

SolidWorks

# **SHEET METAL DESIGN**

July 28, 2009

# Summary

This presentation will outline

- ① Benefits of using SolidWorks Sheet Metal Tools
- ① Sheet Metal Toolbars
  - Design Tools
- ① Design Library
  - Modifying Features
- ① Sheet Metal Drawing
  - Bend Tables
  - K Factors
- ① Tips for Sheet Metal Design

# Introduction

## Advantages of SolidWorks Sheet Metal Tools

- ① Design Intent for Sheet Metal Fabrication
- ① Specified tools for sheet metal operations and common features
- ① Bend Factor – K Factor Calculations
- ① Visual aids – Flatten Features
- ① Link features to sheet thickness
- ① 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





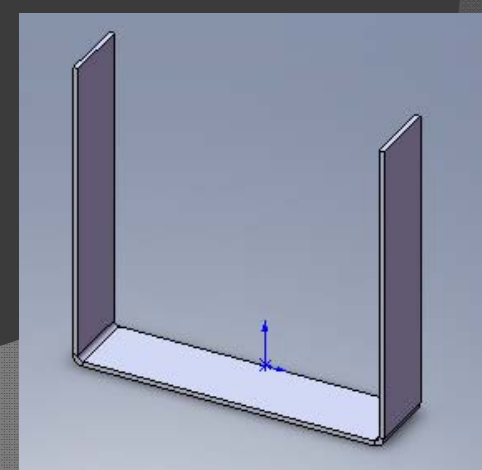
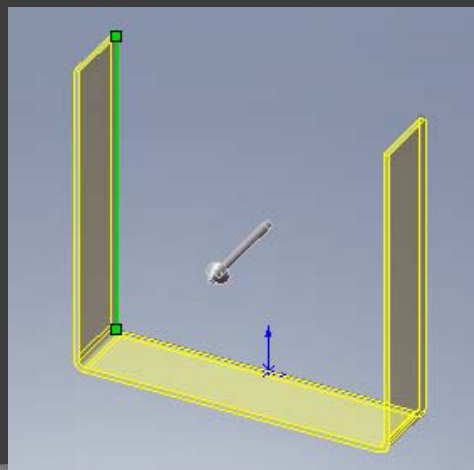
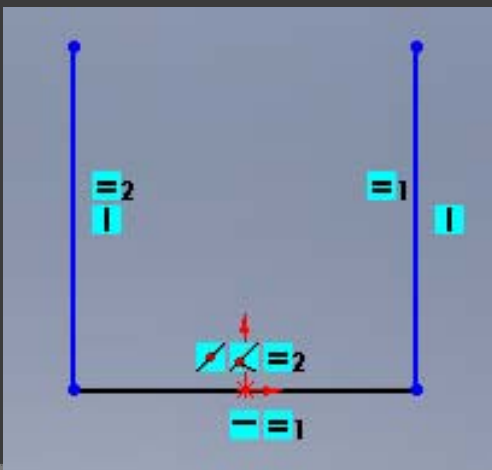
# Base Flange

- ⦿ 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



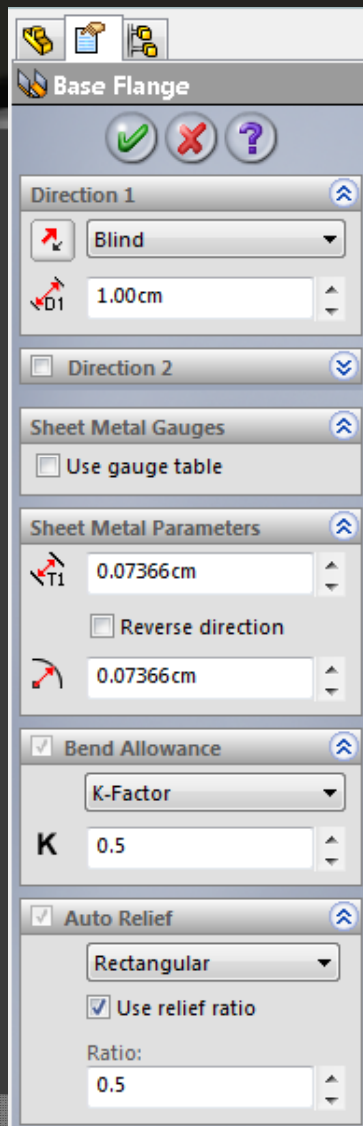
# Base Flange – How to




- 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

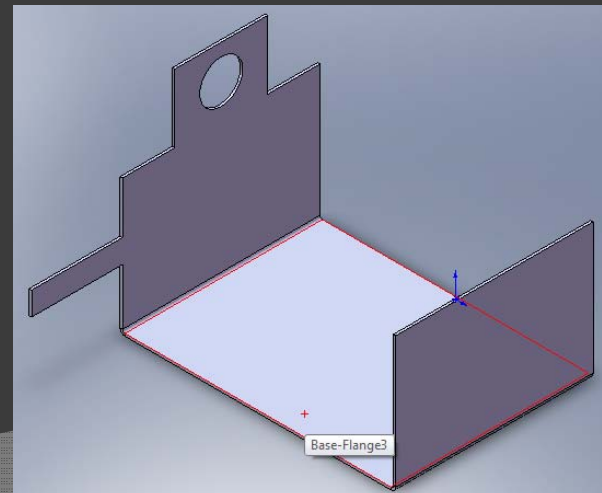
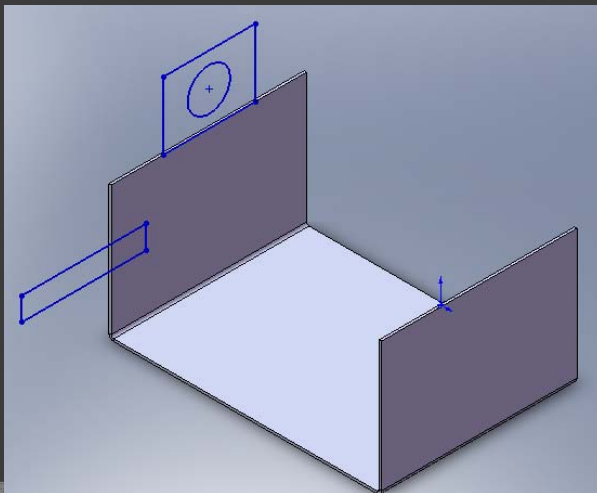
- ⦿ Adds a Tab to the sheet metal part
- ⦿ Thickness of tab linked to thickness of base flange
- ⦿ Depth automatically coincides with part
- ⦿ Feature is created from a sketch
  - single closed
  - multiple closed
  - multiple-enclosed
- ⦿ Sketch must be perpendicular to thickness of part





