SolidWorks

SHEET METAL DESIGN

July 28, 2009

Summary

This presentation will outline

- Benefits of using SolidWorks Sheet Metal Tools
- Sheet Metal Toolbars
  - Design Tools
- Design Library
  - Modifying Features
- Sheet Metal Drawing
  - Bend Tables
  - K Factors
- Tips for Sheet Metal Design

Introduction

Advantages of SolidWorks Sheet Metal Tools

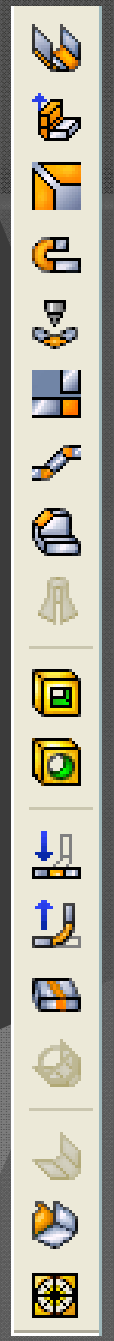
- Design Intent for Sheet Metal Fabrication
- Specified tools for sheet metal operations and common features
- Bend Factor – K Factor Calculations
- Visual aids – Flatten Features
- Link features to sheet thickness
- Automatically closed corners

Sheet Metal Toolbar

Locating the Toolbar:

View → Toolbars → Sheet Metal

- Base Flange or Tab
- Edge Flange
- Miter Flange
- Hem
- Sketched Bend
- Closed Corner
- Jog
- Break-Corner/Corner-Trim
- Lofted Bend
- Unfold
- Fold
- Flatten
- No Bends
- Insert Bends
- Rip
- Vent



Bend Position

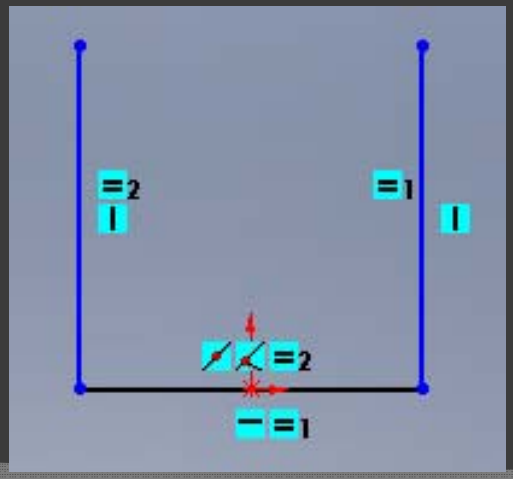
- Must select bend position for Miter Flange, Edge Flange, Sketched Bend, Hem, or Jog
- 5 options:
  -  Material Inside
  -  Material Outside
  -  Bend Outside
  -  Bend from Virtual Sharp
  -  Bend Centerline

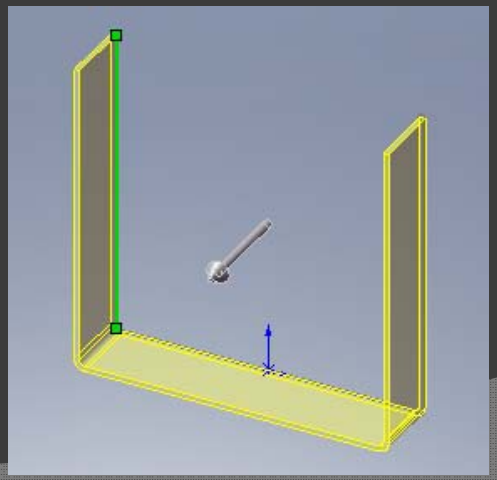
Base Flange

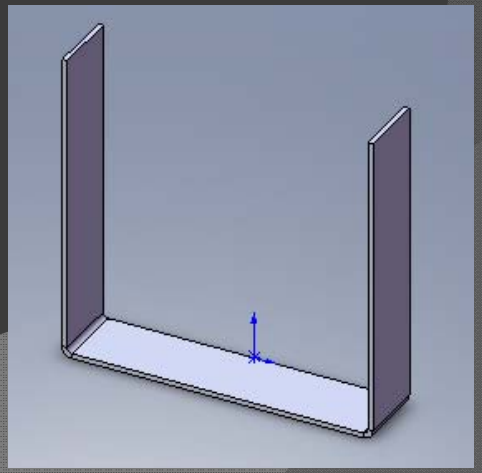
- First feature in a sheet metal part
  - Marks part as a sheet metal part
- Only one Base Flange per part
- Sets default thickness and bend radius for part
- Feature is created from a sketch
  - Sketch can be:
    - single open
    - Single closed
    - multiple-enclosed

Base Flange – How to

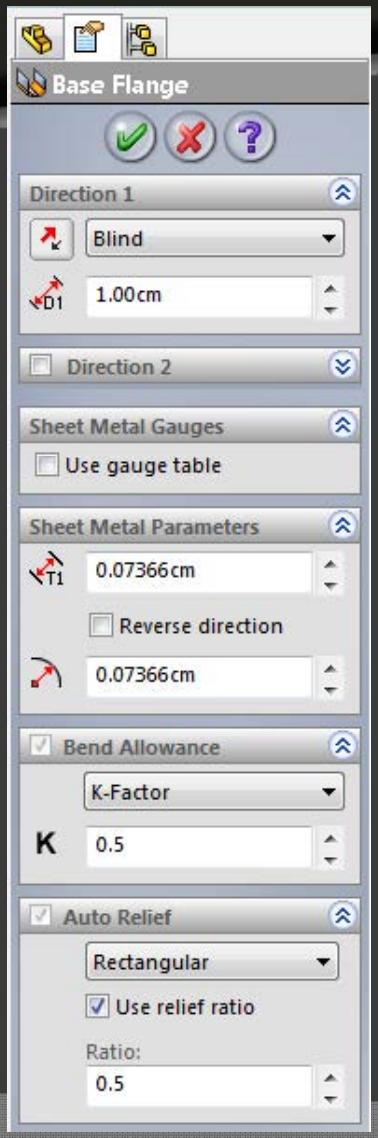
- Create sketch
- Click  Base-Flange/Tab
- Set parameters in base flange property manager
- Click  when complete







Base Flange Property Manager



- Direction 1 and 2 set:
  - End Condition
  - Depth
- Sheet Metal Parameters:
  - Thickness
  - Bend Radius
- Bend Allowance type
- Auto Relief type
  - Relief ratio
  - User defined values for Relief Width and Depth

Base Flange

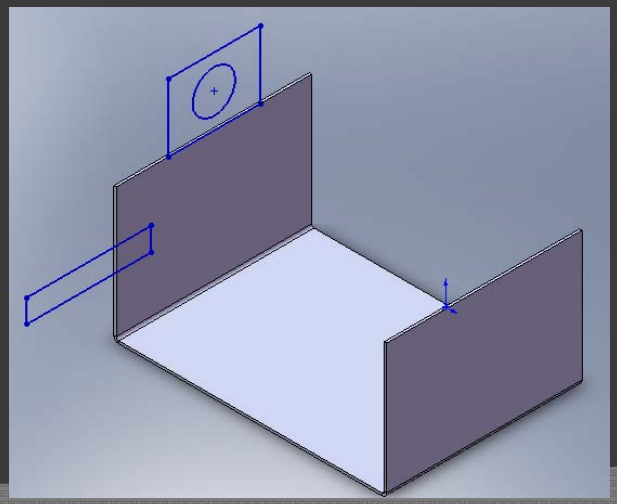
- Features created in the FeatureManager design tree:
  - Sheet-Metal – default bend parameters
  - Base-Flange – first solid features of the part
  - Flat-Pattern – flattens sheet metal part
    - Initially suppressed by default
    - New features are automatically inserted above Flat-Pattern in design tree
    - If unsuppressed, new features are not added to folded part

