**Sheet metal tutorial**

In the following tutorial you will cover the basic features of the Solid Works sheet metal tool by modelling the component shown opposite.

**Activating Sheet metal mode**

Sheet metal components are modelled in the part environment by activating the Sheet metal tool bar.

To do so right click in a blank section of the Solid Works boundary and select the Sheet Metal toolbar.

This bring up the floating sheet metal toolbar shown opposite, The sheet metal options may also be activated in the command manager.

**Creating the first feature**

The first feature to be created will be a flat rectangular section 100 x 60mm. Working on the top plane create this rectangle shown. On completion select the base flange / tab icon the create this first Tab feature.

A flat portion is generally referred to as a tab feature.

At this point we will be given the opportunity to specify a number of parameters. The thickness parameter will determine the thickness for the entire component.

Specify a thickness of **2mm**.

To set the bend radius Right click on the first sheet metal icon in the command manager and specify a bend radius or 1mm.
As the material is being folded, metal on the outside stretches while material on the inside compresses. If the material stretched and compressed by similar amounts the neutral axis would be in the middle $K = 0.5$. However for practical reasons such as friction against the tool etc., the position of the Neutral axis can be effected. In this case specify a $K$ factor of 0.35.

Adding a second tab
Next we will add another flat portion as a continuation of the existing part. Again this is referred to as a tab feature.

On completion of the sketch. Select the tab icon.

The icon does not convey very well the concept of a tab. A tab might be better represented by this icon.

The feature once created is given the following symbol in the feature manager.

Adding a flange
Next we will add a flange. A flange is material which is added at some angle to the exiting material, generally 90 degrees but any angle may be used.

To add a flange select the flange tool, then select the edge to which the flange is to be added. (Bottom edge shown) Five options are available to determine the position of the flange relative to the selected edge. The first three are the most important.
1. Flange inside. Outside face of vertical portion is in line with selected edge

2. Flange outside. Inside face of flange is in line with edge. (bend partially inside)

3. Both Bend and flange outside.

Use this option in this case.

A number of options are available regarding how to specify flange length.

The flange length may be from the “nearside” face or the “farside” face.

In this case we will specify **20mm** from the nearest face so that flange will be 20mm from the underside of the existing plate.

**Adding the side Flange**

Next we will add the side flange as shown. This time we will keep all material inside the existing edge using the material inside option.

We will make this flange the same length as the previous flange so rather than specifying the length a second time, set the length by using the up to vertex option and select a corner at the bottom of the existing flange.

On closer examination you will see tearing of the corner.

This may be eliminated by selecting

> Trim side bends
**Cutting across a fold**

We will now create a cut which straddles a bend. This hole will be cut while still flat, and while sizes of the resulting hole while flat will be known. The exact size following folding is not known due to complexities of bend behaviour.

To create the correct size cut-out, the cut will be applied to the flattened shape. To create the cut you will:

1. Un-bend the component
2. Create the cut
3. Re-bend the component

To unbend or unfold select the unfold icon.

For the face to fix select Face **A** above. And for the bend to straighten select Bend **B** above. (The user may select any number of bend or all bends if required)

Once unfolded the component will look as shown.

Next create the rectangular sketch shown opposite and create an extruded cut.

Apply **5mm** fillets to each corner.

The finished cut will look as shown.

Once complete you are now ready to re-bend the component. To do so select the re-bend icon.

For the Face to fix and bends to fold, use the same selections as before.
**Adding the back Flange**
Next create the back flange, giving it a height of 30mm. using the material outside option.

Next create a side flange.

**Adding the rear side Flange**
While the flange is initially the full length of the edge, we will modify the flanged to reduce its size. This will be achieved by selecting “Edit Flange Profile”

When in Edit mode you are free to drag the end points of the sketch and to add dimensions. When finished select Finish in the dialog box shown below.

Specify the material inside option and add fillets of appropriate size to round the end and add a 6mm hole to the flange.

On closer examination we can see that relief is provided in the form of a rectangular recess.

We will modify this to a rounded or obround recess. To do this right click on the flange in the feature manager and modify the flange properties.

Activate custom relief type and choose Obround the choose Accept. The resulting relief will now look as shown opposite.
**Adding the rear side Flange**

To make the second rear flange we will mirror image the one just created.

To do this we will create a plane parallel to front plane passing through the midpoint on the back flange.

Once created use the mirror command to mirror the rear flange and its associated hole and fillets. Ensure that all features are mirrored in a single operation.

The result geometry should look as shown.

**Creating an angled flange**

In this step you will create an angled flange.

Working on the corner shown, create a regular flange pointing downward by **16mm**. Modify its profile so that it extends just **25mm** from the corner. Use the material outside option.

Finally modify the flange angle so that the flange points downwards at an angle of **70 degrees**.

On the adjacent edge create another flange **30mm** long this time pointing straight downwards by **15mm**. Where gaps exist sometimes it is necessary to close this gaps. This may be necessary to accommodate welding to product water tight container.

This is achieved using the close corners tool.
**Closing corners**

A number of options are available. These determine whether the faces meet edge to edge or whether one overlaps the other.

To close the corner select the relevant faces of both edges and then apply to appropriate corner condition so that the corner looks as shown opposite.

**Adding a Jog feature**

A jog features is one which allow additional material to be added to add a step to a sheet metal feature without effecting its lateral position.

This feature represents a fixing bracket and while the lateral position of the hole is correct the screw is too short to reach to hole in its current position. This can be rectified by adding a Jog feature.

To add a jog, draw a line at the position shown 10mm from the base feature.

As before the vertical portion of the jog may be inside or inside of this line.

Also the dimension may be one of the following.
- Distance between equivalent surfaces
- Distance between nearest surfaces
- Distance between furthest surfaces.

Specify a distance downward of 10mm between equivalent faces.
**Bending existing features**

Working on the top face of the original tab draw the sketch shown opposite.

On completing create a cut-out. Into this recess add the following sketch.

Create a tab feature and add a 6mm hole as shown.

In this case we will apply a bend to an existing feature. This is done using a sketch bend.

To apply a sketch bend draw a line in the position shown. Next choose the sketch bend icon.

For the first selection select that portion of the material which will remain stationary. (Represented by black dot below).

The same options of material inside/material outside apply. We will use bend outside to ensure that all material deformation is to the right of this line.

Reproduce this feature on the other side by whatever means is most convenient.
Drafting a Sheet metal part.

To create a drawing of a sheet metal part create orthographic and pictorial views in the usual way.

Create a plan, elevation, end view and isometric. Change tangent edges where appropriate.

Showing the sheet metal component in the flattened state.

To create a drawing of a sheet metal part create orthographic and pictorial views in the usual way.

Create a plan, elevation, end view and isometric. Change tangent edges where appropriate.