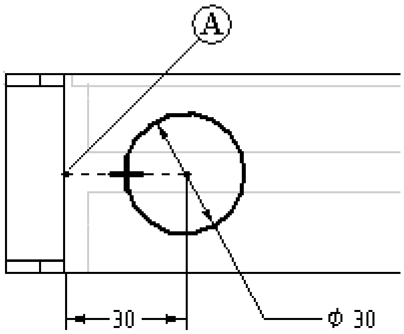
Create circular cutout

Create a circular-shaped cutout and remove a finite amount of material from the part. The Hole command could be used here, however in this step the Cut command and a circular profile is used.

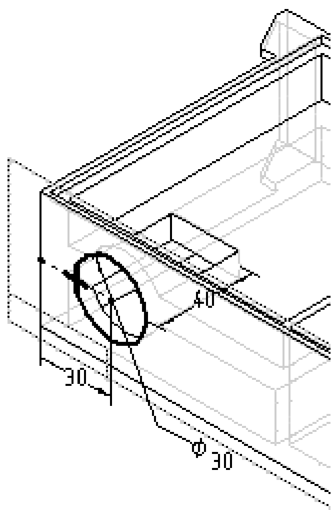
- Choose the Cut command.
- Select the profile plane as shown.



- Draw and dimension the profile. Center the circle on midpoint of line (A).



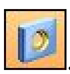
- Choose Close Sketch.
- In the Distance box, type 40 for the extent and position the cutout into the part.



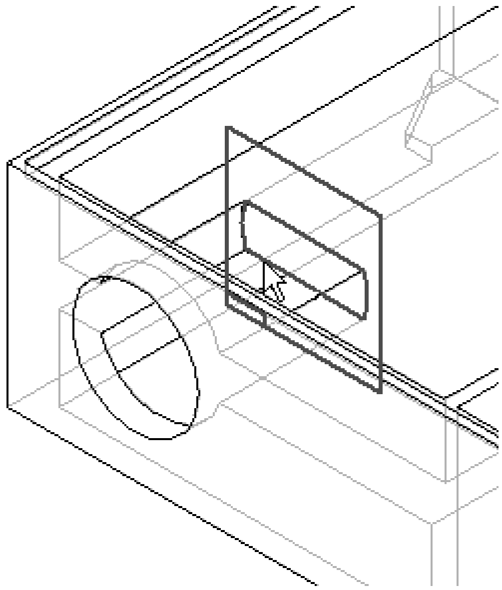
- Click Finish.

<u>Construct a hole</u>

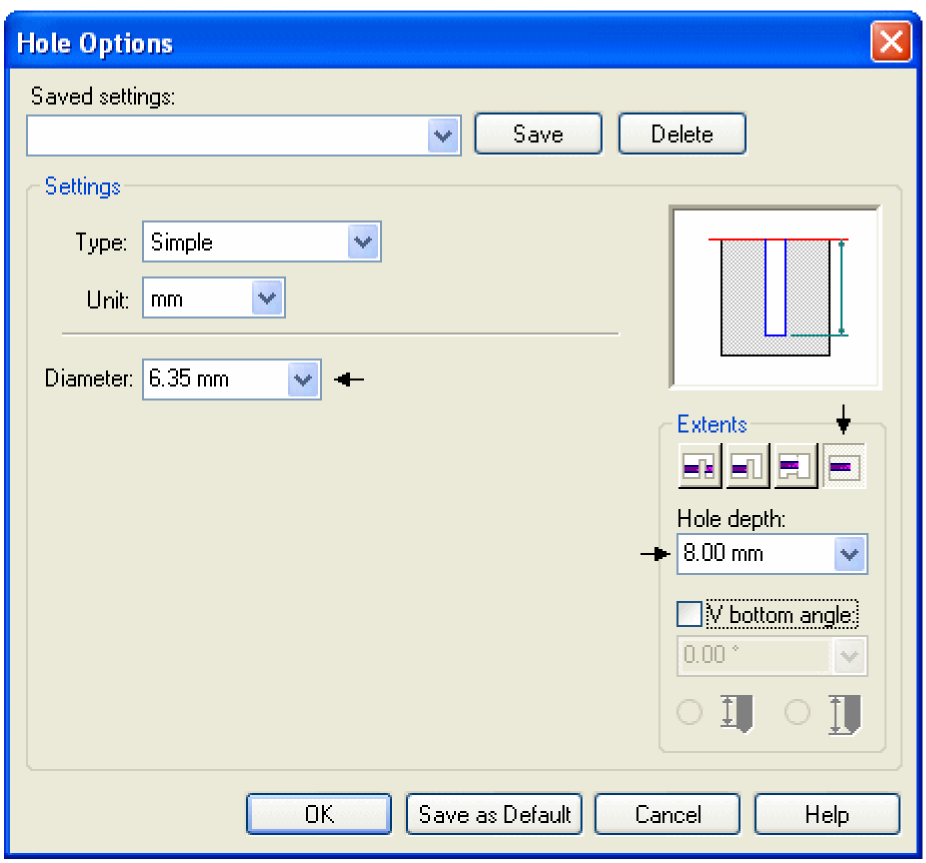
Construct a hole at the rear of the cutout created in the previous step.

- Choose the Hole command .

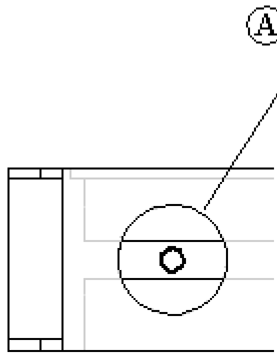
- Select the profile plane as shown.



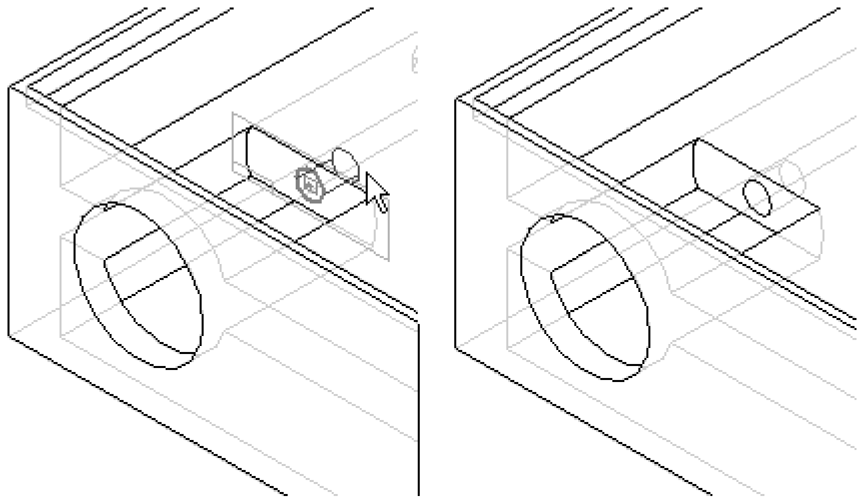
- On the command bar, click the Hole Options button . Type 6.35 for the Diameter, select the Finite extent and Hole Depth of 8. Click OK.



- Place the hole centered on circle (A).



- Choose Close Sketch.
- Position the extent to the right as shown and click.



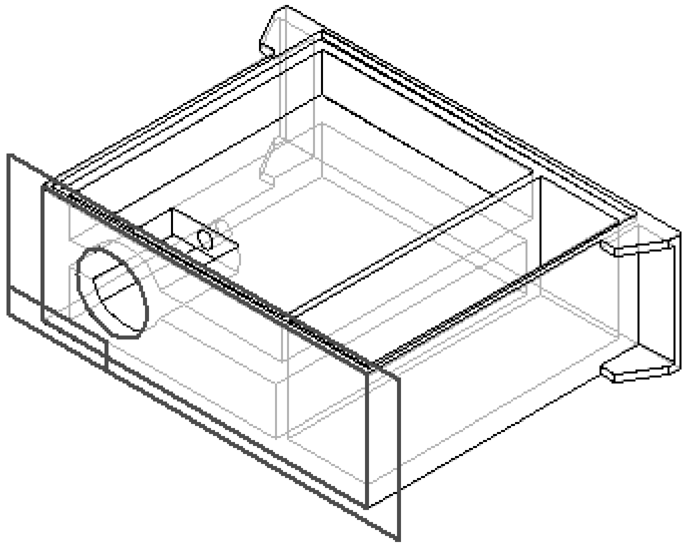
- Click Finish.

Create a cutout

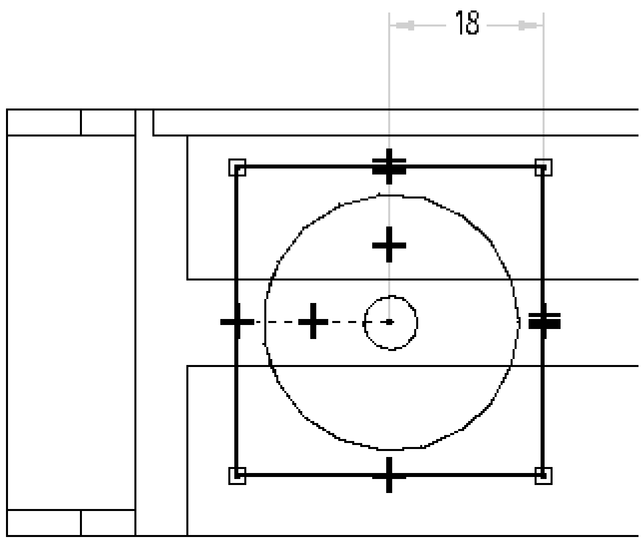
Create another cutout on the part. This cutout will surround the circular cutout created earlier.

- Choose the Cut command.

- Select the profile plane as shown.

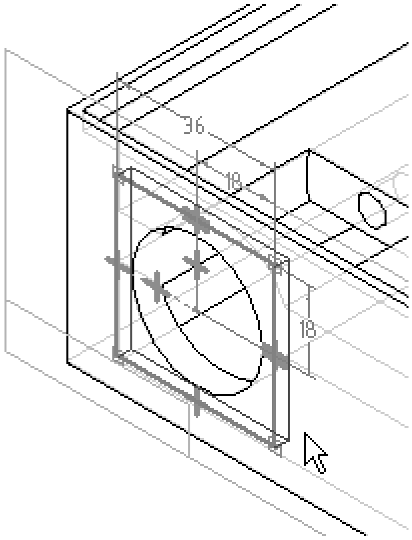


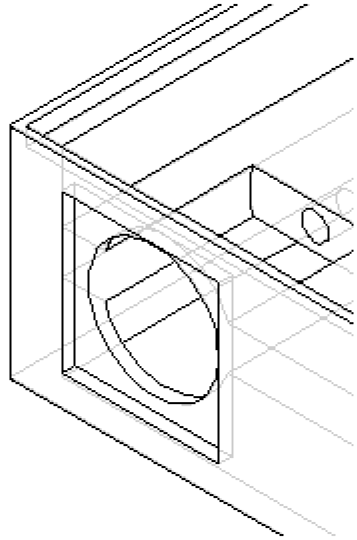
- Draw and dimension the profile. Use Horizontal/Vertical and Equal relationships to center the square profile around the circular cutout from the previous step.



- Choose Close Sketch.
- Click the Finite Extent button, and type 3 in the Distance field.

- Position the cursor so that material is removed from the part and click.



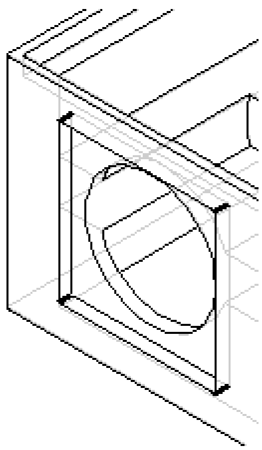


- Click Finish.

Add rounds

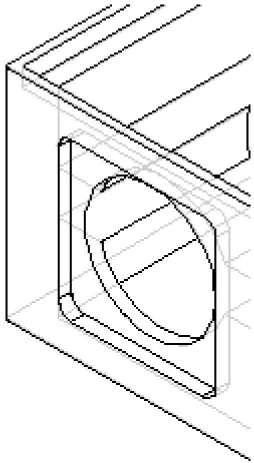
Add rounds to the cutout.

- Choose the Round command.
- Select the four edges as shown.



- Type 3 in the Radius field, and then click the Accept button.

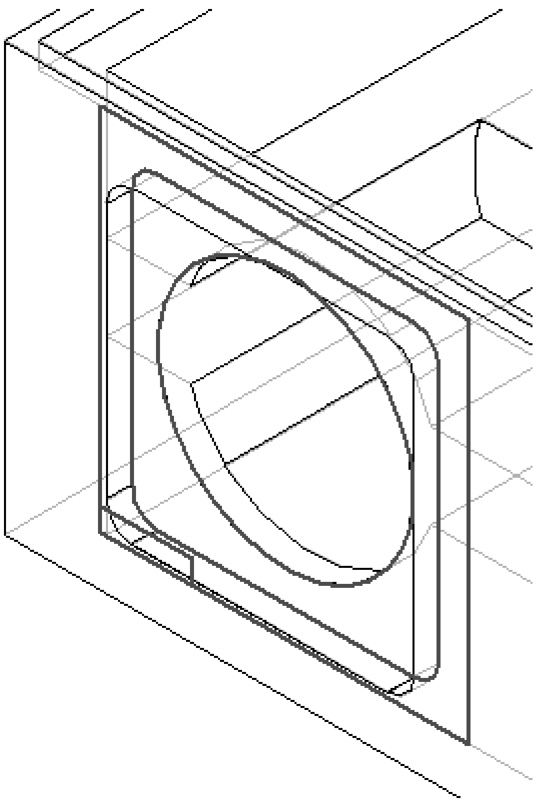
- Click Preview and Finish.



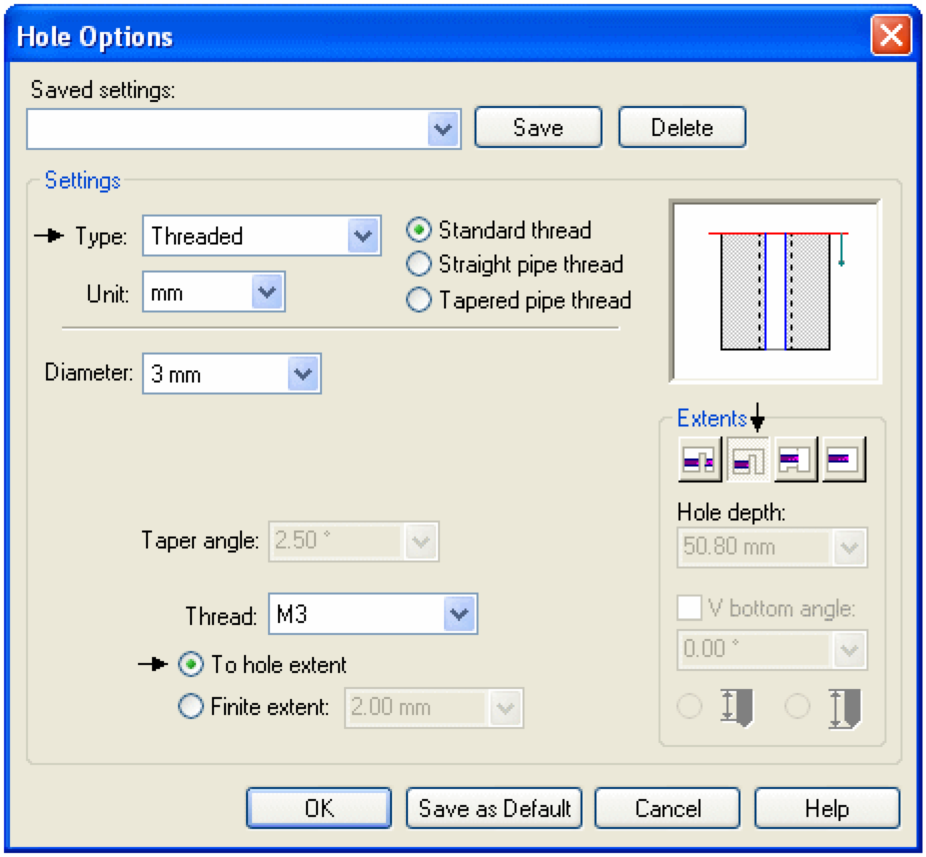
Add holes

Add a series of holes to the surface created by the rectangular cutout.

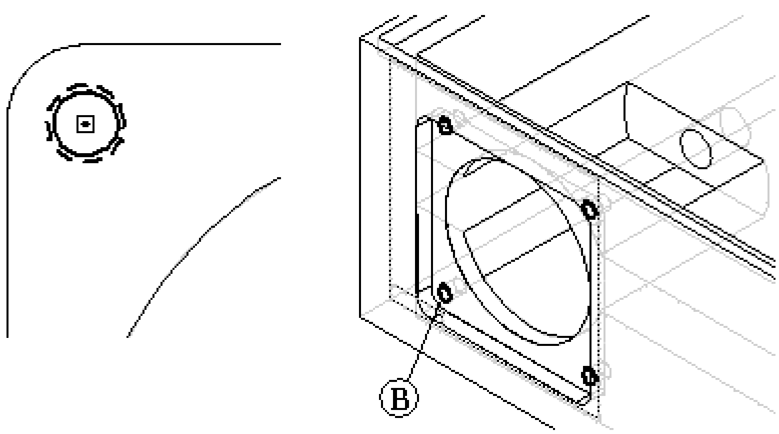
- Choose the Hole command.
- Select the profile plane as shown.



- Click the Hole Options button and set the options as shown. Click OK.

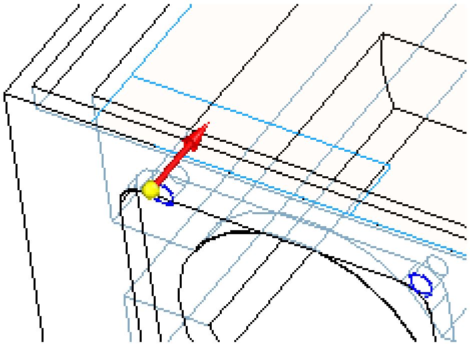


- Place four holes as shown (B). Center the holes on the rounds you created in the previous step. The dashed line around the hole profile indicates a threaded hole.

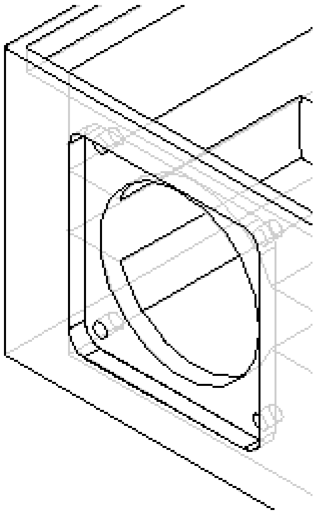


- Choose Close Sketch.

- Position the direction arrow to point towards the interior of the part and click.



- Click Finish.

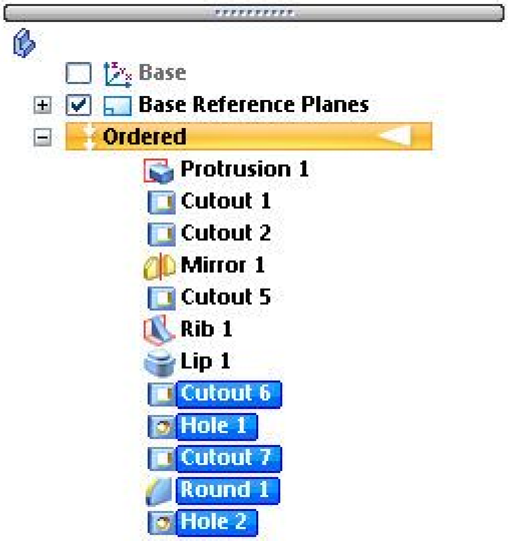


Create a pattern of features

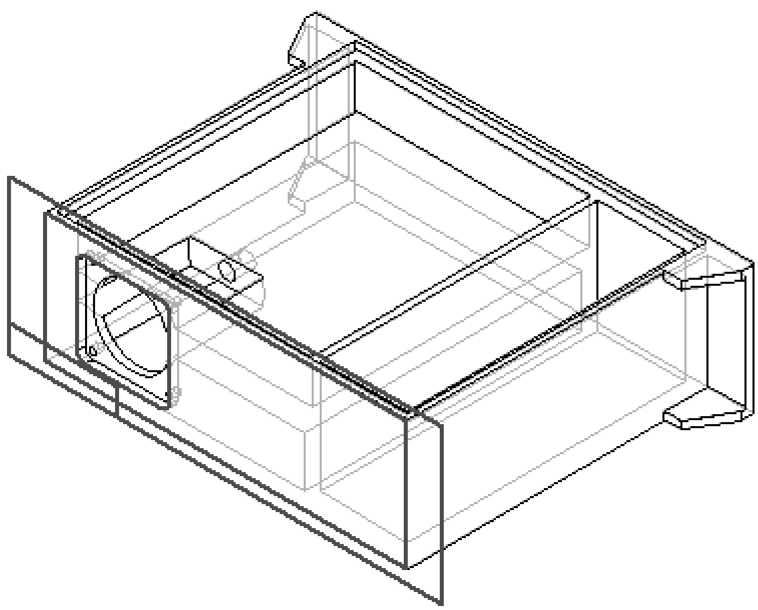
Pattern the five features, which include the circular cutout, single hole, square cutout, rounds, and series of four holes.

- Choose the Pattern command and on the command bar, click the Smart option.

- Select the features shown below to pattern.

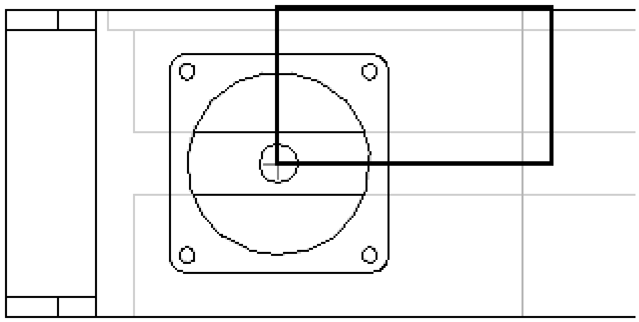


- Click the Accept button.
- Select the pattern reference plane as shown.

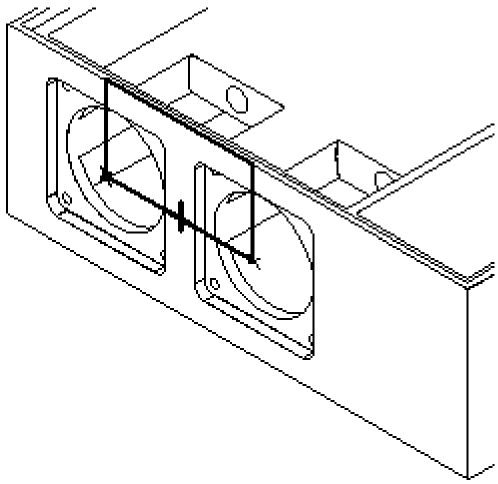


- On the command bar, type the following patterning parameter values.

- Define the pattern profile by selecting the first point in the center of the small hole and then position the rectangle as shown.



- Choose Close Sketch.



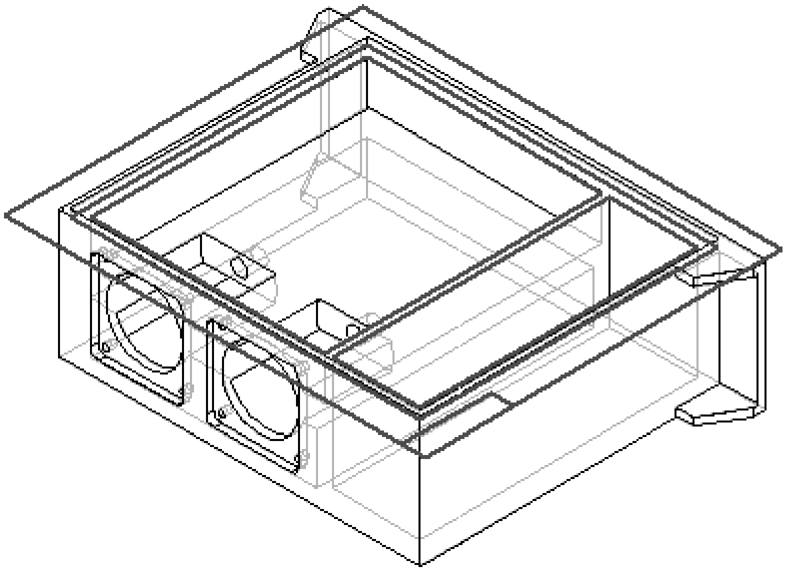
- Click Finish.

Create an extrusion

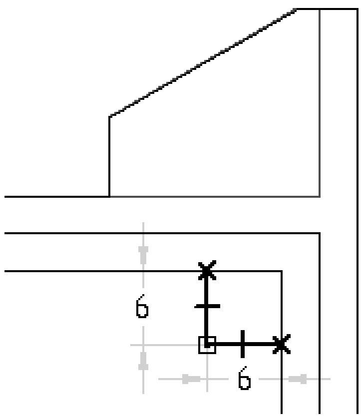
Use the Extrusion command to add material in the corner of the part. It serves as a boss for the model.

- Choose the Extrude command.

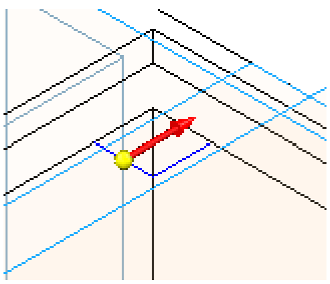
- Select the profile plane as shown.



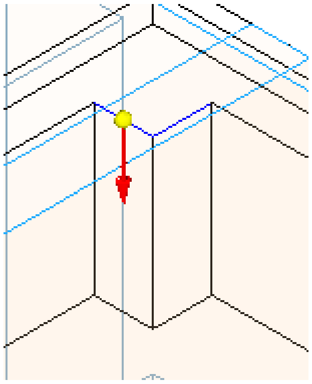
- Draw and dimension the profile.



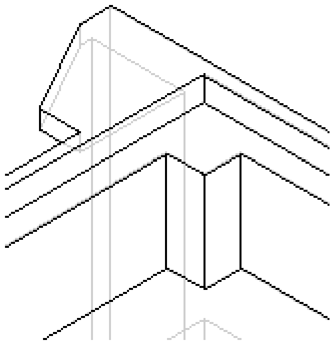
- Choose Close Sketch.
- Position the direction arrow as shown and click.



- For the extent step, on the command bar, click the Through Next button. Position the cursor so that the material is added below the profile as shown and click.



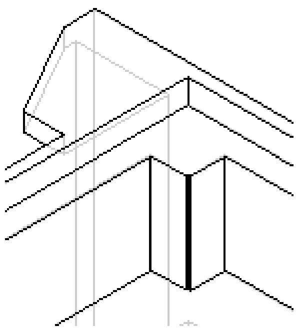
- Click Finish.



