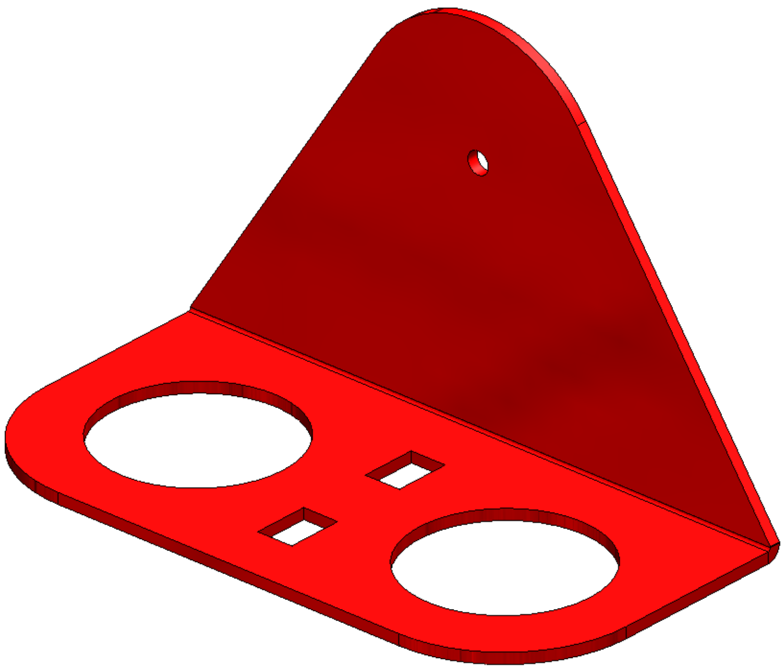
Toothbrush Holder



Prerequisite Knowledge

Previous knowledge of the following commands is required to complete this lesson; **Sketch (**Line, Centerline, Circle, Add Relations, Smart Dimension,), **Extrude Boss/Base,** and **Edit Materials.** A basic knowledge of the drawing environment is also required

Focus of the Lesson

This lesson focuses on designing a sheet metal part from the flattened state. In this case, you create a sheet metal part and then insert bend lines on which to fold the part.

Commands Used

This lesson includes **Sketch** (Line/Centerline, Circle, Mirror Entities, Add Relations, and Smart Dimension), ***Base Flange, Extruded Cut, Sketched Bend*** and **Edit Material.**

A drawing of the sheet metal part will also be created.

New File

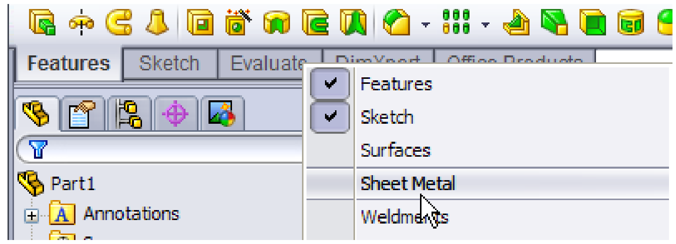
Create a new part file.

Save File

Save the file as '**Toothbrush Holder'** to a folder called **'Holder exercise'**
(**Continue to save periodically throughout the exercise**)

Getting Started In order to begin working with **Sheet Metal** you must first activate the sheet metal tab on the command manager.

To activate this tab, right click on the command manager. Choose **Sheet Metal** from the drop-down list.



The Sheet Metal tab is now active on the command manager.

Note: *The Sheet Metal commands are also available from the drop down menu by selecting "**Insert"** and "**Sheet Metal"…***

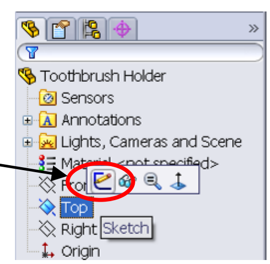
Creating a sketch: How do we start to model the toothbrush holder as a sheet metal part?

We begin by creating a sketch to generate the rectangular piece of acrylic required to manufacture the artefact.

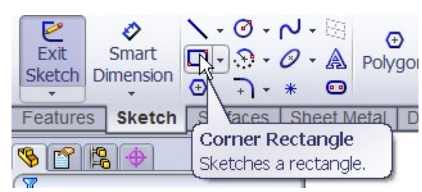
What plane will this sketch be created on?

Because the material sits on the horizontal plane while we carry out the work, we will create a sketch on the Top Plane.

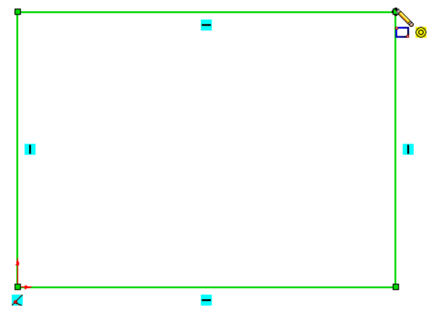
Create a rectangular sketch on the Top Plane. Left click On the 'Top' plane and click on the sketch icon



From the Sketch toolbar, select the Corner Rectangle.

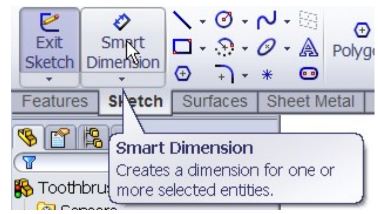


Left click on the Origin, move the cursor diagonally and left click on the opposite vertex to create the rectangle.



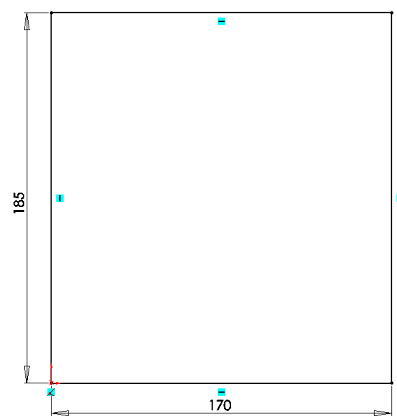
(Press 'Esc' to exit the Corner Rectangle command)

Note the automatic relations that are added to the sketch. If these are not shown, go View/Sketch Relations on the dropdown menu.



Select Smart Dimension from the Sketch toolbar and dimension the rectangle as shown.

Remember always to dimension from the shortest to the longest distances.



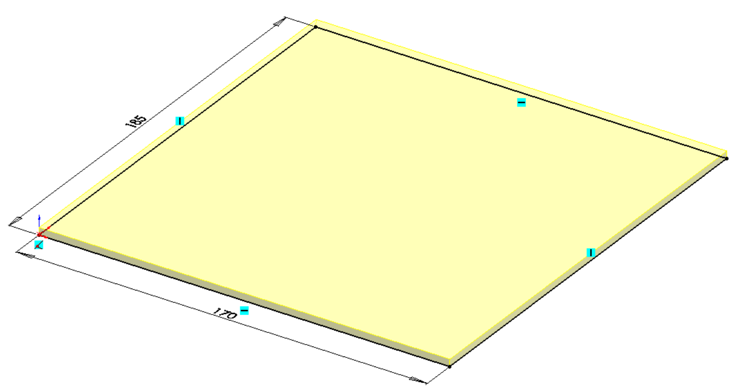
The sketch lines turn black when **fully defined.**

Exit the sketch 

Sheet Metal Feature:

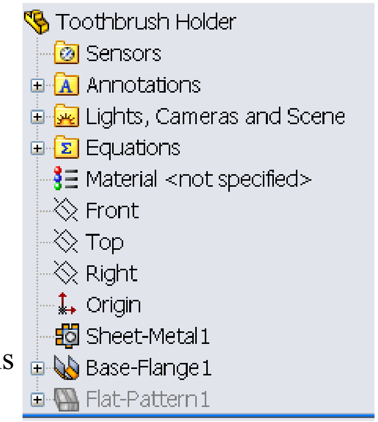
To create a sheet metal feature, click the **Sheet Metal** tab on the Command Manager and choose **Base Flange**

Enter a value of 3mm for **thickness** in the Base Flange options dialog box



Click Ok 

About Base Flange A base flange is the first feature in a new sheet metal part. When you add a base flange feature to a SolidWorks part, the part is marked as a sheet metal part. Bends are added wherever appropriate, and sheet metal specific features are added to the FeatureManager design tree.



