SolidWorks Part 4

Prof. Steven S. Saliterman
Introductory Medical Device Prototyping
Department of Biomedical Engineering, University of Minnesota
http://saliterman.umn.edu/
Starting The Tutorials

- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials
Select Basic Techniques

Sheet Metal
Creating the Base-Flange

To create a sheet metal part, you sketch an open profile and use the base flange feature to create the flange feature and the bends. When you develop a sheet metal part, you generally design the part in the folded state. This allows you to capture the design intent and the dimensions of the finished part.

1. Open a new part.
2. Click Base Flange (Tab: Sheet Metal toolbar).
3. Select the Front plane.
4. A sketch opens on the Front plane.
5. Sketch and dimension the profile.

To draw the line with its midpoint at the origin, click Line (Sketch toolbar). In the sketch, use PropertyManager, under Options, select Midpoint line. Click on the origin to place the midpoint, then click again to place an endpoint.

6. Click Finish Sketch (Sketch toolbar).
7. In the PropertyManager, under Direction 1:
   a. Select Blind in tab Condition.
   b. Set Depth to 150.
8. Under Sheet Metal Parameters:
   a. Set thickness to 3.
   b. Set Bend Radius to 3.
9. Click
   The sketch is created, and the bends are added.
Making the Extruded-Cut

Adding a Miter Flange

You can add flanges to your sheet metal part with corners that are automatically mitered. First you add a notch to limit the propagation of the miter flange. Then you add and dimension the sketch for the miter flange.

1. Click Extruded Cut (Features toolbar), and select the bottom face.
2. Sketch and dimension a circle with its center coincident to the midpoint of the edge, as shown.
3. Close the sketch.
4. In the PropertyManager, click Link to thickness under Direction 1, and then click.
5. Click Wireframe (View toolbar).
The Extruded Cut

Adding a Miter Flange

You can add flanges to your sheet metal part with corners that are automatically filleted. First you add a notch to limit the propagation of the miter flange. Then you add and dimension the sketch for the miter flange.

1. Click Extruded Cut (Features toolbar), and select the bottom face.
2. Sketch and dimension a circle with its center coincident to the midpoint of the edge, as shown.
3. Close the sketch.
4. In the PropertyManager, click Link to thickness under Direction 1, and then click ✓.
5. Click Wireframe (View toolbar).
Wire-Frame View

Setting Up the Miter Flange

1. Click Miter Flange (Sheet Metal toolbar).
2. Select the inside vertical edge as shown to create a sketch plane normal to the selected edge with its origin at the closest endpoint of the edge. Make sure to select the upper portion of the edge.
3. Click Bottom (Standard Views toolbar).
4. Click Zoom to Area (View toolbar) and zoom to the sketch origin.
5. Starting at the origin, sketch as shown.
   a. Horizontal Line (Sketch toolbar), with an approximate length of 20.
   b. Tangent Arc (Sketch toolbar), with an approximate radius of 10.
Making a Path for the Miter

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Miter

Completing the Miter Flange

Now you create the miter flange.

1. Click Isometric (Standard Views toolbar).
2. Click Exit Sketch (Sketch toolbar).
3. Click Propagate
   The miter flange is propagated to the tangent edges, stopping at the notch.
4. In Miter Parameters, under Flange position, click Material Outside.
5. Click .
6. Click Shaded With Edges (View toolbar).
Propagation of Miter

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Miter Shown with Edges
Mirrored Geometry

1. Click "Back" (Standard Views toolbar).
2. Zoom to the right corner of the model's back face.
3. Click "Mirror" (Features toolbar).
4. Select one of the back planar faces as shown for "Mirror Face/Plane".
5. Click " Bodies to Mirror", and select the sheet metal body as shown for "Solid/Surface bodies to Mirror".
6. Click "OK".
7. Click "Isometric" (Standard Views toolbar).
8. Expand Mirror in the FeatureManager design tree to view new bends for the mirrored geometry.
Creating Edge Flange

1. Drag the edge flange as shown, and click.

2. In the PropertyManager:
   a. Under Flange Length, set Length to 30.
   b. Under Flange Position select Material Outside.
   c. Under Flange Position select Offset.
   d. Under Flange Position set Offset Distance to 11.
   e. Under Flange Parameters, click Edit Flange Profile to display the Profile Sketch dialog box.

3. Select the end points along the base flange (inside edge) and drag them towards the center as shown.

Repositioning the end points removes the relation between the width of the base flange and the width of the edge flange.
Finished Edge Flange
Mirroring a Sheet Metal Feature

You can mirror sheet metal bodies as well as selected sheet metal features. Use the Right plane to mirror the edge flange feature.

The plane used to mirror the sheet metal feature must be centered between the edges of the base flange.

1. Click Mirror (Features toolbar).
2. Expand the FeatureManager design tree, and in the PropertyManager:
   a. Under Mirror Face/Plane, select Right Plane for Mirror Face/Plane.
   b. Under Features to Mirror, select Edge-Flange1 for Features to Mirror.