Apply a round

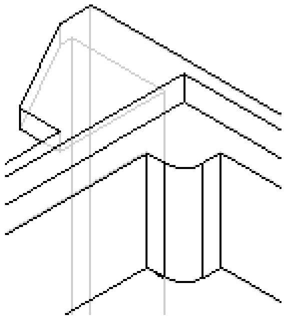
Apply a round to the material added in the previous step.

- Choose the Round command.
- Select the edge shown.



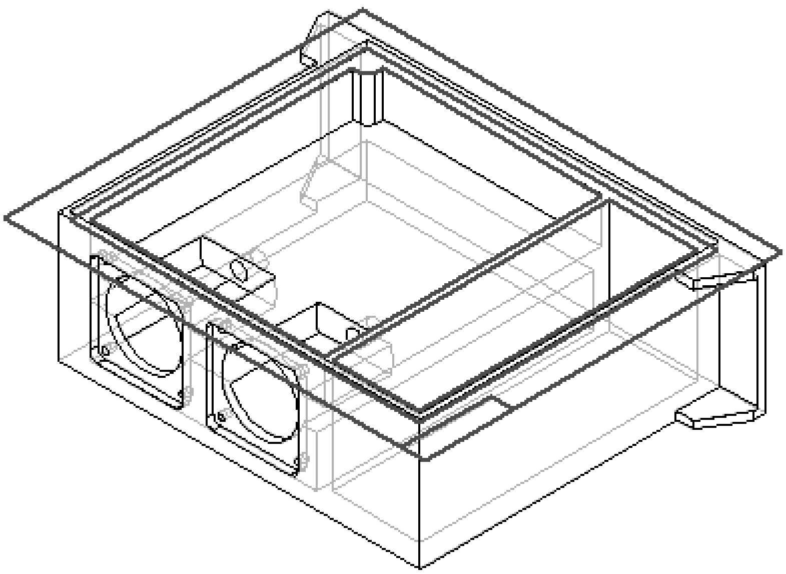
- Type 3 in the Radius field, and then click the Accept button.

- Click Preview and Finish.

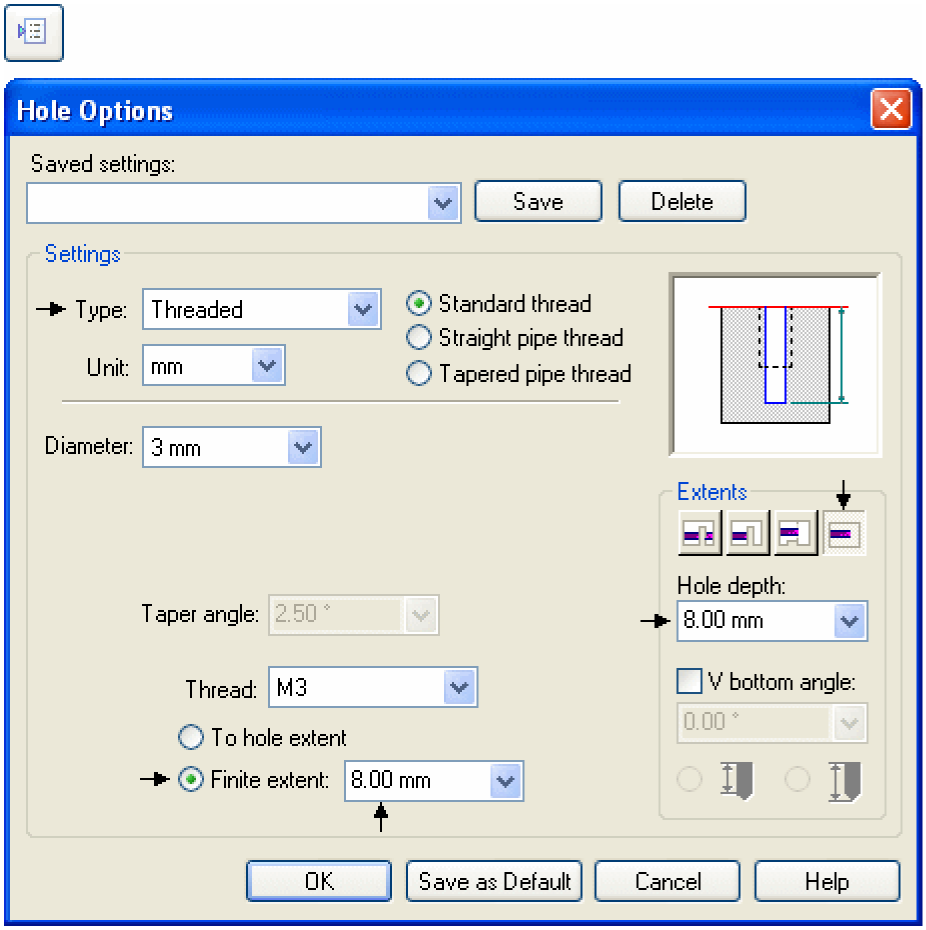


Add a threaded hole

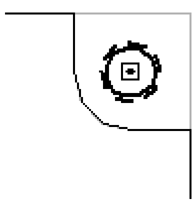
- Choose the Hole command.
- Select the profile plane as shown.



- Click the Hole Options button and set the options as shown. Click OK.

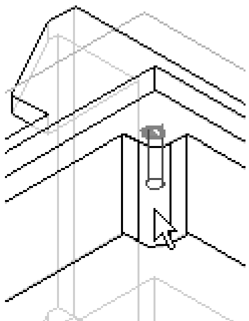


- Place the hole concentric with the arc.

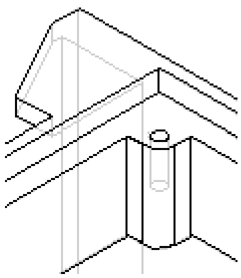


- Choose Close Sketch.

- Position the cursor so that the extent is defined as shown and click.



- Click Finish.

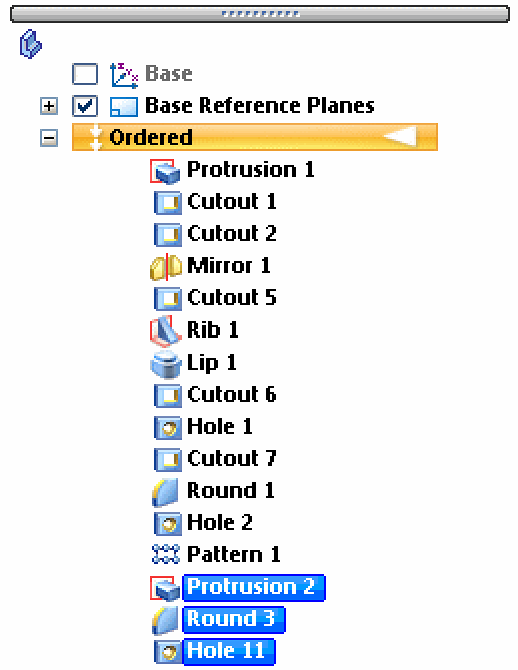


<u>Mirror features</u>

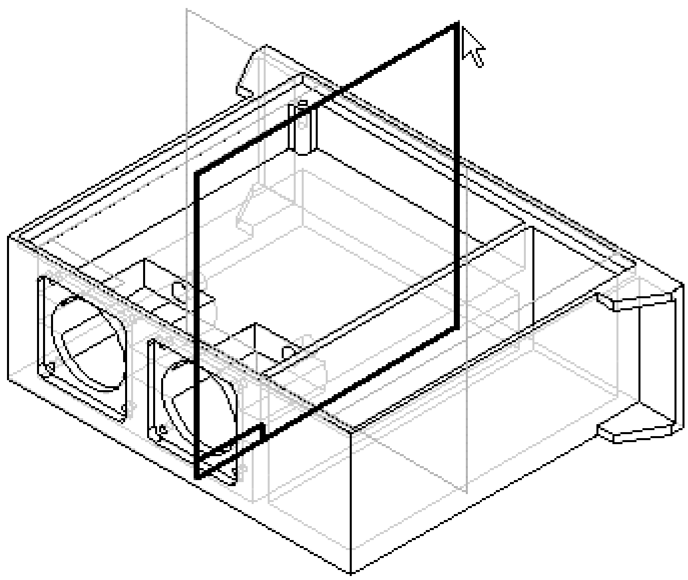
Mirror the features created in the previous steps. These include the rectangular boss, round, and hole.

- Choose the Mirror Copy Feature command.
- Click the Smart button.

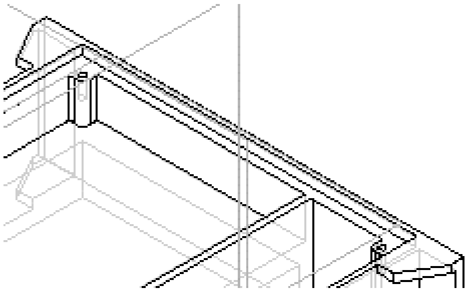
- In PathFinder, select the last three features constructed, protrusion, round and hole. Click the Accept button.



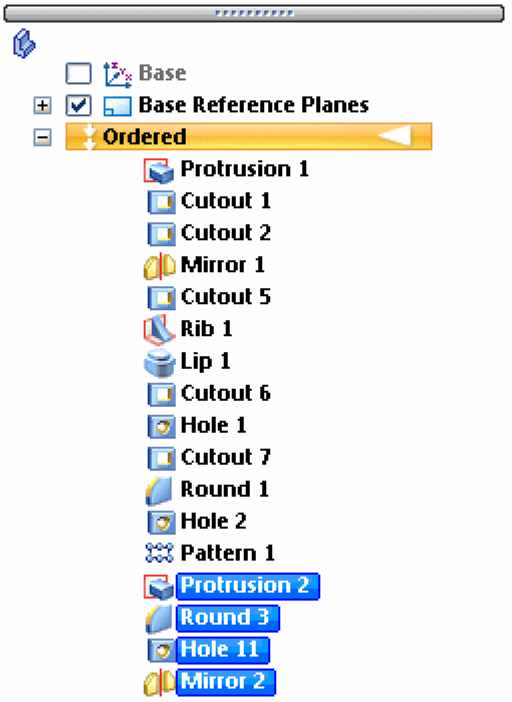
- Select the reference plane shown as the plane to mirror the features about.



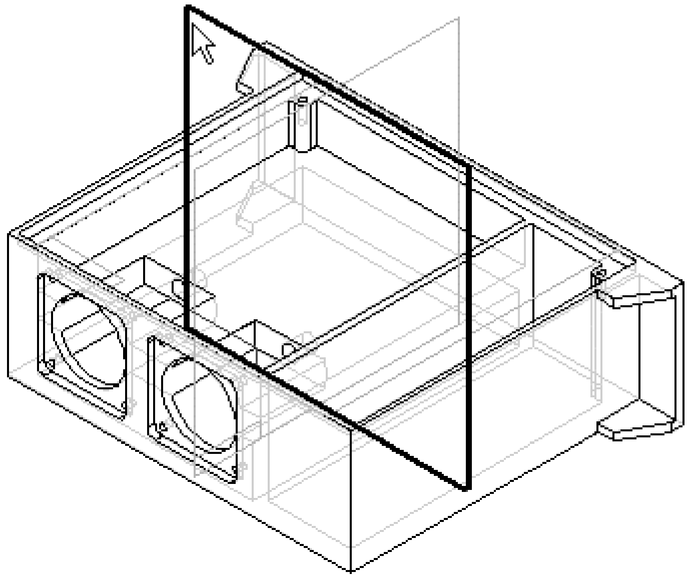
- Click Finish.



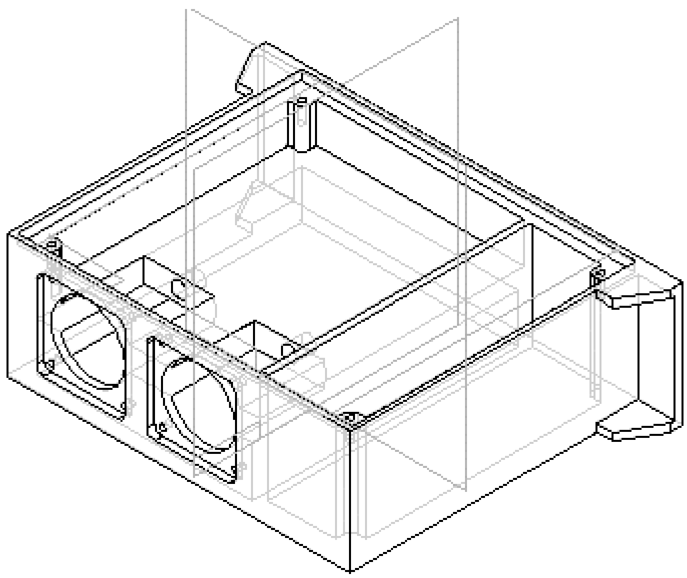
- Choose the Mirror Copy Feature command.
- Click the Smart button.
- In the PathFinder, select the protrusion, round, hole and mirror features. Click the Accept button.



- Select the reference plane shown as the plane to mirror the features about.



- Click Finish.



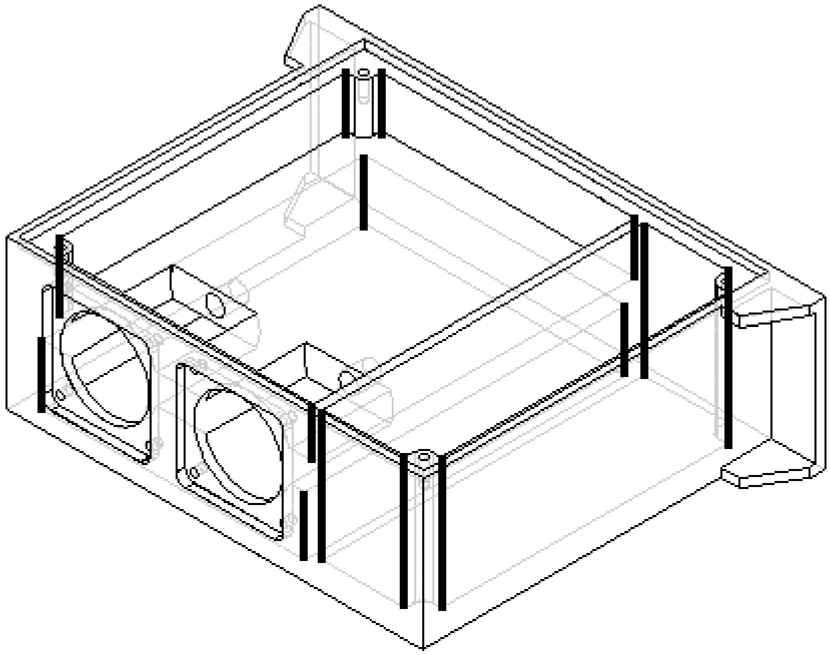
Note

In order to save time, you may stop at this point. The remainder of the activity covers adding more rounds and holes. Save the file at this point and finish later.

Add rounds to the inside edges

- Choose the Round command.

- Select the edges shown.



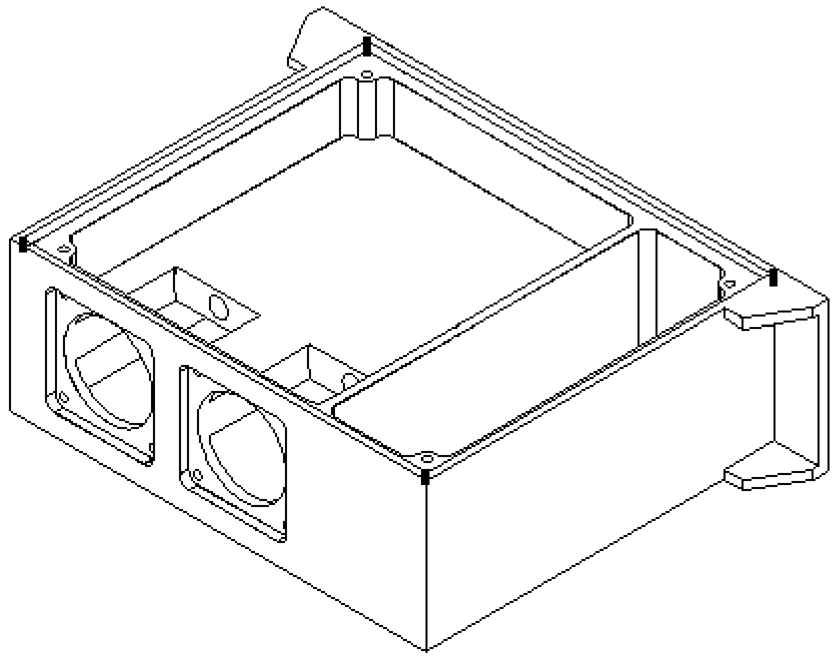
- Type 3 in the Radius field. Click the Accept button.
- Click Preview and Finish.

Add rounds

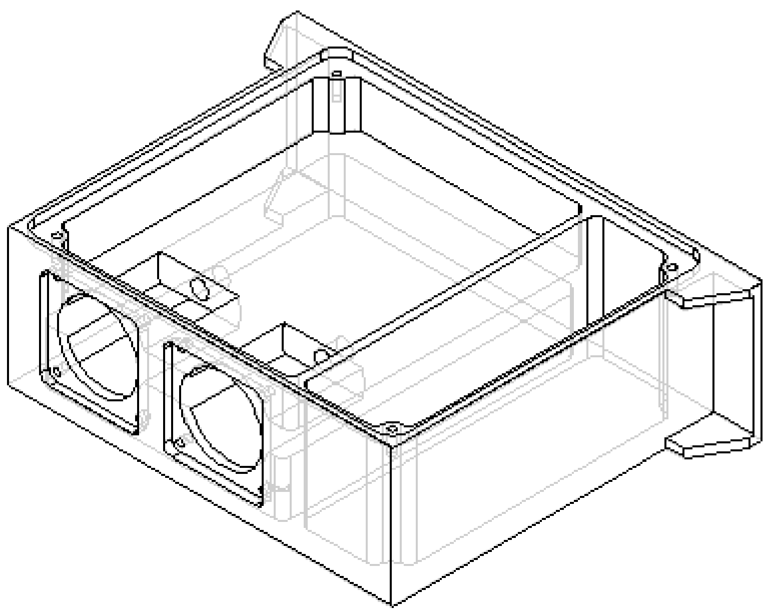
Add rounds to more of the interior edges of the part.

- Choose the Round command.

- Select the edges shown.



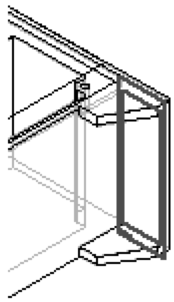
- Type 6 in the Radius field. Click the Accept button.
- Click Preview and Finish.



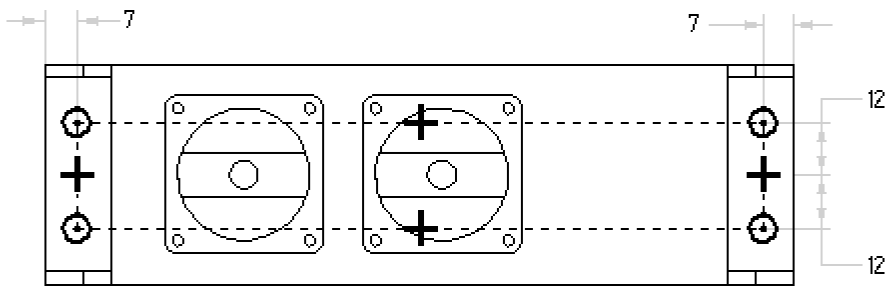
Add holes to the part

- Choose the Hole command.

- Select the profile plane as shown.

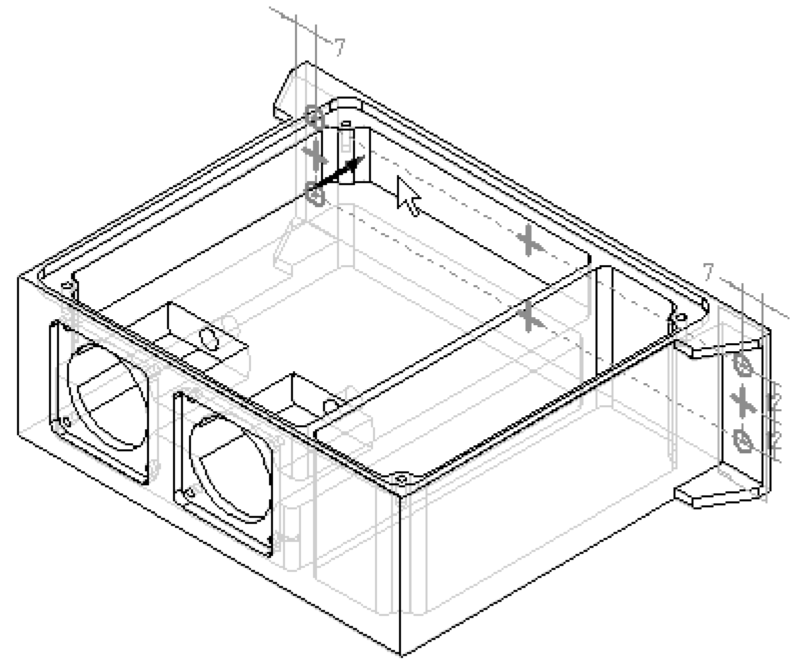


- On the Main toolbar, click Fit.
- Click the Hole Options button. Type 6.35 for the hole diameter and click OK.
- Place and dimension four holes.

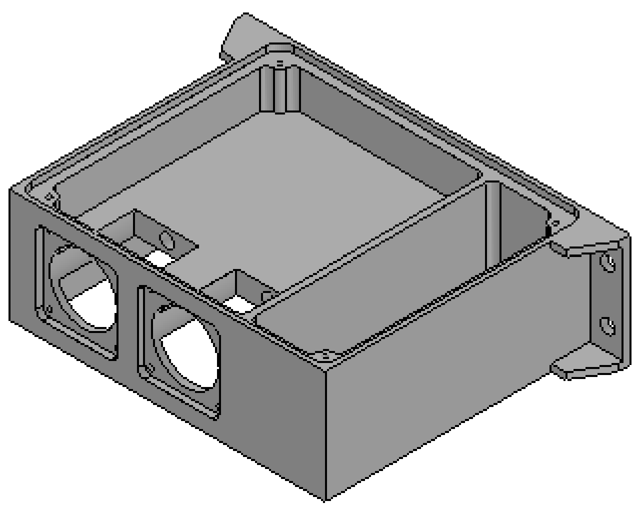


- Choose Close Sketch.
- Click the Through All button.

- Position the cursor so that the direction arrow is displayed as shown, and click.



- Click Finish.
- Close and save the file. This completes the activity.



Summary

In this activity you modeled a machined part that included cutouts, rounds, patterns, mirror copied features, ribs, lip and holes. In this activity, non profile based features were used to more efficiently model the machined part.

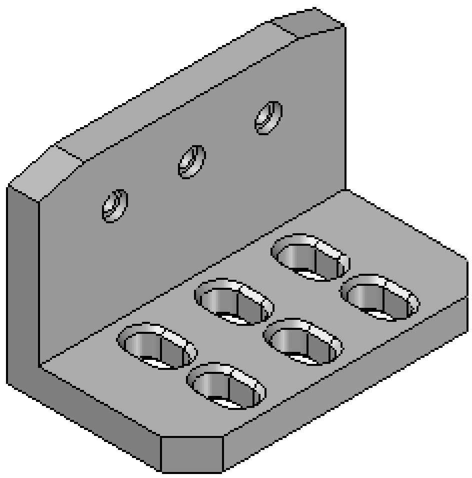
Constructing a bracket

In this activity, construct a solid model and create holes, chamfer, and pattern features.

Activity: Constructing a bracket

Objectives

In this activity you will construct a solid model and create holes, chamfer, and pattern features.



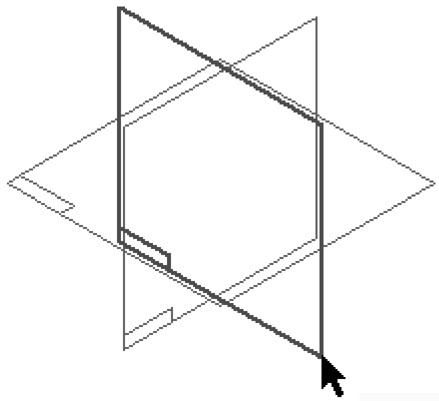
- Create a new ISO part file.
- Make sure you are in the ordered environment.

<u>Construct the base feature</u>

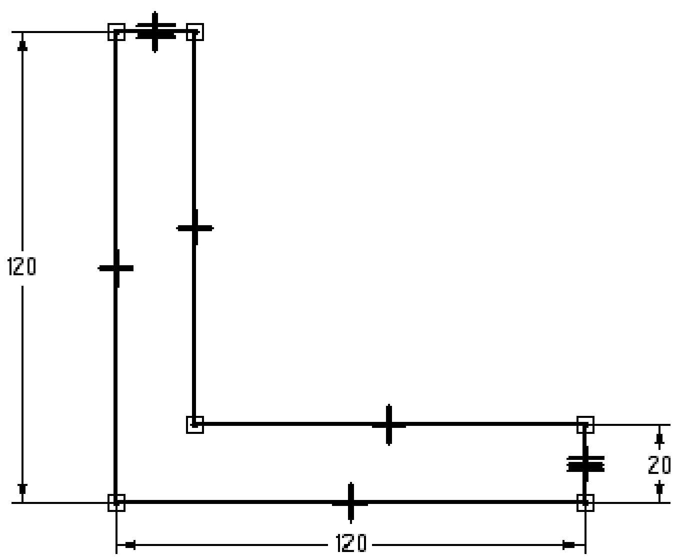
Create an L-shaped extrusion as the base feature. In subsequent steps, use additional features to create the final part shown above.

- Choose the Extrude command.
- Turn on the display of the base reference planes.

- Set the Create-from option to Coincident Plane, and select the reference plane shown.

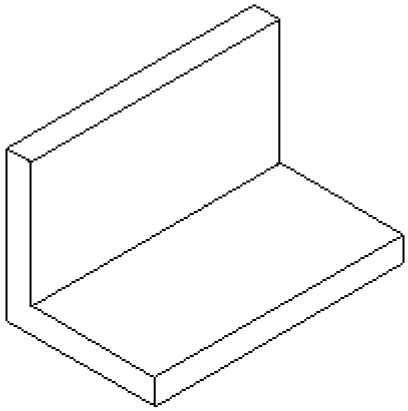


- Hide all reference planes.
- Draw the profile.



- Use an equal relationship, as shown above, to make the two shorter lines equal to one another.
- Choose Close Sketch to complete the profile.
- On command bar. click the Symmetric Extent button. Type 200 in the Distance field and press the Enter key.
- Fit the view.

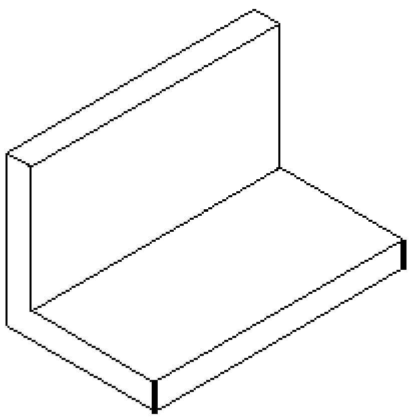
- Click Finish.



