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.

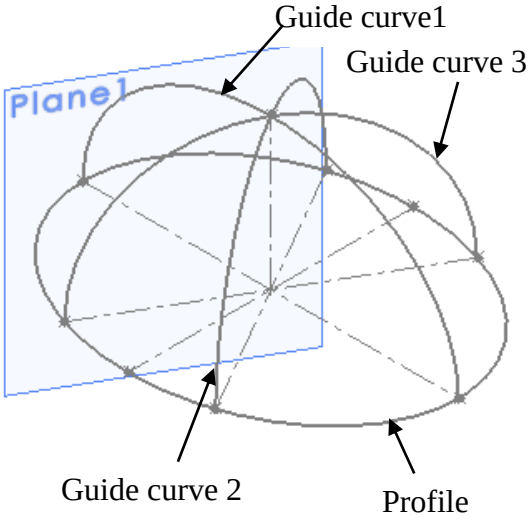
Introduction

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.



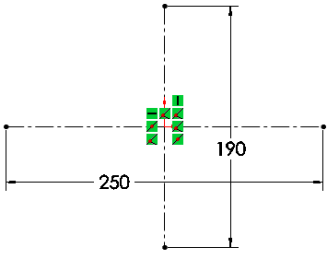
Once the profile and guide curves are drawn, the ***Surface Fill*** tool is used to generate the ***hat section***.

Sketch 1

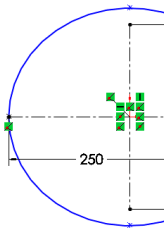
We begin by creating a **sketch** on the **Top Plane**.

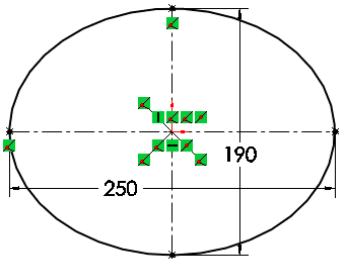
Creating the Base Profile Of the Baseball cap

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.**



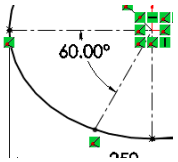
From the **sketch** toolbar select **ellipse**.
Make the centre of the ellipse **coincident** with the **Origin** and make the ends of the major axis **coincident** with the ends of the horizontal centreline. Drop the ends of the minor axis beyond the ends of the vertical centreline. (If you were to 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.

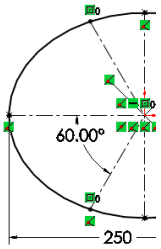




If it is decided to change the dimensions of the centrelines, the major and minor axes will automatically update to reflect this.

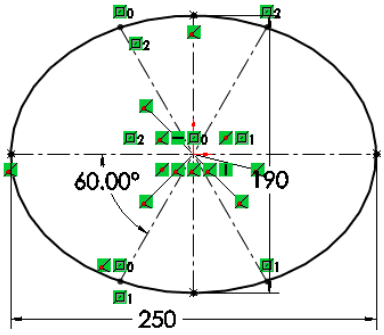
Sketch the centreline shown with the relations applied. **Smart dimension** the angle indicated. These lines will be used in the setting up of planes to contain guide profiles for the body of the cap.





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.

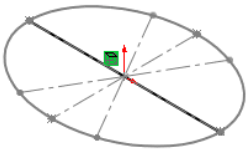
Confirm the sketch. 

Sketch 2
Guide curve from front to back

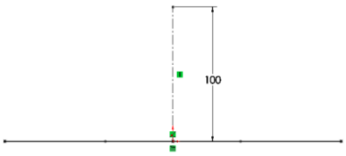
Create a sketch on the **Front Plane**.
If the length of the base of the cap is changed, the curve forming the profile from front to back will need to update to reflect this change. In order for this to happen we will use the centreline(major axis) from sketch 1.

Convert Entities

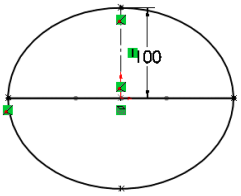
Choose an ***isometric view***, select the major axis, select **convert entities** from the **sketch** toolbar.
N.B. The line will revert to a solid line so it must be changed back to a construction line, use the ***property manager.***



Choose a ***front view***. Draw the centreline and **smart dimension** as shown.