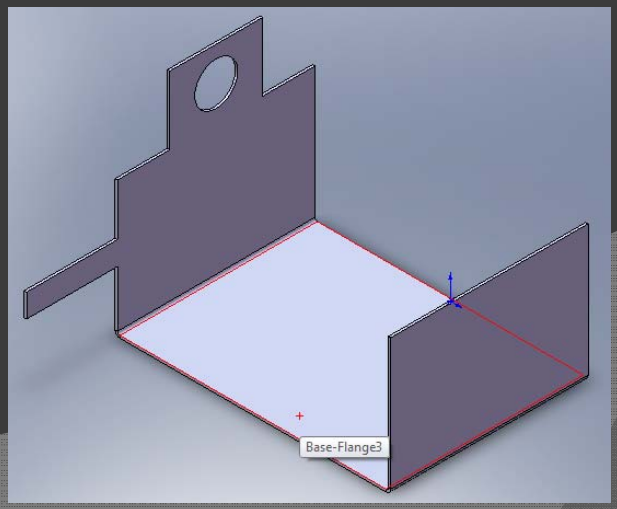
Tab

- Adds a Tab to the sheet metal part
- Thickness of tab linked to thickness of base flange
- Depth automatically coincides with part
- Feature is created from a sketch
  - single closed
  - multiple closed
  - multiple-enclosed
- Sketch must perpendicular to thickness of part

Tab

- Create sketch perpendicular to thickness of part
- Click Base-Flange/Tab
- Tab is added to the part
  - Depth and direction automatically set to match base flange



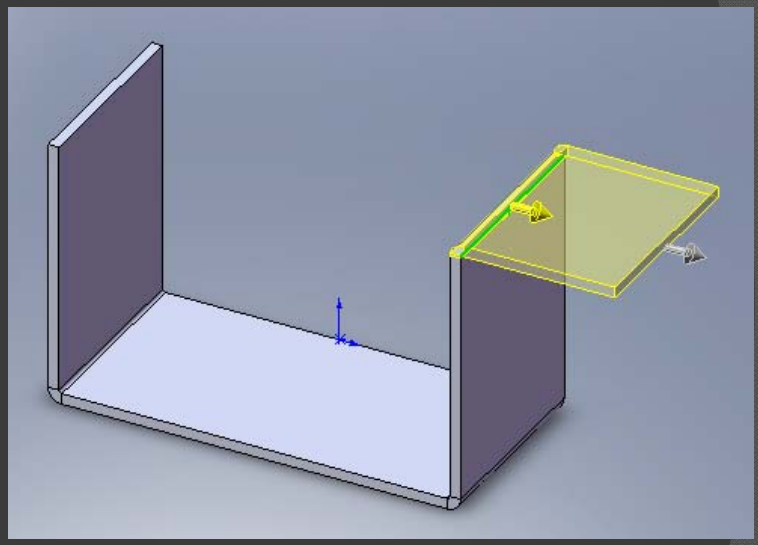


Edge Flange

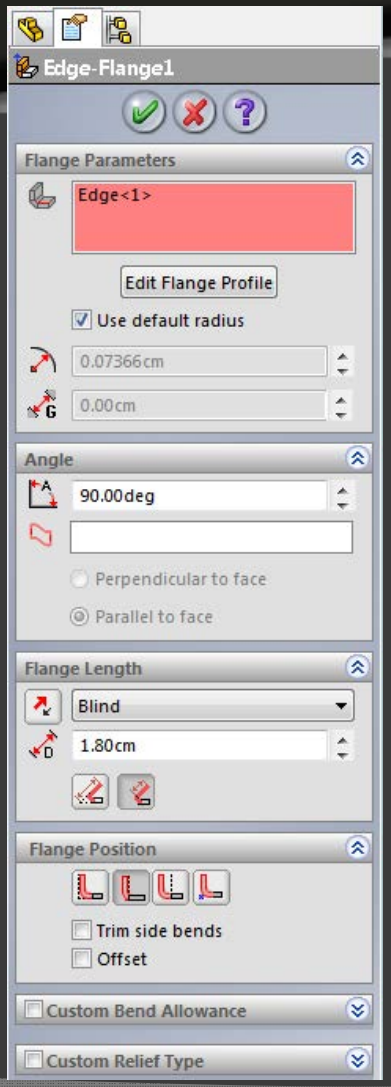
- Adds wall to an edge of sheet metal part
- Can add linear and curved edge flanges
- Thickness linked to part

Edge Flange – How to (Linear)

- Select Edge Flange
- Select 1 or more outer edges
  - Drag the edge by handle
- Set parameters in edge flange property manager



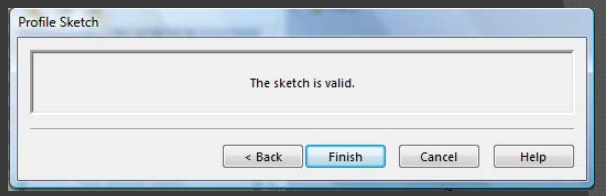
Edge Flange Property Manager

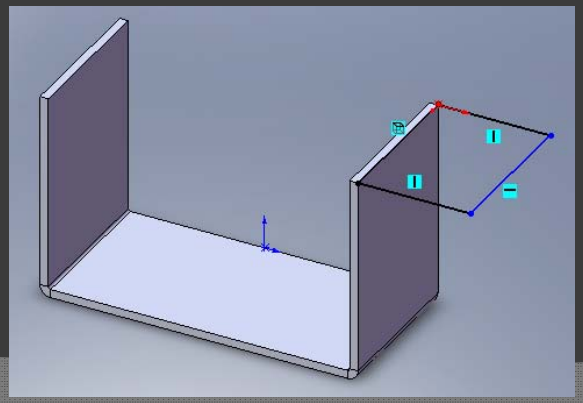


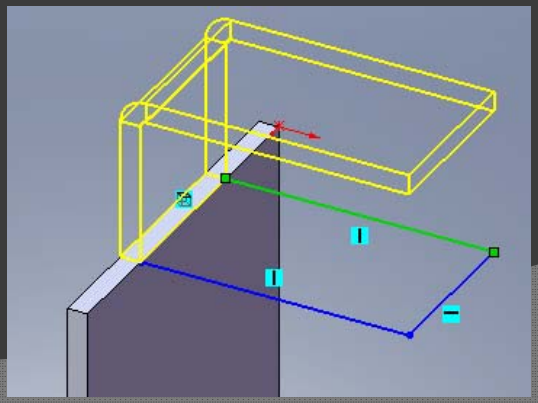
- Flange Parameters
  - Edit Flange Profile
  - User can define Bend Radius and Gap distance
- Angles
- Flange Length
- Flange Position
  - Bend Position
  - Offset
- Custom Bend Allowance
- Custom Bend Type

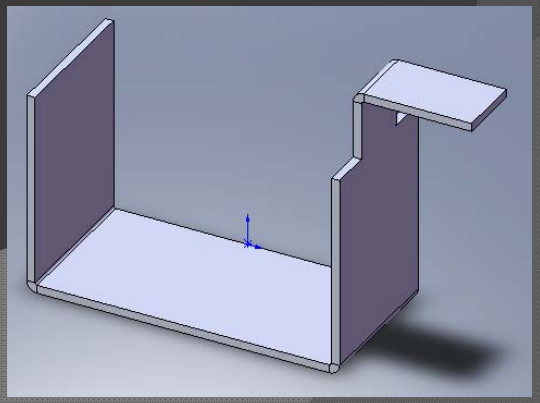
Edge Flange – How to (Linear)

- Select Edit Flange Profile
  - Profile Sketch dialog box opens
  - Modify sketch
  - Select Back to accept changes and continue editing
  - Select Finish to close Profile Sketch dialog box







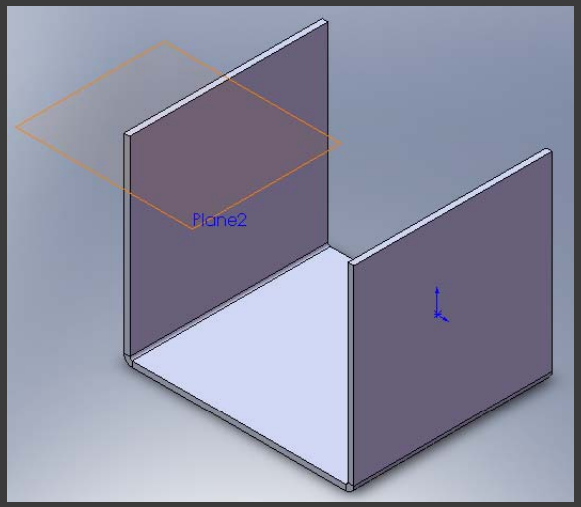


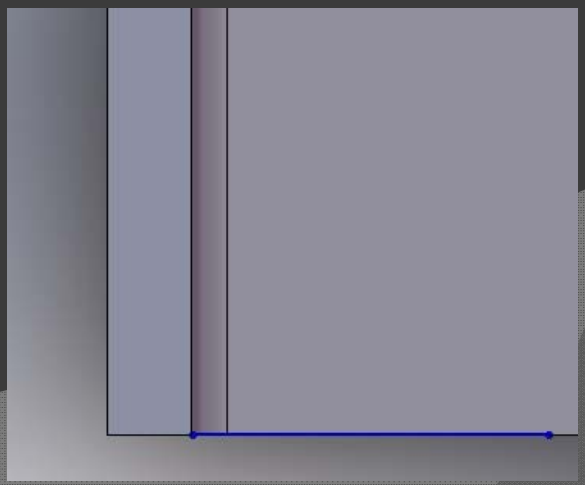
Miter Flange

- Adds a series of flanges about one or more edges
- Performed so flanges are flush
- Sketch can contain:
  - Lines
  - Arcs
  - Multiple continuous lines
- Flange can be made on series of tangent or non-tangent edges

