Advanced CAD Modelling Course
(SolidWorks 2009)

Published by:
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Please note

- Screenshots used in this manual may appear different to those on computer screens used by participants; variations in versions of the software and differing operating systems may be in use.
- Screenshots and software titles used throughout the manual are from a PC using Microsoft Windows XP©.
- Participants using other operating systems may encounter some differences in screen presentation and layout.

Throughout this module reference may be made to software titles and suppliers of Internet services. These references are made purely to illustrate or expound course content. Any such reference does not imply any endorsement by the NCTE of a product or company. The reader should be aware that typically there are many products and companies providing similar services in areas related to ICT. Participants should be as informed as possible before making decisions on purchases of ICT products or services.

Advanced CAD Modelling Course – SolidWorks 2009

Table of Contents

Advanced CAD Modelling Course – SolidWorks 2009

Introduction

The revised syllabuses for Design and Communication Graphics (previously Technical Drawing) and a new subject at Leaving Certificate level, Technology, were introduced to the senior cycle curriculum in September 2007 and were examined for the first time in 2009.

A Technology Subjects Support Service (T4) was established to support schools in the implementation of the revised/new syllabuses. The support service has rolled out the intensive phase of a professional development programme for teachers that consisted of a nine day in-service programme over three years. A significant component of this professional development was focused on Computer Aided Design (CAD). Technologies and methodologies including powerful design tools such as parametric CAD are being utilised in the revised and new technology subjects and forms part of a significant assessment component in Design and Communication Graphics. Indeed, SolidWorks parametric CAD software is being used in second level schools providing the technology subjects at both senior and junior cycle. The application integrates the development of Technology and IT skills with a variety of basic and advanced features.

Despite the very successful roll-out of the T4 professional development programme a request for additional CAD and other ICT courses have been made by the teachers of the technology subjects. This request was prompted in part by the upgrade of SolidWorks 2006 to the 2009 Education Version which was made available to all schools in March 2009. In response, the NCTE in collaboration with T4 and the Education Centres Network hope to satisfy the demand by providing additional courses and opportunities for further professional development. The Advanced CAD Modelling Course (SolidWorks 2009) is complementary to the work of the T4 professional development programme and the other three CAD Modules that were developed by the NCTE in collaboration with the Technology Subjects Teacher Professional Network.

The Advanced CAD Modelling Course has been designed to allow participants to do all or some of the exercises in the four broad areas covered – Sheet Metal Features, Curves, Surfaces and Photorealistic images. It is envisaged that this course will be delivered over 15 hours (6 evenings x 2.5 hours) but the pace of progress will be determined by the skill level of the participants. Each group is recommended to discuss the course schedule with the tutor in order to reflect their unique needs, interests and ability level. Prior to commencing the Advanced CAD Modelling Course, it is assumed that participants will have completed CAD Module 2 and 3 and have developed a broad base of basic ICT skills. An opportunity will be provided for participants to share resource materials developed during the course.

While this course is limited in the range and depth of topics it can cover, the NCTE has a wider and more detailed range of courses which address other areas of ICT. Details of these courses can be found at www.ncte.ie/training or through your local education centre. A more extensive range of CAD resource that was developed during the course of the T4 professional programme is available on www.t4.ie or by contacting your local T4 Regional Development Officer.

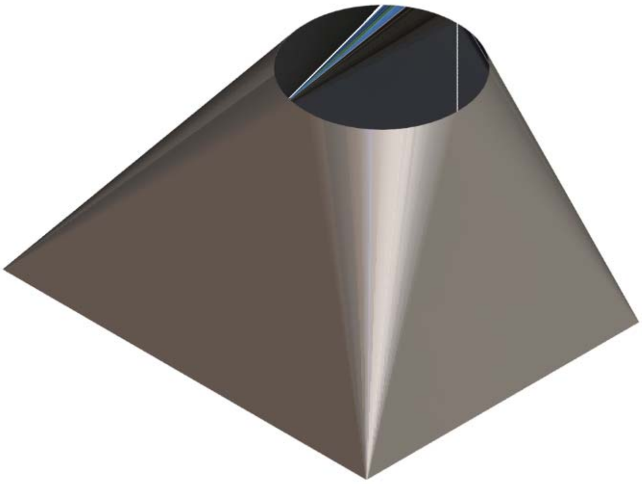
Duration

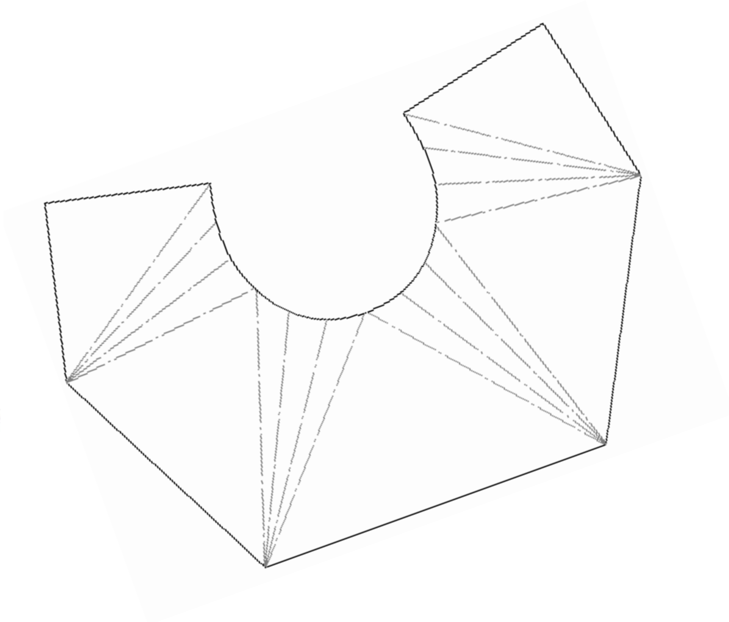
15 hours

Objectives

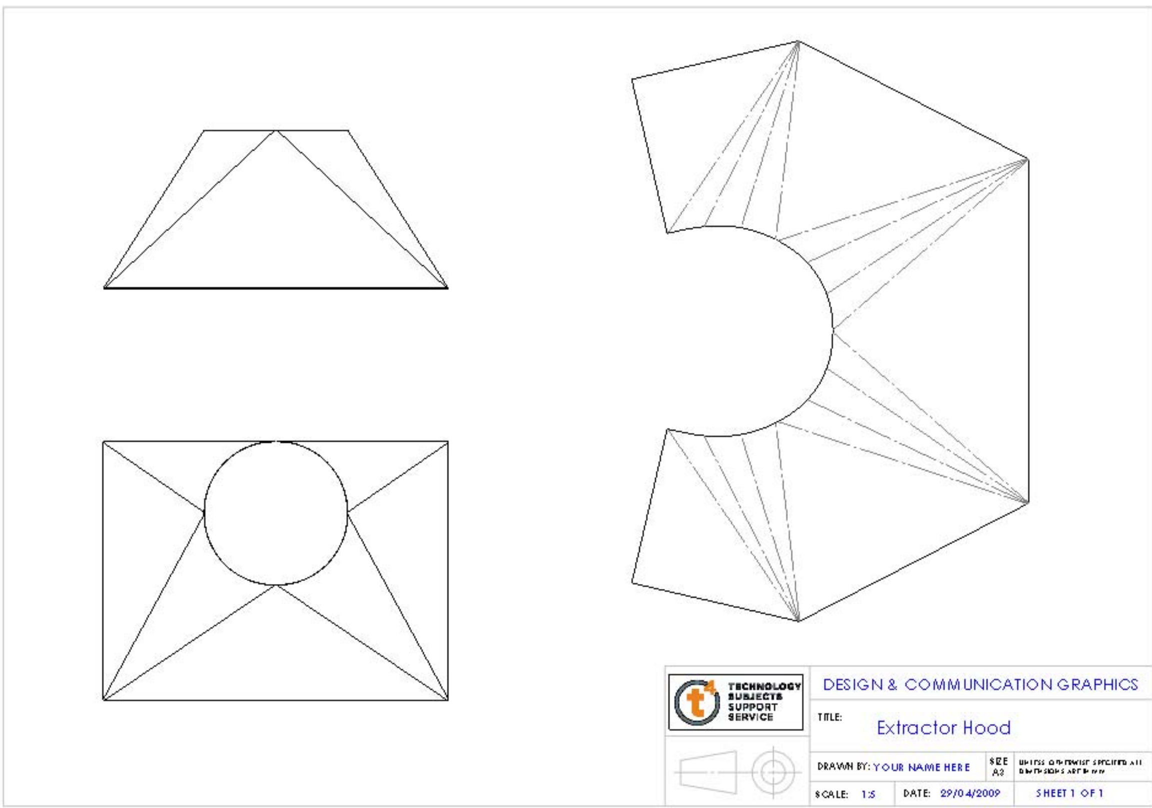
This course aims to enable the participant to:

- Build on and reinforce the CAD skills developed to date
- Explore the more advanced features and tools within SolidWorks 2009 by:
  - Building sheet metal type models and creating their development by flattening the models using the Sheet Metal features
  - Creating 'Curved Features' which explores the uses of splines, helices, and composite curves to model everyday items and geometric problems
  - Using the Surface toolbar to create faces and features which may not be conveniently produced using solid modelling techniques
  - Producing high quality photorealistic images using PhotoWorks and PhotoView 360 of CAD models
- Examine the application of these advanced CAD features in enhancing the teaching and learning of the core geometry and its applications

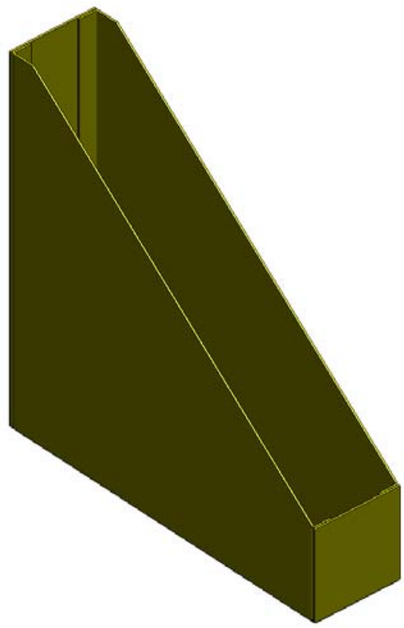




Introduction to Sheet Metal Features
SolidWorks 2009

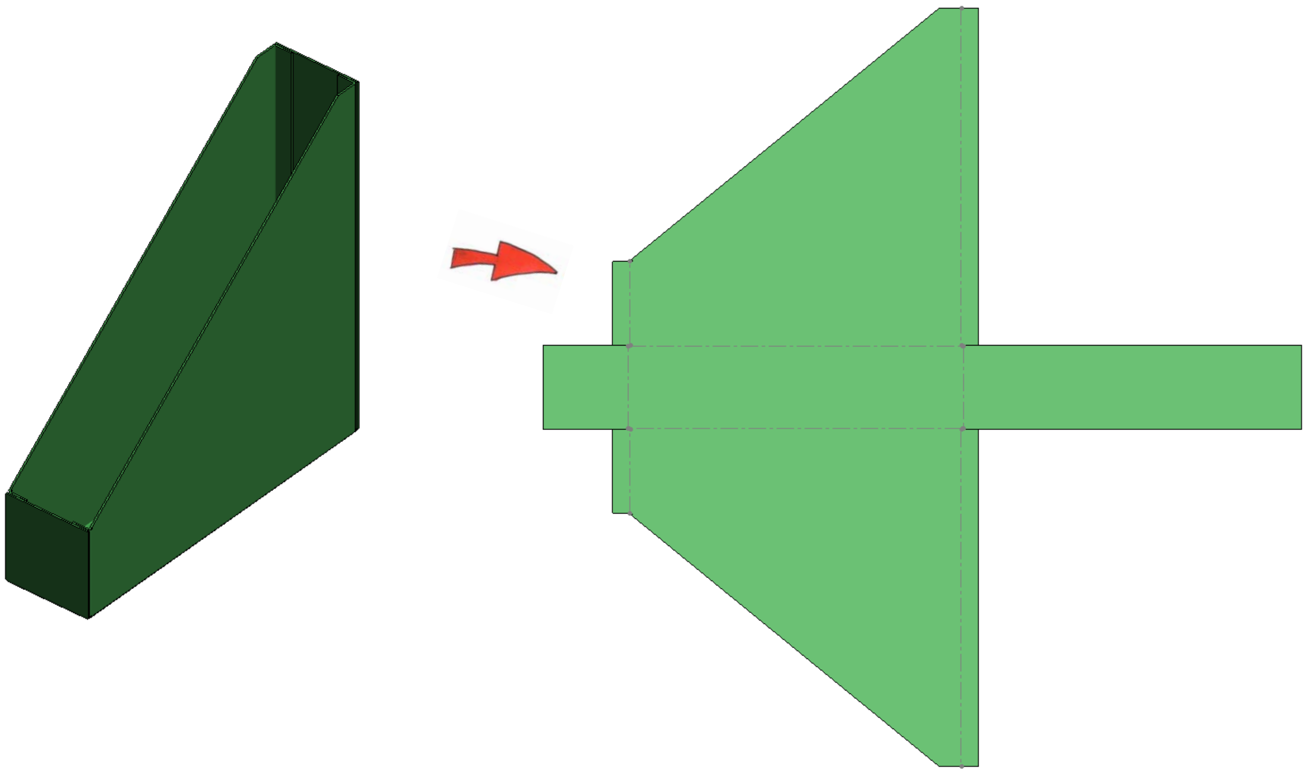


BASE FLANGE METHOD - MAGAZINE FILE.



Sheet Metal

The sheet metal feature within SolidWorks enables the user to build a sheet metal model, using a variety of sheet metal features. The development of the model can be created by flattening the model as a whole or by flattening individual bends.



Prerequisite knowledge

To complete this model you should have a working knowledge of Solidworks 2006/2009.

Focus of lesson

This lesson focuses on using the base flange approach to sheet metal. Commands used include ***Base Flange, Edge Flange, Corners and Extruded Cut.***

Getting started.
New File

Create a new part file.

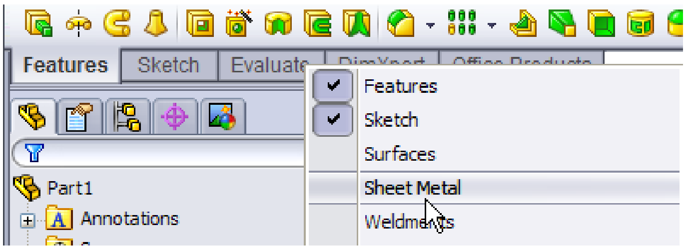
Save File

Save the file to a chosen location as **Magazine File**.

Getting Started

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

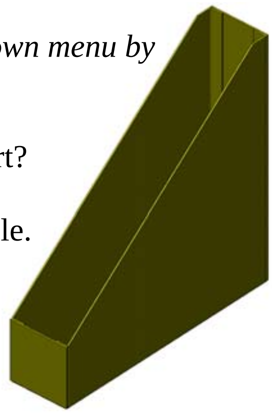
To activate this tab, right click on the command manager. Choose **Sheet Metal** from the drop-down list.



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**"…*



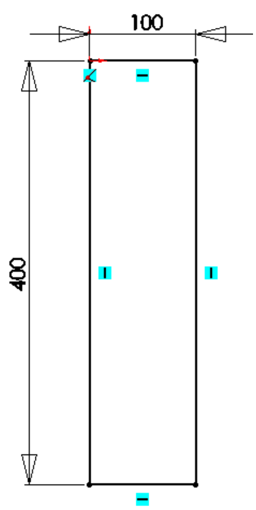
Creating a sketch:

How do we start to model the magazine file as a sheet metal part?

We will begin by creating a sketch to generate the base of the file.

What plane will this sketch be created on?

Because the file sits on the horizontal plane we will create a sketch on the Top Plane.



Sketch:

Create a rectangular sketch on the top plane placing the top left hand corner coincident with the origin.

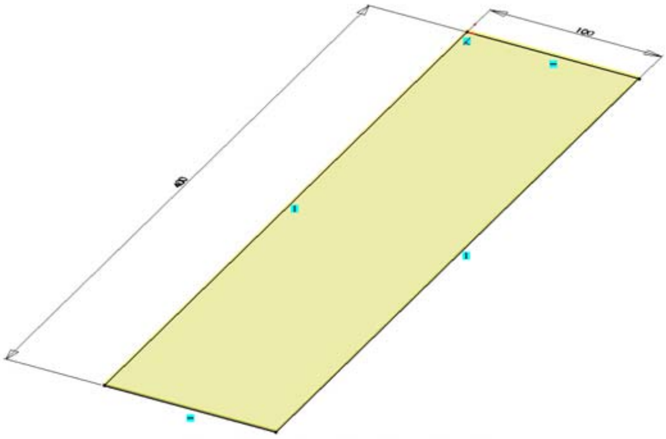
Smart dimension the rectangle as shown.
Height – 400mm & Width – 100mm

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 1.50mm for **thickness** in the Base Flange options dialog box

Click Ok



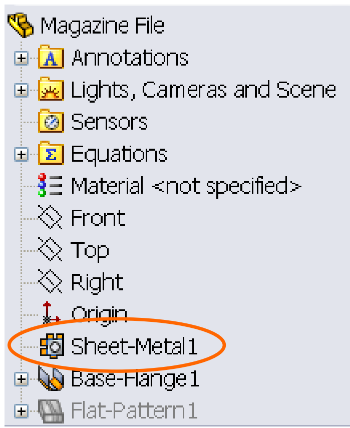
About Base Flange

When a base flange feature is created SolidWorks immediately recognises this part as a sheet metal part.

Only one base flange feature may be inserted for each sheet metal part document.

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

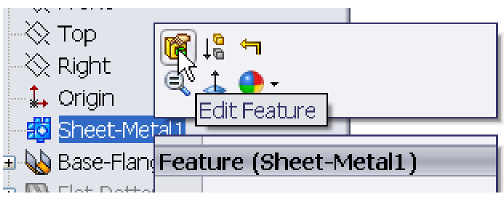
Sheet-Metal1: is automatically added above the Base flange feature. It holds the default sheet metal settings such as sheet metal thickness, radius etc.



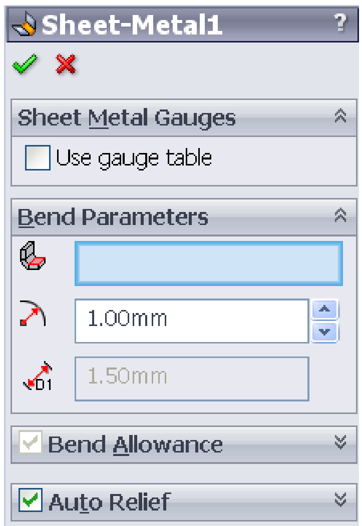
Sheet-Metal1 will remain at the top of the feature manager design tree

Sheet-Metal 1

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



The sheet metal settings may be changed here.



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.

Adding the vertical faces

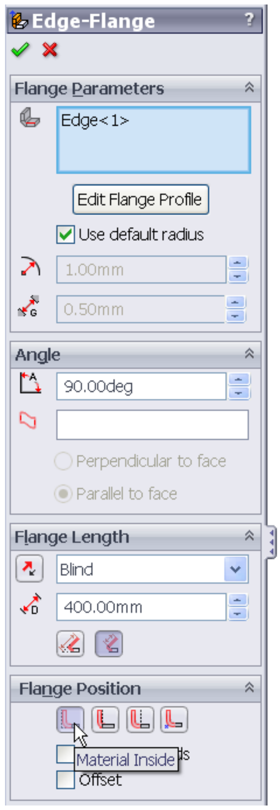
We will create the vertical faces using **Edge Flange** 

Edge Flange

Edge flange is used to create a 90° bend to a selected edge, in the direction and distance specified, using the thickness of the part. The shape of the flange by default is rectangular. This may be edited to a custom profile also.

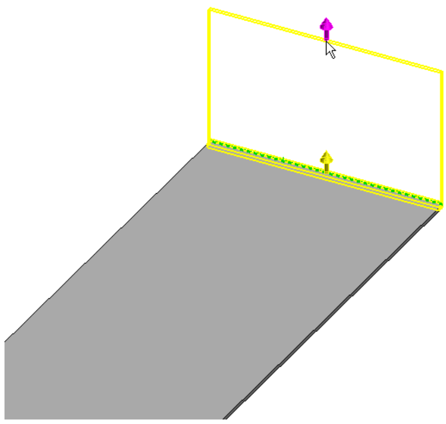
Adding an Edge Flange

Select **Edge Flange**  from the sheet metal toolbar.



Choose the back edge of the **base flange** as the edge on which you wish to create the **edge flange**.

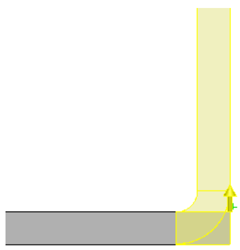
Drag the edge upwards and left click to indicate direction and an initial value for length.



The **default radius** of **1mm** is used. The **gap distance** is greyed out as there is only one selection.

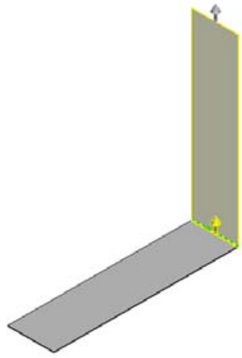
The **flange angle** is set to **90°.**

Enter a value of **400mm** for **Flange Length**



Choosing **material inside** will ensure that the face of the edge flange, when bent, will be flush with the original edge of the base flange, as indicated opposite. A preview of the proposed flange is displayed.

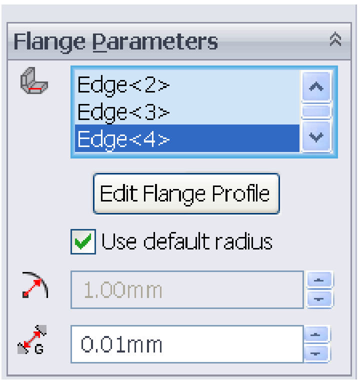
We wish to add further selections to the feature.

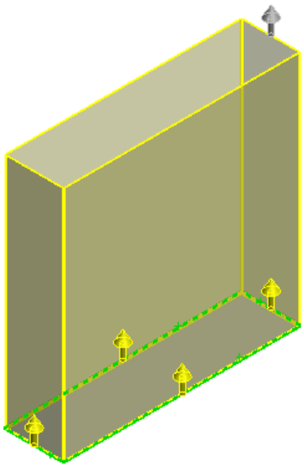


Adding further edges.

Choose the edges indicated to create further edge flanges, using the same parameters.

Enter a **Gap Distance** of **0.01mm. Gap distance** refers to the distance between adjacent edge flanges and must be greater than **0mm**.





Gap Distance

Click **OK**.

Creating cut edge:

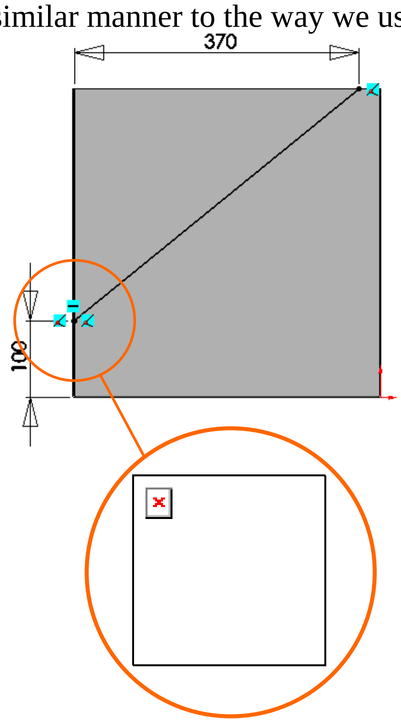
In order to complete the shape of the magazine file we must cut a section from the rectangular prism which we have created.

Extruded Cut

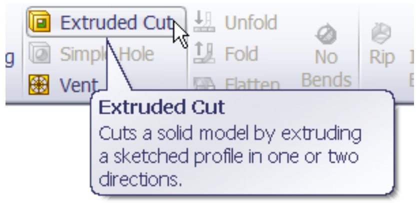
Extruded cut within Sheet Metal is used in a similar manner to the way we use it in dealing with solid models.

We will begin by creating a sketch of the profile used to create the cut on the right face of the box.

Note: the enlarged detail of the sketch shows a horizontal line coincident with the edge of the front face and the endpoint of the inclined line on the right face



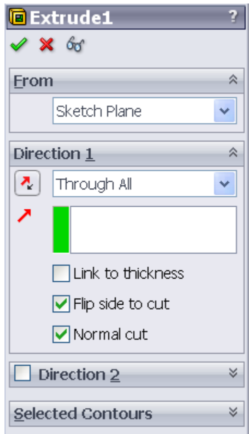
Select **Extrude Cut** from the **Sheet Metal** tab.

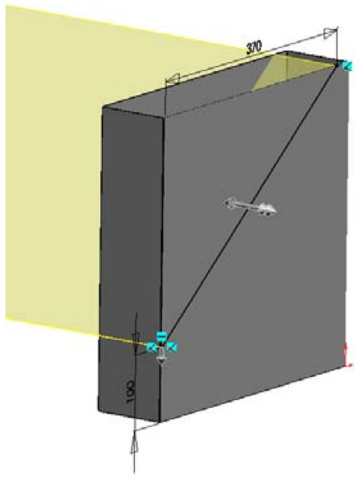


The line sketch is automatically selected. From the options list deselect the **Direction 2** box.

In the direction 1 box select the **Through All** end condition.

Select "**Flip side to cut**", if necessary, to remove what is above the plane and keep what is underneath.





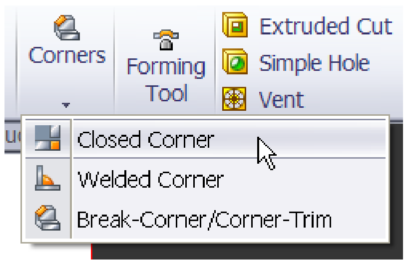
Choose **OK**

Corners

Zoom into the lower right hand corner.
You will notice that the corner is open.

The faces may be extended to close the corner using the **Corners** feature on the **Sheet Metal** toolbar.

Choose **Corners, Closed Corner.**

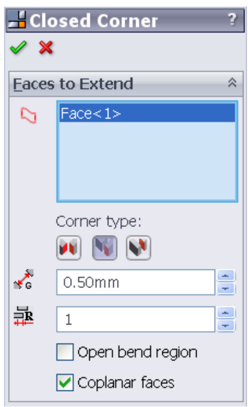


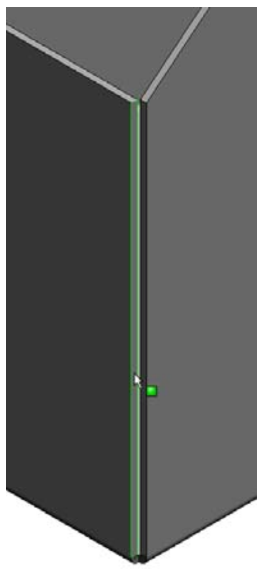
Select the face indicated.

Choose a **gap distance** of **0.5mm**

Select **Overlap** as **corner type**

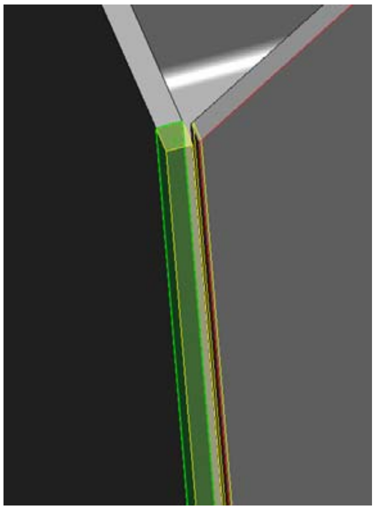
An **overlap ratio** of **1** will ensure that the two faces overlap completely.





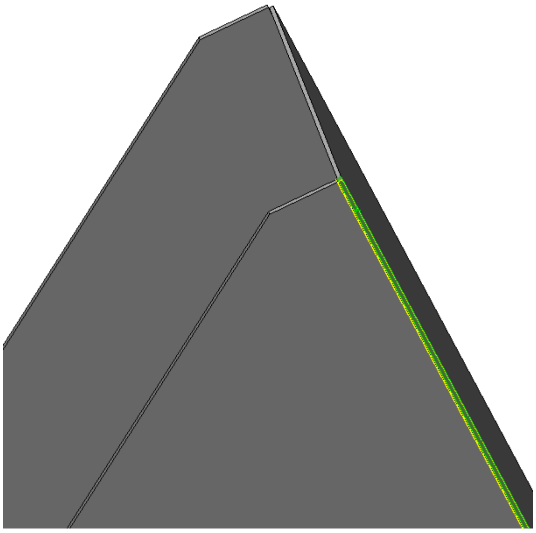
A preview will be displayed as shown opposite.

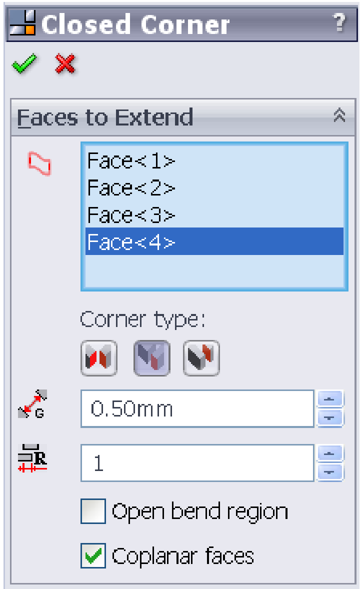
Note: The gap distance of 0.5mm can be seen clearly at this stage.



Selecting further faces

Rotate the model and choose the face as indicated below.

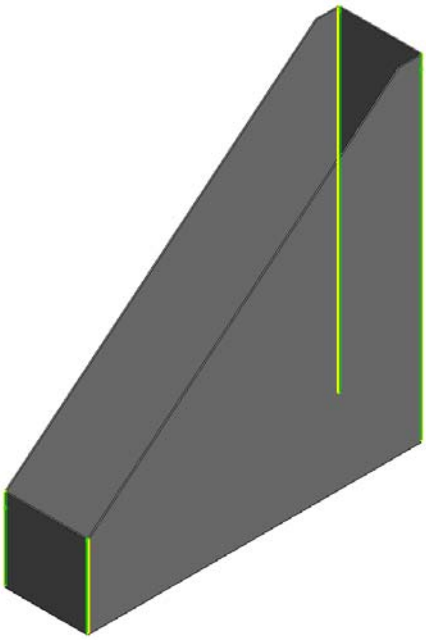


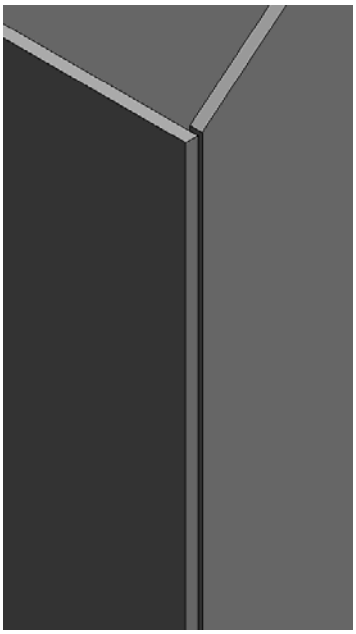


Rotate the model and choose the corresponding faces on the opposite side.

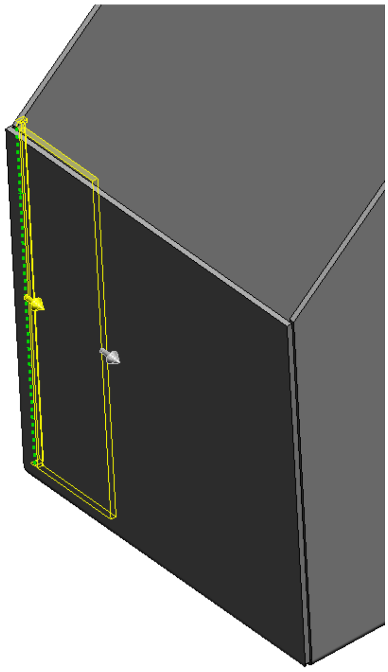
In total you should now have 4 faces selected.

Choose **OK.** The corners now appear as shown below.





Adding tabs To complete the part, tabs must be added to vertical edges of the side pieces.



We will use **Edge Flange** to add these tabs.

Choose **Edge Flange** and select the internal corner of the vertical side.

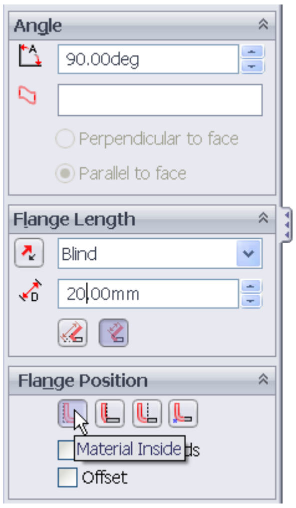
Drag the corner in the direction shown and left left click to assign an initial distance.

Input the following;

Angle - **90°**

Flange Length – **20mm**

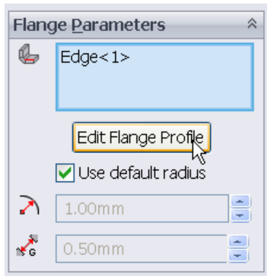
Flange Position – **Material Inside**



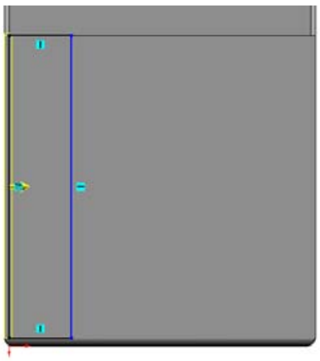
Edit Flange Profile

Choose **Front View.**

Select **Edit Flange Profile** from the **Flange Parameters**

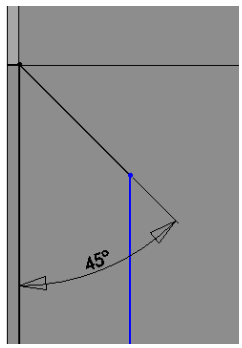


The sketch used to create the flange appears along with the **profile sketch dialog box** shown.



We must edit this sketch in order to edit the shape of the flange.

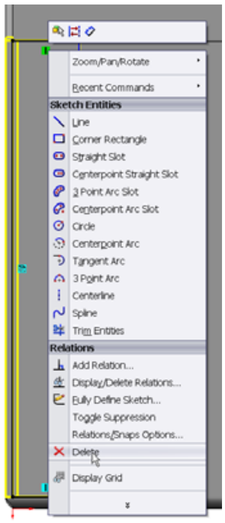
We are going to edit the sketch so that the top is chamfered at 45°, as shown.



How will we achieve this?

Editing the sketch.

We must first remove the automatic relations from the line.

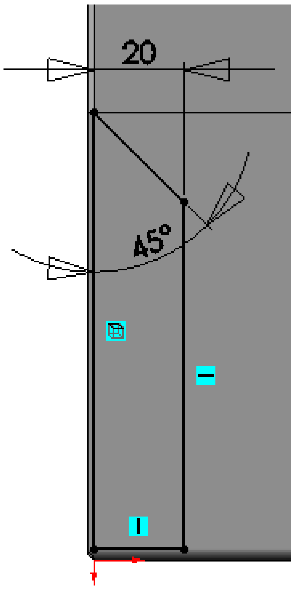


To delete the relation; right click on the relation icon, displayed in green when highlighted, and select delete from the dropdown list. Alternatively choose delete from the keyboard.

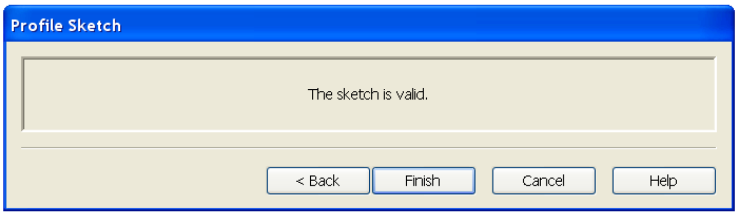
Removing the relations allows us to add angular dimensions to the line.

Smart Dimension

Smart dimension the angle as shown. Add the length of the flange, 20mm.



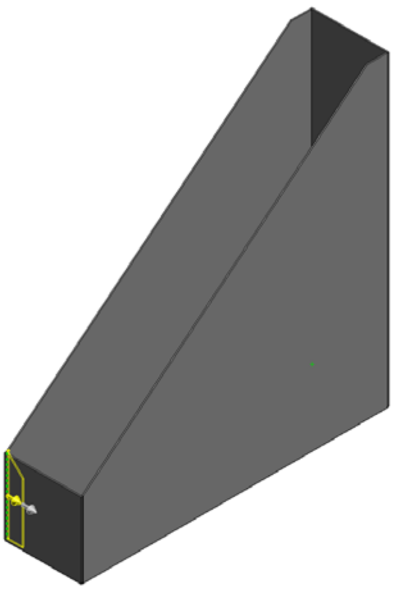
The **profile sketch dialog box** will indicate whether the sketch may be used to create the flange or not.



Choose **Finish**

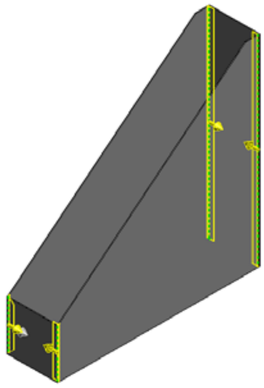
A preview of the customised tab is displayed.

Choose **Isometric View.**



Selecting other edges

We want to add a similar tab to the other 3 internal corners.

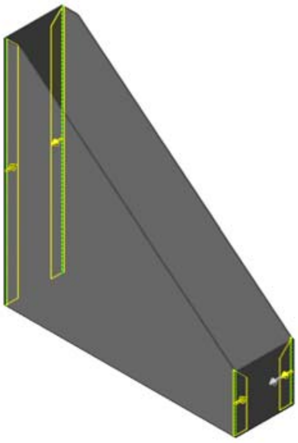


Using rotate and zoom commands select the remaining corners, as indicated opposite.

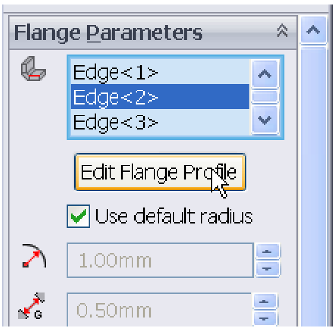
Unfortunately, each of the sketches defining the Individual flange profiles must be edited separately.

Choose **Edge 2** and select **Edit Flange Profile**

Edit the profile as before.



Repeat the procedure for edges 3 & 4.



Choose **OK.**

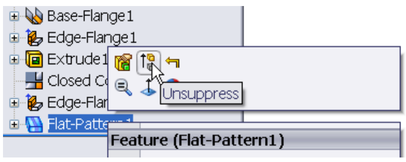
Flat-pattern

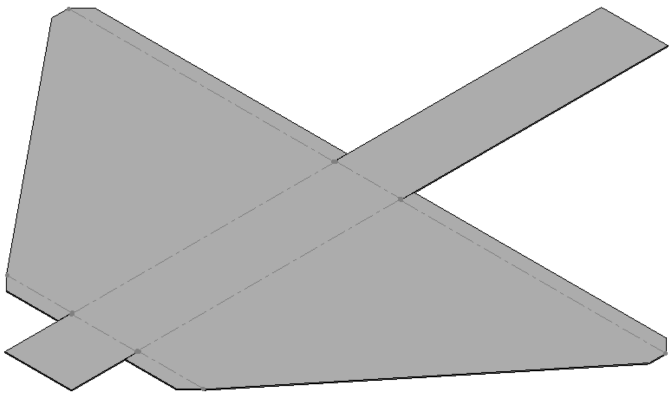
Remember the **Flat-pattern1** feature discussed earlier?
It is added to the bottom of the feature manager design tree when we create a sheet metal part. As sheet metal features are added to the part it remains at the bottom. You will also notice that it is greyed out or **suppressed.**

Let's see what happens when we unsuppress it!

Unsuppress Flat-pattern

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

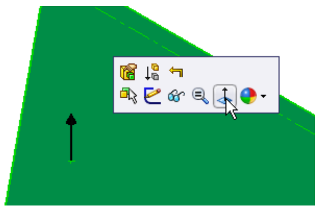




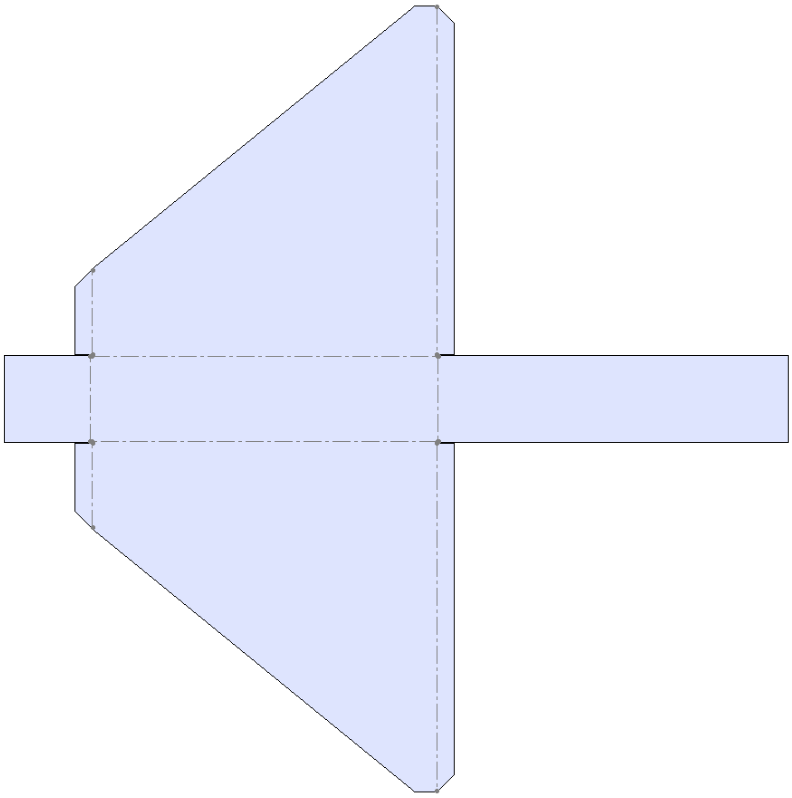
The sheet metal model flattens out into the surface development used to create it.

All of the bend lines are displayed.

Select the top face and choose **Normal To**

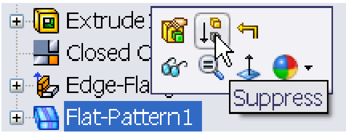


Complete surface development



Suppress

Left click on **Flat-Pattern1** and choose **Suppress** to return to the sheet metal



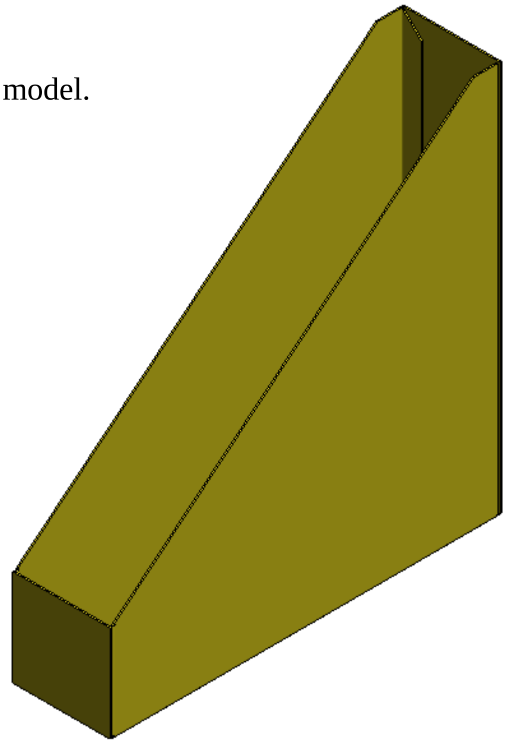
Add appearance:

Add an appropriate appearance to the model.

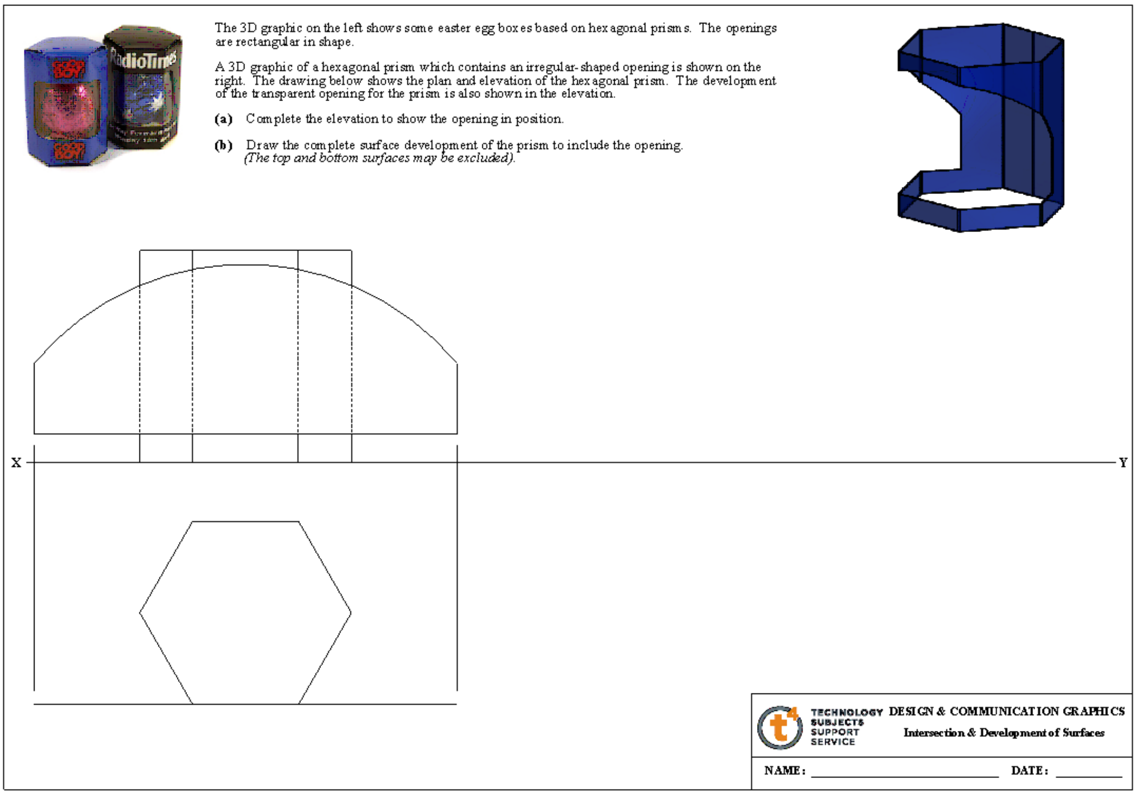
Choose **Yellow Low Gloss Plastic**

Save the completed part

Lesson Complete!



Envelopments and Developments of Surfaces



Prerequisite knowledge: Exercise 1 - **Magazine File** should be completed in advance of this exercise.

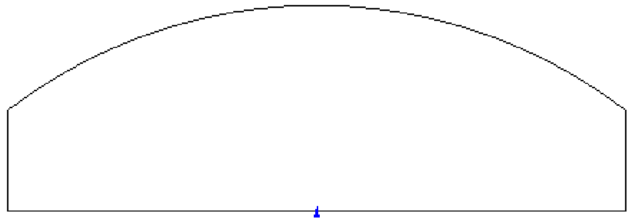
Focus of lesson: To further explore the tools available within sheet metal and investigate how these tools may be used to enhance the teaching of geometry.

Commands Used: This lesson includes Sketching, Base Flange**,** Flatten, Extruded Cut, **Fold and Unfold.**

Problem: A worksheet is presented based on a hexagonal prism and irregular shaped opening.
The hexagonal prism, shown overleaf, is to be produced complete with the given window removed. The solution requires us to generate the hexagonal prism, unfold it, add the true shape of the cut out to the development and then refold the model. The orthographic views will be generated from this model.

We will omit the top and bottom surfaces of the prism from the model as the problem does not require them.

We will explore the use of SolidWorks in completing the problem, focusing on sheet metal features

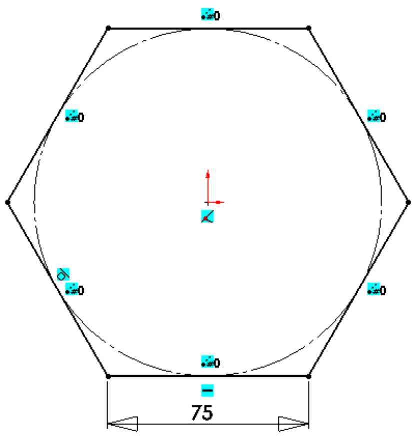


How will we create the solution?
To complete the solution to this problem we must first create the development of the hexagonal prism.

Getting started:

We will begin by creating the model of the prism and then derive the development from it. To start we will draw the base profile of the prism, a hexagon.
Create a new SolidWorks part document and save it as **Hexagonal Prism**.

What Plane will we sketch on?
As the hexagonal prism sits on the horizontal plane we will begin sketching on the **Top Plane.**



Sketch:

Create a hexagonal sketch on the top plane, placing the centre of the hexagon coincident with the origin.

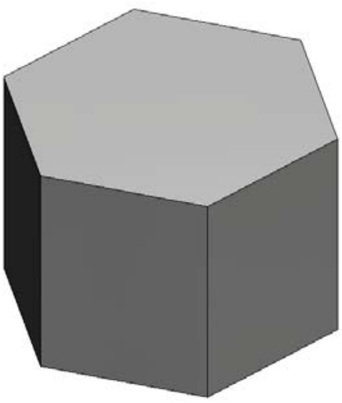
Add Relation

Add a **horizontal relation** to one of its sides, and a side length of 75mm.

