

SolidWorks® 2005

Sheet Metal and Weldments

SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA

© 1995-2004, SolidWorks Corporation

300 Baker Avenue
Concord, Massachusetts 01742 USA
All Rights Reserved

U.S. Patents 5,815,154; 6,219,049; 6,219,055;
6,603,486; and 6,611,725; and certain other foreign
patents, including EP 1,116,190 and JP 3,517,643.
U.S. and foreign patents pending.

SolidWorks Corporation is a Dassault Systemes S.A.
(Nasdaq:DASTY) company.

The information and the software discussed in this
document are subject to change without notice and
should not be considered commitments by
SolidWorks Corporation.

No material may be reproduced or transmitted in any
form or by any means, electronic or mechanical, for
any purpose without the express written permission
of SolidWorks Corporation.

The software discussed in this document is furnished
under a license and may be used or copied only in
accordance with the terms of this license. All
warranties given by SolidWorks Corporation as to
the software and documentation are set forth in the
SolidWorks Corporation License and Subscription
Service Agreement, and nothing stated in, or implied
by, this document or its contents shall be considered
or deemed a modification or amendment of such
warranties.

SolidWorks, PDMWorks, and 3D PartStream.NET,
and the eDrawings logo are registered trademarks of
SolidWorks Corporation.

SolidWorks 2005 is a product name of SolidWorks
Corporation.

COSMOSXpress, DWGEditor, eDrawings, Feature
Palette, PhotoWorks, and XchangeWorks are
trademarks, 3D ContentCentral is a service mark,
and FeatureManager is a jointly owned registered
trademark of SolidWorks Corporation.

COSMOS, COSMOSWorks, COSMOSMotion, and
COSMOSFloWorks are trademarks of Structural
Research and Analysis Corporation.

FeatureWorks is a registered trademark of
Geometric Software Solutions Co. Limited.

ACIS is a registered trademark of Spatial
Corporation.

GLOBEtrrotter and FLEXIm are registered
trademarks of Globetrotter Software, Inc.

Other brand or product names are trademarks or
registered trademarks of their respective holders.

COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY

U.S. Government Restricted Rights. Use,
duplication, or disclosure by the government is
subject to restrictions as set forth in FAR 52.227-19
(Commercial Computer Software - Restricted
Rights), DFARS 227.7202 (Commercial Computer
Software and Commercial Computer Software
Documentation), and in the license agreement, as
applicable.

Contractor/Manufacturer:
SolidWorks Corporation, 300 Baker Avenue,
Concord, Massachusetts 01742 USA

Portions of this software are copyrighted by and are
the property of Electronic Data Systems Corporation
or its subsidiaries

Portions of this software © 1988, 2000 Aladdin
Enterprises.

Portions of this software © 1996, 2001 Artifex
Software, Inc.

Portions of this software © 2001 artofcode LLC.

Portions of this software © 2004 Bluebeam
Software, Inc.

Portions of this software © 1999, 2002-2004
ComponentOne

Portions of this software © 1990-2004 D-Cubed
Limited.

Portions of this product are distributed under license
from DC Micro Development, Copyright © 1994-
2002 DC Micro Development, Inc. All rights
reserved

Portions © eHelp Corporation. All rights reserved.
Portions of this software © 1998-2004 Geometric
Software Solutions Co. Limited.

Portions of this software © 1986-2004 mental
images GmbH & Co. KG

Portions of this software © 1996 Microsoft
Corporation. All Rights Reserved.

Portions of this software © 2004 Priware Limited

Portions of this software © 2001, SIMULOG.

Portions of this software © 1995-2004 Spatial
Corporation.

Portions of this software © 2003-2004, Structural
Research & Analysis Corp.

Portions of this software © 1997-2004 Tech Soft
America.

Portions of this software © 1999-2004 Viewpoint
Corporation.

Portions of this software © 1994-2004, Visual
Kinematics, Inc.

This software is based in part on the work of the
Independent JPEG group.

All Rights Reserved

Document Number: PMT0044-ENG

Table of Contents

Introduction

About This Course	3
Prerequisites	3
Course Design Philosophy	3
Using this Book	3
About the CD	4
Windows® 2000	4
Conventions Used in this Book	4

Lesson 1:

Modeling Sheet Metal Parts

Sheet Metal Methods	9
Stages in the Process	9
Sheet Metal Toolbar	9
Designing with Sheet Metal Features	10
Flanges	10
Base Flange	11
Sheet Metal Features	13
Sheet-Metal Feature	13
Flat-Pattern Feature	14
Miter Flange	14
Edge Flanges	17
Adding a Tab	21
Flat Pattern	22
Flat Pattern Options	23
Cuts	25
Cuts in the Folded Model	25
Unfold and Fold	26

Sheet Metal Forming Tools	29
Standard Tools	29
Using a Standard Forming Tool	29
How They Work	30
Copy and Modify a Forming Tool	31
Creating a Custom Forming Tool	34
Bend Angles	38
Sheet Metal Parts in Drawings	38
Closed Corners and Miter Flanges	40
Individual Edges with a Miter Flange	41
Closing a Corner	42
Designing in Flat	43
Existing Rounds	47
Base Flange	47
Using Symmetry	47
Manual Relief Cut	48
Break Corner	50
Jog Feature	50
Fix Projected Length	52
Lofted Bends	54
Bend Deviation	55
Exercise 1: Sheet Metal Bends	57
Exercise 2: Sheet Metal Relief	58
Exercise 3: Sheet Metal from Flat	60
Exercise 4: Sheet Metal Flanges and Bends	63
Exercise 5: Assorted Framing Hangers	65

Lesson 2:

Converting Parts to Sheet Metal

Sheet Metal Topics	71
Stages in the Process	71
Converting Legacy Sheet Metal Parts	71
Recommended Practices	72
Recognize Bends Method	73
Importing	73
Opening IGES Files	73
Document Template	74
Report Files	74
Using the Rip Feature	75
Adding Bends in Place of Sharp Corners	76
Sheet Metal Features	78
New Features	78
Switching Between States	79
Making Changes	79
Unrolling Cones and Cylinders	83
Process Plans	86
Bends	86

Configurations	86
Exercise 6: Importing and Converting	89
Exercise 7: Process Planning	92
Lesson 3:	
Modeling Sheet Metal In the Context of an Assembly	
Sheet Metal Topics	95
Top-Down Assembly Modeling	95
Stages in the Process	95
In-Context Sheet Metal Parts	96
Adding a New Part into an Assembly	96
Results of Insert, Component, New Part	97
Appearance During Part Editing	99
Color of the Part Being Edited	99
Building Edge Flanges In Context	100
Building Miter Flanges In Context	102
Additional Cut Features	104
Hems	107
Edit Assembly	108
Exercise 8: Sheet Metal in an Assembly	111
Exercise 9: Jogs and Hems	115
Exercise 10: U-Bolt In-Context	118
Exercise 11: Tube Brace	119
Weldments	
Weldments	125
Weldments Toolbar	125
Weldment Feature	125
Structural Members	126
Corner Treatments	127
Sketch Considerations	129
Trimming Structural Members	132
Adding Plates	133
Weld Beads	134
Using Symmetry	135
Gussets and End Caps	136
Gusset Profile and Thickness	136
Locating the Gusset	137
End Cap Parameters	137
Profile Sketches	140
Custom Properties	140
Working with Weldments	143
Sub-weldments	143
Non-structural Components	144
Saving Bodies as Separate Parts	144
Post-assembly Machining Operations	145
Managing the Cut List	145
Weld Beads	145

Balloons	145
Automatically Generating the Cut List.....	145
Custom Properties.....	146
List of Properties.....	147
Cut List Properties	148
Weldment Drawings	148
Drawing Views of Individual Bodies.....	149
Cut List Tables	151
Working with Pipes and Tubing	155
3D Sketching.....	155
Merge Arc Segment Bodies	160
Weldments and Sheet Metal in Assemblies	166
Sheet Metal or Structural Member?	172
Reusing In-context Parts.....	173
Exercise 12: Creating a Weldment	175
Exercise 13: Bent Tubing, Sheet Metal, and Assemblies.....	190

Introduction

About This Course

The goal of this course is to teach you how to build sheet metal parts using SolidWorks mechanical design automation software.

The sheet metal application in SolidWorks 2005 is so robust and feature rich that it is impractical to cover every minute detail and still have the course be a reasonable length. Therefore, the focus of this course is on the fundamental skills and concepts central to successfully building sheet metal parts. You should view the training course manual as a supplement to, not a replacement for, the system documentation and on-line help. Once you have developed a good foundation in basic skills, you can refer to the on-line help for information on less frequently used command options.

Prerequisites

Students attending this course are expected to have the following:

- Q Mechanical design experience.
- Q Completed the course *SolidWorks Essentials: Parts and Assemblies*.
- Q Experience with the Windows™ operating system.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. Rather than focus on individual features and functions, a process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a design task.

Using this Book

This training manual is intended to be used in a classroom environment under the guidance of an experienced SolidWorks instructor. It is not intended to be a self-paced tutorial. The examples and case studies are designed to be demonstrated “live” by the instructor.

Laboratory Exercises

Laboratory exercises give you the opportunity to apply and practice the material covered during the lecture/demonstration portion of the course. They are designed to represent typical design and modeling situations while being modest enough to be completed during class time. You should note that many students work at different paces. Therefore, we have included more lab exercises than you can reasonably expect to complete during the course. This ensures that even the fastest student will not run out of exercises.

A Note About Dimensions

The drawings and dimensions given in the lab exercises are not intended to reflect any particular drafting standard. In fact, sometimes dimensions are given in a fashion that would never be considered acceptable in industry. The reason for this is the labs are designed to encourage you to apply the information covered in class and to employ and reinforce certain techniques in modeling. As a result, the drawings and dimensions in the exercises are done in a way that compliments this objective.

About the CD

Bound inside the rear cover is a CD containing copies of the various files that are used throughout this course. They are organized by lesson number. The *Case Study* folder within each lesson contains the files your instructor uses while presenting the lessons. The *Exercises* folder contains any files that are required for doing the laboratory exercises.

Windows® 2000

The screen shots in this manual were made using SolidWorks 2005 running on Windows® 2000. If you are running on a different version of Windows, you may notice subtle differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Conventions Used in this Book

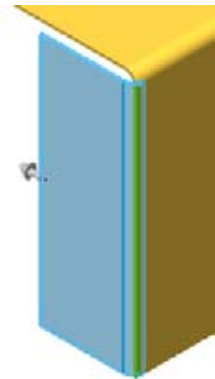
This manual uses the following typographic conventions:

Convention	Meaning
Bold Sans Serif	SolidWorks commands and options appear in this style. For example, Insert, Boss means choose the Boss option from the Insert menu.
Typewriter	Feature names and file names appear in this style. For example, <i>Sketch1</i> .
===== 17 Do this step =====	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

Use of Color

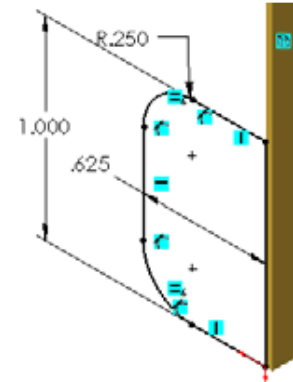
The SolidWorks 2005 user interface makes extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of SolidWorks 2005. To take maximum advantage of this, the training manuals are printed in full color.

Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the result of an operation in a different color, even though by default, the SolidWorks software would not display the results in that way.

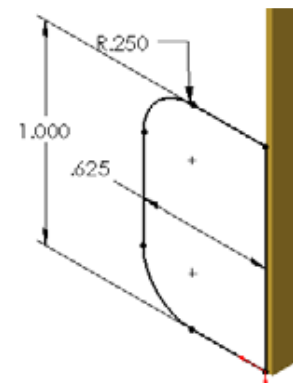


Displaying Sketch Relations

SolidWorks 2005 has the capability to automatically display graphic representations of sketch relations as shown in the illustration at the right. These can be turned on and off using the command **View, Sketch Relations**.



In most of the examples in this book, the graphic display of the sketch relations has been turned off.



Lesson 1

Modeling Sheet Metal Parts

Upon successful completion of this lesson, you will be able to:

- Q Create a sheet metal part using a base flange.
- Q Add the sheet metal specific flange features such as the edge and miter flanges.
- Q Close corners by extending faces.
- Q Use forming tools and the Design Library to create formed features such as ribs, louvers, and lances.
- Q Create your own custom forming tools.
- Q Design sheet metal parts in the flat and add bends using sketched bends.

Sheet Metal Methods

- Fundamentally, there are two approaches for building sheet metal parts:
- Q Modeling the part as a sheet metal part from the very beginning, starting with the very first feature — the base flange feature. This method takes advantage of all the functionality and specialized tools, commands, and options in the sheet metal application. This is the preferred method for almost all parts and that is what this lesson covers.
 - Q Transforming a part that was built conventionally into a sheet metal part so it can be flattened, and so that sheet metal-specific features can be applied to it. Converting an imported part into a sheet metal part is a typical example of when this approach makes sense. This method is covered in *Lesson 2: Converting Parts to Sheet Metal*.

Stages in the Process

Some key stages in the modeling process are given in the following list. Each of these topics comprises a section in the lesson.

- Q **Designing with Sheet Metal Features**
Features that are specifically sheet metal are used to create parts. These include several types of flanges, tabs, added bends and flattening tools.
- Q **Using Forming Tools**
Use forming tools to shape the sheet metal part. The tools can be used as is, modified, or created from scratch.
- Q **Sheet Metal Drawings**
Creating a drawing using a sheet metal part has some unique options.
- Q **Using Symmetry**
Symmetric models can be created as partial models and mirrored using **Mirror Bodies**.
- Q **Existing Rounds**
Sketches that contain rounds are processed by the base flange feature and converted to bends.
- Q **Designing in the Flat**
Sheet metal parts can be designed as a flat pattern and folded later.

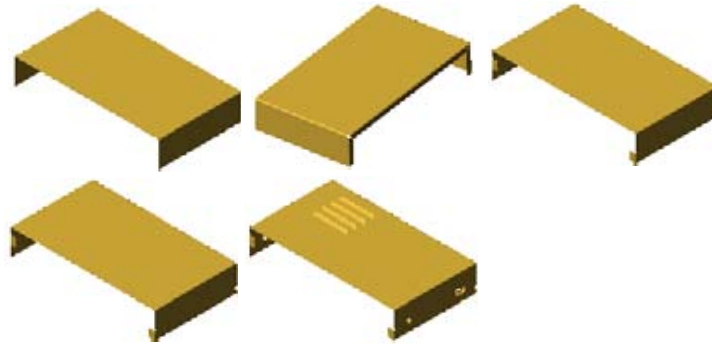
Sheet Metal Toolbar

The Sheet Metal toolbar contains shortcuts for all the sheet metal commands. These commands can also be accessed via the **Insert, Sheet Metal** menu.



Designing with Sheet Metal Features

Sheet Metal parts constitute a very specific type of solid model. They have the characteristics of being thin walled with rounded corners. The wall thickness is constant throughout the model and the bends are applied using a selected bend radius value. When relief is needed, it is added automatically. SolidWorks takes care of these requirements for you using specific sheet metal tools.

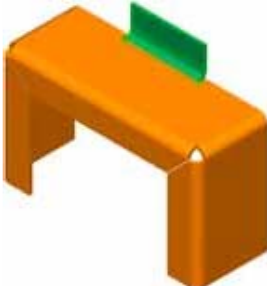
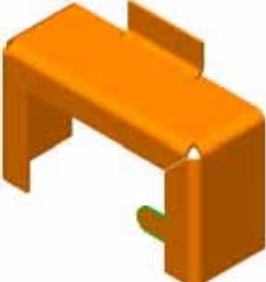


Flanges

The fundamental sheet metal feature is a flange. In this example, different types of flanges will be used to construct a sheet metal part. A description of the different flange types is offered below.

SolidWorks Sheet Metal has four different types of flanges that can be used to create parts. The flanges add material of predefined thickness in different ways.

<p>Base Flange</p>	<p>The Base Flange is used to create the base feature for the sheet metal part. It works similarly to the Extrude feature but it automatically adds bends using a specific Bend Radius.</p>	A 3D model of a base flange, which is a rectangular sheet metal part with two vertical flanges extending from the top edge. The flanges are bent at a 90-degree angle.
<p>Miter Flange</p>	<p>The Miter Flange is used to create flanges that connect at an angle to existing edges of the model. It has options to follow a tangent set of edges, create rips where needed or offset the ends.</p>	A 3D model of a miter flange, which is a rectangular sheet metal part with two vertical flanges extending from the top edge. The flanges are bent at an angle (miter) to the top edge of the part.

<p>Edge Flange</p>	<p>An Edge Flange adds material to create a single flange at an angle to an existing edge. The angle and the profile sketch can be modified.</p>	
<p>Tab (Boss Flange)</p>	<p>A Tab can be used to add a boss feature that is always the same thickness as the rest of the model. The tab is sketched on an existing face.</p>	

Procedure

The model to be created is a cover for a stereo receiver. It uses all four types of flanges, as well as cuts and forming tools.



Base Flange

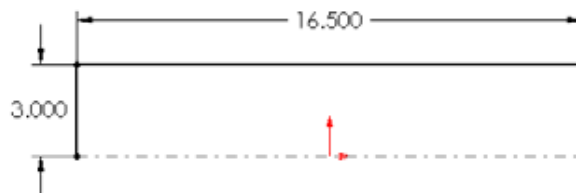
The **Base Flange** is the base feature of a sheet metal part. It creates the initial solid and defines the settings that are used for subsequent sheet metal features. The feature is a variation on the familiar Boss Extrude feature.

1 New part.

Create a new part with units **inches**. Name the part *Cover*.

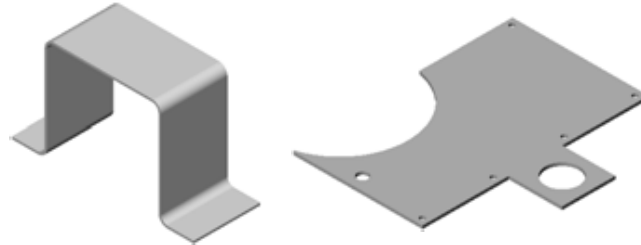
2 Sketch.

Sketch a rectangle using the **Front** plane. Change the bottom line to construction geometry. Relate the bottom line to the **Origin** using a **Midpoint** relation.




**Introducing:
Base Flange**

The **Base Flange** is used to create the base feature of a sheet metal part. The flange is created as an extrusion with a **Thickness** value and **Bend Radius**. An open contour is treated like a thin feature extrusion. A closed contour is treated as the outline of a flat plate. This is useful if you want to design sheet metal parts from a flat pattern.



Where to Find It

- Q From the menu choose: **Insert, Sheet Metal, Base Flange....**
- Q Or, click the  tool on the Sheet Metal toolbar.



3 Base Flange.

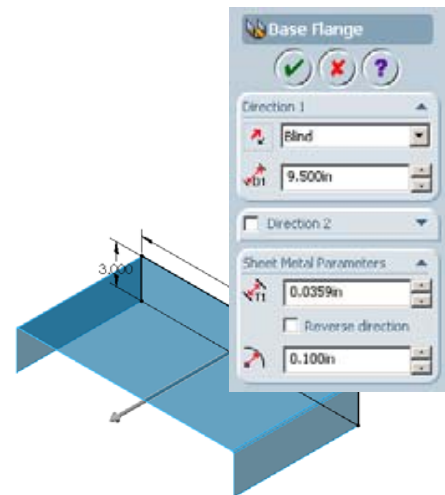
Click the **Base Flange** tool and change the settings.


- Q **End Condition = Blind**

- Q  (Depth) = 9.5”

- Q  (Thickness) = 0.0359”

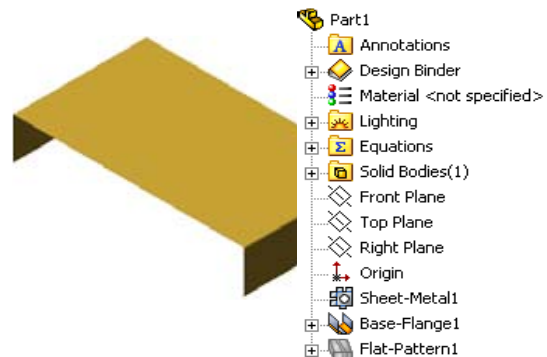
- Q  (Bend Radius) = 0.1”
- Verify that the thickness is applied to the outside of the sketch. If it is not, use **Reverse Direction**  to change it.



Click **OK**  to create the flange.

4 Completed base flange.

The base flange is listed in the FeatureManager as Base-Flange1. Note that two other features have also been added: Sheet-Metal1 and Flat-Pattern1. These features will be explained on the following pages.



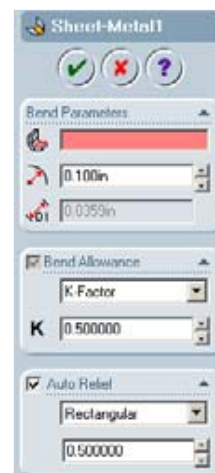
Sheet Metal Features

Sheet-Metal Feature



Certain specialized features are generated with the Base Flange that occur only in sheet metal parts. They are used for default settings and control of the part.

The Sheet-Metal feature contains the settings used for the part. This includes the default bend radius, the material thickness, how the bend allowance is calculated, and how automatic bend relief is generated. The information comes from the choices made in the base flange and other default settings. Use **Edit Feature** to change it.



Explaining the Bend Parameters

For **Bend Allowance** there are four options:

- Q **Bend Table** is a material specific (steel, aluminum, etc.) table that contains bend calculations based on thickness and bend radius. Bend tables are Excel files and have the *.xls file extension. Use **Insert, Sheet Metal, Bend Table, From File** or **New** to add one. They can also be chosen from the **Use bend table** pull-down in the Sheet-Metal dialog.

You can create your own **Bend Tables** using one of the sample files and Excel. Samples can be found under the installation directory and: lang\Bend Allowance and **Bend Deduction** methods.

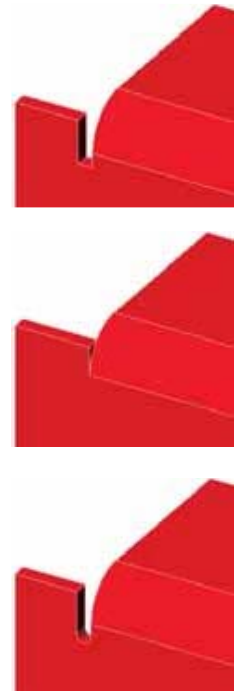
Radius	1/8	3/32	1/4	5/16	3/8
1/32	0.06	0.071	0.082	0.093	0.1
1/16	0.093	0.104	0.115	0.126	0.137
3/32	0.126	0.137	0.148	0.159	0.17
1/8	0.159	0.17	0.181	0.192	0.203
3/16	0.192	0.203	0.214	0.225	0.236
1/4	0.225	0.236	0.247	0.258	0.269
5/16	0.258	0.269	0.28	0.291	0.302
3/8	0.28	0.291	0.302	0.313	0.324
7/16	0.313	0.324	0.335	0.346	0.357
1/2	0.346	0.357	0.368	0.379	0.39
5/8	0.379	0.39	0.401	0.412	0.423
3/4	0.401	0.412	0.423	0.434	0.445
7/8	0.434	0.445	0.456	0.467	0.478
1	0.467	0.478	0.489	0.5	0.511

- Q **K-Factor** is a constant used in the bend calculations. It is the ratio that represents the location of the neutral sheet, measured from the inside, with respect to the thickness of the sheet metal.
- Q **Bend Allowance** and **Bend Deduction** allow you to enter a value based on your experience and shop practices.

Examining the Auto Relief Options

For **Auto Relief** there are three options:

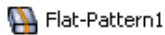
- Q **Rectangular** (top picture) creates a rectangular shaped cut around the edges that require bend relief.
- Q **Tear** (middle picture) creates a rip or tear relief that creates torn edges and faces, but no cut.
- Q **Oround** (bottom picture) creates a rounded end cut around the edges that require bend relief.



The **Offset Ratio** relates the size of a rectangular or oround relief cut to the material thickness. The default ratio of 0.5 means the relief cut will be $\frac{1}{2}$ of the material thickness. For example, if the thickness of the sheet metal part is 0.5mm, the relief cut will be 0.25mm. In the illustrations at the right, the **Offset Ratio** was set to 1.0 so that the relief cuts would be more visible.

The **Fixed edge or face:** selection identifies the face or edge that remains stationary during flattening. This selection is blank when you build a sheet metal part using the base flange feature. It is only used in special cases such as unrolling a cone or cylinder, or converting a conventional part to a sheet metal part. See *Unrolling Cones and Cylinders* on page 83 for more information.

Flat-Pattern Feature

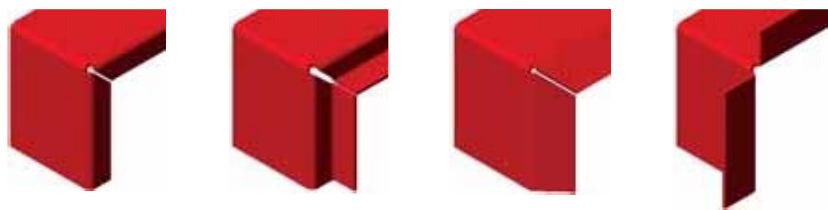


This feature is used to switch between the folded and flattened states of the model. It is suppressed by default.

- Q Suppressed = the part is shown in its *folded* state.
- Q Unsuppressed = the part is shown in its *flattened* state.

Miter Flange

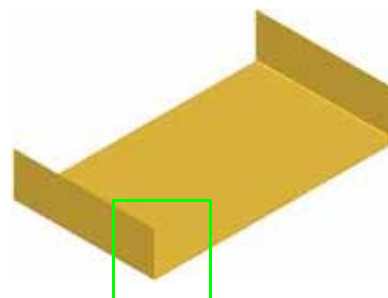
The **Miter Flange** is used to add flanges that connect and automatically create the necessary corner rips. You have to sketch the profile of the flange on a plane that is normal to the edge.



Miter Flange Examples

5 Flip the part over.

Hold down the **Shift** key, and press the up arrow twice. Then zoom in on the area indicated.

**6 Sketch normal to a curve.**

Normally, you have to create a plane normal to the curve, and then open a sketch on that plane. However, there is a shortcut.

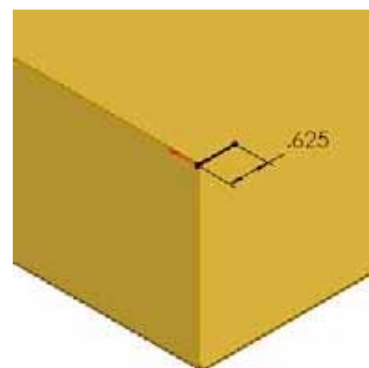
Select the outer edge and click **Insert, Sketch**. A sketch is automatically created normal to the nearest endpoint. When you exit the sketch, the system will create the plane and add it to the FeatureManager design tree.

**7 Flange profile.**

Sketch a horizontal line **0.625"** long, attached to the *outside* vertex. This is the profile of the miter flange.


Note

Note that the sketch can be more complex than just a single line. However, it must form an *open* profile.


**Introducing:
Miter Flange**

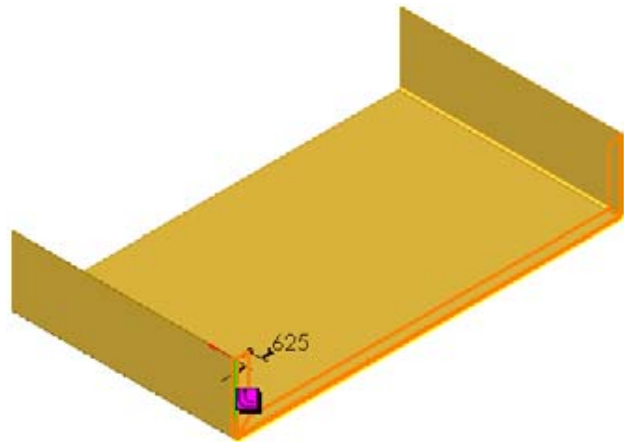
The **Miter Flange** is used to create one or more interconnected flanges. The flanges can be attached to multiple edges and are automatically ripped to allow unfolding of the part. Settings allow the flanges to be placed inside or outside of the model.

Where to Find It


- Q From the menu choose: **Insert, Sheet Metal, Miter Flange....**
- Q Or, click the  tool on the Sheet Metal toolbar.

Miter Flange


- 8 Flange propagation.**
Click the **Miter Flange** tool and a preview of the flange appears.
Click the **Propagate** marker  to automatically select tangent edges and continue the flange around the edge of the part.



Note




The **Propagate** marker acts as a toggle. It is cyan if the **Propagate** option is turned off. It is magenta  when it is turned on.

- 9 Miter Flange settings.**
In the PropertyManager, set the following:

- Q **Use default radius = On**
- Q **Flange position = **
- Q **Gap distance = 0.01"**

Flange Positions

There are three options for positioning the flange:

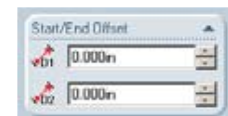
- Q **Material Inside** 
- Q **Material Outside** 
- Q **Bend Outside** 



Start/End Offsets

There are two options for offsetting the start or end distance:

- Q **Start Offset Distance**
- Q **End Offset Distance**



Custom Bend Allowance

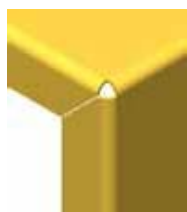
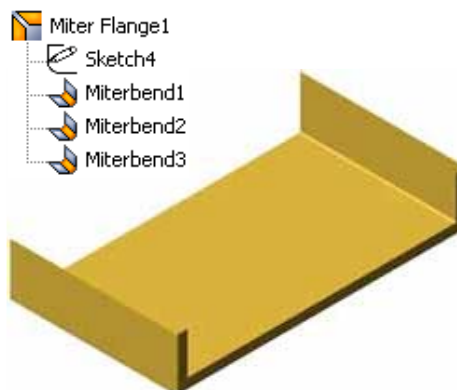
A **Custom Bend Allowance**, differing from the default, can be set for the bends of this feature.



10 Click OK to create the miter flange.

The **Miter Flange** command creates the flanges and bends. Although it is a single feature, each bend can be unfolded individually.

Gaps and bend relief are created where the flanges meet.

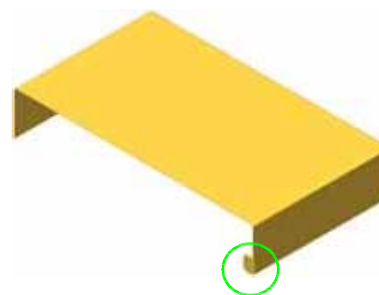


Edge Flanges

The **Edge Flange** tool can be used to dynamically add flanges to sheet metal edges. A selected edge can be dragged to set the size and direction of the flange. Settings allow you to change:

- Q **Edge flange profile**
- Q **Angle**
- Q **Flange length**
- Q **Flange position**


Default setting for the **Relief type** and **Bend radius** can be overridden. There are many variations on the use of the **Edge Flange**. Here are some possibilities.



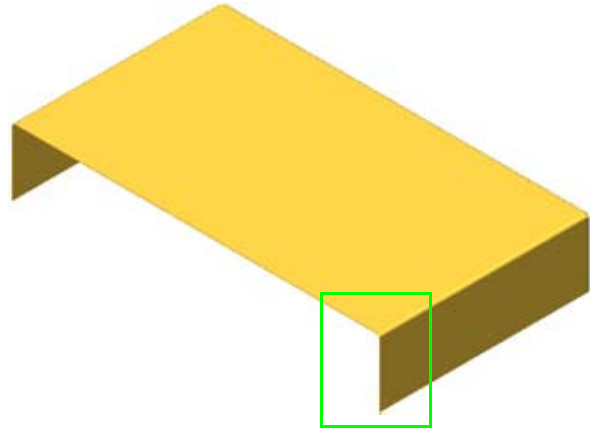
Introducing: Edge Flange


The **Edge Flange** adds a single flange to an edge.

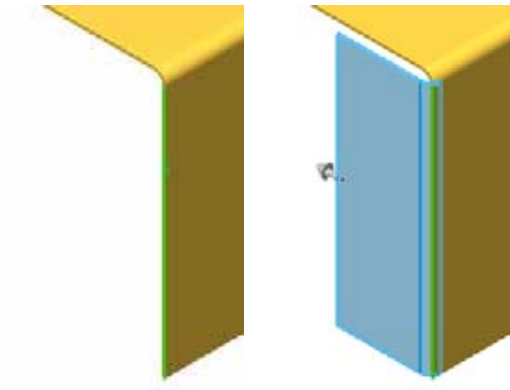
Where to Find It

- Q From the menu choose: **Insert, Sheet Metal, Edge Flange....**
- Q Or, click the  tool on the Sheet Metal toolbar.

- 11 Change the view.**
Return to a standard
Isometric view.
Zoom in on the area
indicated.





- 12 Flange direction.**
Select one of the vertical
edges and click the **Edge
Flange** tool . Drag the
cursor to the left and set the
flange length at **0.90"**.



13 Flange parameters.

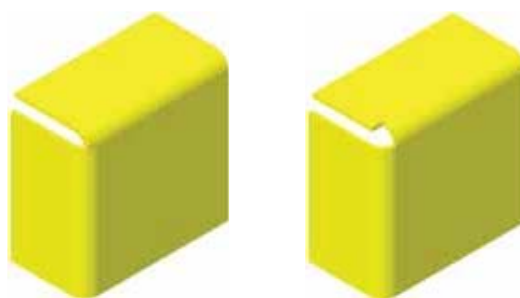
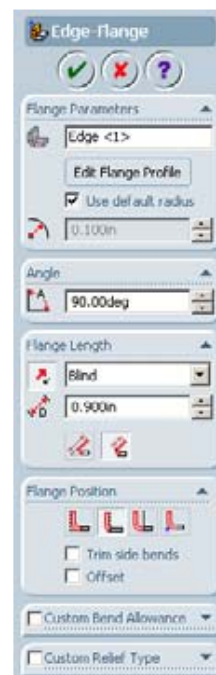
Settings for the **Angle** and **Flange Position** are set through the PropertyManager.

- Q **Angle = 90°**
- Q **Flange position =  Material outside**
- Q **Flange length =  measured to the Inner Virtual Sharp**

The **Flange length** will be **Blind** and the shape will be determined by using the **Edit flange profile** option.

Custom Bend Allowances and **Relief Types** are also available.

The **Trim side bends** option is used when an edge flange contacts an existing flange. In the examples below, the option is *off* for the example on the left; *on* for the one on the right.

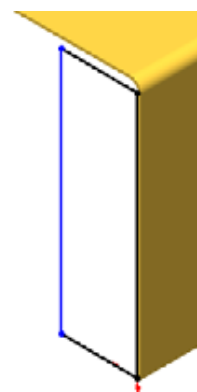


14 Flange profile.

Click **Edit flange profile** to change the default rectangular shape. The **Profile Sketch** dialog appears.

Note

The sketch will always include a line segment converted from the existing edge that lies in the direction of the flange, regardless of the edge you selected in step 12.



15 Fully define the sketch.

Drag the geometry and add dimensions and sketch fillets to fully define it as shown.

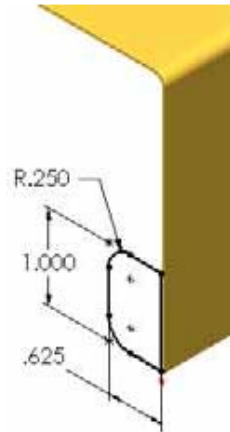
16 Continue.

Click **Back** on the **Profile Sketch** dialog. This takes you out of edit sketch mode, leaving the PropertyManager open so you can set any other parameters for the flange.

Click **OK** to create the flange and close the PropertyManager.

Note

If you click **Finish** on the **Profile Sketch** dialog, it will automatically exit the sketch, create the flange, and close the PropertyManager. If you then need to make changes to the other parameters of the flange, right-click the flange, and select **Edit Feature**.



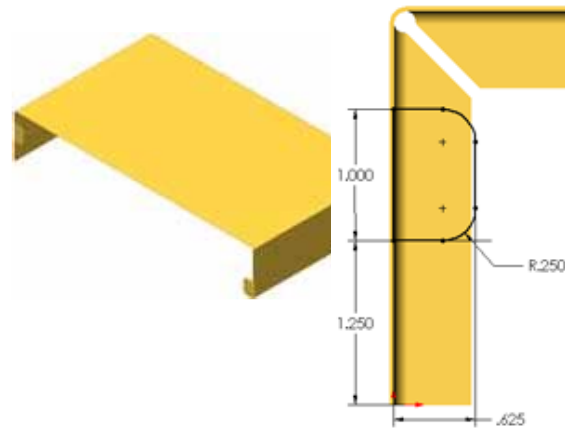
17 Completed edge flange.

Like any SolidWorks feature, the edge flange can be edited using **Edit Feature**.



18 Second flange.

Add another **Edge Flange** on the opposite side of the part using a similar procedure. Notice that the position of the profile is slightly different.



Adding a Tab

The **Tab** or **Boss Flange** is used to add a boss that is sketched on a face and extruded the thickness of the sheet metal. There is no dialog because the extrusion direction and thickness are known.

Introducing: Tab

The **Tab** adds a boss to a face.

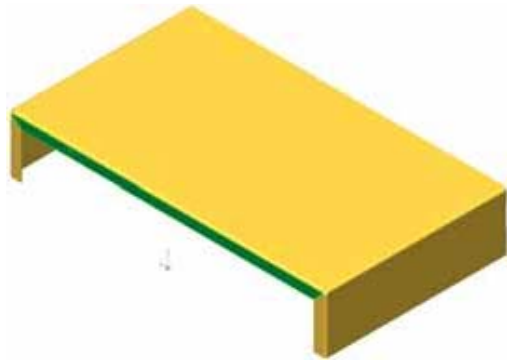
Where to Find It

Q From the menu choose: **Insert, Sheet Metal, Tab...**

Q Or, click **Base-Flange/Tab**  on the Sheet Metal toolbar.

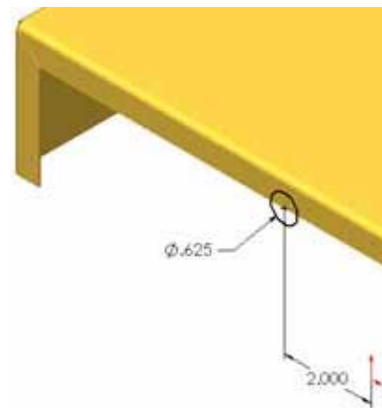
19 New sketch.

Select the upper face formed by the miter flange and insert a sketch.




20 Circular profile.

Sketch a circle whose center is coincident with the edge of the flange. Dimensioned it as shown.



21 Tab.

Click **Base-Flange/Tab**

 to create the tab feature. The direction of the extrusion and the depth are determined from the model.




Flat Pattern

The flat pattern can be seen at any time during the modeling process. Simply *unsuppress* the last feature in the FeatureManager design tree — the Flat-*Pattern* feature.

22 Unsuppress Flat-*Pattern*1.

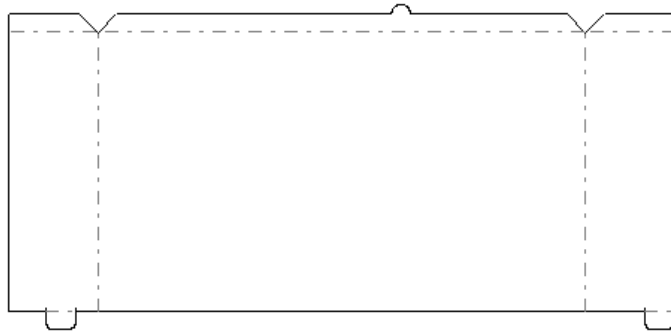
Unsuppressing a feature can be done in several ways. One way is to right-click the Flat-*Pattern*1 feature, and choose **Unsuppress** from the shortcut menu.

Note

The **Flattened** tool  can be used to perform the same unsuppress/suppress procedure.

23 Flat pattern.

The flat pattern, complete with bend lines, appears true size and shape in the Top view orientation.



Flat Pattern Options

The Flat-*Pattern* feature includes several options for the appearance and treatment of the flat pattern. To access these options, right-click on the Flat-*Pattern* feature, and choose **Edit Feature**.

Q Merge faces

When the **Merge faces** option is selected, faces that are planar and coincident in the flat pattern are merged. No edges are shown in the bend regions.



If you clear this check box, the tangent edges of the flattened bends appear.

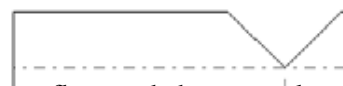


Q Simplify bends

When **Simplify Bends** is enabled, curves in the bend region are represented as linear edges in the flat pattern, simplifying the model geometry. When this option is not selected, compound curves remain in the flat pattern.

Q Corner treatment

When you flatten a sheet metal part by unsuppressing the Flat-*Pattern* feature, corner treatments are automatically applied to create a clean, flattened sheet metal part. The corner treatments are applied so the flat pattern is correct for manufacturing.



If you clear this check box, the flat pattern is shown *without* the corner treatments.



Q Add corner-trim

Selecting the **Add corner-trim** check box adds a new feature to the tree, positioned after the Flat-*Pattern* feature and dependent upon it.



This feature can be used to add break corners and relief options to the flat pattern.

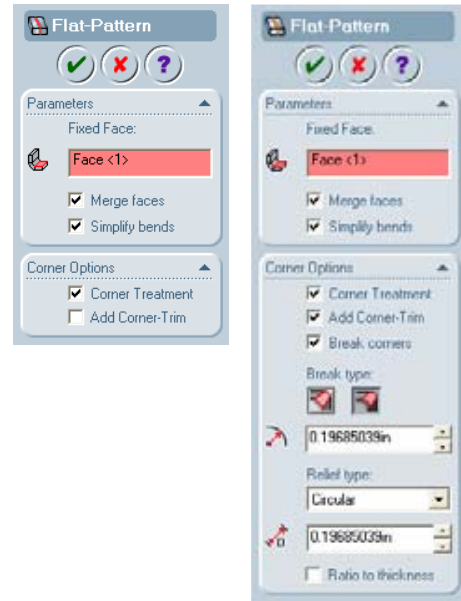


24 Flat pattern options.

Right-click the Flat-Pattern1 feature, and select **Edit Feature**. There are three options that are selected by default:

- Q **Merge faces**
- Q **Simplify bends**
- Q **Corner treatment**

Clicking **Add corner-trim** brings up settings for using **Break corners** and **Relief types**.



25 Corner-trim settings.

Editing the definition of the Corner-Trim1 feature allows you to change individual corners.

- Q **Break Corners**
Break selected edges with fillets or chamfers and optionally filter **Internal corners only**.
- Q **Relief Options**
Set relief shapes for edges. The available types are **Circular**, **Square** or **Bend Waist**.



26 Suppress.

Go back to the folded model by suppressing the Flat-Pattern1 feature.



Cuts

So far, all the tools we have used have created flanges (bosses) and added material to the part. Cuts can be made to the “folded” or “flattened” versions of the model.

Cuts in the Folded Model

Cuts that are made to the folded model can be made at any time on any face of the model. No special precautions are needed.

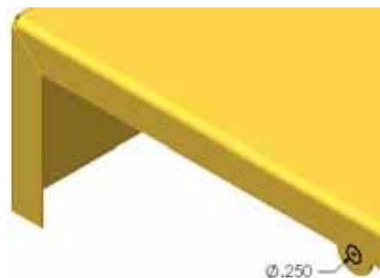
27 Sketch face.

Select the face indicated and click **Insert, Sketch**.



28 Sketch.

Sketch a **0.25”** diameter circle concentric to the circular edge.



29 Normal cut.

Create a **Blind** cut using the default **Normal cut** option and **Link to thickness**. These options ensure that the cut is normal to the sheet metal and the same thickness as the material.



Note

The end condition **Through All** may cut through other areas of the model, creating an undesirable result.

Unfold and Fold


Any cuts that must occur in the flattened model will need a pair of additional features. The **Unfold** and **Fold** commands precede and follow the cut feature. This allows the part to be unfolded locally, cut, and then refolded.

Introducing: Unfold

Unfold is used to flatten one or more bends. It is used in conjunction with **Fold** to temporarily flatten a portion of the model.

Where to Find It


Q From the menu choose: **Insert, Sheet Metal, Unfold...**

Q Or, click the  tool on the Sheet Metal toolbar.

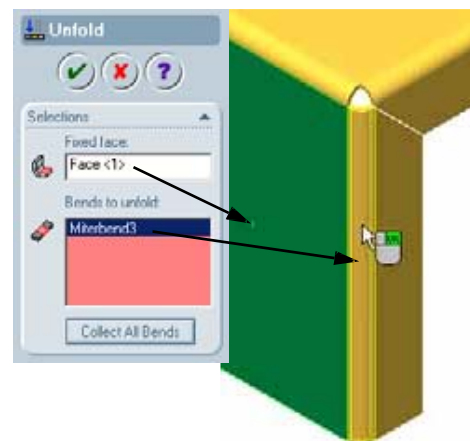
Note

The **Unfold** and **Fold** features are used in the *design* portion of the model. For features that are specifically for the manufacturing of the model (such as corner treatments) use the Flat-Pattern feature. Unsuppress the Flat-Pattern feature and add the manufacturing-specific features. The features will be added after that feature and will be children of it. Suppressing the Flat-Pattern feature again will suppress the added features as well.

30 Unfold.

Click the **Unfold** tool , and select the **Fixed face**. Click in **Bends to unfold** and select the bend indicated.

Click **OK**.



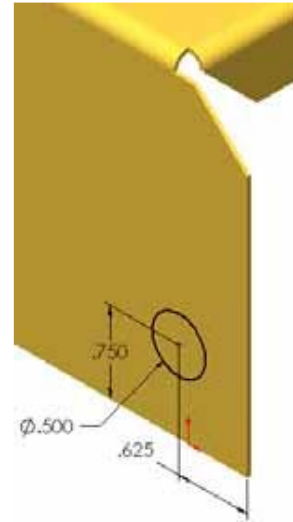
31 Result.

The selected bend is flattened, keeping the fixed face in position.



32 Sketch.

Sketch on the fixed face using the dimensions shown.

**33 Cut.**


Extrude the sketch as a **Blind** cut using **Link to thickness**.




Introducing: Fold

Fold is used to refold one or more unfolded bends. It is usually used in conjunction with **Unfold**.

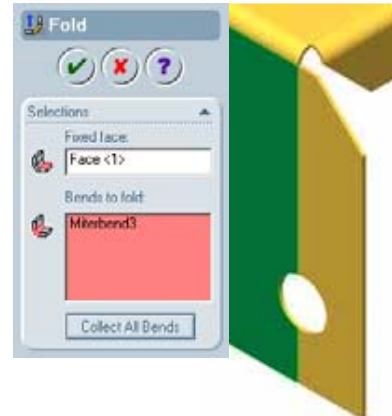
Where to Find It

- Q From the menu choose: **Insert, Sheet Metal, Fold...**
- Q Or, click the  tool on the Sheet Metal toolbar.

34 Fold.

Refold the bend using **Fold** . Select the same **Fixed face** that was used in the unfold operation. Rather than selecting individual bends, click **Collect All Bends** to have the system identify the bends.

Click **OK**.



35 Resulting cut.

The cut that was made across the bend then refolded.

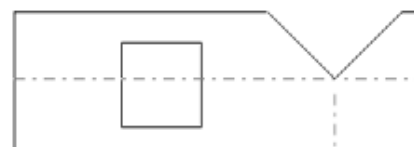
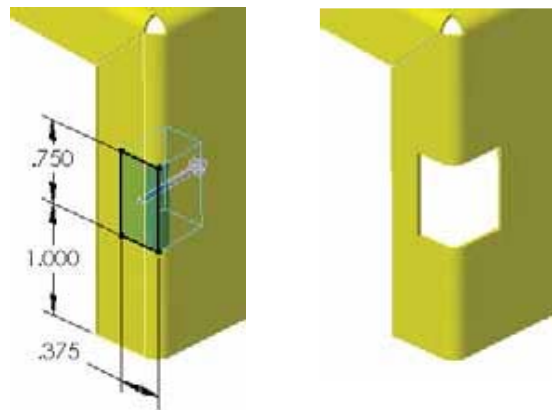


Note

Cuts across bends can be made without unfolding. If the cut does not require the model to be in the flattened state during the cut, it can be sketched directly on a face of the model.

In this example, a rectangular cut is made from the flange through the bend, depth **0.5**".

The flat pattern processes the cut and flattens it.



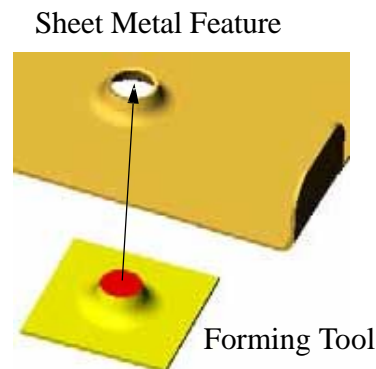
Sheet Metal Forming Tools

Forming tools are used to bend, stretch, or otherwise form sheet metal. SolidWorks supplies a number of sample tools in the Design Library. In addition, you can edit existing tools or create new tools.

You were introduced to the Design Library in the course *SolidWorks Essentials: Parts and Assemblies*.

Standard Tools

A set of forming tools is included with the SolidWorks software. The set includes embosses, extruded flanges, louvers, ribs and lances. You can add more of your own design.



Using a Standard Forming Tool

Standard features refer to those that come with the software, whether they are parts or library features. In this example, a standard sheet metal forming tool will be added to our part.

The individual forming tools are parts specially designed for use in creating form features. They are intended to be used like other library features: to be dragged and dropped onto a face of a model. However, forming tools can only be dropped into sheet metal parts.

In general, there are five steps to adding a forming tool using the Design Library:

1. Drag the tool from the Design Library onto the appropriate face of the model.
2. Use the **Tab** key to reverse the direction of the form, if necessary.
3. Drop the tool by releasing the mouse button.
4. Edit the sketch. This includes repairing dangling relations/dimensions or adding new relations/dimensions. The sketch is generally fully defined at the end of this stage.
5. Form features are *not* sized interactively as you add them to the model. After editing the sketch, they are added at their default size. You can edit the definition of the form feature to change its dimensions.

How They Work

Forming tools act as dies that bend, stretch, or otherwise deform sheet metal. The face to which you apply the forming tool corresponds to the stopping surface of the tool itself. By default, the tool travels inward towards the face. The material is deformed when the tool strikes the face where it is dropped.

Drag and Drop

The drag and drop operation is very simple. Just drag the feature from the Design Library onto the face of the active part and drop it. A preview of the feature on the face is shown at the cursor.

Push or Pull?

Form features can be pushed through from the face on which they are dropped or pulled out from the opposite side of the material. By default, SolidWorks will push them into the face on which they are dropped. If you press the **Tab** key *before* you drop the feature, the form feature will be pulled out. Pressing **Tab** key repeatedly will toggle this behavior with the preview graphics updating accordingly. Observe the preview as you toggle the direction.

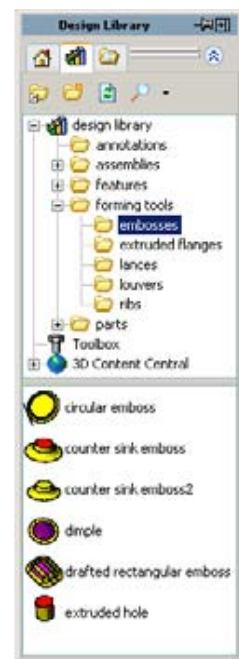
36 Sheet metal forming tools.

Expand the Forming Tools folder in the Design Library to see the available folders. Click the Embosses folder.

Note

Forming tools and regular SolidWorks part files both have the same *.sldprt file extension. In the Design Library, *.sldprt files are treated as regular part files unless you mark the folder as a forming tools folder. By default, when you install the SolidWorks software, the folder sldworks\data\Design Library\Forming Tools and its sub-folders are already marked as forming tools folders.

To manually mark a folder as a forming tools folder, right-click the folder in the Design Library window and select **Forming Tools Folder** from the shortcut menu.

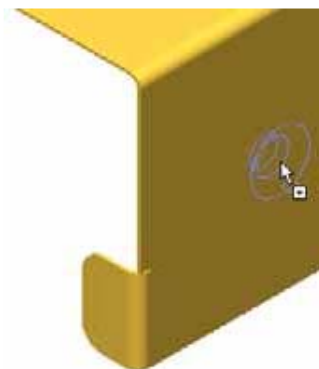


37 Drag and drop.

Drag the counter sink emboss onto the model face.