Miter Flange – How to

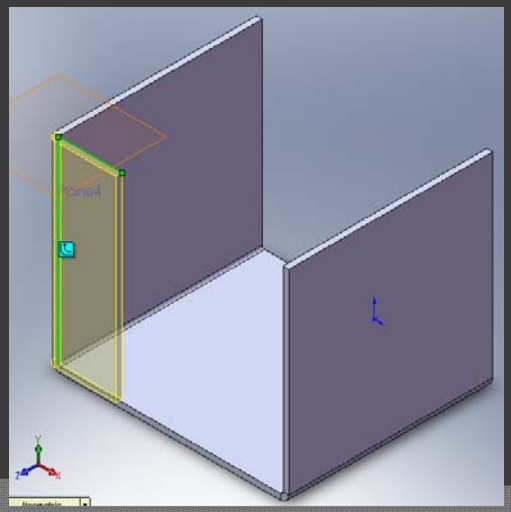
- Select Miter Flange
- Select edge of part
  - Note: sketch plane is created normal to selected edge with origin at closest endpoint
- Create appropriate sketch

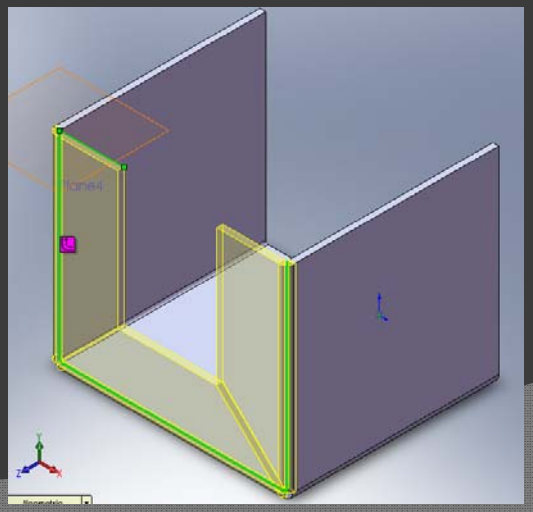


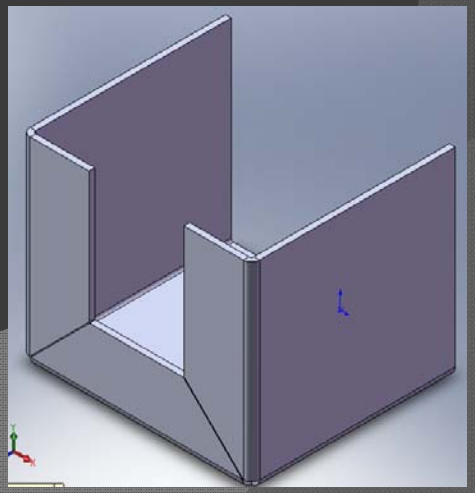


Miter Flange – How to

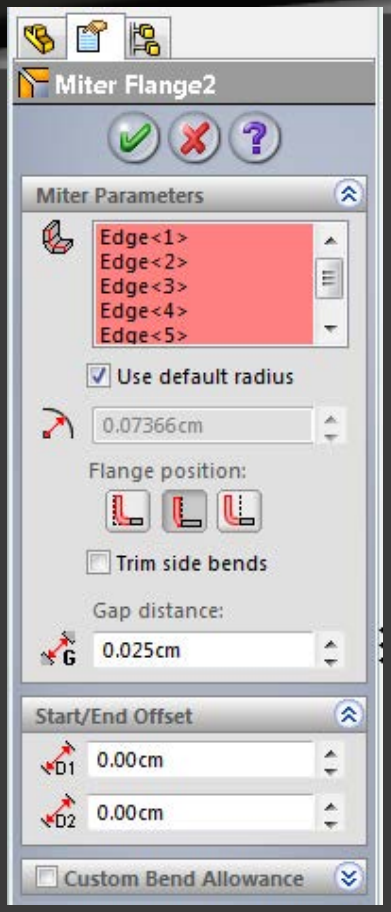
- Click to close Sketch
  - Miter flange is applied to initial edge
- Select edges to apply miter flange
  - Click Propagate to select all tangent edges
- Set parameters in miter flange property manager







Miter Flange Property Manager

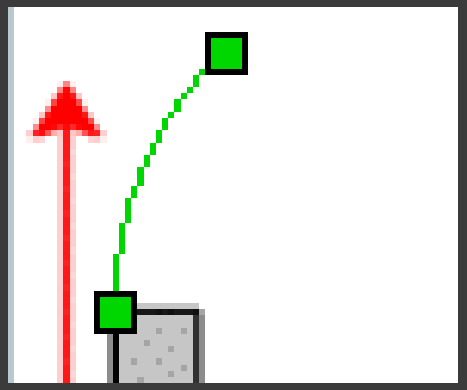


- Miter Parameters
  - User can define Bend Radius
  - Bend Position
  - Gap distance
- Start/End Offset
  - If offset other then zero, option to set Custom Relief Type
- Custom Bend Allowance

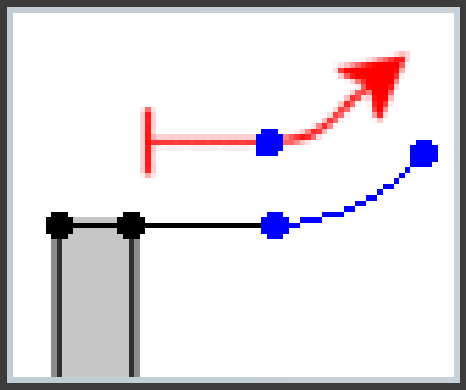
Miter Flange – Arc Sketches

- Arc can be tangent to long edge of part
- If tangent to thickness, requires small sketch line