These three pieces of information are needed to fully define the sketch

Base Flange

If a base flange is created using this closed sketch it will not allow us to create the thin walled sheet metal hexagonal prism we require.



Instead it will generate a solid prism as shown.

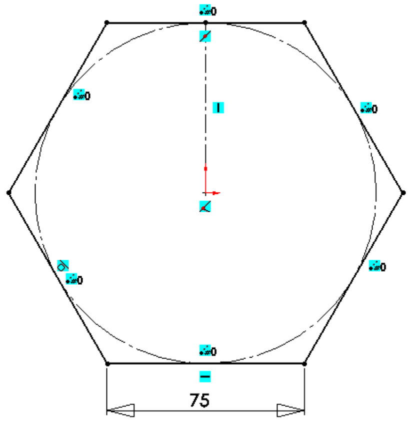
In order for SolidWorks to create the sheet metal container, a joint edge, or break, must be placed in it.

This joint will be the position from which the prism will later be developed.

Breaking the sketch

How will we create a break in the sketch?

We will use the **trim** command to create the break in the sketch.



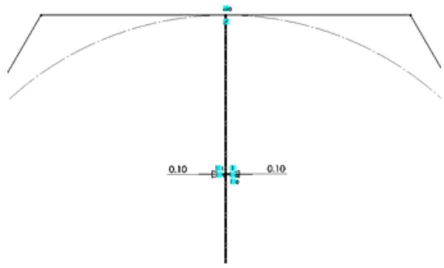
Centreline

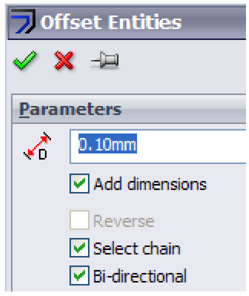
Select **Centreline** and draw a line from the origin to the midpoint of the top edge of the hexagon.

Offset

Offset this line **0.1mm**.

Choose **Bi directional**. This will offset the line at both sides of the original.





For Construction

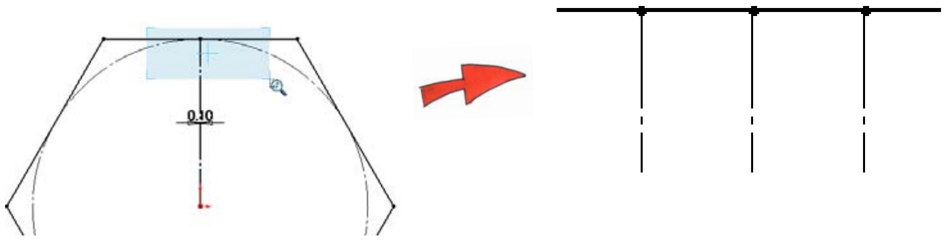
These lines are for construction purposes only and will not form part of the feature afterwards. For this reason they must be marked as construction lines.

Zoom in, select the offset lines and choose **For Construction** from the **Line properties dialog box**. These lines will now change to a chain line type.

Trim

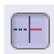
In order to trim the sketch we will need to zoom in.

Select the Zoom to area  icon

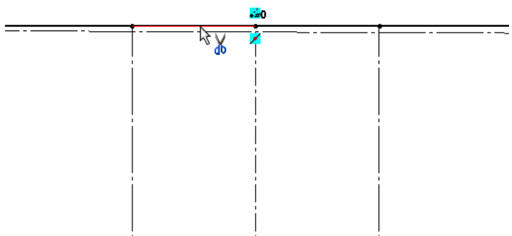


Trim (Contd.)

Choose **Trim Entities** 

Select **Trim to closest** 

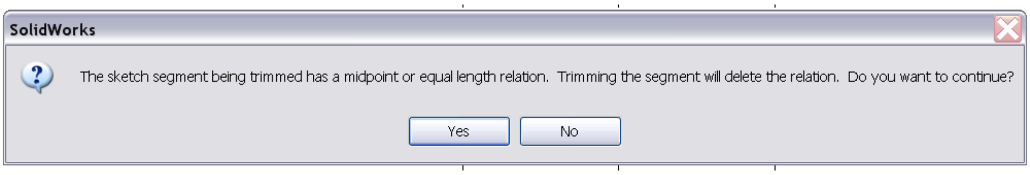
Select the portion of the hexagonal sketch which lies between the two offset lines.



SolidWorks warning

SolidWorks gives us a warning because the original centreline has a midpoint relation with the line we are trying to trim. Trimming the line will delete the relation.

Choose **Yes** when the SolidWorks dialog box appears.



Choose **OK**. Exit the sketch

Create feature:

What will we use to create the feature from the hexagonal sketch?

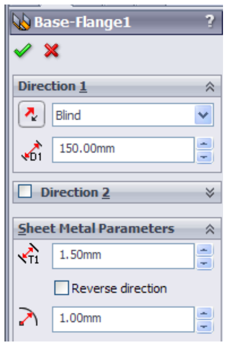
Base Flange:

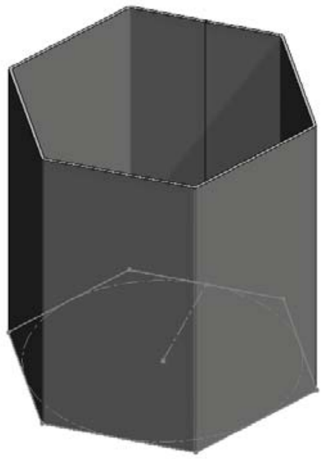
Select **Base flange** from the sheet metal toolbar.

Direction 1: **150mm**

Thickness: **1.5mm**

Bend radius: **1mm**





Adding Window feature

In order to create the sketch for the window feature we need to flatten the prism. In the previous exercise we **unsuppressed** the **flat pattern** feature to develop the model.

Unsuppressing the flat-pattern feature is not suitable in this instance. Why?

If we create a sketch on the surface of the unsuppressed flat-pattern feature and then extrude cut that sketch, the feature created will be added to the feature manager tree **below** the flat pattern feature.

Because the new feature is created below the flat-pattern feature it will not be displayed when the model is suppressed.

About Unfold/Fold

Unfold can be used to flatten a model, allowing you to create sheet metal features which cross bend lines eg. a hole. **Fold** takes the unfolded model in the flattened state and refolds it.

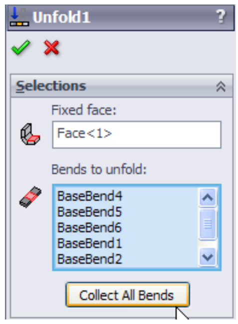
Any feature created in the unfolded state will appear above the flat-pattern feature in the featuremanager design tree and hence will appear when the flat-pattern feature is suppressed.

Unfold

Select Unfold [Unfold] from the sheet metal toolbar *or* choose ***Insert, Sheet metal, Unfold…***

Choose the following options;

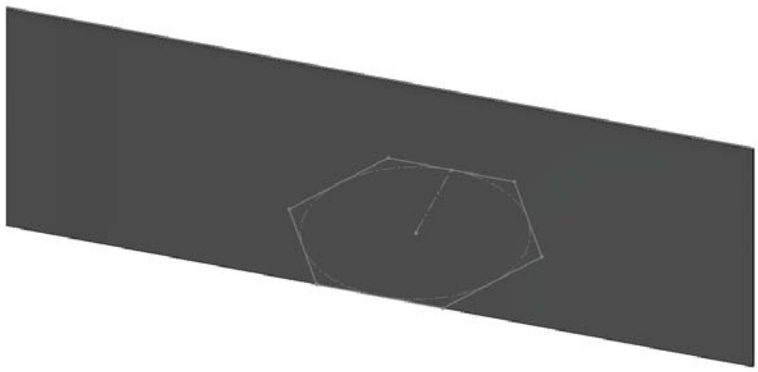
Fixed face: This will be the only surface which remains stationary. Choose the front face as shown.



Bends to Unfold: These may be selected individually from the graphics area or choose all bends.
In this case we wish to unfold all the bends so we select **Collect All Bends.**

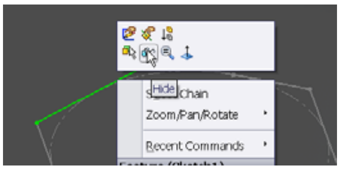
SolidWorks will automatically select all bends from the model. Choose **OK**

The model is now unfolded. Notice how similar unfolding is to flattening, however unfolding allows us to add sheet metal features and include them in the flat-pattern feature. Flattening does not allow us to do this.



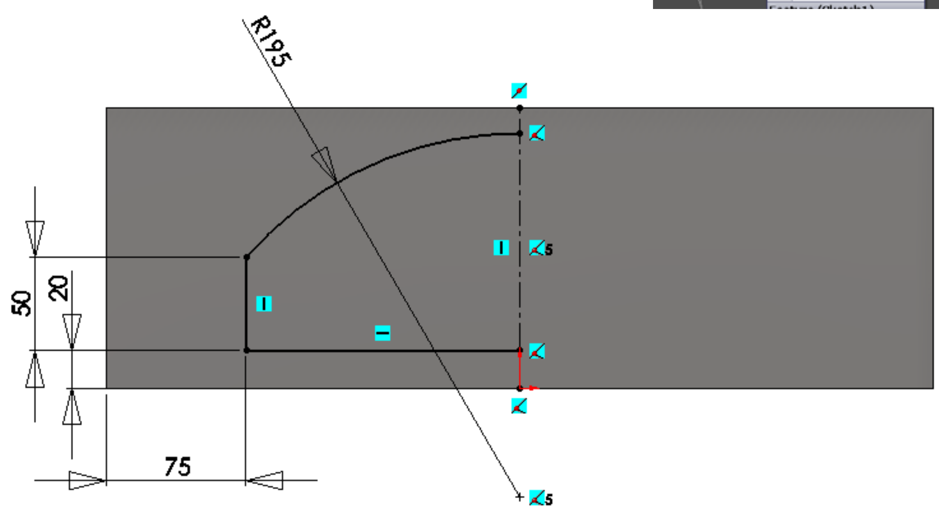
Hide Sketch

The sketch used to create the base-flange may be hidden by clicking on it and choosing **hide**.



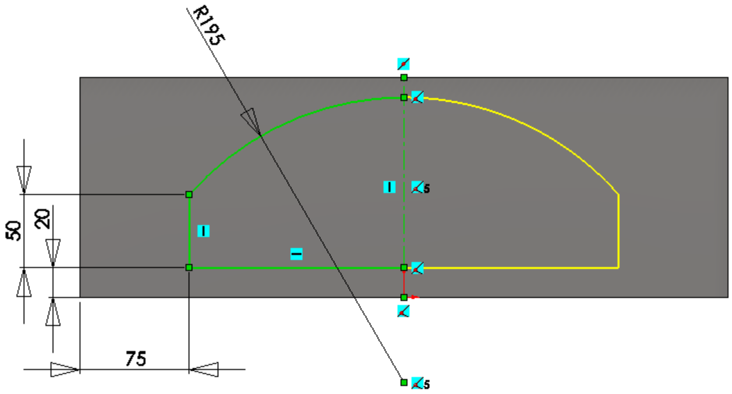
Sketch:

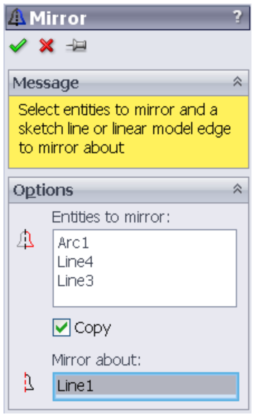
Create the following sketch on the front face of the unfolded sheet metal part.



Mirror

Choose **Mirror** . Mirror the sketch across the centreline





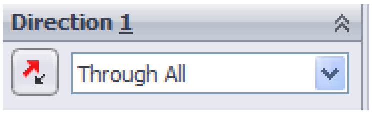
Choose **OK.** Exit the sketch

Cut Extrude

To remove the material from the development we will create an **Extruded Cut** feature.

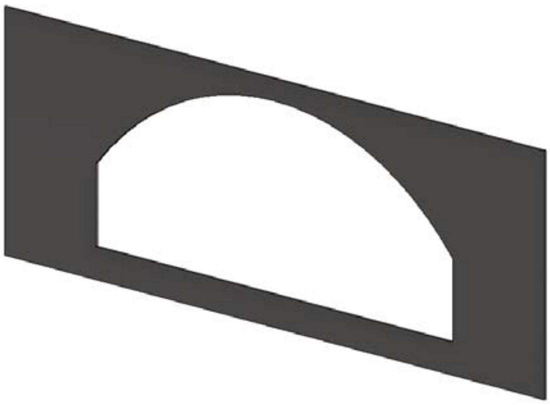
Choose **Extruded cut** from the sheet metal toolbar.

Choose the sketched window.



Select the **Through All** end condition.

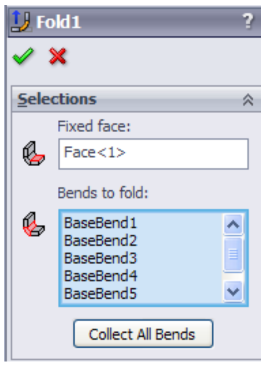
Click OK.



Refold

To refold the prism select **Fold** Fold from the sheet metal toolbar *or* Choose **Insert, Sheet metal, Fold** from the drop-down menu.

Now that the window has been added to the model, we can refold the model.

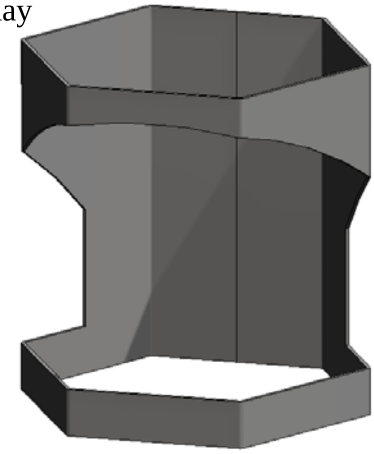


Folding the hexagonal prism follows a similar procedure to unfolding it.

We must indicate which surface is to remain stationary and which bends are to be folded.

Fixed Face

By default SolidWorks chooses the same **fixed face** as was used to unfold the model. A different face may be chosen if you wish.



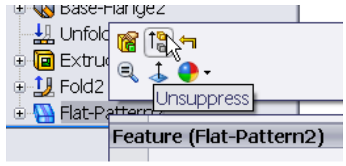
Bends to fold

Choose **Collect All Bends.**

The model will fold along the bend lines and the folded model will include the cut-out.

Flatten the model

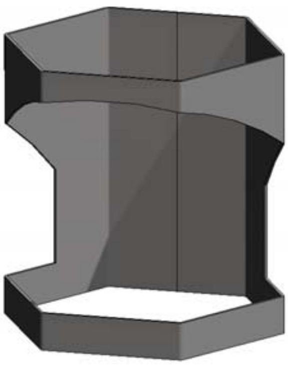
Right click on the flat-pattern feature and choose **unsuppress** 

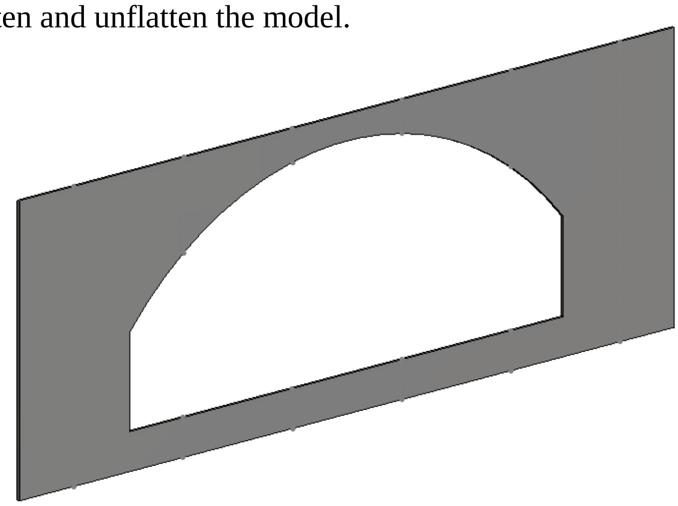


or choose flatten 

from the sheet metal toolbar.

Note: This tool will both flatten and unflatten the model.

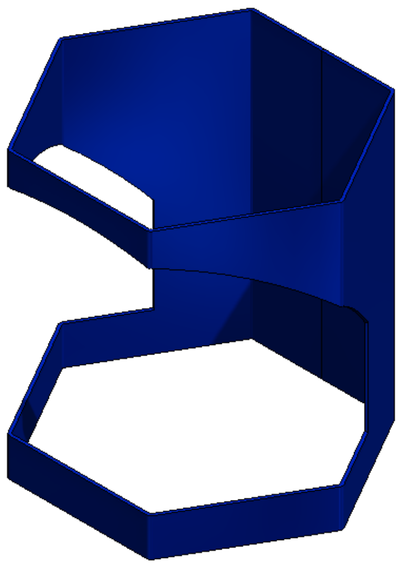




Edit Appearance:

Choose **Blue Medium Gloss Plastic**

Save the completed par.t



Creating a drawing:

As with any Solidworks part, a drawing may be produced by selecting **File, Create drawing from Part/Assembly** *or*

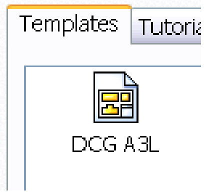
Choose  from the standard toolbar.

When a drawing is created from a sheet metal part, in addition to the standard views, the opportunity exists to create a view which displays the development of that part.

Create Drawing:

With the part file open select **File, Create drawing from part/assembly.**

Select **DCG A3L** as the drawing template you wish to use.



Standard Views

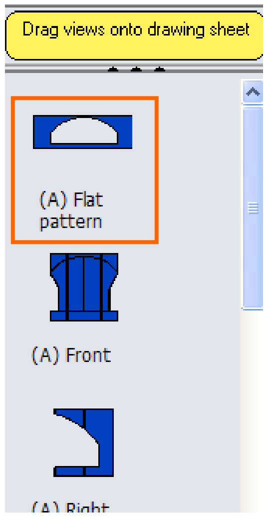
All of the standard drawing views are displayed in the **Task Pane.**

Any of these views may be dragged and dropped onto the drawing sheet.

Flat pattern

Flat Pattern is included in these views.

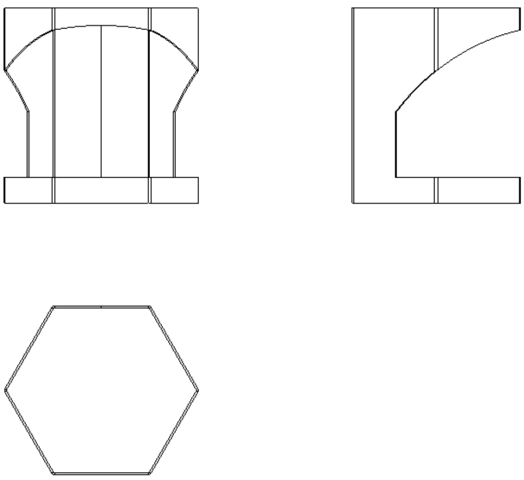
Dragging this icon into the sheet will generate a view displaying the development of the prism.



Creating the view layout

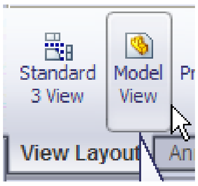
To generate the solution we want to create 3 orthographic views and add the surface development of the cut hexagonal prism.

Drag the **Front View** onto the sheet and project an End view and Plan view from the parent view. Use an appropriate scale.



Adding the surface development:

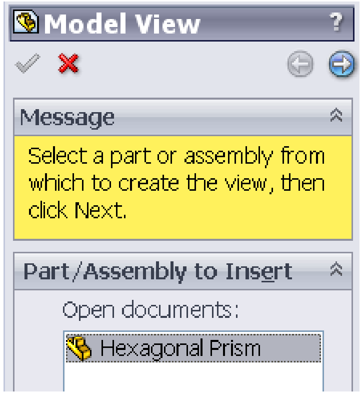
To add the surface development, select **Model View** from the **View Layout** toolbar.



Model View

Choose the hexagonal prism from the **Open documents** list in the Model View options dialog box**.**

Select **next** to proceed



Flat Pattern

Select the **Flat Pattern** option

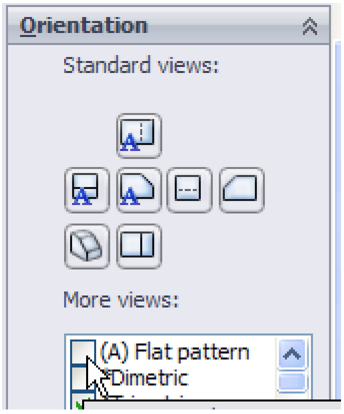
Position the flat pattern view on the drawing sheet.

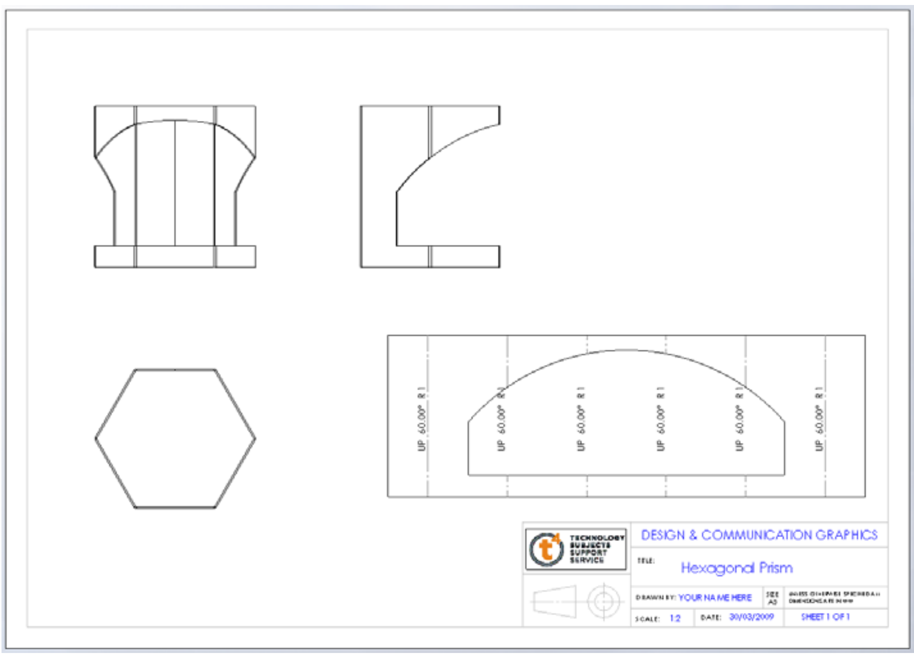
Choose **OK**.

Positioning the views

Drag the views to position them on the sheet.

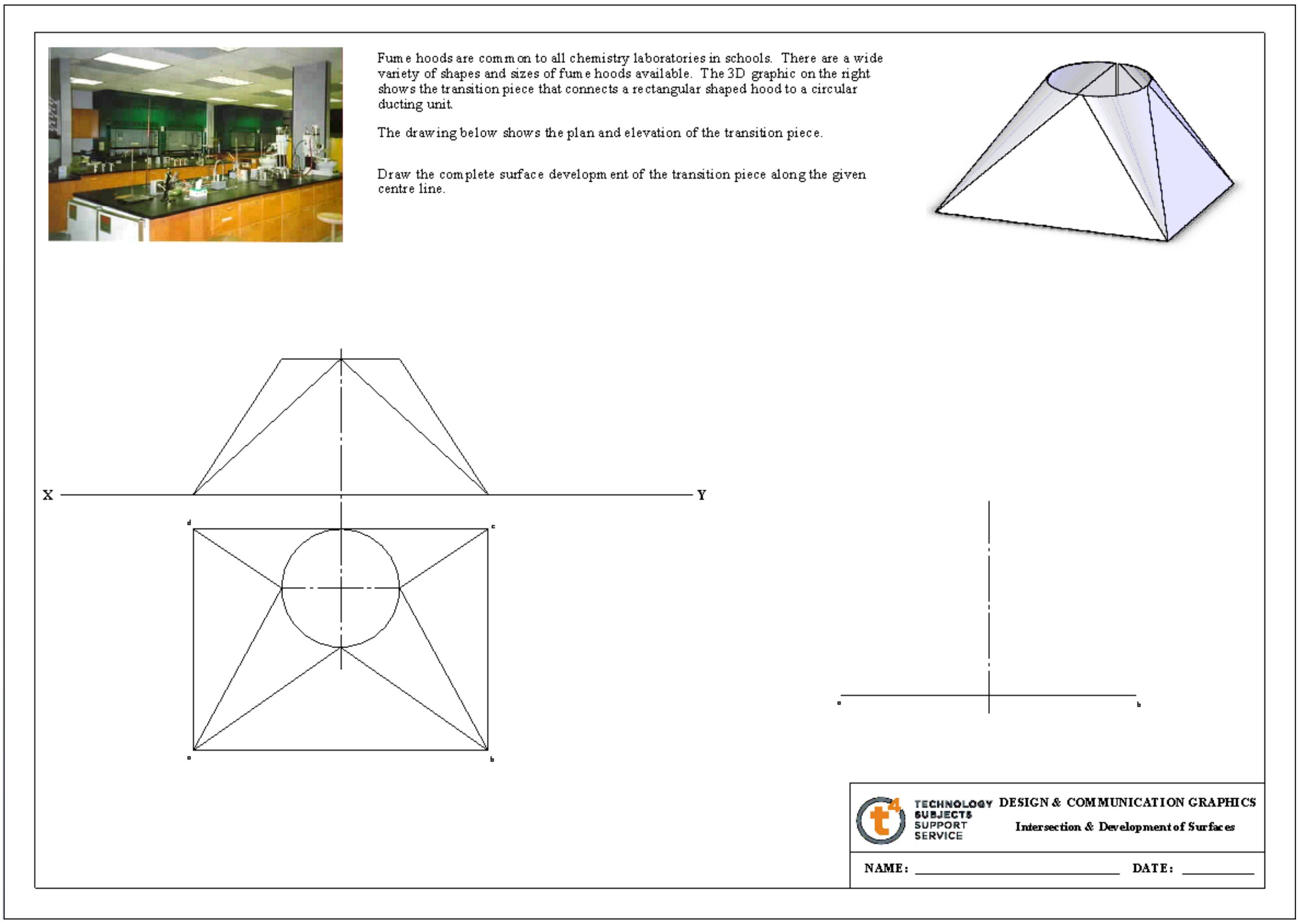
Note: The text may be removed from the surface development by selecting it and choosing **Hide**





Save all SolidWorks Documents. Lesson Complete!

Transition Piece Development – Extraction Hood

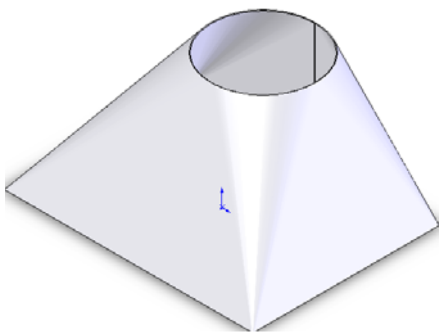


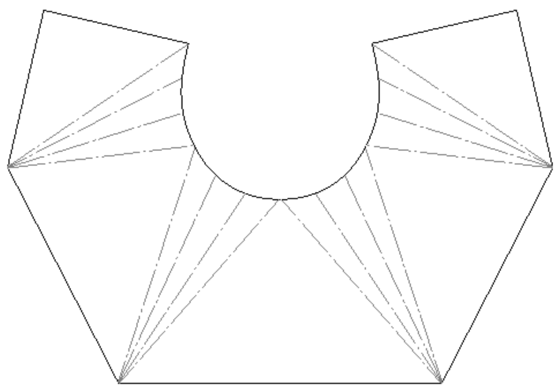
Prerequisite knowledge: Exercise 1 – "**Magazine File**" and Exercise 2 "**Easter Egg Box**" should be completed before attempting this exercise.

Focus of lesson: To further explore the tools available within sheet metal, through the completion of a transition piece exercise.

Commands Used: This lesson includes Sketching and **Lofted Bend, Fold** and **Unfold.** It works through the creation and development of a transition piece.

Development of transition part





Getting started.

New File Create a new part file.

Save File Save the file to a chosen location as **Extractor Hood.**

Creating a sketch:

How will we create the transition piece? *(Extractor Hood)*
The extractor hood is created in a similar way to the way that pyramids are created as Solidworks parts, as a loft.
Loft is also available in Sheet metal and is called **Lofted Bend** 

What is required to create a loft?
Previously, when we used lofts to create pyramids, we created two sketches; a sketch of the base profile and one of the top profile. We then used these two sketches to create a feature. The procedure is the same in sheet metal.

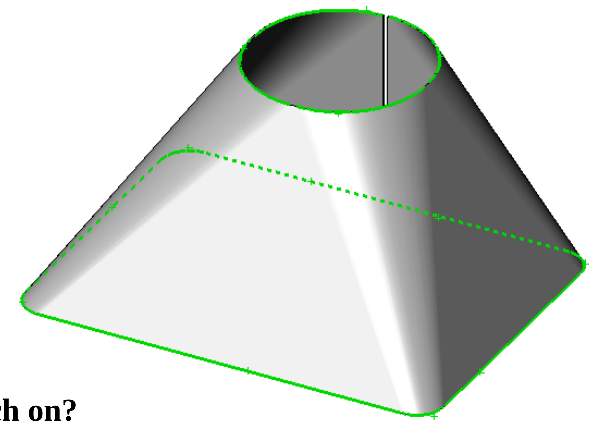
What sketches do we need to represent the profile of the base and top of the extractor hood?

What shape is the Base profile?

What shape is the Top profile?

Base: Rectangular in shape

Top: Circular in shape.



Creating Base Sketch:

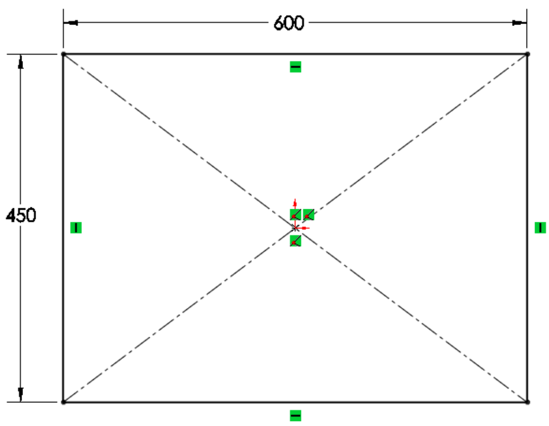
What Plane will we create the Sketch on?

Because the hood sits on the Horizontal plane we will sketch the profile on the top plane.

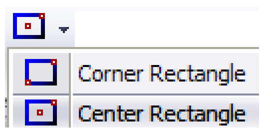
Create the sketch shown on the top plane

Width: **600mm**

Depth: **450mm**

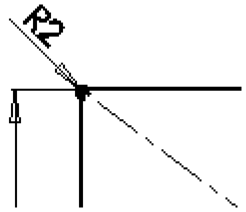


Note: A Centre rectangle was used to create the rectangle. This will allow us to use the origin to reference both sketches, ensuring they are correctly aligned. It will also allow us to use the origin when creating breaks in the sketches.



Sketch Fillet:

In sheet metal the **lofted bend** command can only create a feature from sketches which have rounded edges. Although the hood has a rectangular base sketch, we will have to create a fillet at each corner to create the sheet metal part in SolidWorks. We will use a 2**mm** radius for the purposes of the sketch fillet. Select sketch fillet and add the 2 mm radius to each corner.



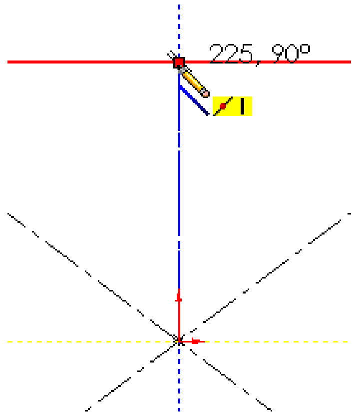
Sheet metal sketches:

As this sketch will be used to create a sheet metal feature a break must be added. This break in the sketch will later allow Solidworks to develop the completed model.

Break the sketch

Where is the best place to create a break in the sketch?
In real life the joint or break in the hood would be kept out of view of the user and would be at the back of the hood. For the same reasons we will create the break in the sketch at the back.

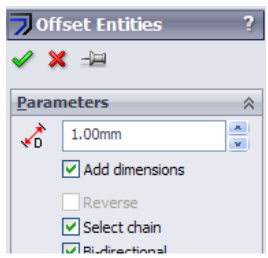
To create the break in the sketch first draw a centre line from the origin to the midpoint of the back line of the sketch.



Offset

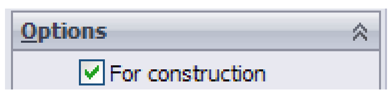
Offset 2 lines **1mm** either side of the centreline. These lines will be used to trim the sketch.

Select offset, enter a value of **1mm** and check the **Bi – directional** option.

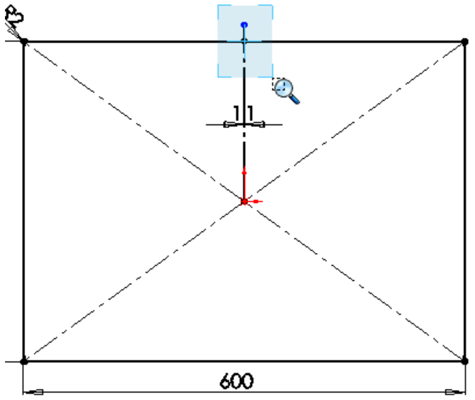


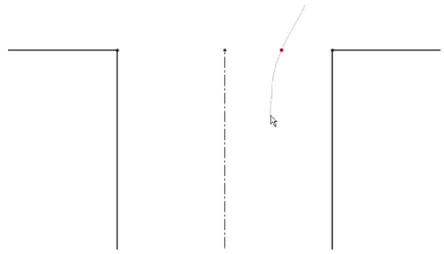
The lines that we have just drawn will allow us to break or trim our initial sketch. They are not part of the finished sketch and must be converted to construction lines.

To do this left click on the lines and check **"For Construction"** from the options box.



Trim Sketch: Zoom into the area between the lines we have just created, and use **Power Trim** to remove the material.



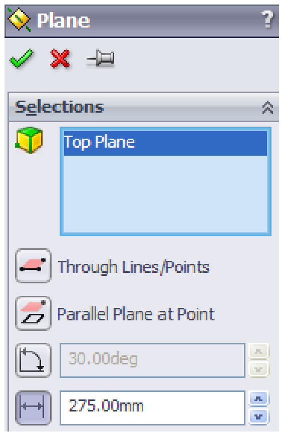


Base profile sketch complete!

Exit Sketch Exit the sketch. Rename the sketch **base profile.**

Creating Top Profile: The top profile is positioned a height of 275mm above the base profile.

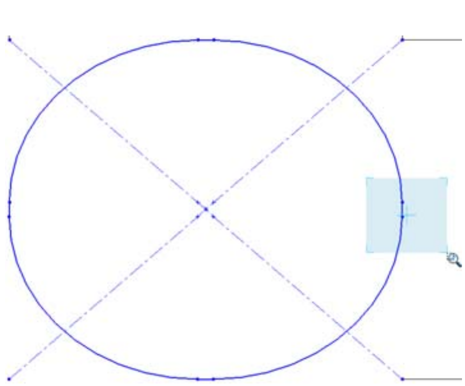
How will we create the profile 275mm above the base profile?
In order to create this sketch we must first create a plane on which to draw the sketch. This new plane will be a height of **275mm** above the top plane.



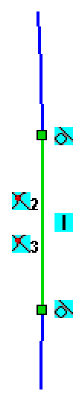
From **Features** select **Reference geometry** and **Plane.**

Create a plane **275mm** above the Top plane.

Creating the Sketch: **Note:** Earlier we mentioned that the top profile was circular in shape. However if we use the circle command to create the top profile, the finished feature will develop without fold lines. In order to create triangulation in the development the top profile must have an equal number of curved and straight sections as the bottom profile.



Circular profile with four flat sections.



Flat Section of curve (Highlighted)

Creating the circular section with the flat sections included as shown.
We could use the circle command and cut sections from it – rejoining these sections with straight lines. Would this be the best way to produce the profile?

Alternatively

Creating top profile

We begin by drawing a square, in this case 250mm side length. Create a sketch fillet, of 123mm, on the four corners. This will leave 4 flat sections of 1mm on the four sides.

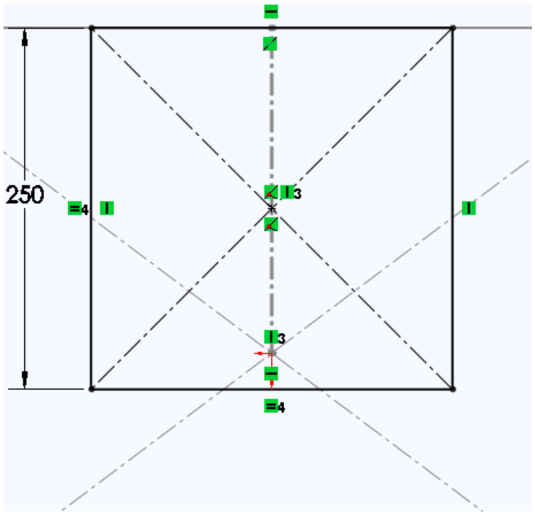
Create Sketch:

On the new plane create a **centre rectangle** sketch.

Adding Relations

The sketch may be fully defined by smart dimensioning one side and adding appropriate relations.

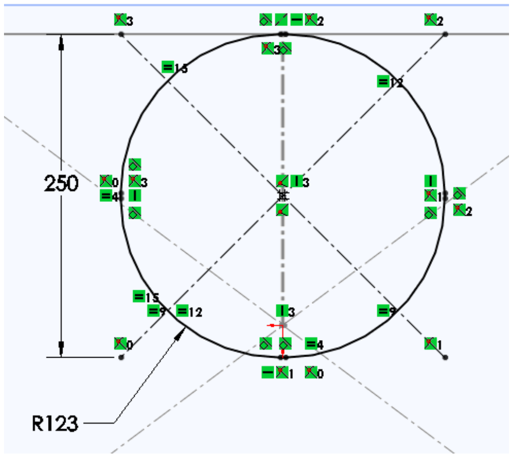
Note the relations added

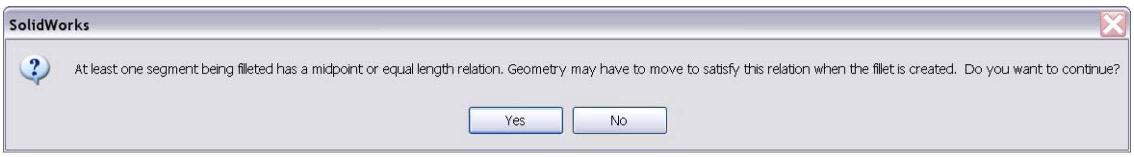


Sketch Fillet

Add a sketch fillet of **123mm** to each of the corners.

Should the SolidWorks warning shown below appear; Choose **Yes**

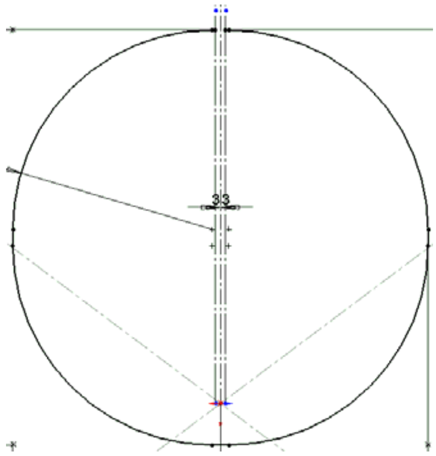




Break the sketch

Just as in the base profile, a break must be added to the sketch profile.

The same procedure may be adopted as used previously or alternatively you may take the approach outlined overleaf.



Alternatively:

We can use the break lines from the first sketch to create a break in this sketch using the convert entities 

sketch command.

Convert Entities

This command allows elements from previous sketches or model edges to be converted into sketch entities. In this case it uses the break lines from the base profile sketch to create sketch segments in the current sketch.

Note: When using the Convert Entities command, you must pre select the elements of the sketch you wish to convert before selecting convert entities 

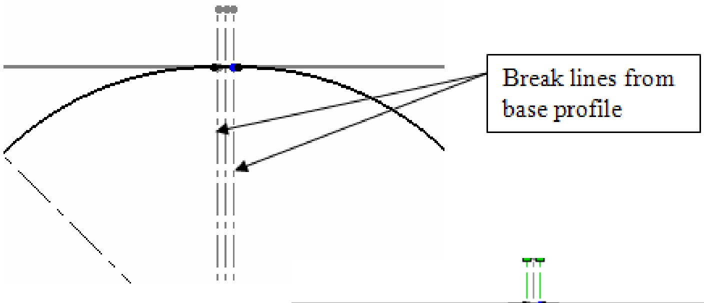
Also

Using this command means that any changes made to the distance between the break lines in the base profile automatically updates in the top profile

Select sketch elements:

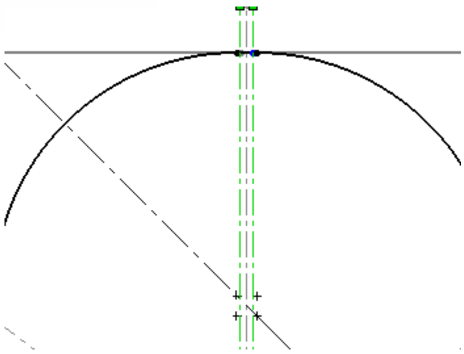
What elements of the base profile sketch do we wish to convert for use in the top profile sketch?

The break lines from the base profile.



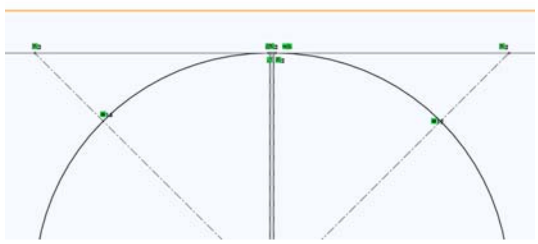
Multiple Selections

Select the break lines - To select more than one line from the sketch you must hold down the **Ctrl** button while selecting the lines.

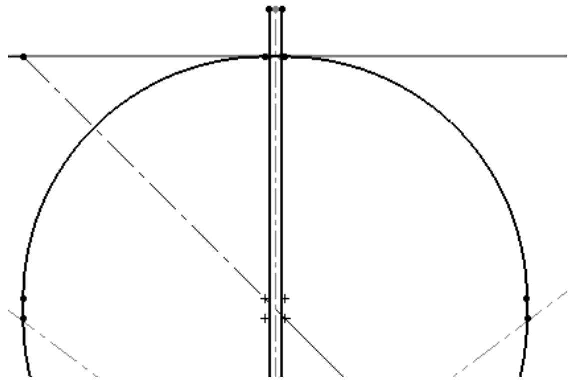


Convert Entities

With the break lines and the top profile Selected, choose Convert Entities 

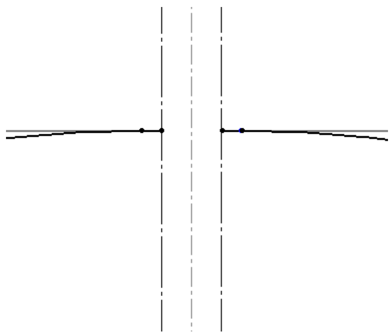


This will now convert the break lines into sketch lines within the top profile sketch.



Construction Geometry

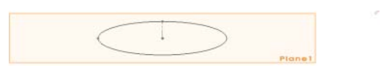
Note: these new lines must now be changed into construction lines, and then the area between them must be trimmed in the same way as we did in the first sketch.



Exit the sketch and rename **top profile**

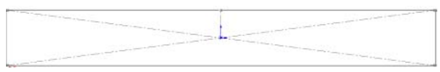
Completed Sketches

The completed sketches of the top and base profiles.



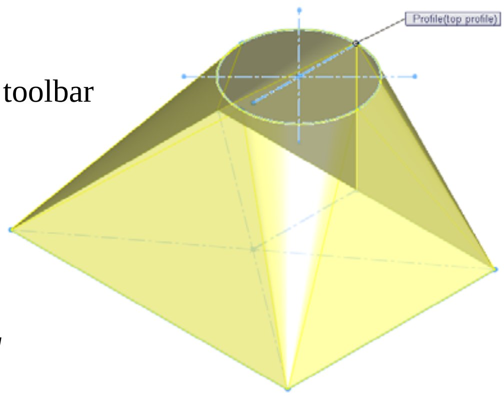
Creating the Feature:

Note: In order to create a lofted bend feature we must have exited both sketches, as both will be used to create the feature.



Lofted Bend

Select **Lofted bend** from the sheet metal toolbar



Profiles

Select the sketch profiles to create the loft
Sketch1 - Base Profile
Sketch2 - Top Profile

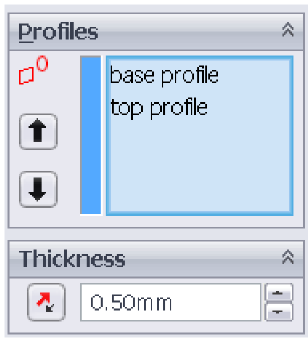
***Note**: When choosing the two sketches pick corresponding parts of both sketches to avoid forming a warped surface.*

Thickness

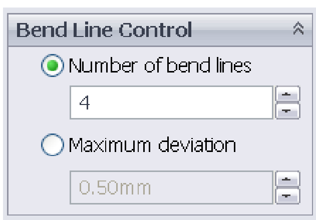
Enter a **Thickness** of **0.5mm**

Bend Lines

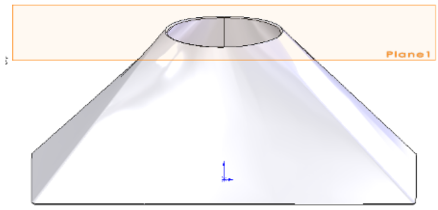
The number of bend lines used to create the transition piece may be controlled here.



Choose 4 bend lines.



Click **OK** to confirm

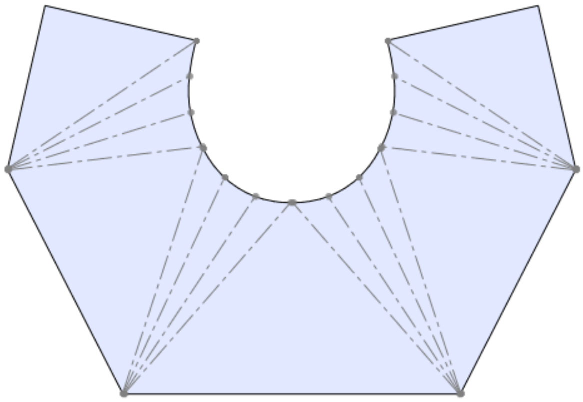


Hiding the Plane: Left click on **plane1** in the feature manager tree and select **Hide**

Flatten Choose **Flatten**

The transition piece development is displayed.

Note: 4 bend lines at each transition.



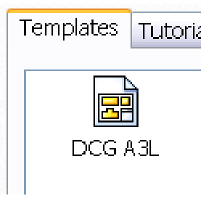
Creating a drawing: As with any Solidworks part, a drawing may be produced by selecting **File, Create drawing from Part/Assembly** *or*

Choose Make Drawing from Part/Assembly from the standard toolbar

When a drawing is created from a sheet metal part, in addition to the standard views, the opportunity exists to create a view which displays the development of that part.

Create Drawing: With the part file open select **File, Create drawing from part/assembly.**

Select **DCG A3L** as the drawing template you wish to use.

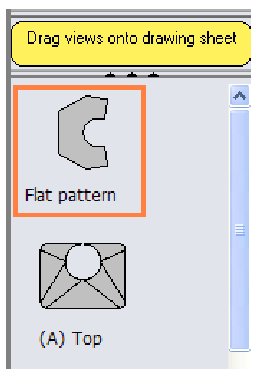


Standard Views All of the standard drawing views are displayed in the **Task Pane**

Any of these views may be dragged and dropped onto the drawing sheet.

Flat pattern **Flat Pattern** is included in these views.

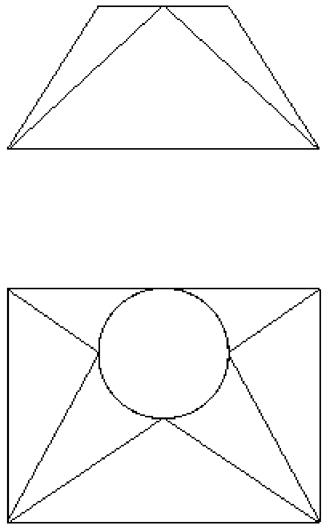
Dragging this icon into the sheet will generate a view displaying the development of the prism.



Creating the view layout

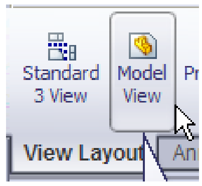
To generate the solution we want to create 3 orthographic views and add the surface development of the transition piece

Drag the **Front View** onto the sheet and project a Plan view from the parent view. Use a scale of **1:5**



Adding the surface development:

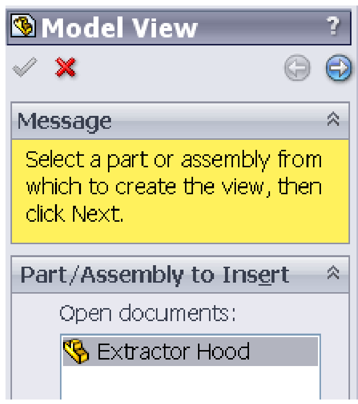
To add the surface development, select **Model View** from the **View Layout** toolbar.