# 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






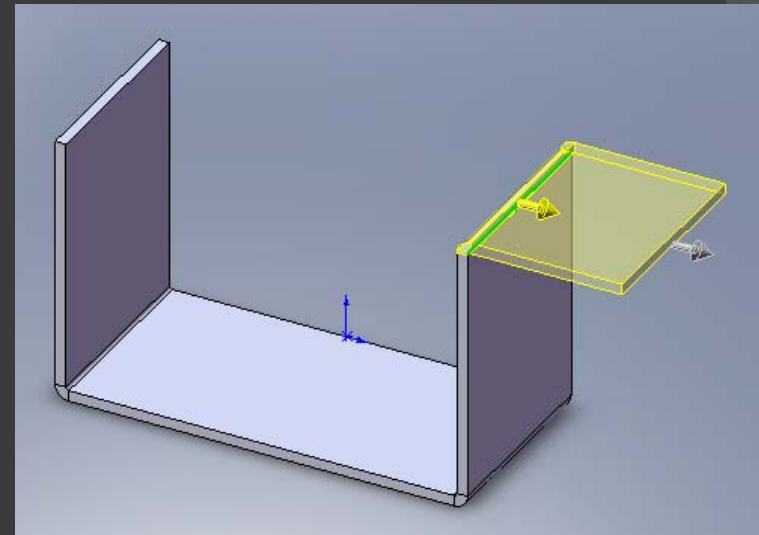
# Edge Flange

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



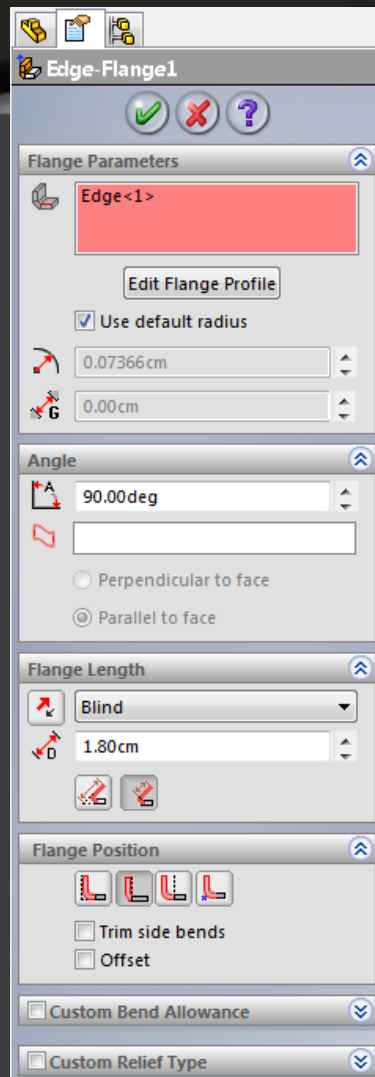
# Edge Flange – How to (Linear)



- ◉ 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

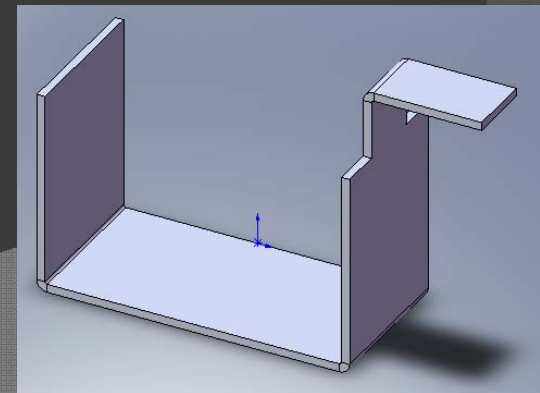
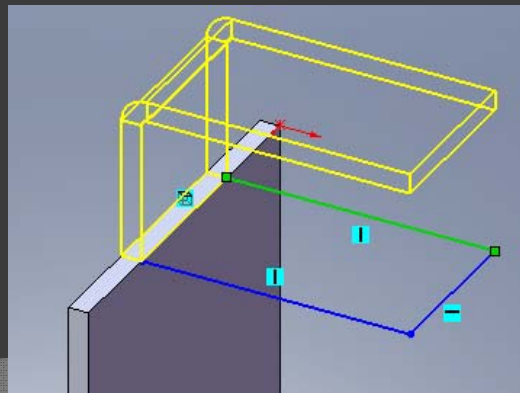
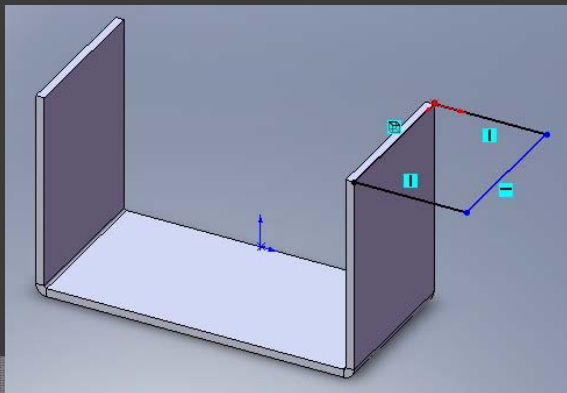
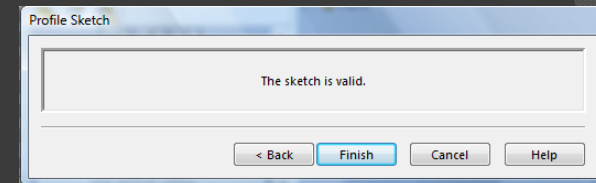


- Flange Parameters
  - Edit Flange Profile
  - User can define  Bend Radius and  Gap distance
- Angles
- Flange Length
- Flange Position
  - Bend Position
  - Offset
- Custom Bend Allowance
- Custom Bend Type



# 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

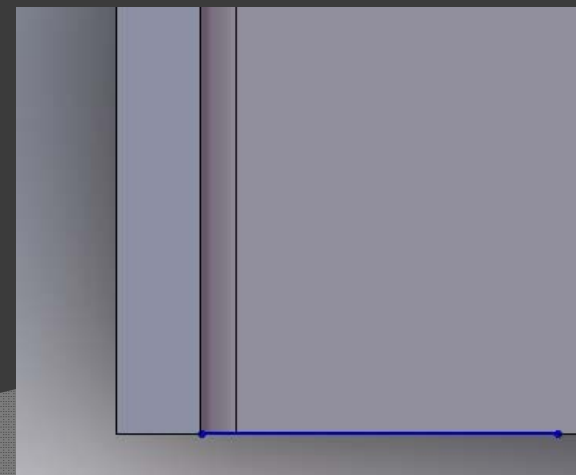
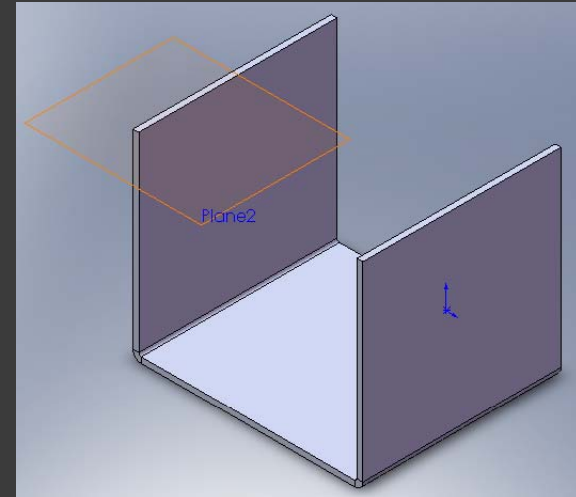
- ◉ Adds a series of flanges about one or more edges
- ◉ Performed so flanges are flush
- ◉ Sketch can contain:
  - Lines
  - Arcs
  - Multiple continuous lines
- ◉ Flange can be made on series of tangent or non-tangent edges