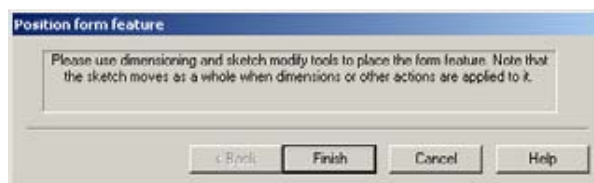
Check the orientation of the feature. Since we want to push the form feature *into* the target face, there is no need to press the **Tab** key.

Drop the feature by releasing the mouse button.



38 Edit the sketch.

You are now in **Edit Sketch** mode. A message box appears asking you to position the form feature.



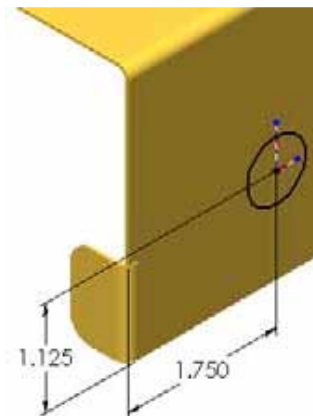
The profile of the feature and two centerlines (for positioning) appear.

*Do not click **Finish** yet.*

39 Locate.

Locate the sketch from the edges of the model using the dimensions shown.

Click **Finish** on the **Position form feature** dialog.

**40 Completed form feature.**

The form feature is added to the model oriented in the chosen direction.



Copy and Modify a Forming Tool

The existing forming tools cover a wide range of types but you cannot interactively edit their size as you add them. After they are added, the dimensions can be changed to resize them. If the forming tool will be used multiple times, it may be better to make a modified version with the correct dimensions. It is generally easier to modify a copy of a tool than to create a new tool from scratch.


41 Edit the tool.

To open and edit a forming tool, right-click the icon, and choose **Open**. In this example, choose the Louvers folder and the louver feature.

42 Save As.

Save a copy of the louver forming tool as LongLouver. By default, the new part is saved in the same folder as the original.

Click **Save**.

To refresh the Design Library so you can see the LongLouver, click **Reload**  on the toolbar in the Design Library.



43 Units.

Change the units to **inches**.

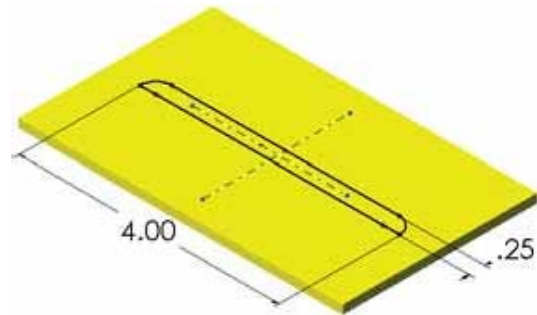
44 Enlarge base feature.

Make Base-Extrude long enough to enclose the updated Layout Sketch; at least 4.5 inches.

45 Edit Sketch.

Edit Layout Sketch and resize the length and width as indicated.

Exit the sketch.




46 Fillet.

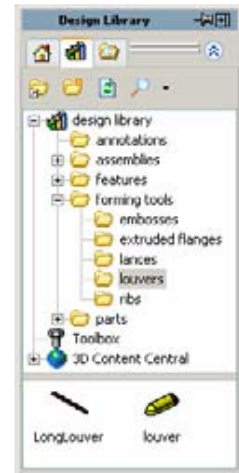
Change the fillet radius to **0.10"** and rebuild the model.

Save and Close the modified part.



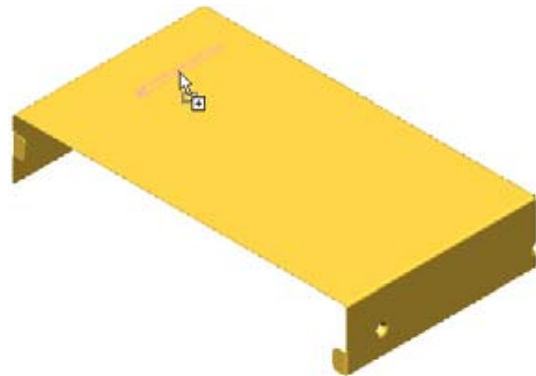
47 Design Library.

Refresh the Design Library using the  button. The updated image of the modified forming tool, LongLouver, appears.

**48 Drag and drop on face.**

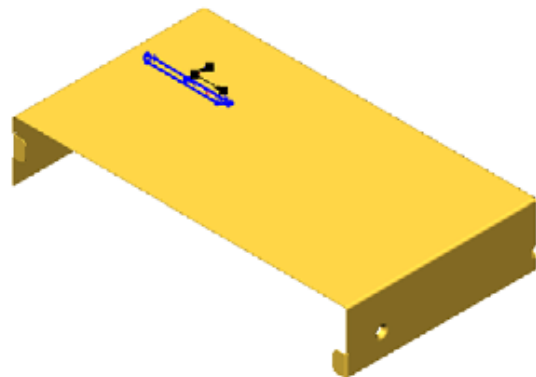
Drag the LongLouver from the Design Library onto the top face of the part.

Use the **Tab** key to ensure that the louver opening is *above* the top face.

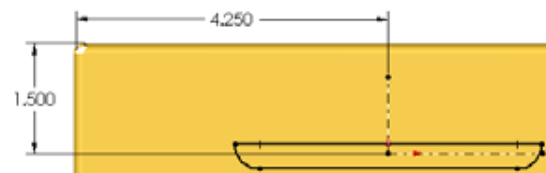
**49 Modify Sketch.**

Use **Tools, Sketch Tools, Modify...** to rotate the sketch so that the straight edge of the sketch faces the rear of the model.

You can rotate the sketch either **270°** or **-90°**.

**50 Dimensions.**

Dimensions and relations can be used to fully define the sketch. Use the construction geometry within the sketch to orient and locate sketch.



Click **Finish** to complete the process.

51 Completed form feature.

The part shows the form feature placed on the upper face.

Note

Unlike a library feature, the dimensions of a forming tool can only be changed *after* you complete the process of inserting it.

To change the dimensions of a form feature after it has been applied, double-click the feature, either in the graphics window or the FeatureManager design tree. The available dimensions will appear. Change their values and rebuild the part.



52 Pattern.

Pattern the louver using 1" spacing and a total of 4 instances.



Creating a Custom Forming Tool

The face to which you apply the forming tool corresponds to the stopping surface of the tool itself. The direction of travel of the tool is normal to the stopping surface. When you create a forming tool, you must keep these facts in mind.

The forming tool is created by building a specialized part that represents the shape of the forming die. The horizontal plane (the Top reference plane) of this part is the stopping surface of the forming tool. The direction of travel of the forming tool is along the vertical axis, normal to the Top reference plane. When you build the forming tool, the entire tool must lie entirely on one side of the stopping surface. Finally, give some thought to the relationship between the forming tool and the origin of the part in which you build it. The origin determines where on the sheet metal face the tool drops when you apply it to a sheet metal part.

Note

For the next portion of this example, you can use the partially completed part `PartialFormFeat`.

53 Open PartialFormFeat.

The part contains boss and fillet features.


Important!

When adding fillets, keep in mind that the minimum radius of curvature of the forming tool must be greater than the thickness of the sheet metal part to which the tool is applied. To determine the minimum radius of curvature, use **Tools, Check**.

**Introducing:
Split Lines**

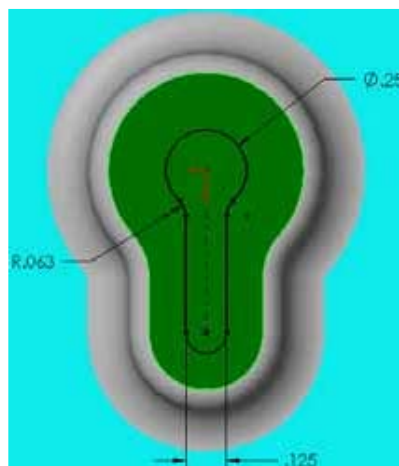
Insert, Curve, Split Lines uses one or more curves to split one model face into two. The curves are sketched on a plane and project onto the faces to be split.

Where to Find It

- Q From the **Insert** menu, select **Curve, Split Line...**
- Q Or from the Curves toolbar, click the **Split Line**  tool.

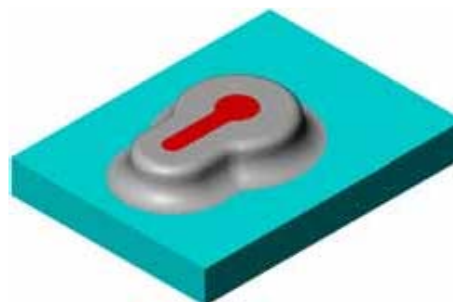
54 Split Line.

Create a sketch for the **Split Line** function. Use the sketch to split and create a new face. This new face will be marked to represent a cut in the form feature.

**55 Face color.**

Color the new face **red**.

This indicates to the SolidWorks software that the face punches a hole in the sheet metal it is applied to.

**Note**

The color must be pure red (255, 0, 0) in order to work properly.

56 Remove base geometry.

Using a cut, slice off the base feature geometry from the model.

Tip

An easy way to remove the base feature is to select the stopping face of the forming tool — the upper face of the base feature — open a sketch and click **Convert Entities**. Extrude a cut away from the forming tool, using the **Through All** end condition.



57 Positioning sketch.

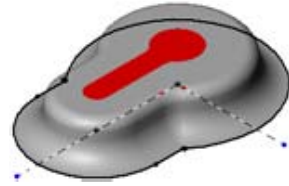
A useful feature is a positioning sketch that shows the outline of the forming tool and uses centerlines to denote the positive directions of the horizontal and vertical axes of the feature. These may help in placing and orienting the feature.

Tip

One method of creating the positioning sketch is to open a sketch on the face of the forming tool and use **Convert Entities** to replicate the outline of the feature.

Another Tip

It is a good idea to (a) rename the positioning sketch and (b) hide the positioning sketch so that it does not appear in the thumbnail graphic of the forming tool displayed in the Design Library.



58 Save.

Save the part as Keyhole in the `sldworks\data\Design Library\Forming Tools\embosses` folder. This is one of the standard folders that is created and used by the SolidWorks software.

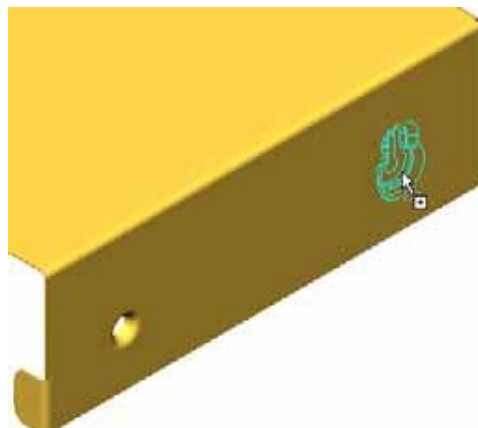
Note

Forming tools and regular SolidWorks part files both have the same *.sldprt file extension. In the Design Library, *.sldprt files are treated as regular part files unless you mark the folder as a forming tools folder. By default, when you install the SolidWorks software, the folder `sldworks\data\Design Library\Forming Tools` and its sub-folders are already marked as forming tools folders.

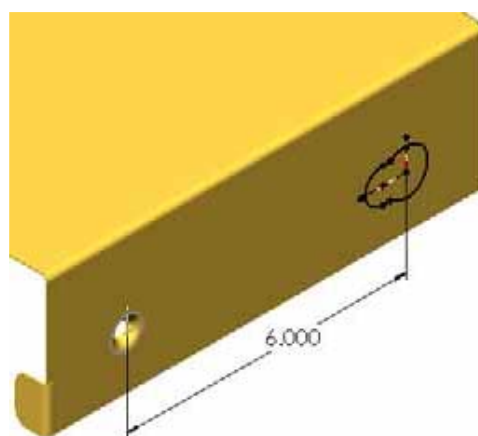
To manually mark a folder as a forming tools folder, right-click the folder in the Design Library window and select **Forming Tools Folder** from the shortcut menu.

59 Drag and drop.

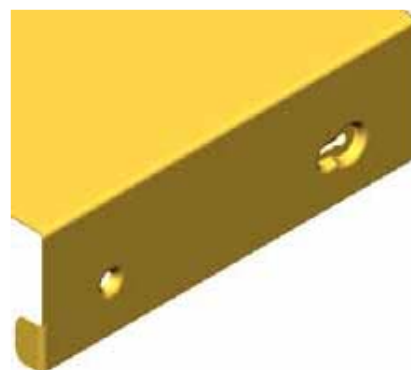
Drag and drop the Keyhole from the embosses folder onto the model.

**60 Orientation, relations and dimensions.**

Use **Modify Sketch** to orient the feature. Show the sketch of the counter sink emboss1 and add a **Collinear** relation between the centerlines of the two features. Add a dimension to fully define the sketch.

**61 Finished feature.**

Click **Finish** to complete the addition of the Keyhole feature.

**62 Mirroring the features.**

Mirror the Keyhole and counter sink emboss1 features using the Right reference plane.

63 Save the part.

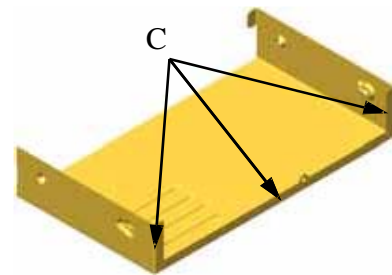
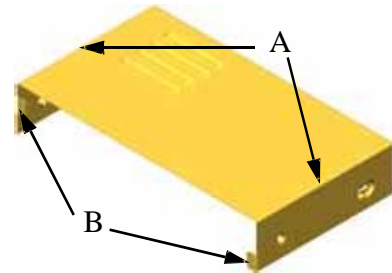
Bend Angles

How you edit the bends in the part depends on how they were created. For example, consider the bends in this part:

Bend A: These angles are controlled by the sketch of the **Base Flange**. To change the angle, edit the sketch. If it was sketched with horizontal, vertical, or similar relations, you would have to delete the relations and add angular dimensions instead.

Bend B: The angle of these bends is determined by the fact that we created an **Edge Flange**. By default the flange is perpendicular to the top face of the part, so this angle is set at **90°**. But can be changed to any value.

Bend C: The three flanges that comprise the **Miter Flange** depend on the angle of the sketch used. To change the angle, edit the sketch.



Sheet Metal Parts in Drawings

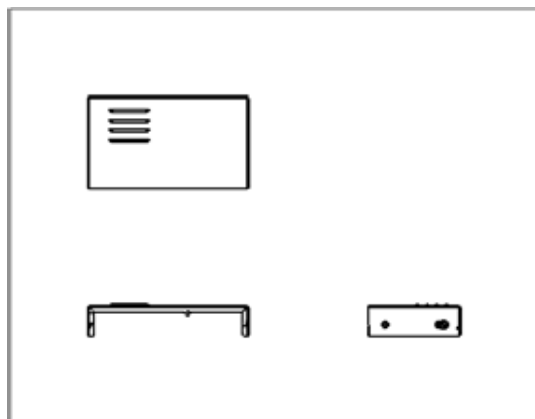
Sheet Metal parts can be unfolded at any time during the design process. In addition, when you create a drawing of a sheet metal part, the system automatically makes a configuration of the flattened part. This configuration can be used to make a flat pattern view.

1 New drawing.

Using **Make Drawing from Part/Assembly**, create a new drawing size **A-Landscape** (279.4 mm x 215.9 mm) with no sheet format. The scale should be **1:5**.

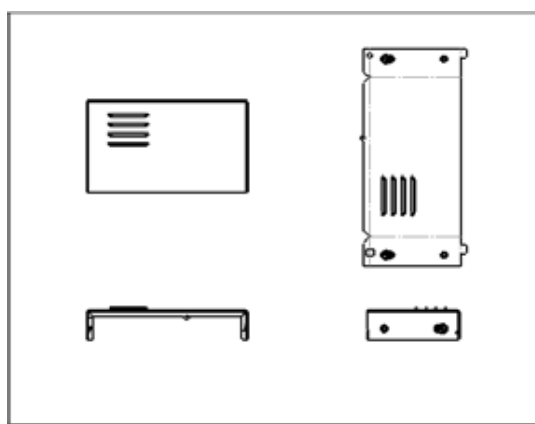
2 3 Views.

Select the **Front** view of the Cover and place it on the sheet. Drag and place **Top** and **Right** projection views as well.




3 Flat Pattern view.

Add a new **Model** view to the drawing. Choose the Flat Pattern view. This choice automatically uses the configuration DefaultSM-Flat-Pattern that was created by the software.



Tip

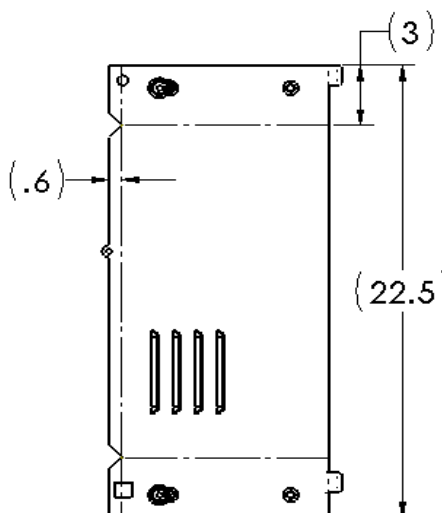
If you do not like the default orientation of the flat pattern view, you can easily rotate the view. One method is to select the view and click **Rotate View** . A dialog box will appear in which you enter the angle of rotation— usually positive or negative 90°.

4 Reference Dimensions.

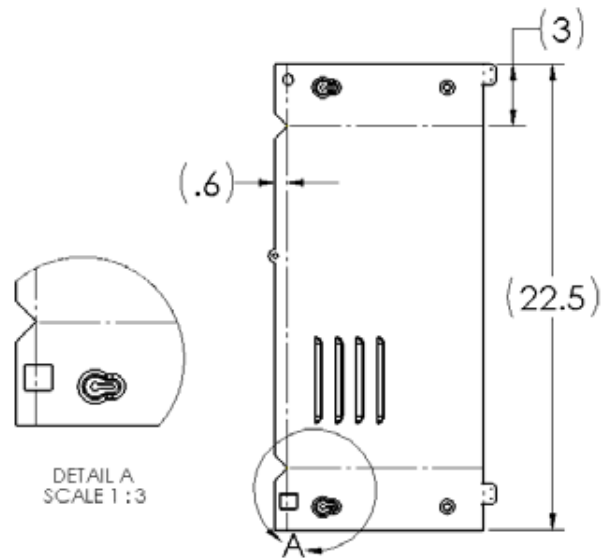
Dimensions can be imported or added as driven (reference) dimensions.

Note

Bend lines can be dimensioned to like any other model edge.



- 5 **Additional view.**
Additional views such as details, sections or cropped views can be added.
- 6 **Save and close.**
Save and close the files.

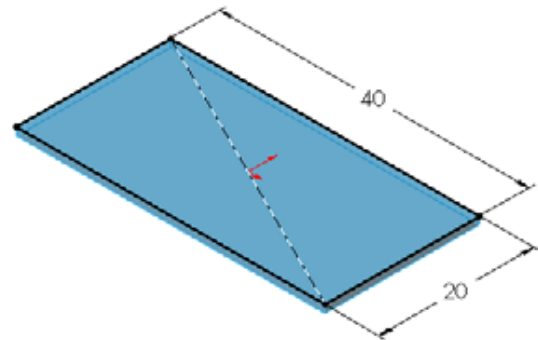


Closed Corners and Miter Flanges

Adding a **Miter Flange** leaves a gap between the individual flanges. Although the gap distance can be set within the miter flange dialog, some amount of gap must be created.

These gaps can be closed using the **Closed Corner** dialog. It allows three types of corners to be created from the gap.

- 1 **Base Flange.**
Create a rectangle centered on the Origin and sized as shown. Use it to insert a **Base Flange** with **1mm** thickness.



Note

When making a base flange from a closed contour, only the thickness option is available.

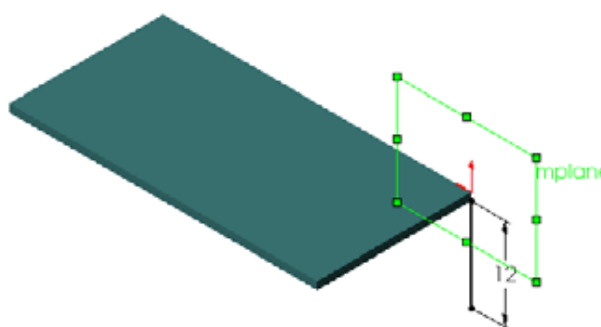
2 Sheet Metal Feature.

The **Bend Radius** has assumed a default value. Edit the Sheet-Metal1 feature. Set the **Default bend radius** to **1.5mm**.

Click **OK**.

**3 Plane and sketch.**

Create a new plane and sketch normal to the edge of the solid. Sketch a single line for the profile of the miter flange.

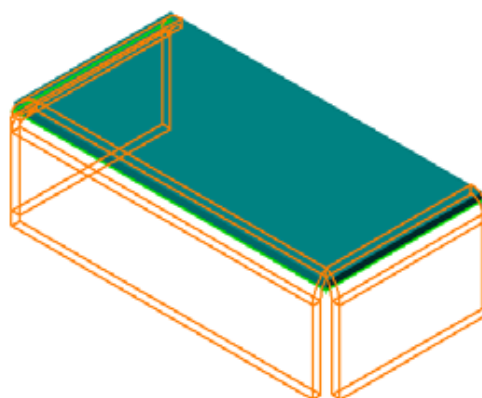
**Individual Edges with a Miter Flange**

Like fillets, the miter flange can be made to propagate along tangent edges. When edges are not tangent but are continuous, they can be selected individually.

4 Miter selections.

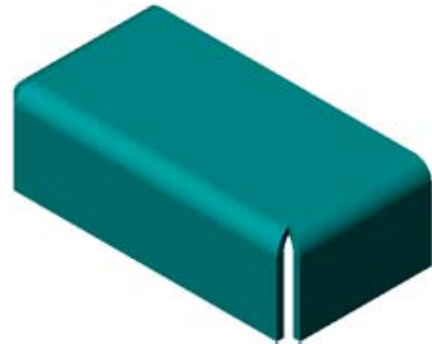
Select three connected edges of the model to continue the path of the miter flange around two sharp corners. Set the **Gap distance** to **1mm**.

Use **Material Outside** for the flange position, and click **OK**.





5 Corners.

The corners created between the miter flanges use the gap setting.



Closing a Corner


The closed corner option changes the open corner to a closed one by extending either or both of the existing faces, and trimming them to the corner. You can choose from three possible results:

- Q **Butt** 
- Q **Overlap** 
- Q **Underlap** 


**Introducing:
Closed Corner**

Closed Corner is used to extend one or both faces to the theoretical corner.

Where to Find It

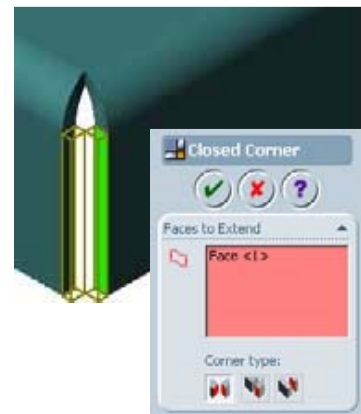
- Q From the menu choose: **Insert, Sheet Metal, Closed Corner...**
- Q Or, click the  tool on the Sheet Metal toolbar.

6 Select Face.

Select one planar face at the corner and set the **Corner type** to **Butt** 

Click **OK**.

7 Completed corner.

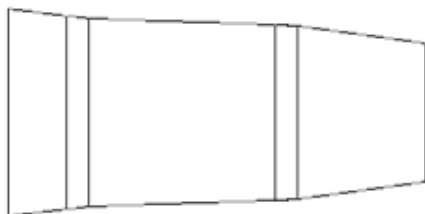


8 Close.

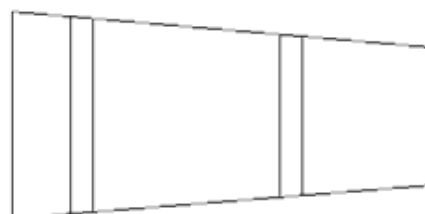
Close the file without saving the changes.

Designing in Flat

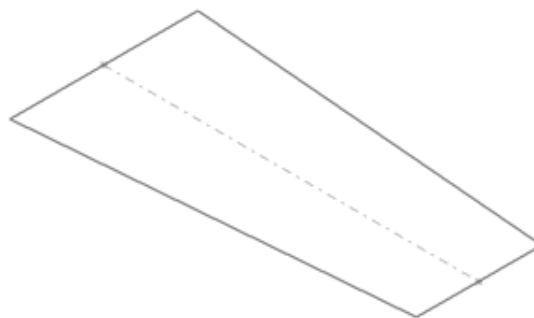
Many times it is desirable to design sheet metal parts in the flat rather than bent state. For example, consider this tapered, bent bracket shown at the right. If we were to model this in its bent state, the flat pattern would look something like:



By modeling it in the flat we can simplify the shape of the blank, reducing manufacturing costs. We are able to do this because SolidWorks allows you to work from a flat piece of material, adding bends as you go. This simple example will show the method used to design in the flat and develop the folded model.



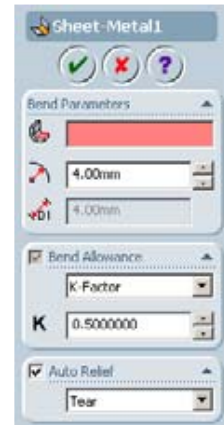
-
- 1 Open the part.**
Open the part sheet metal flat. It contains a sketch.
 - 2 Base flange.**
Select the sketch and use **Insert, Sheet Metal, Base Flange** with a **thickness of 4mm**.



3 Sheet metal feature.

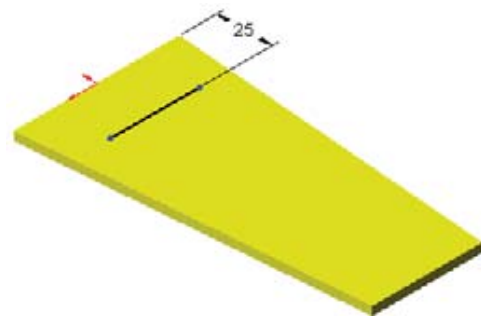
Edit the Sheet-Metal1 feature. Set the **Default bend radius** to 4mm.

Click **OK**.



4 First bend line.

Create a new sketch on the top face and add a bend line to the flat model. Use a vertical line dimensioned from the edge of the model




Note

Bend lines can be added to *any* sheet metal part in the same way. If multiple bend lines are used in the same sketch, all the bends will be in the same direction.

**Introducing:
Sketched Bends**

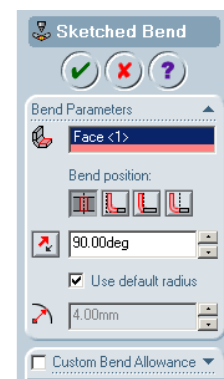
Sketched Bend is used to add a bend line to a flat portion of a sheet metal part. An existing sketch with one or more bend lines must be used.

Where to Find It

- Q From the menu choose: **Insert, Sheet Metal, Sketched Bend....**
- Q Or, click **Sketched Bend**  on the Sheet Metal toolbar.

5 Sketched Bend.

Click the **Sketched Bend** tool 



6 Fixed face.

Given a rectangular plate with a bend line dividing it, which of the two faces remains stationary? A fixed face will be chosen to determine this.





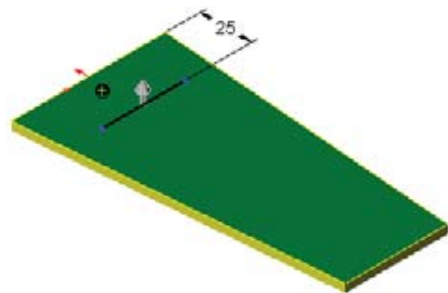
This?



Or this?

7 Settings.

Select the smaller area as the **Fixed face**. A symbol  appears indicating the selection. Set the bend angle to **75°**. The arrow indicates the direction of the bend. Use the  button to reverse the direction of the bend, if desired.



Set the **Bend position** to **Bend centerline**  and check **Use default radius**.

Click **OK**.

8 Bend.

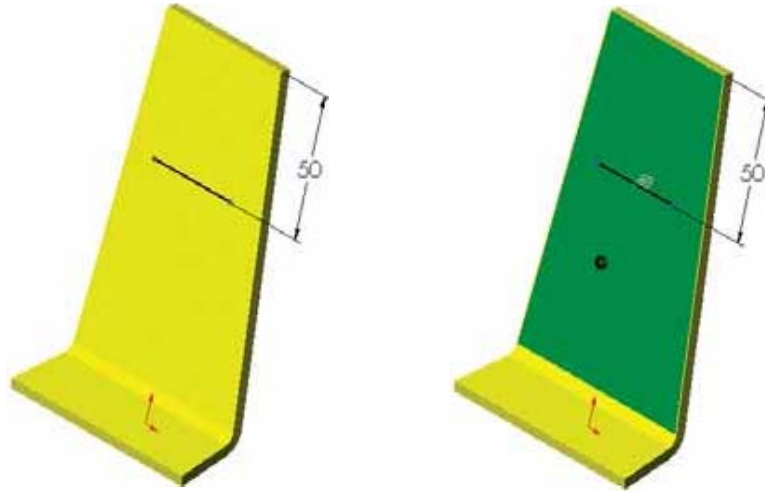
The selected portion of the face remains stationary as the other bends upwards at an angle of **75°**.



9 Second bend.

Add a second bend line.

Select the center section as the **Fixed face** and use the same angle, radius, and bend position as before. Bend in the opposite direction.



10 Results.

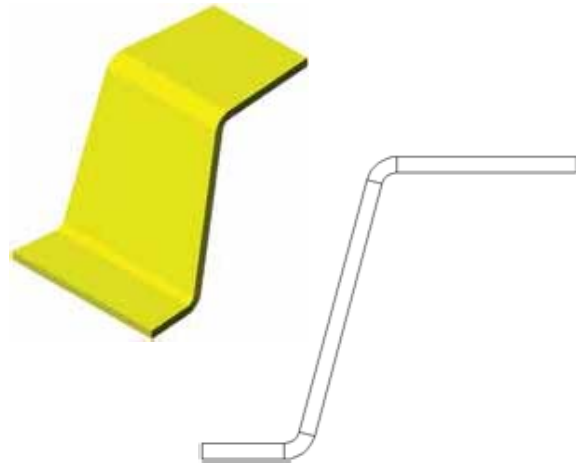
The complimentary bend angles force the center section to be angled and the ends to be horizontal.

Tip

You can use **Link Values** to keep the bend angles equal.

11 Exit the part.

You can save or discard the changes, as you wish.



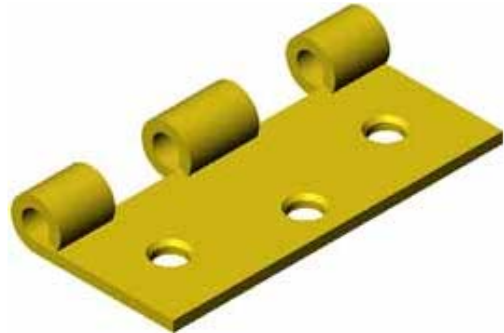
Existing Rounds

If a sketch has arcs in it, SolidWorks will convert them to sheet metal bends and flatten them. The value of the arc radius is used in place of the default **Bend Radius** value.

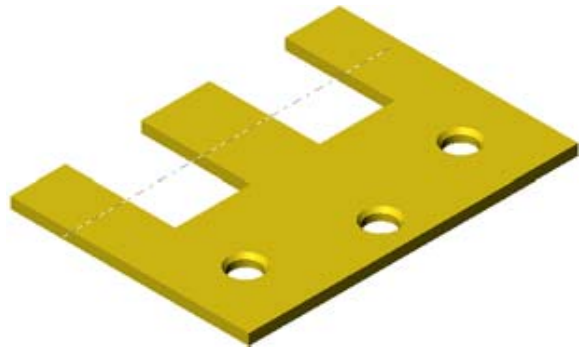
Base Flange

The Base Flange uses a standard open contour sketch for extrusion.

If the sketch contains arc shapes, they are interpreted as bends.

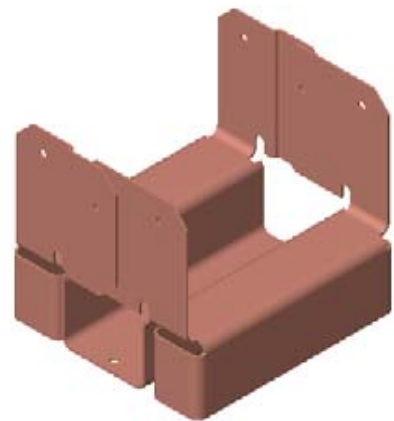


The bends are flattened in the same way as bends that were created from sharp corners.



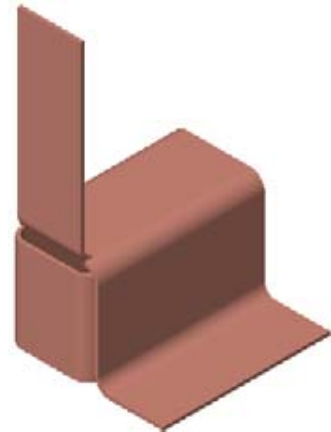
Using Symmetry

Parts that are symmetrical can be created as “half-models” and completed using **Insert, Pattern/Mirror, Mirror...** and selecting the **Bodies to Mirror** option. The **Flat Pattern** will recognize the mirrored features.



1 Open part.

Open the part `SM_Symmetry`. It is a sheet metal part that contains the base flange and edge flange features.



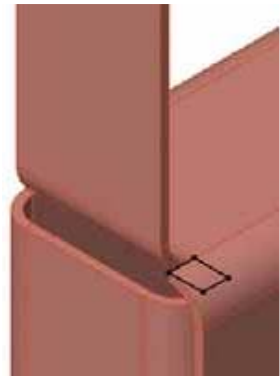
Manual Relief Cut

To avoid problems in flattening, it is sometimes necessary to add bend relief cuts manually.

2 Sketch.

Sketch a rectangle using vertices and edges to fully define it.

Use **Shaded with Edges** view mode to help with edge selection.

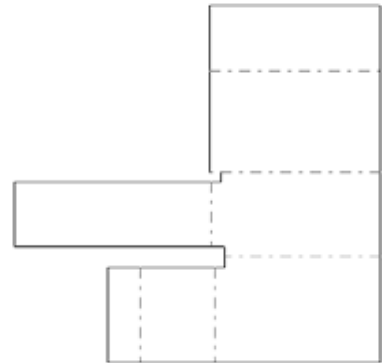


3 Extrude Normal Cut.

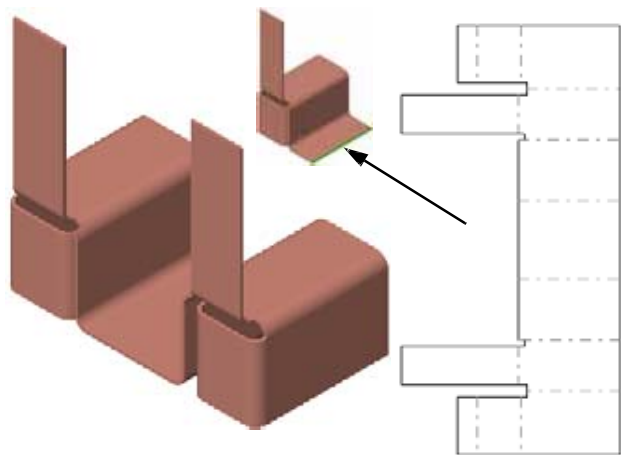
Using the **Normal cut** and **Link to thickness** options, create a cut feature. The cut is made normal to the model faces.



- 4 **Flat Pattern.**
Show the flat pattern of the part.

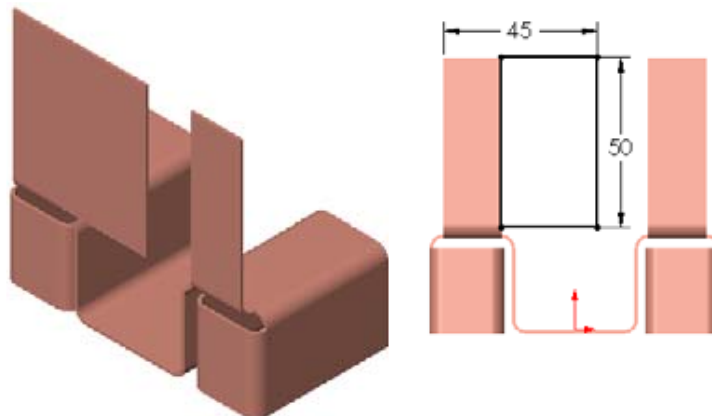


- 5 **Mirror.**
Refold the part, and use **Insert, Pattern/Mirror, Mirror...** with the **Bodies to Mirror** option. Note in the inset picture, which face is used for the mirror plane.

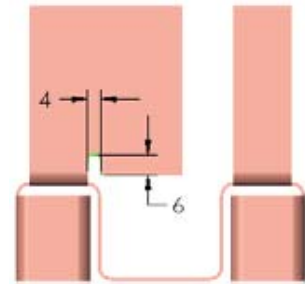


The flat pattern updates accordingly.

- 6 **Tab.**
Add a tab, sketching on the left vertical flange.



- 7 **Tombstone relief cut.**
Create a relief cut using the shape shown. Set the end condition to **Blind** and use **Link to thickness**.




Break Corner


Use **Break Corner** command to create chamfered or filleted corners.

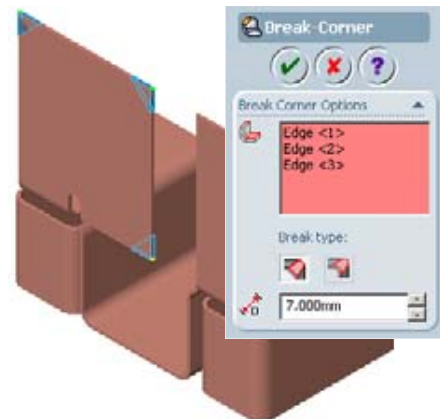
Introducing: Break Corner

Break Corner is used to add rounds or chamfers to the sheet metal part. The command is easier to use than the **Fillet** or **Chamfer** commands because it makes edge selection easier and it only removes material.

Where to Find It

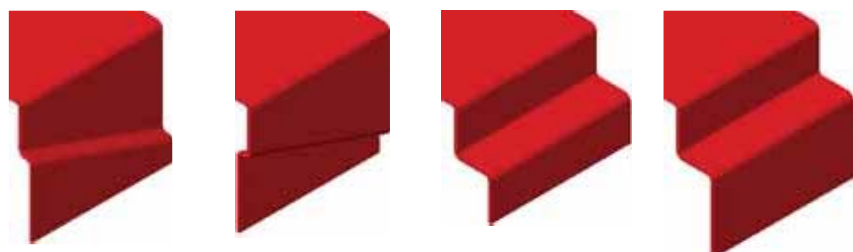
- Q From the menu choose: **Insert, Sheet Metal, Break Corner...**
- Q Or, click **Break Corner**  on the Sheet Metal toolbar.

-
- 8 **Chamfers.**
Click **Break Corner** , and select the **Chamfer** option. Set the **Distance** to **7mm** and select the short edges as shown.
Click **OK**.



Jog Feature

The **Jog** feature is used to add a jog or an offset to an existing flange. A jog is also sometimes referred to as an offset or a joggle.




Jog and Offset Examples

Introducing: Jog

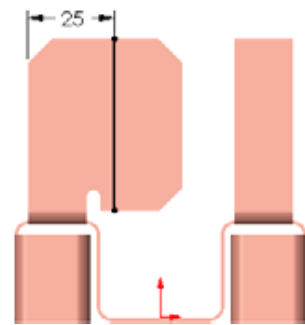
Jog is used to add a pair of bends and a flat to a flange. A sketched line is used to position it with a **Jog Offset** distance.


Where to Find It

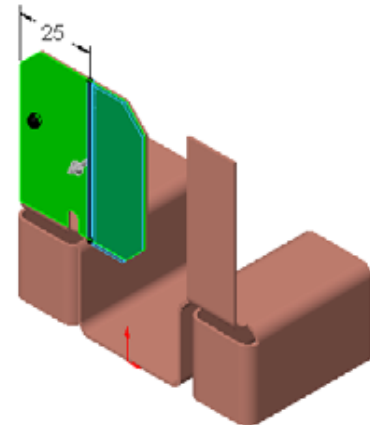
- Q From the menu choose: **Insert, Sheet Metal, Jog...**
- Q Or, click **Jog**  on the Sheet Metal toolbar.

9 Sketch for Jog.






Create a sketch on the flat face created by the tab and sketch a vertical line. Add a dimension to position it relative to the edge of the tab.

**10 Face selection.**

Click **Jog** , and select the face to the left of the sketched line. The arrow should point in the direction shown.

**11 Jog settings.**

In the PropertyManager, set the following:

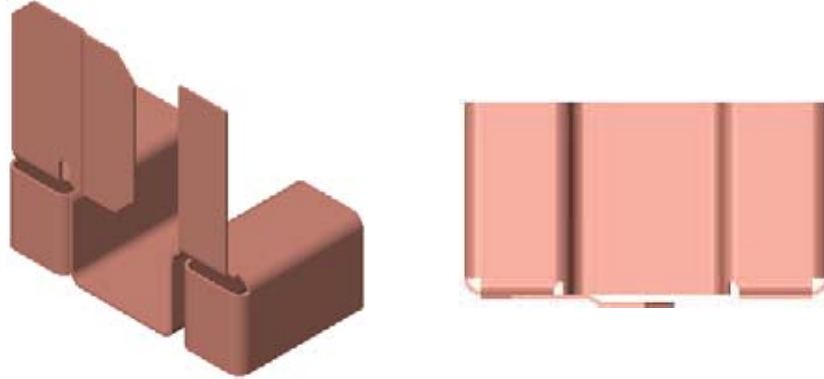
- Q **Use default radius = Off**
- Q **Radius value**  = 0.5mm
- Q **End condition = Blind**
- Q **Depth**  = 0.5mm
- Q **Dimension position** 
- Q **Fix projected length = On**
- Q **Jog position**  = Bend Centerline
- Q **Jog Angle**  = 30°

Click **OK**.



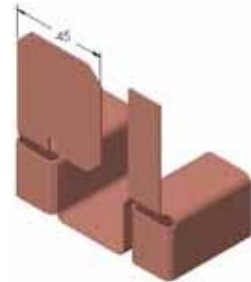
12 Result.

The resulting **Jog** feature,  Jog1, creates bends and flats in the flange.

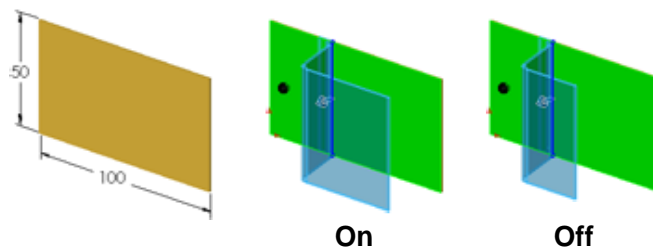


Fix Projected Length

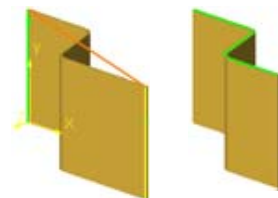
The option **Fix projected length** is useful when designing parts in their folded, 3D state. For example, in step 6, we defined overall width of the tab to be **45mm**. That is our design intent – even after we add the jog. Rather than trying to calculate how much additional material is needed for the jog, the option **Fix projected length** keeps the overall dimension correct after the jog is added.



Consider the jog shown below. The preview shows the difference between having the option **Fix projected length** on, or off.



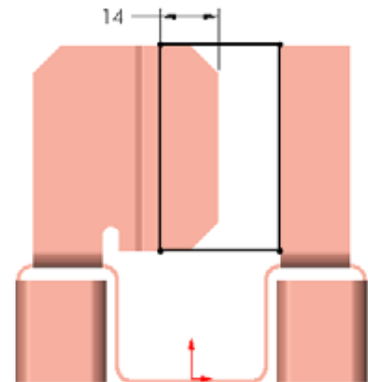
If you use **Tools, Measure** on the two different results, having the **Fix projected length** option *on* gives a Delta X distance of 100mm and a length along the edges (use **Select Tangency**) of 119.741mm.



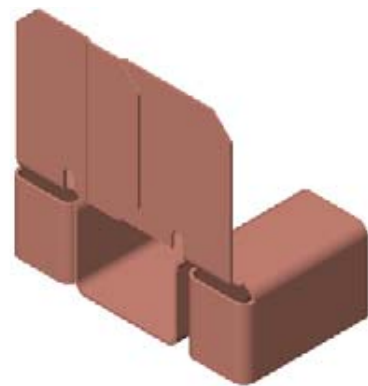
With the option *off*, Delta X is 80.259mm but the length measured along the edges is 100mm.

13 Tab.

Add another **Tab** using the right vertical flange for the sketch plane. Dimension the overlap.

**14 Copy and paste feature.**

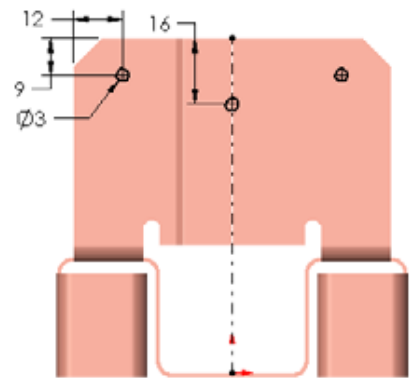
Features in Sheet Metal parts cannot be mirrored individually. To make another tombstone relief cut, copy and paste the existing cut (step 7) and repair the dangling relation.

**15 Break corners.**

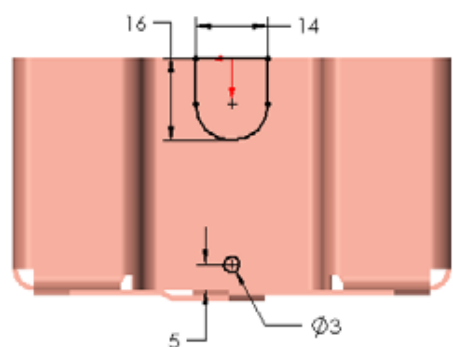
Also, add chamfers using **Break Corner**.

16 Add holes.

Sketch as shown and extrude using the **Through All** end condition.

**17 Cut.**

Sketch and extrude a **Blind** cut as shown. Use **Link to thickness**.

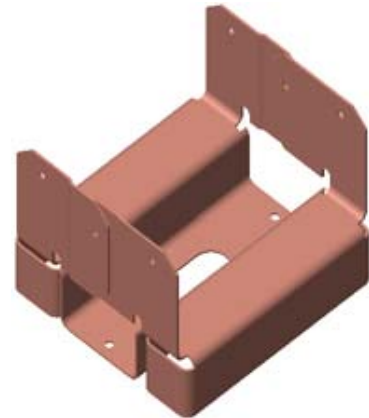


18 Mirror.

Using **Mirror** again, the part is completed.

19 Exit the part.

You can save or discard the changes, as you wish.



Lofted Bends

The **Lofted Bends** tool is used to create a sheet metal part through a lofting process. The resulting part can be flattened and unflattened. This feature has the following requirements:


- Q Only open contours may be used as sketch profiles.
- Q The gaps in the profiles should be aligned for flat pattern accuracy.
- Q The sketches cannot contain sharp corners.
- Q No more than two sketch profiles are allowed.
- Q Guide curve lofts are not supported.
- Q Centerline lofts are not supported.

Introducing: Lofted Bends

Lofted Bends uses two open contour sketch profiles to create a sheet metal loft feature. For more information on loft features, see *Basic Lofting* in the *Lofts* lesson of the *Advanced Part Modeling* manual.

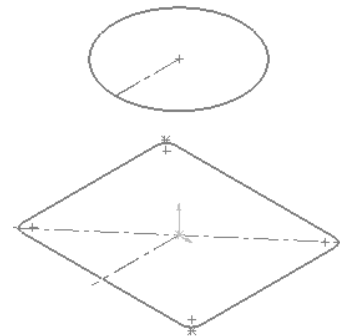
The Design Library contains several pre-made sheet metal parts created with lofted bends. These parts are located in the parts\sheetmetal\lofted bends folder.


Where to Find It

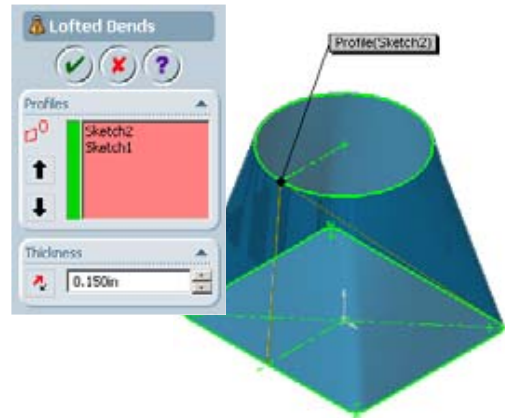
- Q From the Sheet Metal toolbar choose the **Lofted Bends** tool 
- Q Or, click **Insert, Sheet Metal, Lofted Bends....**

1 Open the part named Lofted Bends.

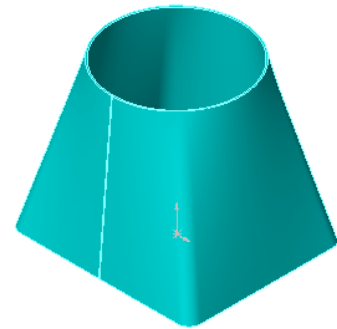
The part **Lofted Bends** consists of two sketches and a related plane.



- 2 Click Lofted Bends** 
Select the profiles near common points to orient the loft. Set the **Thickness** to **0.15 inches**.
Click **OK**.



- 3 Completed feature.**
The completed feature is shaped like a thin walled loft.



Bend Deviation

Generally, lofted bends create deformations in the flat pattern. You can measure these deformations in the **Bend Deviation** PropertyManager. The **Bend Deviation** PropertyManager displays the surface area and curve lengths of the lofted bend.

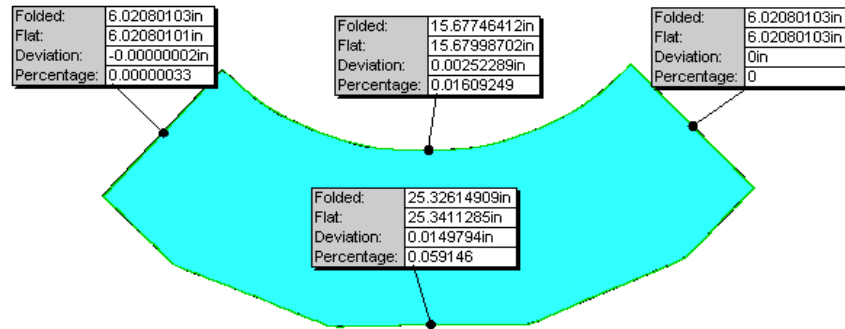
- 4 Unsuppress.**
Right-click on Flat - Pattern1 in the FeatureManager and select **Unsuppress**.



5 **Bend Deviation.**

Right-click on Flatten-<FreeForm Bend>1 and select **Bend Deviation**.

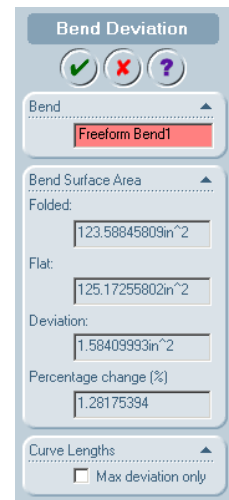
In the graphics area, the corresponding values are displayed for the bend deviation of each edge.



The **Bend Deviation** PropertyManager appears and displays the following:

Bend Surface Area

- Q **Folded**
Surface area of the lofted bend when in the folded state.
- Q **Flat**
Surface area of the lofted bend when in the flattened state.
- Q **Deviation**
Flat value minus the **Folded** value.
- Q **Percentage change (%)**
Deviation value divided by the **Folded** value, multiplied by 100.



Curve Lengths

- Q **Max Deviation Only**
Select this check box to show only the maximum deviation of each edge.