Model View

Choose **Extractor Hood** from the **Open documents** list in the Model View options dialog box**.**

Select **next** to proceed

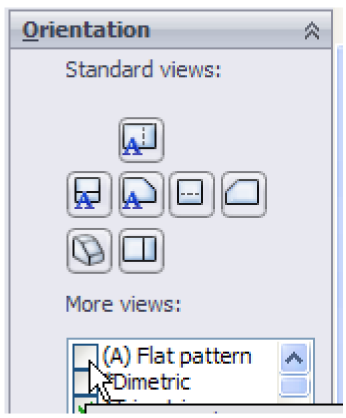


Flat Pattern

Select the **Flat Pattern** option

Position the flat pattern view on the drawing sheet.

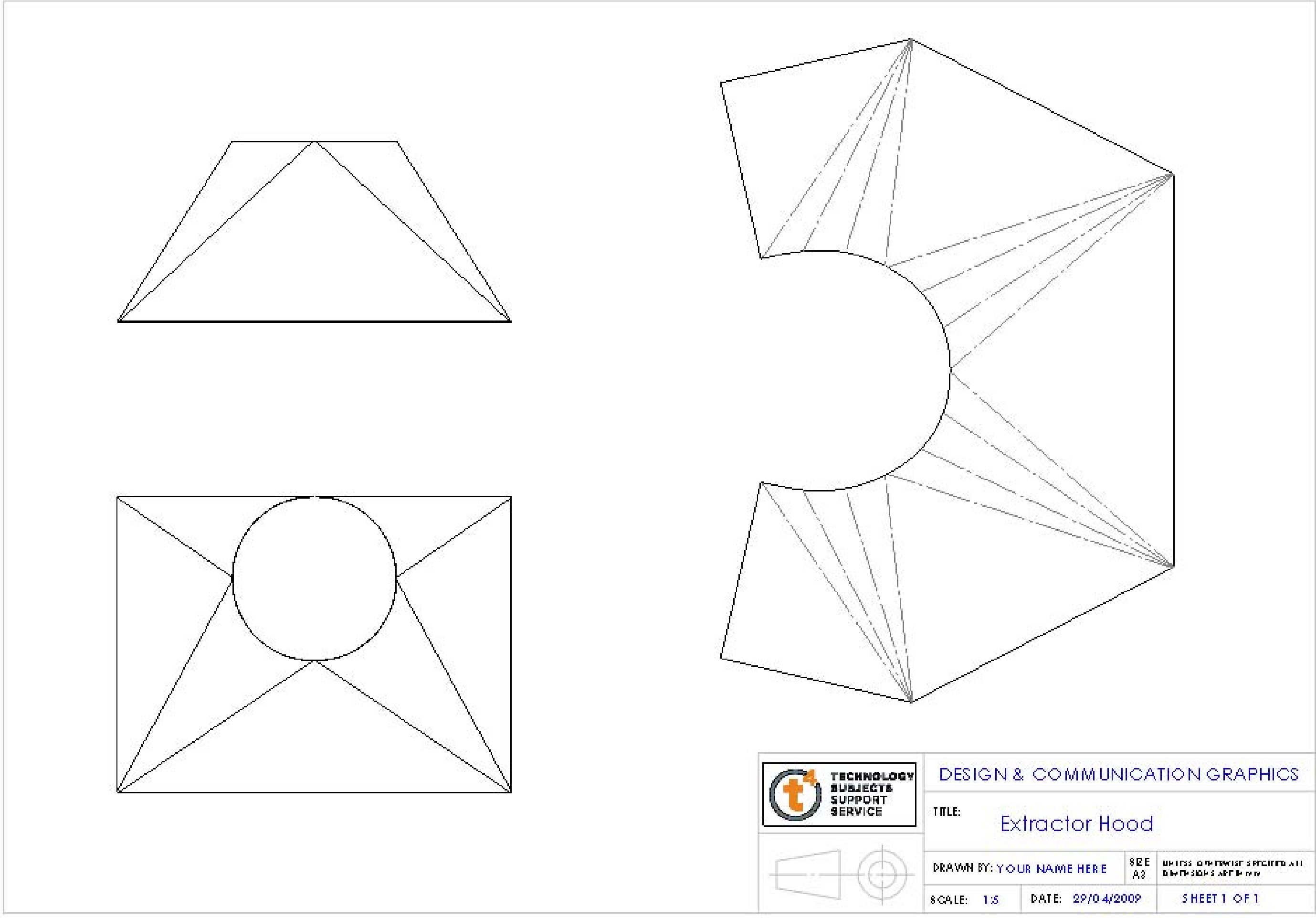
Choose **OK**.



Positioning the views

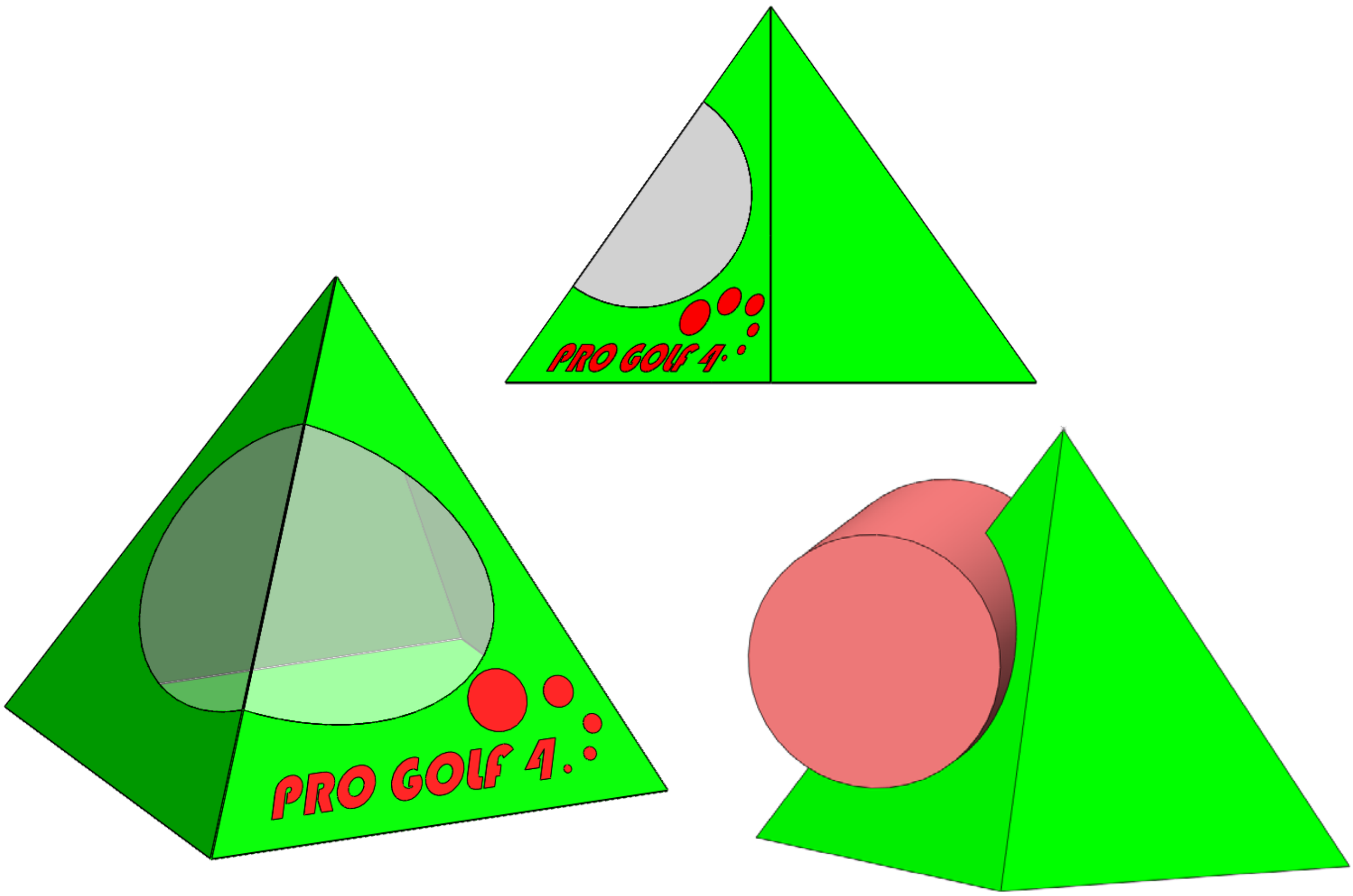
Drag the views to position them on the sheet.

Note*: The text may be removed from the surface development by right clicking on it and choosing* ***Hide.***



Save & Close Save all SolidWorks Documents. Lesson Complete!

Convert to Sheet Metal - Golf Ball Package



Prerequisite knowledge: Sheet metal exercises 1, 2, and 3, should be completed in advance of this exercise.

Focus of lesson: To convert solid features to sheet metal and to use sheet metal to create a surface development.

Commands Used: This lesson includes Sketching, Lofted Bend**,** Flatten, Extruded Cut and **Convert to Sheet Metal.**

Problem: The development of the Golf ball package is to be created using SolidWorks. The box is based on a square based pyramid. The window is created with a cut generated by a cylinder. Generate the pyramid, removing the cut material for the window. Retrieve the development of the box using sheet metal features.

New File Create a new part file.

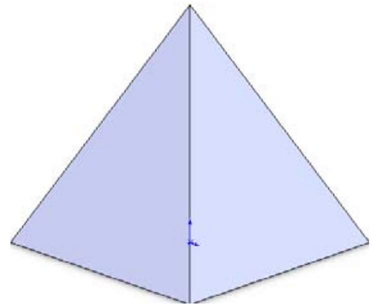
Save File Save the file to a chosen location as **Package development.**
We will explore the use of SolidWorks in completing the problem, focusing on sheet metal features.

The approach

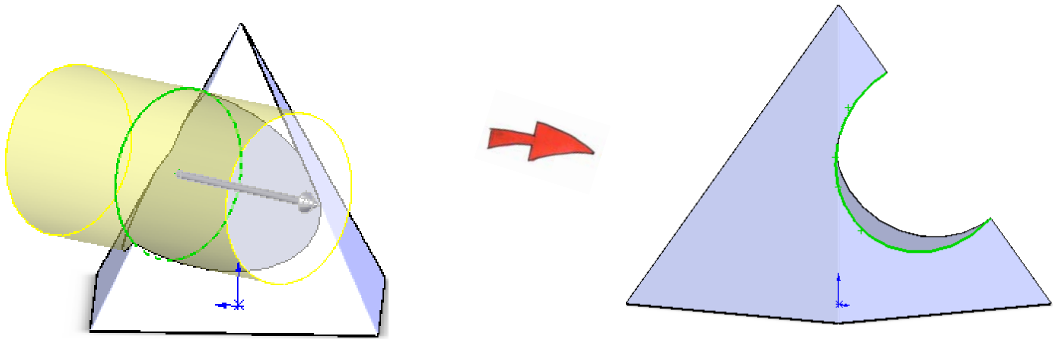
The square based pyramid must first be modelled and the development of the package created from it. As the pyramid is to be cut with a cylindrical feature, we will create it as a solid. We will then convert it to a sheet metal component in order to generate its development.

What geometric shapes are used?

The main body of the package is created by modelling a square based pyramid.



The cut surface is generated by cutting the pyramid with a cylinder to give a circular cut when viewed from the right or left.



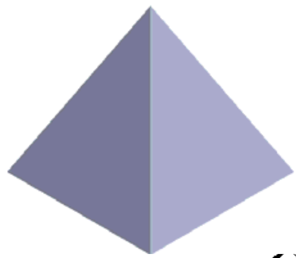
Getting Started

How will we create the pyramid?
In exercise three we created the transition piece using lofted bend. We will use lofted bend again to create the pyramid.

What two sketch profiles must we create?

Base: Square

Top: Point

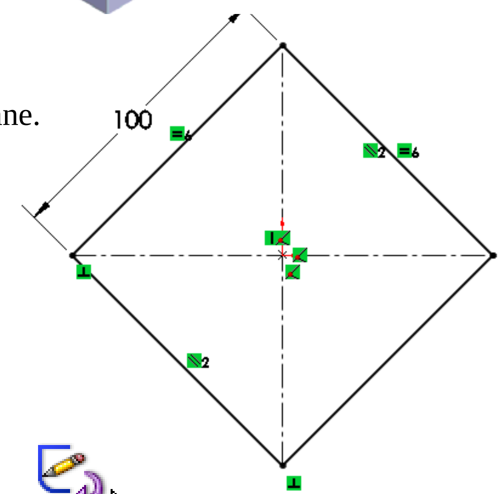


Creating Base Sketch:

What Plane will we create the Sketch on?
Because the pyramid sits on the Horizontal Plane, we will create our sketch on the top plane.

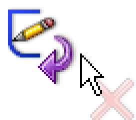
Create the sketch shown on the top plane.
Use only the dimension shown.
Add appropriate relations

Side: **100mm**



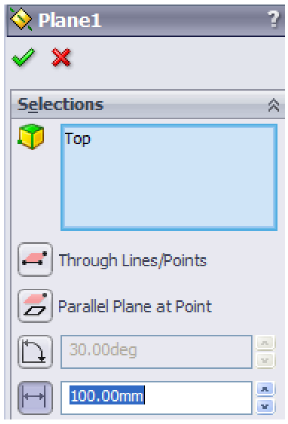
Exit Sketch

In order to create the second sketch we must first exit this sketch.



Creating Top Profile:

The top profile is positioned a height of 100mm above the base profile.



How will we create the profile 100mm above the base profile?
We must begin by creating a plane on which to sketch.

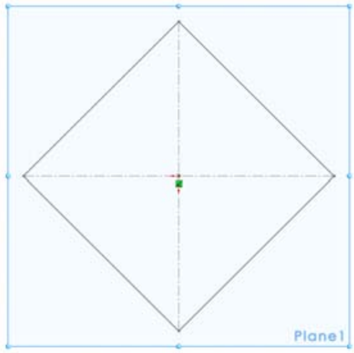
From **Features** select **Reference geometry** and **Plane.**

Create a plane **100mm** above the Top plane.

Creating sketch

As the pyramid forms a point at the apex, we will use **point** to create the top profile.

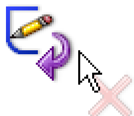
Note: by creating the base profile with the origin as centre it allows us to use the origin to align both profile sketches.



Select the new plane as the sketch plane. From the sketch toolbar select **point** Position a point on the origin as shown. This will ensure that the point is positioned directly over the centre of the rectangular base thus producing a right pyramid.

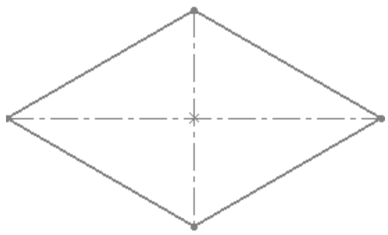
Exit the Sketch

Exit sketch.



Creating the Feature:

Note: In order to create a lofted bend feature we must have exited both sketches, as both will be used to create the feature.



Loft

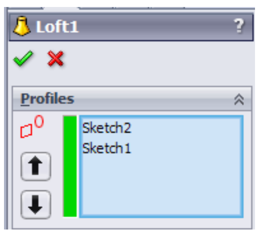
Select **Loft** from the features toolbar

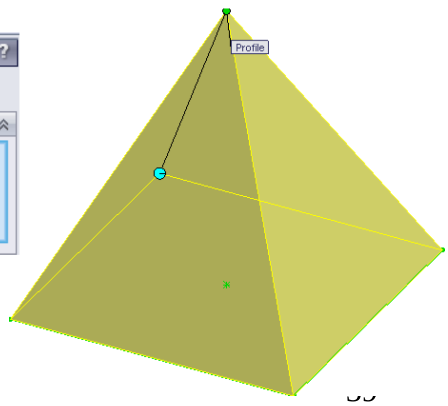
Profiles

Select the sketch profiles to create the loft

Sketch1: Base Profile
Sketch2: Top Profile

Click **OK** to confirm



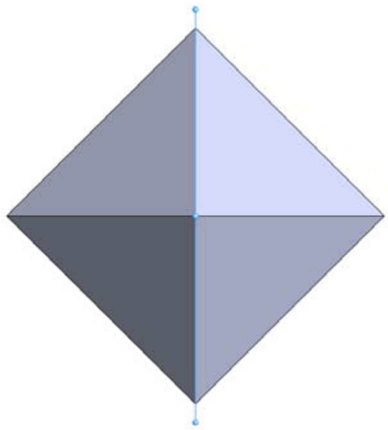


Creating the Cut

The pyramid is to be cut by a cylinder. In order to create this cut we will need to sketch a circle to extrude cut through the pyramid.

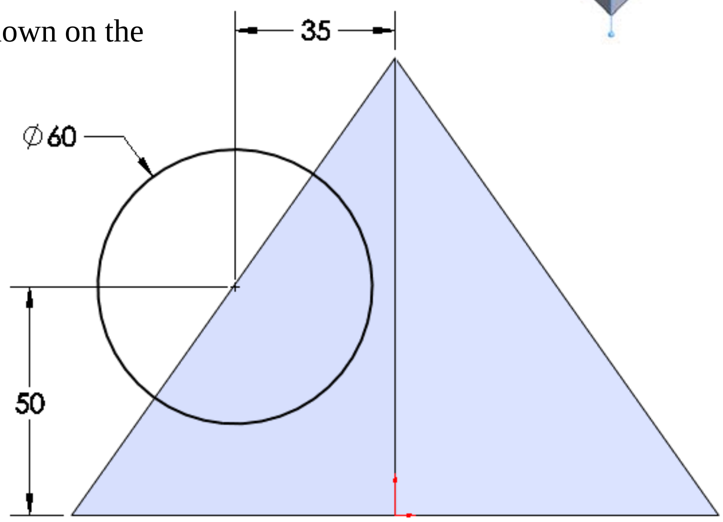
The cylinder cuts through the pyramid therefore the sketch is placed on the right plane which cuts the pyramid symmetrically. A mid- plane extrusion will be used to create the cut.

Due to the prudent location of the origin we can use the Right plane on which to sketch.



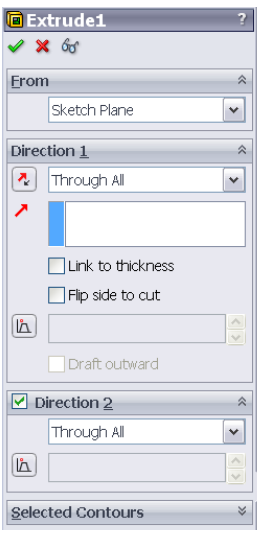
Create Sketch

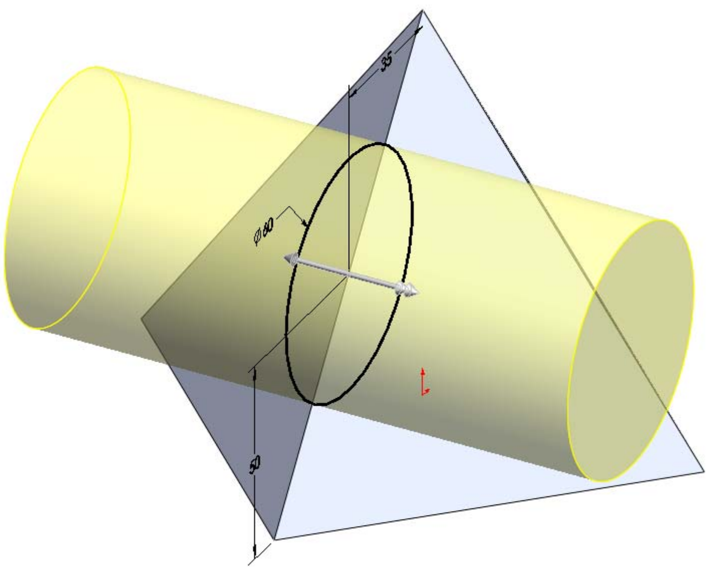
Create the sketch shown on the Right plane



Creating the feature

Create an Extruded Cut feature using a **Through All** end condition in both directions.





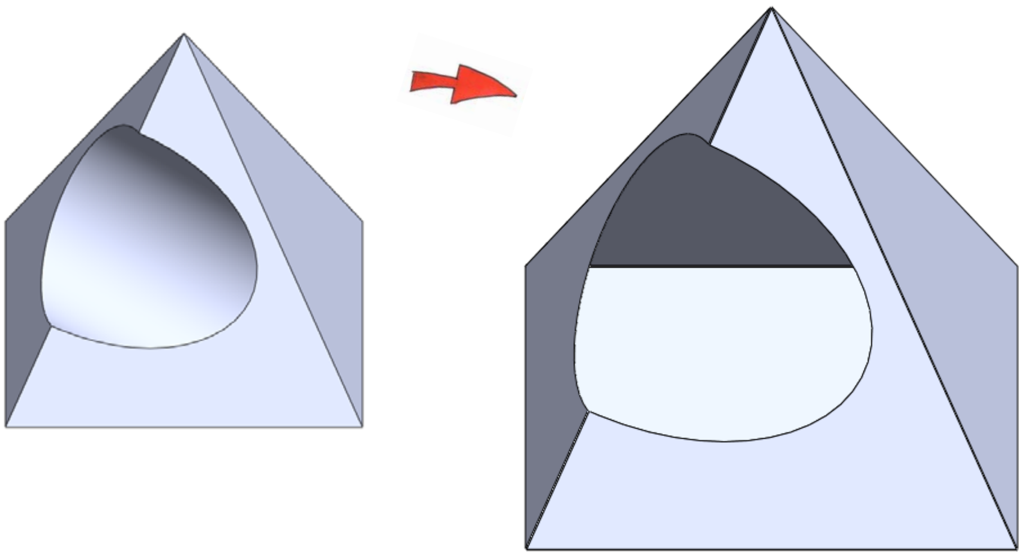
Click **OK.**

Surface Development

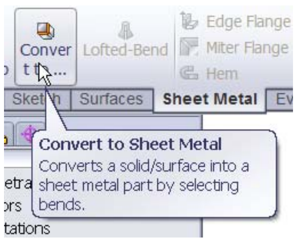
A cut pyramid has now been created but SolidWorks cannot create a development from a solid model. We must convert the model to a sheet metal part in order to retrieve the surface development.

Convert to sheet metal

In Solidworks 2009 solid models can be converted to sheet metal. Using this feature will allow us to create a development from this model.



Choose **Convert to sheet metal** from the sheet metal toolbar.



When converting to sheet metal you will be required to choose;

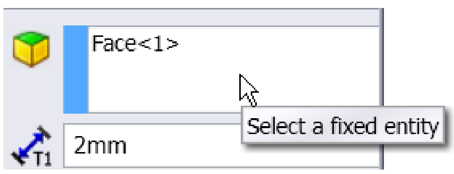
- a fixed face, about which the development will be created
- corners which will become bends
- edges along which you wish to cut or rip the feature to flatten it out

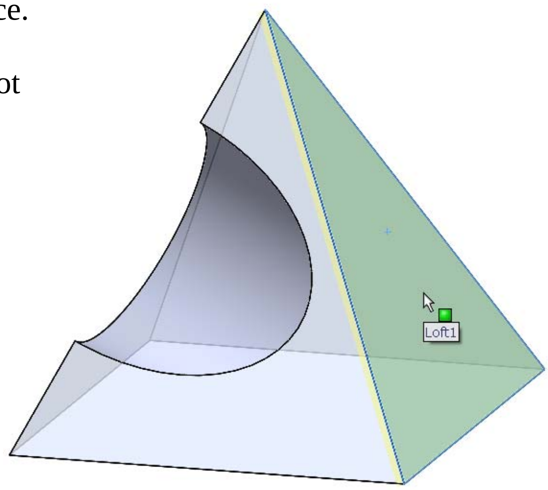
Selecting a fixed face:

Any face may be selected as the fixed face.

We will select one of the faces that are not cut by the cylindrical hole.

Face 1 is entered as the selected face.

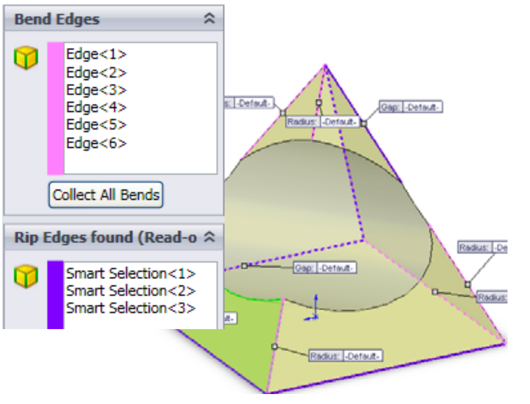




Selecting Bend Edges:

The bend edges must be selected.
Select each of the bends on the model.

As the bends are selected, SolidWorks will automatically select the edges which will be used as rip edges.



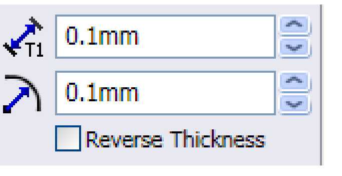
Further parameters

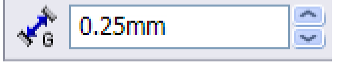
The following parameters must be set;

Sheet thickness: **0.1mm**

Bend radius: **0.1mm**

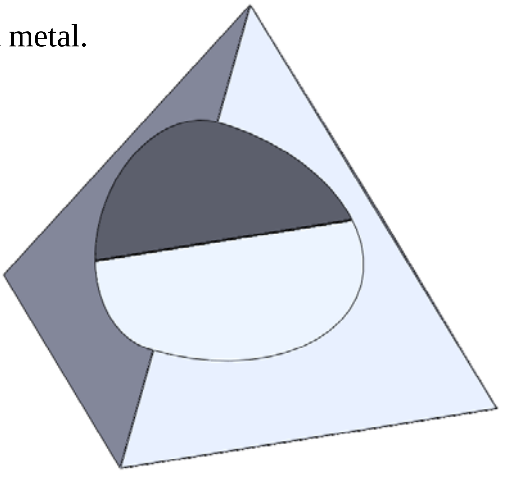
Gap size: **0.25mm**





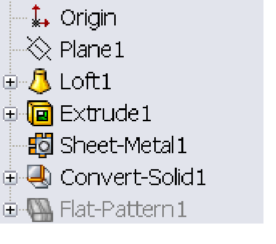
Click **OK.**

The pyramid has now been converted to sheet metal.



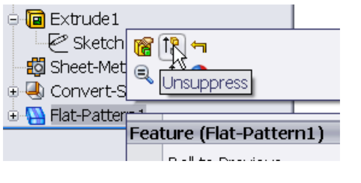
Design Tree

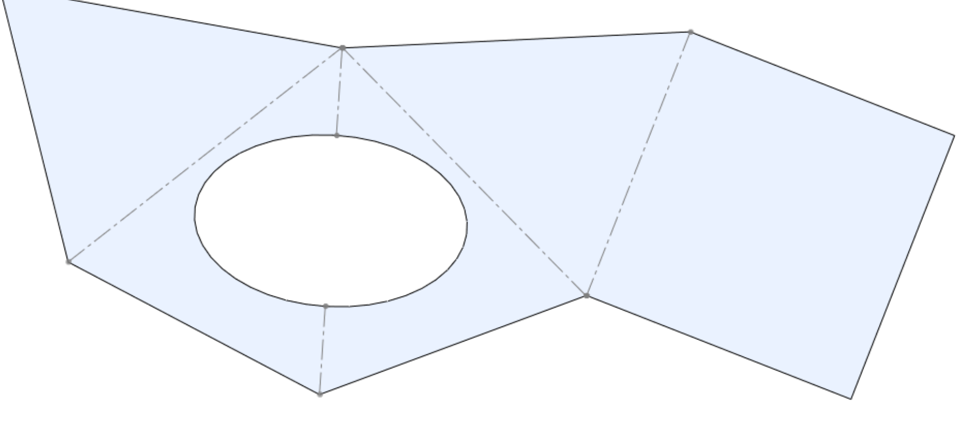
Sheet Metal features have been added to the Feature Manager Design Tree.

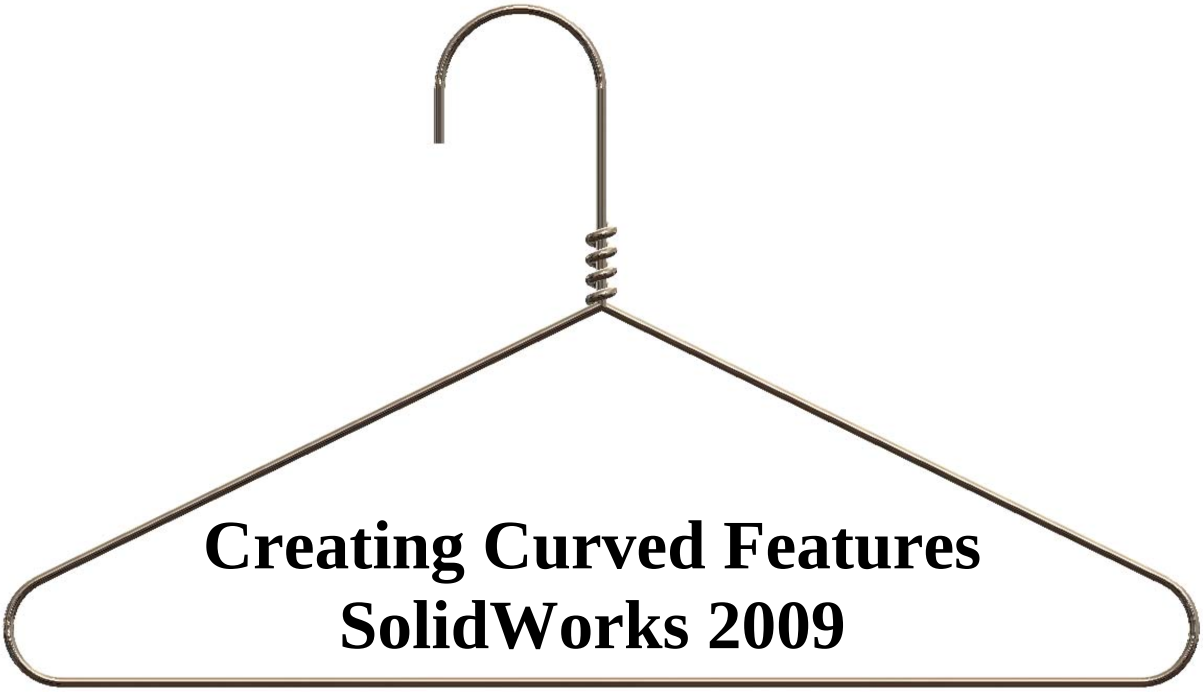


Surface Development

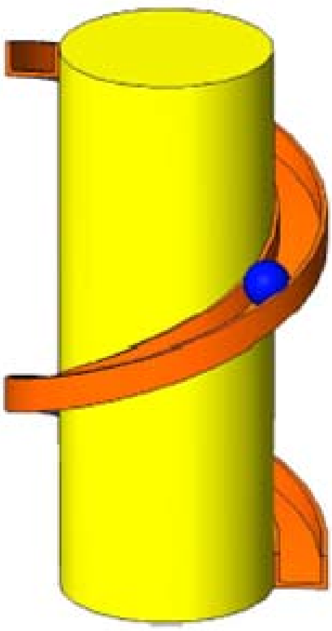
Right click on **Flat-pattern** in the design tree and choose **Unsuppress.**







Creating Curved Features
SolidWorks 2009

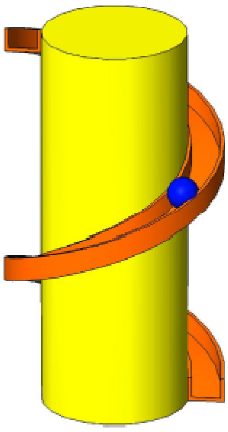


Helical Slide Exercise

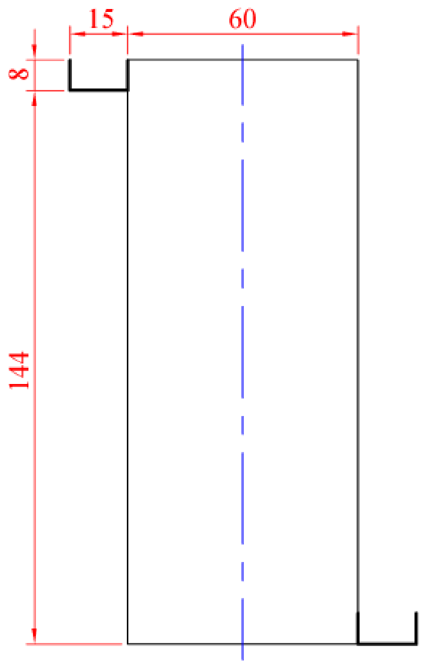
(b) The 3D graphic below shows a helical marble chute from a child's toy. The relative positions of the rectangular ends of the chute are shown in fig C-1(b). The helix rotates about the central support column through an angle of 540° as shown.

Draw the plan and elevation of the column and chute showing all helical curves.

(Hidden detail need not be shown and the wall thickness of the chute should be ignored).



Scale 1:1



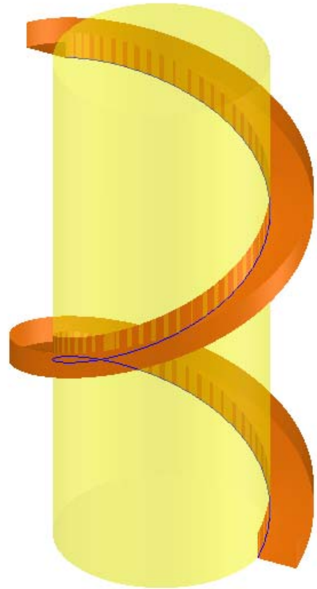


Fig. C-1(b)

Prerequisite knowledge A basic knowledge of SolidWorks 2009 is required – use of sketching and extrude boss/base.

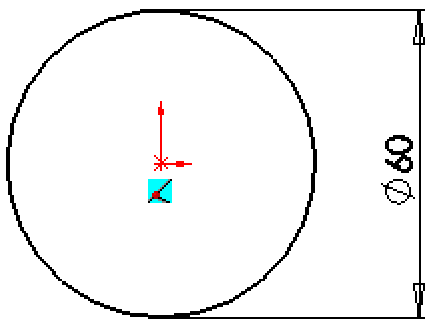
Focus of Lesson This lesson will focus on creating a 3-dimensional solution to the geometry problem posed using SolidWorks.

Commands Used This lesson includes ***Sketching, Helix, Surface Sweep, Sketch Sharing*** *and* ***Extrude Boss/Base.***

Getting Started A helix is a curve which lies on the surface of a cylinder or cone. A cylindrical helix is formed by a point moving uniformly at a constant rate around a cylinder.

Create Sketch Create a sketch of a circle on the Top plane using the dimensions shown. Position the sketch so that the origin is coincident with the centre point of the circle.

This circle will be the basis for generating the helix.

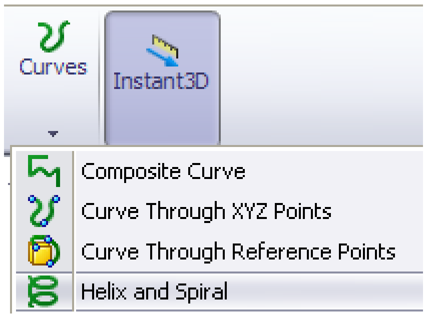


Exit the sketch and rename it to **Profile of Helix** in the **Design tree.**

Creating Helix

Select the profile and choose **Helix and Spiral** from the features toolbar under **Curves**.

Or choose;
Insert, Curves, Helix/Spiral



Helix/Spiral Property Manager

The **Height and Revolution** option will be used to create the helix.

Parameters

Height: 144mm, given in the question.
Height defines the height from the profile.

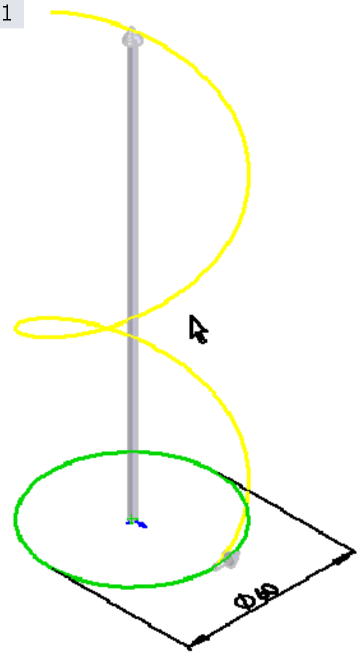
Reverse direction brings the height above or below the plane the profile exists in.

Revolutions: 1.5
Revolutions defines the number of turns, in this particular case we require 1.5 revolutions.

Start Angle defines the start point of the helix on the sketched circle.
Start angle: 90°

Choose **OK**

The profile sketch is incorporated into the feature and so is out of our view.

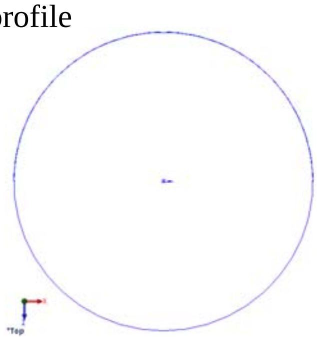


Rename feature

Rename the **Helix/Spiral** as **Slide Path.**

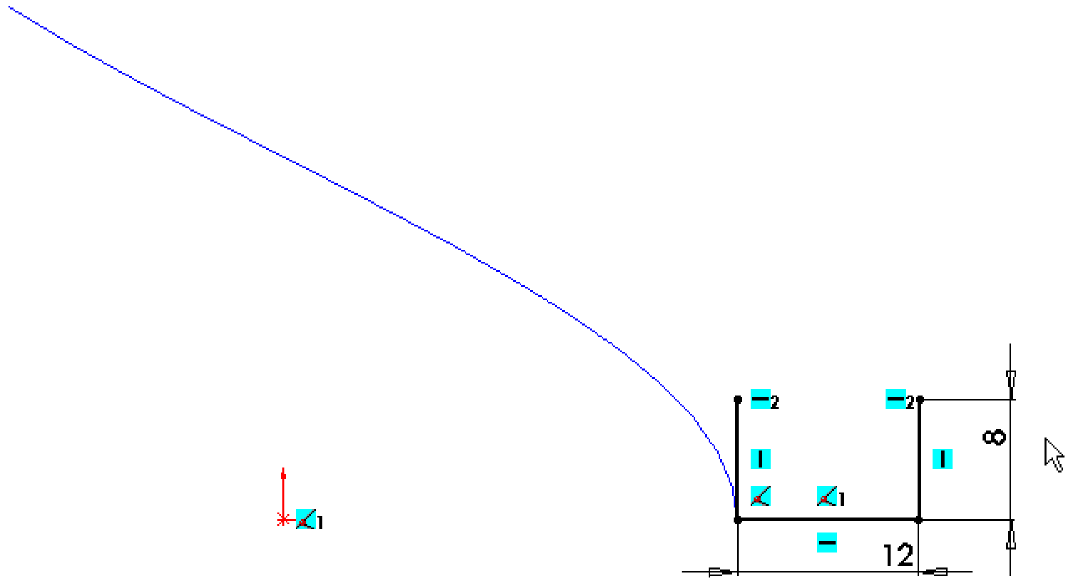
Choose **Top View.**

The helix appears as the profile circle in plan.



Slide

Select front plane and create the following sketch.
Ensure that the endpoint of the horizontal line is coincident with the start point of the helix as shown.



Rename the sketch **Slide Profile**.

Surface Sweep

Surface Sweep command works in the same way as the feature **Swept Boss/Base** however the surface produced is **infinitely thin**. The question outlined at the start directs us to ignore wall thickness of the chute.

Swept Boss/Base uses a **closed sketch** and produces a sweep which has thickness. Swept Boss/Base cannot use open sketches as we have here.

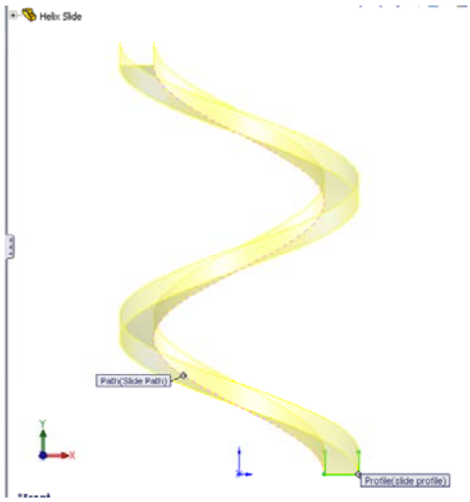
Locating the command

Add in the surfaces tool bar 

to the features command manager.

Locate the swept surface option under this menu. 

Profile - slide profile
Path - Slide path

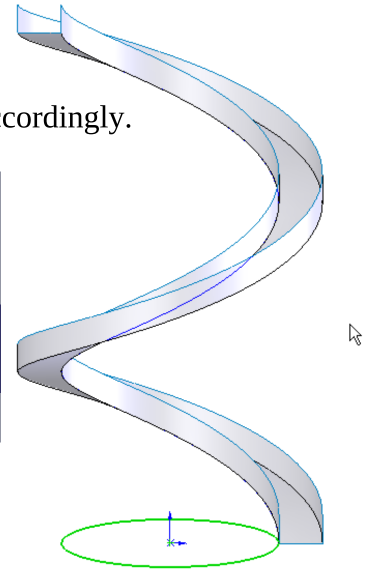


Choose **OK**
Rename the surface sweep as **Slide.**

Finish Slide

Identify the Profile of Helix sketch.
This sketch will be used to generate the cylinder.

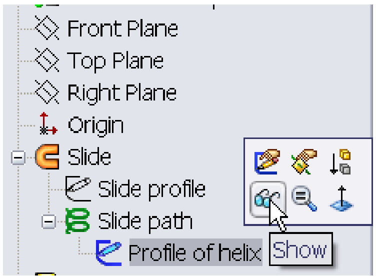
Note: The advantage of using this circle is; should the helical diameter change, the cylinder will update accordingly.



How to find the circle:

Click the plus sign beside *Slide*

Click the plus sign beside *slide path*

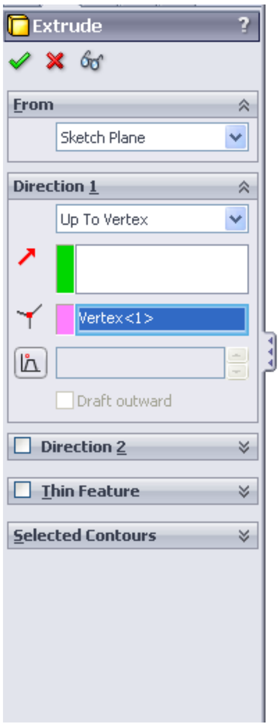


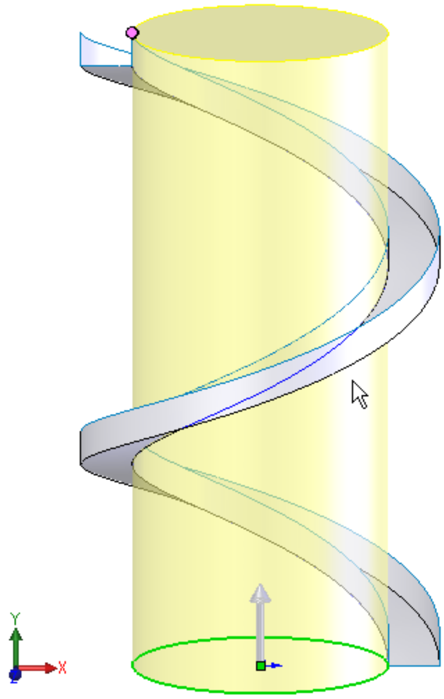
Right click on *Profile of Helix* and select show.

Extruded Support

Extrude this sketch using the feature **Extrude Boss/Base** with the end constraint set as **Up to Vertex**. Choose the top vertex on the slide to be your height. This will keep the extrude referenced to the slide if the height of the helix (slide) is altered.

Rename the extrude **Slide support.**





Note -

Notice the icon representing the sketch for *Profile of Helix*.

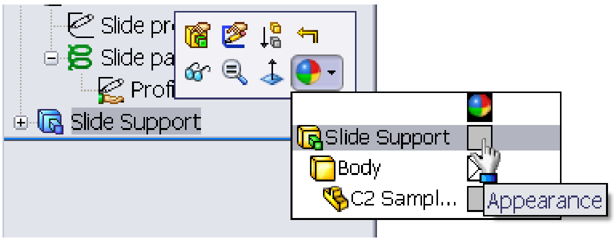
In the same way as shared folders Shared Documents represented on your computer shared sketches have a similar icon to describe the fact that they are shared.

Appearance

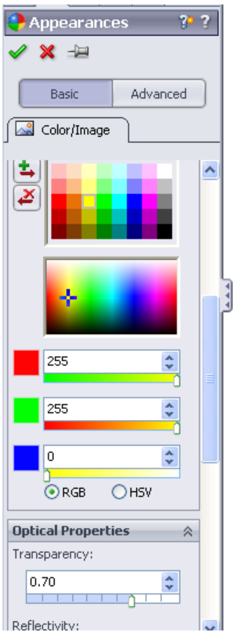
The solid is now complete. However, we can see there is a problem with the surface of the cylinder and the surface of the inner wall of the slide – to accurately complete the question we will manipulate the transparency of the cylinder and allow the slide to overlay it.

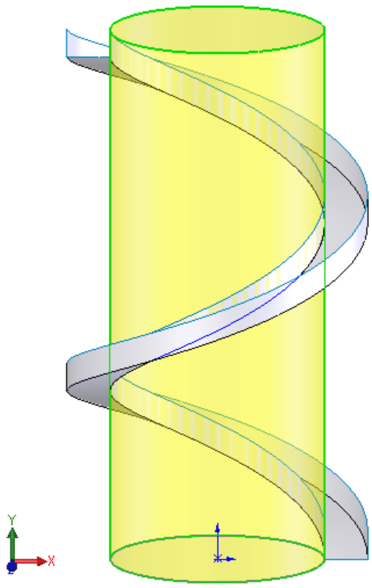
Transparency

Right click on the feature called **Slide Support** in the feature manager design tree and select **Appearance**.



Choose an appropriate colour from the **color palette**.

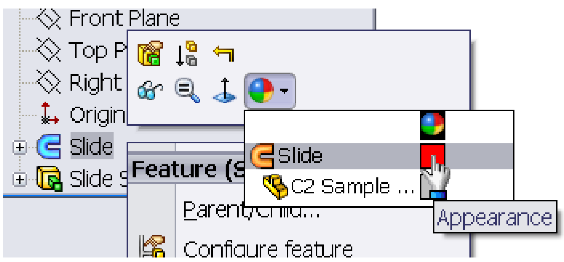


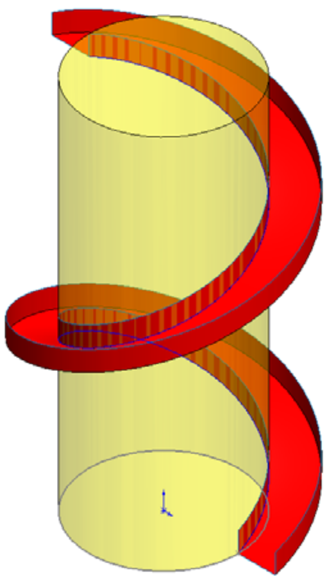


Transparency is present under Optical Properties - Set the Transparency to 0.70

Slide Colour

Apply a colour to the slide in the same way.



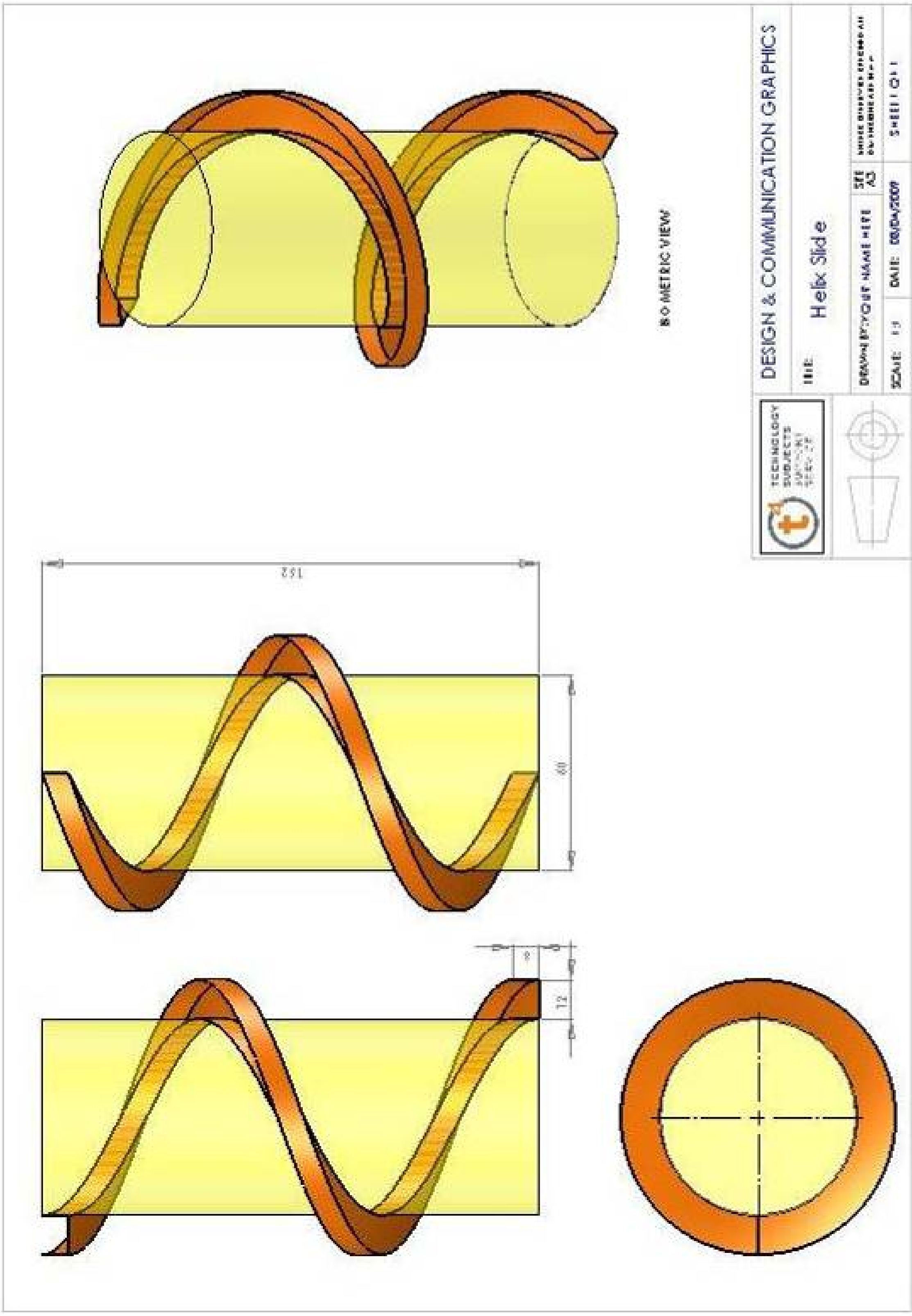


Note: No transparency required for the slide

Drawing

Make a drawing file from the part

Ensure to give the drawing the **shaded with edges** display under **display style –** a surface in SolidWorks has no thickness therefore will not be seen unless the above step is taken.