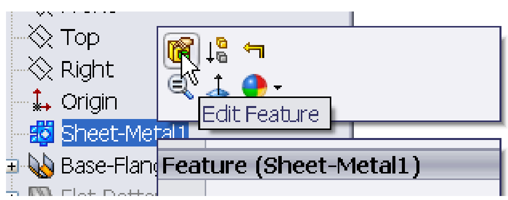
The Base-Flange feature is created from a sketch. The sketch can be a single open, single closed or multiple-enclosed profiles. There can be only one base flange feature in a SolidWorks part.
The thickness and bend radius of the Base-Flange feature become the default values for the other sheet metal features.

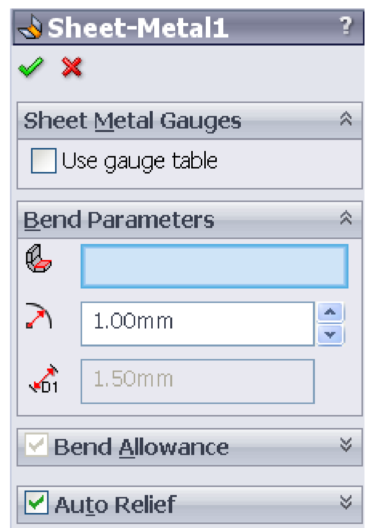
When a base flange feature is created a number of items are added to the feature manager design tree.

Sheet-Metal1: is automatically added above the Base flange feature. It holds the default sheet metal settings such as sheet metal thickness, radius etc.

Sheet-Metal1 will remain at the top of the feature manager design tree (under 'Origin')

Sheet-Metal 1 Right click on **Sheet-Metal 1** and choose **Edit Feature**





The sheet metal settings may be changed here.

Set the bend radius to 1mm in the Bend Parameters
Choose **OK**

Flat-Pattern Feature This is added below the base flange feature. It has a couple of special properties that are not found with other features.

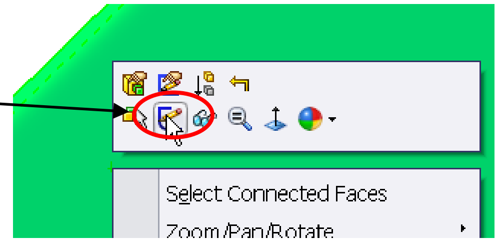
Unlike other features, flat-pattern will remain at the bottom of the tree. Other sheet metal features, when added, will appear overhead even though they are added after its creation. Secondly, the feature is suppressed when added to the design tree.

We will look further at this feature as we work through this exercise.

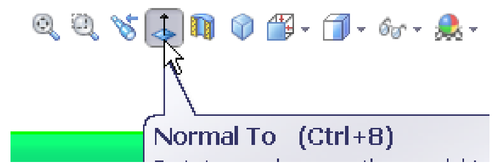
Creating the Rectangular holes

A sketch needs to be created on the top face of the Base Flange so that the rectangular holes can be formed.

Right click on the top face and select the sketch icon



Select ‘Normal To’ from the Heads-Up Toolbar



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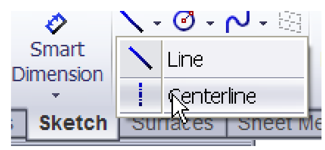
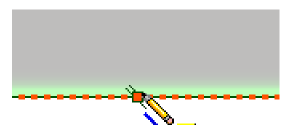
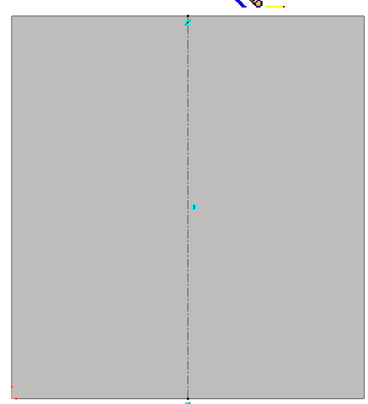
Select the ‘Centerline’ command from the Sketch Toolbar (use the down arrow beside Line command)



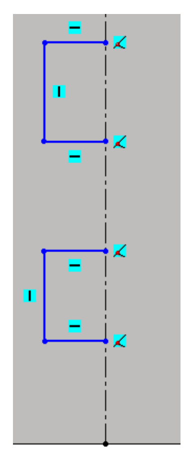
Hover over the edge of the base flange and the midpoint will appear



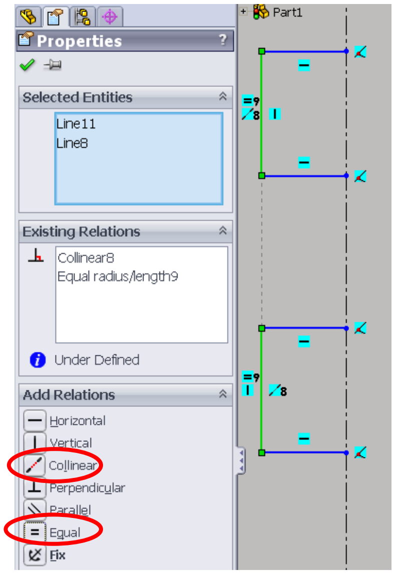
Sketch the vertical centerline



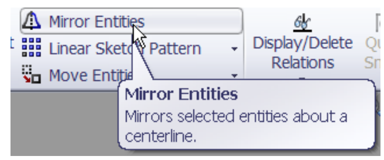
Using the Line Command, sketch the lines shown opposite



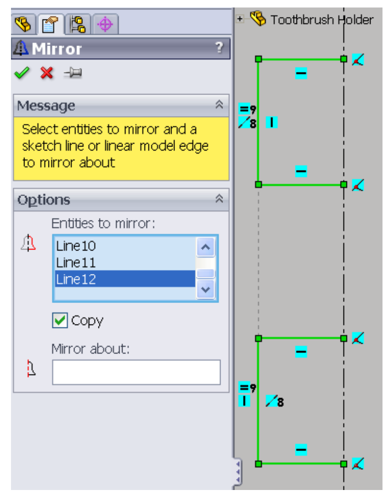
Select the two vertical lines by holding down the ‘Ctrl’ key. Add a ‘Collinear’ and ‘Equal’ relation between the two lines.



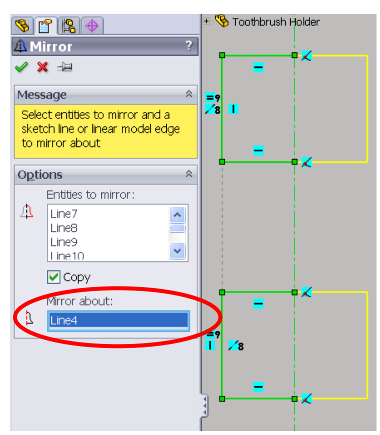
Select ‘Mirror Entities’ from the sketch toolbar



Select the 6 lines at the ‘Entities to mirror’

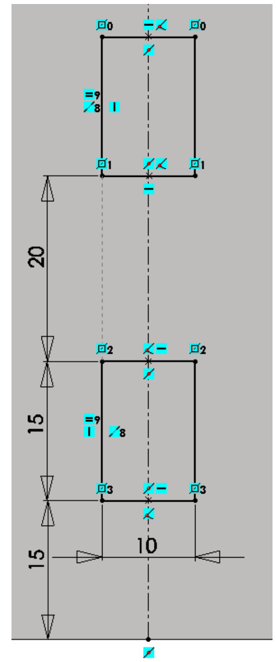


Click on the ‘Mirror about’ dialogue box and select the centerline to mirror the sketch entities and select OK.

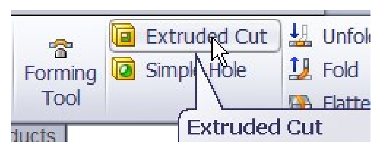


Dimension the sketch as shown below.
Note that the sketch is fully defined.

Exit the sketch.



Extruded Cut Select Extruded Cut from the Sheet Metal toolbar



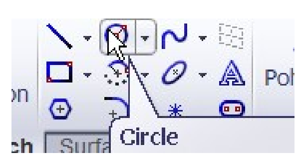
Select the sketch containing the rectangles and select 'Through All' as the end condition.