# Miter Flange – How to

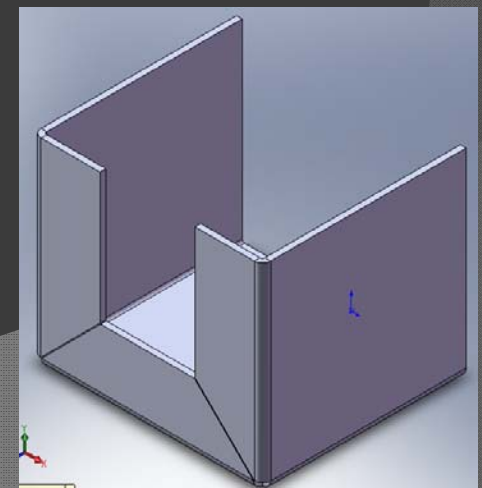
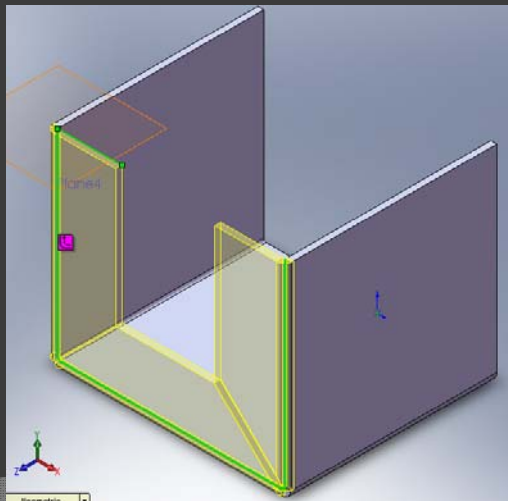
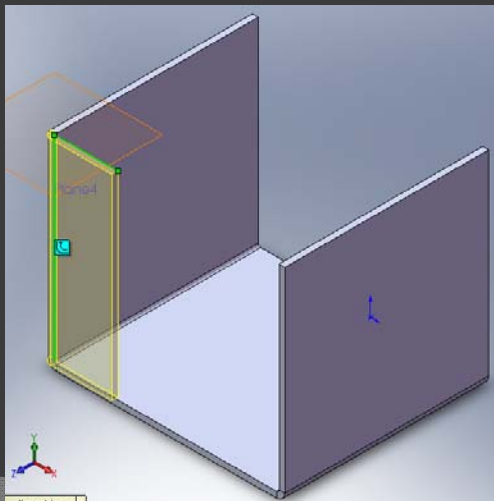
- Select  Miter Flange
- Select edge of part
  - Note: sketch plane is created normal to selected edge with origin at closest endpoint
- Create appropriate sketch





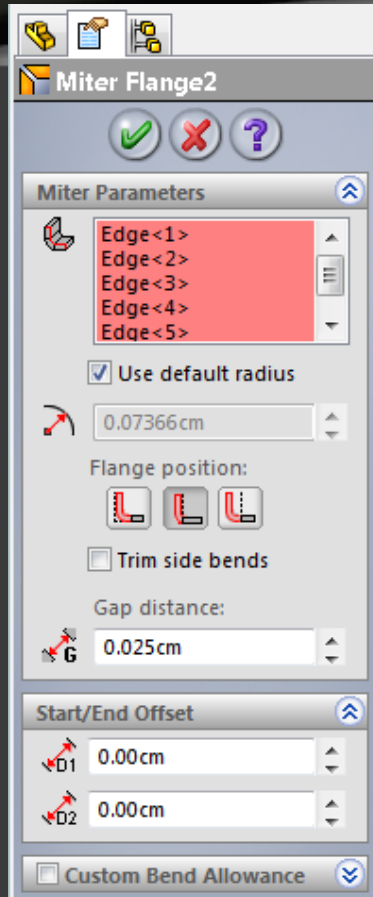
# Miter Flange – How to

- 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 than zero, option to set Custom Relief Type

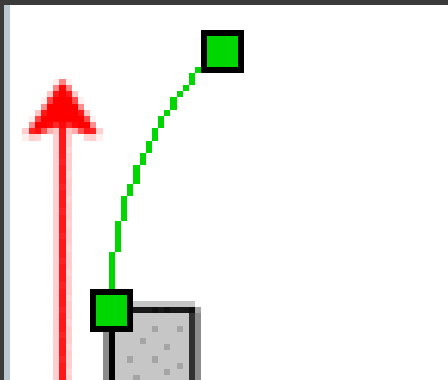
## ○ Custom Bend Allowance



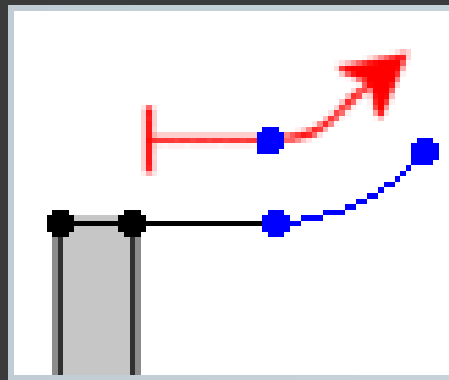
# Miter Flange – Arc Sketches

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

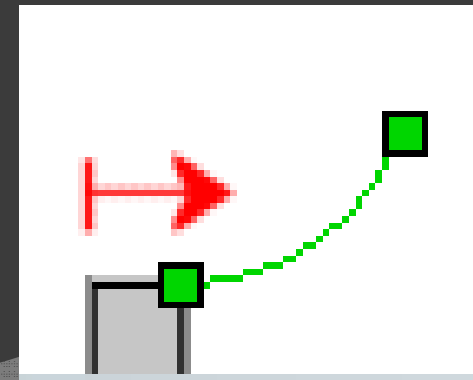
Valid Sketch:  
Arc tangent to  
long edge



Valid Sketch:  
Sketch line between  
arc and part



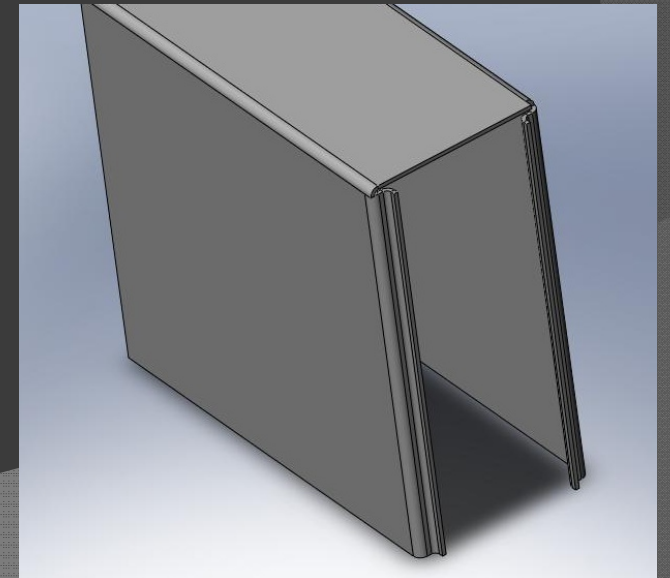
Invalid Sketch:  
Arc tangent to  
thickness