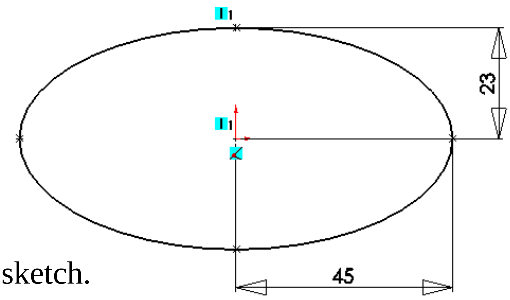
Projected Curve & Lofted Boss/Base – Hand Soap Bottle

Prerequisite knowledge A basic knowledge of SolidWorks 2009 is required – use of *sketching* and *extrude boss/base*.

Focus of Lesson This lesson will focus on using the following feature commands-
Lofted boss/base with 3 guide curves, Spline and Project Curve.

Commands Used This lesson includes **Sketching (Spline), *Loft boss/base, projected curve, Swept boss/base and Extrude boss/base.***

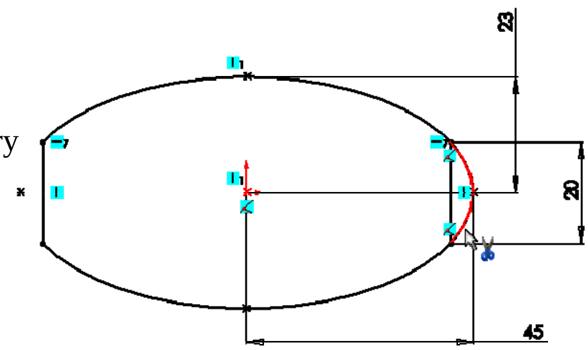
Getting Started Create the following sketch of an ellipse on the top plane using the dimensions shown. Position the sketch such that the origin is at the centre of the sketch.



Note the vertical relation to fully define the sketch.

Save ***Save the part as Soap Bottle***

Add the two vertical lines as shown. Smart dimension and add the necessary relations to fully define the sketch.

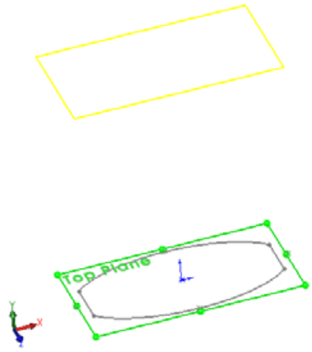


Trim the ellipse using the two lines
Exit sketch and rename it **profile1.**

Creating Profile 2

The second sketch used to define the loft must be created at a height of 105mm above **profile1**.

We must first create a plane on which to sketch.



Offset Plane

Choose **Insert, Reference Geometry, Plane**

Define Plane

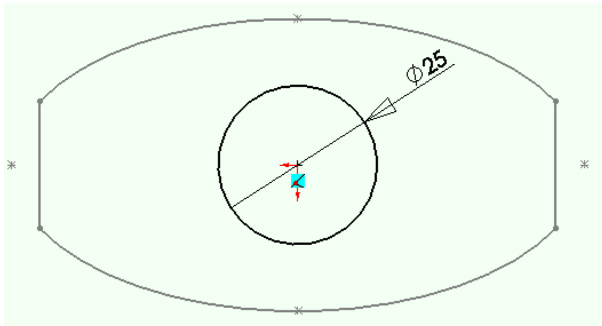
Selections:

Top Plane
Offset Distance: **105mm**.

Sketch

Create the circular sketch shown on Plane 1.

Close sketch and rename it as **Profile2.**



Creating the bottle

We are going to use **Lofted Boss/Base** to create the bottle shape, however we will direct the shape of the transition between the two sketches using **guide curves**.

The guide curves will be generated as sketches using the **Spline** tool.

Splines

Spline draws a freeform curve. Splines may form either a single closed loop or an open loop. In either case the spline is not allowed to cross itself.

You can draw a spline by clicking each location where you want to add a control point. Splines are used mainly for freeform complex shapes in 2d and 3d sketches, although you can also use them for anything that you would use other sketch elements for.

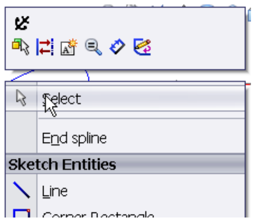
Creating Guide Curves

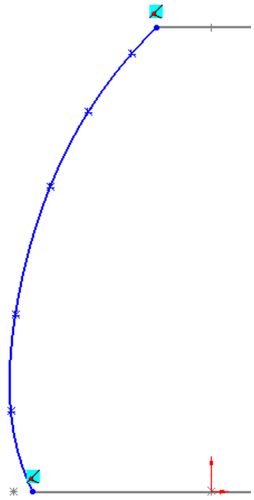
Hide **Plane 1.** Choose **Front View.**
Create a sketch on the front plane. Choose Spline from the sketch toolbar.

Sketch a spline with its start and end points coincident with profile1 and profile2 respectively.

Ensure there are **7 points** in total on this **spline.** Start and finish with 5 in between.

Note: To end the spline; right click and choose **Select.**





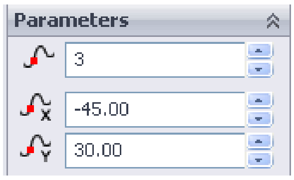
Cartesian Method

We will use *Cartesian Coordinates* to identify the position of **spline points** and hence drive the shape of the spline.

To give the spline points their Cartesian Coordinates double click on the spline.

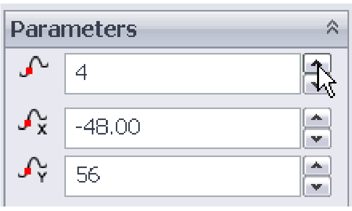
Spline point number - where on the spline the point is located

X and ***Y axis*** – based on the ***Cartesian Coordinate System.***



Using the values in the table, index through the seven spline point numbers and assign the coordinates for each spline point.

Point	X Axis	Y Axis
1	**-40.524**	**0**
2	**-44**	**10.5**
3	**-48**	**37**
4	**-48**	**56**
5	**-40.50**	**81**
6	**-25**	**100**
7	**-12.5**	**105**



Exit the sketch and rename as **guide curve1.**

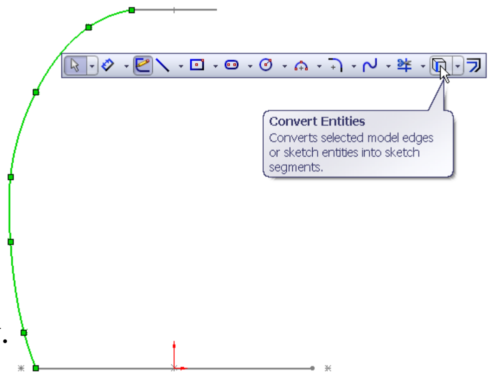
Guide Curve 2

The guide curve on the opposite side will be a mirror image of **guide curve1** as the bottle is symmetrical.

Create a new sketch on the **front plane**.

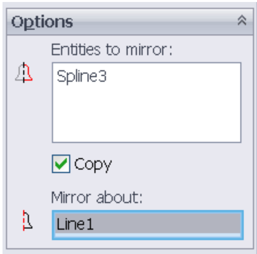
Select **guide curve1** and choose **Convert Entities**

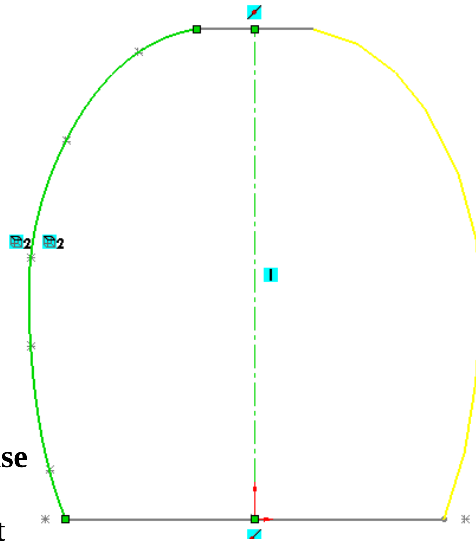
This sketch will be projected onto the sketch plane and may now be used as sketch geometry.



Mirror Entities

Add in a **centreline** as shown and **mirror** the curve about the centerline.



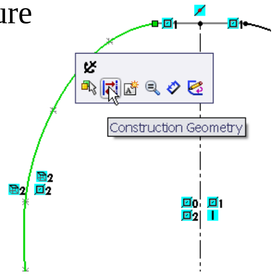


For Construction

The curve on the right hand side will be used as a guide curve to create the **Lofted Boss/Base**

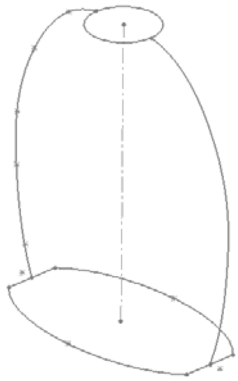
The *'Convert Entities'* curve, on the left, must be marked as **Construction Geometry** to ensure it is ignored when using the sketch as a guide.

Select the curve on the left and choose **Construction Geometry** in the pop-up menu.



Exit the sketch and rename **as guide curve2.**

Note: If Guide Curve1 is edited guide curve2 will automatically update to reflect those changes.



Guide curves 3 & 4

Guide curves 1 & 2 will drive the shape of the left and right hand sides of the bottle. Sketches representing guide curves 3 & 4 must be created to drive the profile of the front and back of the bottle.

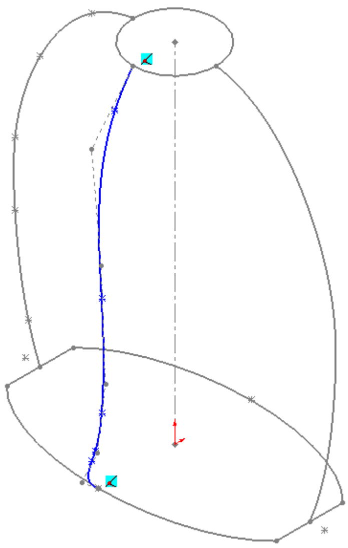
Guide curves 3 & 4 will be created using the same steps with different spline point coordinates.

Create Sketch

Create a sketch on the **Right plane**.

Sketch the spline shown using the following **Cartesian Coordinates**.

Point	X Axis	Y Axis
1	**-23**	**0**
2	**-25**	**8**
3	**-24**	**10**
4	**-22**	**19**
5	**-22**	**49**
6	**-18**	**96**
7	**-12.5**	**105**



Exit Sketch

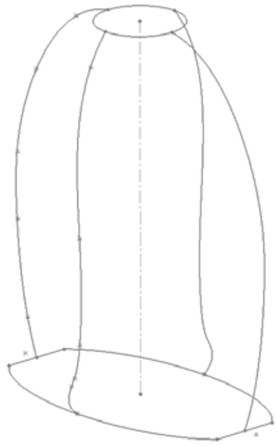
Exit the sketch and rename **guide curve3**

Guide Curve4 — **Guide Curve4** will be created in the same way as **guide curve2** as the bottle is again symmetrical in this direction.

Create sketch — Create a new sketch on the **Right plane**. Choose **Right View.**

Convert Entities — Use **Convert Entities** to convert **guide curve3** onto the plane.

Sketch the vertical centerline, coincident with the origin and **mirror** the curve about the centreline.



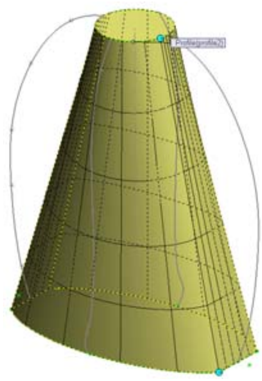
Remember: The *'Convert Entities'* curve, on the left, must be marked as **Construction Geometry**.

Select the curve on the left and choose **Construction Geometry** in the pop-up menu.

Exit the sketch and rename **guide curve4.**

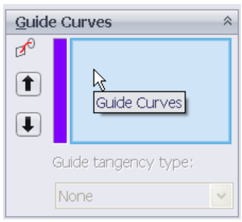
The model should now appear as shown.

Lofted Boss/Base — **Choose Lofted Boss/Base** and create a Loft between Profile1 and Profile2.

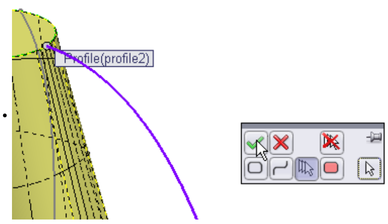


Without the influence of the guide curves the loft will appear as shown.

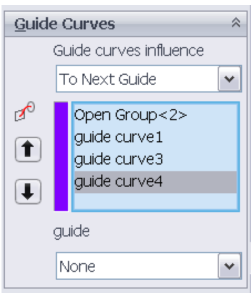
Guide Curves — Highlight the **Guide Curves** dialog box

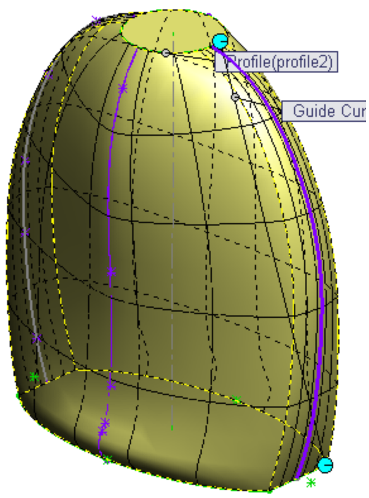


Select one of the guide curves and choose **OK**



Repeat the procedure to capture all of the guide curves.

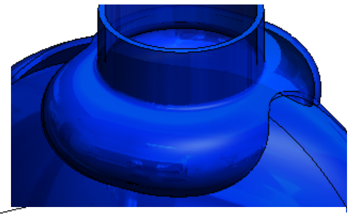




Rename **loft** as **Body of Bottle**

Neck Feature

We will use a **Swept Boss/Base** to create the feature shown on the neck of the bottle.



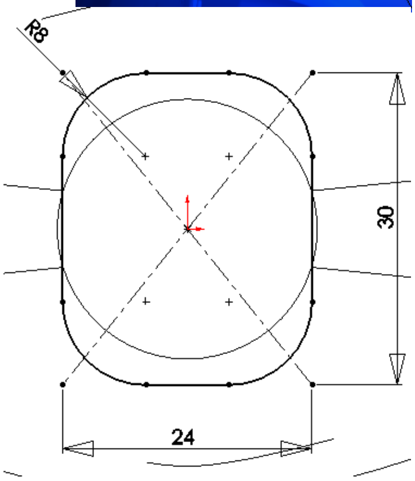
In order to generate a sweep we must create a **profile** and a **path** around which to sweep the profile.

These will be created as two separate sketches.

Path Sketch

Create the sketch shown on the top surface of the bottle using centre rectangle, coincident with the origin, and a sketch fillet of 8mm.

Exit the sketch and rename **neck path.**

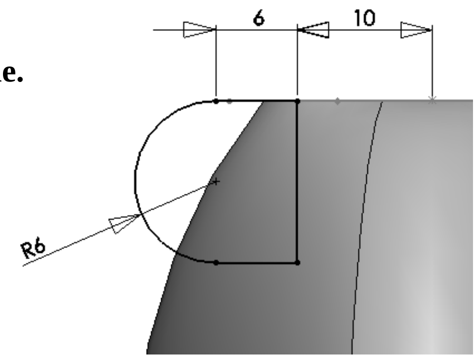


Profile Sketch

Choose **Right View**.

Create the sketch shown on the **Right plane.**

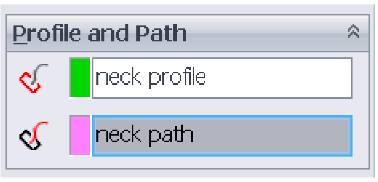
Exit the sketch and rename **neck profile.**

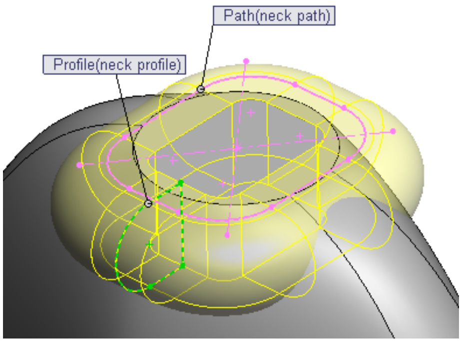


Neck of Bottle

Choose Swept Boss/Base

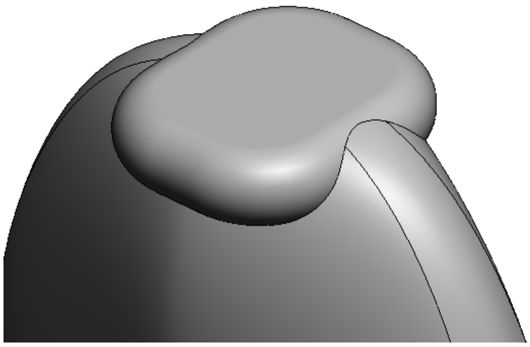
Choose the profile and path as indicated below.





Choose **OK.**

Rename the feature **Bottle Neck.**



Cylindrical Feature

We will use **profile2**, a circle of Ø25, which was used to define the loft earlier, as the sketch to create the cylindrical feature on the top of the bottle.

Show Sketch

Expand the **Body of bottle** feature.

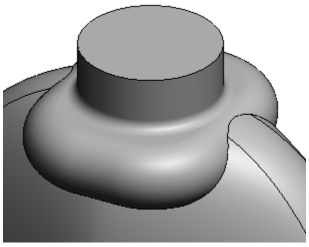
Select p**rofile2** and choose **Show.**

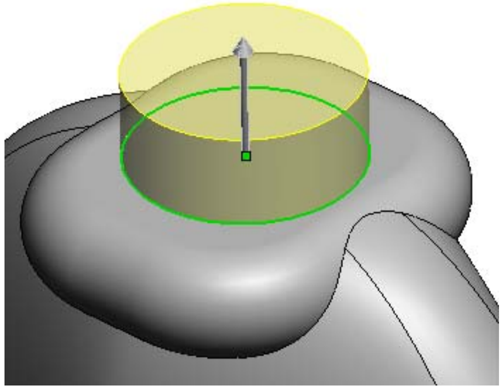
Extruded Boss/Base

Extrude this sketch a **distance** of **10mm.**

Fillet

Add a **2mm fillet** around the base of the cylindrical feature.





Embossed Label

The label on the front of the bottle is contained within a raised profile. This embossed profile will be created as a **Swept Boss/Base**. The **profile** for the sweep will be a circle of diameter 2.5mm. The **path** must be positioned on the face of the bottle.

Because the face of the bottle is not planar we must pursue another method of creating the path.

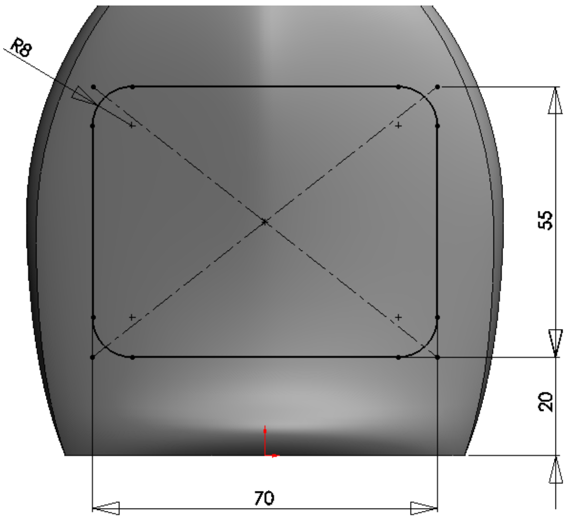
Projected Curve

Allows you to project a sketch onto a model face to create a 3D curve.

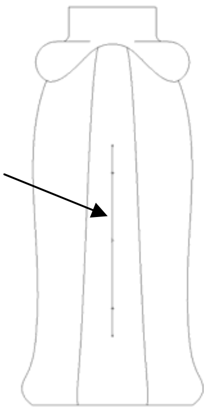
We will begin by creating the sketch on the **Front plane**.
Ensure that the centre of the rectangle is vertically above the origin.

Use sketch fillet to apply an 8mm fillet to the corners of the rectangle.

Close the sketch and rename it **label**.



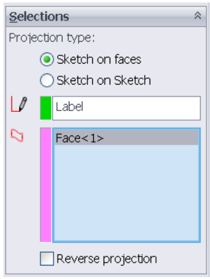
Position of sketch

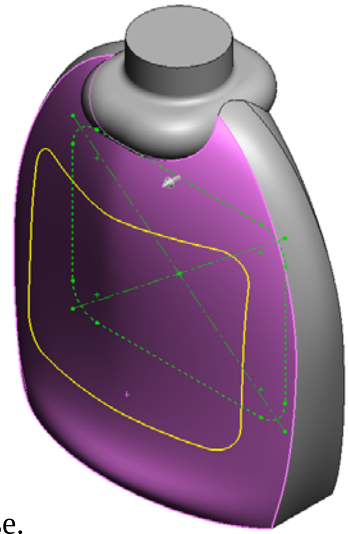


Projected Curve

Choose **Projected curve** from the **Curves** menu, on the features toolbar.

Selections:





Sketch on faces.

Label as the **sketch to project**

Front face as **Projection Faces**

The sketch will project onto the face and will be represented as Curve1 in the design tree.

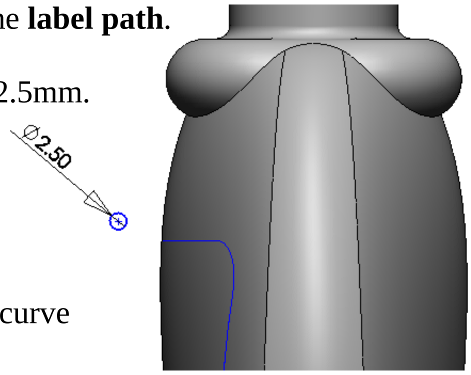
This 3D curve will act as the path for the Swept Boss/Base.

Rename the feature **label path**

Label profile

The profile for the label is a circle of Ø2.5mm. This sketch will be positioned on the right plane perpendicular to the **label path**.

Create a sketch on the right plane of a circle Ø 2.5mm.

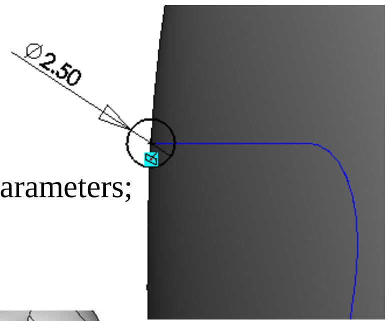


Pierce Relation

The **Pierce** relation will make the **centre of the circle** coincident with the **label path**.

Select the centre of the circle and the projected curve using ctrl.

Choose Pierce from the properties dialog box.

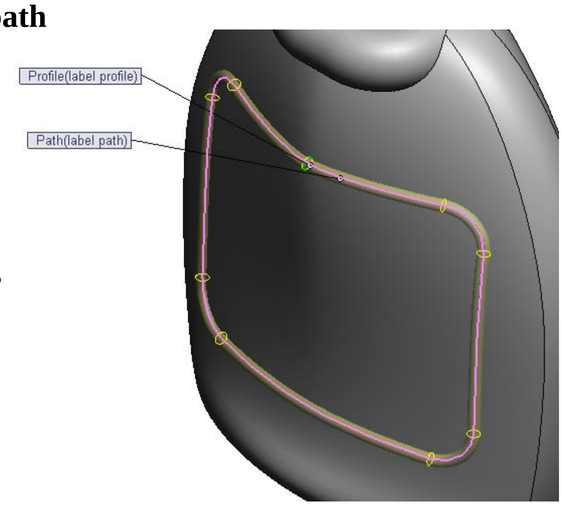


Exit the sketch and rename it label **profile.**

Swept boss/base

Create a swept boss/base feature using the following parameters;

Profile: **label profile**
Path: **label path**



Choose **OK.**

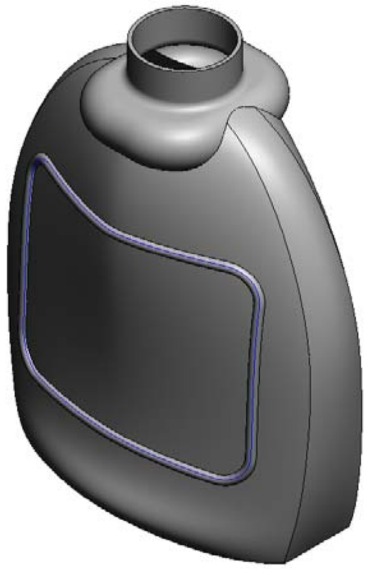
Rename the feature as **label area.**

Shell bottle **Shell** the entire feature using a **thickness** of **1mm**, choosing the top face as **faces to remove.**

Appearance Apply an appropriate appearance to the part.

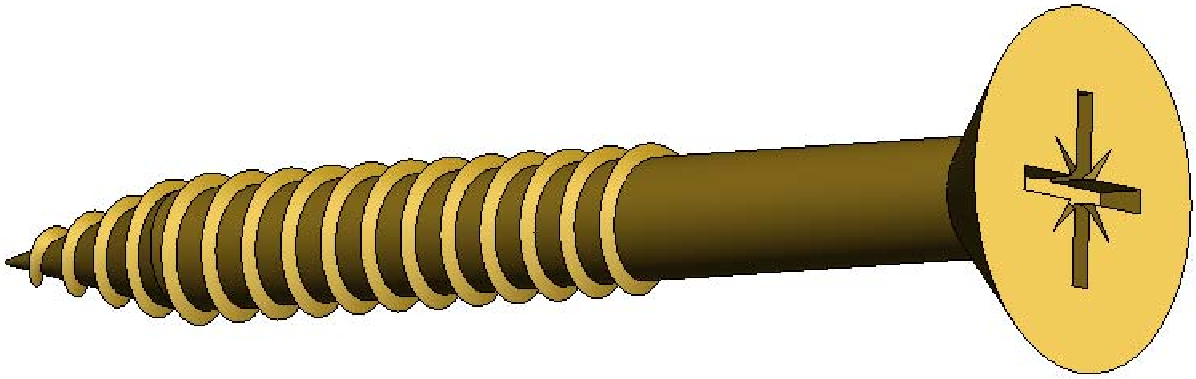
Use optical properties to reduce the **transparency** of the part.

Apply different appearance settings to the label face with transparency setting of zero.



Lesson Complete!

Introduction to Curves & Splines - Countersunk Screw



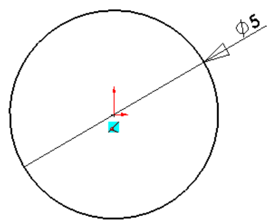
Prerequisite knowledge A basic knowledge of Solidworks 2009 is required – use of *sketching* and *extrude boss/base*.

Focus of Lesson This lesson will focus on using the following feature commands; **Helix** and **Circular Pattern.**

Commands Used This lesson includes **Sketching, *Extrude boss/base, Helix, Swept boss/base*** *and* ***Extruded Cut*** *with* ***Circular Pattern.***

Getting Started We will begin by creating the straight helical portion of the screw.

Create a sketch of a circle on the Top plane of diameter 5mm.

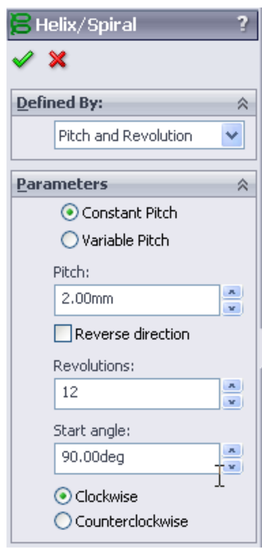


Position the sketch such that the origin is coincident with the center point of the circle.

Exit sketch and rename it **profile1.**

Save as *Save the part as **Countersunk screw***

Helix Use **profile1** as the sketch to create the Helix.



Choose the **Pitch and Revolution** option as the means of defining the helix.

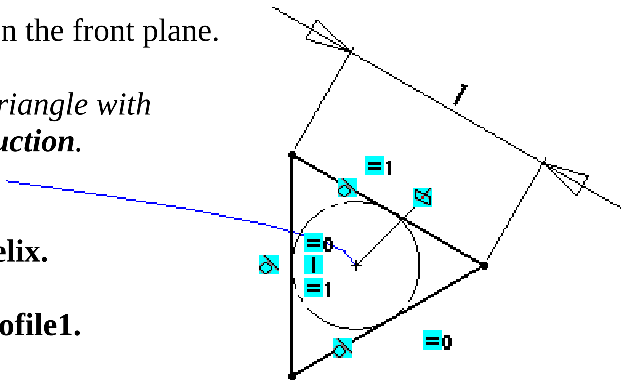
Set the **Pitch** to **2mm** and **Revolutions** to **12**.

The **Start angle**, ie where to start the first turn on the sketch circle, is **90°**. Choose **OK.**

Rename the feature **straight helix.**

Create Sketch We will create the profile of the screw thread next.

Sketch the equilateral triangle shown on the front plane.



*The sketch is based on an equilateral triangle with an inscribed circle, marked **for construction**.*

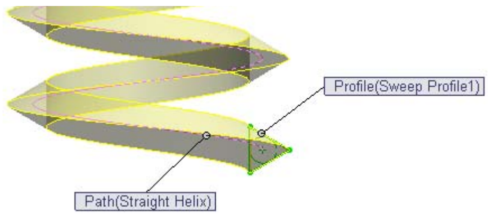
Ensure to pierce the centre point of the construction circle with the **straight helix.**

Close the sketch and rename **sweep profile1.**

Swept boss/base

Use **sweep profile1** as the **profile** and the **straight helix** as the **path.**

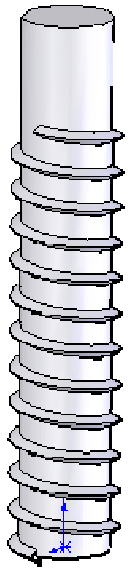
Rename the sweep as **thread1.**



Extrude boss/base

Select the **profile1** sketch from the feature manager design tree; choose **Show**.

This sketch will be used to generate the cylindrical part of the screw.



Extrude upwards a distance of 40mm**.**

Rename the extruded feature **body of screw.**

The lower portion of the screw has a tapered helix. We will again use the Helix/Spiral feature to create this, with some different parameters.

Create Sketch

Select the Top plane and create the same sketch as p**rofile1**; Circle diameter 5mm, origin coincident with the circle centre.

Rename the sketch **profile2**

Tapered Helix

Defined by: **Pitch and Revolution**

Pitch: **2mm**

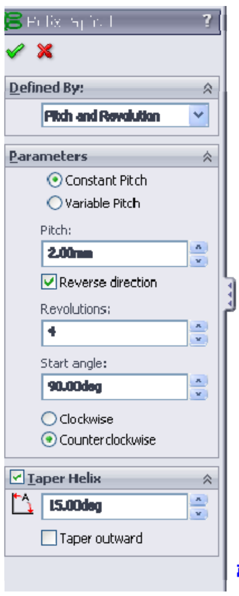
Reverse Direction

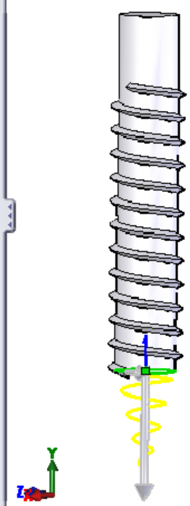
Revolutions: **4**

Start angle: **90°**

Counterclockwise

Tapered Helix of **15°.** Choose **OK**



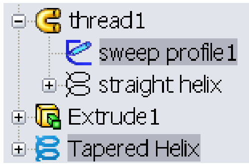


Rename

Rename feature **tapered helix**

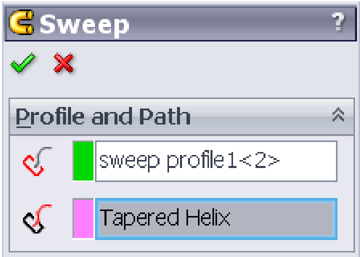
Swept boss/base

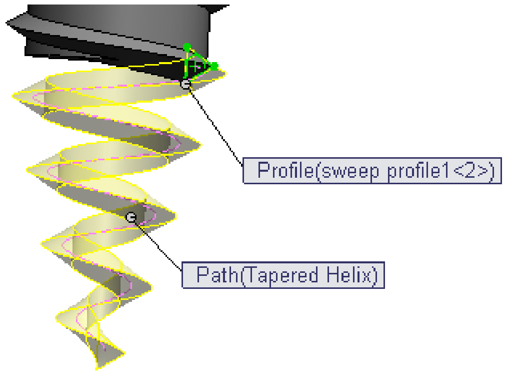
Select the sketch previously used as the profile for **thread1.**



Choose **Swept Boss/Base.**

Sweep profile1 will appear as **profile** as it was preselected. Use the **tapered helix** as the **path**.





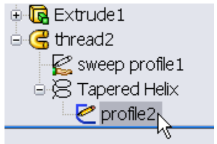
Rename

Rename the feature **thread2**

Extruded Boss/Base

We will use **Extruded Boss/Base** to add the material for the body of the screw in the tapered section.

Choose **profile2** as the sketch to be extruded.



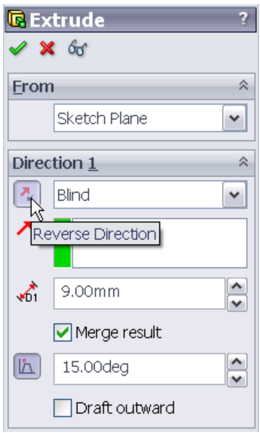
End Condition: Blind

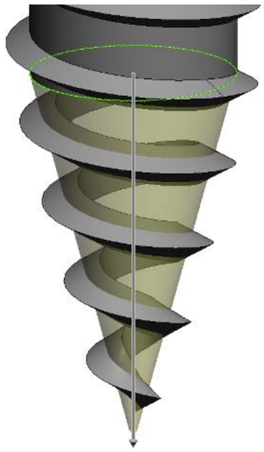
Distance: 9mm

Draft angle: 15°

Reverse direction.

Note: The angle matches that of the tapered helix





Rename

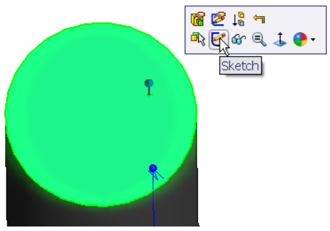
Rename **tapered portion**

Head of Screw

To create the head of the screw we will sketch a circle on the top surface of the body of the screw and extrude it at a draft angle to the required distance.

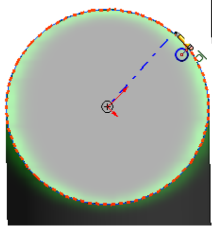
Create sketch

Create a sketch on the top surface of the body of the screw.

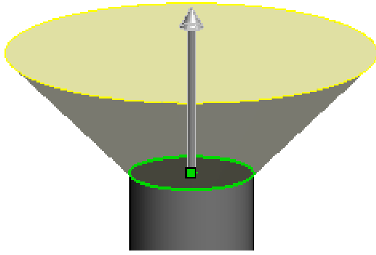


Sketch a circle of equal diameter to the screw body.

Note: Ensure to capture the necessary automatic relations.



Extruded Boss/Base Extrude this sketch with the following parameters;



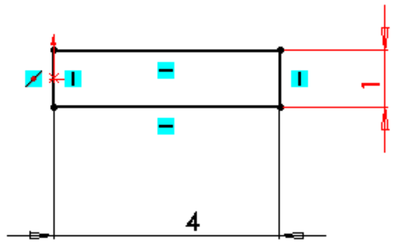
Rename feature **Head of Screw**

Screwdriver cutout Because the cutout for the screwdriver is symmetrical across two axes, we can create one simple sketch, create an extruded cut feature from that sketch and then create a circular pattern using that feature.

Create sketch Select the top surface on the **head of screw** and complete the following sketch.

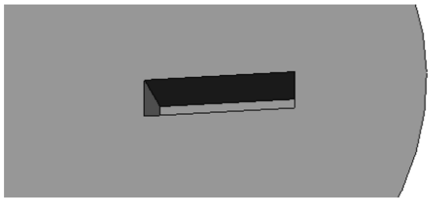
Ensure the origin is the **midpoint** of the vertical line on the left side of rectangle.

Close the sketch and rename as **head sketch**



Extruded Cut **Cut extrude** the sketch down into **head of screw** a **distance** of **2mm**.

Rename the feature **screw head cut.**



Circular Pattern 1 In order to create the circular pattern we must have an axis around which to pattern.

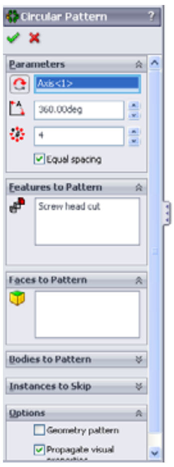
Temporary Axes Choose **View, Temporary Axes.**
An axis for the part will appear as a blue chain line.

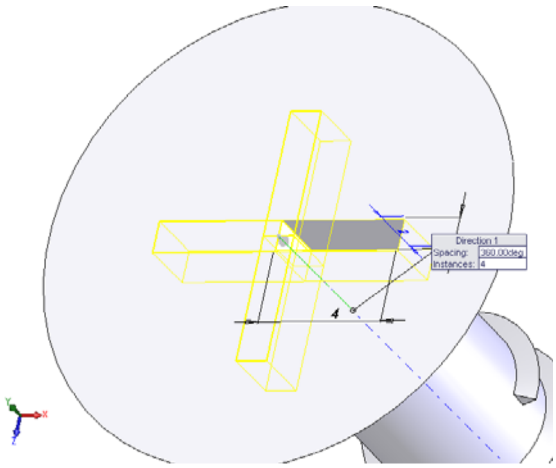
Circular Pattern Select **circular pattern** from the features tools.

Select the **temporary axis** as the **Pattern Axis**

Angle: 360°
Number of instances: 4
Equal Spacing

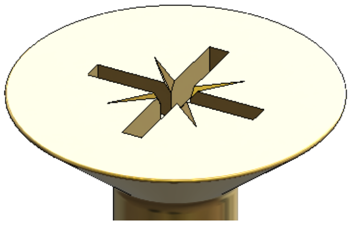
Choose **screw head cut** as the **features to pattern.**
Choose **OK**





Circular pattern 2

Taking a look at the completed model, we can see that there is a second cut to be made to complete the head of the screw.

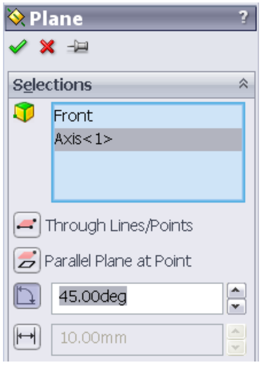


In order to create this cut we must first create the triangular profile of the cut.

This profile will be created on a plane at 45° to the front plane.

Reference geometry

To create the plane choose **Insert, Reference Geometry, Plane**



This plane will be defined as making an angle of **45°** to the **front plane** and the **temporary axis lies on it**.

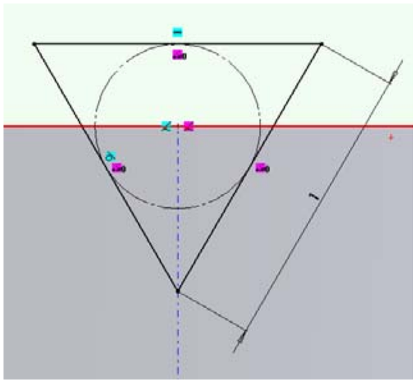
Reference Entities: **Front Plane**
Temporary Axis

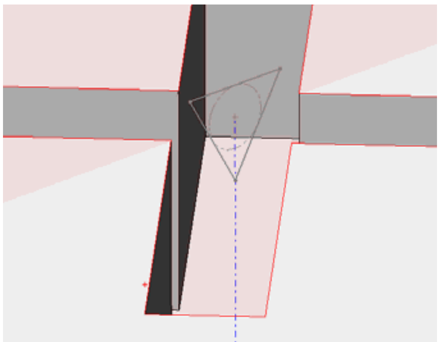
Angle: **45°**

Create Sketch

Create the sketch shown on **plane 1**.

Exit the sketch, rotate the model and note the position of the sketch.



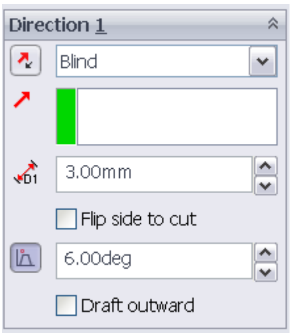


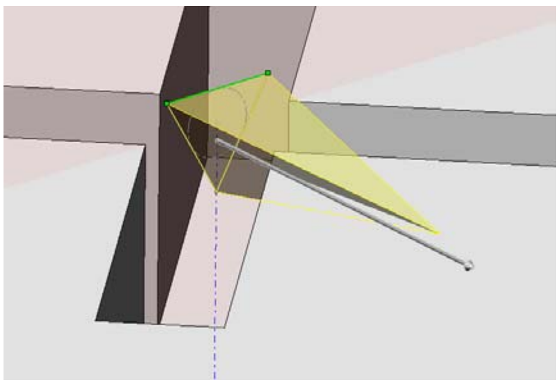
Rename

Rename the sketch as **head sketch1**

Extruded Cut

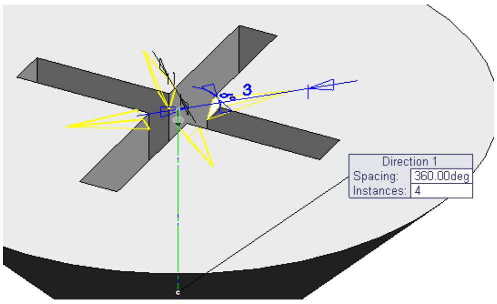
Extrude cut this sketch with the following parameters:
Distance: **3mm**
Draft: **6°**





Rename Rename the feature **screw head cut1**

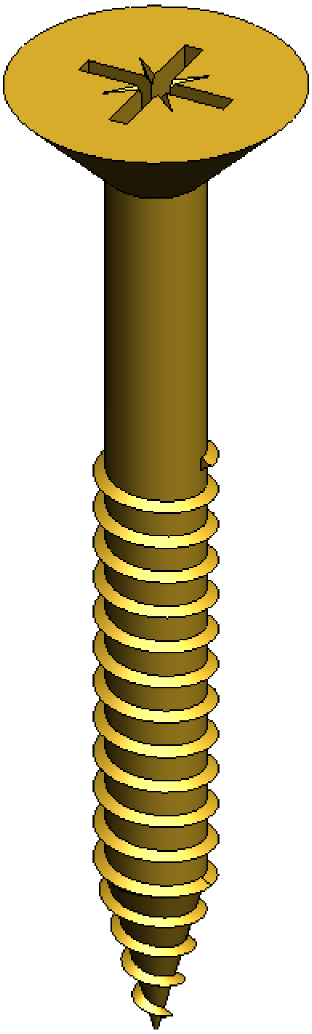
Circular pattern Create a **Circular Pattern** as before using the temporary axis as the **pattern axis** with this cut as the **feature to pattern.**



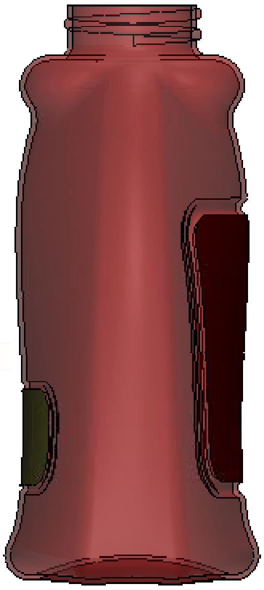
Hide **plane1** and the **temporary axes**.

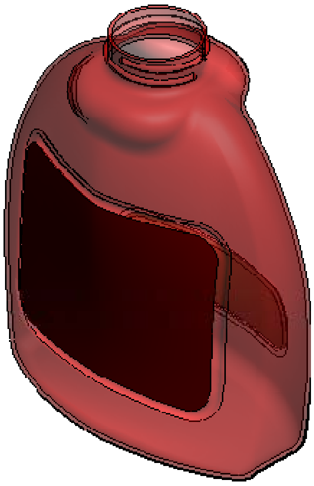
Appearance Apply a **brass** appearance to the screw.

Lesson Complete!



Modifying Curved Features -Hand soap Bottle Modifications





Prerequisite knowledge A basic knowledge of SolidWorks 2009 is required – use of *sketching* and *extrude boss/base*.

Focus of Lesson This lesson will focus on applying modifications to an existing SolidWorks part.

Commands Used This lesson includes **Sketching, *Dome, Project curve, Swept cut and Helix.***

Getting Started Open 'Soap Bottle' part as created in lesson 2.

Save *Save the part as **Soap Bottle Modified***

Suppress Shell Right click on the **Shell** feature in the featuremanager design tree and choose to **Suppress**.

This will disable the shell feature, making the bottle solid We can unsuppress the feature after the modifications have been completed

Delete Sweep (Label area) Delete the **label area** sweep completed on the model keeping the ***label path*** and ***label profile***. We will use the profile and path to create a **swept cut.**

We can use the same path and profile to create a **Swept Cut** which will remove the **label profile** from the bottle following the **label path**

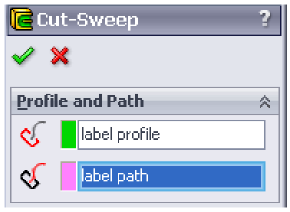
Swept Cut

Select **Swept Cut** from the features toolbar.
Choose:

label profile as the **profile**
label path as the **path**

Choose **OK.**

Rename swept cut as **label indent.**



Dome

The base of the model is flat. In reality the underneath of the soap bottle is curved inwards. This curved geometry may be added using the **Dome** feature.

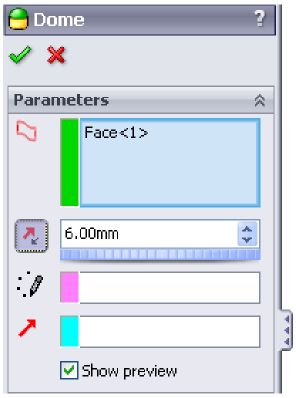
The dome feature allows you to add a dome to planar and non-planar surfaces.

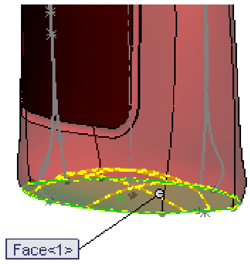
Adding the dome

Select the **bottom surface** of the bottle and choose **dome**

Apply the following settings:

Distance **6mm**





Reverse direction

Choose **Reverse Direction**

This ensures the dome is applied to the inside of the bottle and not protruding below the model.
Choose **OK.**

Rename the feature as **Base.**

Fillet

We will use the **fillet** command to add more rounded contours to the model to mimic the original model.

Choose **Fillet.**

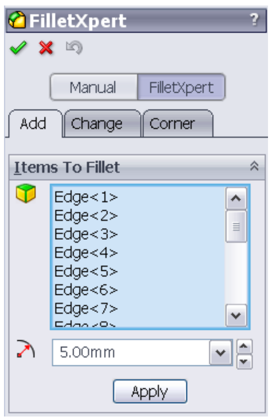
Select the **FilletXpert** tab.

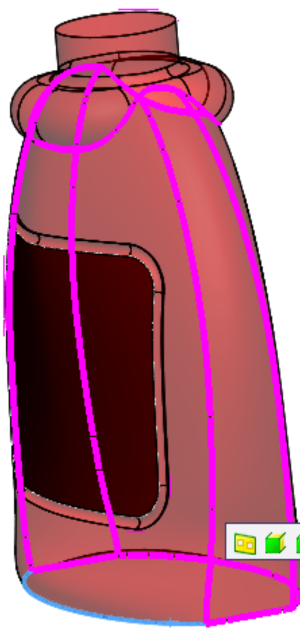
Choose a **radius** of **5mm.**

Select an edge at the base. A pop-up menu appears with a number of pre-selected edges based on the chosen edge.

From the pop-up menu choose **Connected 19, Edges.**

Choose **OK.**

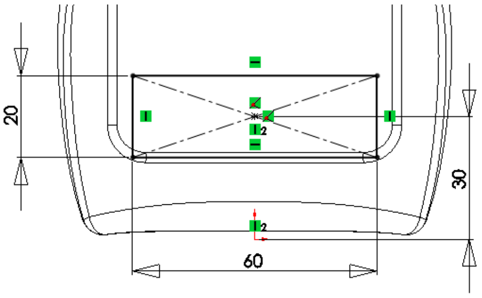




Rear Label

We will add another label area to the rear of the bottle using the same procedure as was used to create the front label area.

