# PDST

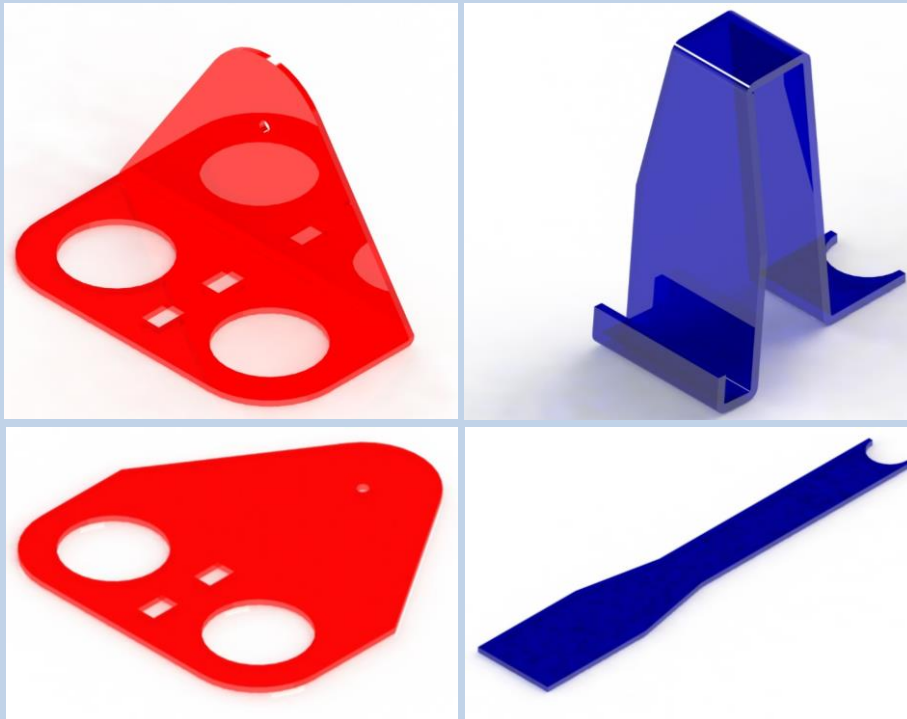**Professional Development Service for Teachers** | **An tSeirbhís um Fhorbairt Ghairmiúil do Mhúinteoirí**

# Introduction to Solidworks

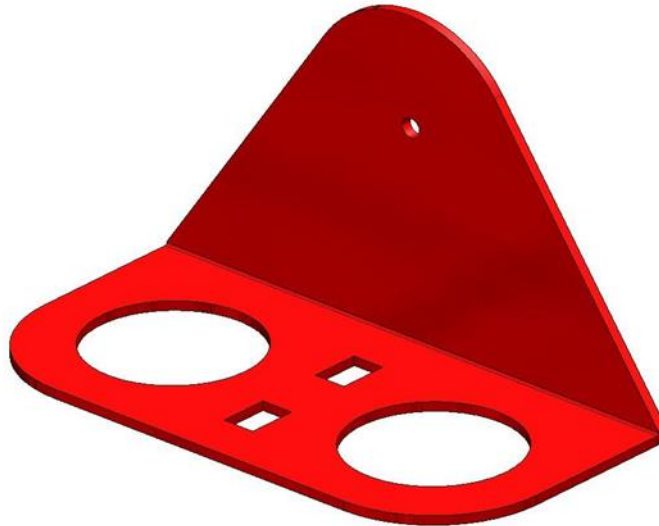# Sheet-Metal

## No1: Toothbrush Holder

## No2: Phone Holder

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# Exercise 1: Toothbrush Holder

## Introduction

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.

## Learning Intentions

At the end of these exercises, you should be able to:

- Create a sheet metal part, using **Base Flange, Extruded Cut, Sketched Bend** and **Edit Material** commands

- Create a drawing of the sheet metal part

-

## 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

## Saving Your Work

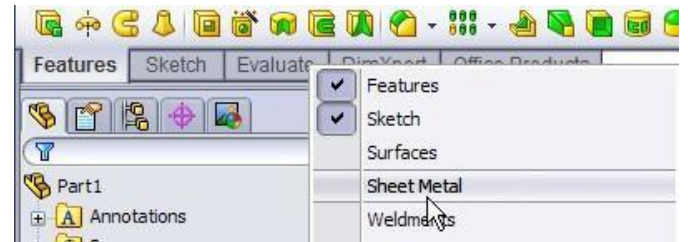Save the file as **Toothbrush Holder** to a folder called **Toothbrush Holder Exercise.**

(**Continue to save periodically throughout the exercise**)
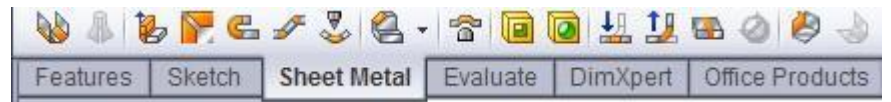
# Creating the Sheet Metal Part

## 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.

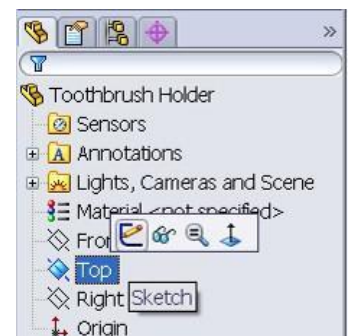The Sheet Metal tab is now active on the command manager.

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

## Creating a sketch

Begin by creating a sketch to generate the rectangular piece of acrylic required to manufacture the artefact.
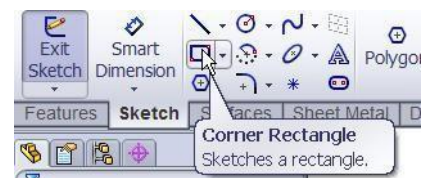
Because the material sits on the horizontal plane while we carry out the work, we will create a sketch on the **Top Plane**.

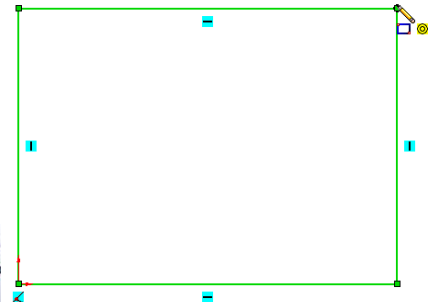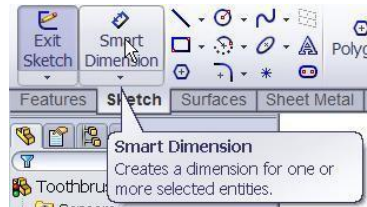Left click on the 'Top' plane and click on the sketch icon

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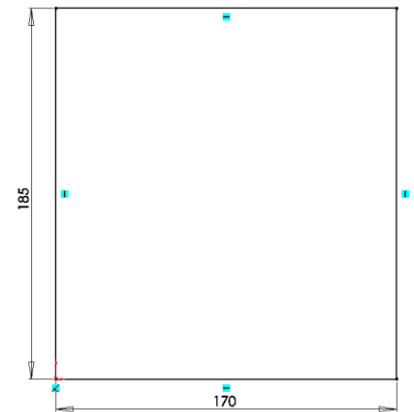
(Press 'Esc' to exit the Corner Rectangle command)

**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.

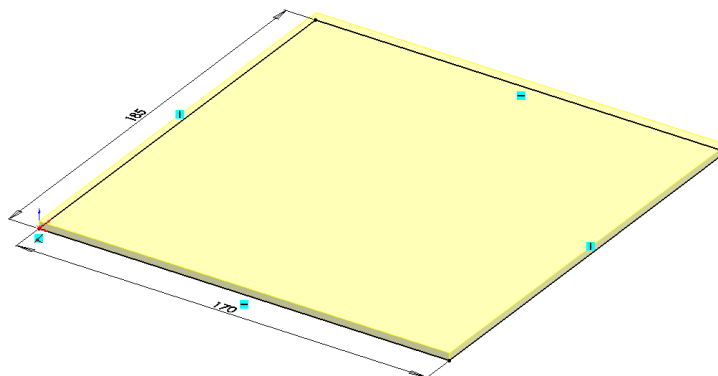The sketch lines turn black when **fully defined**

Exit the sketch

## Sheet Metal

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

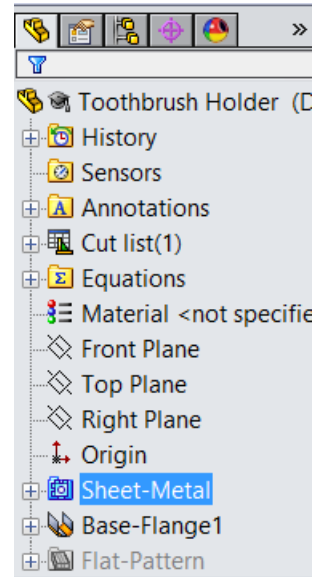Enter a value of 3mm for **thickness** in the Base Flange options dialog box

**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 Feature Manager 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.
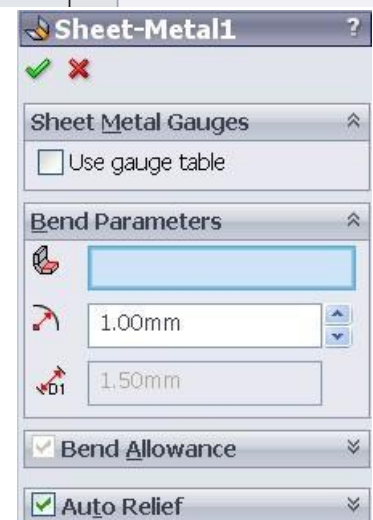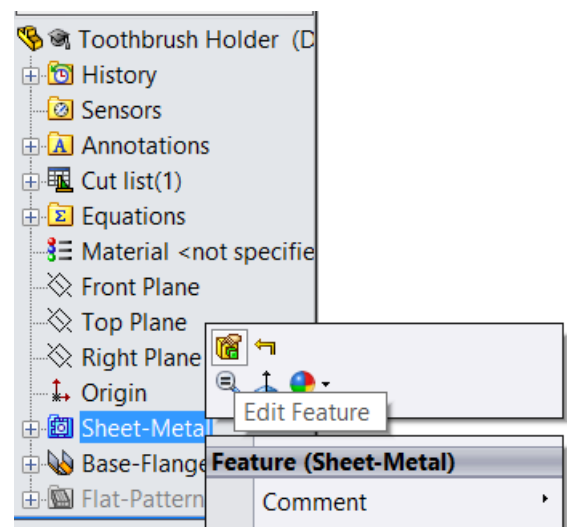
**Sheet-Metal1 Feature:**

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

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

- 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
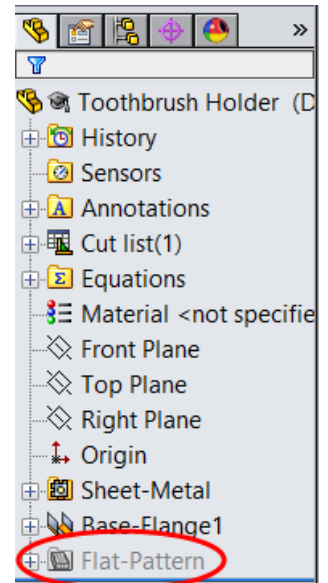
Choose **OK**

**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.

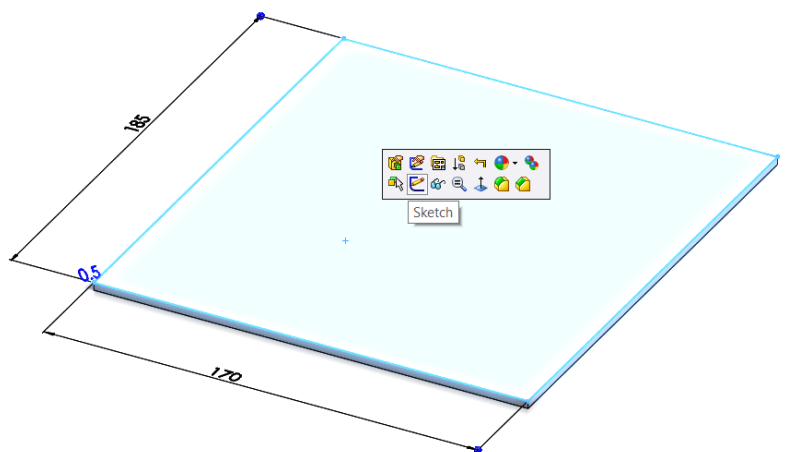Unlike other features, flat-pattern will remain at the bottom of the tree. Other sheet metal features, when added, will appear overhead even though they are added after its creation. Secondly, the feature is suppressed when added to the design tree.

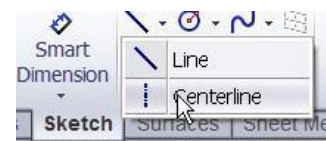We will look further at this feature as we work through this exercise.