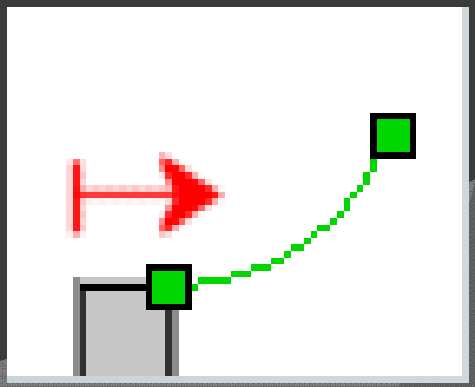
Valid Sketch:
Arc tangent to long edge



Valid Sketch:
Sketch line between arc and part

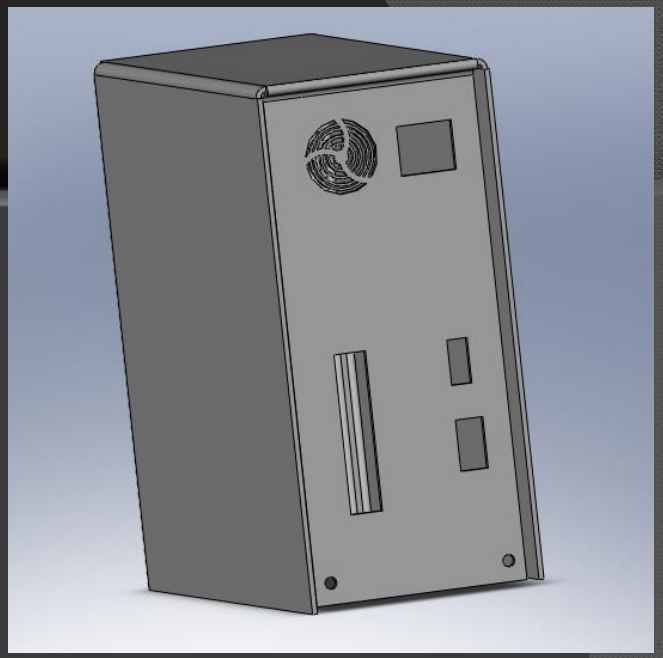


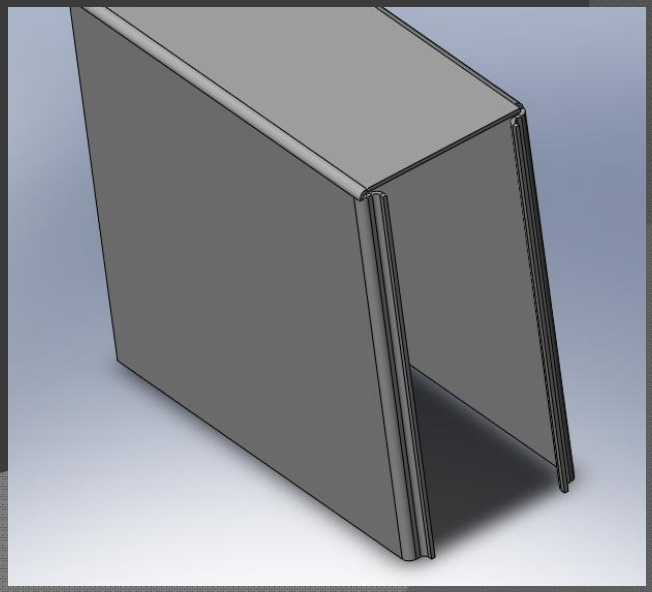
Invalid Sketch:
Arc tangent to thickness



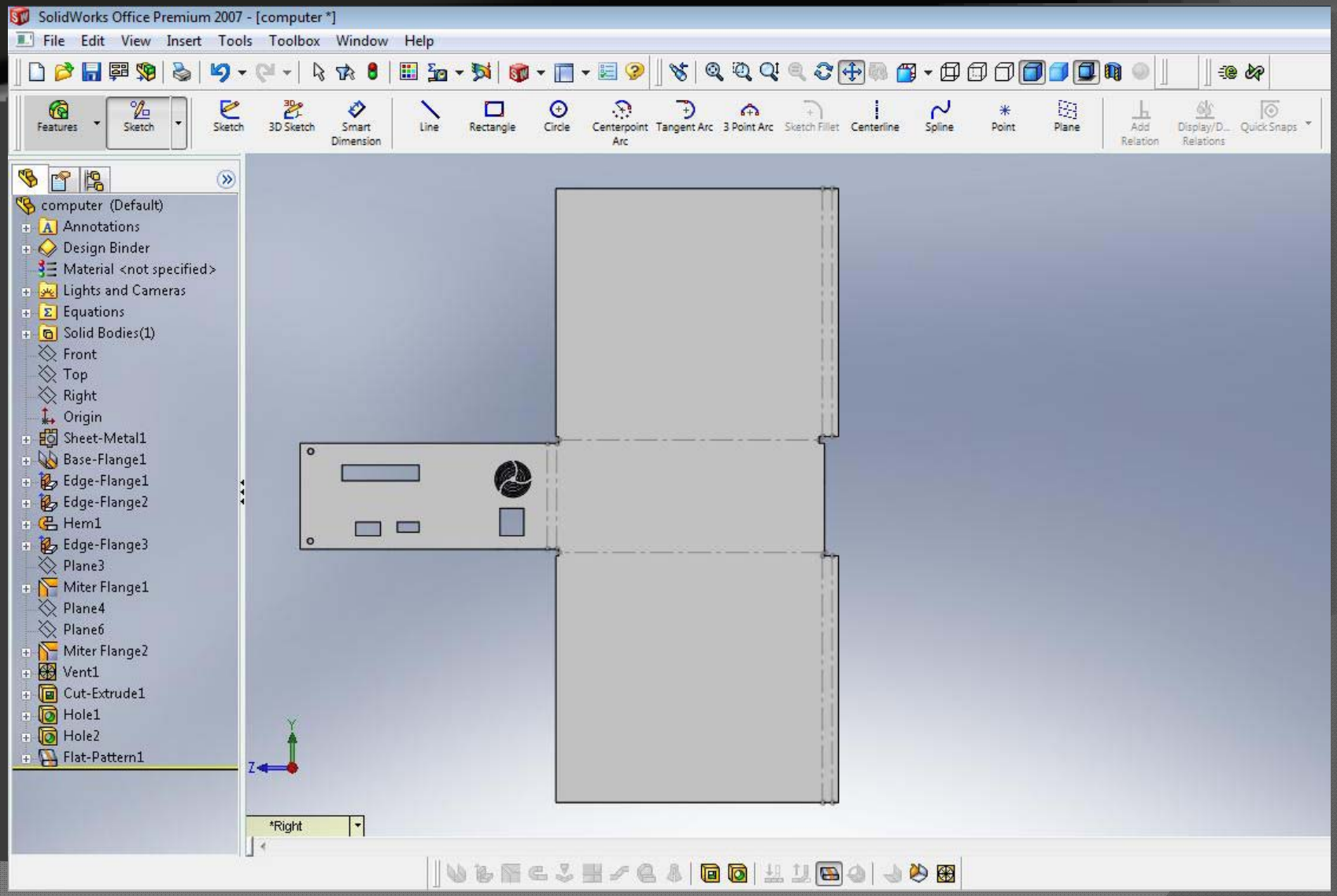
Flatten

- Shows the flat pattern for the existing sheet metal part.
  - Useful for identifying interferences
  - Identifies impossible bends

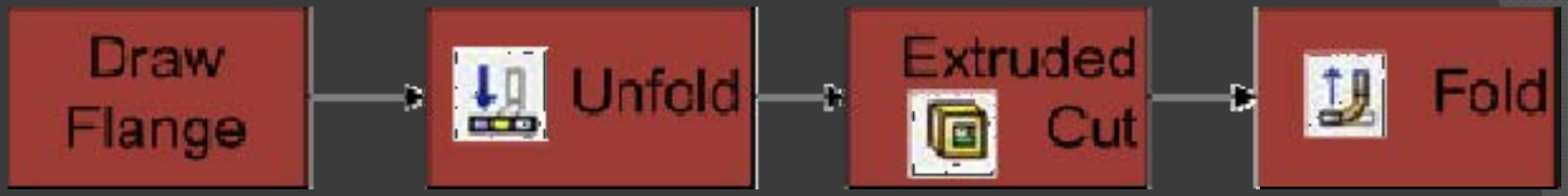




Flatten


SolidWorks Office Premium 2007 - [computer *]
File Edit View Insert Tools Toolbox Window Help
Features
Sketch
3D Sketch
Smart Dimension
Line
Rectangle
Circle
Centerpoint Arc
Tangent Arc
3 Point Arc
Sketch Fillet
Centerline
Spline
Point
Plane
Add Relation
Display/D... Relations
Quick Snaps
computer (Default)
Annotations
Design Binder
Material <not specified>
Lights and Cameras
Equations
Solid Bodies(1)
Front
Top
Right
Origin
Sheet-Metal1
Base-Flange1
Edge-Flange1
Edge-Flange2
Hem1
Edge-Flange3
Plane3
Miter Flange1
Plane4
Plane6
Miter Flange2
Vent1
Cut-Extrude1
Hole1
Hole2
Flat-Pattern1
*Right