Select 

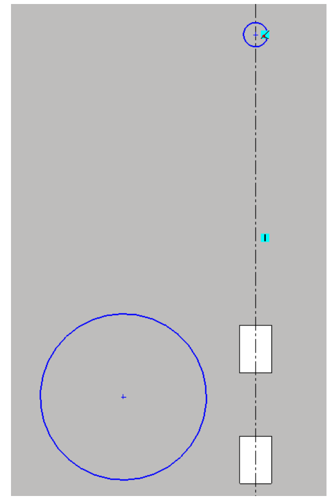
Rename Feature Double click on the 'Extrude 1' feature and rename as 'Rectangular holes'

Circular Holes Create a sketch on the top face of the base flange. Draw a vertical centerline as described earlier.

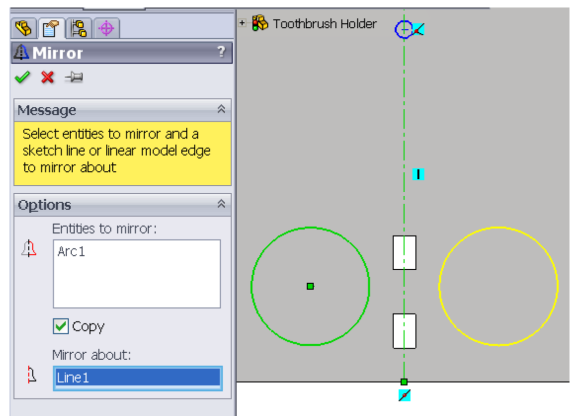
Select the 'Circle' command from the sketch toolbar



Create two circles, one of which is coincident with the centerline

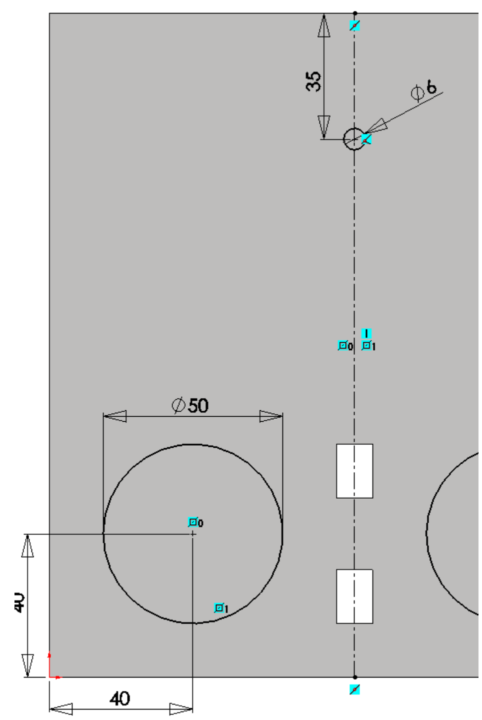


Use ‘Mirror Entities’ to create a circle on the right of the centerline



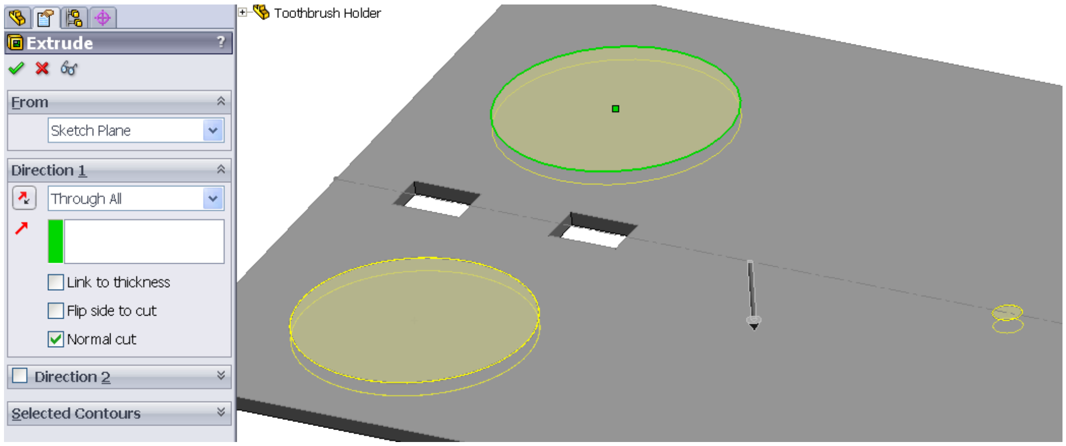
Smart dimension as shown opposite

Exit the sketch



Extruded Cut

Select Extruded Cut from the Sheet Metal toolbar. Select the previous sketch in the graphics area and choose ‘Through All’ as the end condition.



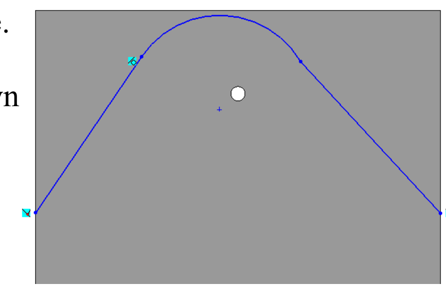
Rename feature

Rename this feature as ‘Circular holes’

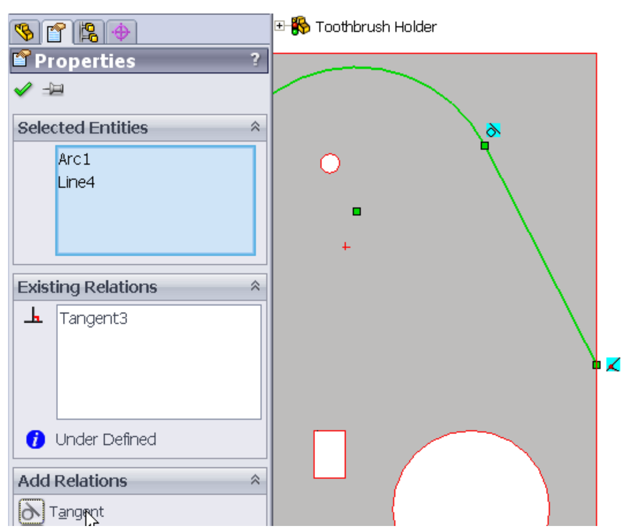
Shaping

Create a sketch on the top face of the base flange.

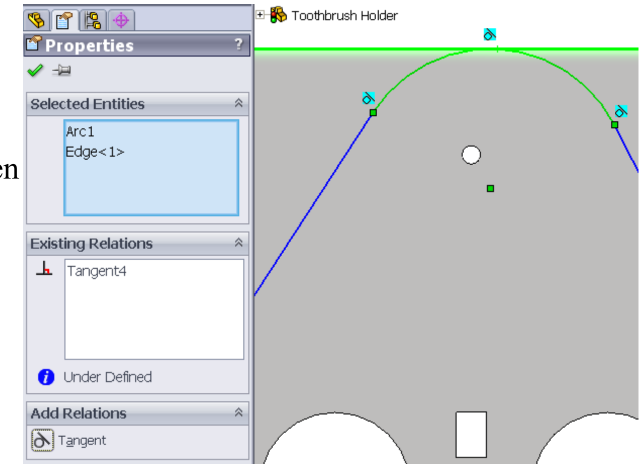
Using the Line command, create the sketch shown opposite



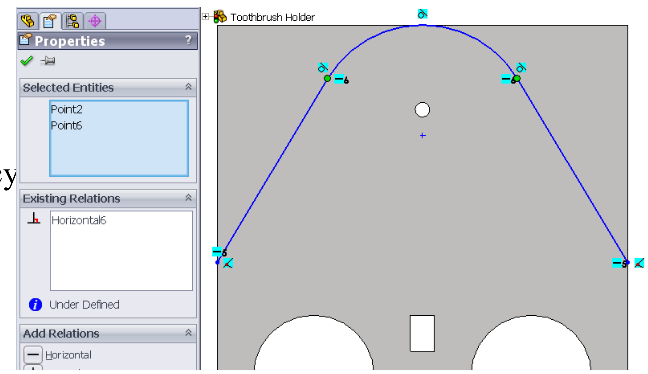
Select the line segment and arc by holding down the ‘CTRL’ key and apply a ‘Tangent’ relation



Apply a 'Tangent' relation between the arc and edge of base flange

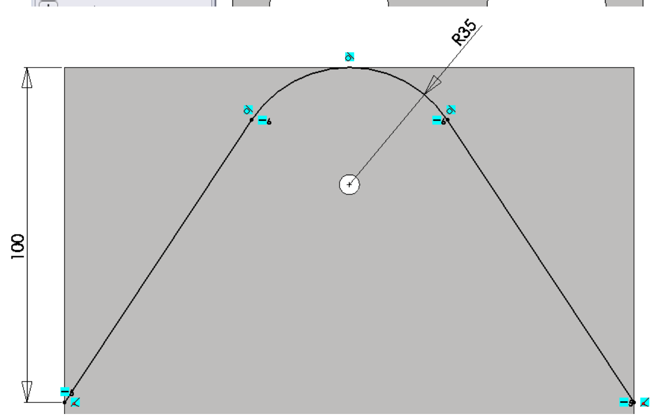


Apply a 'Horizontal' relation between the endpoints of the line segments and the points of tangency in turn.