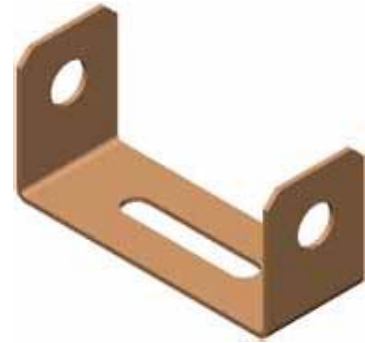
6 **Click OK.**

7 **Exit the part.**

You can save or discard the changes, as you wish.

Exercise 1: Sheet Metal Bends

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.



This lab uses the following skills:

- Q Sheet Metal
- Q Base Flange

Units: **millimeters**

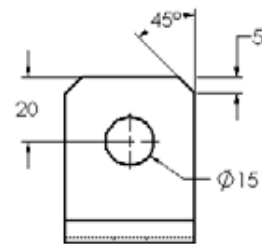
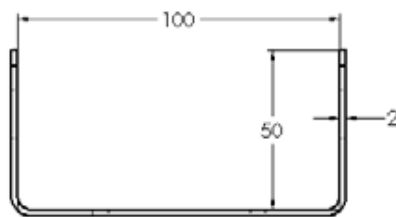
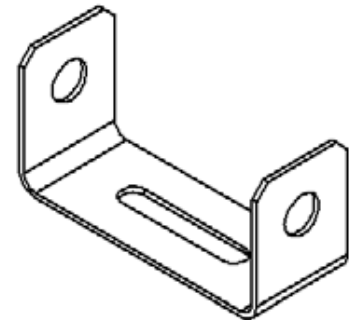
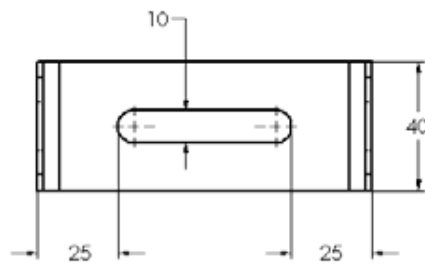
Design Intent

The design intent for this part is as follows:

1. Material thickness is constant **2mm**.
2. Part is symmetrical.
3. Round holes are equal diameter and placement.
4. Bend radius **2mm**.

Dimensioned Views

Use the following graphics with the design intent to create the part.



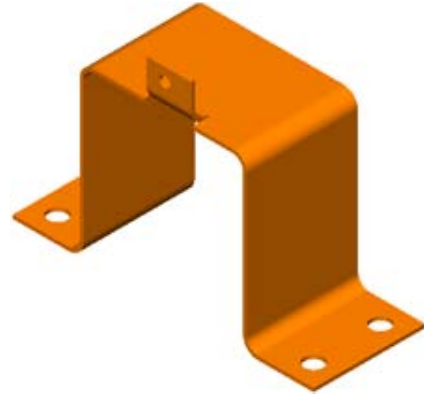
Exercise 2: Sheet Metal Relief

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.

This lab uses the following skills:

- Q Sheet Metal
- Q Base Flange
- Q Edge Flange

Units: **millimeters**



Design Intent

The design intent for this part is as follows:

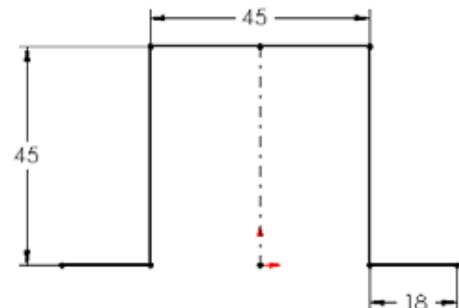
1. Part is formed from a single sheet.
2. Four mounting holes are equal diameter and are located with respect to the ends of the bracket.
3. Bends are equal.
4. Tab is added with bend relief.

Lab Steps

Open a new part using the Part_MM template.

1 Profile sketch.

Using the Front reference plane, sketch the full profile of the part as single lines.



2 Base flange.

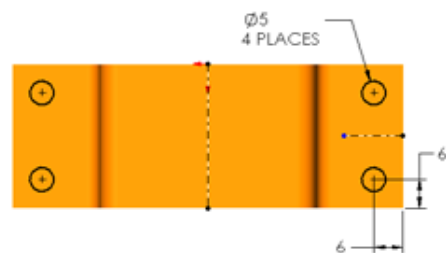
Extrude the **Base Flange** to a depth of **30mm**. Use a **Default Bend Radius** of **3mm** and a **K Factor** of **0.5**.

The **Thickness** is **1.0mm**, offset to the *inside* of the profile.



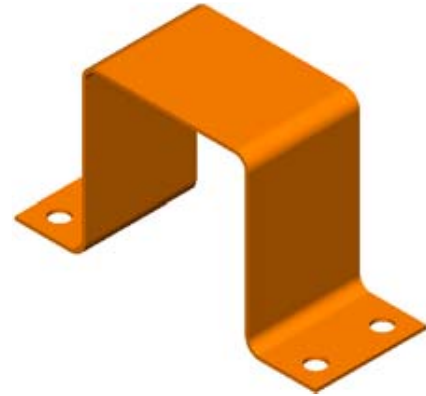
3 Sketch holes.

Using relations, sketch and dimension four **5mm** circles as shown.



4 Create cuts.

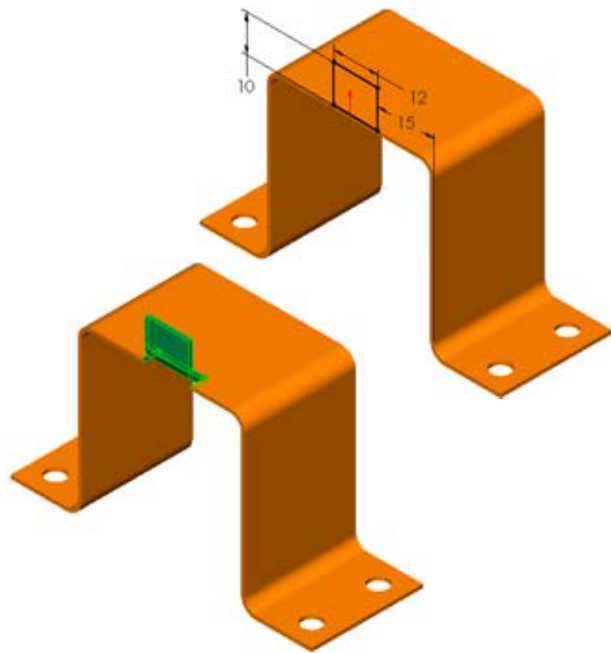
Extrude the sketch **Up to Next** to create the holes.



5 Add an edge flange.

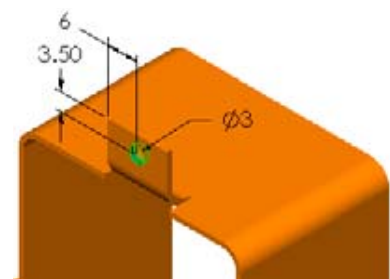
Use the **Edge Flange** tool and add a flange using the dimensions shown at right.

The material is *inside* the model.



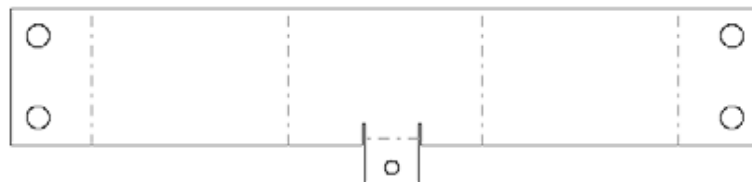
6 Add cut.

Create a circular cut through the tab. Dimension the circle as shown.



7 Flat pattern.

The flat pattern (shown rotated) should appear as illustrated below.



8 Save and close the file.

Exercise 3: Sheet Metal from Flat

Create this part using the dimensions provided. This is a sheet metal part that is designed in the flat.

This lab uses the following skills:

- Q Sheet Metal
- Q Base Flange
- Q Form Features

Units: **millimeters**



Design Intent

The design intent for this part is as follows:

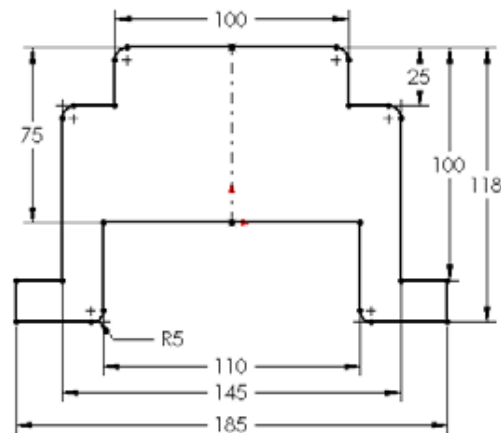
1. Part is formed from a single sheet.
2. Part is designed in the flat.
3. A form feature is used to create the lance.

Lab Steps

Open a new part using the Part_MM template.

1 Profile sketch.

Using the Front reference plane, sketch the full profile of the flat pattern. Use a centerline and mirroring to create the geometry.

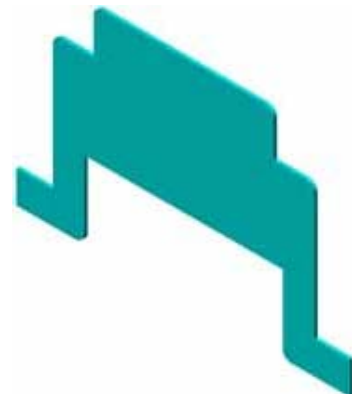


2 Extrusion.

Extrude the **Base Flange** sketch to a depth of **2mm**.

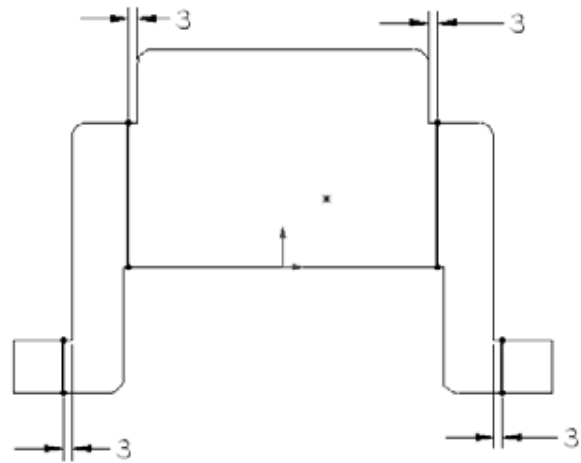
3 Edit Sheet Metal feature.

Set the **Default bend radius** to **2mm**.




4 Add bend lines.

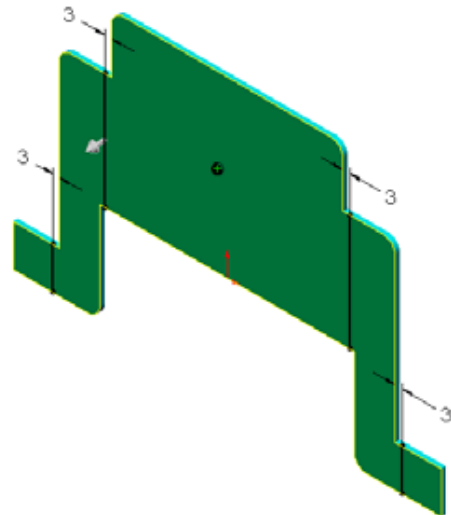
Select the top face of the model and sketch bend lines.



5 Create sketched bends.

Click the **Sketched Bends** tool  and choose the center portion of the face as the **Fixed face**.

All the bends are added as 90° bends, bending up.



6 Cuts.

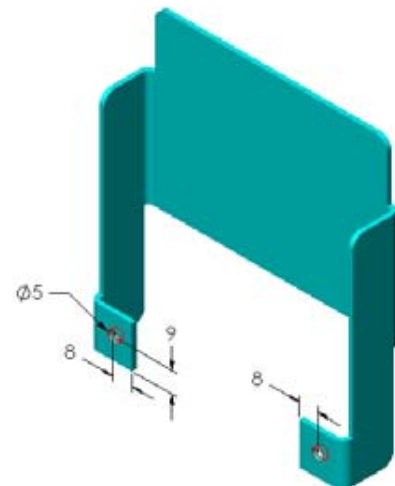
Add circular cuts to the small end faces.

7 Forming tools.

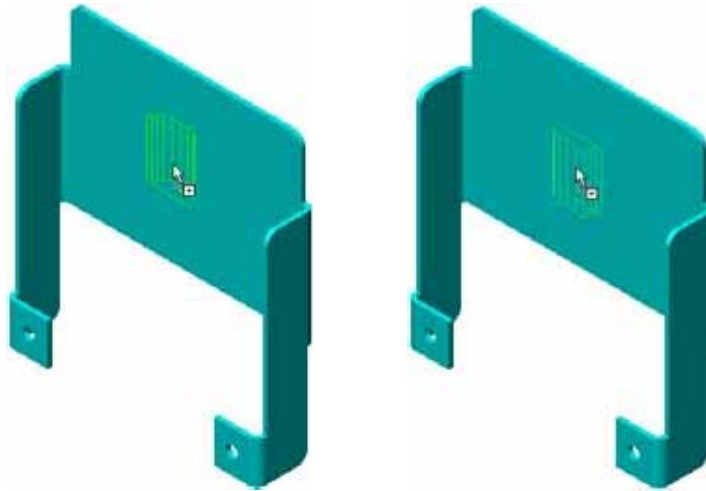
Open the folder **Forming Tools** and the sub-folder **Lances**. One of the lance features, **lance & form shovel**, will be used.

8 Drag and drop feature.

Drag the **lance & form shovel** feature onto the front face of the part; *don't drop it yet.*



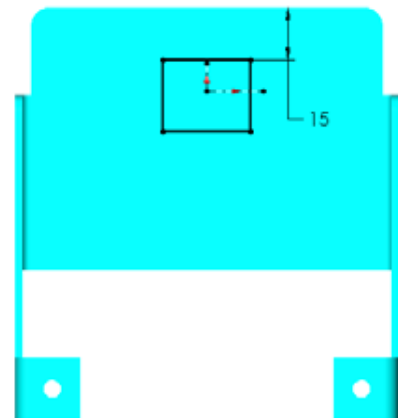
Use the **Tab** key to reverse the direction of the form feature, so that it enters from the opposite side.



9 Sketch.

Use **Modify Sketch** to rotate the sketch. Fully define the sketch by adding a relation to center it and a dimension, as shown.

Click **Finish** to add the feature.

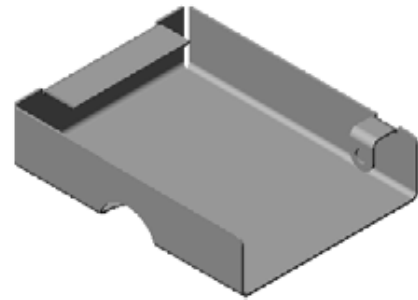


10 Save and close the file.



Exercise 4: Sheet Metal Flanges and Bends

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.



This lab uses the following skills:

- Q Base Flange
- Q Miter Flange
- Q Edge Flange
- Q Fold
- Q Unfold
- Q 3D Bends

Units: **millimeters**

Design Intent

The design intent for this part is as follows:

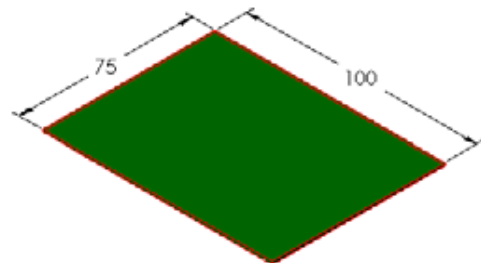
1. All bends are equal radius (**1mm**).
2. Tabs are added with bend relief
3. Part is asymmetrical.

Lab Steps

Open a new part using the Part_MM template.

1 Base flange.

Create the geometry shown using a base flange. The thickness is **1mm** upwards.

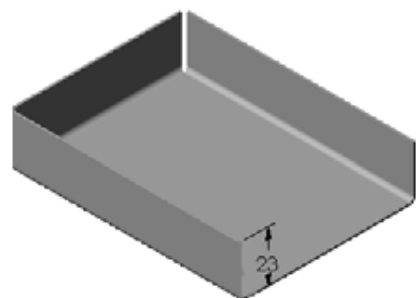


2 Edit Sheet Metal feature.

Set the **Default bend radius** to **1mm**.

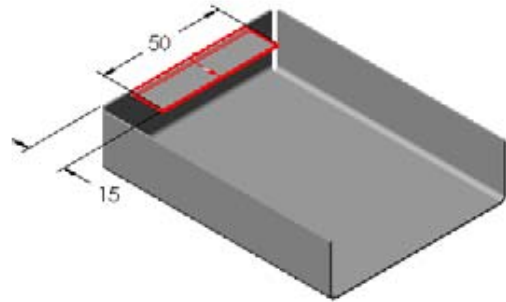
3 Miter flange.

Add a miter flange using the dimension provided. The flange is applied to three sides of the model.



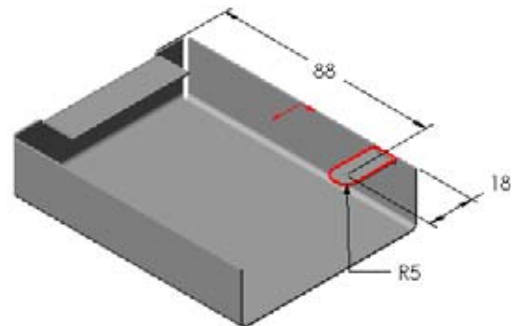
4 Edge flange.

Add an edge flange using the dimensions provided. The tab is flush with the top of the model.



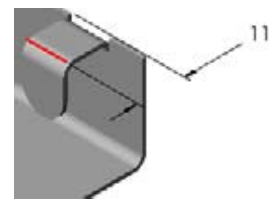
5 Rounded flange.

Using the standard rectangular flange, modify the sketch to have a rounded end.



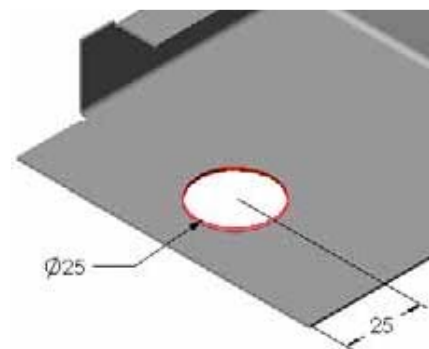
6 Sketched bends.

Create the bend line as dimensioned. Use it to bend the flange.

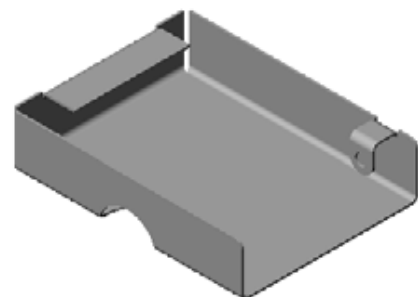


7 Cut across a bend.

Use **Fold** and **Unfold** to flatten a bend of the model and add a cut.



8 Save and close the file.

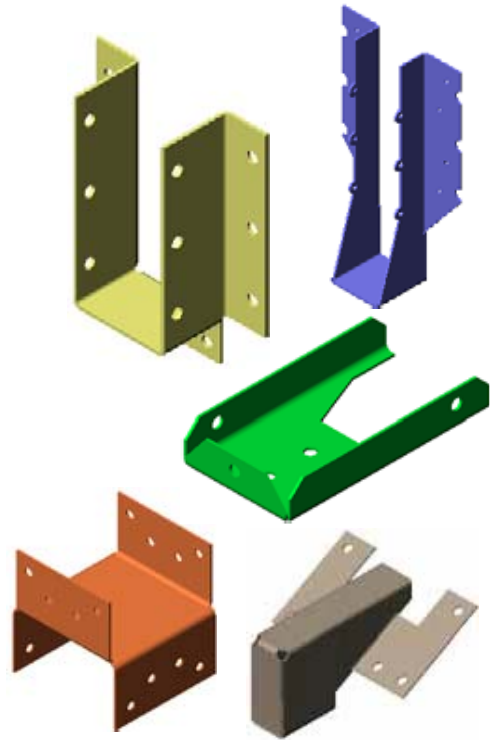


Exercise 5: Assorted Framing Hangers

Create these parts using the drawings provided and your own dimensions. Use sheet metal features.

This lab uses the following skills:

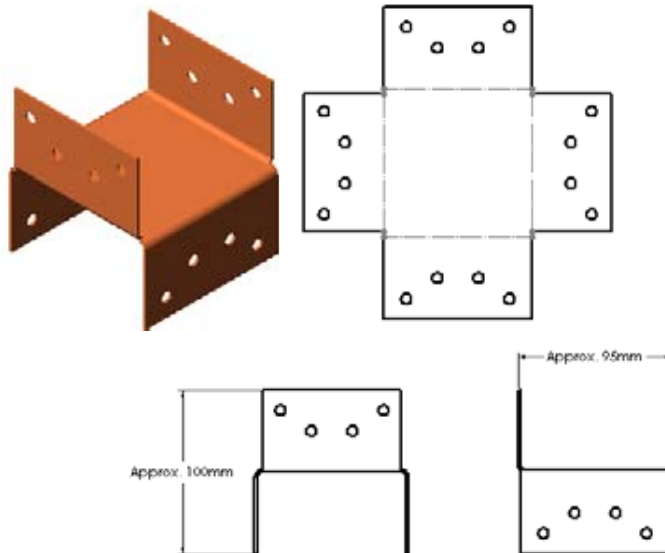
- Q Sheet Metal features



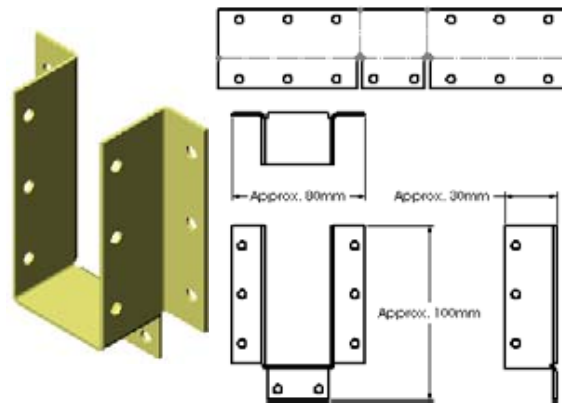
Directions

For the following exercises, use the accompanying graphics to create the part. Add as many or as few details as you wish. Only a few approximate dimensions are given. You should improvise the rest.

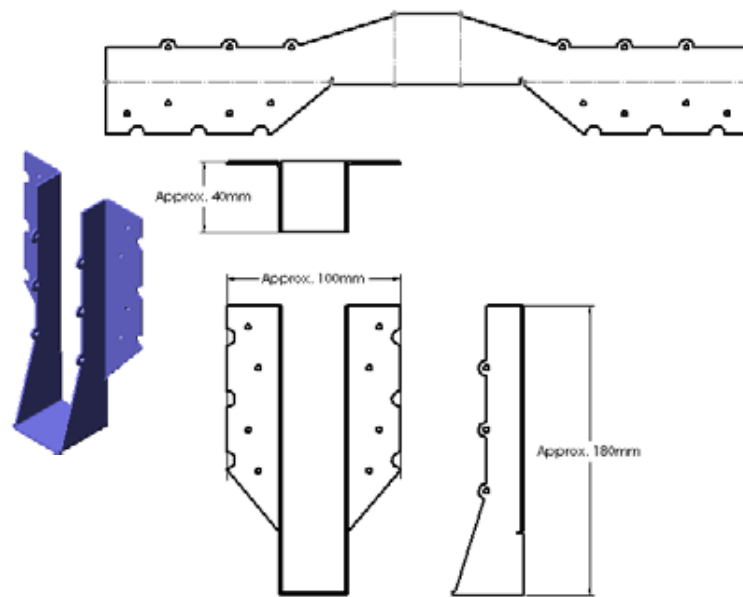
Post Cap



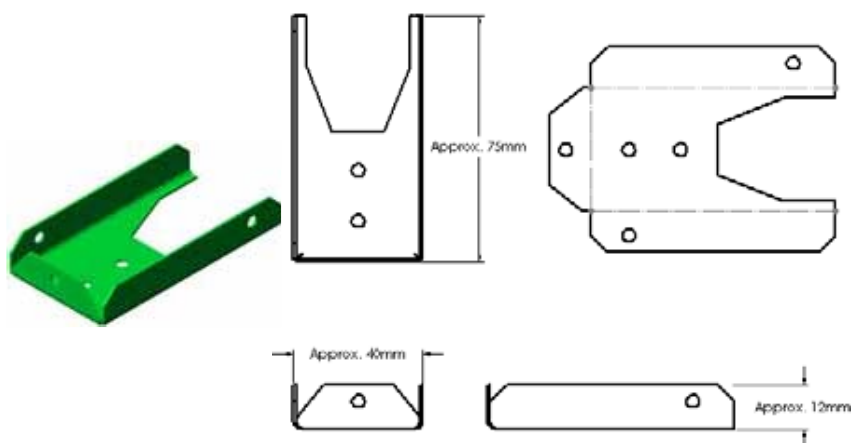
Heavy Duty Joist Hanger



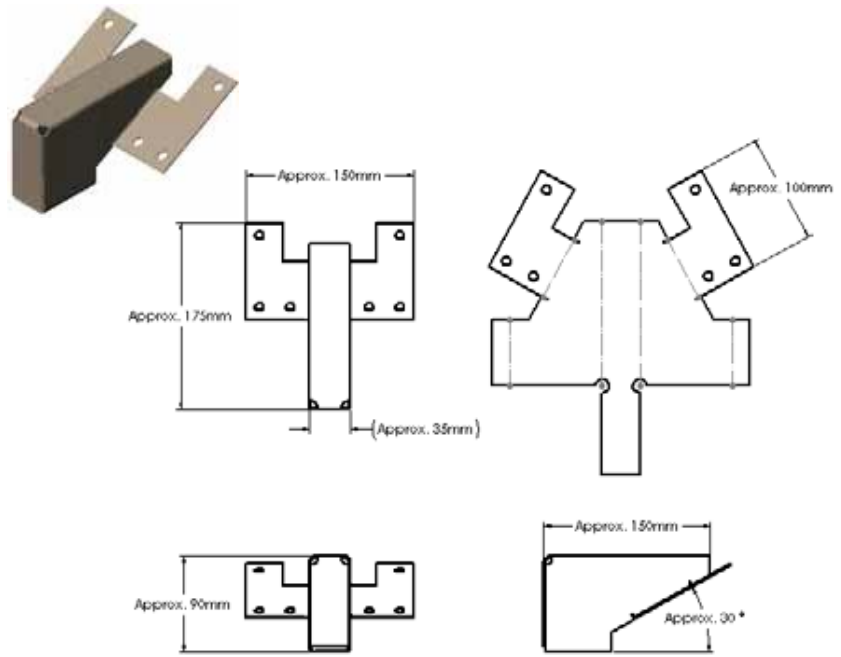
Standard Joist Hanger



End Clip



Roof Bracket



Lesson 2

Converting Parts to Sheet Metal

Upon successful completion of this lesson, you will be able to:

- Q Convert a legacy sheet metal part.
- Q Import an IGES file into SolidWorks.
- Q Recognize the bends in a non-sheet metal part.
- Q Rip the corners of a thin-walled part so that it can be recognized as a sheet metal part.
- Q Display the bend sequence using configurations.

Sheet Metal Topics


This lesson deals with the conversion of parts to sheet metal. These parts are modeled using conventional techniques, and then converted to sheet metal parts by recognizing the bends in the part.

Stages in the Process

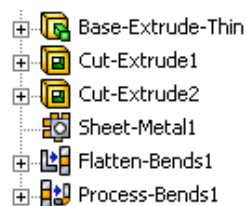
Some key stages in the modeling process are shown in the following list. Each of these topics comprises a section in the lesson.

- Q **Converting legacy sheet metal parts**
Sheet Metal parts created before SolidWorks 2001 contain different sheet metal features than those created with the Base Flange, but they can be converted. This method is useful for parts that were created with shells, cylinders, or cones.
- Q **IGES files**
When an IGES file is opened in SolidWorks, it is converted to a solid model if possible. The model is a “dumb” solid that is made up of a single monolithic feature with no parametric data.
- Q **Rip feature**
The **Rip** tool breaks a solid along a corner to make it suitable for adding bends.
- Q **Recognizing bends**
The **Recognize bends** method takes an existing solid and converts it into a sheet metal part. In doing this bends are added to the model.

Converting Legacy Sheet Metal Parts

SolidWorks parts that were built prior to SolidWorks 2001 used a different method of representing sheet metal features. These parts can be converted to the current method by simply adding a sheet metal-specific feature such as an edge flange or miter flange. Flattening will then be controlled by the  Flat-Feature1 feature.

Each legacy part contains three sheet metal specific features: Sheet-Metal, Flatten-Bends and Process-Bends. These features control the “sharp”, “flat” and “bent” states of the model.

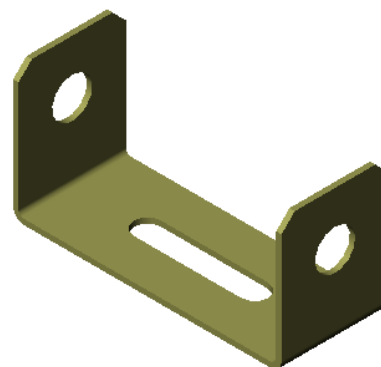


Note

The **Recognize Bends** method will be discussed in detail in the following section.

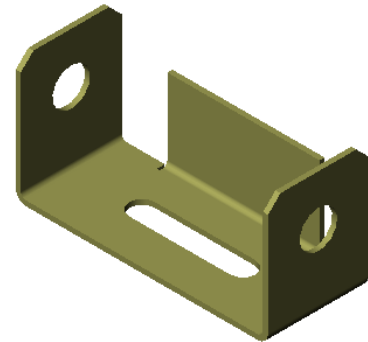
1 Open LegacySM.

The LegacySM part was created as a sheet metal part prior to SolidWorks 2001.



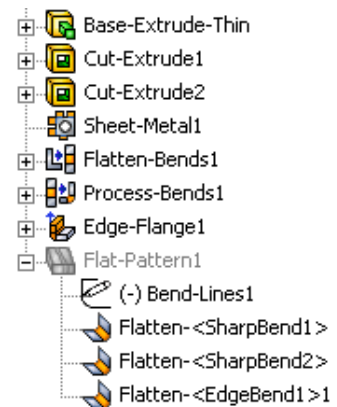
2 Add an Edge Flange.

Add a **Edge Flange** to the model. The size and position is not important at this time.



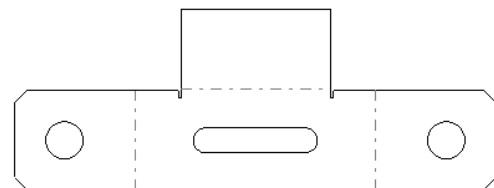
3 FeatureManager.

Two features are added to the model: **Edge-Flange1** and **Flat-Pattern1**. All of the bends in the part have been converted and are listed under the **Flat-Pattern1** feature. The two legacy bends are listed with the **SharpBend** suffix.



4 Flatten.

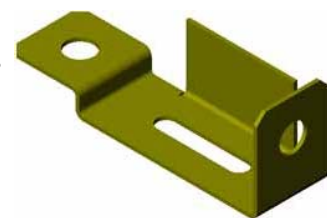
Now the part has been converted and it can be flattened by unsuppressing the **Flat-Pattern1** feature.



Recommended Practices

Once a legacy sheet metal part has been converted to the current format, there are some things you should, and should not, do when making modifications.

1. Adding more bends: Although you can do this by editing **Flat-Sketch1** which is located under the **Process-Bends1** feature, it is not recommended. First of all, editing **Flat-Sketch1** rolls the part back *before* the **Edge Flange**, so the flattened representation of the part is incomplete. This can lead to errors in your design work. Secondly, performance is reduced because you have to wait for the rollback operation.
 Recommended practice: Use the **Sketched Bend** feature to add bends.



2. Editing legacy bend radii: The bends that were part of the legacy part are controlled by the SharpBend features that are embedded under Flatten-Bends1.

Recommended practice: Edit the definition of the SharpBend you want to change.



Recognize Bends Method

Another method of creating sheet metal parts is the **Recognize Bends** method. This method has existed within SolidWorks since SolidWorks 97 and uses three standard sheet metal features to control the states of the sheet metal process. Models never began as sheet metal parts, they were converted to them.

This method is still suited for:

- Q Importing geometry for sheet metal
- Q Shelled SolidWorks parts for sheet metal

Note

Legacy SolidWorks sheet metal parts (pre-SolidWorks 2001) can be converted to the SolidWorks 2001 format as was shown in the preceding example. The addition of a sheet metal feature (Edge Flange, Miter Flange, etc.) completes the conversion automatically.

Importing

In the next example, we will import a neutral format file (IGES) and modify it so that it can be processed as a sheet metal part. The following are the major topics in this example:

- Q Opening an IGES file
- Q Using the Rip feature
- Q Recognizing bends
- Q Sheet Metal part
- Q Auto Relief
- Q Adding features
- Q Additional bends

Opening IGES Files

Files created in, or saved as, other formats can be opened in SolidWorks and edited. SolidWorks uses the standard **Open** dialog to access these files. This example uses an IGES format file that consists of surfaces that are intended to describe a solid.

1 Open an IGES file.

Click **File, Open**, and set **Files of type** to IGES (*.igs;*iges). Select the file IGESimport, but do not click **Open** yet.

2 Options.

Click the **Options** button and use the following settings: **Surface/solid entities**, **Try forming solid(s)**, and **Perform full entity check and repair errors**. Click **OK**.

Note These options can be set for all imports using **Tools, Options, Import Options**.

3 Open.

Click **Open** to open the file as a SolidWorks part. The **Progress...** window appears and status messages are displayed.

Document Template

Importing an IGES file creates a new SolidWorks part document. Somehow, the system has to determine which template to use. There are two choices:

1. You will be prompted to select a template, just as if you were opening a new part document.
2. Or, the system will use a predetermined template.

Which of these events occurs is controlled by **Tools, Options, System Options, Default Templates**.

Report Files

Whenever a file is imported into SolidWorks, a file is generated: the **Report File** (filename .RPT). If there are errors during the import process, a second file is created: the **Error File** (filename .ERR). Both files are text files and can be read with any text editor.

The examples below are from an IGES file that opened in SolidWorks but *failed* to construct a solid.

Contents of the Error File

The **Error File** will list any errors that occurred during the open process. Suggestions and settings are also listed.

An error would typically have a line like that below:

```
WARNING: Unable to create solid from trimmed surfaces.
```

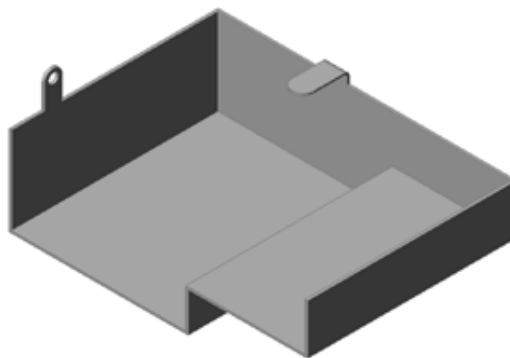
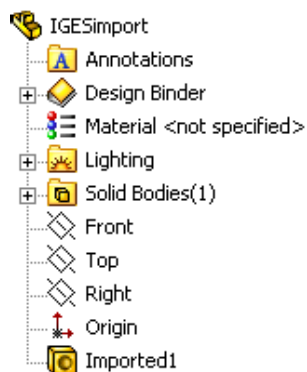
Contents of the Report File

The **Report File** contains:

- Q General information about the IGES file
- Q Entity processing information
- Q A analysis of the IGES file
- Q An entity summary showing the quantity of each type of entity and how many were converted
- Q A results summary

4 Imported solid.

The surfaces have been knitted into a single solid. The single feature is listed as Imported1.



The solid is considered a “dumb” solid because it contains no parametric information or individual features. However you can still add to it using standard SolidWorks sketches and features.

Using the Rip Feature

The **Rip** feature can be used to open a box by cutting the corners of the solid with a very tiny gap (0.05 times the material




thickness) allowing the part to be flattened as a sheet metal part. The rip feature can create three types of corners: shortening either wall or both at the edge where it is applied.

Introducing: Insert Rip

Insert Rip is used to rip edges of the model. Rips can also be created from within the **Insert Bends** command.

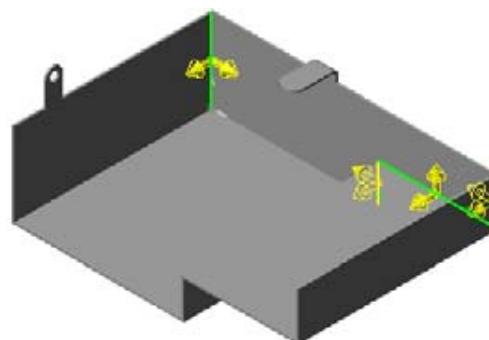
Where to Find It

- Q From the menu, click **Insert, Sheet Metal, Rip...**
- Q From the Sheet Metal toolbar, click the **Insert Rip**  tool.

5 Edge selection.

Click the **Rip** tool and select the edges identified on the right.

Note that all the edges are *inside* edges of the model.



Tip

Before initiating the **Rip** operation, click **Tools, Options**. On the **System Options** tab, click **Display/Selection**. Select the **Allow selection in HLR and shaded modes** check box. This will allow you to select the inside edges of the model even though they are hidden from view.

Note

You will probably want to turn this option off when you are done ripping the model.

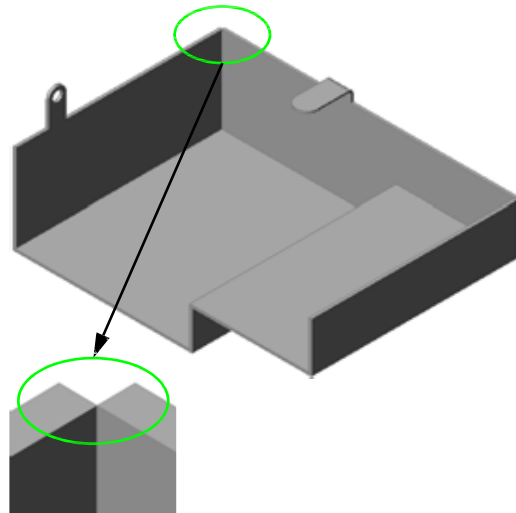
6 Settings.

Use the default settings on the dialog. The change direction button is used to switch between the three types of joints. The default “both” will be used. Click **OK**.



7 Resulting rips.

The select edges are ripped with both flanges being shortened to a common corner.




Adding Bends in Place of Sharp Corners

Introducing: Insert Bends

The next step will be to add bends to the sharp cornered part. During this process you will determine how the bend allowance is calculated and specify the default bend radius.

Insert Bends creates a Sheet Metal part from a thin feature or shelled part. The Sheet Metal, Flatten Bend and Process Bend features are added by this command. These features make editing the bends and flat pattern possible.

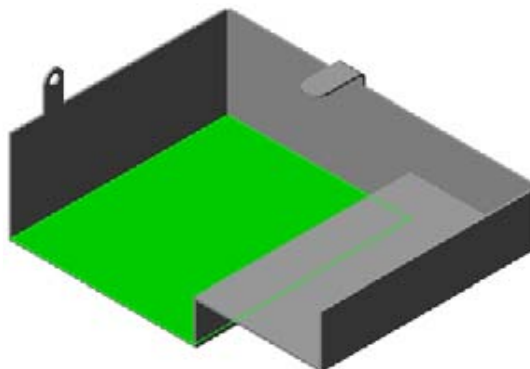
Where to Find It

- Q From the menu choose: **Insert, Sheet Metal, Bends....**
- Q Or, click **Insert Bends**  on the Sheet Metal toolbar.

8 Insert Bends.
 Select the face indicated as the **Fixed face**.

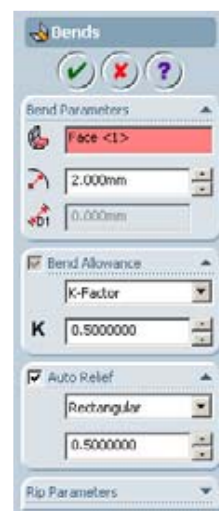
Click **Insert, Sheet Metal, Bends**.

When the system unfolds the sheet metal part, the other faces will be rotated to lie in the plane of the selected stationary face.



9 Sheet Metal settings.
 The settings used are as follows:

- Bend Radius = 2mm**
- Bend allowance = k-factor of 0.5**
- Auto Relief = On**
- Rectangular**
- Offset ratio = 0.5mm**



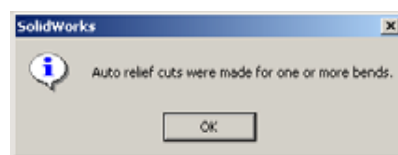
Note

The **Rip Parameters** have the same function as the **Rip** command. This allows you to rip the corners and recognize the bends all in one operation. Since we already ripped the corners of this part, this option will not be used in this step.

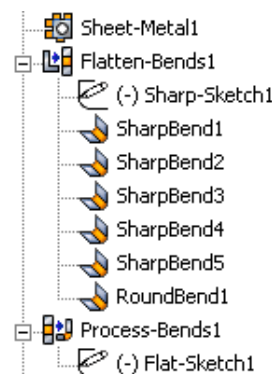
Bend Allowance

To review the options for Bend Allowance, see *Explaining the Bend Parameters* on page 13. Click **OK**.

10 Message.
 The corners created by the rip need relief in order to be folded. The system adds the relief automatically where it is needed.



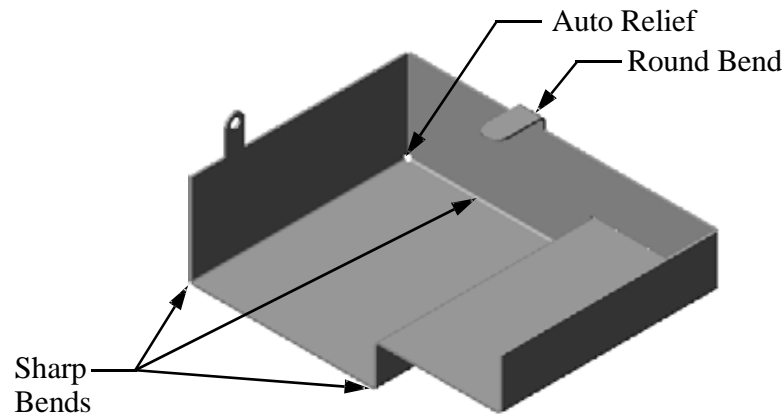
11 Results.
 Three features (Sheet-Metal1, Flatten-Bends1 and Process-Bends1) have been added to the part. They will be explained in the following section.



Sheet Metal Features

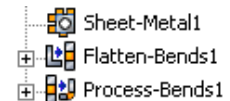
The imported part has been processed or converted into a sheet metal part. To continue working with it, there are a few things you should know. Specifically, what are the specialized sheet metal features, and what do they do?

Sharp corners are converted into **SharpBend** sub-features, using the default **Bend Radius** value. Existing bends are converted into **RoundBend** sub-features, using their current radii. Both types of bends are found under the **Flatten-Bends** feature.



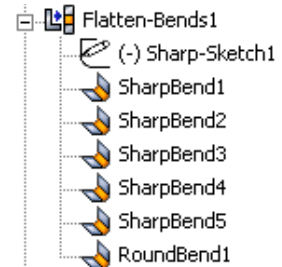
New Features

As a result of the **Insert Bends** operation, several new features appear in the FeatureManager design tree. These represent what can be thought of as a process plan for the sheet metal part. The system applied two distinct operations to the sheet metal part. First, it calculated the bends and created the flat pattern. Then it re-folded the flat pattern to produce the formed representation of the part.

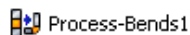


This feature represents the **Insert Bends** operation. You use **Edit Definition** on this feature to regain access to the **Sheet Metal** dialog and makes any changes to the default bend radius, method of calculating the bend allowance, default type of bend relief, or to change which face or edge remains stationary during flattening and folding.

This feature represents the flattened part. This is where the system stores information related to the “conversion” of the **Sharp** and **Round** corners into the **Bends**. If you expand the listing of this feature you will see features for each of the bends that replaced the sharp and rounded corners.



Editing a **Sharp Bend** feature gives you access to the **Bend From Sharp** dialog where you can override the default bend parameters.



This represents the transformation of the flat pattern into the finished, formed part.

Switching Between States

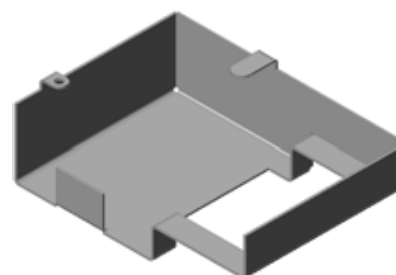
There are two ways to move around within the chronology of the sheet metal process and to switch among the sharp corner, the flattened and the processed states of the part.

- Q Use **Rollback**: Dragging the rollback bar before the Flatten-Bends1 feature represents the part in its sharp corner state. Rolling back to before the Process-Bends1 feature represents the part in its flattened state.
- Q Use tools on the Sheet Metal toolbar. The **No Bends** tool rolls the part back to its sharp corner state. The **Flatten** tool rolls the part back to its flattened state. The nice thing about using the tools is that they act like toggles. Clicking them once rolls the part back. Clicking them a second time rolls the part forward again.

Making Changes

This part requires some modifications. When you have to make changes to legacy type sheet metal parts, there is a simple rule you should follow:

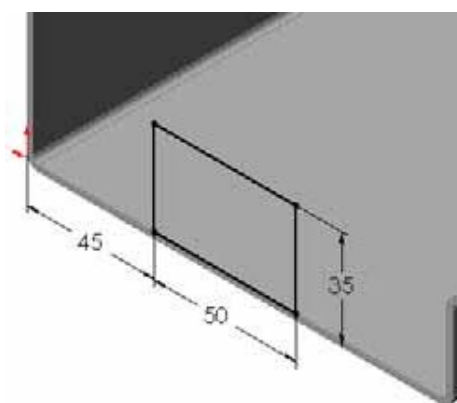
Rule: Make the changes that utilize the new sheet metal features first. For example, in the illustration at the right, add the edge flange *before* attempting to make the rectangular cut across the bends. This converts the part to the new sheet metal format and makes creating the rectangular cut a great deal easier.



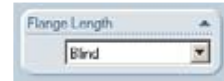
12 Edge flange.

Insert an **Edge Flange**. Edit the flange profile sketch as shown. Settings for the **Angle** and **Flange Position** are set through the PropertyManager.

- Q **Angle = 90°**
- Q **Flange position = Material inside**




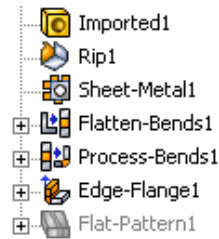
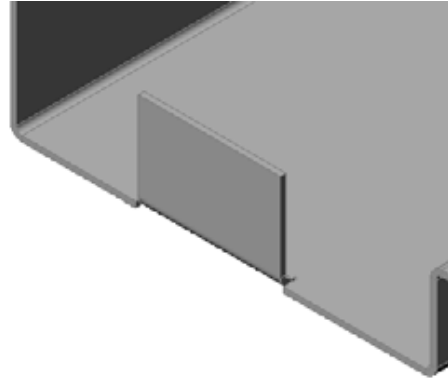
The **Flange Length** will be **Blind**. However, notice that the **Reverse Direction** button and the **Length** field have disappeared from the PropertyManager.



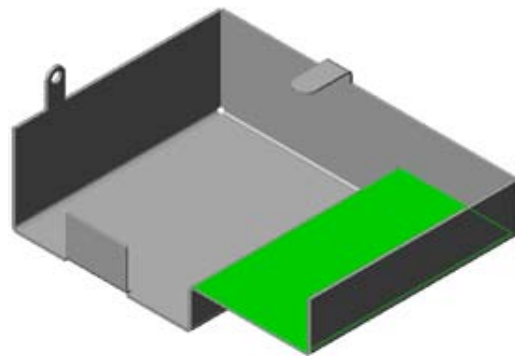
This is because these aspects of the flange are now controlled by the **35mm** dimension in the flange profile sketch.

13 Results.

The system creates the flange and adds the appropriate bend relief. In addition, the system converts the part to the new sheet metal format. This is evident by the appearance of the  Flat-Pattern1 feature.

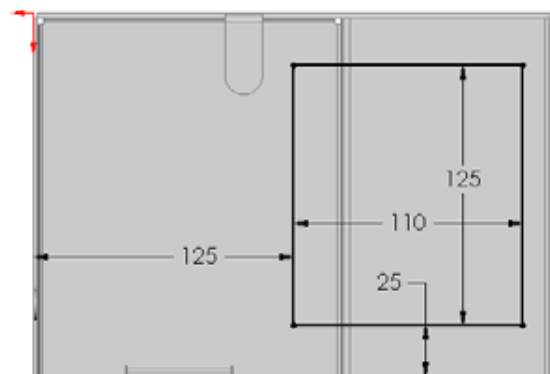


14 Open a sketch on the indicated face.



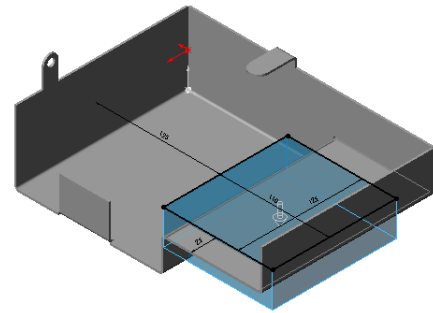
15 Sketch.

Switch to a Top view, and sketch a rectangle using the dimensions shown.




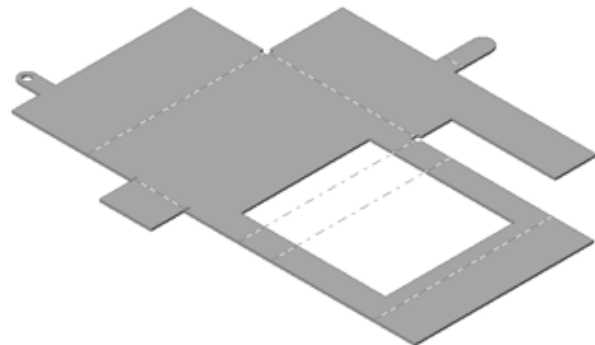
16 Cut.

Use **Through All** end condition.



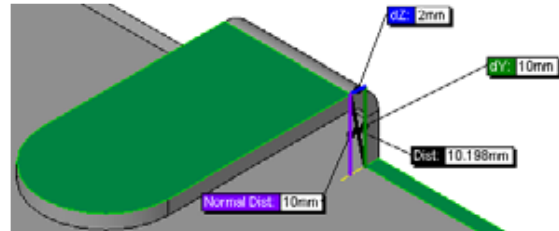
17 Flatten.

Click  to show the part in its flattened state. Because the part has been converted to the new sheet metal format, the cut is automatically positioned in the correct location in the FeatureManager design tree so that it is represented in the flat pattern. Refold the part.



18 Measure.

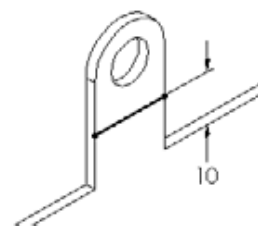
Zoom in on the bent tab and measure the distance between it and the upper edge of the part. The normal distance is **10mm**.




Turn off the **Measure** tool.

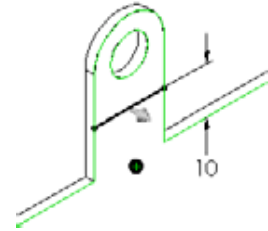
19 Sketch a bend line.


Zoom in on the other tab, with the hole in it. Open a sketch on the inside face of the part, and sketch a line as shown.



20 Sketched Bend.

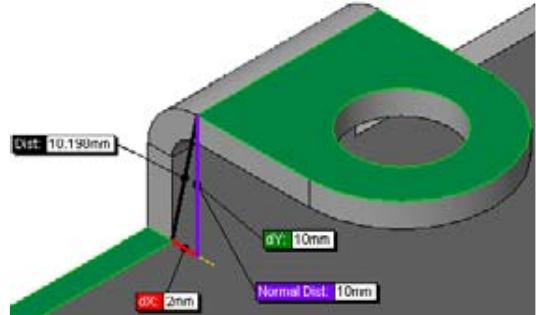
Click the **Sketched Bend** tool  Use the default radius. Select the inside face of the part as the **Fixed face**.



For the **Bend position**, click  **Material Inside**. This means that after the tab is bent, its upper or outside face will line up with the sketched bend line.