# Flatten

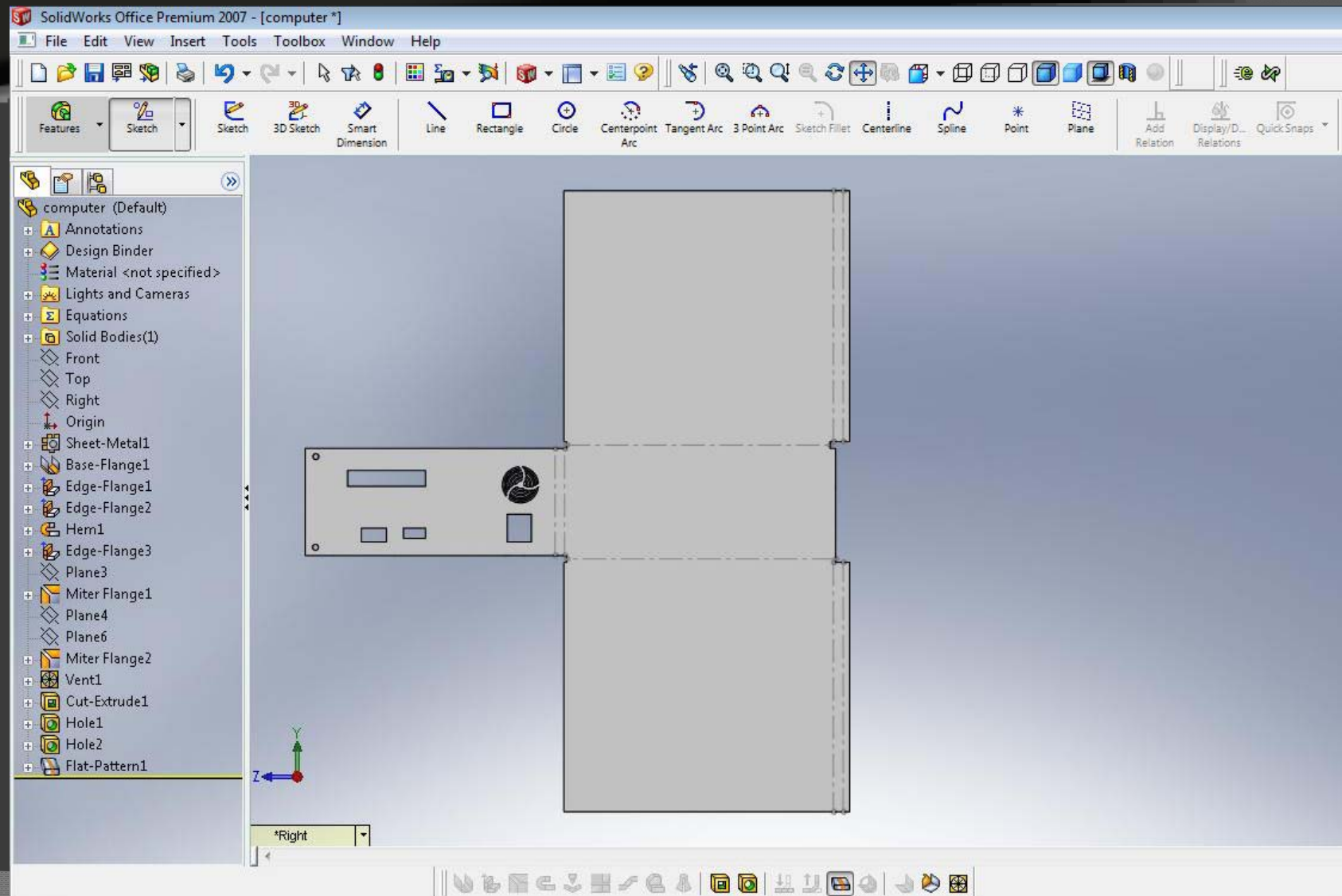
- Shows the flat pattern for the existing sheet metal part.
  - Useful for identifying interferences
  - Identifies impossible bends