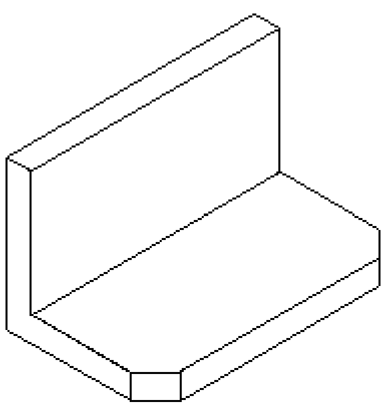
Add a chamfer feature

Add a chamfer treatment feature to the base feature.

- In the Solids group, on the Round drop list, choose the Chamfer command.
- Select the two short vertical edges on the front of the part as shown.



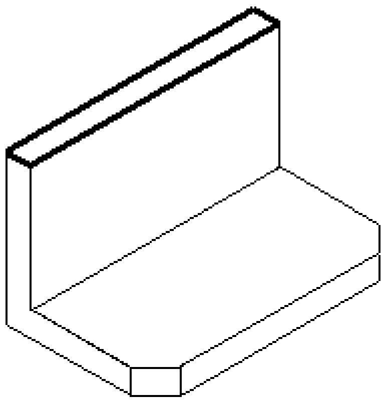
- On the command bar, type 20 in the Setback field and click the Accept button.
- Click Finish.



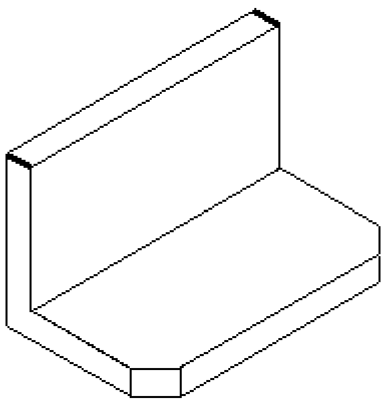
Add chamfer feature

Change the chamfer option settings and add another set of chamfers with an angle and setback.

- Choose the Chamfer command.
- On command bar, click the Chamfer Options button. Click the Angle and setback option and then click OK.
- Notice that after setting the Angle and Setback option, the command bar changes to include the Select Face step.
- Select the top face and then on the command bar click the Accept button.

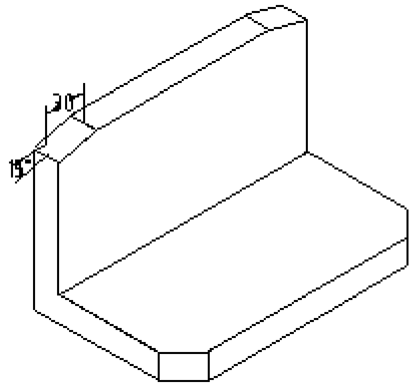


- Select the short edge on each end of the top face.



- Type 30 in the Setback field and type 15 in the Angle field.

- Click the Accept button to apply these values.

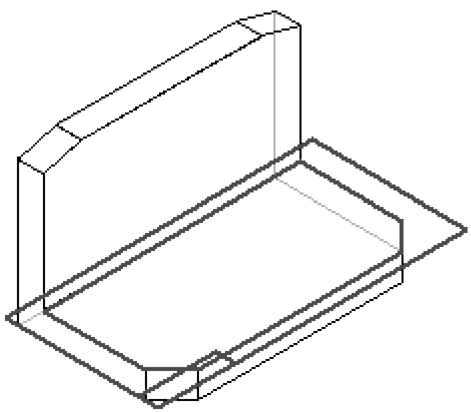


- Click Finish.
- Save the file as *angle.par*.

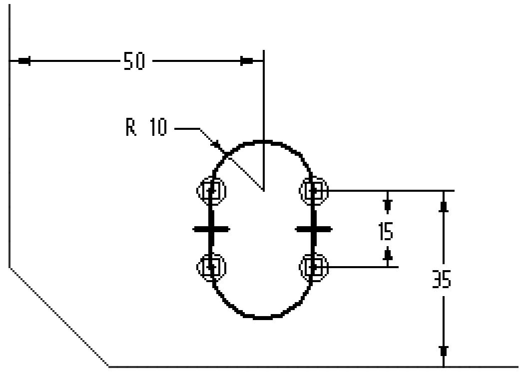
Construct a cutout

Construct a cutout on the front horizontal face shown.

- Choose the Cut command.
- Select the horizontal face shown to define the reference plane.

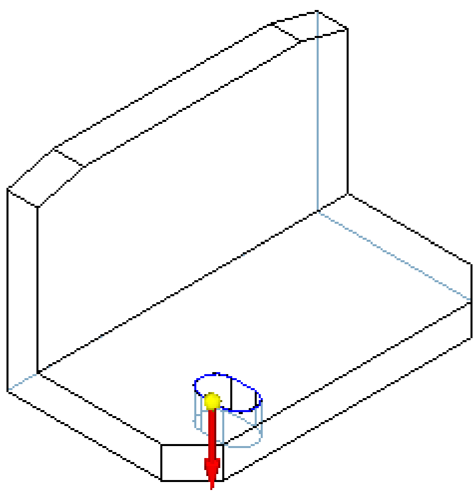


- Draw the profile. Use the Line command and toggle between the Line and Arc modes.



- Choose Close Sketch.

- On command bar, click the Through Next option, and position the cursor to project the cutout downward.



- Click Finish.

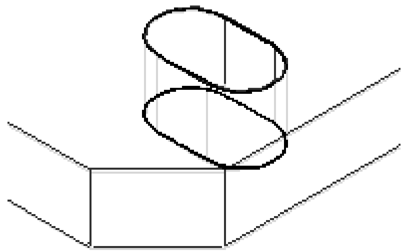
Add a chamfer

Add a chamfer to the cutout constructed in the previous step.

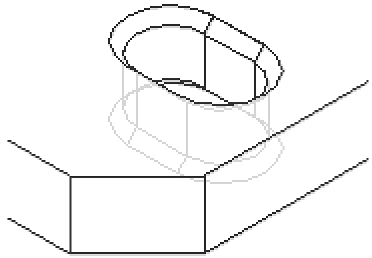
- Choose the Chamfer command.

- On command bar, change the chamfer setting to Equal setbacks..

- Select the top and bottom edges of the cutout.



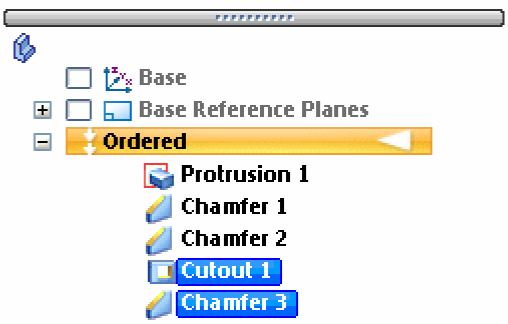
- In the Setback box, type 3 and click the Accept button.
- Click Finish.



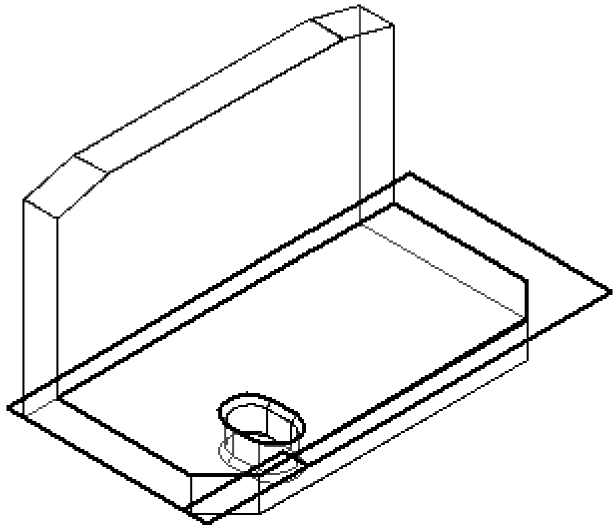
<u>Pattern features</u>

Pattern the cutout and chamfer. Since the cutout is the parent feature of the chamfer, the cutout must be patterned with the chamfer.

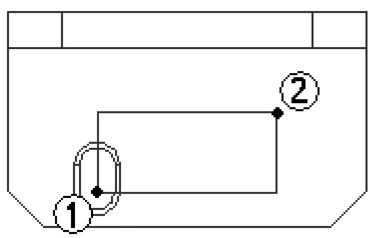
- Choose the Pattern command and on command bar, click the Smart option.
- On PathFinder, select Cutout 1 and Chamfer 3 as the features to pattern. Click the Accept button.



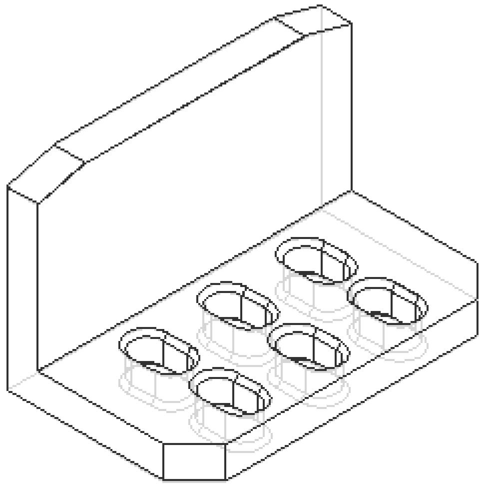
- Select the reference plane to place the pattern on. Use the same profile plane that was used for the Cutout feature.



- In the Features group, click the Rectangular Pattern command.
- Set the Pattern Type to Fixed. Set the X count to 3 and the Y count to 2. Type 50 for the X spacing and 45 for the Y spacing. Press the Enter key.
- Click the center of the arc in the bottom of the cutout to define the start point of the pattern profile (1), and then position the rectangle defining the pattern up and to the right (2).



- Choose Close sketch.

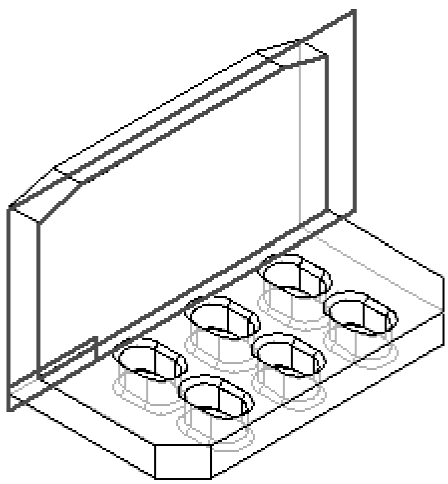


- Click Finish to complete the feature.
- Save the file.

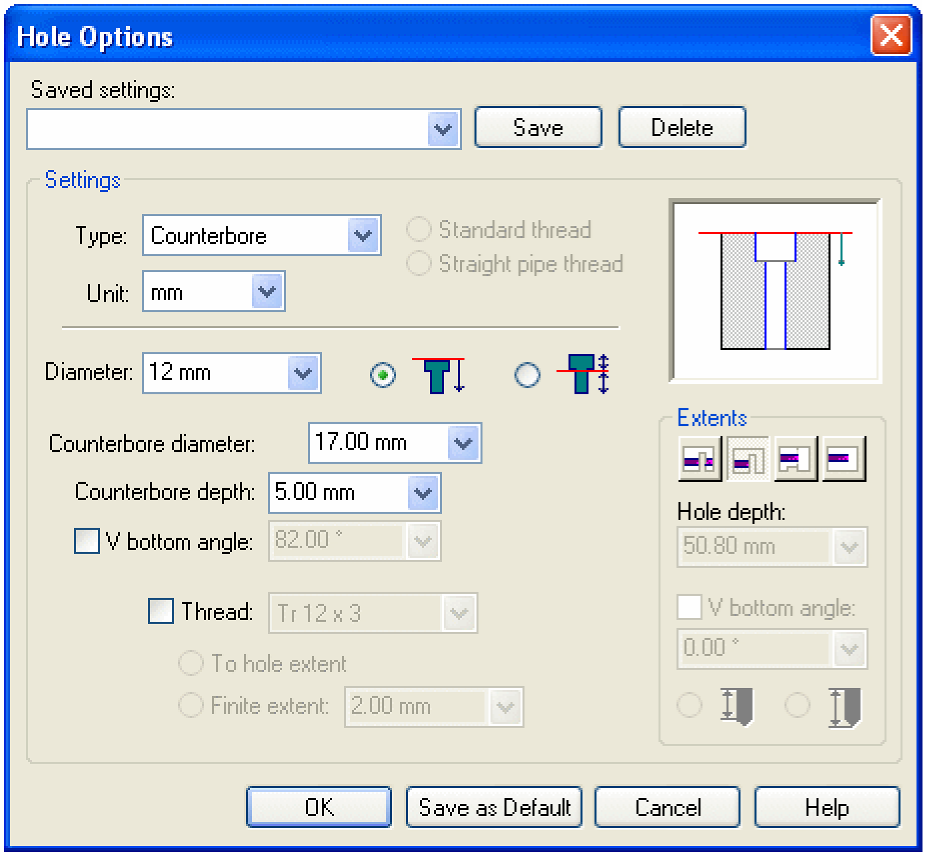
Add hole features

Add holes to the vertical front face of the part.

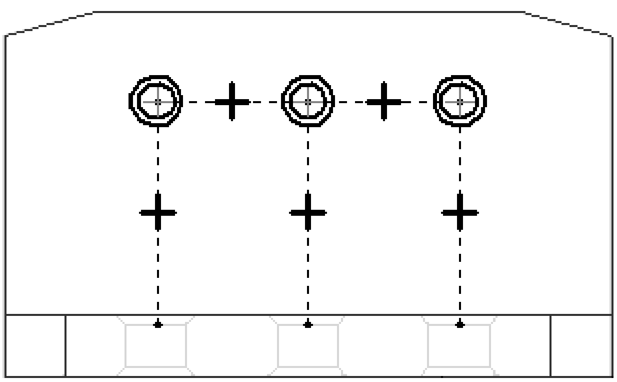
- Choose the Hole command.
- Select the front vertical face of the bracket as shown.



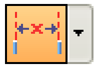
- Click the Hole Options button and set the options shown and click OK.

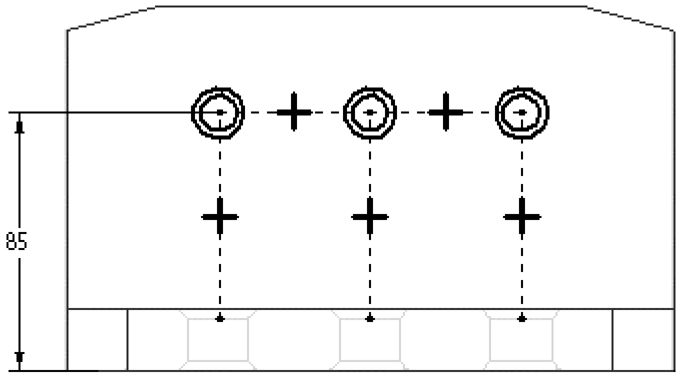


- Place a hole centered over each slot. Align the holes as shown.

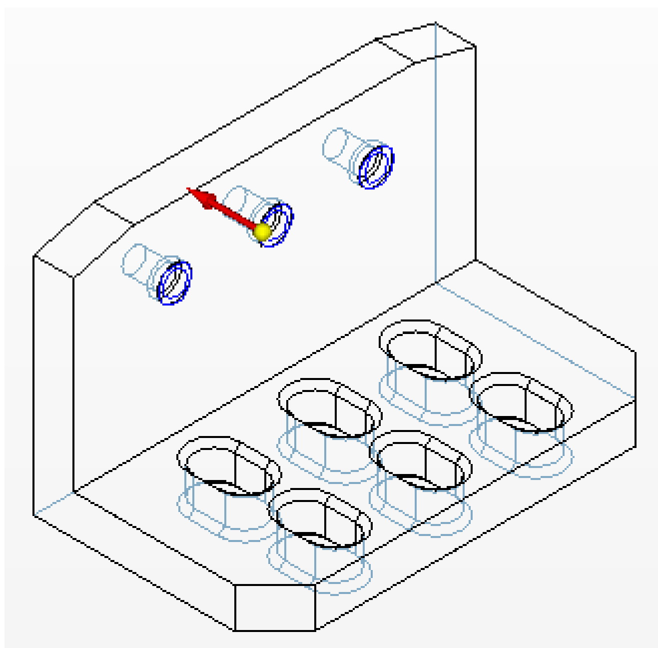


- Dimension the location of the holes as shown.





- Choose Close Sketch.
- Specify the extent direction shown in the illustration.



- Click Finish.
- Save and close this file. This completes the activity.

Summary

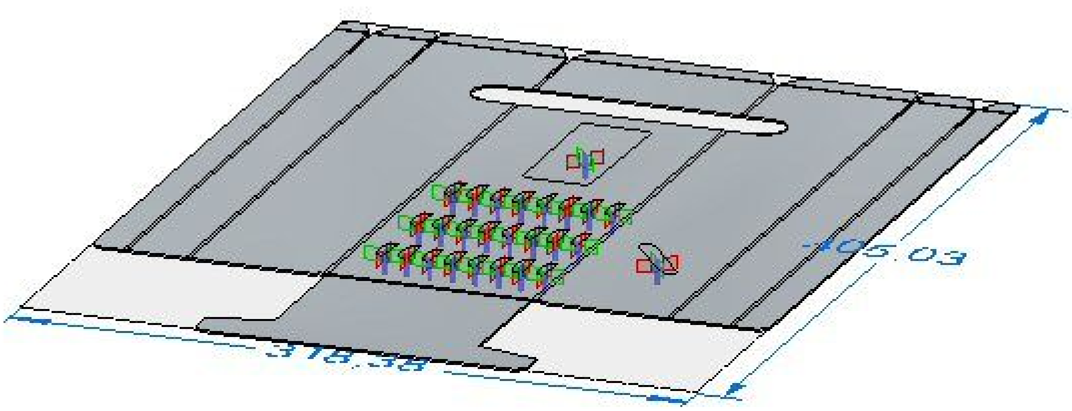
In this activity you learned how to create a chamfer feature and to create a pattern consisting of more than one feature. You used the hole command to create the counterbored holes in the bracket.

Lesson

12 *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

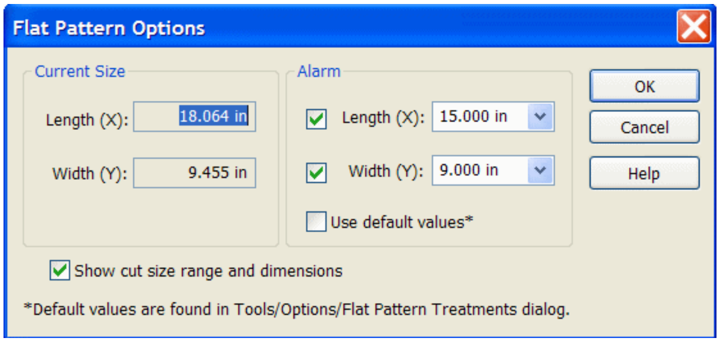
When flattening sheet metal parts, the system adds bend relief to the flat pattern. This system-generated bend relief can cause problems to downstream manufacturing processes such as punching and nesting. While working in the Sheet Metal environment, you can set options on the Flat Pattern Treatments 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.

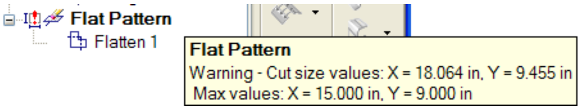
If you change the options on this tab after a flat pattern is generated it recomputes the flat pattern, or updates the flat pattern.

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

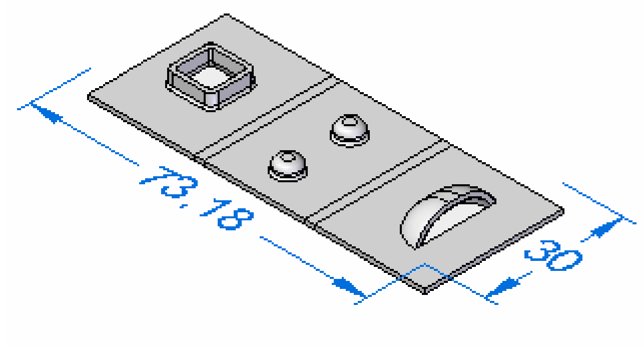
If you want to remove a deformation feature after you flatten a part using the Flatten and Part Copy commands, you can construct a cutout feature that is sized according to the area the deformation feature occupied. In many cases, you can use the Include command to create a cutout profile that is associatively 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.

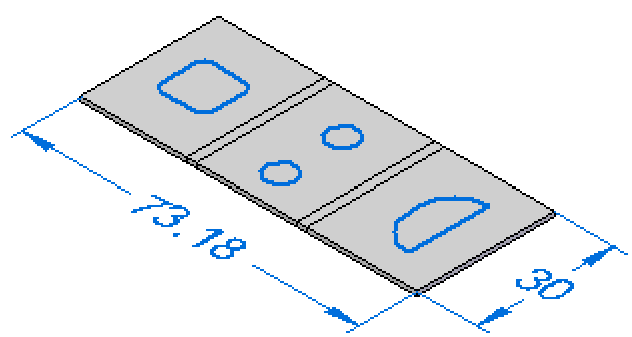
Displaying deformation features in the flat pattern

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.

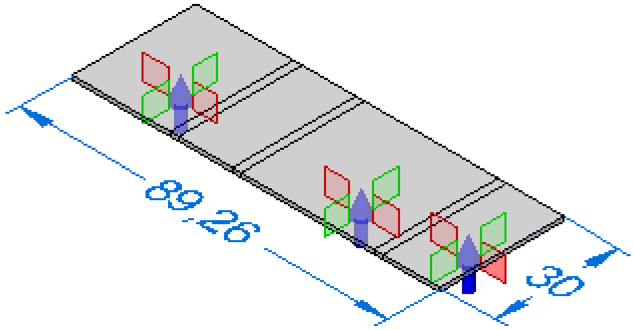
You can display the deformation features as a formed feature,



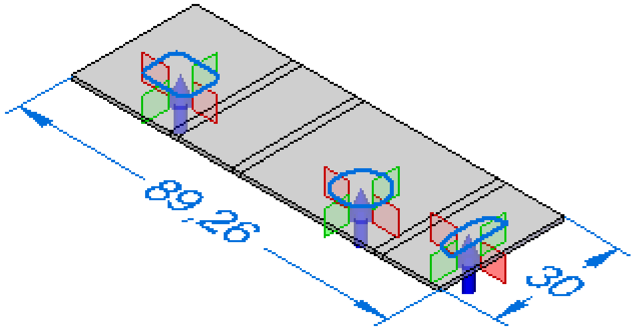
as a feature loop,



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.

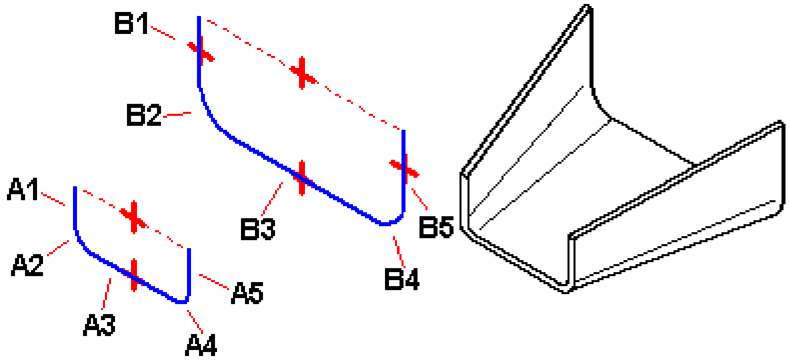
When saving the document to .dxf format:

- 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

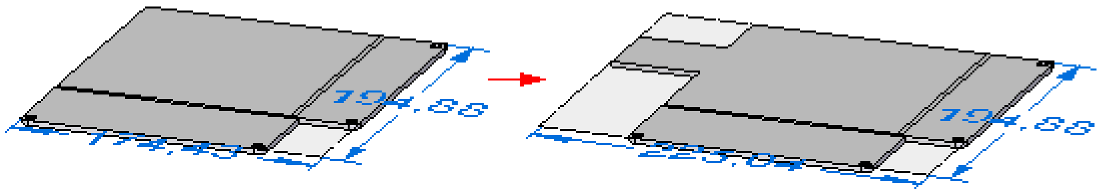
  If the lofted flange contains faces that would prevent it from being flattened, a gray arrow is displayed adjacent to the feature on the PathFinder tab. If you pause your cursor over the feature in PathFinder, a message is displayed in the status bar describing the problem.

PMI dimensions in the flat pattern

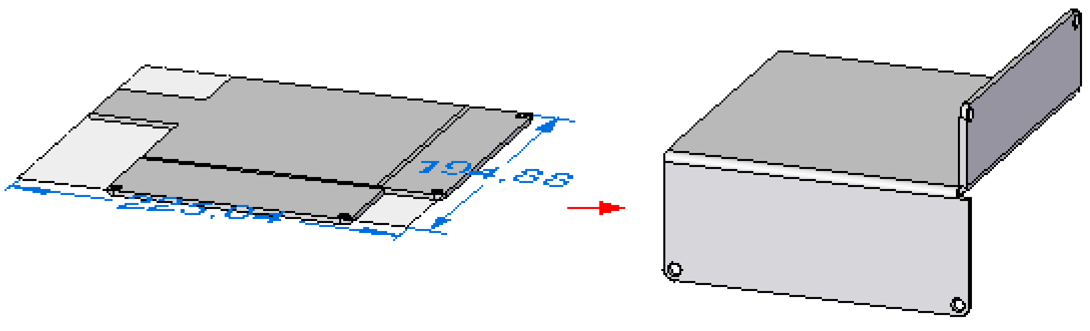
When a flat pattern is created, PMI dimensions are placed as driven dimensions. In other words, these dimensions are for reference only and cannot be changed while in the flat pattern. If you select a PMI dimension in the flat pattern, all fields on the dimension edit control are disabled. If you make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

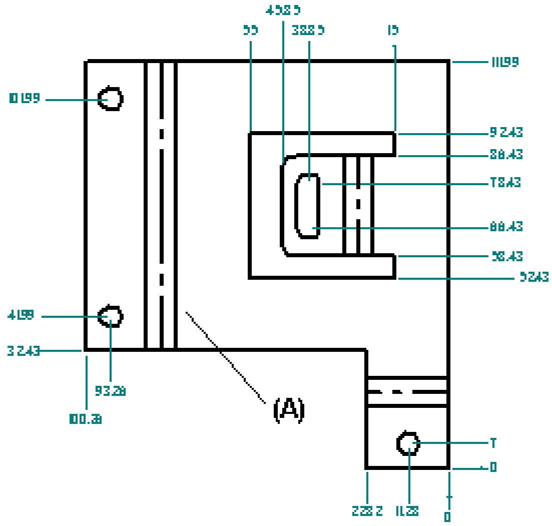
- Bend up centerlines are saved to a layer named UpCenterlines. This layer contains the bend centerlines of all linear and conical bends that are in the up direction relative to the selected output face. These lines are generated in the flatten process and do not exist in the model. The line style associated 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.