3. Click .
Mirrored Feature
Adding a Tab

1. Select the face as shown, then click Shape Flange/Tab (Dihedral Metal toolbar).
2. Sketch a rectangle, making one edge of the rectangle coincident to the upper edge of the edge flange.
3. Dimension the rectangle to stick out 20 from the edge flange and be 40 long.
4. Select the Dimension tool.
5. Add a consistent rotation between the midpoint of one of the long lines of the rectangle and the edge flange:
   a. Right-click one of the long lines of the rectangle, and click Select Midpoint.
   b. Click Add Relation (Dimensions/Relations toolbar).
   c. Right-click the long edge of the mirrored edge flange, and click Select Mipoint.
6. In the PropertyManager, under Add Relations, click Horizontal, then click.
Finished Tab
Bending a Tab with Sketched Bend

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Unfolding a Bend

Adding a Cut Across a Bend

To cut across a bend, you start by unfolding only the bend that you want to cut across. Unfolding only one bend results in faster system performance.

1. Click Unfold [Sheet Metal toolbar].
2. In the graphics area, select the face and bend as shown for Fixed face and Bends to unfold.
   You can only select bends when the Bends to unfold box is active.
3. Click to unfold the selected bend.
Adding a Collinear Relationship
Extruded Cut Through All
Finished Cut
Fold – Collect All Bends
Creating an Edge Flange

1. Click Edge Flange (Sheet Metal toolbar).
2. Select the edge as shown.
3. In the PropertyManager:
   a. Under Angle, set Flange Angle to 15.
   b. Under Flange Length, cut Length End Condition to filled.
   c. Under Flange Length, if the face is down, click Reverse Direction.
   d. Under Flange Length, set Length to 0.5.
   e. Under Flange Projection, select Mainlet Direction.
   f. Under Custom Relief Type, in Relief Type, select Rectangle.
   g. Under Custom Relief Type, set one relief ratio.
   h. Under Custom Relief Type, set Radius to 0.5.

4. Click.
Closing the Corner with Butt Joint

1. Click **Closed Corner** (Sheet Metal toolbar).
2. Select the face of the base flange as shown for **Faces to Extend**.
3. Select **Butt** for **Corner type**.
4. Click **OK**.
Flatten Sheet

Flattening and Folding the Part

You can flatten all the bands of a sheet metal part at once.

1. Verify that the rollback bar is rolled to the end, then click Flatten (Sheet Metal toolbar).
   This is the same as unsuppressing the Flat Pattern feature that was created with the Base Flange feature.
   The flattened sheet metal part appears with all of the bend lines shown. You may also see a bounding box around the part. The bounding box is the smallest rectangle in which the flat pattern can fit.

2. To fold the part back up, click Flatten again.

3. Save the part.
Drawing from Part–Assembly, Scale 1–3
Model View

Next, add a view of the unfolded model. A flat pattern view is automatically added in the PropertyManager when you create a sheet metal part.

1. Click Model View (Drawing toolbar).
2. In the PropertyManager:
   a. Click 
   b. Under Orientation, in More views, select (A) Flat pattern.
   c. Under Scale, select Use custom scale. Select User Defined in the list and type 1.18.
   d. Click to place the view in the drawing sheet.

The view orientation of flat patterns in drawings varies, depending on such factors as the way you extruded the base shape. To rotate the view, select the drawing view, click Rotate View (View toolbar), and make your choice in the dialog box. You may also need to use Flip view in the PropertyManager.

3. Click .
4. Save the drawing as Sheet Meta.

Click Save All to save both the drawing and the updated model if a message box notifies you that the model referenced in the drawing was modified.
Adjusting Notes Font
Adding Leaders and Rotating Notes

2. Right-click and select ridge.
3. Flared steps 1 and 2 with the outer flange bend line index at the top, as shown.
4. Select UP 90.00° R1 from the outer flange located at the top and drag outside the part’s shape to the M1.
5. In the PropertiesManager, under Leaders, click Leader R1.

   You can change the angle of a bend note, the leader type, and the arrow style.

   Repeat steps 4 and 5 with the three remaining inner flange notes.
7. Click to close the PropertiesManager.
8. Draw and select the five horizontal extensions of UP 90.00° R1 and DOWN 90.00° R1 along the center of this drawing.
9. In the PropertiesManager, under Text Format, set Angle 2 to 5.
10. Click to allow the bend notes in the drawing view approximately resembles the image below.

11. Save the drawing.
Summary

Concepts reviewed:
- Creating a base-flange.
- Creating a path for a miter.
- Mirroring geometry.
- Creating an edge flange.
- Mirroring a feature.
- Adding a tab.
- Bending with a sketched bend.
- Unfolding a to make a cut, refolding.
- Flattening sheet.
- Making a drawing and modifying the notes with leaders.