# Flatten





# Modification of Flanges

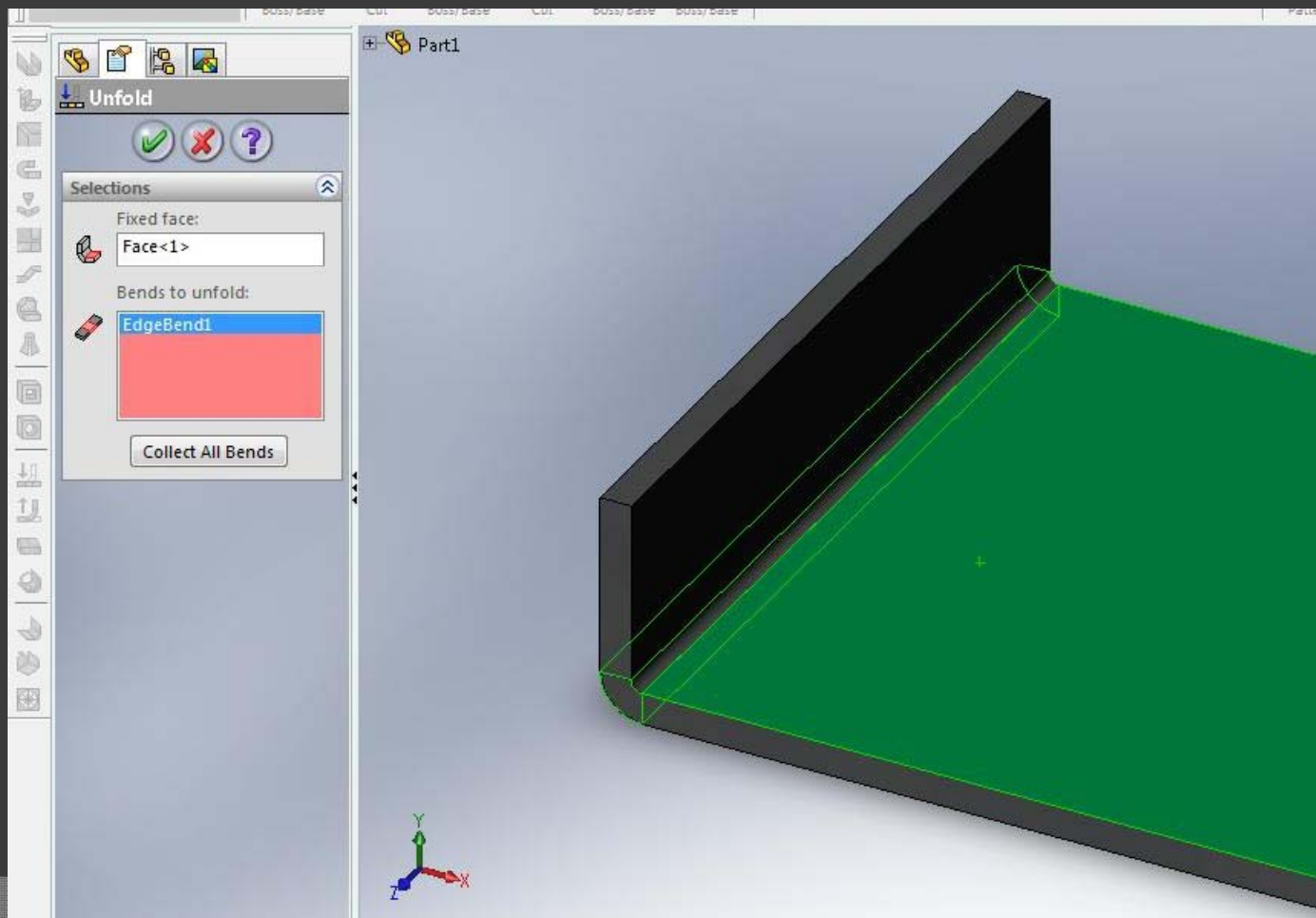


- ⦿ Must Unfold before attempting to cut across a bend or curved face.
- ⦿ Extrude the cut onto the unfolded face.
- ⦿ Insert a Fold – SolidWorks automatically contours the cut to match the folded face.
- ⦿ Flatten to identify interferences.

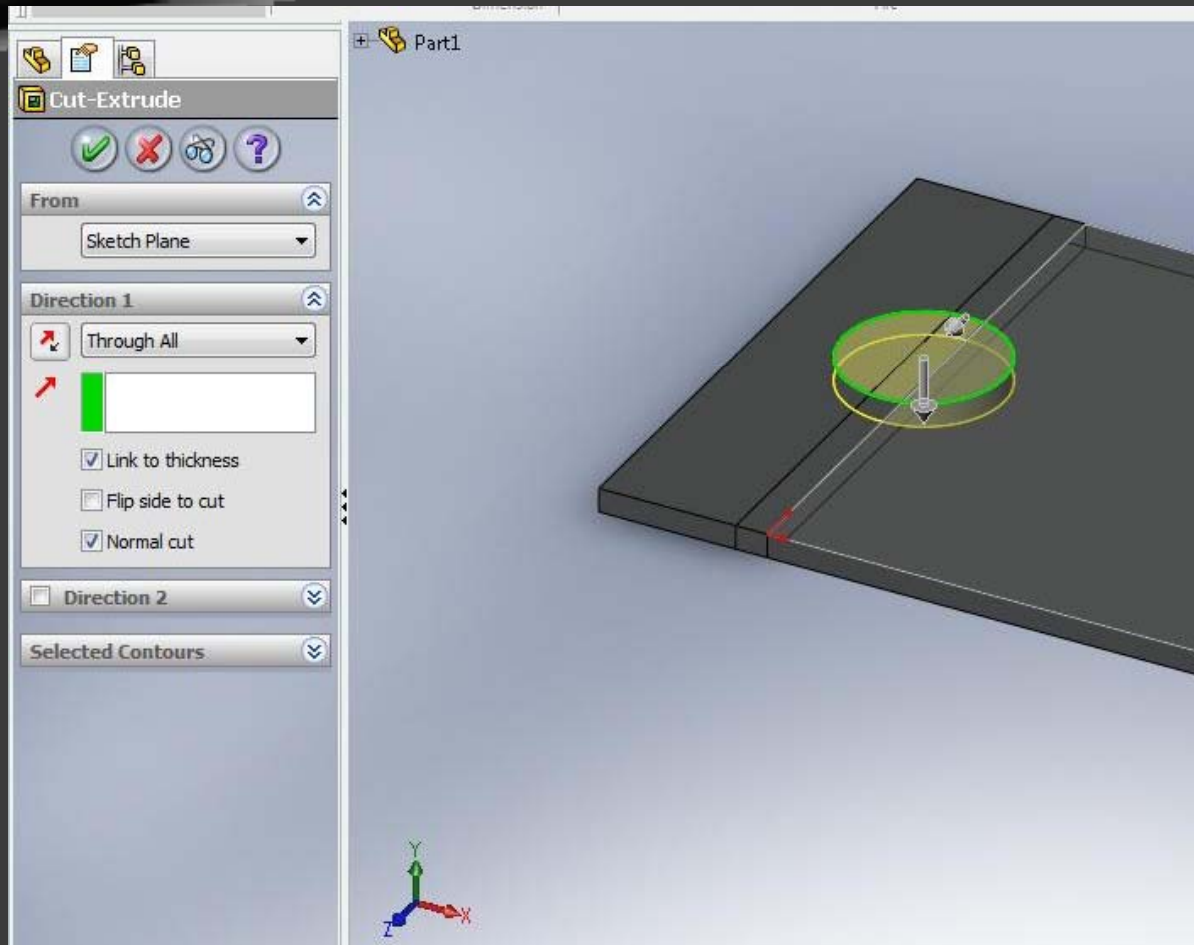


# Unfold

Selecting the Unfold icon opens the fold interface:



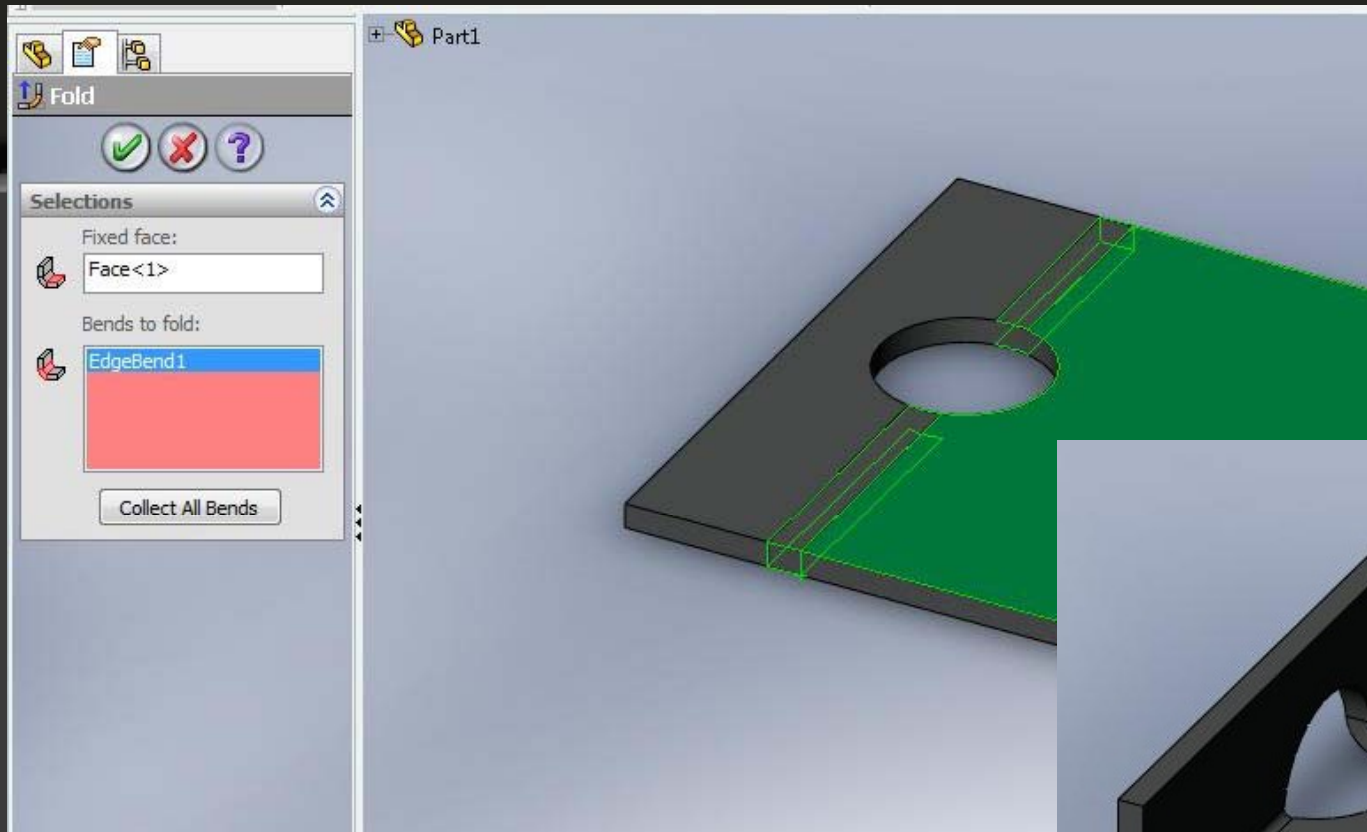
# Extruded Cut



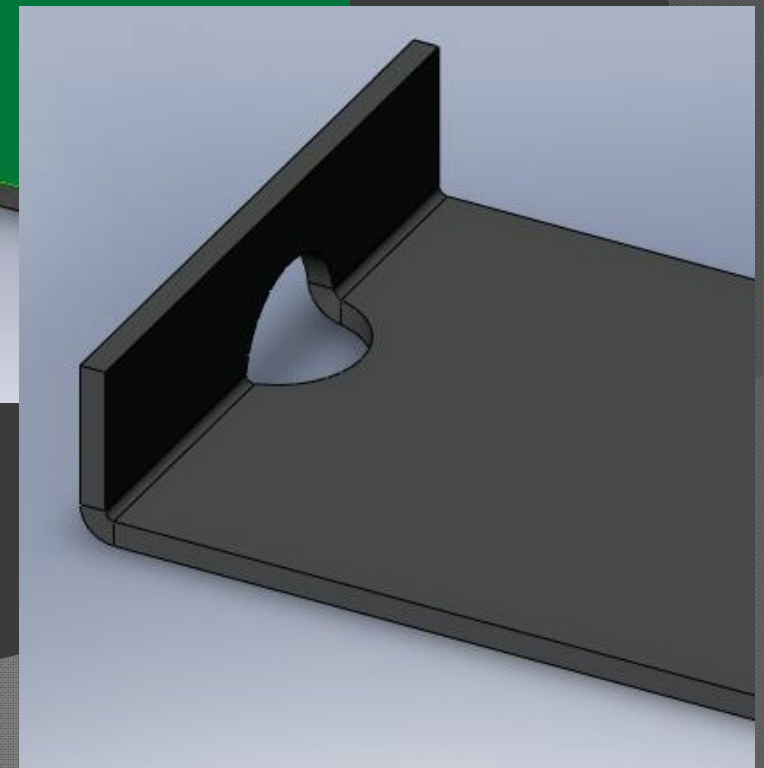
- Select face to insert cut on
- Sketch the cut
- Select Link to thickness and Normal cut.
  - Allows for material changes in the future



# Fold



- Select face to fix
- Select bend to be folded





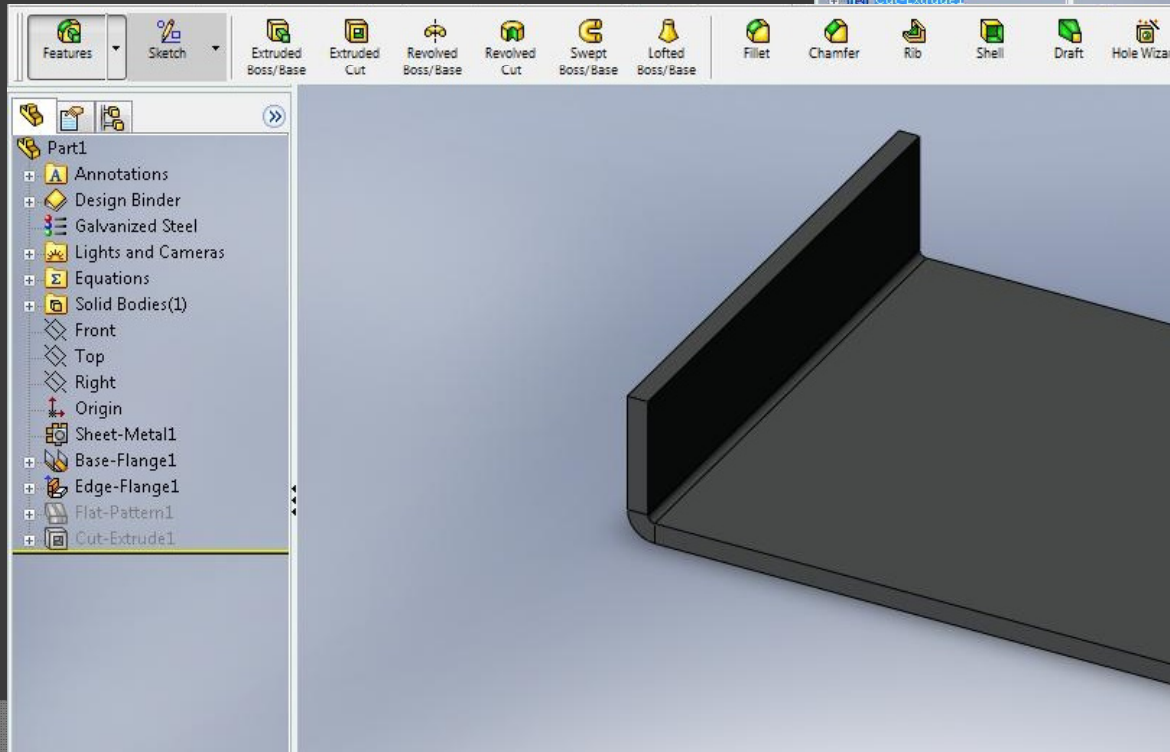
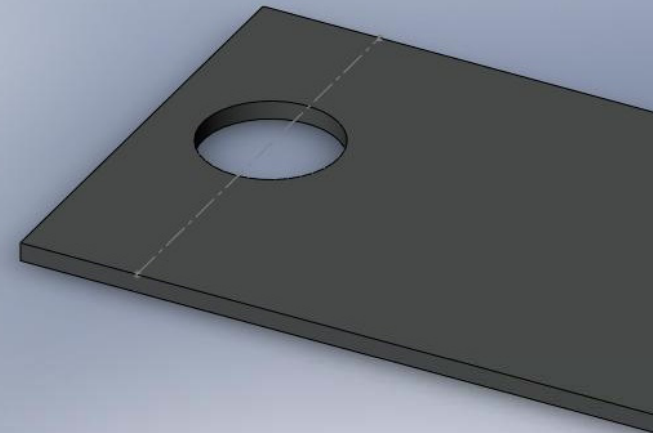
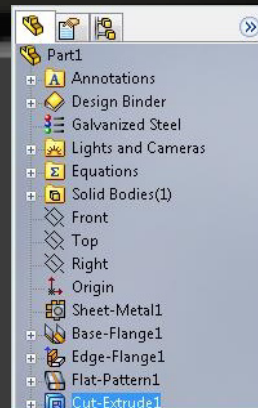
# Incorrect Method