Dimension the sketch as shown opposite.

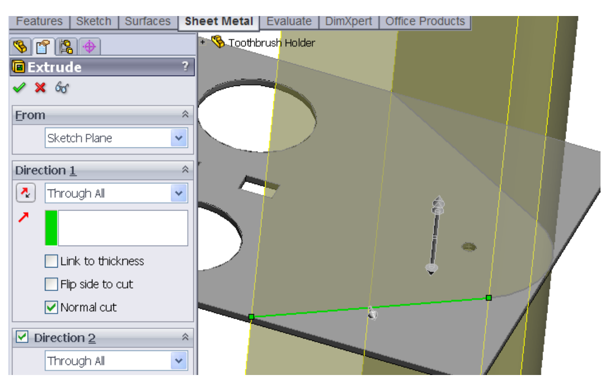
Exit the sketch.



Extruded Cut

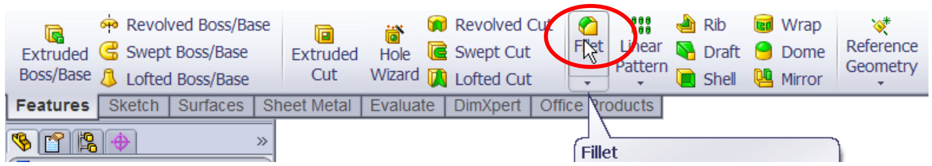
Select Extruded Cut from the Sheet Metal toolbar. Select the previous sketch in the graphics area and choose 'Through All' as the end condition.

Rename as 'Shaping'

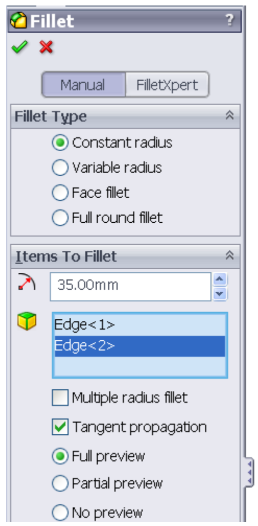


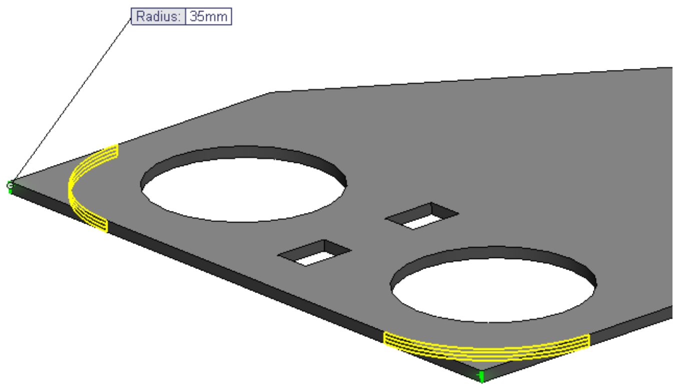
Filets

Select the 'Fillet' from the **'Features'** toolbar



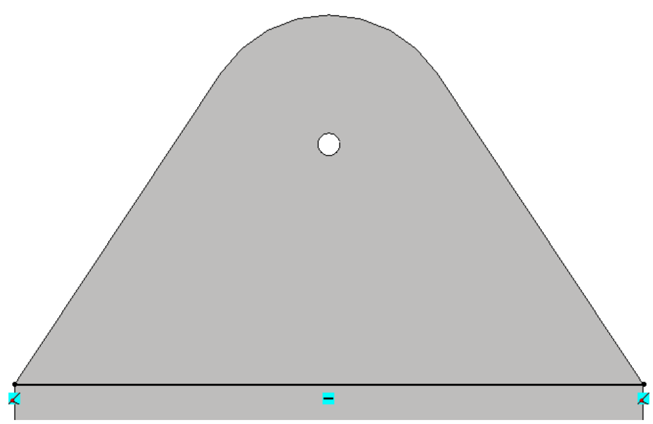
Choose the settings in the property manager as indicated below



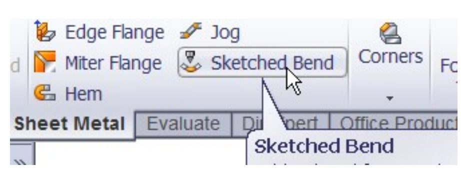


Sketched Bend

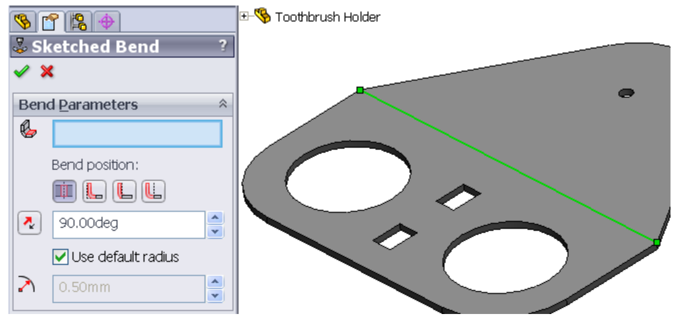
Create a sketch on the top face of the base flange.
Using the line command, sketch a line coincident with the endpoints of the shaping.
This line will be used as the b



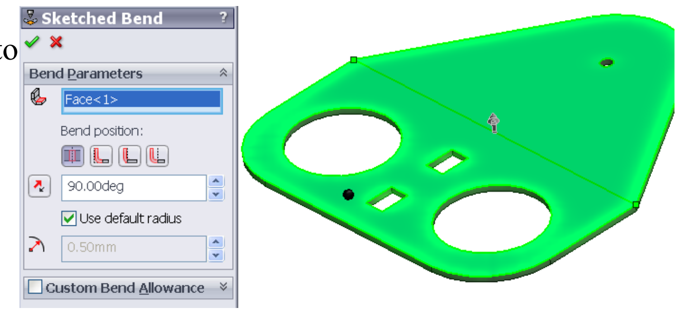
Select 'Sketched Bend' from the Sheet Metal Toolbar.

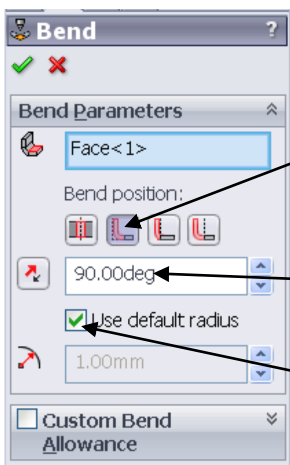


Select the bending line



Select the face that you wish to remain horizontal after the bending process





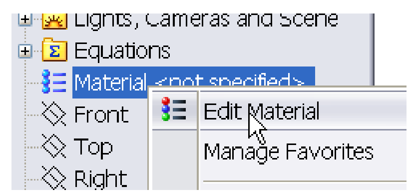
Select 'Material Inside' as the Bend Position

Select 90° as the bending angle

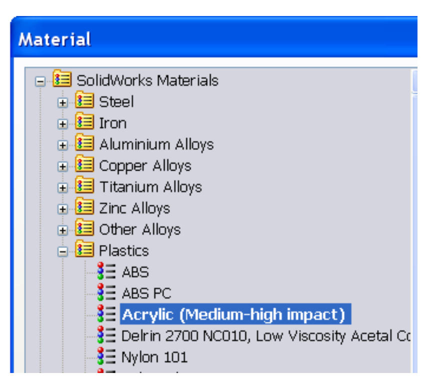
Choose the default radius as the bending radius

Edit Material

Right click on 'Materials <not specified> in the Design Tree and select 'Edit Material'