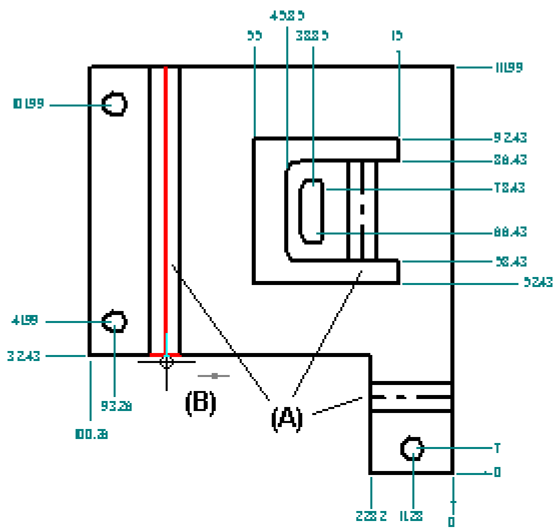


To apply the template, you must run the Drawing View Wizard and set the Part and Sheet Metal Drawing View Options on the first page of the Drawing View Creation Wizard to *Flat Pattern*.

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.

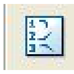
Updating flat pattern drawings

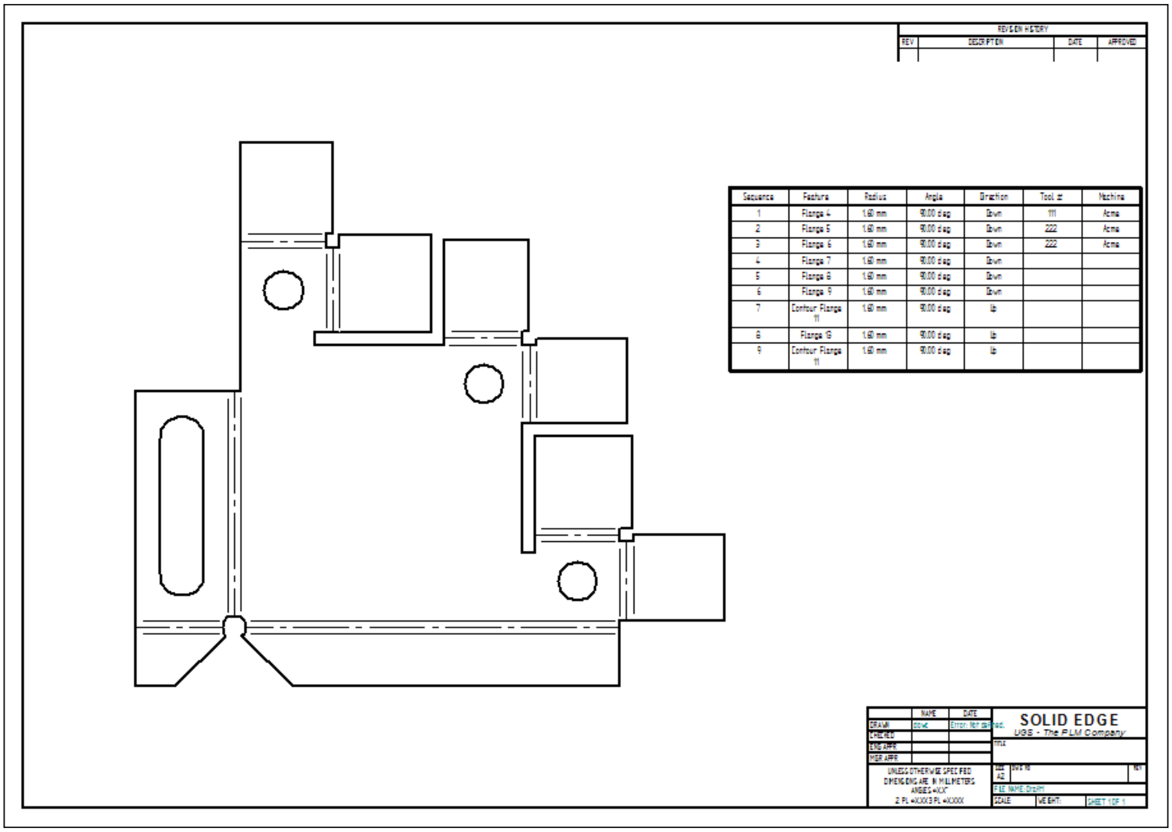
When you make design changes to a folded sheet metal part, you need to update the associatively flattened part first, and then update the flattened drawing to see the changes. When you open the flattened part document, an out of date symbol is displayed adjacent to the base feature in the PathFinder 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.





Sequence	Feature	Radius	Angle	Direction	Tool #	Machine
1	Flange 4	1.60 mm	90.00 deg	Down	111	Acme
2	Flange 5	1.60 mm	90.00 deg	Down	222	Acme
3	Flange 6	1.60 mm	90.00 deg	Down	222	Acme
4	Flange 7	1.60 mm	90.00 deg	Down		
5	Flange 8	1.60 mm	90.00 deg	Down		
6	Flange 9	1.60 mm	90.00 deg	Down		
7	Contour Flange 11	1.60 mm	90.00 deg	Up		
8	Flange 13	1.60 mm	90.00 deg	Up		
9	Contour Flange 11	1.60 mm	90.00 deg	Up		

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

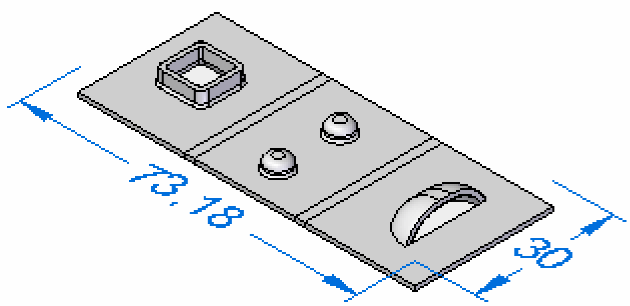
Specifies the minimum value for the arc created from the B-spline curve.

Deviational Tolerance

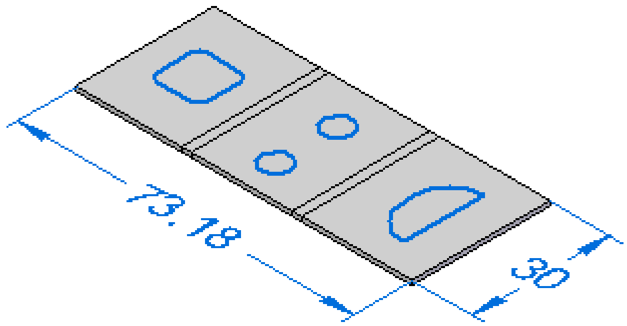
Specifies the deviation tolerance for the B-spline curve.

Formed Feature Display

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



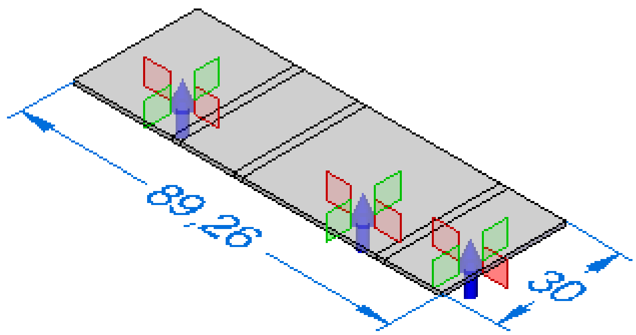
As Feature Loops	Displays the sheet metal features as feature loops in the flat pattern.



Note

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

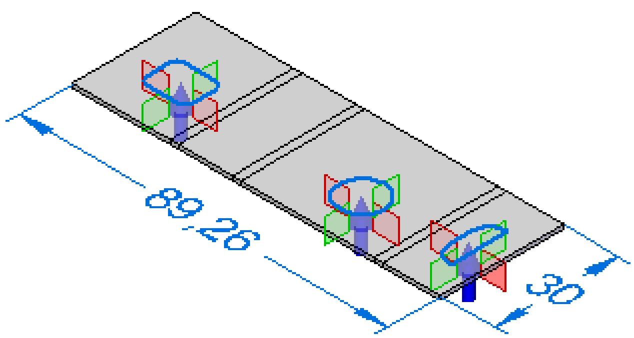
As Feature Origin	Displays the sheet metal features as feature origins in the flat pattern.



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.

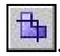
Maintain Holes in the Flat Pattern

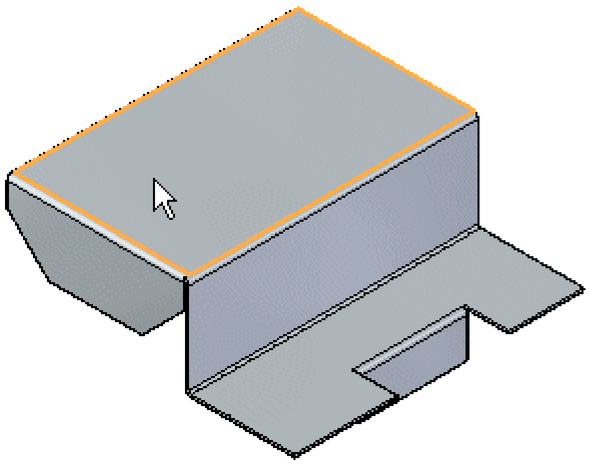
When a hole is placed on a non-orthogonal surface or angle, the resulting hole is elliptical when flattened. Creates the flat pattern with actual round holes.

Save as Flat Command

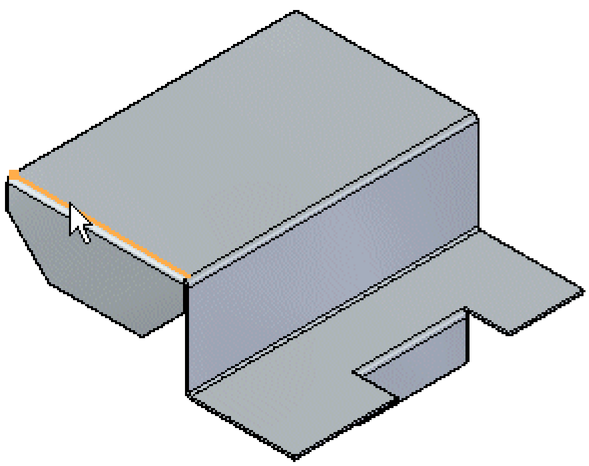
Use Existing Flat Pattern (Use Folded Model if Not Defined)	Creates the flat pattern definition based on an existing flat pattern. Any material added or removed in the flat pattern environment is included when the flat is saved. If no flat pattern exists, the folded model is used to define the pattern.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



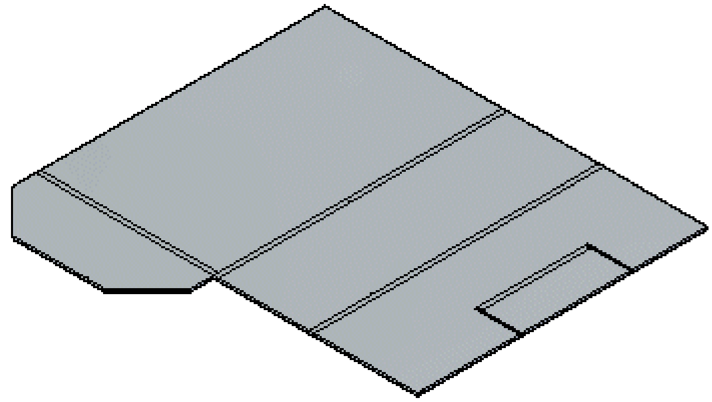
4. Click an edge to define the X axis and origin.



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.

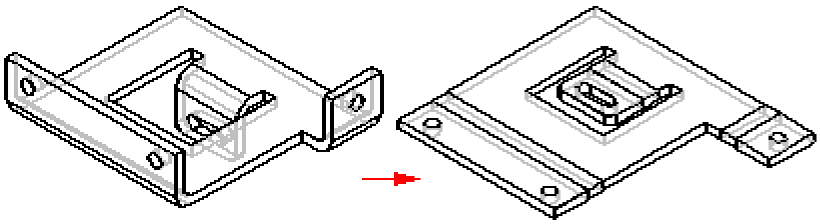
- The flat pattern created with the Flatten command contains all bend centerline information used to create bend centerlines in Draft drawing views. It also contains the same information used by the Save As Flat command.

- 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

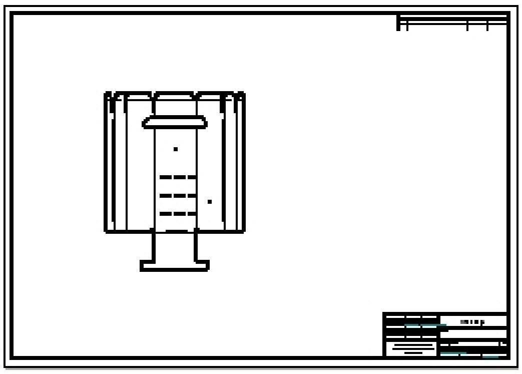
Bend lines are not automatically created with this command. If you want to automatically create bend lines, you must use the Insert Part Copy command to create a flat .psm file. You can then place a part view for the flat .psm file in a draft file.

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.



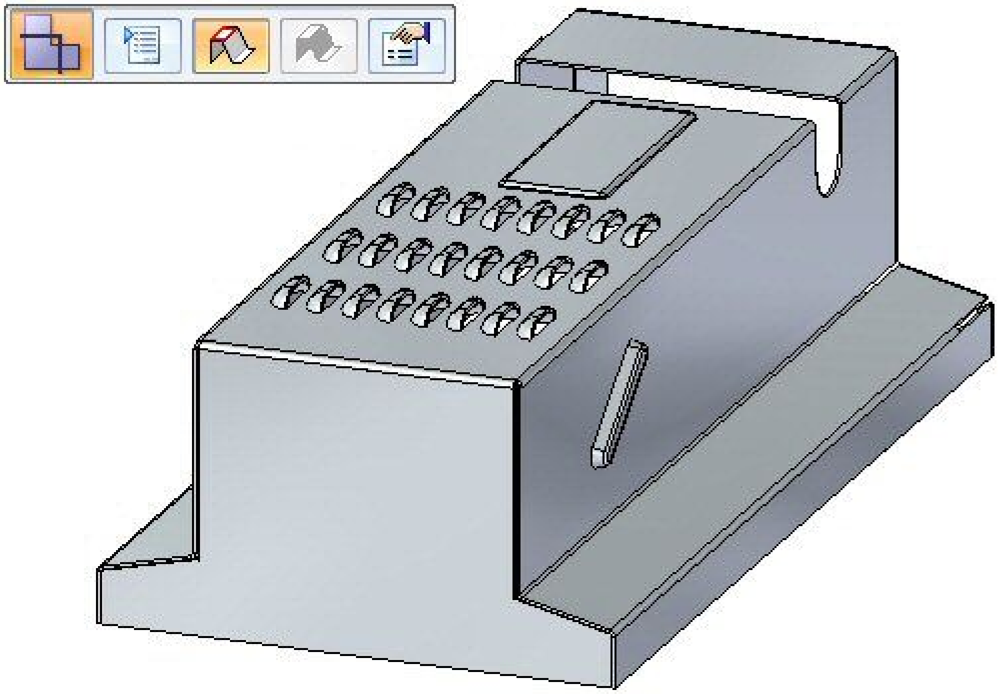
Activity: Creating a flat pattern from a sheet metal part

Open a sheet metal file

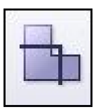
- Start Solid Edge ST4.
- Click the  **Application** button ® **Open** ® *flatpattern_activity.psm*.
- Proceed to the next step.

Creating a flat pattern

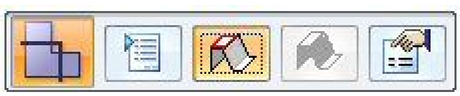
- Click Tools® Model® Flat Pattern.



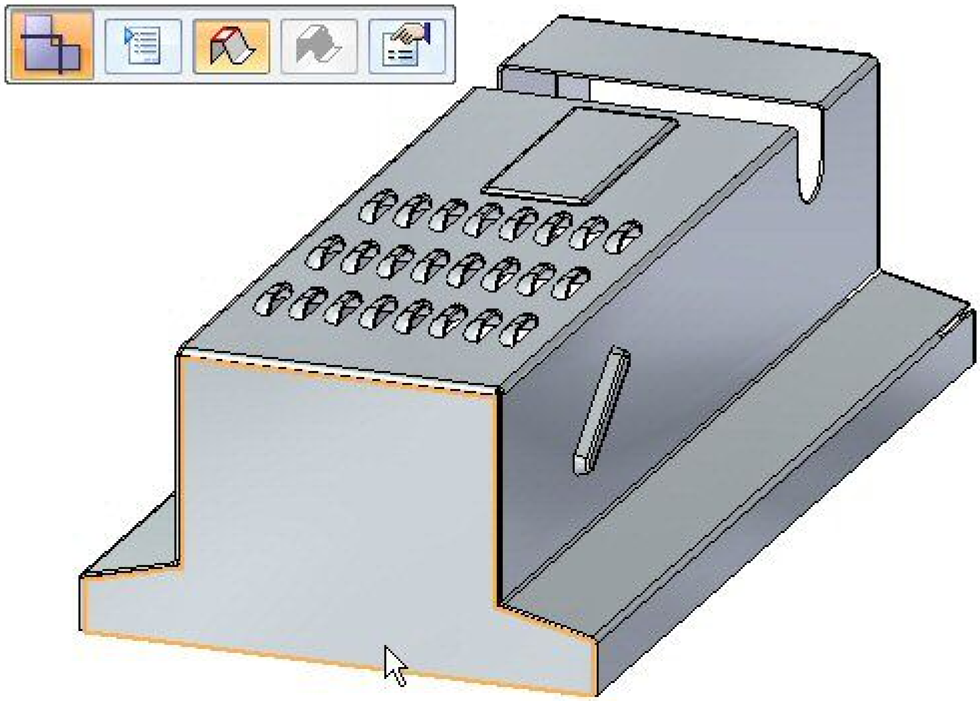
- Click Tools® Flat® Flatten.



The Flat Pattern command bar is displayed.



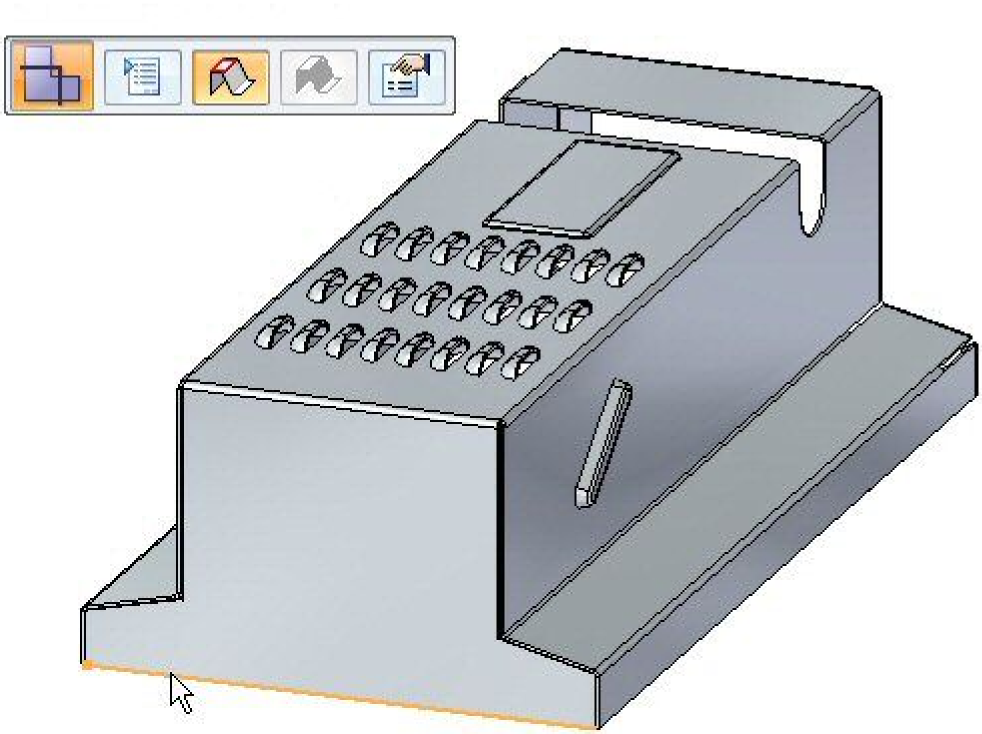
- Select the face shown.



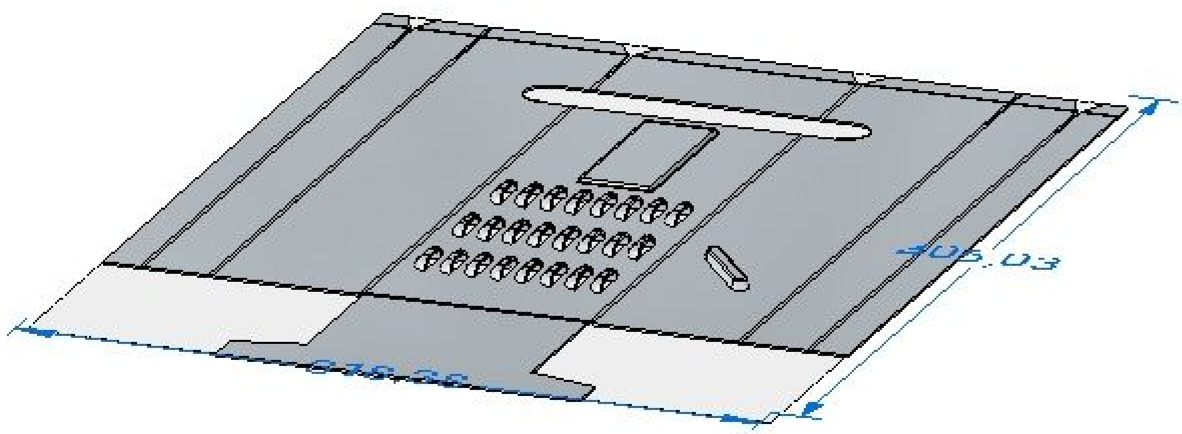
- Select the edge shown to orient the flat pattern.

  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.

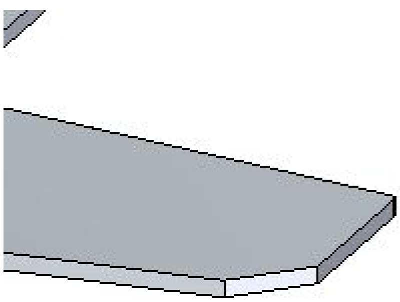
  

  Note

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

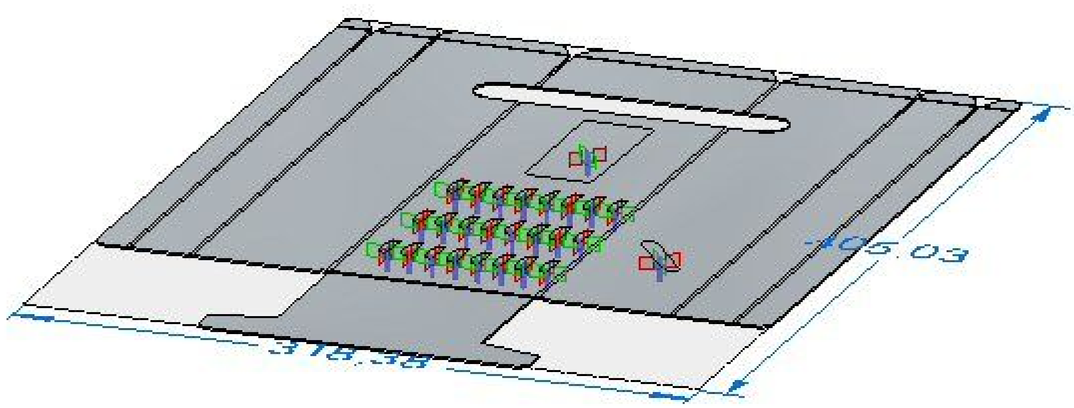
- Set Outside Corner Treatments to Chamfer and set the value to 4.00 mm. Click Apply. Observe the results. Chamfers are applied to the exterior corners that do not contain rounds.

  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

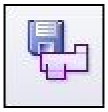
When placing a flat pattern onto a drawing sheet in Solid Edge Draft, this display controls the what geometry is placed. If the sketches or 3D geometry is desired on the drawing sheet, the option will need to be set here.

- 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

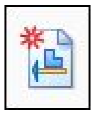
  Some numerical control machines can read a .dxf file directly. The flat pattern can also be saved as a Solid Edge part file.

- Proceed to the next step.

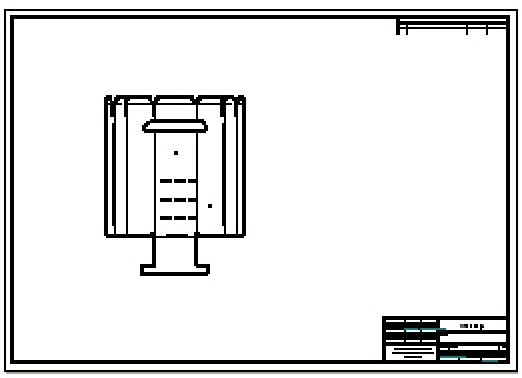
Placing a flat pattern on a drawing sheet

- 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.

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.

Lesson

13 Ordered sheet metal activities

Activity: Editing an ordered flange feature

Overview

In this activity, you will create a flange and then edit the default flange profile to accommodate the shape of a contour flange.

Objectives

After completing this activity, you will be able to edit a flange.

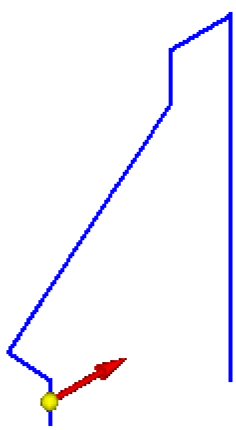
Activity

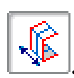
Step 1: Open *flange_edit.psm*. Use Save As to create a duplicate file called *flange_edit2.psm*. Initially, you will work in this new file. Later you will open the original file.

> **Note**
>
> When you Save As and define a new name, this new file will be active.

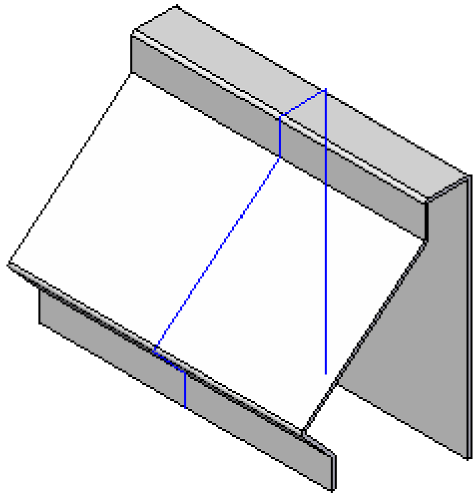
Step 2: Create a contour flange using the sketch provided.