Flatten



Extruded Cut

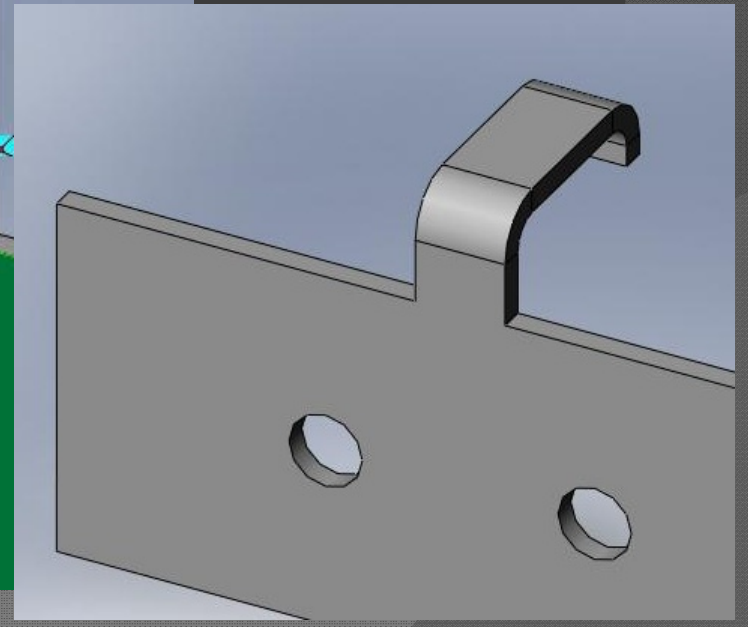
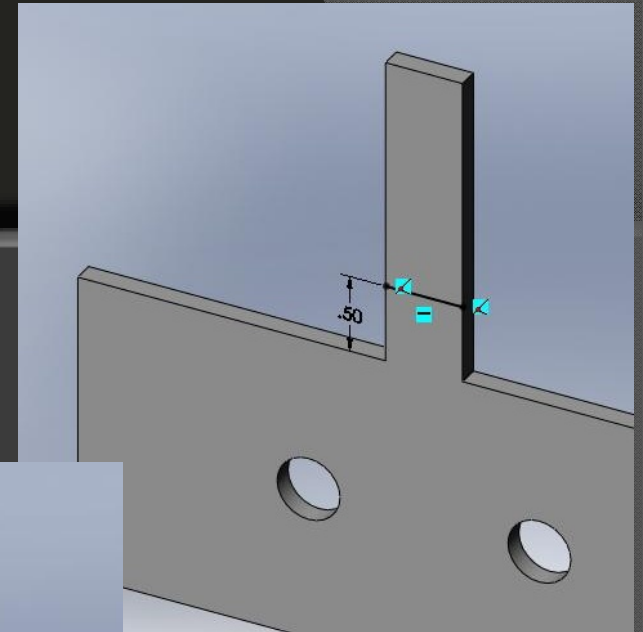
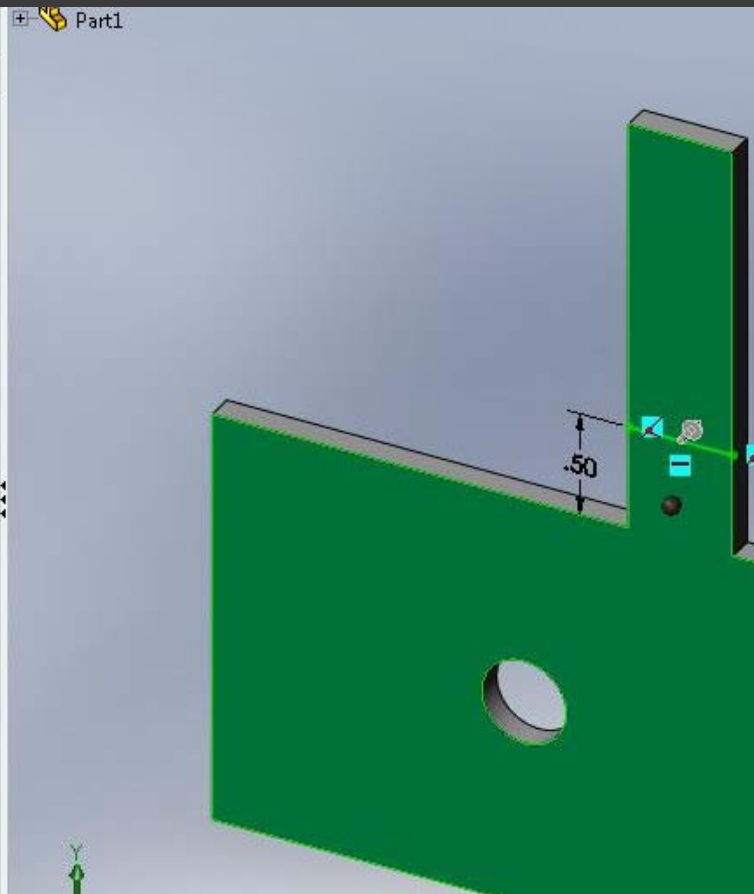
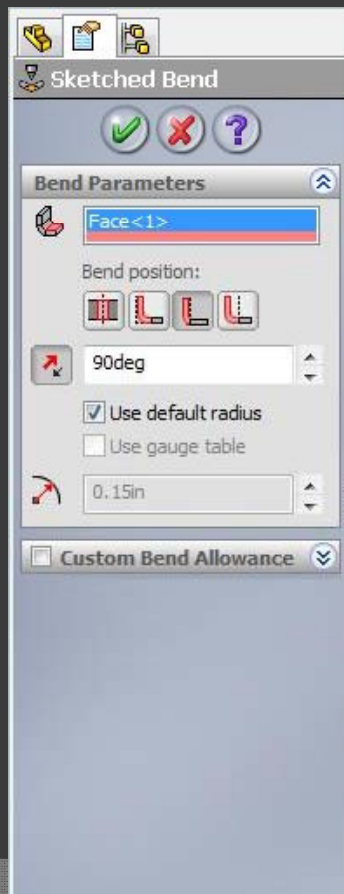