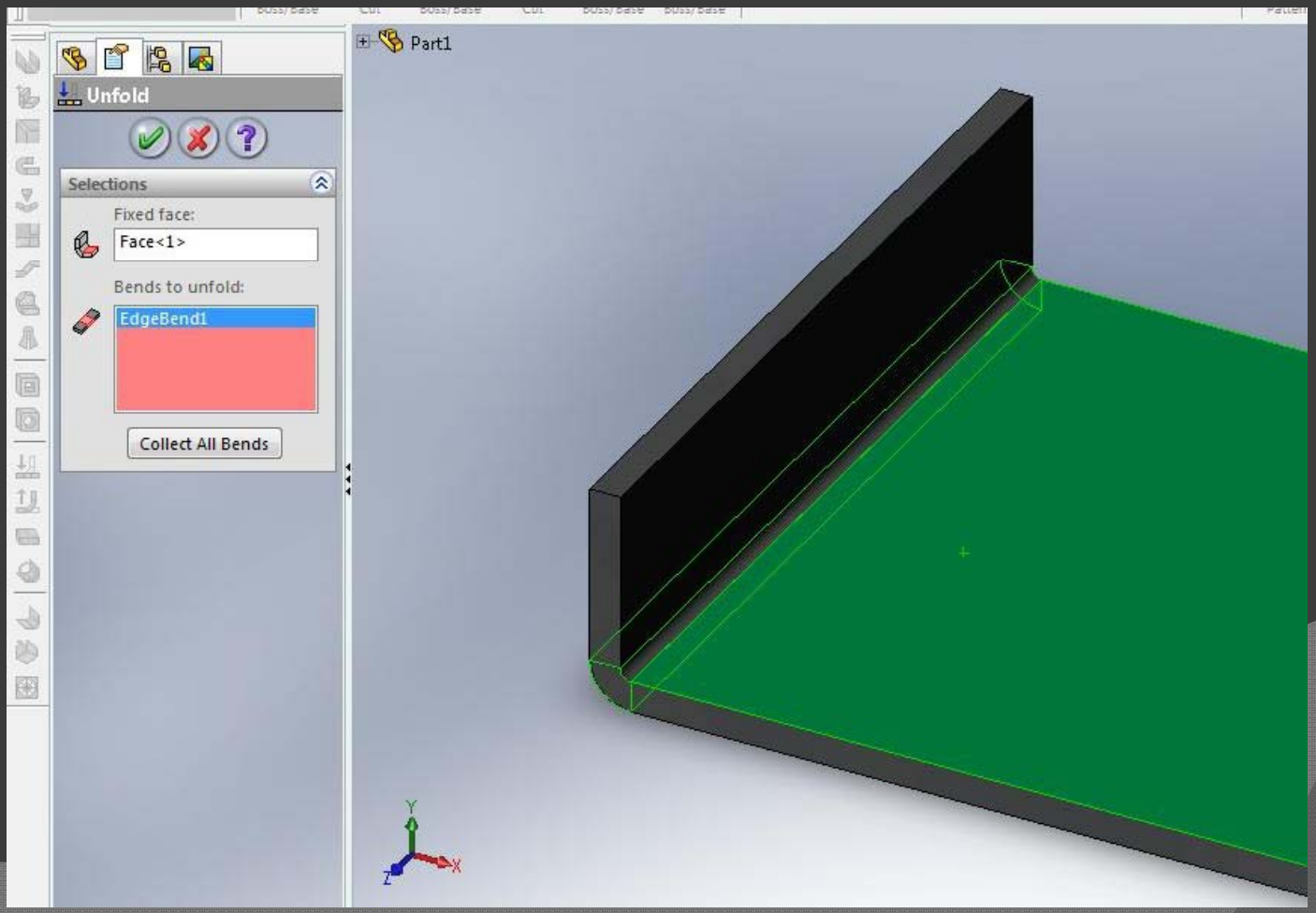
Modification of Flanges



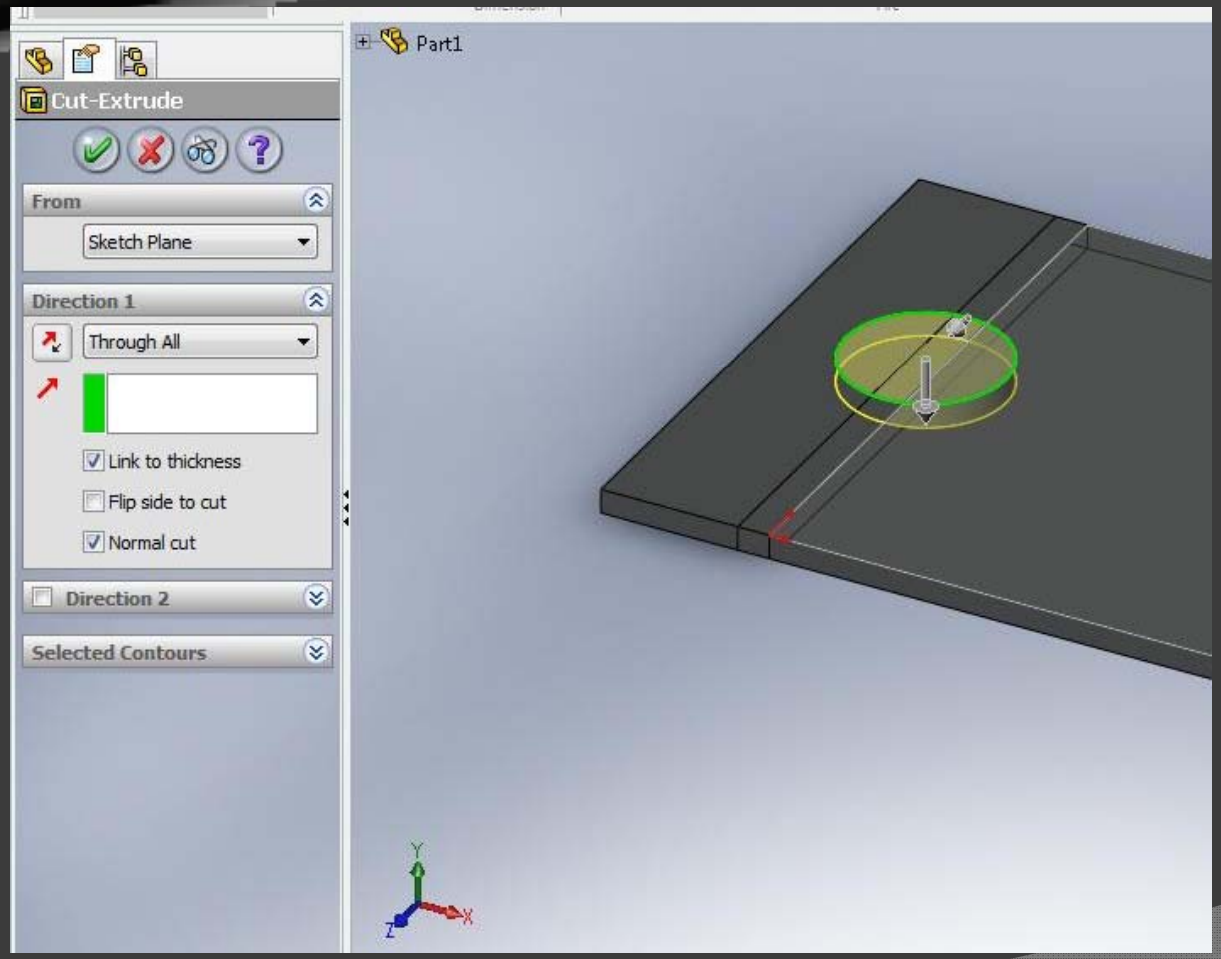
- Must Unfold before attempting to cut across a bend or curved face.
- Extrude the cut onto the unfolded face.
- Insert a Fold – SolidWorks automatically contours the cut to match the folded face.
- Flatten to identify interferences.

Unfold

Selecting the Unfold icon opens the fold interface:

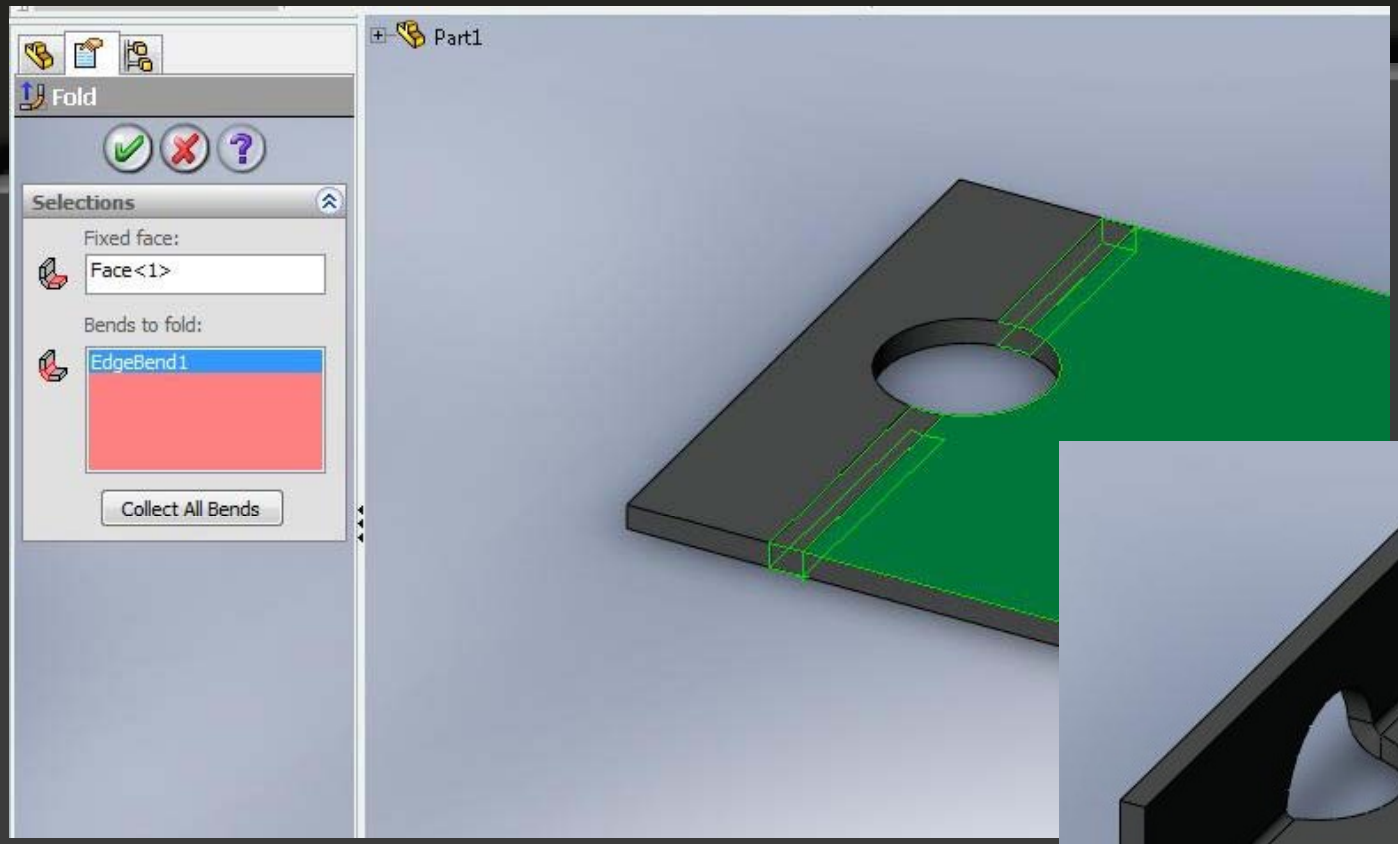


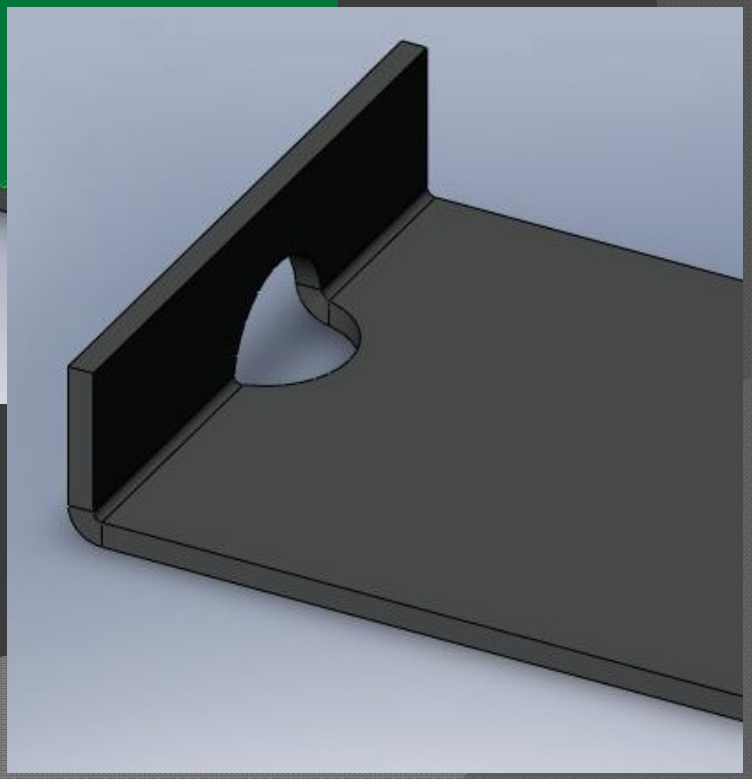
Extruded Cut



- Select face to insert cut on
- Sketch the cut
- Select Link to thickness and Normal cut.
  - Allows for material changes in the future