Create the sketch shown on the front plane.

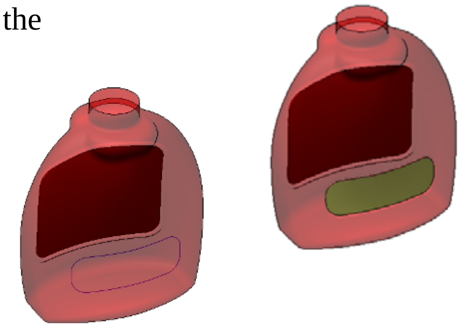


Sketch Fillet

Apply an **8mm sketch fillet** to all corners of the rectangle

Project Curve

Using **Project Curve**, project the sketch onto the back face of the bottle.



Cut Sweep

Create a profile for the cut sweep.

The **Profile** is a Ø1mm circle drawn on the right plane and pierced with the path.

Path – the projected curve just created.

Rename the feature **rear label indent.**

Helix (Thread)

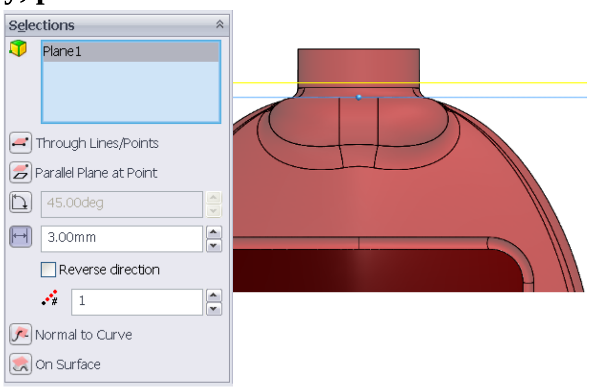
A **helix** is required to create the thread on the top of the bottle. This helix will be based on a profile circle located 3 mm above the neck of the bottle.

Insert Plane

Choose **Insert, Reference geometry, plane…**

Select **Plane1** as **reference entities** and a **distance** of **3mm.**

Choose **OK.**



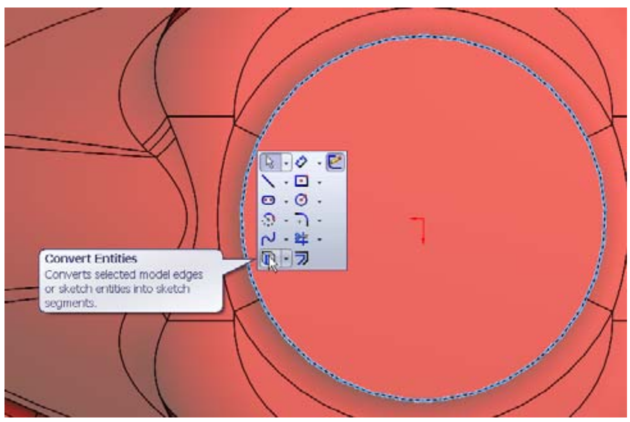
Create Sketch

Create a sketch on **Plane2.**

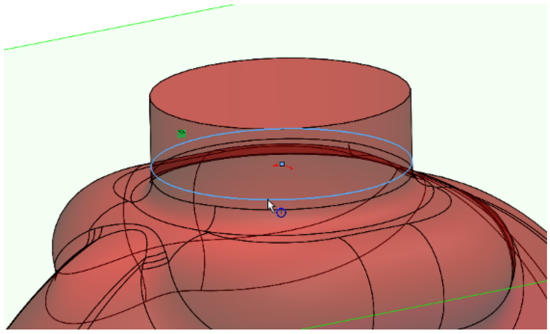
Choose **TopView.**

Convert Entities

Select the circular profile and choose **Convert Entities.**



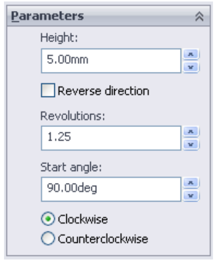
This will create a circular profile sketch on plane2



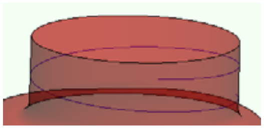
Exit the sketch.

Show/Hide — **Hide Plane2.**

Helix — Create the **helix** using the **height and revolution** as the definition.

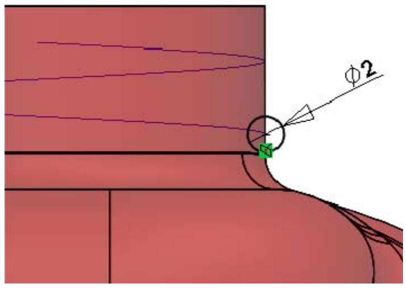


Height: **5mm**
Revolution **1.25**
Start angle: **90°**



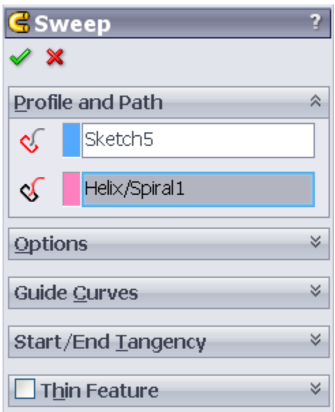
Swept boss/base — Create a circular sketch, Ø2mm, on the front plane.

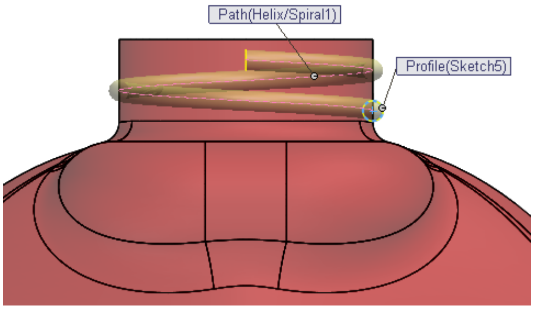
Pierce the centre point with the **Helix** just created.

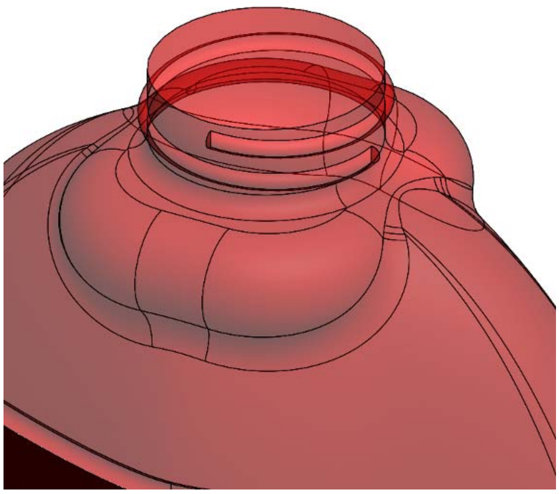


Swept Boss/Base — Create a swept boss/base using the following parameters:

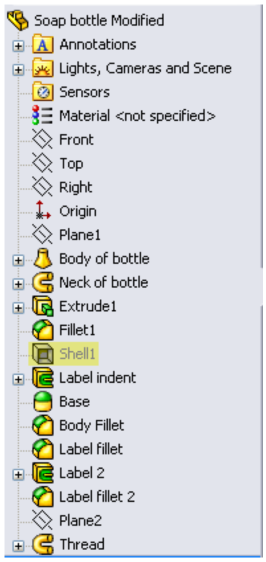
circular sketch as the **profile**
helix as the **path**

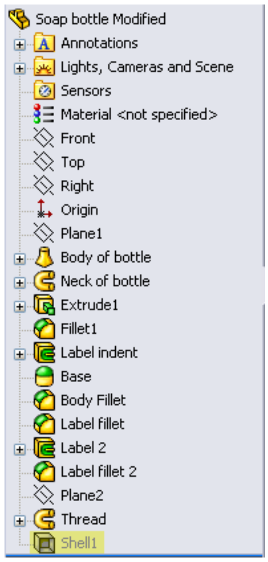






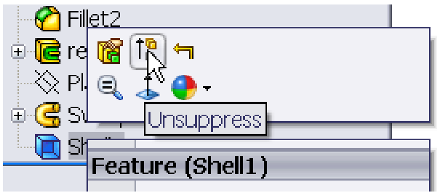
Shell Move the **shell** from its position in the design tree to the end of the list in the design tree.





Drag & drop Select and hold the feature in the featuremanager design tree. Drag the mouse to the end of the list and release.

Unsuppress Right click on the **shell 1** and **unsuppress**.

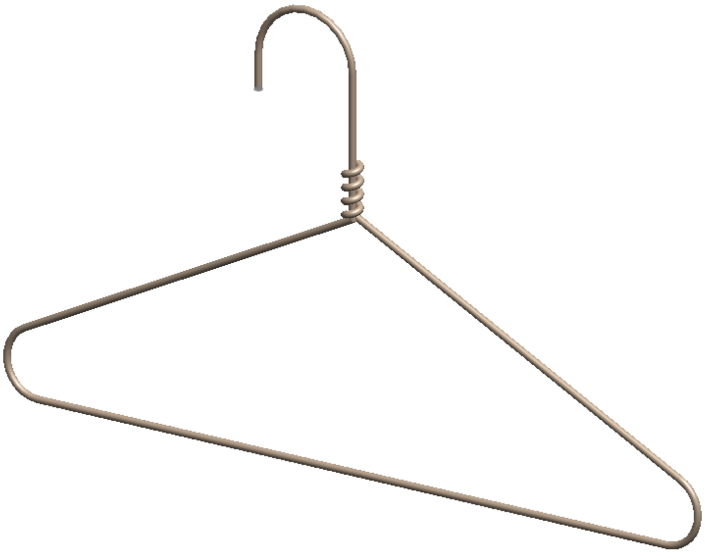


The object is shelled as before.

Display Style Choose **Shaded** as the display style.

Lesson Complete!

Composite Curve – Wire Clothes Hanger



Prerequisite knowledge A knowledge of SolidWorks 2006/2009 is required – use of *sketching*, helix and *swept boss/base*.

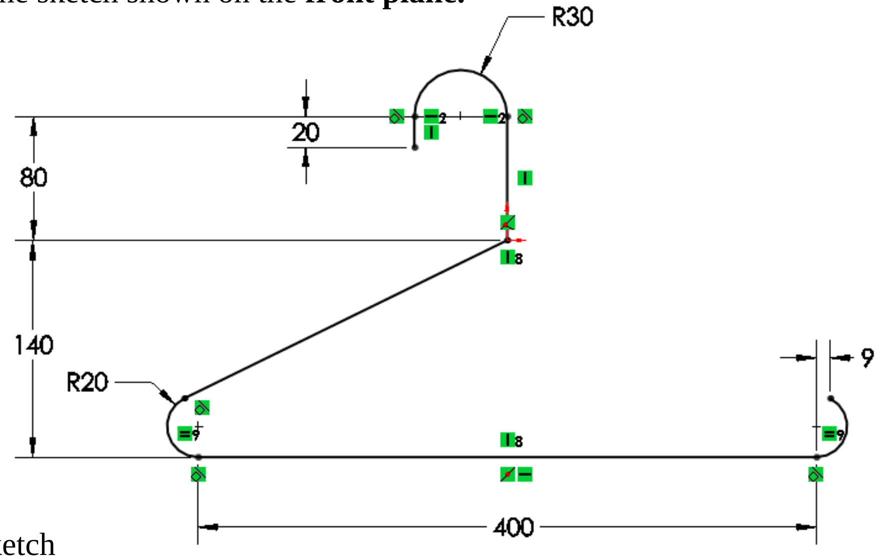
Focus of Lesson This lesson will focus on using **Composite Curve**.

Commands Used This lesson includes ***Helix, 3D Sketch, Composite Curve & Swept Boss/Base.***

The hanger will be created using Swept Boss/Base. The profile will be a circle. The path will be made up of 3 separate sketches. These 3 sketches will have to be joined to form one curve in order to use it as the sweep path.

Getting Started Create a new part file & save as **Wire Clothes Hanger**.

Create Sketch Create the sketch shown on the **front plane.**

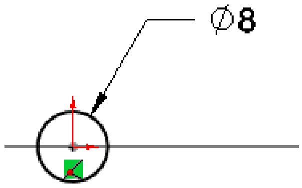


Exit Sketch

Helix The neck of the clothes hanger is based on a helix. The helix is based on a Ø8mm circle.

Create sketch Create the sketch shown on the Top plane.

Note the prudent positioning of the origin at the outset.



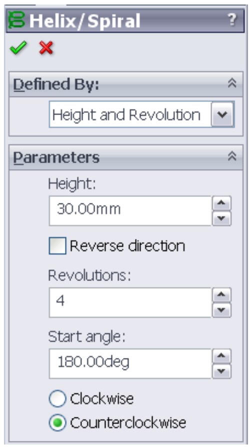
Helix Choose **Curves, Helix and Spiral.**

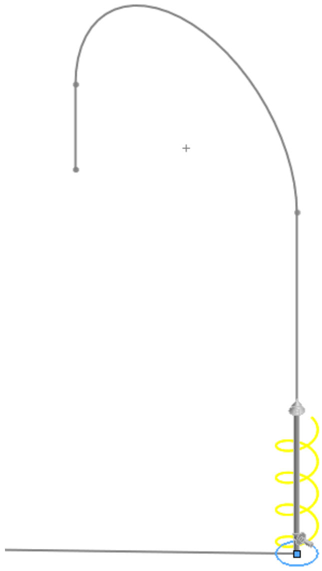
Choose the circular sketch as the basis for the helix.

Define the helix using the following parameters;

Height and Revolution

Height: **30mm**
Revolutions: **4**
Start Angle: **180°**
Counter Clockwise

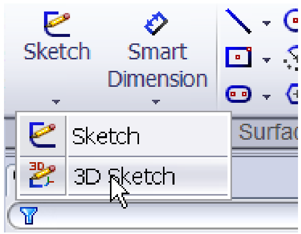




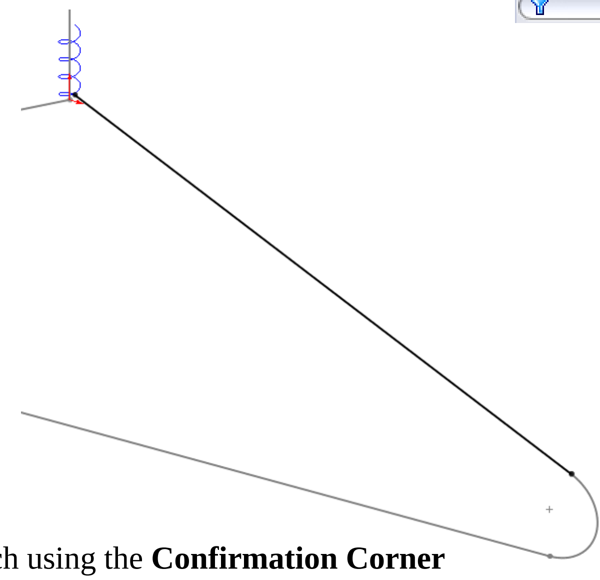
Choose **OK.**

3D Sketch To complete the path we must join the helical curve to the initial sketch. We will create a 3D sketch line to join their endpoints.

Choose 3D sketch from the Sketch toolbar. Select **line**.



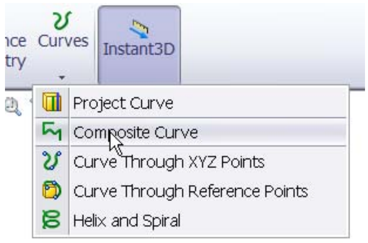
Add a 3D sketch line coincident with the endpoint of the helix and sketch1, as shown.



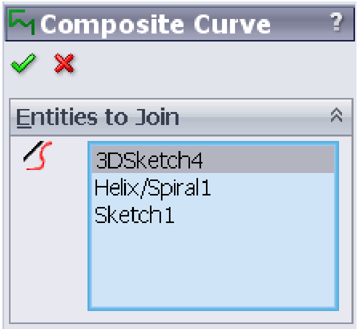
Exit the 3D sketch using the **Confirmation Corner**

Composite Curve At the moment the sweep path is made up of Sketch1, the helix and the 3D sketch joining the endpoints of both. As discussed earlier, these 3 entities must be joined together to form one curve. The **Composite Curve** feature enables us to do this.

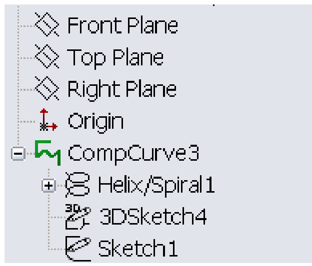
Choose Composite Curve from the features toolbar.



Make Selections Select the aforementioned as **entities to join.**



Choose **OK.** The 3 entities will be consumed under the composite curve feature in the design tree.



Rename the feature **hanger path.**

Hanger Cross-section The cross-section of the hanger is a circle of **Ø4mm.** This will act as the profile for the swept boss/base.

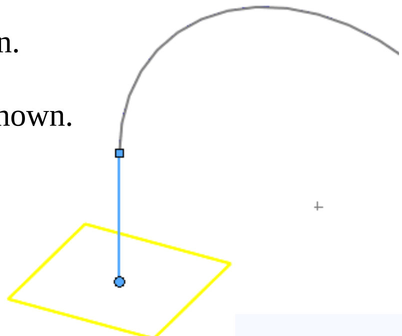
Insert Plane Prior to creating the sketch representing the circular cross section we must first create a plane on which to sketch.

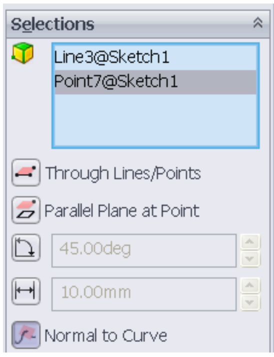
Choose **Insert, Reference Geometry, Plane.**

Select **Normal to curve** option.

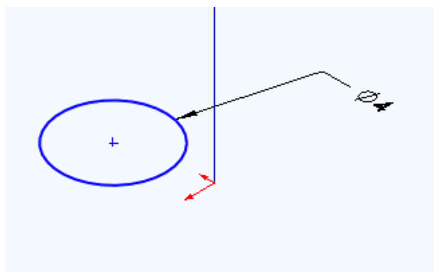
Choose the line and point as shown.

Choose **OK.**

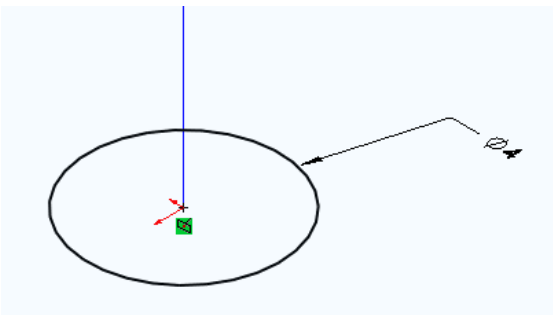




Create Sketch Create a circular sketch of **Ø4mm** on the plane.



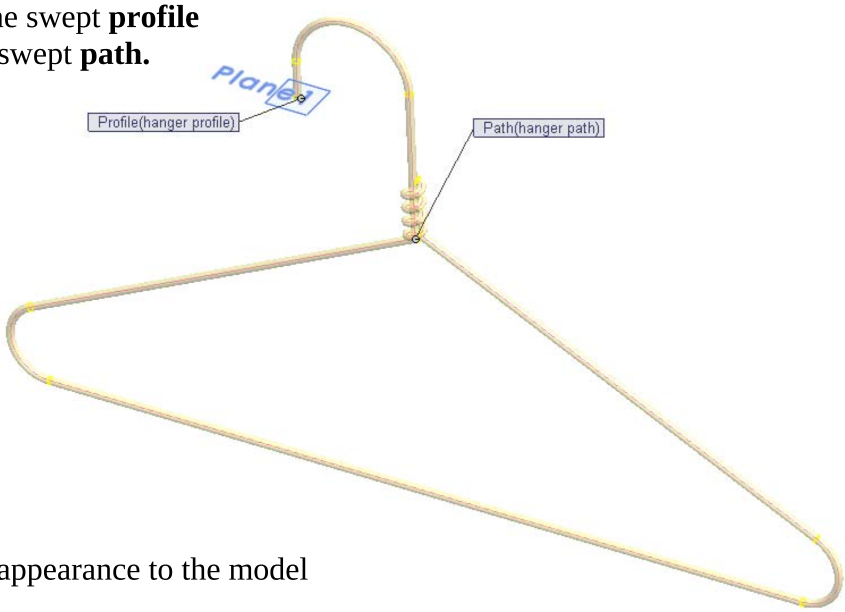
Add a pierce relation between the centre of the circle and the composite curve.



Rename the sketch **hanger profile.**

Swept Boss/Base Create a swept boss/base using;

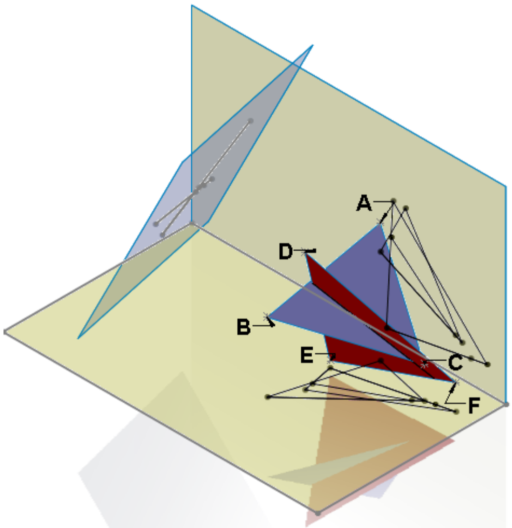
hanger profile as the swept **profile**
hanger path as the swept **path.**

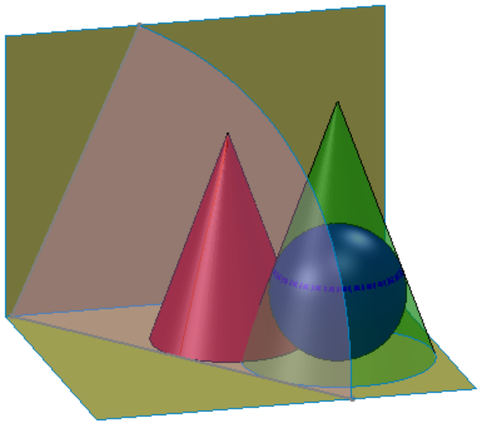


Show/hide **Hide** Plane1.

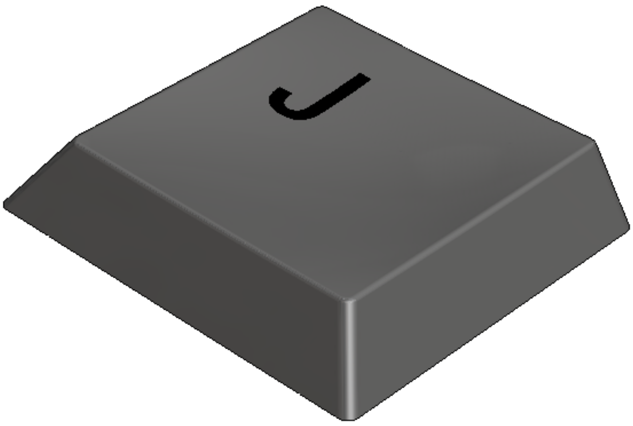
Appearance Add an appropriate appearance to the model

Lesson Complete!

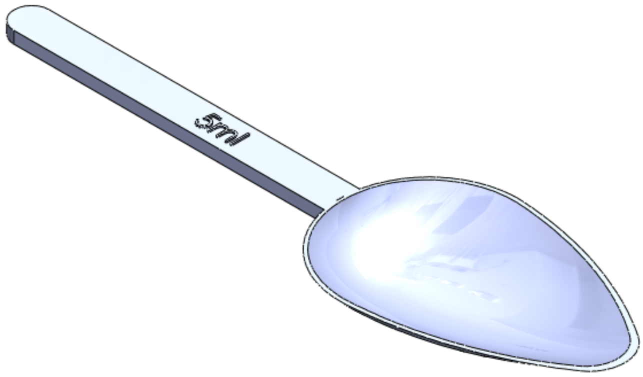




Working with Surfaces
SolidWorks 2009



Plastic Medicine Spoon



Surface Modelling

In surface modelling a model is built face by face. Faces created by surface features may knit together to enclose a volume, which may be turned into a solid model.

Surface modelling is used to create faces and features which may not be conveniently produced using solid modelling techniques. Surface tools are employed in situations where they make it easier, more efficient or even possible to complete the task at hand.

The focus of this exercise is to give a basic introduction to surfaces and explore the functionality of some surfacing tools.

As we work through the exercise we will explain the terminology associated with surfaces.

Prerequisite knowledge

To complete this model you should have a working knowledge of Solidworks 2006/2009.

Focus of lesson

This lesson focuses on using the following surface tools; ***Filled Surface, Surface Thicken*** *and* ***Cut with Surface*** as well as ***Shell*** *and* ***Extrude*** feature tools.

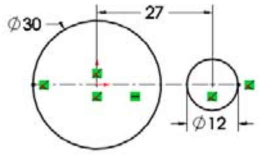
Getting started

New File Create a new part file and save it as **Plastic Medicine Spoon** in the desired location.

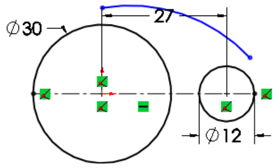
New Sketch We are going to begin by creating a sketch to represent the top profile of the spoon. Create a sketch on the Top plane.

Steps required Sketch the centreline as shown.

Sketch a circle of diameter 30mm with its centre coincident with the origin. Sketch a diameter 12mm with its centre Coincident with the centreline and dimension as shown.

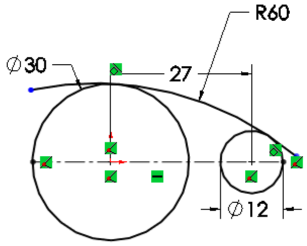


Create the **centrepoint** arc shown.

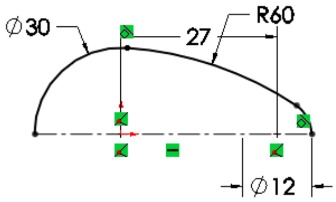


Smart dimension 60mm.

Apply a **tangent relation** between the arc and the two circles.

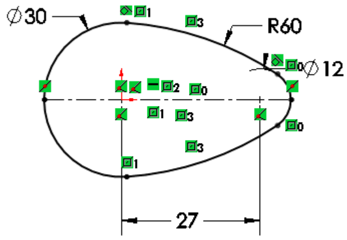


Trim the sketch using **Power Trim**.



Using **Mirror** copy the sketch across the centreline.

We have now created the profile for the outline of the spoon.



Confirm the sketch.  Rename the sketch **top profile.**

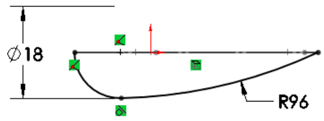
Sketch 2 Having created the top profile we will now sketch the front profile of the spoon. Create a new sketch on the front plane.

Steps required. We need to use the centreline from sketch 1 in sketch 2. To do this we use the **convert entities** command. Select the centreline, choose **convert entities.** 
The centreline becomes an entity within the current sketch and appears as a line. Use line properties to convert it back to ‘for construction’. It will now appear as a chain line within the current sketch.

The centreline can now be used to create the relations required to define the sketch.

Use circle and centrepoint arc create the following sketch.

Smart dimension the sketch and apply the relations shown.



Now we have the profile for the curvature of the spoon when viewed from the front.

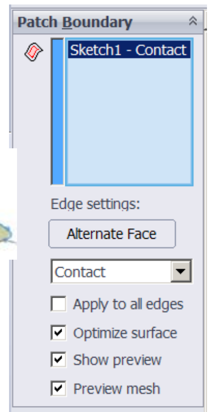
Confirm the sketch. Rename the sketch **front profile.**

Filled Surface

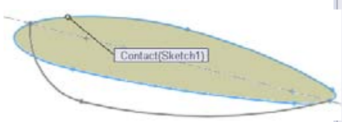
Filled surface enables you to create a surface or 'Patch' defined by edges, lines or curves.

Steps required.

Select the filled surface tool from the surfaces toolbar.

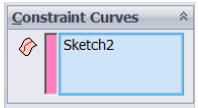


Select **top profile** as the patch boundary. This is the outline of the spoon.



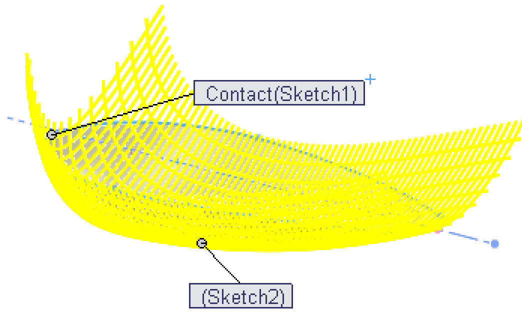
Check **Optimize surface** to ensure that the patch or surface will finish at this outline.

A planar surface is now created within the boundary of this sketch.



Select **front profile** as the constraint curve. This curve will direct the surface to 'bend' to give the curvature required to create the surface.

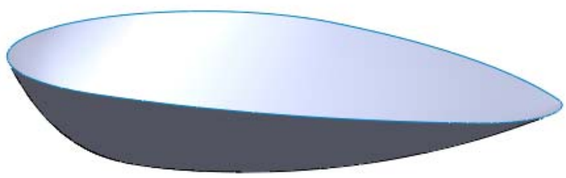
Select OK



The surface is now created!

Rename the feature

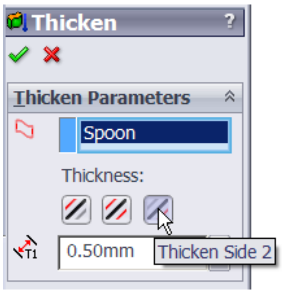
Rename the feature as **Spoon.**



From Surface To Solid

When a surface is created in SolidWorks it has no thickness. If we take a section view through the surface we can see this.

To give the spoon wall a thickness we use the **Thicken** command from the surfaces toolbar.



Select **Spoon** as the surface to thicken.

Set the thickness to 0.5mm.

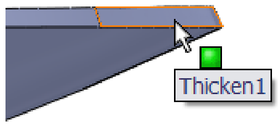
We have three options to choose from in order to thicken the surface.
Selecting ‘Thicken side 1’ will thicken to the outside,
‘Thicken both sides’ will thicken either side of the surface,
‘Thicken side 2’ will thicken to the inside.

Choose **Thicken Side 2**

Select OK.

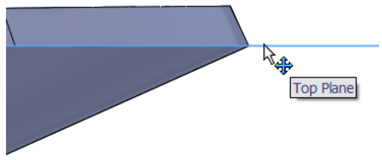
Trimming back Surface thicken

You will notice that the surface thicken gives a bevelled finish to the edge of the spoon.

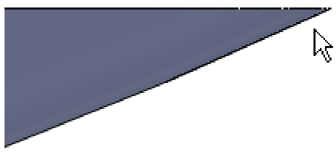


To correct this we will use **Cut with surface** to trim the excess off the top edge.

Cut with surface removes unwanted material by cutting with a surface or a plane.



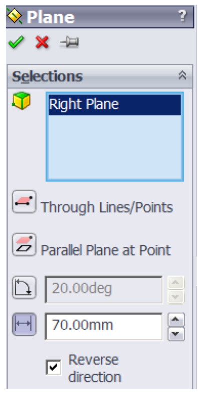
As the **Top Plane** will conveniently cut the solid in this case it will be used as the cutting plane.
Reverse direction if required.



Select OK.

The edge is now planar.

Creating the Handle

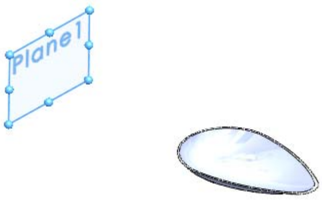


To create the handle we must first set up a plane parallel to the **Right Plane** and create a sketch on that.

From the **Surfaces** menu select **reference geometry, plane.**

Select the **Right Plane** as a reference entity.

Check **Reverse direction**.
Set the **distance** to **70mm**.

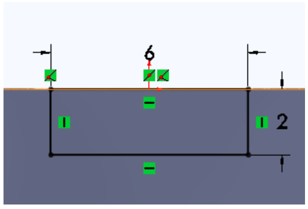


Select OK

Sketch

Sketch a **corner rectangle** on **Plane1.**

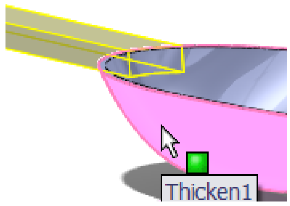
Apply a coincident relation between the midpoint of the top line of the rectangle and the origin as shown.



Confirm the sketch.

Extruding the Profile

Select **Extruded Boss/Base** from the features menu. Select **Up to Surface** as the **end condition**. Select the outer surface of the spoon as the **face/plane** to extrude to.



Select OK

Rename the feature

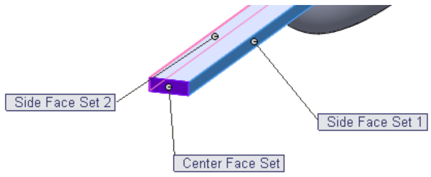
Rename the feature as **'Handle'.**

Hide Plane 1

Click on **Plane 1** and select **hide.**

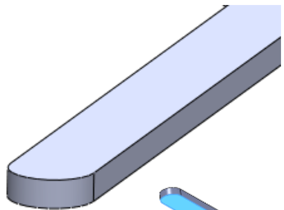
Fillet

Apply a **Full round fillet** to the sides of the handle. Select the faces shown.



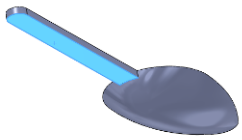
Select OK

The end of the handle is now rounded shown.



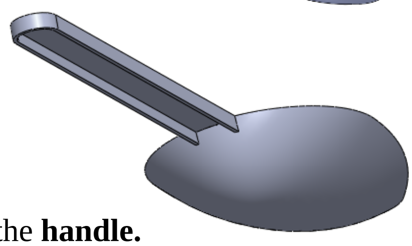
Shell Feature

Select **Shell** from the **Features** menu. Select the underside of the ***Handle*** as the faces to remove.



Set the thickness to 0.5mm.

Select OK

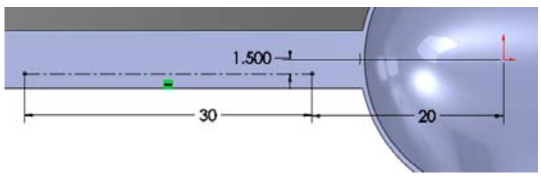


Creating text Feature

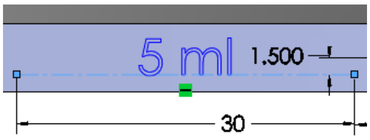
Create the following sketch on the top surface of the **handle.**

Select **Text** from the **sketch** menu.

Make the following selections:

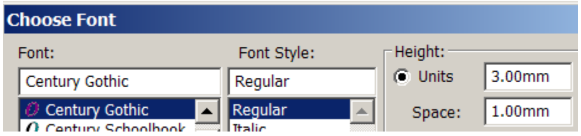


Select the **centreline** as the line for the text to follow.



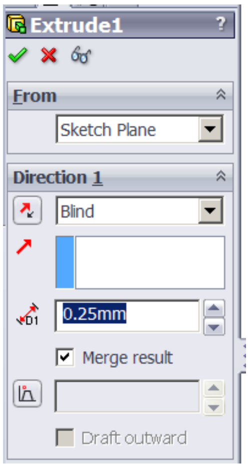
Type in **5 ml** as the text.

Select ***Century Gothic*** as font, set the units to 3mm and centre align the text.



Select OK

Extrude Text Select **Extrude** from the **Features** manager

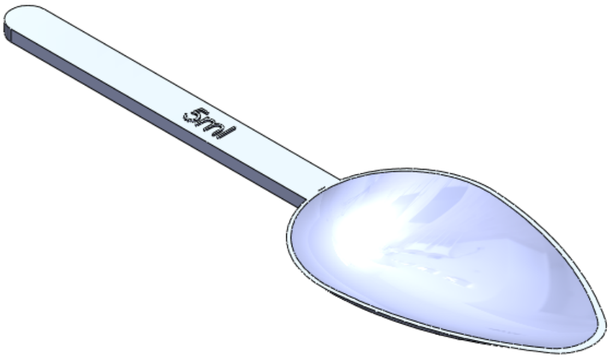


Set the thickness to **0.25mm**.

Select OK

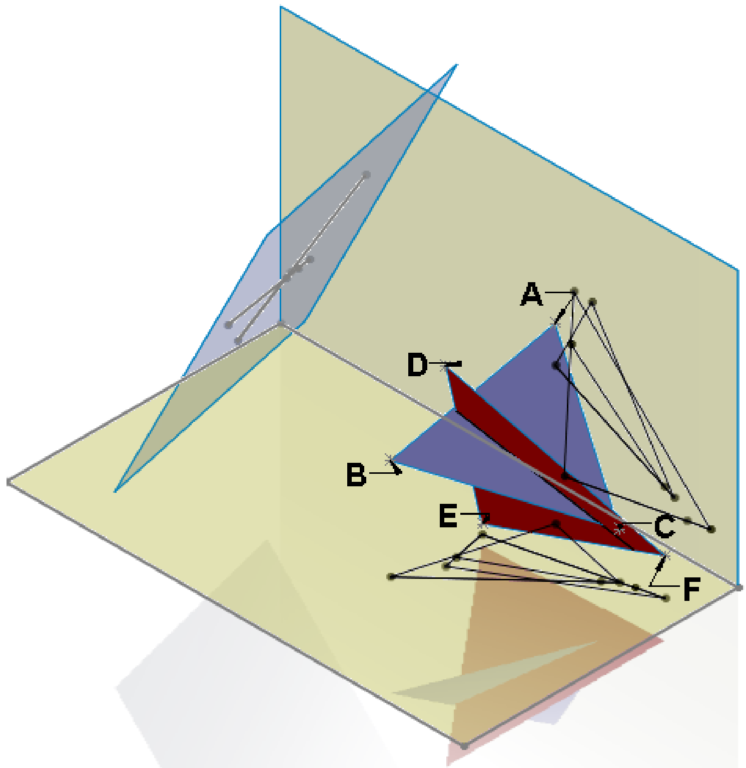
Rename Feature Rename the feature as **Text.**

Select Material Set the material as **PTFE**.



Exercise complete!

Intersecting Lamina



Prerequisite knowledge To complete this model you should have a working knowledge of Solidworks 2006/2009.

Focus of lesson This lesson focuses on using SolidWorks to solve a geometrical problem. The following **Surfaces** tools are used: **Planar Surface, Ruled Surface.**

Problem The horizontal and vertical coordinates for two intersecting planes **ABC** and **DEF** are given below.

A	=	170	---	95	---	20
B	=	215	---	25	---	30
C	=	150	---	55	---	90
D	=	235	---	20	---	25
E	=	155	---	5	---	45
F	=	160	---	95	---	70

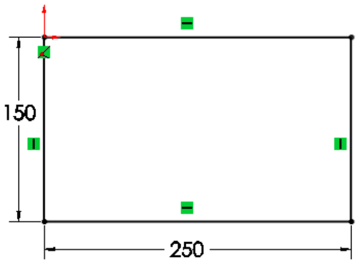
(a) Draw the plan and elevation of the intersecting planes

(b) Determine the line of intersection between the planes

(c) Determine the dihedral angle between the planes

New File — Create a new part file and save it as **Intersecting Lamina** in the desired location.

New Sketch — We are going to begin by creating a sketch to represent the outline of portion of the **Horizontal Plane**.

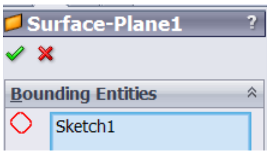


Create the sketch shown on the Top plane.

Smart Dimension as shown.

We want to transform this rectangle into a **Planar Surface.**

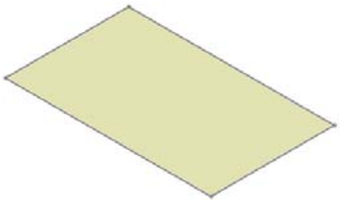
Planar Surface — Select **Planar Surface** Planar Surface from the **Surfaces** toolbar.



Select **Sketch 1** as the **Bounding Entities.**

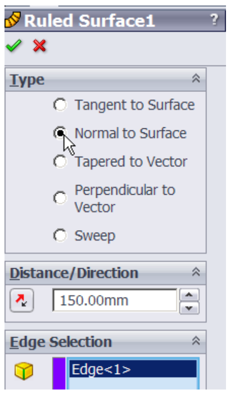
Select **OK**

Rename Feature — Rename the feature as **Horizontal Plane.**



We have now created a portion of the horizontal plane. This planar surface has no thickness but can be used as a datum for measurements, a surface to project views onto, or a surface to sketch on.

The Vertical Plane — To create the vertical plane we use **Ruled Surface.** **Ruled Surface** command Ruled Surface creates surfaces that extend out in a specified direction and distance from selected edges.

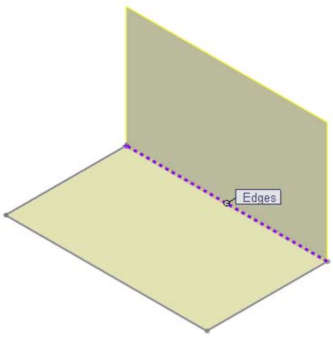


Ruled Surface — Select **Ruled Surface** from the **Surfaces** toolbar.

Select **Normal to Surface** as the **Type**. This will create a ruled surface at 90 degrees to another surface at a specified edge.

Set the **distance** to 150mm. This will extend the surface out 150mm from the selected edge. The width of the surface will be determined by the length of the edge selected.

Select the edge of the **Horizontal Plane** shown as the edge to set up the ruled surface from.

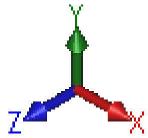


Select **OK.**

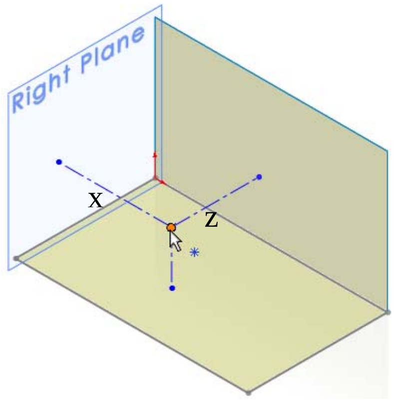
Rename Feature — Rename the feature as **Vertical Plane**.

Positioning Co-Ordinates

In order to use the **XYZ** Co-ordinates to position the points **A, B** and **C**, select **3DSketch** from the **Sketch** toolbar.



The co-ordinates are positioned as follows:
The **X** value is the distance of the point from the **Origin** or the **Right Plane**.

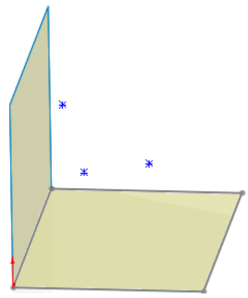


The **Y** value is the distance of the point from the **Horizontal Plane**.

The **Z** value is the distance of the point from the **Vertical Plane**.

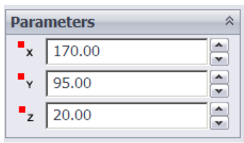
Using the **front view** we can see the **X** and **Y** values; using **top view** we can see the **X** and **Z** values; use the **right view** to see the **Y** and **Z** values.

Rotate the sketch as shown so that when the points are dropped in we can avoid making them coincident with the two planes.



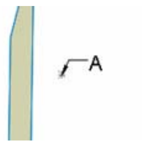
Using the **Point** command from the **Sketch** toolbar, drop **3** points into the sketch as shown. Select **OK** and press escape to exit the **Point** command.

We must now enter the coordinates of the points. Select one of the points to access its co-ordinates or **parameters**.



Enter the following values for the point.

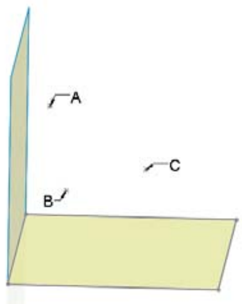
Select **OK**.



Labelling the Coordinates

From the **Insert** dropdown menu, select **Annotations, Note** Note...

Make the endpoint of the arrowhead coincident with the points as shown, select **Arial** as the text, and label the point **A**.



Repeat this procedure for points **B** and **C** entering the co-ordinates as given at the beginning of the exercise.

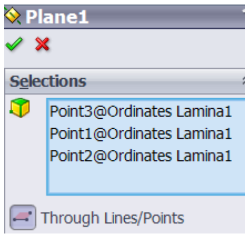
N.B. to change the values of the coordinates you must first hide the **annotations**. This is done from the ***heads up toolbar***. Select **hide/show items**, de-select **annotations display.**

Rename Feature

Rename the **3DSketch** as **Ordinates Lamina ABC**.

Insert Plane

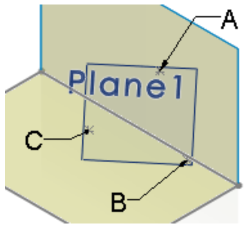
We now want to create a lamina containing all three points. In order to do this we must create a **Plane** which contains all three points.



Select **reference geometry, plane**

Choose **through lines/points** as entities for selection.
Select the three points **A, B** and **C.**

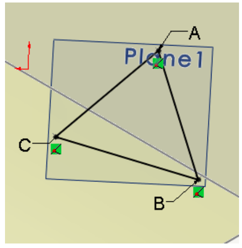
Select **OK.**



Creating the Lamina

We can now create the lamina on this plane. Using the **line** command, create the following sketch on **Plane1**.

(Note the automatic relations)

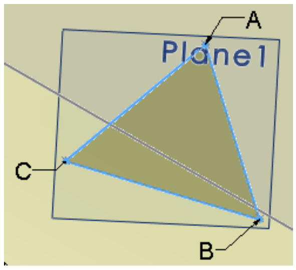


Rename Sketch

Rename the sketch as **sketch abc**.

Planar Surface

Select **Planar Surface** from the **Surfaces** toolbar. We use **planar surface** so that if needs be we can sketch on it or project lines/points onto it.



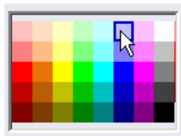
Select **sketch abc** as **bounding entities**.

Rename Feature

Rename the feature as **ABC**

Select **OK.**

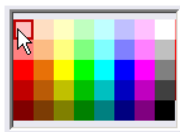
Hide **Plane1.**



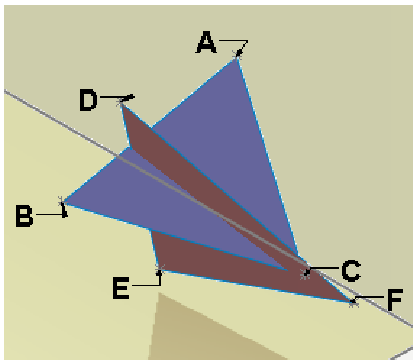
Set the colour of **ABC** as shown.

Lamina DEF

Create **Lamina DEF** following the same steps, and using its coordinates as given at the beginning of the exercise.



Set the colour of **DEF** as shown.



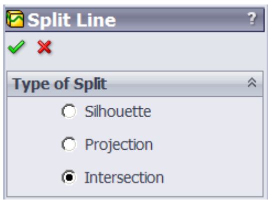
We have now created the intersecting lamina.

Line of Intersection

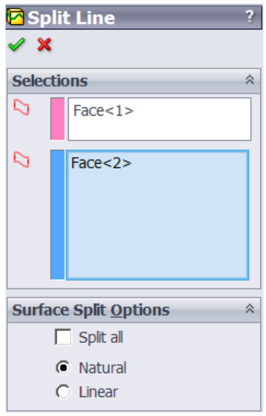
To determine the line of intersection between the two planes we use the **Splitline** command. When using **Splitline** in this situation we select one lamina as the **cutting plane** and the other as the **cut plane**.

From the **Surfaces** toolbar select **Curves**, 

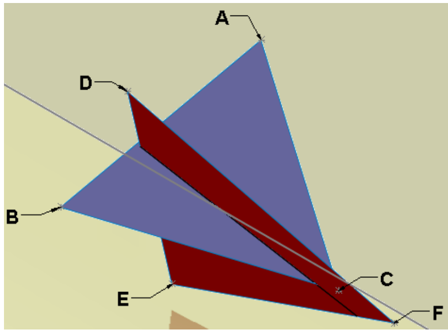
Splitline.



Under **Type of Split** choose **Intersection.**



Choose **Lamina ABC** as the cutting or ***Splitting plane,*** and **Lamina ADE** as the ***Plane to be split.***



In the **Surface Split Options,** choose ***Natural.***

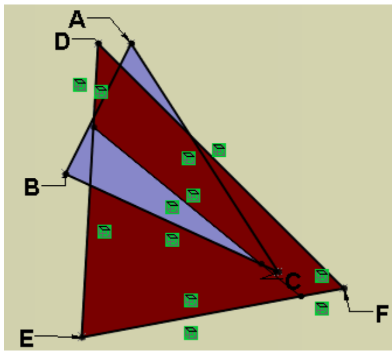
Choose **OK**

Rename Feature

Rename the feature as **Line of Intersection.**

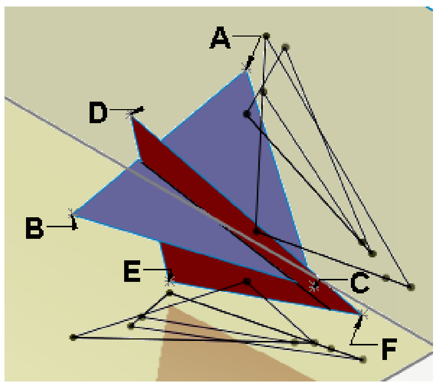
Creating the Orthographic Views

The orthographic projection of the intersecting lamina can be created using the **Convert Entities** command. 