# Sketched Bend

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

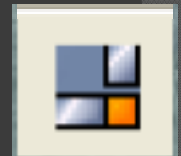


# Other Functions



- 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



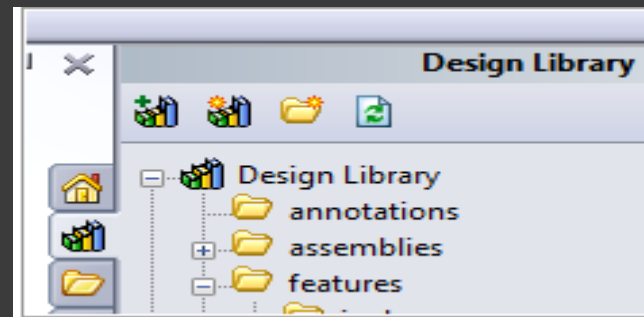


# Design Library

- 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

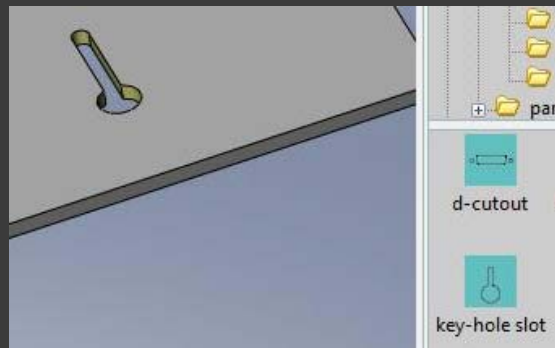
- In order to access the Design Library ensure that the Task Pane is enabled. (click View -> Task Pane)
- When enabled, the Task Pane is usually anchored to the right side of the graphics area.



- 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

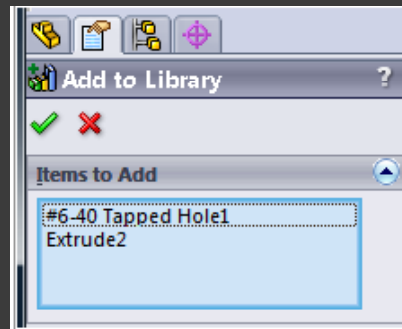
- Go to the Design Library and make the Forming Tools Folder the current folder by using the context menu.
- Drag and Drop the feature to the desired surface.



- Apply the Geometric Constraints and Dimensions for locating the feature

# Add Feature to Design Library

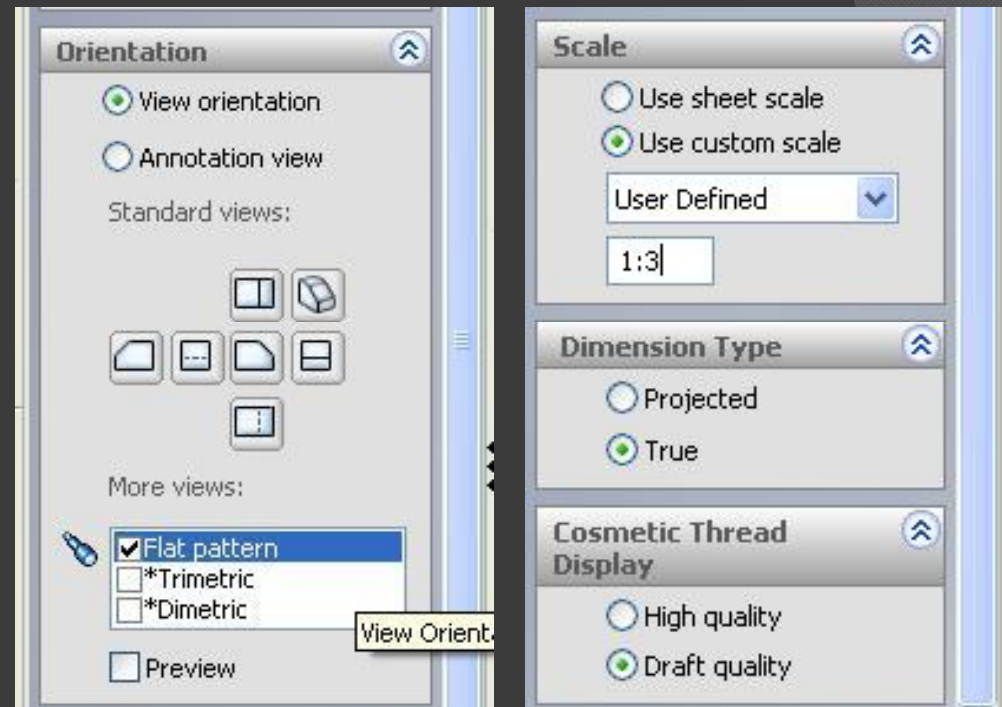
- In the FeatureManager, select the features while holding down the CTRL key and dragging the features into the lower pane of the Design Library



- 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

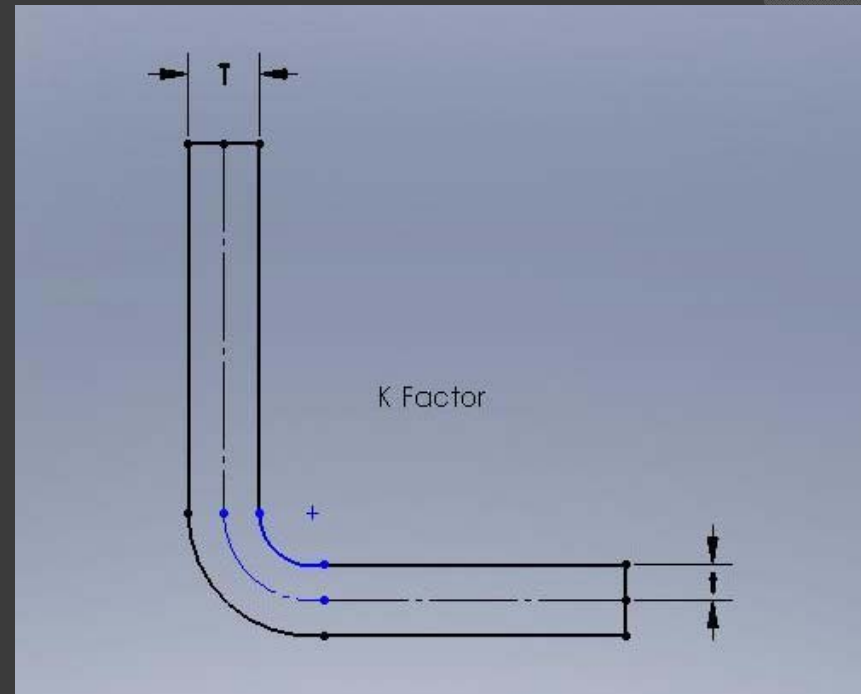
- Insert a flat pattern view
- Modify flat pattern configurations





# 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

- ① Use symmetry when applicable
- ① Edit a flange after its been created
- ① Always link features to the thickness
- ① Long load time
- ① 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?