**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
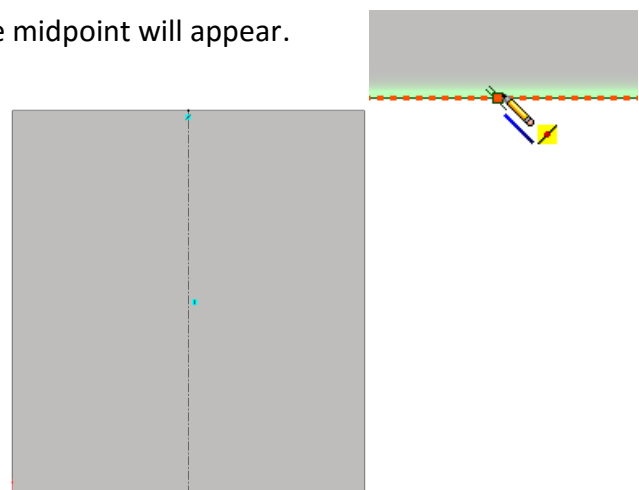


Select **Normal To** ⭥ from the view selector (press spacebar)

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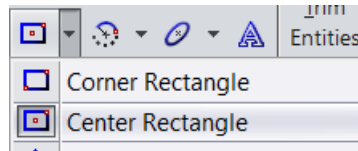


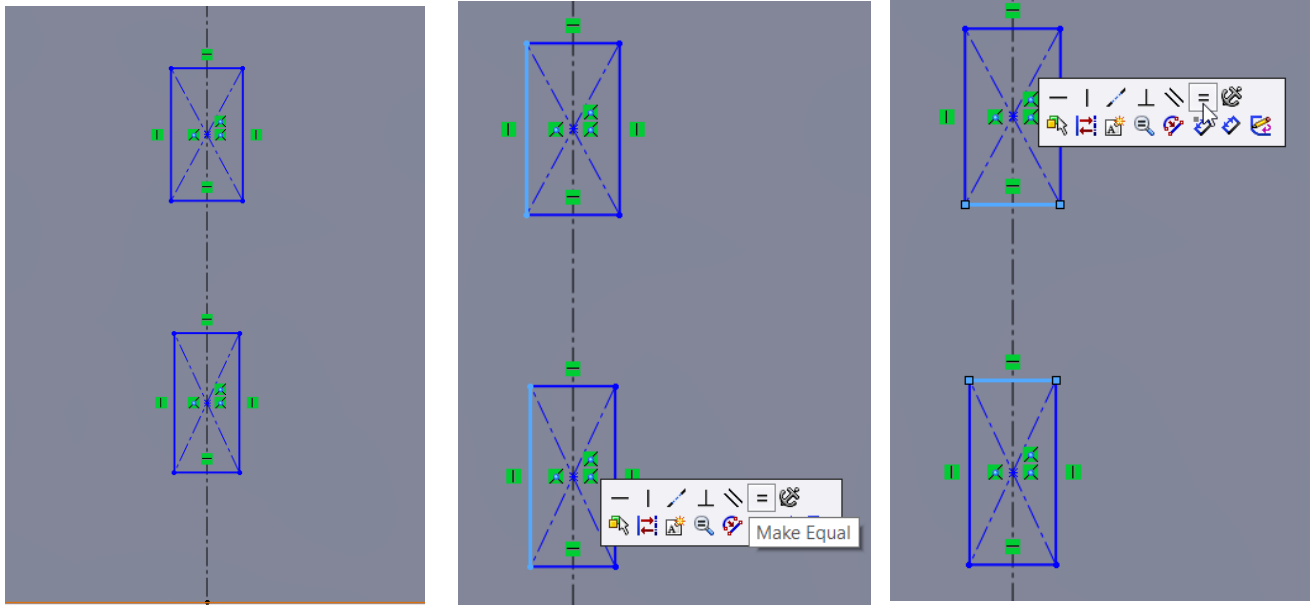Hover over the edge of the base flange and the midpoint will appear. Sketch the vertical centerline

Using the Center rectangle command draw two rectangles as shown
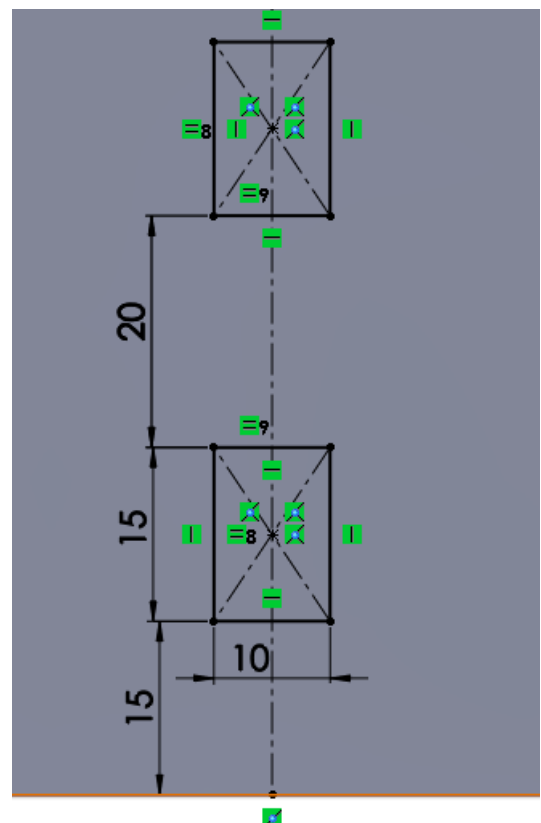
Add an equal relations to two vertical lines (hold the Ctrl button and select the lines) Then add an equal relation to two horizontal lines.



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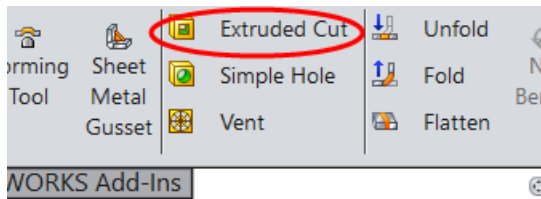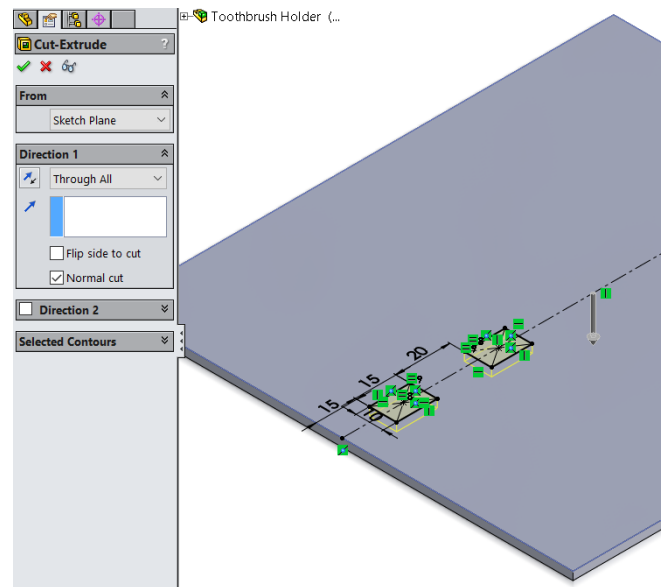
Exit the sketch

## Extruded Cut

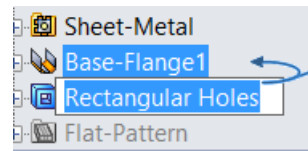Select Extruded Sheet Metal Toolbar



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

Select ![checkmark]



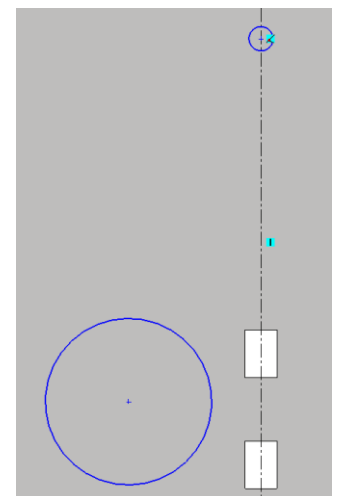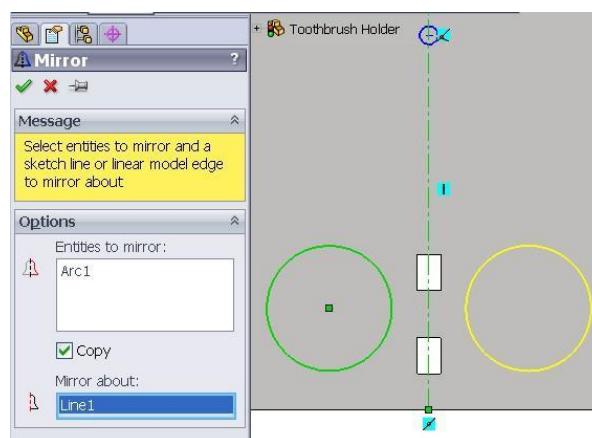Double click on the 'Extrude 1' feature and rename as **Rectangular holes**



## 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 centreline and one larger near the rectangular holes

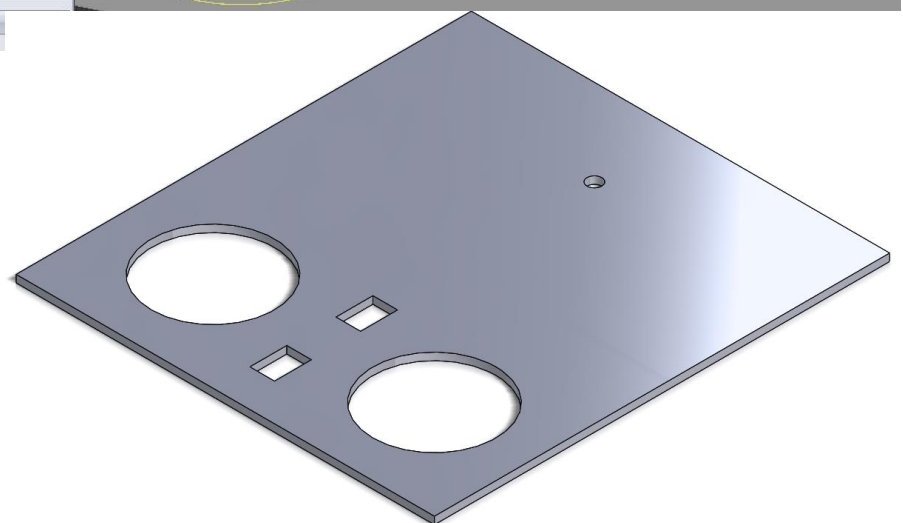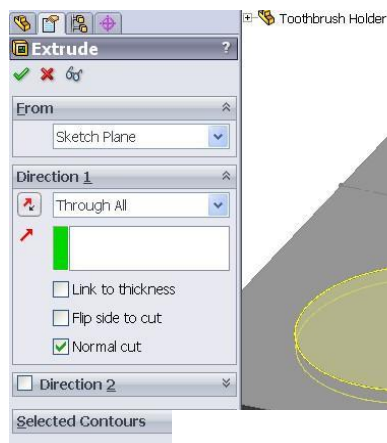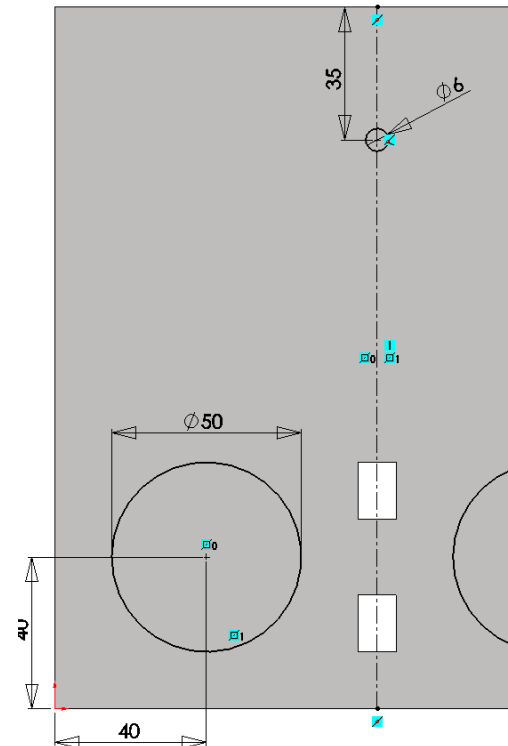Use 'Mirror Entities' to create a circle on the right of the centerline

Smart dimension as shown opposite

Exit the sketch

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 this feature as

**Circular holes**

## Shaping of Holder

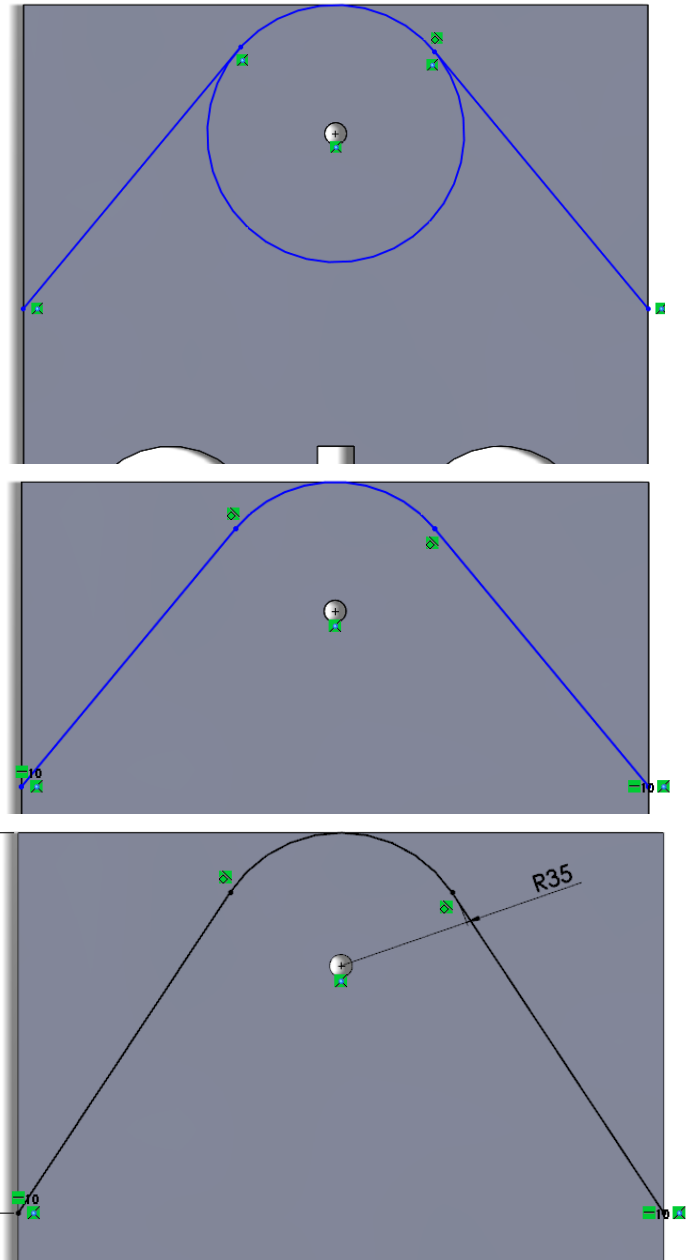Create a sketch on the top face of the base flange as shown

The circle should have its centre coincident with the existing hole and tangential to the edge.