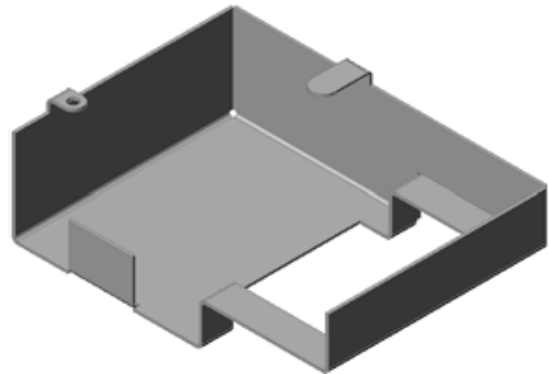
21 Results.

Use **Tools, Measure** to verify that the tab is in fact, **10mm** above the upper edge of the part.



22 Finished part.

The illustration at the right shows the finished part after the three modifications have been made.



23 Save and close the part.


Unrolling Cones and Cylinders

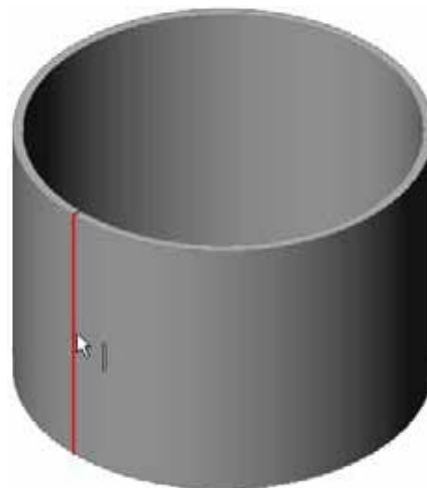
To “unroll” a conical or cylindrical shape, the **Recognize Bends** method must be used. A cut of minimal thickness must be made in the direction of the cone’s center axis and through a wall. The “fixed face” must be an edge for the unrolling operation to work.

1 Open the part Unroll.


The Unroll part contains an arc that was extruded as a thin feature. The arc has a 1° gap, creating a gap in the wall of the cylinder.

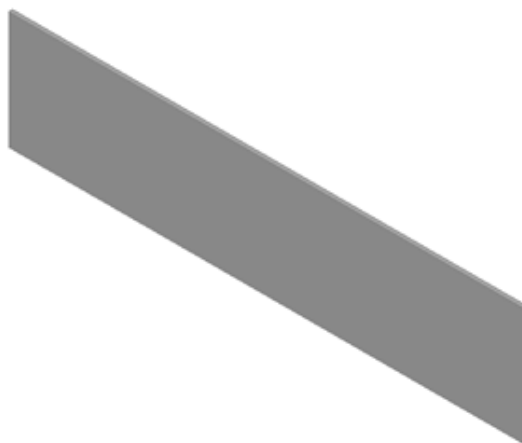
2 Recognize bends.

Click **Insert Bends**  and select the model *edge* indicated. This is the **Fixed edge or face**. The **Bend radius** and the **Auto Relief** options have no bearing on this example. Click **OK**.



3 Flattened.

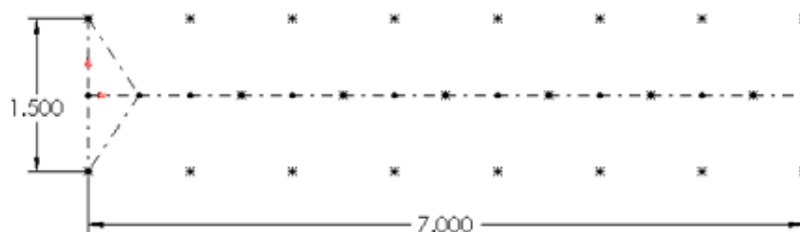
Click **Flatten**  to show the part in its flattened state. Notice that the part is rolled back to *before* the Process-Bends1 feature.



4 Copy a sketch.

Open the part Pattern Sketch. It contains a sketch that will be used for a sketch driven pattern.

Select the sketch and copy it to the clipboard. You can use **Edit, Copy**, or press **Ctrl+C**.



5 Paste the sketch.

Switch back to the unrolled cylinder part window.

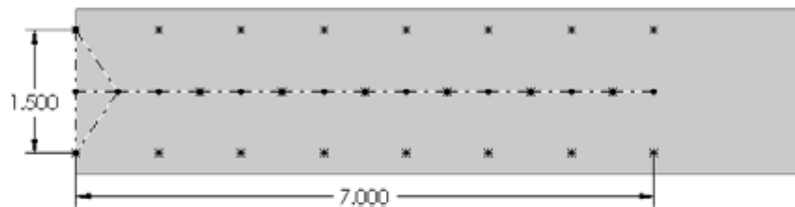
Change to a Front view.

Click the flattened face, and paste the sketch.



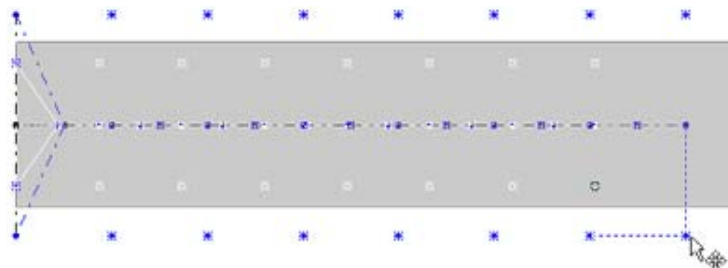
6 Edit the sketch.

Add a **Midpoint** relation between the leftmost end of the horizontal centerline and the leftmost vertical edge of the part.



7 Delete the dimensions.

Delete the two dimensions. Drag one of the sketch points to understand what geometric relations are present, and how the sketch behaves.



8 Midpoint relation and dimension.

Add a **Midpoint** relation between the rightmost end of the horizontal centerline and the rightmost vertical edge of the part.

Add a linear dimension as shown. The sketch is fully defined.



9 Exit the sketch.

10 Sketch.

Select the flat face of the part and insert a *new* sketch. Sketch a circle and dimension it as shown.

**11 Cut.**

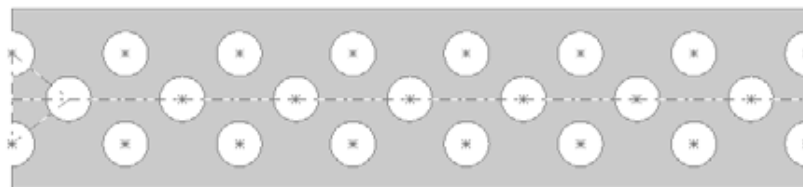
Extrude the circle as a cut. Use **Link to thickness**.

**12 Pattern.**


Click **Insert, Pattern/Mirror, Sketch Driven Pattern**, or click **Sketch Driven Pattern**  on the Features toolbar.

For **Features to Pattern**, select the cut you made in the previous step. Click in the **Selections** list, and select the pattern sketch.

Click **OK**.

**13 Results.**

Hide the pattern sketch, and click

Flatten  to re-roll the part.

14 Save and close.

Process Plans

Adding configurations to the sheet metal part allows you to suppress or unsuppress one or more bends at a time.

Bends

Bends are created in many sheet metal features. They are listed as either Bends or Flange-Bends in the FeatureManager design tree.

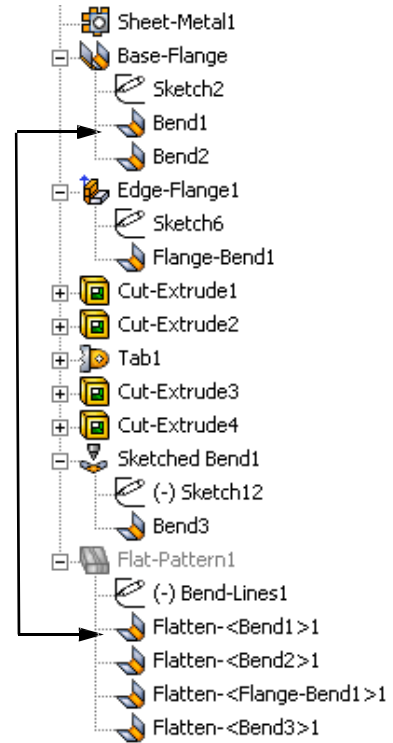
1 Where are the bends?

Open the part ProcessPlan. Expand the features to find the bends. The bends Bend1 and Bend2 created under the Base-Flange feature are converted sharp bends.

The bend under Edge-Flange1 named Flange-Bend1 is an additional bend added by the **Edge Flange** tool.

Bend3 was created during the **Sketched Bend** procedure.

The bends are linked to those listed under the Flat-Pattern1 feature. The arrows show that Bend1 is the same bend as Flatten-<Bend1>1.



Configurations

Configurations are used to capture different representations of the sheet metal part.


2 Configurations.

The only configuration in the part is FOLDED.

3 Make a new configuration.

Add a configuration and name it FLAT.

4 Flatten the part.

Click  to flatten the part. This *unsuppresses* the Flat-Pattern1 feature. There are now two configurations — FLAT and FOLDED.



5 Copy the FLAT configuration.

Make three copies of the FLAT configuration. Name them:

Q STEP1

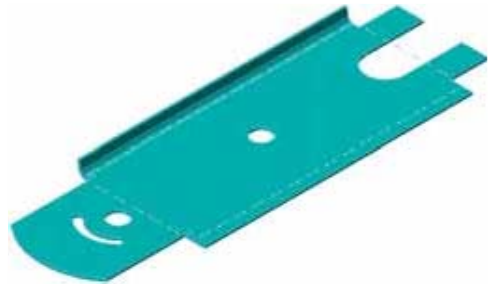
Q STEP2

Q STEP3

6 Create STEP1.

Make STEP1 the active configuration. The part should still be in its flattened state.

Suppress the
Flatten-<Bend1>1
feature.

**7 Create STEP2.**

Make STEP2 the active configuration.

Suppress:
Flatten-<Bend1>1
Flatten-<Bend2>1

**8 Create STEP3.**

Make STEP3 the active configuration.

Suppress:
Flatten-<Bend1>1
Flatten-<Bend2>1
Flatten-<Flange-Bend1>1

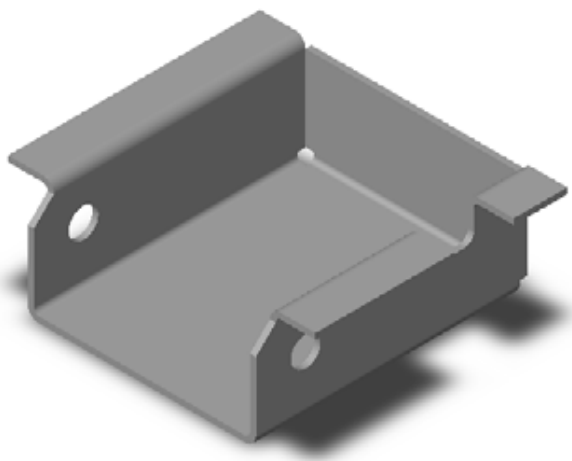


Exercise 6: Importing and Converting

Use an existing IGES file and create a sheet metal part.

This lab reinforces the following skills:

- Q Import
- Q Recognizing Bends
- Q Converting Sheet Metal models
- Q Corner Trim



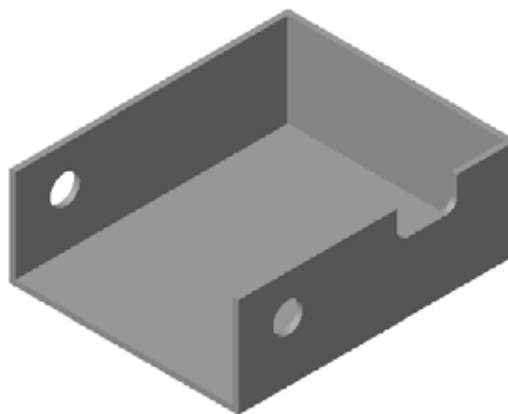
Procedure

Use the following procedure.

1 Open an existing IGES file.

Open the existing IGES file named `igesLab` using the default setting of **Try Forming Solid(s)**.

A thin walled solid is created.

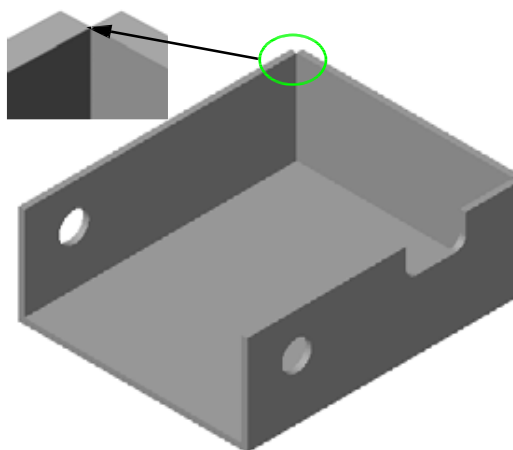


2 Units.

Set the units of the model to **inches**.

3 Rips.

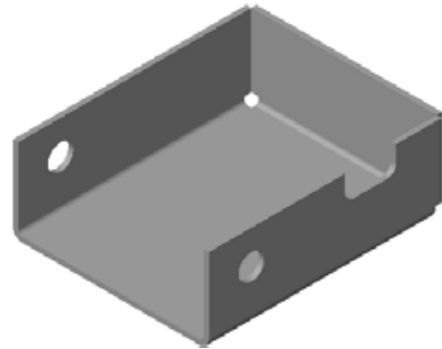
Rip the two rear corners of the solid. The rips should cut both flanges.




4 Recognize Bends.

Use the **Insert Bends** tool to recognize the bends using a **Bend Radius** of **1/16"**.

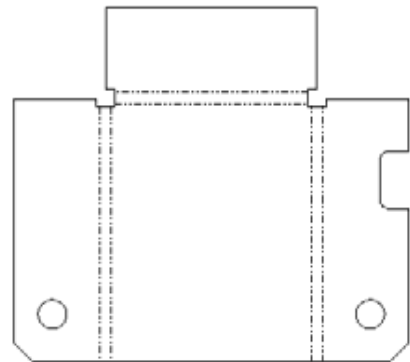
Note the auto relief cuts have been made in addition to the bends.



5 Flat Pattern.

Click **Flatten**  to flatten the part. The flat pattern appears.

Click **Flatten**  again to refold the part.

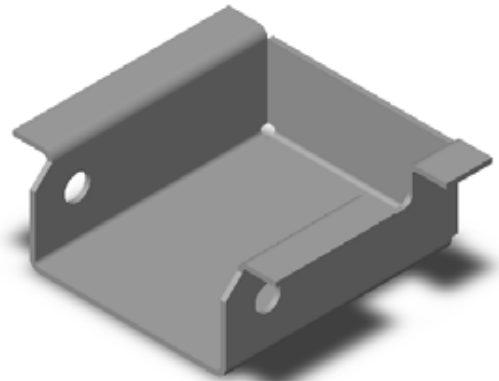


6 Modifications.


Break the corners with a **0.25"** chamfer.

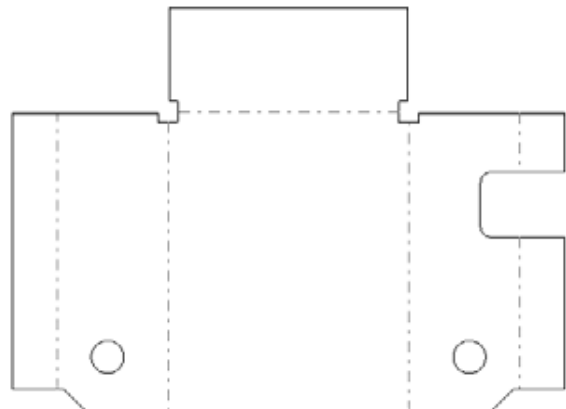
Use the **Edge Flange** tool to add three flanges.

Use flanges that are **1/2"** wide with **Bends Outside**.

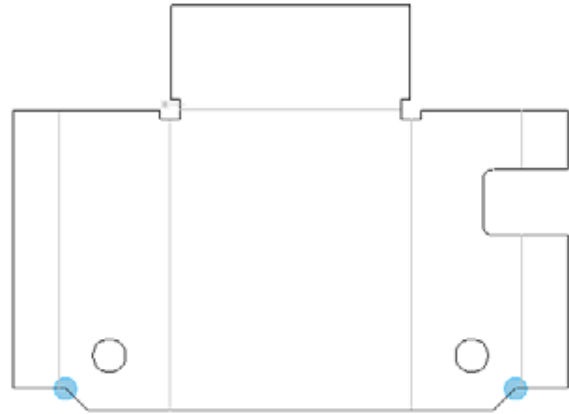


7 Flatten.

Click **Flatten**  to flatten the part.



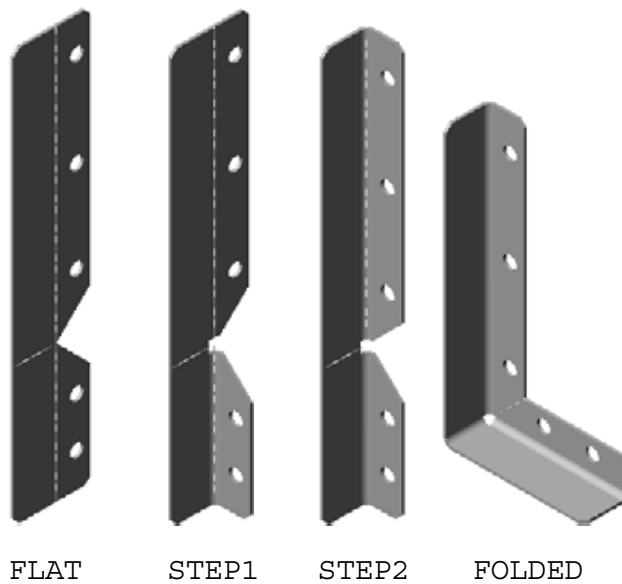
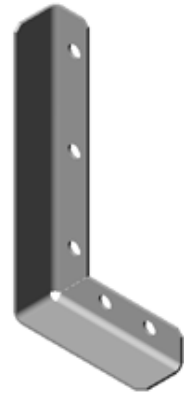
- 8 **Corner-Trim.**
Add a **Corner-Trim**
feature with **Circular**
Relief Type as shown.
Use a diameter of
0.125".
- 9 **Save and close.**



Exercise 7: Process Planning

Modify this part to add configurations that represent the process plan. Suppress and unsuppress features and bends to create the differences between the configurations. This lab uses the following skills:

- Q Adding configurations
- Q Suppressing and unsuppressing features



Lab Steps

Open the existing part named ProcessPlanning.

- 1 Bends.**
Locate the bends within the FeatureManager tree. Expand the Flat-Pattern1 feature to see them. There are three bend features.
- 2 Configurations.**
Create configurations (see above) to capture stages of the bending process.
- 3 Individual configurations.**
In each new configuration, *unsuppress* the Flat-Pattern1 feature and *suppress* the bends that you want to see.
- 4 Save and close the file.**



Lesson 3

Modeling Sheet Metal In the Context of an Assembly

Upon successful completion of this lesson, you will be able to:

- Q Create new sheet metal parts in the context of an assembly.
- Q Add edge flanges with in-context references.
- Q Add hems to edges.
- Q Edit miter flanges and modify the start and end offsets.

Sheet Metal Topics

This lesson deals with creating sheet metal parts in the context of an assembly. Sheet metal features use some of the same in-context options as standard features.

Top-Down Assembly Modeling

SolidWorks supports two assembly modeling methods: *Bottom-Up* and *Top-Down*. Bottom-Up assembly modeling was covered in the basic course *SolidWorks Essentials: Parts and Assemblies*, and consists of assembling pre-built parts in an assembly file.

There is considerable power and advantage to being able to design and model parts in the context of the assembly where they will be used. This is called Top-Down assembly modeling.

There is More to Learn

Designing and building component parts in the context of an assembly is a broad topic. There is much more to assembly-centric design than is covered in this lesson. For a more in-depth exploration of top-down assembly design, you should take the course *Advanced Assembly Modeling*.

Stages in the Process

Some key stages in the modeling process are shown in the following list. Each of these topics comprises a section in the lesson.

- Q **Insert New Parts**
Sheet metal parts can be created in context like other SolidWorks parts.
- Q **Edit Component**
Edit Component is used to access the sheet metal features from the assembly context.
- Q **Base Flange**
Base flanges have the same end conditions as a standard extrusion. **Up to Next**, **Up To Surface** and **Offset From Surface** are available.
- Q **Edge Flange.**
Using the **Offset** option allows you to attach an edge flange to existing faces in the assembly.
- Q **Flange position**
The **Flange position** options for miter and edge flanges allow you to position the flanges relative to the part.

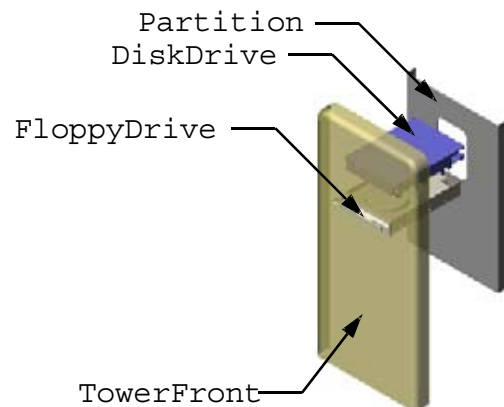
In-Context Sheet Metal Parts

Sheet Metal parts can have in-context features or be created in the context of an assembly. The advantages of using the top down approach are apparent with sheet metal. There are also some special considerations you should keep in mind.

In this example, an enclosure for a disk and floppy drive is created. The enclosure is a sheet metal part that is attached to the TowerFront and Partition components.

- 1 Open assembly.**
Open the existing assembly TopDownSM. It contains four components.

Hide the TowerFront component.




Adding a New Part into an Assembly

New parts can be added to an assembly as needed. These new parts can be created in the context of the assembly, using the geometry and locations of existing parts to build upon. They will appear in the FeatureManager design tree as component parts, with a full listing of their features.

Introducing: Insert Component

Insert, Component, New Part creates a new part and component in the assembly. The new part is named and then mated to a plane or planar face of an existing part in the assembly.

Where to Find It

- Q From the menu choose: **Insert, Component, New Part...**
- Q Or, click **New Part**  on the Assembly toolbar.

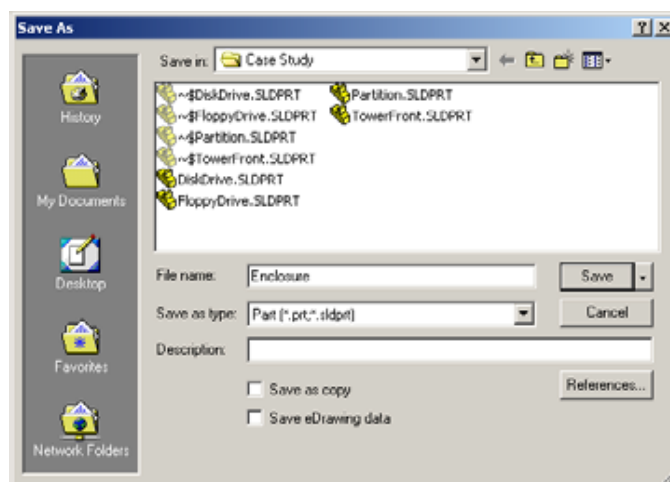
Default Template

Since this command creates a new part document, you have the option of specifying a template or allowing the system to use the default template. This choice is determined by means of **Tools, Options, System Options, Default Templates**.

2 Click Insert, Component, New Part...

The **Save As** dialog box opens. Enter the name **Enclosure** in the **File Name** field. You can also create and change directories to put the file in, if required.

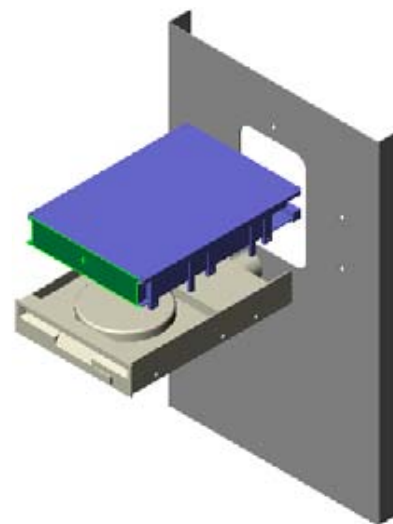
Click **Save**.

**3 The face/plane cursor.**

A new cursor appears, indicating that a plane or planar face must be selected. In the next step, a planar face will be selected.

4 Plane selection.

Select the front face of the **DiskDrive**. This plane will correspond to the **Front** plane of the new part.

**Results of Insert, Component, New Part**

When a new part is inserted into an assembly, several things happen:

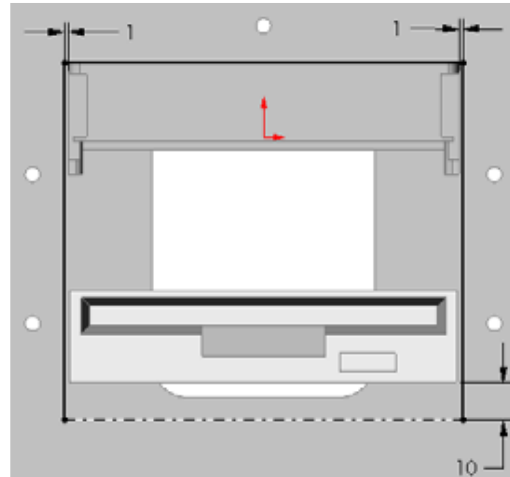
- Q The new part is created.
- Q The new part appears in the FeatureManager as a component of the assembly.

- Q The Front reference plane of the new part is made coincident with the face or plane that you selected.
- Q You are switched into edit part mode.
- Q A new sketch is opened on the selected face.
- Q A new mate, named InPlace1, is added fully defining the component.

5 Sketch.

Click  **Shaded with Edges**.

Sketch the Enclosure profile using edges of the existing components.



6 Base Flange.


Extrude the sketch **Up To Surface** (Partition) using the **Base Flange** tool.

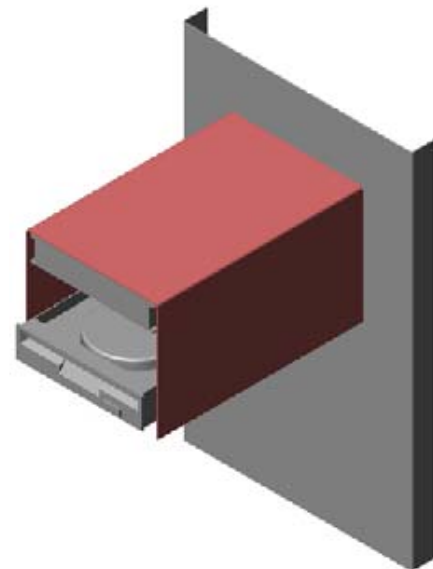
Thickness = 1mm

Make sure the material is added to the *outside* of the sketch. Otherwise, the flange will interfere with the DiskDrive.

Bend Radius = 1mm


The external references created between components are listed as features at the bottom of the FeatureManager design tree

 Update Sketch1 in Enclosure. They are called **Update Holders**.



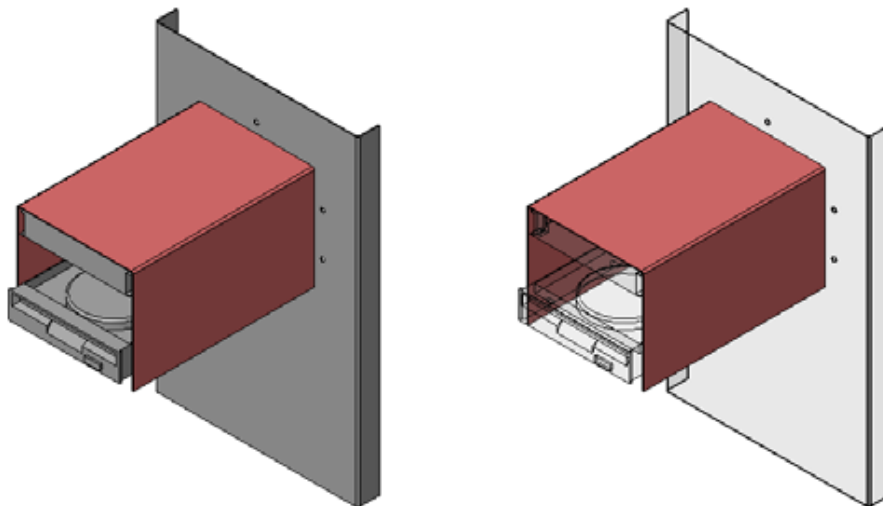
Important!


Tip

The display of the **Update Holders** can be toggled on and off. While in **Edit Assembly** mode, right-click the top-level assembly icon  , and select **Hide Update Holders** from the shortcut menu.

Appearance During Part Editing

You can control the appearance of the parts that are *not* being edited. The assembly can either be opaque or transparent. If some components already have transparency applied to them, that can be maintained giving a mixture of opaque and transparent components.



To switch between opaque and transparent displays, click **Assembly Transparency**  on the Assembly toolbar and from the menu, select either:

- Q **Opaque**
- Q **Maintain Transparency**
- Q **Force Transparency**

Or, right-click any blank area in the graphics window and click **Assembly Transparency**. On the shortcut menu select **Opaque**, **Maintain Transparency**, or **Force Transparency**

To set the degree of transparency, click **Tools, Options, System Options, Display/Selection**. Under **Assembly transparency for in-context edit**, select **Force assembly transparency**, and move the slider to the desired transparency level. The illustration above shows 90% transparency.

Color of the Part Being Edited

To change the color of the part being edited:


1. Click **Tools, Options, System Options, Colors**. In the list of **System colors**, scroll down and select **Assembly, Edit Part**.
2. Click **Edit** and select the desired color.
3. Click **OK** to close the color palette.
4. Select **Use specified colors when editing parts in assemblies**.
5. Click **OK** to close the **Options** dialog box.

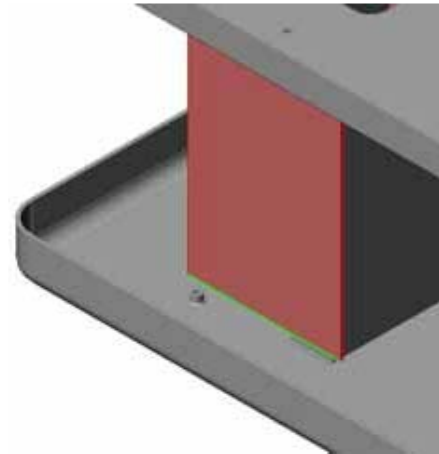
Building Edge Flanges In Context

Edge flanges can be created in context by using external references for the flange profile and offset position. Next we will add an **Edge Flange** to the Enclosure.

7 Edge Flange.

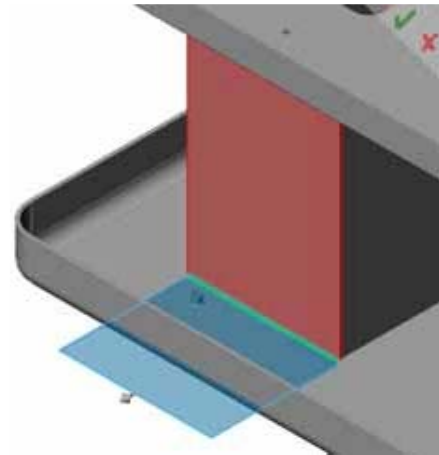
Show the TowerFront and zoom in on the upper boss.

Select the lower edge of the Enclosure, and click  to create an edge flange.




8 Direction.

Drag the edge flange out away from the model.




9 Flange Position.

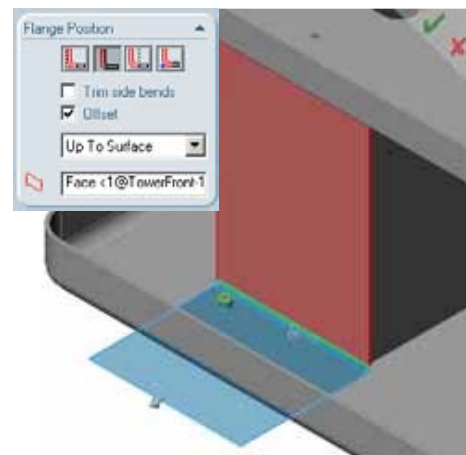
Expand the flange position section and change the settings:

Position =  **Material Outside**

Offset = Enabled

End Condition = Up To Surface

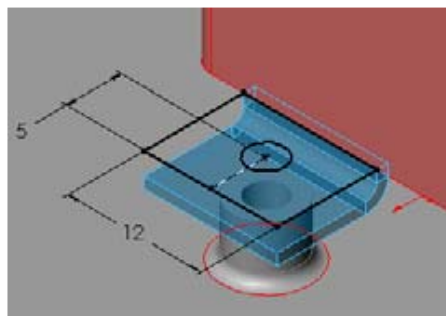
 **(Surface)** = Click the top face of the circular boss.



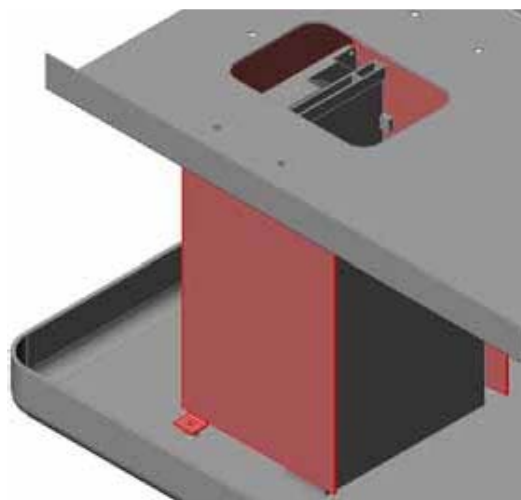
10 Flange Profile.

Click **Edit flange profile**, and center the flange profile over the circular boss.

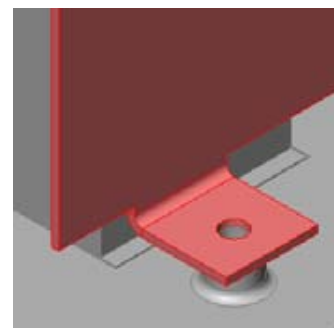
Use **Convert Entities** to copy the edge of the hole in the boss into the sketch of the flange profile. Click **OK**.

**11 Completed edge flange.**

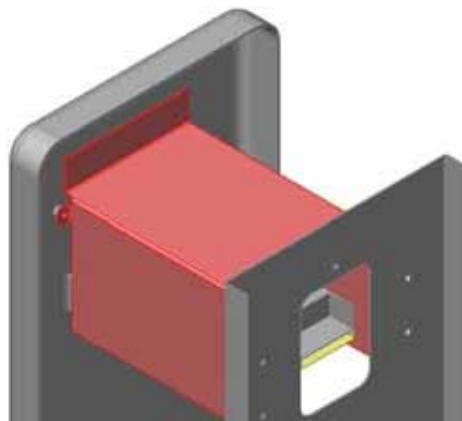
The completed flange is shown at right. It includes a circular hole that is the same size as the hole in the boss.

**12 Second flange.**

Create a second edge flange on the opposite side. Use the same procedure and settings.

**13 Top flange.**

Add an edge flange on the top edge of the part. Use the **Offset** and **Up To Surface** options (select the inside face of the TowerFront part), and a **Flange Length** of 25mm.



Building Miter Flanges In Context

Miter Flanges can also be created in context. Although the flanges cannot be offset like the edge flange, the position of the flange can be controlled relative to the material of the part.

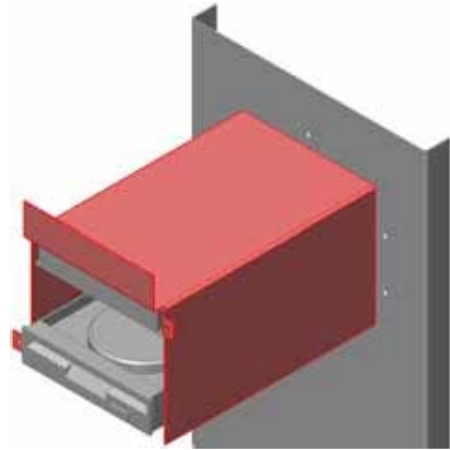
14 Zoom in.

Switch back to the standard isometric view and zoom in on the rear of the part.

Remember that the base flange of the Enclosure was extruded **Up to Surface**. It ends at the face of the Partition.

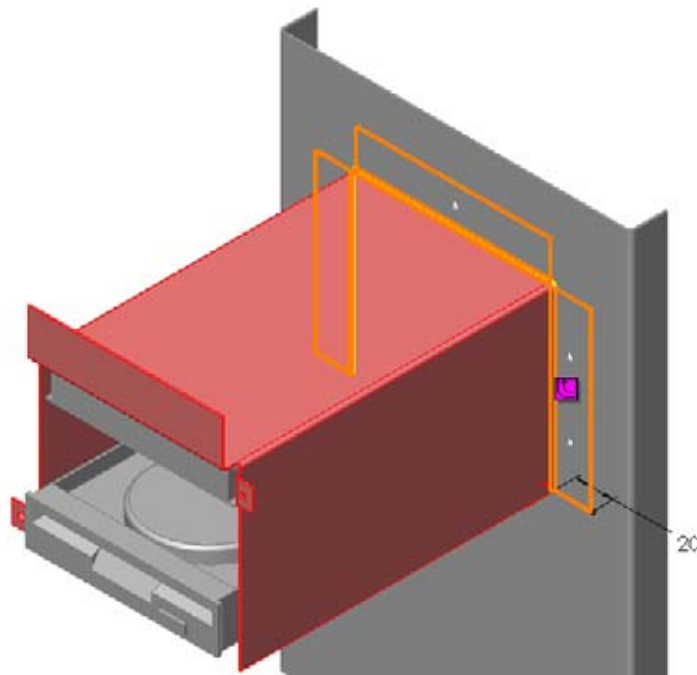
Note

The TowerFront is hidden in this and the following illustrations.



15 Flange sketch.







Create a single line sketch for the miter flange. Propagate the flanges around all three sides of the Enclosure.



16 Flange Position.

Change to a Top orientation to choose a **Flange Position**.

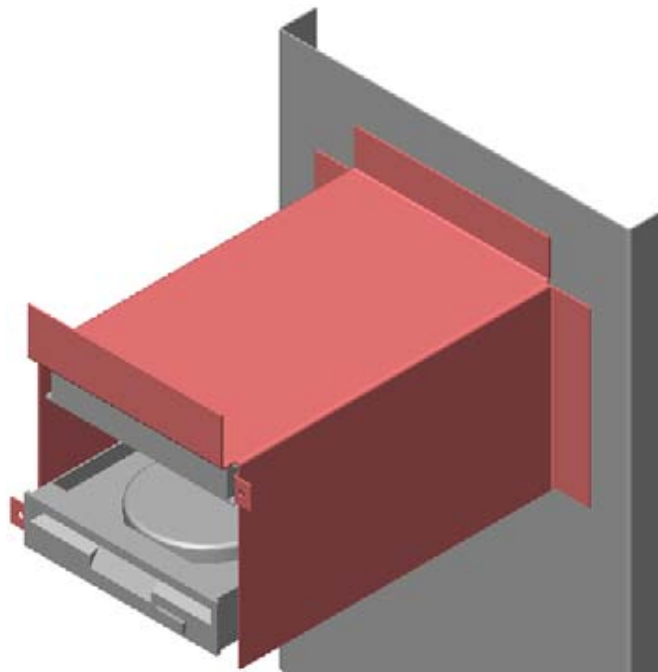
The three options are:

- Q **Material Inside**  
- Q **Material Outside**  
- Q **Bend Outside**  

The **Material Inside** option is the correct one.

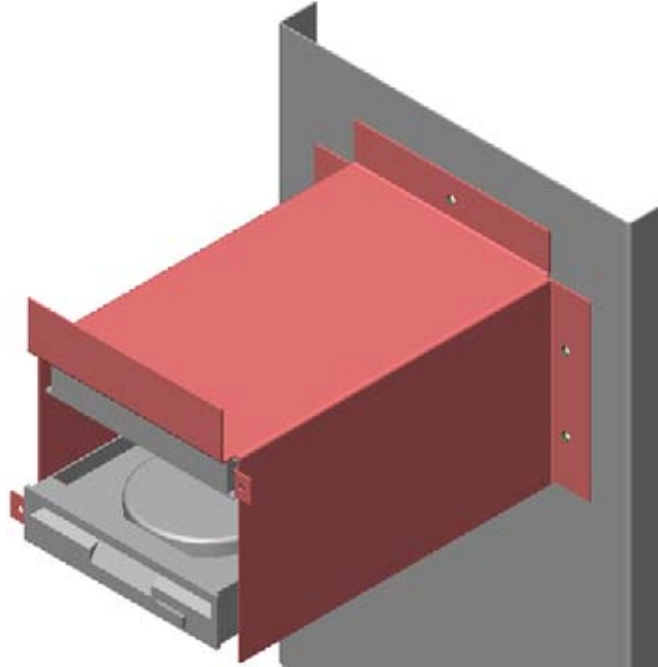
17 Completed miter flange.

Click **OK** to create the flange.



18 In-context holes.

Add in-context holes to the flanges. Convert the edges of the holes in the Partition and extrude a cut using the options **Blind** and **Link to thickness**.



Additional Cut Features

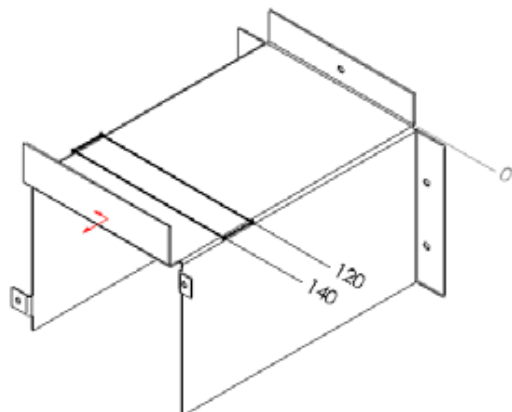
Some additional cut features and patterns will be added to the part. These features have no external references and can be done either in the context of the assembly, or from within the part itself. Since there is no real advantage to making these features in the context of the assembly, we will open the part.

The first cut is across bend lines but does not require the model to be flattened. The second uses a standard Design Library shape.

19 Open the part named Enclosure.

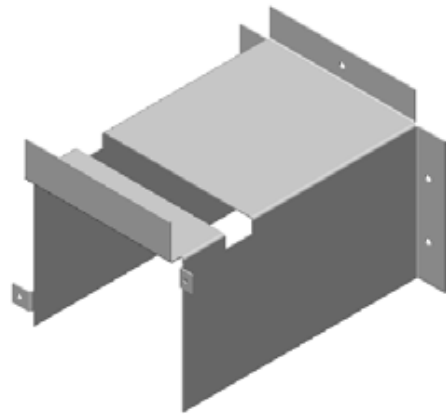
Right-click the Enclosure, and select **Open Part** from the short-cut menu.

Insert a sketch on the top face. Add a rectangle and use ordinate dimensions to fully define it.



20 Cut.

Extrude the cut **10mm** into the solid.



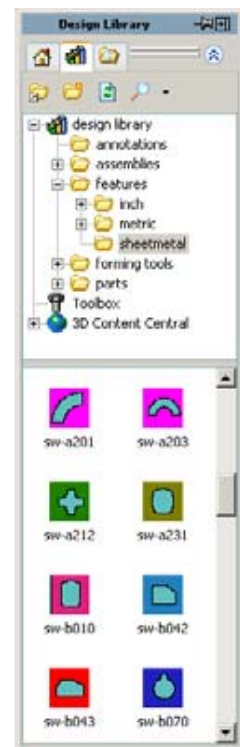
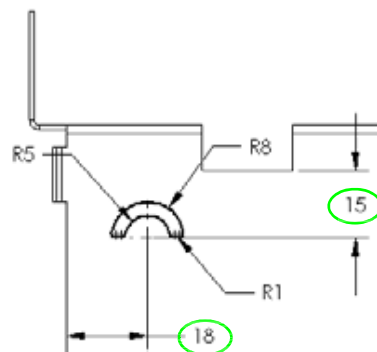
21 Design Library.

Open the **Design Library** to **Features** and the **Sheetmetal** folder.

Choose the feature **sw-a203** from the list.

22 Drag and drop.

Drag and drop the feature onto the side face of the model. Position the sketch as shown.



Click **Back**.

Click **Override dimension value** and change the values of dimensions:

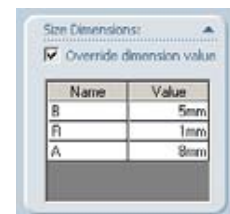
B = 5mm

R = 1mm

A = 8mm

Note

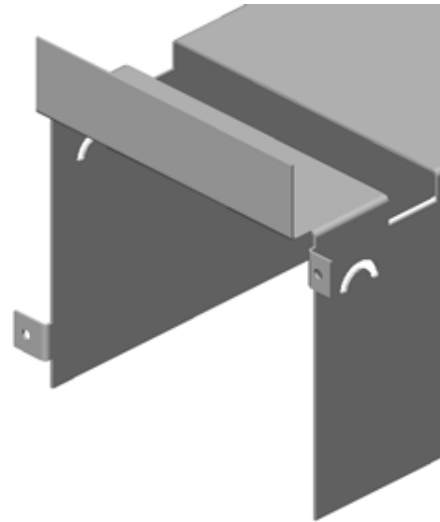
Many library features contain multiple configurations although this one does not. When you override the dimensions of the active configuration, the system creates a new, custom configuration.



23 Click OK.

This sheet metal feature uses the end condition **Up to Next**. This makes sense. **Up to Next** ensures that the feature cuts through the material regardless of its thickness. However, we want the feature to cut all the way through *both* sides of the part.

Edit the feature and change the end condition to **Through all**.

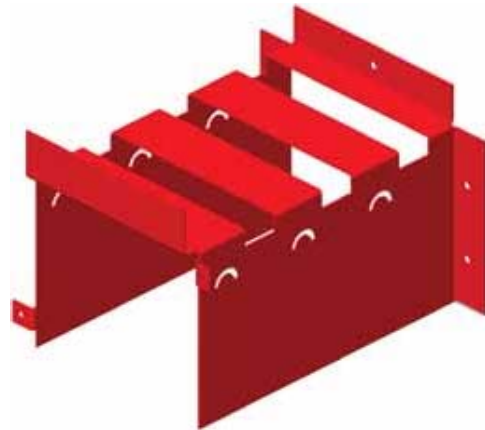


24 Change the part color.

Because gray is boring.

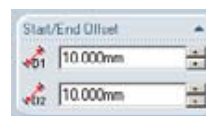
25 Pattern.

Pattern both cuts using a **Linear Pattern** with spacing of **51mm**.



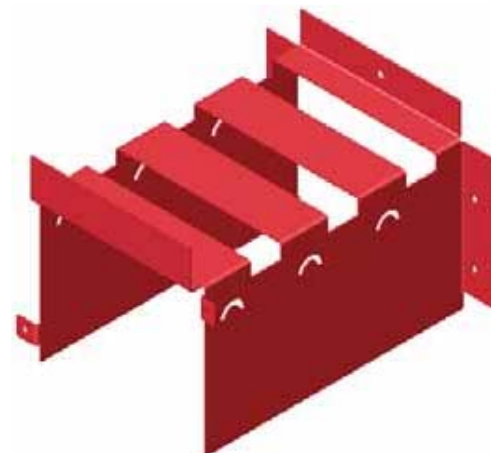
26 Modify the Miter Flange.

Use **Edit Feature** to modify the Miter Flange1 feature. Set both **Start/End Offset** values to **10mm**.



Click **OK**.

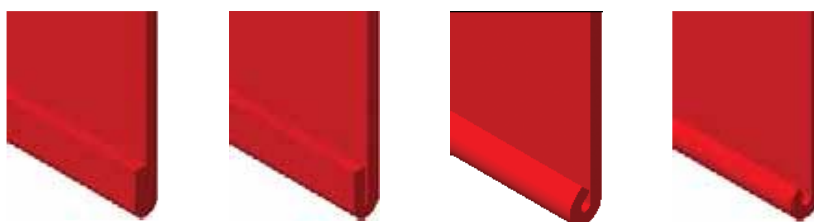
Shortening the miter flanges leaves room for the next feature, a hem.



Hems

The **Hem** tool is used to fold over an edge of the model using one of several shapes. Hems are applied to existing edges. Here are some additional items to note about the **Hem** tool:

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




Hem Examples

Introducing: Hem





The **Hem** is used to add a bend at an edge, doubling the material thickness there. Hems can be applied to multiple edges.

Where to Find It

- Q From the menu choose: **Insert, Sheet Metal, Hem...**
- Q Or, click **Hem**  on the Sheet Metal toolbar.

27 Click Hem , or click Insert, Sheet Metal, Hem.



In the PropertyManager, set the following:

- Q **Position** = 
- Q **Type** = 
- Q **Angle** =  200°
- Q **Radius** =  0.50mm

Select the two bottom *outside* edges.


Positioning

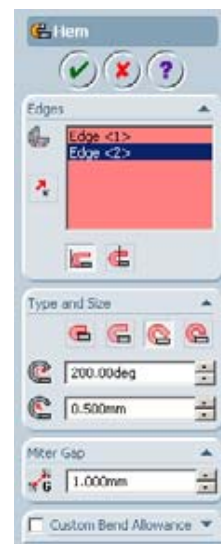
There are two options for how the hem is added to the edge:

- Q **Material Inside** 
- Q **Bend Outside** 

Hem Type and Size

There are four options for the shape of the hem:


- Q **Closed** 
- Q **Teardrop** 
- Q **Open** 
- Q **Rolled** 

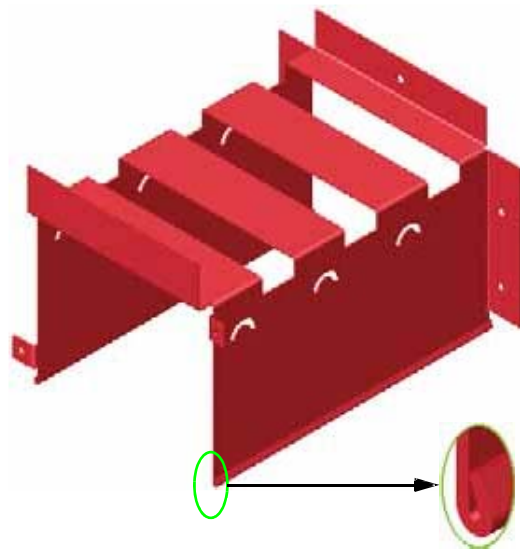
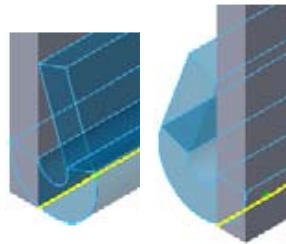


Hems

Review

28 Click OK to complete the hem.
Both selected edges receive hems on the outside.

The hem is applied to the edge you select. If you want the hem on the outside, pick the outside edge or click **Reverse Direction** .

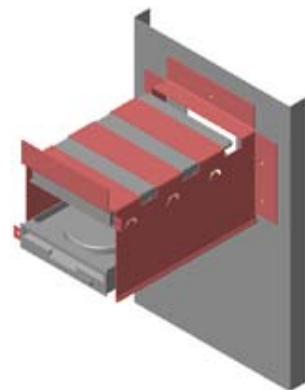


29 Return to the assembly.
Save the changes to the part and return to the assembly document.

The changes to the Enclosure propagate to the assembly.

Note

We are still in edit part mode.




Edit Assembly

Two commands, **Edit Component** and **Edit Assembly**, are used to switch back and forth between editing one component in an assembly and editing the assembly itself. A component of an assembly can either be a single part or a sub-assembly. When we inserted the new part, the Enclosure, we were automatically placed in edit part mode. Now we need to leave edit part mode and return to editing the assembly.


Introducing: Edit Assembly

Edit Assembly is used to switch between editing a part, and editing the assembly itself.

Where to Find It

- Q Right-click the top-level node or in the graphics window, and select **Edit Assembly** from the shortcut menu.
- Q Or, from the Assembly toolbar, click **Edit Component** 

Tip

Edit Component  is a toggle. It switches you between **Edit Part** or **Edit Sub-assembly** mode and **Edit Assembly** mode. It also acts as a

visual indicator of which mode you are in. It is depressed or pushed in when you are in **Edit Part/Edit Sub-assembly** mode.