We must first of all create a new sketch on the **Vertical plane**. Select a **front** view.

Hold down the **Ctrl** key and select all of the lines on the sketch, including the line of intersection. Now select **Convert Entities.** The elevation is created on the vertical plane.



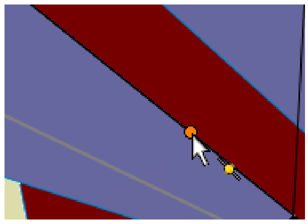
Confirm the sketch.

Rename Sketch

Rename the sketch as **Elevation.**

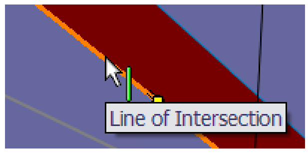
Repeat the process for the **plan** view, creating the sketch on the **horizontal plane** this time.

Finding the Dihedral Angle

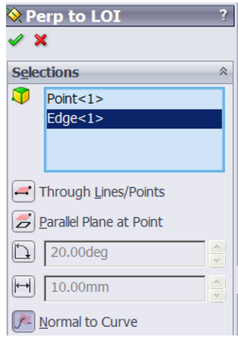


In order to find the dihedral angle, i.e. the angle between the two lamina we must take a point view of the **Line of Intersection**. In order to capture this view we must set up a plane perpendicular to the **Line of Intersection**.

Choose **reference geometry, plane**



Select the **midpoint** of the **Line of intersection** as one of the selections. We do this so that if we enter different values for the coordinates, the plane will always set up on the midpoint of the **line of intersection.**
N.B. this plane is for construction only. We will set up the plane required for the auxiliary view parallel to this.



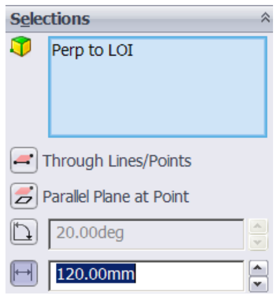
Select the **Line of Intersection** as the other selection.

Choose **normal to curve** so that the plane will be perpendicular to the **Line of Intersection.**

Rename Feature

Rename the plane as **Perp. to LOI**.

We now have the construction plane set up. The next step is to set up the plane onto which the auxiliary view will be projected.

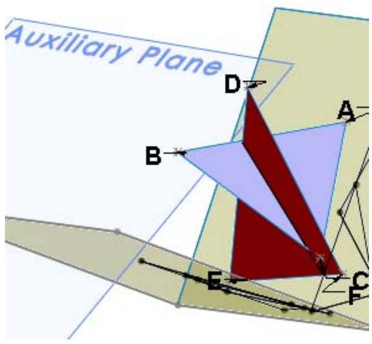


Once again select **reference geometry, plane**

Select **Perp to LOI** as the reference plane, set the distance to **120mm.**

Choose **OK.**

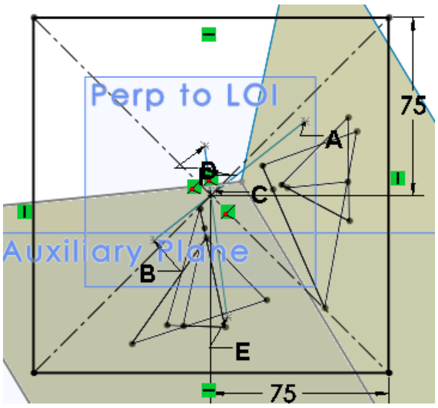
We now have the auxiliary plane that will Contain the required view. You will notice that the plane produced can be to the right or left of the initial plane, but it remains parallel to the plane. In order that the projected view will appear directly on the auxiliary plane we will create a portion of the auxiliary plane which has its centrepoint **coincident** with the **Line of Intersection**.



Therefore if we want to change the coordinates to suit another similar problem, the auxiliary plane will always have its centrepoint coincident with the point view of the line of intersection.

Sketch

Create a sketch on the **Auxiliary Plane**. Select **Normal to.** We are now looking at a point view of the **line of Intersection**.



Select **Centre Rectangle** and make its centre Coincident with the point view of the **line of Intersection.** Note the relations.

Smart Dimension as indicated.

If the coordinates of the lamina are changed to suit another problem, this portion of the auxiliary plane will always have the point view of the line intersection coincident with its centrepoint.

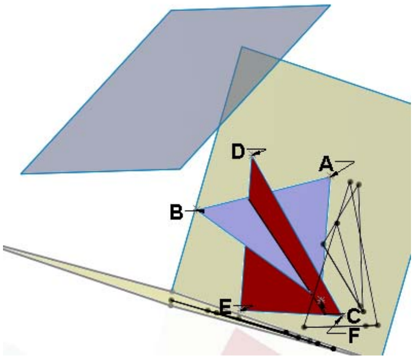
Confirm the sketch.

Rename Sketch

Rename the sketch as **sketch of portion of AP**

Planar Surface

Select **Planar Surface** from the **surfaces** toolbar.



Choose **sketch of portion of AP** as the **Bounding entities**.

Hide **Auxiliary Plane** and **Perp to LOI**

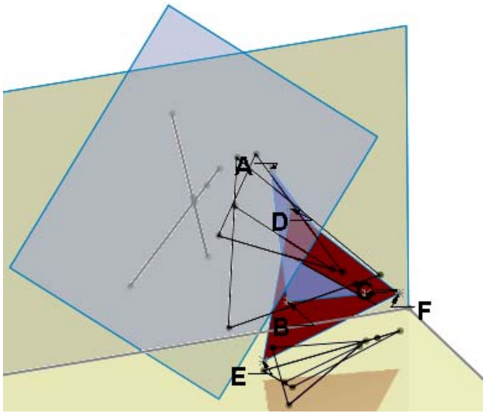
Choose **OK.** ✔

Rename Feature

Rename the feature as **Portion of AP.**

Auxiliary View

Create the auxiliary view on the **Portion of AP** using **Convert Entities** as in the **Orthographic Views**.



We can now see the angle between the **Lamina ABC** and **DEF**.

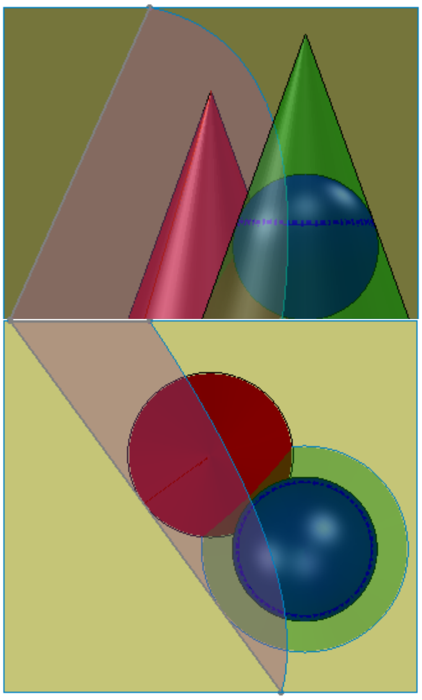
Rename Sketch

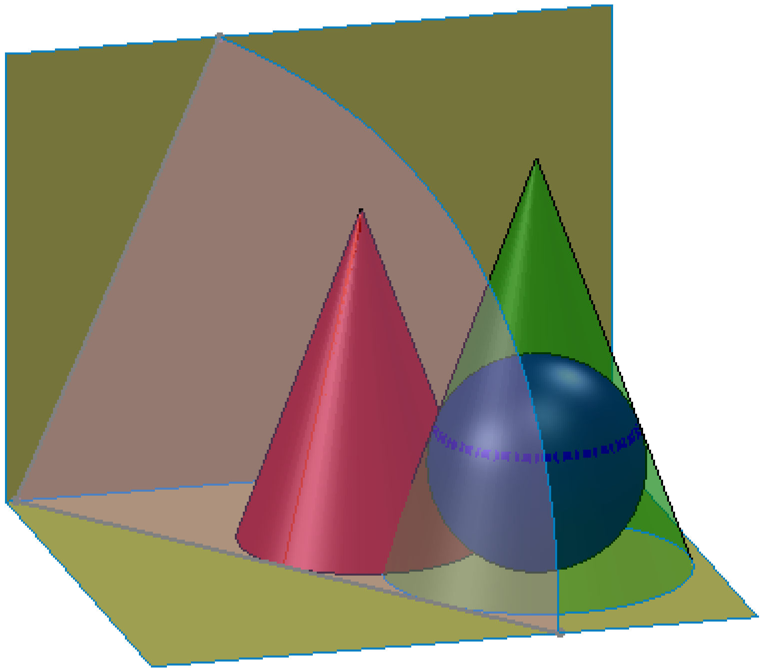
Rename the sketch as **Dihedral Angle.**

This model can now be used to solve similar problems involving intersecting lamina.

Exercise complete!

Tangent Planes





Prerequisite knowledge To complete this model you should have a working knowledge of Solidworks 2006/2009.

Focus of lesson This lesson focuses on using SolidWorks to solve a geometrical problem. The following **Surfaces** tools are used: **Planar Surface, Ruled Surface, Surface Revolve and Filled Surface.**

Getting started.

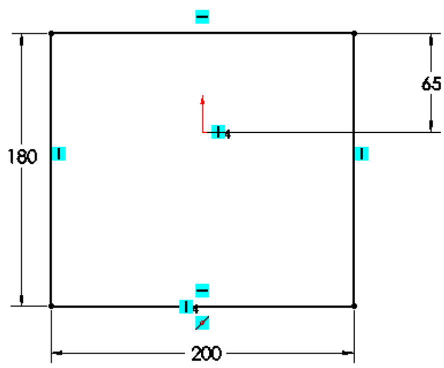
New File Create a new part file and save it as **Tangent Planes** in the desired location.

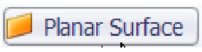
New Sketch We are going to begin by creating a sketch to represent the outline of portion of the **Horizontal Plane**.

Create the sketch shown on the Top plane.

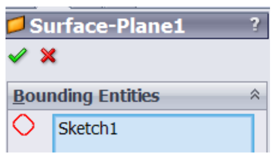
Smart Dimension and apply the relations as shown.

We want to transform this rectangle into a **Planar Surface.**



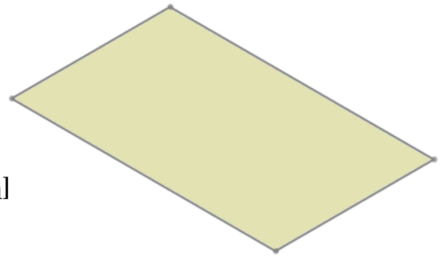
Planar Surface Select **Planar Surface** 

from the **Surfaces** toolbar.



Select **Sketch 1** as the **Bounding Entities.**

Select **OK**



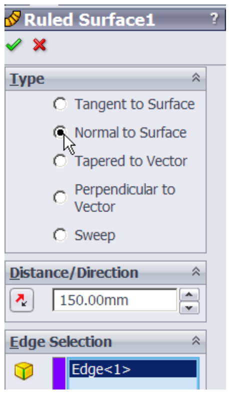
Rename Feature

Rename the feature as **Horizontal Plane.**

We have now created a portion of the horizontal plane. This planar surface has no thickness but can be used as a datum for measurements, a surface to project views onto, or a surface to sketch on.

The Vertical Plane

To create the vertical plane we use **Ruled Surface.** **Ruled Surface** command Ruled Surface creates surfaces that extend out in a specified direction and distance from selected edges.

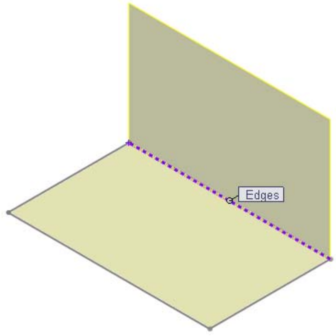


Ruled Surface

Select **Ruled Surface** from the **Surfaces** toolbar.

Select **Normal to Surface** as the **Type**. This will create a ruled surface at 90 degrees to another surface at a specified edge.

Set the **distance** to 150mm. This will extend the surface out 150mm from the selected edge. The width of the surface will be determined by the length of the edge selected.



Select the edge of the **Horizontal Plane** shown as the edge to set up the ruled surface from.

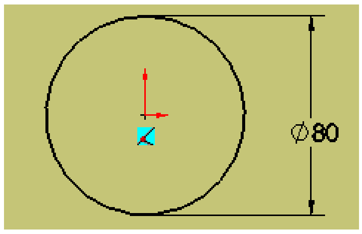
Select **OK.**

Rename Feature

Rename the feature as **Vertical Plane**.

The Cone

We will use ***Lofted surface*** to create the cone. We also require a plane to contain the apex of the cone. (Alternatively we could use ***3D Sketch***)



Sketch

Create the sketch shown on the ***Horizontal Plane.***

Smart Dimension as indicated.

This sketch forms the base of the cone.

Rename Sketch

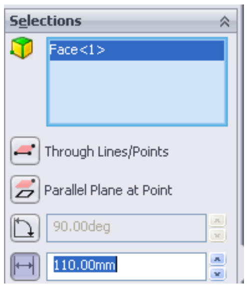
Rename the sketch as ***Trace of cone.***

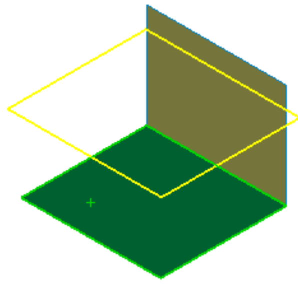
Insert Plane

We must now insert a horizontal plane equal in altitude to the height of the cone.

From the ***Surfaces*** menu select ***Reference Geometry,Plane***.

Make the following selections:



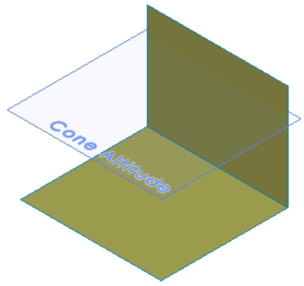


Choose the ***horizontal plane*** as the ***reference entity.***

Set the distance to 110 which is the altitude of the cone.

Select **OK**.

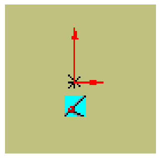
Rename Feature Rename the feature as ***Cone Altitude.***



This plane will contain the apex of the cone.

Sketch Create a new sketch on ***Cone Altitude.***

From the ***Sketch*** menu select ***Point.***
Make the point ***coincident*** with the ***origin.***



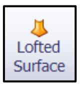
Confirm the Sketch.

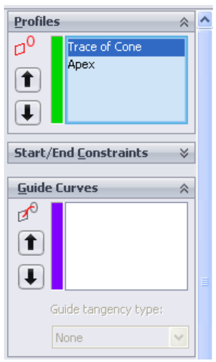
Rename Sketch Rename the sketch as ***Apex.***

Lofted Surface We will now create a ***Lofted Surface*** between the ***Trace of the cone*** and the ***Apex.***

About Lofted Surface The difference between lofted surfaces and lofted solids is that surfaces can use edges and curved features between which to loft rather than simply sketches and faces as is the case with solids. Guide Curves may be added, if necessary, to influence the resultant surface.

Creating the loft Choose **Lofted Surface** from the Surfaces toolbar

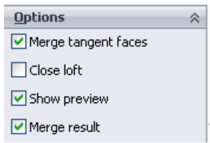




Select the two sketches as **Profiles**

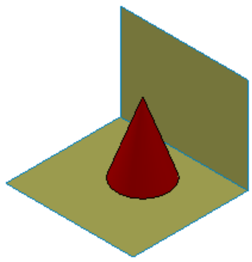
We will not be using any guide curves.

Check - **Merge tangent faces.**
This will create one surface between the two profiles as opposed to a series of individual surfaces.



Show preview – will provide a preview of the loft.

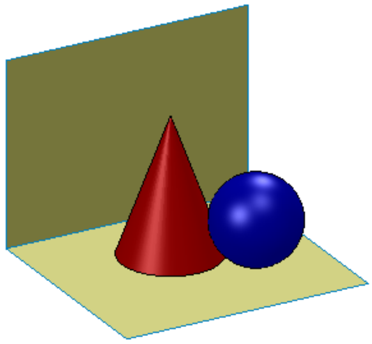
Choose **OK**



Hide Hide ***Cone Altitude.***

Appearance Apply a red colour as shown to the cone.

Rename Feature Rename the feature as ***Cone.***

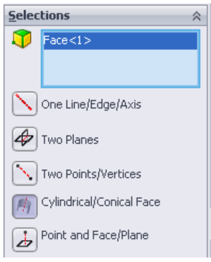


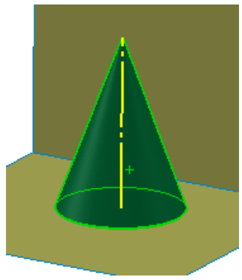
Setting Up The Sphere

The ***sphere*** is in contact with the ***cone***. The point of contact between the ***sphere*** and the ***cone*** is contained on a vertical plane which also contains the vertical axes of both the ***sphere*** and the ***cone***. The plane is inclined at ***45 degrees*** to the ***vertical plane***.

Axis of Cone

To define the plane we will use the axis of the ***cone*** To view the axis of the ***cone*** select ***Reference Geometry, Axis.*** Axis





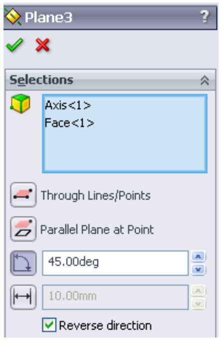
Make the following selections:

Choose ***Cylindrical/Conical Face***

Then select the conical face of the ***Cone*** for the ***Reference Entities.***

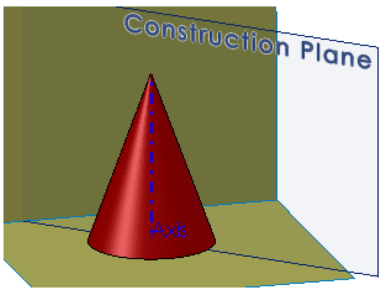
Insert Plane

We must now insert a vertical plane which contains the axis of the ***cone*** and is inclined at 45 degrees to the ***vertical plane***.



From the ***Surfaces*** menu select ***Reference Geometry,Plane***. Make the following selections:

Choose the ***vertical plane*** and the ***Axis*** of the ***Cone*** as the ***reference entities.***



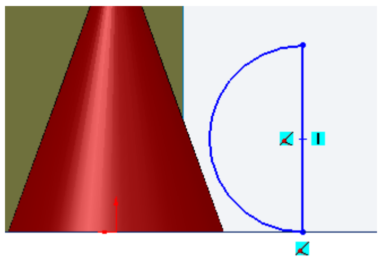
Set the angle to 45 degrees.

Reverse direction if required.

Select **OK**.

Hide

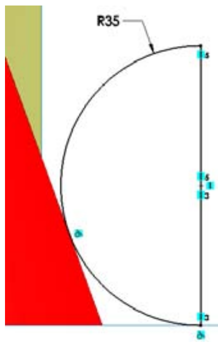
Hide the axis of the cone.



Sketch

Create a sketch on the ***Construction plane.***

Using the ***Line*** command set up the sketch opposite. Note the automatic relations.



Smart Dimension (radius = 35mm) and apply the relations as indicated. These relations ensure that the sphere and cone are tangent to each other.

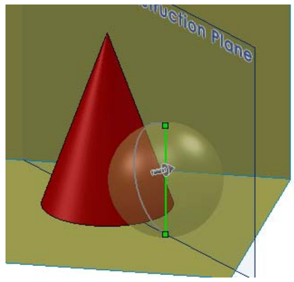
Confirm the Sketch.

Rename Sketch

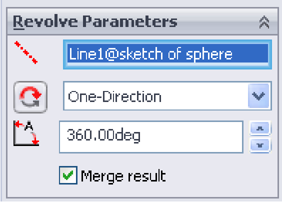
Rename the sketch as ***Sketch of Sphere.***

Revolve

From the ***Features*** menu select ***Revolve.***



Select the diameter line of the Semi-circle as the ***axis of revolution.***



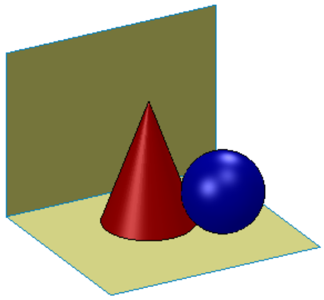
Select **OK.**

Appearance

Apply a blue colour to the ***Sphere***.

Rename Feature

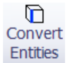
Rename the feature as ***Sphere.***



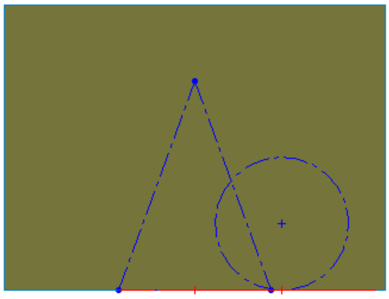
We now have the ***"Solids in Contact"*** portion set up.

Creating the Orthographic Views

The orthographic projection of the Solids in Contact can be created using the ***Convert Entities*** command.



We must first of all create a new sketch on the ***Vertical plane***. Select a ***front*** view.

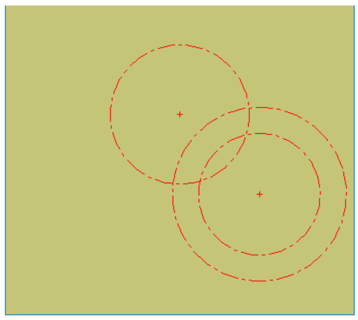


Hold down the ***Ctrl*** key and select all of the lines on the sketch. Now select ***Convert Entities.*** The elevation is created on the vertical plane.(Solids hidden for illustration)

Confirm the sketch.

Rename Sketch

Rename the sketch as ***Elevation.***



Repeat the process for the ***plan*** view, creating the sketch on the ***horizontal plane*** this time.

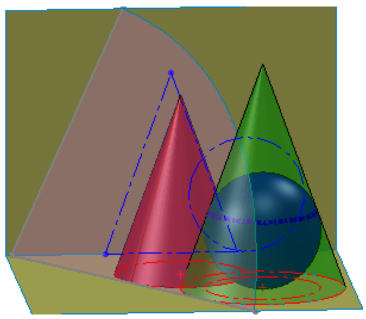
Confirm the sketch.

Rename Sketch

Rename the sketch as ***Plan.***

Setting up the Tangent Plane

In order to set up the tangent plane, we will have to create a construction cone which fits down over the ***Sphere***; is tangent to the ***Sphere*** and has the same base angle as the ***Cone***. The tangent plane will rest against both of these cones.

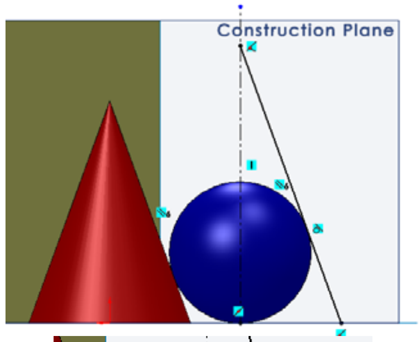


Sketch

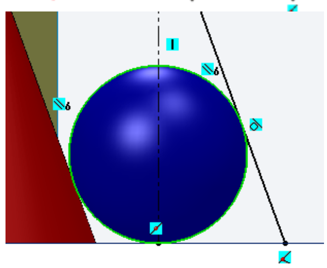
Create a sketch on the ***Construction Plane.***

Set up the centreline first. This will be the axis of the ***construction cone***.

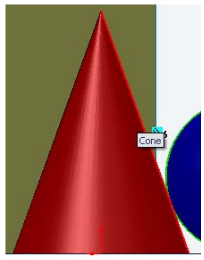
Sketch the line then which will be the ***generator*** of the ***construction cone*** and will be the line of intersection between the ***construction plane*** and the ***construction cone.*** (True Length)
Make one of the endpoints of the ***generator line coincident*** with the ***centreline*** as shown, and the other ***coincident*** with the groundline.



A ***tangent relation*** is applied between the ***sphere*** and the ***generator line***. The intersection of the ***construction plane*** and the ***sphere*** produces a circle so it is this circle that is selected for the relation. (True Length)



A ***parallel relation*** is applied between the ***generator line*** and the generator of the ***cone*** which contains the point of contact, which is also the line of intersection between the ***construction plane*** and the ***cone***.



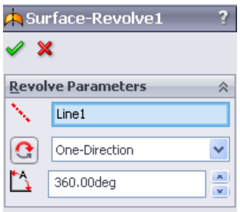
Confirm the sketch.

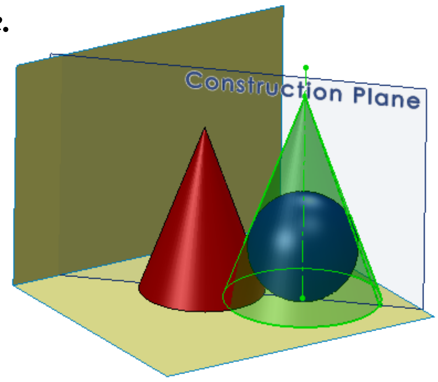
Rename Sketch

Rename the sketch as ***Generator Line.***

Surface Revolve

From the ***Surfaces*** menu select ***surface revolve. Surface revolve*** acts in the same manner as the ***revolve*** command in the ***features*** menu, but instead of producing a solid body it produces a hollow body.





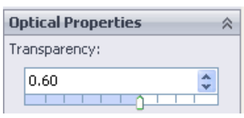
Select the centreline as the ***axis of revolution.***

Select OK.

Appearance

Apply a green colour to the ***Construction cone.***

Change the ***Transparency setting*** to ***0.6.*** This enables us to view the ***sphere*** "inside" the ***Construction cone***.

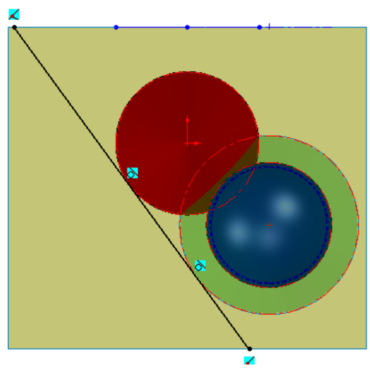


Hide

Hide the ***Construction Plane.***

Horizontal Trace

Create a sketch on the horizontal plane and using the line command draw the line shown Tangential to the bases of both cones i.e. ***HT.***

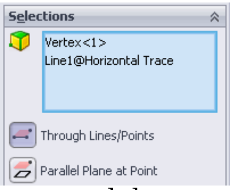


Rename Sketch

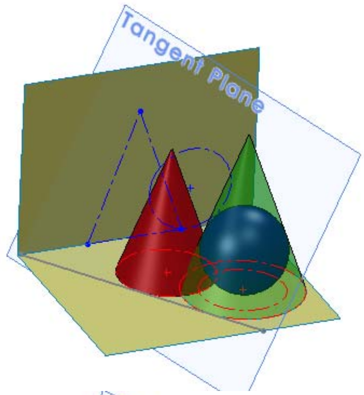
Rename the sketch as ***Horizontal Trace.***

Insert Plane

From the ***reference geometry*** tool select***Plane.***



Select ***Through Lines/Points*** as the end condition.



Choose the apex of either cone and the ***Horizontal trace*** as the ***reference entities***.

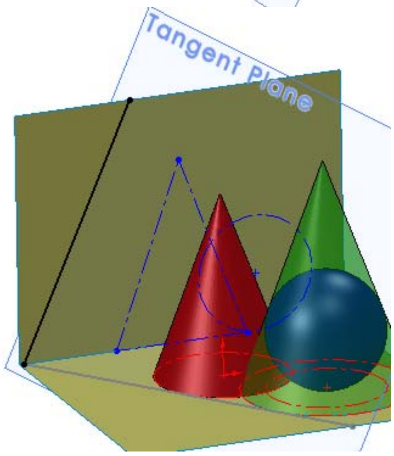
Select OK.

Rename Feature

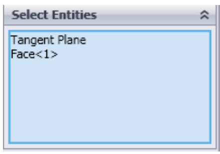
Rename the feature as ***Tangent Plane.***

Vertical Trace

From the ***tools*** menu select ***sketch tools***, ***intersection curve***.



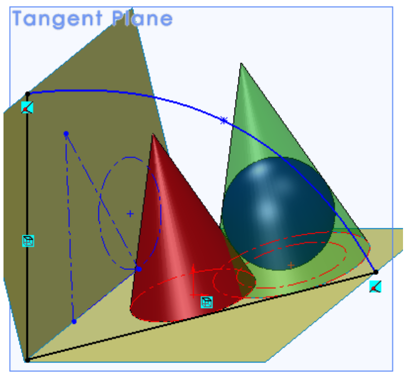
Select the ***tangent plane*** and the ***vertical plane*** as the ***entities to generate the intersection curve***



Select OK.

Rename Sketch

Rename the sketch as ***Vertical Trace.***



Portion Of Tangent Plane

Using the ***spline*** tool and ***convert entities*** create the sketch shown on the ***tangent plane***.

Confirm the sketch.

Rename Sketch

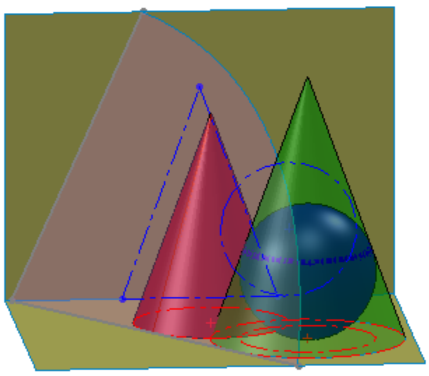
Rename the sketch as ***sketch of portion of tangent plane***

Hide

Hide the ***tangent plane***.

Planar Surface

From the ***surfaces*** menu select ***planar surface***



Select the ***sketch of portion of the tangent plane*** as the ***bounding entities.***

Select OK.

Rename Feature

Rename the feature as ***portion of Tangent Plane.***

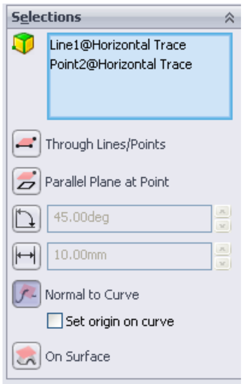
Plane Perpendicular To Horizontal Trace

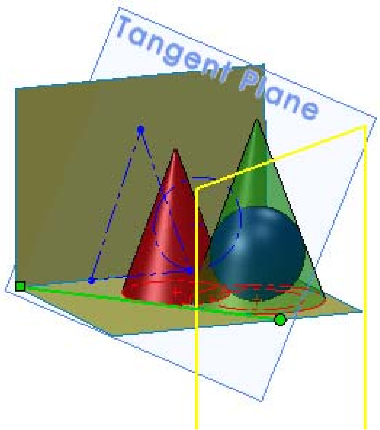
We now want to set up a plane onto which an auxiliary view could be projected showing a line view of the ***tangent plane,*** and the true inclination of the ***tangent plane*** to the ***horizontal plane***. Therefore we need to set up a plane perpendicular to the ***horizontal trace.***

Once again select ***reference geometry, plane.***

Choose ***normal to curve*** as the end condition.

Select the ***horizontal trace*** and the end point of the ***horizontal trace*** as the ***reference entities.***



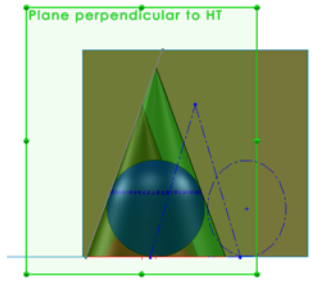


Select **OK.**

Rename Feature

Rename the feature as ***Plane Perpendicular To HT.***

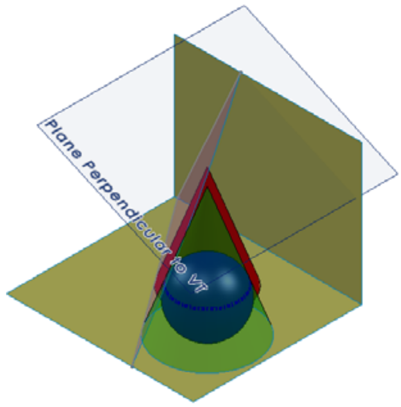
If we want to view the true inclination of the Tangent Plane to the horizontal plane, simply Select the ***Plane Perpendicular To HT*** and choose a ***normal to*** view.



Plane Perpendicular To Vertical Trace

Repeat the steps above to create a plane perpendicular to the ***Vertical Trace.***

This time select the ***vertical trace*** and its endpoint as ***the reference entities.***



Exercise complete!

Keyboard Button.

Surface Modelling

In surface modelling a model is built face by face. Faces created by surface features may knit together to enclose a volume, which may be turned into a solid model.

Surface modelling is used to create faces and features which may not be conveniently produced using solid modelling techniques. Surface tools are employed in situations where they make it easier, more efficient or possible to complete the task at hand.

The focus of this exercise is to give a basic introduction to surfaces and explore the functionality of some surfacing tools.

As we work through the exercise we will explain the terminology associated with surfaces.

Prerequisite knowledge

To complete this model you should have a working knowledge of Solidworks 2006/2009.

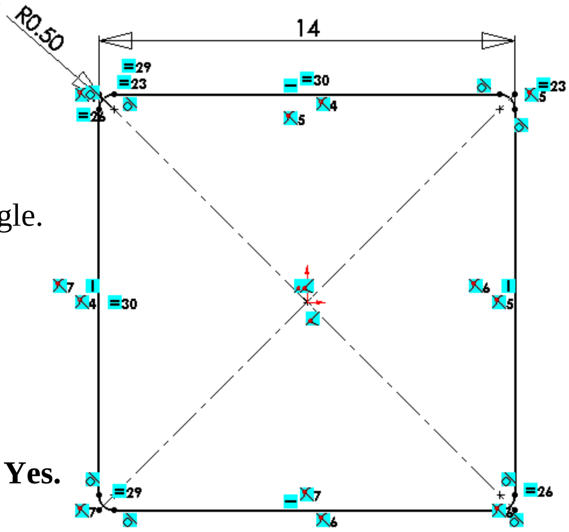
Focus of lesson

This lesson focuses on using the following surface tools;
Surface Loft, Planar Surface, Filled Surface, Surface Knit, Surface Offset, Replace Face, as well as ***Shell*** *and* ***Extrude*** feature tools.

Getting started.

New File Create a new part file and save it as **Keyboard Button** in the desired location.

New Sketch Using **Centre Rectangle** , create the sketch shown on the **Top plane**.
This sketch will represent the profile of the top of the button



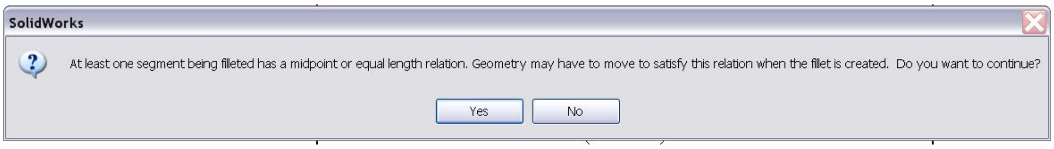
Steps required Apply an **Equals Relation** between the horizontal and vertical sides of the rectangle.

Smart dimension one of the sides **14mm**

Add **sketch fillets** of **0.5mm** to the four corners.

Should the warning below appear choose **Yes.**

Exit the sketch.



Inserting a plane We want to insert a plane parallel to the top plane at a distance of 3mm below it. We will sketch the profile of the base of the button on this plane.

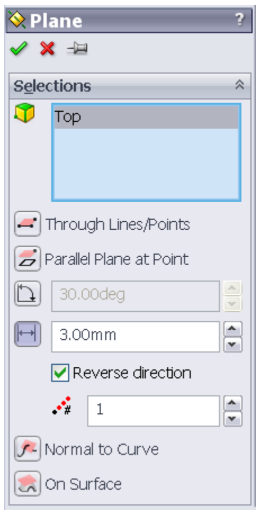
Choose **Insert, Reference Geometry, Plane…** from the drop down menu or choose **Plane** from the pop-up menu.
Note: The pop-up menu is accessed by pressing 's' on the keyboard.

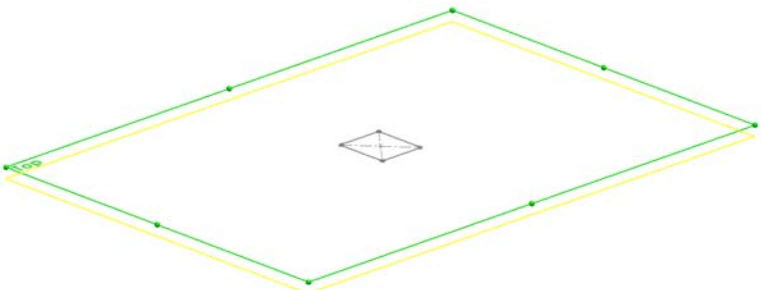
Choose the **Top plane** as the **Reference Entity**

Insert a **distance** of **3mm**

Toggle **Reverse direction** to create the new plane below the Top plane if required. Choose **OK**

Note – You may need to zoom out to see the preview of the parallel plane, as shown below.



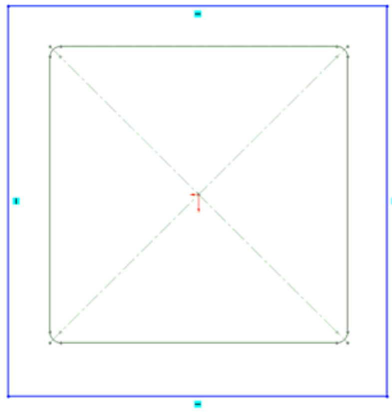


New Sketch

Create a sketch on the offset plane.

Choose **Top view**.

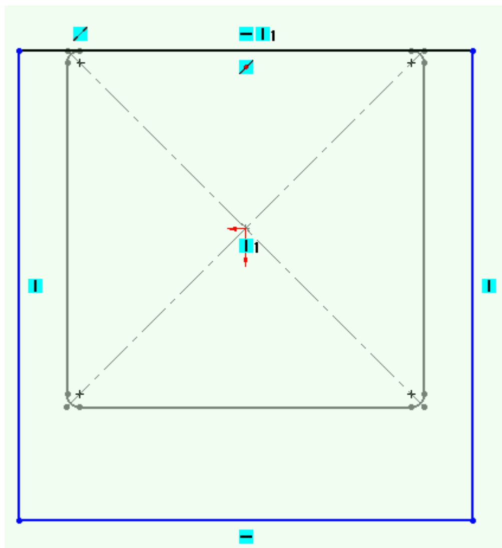
Using **Corner Rectangle,** create the rectangle shown.



Add Relations

Add a **Vertical relation** between the midpoint of the top horizontal line of the rectangle and the origin.

Apply a **Collinear relation** between the top horizontal lines of both sketches.

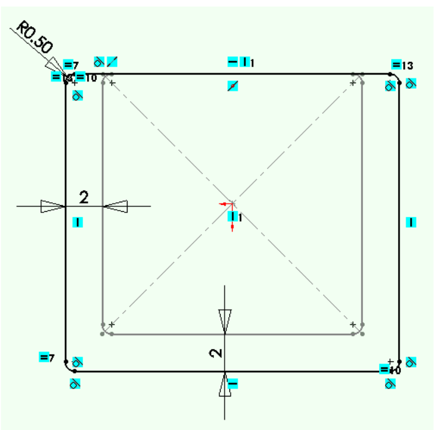


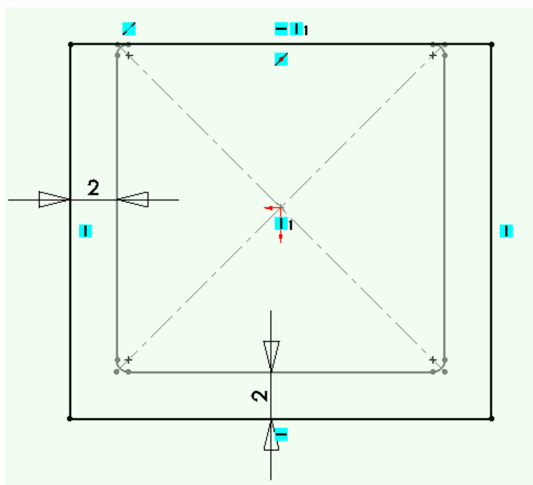
Smart Dimension

Smart Dimension the sketch as shown below right.

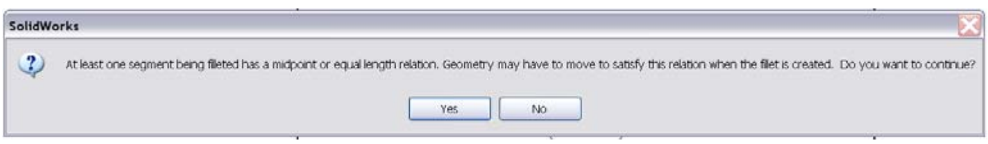
Sketch Fillet

Create a **0.5mm** sketch fillet on the four corners of the rectangle.





Choose **Yes** for the warning message as before. Exit the sketch.



Hide Plane1

To hide plane 1; left click on **plane1** on the featuremanager design tree and choose **Hide**

Orientation

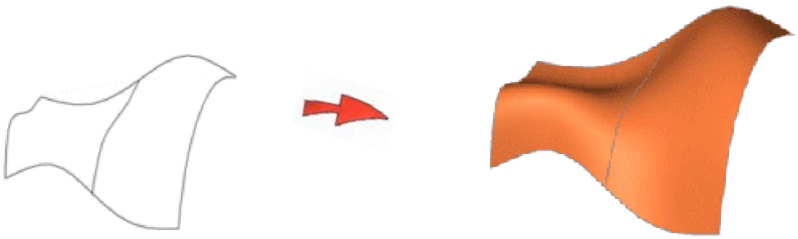
Choose **Isometric view** 

About Lofted Surface

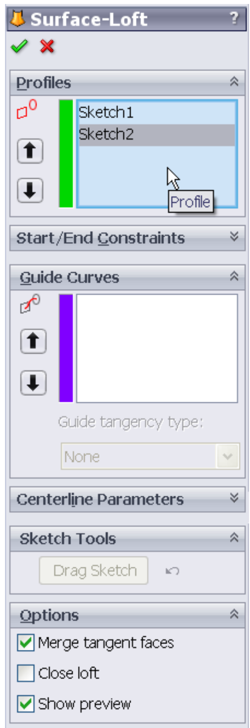
The difference between lofted surfaces and lofted solids is that surfaces can use edges and curved features between which to loft rather than simply sketches and faces as is the case with solids.

Profiles Simple Loft

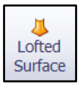
Guide Curves may be added, if necessary, to influence the resultant surface.



Profiles with guide curves Loft using guide curves



Creating the loft

Choose **Lofted Surface** 

from the Surfaces toolbar

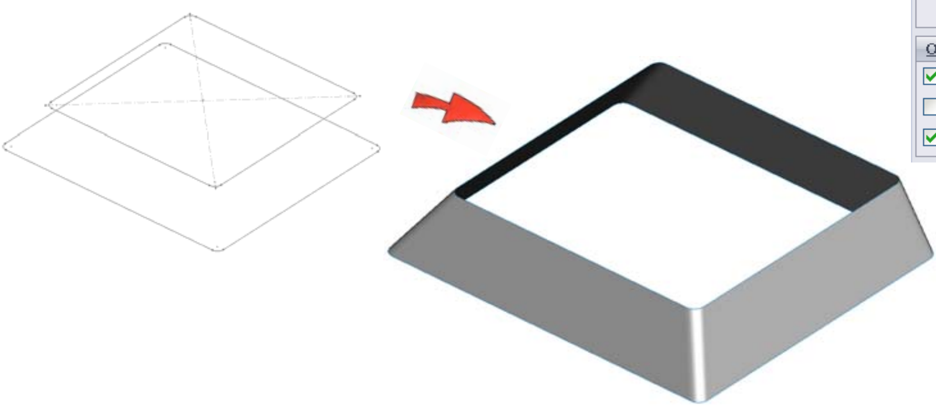
Select the two sketches as **Profiles**

We will not be using any guide curves in this example.

Check - **Merge tangent faces** – This will create one surface between the two profiles as opposed to a series of individual surfaces.

Show preview – will provide a preview of the loft.

Choose **OK** 



Lofted Surface

Examine the model. You will notice that the model is made completely of surfaces, which have zero thickness.

Image Quality

Zoom into the filleted corners of the model. Circular edges may be appearing as straight line segments. This may be corrected by increasing the image quality.

Choose **Tools, Options, Document properties, Image Quality**

Drag the **Shaded and Draft Quality** slider to a higher setting. Choose **OK**

Next Step

We now want to close off the base of the button by adding another surface. This surface is going to be defined by the edge of the base of the button.
To complete this task we are going to use a **Planar Surface.**

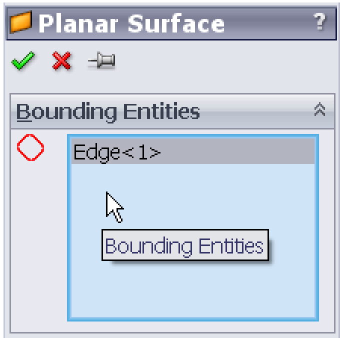
About Planar Surface

Planar surfaces are by definition planar. They may be sketched upon or used as a plane for mirroring. Planar surfaces may be defined in SolidWorks by the same means as they are defined in plane geometry, including two parallel lines.

More commonly planar surfaces are defined using a closed sketch e.g. rectangle.

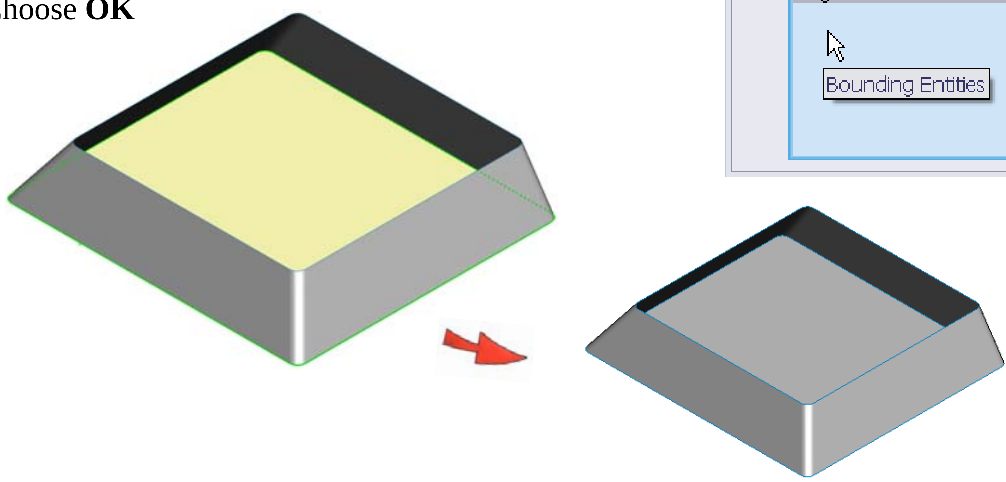
Planar surface

Choose **Planar Surface**  from the Surfaces toolbar.



Select the base edge as **Bounding Entities**

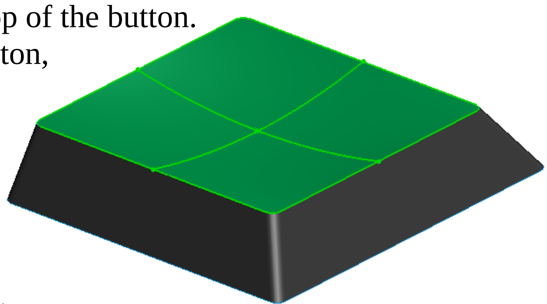
Choose **OK**



A planar surface has now been added to the base of the model.

Top Surface

The next step is to create a surface at the top of the button.
You will notice from the graphic of the button,
that the top surface of the button is not flat,
but concave.



For this reason planar surface will not
create this surface for us.

The surface is defined by the top edge and two
constraint curves, which direct the curvature of the surface in both directions.
This surface is created using **Filled Surface.**

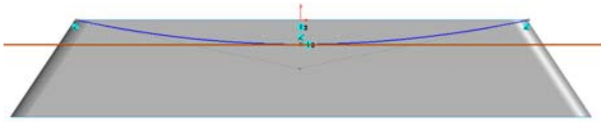
About Filled Surface

The **Filled Surface** is intended to be used to fill gaps in surface bodies. Constraint curves may be used to drive the shape of the fill between existing boundaries.

Constraint Curves

We must first create the two constraint curves which will be used to define the filled surface.

The constraint curve is defined by the existing surfaces along with a plane which is placed at the lowest point of its curvature.



We will begin by creating this plane.

Insert plane

Insert a parallel plane at a distance of **0.75mm** below the **Top plane**, as described in page 2 of this document.

Create sketch

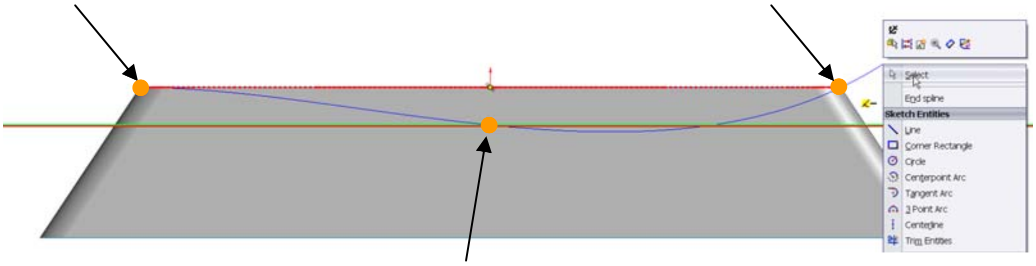
Create a sketch on the **Front plane**. Choose **Front view**.