- ☐ On the Home tab® Sheet Metal group, choose the Contour Flange command .
- ☐ On the command bar in the Sketch Step, click the Select from Sketch option.
- ☐ Select Sketch 1.
- ☐ Click the Accept button.
- ☐ Position the direction arrow pointing inside the sketch. Click to accept the inside of the sketch for material addition.



- ☐ Click the Symmetric Extent button .

☐ In the Distance box, type 250, and press the **Enter** key.



☐ Click Finish.

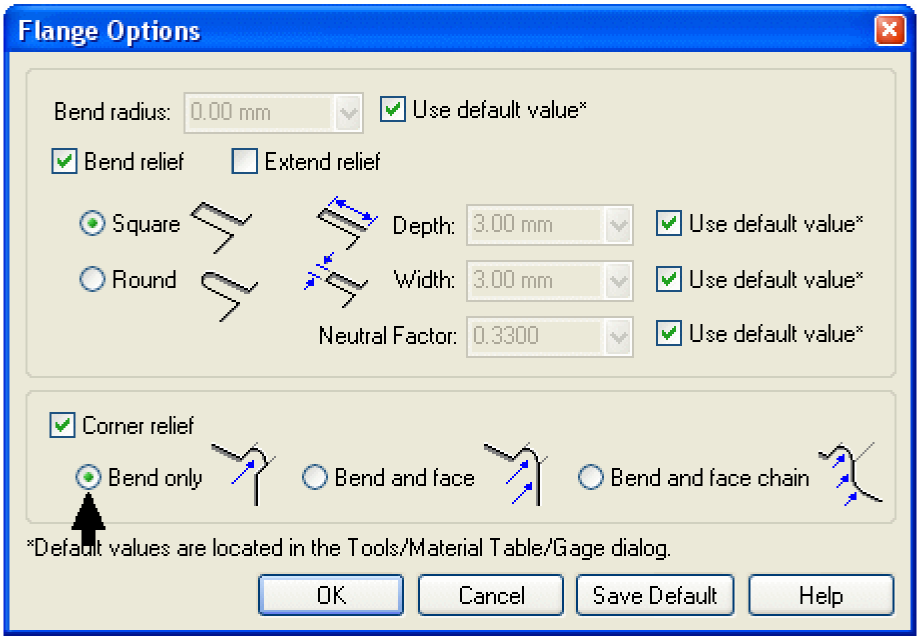
☐ Hide all sketches.

Step 3: Create a flange.

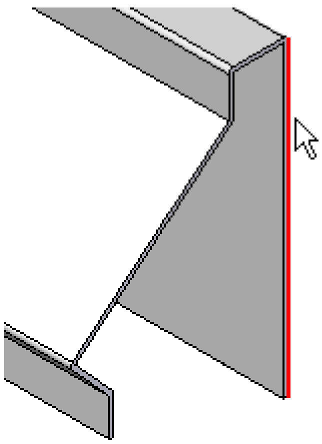
☐ Click the Flange command .

☐ On the command bar, click the Flange Options button.

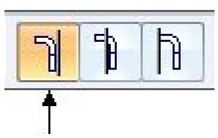
☐ Set the Corner relief option to Bend only and click OK.



☐ For the thickness face edge, select the outside edge on the right end of the model.

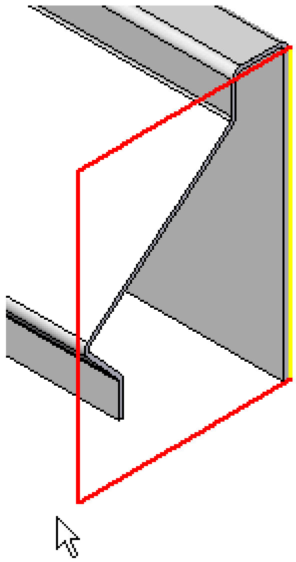


☐ Set the Material Inside option.

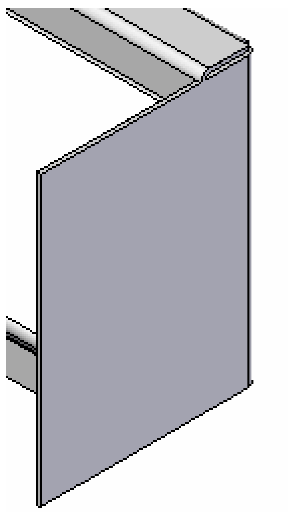


☐ For the extent step, in the Distance box type 150 and press the **Enter** key.

☐ Click to place the flange toward the front of the model.



☐ At this point, you could go to the edit flange profile step or Finish the flange. We will Finish the flange and then go back and edit the flange profile. Click Finish.

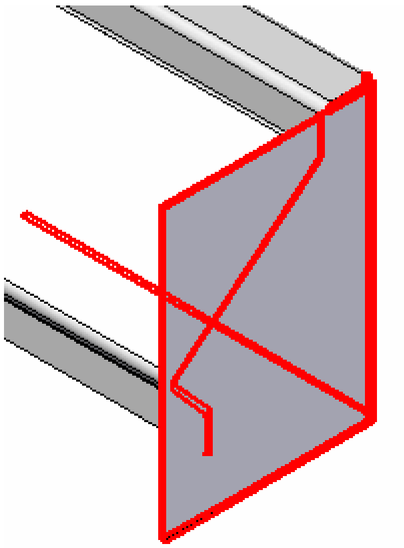


☐ Save the file.

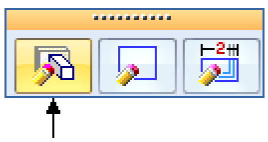
Step 4: Edit the flange profile to accommodate the shape of the contour flange.

☐ Click the Select tool .

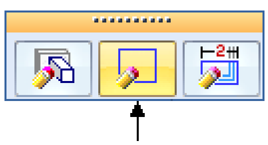
☐ Select the flange.



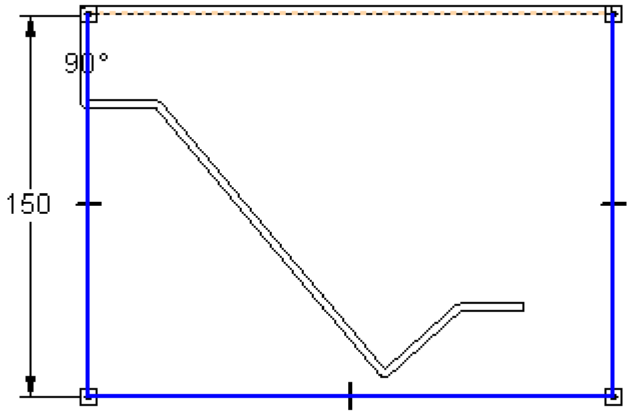
☐ There are two ways to enter the profile mode to edit the flange profile. You can click the Edit Definition button and then on the command bar, click the Flange profile step.



☐ You can also click the Edit Profile button to go directly to the profile.



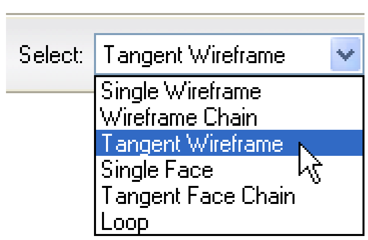
☐ Click the Edit Profile button.



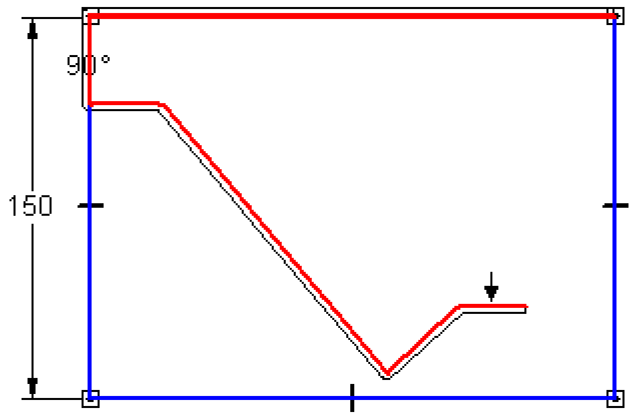
☐ On the Home tab® Draw group, click the Include command. You will use this command to draw a profile that precisely matches the shape of existing contour flange edges.

☐ On the Include Options dialog box, set the "Include with offset" option and click OK.

☐ Set the Select filter to Tangent Wireframe.



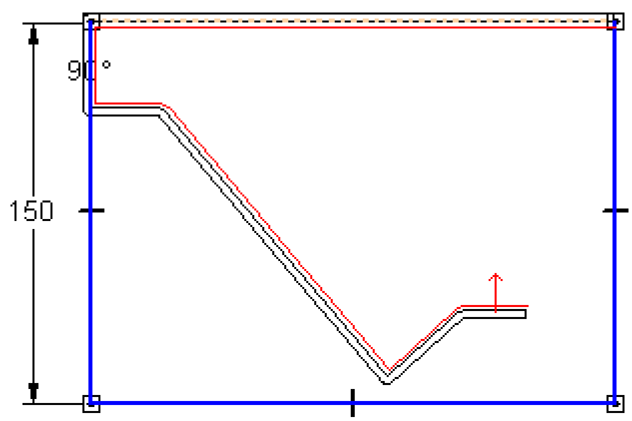
☐ Select the interior edge of the model as shown and click the Accept button.



☐ The Tangent Wireframe option will create extra lines that you will delete later.

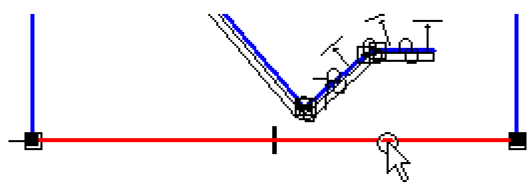
☐ On the command bar, in the Distance box, type 2 and press the **Enter** key.

☐ Position the cursor so that the arrow points inside. Click to place the profile elements inside of the model.



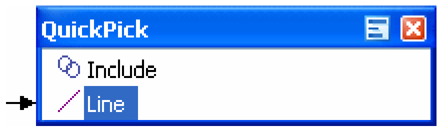
☐ On the Home tab® Select group, click the Select tool .

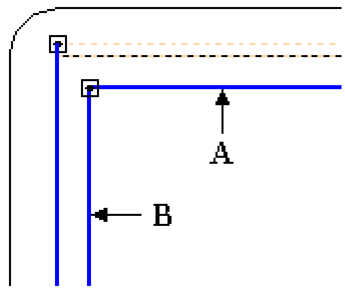
☐ Delete the bottom horizontal line of the original flange profile.

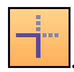


☐ Zoom in to the top left corner.

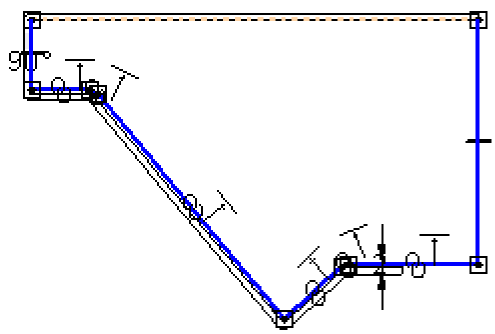
☐ Delete lines A and B created with the Include command. Use QuickPick for selecting the horizontal line (A). You will get two choices to select: 1) the include/offset relationships and 2) the horizontal line. Make sure you select the horizontal line.





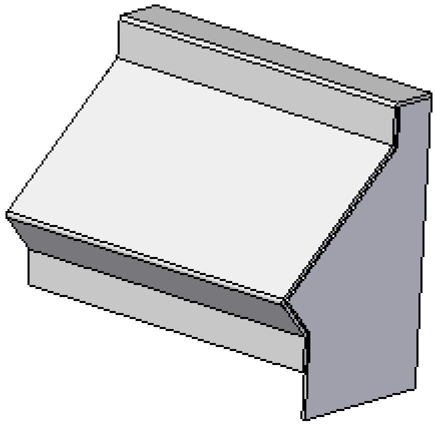
☐ Click the Trim Corner command .

☐ Trim and join both ends of the included profile with the two original vertical lines. This will complete the profile as shown in the illustration below.



☐ Click Close Sketch to complete the profile.

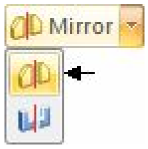
☐ Click Finish again to complete the flange feature.



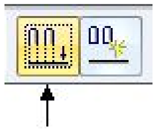
Step 5: Mirror the last flange you created to the other side of the sheet metal part.

☐ Turn on the display of the base reference planes.

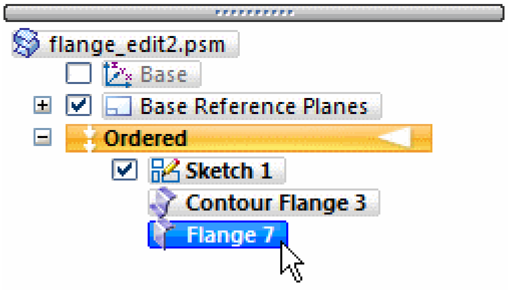
☐ On the Home tab® Pattern group, click the Mirror Copy Feature command.



☐ On the Mirror Copy feature command bar, click the Smart button.

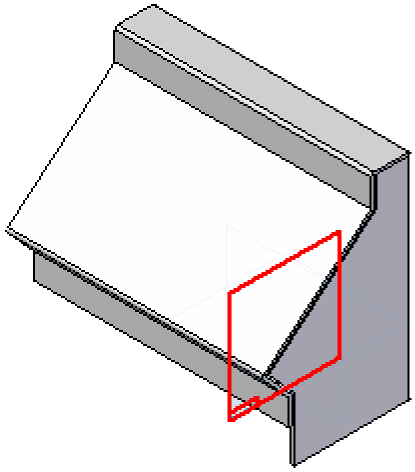


☐ On Pathfinder, select the flange feature.

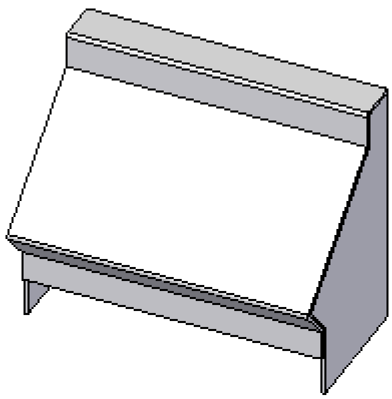


☐ On the command bar, click the Accept button.

☐ Select the reference plane shown.



☐ Click Finish.

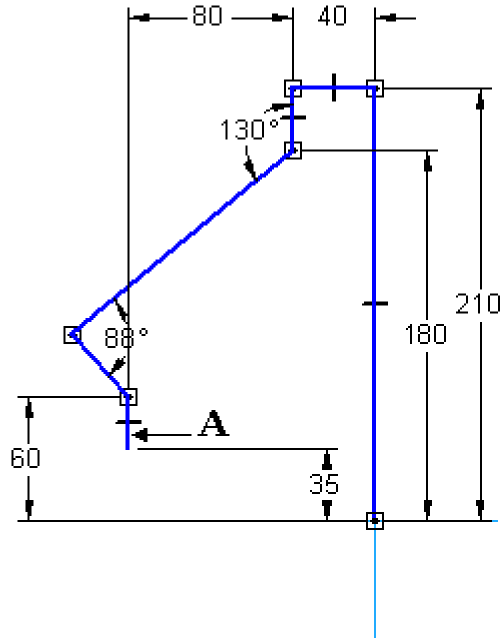


☐ Save and close the file.

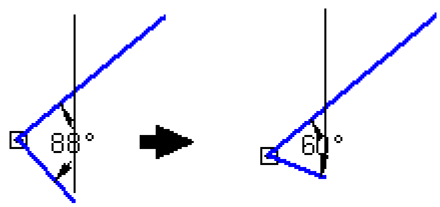
Step 6: Use an alternate method of creation for the contour flange.

☐ Open the file *flange_edit.psm*.

☐ Edit sketch named “Sketch 1”. Delete line A.



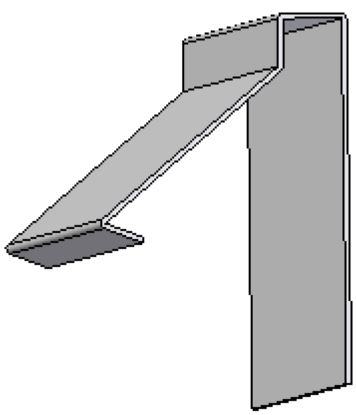
☐ Edit the 88° dimension to 60°.



☐ Click Close Sketch and Finish.

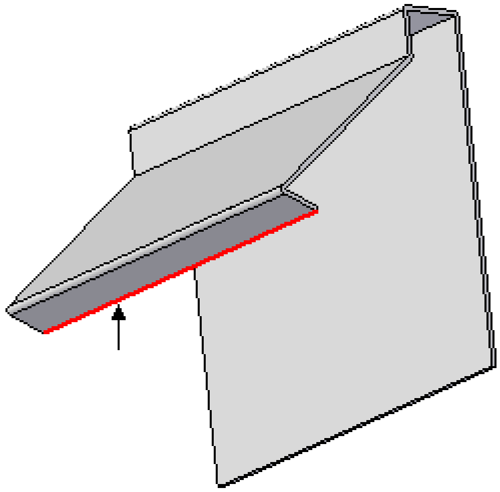
Step 7: Create a contour flange using the modified sketch.

☐ Create a contour flange with a symmetric extent of 250 mm. Add material inside the profile.

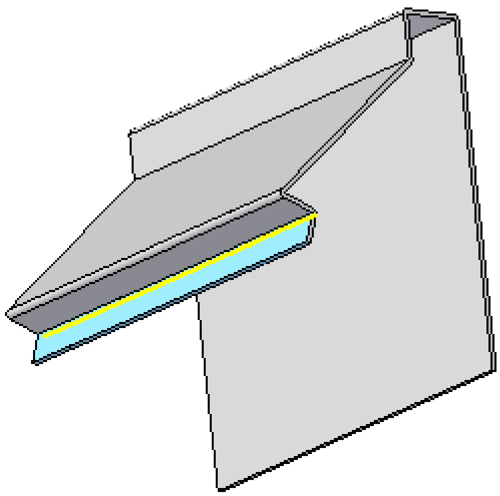


Step 8: Use the Match Face option in the Flange command.

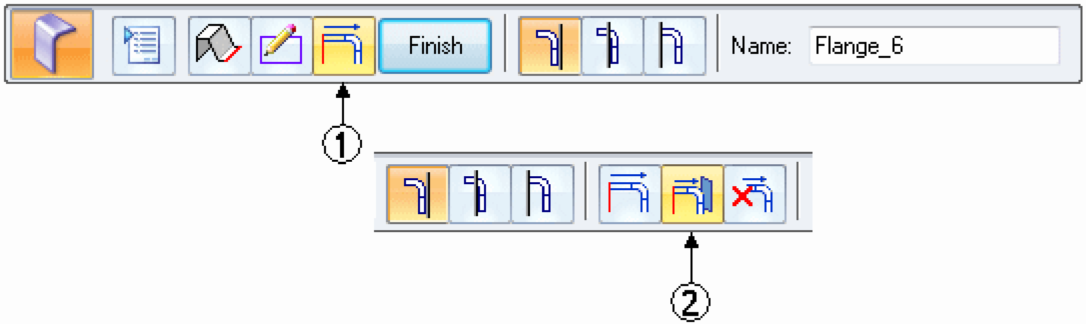
- ☐ Click the Flange command .
- ☐ Select the edge shown.



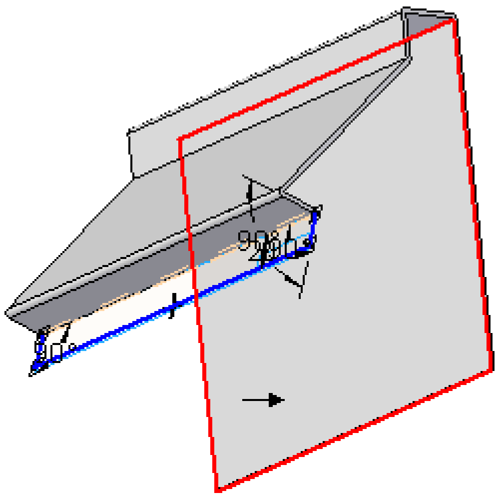
- ☐ Type 20 mm in the Distance field and set the direction as shown.



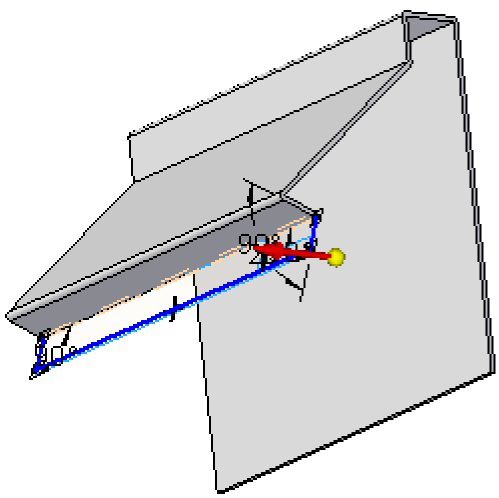
☐ In the Offset Step (1), click the Match Face button (2) and type 40 mm in the Offset field.



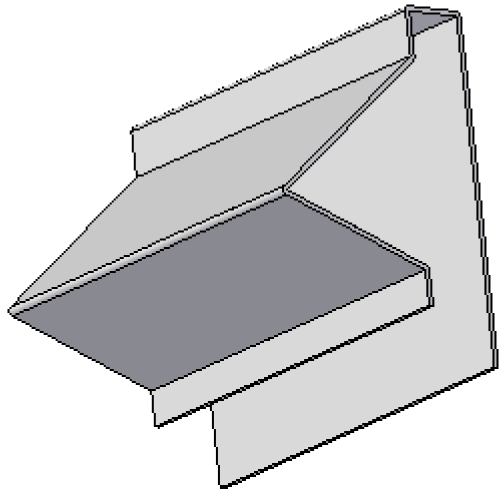
☐ Select the inside back face as the “match face”.

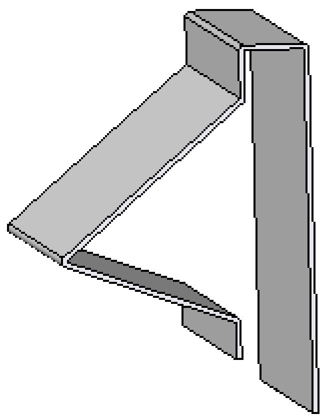


☐ Set the direction arrow towards the flange.

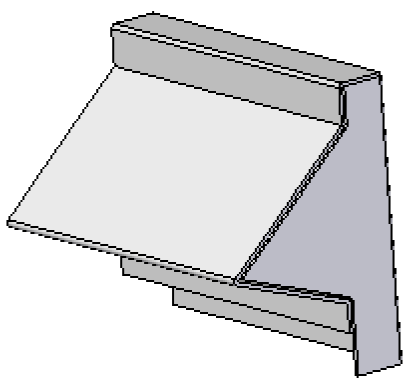


☐ Click Finish. The flange has been matched to the back face.





Step 9: You can repeat Steps 3–5 to complete the enclosure.



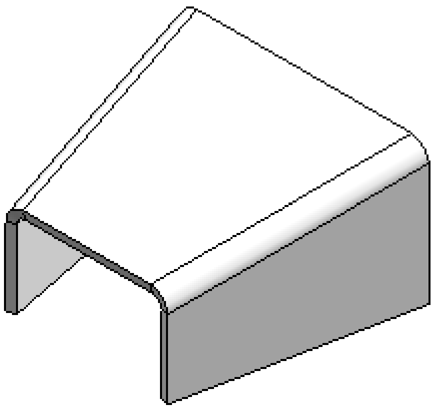
Activity Summary

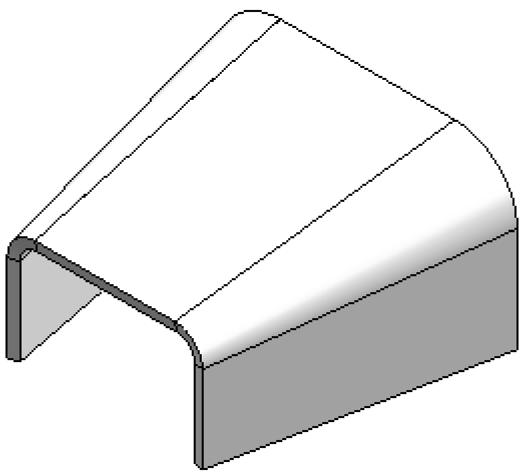
In this activity, you learned how to construct a flange, then modify the profile using existing edges to correctly create the desired geometry of the new flange.

Activity: Using the ordered lofted flange command

Overview

In this activity you will use the Lofted Flange command to construct a base feature.





Objectives

After completing this activity, you will be able to construct a lofted flange.

Activity

Step 1: Open *loft.psm*.

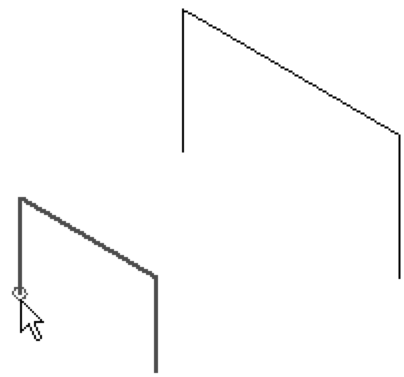
Step 2: Construct a lofted flange.

☐ On the Home tab® Sheet Metal group, click the Lofted Flange command.

☐ On the command bar, ensure that the Select from Sketch/Part Edges option is set.

☐ For the first cross section, position the mouse cursor over the left end of Sketch 1.

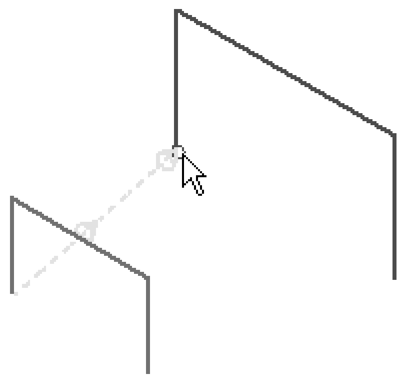
☐ Click when a red dot displays on the profile.



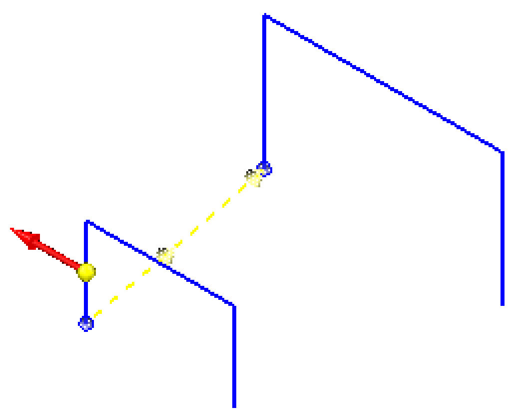
☐ Click the Accept button to accept the profile and start point.