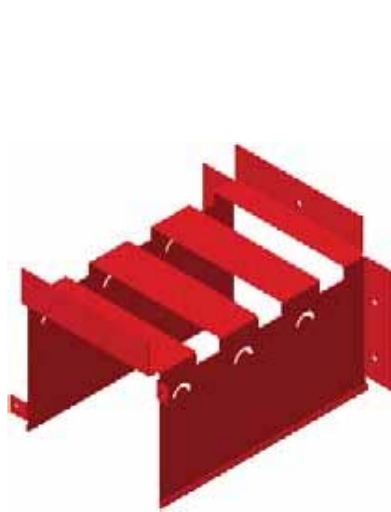
30 Switch to Edit Assembly mode.

Right-click in the graphics area, and select **Edit Assembly** from the short-cut menu.

31 Finished.

Switch back to the Enclosure window and flatten the part.

Notice how the part is flattened, and the orientation of the result. This is controlled by the face that is defined as the **Fixed Face**.



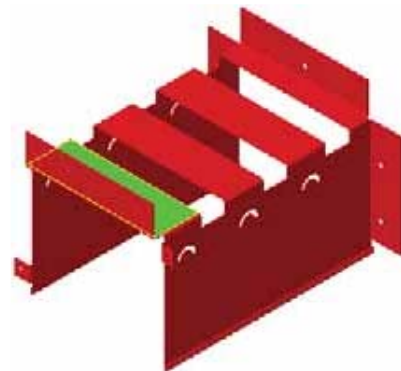
Before



After

32 Redefine the Fixed Face.

Edit the definition of the Flat-Pattern1 feature, and select the face shown as the **Fixed Face**.

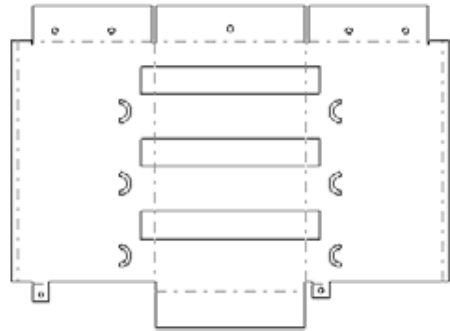


33 Flatten the part.

This is a better result.

34 Refold the part.

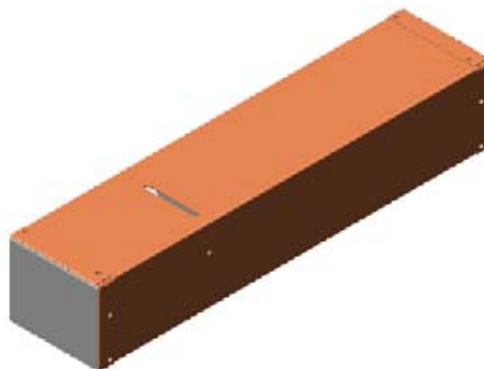
35 Save and close all files.



Exercise 8: Sheet Metal in an Assembly

Use an existing assembly to form in-context parts. Two plates are added to connect the end pieces in the assembly. This lab reinforces the following skills:

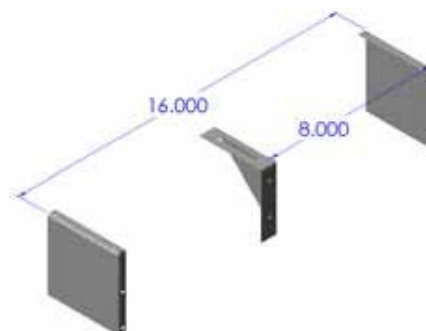
- Q Insert a new part into an assembly
- Q Sheet metal in context
- Q Hems
- Q Propagating changes through parts to the assembly



Procedure

Use the following procedure.

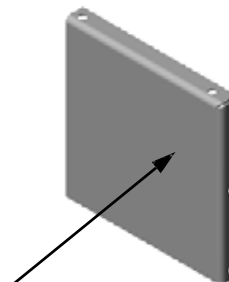
- 1 **Open an existing assembly named SheetMetalAssy.** The assembly consists of three components mated as shown at the right.



Building the Plate

Build the cover plate using in-context sketches based on model edges.

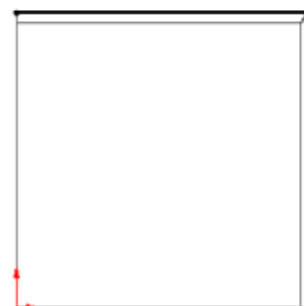
- 2 **Insert new part.**
Insert a new part named TopPlate into the assembly. Position it on the indicated face of the LeftEndPlate. An **Inplace** mate is created and you are ready to sketch.



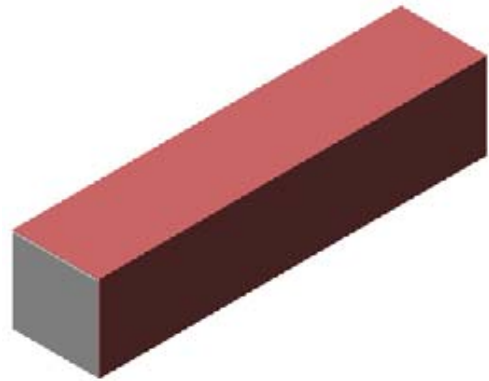
Units

Although the assembly **units** are inches, the TopPlate is modeled in **millimeters**.

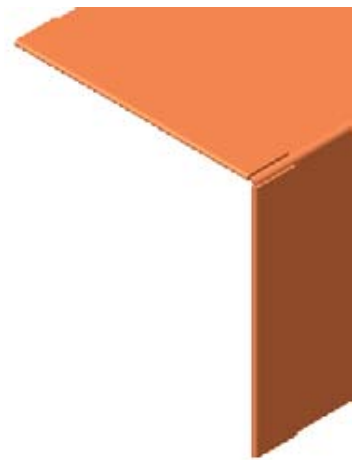
- 3 **Sketch the TopPlate profile.**
Create the profile using existing edges. The profile is as high and as wide as the LeftEndPlate.



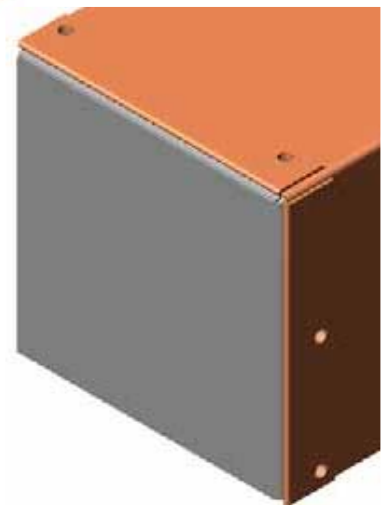
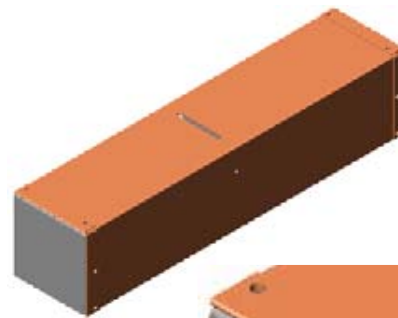
- 4 Extrude a base flange.**
Extrude the sketch as a **Base Flange** whose length equals the distance between the outside faces of the two end plates.
Use a thickness and bend radius of **1mm**. Orient the thickness *away* from the existing end plates.



- 5 Hems.**
Add hems that are **Closed** and **Length 16mm** to both ends.



- 6 Add in-context holes.**
Add holes that are the same size, shape, and location as those on the two end plates and the CenterBrace.



Changes to the End Plates

Changes to the existing end plates will cause changes in the cover plate. The in-context references will keep the design intent intact.

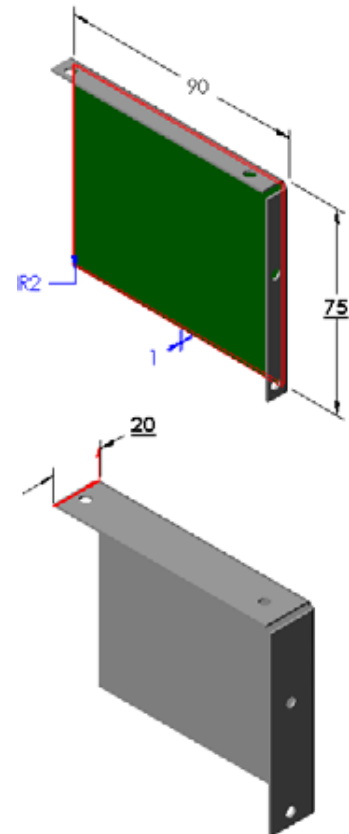
7 Open Part.

Switch from editing the TopPlate to working on the Right End Plate. Right-click the component RightEndPlate, and select **Open Part**.

8 Modify the RightEndPlate part.

The RightEndPlate part is also the master for the LeftEndPlate part. Make the changes shown to the part's overall size (Base-Flange and Miter-Flange1 features).

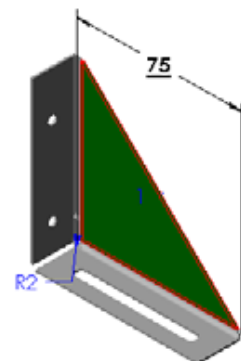
Save and **Close** the part.



9 Modify the CenterBrace part.

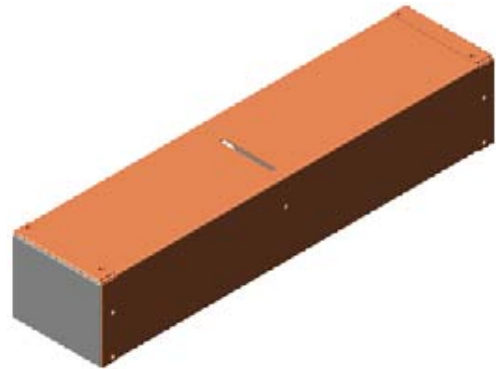
Change from editing the RightEndPlate to working on the CenterBrace by right-clicking on the component, and selecting **Open Part**. Make the change to the part's overall size as indicated by the underlined text.

10 Save and Close the part.



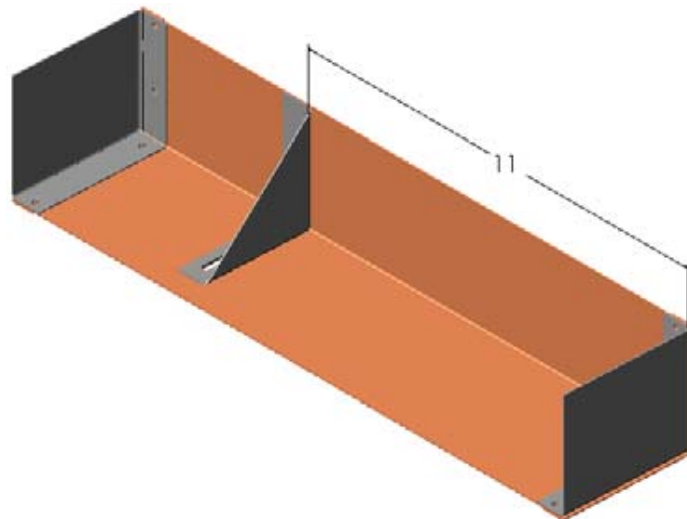
11 Result.

The resulting assembly shows the results of the changes made to the RightEndPlate and CenterBrace parts.



12 Optional.

Reposition the CenterBrace component between the RightEndPlate and LeftEndPlate to a distance of 11".



The in-context cuts should update to reflect the new position.

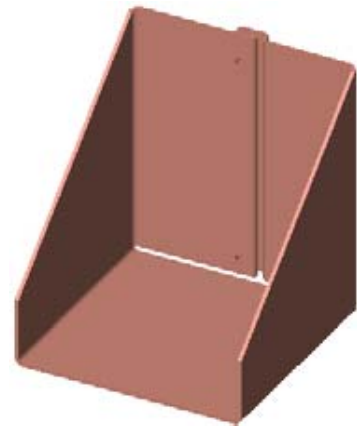
13 Save and close the assembly.

Exercise 9: Jogs and Hems

Use the **Jog** and **Hem** features to create a sheet metal part.

This lab reinforces the following skills:

- Q Base and Edge Flanges
- Q Jog Feature
- Q Hem Feature

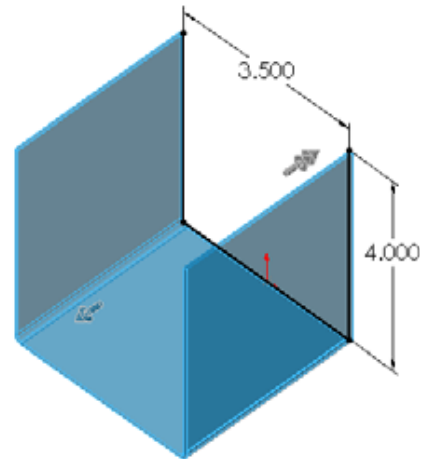


Procedure

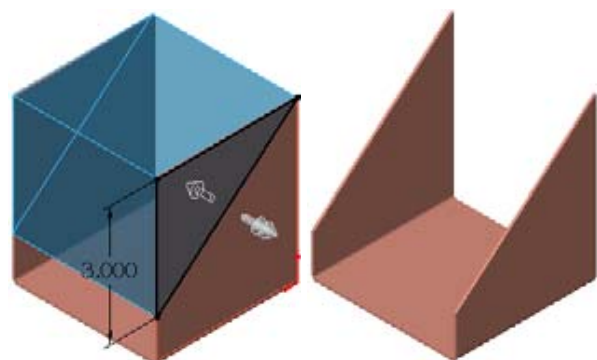
Use the following procedure.

- 1 Open a new part.**
Open a new part with units **inches**.
- 2 Base Flange.**
Create a sketch and base flange using **Thickness** and **Bend Radius** of **0.0625"**.

The thickness is applied to the *outside* of the sketch and the **Depth** is **3.5"**.



- 3 Cut.**
Create a cut as shown.



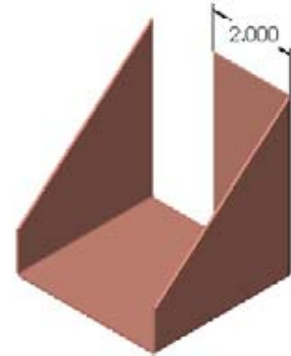
4 Rightmost edge flange.

Add an edge flange as shown using:

- Q Use default radius
- Q Angle = 90°
- Q Flange Length = 2"
- Q Flange Position = Material Outside

Note

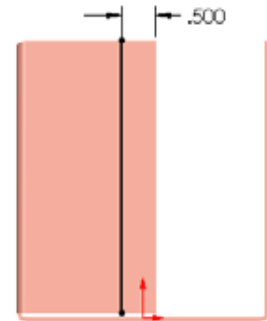
The length of the flange is dimensioned from the *outside* face of the model. If you choose to define the flange length within the **Edge Flange** dialog rather than by adding a dimension through **Edit sketch profile**, then **Outer virtual sharp** should be used.

**5 Jog.**

Sketch the **Jog** line on the flange. Position the line **0.5"** from the edge of the flange.

Create the jog using:

- Q Use default radius
- Q Offset Distance = 0.188"
- Q Dimension position = Outside offset
- Q Fix projected length = On
- Q Jog Position = Bend centerline
- Q Jog Angle = 70°

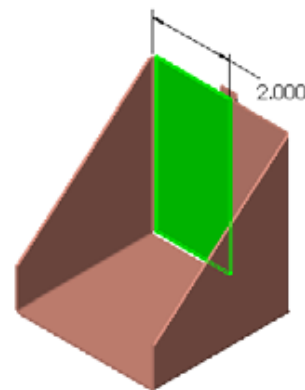
**6 Leftmost edge flange.**

Add an edge flange as shown using:

- Q Use default radius
- Q Angle = 90°
- Q Flange length = 2"
- Q Flange position = Material outside

Note

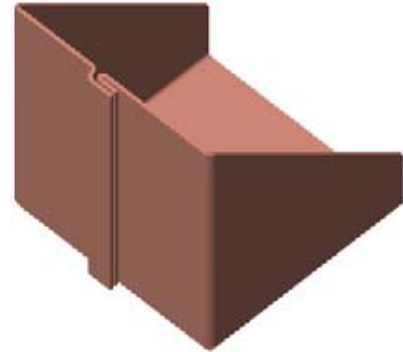
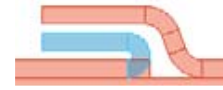
The length of the flange is dimensioned from the *outside* face of the model, as before.



7 Hem.

Add a hem to the outside edge of the flange using:

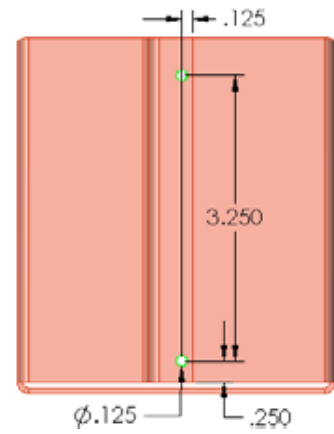
- Q **Material inside**
- Q **Type = Open**
- Q **Length = 0.375"**
- Q **Gap Distance = 0.03"**



8 Holes.

Create two circular holes through the hem and the jog.

9 Save and close the part.

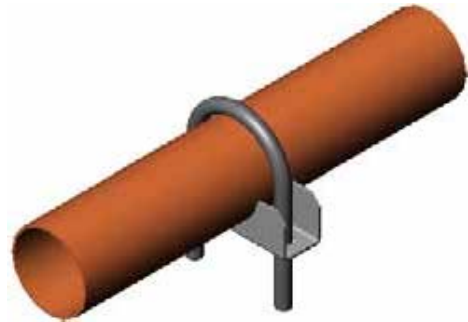


**Exercise 10:
U-Bolt In-
Context**

Add an in-context sheet metal component to this assembly.

This lab uses the following skills:

- Q New part in-context
- Q Base Flange
- Q Break Corner Chamfer
- Q In-context features



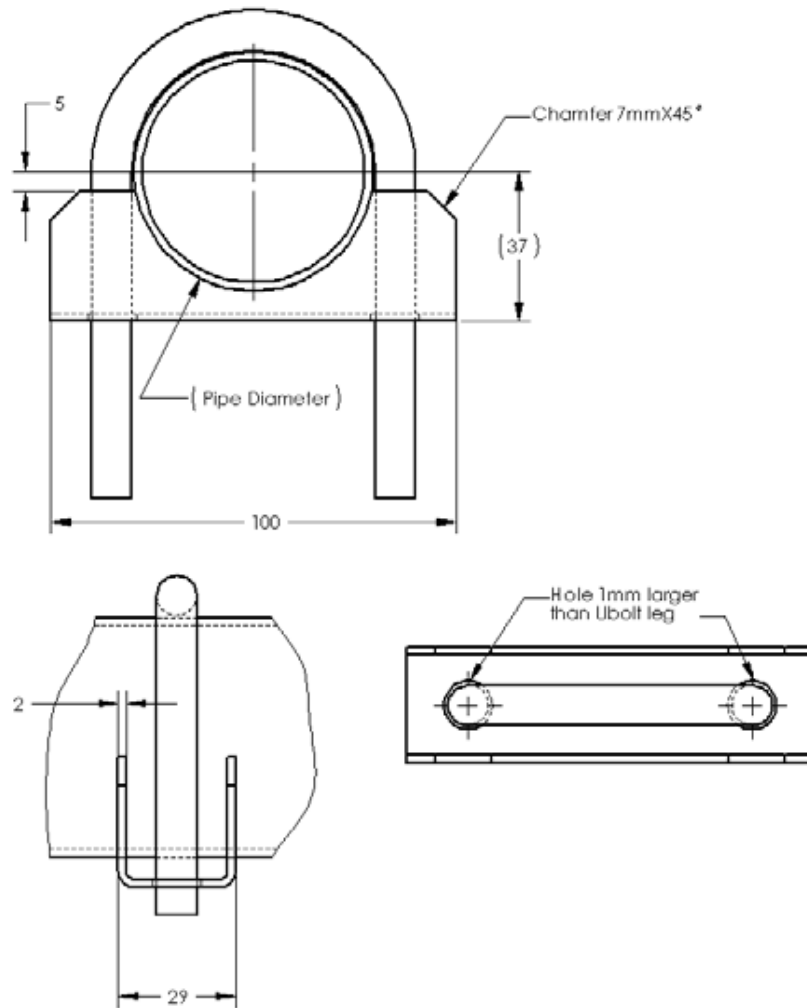
Design Intent

The design intent for this part is as follows:

1. All bends are equal radius. Bend Radius = **2mm**.
2. Part is symmetrical.

Procedure

Open an existing assembly named `UboltAssy`. Use the dimensions below to create the `UboltClamp` component in-context.



Exercise 11: Tube Brace

Use an existing assembly to form an in-context part.

This lab reinforces the following skills:

- Q Building sheet metal in-context
- Q Interference Detection
- Q Using Mirror All



Procedure

Use the following procedure.

- 1 **Open an existing assembly named HandleAssy.** The assembly consists of four components mated as shown at the right.



Building the Connector

Build the Connector using in-context sketches based on model edges.

- 2 **Insert new part.**

Insert a new part named Connector into the assembly. Position it on the reference plane TubeSurface of the Tube<1> component. An **Inplace** mate is created and you are ready to sketch.

Units

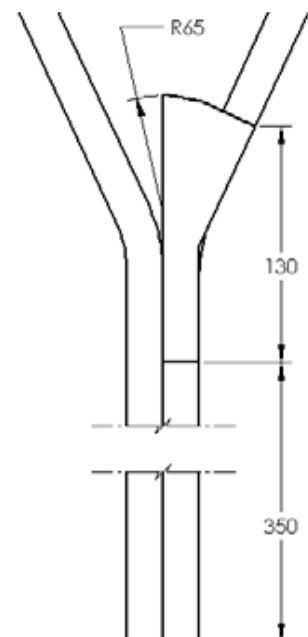
The Connector is modeled in **millimeters**.

- 3 **Sketch.**

Create a sketch for the **Base Flange** of the Connector. Create lines **Collinear** with the silhouette edges of the tubes.

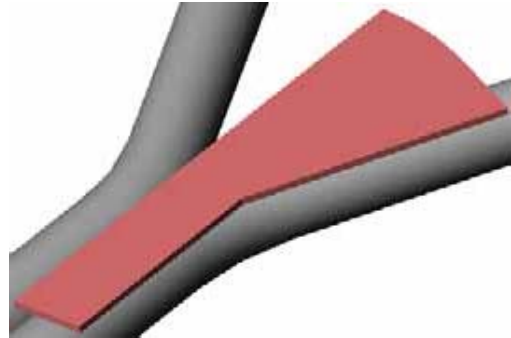
Note

The **350mm** dimension measures to the bottom of the Tube.

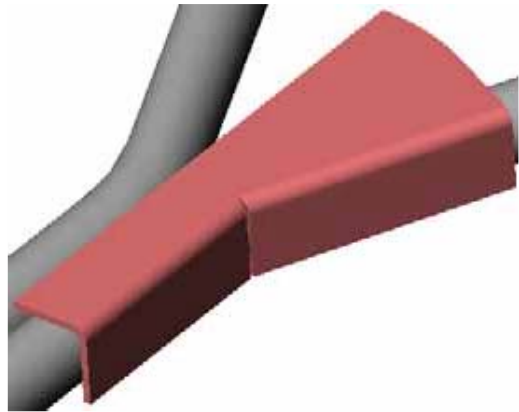


4 Base Flange.

Create the base flange using a **Bend Radius** and **Thickness** of **2mm**. The direction of the extrusion is *away* from the tubes.

**5 Miter Flange.**

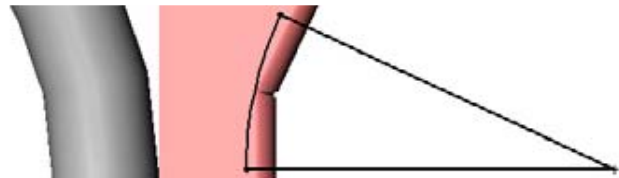
Add a miter flange along the outer edge of the base feature. Set the material to the *outside* and the depth the same as the tube. Use a **Gap** of **1mm**.

**6 Interference.**

Check the interferences using all components in the assembly. There are interferences near the bend line between the Tube and the part being edited.

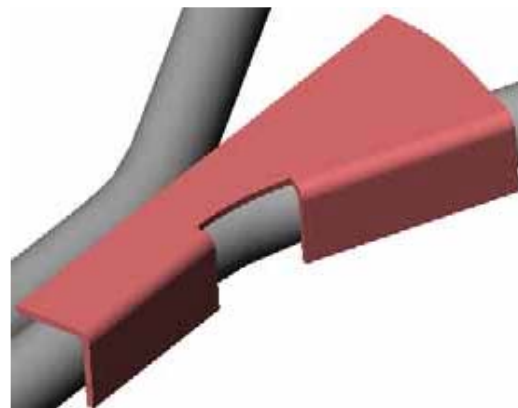
7 Cut.

Create a cut to remove the interferences.

**Tip**

Offset the silhouette edge of the bend in the Tube.

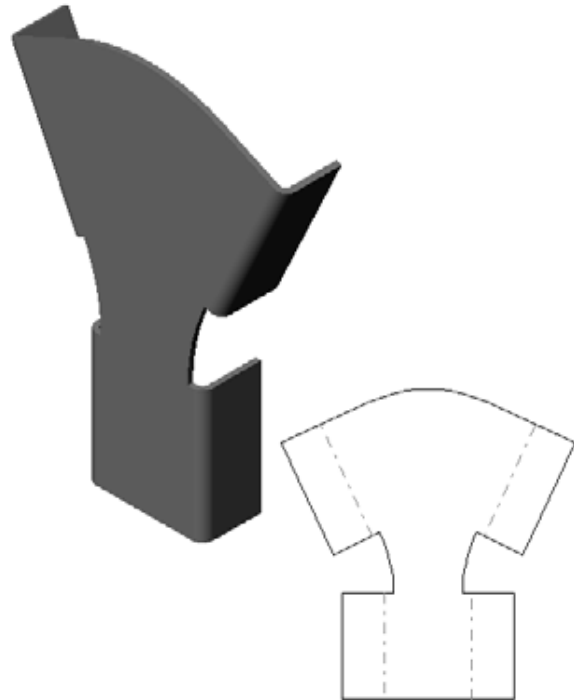
Cut using the **Through All** end condition.
Recheck for interferences.



8 Open part.

Switch to editing the assembly and open the Connector part.

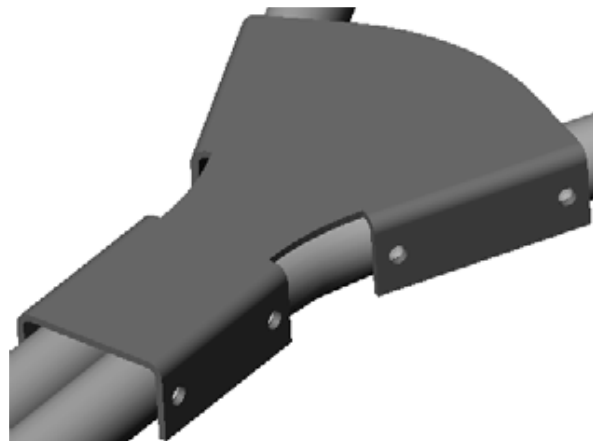
Use **Mirror Bodies** to create the other half of the part. Check the flat pattern.



9 In-context cuts.

Optionally add in-context cuts for drilled holes through the Connector and Tubes.

All holes are **5mm** diameter, **6mm** from the edges of the flanges, and centered on the centerline of the tubes.



10 Save and close.

Lesson 4 Weldments

Upon successful completion of this lesson, you will be able to:

- Q Create weldments.
- Q Create library parts for structural member profiles.
- Q Manage corner treatments and trimming of structural members.
- Q Create fillet weld beads.
- Q Create gussets and end caps.
- Q Represent post-assembly machining operations.
- Q Manage custom properties.
- Q Incorporate non-structural member components such as castings and sheet metal brackets into weldments.
- Q Create and manage sub-weldments.
- Q Create drawings of weldments.
- Q Create a cut list table for a weldment drawing.

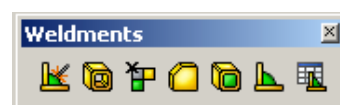
Weldments

A weldment is an object made up of several parts welded together. Although it is really an assembly, many prefer to represent it as a single part for BOM purposes. To achieve this, we will model weldments as multibody parts. A special **Weldment** feature designates a multibody part as a weldment. This enables the weldment environment giving you access to a number of specialized tools and capabilities. You can:

- Q Insert structural members, gussets, end caps, and fillet weld beads.
- Q Use a special-purpose tool for trimming or extending structural members.
- Q Manage cut lists and create cut lists on drawings.
- Q Group bodies as sub-weldments.

Weldments Toolbar

The specialized weldment commands are grouped on the Weldments toolbar. They are also accessible via **Insert, Weldments**.



Weldment Feature

Adding the weldment feature to a part does the following:

- Q **Configurations:** The description <As Machined> is added to the active configuration. A new derived configuration with the same name is added with the description <As Welded>. Once the part is marked as a weldment, any subsequent top level configurations you create automatically have a corresponding <As Welded> derived configuration.
- Q The **Merge Result** check box is automatically cleared for all subsequent features that use this option. This way when you create solid bodies representing components in the weldment they remain as separate discrete bodies.
- Q **Custom properties:** The weldment feature acts as a holder for a default list of custom properties that are inherited by each cut list item.
- Q The specialized weldment commands are enabled.

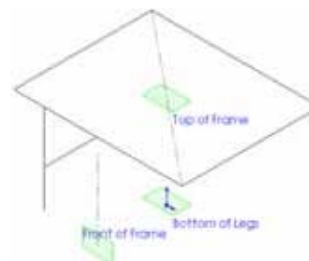
You can insert only one weldment feature per part. It will always appear as the first feature regardless of when you insert it.

Where to Find It


- Q Click **Weldment**  on the Weldments toolbar.
- Q Or, click **Insert, Weldments, Weldment**.

1 Open part.

Open the existing part Conveyor Frame. This part already contains some layout sketches that will be used to create the structural members.



2 Weldment feature.

Click **Weldment**  on the Weldments toolbar.

The Weldment feature is added to the FeatureManager design tree.

Tip


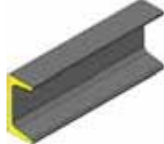
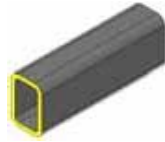

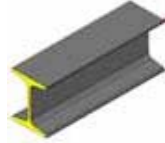


If you do not insert the weldment feature, one will be added automatically when you insert the first structural member.

Structural Members

You insert structural members using line and arc segments in 2D and 3D sketches. This technique lets you create a wireframe layout of the structural members. Inserting a structural member follows this general procedure:

1. Specify a profile.
2. Select the sketch segments.
3. Specify the orientation and position of the profile if necessary.
4. Specify the corner conditions between structural members.

The available profiles are shown in the following table. You can also create your own library of profiles.

Angle Iron		C Channel	
Rectangular Tube		Pipe	
S Section		Square Tube	
Tee			

Introducing: Insert Structural Member

Use a combination of 2D and 3D sketches to lay out the paths for the structural members.

Where to Find It

- Q Click **Structural Member**  on the Weldments toolbar.
- Q Or, click **Insert, Weldments, Structural Member**.

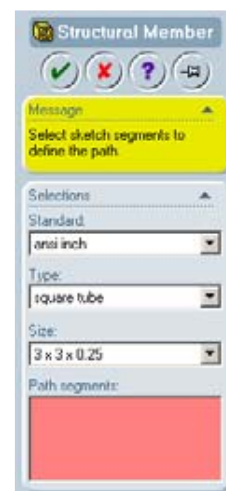
3 Inserting a structural member.

Click **Structural Member**  or, click **Insert, Weldments, Structural Member**.

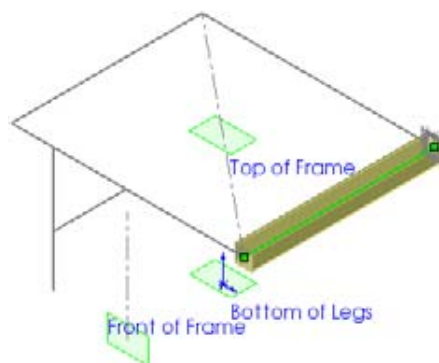
For **Standard**, select **ansi inch**.

For **Type**, select **square tube**.


For **Size**, select **3 x 3 x 0.25**.

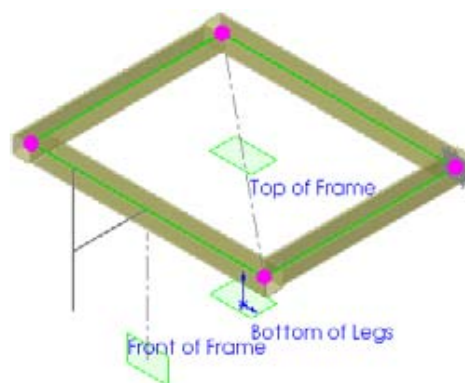
**4 Select the first path segment.**

The system creates a plane normal to the segment and applies the sketch of the profile to it. A preview of the structural member appears.


**5 Select the remaining segments.**

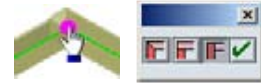
Select the remaining three segments that define the top of the frame.

Select the **Apply corner treatment** check box and select **End Butt2** .

**Corner Treatments**

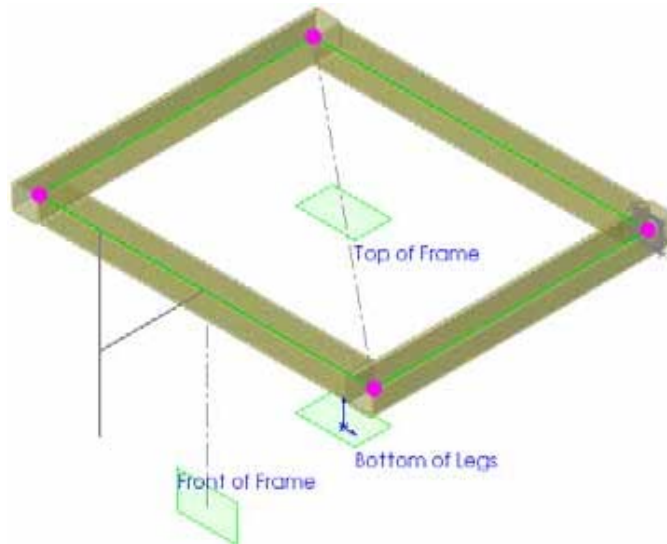
You may want the corner treatments for some corners to be different than the default, system applied corner treatment. To do this you can override the default corner treatment individually.

Click the sphere that appears at the corner. A toolbar appears. Select the appropriate corner treatment and click **OK** .



6 Corner treatment.

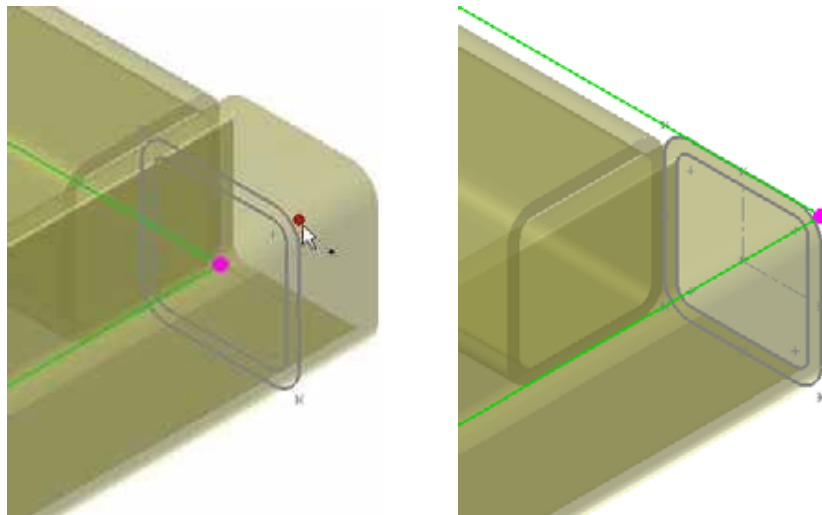
Change the corner treatments to match the illustration below.



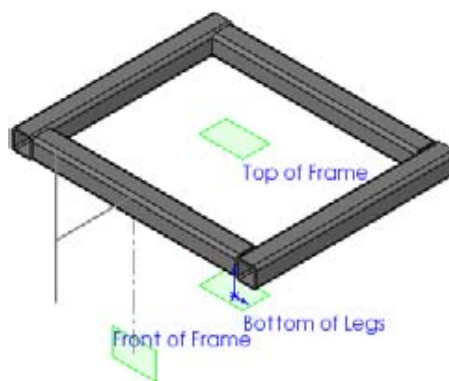
7 Locating the profile.

By default the profile sketch is located with its sketch origin on the path. You can reposition the profile so that any sketch point or vertex is aligned with the path segment.

Click **Locate Profile**. The display zooms in to show the profile sketch. Click the reference point located at the virtual sharp of the upper-right corner. The sketch profile is repositioned.



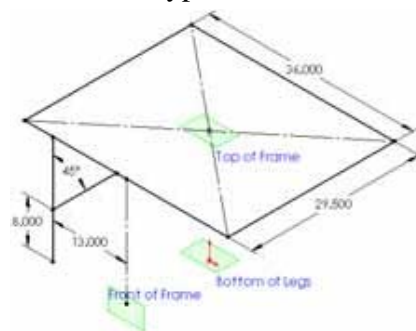
- 8 **Click OK.**
The system creates the four structural members as individual solid bodies.
- 9 **Zoom to Fit and hide the Upper Frame sketch.**



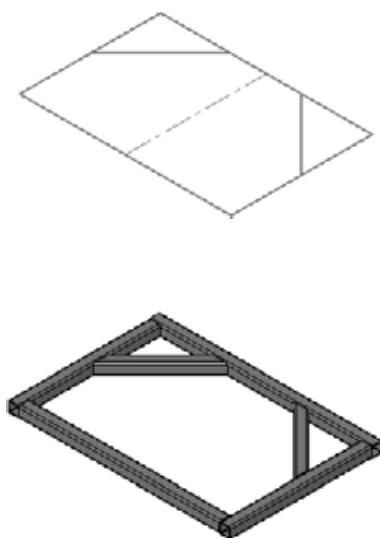
Sketch Considerations

There are some things you should keep in mind about layout sketches and how they can be used for structural members.

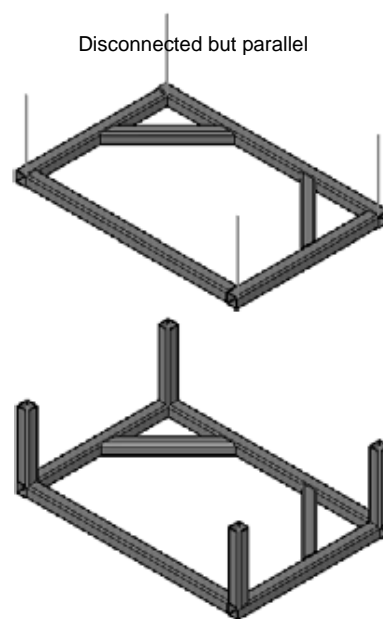
- Q You can use 2D, 3D, or a combination of both types of sketches.
- Q You have to balance the ease of sketching against any benefit that might come from having all the paths in one sketch. For example, the conveyor frame we are working on has two 2D sketches: one for the top of the frame, and one for the legs. This could have been done in a single 3D sketch.
- Q Path segments must be either:
 1. Connected.
 2. Disconnected but parallel to each other.



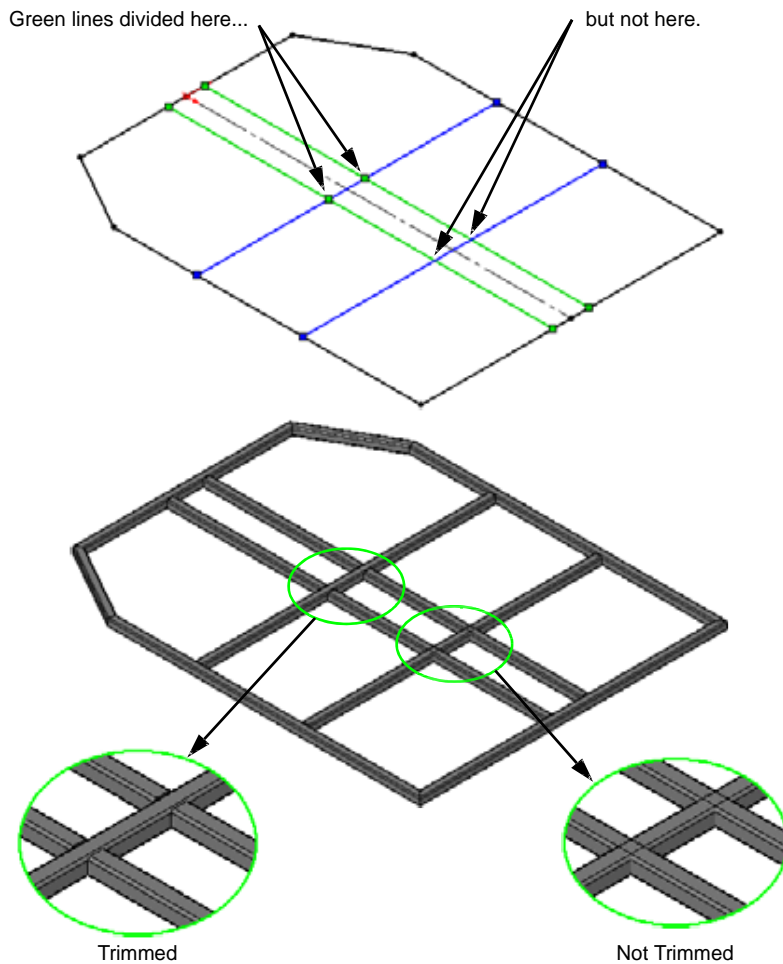
Connected



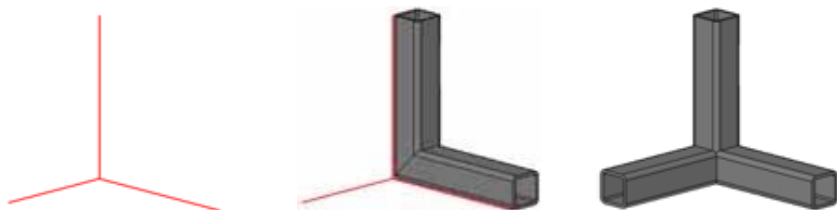
Disconnected but parallel



- Q How path segments are divided determines how the structural members will be trimmed at their intersections. For example, in the illustration below, the two blue lines are each one segment. The two green lines are divided at the intersection with the first blue line, but not the second.



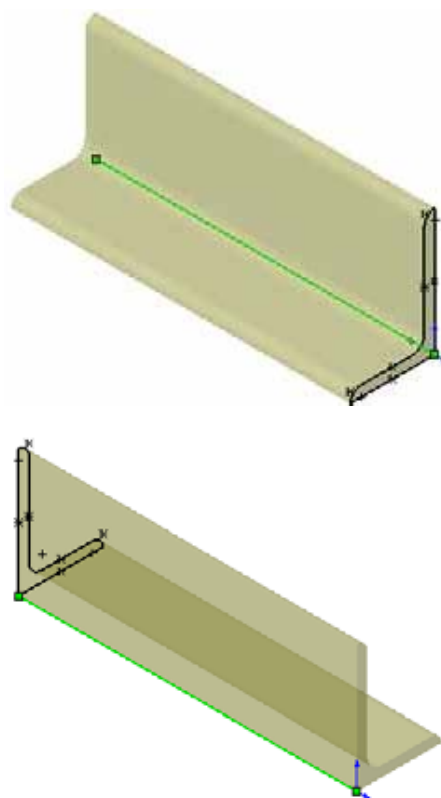
- Q In order for path segments to be considered connected, they must be in the same sketch.
- Q You cannot select more than two path segments that share a common vertex in a single operation. To create the corner shown below, you will have to insert two structural member features.



- Q One layout sketch can be used for several structural member features.

Q The profile is positioned at the *start* point of the path segment. This has implications when working with asymmetrical profiles. For example, consider the 3 x 2 angle iron in the upper illustration at the right. The **Locate Profile** option allows you to rotate and/or position the profile. It does not allow you to *mirror* the profile. If the result you want looks like the lower illustration, you have two options:

1. Sketch the path segment going in the opposite direction.
2. Create a mirrored version of the profile sketch.



10 Create the leg and brace.

Click **Structural Member** 

For **Standard**, select **ansi inch**.

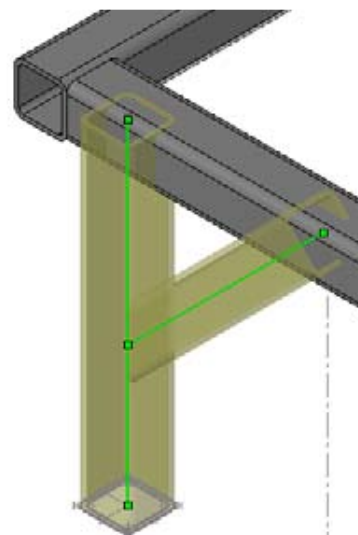
For **Type**, select **square tube**.

For **Size**, select **3 x 3 x 0.25**.

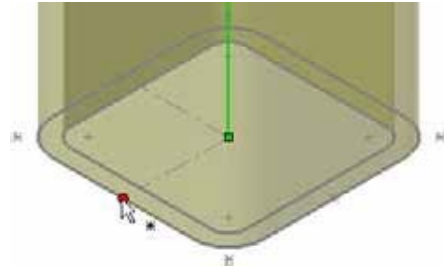
Select the vertical sketch segment, and then the angled one.

Note

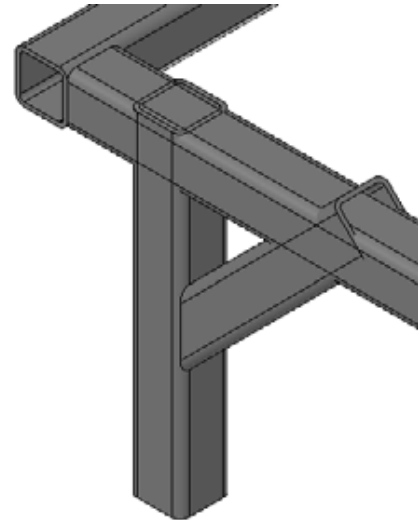
The order in which you select the sketch segments is important. The endpoint of the second segment must touch the first segment. Picking the angled segment first would not have worked because the endpoints of the vertical segment do not touch the angled one.



- 11 Position the profile.**
Click **Locate Profile**, and select the midpoint of the edge.



- 12 Click OK.**
The results are shown at the



Trimming Structural Members

Introducing: Trim/Extend

Where to Find It


When structural members are inserted as part of a single feature, trimming is done automatically. However, many times structural members are inserted during separate operations. When this happens it is usually necessary to trim these structural members with respect to the existing structural members.

Trim/Extend trims structural members. The same set of corner treatments that were available when inserting structural members are available for trimming. You can trim a structural member to the intersection of the solid body of an existing structural member, or to the intersection of a planar face.

- Q Click **Trim/Extend**  on the Weldments toolbar.
- Q Or, click **Insert, Weldments, Trim/Extend**.

13 Trimming structural members.

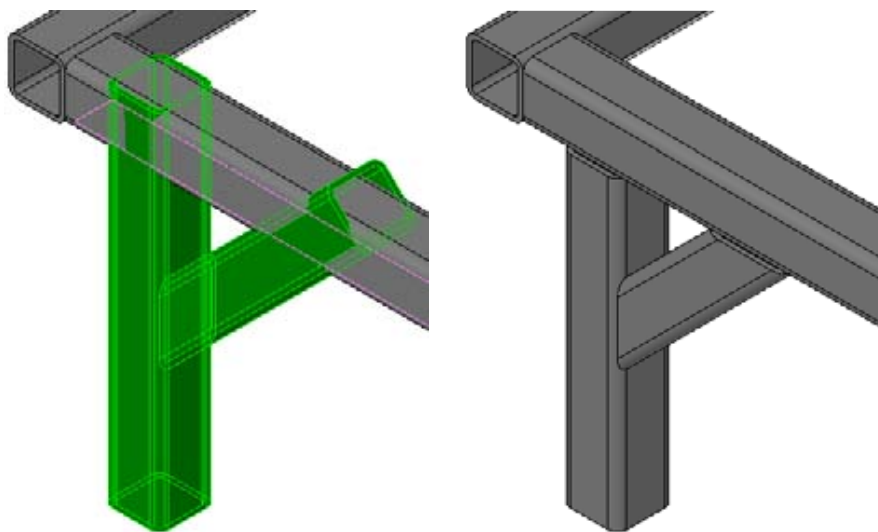
Click **Trim/Extend**  or, click **Insert, Weldments, Trim/Extend**.

For **Corner treatment**, select **End Trim** 

For **Bodies to be Trimmed**, select the leg and the angled brace.

For **Trimming Boundary**, click **Planar face** and select the planar face on the underside of the horizontal square tube.

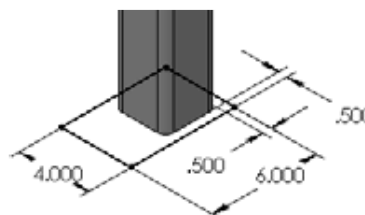
Click **OK**.

**Adding Plates**

Not all the components in a weldment are structural members. In this section we will create a simple plate. Later in this lesson we will examine how to deal with castings and fabricated items like brackets.

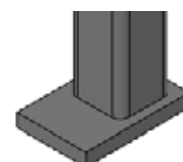
14 Sketch the plate.

Select the plane **Bottom of Legs** and open a sketch. Sketch a rectangle as shown.

**15 Extrude.**

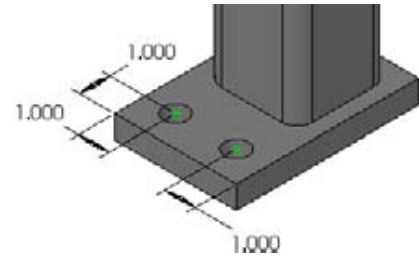
Use the end condition **Blind** and a **Depth** of **0.75"**. Extrude downward, away from the legs.

Name the feature **Plate**.



16 Add the holes.

Use the **Hole Wizard** to add two clearance holes for 0.75” bolts. Locate the holes as shown in the illustration.



Weld Beads

The weld bead is a solid body and as such is included in operations such as mass property calculations. At this time only fillet weld beads are supported.

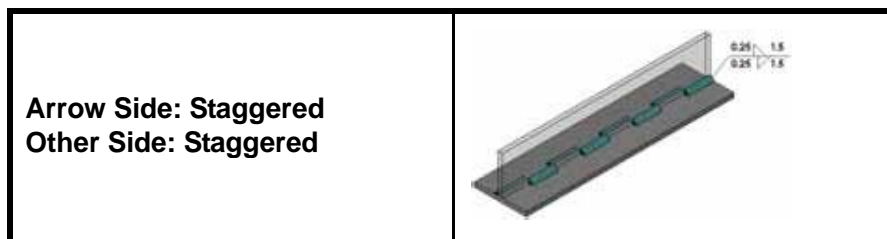
**Introducing:
Fillet Bead**

You can have a gap between selected faces or bodies as long as the gap is smaller than the specified weld bead size.

You can specify whether you want a full length or intermittent weld bead. You can weld one side or both sides. It is possible to combine full length and intermittent weld beads on opposite sides. If you create intermittent beads on both sides, you can stagger them if desired.

Some examples of weld beads are shown in the following table. Note that the vertical plate has been made transparent for illustration purposes.

<p>Arrow Side: Full Length Other Side: None</p>	
<p>Arrow Side: Full Length Other Side: Intermittent</p>	
<p>Arrow Side: Intermittent Other Side: Intermittent</p>	




You can suppress all weld beads in the part by right-clicking the Weldment feature and selecting **Suppress Weld Beads**.

Where to Find It

- Q Click **Fillet Bead**  on the Weldments toolbar.
- Q Or, click **Insert, Weldments, Fillet Bead**.

17 Fillet weld bead.

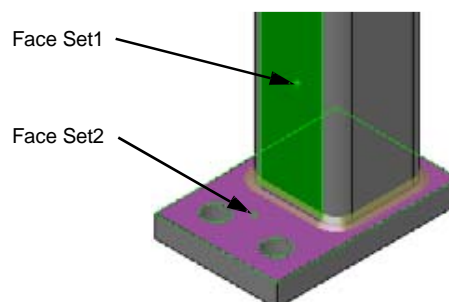
Click **Fillet Bead**  on the Weldments toolbar, or click **Insert, Weldments, Fillet Bead**.