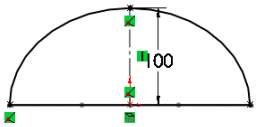


Draw the ellipse with the endpoints of the major axis coincident with the ends of the horizontal centreline. As with **sketch 1** – manually apply the relation afterwards.

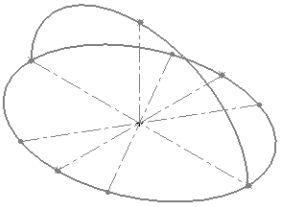


Trim

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.

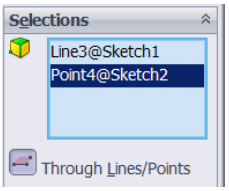


Side profiles

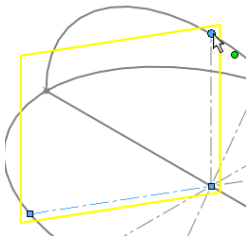
The profiles for the sides of the baseball cap are needed to give the cap the required shape. Were we to form the surface without them, it would end up rather spoon shaped! The profiles will be drawn on two separate planes, which will be set up using the 60 degree lines from sketch 1 and the endpoint of the 100mm vertical line from sketch 2.

Insert Plane

Select **reference geometry, plane**. Choose the following selections.

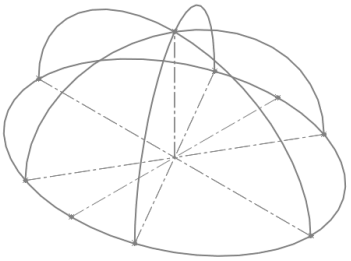


Choose **OK**.



Sketch 3

Create the sketch on this plane in the same fashion used for sketch 2. Use **convert entities**



Hide

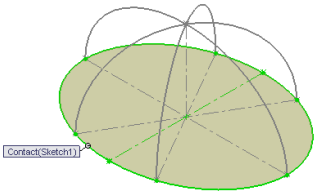
Hide **plane 1** when the sketch is completed.

Sketch 4

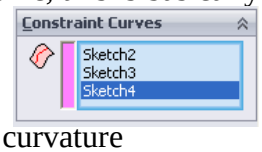
Repeat the process to complete sketch 4.

Filled Surface

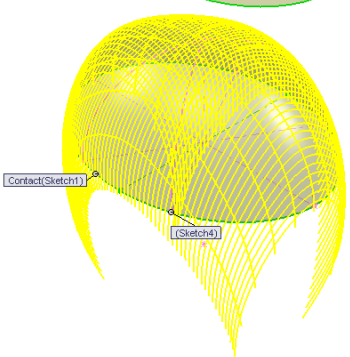
Select **Filled Surface** from the **Surfaces** toolbar.



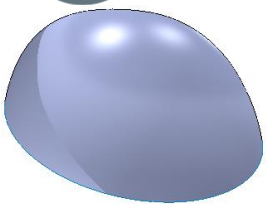
Use ***sketch1*** as the **patch boundary**.
As can be seen from the graphic, this is basically a planar surface.
Select ***sketches 2,3*** and ***4*** as the **guide curves**.
These curves will define the curvature of the surface we require.



The **patch boundary** acts like a cutting plane as it trims off the excess surface created by the **guide curves.**



Select **OK**.



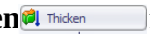
Rename Feature

Rename the feature as **Hat Section**.

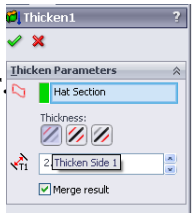
Rotate

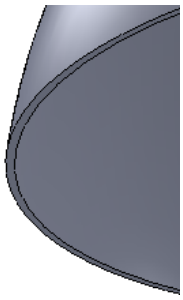
If the ***Hat Section*** is rotated and viewed from underneath it can clearly be seen that it has no thickness.