Spline

Choose **Spline** from the sketch toolbar. Three points will be used to define the spline. Ensure to capture the co-incident relations.

Point 1

Point 3



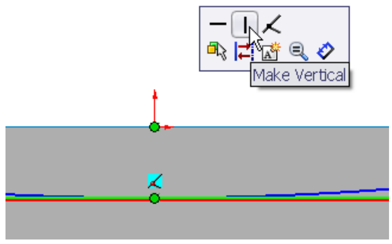
Point 2

End Spline

To end the spline; **Right Click** and choose **Select**.

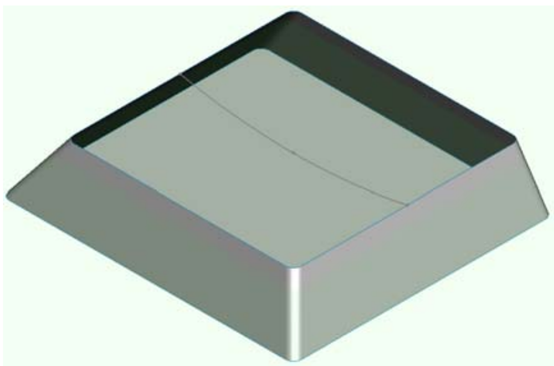
Add Relation

Add a **Vertical relation** between the lowest point of curvature of the spline and the origin.



Exit the sketch and choose an Isometric View.

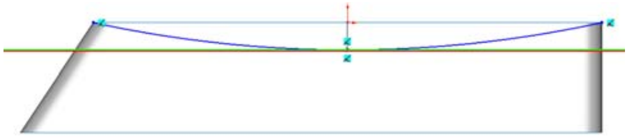
The sketch will appear as shown.



Creating the other Constraint Curve

Create a sketch on the **Right plane**
Choose **Right View**

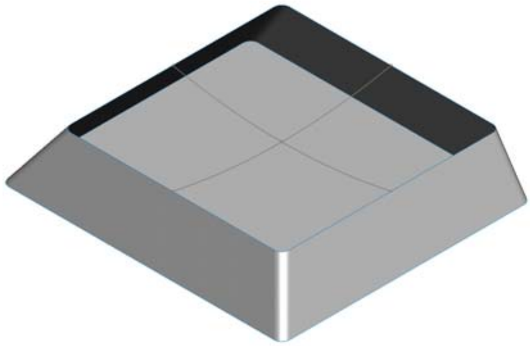
Using a similar procedure as before, sketch the spline shown opposite.



Exit the sketch and hide the plane shown.

Choose Isometric View.

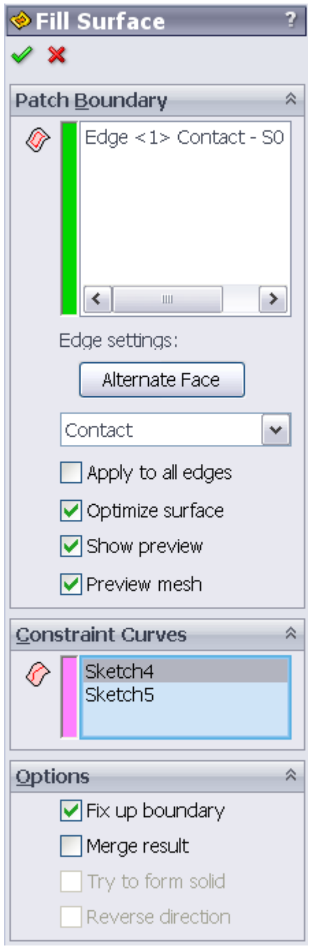
We now have the necessary geometry to create the filled surface.



Filled Surface

Choose **Filled Surface** 

from the surfaces toolbar.



Select the top edge of the model as the **Patch Boundaries**

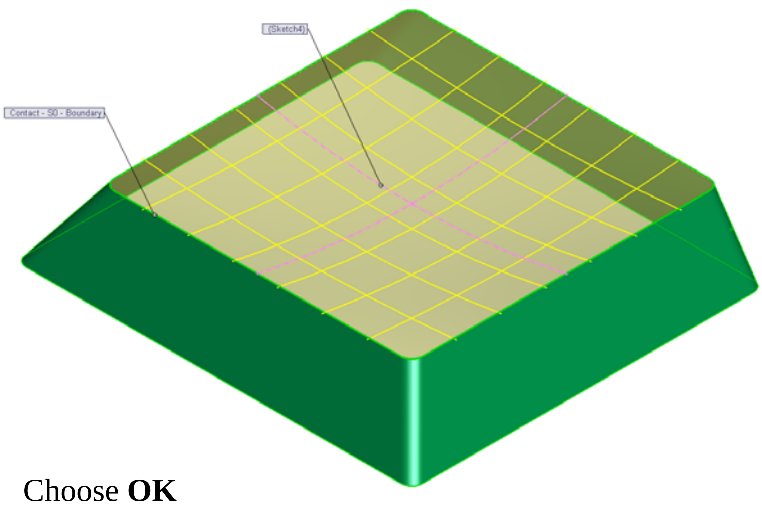
Choosing **Optimize surface** will ensure that SolidWorks tries to fit the patch surface within the limits of the boundary.

Select **Show Preview** and **Preview mesh**

Select the two sketches as **Constraint Curves**

We will choose not to **merge result** in this case. Therefore the surface produced may be chosen independently of the others.

A preview of the surface is displayed in the graphics area.



Choose **OK**

Fillet Surface

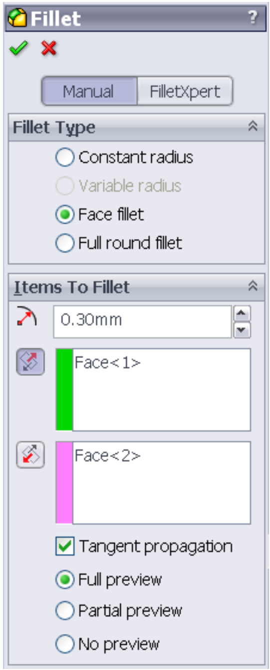
Surface fillet works in a very similar manner to that of Solid Fillet.

To apply a fillet to the top edge of the button;

Choose **Fillet** from the Surface toolbar.

Select the **Manual** tab. Choose **face fillet**

Insert a **Fillet radius** of **0.3mm**

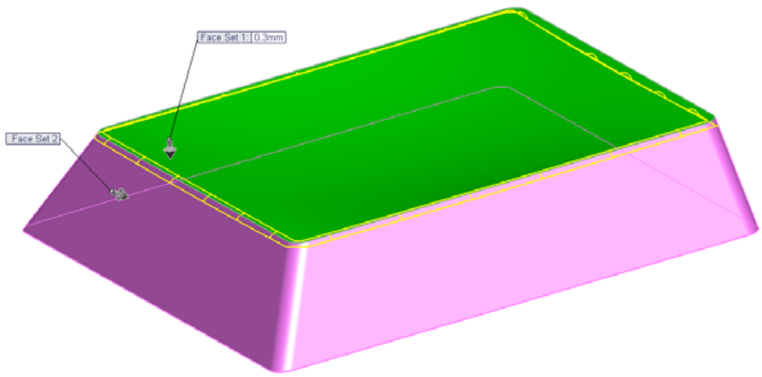


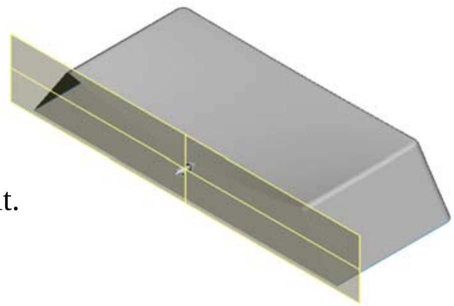
Select surfaces

Select the top face as **face set 1** and the side as **face set 2**

Ensure that the direction arrows are pointing inward.
These may be toggled by choosing **reverse face normal**

Check **Full preview.** Choose **OK**

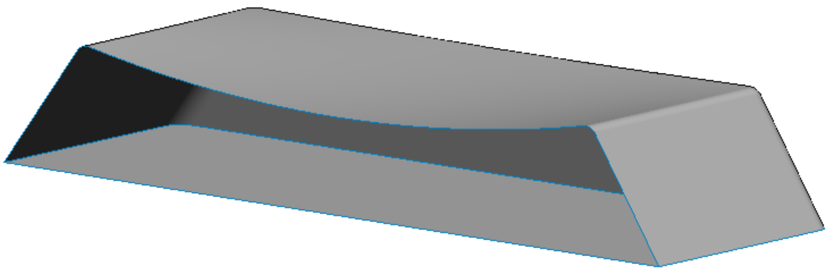




Section

Choose **Section** from the heads-up toolbar.

A section is created using the front plane by default.
Choose **OK**.



Examine the model

You will notice that the model is completely hollow inside and is enclosed by the surfaces created.

Deselect Section View to return to a view of the complete model.

Creating a Solid

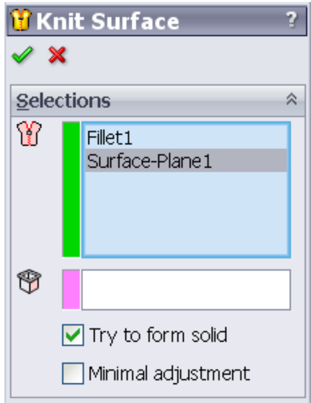
As discussed previously, the aim of surface modelling is commonly to create a solid. A solid model may be generated from a surface model using **Knit Surface.**

About Knit Surface

Knit joins multiple surface bodies into a single surface body. It also has the option to create a solid if the resultant surface body satisfies the requirements; *a fully enclosed volume without gaps or overlaps.* Surface bodies must intersect edge to edge.

Knit Surface

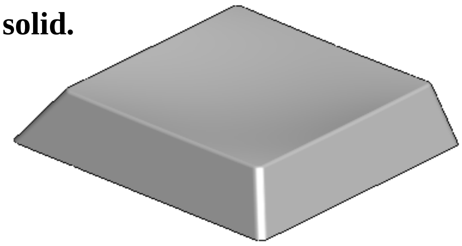
Choose **Knit Surface** from the surfaces toolbar.



Choose the fillet and base surface as **Surfaces and Faces to Knit.**

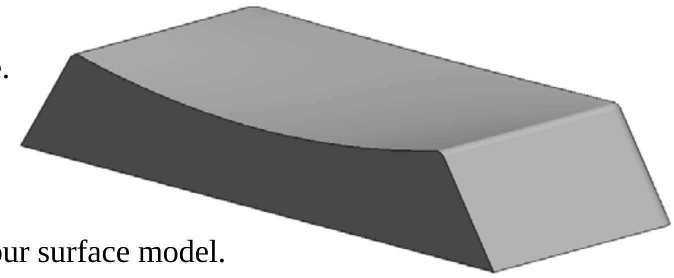
Check **Try to form solid.**

Choose **OK.**



Section

Section the model as before.



You will notice that the model is now completely solid.

Knit has created a solid from our surface model.

Deselect Section View to return to a view of the complete model.

New surfaces

We are now going to create two new surfaces. These surfaces will be created as copies of existing surfaces. We are going to use **Offset Surface** to create these new surfaces.

Offset Surface

Offset Surface does in 3D what Offset Sketch does in 2D. When a surface is offset a new surface is generated a distance of the offset away from the original surface.

Similar to Sketch Offset, an offset surface will fail if it is offset in the direction of decreasing radius and the offset is greater than the smallest radius of curvature of the surface.

One way of troubleshooting a failing offset surface is to use **Tools, Check…** to find the minimum radius.

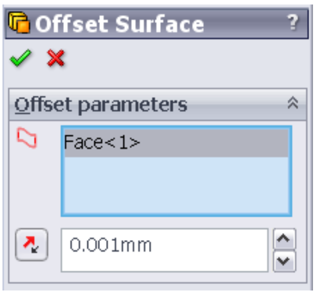
A copy of a surface will be generated if it is offset by a distance of zero.

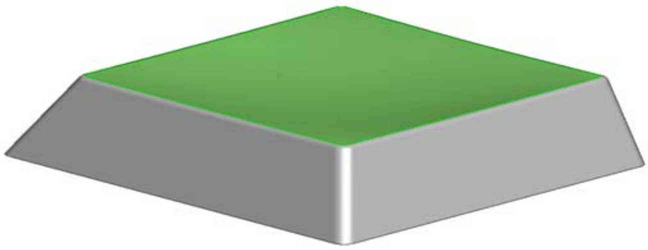
To continue, we will now offset both the top and bottom surface of the model.

Offset Surface

Choose **Offset Surface** from the surfaces toolbar.

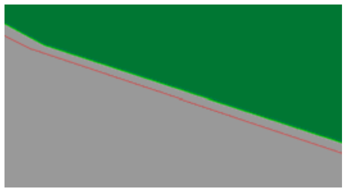
Select the top face as **Surfaces or Faces to Offset** with an offset distance of **0.001mm.**





Zoom

Zoom in to ensure that the surface is being offset **above** the existing surface.



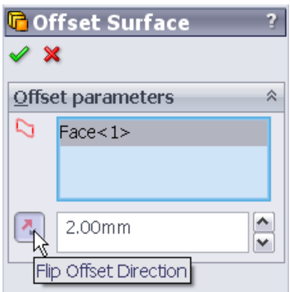
If below, choose **Flip Offset Direction** to reverse it.

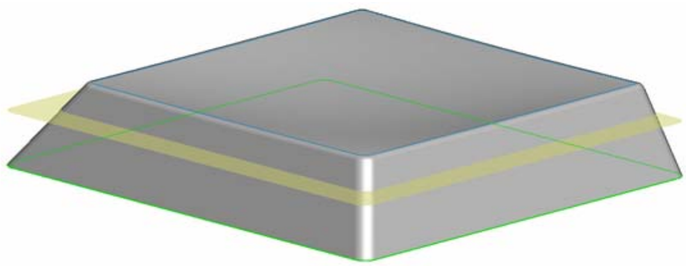
This will create a copy of the top surface, offset a distance of 0.001mm above it. This surface will be used when applying the letter ‘J’ to the button.

Offset Surface 2

We will now repeat this procedure to create an offset surface from the **base surface** at a distance of **2mm**. Choose **Flip Offset Direction** if required.

Choose **OK**

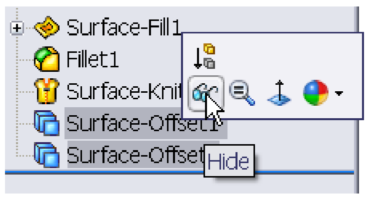




This surface will be used later to replace a surface created when we shell the button.

Hide Surfaces

Choose **Surface Offset1** from the feature manager design tree. Hold Ctrl and select **Surface Offset2.** Release **Ctrl** and choose **Hide**.



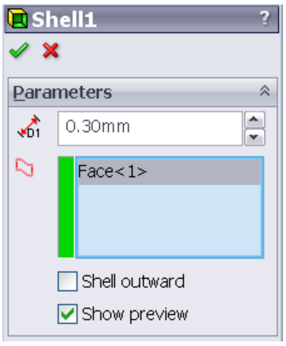
The two surfaces will be hidden.

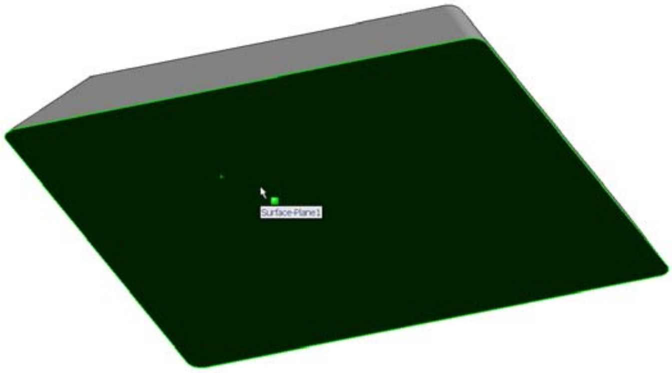
Shell

Choose **Shell** from the **feature** toolbar.

Insert a **distance** of **0.3mm.** Rotate the model and choose the base surface as **faces to remove.**

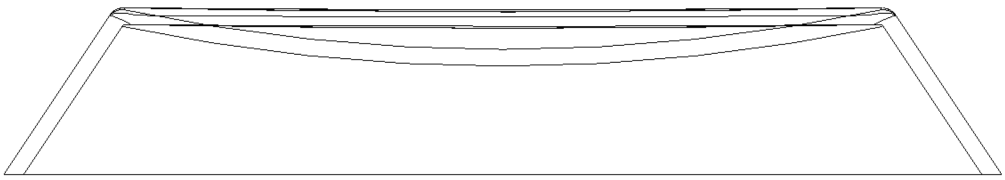
Choose **Show preview.** Select **OK.**





Orientation

Choose a **front view** orientation and select **wireframe display** from the heads-up toolbar**.**



The wireframe display shows the internal walls created by the shell feature.

Because the top surface is curved, the internal top surface is also curved. In reality this is not the case. The internal surface is planar.

We will now replace the internal curved surface with the offset surface created from the base. To do so we will use the **Replace Face** tool.

About Replace Face

Replace Face replaces selected faces of a solid or surface body with a selected surface.

If this was to be done manually it would involve **deleting** existing faces of the solid, **extend** and **trim** the new faces and then **knit** the new faces together.

Orientation and display

Choose **Shaded with edges** as display style.

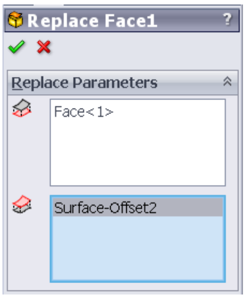
Rotate the model to an orientation which will allow you to choose the curved surface on the inside.

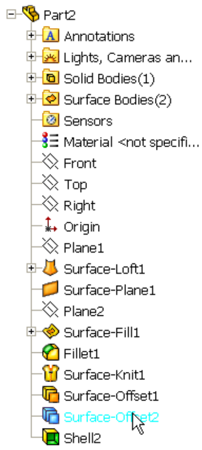
Replace Face

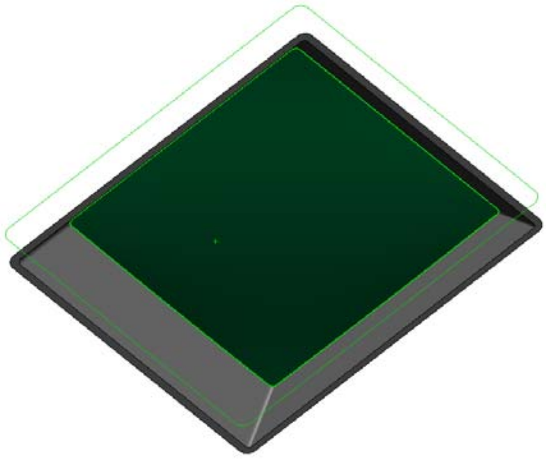
Choose **Replace Face** from the surfaces toolbar.

Select the curved surface of the model as **the target face for replacement**

Highlight the selection box for **Replacement Surfaces**. Expand the feature manager design tree in the graphics area and choose **Surface-Offset2.**





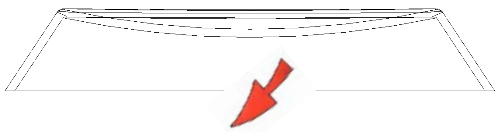


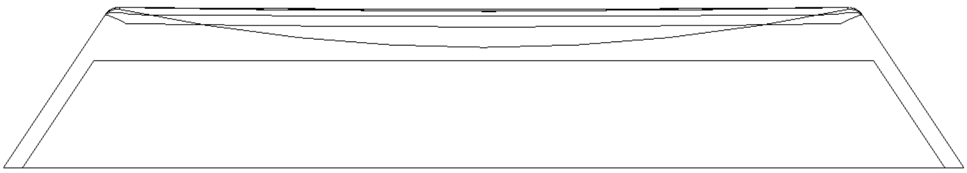
Choose **OK**

Orientation

Choose a **front view** orientation and select **wireframe display** from the heads-up toolbar**.**

The top surface inside is no longer curved.
It has been replaced by a flat planar surface.





Orientation and display

Choose **Shaded with edges** as display style. Select an **Isometric View.**

Adding text

The final stage in completing the exercise is to add the letter ‘J’ to the curved surface of the button.

In order to do so we must first create a sketch, of the letter, on the top plane.

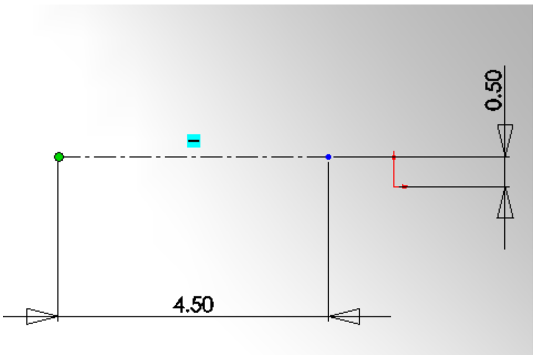
We will then extrude the letter between the offset face and the top face of the button.

Creating the sketch

Create a sketch on the **Top plane**. Choose **Top View.**

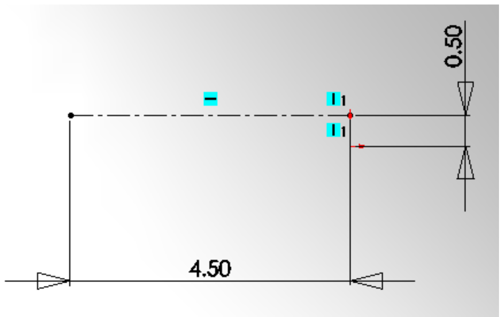
Centreline

Sketch a horizontal **Centreline** and Smart dimension as shown.



Add Relation

Add a **Vertical Relation** between the endpoint of the line and the origin.



Text

Choose **Text** from the sketch toolbar.

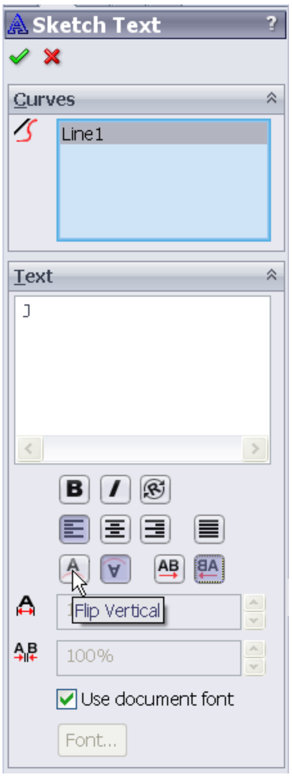
Select the centreline sketch as the guide.

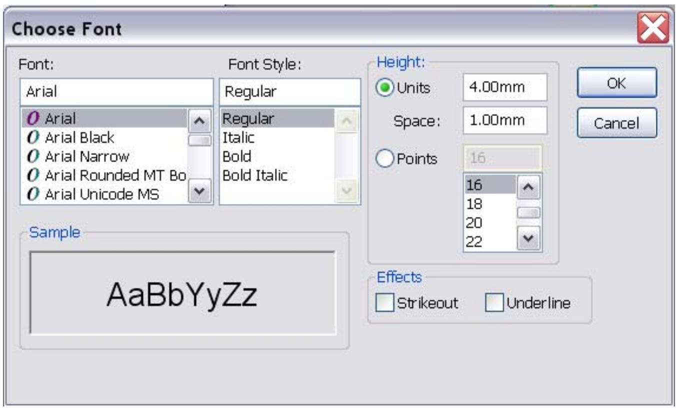
Enter 'J' as **TEXT**

Note the preview in the graphics area.
You may have to use **Flip Vertical** or **Flip Horizontal** to orientate the text correctly.

Deselect **Use document Font** and select Font...

Choose the settings displayed below.





Choose **OK.** Choose **OK** Exit the sketch.

Extrude We are now going to create the extruded feature from the text sketch.

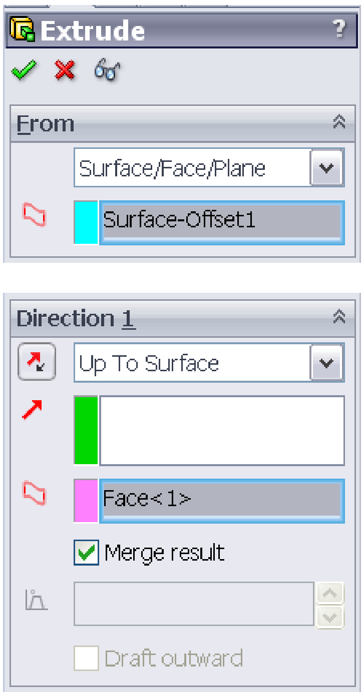
Orientation and display Select an **Isometric View.**

Note: *The sketch is contained on the Top Plane. Underneath the top plane we have the top surface of the button and also Surface-Offset1 which is 0.001mm above it.*
When we generate the extruded feature we want it to start on the offset surface and finish on the top surface of the part, even though the sketch is contained on the top plane. i.e. The extruded feature will be 0.001mm high.

Extrude Text Choose **Extruded Boss/Base** from the feature toolbar.

Select the sketch containing the text.
Tip *– It may be easier to select the centreline, contained within the sketch rather than the text itself.*

Ensure that the extruded boss is downward. To do so toggle **Reverse direction**



To start the extrude on the offset surface choose **Surface/Face/Plane** as the **Starting Condition.**

Select **Surface-Offset1**, as the starting surface, from the featuremanager design tree.

To identify the end condition;

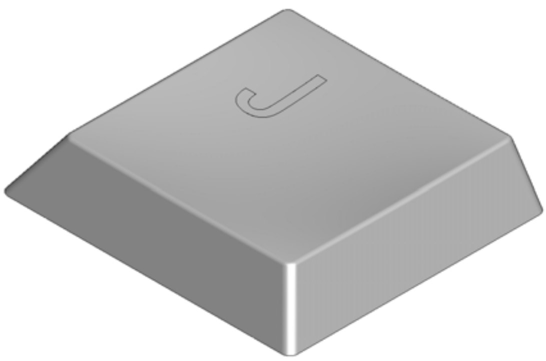
Choose **Up to Surface** from the drop-down menu

Select the top face of the model as **Face/Plane.**

Tip – you may pick the top surface of the model directly from the model as the offset surface, which is located above it, is hidden.

Choose **Merge result.** Choose **OK**

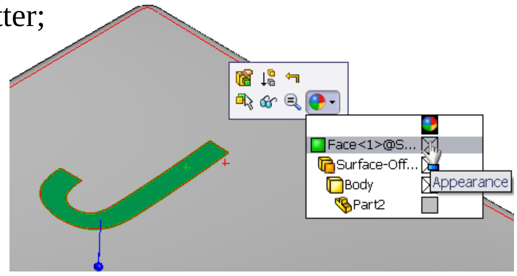
The letter is now displayed as shown below.



Appearance settings

To change the appearance colour of the letter; left click on the face of the letter and choose **Appearance, Face…**

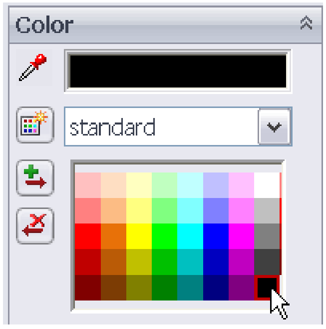
Tip – Choosing face will only apply the colour change to that face.



Face Colour

Choose Black from the colour palette on the left hand side.

Choose **OK**

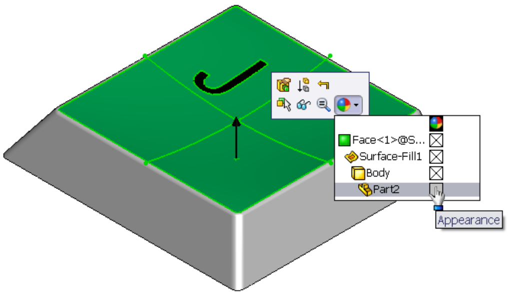


Part Colour

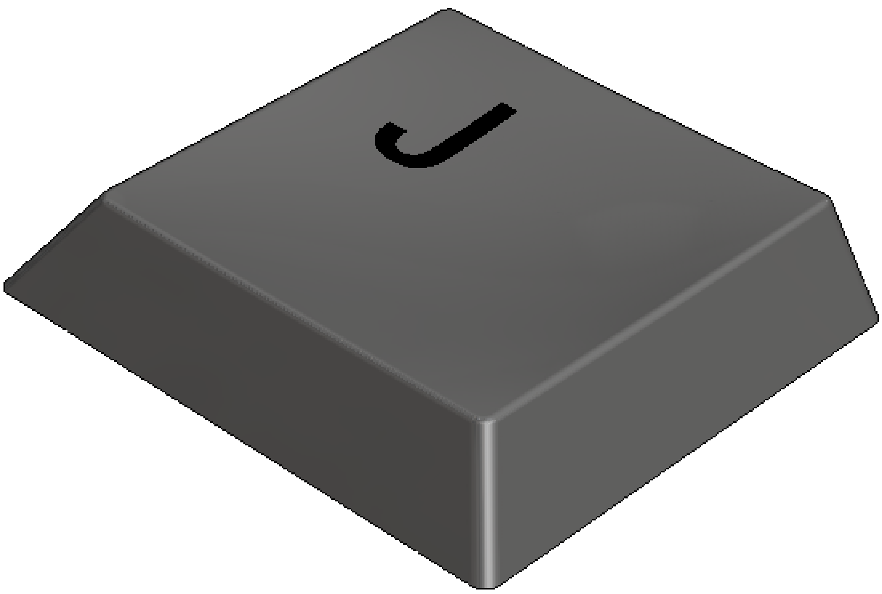
To change the part colour;
Right click on any face and choose **Appearance, Part…**

Choose a colour from the palette.

Choose **OK**



Note – The colour of the letter will not change because face colour overrides part colour



Lesson Complete!

Baseball Cap

Prerequisite knowledge To complete this model you should have a working knowledge of Solidworks 2006/2009.

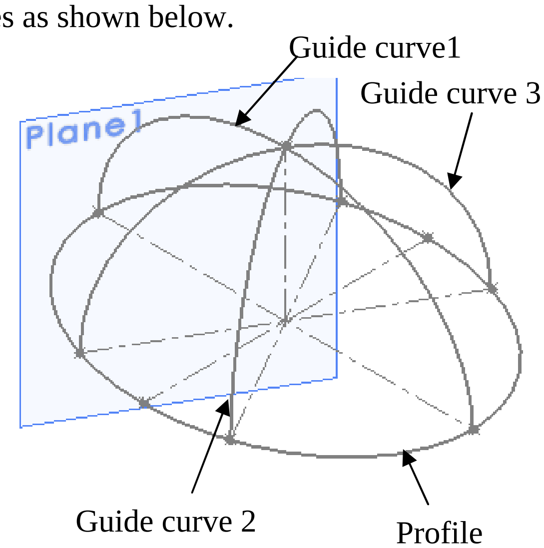
Focus of lesson This lesson focuses on using the following **Surfaces** tools: **Filled Surface, Surface Thicken, Surface Offset** as well as the usual **sketch** and **feature** Tools.

Introduction We will begin with the hat section of the baseball cap. First let us explore the geometry of the hat section. This will be created using a surface fill based on an elliptical profile and three guide curves as shown below.

Profile: this ellipse forms the boundary of the surface and is drawn on the ***Top Plane.***

Guide Curve 1: drawn on the ***Front Plane.*** Its endpoints are ***coincident*** with the major and minor axes of the profile ellipse.

Guide curves 2 & 3:drawn on planes which will be defined using the 60 degree lines and the vertical centreline.



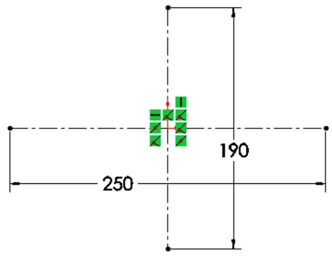
Once the profile and guide curves are drawn, the ***Surface Fill*** tool is used to generate the ***hat section***.

Sketch 1

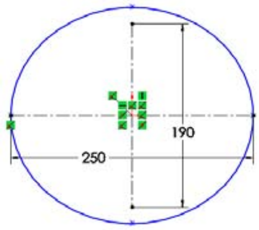
Creating the Base Profile Of the Baseball cap

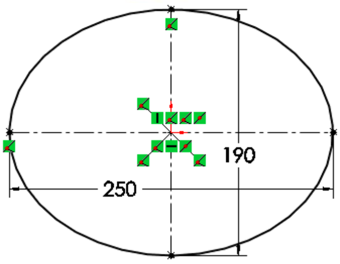
We begin by creating a **sketch** on the **Top Plane**.

Sketch the two **centrelines** shown, which will act as the major and minor axes of the profile ellipse.**Smart Dimension** as indicated and make the midpoints of both **coincident** with the **Origin.**



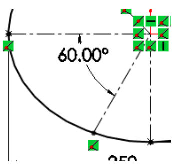
From the **sketch** toolbar select **ellipse**.
Make the centre of the ellipse **coincident** with the **Origin** and make the ends of the major axis **coincident** with the ends of the horizontal centreline. Drop the ends of the minor axis beyond the ends of the vertical centreline. (If you were to drop the ends of the minor axis onto these points, the automatic coincident relations will not be added). We have to apply this coincident relation manually.



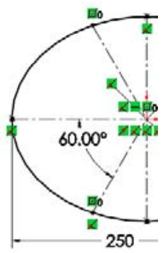


If it is decided to change the dimensions of the centrelines, the major and minor axes will automatically update to reflect this.

Sketch the centreline shown with the relations applied. **Smart dimension** the angle indicated. These lines will be used in the setting up of planes to contain guide profiles for the body of the cap.

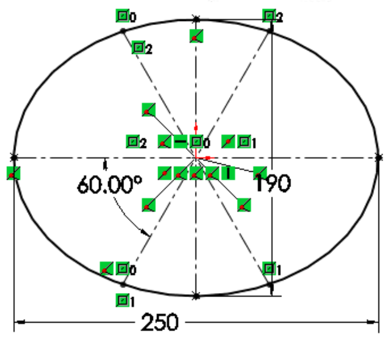


Mirror the 60 degree centreline about the major axis as shown.



Mirror both of these lines now about the minor axis. Note the automatic relations.

Should we decide to change the angle, or the dimensions of the ellipse these lines will automatically reflect these changes because of the relations applied.



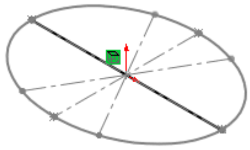
Confirm the sketch.

Sketch 2
Guide curve from front to back

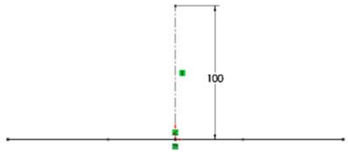
Create a sketch on the **Front Plane**.
If the length of the base of the cap is changed, the curve forming the profile from front to back will need to update to reflect this change. In order for this to happen we will use the centreline(major axis) from sketch 1.

Convert Entities

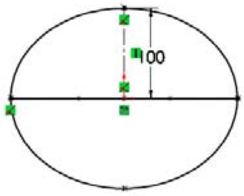
Choose an ***isometric view***, select the major axis, select **convert entities** from the **sketch** toolbar.
N.B. The line will revert to a solid line so it must be changed back to a construction line, use the ***property manager.***



Choose a ***front view***. Draw the centreline and **smart dimension** as shown.



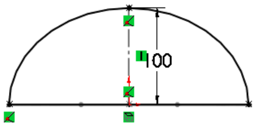
Draw the ellipse with the endpoints of the major axis coincident with the ends of the horizontal centreline. As with **sketch 1** – manually apply the relation afterwards.

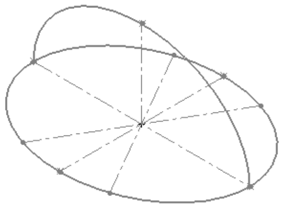


Trim

Trim the sketch to form the semi-ellipse shown. This curve will be the profile of the baseball cap from front to back.

Confirm the sketch.





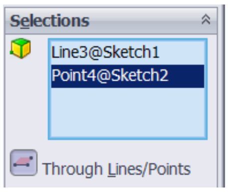
Side profiles

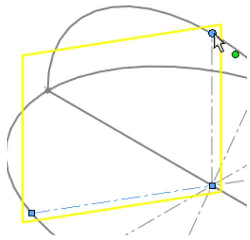
The profiles for the sides of the baseball cap are needed to give the cap the required shape. Were we to form the surface without them, it would end up rather spoon shaped! The profiles will be drawn on two separate planes, which will be set up using the 60 degree lines from sketch 1 and the endpoint of the 100mm vertical line from sketch 2.

Insert Plane

Select **reference geometry, plane.** Choose the following selections.

Choose **OK**.





Sketch 3

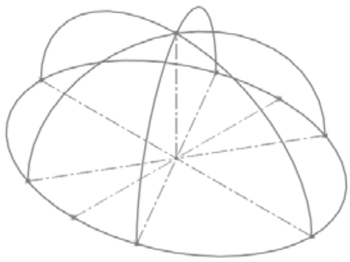
Create the sketch on this plane in the same fashion used for sketch 2. Use **convert entities**

Hide

Hide **plane 1** when the sketch is completed.

Sketch 4

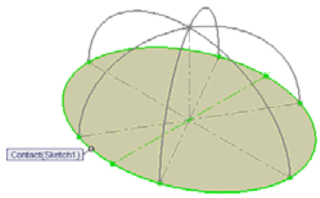
Repeat the process to complete sketch 4.

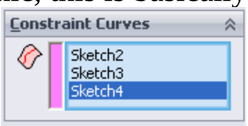


Filled Surface

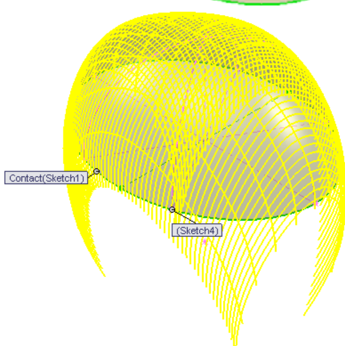
Select **Filled Surface** from the **Surfaces** toolbar.

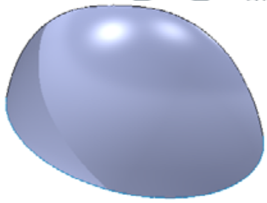
Use ***sketch1*** as the **patch boundary**.
As can be seen from the graphic, this is basically a planar surface.
Select ***sketches 2,3*** and ***4*** as the **guide curves**.
These curves will define the curvature of the surface we require.





The **patch boundary** acts like a cutting plane as it trims off the excess surface created by the **guide curves.**



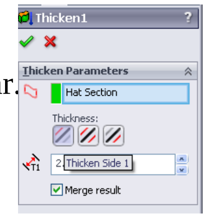


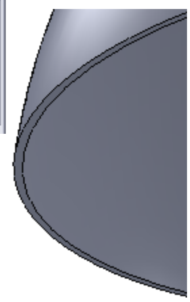
Select **OK.**

Rename Feature Rename the feature as **Hat Section**.

Rotate If the ***Hat Section*** is rotated and viewed from underneath it can clearly be seen that it has no thickness.

Surface Thicken To give the **Hat Section** a thickness, select **Thicken** Thicken from the **surfaces** toolbar.



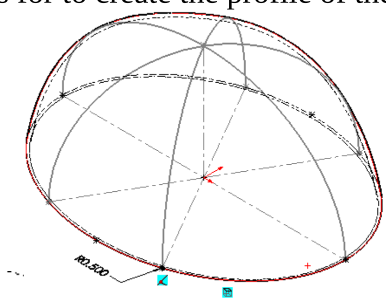


Select the **Hat Section** as the **surface to thicken.**

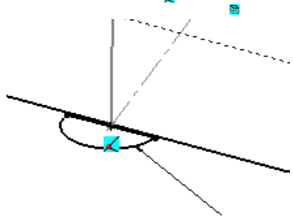
Select ***2mm*** as the thickness.

Select ***Thicken Side 1*** so that the surface thickens towards the inside of the hat to keep the dimensions of the outer surface unchanged.

Creating the Stitching In order to create the stitching effect we will sweep a semicircular profile along the existing guide curves. We will also use the intersection of these guide curves with the base of the hat section as coincident points for to create the profile of the stitching.



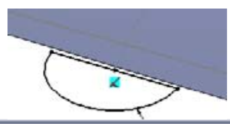
Sketch We will begin by creating the semi-circular profile sketch on the ***Top Plane.***Choose to ***show sketch1.*** Choose a ***Normal To*** view.
Make the centre point of the circle coincident with the intersection point as shown.



Use ***Convert Entities*** to use the ellipse to trim the circle.

Smart Dimension to diameter 1mm.

Confirm the sketch.



Stitching 1 To create the stitching, select ***Swept Boss/Base*** from the ***Features*** menu. Swept Boss/Base In order to create the sweep a profile and a path are required.
Use ***sketch 5*** as the profile. Select ***sketch 4*** as the path along which the profile will travel.

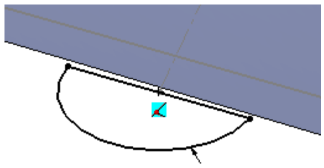
Select **OK.**

Rename Feature Rename the feature as ***Stitching1.***

You will now notice an open hand under ***sketch 4***in both the ***Hat Section*** and the ***Stitching***. This indicates that the sketch is being shared by two features. If the dimensions of the ***hat section*** are changed the ***stitching*** will automatically update as a result of this selection.

Stitching 2

Repeat this process for ***Stitching 2.*** First create the sketch of the semi-circle with its centre point coincident with the intersection of ***sketch3*** and ***sketch1.***



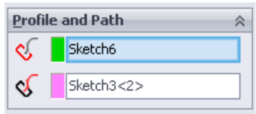
Smart Dimension to diameter 1mm.

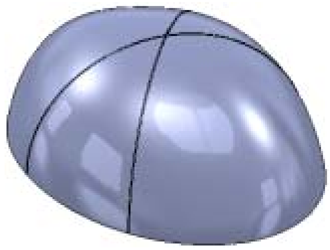
Use ***Convert Entities*** to use the ellipse to trim the circle.

Confirm the sketch.

Stitching 2

To create the stitching, select ***Swept Boss/Base*** from the ***Features*** menu. Swept Boss/Base
In order to create the sweep a profile and a path are required. Use ***sketch 6*** as the profile.
Select ***sketch 3*** as the path along which the profile will travel.





Select **OK.**

Rename Feature

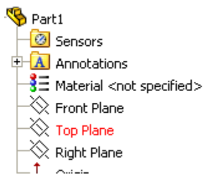
Rename the feature as ***Stitching2.***

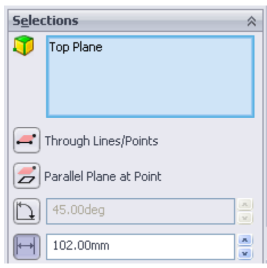
Creating the Button

In order to create the button feature on top of the cap we need to set up a plane on which to create the sketch, as we can't sketch on the surface of the cap. By creating the sketch on this plane, it can then be extruded to the surface of the cap.

Insert Plane

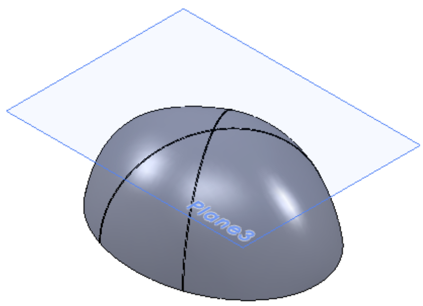
Choose ***Plane*** Plane from the ***Reference Geometry*** commands on the ***Features*** menu.





Choose ***Top Plane*** as the first Reference.

Set the distance as 102mm. This puts the plane 2mm above the highest point of the ***hat section***.

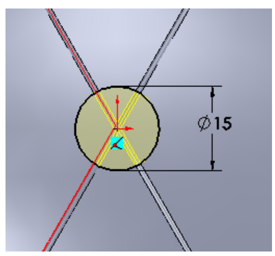


Reverse direction as required.

Select **OK.**

Sketch

Create the sketch shown on ***Plane 3.*** The centre of the circle is coincident with the ***Origin.***



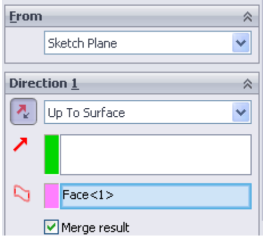
Confirm the sketch.

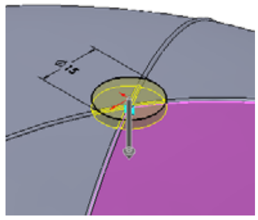
Extrude

Select ***Extrude*** from the ***Features*** menu.

Make the following selections:

For ***direction 1*** choose **Up to Surface.**

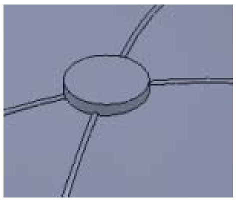




Select any one of the outer faces of the ***hat section***. This will cause the extrusion to stop when it reaches this surface.

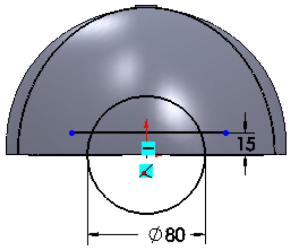
Select **OK.**

Rename Feature Rename the feature as ***Button***.



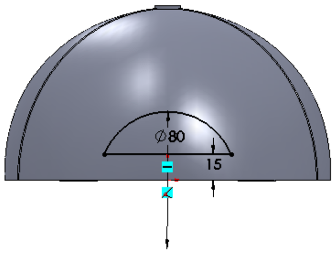
Hide Hide ***Plane 3.***

Rear Hole In order to create the ***rear hole*** cut on the back of the cap, once again we will have to create the sketch on a plane and ***extrude cut*** up to the surface as we can't sketch on the curved surface.



Create this sketch on the ***Right Plane***, making the centre of the circle coincident with the ***origin***.

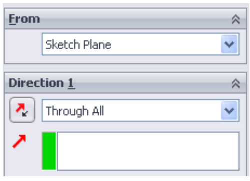
Sketch a horizontal line and ***smart dimension*** as shown.

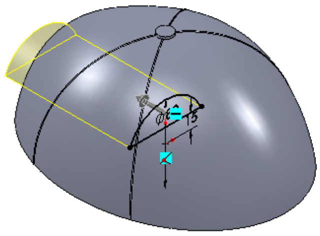


Using the ***Trim*** command trim the ends of the line and the circle.

Confirm the sketch

From the ***Features*** menu select ***Extruded Cut.***

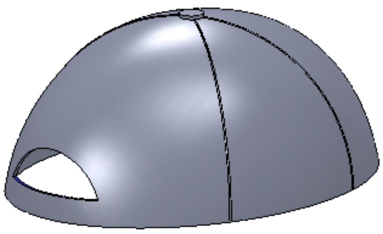




Set the end condition to ***through all*** to ensure that the cut passes through the entire hat structure.

Flip direction as required, to make sure that the cut is made to the rear.

Select **OK.**

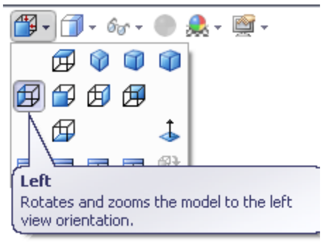


Rename Feature Rename the feature as ***Rear Hole.***

Adjustment Strap The adjustment strap of a baseball cap is usually made from a different material to that of the cap. We already have the strap drawn as part of the ***hat section.*** We can use the ***Split line*** tool to separate the strap section from the hat surface, and apply a different appearance to the strap. The ***Split Line*** tool projects an entity (sketch, solid, surface, face, plane, or surface spline) to surfaces, or curved or planar faces. It divides a selected face into multiple separate faces.

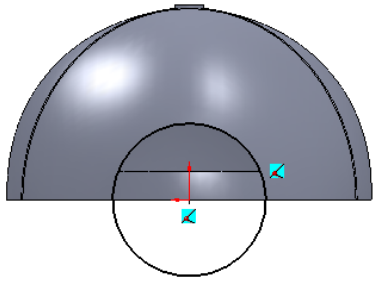
Sketch

We must first create a sketch to use as the line for the ***Split line*** command.



Take a ***Left View*** to enable us to view the ***rear hole*** of the ***Hat Section.***

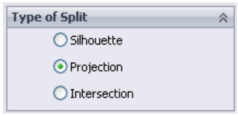
Create the sketch shown opposite on the ***Right Plane,*** Having its centre on the ***Origin*** and making the circumference of the circle coincident with the point indicated.



Confirm the sketch

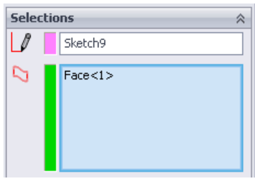
Split Line

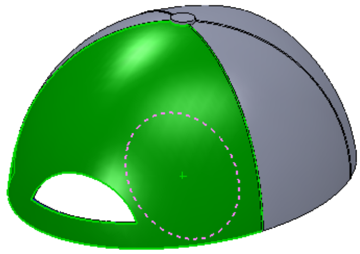
Select ***Curves, Split Line,*** from the ***Features*** menu.



Under ***Type of Split*** select ***projection***; as we want to project the sketch onto the curved surface to create the split.

Make the following selections :





Sketch 9 as the sketch to project

Face 1 as the face to split.

Select **OK.**

Rename Feature

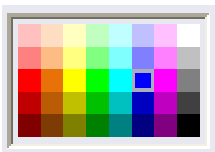
Rename the feature as ***Adjustment Strap Outside.***

Appearance

It appears as though nothing has changed because we don't see a line appearing to separate the two surfaces, so for clarity we will add a colour to the strap portion.