Under **Arrow Side**, select the following:

- Q **Bead type = Full length**
- Q **Fillet size = 0.25"**
- Q **Tangent propagation: selected**

18 Select faces.

For **Face Set1** select the front, flat face of the leg. For **Face Set2** select the upper face of the Plate.

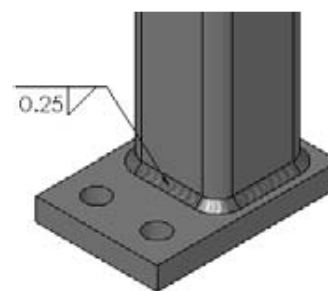


19 Click OK.

The results are shown at the right.

Tip


When you insert a weld bead, the display of annotations is automatically turned on. You can turn them off by clicking **Tools, Options, Document Properties, Annotations Display**. Under **Display filter**, clear the **Welds** check box.



Using Symmetry

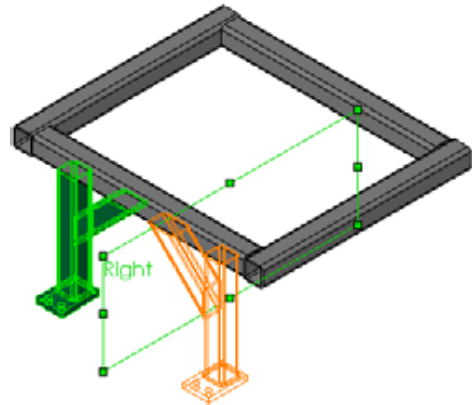
Structural members can be mirrored and patterned in the same way as any other solid body.

20 Mirror.

Click **Mirror** . For the **Mirror Face/Plane**, select the **Right** reference plane.

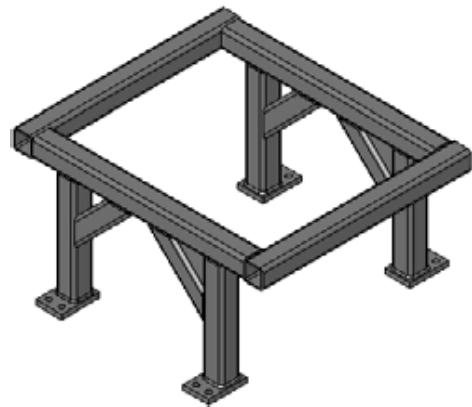
Under **Bodies to Mirror**, select the leg, the angled brace, the weld bead, and the plate.

Click **OK**.



21 Mirror again.

Use the **Front** reference plane to mirror the front legs as shown.



Gussets and End Caps

Gussets and end caps are common features in weldments. Building them manually is tedious. The weldment environment has specialized tools that greatly simplify and speed up creating these features.

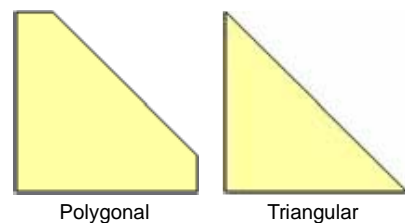
Introducing: Gusset

To insert a gusset you must select two planar faces whose included angle is defined as: $0^\circ < \text{included angle} < 180^\circ$. The faces must touch each other.

Gusset Profile and Thickness

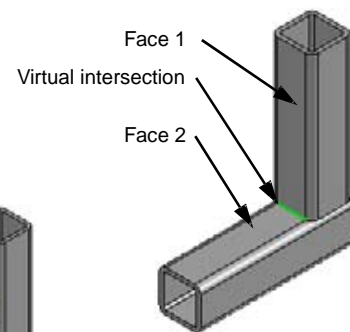
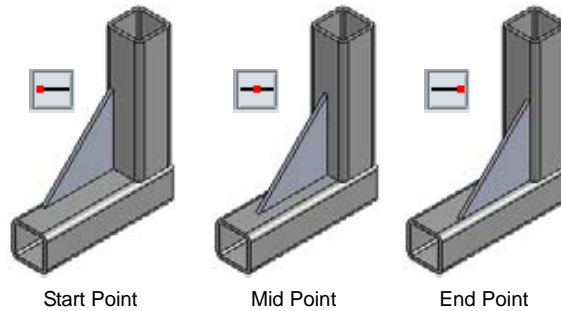
There are two gusset profiles. You specify the dimensions via the PropertyManager.

The thickness of the gusset is set in the same manner as for a rib feature:



Locating the Gusset

When you select two planar faces, the system computes their virtual intersection. The gusset is located along that intersection. There are three possible positions:



In addition, you can specify an offset distance from whatever position you select.

Note

The use of gussets is not limited to weldment type parts. You can use the gusset feature in any part whether multibody or not. The gusset will be created as a separate solid body.

Where to Find It

- Q Click **Gusset**  on the Weldments toolbar.
- Q Or, click **Insert, Weldments, Gusset**.

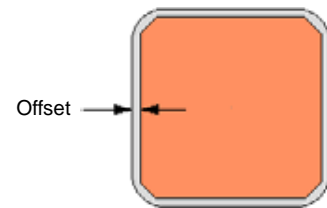
**Introducing:
End Cap**

End caps are metal plates that are welded across the openings of square or rectangular tubes. They are used to close off the openings of the tubes keeping out dirt, debris or other contaminants.

**End Cap
Parameters**

The size and shape of the end cap is determined by the following:

- Q Size – determined by offsetting the outside wall of the tube. The default offset is one half of the wall thickness.
- Q Thickness – specify the thickness in the PropertyManager.
- Q Corners – you can chamfer the corners or not. If chamfered, specify the size of the chamfer.


**Where to Find It**


- Q Click **End Cap**  on the Weldments toolbar.
- Q Or, click **Insert, Weldments, End Cap**.

Note

Gussets and end caps are children of the structural members they reference. Deleting a structural member deletes any associated gussets or end caps.


22 Insert a gusset.

Click **Gusset**  on the Weldments toolbar, or click **Insert, Weldments, Gusset**.

Click **Polygonal Profile** 
Set **d1** and **d2** to **5.0** inches.

Set **d3** to **1.0** inch.

Set **Profile Angle (a)** to **45°**.

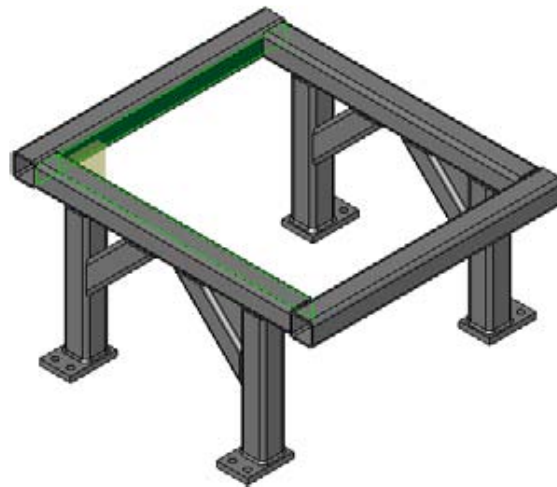
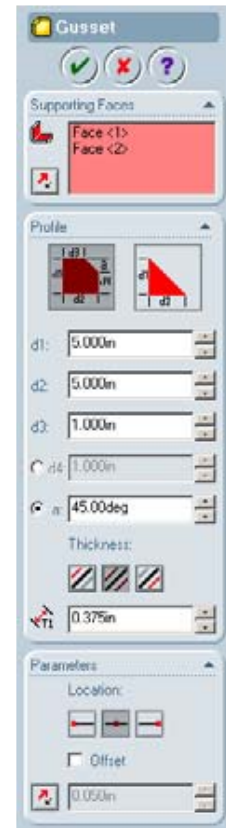
Set **Gusset Thickness** to **0.375** inches and click **Both Sides** .

For **Location**, click **Mid Point** .

Select the faces shown in the illustration below.

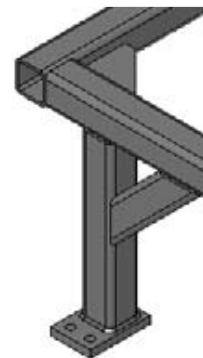
Tip

If you use **Select Other**, be careful not to select the inside face of the tube.




23 Click OK.

The results are shown in the illustration at the right.



24 Insert an end cap.

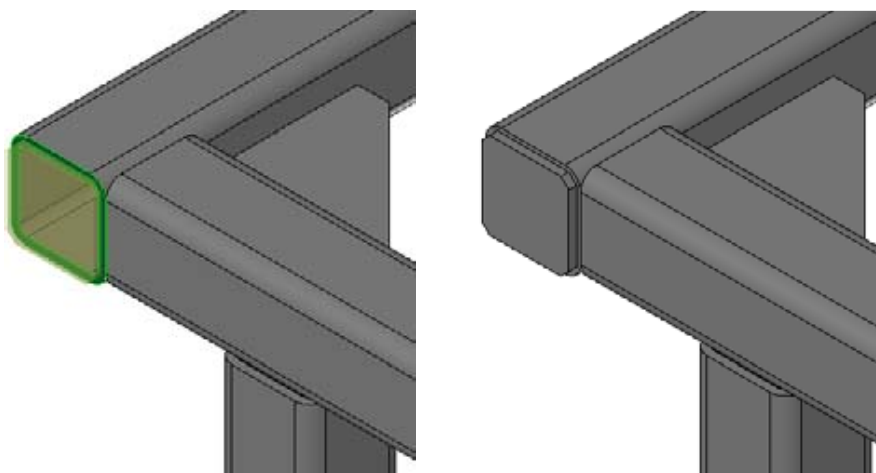
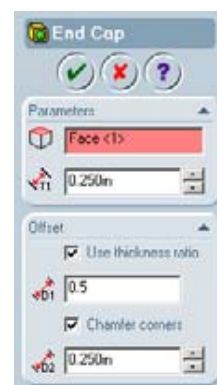
Click **End Cap**  on the Weldments toolbar, or click **Insert, Weldments, End Cap**.

Set **Thickness** to **0.25** inches.

Under **Offset**, select **Use thickness ratio** and verify that the **Thickness Ratio** is set to **0.5**.

Select **Chamfer corners** and set the **Chamfer Distance** to **0.25** inches.

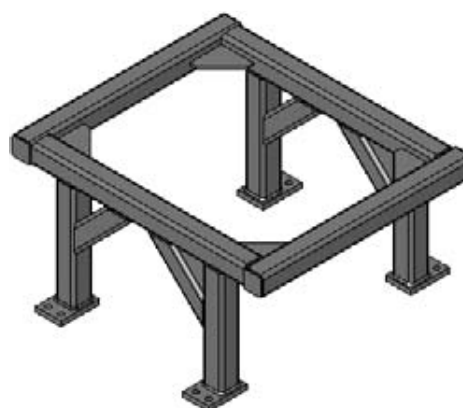
Select the end face of the tube as shown in the illustration below. Then click **OK**.

**25 Reorder.**

Drag the **Gusset1** and **End cap1** features so they come before the **Mirror1** feature in the FeatureManager design tree.

26 Edit the Mirror features.

Edit the definition of both **Mirror** features and include the gusset and end cap in the **Bodies to Mirror** list.



Profile Sketches

The profile sketch for a structural member is a library feature part that contains only a sketch. You can create your own profiles to supplement those provided with the system. The search path for weldment profiles is controlled by **Tools, Options, System Options, File Locations, Weldment Profiles**. The default weldment profiles supplied with the software are located in `install_directory\data\weldment profiles`.

Tip

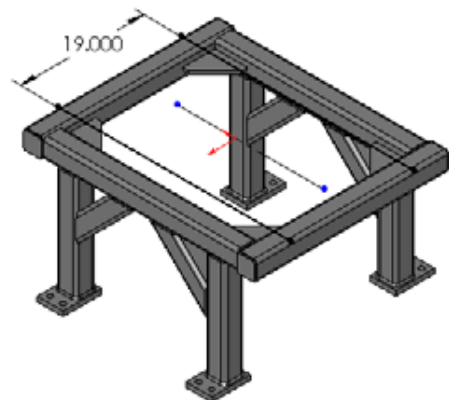
If you customize the weldment profiles or create new ones, it is good practice to store the weldment profiles outside of the SolidWorks installation directory. When performing a clean installation of SolidWorks, the files in the installation directory will be deleted. If the weldment profiles are stored *outside* this directory, they will not be deleted and will still be available for use in the new installation.

Custom Properties

The profile library parts should carry any custom properties that are commonly used, unique to the profile, and that should be imported into the cut list. For example, the profiles supplied with the software have a custom property named **Description**.


27 Sketch.

Select the reference plane Top of Frame and insert a sketch. Sketch two lines as shown. Use mirroring to make them symmetrical with respect to the origin.



28 Exit the sketch.

29 Insert a structural member.


Click **Structural Member**  or, click **Insert, Weldments, Structural Member**.

For **Standard**, select **ansi inch** and for **Type**, select **angle iron**.

Check the list of available sizes. The size we want is **3 x 3 x 0.25** but it is not listed.

Click **Cancel** to close the PropertyManager.

30 Open the library part.

Click **Open**  or click **File, Open**.

Set **Files of type** to **Lib Feat Part (*.lfp, *.sldlfp)** and browse to the folder: `data\weldment profiles\ansi inch\angle iron`. Select the part **3 x 2 x 0.25** and click **Open**.

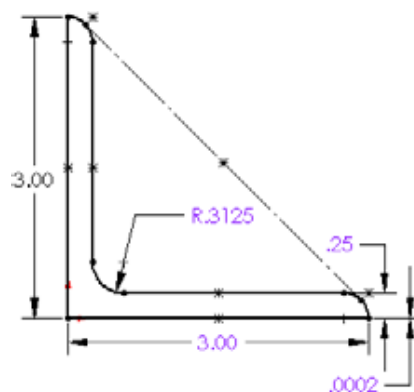
31 Edit the sketch.

Change the horizontal dimension from 2.00" to 3.00".

Sketch a centerline tangent to the two arcs.

Insert a point at the midpoint of the centerline.

Exit the sketch.



Note

The centerline and point will be used to position the structural member later in this case study.

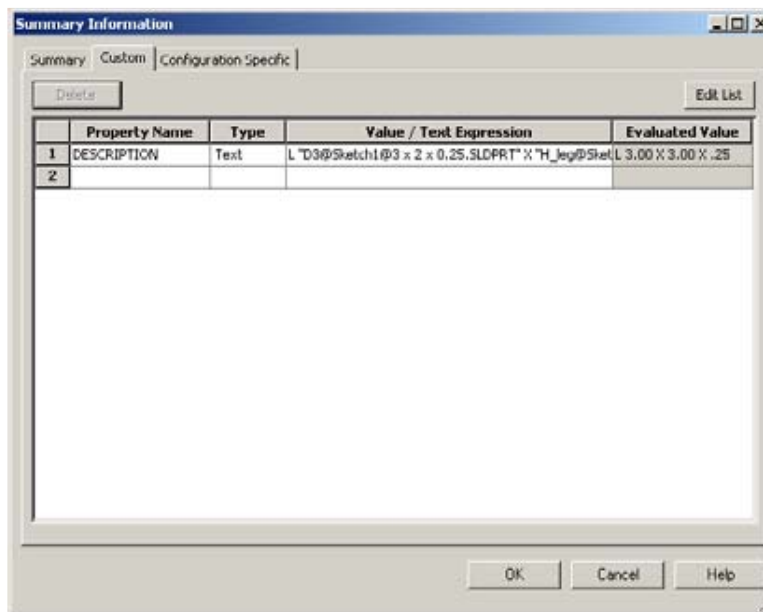
32 File, Save As.

Save the modified library part as 3 x 3 x 0.25.sldlfp.

33 Custom properties.

Click **File, Properties**. Click the **Custom** tab.

Examine the information for the property name **DESCRIPTION**. In the **Value / Text Expression** field, notice the use of associative dimensions. This is what caused the description to update automatically when the dimension was changed.



It is important to have a description associated with the profile because this custom property will be used when the cut list is generated.

For more information about custom properties, see *Custom Properties* on page 146.

34 Close the library part.

35 Insert a structural member.

Click **Structural Member**  or, click **Insert, Weldments, Structural Member**.

For **Standard**, select **ansi inch**.

For **Type**, select **angle iron**.

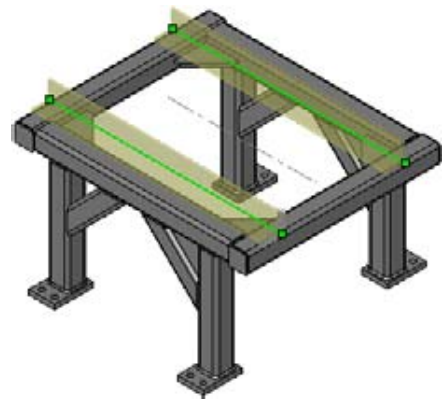
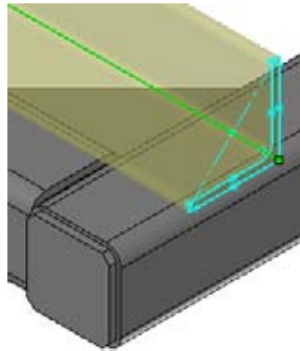
For **Size**, select **3 x 3 x 0.25**. This is the profile we just created.

36 Select the path segments.

37 Click **Locate Profile**.

The profile sketch is shown in color for clarity.

Note

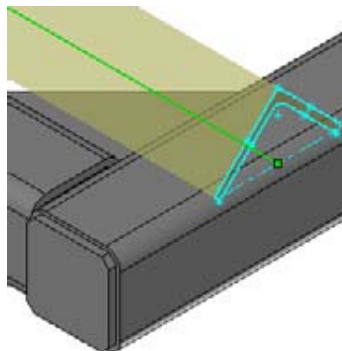
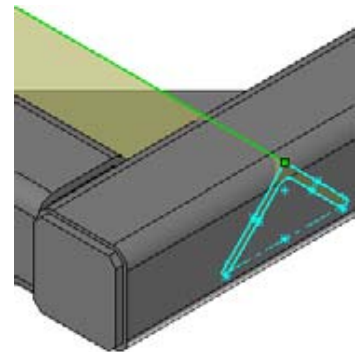


38 Rotate the profile.

Under **Settings**, enter **135°** for the **Rotation Angle**.

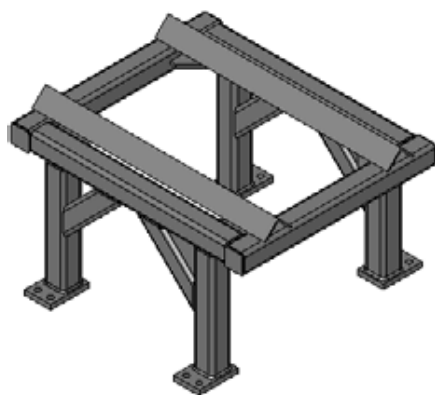
39 Position the profile.

Select the point that is at the midpoint of the centerline.



40 Click OK.

The results are shown in the illustration below.



Working with Weldments

This section addresses several topics:

- Q Creating sub-weldments.
- Q Dealing with components that are not structural members such as castings and bent metal brackets.
- Q Saving solid bodies out to parts – whether individual bodies or sub-weldments.

Sub-weldments

Sometimes it is necessary to break large weldments into smaller sections. Typically this is done to facilitate transportation. These smaller sections are called sub-weldments.



Introducing: Create Sub-weldment


Sub-weldments are folders into which you put the associated solid bodies. You can save a sub-weldment as a separate multibody part. You can then make a drawing of the sub-weldment.

Where to Find It

- Q Right-click one or more solid bodies and select **Create Sub-weldment**.

Tip

If you right-click on a body in the graphics area, **Create Sub-weldment** *does not* appear on the shortcut menu. However, if you turn on **Filter Solid Bodies**  it will appear, and then you can create sub-weldments directly from the graphics window. Since this is so handy, you might want to customize the Weldments toolbar by adding the **Filter Solid Bodies**  button.

Make sure the Weldments toolbar is visible on the screen. Click **Tools, Customize, Commands**. Select **Selection Filter** from the **Categories** list. Drag the **Filter Solid Bodies**  button onto the Weldments toolbar and click **OK**.

Non-structural Components

To incorporate components that are not structural members into a weldment you should make an assembly. The weldment part would be the first component in the assembly. Then you add the other components such as castings or bent metal brackets. You use this procedure for several reasons:

- Q Castings. Castings are treated as separate parts, not bodies within a part. It is much easier to produce the necessary drawings of a casting if it is a separate part.
- Q Bent metal brackets. In order to take advantage of the sheet metal capabilities of the SolidWorks software, bent metal brackets should be modeled as separate parts, not as a bodies within a weldment part. This allows you to take advantage of the unfolding and flat pattern tools within the sheet metal application. It also facilitates making the necessary drawings of the brackets.
- Q Bill of materials. The weldment, castings, and brackets would all be listed as single items on a bill of materials.

Saving Bodies as Separate Parts

You can save any solid body as a separate part file. You can also save a sub-weldment folder as a multibody part.

Introducing: Insert into New Part

Insert into New Part allows you to save individual structural members (solid bodies) as separate part files. Each resulting part file is linked by an external reference back to the source part - the weldment. A `Stock-<source part name>` feature appears in the saved part. This feature carries the external reference. For more information about external references, refer to the *Advanced Assembly Modeling* training course.

Default Templates

Insert into New Part creates a new SolidWorks part. You have the option of specifying a document template or allowing the system to use the default template. This choice is determined by the settings in **Tools, Options, System Options, Default Templates**.

Note

If you select multiple structural members, the `Cut List` folder, or a `Sub-weldment` folder, the saved part will be a multibody part with a `Stock` feature for each body. This enables you to save a sub-weldment as a separate part.

Any changes you make to the source part will propagate to the saved parts.

Where to Find It

- Q Right-click what you want to save – the structural member or the folder – and select **Insert into New Part** from the shortcut menu.

For more information about saving solid bodies as parts and assemblies, see *Saving Solid Bodies as Parts and Assemblies* in the *Advanced Part Modeling* book.

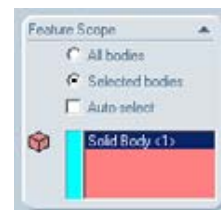
**Post-assembly
Machining
Operations**

Use traditional cut features to remove material representing post-assembly machining operations. Use the feature scope to control which solid bodies are affected.

Feature Scope

The **Feature Scope** allows you to select which bodies are affected by a feature. The **Feature Scope** option exists in the following commands:

- | | |
|--------------------|-----------|
| Q Extrude | Q Revolve |
| Q Sweep | Q Loft |
| Q Cut with Surface | Q Thicken |



Expand the **Feature Scope** group box. Clear the **Auto-select** check box. Select the bottom half of the clamp and click **OK**.

For more information about feature scope, refer to the topic *Feature Scope* in the Advanced Part Modeling book.

Important!

Be sure to suppress the cut features in the As Welded configuration.

**Managing the
Cut List**

A cut list is like a bill of materials for a multibody part. Like items are grouped together in a special folder called a *cut list item*. These folders inherit properties from the sketch profiles and the weldment feature. In addition you can assign other properties as needed. For example, the description of a structural member comes from the profile sketch. Gussets do not have sketches. Therefore, to add a description for a gusset, you add it as a custom property of the cut list item.

Weld Beads

Weld beads are not put in a cut list item because you do not want them listed in the cut list table on the drawing.

Balloons

In order to create balloons on the drawing, the bodies must belong to a cut list item.

**Automatically
Generating the Cut
List**

SolidWorks can automatically generate the cut list. The system identifies the bodies that are the same – have the same profile, length, corner treatment, etc. – and groups them into cut list items.

**Introducing: Update
Cut List**

In the weldment part you create one or more cut list items. This enables you to insert a cut list table on the drawing and create balloons calling out the various items on the drawing. The process of putting solid bodies into cut list folders is automatic.

Where to Find It

- Q Right-click the Cut List feature in the FeatureManager design tree, and select **Update** from the shortcut menu.

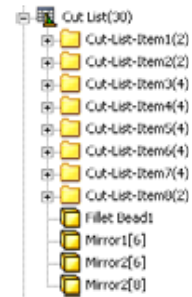
Note

If the resulting cut list is not as you want it, you can manually make changes by dragging and dropping solid bodies from one folder to another. However, this will cause the system to disable the automatic update. You will still be able to manually create cut list items by right-clicking one or more solid bodies and selecting **Create cut list item** from the shortcut menu.

41 Generate the cut list.

Right-click the Cut List feature in the FeatureManager design tree, and select **Update** from the shortcut menu.

The system automatically generates the cut list items.



42 Renaming the cut list items.

The default cut list item names are not very descriptive. Use the following table as a guide for renaming the cut list items.

Name of Cut List Item	Selected Bodies	Name of Cut List Item	Selected Bodies
SIDE TUBES		FRONT/REAR TUBES	
ANGLED BRACES		LEGS	
RAILS		BASE PLATES	
END CAPS		GUSSETS	

Custom Properties

Custom properties can be assigned and controlled from three different sources:

- Q The profile sketch: You should assign to the profile sketch any properties that are commonly used, unique to the profile, and that you want inherited by the cut list item. An example is **DESCRIPTION** as was seen in step 33 on page 141.

- Q The weldment feature: The properties assigned to the weldment feature are inherited by all cut list items. This allows you to assign a property with a “default” value. Then you can simply edit the value for each cut list item. An example would be **COST** with a value of **TBD** (to be determined).
- Q The cut list item: The cut list item inherits custom properties from the profile sketch and weldment feature. You can assign additional properties or edit the values of the existing ones. For example, you can add the property **WEIGHT** and link it to the part’s mass properties. The system correctly computes the weight of the first solid body in the cut list item. In addition, the system computes and adds the properties **LENGTH** and **ANGLE**.

Note In SolidWorks 2005 only one material can be assigned to a part. It is not possible to assign different materials to separate bodies. This single material will be used in all calculations involving density.

List of Properties The list of properties associated with weldments is maintained in the `weldmentproperties.txt` file. It is located in `install_directory\lang\<language>\weldments`.

Capitalization Most drafting departments have standards regarding capitalization of text in tables and other annotations. The format of the weldment properties text file determines how the properties appear in the cut list table on the drawing. If necessary, reformat the text file to meet your needs.

43 Define the material.

Right-click **Material** feature in the FeatureManager design tree, and select **Plain Carbon Steel** from the shortcut menu.

44 Custom properties.

Right-click the cut list item **Angled Braces** and select **Properties**. The **LENGTH** property was computed automatically and added by the system. This represents the cut length of the structural member. You cannot change it or delete it. Likewise the **ANGLE** properties were computed and added by the system.

The **DESCRIPTION** property is inherited from the sketch profile. Click in the empty cell in the Property Name column and select **WEIGHT** from the list of properties. In the **Value / Text Expression** field, select **Mass**. The field will be filled in with the expression: `"SW-Mass@@@Angled Braces@Conveyor Frame.SLDPRT"`. The **Evaluated Value** reads **6.965**.

Click **OK**.

Optional

Add the **WEIGHT** property to the remaining cut list items. If you do not, the weights of the other cut list items will not be displayed in the cut list table on the drawing (see step 55). Only the Angled Braces cut list item will have a weight displayed.

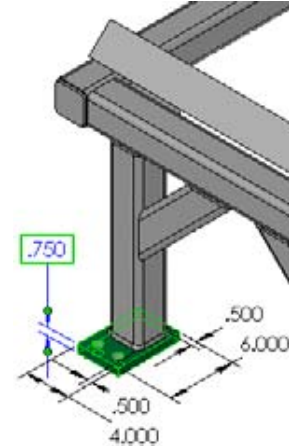
45 Description for Base Plates cut list item.

Right-click the Base Plates cut list item and select **Properties**.

From the list of properties select **DESCRIPTION**. Click in the **Value** field. Double-click the Plate feature to display its dimensions.

Select the **.750"** dimension. Type a space. Then type: **IN. PLATE**.

The value should read:
"D1@Plate@Conveyor Frame .SLDPRT" IN. PLATE.
Click **OK**.



Cut List Properties

At this time you would continue this process to add or edit any of the properties associated with the cut list items. For example, you would add descriptions to the gussets and end caps. Since they have no profile sketches a description is not supplied automatically.

46 Descriptions for End Caps and Gussets cut list items.

Repeat the procedure in step 45 for the End Caps and Gussets cut list items. The end caps are **.25"** thick and the gussets are **.375"**.

Weldment Drawings

Making a drawing of a weldment is the same as making a drawing of any other part but with two exceptions:

- Q Adding views of individual solid bodies.
- Q Adding a cut list table.


This section is not intended to be a comprehensive treatment of drawing production. The intent is only to illustrate those topics that are unique to weldments. For a thorough discussion of creating engineering drawings you should attend the course *SolidWorks Essentials: Drawings*.

47 Setup.


Click **Tools, Options, System Options, Drawings, Display Style**. For **Tangent edges in new views**, select **Use font**.

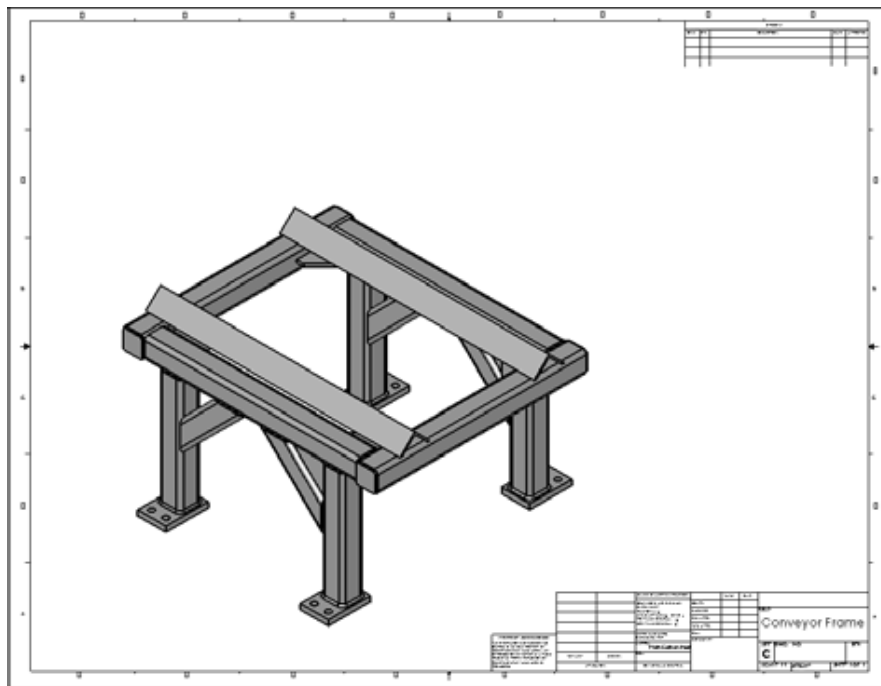
Click **OK**.

48 Create a drawing.

Click **Make Drawing from Part**  on the Standard toolbar. Click the **Training Templates** tab. Select the **C_Size_ANSI_Inch** template and click **OK**.


Under **Orientation** select **Isometric**. Under **Scale**, click **Use custom scale** and set the scale to **1:4**.

Position the view on the drawing. Click **Shaded with Edges** 

**Drawing Views of Individual Bodies**

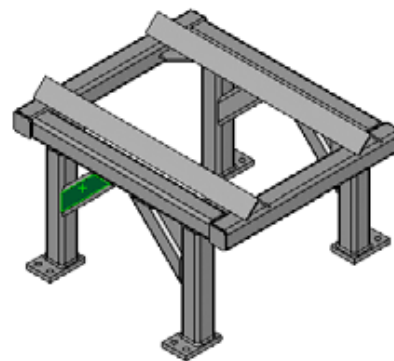
When you create a relative view, if you select both orientation faces from the same body, you specify whether you want the view to be of the selected body or the entire part. The default is the selected body.

49 Relative view.

Click **Relative View**  on the Drawing toolbar.

Press **Ctrl+Tab** to switch back to the part document window.

Make sure **First orientation** is set to **Front** and select the front face of the angled brace as shown.

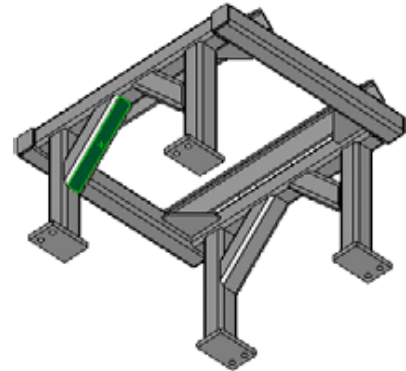


50 Second orientation.

Set **Second orientation** to **Bottom**.
Rotate the view and select the face
shown.

Click **OK**.

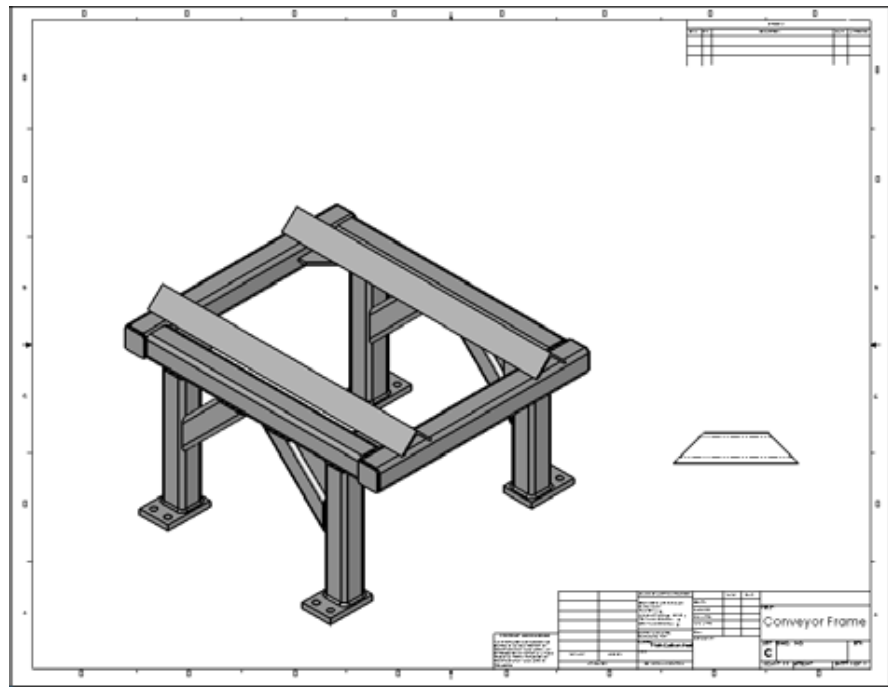
The drawing document window
automatically appears.



51 Scale.

Select **Use custom scale** and set the scale to **1:4**.


Position the view on the drawing as shown below and click **OK** to close
the **Relative View** PropertyManager.

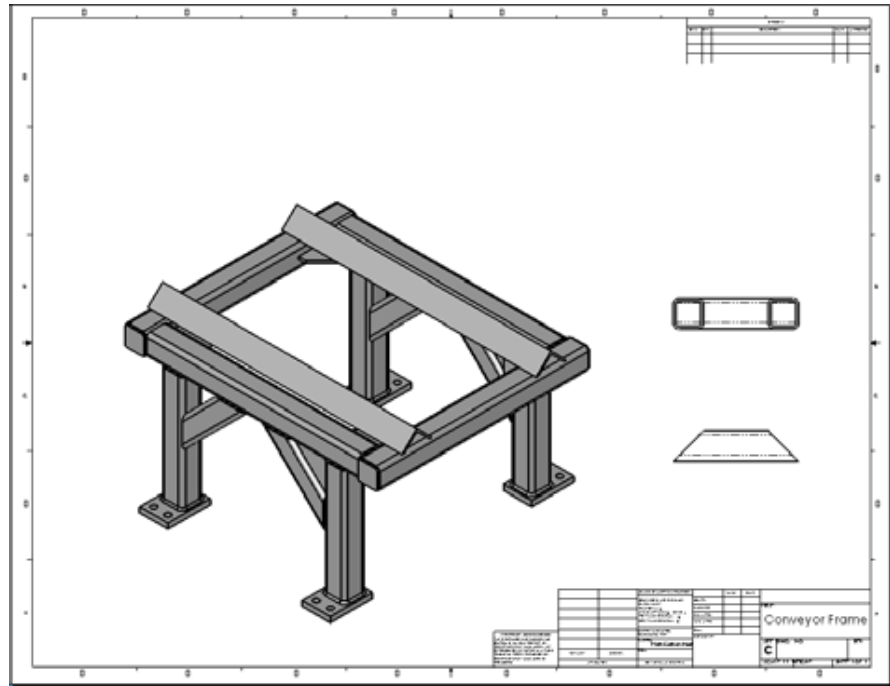


Note

Currently **Insert, Model Items** does not work on single body views.
This is because dimensions are associated with a feature, not a body.
For example, in step **10** on page 131 when we created the leg and
angled brace, one structural member feature created two solid bodies.
Subsequently, the angled brace was mirrored creating a total of 4 solid
bodies. At this time there is no way to associate the dimensions from
the profile sketch (which was normal to the path segment of the leg)
and display them correctly in a drawing view of the angled brace.

52 Projected view.

Click **Projected View**  on the Drawing toolbar. Select the view of the angled brace and position the projected view above it as shown below.

**Cut List Tables**

Insert a cut list table to create a list of the components in the weldment. The properties associated with the cut list items are imported into the table.


Where to Find It

- Q Click **Weldment Cut List**  on the Table toolbar.
- Q Or, click **Insert, Tables, Weldment Cut List**.

53 Setup.

Click **Tools, Options, Document Properties, Annotations Font**. Click **Tables** and set the font to **14 pt. Century Gothic**. Click **OK**.

54 Cut list table.

Click **Weldment Cut List**  on the Table toolbar, or click **Insert, Tables, Weldment Cut List**.

Select the isometric view of the weldment.

Under **Table Anchor**, select **Top Left**  and select **Attach to anchor**.

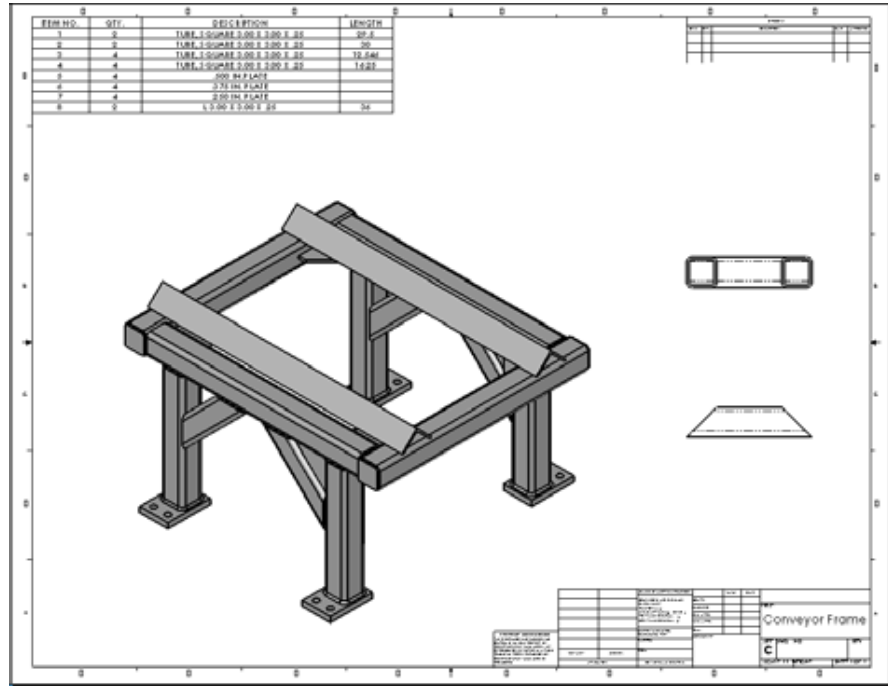
Click **OK**.

55 Results.

The table appears in the upper left corner of the drawing.

Note

In step 44, if you did not add the **WEIGHT** property to *all* the cut list items, then only Angled Braces, will have a weight displayed in the table.



Cut List Table Commands

There are several specialized commands for working with cut list tables. They are accessed by right-clicking in the table and selecting the command from the shortcut menu. The commands are:

Q Insert Column Right Column Left Row Above Row Below	Q Select Table Column Row
Q Delete Table Column	Q Formatting Column Width Row Height
Q Sort Ascending Descending	Q Split Horizontally Above Horizontally Below Vertically Left Vertically Right
Q Merge	Q Save as Template

56 Add a column to the table.

Right-click in the column labeled LENGTH and select **Insert, Column Right**.

In the PropertyManager, click **Cut list item property** and select **WEIGHT** from the list.

Click **OK**.

ITEM NO.	QTY.	DESCRIPTION	LENGTH	WEIGHT
1	2	TUBE, SQUARE 3.00 X 3.00 X .25	29.5	21.522
2	2	TUBE, SQUARE 3.00 X 3.00 X .25	30	21.887
3	4	TUBE, SQUARE 3.00 X 3.00 X .25	12.546	6.965
4	4	TUBE, SQUARE 3.00 X 3.00 X .25	16.25	11.856
5	4	.500 IN. PLATE		4.870
6	4	.375 IN. PLATE		1.796
7	4	.250 IN. PLATE		0.524
8	2	L 3.00 X 3.00 X .25	36	14.524

57 Add a second column to the table.

Right-click in the column labeled DESCRIPTION and select **Insert, Column Left**.



In the PropertyManager, select **Cut list item name**.

In the **Title** field edit the text if desired and click **OK**.

ITEM NO.	QTY.	CUT LIST NAME	DESCRIPTION	LENGTH	WEIGHT
1	2	SIDE TUBES	TUBE, SQUARE 3.00 X 3.00 X .25	29.5	21.522
2	2	FRONT/REAR TUBES	TUBE, SQUARE 3.00 X 3.00 X .25	30	21.887
3	4	ANGLED BRACES	TUBE, SQUARE 3.00 X 3.00 X .25	12.546	6.965
4	4	LEGS	TUBE, SQUARE 3.00 X 3.00 X .25	16.25	11.856
5	4	BASE PLATES	.500 IN. PLATE		4.870
6	4	GUSSETS	.375 IN. PLATE		1.796
7	4	END CAPS	.250 IN. PLATE		0.524
8	2	RAILS	L 3.00 X 3.00 X .25	36	14.524

Formatting the Table

There are two ways to change the height of a row and the width of a column. You can:

- Q Drag the cell boundary lines. Position the cursor over the cell boundary line. The cursor changes to  if you are resizing the column width, or  if you are resizing the row height. Drag the cell boundary line to the desired position.
- Q Right-click in the cell and select **Formatting, Column Width** or **Formatting, Row Height**. A dialog box will appear. Enter the appropriate value and click **OK**.

Tip

To change the height of all the rows at once, select the table. Right-click in the blue border. Select **Formatting, Row Height**. The value will be applied to all the rows except the header.

58 Resize columns and rows as desired.

ITEM NO.	QTY.	CUT LIST NAME	DESCRIPTION	LENGTH	WGHT
1	2	SIDE TUBES	TUBE, SQUARE 3.00 X 3.00 X .25	29.5	21.522
2	2	FRONT/REAR TUBES	TUBE, SQUARE 3.00 X 3.00 X .25	30	21.887
3	4	ANGLED BRACES	TUBE, SQUARE 3.00 X 3.00 X .25	12.546	6.965
4	4	LEGS	TUBE, SQUARE 3.00 X 3.00 X .25	16.25	11.856
5	4	BASE PLATES	.500 IN. PLATE		4.870
6	4	GUSSETS	.375 IN. PLATE		1.796
7	4	END CAPS	.250 IN. PLATE		0.524
8	2	RAILS	L 3.00 X 3.00 X .25	36	14.524

59 Set up balloon options.

Click **Tools, Options, Document Properties, Balloons.**

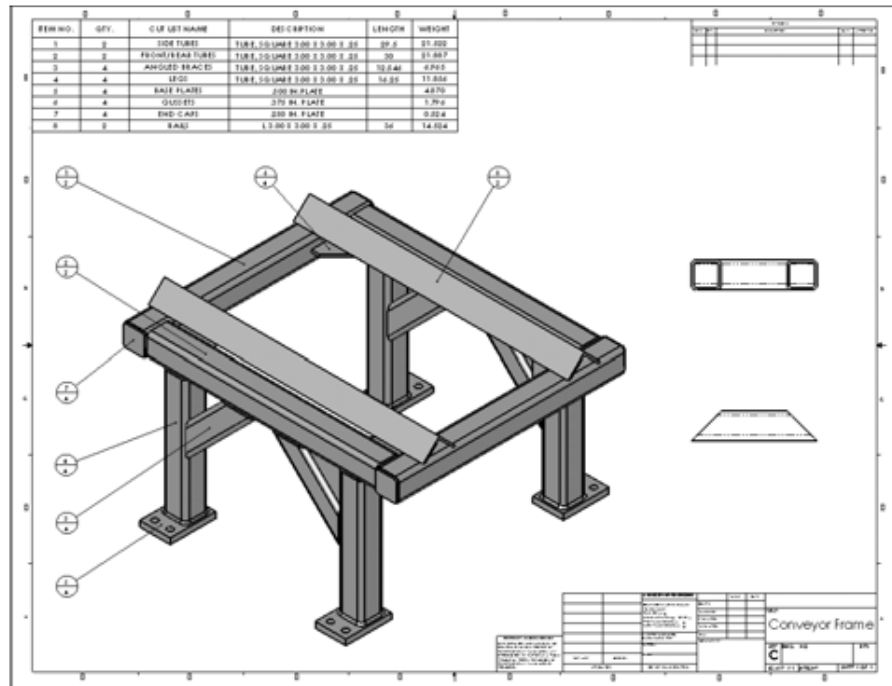
Under **Single balloon**, click **Circular Split Line.**

Under **Balloon text**, set **Upper** to **Item Number** and set **Lower** to **Quantity.**

Click **OK.**

60 Add balloons.

Click **Balloon** on the Annotation toolbar. Add balloons to the isometric view and arrange them as desired.



61 Save and close all files.

Working with Pipes and Tubing

In this section we will examine some final aspects of weldments:

- Q 3D sketching
- Q Bent tubes
- Q Special trimming situations
- Q Combining sheet metal and weldments
- Q Integrating weldments into assemblies



3D Sketching

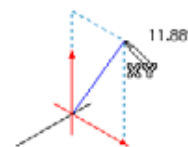
As the name implies, the entities in a 3D sketch are not constrained to a single plane as they are in a traditional 2D sketch. This makes 3D sketches very useful for applications such as weldments. However, 3D sketching can sometimes be a bit of a challenge.

Subset of Sketch Entities and Relations

There are fewer entities and sketch relations available in 3D sketches compared to 2D sketches. For example, only lines, sketch fillets, sketch chamfers, reference points, and splines are available in 3D sketches. The circle, arc, and rectangle tools not available nor is mirroring. Likewise, certain sketch relations such as the equal relation are not available in 3D sketches. However, other relations such as **ParallelYZ**, **ParallelZX** and **AlongZ** are *only* available in a 3D sketch.

Space Handle

When working in a 3D sketch, a graphical assistant is provided to help you maintain your orientation while you sketch on several planes. This assistant is called a **Space Handle**. The space handle appears when the first point of a line or spline is defined on a selected plane. Using the space handle, you can select the axis along which you want to sketch.



By default, you sketch relative to the default coordinate system (Front reference plane) in the model. To switch to one of the other two default planes, press **Tab** while the sketch tool is active. The origin of the current sketch plane is displayed.

Reference Planes

One of the keys to successful 3D sketching is to plan ahead and predefine a number of reference planes that will help you sketch.

Where to Find It


- Q Click **3D Sketch**  on the Sketch toolbar.
- Q Or click **Insert, 3D Sketch**.

-
- 1 **New part.**
Create a new part with units **inches**.
Name the part **Frame**.

2 Reference planes.

Create three reference planes as shown in the illustration at the right. Notice the use of **Link Values** for Plane2 and Plane3.


3 Open a new 3D sketch.

Click the **3D Sketch** tool  to start a new sketch.

4 Select the reference plane.

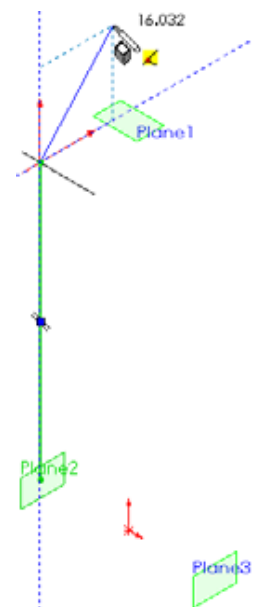
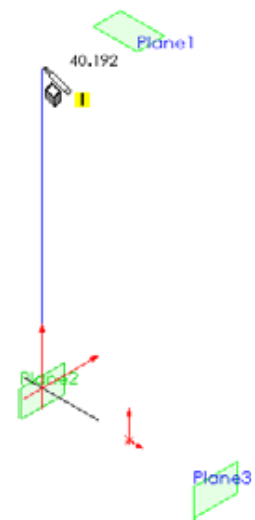
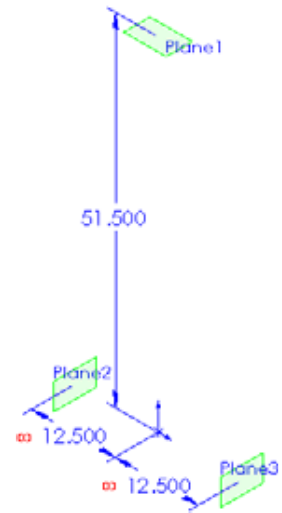
Select Plane2. Preselecting the plane shifts the space handle to that plane from the default plane.

5 Sketching a line.

Click the **Line** tool . The space handle appears on Plane2. Starting at the origin of the space handle, sketch a vertical line approximately **40"** long.

6 Angled line.


Sketch a second line at an angle to the first as shown in the illustration. We are still sketching on Plane2.

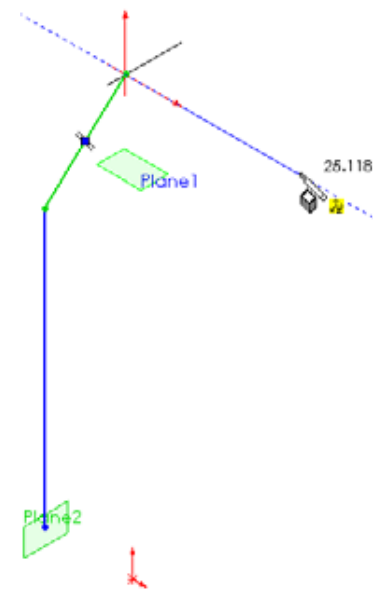


7 Reorient the space handle.

Press the **Tab** key to orient the space handle as shown. Sketch a line approximately **25"** long.

Note

The **X** axis of the space handle is oriented parallel to the **Z** axis of Plane2. This is why the inference symbol next to the cursor indicates the relation **AlongZ** 

**8 Add relations.**

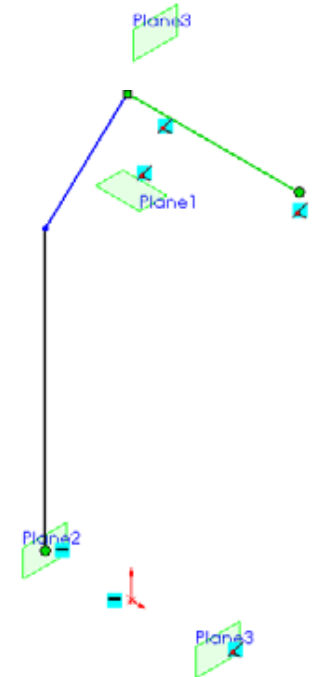
Add a **Horizontal** relation between the bottom end of the first line and the part's origin.

Add a **Coincident** relation between the third line and Plane1.

Add a **Coincident** relation between the end of the third line and Plane3.


Note

The graphics have been enhanced for illustration purposes. Only the callouts for the relations being applied in this step are shown.

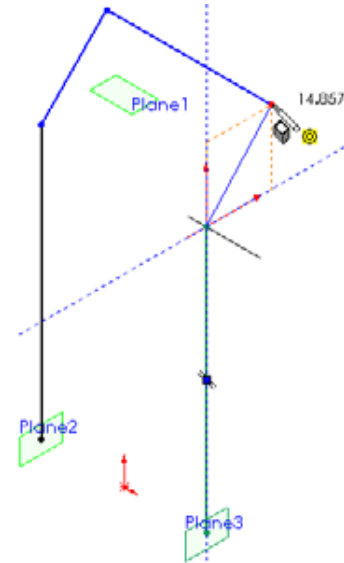


9 Resume sketching.

Select Plane3. The space handle appears on Plane3.