☐ Select the left end of Sketch 2 as the next cross section.

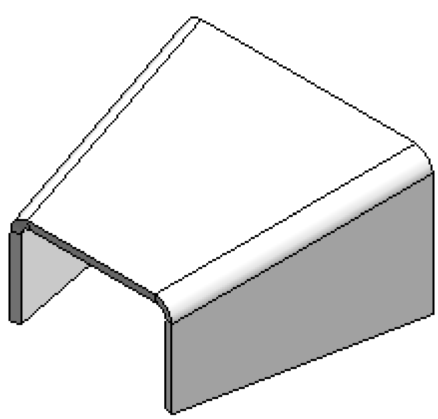
- ☐ Click the Accept button.



- ☐ Position the cursor outside of the profile.
- ☐ When the arrow points outside of the profile, indicating material addition will occur on that side of the profile, click.

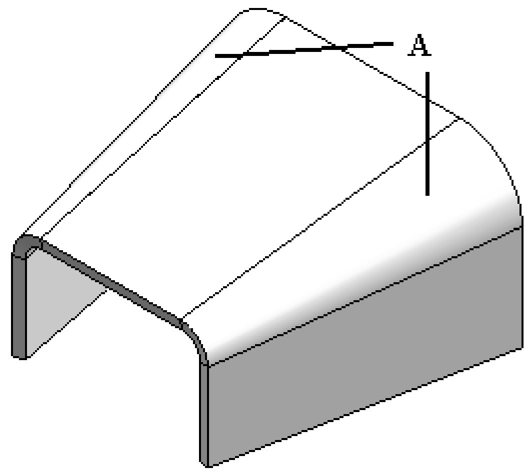


- ☐ Click Finish to complete feature. Notice that sketches are hidden in the illustrations for the purpose of clarity.

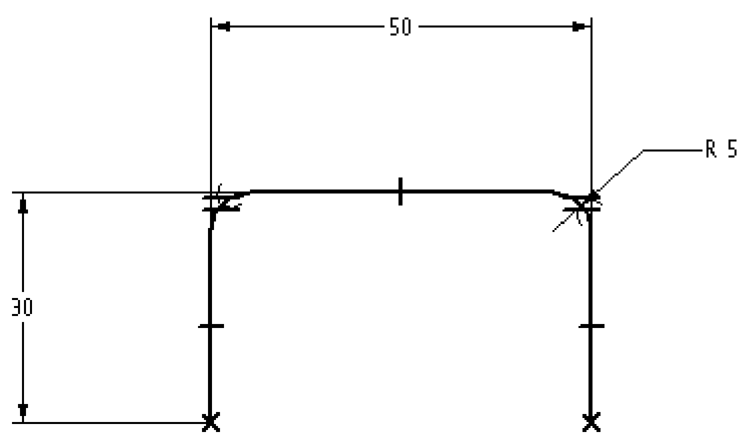


- ☐ Save the file.

Step 3: In the first phase of this activity, the Lofted Flange command inserted bends using the default bend radius defined in the options for this file. Edit the profiles to include fillets that will define the bend size and shape. Notice in the illustration (A) that the profiles have been edited to provide conical shaped bends.



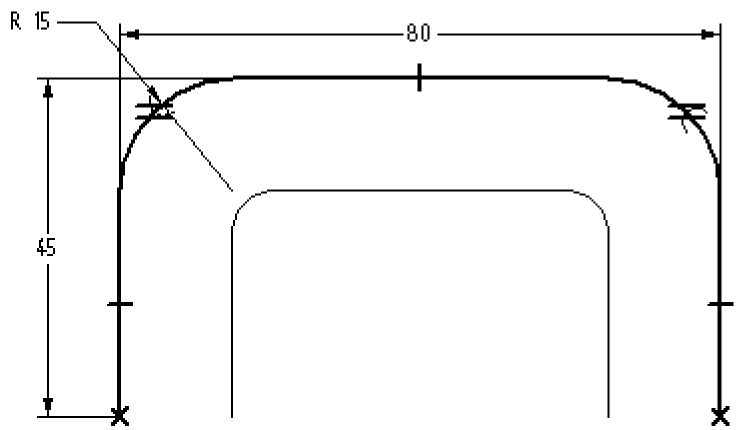
- ☐ Click the Select Tool.
- ☐ Position the mouse cursor over Sketch 1.
- ☐ Right-click to display the shortcut menu.
- ☐ On the shortcut menu, click Edit Profile.
- ☐ Use the Fillet command to add 5 mm fillets at the two vertices.
- ☐ Add a radial dimension to one of the fillets.
- ☐ Add an equal relationship to both.



Note

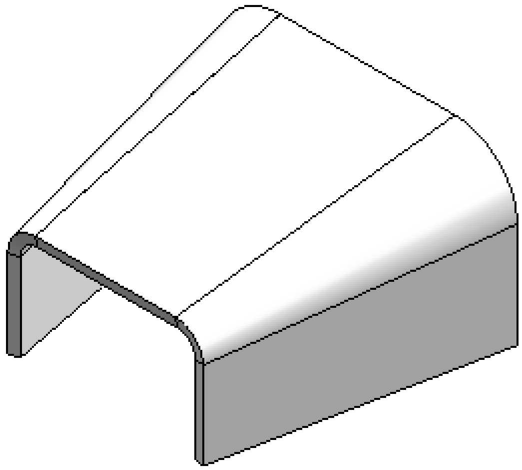
Notice that both the lofted flange solid and Sketch 2 are hidden for clarity in the previous illustration.

- ☐ Click Close Sketch.
- ☐ Click Finish to complete the sketch.
- ☐ If sketches are not listed in Pathfinder, right-click in the Pathfinder window, and on the shortcut menu, click PathFinder Display® Sketches.
- ☐ While still in the Select tool command, right-click Sketch 2 in Pathfinder.
- ☐ On the shortcut menu, click Edit Profile.
- ☐ Use the Fillet command to place 15 mm fillets on the two vertices.
- ☐ Add a radial dimension to one of the fillets.
- ☐ Add an equal relationship to both fillets as shown.



- ☐ Notice that the lofted flange solid is hidden for clarity.
- ☐ Click Close Sketch.

☐ Click Finish to complete the sketch.

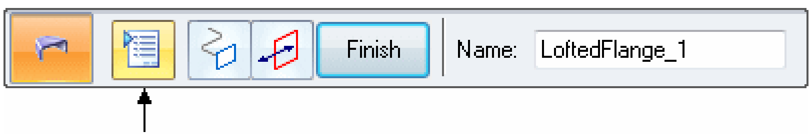


☐ Notice that with different-sized fillets on the vertices of the two profiles, the resulting bends are now conical.

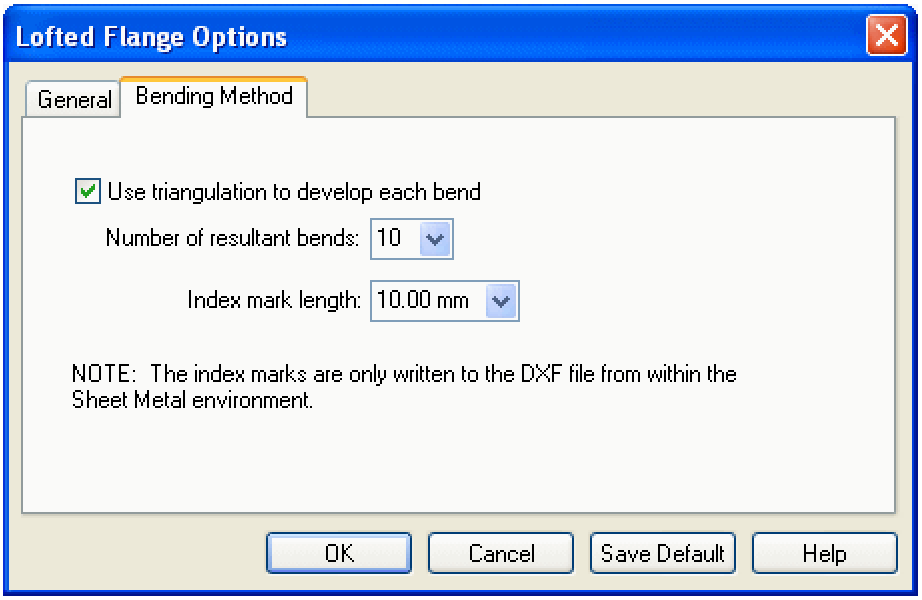
Step 4: Divide the bends. This will produce marks on the metal indicating incremental bend lines within a bend and add index notches. Typical uses of this capability are for modeling and manufacturing transitions such as square to round.

☐ Right-click the lofted flange feature and click Edit Definition.

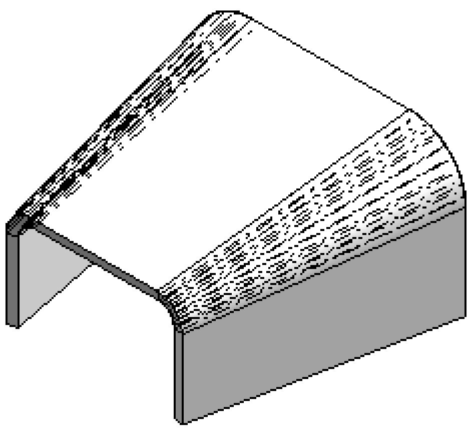
☐ On the command bar, click the Lofted Flange options button.



☐ On the Lofted Flange Options form, click the Bending method tab. Check the "Use triangulation to develop each bend" option. Set 10 as the number of resultant bends and type 10 mm for the Index mark length. Click OK.



☐ The divide lines appear on the bends. Click Finish.



☐ Save the file.

☐ This completes the activity.

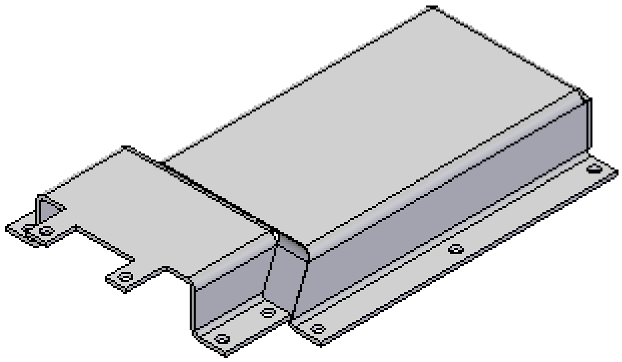
Activity summary

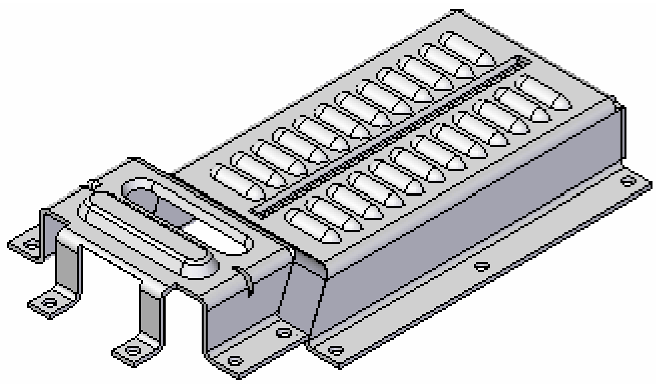
In this activity, you learned how to use sketches to identify geometry to be included in defining the ends of a lofted sheet metal flange. When the sketches were modified the geometry adjusted accordingly.

Activity: Constructing ordered sheet metal features

Overview

In this activity, you will construct the sheet metal features covered in this lesson. To help you work more quickly, you will work with a file that already has sketches drawn for each feature.



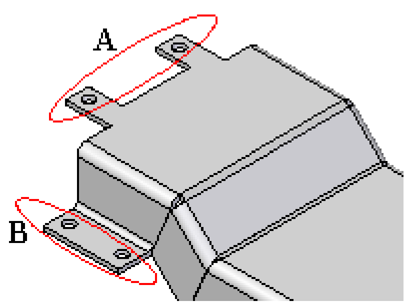


After completing this activity, you will be able to create all of the features shown. Each sketch is listed in PathFinder.

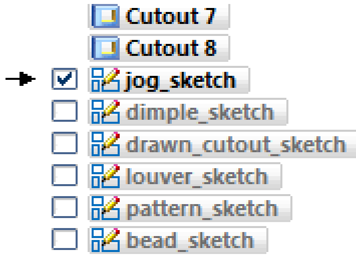
Activity

Step 1: Open *deformation_features.psm*.

Step 2: Create a jog feature using one of the sketches provided. Adjust a pair of mounting holes (A) to be planar with the pair of mounting holes (B).



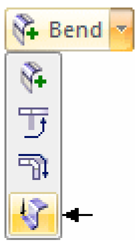
- [ ] In Pathfinder, click on the toggle box for the sketch named **jog_sketch**.



Note

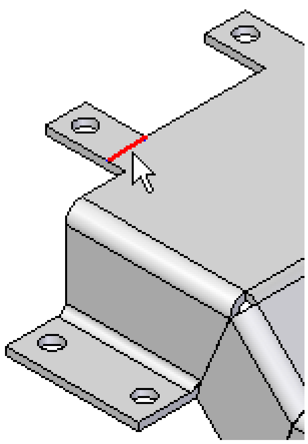
You can hide and show features on Pathfinder by selecting the corresponding toggle box.

- [ ] On the Home tab® Sheet Metal group, on the Bend drop list, choose the Jog command.

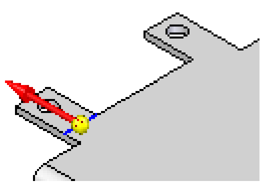


- [ ] On the command bar, click the Select from Sketch option.

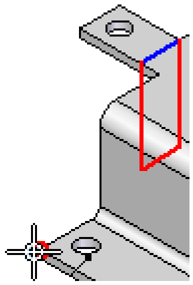
- [ ] Select the sketch and click the Accept button.



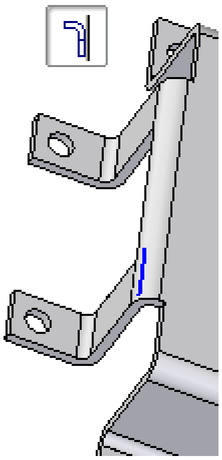
- [ ] Click on the side of the profile shown.



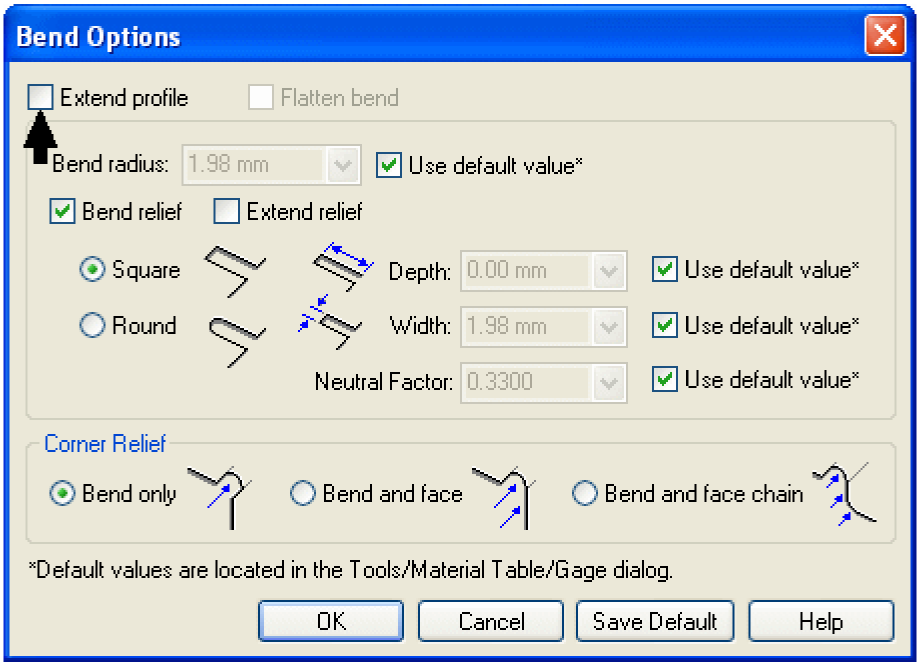
☐ To define the distance, select the keypoint shown (top vertex). This will ensure that the top face of the mounting holes being jogged are planar to the top face of these mounting holes.



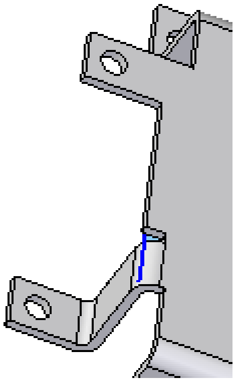
☐ The result shown is with the default setting of “material inside the profile”. Also notice that both mounting holes were included in the jog feature. This is because a Bend option was set to Extend profile. The profile was extended across the entire face.



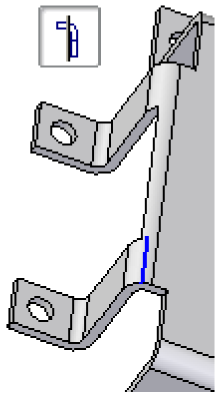
☐ On the command bar, click the Options button. Uncheck the Extend profile option and then click OK.



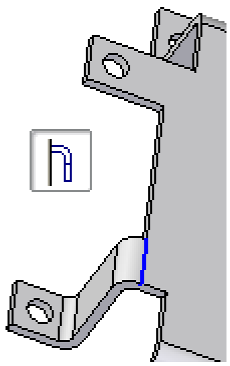
☐ With the Extend profile option turned off, the jog only includes the region of the profile.



☐ Turn on the Extend profile option. On the command bar, click the "Material Outside" option.



☐ On the command bar, click the "Bend Outside" option. Notice that this option only jogs the region defined by the profile. The profile cannot be extended in this particular case.

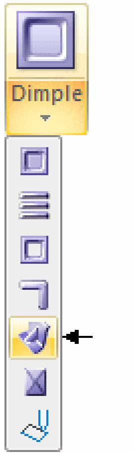


☐ Set the "Material Inside" option and click Finish.

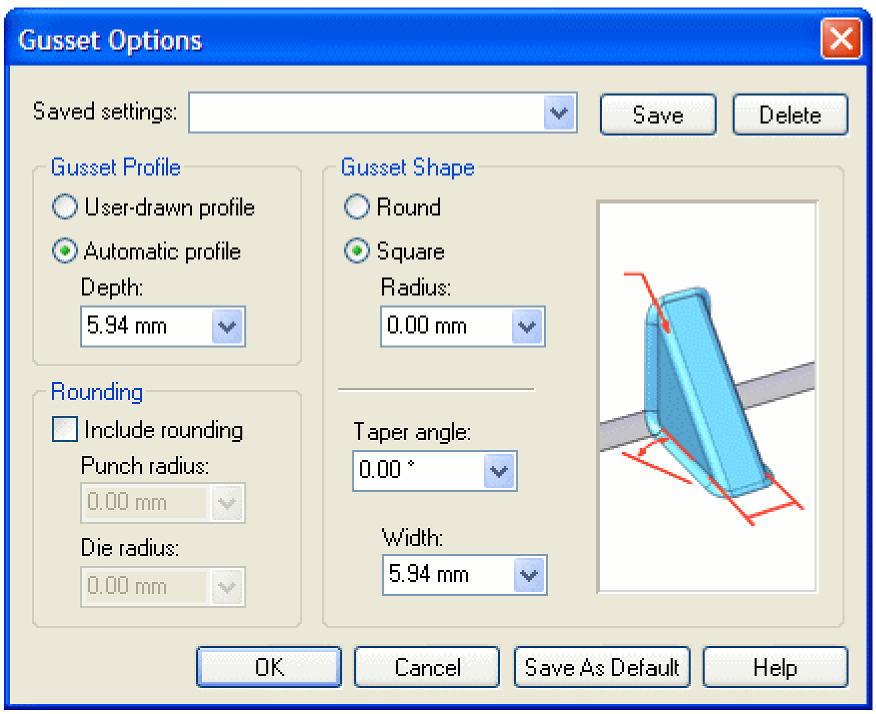
☐ Hide the sketch named **jog_sketch**.

Step 3: Create gussets using both automatic and user-defined methods.

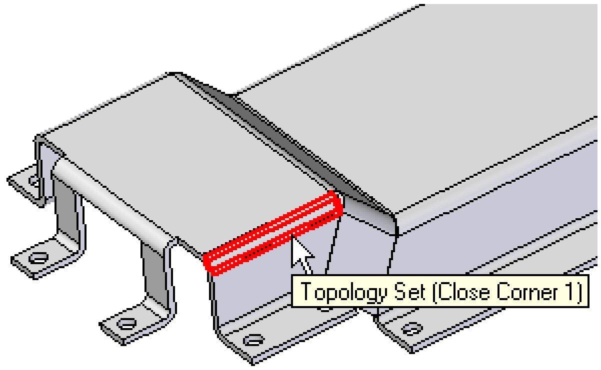
☐ On the Home tab® Sheet Metal group, on the Dimple drop list, choose the Gusset command.



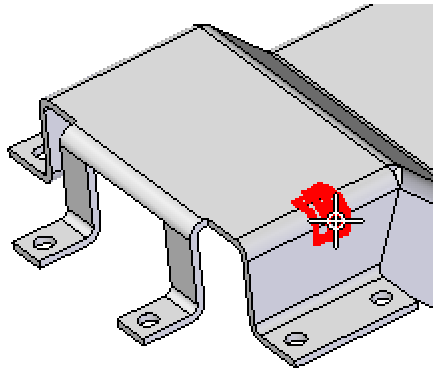
☐ Select the Gusset Options button.



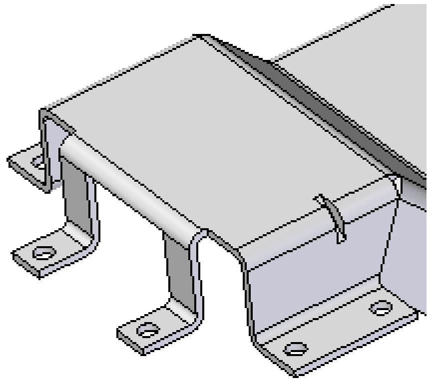
☐ In this particular case, the Automatic profile option is set. Solid Edge will create the profile according to the Depth value. Click OK to accept the default depth value for now. Select the bend shown.



☐ Select a location for the gusset. In this case you will drag it along the bend. However you can enter a distance value.

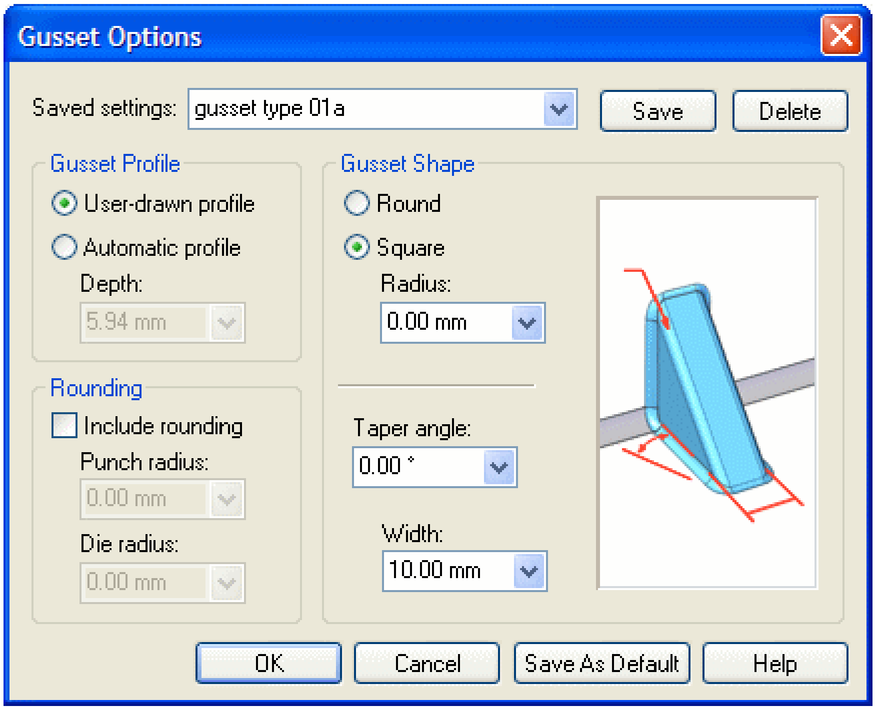


☐ Click Finish.



Step 4: Create a circular-shaped gusset with a User-drawn profile.

☐ On the command bar, select the Gusset Options button and change the Gusset Profile to User-drawn profile. You can save this set of parameters by naming it in the Saved settings field as shown. Enter a Width of 10 mm, then click OK.

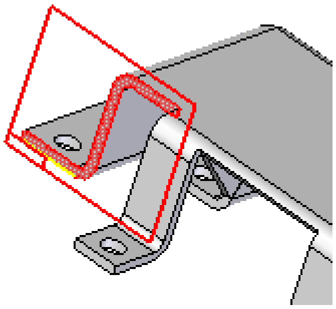


Note that the command bar changes for the User-drawn profile type.

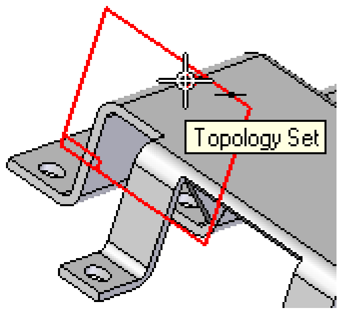
Gusset definition steps

1. Plane selection for drawing the profile
2. Profile step
3. Gusset direction step
4. Gusset side step (user must select this step)

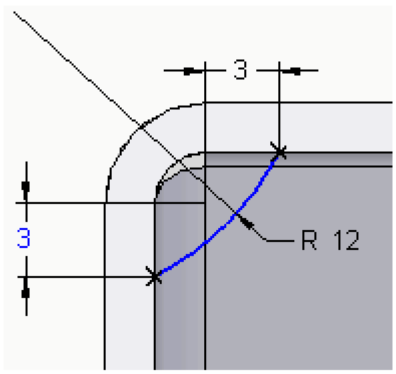
☐ In the Profile Step, you will construct an arc which will represent the final cross-sectional shape of the gusset. First you will be prompted for a sketch plane creation method. Use Parallel Plane option and select the face shown.



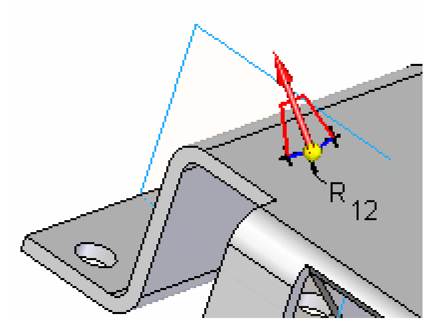
☐ You can drag the plane along the bend, enter a value in the distance field or use a keypoint. Select the midpoint of the bend as shown.