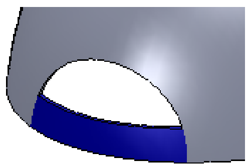
Select the strap portion as shown and select the ***Appearance*** option.

Select **Face<1>@Adj...**



Make the following colour selection.

The division of the surfaces is now much clearer.

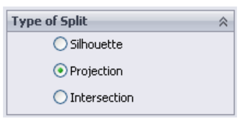


Adjustment Strap Inside

The procedure is repeated for the inside surface of adjustment strap.

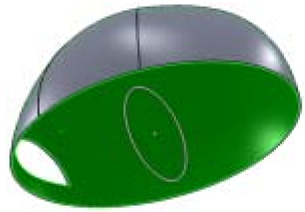
Select ***Curves, Split Line,*** from the ***Features*** menu.

Under ***Type of Split*** select ***projection***; as we want to project the sketch onto the curved surface to create



the split.

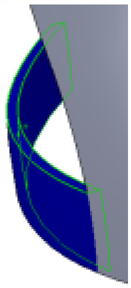
Make the following selections :



Sketch 9 as the sketch to project

The inside surface of the ***Hat section*** as the face to split.

Select **OK.**



Rename Feature

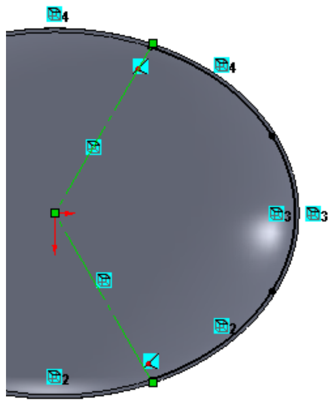
Rename the feature as ***Adjustment Strap Inside.***

Appearance

Again apply a colour to the strap portion. We can apply the colour to the upper surface of the strap as well. The lower edge has not been split. **Do not include!**

The Peak

The peak of the cap is created using the ***Filled Surface*** tool. The boundary for the surface fill is drawn using a combination of ordinates ***(3D Sketch)*** joined up using the ***Spline*** tool and a portion of the elliptical curve on the base of the ***Hat Section.***



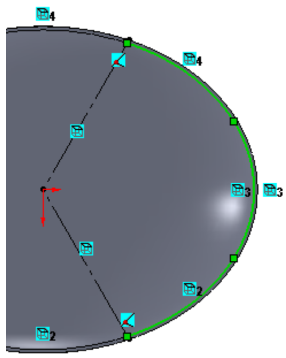
Choose to show ***Sketch 1,*** and <u>choose a ***bottom view.***</u>

Using ***Convert Entities*** convert the lines shown to the new sketch and change their properties to ***for construction.***

Hide

Hide ***sketch 1.***

Select the elliptical edges shown and use ***Convert Entities*** to make them usable in this sketch.
We are using the inner edge of the ***Hat Section*** as a boundary for the ***peak*** to ensure when the surface is thickened it will merge with the surface of the ***Hat Section*** and eliminate the possibility of any space between the ***peak*** and the ***Hat Section.***



Trim

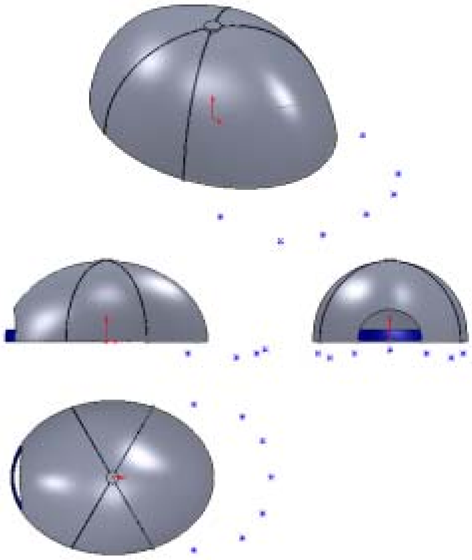
Use the ***Trim*** tool to trim the curve as shown.

Confirm the sketch.

3D Sketch

The peak of a baseball cap is a warped surface. In order to create this effect we will use ***3D Sketch*** to set up points to form the outline of the peak. These points will be joined up using the ***Spline*** tool to form a smooth curve. When setting up points using the ***3d sketch*** tool it is generally done by eye – the points should be viewed from the ***top, front,*** and sides as well as in ***isometric*** while positioning them. When the ***spline*** is added the points and curve can again be pulled into shape, so to speak, to achieve the desired curve. In ***SolidWorks*** a ***spline*** in ***3Dsketch*** tends not to be fully defined. The points can also be set up using XYZ coordinates in ***3Dsketch.*** For convenience in this exercise we will use coordinates to set up the boundary curve for the ***Peak.***

From the ***Sketch*** menu select ***3D Sketch*** to activate the tool.



Now select the ***Point*** command. Select an ***isometric view*** and position the seven random points as illustrated.

Note these points are dropped in randomly for the present. We will apply XYZ values to correctly position them.

As can be seen from the orthographic views the points are randomly positioned but are forming a rough outline of the peak.

XYZ Values

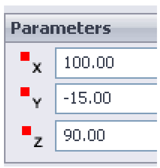
To apply the coordinates to the points we will use the values outlined in the table across.

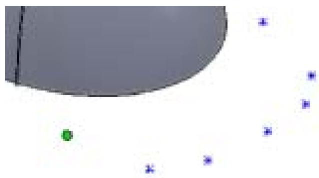
X	Y	Z
100	-15	90
160	-20	75
185	-15	45
195	-10	0
185	-15	-45
160	-20	-75
100	-15	-90

Working from left to right apply the XYZ values to each of the points in turn.

Select the first point by left clicking on it.

Fill in the values for the first point from the table.



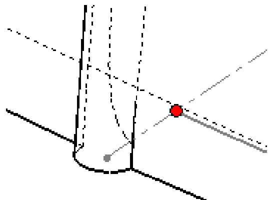


Use the TAB key to move between the XYZ value boxes.

Repeat the process for each of the points.

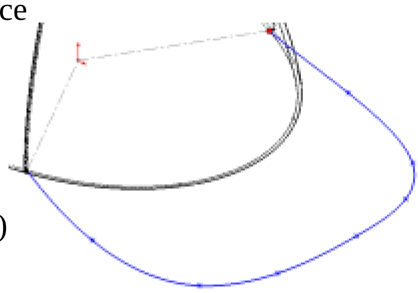
3D Spline

Select ***spline*** from the ***Sketch*** menu.



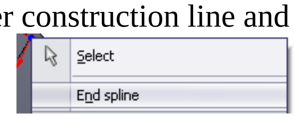
Wireframe used for visibility

Start the spline on the intersection of the construction line and elliptical curve of ***Sketch 10.*** We want the spline to start here so that when the surface is created and thickened it will merge with the surface of the ***Hat Section*** and eliminate the possibility of any space between the ***peak*** and the ***Hat Section.***



Continue to join up the points with the ***spline*** tool taking care to complete the curve in one go.

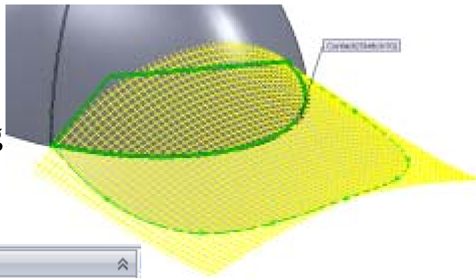
Finish the ***spline*** (Right click and choose end spline) at the intersection of the other construction line and elliptical curve of ***Sketch 10.***



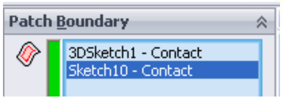
Confirm the sketch.

Filled Surface

From the ***Surfaces*** menu select the ***Filled Surface*** tool. The **Filled Surface** tool constructs a surface patch with any number of sides, within a boundary defined by existing model edges, sketches, or curves, including composite curves.



Select ***3DSketch 1*** and ***sketch 10*** as the ***patch boundaries***.



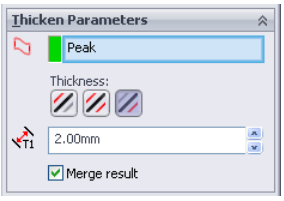
Select **OK.**

We have now created the under-surface of the baseball cap peak.

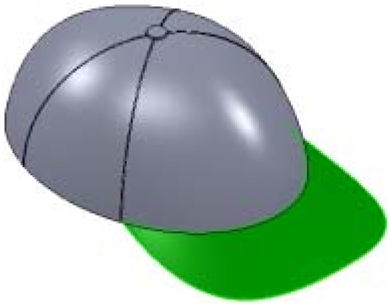
Rename Feature
Surface Thicken

Rename the feature as ***Peak.***
To give the ***Peak*** some depth we will use ***Thicken*** from the ***surfaces*** menu.

Select the ***Peak*** as the surface to thicken.



Set the thickness to ***2mm.***



Choose to ***thicken side 2*** as This will ensure that the Surface of the ***peak*** thickens towards the ***hat section.***

Select **OK.**

Rename Feature

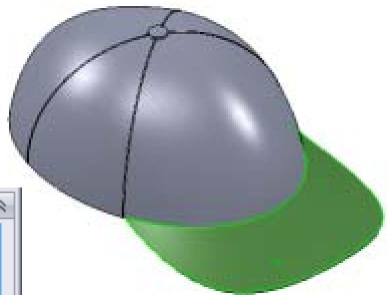
Rename the feature as ***Thicken Peak.***

Nike Logo

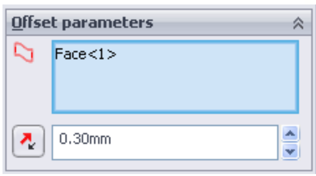
To create the ***Nike*** logo on the peak of the cap we will have to set up the sketch on a plane or planar surface as we can't create a sketch on a curved or warped surface. The top surface of the logo must also run parallel to the top surface of the ***peak*** – as it sits on the ***peak*** of the cap. To satisfy this condition we will set up a ***surface offset***. By doing this the ***extrusion*** can ***begin*** at the offset surface and ***end*** at the top surface of the ***peak.***

Offset Surface

Select ***Offset Surface*** from the ***surfaces*** menu.



Choose the top surface of the ***Peak*** as the ***Surface or face to offset.***

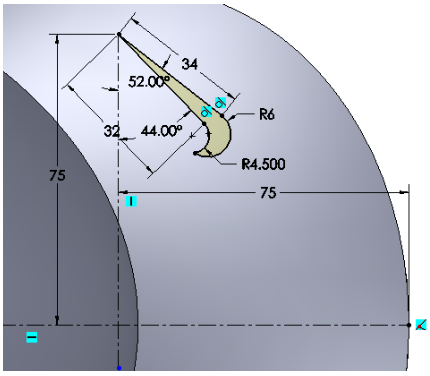


Set the ***thickness*** to ***0.3mm.***

Select **OK.**

Sketch

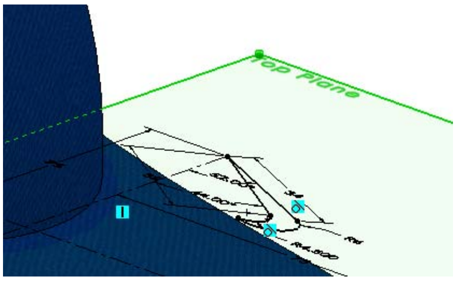
Create the sketch shown on the *Top Plane.*



Smart dimension the sketch as shown.
Note the automatic relations.

Confirm the sketch.

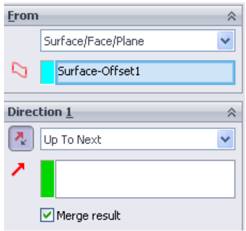
The sketch has been created on the ***Top Plane*** but will be used to create the extrusion from the ***Surface Offset*** to the top surface of the ***Peak.***



Extrude

Select ***Extrude boss/base*** from the ***features*** menu.

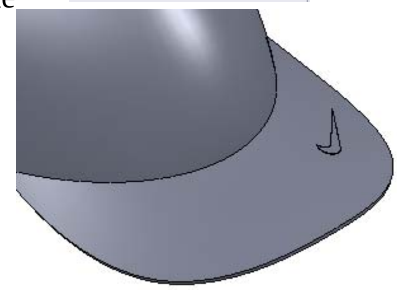
Select ***sketch 11*** as the sketch to use for the extrusion.
Make the following selections:



Set the start condition as ***Surface/face/plane***

Choose ***surface – offset1*** as the start point.

Change the direction of the extrusion so that It is extruding towards the top surface of the ***Peak***.



Set the end condition as ***up to next***.

Select **OK.**

Rename Feature

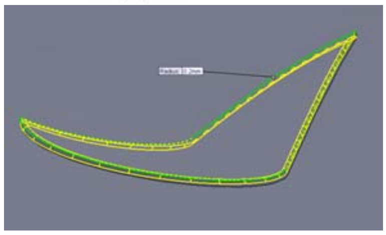
Rename the feature as ***Nike Logo.***

Hide

Hide ***Surface-Offset1.***

Fillet

Apply a ***fillet of 0.2mm*** to the top edges of the logo.

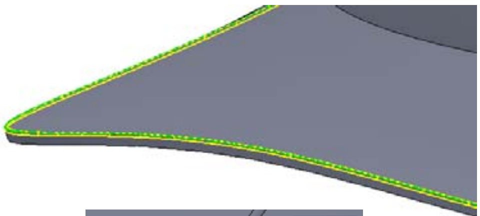


Rename Feature

Rename the feature as ***Logo Fillet.***

Fillet

Apply a ***fillet*** of ***0.5mm*** to the top edge of the ***peak.***

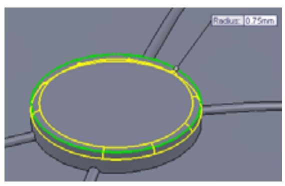


Rename Feature

Rename the feature as ***Peak Fillet.***

Fillet

Apply a ***fillet*** of ***0.75*** to the top edge of the ***button.***

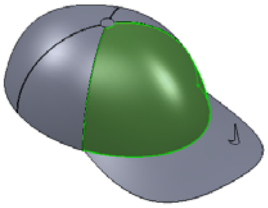


Rename Feature

Rename the feature as ***Button Fillet.***

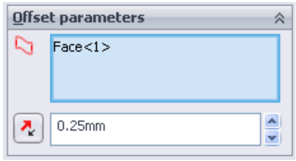
Surface Offset

In order to put the ***NIKE*** text across the front of the ***hat section*** we again have to set up a ***Surface Offset*** as we can't create a sketch on the curved surface.



From the ***Surfaces*** menu select ***surface offset*** as before.

Choose the front of the ***hat section*** as the surface to create the offset from.

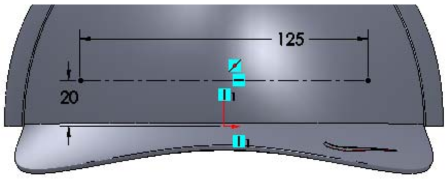


Set the offset distance to ***0.25mm.***

Select **OK.**

Sketch

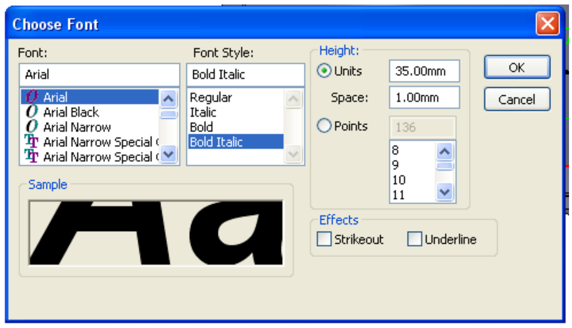
Create the sketch shown on the ***Right Plane.*** Apply a ***vertical*** relation between the ***midpoint*** of the ***centreline*** and the ***origin.***



Smart dimension as shown.

Text

Choose ***Text*** from the ***sketch*** menu.



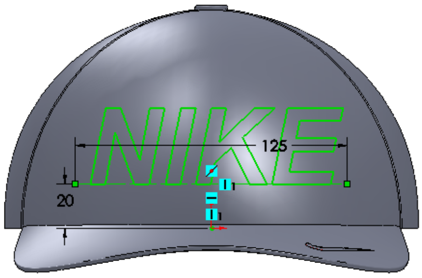
Make the following selections:

Choose the centreline as the line for the text to follow.

Select Arial as the Font.

Set the Font Style to Bold Italic.

Set the units to 35mm.

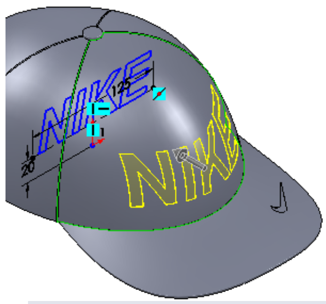


Select **OK.**

Confirm the sketch.

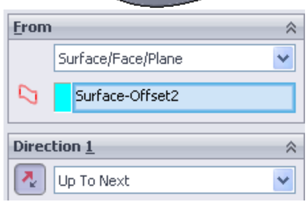
Extrude

Select ***Extrude boss/base*** from the ***features*** menu.



Select ***sketch 12*** as the sketch to use for the extrusion. Make the following selections:

Set the start condition as ***Surface/face/plane***



Choose ***surface – offset 2*** as the start point.

Change the direction of the extrusion so that It is extruding towards the front surface of the ***Hat section***.

Set the end condition as ***up to next***.

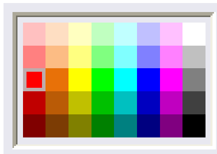
Select **OK.**

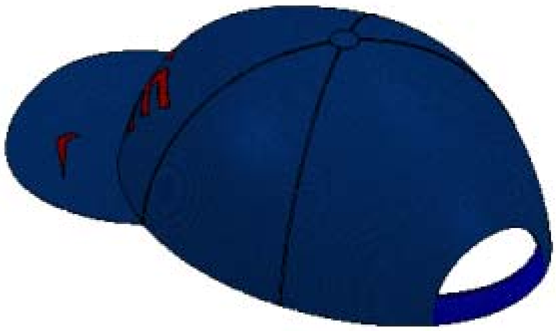
Rename Feature — Rename the feature as ***Nike Text.***

Hide — Hide ***surface – offset 2.***

Apply Material — Under ***fabrics*** choose ***Blue Cotton*** as the material. Apply it to the body of the ***hat section***.

Apply Colour — Select the colour shown across as the colour for the ***NIKE text*** and the ***logo***. Make sure the colour is applied to the ***fillets*** on the ***logo*** as well.

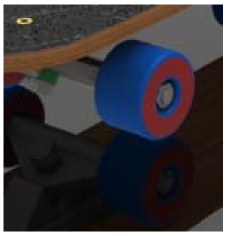
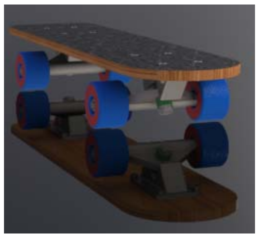
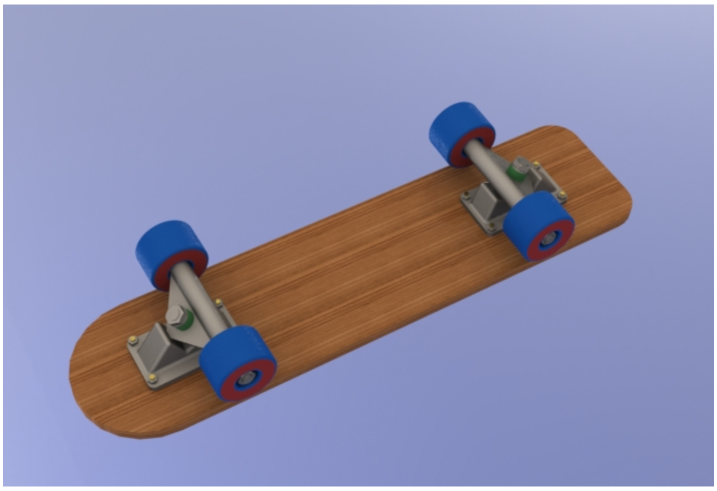
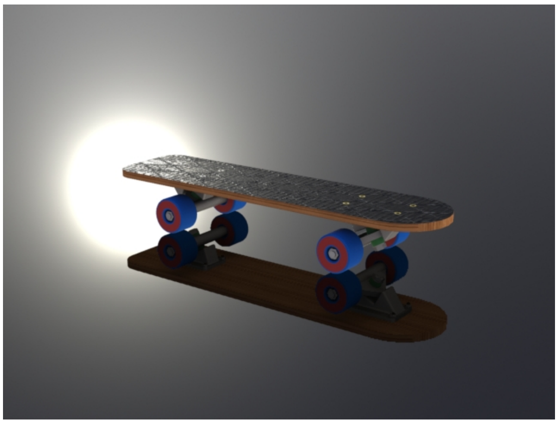




Exercise complete!

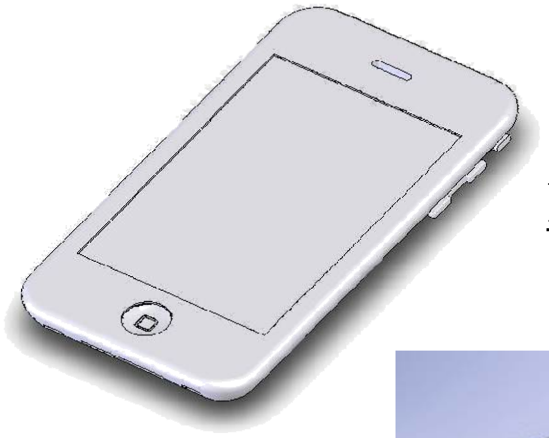
Having completed the exercise it is worth noting the initial positioning of the origin and the subsequent usefulness of the origin and reference planes in completing the exercise.



Creation of Photorealistic Images
PhotoWorks & PhotoView 360



PhotoWorks - iPhone

Prerequisite knowledge A basic knowledge of SolidWorks 2009 is required.

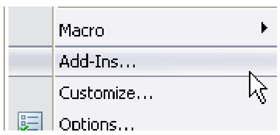
Focus of Lesson This lesson will focus on using PhotoWorks, as part of the SolidWorks program, to generate photorealistic images from a SolidWorks file.

Note ***These notes are created using a SolidWorks part file, however, the same principles apply in creating a photorealistic image from an assembly.***

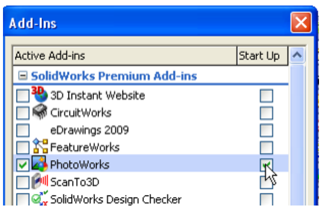
Getting Started PhotoWorks is integrated into SolidWorks 2009 much more so than 2006. Appearances and scenes are applied in the SolidWorks graphics area and are used in PhotoWorks to generate the photorealistic image.

PhotoWorks **PhotoWorks** is an **add-in** of SolidWorks, therefore we must add in the program to begin.

Choose **Tools, Add-ins.**



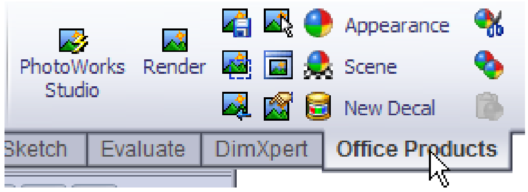
Click on **PhotoWorks** as an **Add in.**



If you want PhotoWorks to appear as an add-in each time you launch SolidWorks; check **Start Up**

Open file Copy the folder called **iphone** from the CD to your computer.
Open the part called **iphone** from the iphone folder.

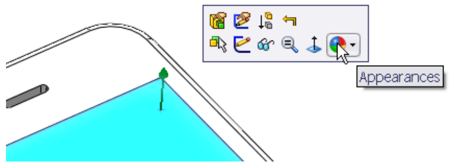
PhotoWorks can now be accessed in the **Office Products** tab in Solidworks



Appearances

Because the iphone is predominantly black in colour, we will add a black appearance to the **entire part**.

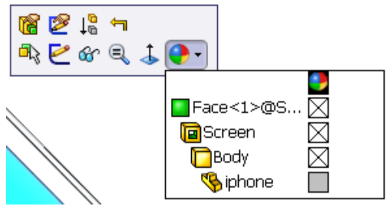
Left click on any of the faces of the iphone
And choose **appearance** from the pop-up menu.



The hierarchy of application of appearances is displayed.

Face, Feature, Body or **Part**

Remember; An appearance applied to a face will override that applied to a feature, which will override that applied to a part…

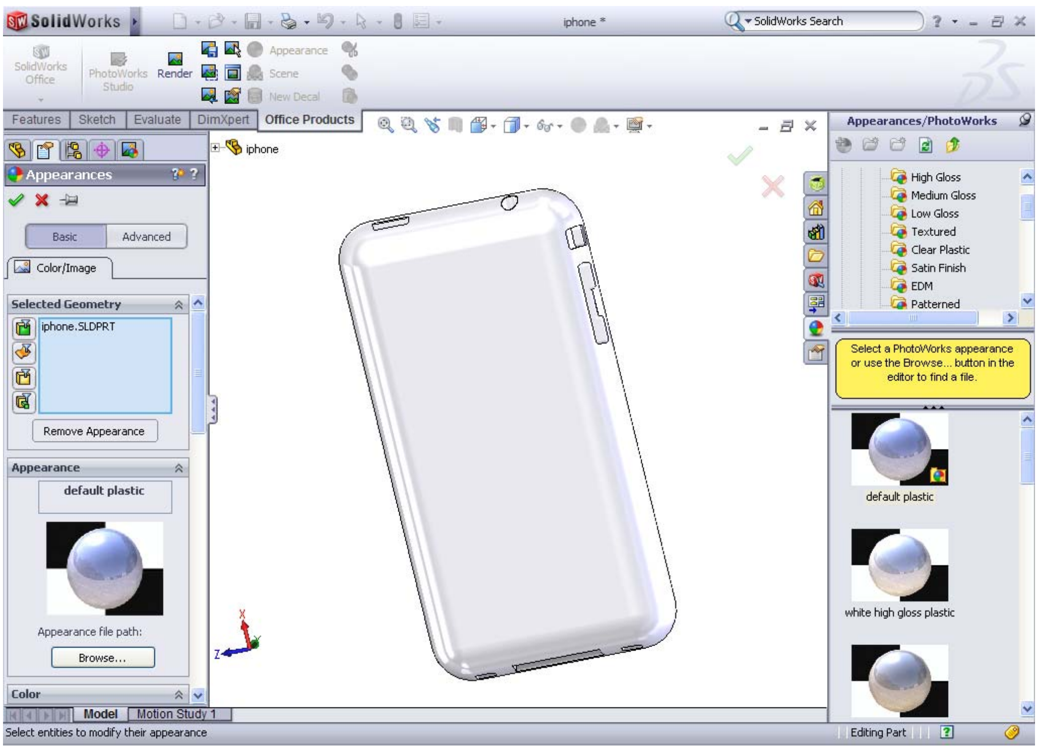


Black appearance

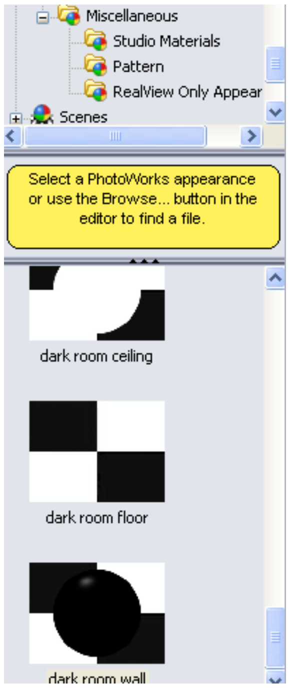
A default appearance has been applied to the part on creation.
Click into the square next to the part name – **iphone.**

(An ☒ indicates that no appearance has been applied)

The **appearances palette** is displayed in the task pane.



The appearances are displayed in a tree like format.

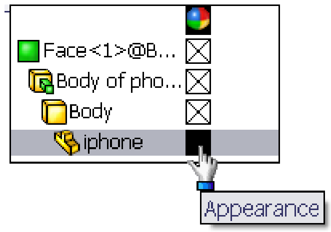


We wish to apply a black appearance to the part.
Expand the tree to locate:

Miscellaneous, Studio materials, Dark room wall

Drag and drop the appearance onto the model.
Choose **OK.**

The appearance is applied to the entire part because the part was preselected from the hierarchy tree.



Silver trim

We will now apply a silver appearance to a number of features of the iphone

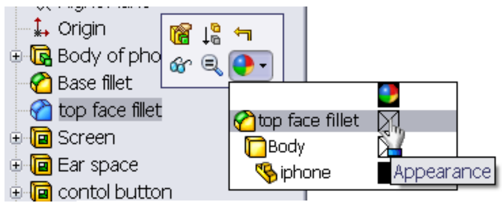
The following image shows the Hierarchy of appearances

The **face** will override the **feature,**
The **feature** overrides the **body**
The **body** overrides the **part.**

The black appearance we have applied to the entire part will be overridden by a different appearance should we apply it to a **face, feature or body**

Left click on the **top face fillet** from the featuremanager design tree.

Choose Appearance and select ☒ next to top face fillet, as shown.



Choose **Metal, Silver, Polished Silver** from the appearances palette in the task pane.

Choose **OK**.

Multiple Selections

We can apply this appearance to multiple feature feature manager design tree.

Whilst holding down control, select each of the following features;

control button, power button, mute button, volume control, volume space, fillet volume, volume button 2 fillet and connection port 2

Release **Ctrl** and choose the ☒ next to the features in the hierarchy display.

Choose **Metal, Silver, Polished Silver** from the appearances palette in the task pane.

Choose **OK.**

The appearance will be applied to all of the preselected features and will override the black appearance applied to the entire part.

Speakers

Apply **Fabric, Carpet, Carpet colour 4 to the** two **speakers** and the **ear space**, in the same way.

Decal

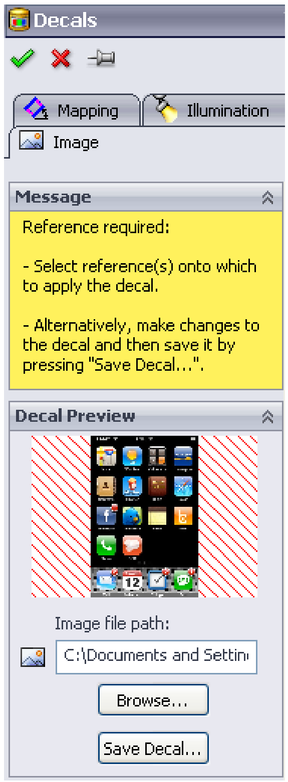
The ability to apply a decal is one defining aspect that is different between PhotoView 360 and PhotoWorks. You can only apply a decal in PhotoWorks.

A decal is any digital image that can be manipulated into being on a face of a solidworks object. (BMP – bitmap files work best)

We are going to apply the iphone interface decal located in the **iphone folder** onto the screen of the iphone.

Applying decal

Choose **PhotoWorks, decal** and browse to find the **homescreen iphone**
(located in the iphone folder**)**



Select the **face** on which we want the decal to be Positioned.

Mapping Image

We now have to map the digital image to the screen to ensure it fits the screen correctly

Select **Mapping** tab in the left window.

Ensure that the map is set as **label.**

Ensure the angle of the decal is correct.

Use the angle adjustments to ensure it fits correctly.

Size/Orientation

Fit to width and fit to height.

You may need to mirror horizontally or vertically.

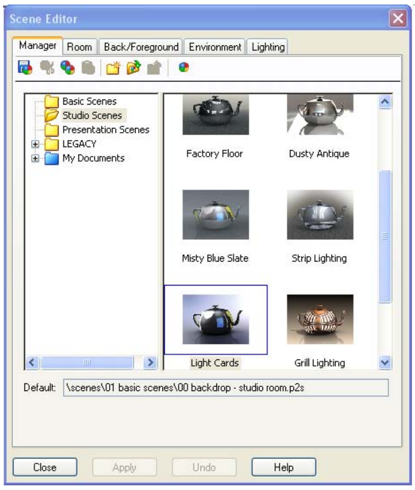
Choose **OK**..

Scene

Choose **PhotoWorks, Scene.**

Choose the desired scene. Choose **Apply.**

Alternatively the scene may be dragged and dropped from the task pane or selected from the heads up toolbar.



Render

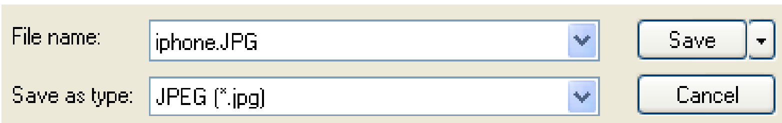
Once the scene has been selected it must be rendered to see the image.

Before rendering orientate the model and zoom in or out to position the required view. Whatever is displayed on screen is what will be captured in the photorealistic image.

Choose **PhotoWorks, Render.**

Save the file

To save the file choose **File, Save as**. Save the model in the iphone folder in an appropriate file format, **JPEG** being the most common.

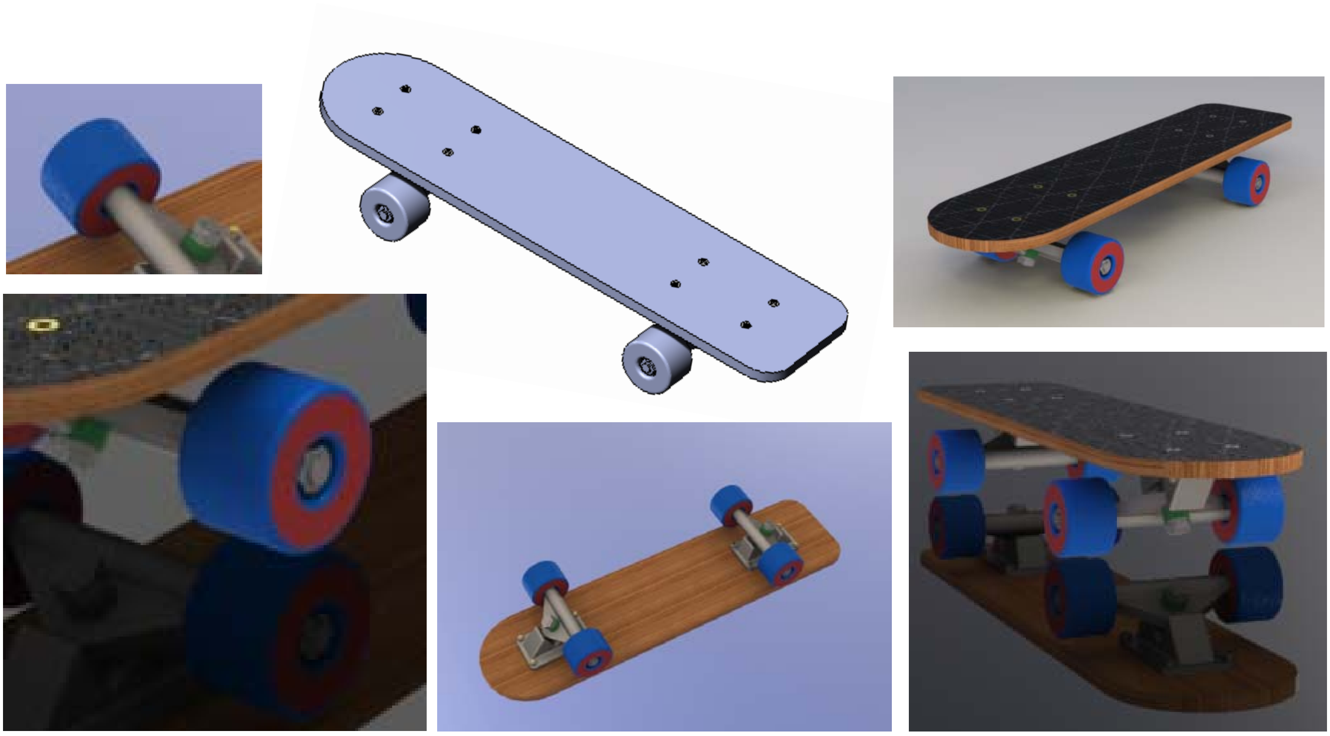


When the model is moved or rotated the rendered model will disappear.

Render to File

Alternatively you may choose **PhotoWorks, Render to File**. This will create the jpeg image file without producing a rendered image on screen.

Photoview 360 – Creating Photorealistic Images



Prerequisite knowledge A basic knowledge of SolidWorks 2009 is required.

Focus of Lesson This lesson will focus on using PhotoView 360 to create photorealistic images from SolidWorks models.

PhotoView 360 PhotoView 360 is a software package, separate to SolidWorks, which generates digital lifelike images (photorealistic images) from SolidWorks files. PhotoView 360 is an easy and quick way to produce high quality photorealistic images.

There are a wider range of appearances in **PhotoView 360** than in **SolidWorks.**

Getting Started **PhotoView 360** is a separate program which loads with SolidWorks 2009.

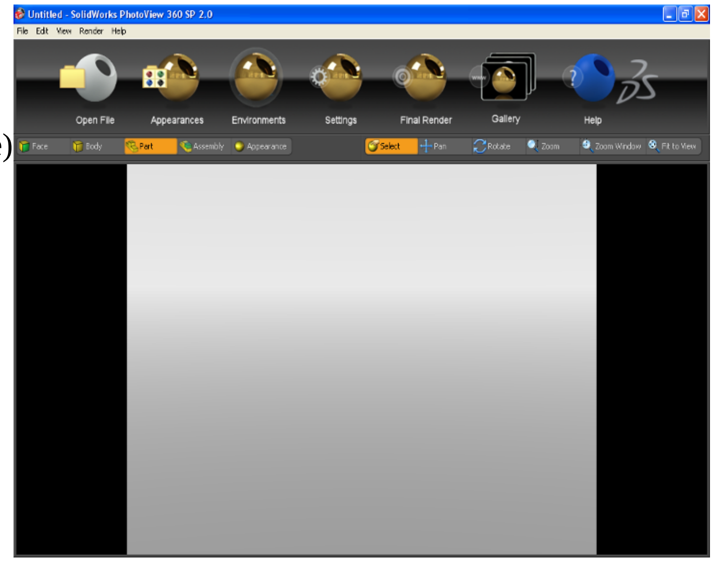
To launch the program; double click on the PhotoView 360 icon located on the desktop.

PhotoView 360 allows you to import a SolidWorks part or assembly and apply particular appearances and scenes to that file.

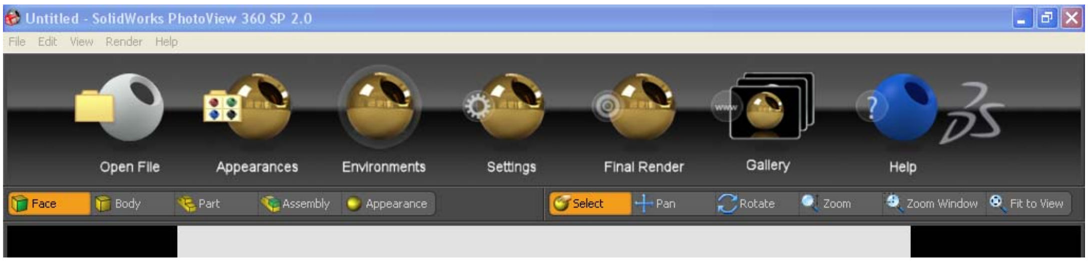
The Interface

When using PhotoView 360 we work from left to right on the main menu.

1. Open file
   (Import SolidWorks File)
2. Apply appearances
3. Apply Environment
4. Edit settings
5. Final render, save file



The main menu is shown below;



Toolbars

There are four options when applying an appearance – you may choose to apply an appearance to an **entire assembly**, **single part**, **body of a part** or an individual **face**.
The appearance will be applied based on whichever option is pre-selected.

There is a second toolbar under the main toolbar. Its function is to move and rotate the model to enable the user to choose particular faces, features or parts as well as positioning the model to capture the photorealistic image.

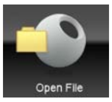
The **roller ball** of the mouse can be used to maneuver a solid in **Photoview 360** window in the same way as **SolidWorks.**

The **select** icon must be highlighted in order to apply appearances to any aspect of the SolidWorks model.

Transfer Folder Copy the folder named **Photoview 360** from the **CD** onto your computer.

Open File Open the assembly named **Skateboard** located in the folder **Photoview 360.** The parts and assembly were created in SolidWorks using the default material appearance.

When you open a SolidWorks file in PhotoView 360, any appearances applied in SolidWorks will be displayed. However, there are enhanced appearances available in PhotoView 360 and it is recommended that these are applied to achieve better results.

Use the **open file** button 

and open the **skateboard** assembly.

Rotate and Pan Practice moving and rotating the model using the various tools in the navigation toolbar.

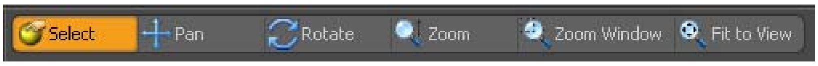
Highlight the command, move to the graphics area, manipulate the positioning of the model.

Applying appearances We are going to apply appearances to each individual part of the model.

Appearances A wooden finish is to be applied to the board of the skateboard.

Highlight the **part** icon in the selection toolbar.

Ensure that **select** is highlighted in the navigation toolbar.



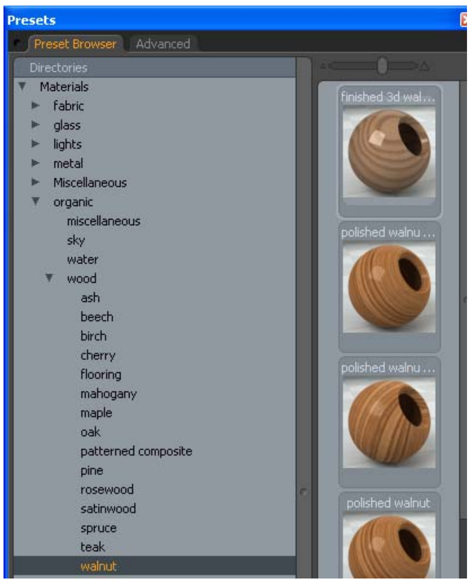
Adding Appearance Click on **appearances** button 

in the main toolbar.

The **Presets** window will appear.
Click on the triangle next to the category name to expand or collapse the selection tree.

Navigate to **Organic, wood, walnut.**

Drag and drop **polished walnut** onto the desired part in the graphics area.



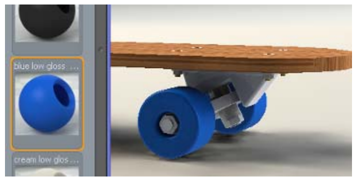
Note: Because **Part** was preselected, the appearance is applied to the entire part not just the face onto which it was dropped.

Edit Appearance

If you wish to change an appearance; drag and drop the chosen appearance onto the part and it will replace the existing one.

Wheels

Employing the same procedure, apply a **blue low gloss plastic** appearance to any of the wheels of the skateboard.

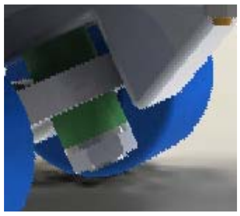


Note: The remaining three wheels will also display that same appearance. This is because these are four occurrences of the same SolidWorks part – **Wheel**.

Bush

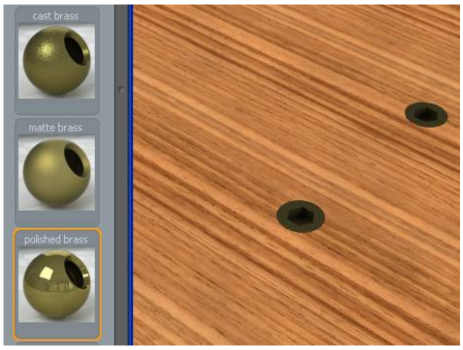
Apply a **green low gloss plastic** appearance to the bushes.

Note: You will have to rotate the model in order to access the bush to drop the appearance onto it.



Hex Bolt

Apply a **polished brass** appearance to the **Hex Bolts and nuts** (Bolts and nuts joining the board to the **bearing**)



Steel elements

All other parts are to have **machined steel** appearance applied.

Face appearance

All appearances applied thus far have been applied to entire parts. We can also constrain an appearance to be applied to a particular face of a part.

Just as in SolidWorks, a hierarchy exits when applying appearances within PhotoView 360. **A face appearance will override an appearance which has been applied to an entire part.**

Wheel face

We will apply an appearance to the face of the wheel to enhance the model.

Pre-select **face** on the selection toolbar.

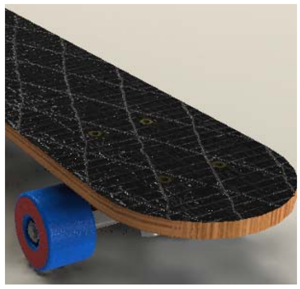
Navigate to **red high gloss plastic** in the Presets window.

Drag and drop the appearance onto the face of the wheel.
Just as before this appearance will be applied to the faces of the four wheels.

Top of skateboard

Choose an appropriate appearance for the **top face** of the **skateboard**

Plastic, Composite, Carbon Fiber Inlay Unidirectional



Environments

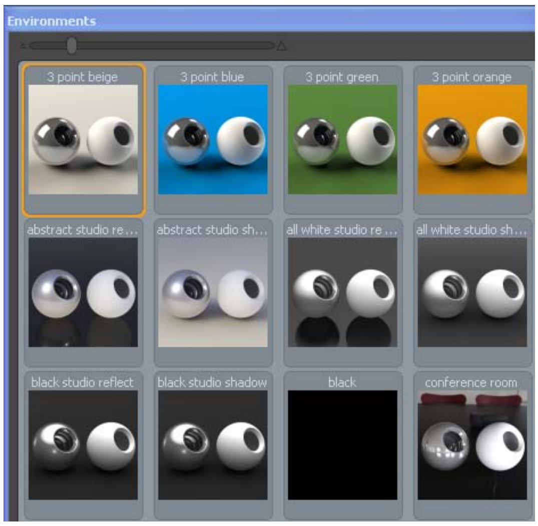
Environments can be looked upon as **backdrops** and can be applied in a similar manner as the appearances are applied to faces or parts.

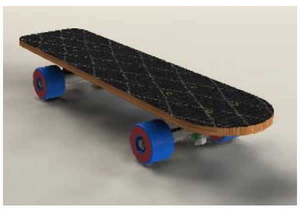
Select **environments** in the main menu.
The **Environments** selection box appears.

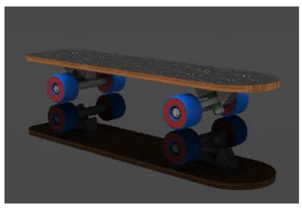
Drag and drop the chosen environment into the graphics area.

Alternatively **double click** on the desired environment.

Dragging and dropping an alternative environment will override the previous selection.





The above are examples of environments that can be applied.

Apply a **daytime** environment.

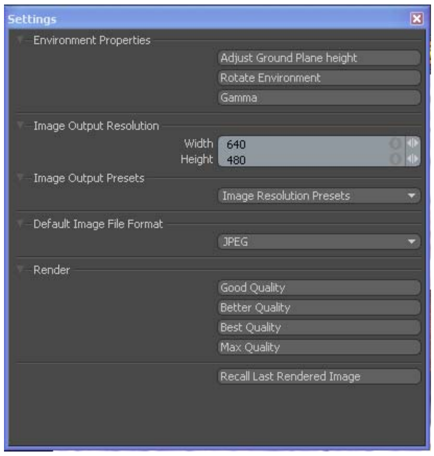
Positioning

Capturing a good photorealistic image can be compared to setting up a camera for taking a photo.

Use the navigation tools to orientate the model so that it fills the majority of the display window and shows good detail of the solid.

Settings for rendering

Settings allows us to set the format in which our image will be saved along with the quality of the image.



Open **settings** option from **main menu**

Environment properties

Adjust Ground Plane height;
Sets the floor height of the environment in relation to the model.
Decreasing this number moves the floor down, increasing the number moves the floor up, closer to the model.

***Rotate Environment**:*
Rotates the environment in relation to the model. This will affect lighting, shadow and shade etc.

Gamma: corrects the output to compensate for the output device ie a printer or monitor.
Image Output Resolution – changes the number of pixels in the final rendering. Increased number of pixels, increases files size and rendering time.
Image output Presets – height and width may be chosen or choose a preset value. 640 x 480 is suitable or an A3 size output.
Default Image File Format – JPEG, BMP, etc
Render – good, better, best, max

The higher the quality of the render, the longer the time it takes to complete the render.

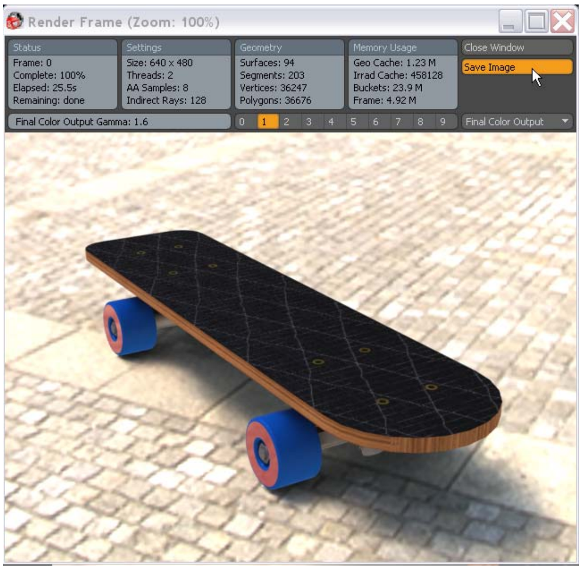
A **better quality** image is sufficient to complete this exercise but sample the other quality images also.

Once the type of render is selected a new window opens with the selected options applied

Render

The render is now complete and a JPEG file can be created from this window.

Save the image as **Skateboard** in the PhotoView 360 folder.



Once rendered, the image will be allocated a number 1 – 9 as indicated above. Selecting that number will allow you to retrieve the image at a later stage, if required.

Add different environments and manipulate the positioning of the model to capture various images of the model.

Zoom and span to specific areas of the model to take photorealistic images of different components