Click the **Line** tool . The space handle appears on Plane3. Starting at the origin of the space handle, sketch a vertical line approximately 40" long.

Sketch a second angled line ending it so that it is **Coincident** with the end of the horizontal line.



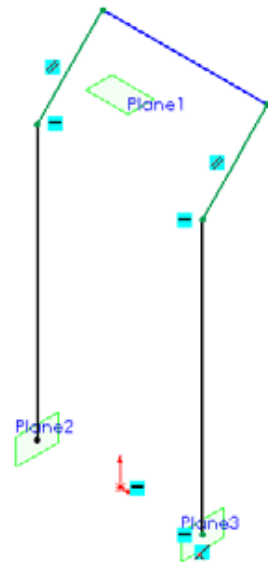
10 Add relations.

Add a **Horizontal** relation between the bottom end of the vertical line and the part's origin. Verify that a **Coincident** relation exists between the endpoint of the vertical line and Plane3. Add a **Horizontal** relation between the uppermost ends of the two vertical lines.

Add a **Parallel** relation between the two angled lines.

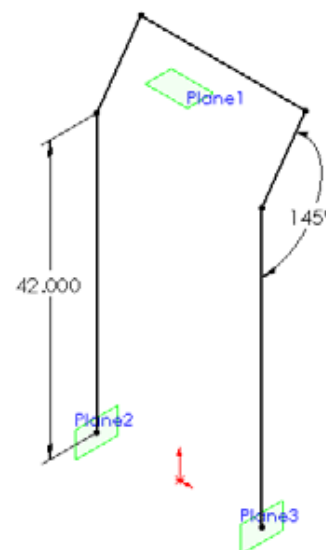
Note

The graphics have been enhanced for illustration purposes.



11 Add dimensions.

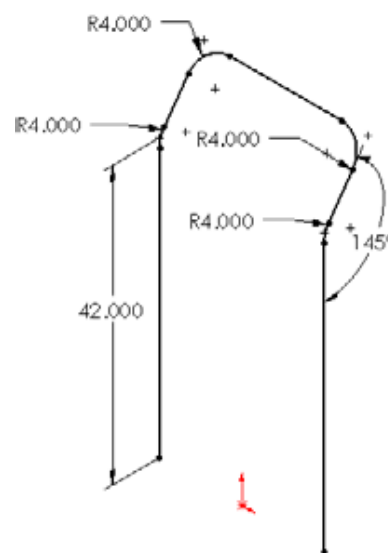
Add the dimensions shown to fully define the sketch.

**12 Add fillets.**

Fillet the four sharp corners with 4" radius fillets.

Note

The reason there is a dimension on each of the fillets is because 3D sketches do not have the **Equal** relation.

**13 Exit the sketch.****14 Weldment profile.**

Rather than take the time to create a custom weldment profile, copy the seamless tube folder from the Case Study folder of this lesson and put it in the data/weldment profiles folder in the SolidWorks installation directory.

Note

To review creating a custom profile, please see *Profile Sketches* on page 140.

15 Preselect the path.

Right-click the vertical line and select **Select Chain** from the shortcut menu.

16 Insert the structural member.

Click **Structural Member**  or, click **Insert, Weldments, Structural Member**.

For **Standard**, select **ansi inch**.

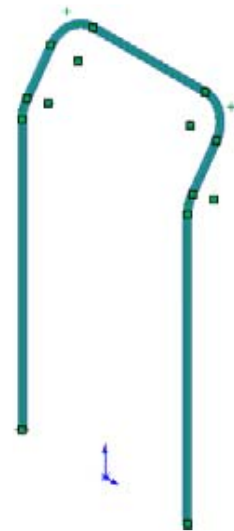
For **Type**, select **seamless tube**.

For **Size**, select **1.050 o.d. x 0.100 wall**.

Since the path segments were preselected, the preview appears automatically.

17 Select the Merge arc segment bodies checkbox.

Then click **OK** to create the structural member.



Merge Arc Segment Bodies

When the path consists of a series of lines and arcs, the option **Merge arc segment bodies** appears in the PropertyManager. Whether or not this option should be checked depends on how the weldment will be fabricated.

If the weldment is fabricated using bent tubing, the option should be checked. This will give you a model where each bent tube is a separate solid body.

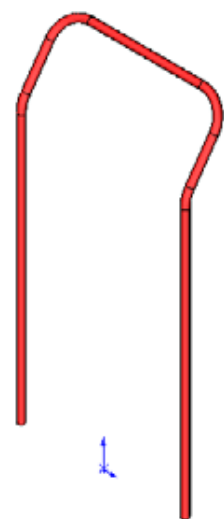
If the weldment is fabricated using elbows and straight tubes, the option should be cleared. This will give you a model where each straight segment is a separate body and each arc segment (elbow) is a separate body.

18 Results.

The structural member is represented as a single solid body.

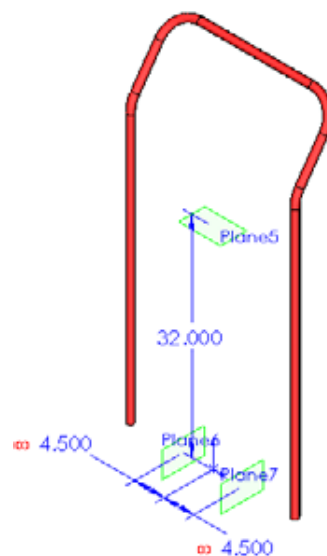
Next Step

Next we will create the sketch for the second structural member.



19 Reference planes.

Create three reference planes as shown in the illustration at the right. Notice the use of **Link Values** for Plane6 and Plane7.



20 2D sketch.

Select Plane6 and open a 2D sketch.

21 Hide solid bodies and show sketch.

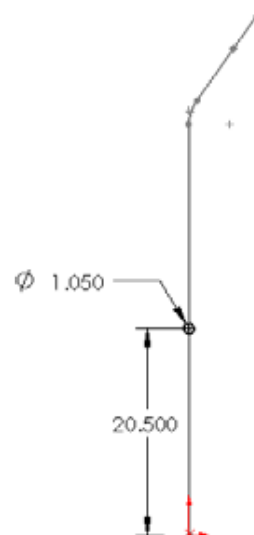
Right-click the Cut List folder and select **Hide Bodies** from the shortcut menu. Verify that the 3D sketch is showing.

22 Change view.

Change to the Right view.

23 Sketch a circle.

Sketch a 1.050" diameter circle and position it as shown. The center of the circle is **Coincident** with the vertical line in the 3D sketch.



Why?

The hand truck has a movable platform (the fold nose extension) that folds up as shown below. The circle will be used to determine the angle of the structural member so it is bent back far enough to provide clearance when the platform is in the up position.



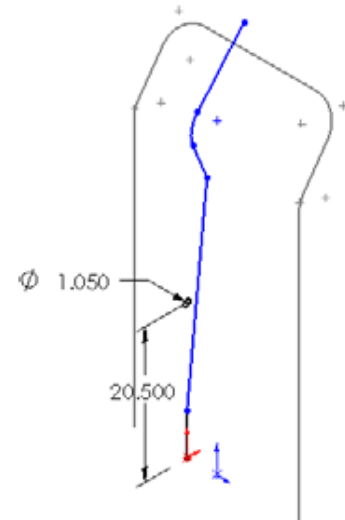
Extension in down position



Extension in up position

24 Continue sketching.

Starting at the sketch origin, sketch a vertical line approximately 5" long. Then sketch the remaining lines and tangent arc as shown.



25 Add relations.

Add a **Coradial** relation between the tangent arc and the 4" fillet in the 3D sketch.

Add a **Parallel** relation between the uppermost line and the angled line in the 3D sketch.

Add a **Pierce** relation between the end of the angled line and the horizontal line in the 3D sketch.



Note

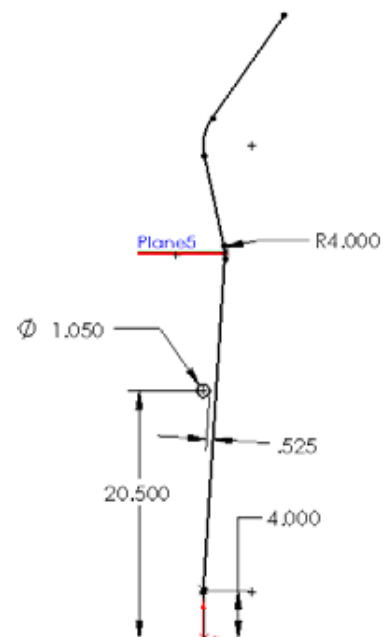
Only the relations being added are shown in this illustration. Other relation callouts such as **Tangent** were removed for clarity.

26 Fully define the sketch.

Add a **Coincident** relation between the uppermost sharp corner and Plane5. Dimension the vertical line so it is 4" long.

Insert 4" fillets on the two sharp corners. Be sure the option **Keep constrained corners** is selected.

27 Exit the sketch.



28 Derived sketch.

Select the 2D sketch and Plane7.

Click **Insert, Derived Sketch**.

29 Fully define the sketch

Add a **Coincident** relation between the lowermost end of the vertical line and the origin.

Add a **Collinear** relation between the vertical line and the Front reference plane.

30 Exit the sketch.

31 Insert two structural members using 1.050" o.d. seamless tube.


This has to be done in two operations because the paths are not connected to each other.

Tip

Preselect the path by right-clicking the first path segment and picking **Select Chain** from the shortcut menu.

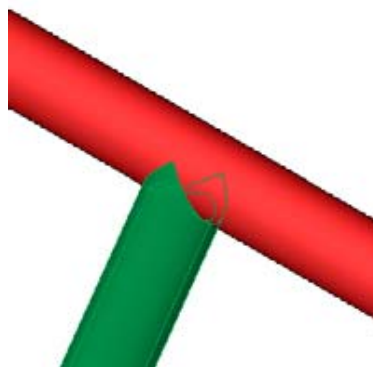
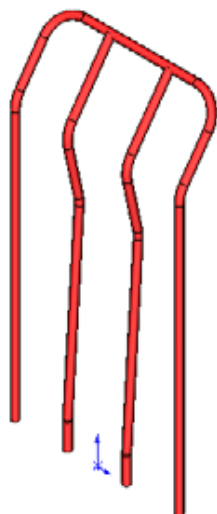
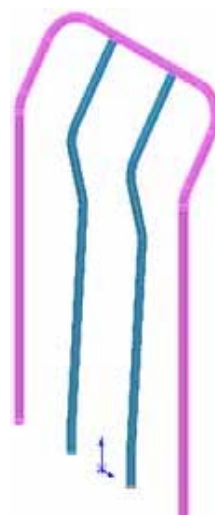
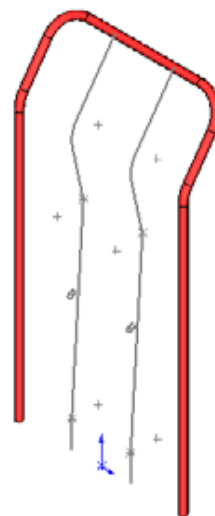
32 Trimming structural members.

Click **Trim/Extend**  or, click **Insert, Weldments, Trim/Extend**.

For **Corner treatment**, select **End Trim**  .
For **Bodies to be Trimmed**, select Structural Member2 and Structural Member3 that were created in the previous step.

For **Trimming Boundary**, click **Bodies** and select Structural Member1.

Click **OK**. The results are shown below.



Close-up of trimmed body

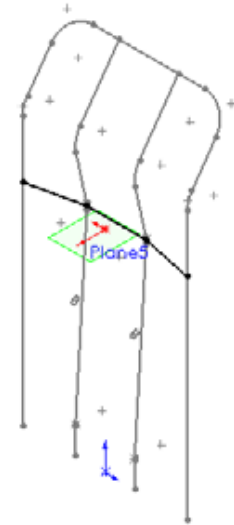
33 New 2D sketch.

Right-click the **Cut List** folder and select **Hide Bodies** from the shortcut menu.

Verify that the three path sketches are showing.

Select **Plane5** and open a 2D sketch.


Sketch three lines as shown. The ends of the lines are constrained to the 3D sketch and the *arcs* in the two 2D sketches using **Pierce** relations.



34 Structural member.

Exit the sketch and insert a structural member using 1.050" o.d. seamless tube.

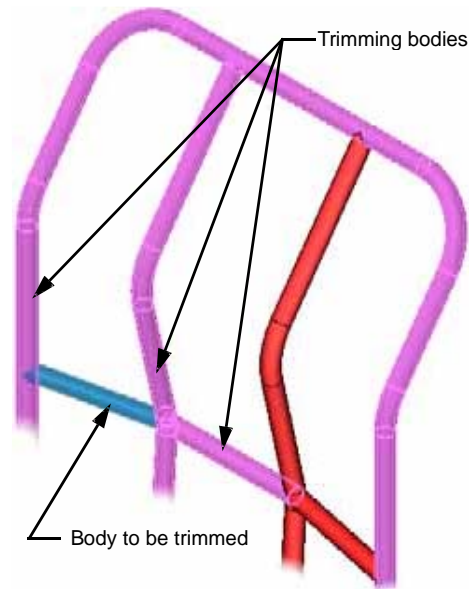
35 Trimming structural members.

Click **Trim/Extend**  or, click **Insert, Weldments, Trim/Extend**.

For **Corner treatment**, select **End Trim** .

For **Bodies to be Trimmed**, select body as shown in the illustration.

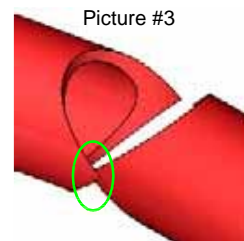
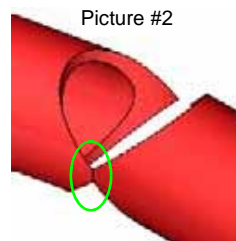
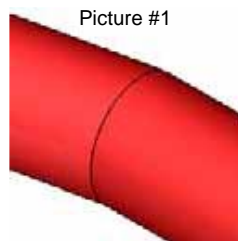
For **Trimming Boundary**, click **Bodies** and select the three bodies as shown in the illustration.



Click **OK**.

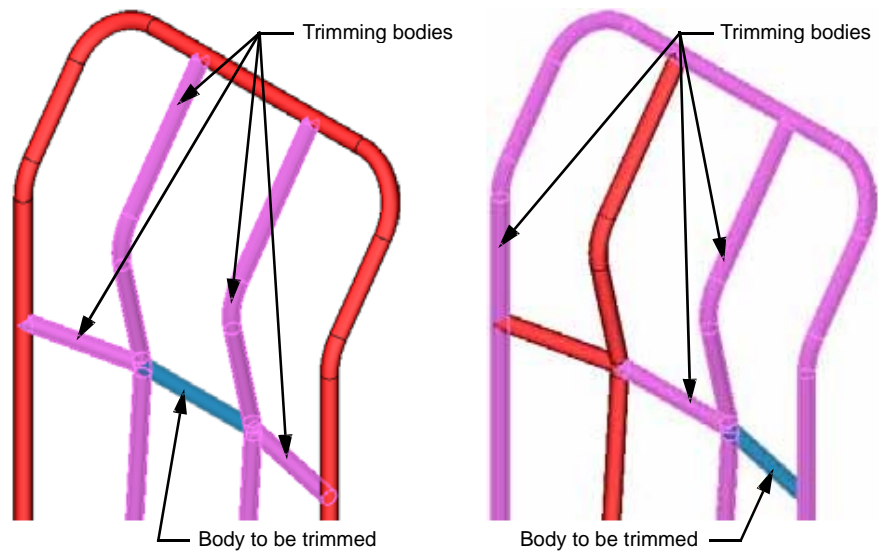
Important!

Structural Member4 consists of three separate solid bodies. When they were created they were automatically trimmed with respect to each other. (Picture #1) When you trim one of the bodies in a structural member, you must select *all* the bodies that intersect with it. If you do not, you will get the result shown in Picture #3.



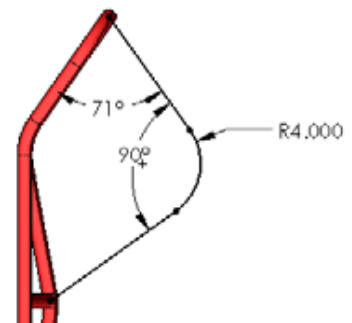
36 Repeat trimming.

Trim the remaining two segments of Structural Member 4.



37 New 2D sketch.

Select the Right reference plane and create the sketch shown in the illustration at the right.



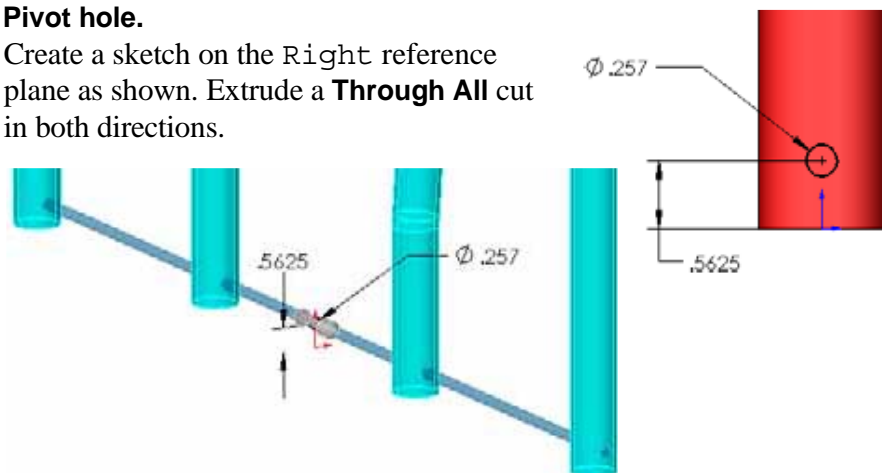
38 Insert a structural member and trim.

Once again use 1.050" o.d. seamless tube.



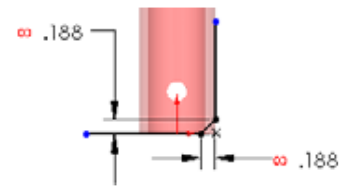
39 Pivot hole.

Create a sketch on the Right reference plane as shown. Extrude a **Through All** cut in both directions.



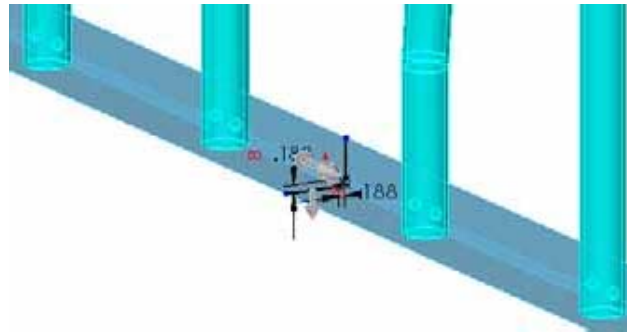
40 Chamfer.

Create a sketch on the Right reference plane as shown. Extrude a **Through All** cut in both directions.



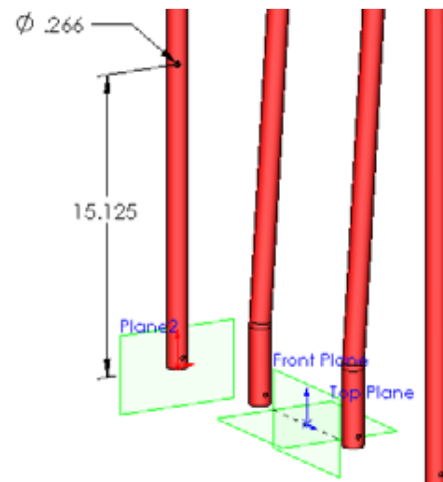
Note

The solid bodies were made transparent for illustration purposes.



41 Hole for retaining pin.

Create a 2D sketch on Plane2 as shown. The center of the circle is **Coincident** with the Front reference plane and **15.125"** above the Top reference plane. Extrude a cut using the **Up To Next** end condition in both directions.



42 Save the part.

Weldments and Sheet Metal in Assemblies

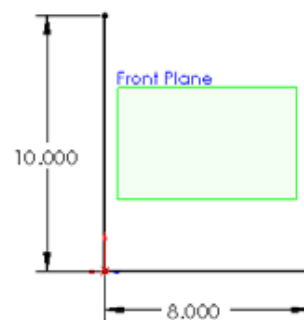
Weldments can be built using top-down assembly modeling techniques in the same way as any other part.

43 New part.

Create a new part with units **inches**. Name the part **Toe Plate**.

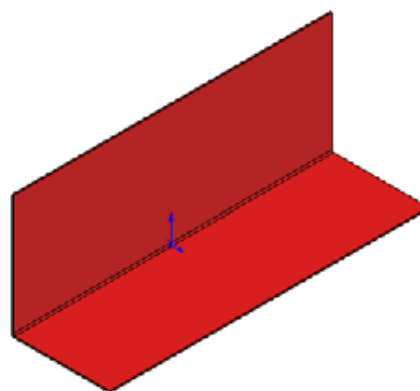
44 Sketch.

Create a sketch on the Front reference plane as shown.

**45 Base flange.**


Extrude a base flange a depth of **26"** using the **Mid Plane** end condition. For **Thickness** use **0.125"**. For the default **Bend Radius**, use **0.125"**.

The material should be added to the inside of the sketch as shown below.

**46 Cutout.**

Create a cutout in the Toe Plate as shown. Use the end condition **Blind** with the **Link to thickness** option.

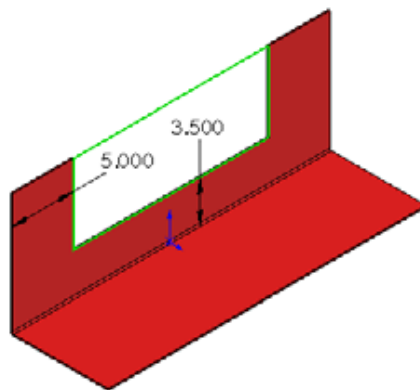
47 Create an assembly.

Click **Make Assembly from Part/Assembly**  and select the Assembly_IN template.


In the **Part/Assembly to Insert** list, make sure the Toe Plate is selected. Click **OK** to insert the component at the origin.

48 Save the assembly.

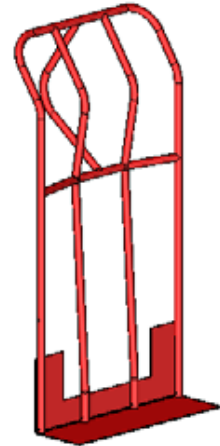
Name the file Extra Wide Fold Nose Truck.



49 Insert a component.

Click **Insert Components**  Add the Frame component to the assembly and mate it as follows:

- Q Right reference plane of the Frame is **Coincident** with the Front reference plane of the assembly.
- Q Vertical face of the Toe Plate is **Tangent** to the cylindrical face of the Frame.
- Q Horizontal face of the Toe Plate is **Coincident** to the bottom face of the structural member of the Frame.

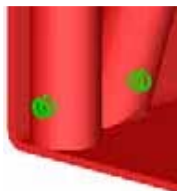
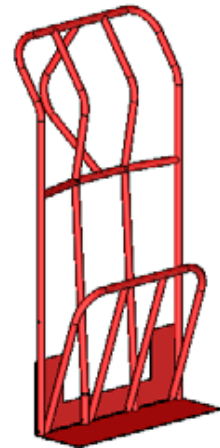


50 Insert a component.


Click **Insert Components**  Click **Browse** and select the Fold Nose Extension.

Mate the component as follows:

- Q Right reference plane of the Fold Nose Extension is **Coincident** with the Front reference plane of the assembly.
- Q The hole in the bottom of the Frame is **Concentric** with the hole in the Fold Nose Extension as shown below.

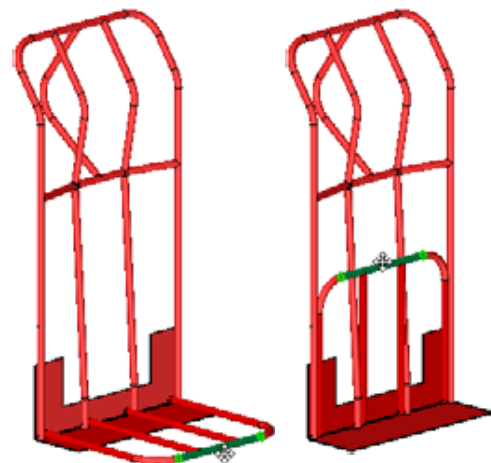


51 Collision detection.

Click **Move Component**  Under **Options**, select **Collision Detection**. Verify that the **Stop at collision** option is also selected.

Drag the Fold Nose Extension between its lower and upper limits of motion.

Leave the Fold Nose Extension in its uppermost position and turn off the **Move Component** command.



52 Add a mate.


Add an **Angle** mate of **0°** between the Top reference plane of the Fold Nose Extension and the Right reference plane of the assembly. This ensures the Fold Nose Extension is in the correct position prior to making the hole for the retaining pin.

53 Test the mate.

Change the angle value to **90°**. Be sure the Fold Nose Extension moves to the correct position when you rebuild the assembly. If it moves the wrong direction, flip the direction of the dimension.

When finished testing, change the value back to **0°**.


54 Edit the part.

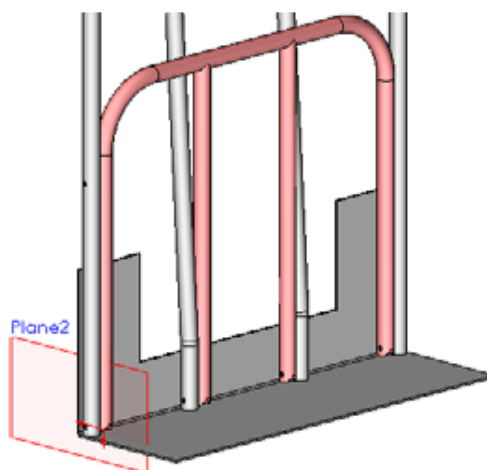
Select the Fold Nose Extension and click **Edit Component** 

55 Hole for retaining pin.

Create a 2D sketch on Plane2 as shown.


In the FeatureManager design tree, expand the listing of the Frame and select the sketch for the retaining pin hole.

Click **Convert Entities**  to copy the selected sketch into the active sketch as shown below.

**56 Extrude a cut.**

Use the **Up To Next** end condition in both directions.

57 Edit assembly.

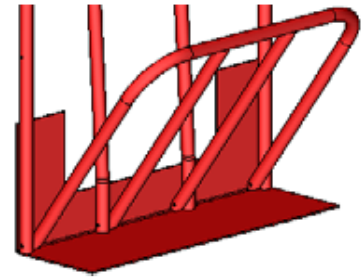
Click **Edit Component**  to switch back to Edit Assembly mode.

58 Suppress the mate.

Suppress the **Angle** mate that was created in step step 52.

59 Drag the component.

Drag the Fold Nose Extension and rebuild the assembly. The cut feature for the retaining pin hole in the Fold Nose Extension is flagged with an error. This is because the location of the hole was defined in the context of the assembly. While in-context features are very handy, they can cause some problems if the components they are in are supposed to move.



The solution is to remove the external references on the cut feature.

60 Unsuppress the Angle mate.

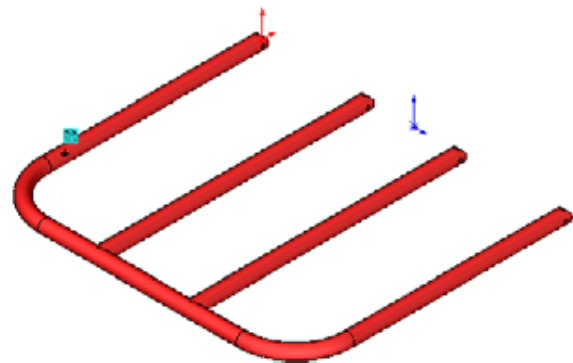
This returns the Fold Nose Extension to its upright position and the cut feature rebuilds correctly.

61 Open the Fold Nose Extension.

Right-click the Fold Nose Extension and select **Open Part** from the shortcut menu.

62 Edit sketch.

Edit the sketch of the cut feature (the retaining pin hole). Click **Display/Delete Relations**. There is only one relation: **On Edge**.

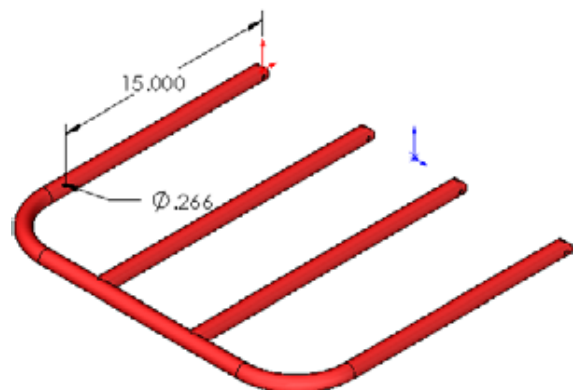


63 Remove the external references.

Delete the **On Edge** relation. The Fold Nose Extension now has no external references.

64 Fully define the sketch.

Dimension the sketch as shown. In addition, add a **Coincident** relation between the center of the circle and the Top reference plane.



65 Save and close the part.

66 Return to the assembly.

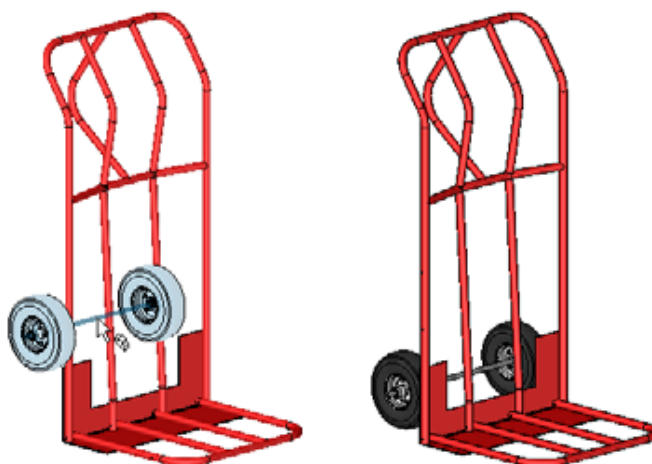
Change the value of the **Angle** mate to **90°**.


67 Insert a sub-assembly.

Click **Insert Components**  Click **Browse** and select the Wheel and Axle Assembly.

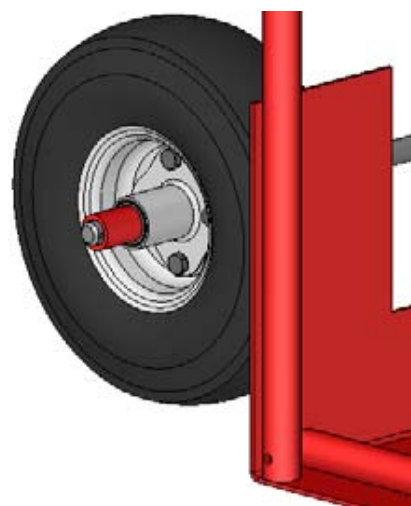
Mate the sub-assembly as follows:

- Q Front reference plane of the Wheel and Axle Assembly is **Coincident** with the Front reference plane of the top-level assembly.
- Q Top reference plane of the Wheel and Axle Assembly is a **Distance of 5.5"** from the Top reference plane of the top-level assembly.
- Q Right reference plane of the Wheel and Axle Assembly is a **Distance of 5.5"** from the Right reference plane of the top-level assembly.

**68 Insert a component.**

Click **Insert Components**  Add the Brace component to the assembly and mate it as follows:

- Q Cylindrical face of the Brace is **Concentric** with the cylindrical face of the Axle.
- Q Front reference plane of the Brace is **Coincident** with Plane2 of the Frame.
- Q Top reference plane of the Brace is **Parallel** with the Top reference plane of the assembly.

**69 Edit the part.**

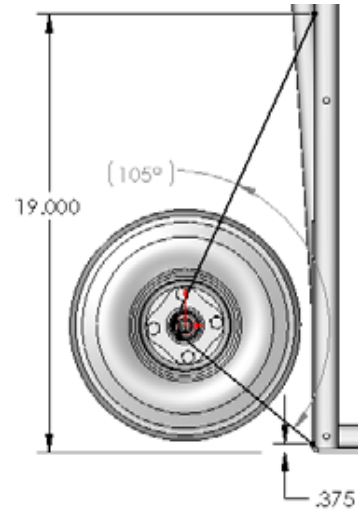
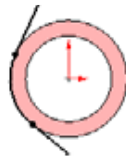
Select the Brace and click **Edit Component** 

70 Create a 2D sketch on the Front reference plane of the Brace.

Select the *outside* edge of the cylinder and click **Convert Entities** .

Sketch two lines **Tangent** to the converted edge and **Coincident** to the silhouette edge of the Frame.

Trim the converted edge as shown:



Sheet Metal or Structural Member?

A part such as the **Brace** can be modeled as a structural member (a weldment) or as a sheet metal part. In fact, the sketch we just created can be used either to extrude a base flange or as the path for a structural member. So based on sketching requirements, there is no difference between the two approaches. So, which approach *is* better?

Q **How is the part going to be fabricated?**

Generally weldments consist of multiple structural members that are cut and then welded together. The weldment tools in SolidWorks excel at creating the complex cuts such as we saw in the tubing of the frame of the hand truck.

This part would be a single structural member – a piece of bar stock or strap iron, cut to some length and bent at some radius and angle. No trimming is required.

Q **What are the drawing requirements?**

Is a flat pattern required? As a sheet metal part we can easily create the flat pattern.

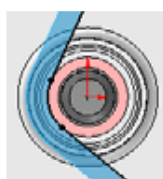
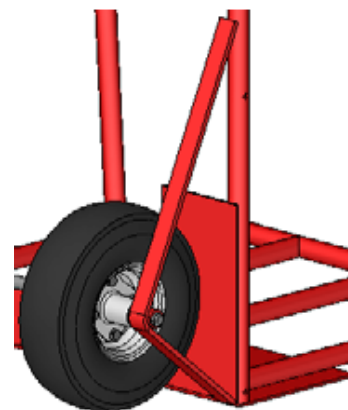
So in this case, using a weldment offers no advantages whereas a sheet metal part gives us the added benefit of the flat pattern.



71 Base flange.

Extrude a base flange a depth of **1"** using the **Mid Plane** end condition. For **Thickness** use **0.25"**. For the default **Bend Radius**, use whatever value appears by default. The bend radius is determined by the radius of the arc in the sketch.

Clear the **Merge result** checkbox. The material should be added to the outside of the sketch as shown below.



Reusing In-context Parts

You must use caution when reusing parts that are modeled in the context of an assembly. If the **Brace** was to be used in another hand truck, changes to the assembly where it was built would cause unexpected changes to the instance of the **Brace** in the other assembly.

Important!

If a component is going to be used in several different assemblies, you should not create external references.

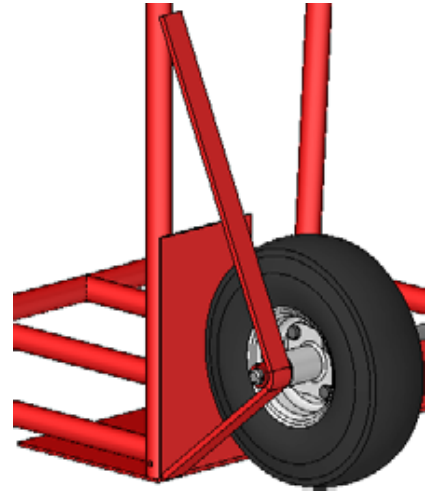
In this case, a second instance of the **Brace** is going to be added in the same orientation as the first – just located on the opposite side of the hand truck. So a change such as changing the distance mates on the **Wheel and Axle Assembly** will not cause any problems. Both instances will change together.

72 Insert a component.

Click **Insert Components** 

Add the **Brace** component to the assembly and mate it as follows:

- Q Cylindrical face of the **Brace** is **Concentric** with the cylindrical face of the **Axel**.
- Q Front reference plane of the **Brace** is **Coincident** with **Plane3** of the **Frame**.
- Q Top reference plane of the **Brace** is **Parallel** with the **Top** reference plane of the assembly.



73 Results.

74 Save and close all files.



Exercise 12: Creating a Weldment

Create this evaporator support weldment by following the steps given.



This lab reinforces the following skills:

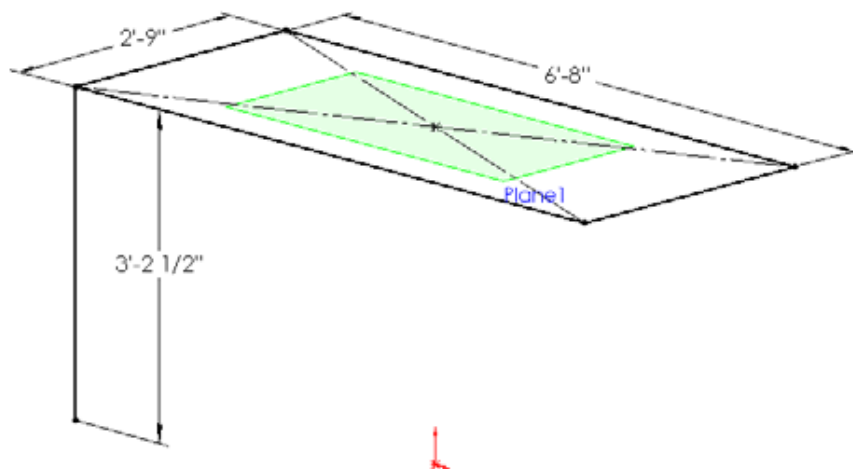
- Q Creating structural members.
- Q Modifying profile sketches for structural members.
- Q Creating sub-weldments.
- Q Creating cut list items.
- Q Creating drawings of weldments.

Units: **Feet & Inches**

Procedure

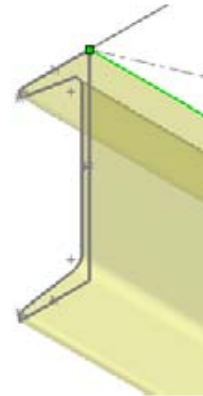
Open a new part using the Part_IN template. Change the units to **Feet & Inches**.

- 1 **Define sketch plane.**
Create a reference plane offset **3'-3"** above the Top reference plane.
- 2 **3D sketch for top of frame and one leg.**
Sketch a rectangle for the top of the frame and a line for the leg as shown in the illustration below.



3 Structural member - C Channel.

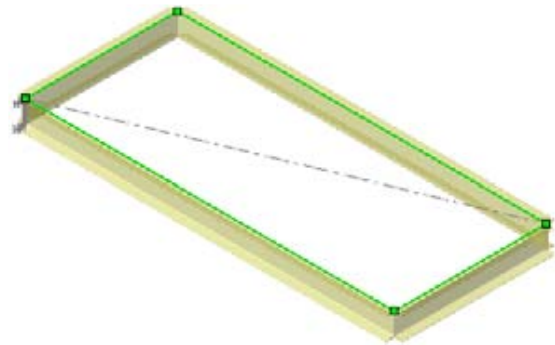
Use ANSI 5 x 6.7 C channel and insert a structural member using the rectangle in the 3D sketch. Rotate and locate the profile as shown.



Note

Depending on how you drew the lines the profile may be positioned at the other end of the line.

- 4 Select remaining sketch segments.**
 Clear the **Apply corner treatment** check box.
 Click **OK**.

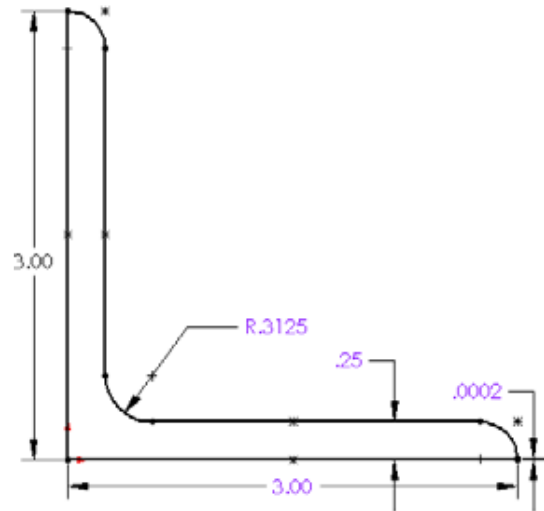


5 Profile sketch.

The legs are 3 x 3 x 0.25 angle iron. This is not one of the default profiles. You will have to create the profile.

Open the library part for the 3 x 2 x 0.25 profile. Modify the sketch as shown and save it as a library part named 3 x 3 x 0.25.

Make sure the file properties correctly reflect the new dimensions and new file name.

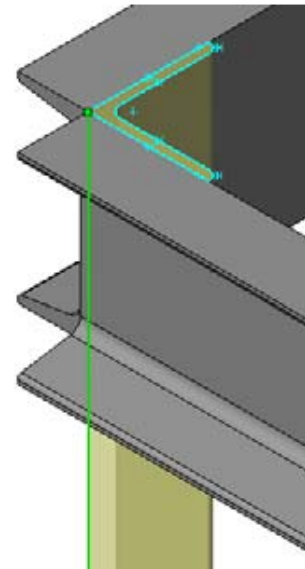


- 6 Insert the structural member for the leg.**
Use the 3 x 3 x 0.25 angle iron profile that was created in the previous step.

Locate the sketch as shown.

Note

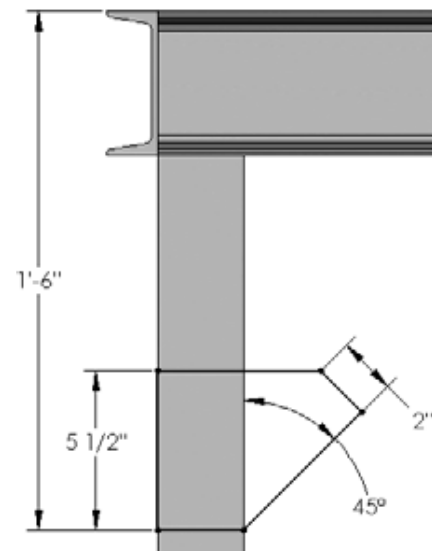
The sketch is shown in color for clarity.



- 7 Make the reinforcing plate.**

A gusset feature will not work here for two reasons:

- Q The shape of the profile doesn't match either of the two that are available with the gusset feature.
- Q To insert a gusset you must select two planar faces whose included angle is between 0° and 180° . The faces must touch each other and must belong to separate bodies. Those conditions do not exist in this situation.

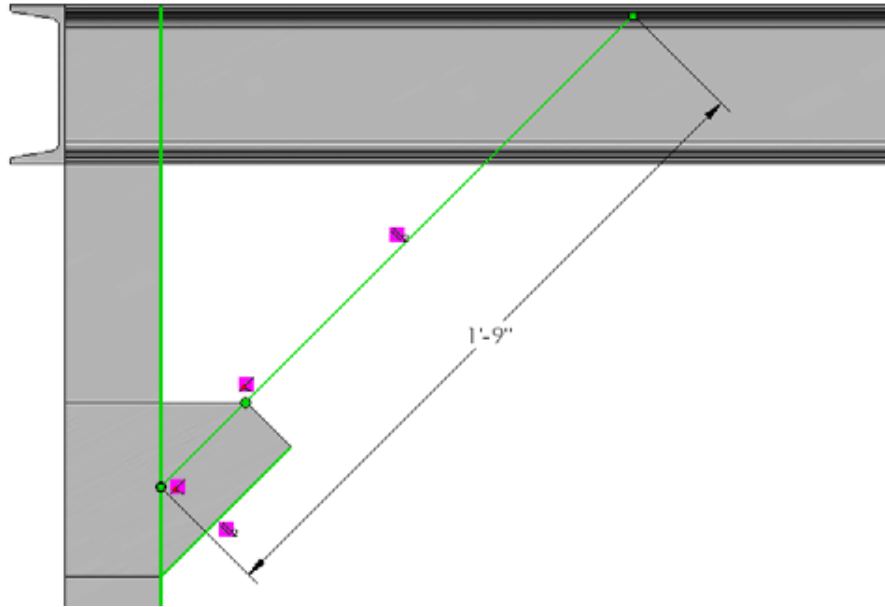


Create the sketch as shown.

Extrude it to a depth of **1/4"**.

8 Sketch for the angled brace.

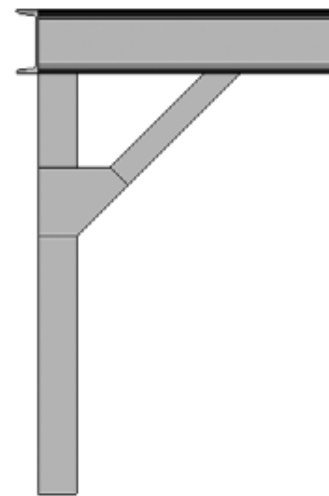
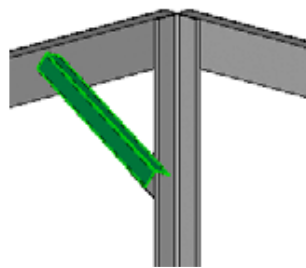
Open a sketch on the *rear* face of the reinforcing plate. Sketch a line as shown below. Note the various geometric relations necessary to fully define the sketch.



9 Structural member.

Use 2 x 2 x 0.125 angle iron.

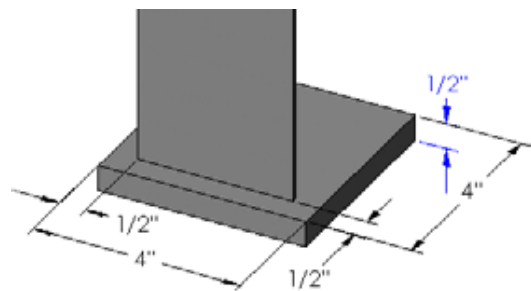
Note the orientation of the profile as shown in the reverse isometric view below.



10 Foot pad.

Create a foot pad using the dimensions shown at the right.

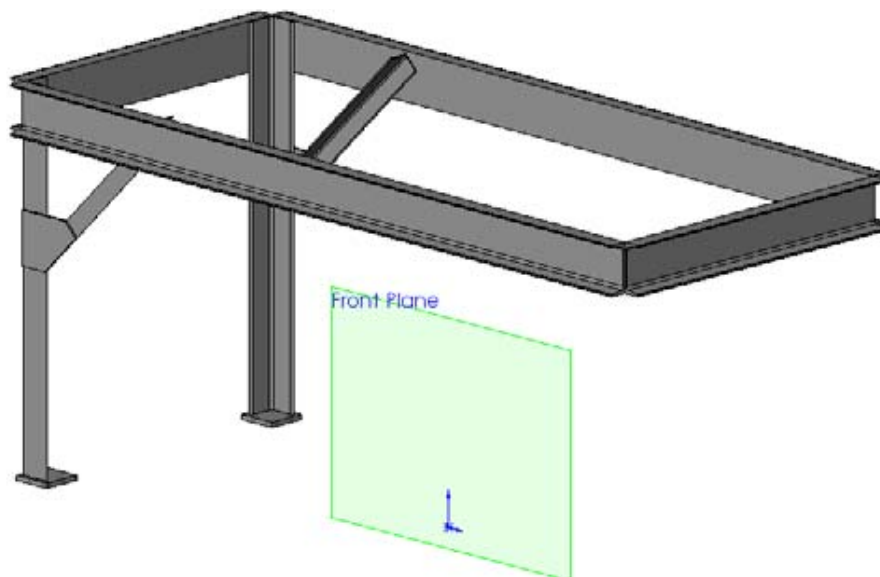
Add a weld bead between the foot pad and the leg.



Optional

11 Mirror.

Mirror the leg components with respect to the Front reference plane. The results are shown in the illustration below.

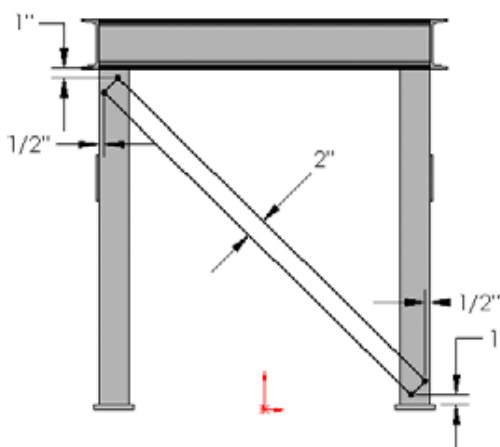


12 Cross brace.

Sketch a polygon and dimension it as shown. Extrude it to a depth of 1/4".

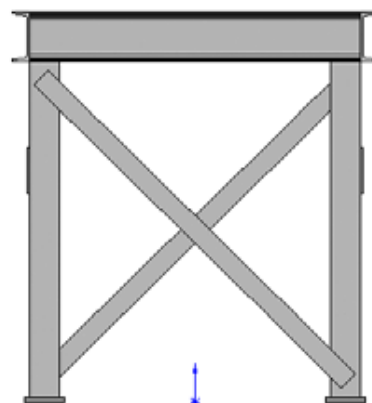
Note

The cross brace could have been created as a structural member. However, that would have required creating a profile sketch and adding it to the weldment profiles folder. Also, the sketch for the path would have been a single line, not a rectangle.



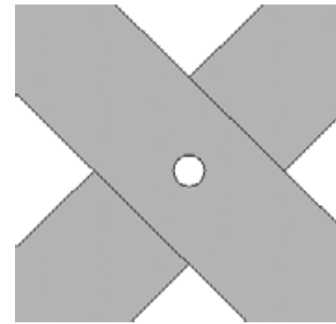
13 Repeat.

Make a second cross brace on the inside face of the legs.



14 Hole Wizard.

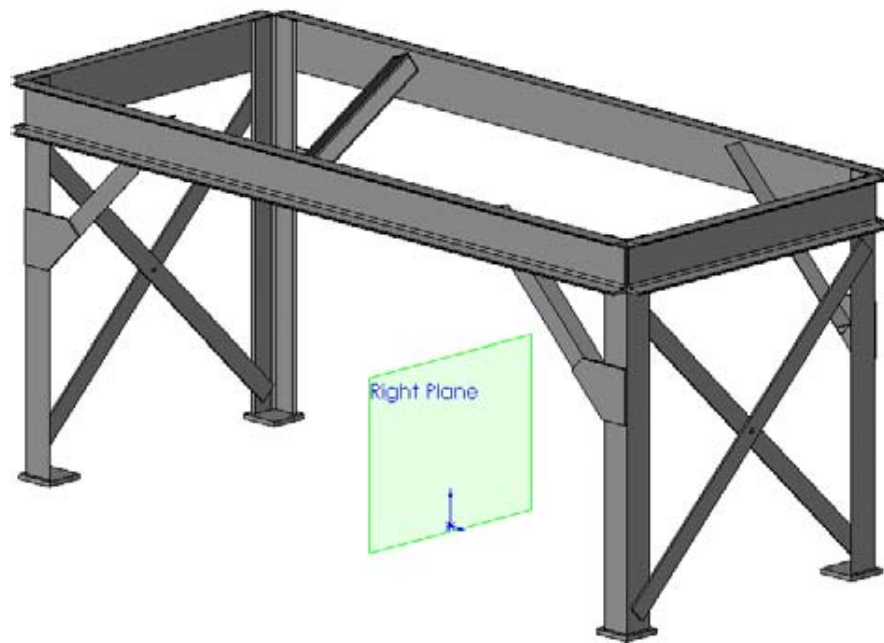
Use the Hole Wizard to make a clearance hole for a $7/16$ " bolt through the two cross braces.



15 Mirror.

Mirror both sets of leg components with respect to the Right reference plane.

The results are shown in the illustration below.

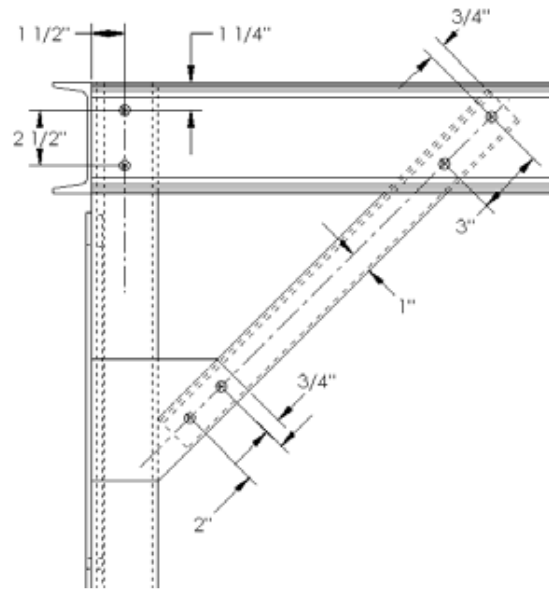


16 Bolt holes.

Use the Hole Wizard to add clearance holes for **7/16"** bolts as shown in the illustration below.