The lines should start coincident with the edge and be have an auto tangent relation to the circle.



Power trim the inside portion of the circle

Apply a Horizontal relation between the endpoints of the line segments and the points of tangency in turn.
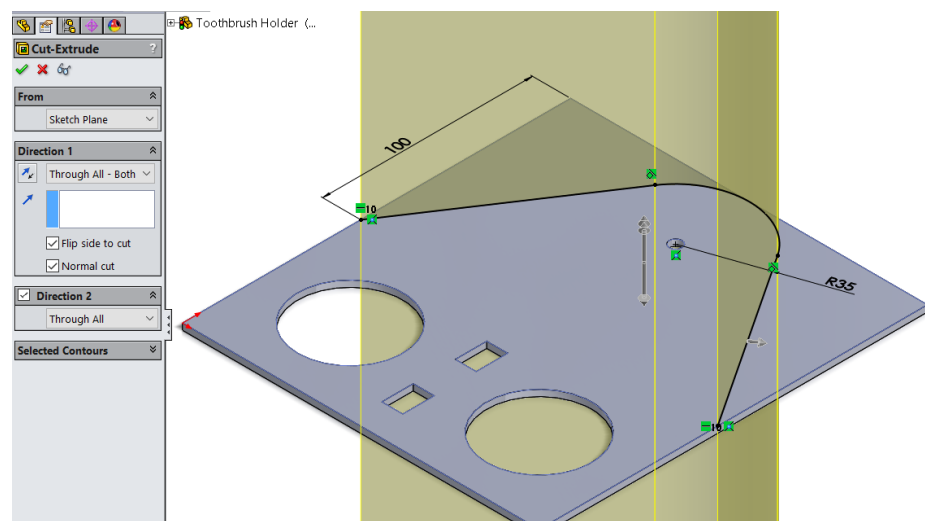
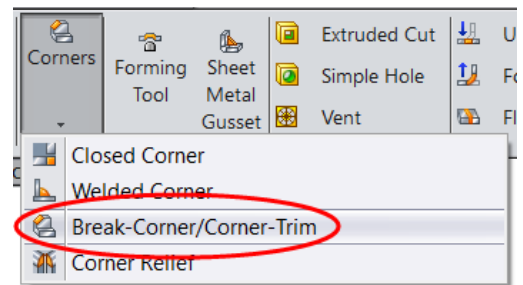Dimension the sketch as shown opposite.





## Extruded Cut

Select **Extruded Cut** from the Sheet Metal toolbar. Select **Through - All-Both** as the direction.
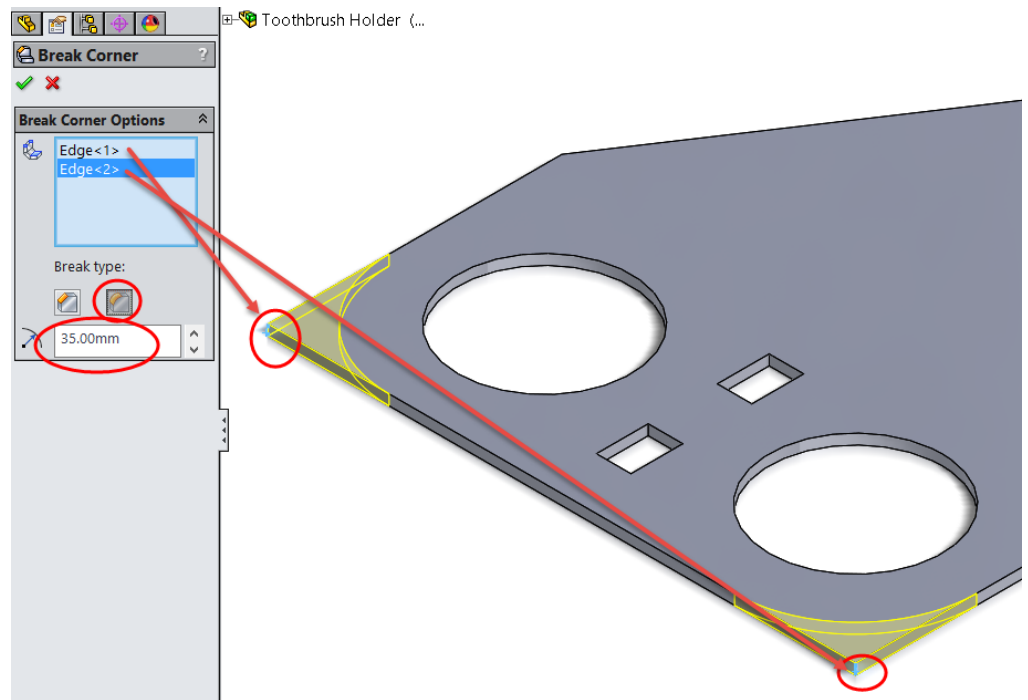
Rename feature as **Shaping**

## Fillets

Select the 'Break-Corner/Corner-Trim' from the **'Sheet Metal'** toolbar

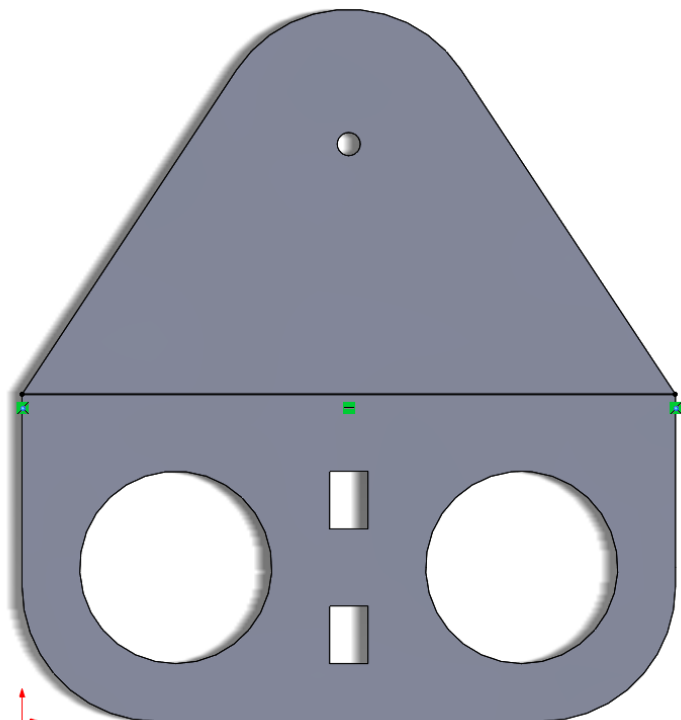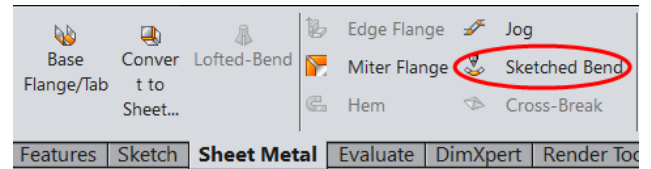Choose the settings in the property manager as shown

## 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 ending line.
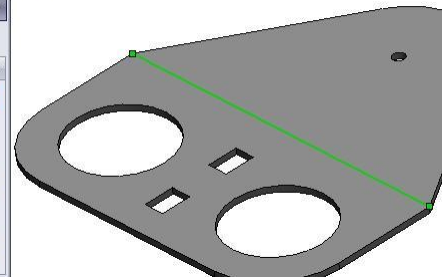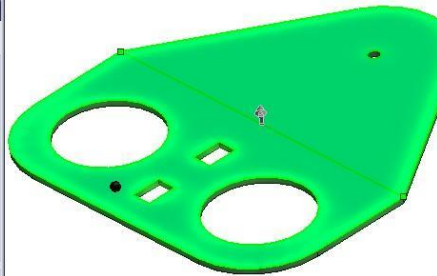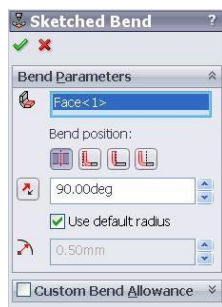
This line will be used as the bend line

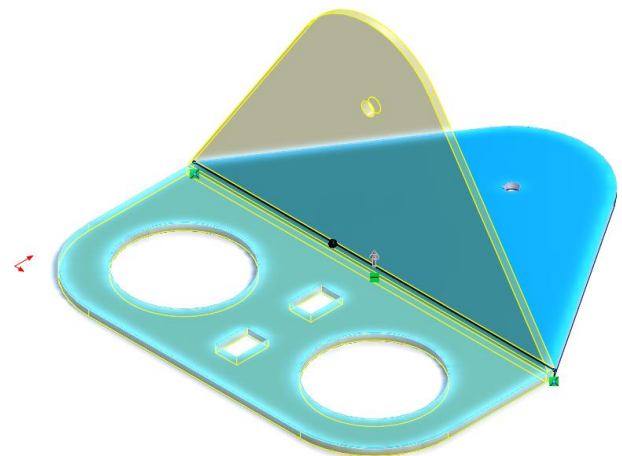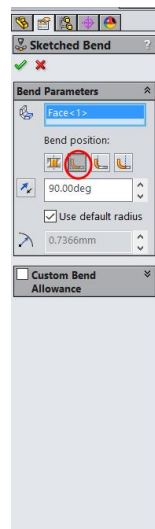Select 'Sketched Bend' from the Sheet MetalToolbar.

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
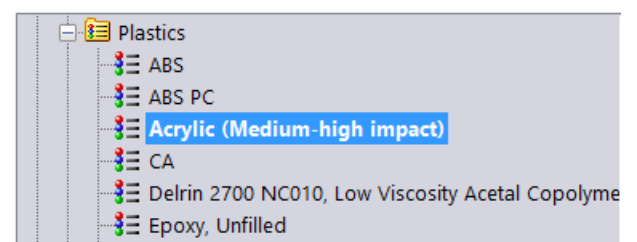
Choose the default radius as the bending radius

**Edit Material**

Right click on **Materials <not specified>** in the Design Tree and select **Edit Material**
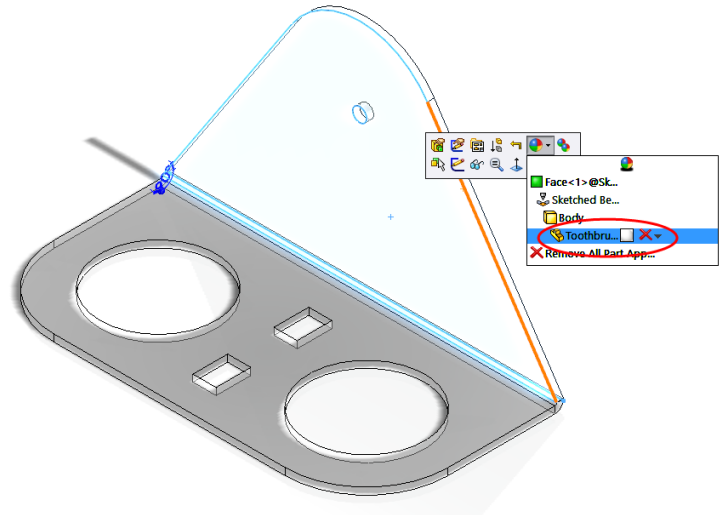
Scroll down to the **Plastics** folder and select

**Acrylic (Medium-high impact)** and choose
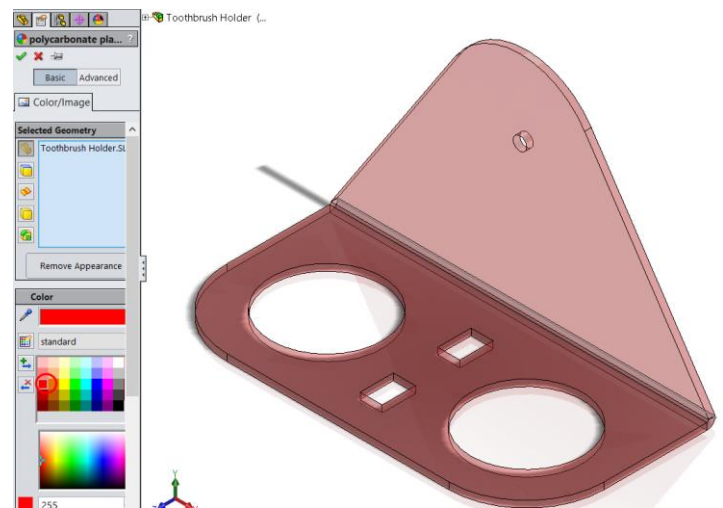
**Apply** and **Close**

Right click on any face of the toothbrush holder go to the appearances icon and select the **Part** to apply an appearance