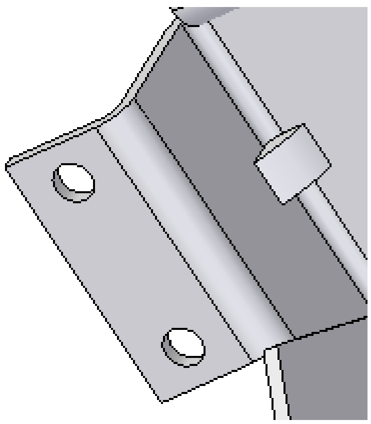
☐ You are now in the profile step. Sketch the arc shown and then click Close Sketch.



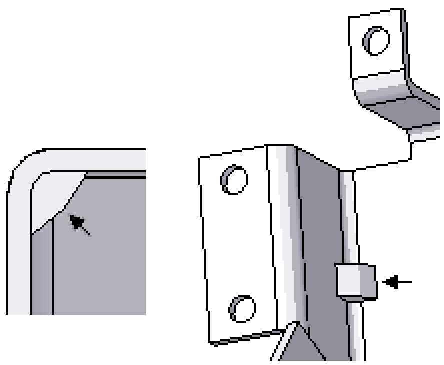
☐ You are now in the direction step. Select the direction shown towards the bend.



☐ The side step defaults to a symmetric extent about the profile plane. If you want to control the side direction, you must click the side step. Otherwise, the side step is skipped and the gusset definition is complete. We will take the default. Click Finish.



Shown are two different perspectives of the result.

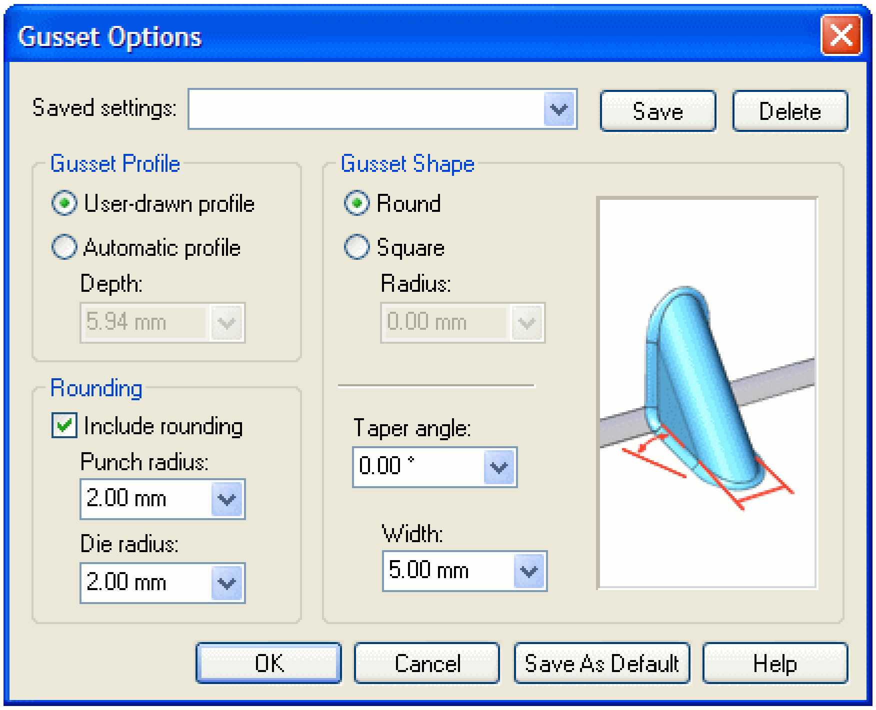


Step 5: Change gusset options for the gusset just created and observe the results.

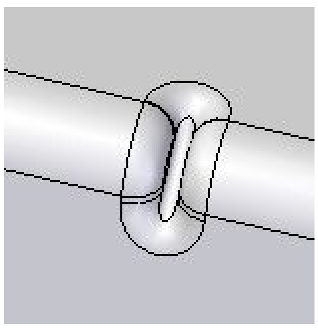
☐ Click the Select tool and then click the gusset feature.

☐ Click Edit Definition.

☐ Click the Gusset options button. Set the options shown.

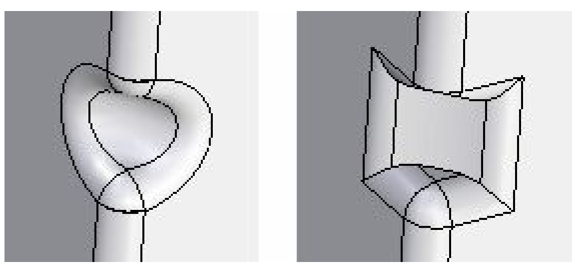


☐ Click OK and view the changes to the gusset.



Note

Looking at the underside of the gusset, you can clearly see the difference between the square versus round shape.



☐ Click Finish.

☐ As gussets are placed, you will see them in the Pathfinder as features.

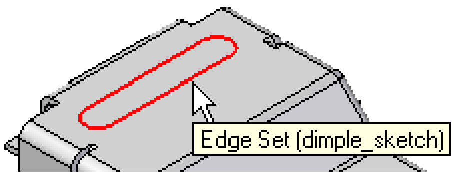
Step 6: Create a dimple feature using a closed profile.

☐ In Pathfinder, show the sketch named **dimple_sketch**.

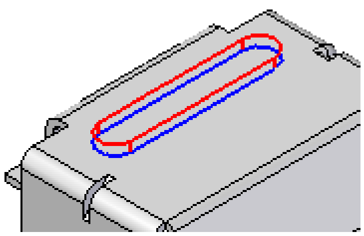
☐ On the Home tab® Sheet Metal group, choose the Dimple command .

☐ Click the Select from Sketch option.

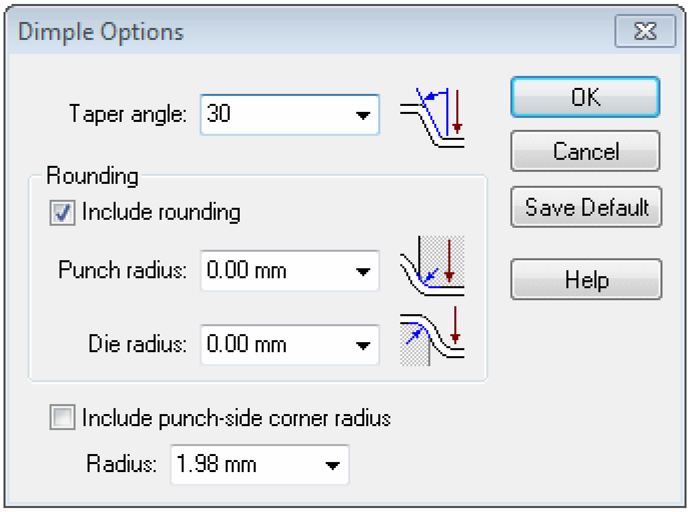
☐ Select the sketch shown and click the Accept button.



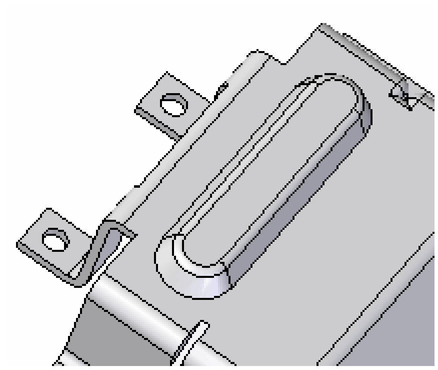
☐ On the command bar, type 3 in the Distance field and then press the **Enter** key. Position the cursor above the profile and click to define the direction.



☐ Click the Options button. On the Dimple Options form, type 30 in the Taper angle field and then click OK.



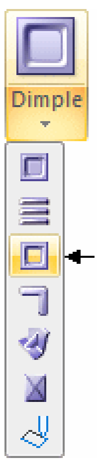
☐ Click Finish and then hide the sketch named **dimple_sketch**.



Step 7: Create a drawn cutout feature using a closed profile.

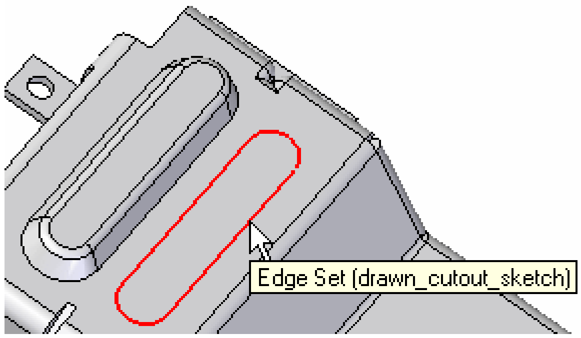
☐ Show the sketch named **drawn_cutout_sketch**.

☐ On the Home tab® Sheet Metal group, on the Dimple drop list, choose the Drawn Cutout command.

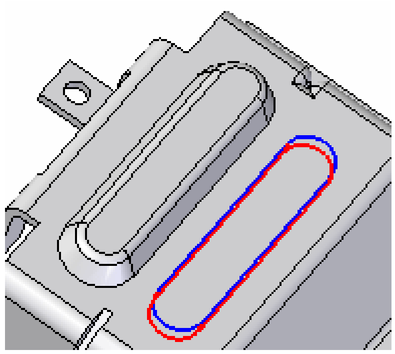


☐ Click the Select from Sketch option.

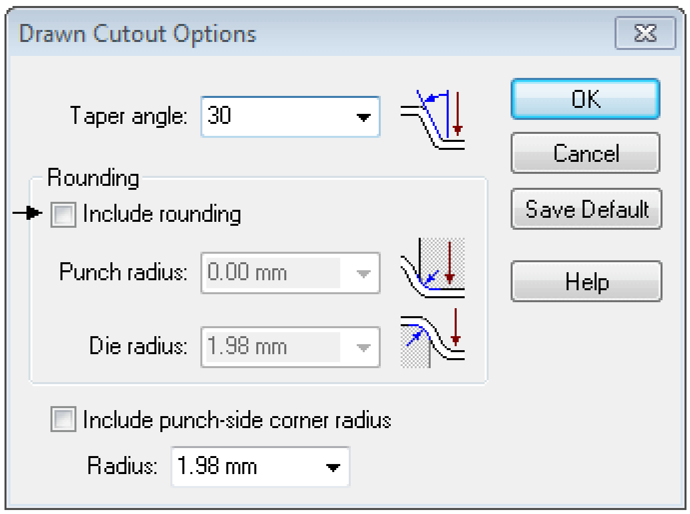
☐ Select the sketch shown, and click the Accept button.



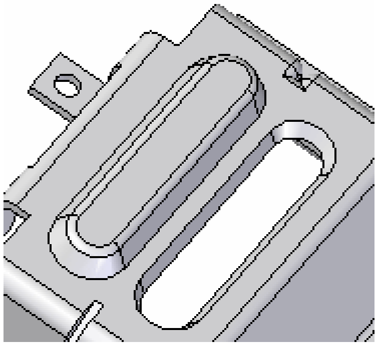
☐ Type 3 in the Distance field and press the **Enter** key. Position the cursor below the profile and click to define the direction.

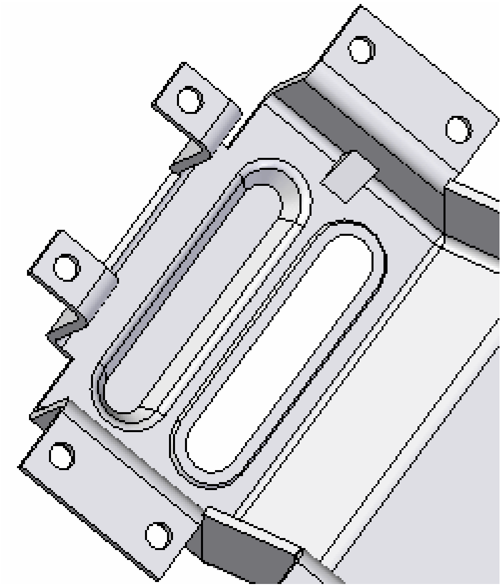


☐ Click the Options button. On the Drawn Cutout Options form, type 30 in the Taper angle field and uncheck the Include rounding option. Click OK.



☐ Click Finish and hide the sketch named **drawn_cutout_sketch**.

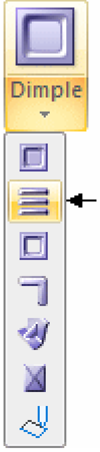




Step 8: Create a louver feature.

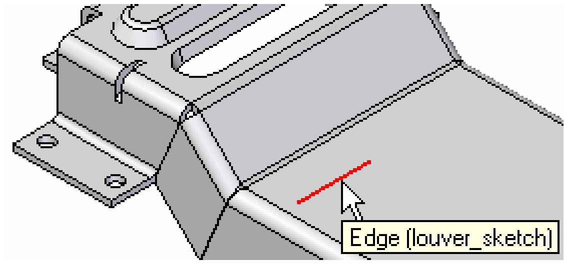
☐ Show the sketch named **louver_sketch**.

☐ On the Home tab® Sheet Metal group, on the Dimple drop list, choose the Louver command.

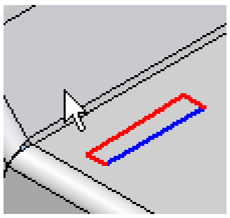


☐ Click the Select from Sketch option.

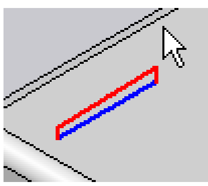
☐ Select the sketch shown and click the Accept button.



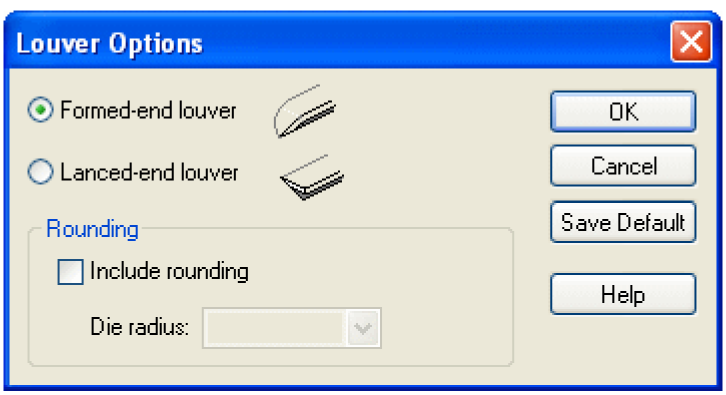
☐ Type 5 in distance field and press the **Enter** key. This defines the depth of the louver. Position cursor as shown and click to define the direction.



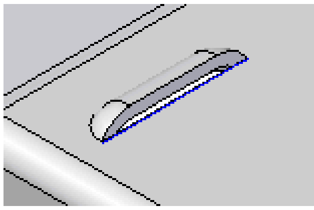
☐ Type 3 in distance field and press the **Enter** key. This defines the height of the louver. Position cursor as shown and click to define the direction.



☐ Click the Options button and make sure the options are set as shown. Click OK.



- ☐ Click Finish.

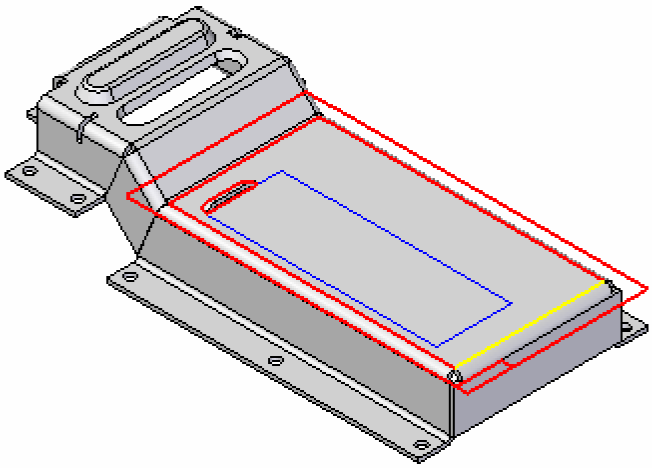


- ☐ Hide the sketch named **louver_sketch**.

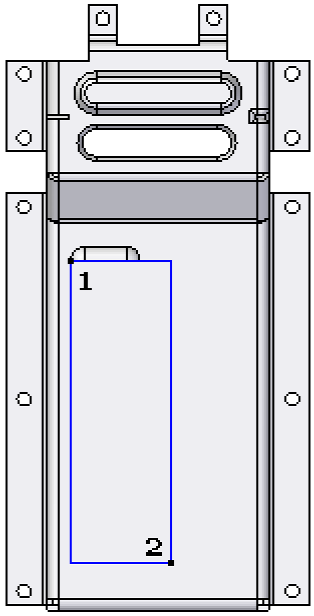
Step 9: Create a pattern of the louver feature.

- ☐ Show the sketch named **pattern_sketch**.
- ☐ On the Home tab® Pattern group, choose the Pattern command .

- ☐ Select the louver feature and then click the Accept button.
- ☐ Select the plane shown.

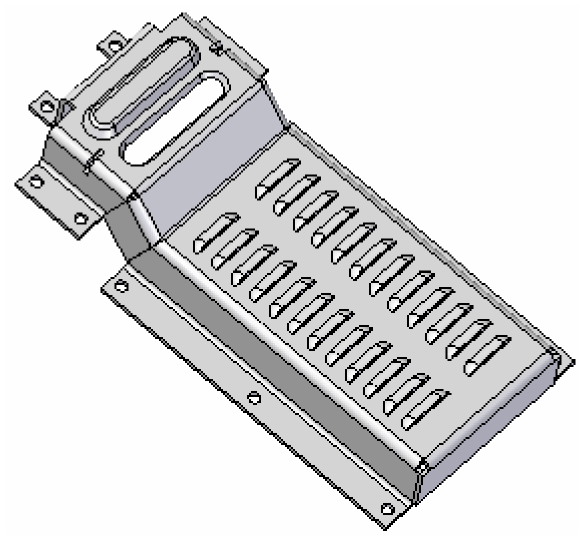


☐ Use the two points shown to define the pattern profile.



☐ On the command bar, set the pattern options as shown and then click Close Sketch.

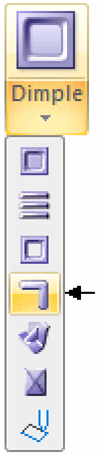
☐ Click Finish and hide the sketch named **pattern_sketch**.



Step 10: Create a bead feature.

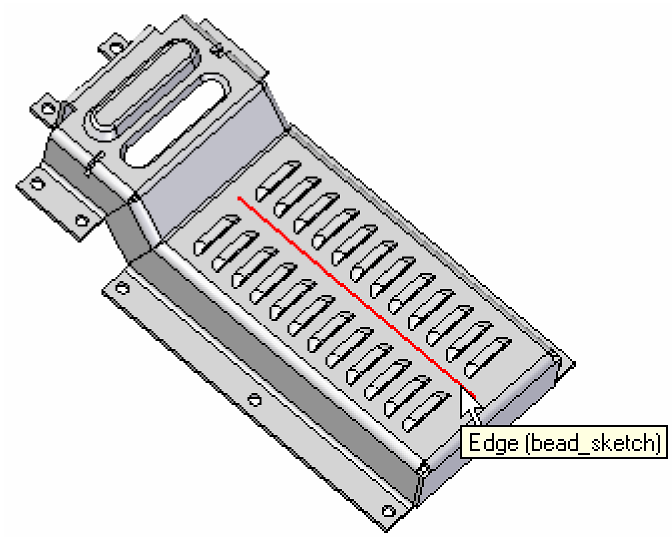
☐ Show the sketch named **bead_sketch**.

☐ On the Home tab® Sheet Metal group, on the Dimple drop list, choose the Bead command.

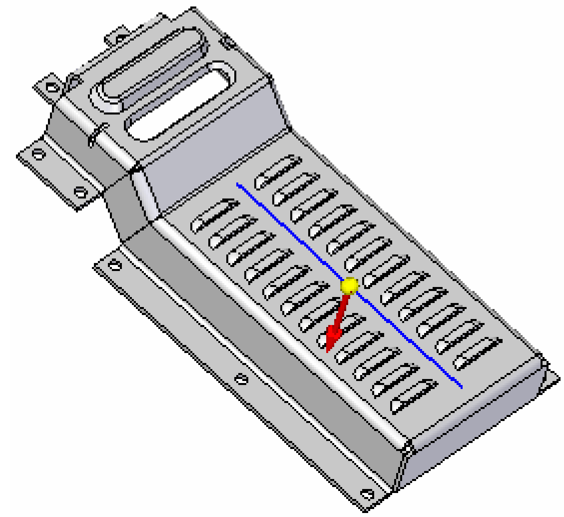


☐ Click the Select from Sketch option.

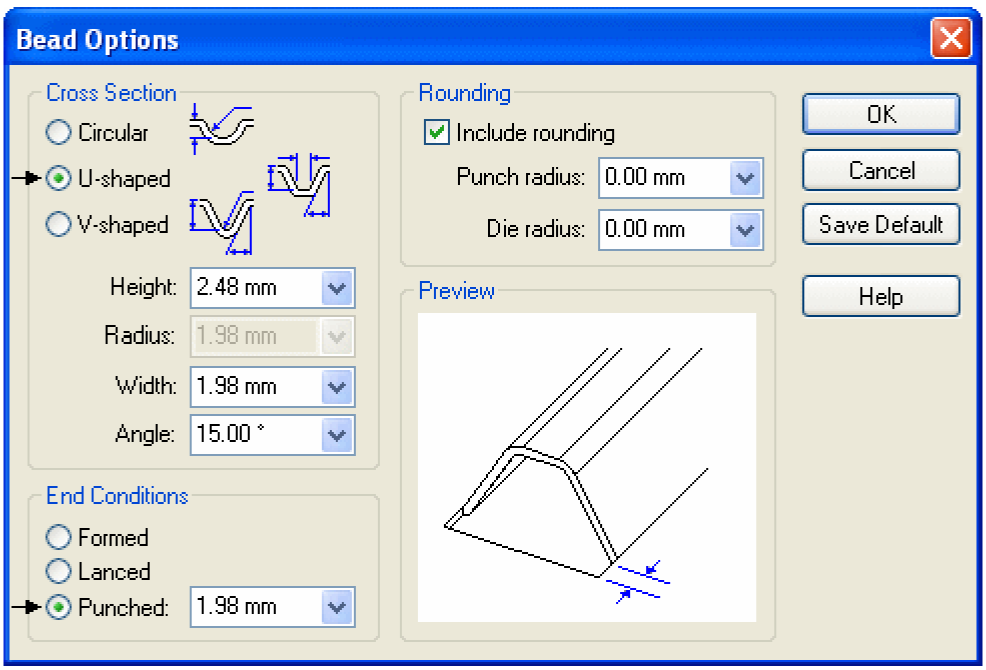
☐ Select the sketch edge shown and click the Accept button.



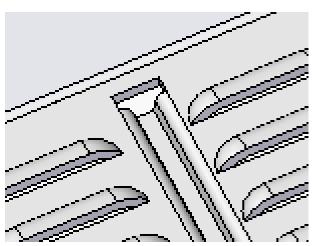
☐ For the side step, position the cursor so that the arrow points as shown, and click.



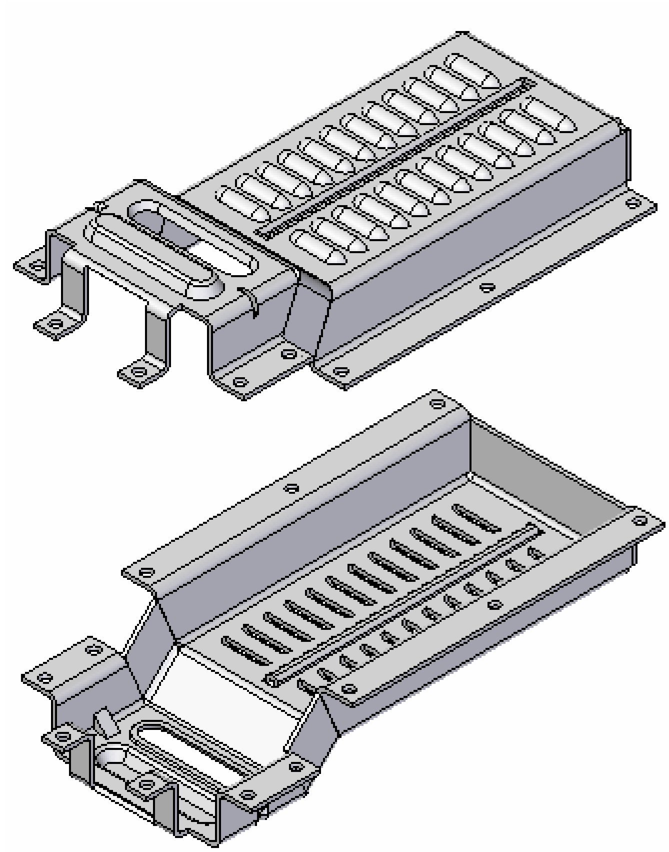
☐ Click the Options button. Set the options as shown and click OK.



☐ Click the Finish button. The bead is complete.



Step 11: Save and close the file. This completes the activity.



Activity summary

In this activity you used sketch geometry to exercise a variety of sheet metal feature commands. You learned the ease of placing jog, dimple, drawn cutout, louver, bead and gusset feature. There are numerous options available with each of these commands. Be sure to experiment with these options to better understand how these commands can be used to produce the required result.

Activity: Designing a brake cover

Overview

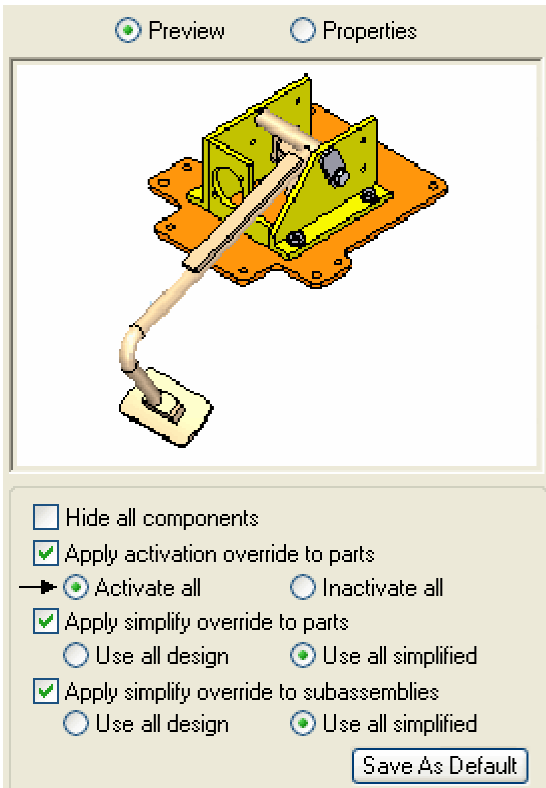
In this activity, you will build a sheet metal cover around a brake assembly. The activity demonstrates the process of designing a sheet metal part in the context of an assembly. The exercise uses the following commands: contour flange, miter option, tab, jog, normal cutout, mirror, break corner and flange.