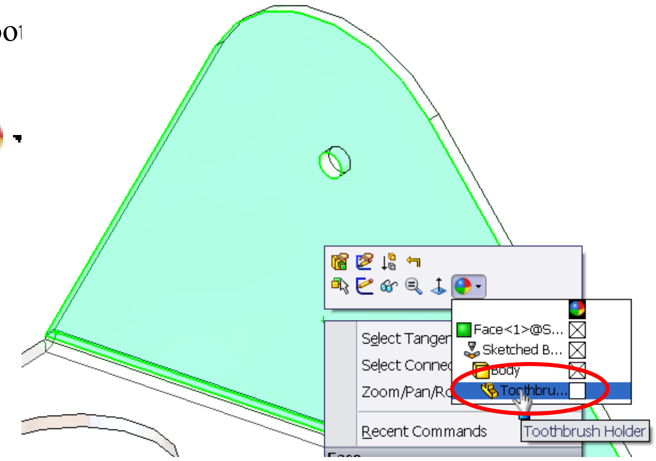


Scroll down to the 'Plastics' folder and select 'Acrylic (Medium-high impact' and choose 'Apply' and 'Close'



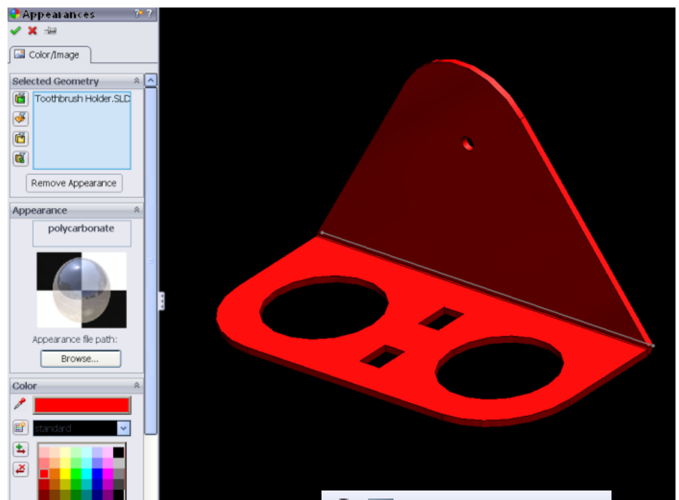
Apply Colour

Right click on any face of the tool



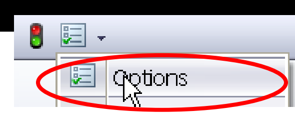
Select the Appearance icon and select the 'Toothbrush' part

Select a colour from the swatch In the Appearances Property Manager

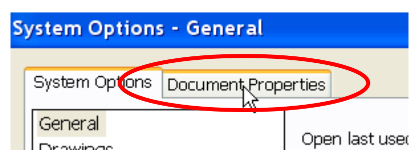


Improve Image Quality

Go to Tools/Options on the standard toolbar

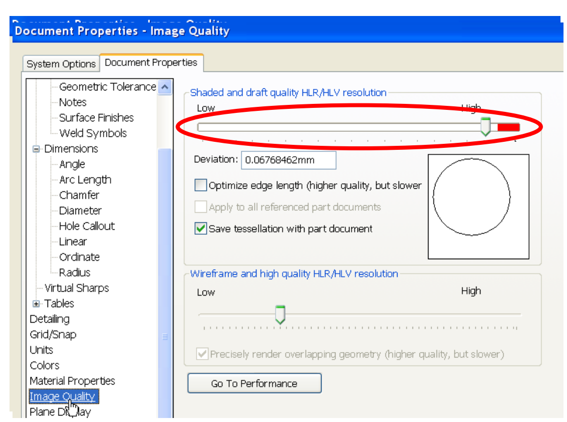


Select 'Document Properties'



Select 'Image Quality' and move the slider to the right.

Select Ok.



Flat-pattern

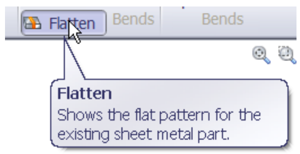
It is added to the bottom of the feature manager design tree when we create a sheet metal part. As sheet metal features are added to the part it remains at the bottom. You will also notice that it is greyed out or **suppressed.**

Unsuppress Flat-pattern

Right click on the feature and choose **Unsuppress** from the pop-up toolbar

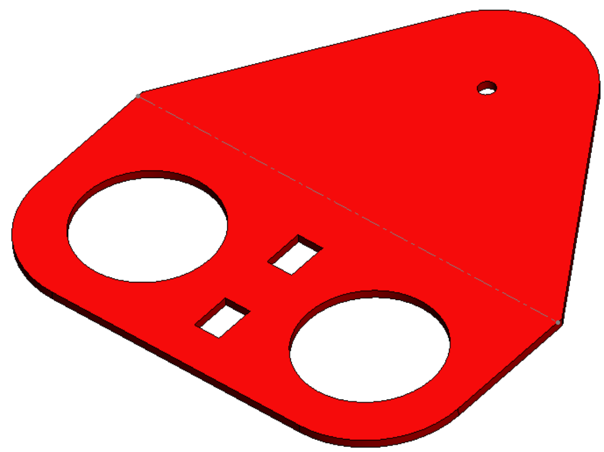
Or

Select 'Flatten' from the sheet metal toolbar



The sheet metal model flattens out into the surface development used to create it.

The bend line is also displayed.



Suppress

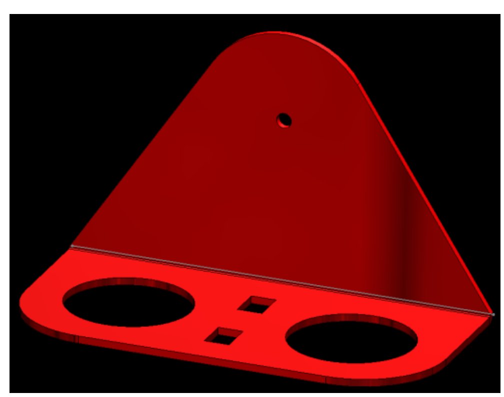
Left click on **Flat-Pattern1** and choose **Suppress** to return to the unflattened state

Or

Click on 'Flatten' in the sheet metal toolbar

Save

Save the 'Toothbrush Holder' part file.



Creating a drawings of the Toothbrush Holder

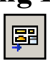
Drawings

SolidWorks enables drawings to be created from parts or assemblies. These drawings are **fully associative** with the parts and assemblies they reference. If the model is changed the drawing will update and vice-versa. A drawing is created using a pre-prepared **template**.

Drawing Templates

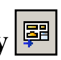
These templates are used when creating presentation drawings. Parameters include sheet size, orientation etc.. The template may include a border, title block projection symbol, and text. When a presentation drawing is to be created using a part model, the template is the starting point. You should refer to the 'Creating Drawings' CAD notes for further information on creating, saving and editing templates

Creating Drawing



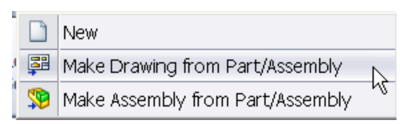
Make Drawing from Part/Assembly takes the current part and steps through the creation of a drawing file and initial drawing views using this part.

Where to find it?

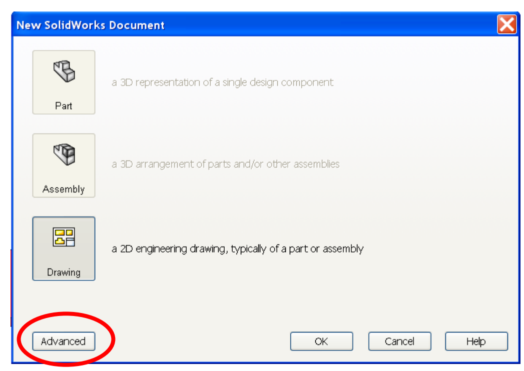
Select **Make Drawing from Part/Assembly**  on the Standard Toolbar *or* choose **File, Make Drawing from Part/Assembly.**

Getting Started

With the 'Toothbrush Holder' part file open,
Select **Make Drawing from Part/Assembly.**



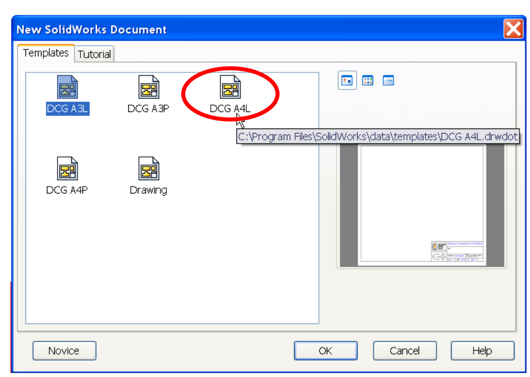
Select **Drawing** and then click **Advanced**