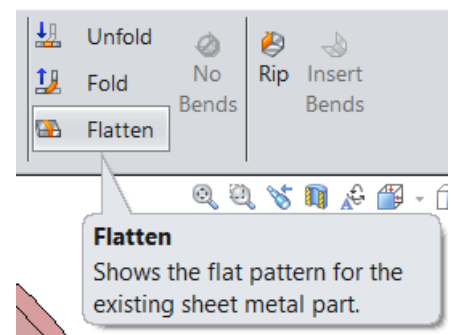Select a colour from the swatch In the Appearances Property Manager

## 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**: Right click on the feature and choose Flat-pattern. Unsuppress from the pop-up toolbar

Or

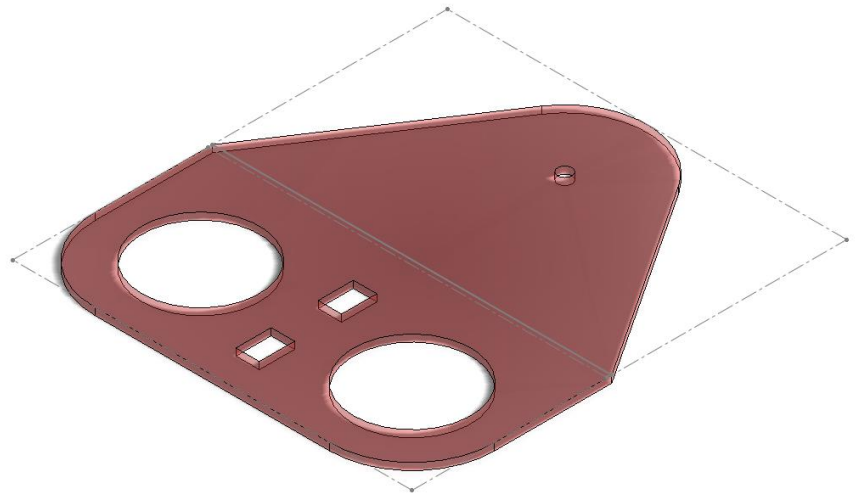Select **Flatten** from the sheet metal toolbar

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

The bend line is also displayed.

Click on **Flatten** in the sheet metal toolbar to suppress the sheet metal feature

Save you work

# Creating a drawing of the Toothbrush Holder

## 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.

The DCG Templates will only be available if they have been saved following the instructions outlined in the on-line resource

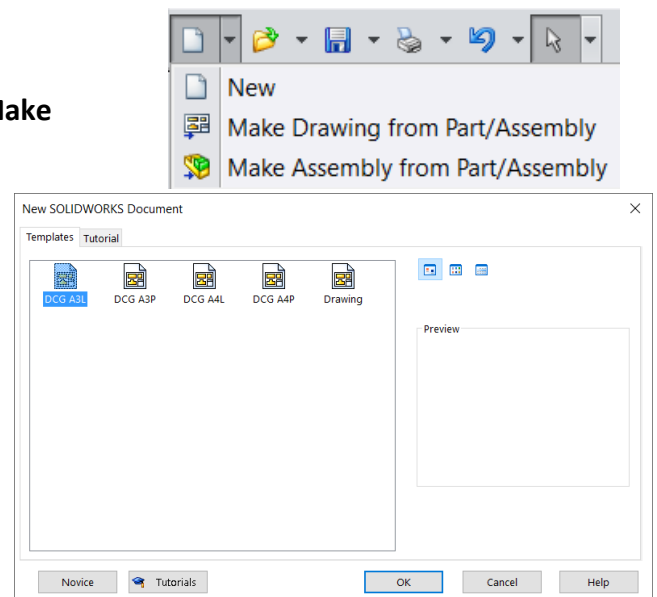http://www.t4.ie/sw/Drawings.html

## Getting Started

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

Select Drawing and then click **Advanced**

Choose the drawing template you wish to use from the list displayed, for example **DCGA4L**
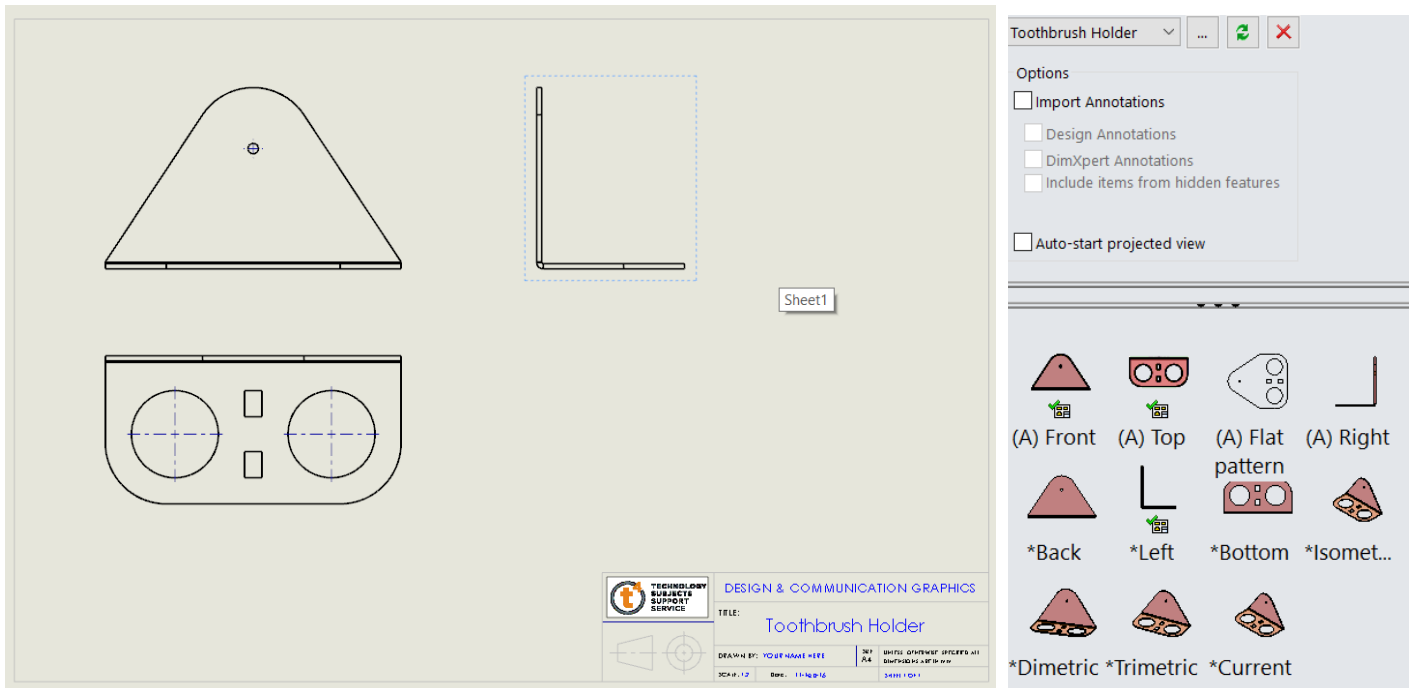
Choose **OK**

Save the drawing. It will automatically save as same name as part.
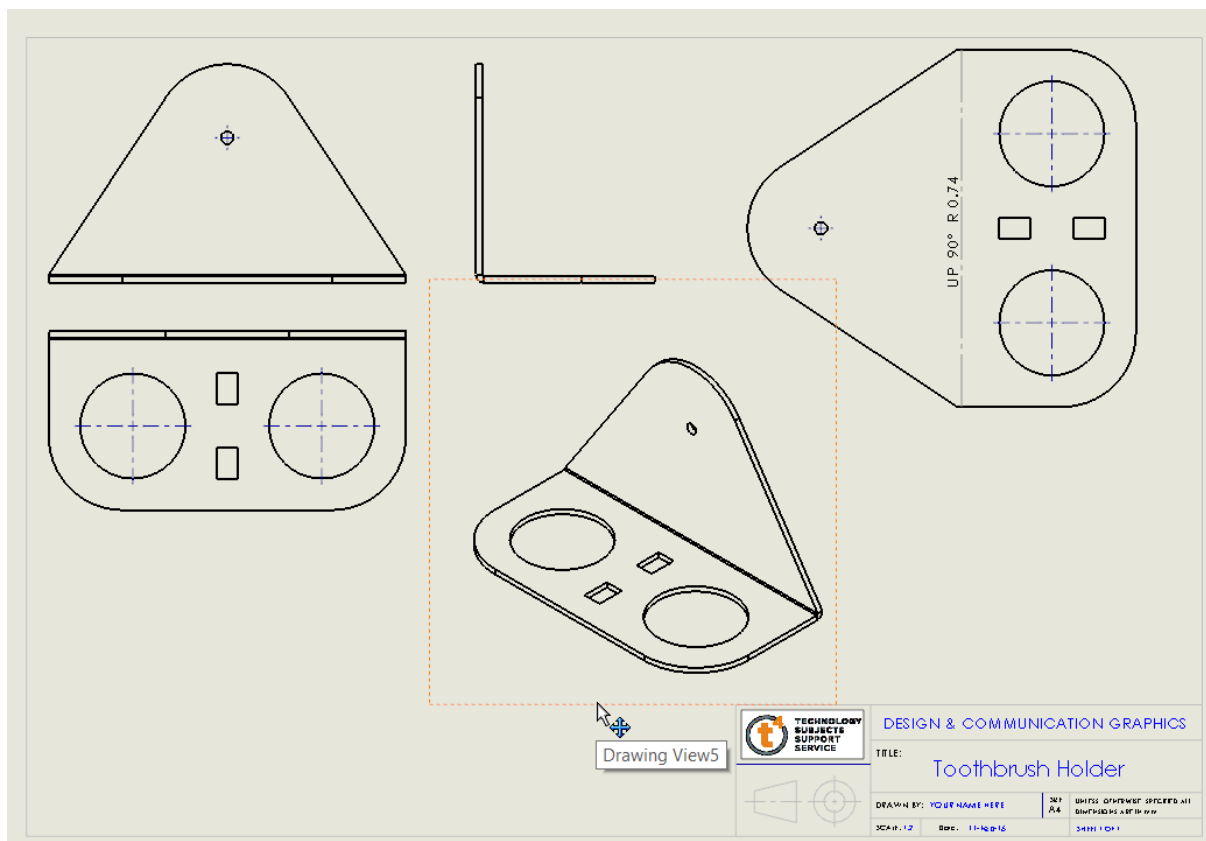
Drag and drop in the three orthographic views from the view palette



Drag and drop in an Isometric view and Flat pattern view

The view can be re-positioned by hovering over box until the pan symbol appears beside the cursor. Hold down the left mouse button to move the view
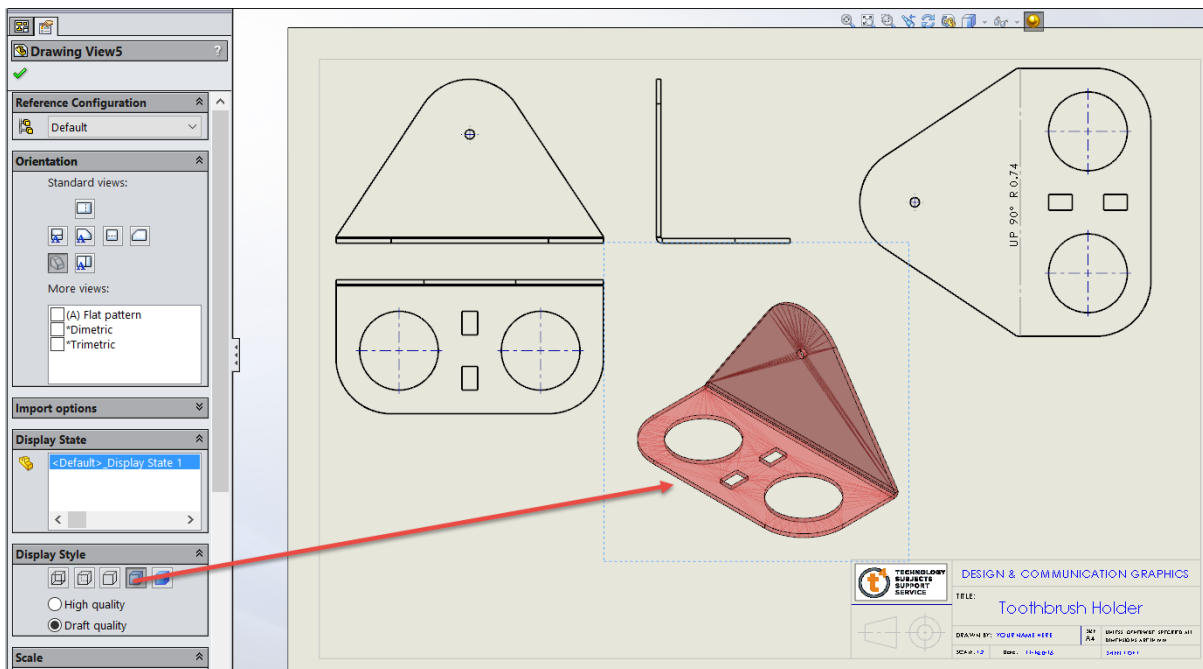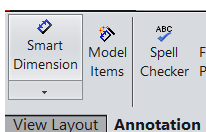
The view can be displayed in a number of ways

- **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.