Note

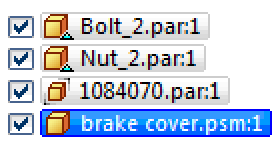
The files needed for this exercise are located in the brake folder in the working folder for this class.

Activity

Step 1: Open *brake.asm*. On the Open File dialog box, click Activate all.



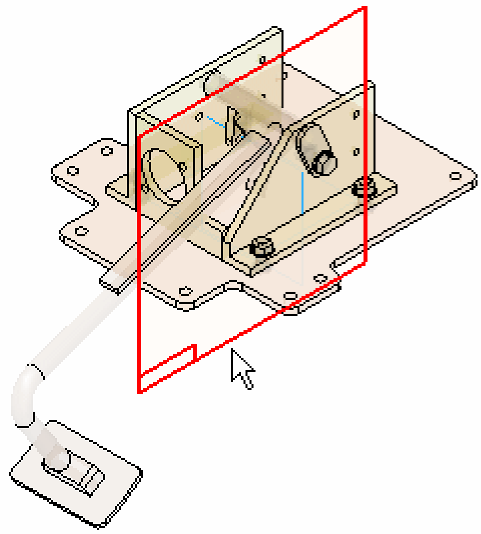
Step 2: In Assembly PathFinder, double-click *brake cover.psm* to In-Place Activate the file.



Step 3: Model the brake assembly cover by first constructing a contour flange as a base feature and then add additional features to the base feature to complete the desired sheet metal part.

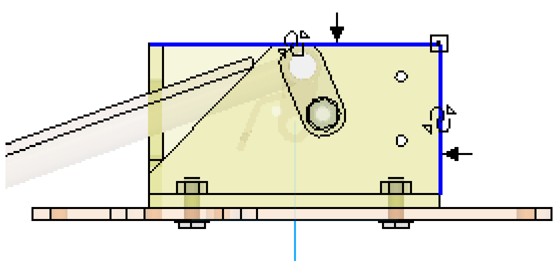
- ☐ Click the Contour Flange command .

- ☐ Select the coincident plane shown to construct the profile of the contour flange. Call this plane the “mirror” plane which is positioned in the center of the two vertical plates of the brake assembly.



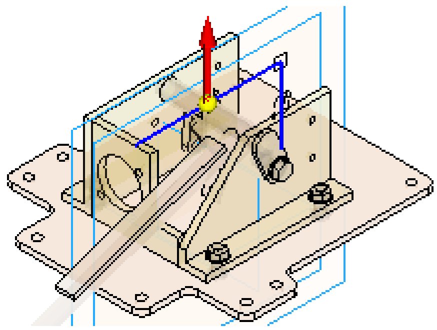
- ☐ Click the Include command  and click OK.

- ☐ Select the two edges shown. These edges will ensure that the contour flange will be a direct fit over the brake assembly plates.

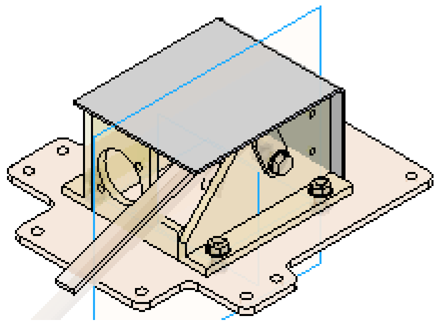


- ☐ Click Close Sketch.

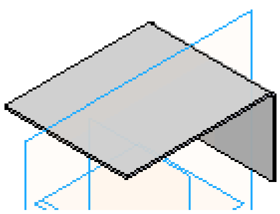
☐ Position arrow such that material is added to the outside of the profile.



☐ For the extent direction, click the Symmetric option and type 140 mm in the Distance field. Press the **Enter** key and then click Finish.



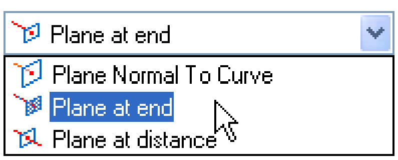
Step 4: Click View® Show® Hide Previous Level to turn off the display of the brake assembly.



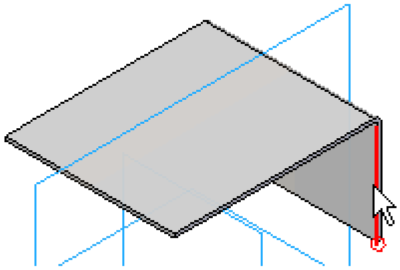
Step 5: Add another contour flange along an edge on one side of the part.

☐ Click the Contour Flange command.

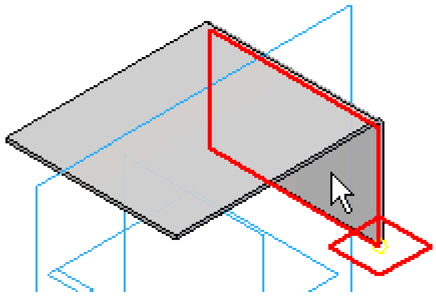
☐ Click the Plane at end option.



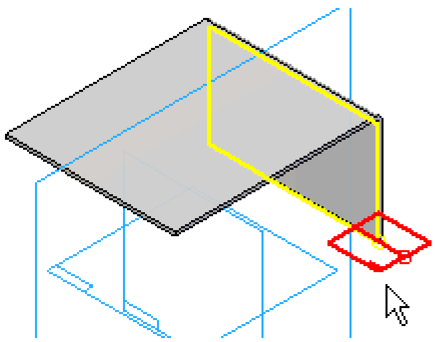
☐ Select the edge shown.



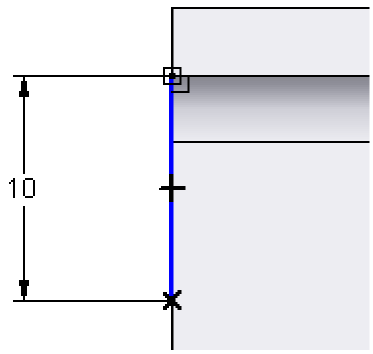
☐ Select the face shown.



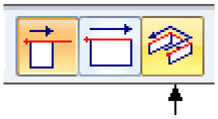
☐ Position the plane as shown by moving the cursor.



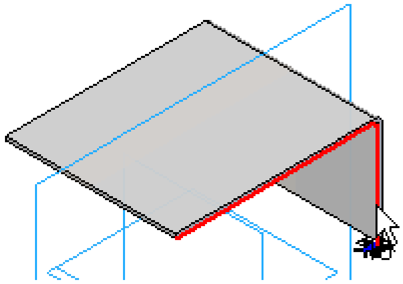
☐ Draw the profile as shown and dimension the line segment. Click Close Sketch.



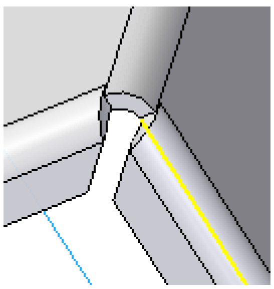
☐ Click the Chain option.



☐ Select the chain shown. Click the Accept button.

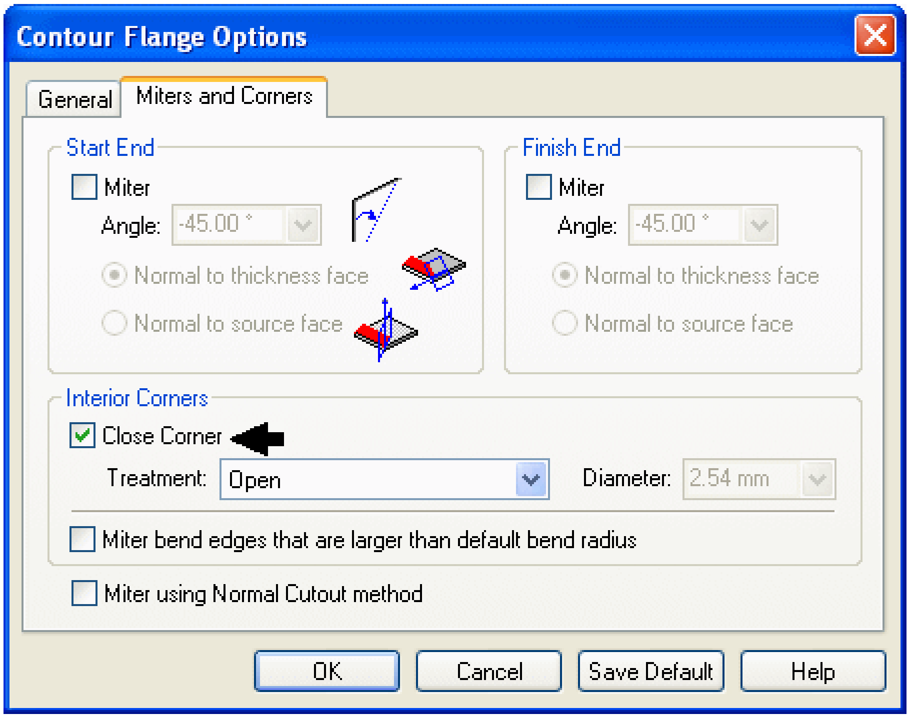


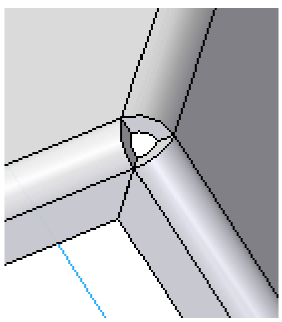
☐ Notice the resulting relief in the corner.



☐ Click the Contour Flange Options button.

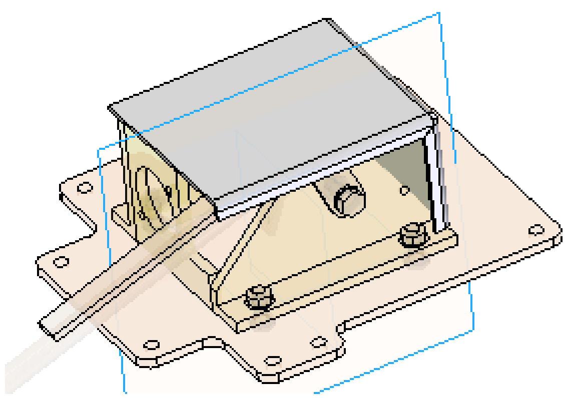
☐ Click the Miters and Corners tab. Click the Close Corner option and leave the Treatment set to Open. Click OK.



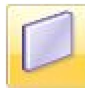


☐ Click Finish.

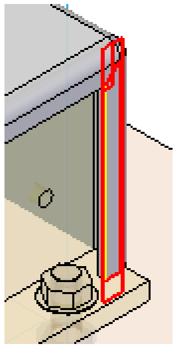
Step 6: Click View® Show® Hide Previous Level to turn on the display of the brake assembly.



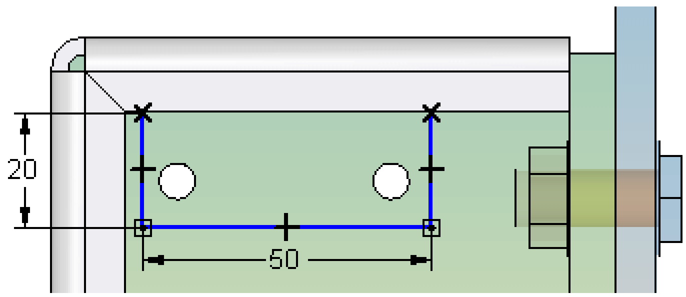
Step 7: Add a tab that will be used as a mounting flange for the cover.

- ☐ Click the Tab command .

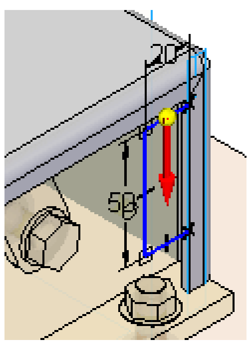
- ☐ Select the outer face on the contour flange as the plane to draw the tab profile on.



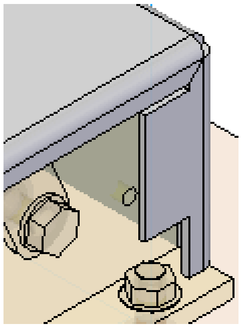
- ☐ Draw the tab profile as shown. The exact location is not important. Make sure the open profile is outside the two holes.



- ☐ Click Close Sketch and position the arrow as shown for the direction to add material.



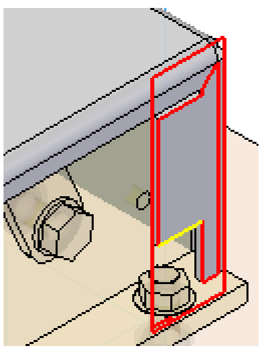
☐ Click Finish.



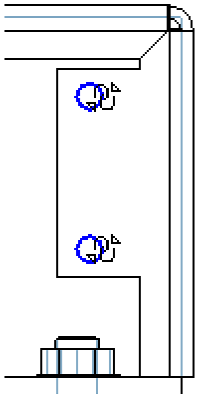
Step 8: Insert two holes in the tab to line up with the holes in brake assembly.

☐ Click the Normal Cutout command on the Hole drop list.

☐ Select the face shown.

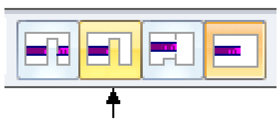


- ☐ Click the Include command and select the two holes as shown. You may need to switch to a Visible and Hidden Edges display to see the holes.

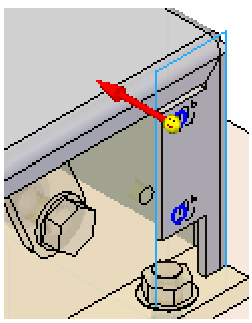


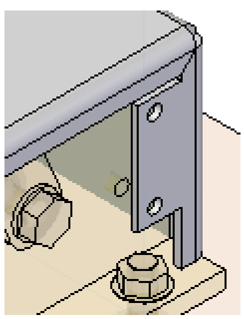
- ☐ Click Close Sketch.

- ☐ Click the Through Next extent option.



- ☐ Select the direction shown and then click Finish.

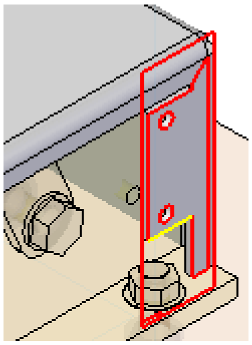




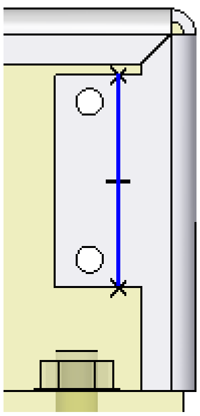
Step 9: The tab and holes on the sheet metal part need to be adjusted to lie on the mounting face of brake assembly. Use the Jog command to accomplish this.

☐ Click the Jog command.

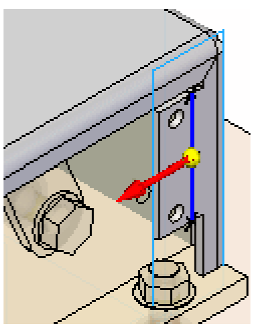
☐ Select the planar face as shown.



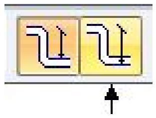
☐ Draw the line as shown (approximate location is OK) and click Close Sketch.



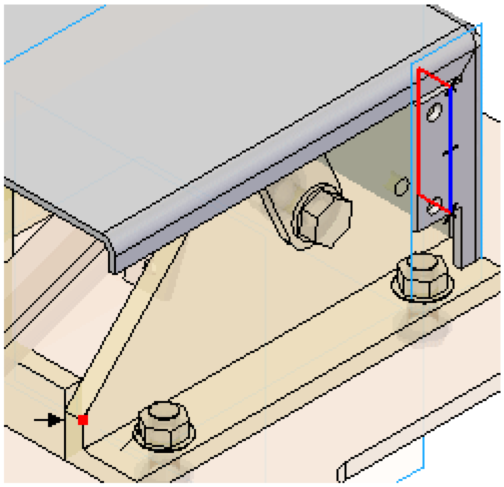
☐ Select side of profile to move as shown.

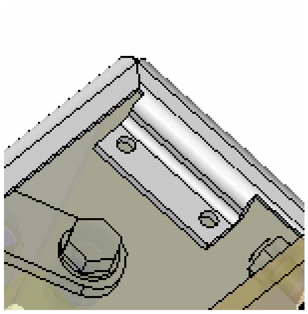


☐ Click the Full Dimension option.



☐ Select the keypoint as shown to define the distance of the jog and then click Finish.



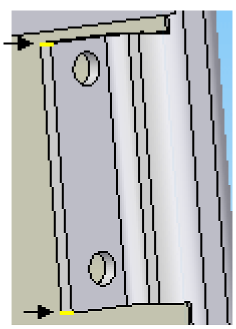


Step 10: Remove the sharp corners on the jog tab.

☐ On the Home tab® Sheet Metal group, choose the Break Corner command 

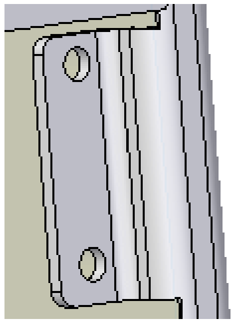
.

☐ Select the edges shown.



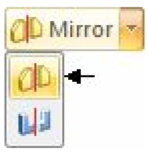
☐ Type 3 in the Break field and then click the Accept button.

☐ Click Preview and then Finish.

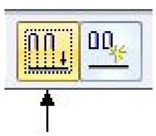


Step 11: The cover is a symmetric part. Mirror the features to the other side of the part.

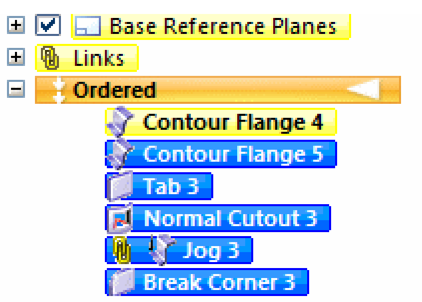
☐ Click the Mirror Copy Feature command.



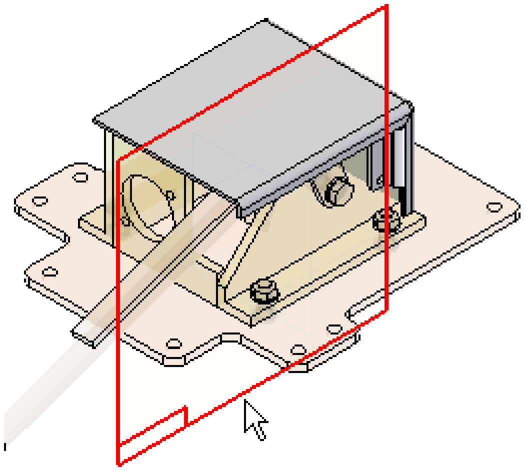
☐ Click the Smart option.

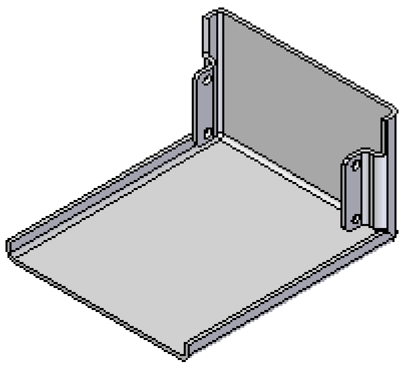


☐ In Pathfinder, select the five features as shown and then click the Accept button.



☐ Select the mirror plane as shown and then click Finish.

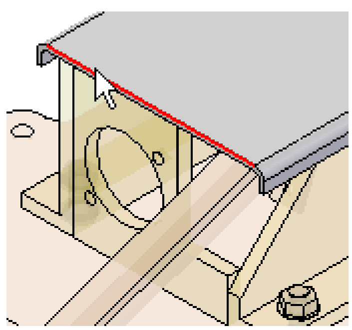




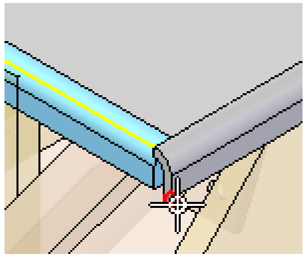
Step 12: Create one more flange and observe the bend options that are available.

☐ Click the Flange command .

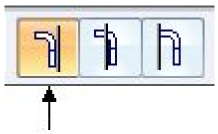
☐ Select the edge shown.

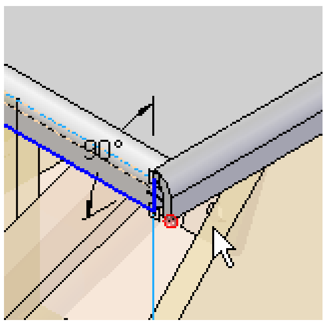


☐ Drag the flange downward and select the keypoint as shown to guarantee that the length will be the same as the other flanges.

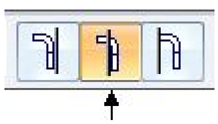


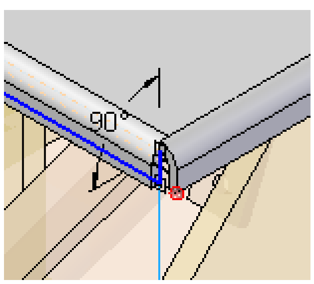
☐ The Material Inside option is selected by default. Notice the result of the flange being created on the inside of the profile.



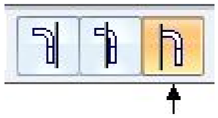


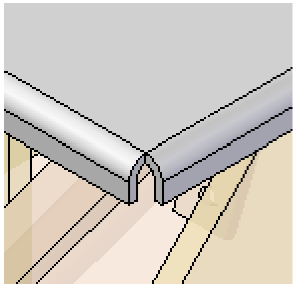
☐ Select the Material Outside option. Notice the result of the flange being created outside of the profile.





☐ Click the Bend Outside option. This option creates the bend and the flange outside the profile.

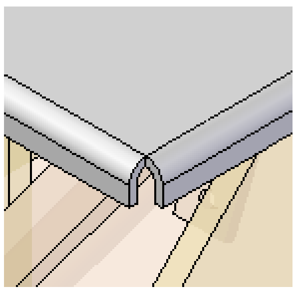




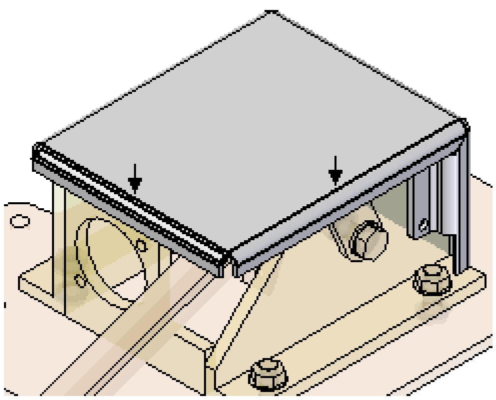
☐ Click Finish.

Step 13: Close the two corners of the three flanges.

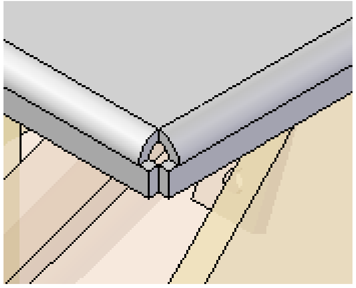
☐ Choose the Close 2-Bend Corner command.



☐ Select the two bends shown and set the options shown on the command bar.



☐ Click the Accept button and then click Finish.

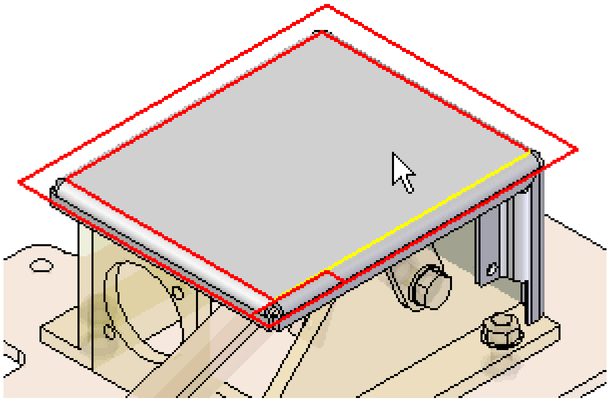


☐ Repeat the close corner step for the other two bends.

Step 14: Create a cutout to allow clearance for the movement of the brake pedal.

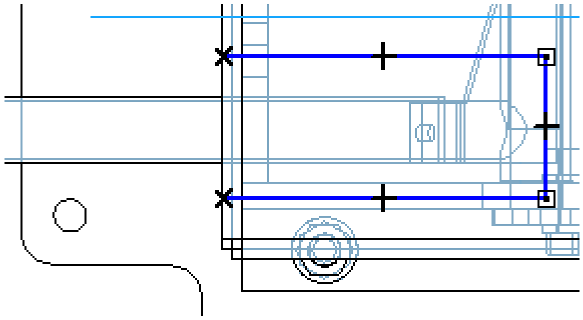
☐ Click the Normal Cutout command.

☐ Select the face shown.

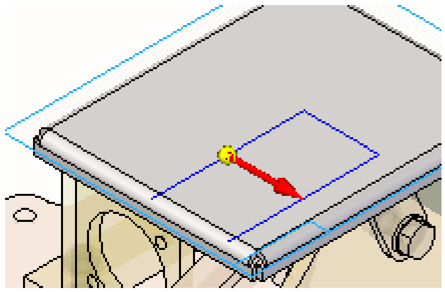


☐ Toggle the display to Visible and Hidden Edges.

☐ Draw an open profile as shown. No dimensions are needed.

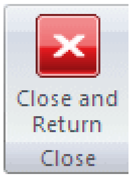


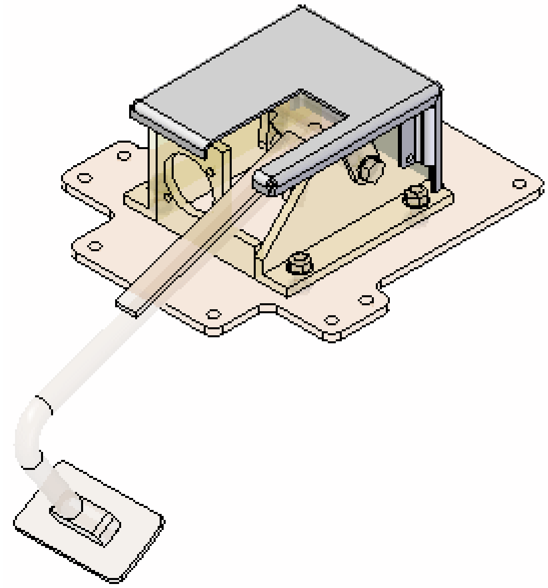
☐ Click Close Sketch and select the direction for material removal.



☐ Select the Through Next extent and position the arrow down. Click Finish.

Step 15: Click File® Close and Return. This will save the sheet metal part *brake cover.psm* and return to the brake assembly *brake.asm*.





Step 16: This completes the activity.

Activity summary

In this activity, you learned how to design a sheet metal part in the context of an assembly.