Choosing a Drawing Template

Choose the drawing template you wish to use from the list displayed – **DCGA4L**

Choose **OK**

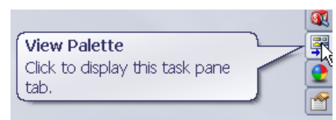


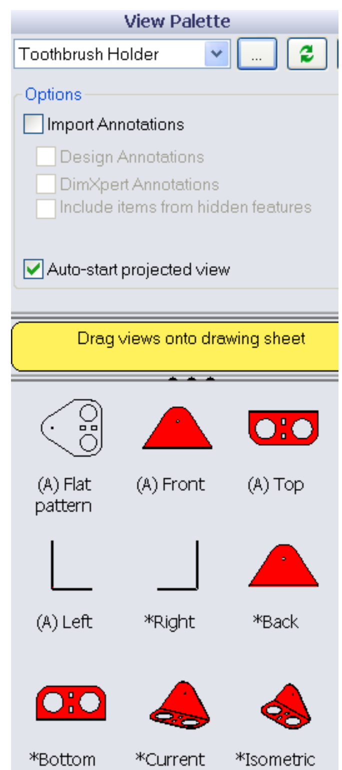
The DCG Templates will only be displayed here if they have been saved following the instructions outlined in the 'Creating Drawings' CAD resource exercise

Drawing Template This template creates an A4 landscape drawing. The sheet format includes a title block, projection symbol, T4 Logo and text.

Note: For detailed instructions on editing the sheet format refer to ***'Customising drawing templates'*** *in the appendix to* ***'Creating Drawings'*** *document*

View Palette The view palette is displayed in the task pane. If this is not shown, click on the icon on the task pane





The file which we are creating the drawing from, **Toothbrush Holder**, is displayed on top.

Ensure **Auto-start projected view** is selected. This will project other views automatically from the parent view.

The standard views are displayed in the view palette as well as any saved views from the part file.

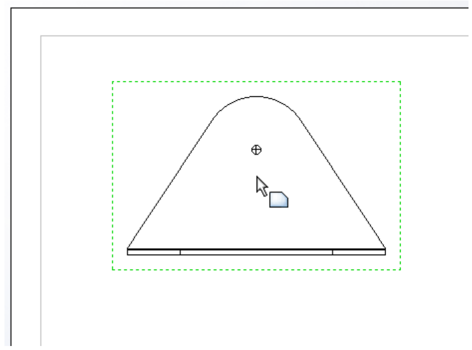
Because this is a sheet metal part, you will notice that a 'Flat pattern' view of the model is also given.

Select and drag the 'Front' view from the view palette into the drawing area.

Position in the top left hand corner, as shown.

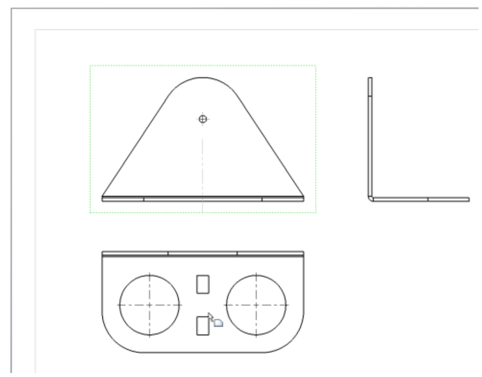
This view is known as the **Parent View**.

The other orthographic views will be projected from this view.



Projected Views

Drag the cursor to the right to project an end view; left click to position the view. Repeat the procedure to project a plan view.

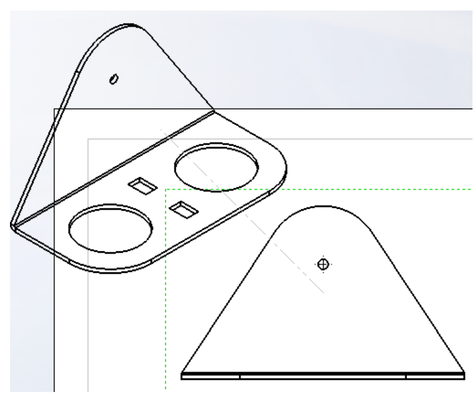


Projecting an Isometric View

A number of isometric views may be projected from this view also.

To project an isometric view, drag the cursor to any of the 4 corners of the front view. A different isometric view of the model will be displayed in each position.

Press 'Esc' to quit projected view

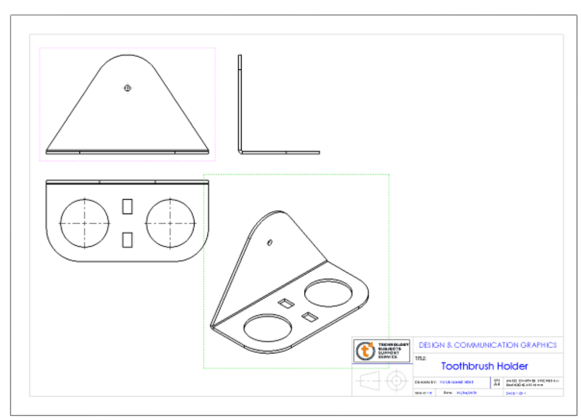


Positioning the View

When the cursor is dragged to position the isometric view on the sheet the projected view changes.

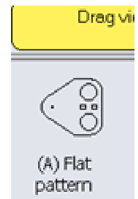
This may be overcome by holding down the **ctrl** key while dragging, thus allowing the current view to be positioned correctly.

Drag the view to a position overhead the title block, left click to position. Choose **OK** in the PropertyManager.



Insert Flat Pattern View

Select and drag the 'Flat pattern' view from the view palette into the drawing area

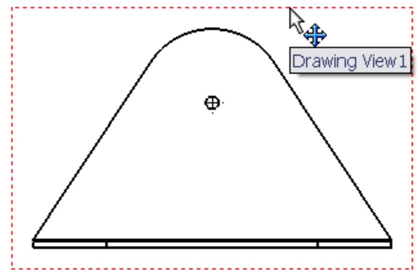


Repositioning Views

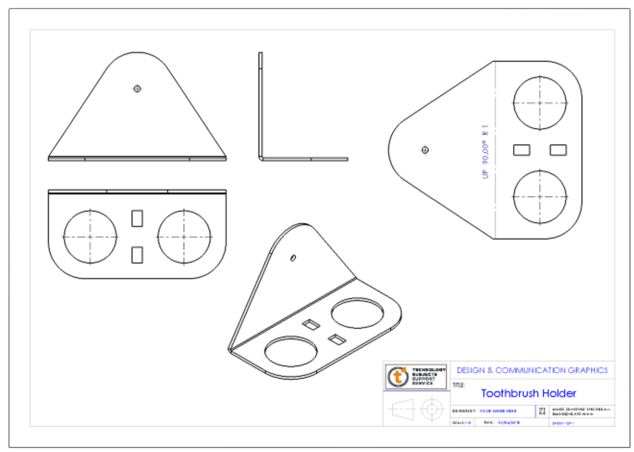
The orthographic views may be moved around the sheet but will maintain alignment.

To reposition a view, move the cursor over the view. A red dotted line will surround the view.

Position the cursor on the red dotted line, hold down the left mouse button and drag.



Reposition the views on the sheet



Display Style

Individual views may be displayed in a number of ways;

- **Wireframe** – Displays all edges.
- **Hidden lines visible** – Displays all edges, hidden lines are visible
- **Hidden lines removed** – Displays edges that are visible at the chosen angle; obscured lines are removed.
- **Shaded with edges** – Displays items in shaded mode with hidden lines removed.
- **Shaded** – Displays items in shaded mode.

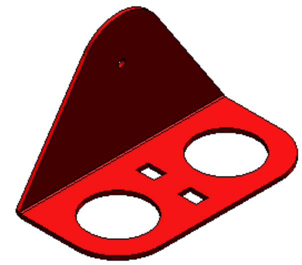
Edit display style

Highlight the **isometric view** by left clicking on the view.

All of the properties of that view appear in the PropertyManager.