Click on the isometric view to change display to **Shaded with edges**
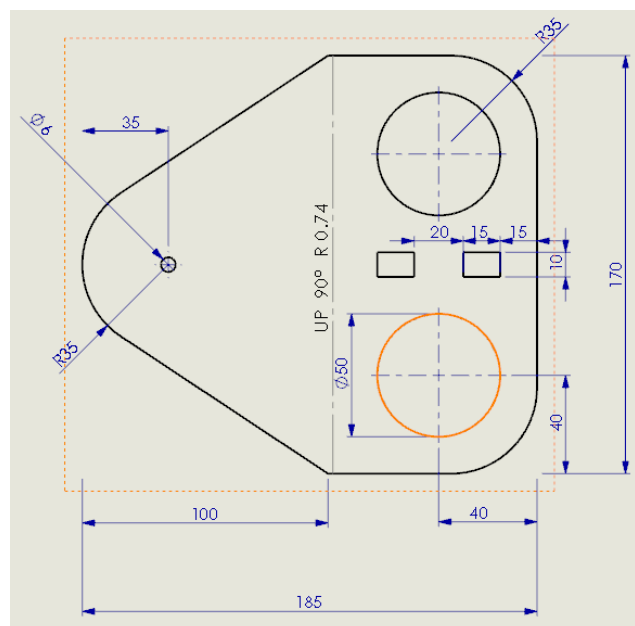


Select **Model Items** in the Annotation tab



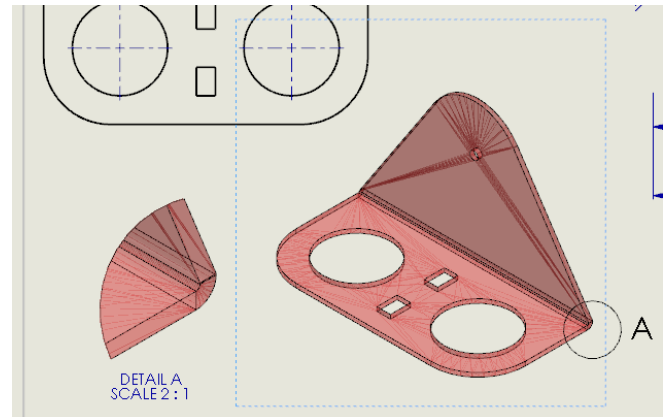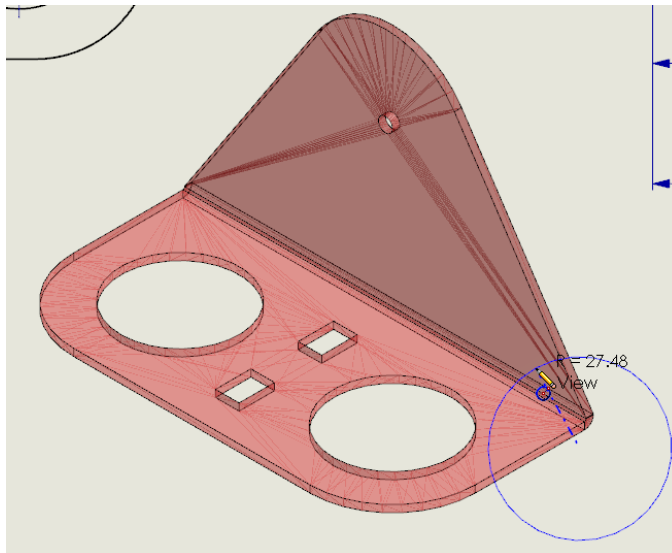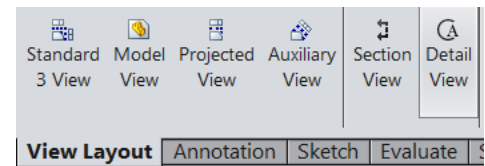Select flat pattern view to import all dimensions.

Move dimensions to improve their appearance

## Detail View

Select Detail View tool and sketch a circle around the

bend of isometric view as shown

The view can then be positioned on the sheet. The scale
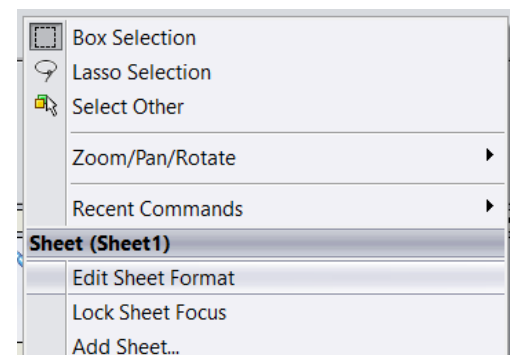
can be changed to **2:1**





## 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
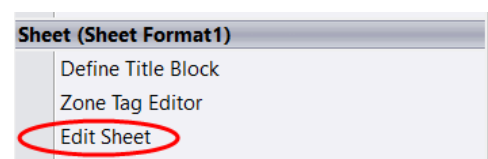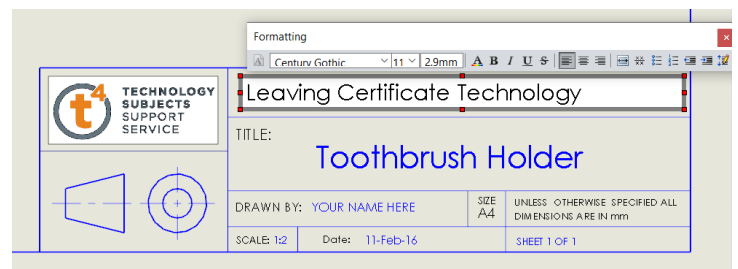
Technology'.



You can also edit the font type, size and

colour while text is highlighted.

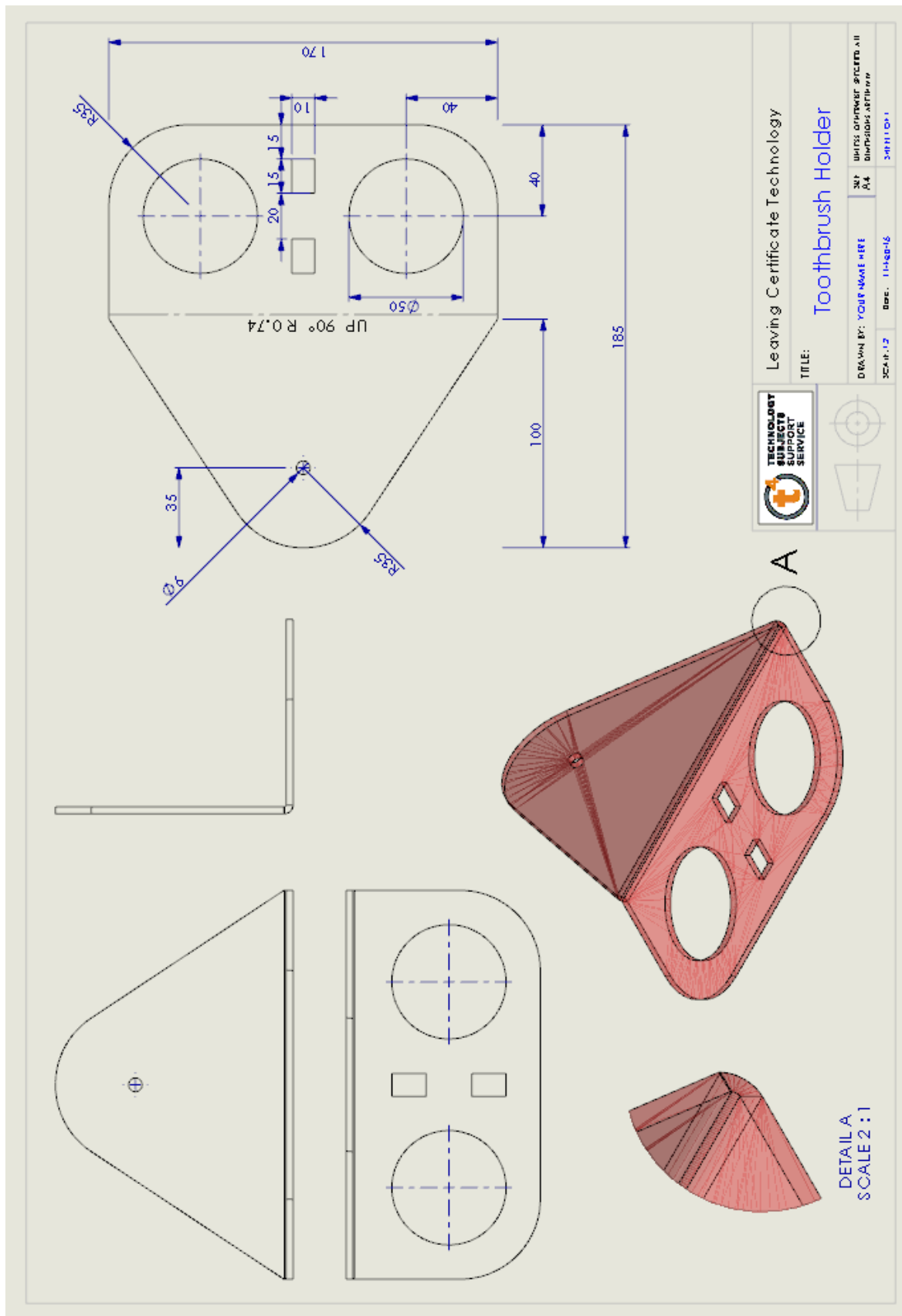Press 'Esc' when you have edited the text.

You can reposition the text box by left

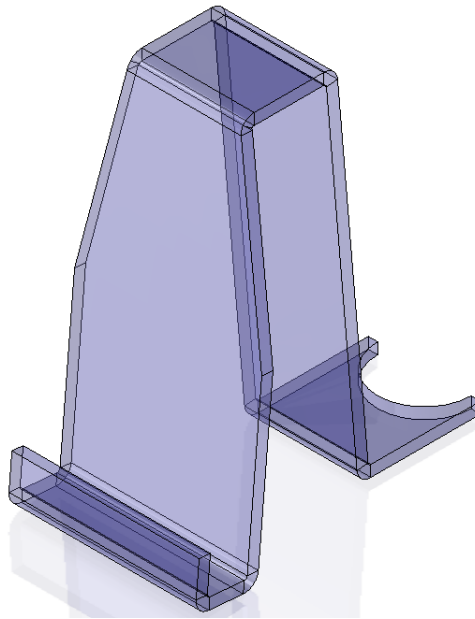clicking on the text and dragging it into the required

position

When finished, right click on the drawing sheet and

select **Edit Sheet,** the drawing views reappear. Save your work

The exercise is now complete

# Exercise 2: Phone Holder



## Introduction

This lesson focuses on designing a sheet metal part from the flattened state.to include a series of bends. The model and drawings will also serve as a classroom for examination questions based on developments and orthographic projection.

## Learning Intentions

At the end of these exercises, you should be able to:

- Create a sheet metal part, using **Base Flange, Extruded Cut, Sketched Bend** and **Edit Material** commands
- Create a drawing worksheet of the exercise
- 

## Prerequisite Knowledge

Introduction to sheet metal commands including a basic knowledge of SolidWorks from the previous exercises (toothbrush holder and childs toy)
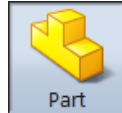
## Saving Your Work

Save the file as **Phone Holder** to a folder called **Phone Holder Exercise.**

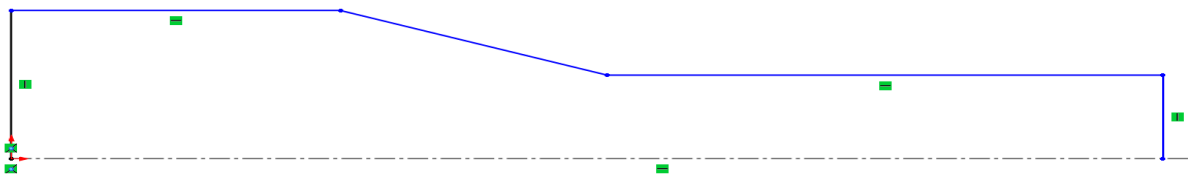(**Continue to save periodically throughout the exercise**)

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## Creating the Sheet Metal Part
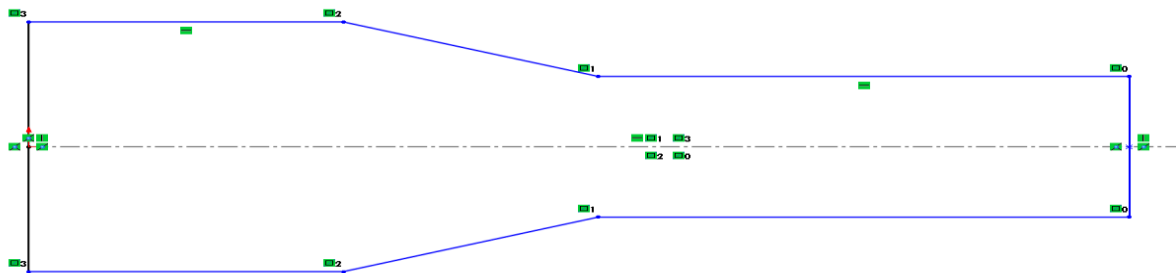
## Creating the Sketch
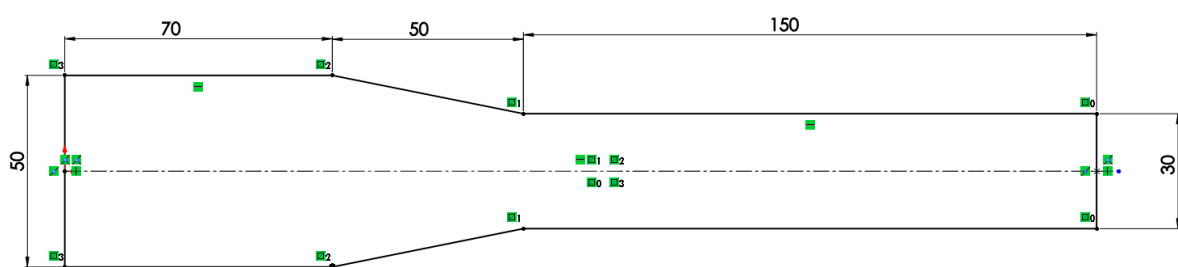
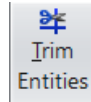Create a new Part. 