Fold

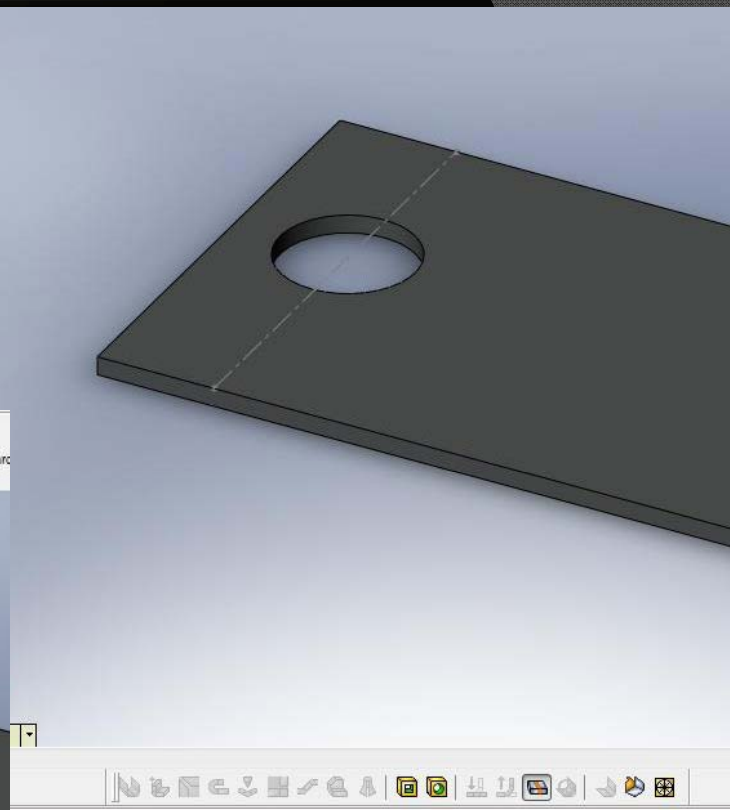


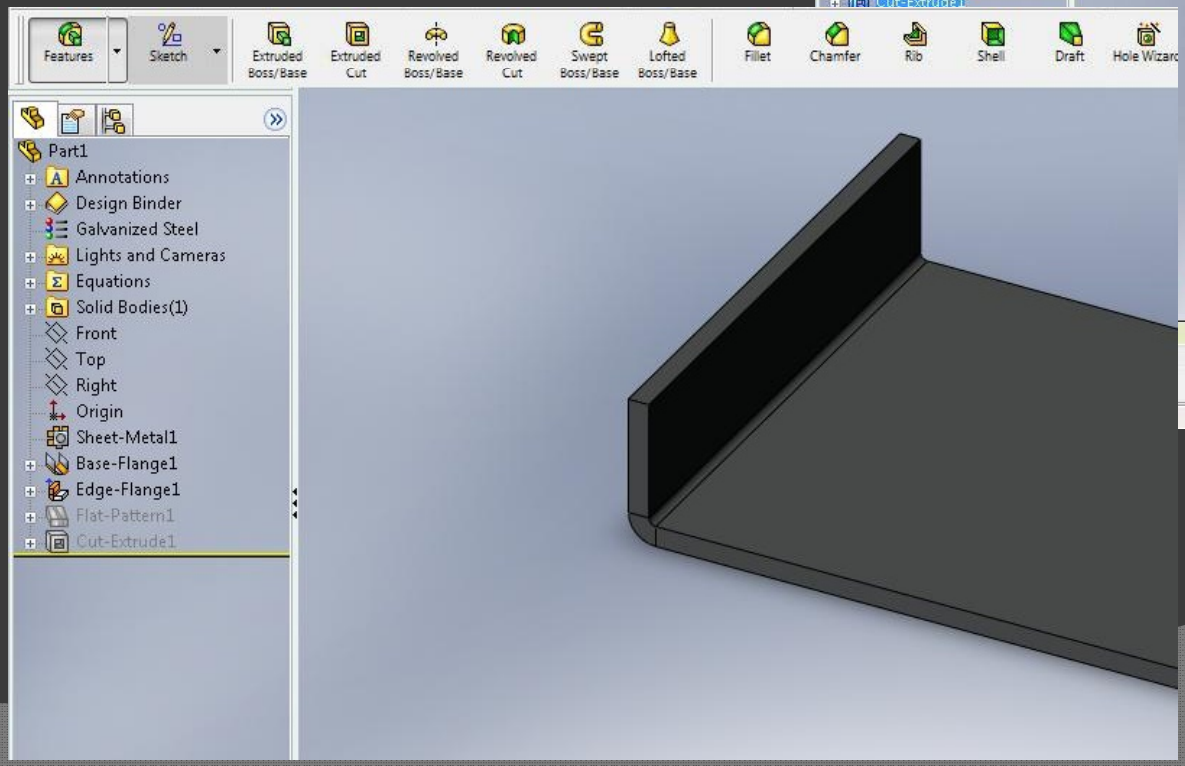


- Select face to fix
- Select bend to be folded

Incorrect Method

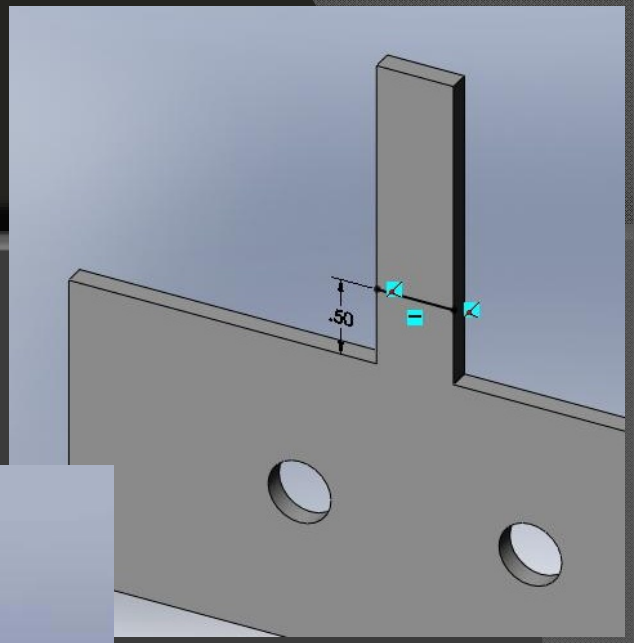
- Flatten
- Extruded Cut

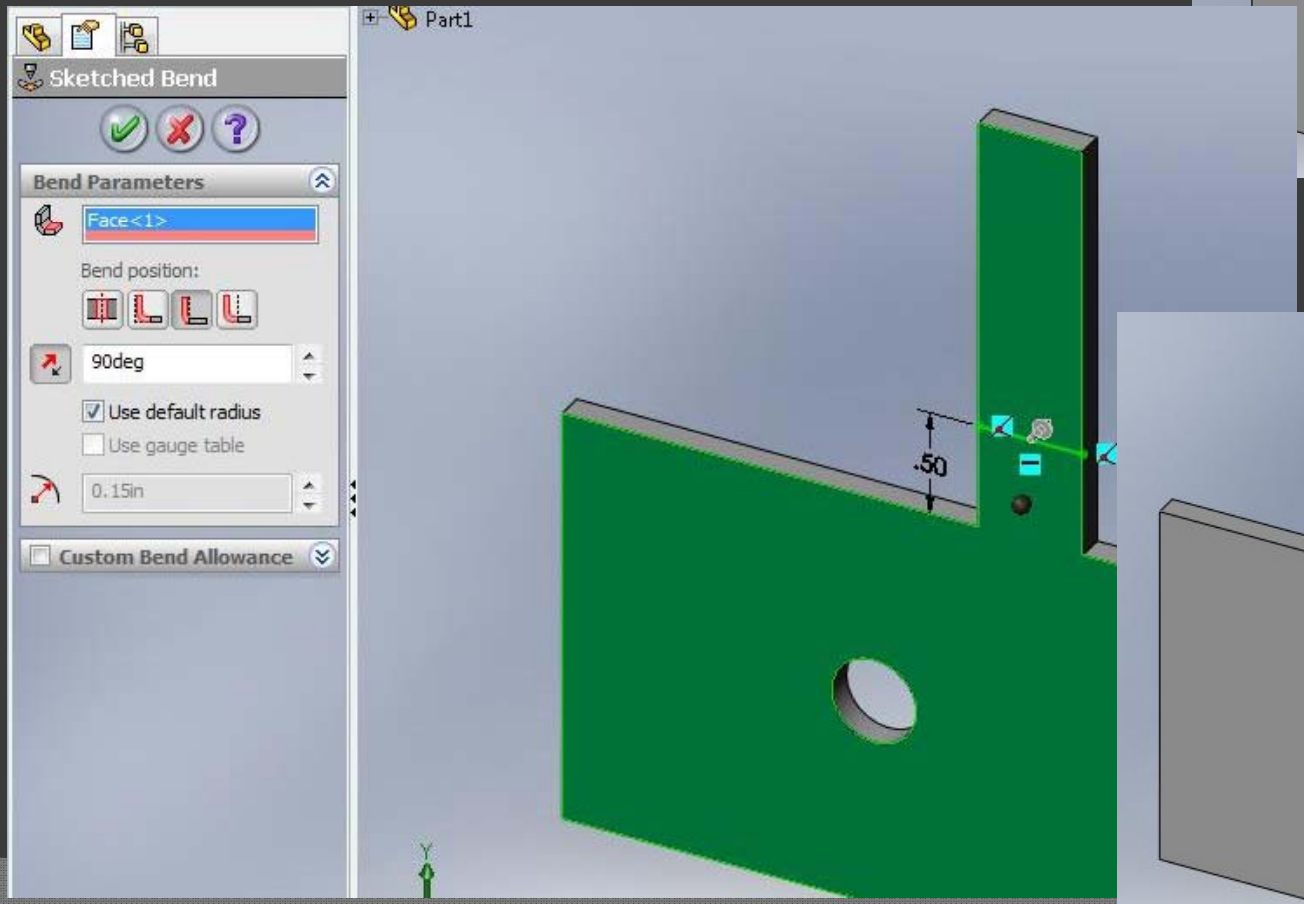


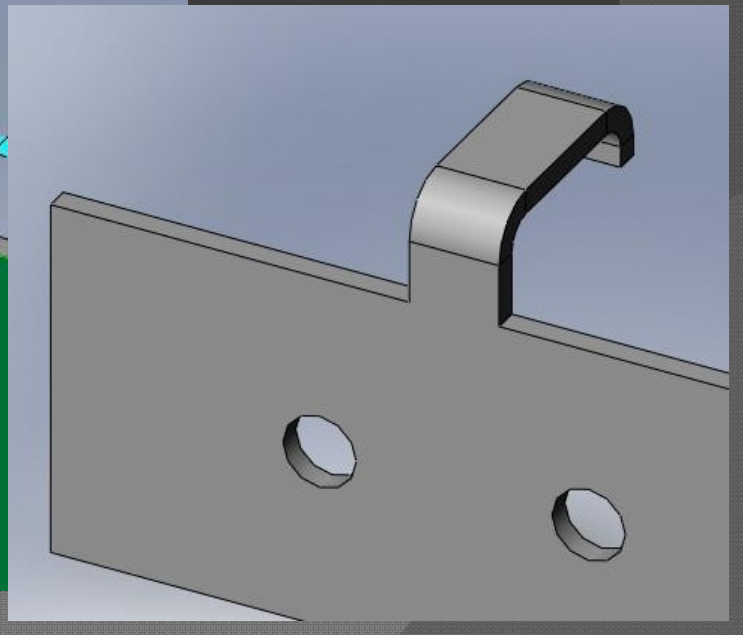


Sketched Bend

- Adds a bend from a selected sketch in a sheet metal part.



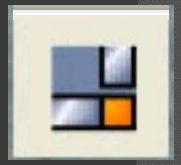




Other Functions

- Hem – Curls the edge of a sheet metal piece

- Closed corner – Extends the face of a sheet metal part



- Jog – Adds two bends from a sketched line in a sheet metal part

Other Functions

- Break-Corner – Cuts material from a face or edge in a sheet metal part

- Lofted-Bend – Creates a sheet metal part between two sketches using a loft feature.

Rip – Creates a gap between two edges in a sheet metal part

Other Functions

- Vent – Uses sketch elements to create a vent for airflow in both a plastic or sheet metal design

- Simple Hole – Creates a cylindrical hole on a planar face.