Choose **Shaded with Edges** from **Display Style** and accept

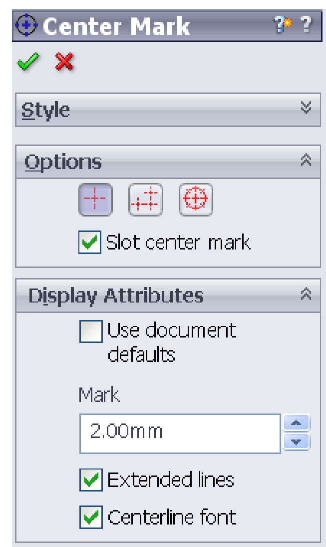
The appearance that is on the part file now is visible on the drawing view

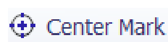


Adding Centre Marks

Centre Marks may need to be placed on circles or arcs in drawings. These may then be used as a reference for dimensioning.

Centre Marks must be added to all the circles in the various views



Choose **Center Mark** 

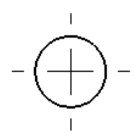
the **Annotation Toolbar.**

Deselect **Use document defaults**

Input a **Mark Size** of **2mm**

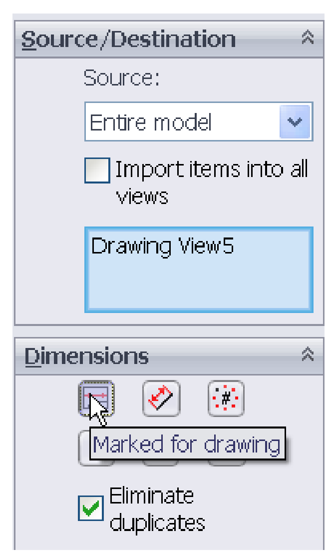
Select the circles representing the two holes in the various views

A centre mark will be added as shown

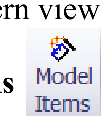


Adding Dimensions

While dimensions can be added to all the views, they will only be added to the flat pattern view in this case



Select the flat pattern view (a green box appears around it)

Select **Model Items** 

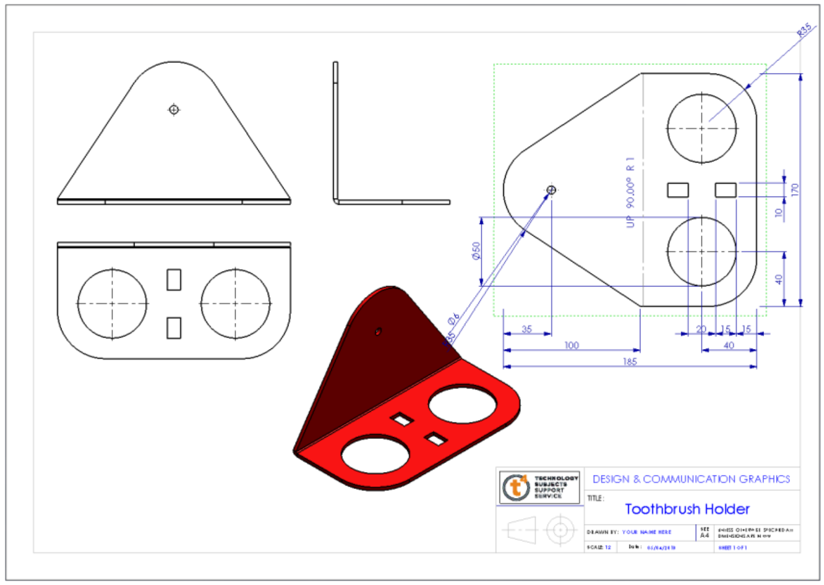
from the **Annotation** toolbar

Choose the **Entire model** as the source from which to import the dimensions.

Choose **Marked for drawing** dimensions from the model.

Choose **OK**

The position of the dimensions may need to be edited

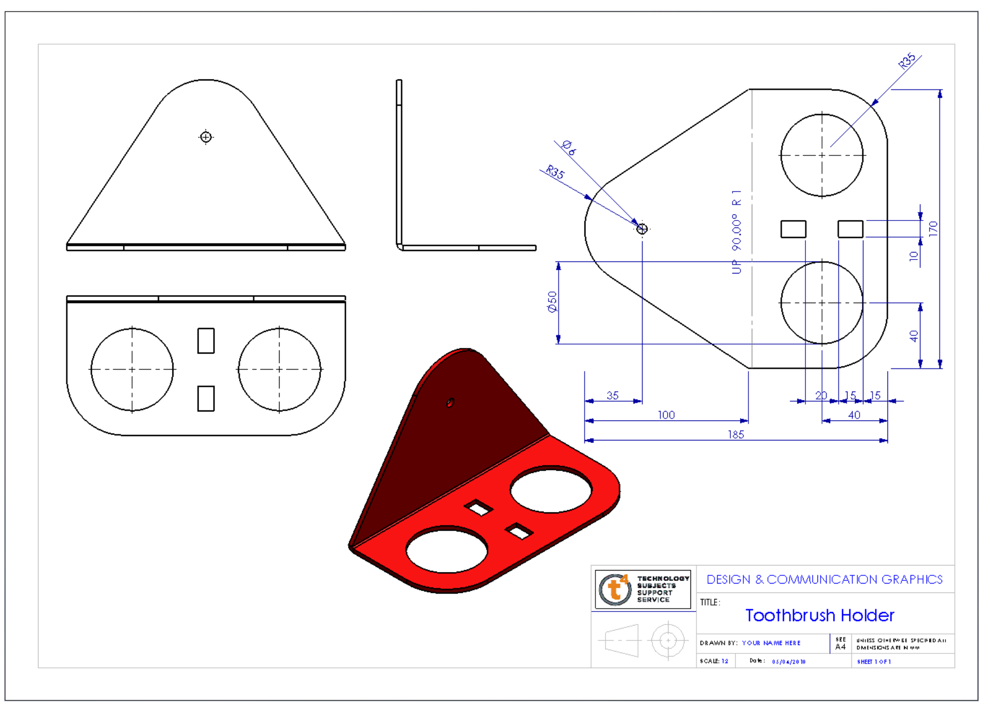


Editing Dimensions

Deleting dimensions To delete a dimension highlight it and press **delete** on the keyboard.

Move dimensions Hold down the left hand mouse button on the dimension and drag. Inference lines will appear to ease alignment of dimensions.

Edit the dimensions by either moving or deleting them, to match the drawing.



Manual Dimensioning

All of the necessary dimensions may not be present on the model. When this occurs the dimension must be inserted manually using **Smart Dimension**. This type of dimension is called a **driven dimension** because its value is driven by the model. *Driven dimensions are shown in a different colour.*

Smart Dimension

Select **Smart Dimension** from the **Annotation** toolbar.

Using the same technique as in dimensioning sketches, select the two edges defining the distance to be dimensioned. Drag and place the dimension.

Ensure that the zoom features are used to make placement and alignment of dimensions easier.

Detail View

A **Detail View** may be created in a drawing to show a portion of a view, usually at an enlarged scale. This detail may be taken from an Orthographic View, an Isometric View, a Section View or another Detail View.

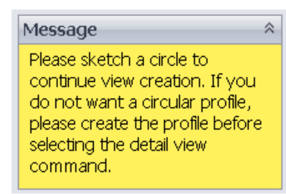
The detail view is determined by the contents of a closed sketch. The default used is a circle

Where to find it

Select **Detail View** from the **View Layout** toolbar.
or
Choose **Insert, Drawing View, Detail.**

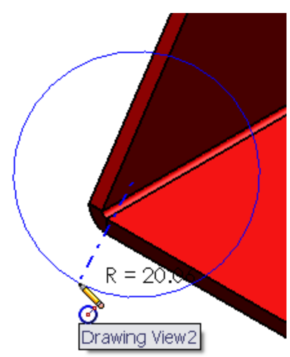
Create a Detail View

Click **Detail View**, the following prompt will appear.



Move the cursor to the approximate position of the centre of the viewing circle. Click and drag the radius.

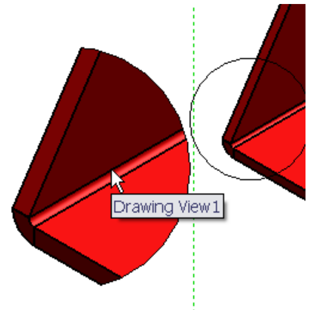
Note – The position and radius of the viewing circle may be altered afterwards.



Positioning the View

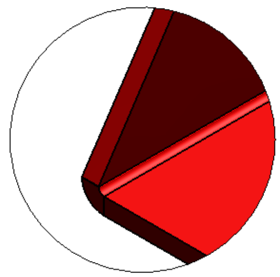
When the viewing circle has been defined the detail will appear.