Draw the following sketch on the **Top Plane**



Mirror the sketch. Mirror Entities .



Dimension the sketch as shown

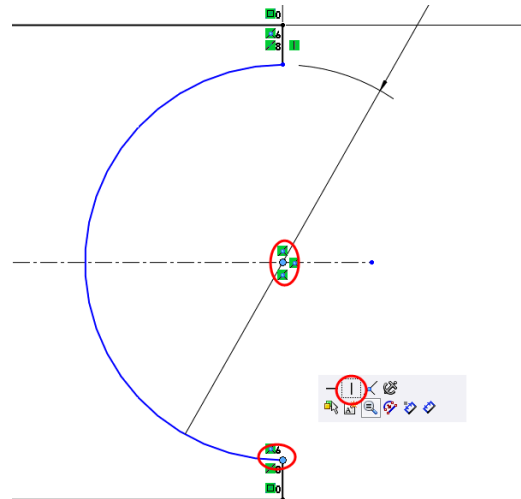Add a **R.25mm** circle at end and trim [Trim Entities] excess lines. Add a vertical relation to the circle centre and an endpoint of the trimmed line.
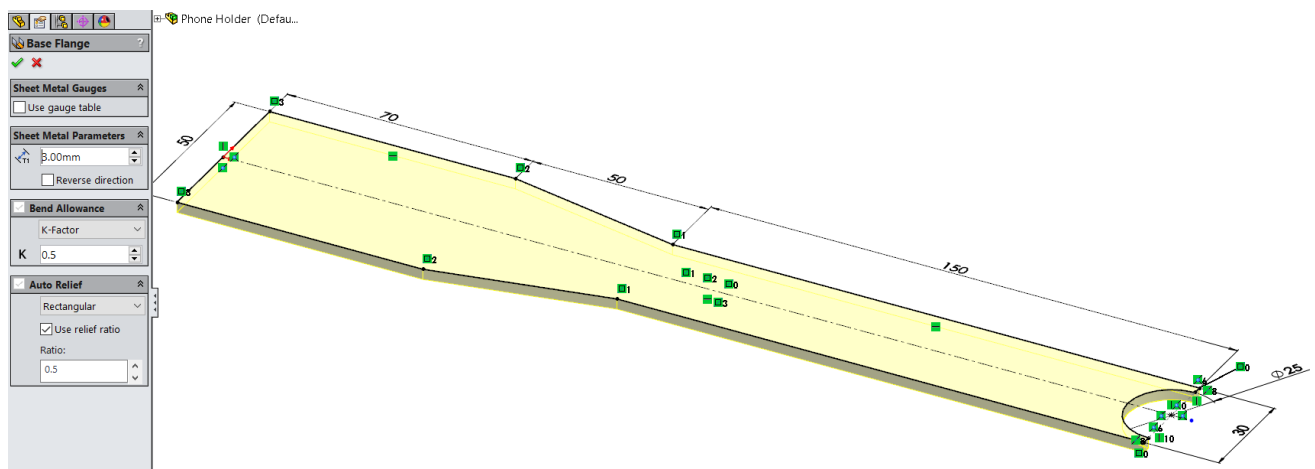
Save your work.

The sketch should be fully defined.

## Create the sheet metal feature

Select the **Sheet Metal** tab and select Base Flange [Base Flange/Tab].

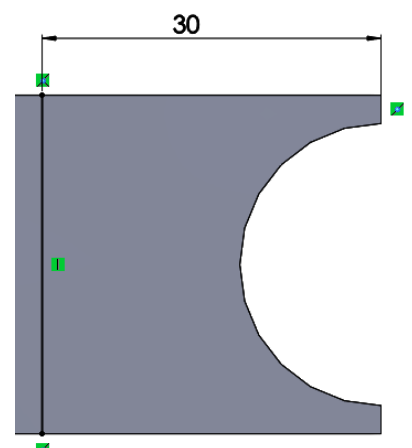The material thickness is **3mm**.



## Create the Sketched bends

> **NOTE:**
>
> The bends will be created in a logical sequence that would occur in "real-life", if you had to make the holder. This will allow the file to be used as a teaching tool to explain the bending sequence

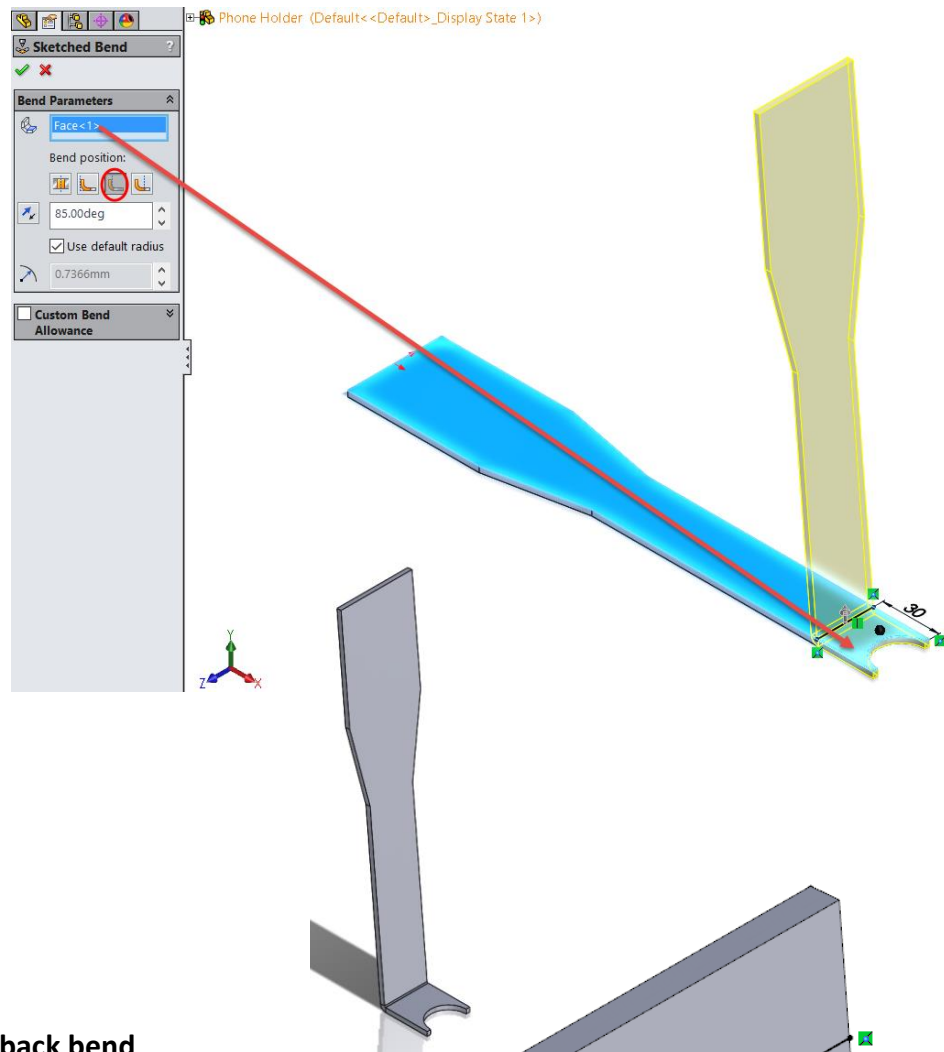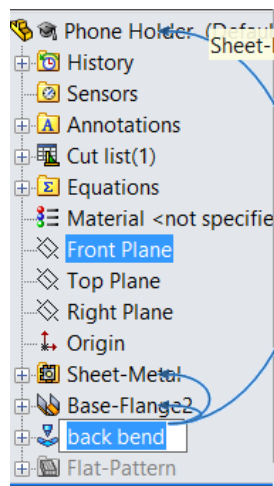Draw a new sketch on the surface as shown

Select the **Sketched Bend** command .

Bend Parameters:

- end to be the fixed face
- **85** degree bend
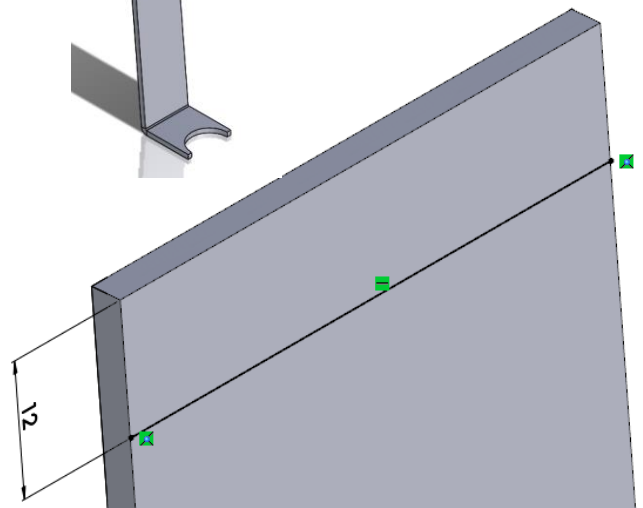- **Material Outside** bend position

Click OK 



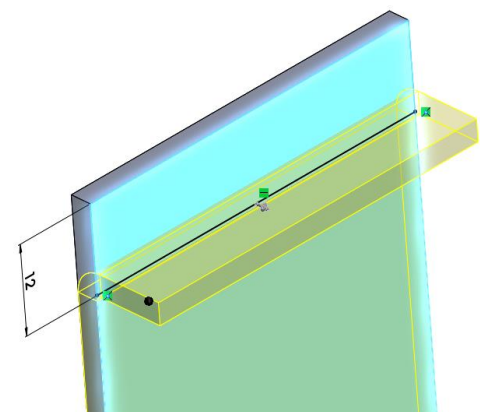Change name of bend feature to **back bend**

(press Fn + F2)

Create another sketch on the other end as shown

Select the **Sketched Bend** command

. Bend Parameters:

- Centre to be the fixed face
- **90** degree bend
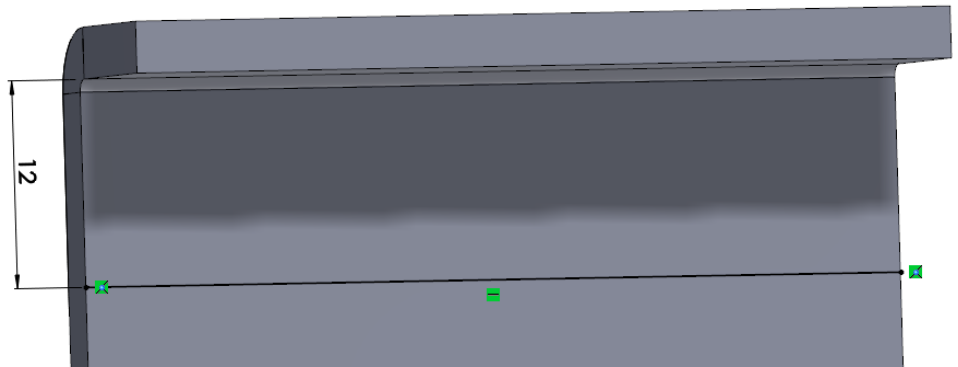- **Material Outside** bend position

Click OK ✔

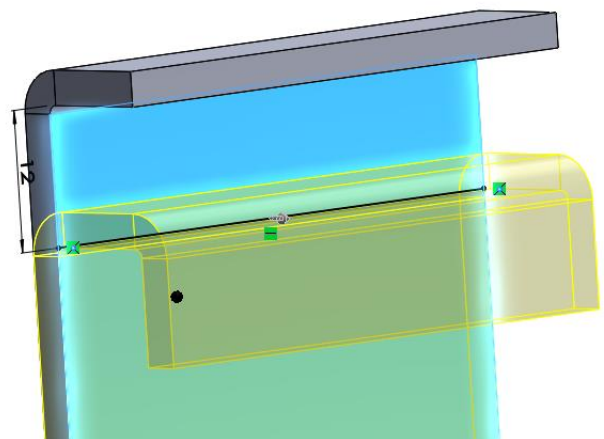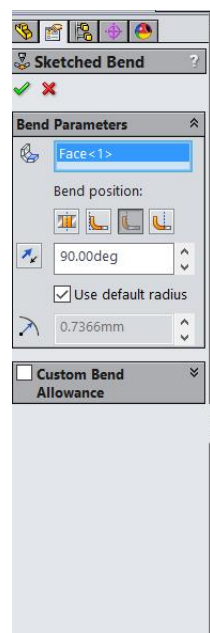Change name of bend feature to **Front bend 1**

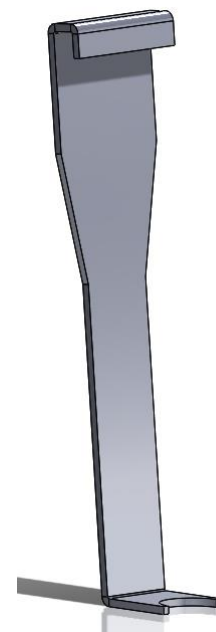Draw another sketch close to previous. Dimension from the bend surface not the curvature edge

Repeat the sketched bend exercise to form the J shape on front of phone holder using the same parameters

Click OK ✔.