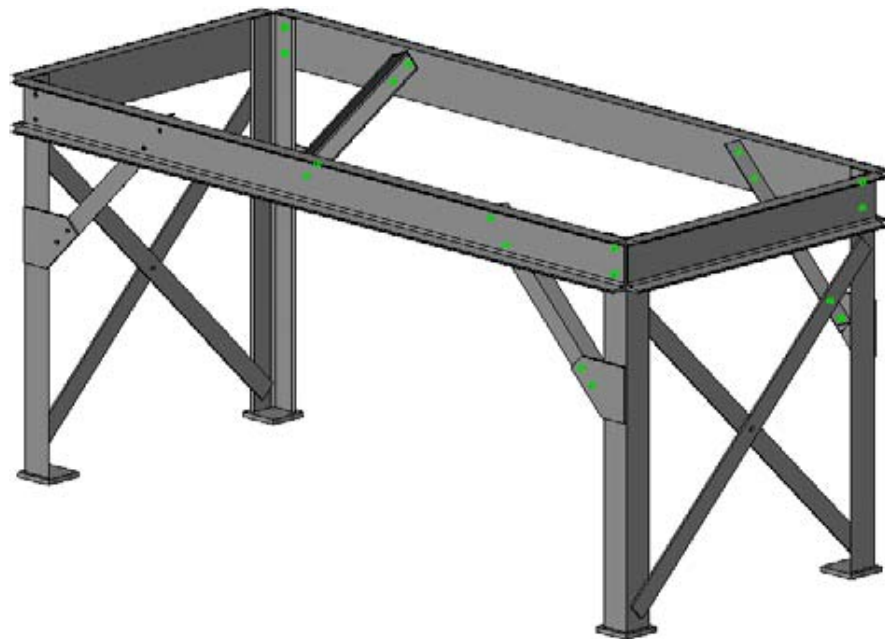
Note

The Hole Wizard does not have a feature scope to control which bodies are affected by the holes. Therefore, do not use the end condition **Through All**. Use a **Blind** hole with a **Depth** of **1"**.



17 Mirror the bolt holes.

This requires two mirroring operations. Once with respect to the Front plane and once with respect to the Right plane.



Note

The leg components and the bolt holes have to be mirrored separately because the **Mirror** command does not support mirroring solid bodies and features in the same operation.

18 Sub-weldment.

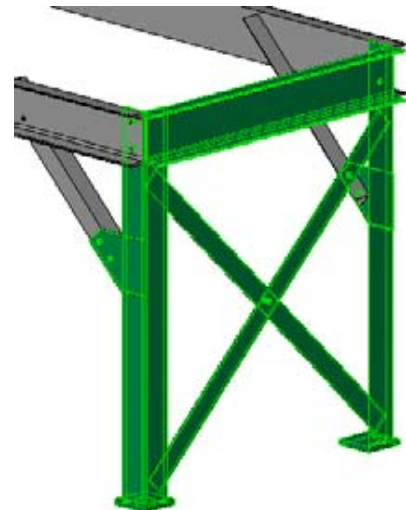
Create a sub-weldment of the bodies that are shown highlighted in the illustration at the right. The sub-weldment consists of:

- Q The 2'-9" long C channel that forms the end of the upper frame.
- Q Two legs.
- Q Two reinforcing plates.
- Q Two cross braces.
- Q Two foot pads.
- Q Any associated weld beads.



19 Repeat.

Create a second sub-weldment identical to the first using the components at the other end of the evaporator support.



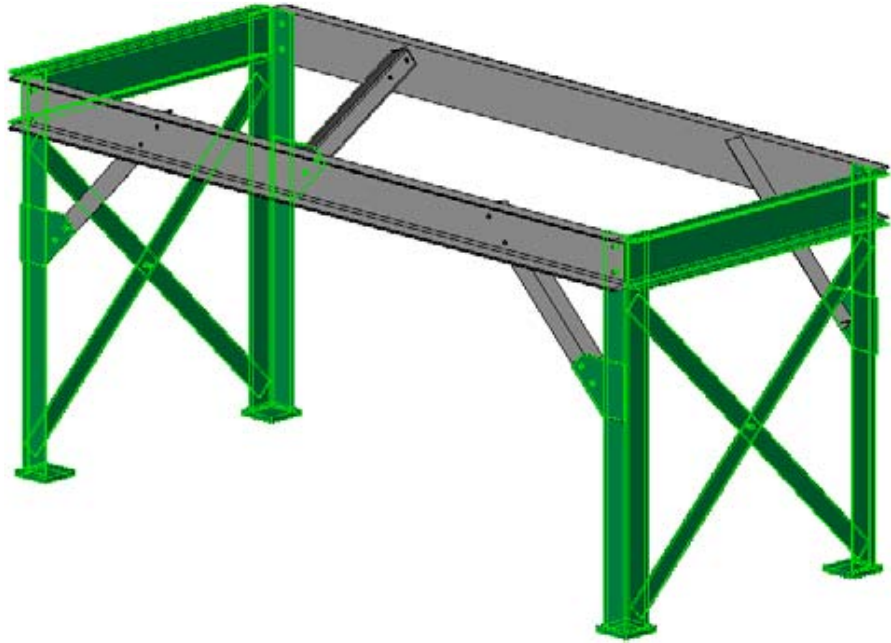
20 Generate the cut list items.

There should be three cut list items as shown in the following illustrations.

In order for the sub-weldments to be listed in the cut list table on the drawing, they have to be in a cut list item.

- ☐ Leg Sub-weldments
- ☐ Sub-weldment1(9)
- ☐ Sub-weldment2(9)

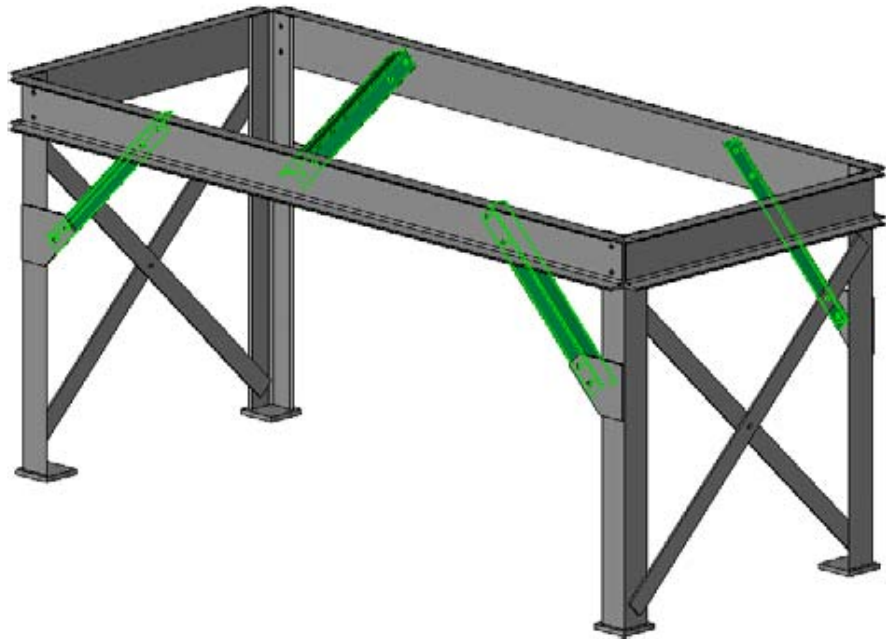
Q Name the cut list item shown below Leg Sub-weldments.



Q Name the cut list item shown below Front/Rear Rails.



Q Name the cut list item shown below Angled Braces.



21 Add the material.

Assign the material **Plain Carbon Steel** to the weldment.

22 Properties.


Verify that the properties associated with the cut list items are correct. **DESCRIPTION** should be imported from the profile sketch of the structural members. **LENGTH** is computed by the system when the cut list items are created.

Add the property **WEIGHT** to the cut list items.

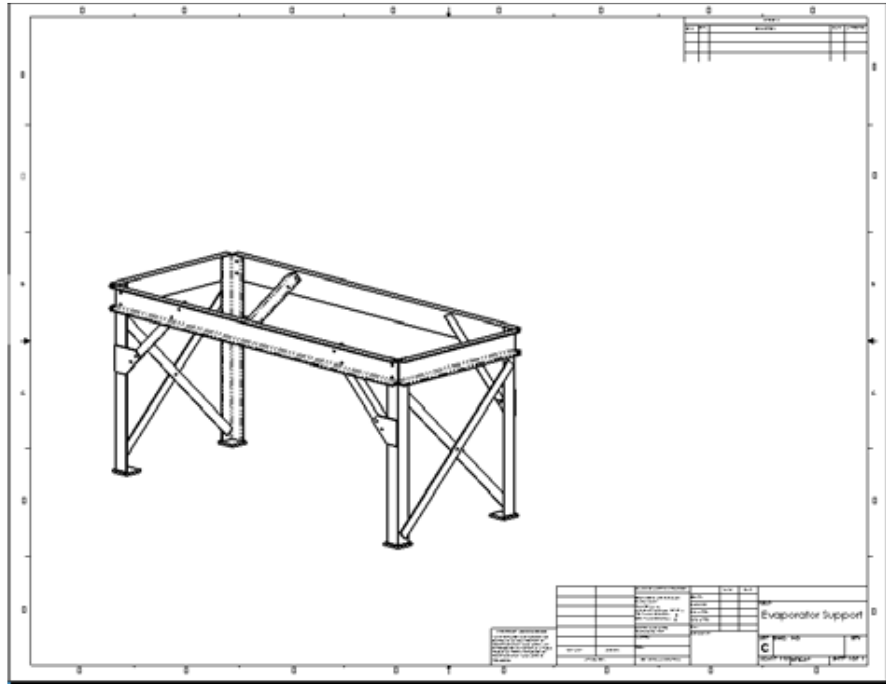
23 Properties.

The cut list item containing the sub-weldments does not inherit any properties. Manually add the property **DESCRIPTION** and give it the value "LEG ASSEMBLY".

24 Create a drawing.

Click **Make Drawing from Part**  on the Standard toolbar. Click the **Training Templates** tab. Select the **C_Size_ANSI_Inch** template and click **OK**.

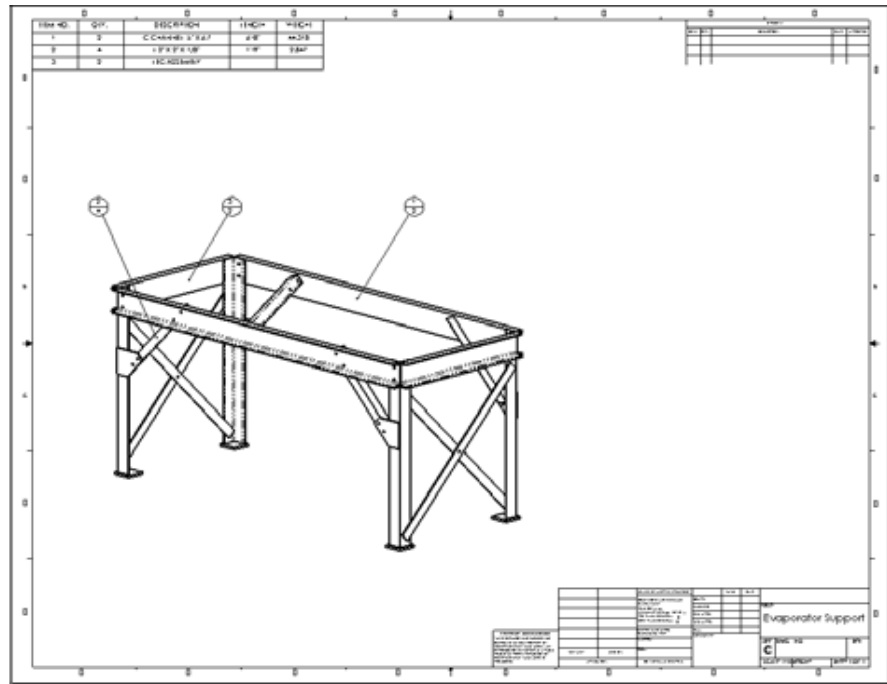
Under **Orientation** select **Dimetric**. Under **Scale**, click **Use custom scale** and set the scale to **1:8**.



25 Add the cut list table.

ITEM NO.	QTY.	DESCRIPTION	LENGTH	WEIGHT
1	2	C CHANNEL, 5" X 6.7	6'-8"	44.318
2	4	L 2" X 2" X 1/8"	1'-9"	2.847
3	2	LEG ASSEMBLY		

26 Add balloons.



27 Save the sub-weldment as a separate file.

Switch back to the weldment part.

Expand the cut list item that contains the two sub-weldments.

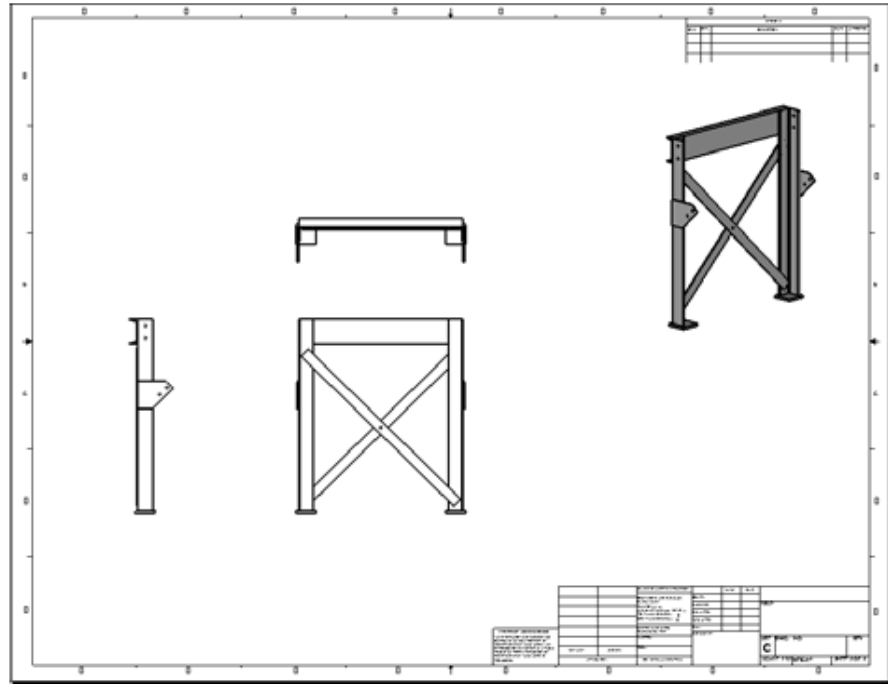
Right-click one of the sub-weldments and select **Save to File**.

Name the file Evaporator Support - Leg Sub-weldment.

28 Make a drawing of the sub-weldment.

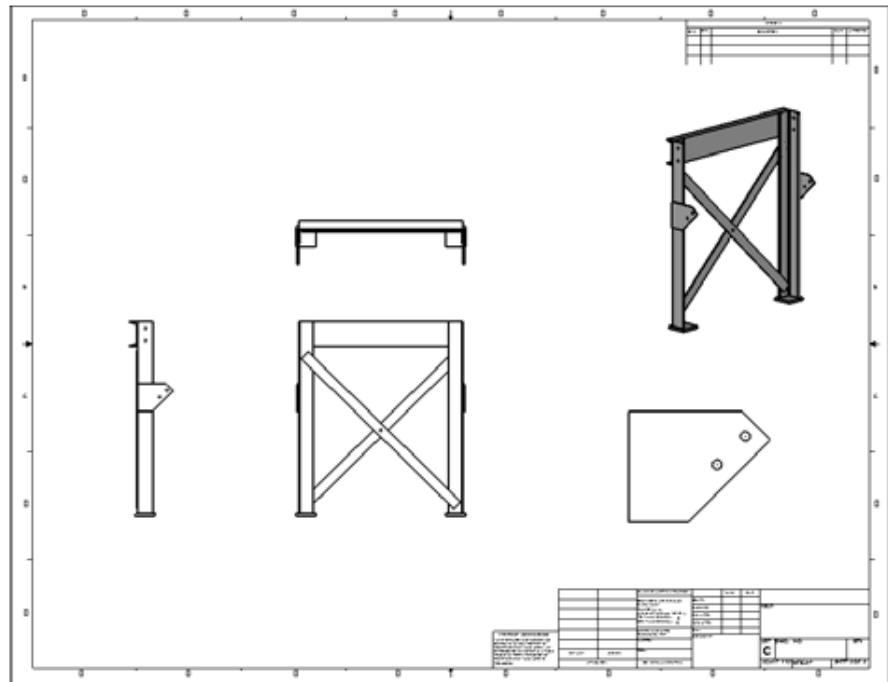
Add a C-size sheet to the existing drawing.

Add three orthographic views and a dimetric view of the sub-weldment. Use a **Scale** of **1:8**.



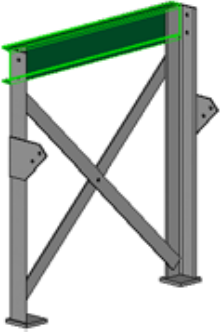
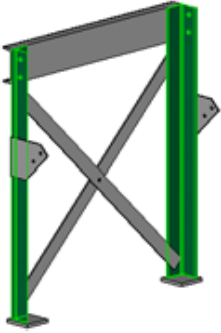
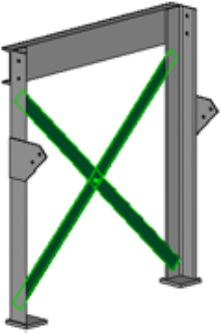
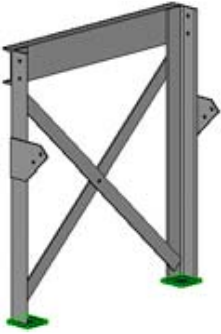
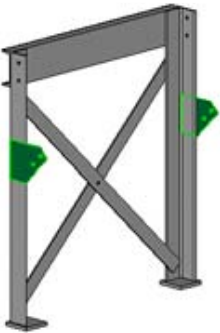
29 Add a relative view of the reinforcing plate.

Use a **Scale** of **1:2**.



30 Create cut list items for sub-weldment.

Switch to the sub-weldment part document and generate the cut list. Use the following table as a guide for renaming the cut list items.

Name of Cut List Item	Selected Bodies	Name of Cut List Item	Selected Bodies
SIDE RAIL, UPPER FRAME		LEGS	
CROSS BRACES		FOOT PADS	
REINFORCING PLATES			

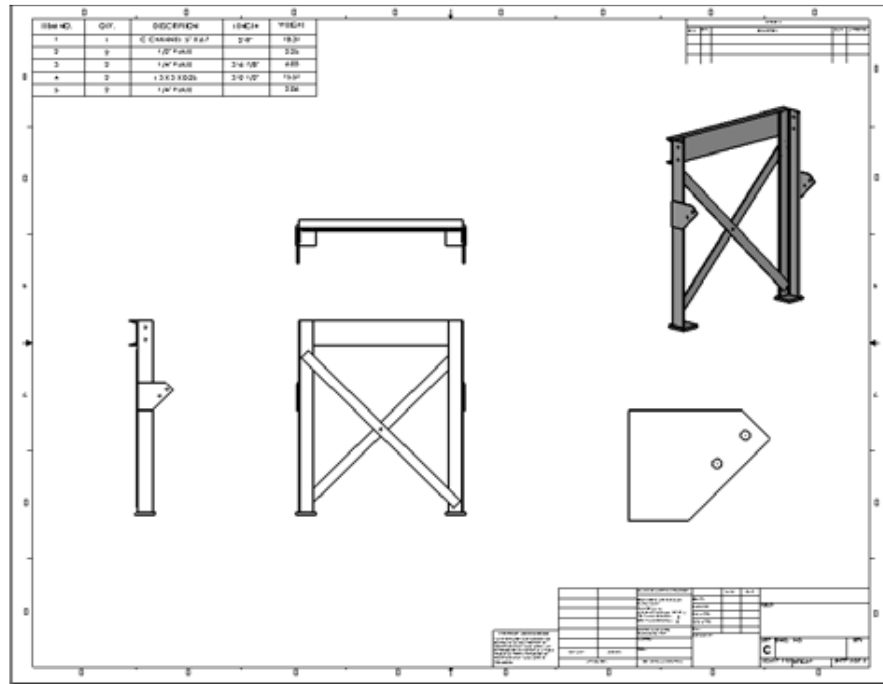
Properties on Sub-weldments

Currently the custom properties such as **Length** and **Description** that are normally associated cut list items do not propagate to the part that is derived from the sub-weldment. Therefore, you have to add the properties manually in order to properly generate the cut list table on the drawing.

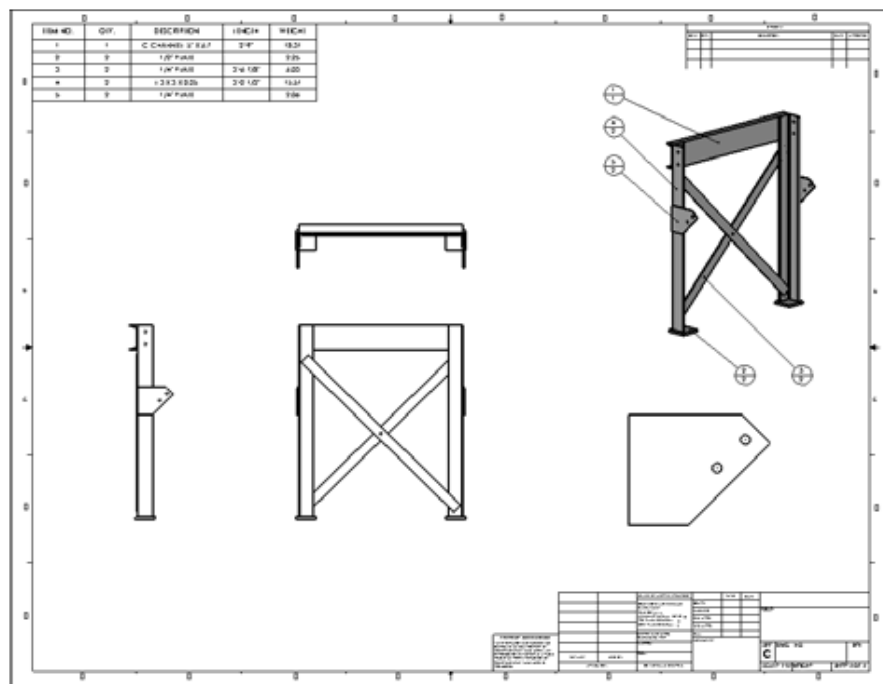
31 Add properties.

Add the properties **DESCRIPTION**, **LENGTH**, and **WEIGHT**.

32 Add a cut list table to the sub-weldment drawing.



33 Add balloons.



**Exercise 13:
Bent Tubing,
Sheet Metal,
and Assemblies**

Create this set of moveable warehouse steps by following the procedure given.



This lab reinforces the following skills:

- Q Creating structural members for bent pipe and tubing
- Q Solving special trimming situations
- Q Creating sheet metal parts
- Q Combining sheet metal and weldments
- Q Integrating weldments into assemblies

Units: **Inches**

Procedure

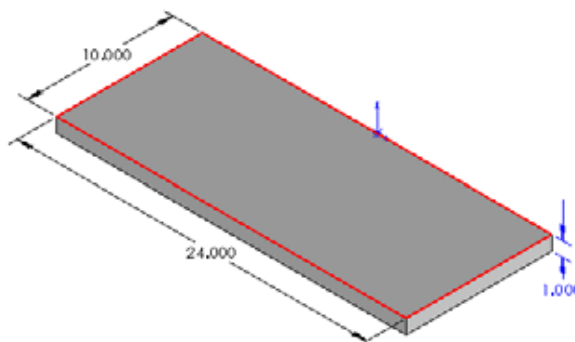
The first part we will create is the stair tread. It will have two configurations: one 10 inches deep and one 7 inches deep.

1 New part.

Create a new part with units **inches**. Name the part Tread.

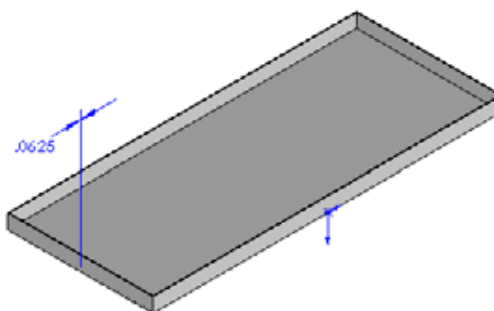
2 Extrude.

Sketch a rectangle 10 inches by 24 inches. Extrude it a depth of 1 inch.



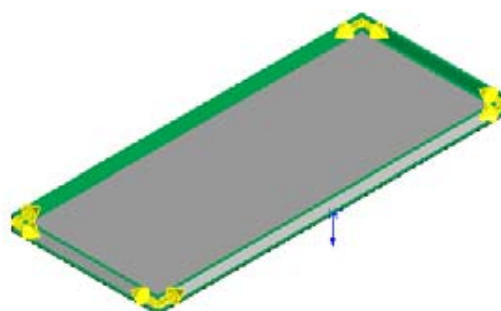
3 Shell.

Shell the part using a thickness of **0.0625**". Remove the face from the underside of the part.




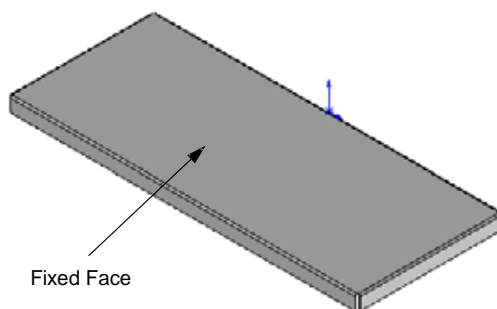
4 Rip.

Rip the part, creating gaps at the four corners as shown in the illustration. Use a gap of **0.125**".



5 Insert bends.

Select the face shown and click **Insert Bends** . For **Bend Radius** use **0.0625**".



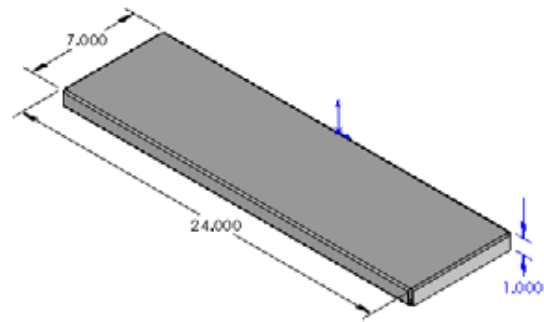
6 Edge flange.

Create an edge flange **0.375**" long at an angle of **90°** as shown. Be sure to position the flange so the material is inside so as not to exceed the 1" height of the tread.

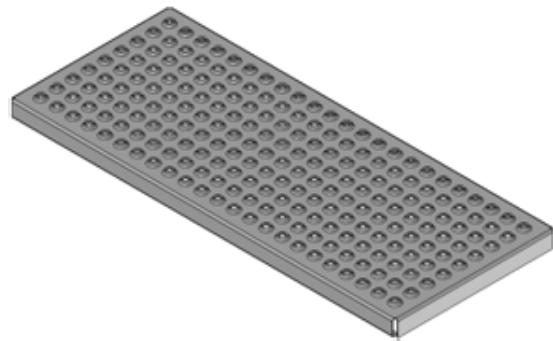


7 Configurations.

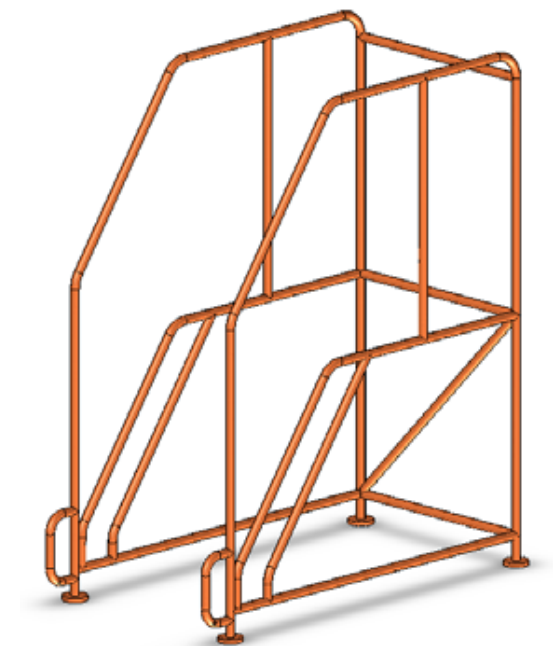
Make a new configuration and modify the dimensions so that the tread measures 7" by 24".

8 Save the part.**Optional**

Metal stair treads typically have some sort of formed feature to provide a non-slip surface. You can drag and drop a counter sink emboss feature from the Design Library and pattern it. However, for performance purposes, it is wise to make two additional, derived configurations that have these features suppressed.

**The Frame**

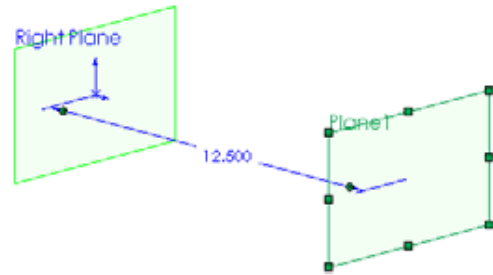
The next part we will build is the frame for the stairs. We will build one side frame, mirror it, and then add the structural members that connect the two sides.

**1 New part.**

Create a new part with units **inches**. Name the part **Frame**.

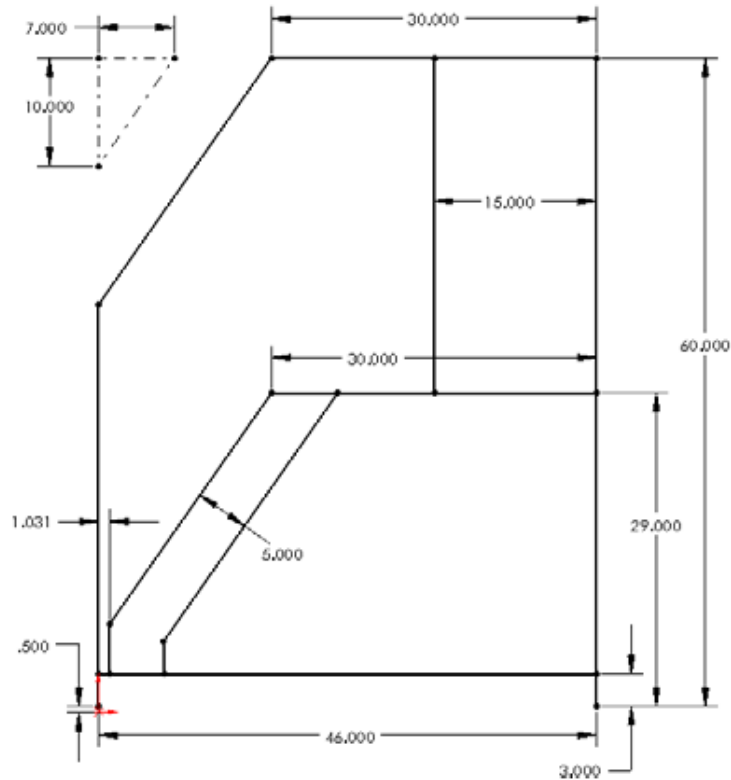
2 Reference plane.

Create a reference plane as shown in the illustration at the right.



3 Sketch.

Create a sketch on Plane1 as shown below.

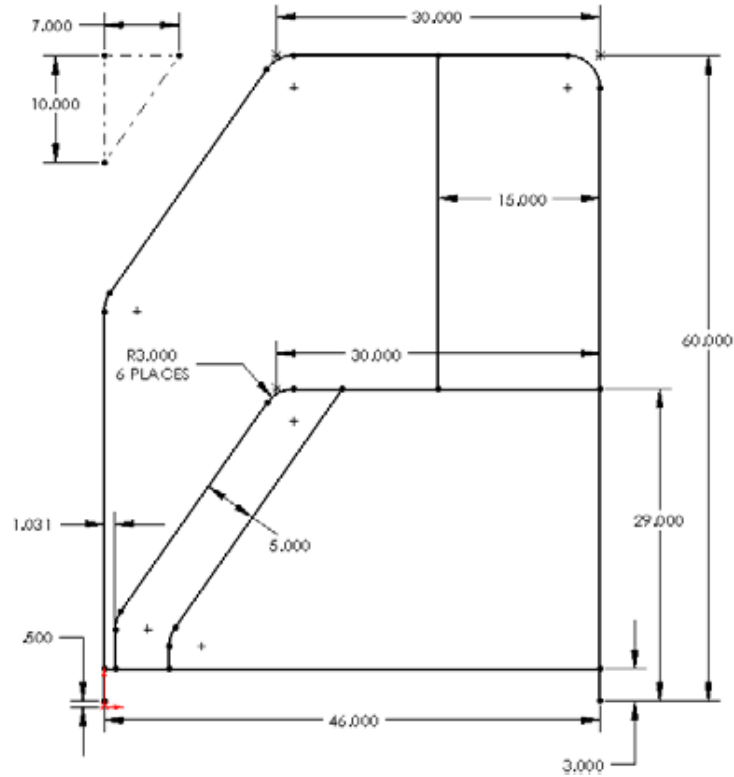


Tip

Stairs are usually specified by their rise and run rather than by some odd angle dimension. An easy way to do this is to sketch a triangle using construction geometry and dimension the rise and run. Then make the stringer of the stairs parallel to the hypotenuse of the triangle.

4 Sketch fillets.

Add six sketch fillets with a radius of 3" as shown below.



5 Finish sketching.

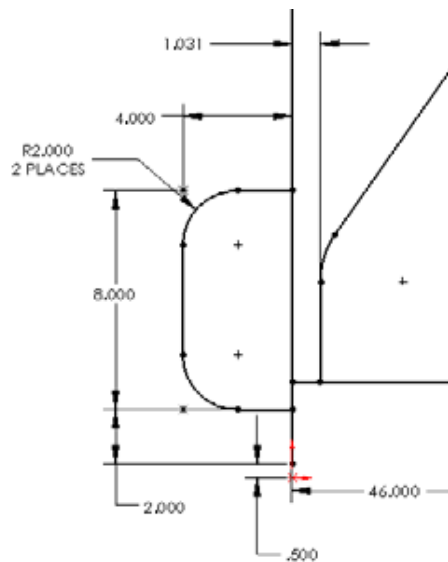
Complete the sketch by adding the geometry as shown at the right.

Note

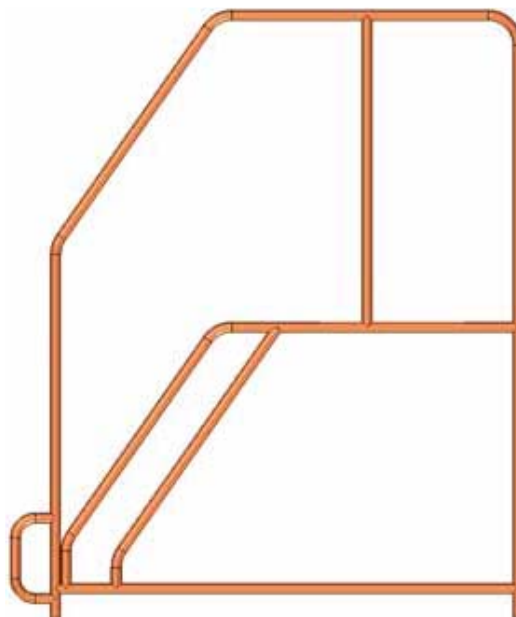
The sketch fillets are 2", not 3" as they were in step 4.

6 Exit the sketch.

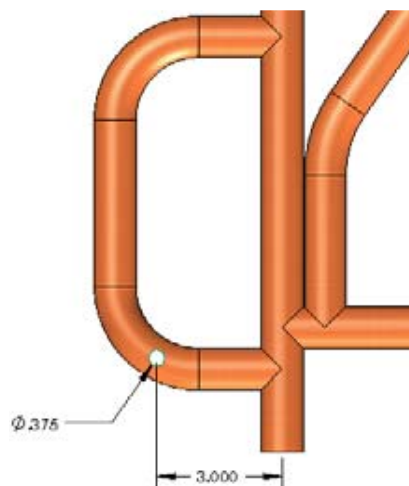
Name the sketch Frame Sketch.



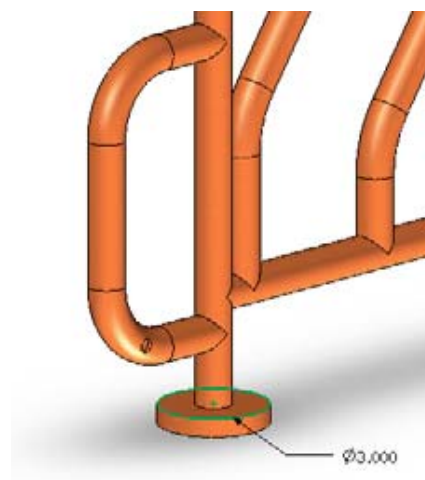
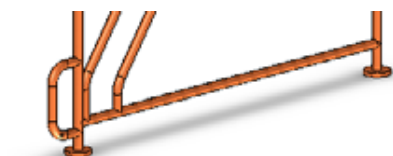
- 7 Structural members.**
 Insert structural members as shown below. This will require several separate operations due to the rules governing path segments. Use ANSI 1.000" o. d. x 0.125" wall seamless tube. Trim the structural members as necessary.



- 8 Extrude a cut.**
 Select Plane1 and open a sketch. Sketch a **0.375"** circle as shown. Extrude a cut using the end condition **Through All** in both directions.

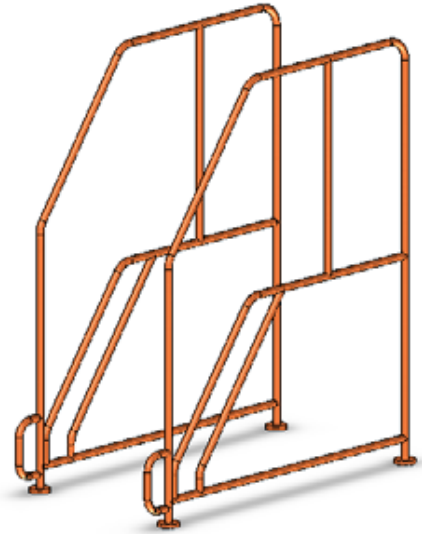


- 9 Add the foot pads.**
 Select the bottom face of the vertical structural member and open a sketch. Sketch a **3"** diameter circle concentric with the bottom end of each vertical segment. Extrude the sketch **0.5"**.



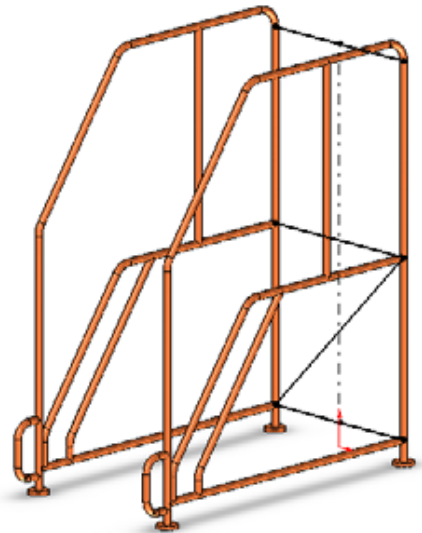
10 Mirror the side frame.

Select the **Right** reference plane and mirror *all* the bodies to create the second side frame.

**11 Sketch the path segments for the crosspieces.**

You can sketch this as a 3D or 2D sketch. One technique using a 2D sketch is to:

1. Create a reference plane parallel to the **Front** plane and coincident with one of the endpoints of the **Frame Sketch**. This way if the dimensions of the **Frame Sketch** change, the position of the reference plane will update.
2. Select this new plane and open a 2D sketch.
3. Sketch a centerline and make it coincident with the **Right** reference plane.
4. Sketch three lines for the horizontal path segments. Make their starting endpoints coincident with the ends of the path segments in the **Frame Sketch**.
5. Make the midpoints of the three lines coincident with the centerline.
6. Finally, sketch the diagonal path segment.




12 Create the structural members and trim them.

Exit the sketch and create the structural members using ANSI 1.000" o. d. x 0.125" wall seamless tube.

Trim the structural members as necessary.

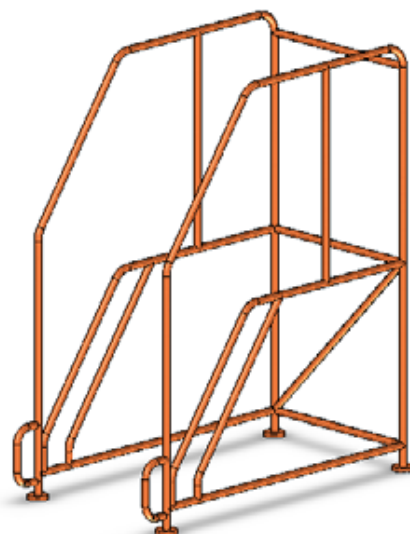
13 Save the part.

14 Create an assembly.

Click **Make Assembly from Part/Assembly**  and select the Assembly_IN template.

In the **Part/Assembly to Insert** list, make sure the Frame is selected.

Click **OK** to insert the component at the origin.



15 Save the assembly.

Name the file *Moveable Steps*.

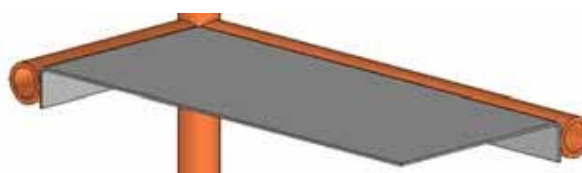
16 Insert a component.

Click **Insert Components** 

Add the Tread component to the assembly. Make sure the 10 in. x 24 in. configuration is active.

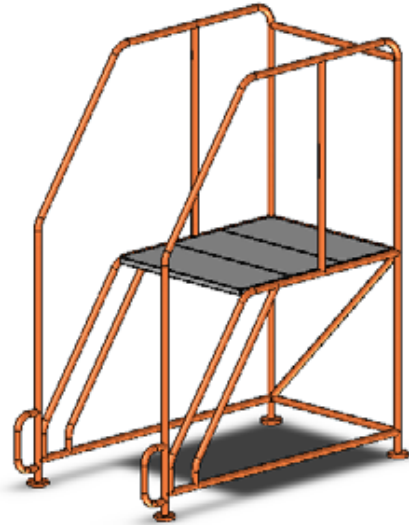
Using the section view below as a guide, mate the Tread as follows:

- Q The uppermost face of the Tread is **Tangent** to the cylindrical face of the middle crosspiece.
- Q The rear face of the Tread is **Tangent** to the cylindrical face of the crosspiece.
- Q The uppermost face of the Tread is **Parallel** to the Top reference plane of the assembly.
- Q The Right reference plane of the Tread is **Coincident** with the Right reference plane of the assembly.



17 Insert two more copies of the Tread.

These instances also use the 10 in. x 24 in. configuration. Mate the treads so they are fully defined and positioned as shown in the illustration at the right.

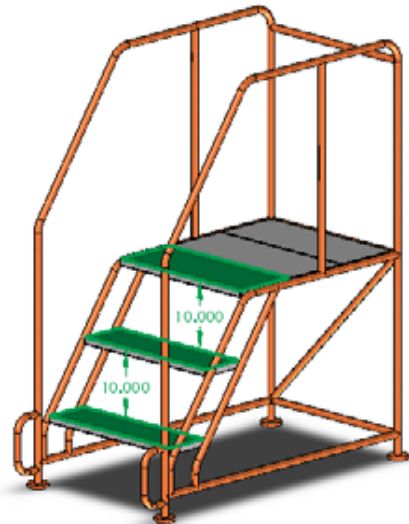


18 Add the components for the steps.

These instances use the 7 in. x 24 in. configuration.

Mate the components as follows:

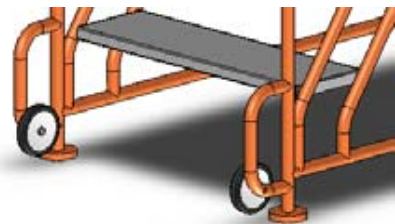
- Q Use a **Distance** mate of **10"** to control the spacing between the uppermost faces of the Tread components.
- Q The rear face of the Tread is **Coincident** to the front face of the Tread above it.
- Q The Right reference plane of the Tread is **Coincident** with the Right reference plane of the assembly.



19 Add the wheels.

Add two instances of the Wheel - 4 in component.

Mate them **Concentric** to the holes and **Tangent** to the Frame as shown in the illustration.



20 Results.



Optional

- Q Generate the cut list for the Frame.
- Q Make a drawing of the Frame and insert a cut list table.
- Q Make an assembly drawing of the Moveable Steps complete with a bill of materials.

21 Save and close all the files.

Index

Numerics

- 3D sketching 155
- space handle 155

A

- adding bend lines 44, 82
- alongZ 157
- assemblies
 - adding components 96, 168
 - adding sub-assemblies 171
 - collision detection 168
 - combining sheet metal and weldments 166
- edit assembly 108
- edit component 108
- edit part 108
- make assembly from part/assembly 167
- propagating changes 108
- removing external references 170
- reusing in-context parts 173
- top-down design 95–110
- update holders 98
- assembly-centric design 95–110
 - removing external references 170
 - reusing in-context parts 173
- auto relief 14

B

- base flange 10–12, 98
- bend allowance
 - bend tables 13
 - K-factor 13
- bend angles 38, 45
- bend deviation 55–56
- bend lines 44, 82
- bend parameters 13, 78
- bend relief 14, 17
 - manual methods 48
 - obround 14
 - offset ratio 14
 - rectangular 14
 - tear 14
- bends
 - fold 26
 - insert 76
 - lofted 54
 - sketched 44, 82
 - suppress and unsuppress 86–87
 - unfold 26

- boss flange 11, 21
- break corner 50

C

- closed corners 40, 42
- collision detection in assemblies 168
- color, face color in forming tools 35
- components
 - adding existing 168
 - adding new 96
- cones, unrolling 83–85
 - See also* lofted bends
- configurations 86–87, 192
- converting legacy sheet metal parts 71
 - recommended practices 72, 79
- corner treatment
 - in flat patterns 23
 - in weldments 127
- corners
 - closed 40, 42
 - gaps 17
 - relief 17
 - rip 75–76
- curves
 - split lines 35
- custom properties 140, 148
- cut list
 - folder 144–145
 - tables 151–154
 - weldments 145
- cuts across bends 26–28, 81
- cuts in sheet metal parts 25
- cylinders, unrolling 83–85

D

- data exchange 73
- derived sketches 163
- Design Library 29, 105
- document templates 74, 96
- drag and drop 30
- drawings
 - sheet metal parts 38–40
 - views of individual bodies 149
 - weldment cut list tables 145, 151–154
 - weldments 148–154

E

- edge flange 11, 17–20, 79, 100–101
- edit flange profile 19, 79

- offset 100
- trim side bends 19

edit

- assembly 108
- bend angles 38
- component 108
- material 147
- part 108
- sub-assembly 108
- end caps, weldments 136
- Excel, bend tables 13
- extend structural member 132, 164–165
- external references 170
- extrude
 - base flange 12
 - cuts 25
 - tabs on sheet metal parts 11, 21

F

- feature scope 145, 181
- FeatureManager design tree
 - cut list folder 144–145
 - materialweldments
 - material 147
 - sheet metal features 13, 78
 - sub-weldment folder 143–144
- features
 - break corner 50
 - end cap 136
 - flanges 10
 - flat pattern 14
 - flatten-bends 78
 - formed, sheet metal 29–37
 - gusset 136
 - hems 107
 - jog 50
 - process-bends 79
 - rip 75–76
 - sheet metal 13, 78
 - split lines 35
 - structural member 126
- file extensions, XLS 13
- fillet welds 134, 145
- flange profile 19, 79
- flanges 10
 - base flange 10–12, 98
 - boss flange 11, 21
 - edge flange 11, 17–20, 79, 100–101
 - miter flange 10, 14–15, 40, 42, 102–

- 103
- position of 16
- flat patterns 22–24
 - corner treatments 23
 - feature 14
 - in drawings 38–40
 - merge faces 23
- fold 26
- folders
 - cut list 144–145
 - cut list item 145
 - sub-weldment 143–144
- formed features 29–37
- G**
- geometric relations
 - alongz 157
- gussets, weldments 136
- H**
- hems 107
- I**
- IGES 73
 - report files 74
- importing a file 73
- in-context design 95–110
 - removing external references 170
- insert
 - bends 76
 - components into an assembly 168
 - solid body into new part 144
 - split line 35
 - structural member 126
 - weldment 125
- insert component, new 96
- J**
- jog 50
 - fix projected length 52
- joggle, *See* jog
- L**
- locate profile (weldments) 128
- lofted bends 54
- M**
- make assembly from part/
 - assembly 167
- material, in weldments 147
- merge
 - arc segment bodies 160
 - faces 23
 - result 125
- methodology 9
- mirroring
 - sheet metal parts and features 37, 47, 49
 - weldments 135, 139, 179–181, 196
- miter flange 10, 14–15, 102–103
 - closed corners 40, 42
 - propagate along edges 16
- multibody parts
 - feature scope 145, 181
- merge arc segment bodies 160
- merge result 125
- weldments 125–174
- N**
- neutral files 73
- O**
- offset ratio 14
- offset, *See* jog
- options 144
- P**
- parts
 - inserting a solid body into a new part 144
 - saving a solid body 144
- patterns
 - mirror 135, 139, 179–181, 196
 - sketch driven 85
- pipes 155
 - merge arc segment bodies 160
- plane, normal to curve 15
- position profile (weldments), *See* locate profile
- process plans 86–87
- properties 140, 148
- R**
- relations
 - alongz 157
- relief, bend 14
- reusing in-context parts 173
- rip feature 75–76
- rollback in sheet metal parts 79
- S**
- saving solid body as a part 144
- sheet metal
 - adding bends 76
 - bend allowance 13
 - bend angles 38
 - bend parameters 13
 - bend relief 14
 - bend table 13
 - converting legacy parts 71
 - designing in the flat 43–46
 - feature 13, 78
 - flat pattern 86–87
 - flatten-bends feature 78
 - formed features 29–37
 - hems 107
 - in-context design 95–110
 - jog 50
 - K-factor 13
 - process plans 86–87
 - process-bends feature 79
 - rip feature 75–76
 - rollback 79
- sheet metal feature 13
- Shift key 15
- sketch driven patterns 85
- sketch relations
 - alongz 157
- sketched bends 44, 82
- sketches
 - 3D 155
 - derived 163
 - normal to curve 15
 - weldment profiles 140
 - weldments 129
- space handle 155
- split line 35
- stock feature 144
- structural members 126, 132, 164–165
 - corner treatments 127
 - locate profile 128
 - profile sketch 140
 - saving as parts 144
 - sketch considerations 129
 - trimming 132, 164–165
- sub-assemblies 171
- sub-weldments 143
 - folder 143–144
 - saving as multibody parts 144
- suppress bends 86–87
- symmetry 135, 139, 179–181, 196
- T**
- Tab key 30
- tabs 11, 21
- tear relief 14
- templates, document 74, 96
- toolbar 9
- tools, options 144
- top-down design 95–110
 - removing external references 170
 - reusing in-context parts 173
- trimming
 - structural members in weldments 132, 164–165
- tubing 155
 - merge arc segment bodies 160
- U**
- unfold 26
- unrolling cones and cylinders 83–85
- unsuppress bends 86–87
- update holders 98
- W**
- weld beads 134, 145
- weldments 125–174
 - bent tubes and pipes 155
 - corner treatments 127
 - custom properties 140, 148
 - cut list tables on drawings 151–154
 - cut lists 145
 - drawings of 148–154
 - end caps 136
 - feature scope 145, 181
 - gussets 136
 - locate profile 128
 - non-structural components 144
 - pipes 155
 - post-assembly machining operations 145, 181
 - profile sketch 140

- properties 140, 148
- sketch considerations 129
- structural members 126, 132, 164–165
- sub-weldments 143
- trimming structural members 132, 164–165
- tubing 155
- weld beads 134, 145
- weldment feature 125
- working with legacy data 71
 - IGES 73
 - recommended practices 72, 79

X

- XLS file extension 13