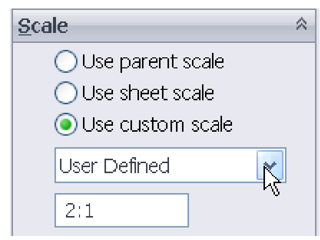
Drag the view to position, click to drop it.



Full Outline

Selecting **Full Outline** will display the viewing circle in the detail view. Select **OK.**





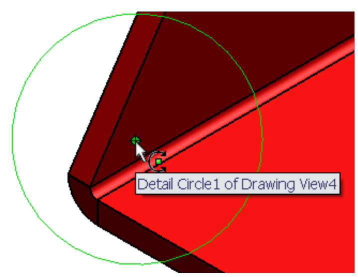
Edit the scale

Select the detail view and apply a 2:1 scale

Editing the centre position

To edit the centre position, click on the circle. A blue dot with crosshairs appears at the centre.

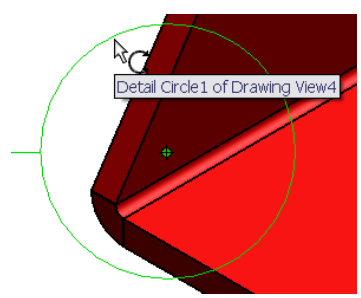
Hold and drag this to reposition the centre.



Editing the radius

Hold and drag the circumference to increase the radius.

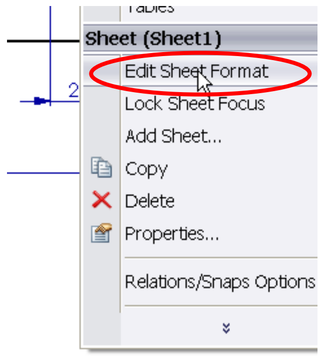
Using these two techniques, whilst displaying the detail view, will establish the detail view to show the required information.

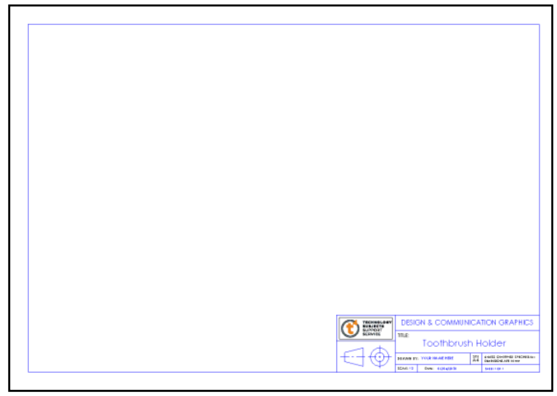


Editing the title block

Right click on a clear area of the drawing sheet

Select 'Edit Sheet Format' – the drawing views will disappear as the sheet background is being viewed





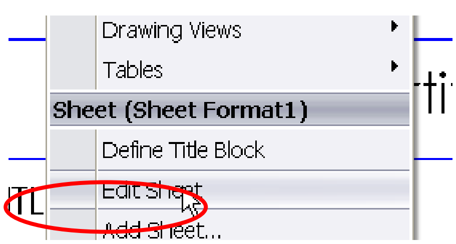
Double click on 'Design and Communication Graphics'

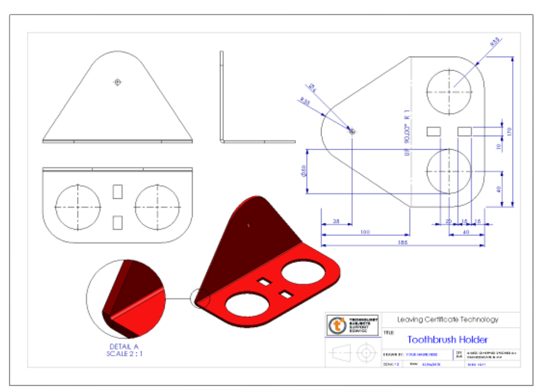
While the text is highlighted, type in 'Leaving Certificate Technology'.

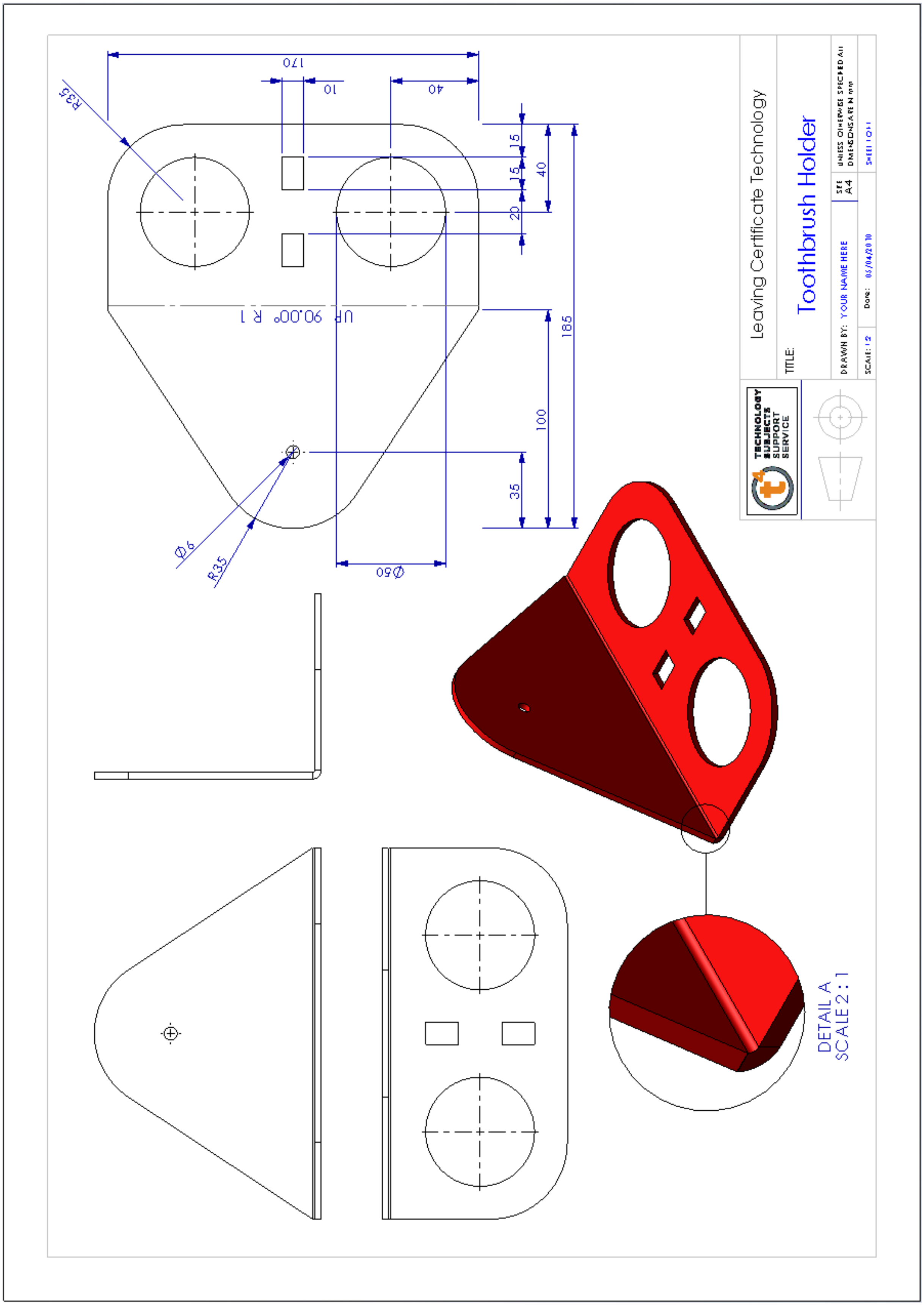
You can also edit the font type, size and colour while text is highlighted.
Press 'Esc' when you have edited the text.

You can reposition the text box by left clicking on the text and dragging it into the required position

When finished, right click on the drawing sheet and select 'Edit Sheet' – the drawing views reappear






Leaving Certificate Technology
TITLE:
Toothbrush Holder
TECHNOLOGY SUBJECTS SUPPORT SERVICE
DETAIL A
SCALE 2 : 1
170
10
40
R35
15
15
20
40
185
100
35
Ø6
R35
Ø50
UP 90.00° R 1