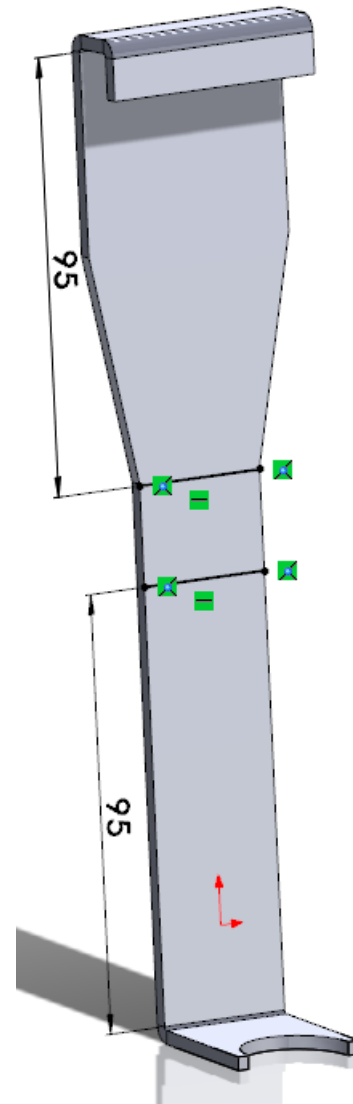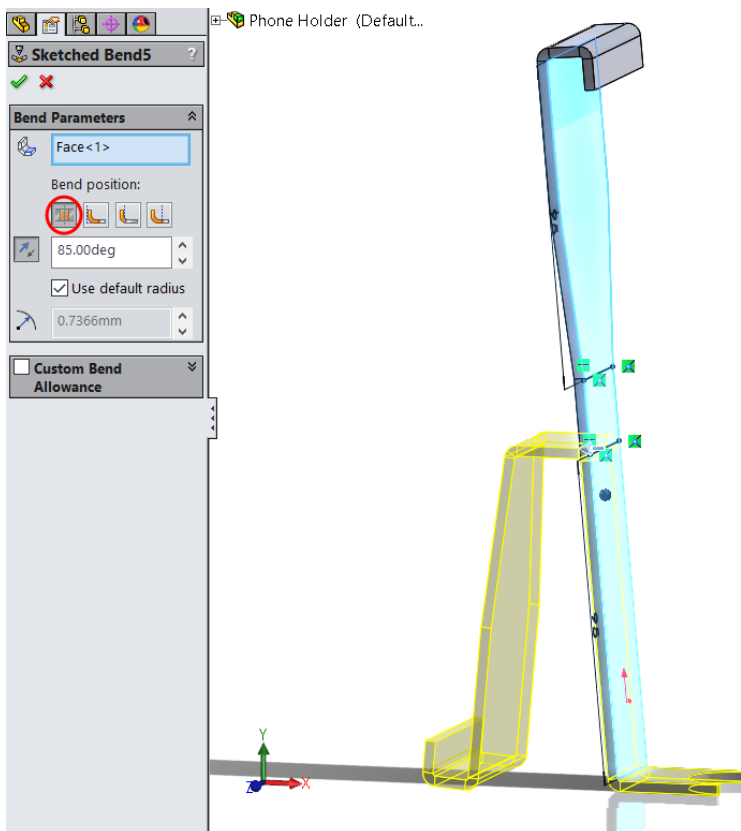Change name of bend feature to **Front bend 2**

Draw a final sketch for the two top bends

Select the **Sketched Bend** command  .

Bend Parameters:

- Rear surface to be the fixed face

- **85** degree bend

- **Bend centreline** bend position



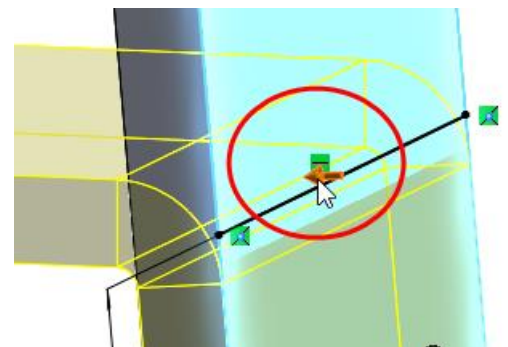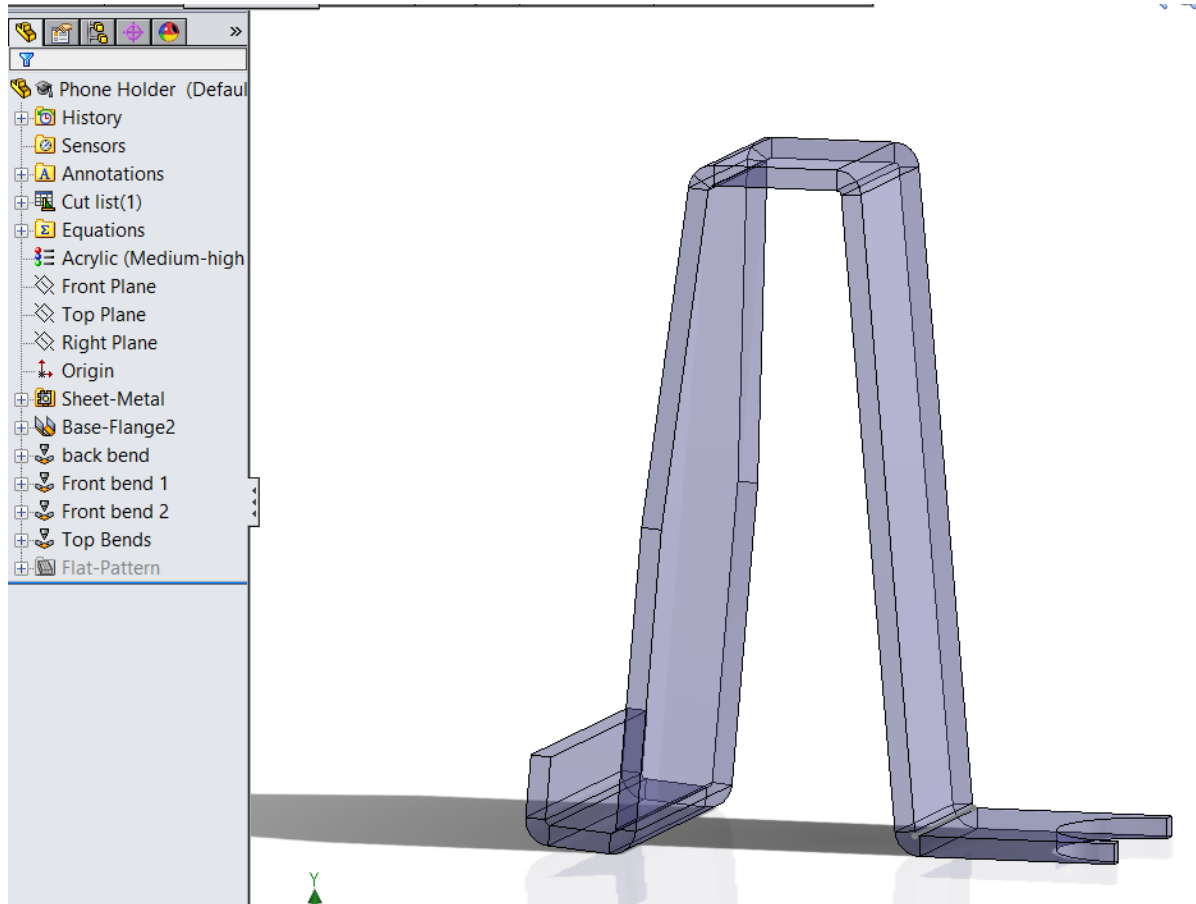You may need to change direction of bend by selecting the arrow on the screen

Click OK  .

Rename bend feature to **Top Bends**
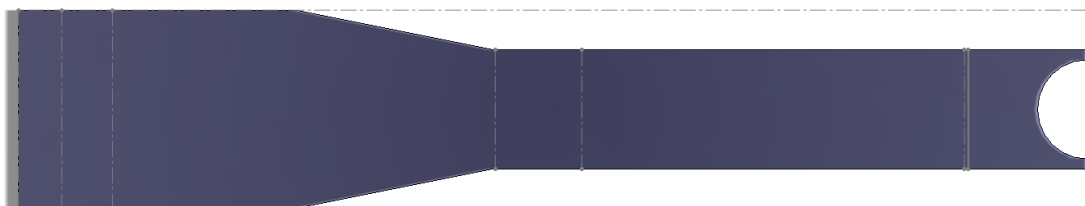
Save your work



---
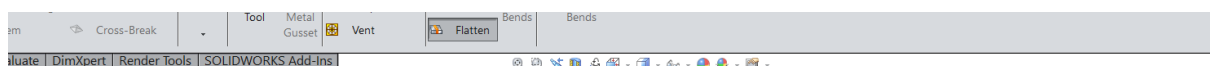
Edit the material to **Acrylic (Medium-high impact).** Edit the appearance to a blue colour.

(see previous exercise for further information)
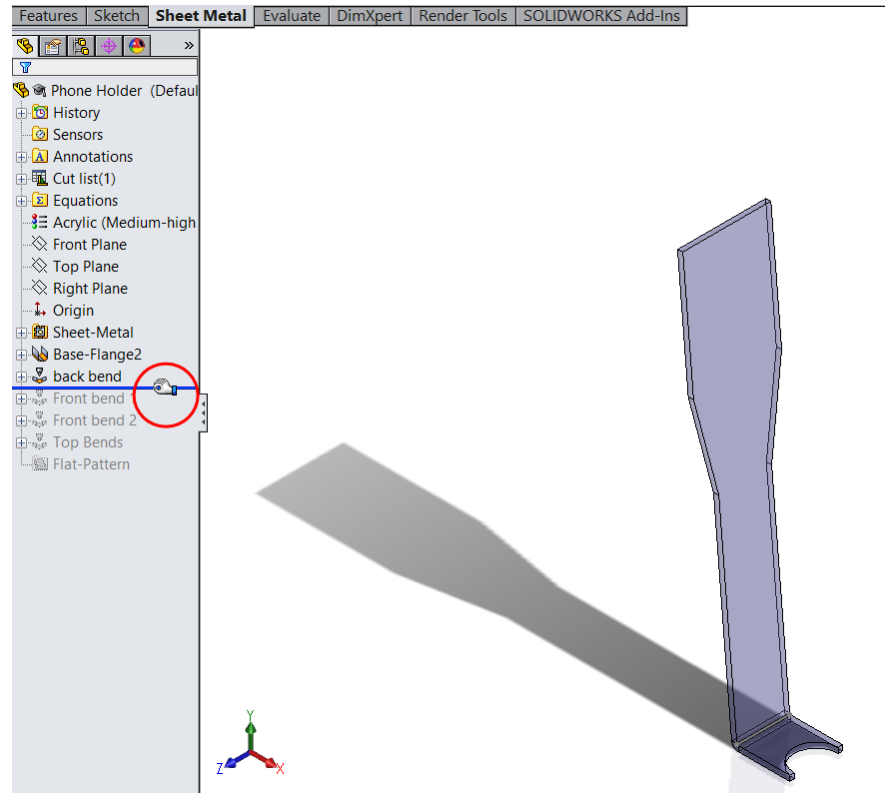


