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 Metal

- Base Flange or Tab
- Edge Flange
- Miter Flange
- Hem
- Sketched Bend
- Closed Corner
- Jog
- Break-Corner/Corners/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
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
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
- 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
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

- $K = \frac{t}{T}$
- % distance of natural line into the material
- Changing $k$ factor changes amount of material in radius
Bending Table

- A bending table in the drawing

<table>
<thead>
<tr>
<th>Angle</th>
<th>Radius / Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.00</td>
</tr>
<tr>
<td>15</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td></td>
</tr>
<tr>
<td>45</td>
<td></td>
</tr>
<tr>
<td>60</td>
<td></td>
</tr>
<tr>
<td>75</td>
<td></td>
</tr>
<tr>
<td>90</td>
<td></td>
</tr>
<tr>
<td>120</td>
<td></td>
</tr>
<tr>
<td>150</td>
<td></td>
</tr>
<tr>
<td>180</td>
<td></td>
</tr>
</tbody>
</table>
Time Saving Tips

- Use symmetry when applicable
- Edit a flange after it's 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?