Other Functions

- Insert Bends – Creates a sheet metal part from the existing part

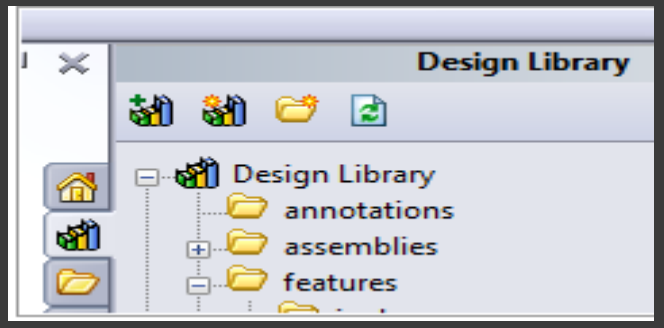
- No Bends – Rolls back all bends in the sheet metal part

Design Library

- The Design Library is a subset of folders in the Task Pane that stores reusable features
- Benefits:
  - Items in the Design Library can quickly be added to SolidWorks documents using drag-and-drop
  - Various forming tools for sheet metal can be stored locally
  - Provides a library of standard components in an easy to access user interface directly in SolidWorks

Accessing the Design Library

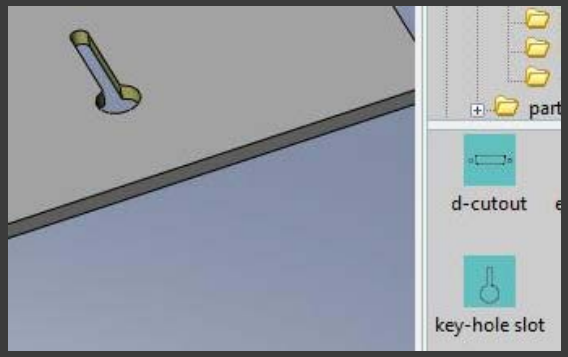
- In order to access the Design Library ensure that the Task Pane is enabled. (click View -> Task Pane)
- When enabled, the Task Pane is usually anchored to the right side of the graphics area.



- In the top pane is a tree structure of the Design Library and in the bottom pane is a list of files available in the Design Library.

Use Feature

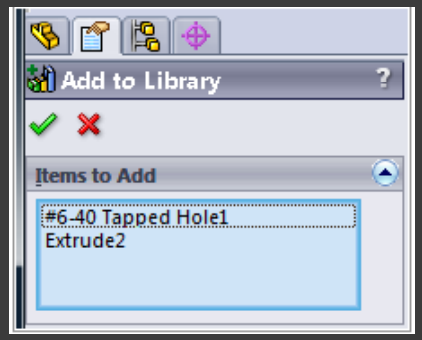
- Go to the Design Library and make the Forming Tools Folder the current folder by using the context menu.
- Drag and Drop the feature to the desired surface.



- Apply the Geometric Constraints and Dimensions for locating the feature

Add Feature to Design Library

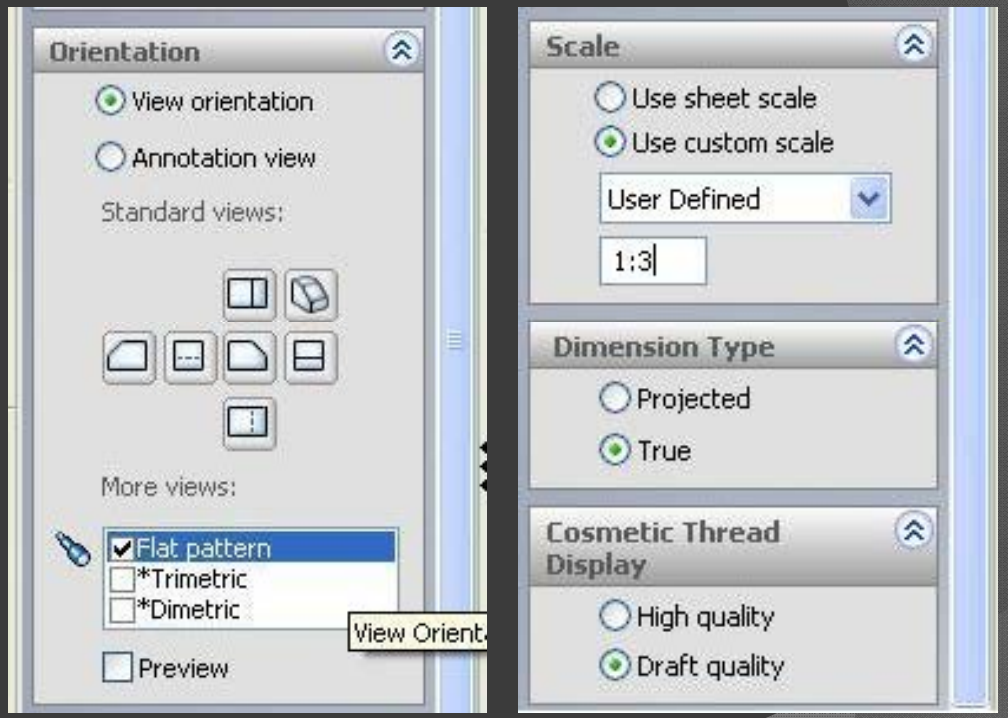
- In the FeatureManager, select the features while holding down the CTRL key and dragging the features into the lower pane of the Design Library



- Enter the File Name that will be displayed and add the description that will be shown
- Click the green check mark and the features will be added

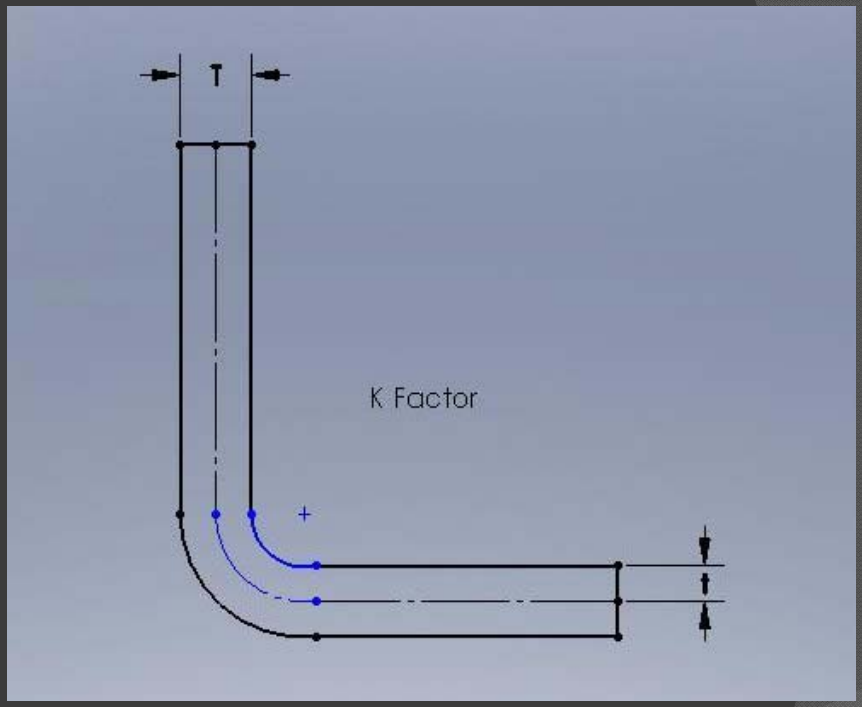
Flattened and Isometric

- Insert a flat pattern view
- Modify flat pattern configurations



K Factor

- K = t/T
- % distance of natural line into the material
- Changing k factor changes amount of material in radius



Bending Table

- A bending table in the drawing

	A	B	C	D	E	F	G	H	I	J	K
1	Type:	K-Factor									
2	Material:	Soft Copper and Soft Brass									
3											
4	Angle	Radius / Thickness									
5		1.00	2.00	3.00	4.00	5.00	6.00	7.00	8.00	9.00	10.00
6	15										
7	30										
8	45										
9	60										
10	75										
11	90										
12	120										
13	150										
14	180										
15											

Sheet1 Sheet2 Sheet3

Time Saving Tips

- Use symmetry when applicable
- Edit a flange after its been created
- Always link features to the thickness
- Long load time
- Clear view palette if used

Conclusion

Advantages of Sheet Metal Modeling in SolidWorks

- Special tools for the application
- Flatten feature
- Tables and bend information
- Design Library features

Questions?