Your **Phone Holder** is now complete

You can use the Flatten feature to show in the flattened state
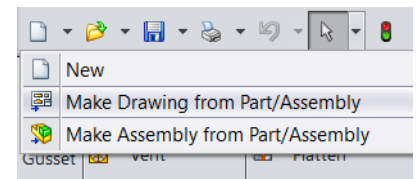
The **Roll back** command allow you to show the sequence of bends. Bring the cursor to the design tree blue line, hold down the left mouse button and move over features to roll back the development of the part.
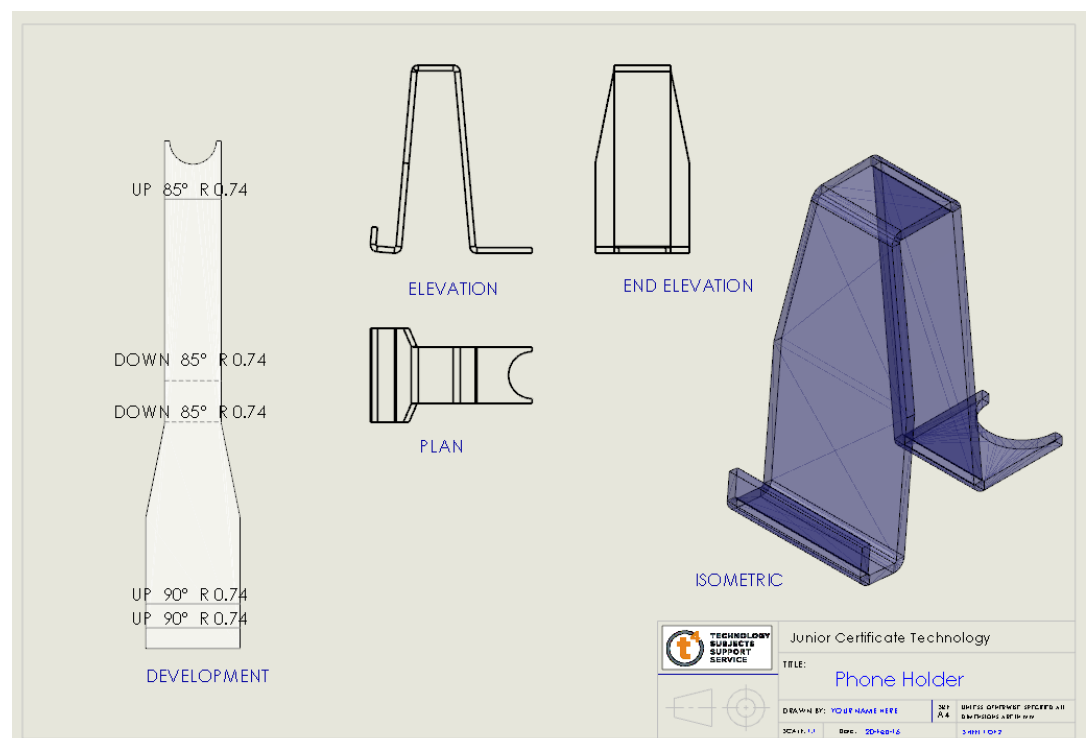


# Creating the Drawings & Worksheets

As in the previous exercise a drawing can be created of the orthographic, isometric and developed views. Save your drawing in the folder created earlier. The sheet template is DCG A4L



The sheet format can be edited as in the earlier exercise

## Creating Worksheets

The question (JC Technology 2012 HL Part B) can be inserted as an image into a sheet, once it has been saved as a picture (use print screen or snip it tool)

To create a worksheet for **Part (i),**

- drag and drop the flattened view of the part into the space. The text can be hidden by right clicking and select **Hide**.

- Use the Break
  tool  to
  remove the
  middle of the
  view, increase
  the gap to
  110mm
- Add text to
  sheet



**Broken View**

**Message**

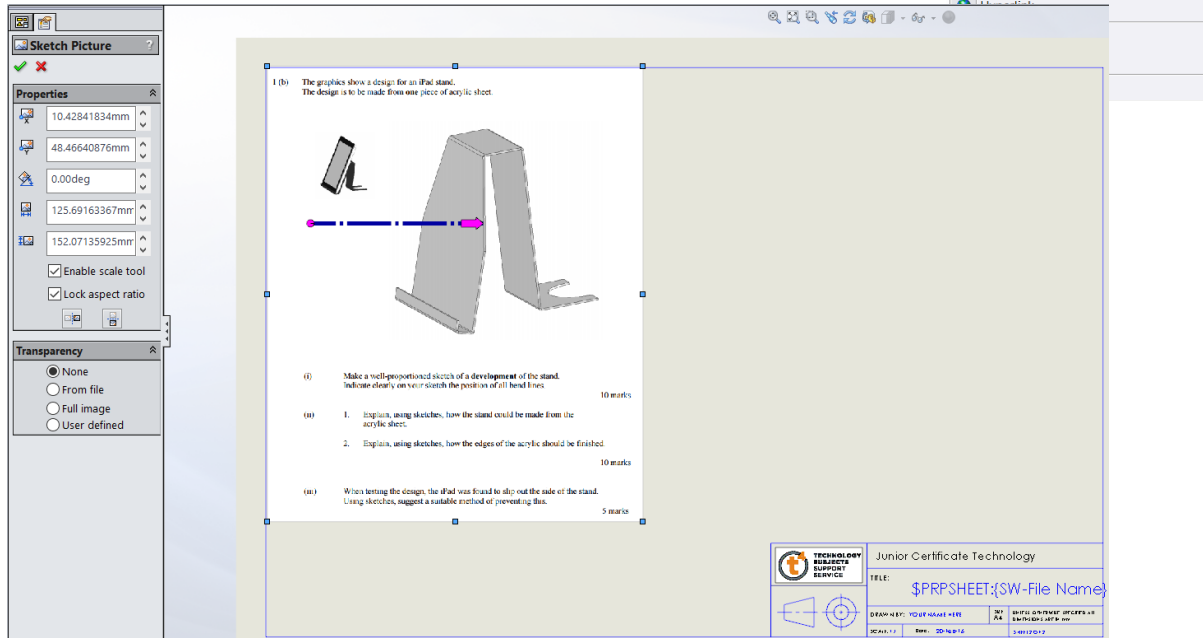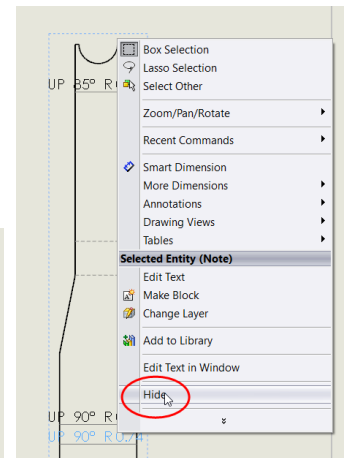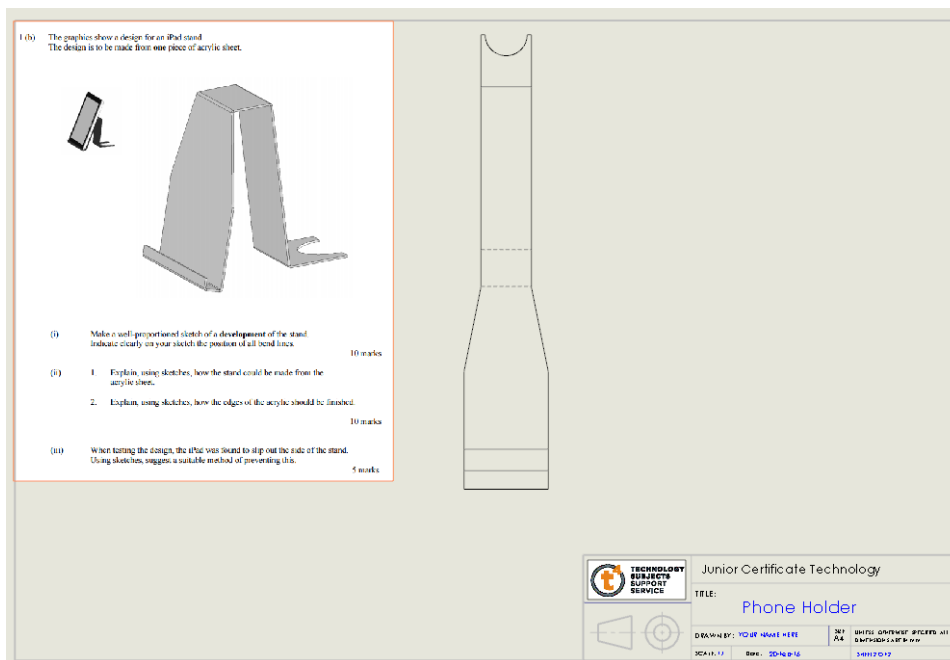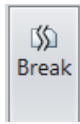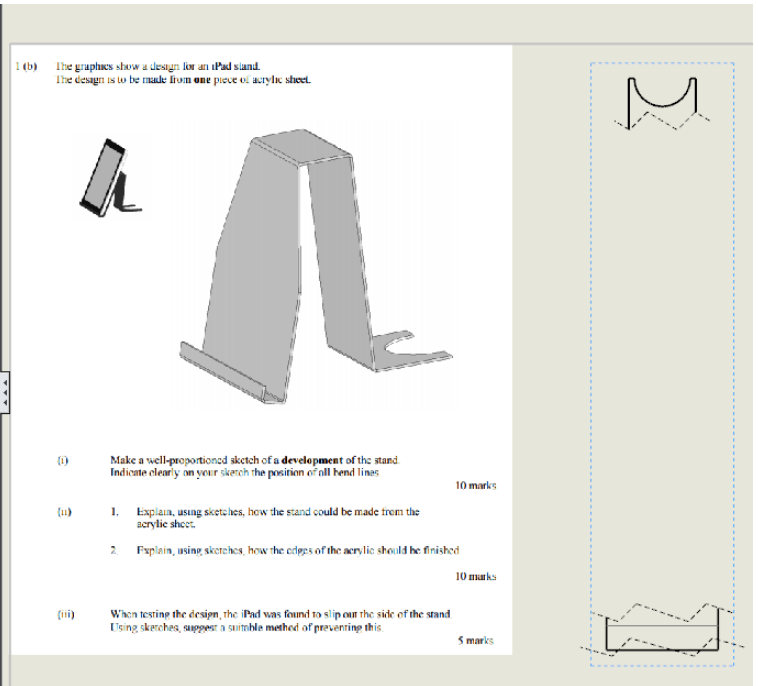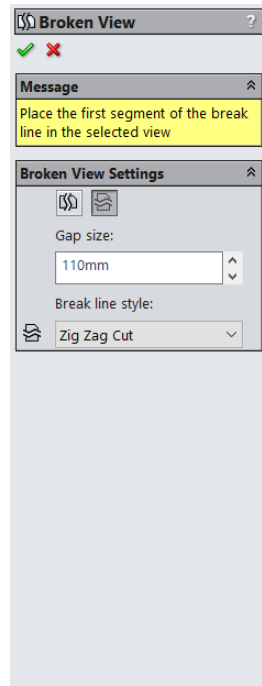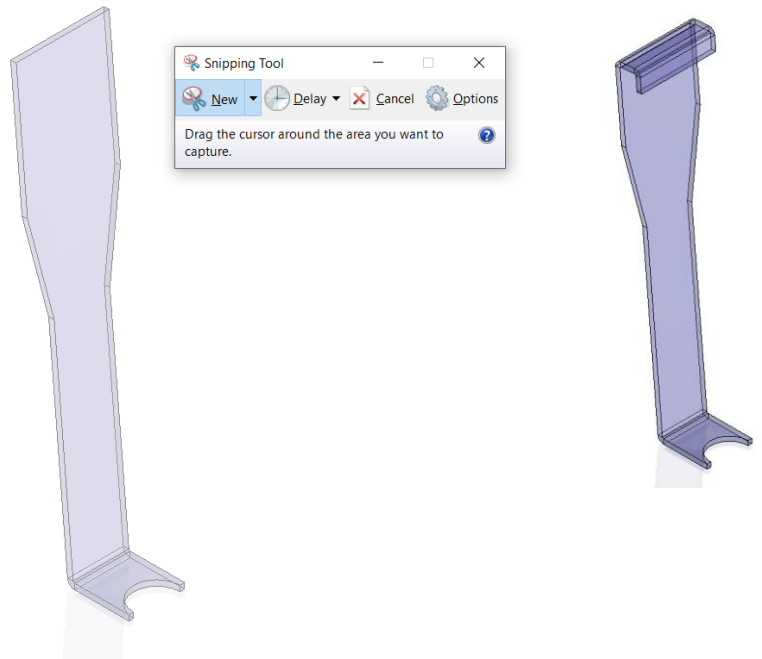Place the first segment of the break line in the selected view

**Broken View Settings**

Gap size:

110mm

Break line style:

Zig Zag Cut



1 (b)    The graphics show a design for an iPad stand.
The design is to be made from **one** piece of acrylic sheet.

(i)    Make a well-proportioned sketch of a **development** of the stand.
Indicate clearly on your sketch the position of all bend lines

10 marks

(ii)    1.    Explain, using sketches, how the stand could be made from the acrylic sheet.

2.    Explain, using sketches, how the edges of the acrylic should be finished

10 marks

(iii)    When testing the design, the iPad was found to slip out the side of the stand.
Using sketches, suggest a suitable method of preventing this.

5 marks

Part (i)
Complete development

**TECHNOLOGY SUBJECTS SUPPORT SERVICE**

Junior Certificate Technology

TITLE:

**Phone Holder**

DRAWN BY: YOUR NAME HERE    SET AA    UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm

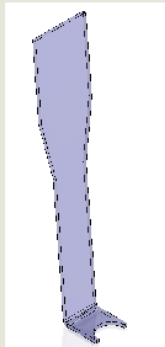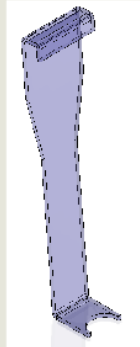SCALE: 1:1    DATE: 20Feb18    SHEET 2 OF 2

## Add another sheet for Part (ii)

Save a number of roll back views using the windows Snipping Tool Insert them into the drawing sheet.

Add some text to complete the worksheet. Add a sketch line to divide the sheet





Part (ii) A

Explain how to out out the shape from acrylic:

Bending the Acrylic



Step 1
Bend outside faces

Step 2
Bend 2 faces
at front to form lip

Step 3:

Draw a neat, labelled sketch of **Equipment** used to form Bends

Junior Certificate Technology

TITLE:

DRAWN BY: YOUR NAME HERE

SCALE: 1:1     DATE:   20-feb-16

Further sheets can be added to complete the other parts of the question

The exercise is now complete.

Save your work