Surface Thicken

To give the **Hat Section** a thickness, select **Thicken** 

from the **surfaces** toolbar.





Select the **Hat Section** as the **surface to thicken.**

Select ***2mm*** as the thickness.

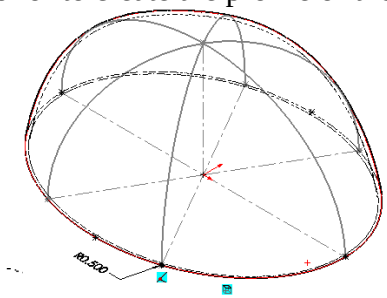
Select ***Thicken Side 1*** so that the surface thickens towards the inside of the hat to keep the dimensions of the outer surface unchanged.

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.

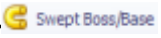


Use ***Convert Entities*** to use the ellipse to trim the circle.

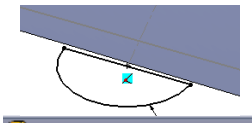
Smart Dimension to diameter 1mm.

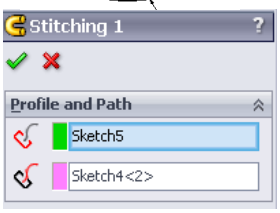
Confirm the sketch. 

Stitching 1

To create the stitching, select ***Swept Boss/Base*** from the ***Features*** menu. 

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.

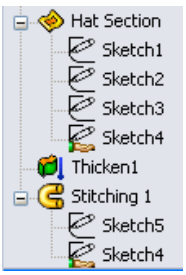




Select **OK.**

Rename Feature

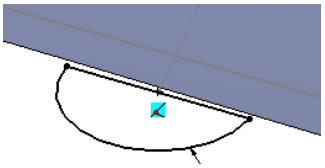
Rename the feature as ***Stitching1.***



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.***



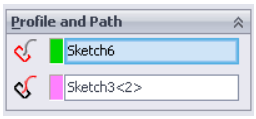
Smart Dimension to diameter 1mm.

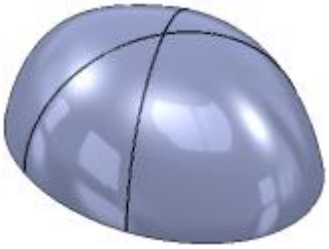
Use ***Convert Entities*** to use the ellipse to trim the circle.

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

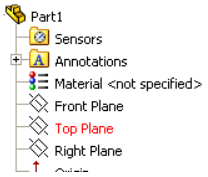
Rename the feature as ***Stitching2.***

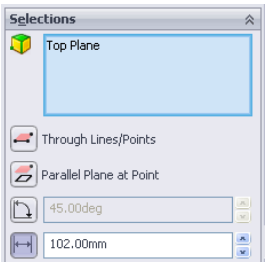
Creating the Button

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.



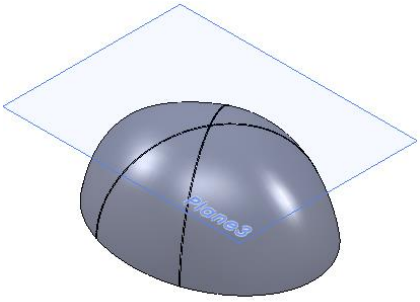


Choose ***Top Plane*** as the first Reference.

Set the distance as 102mm. This puts the plane 2mm above the highest point of the ***hat section***.

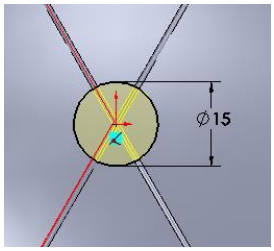
Reverse direction as required.

Select **OK.**



Sketch

Create the sketch shown on ***Plane 3.***
The centre of the circle is coincident with the ***Origin.***



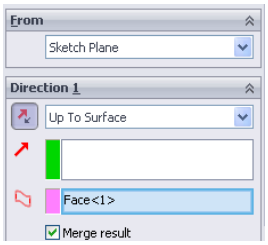
Confirm the sketch.

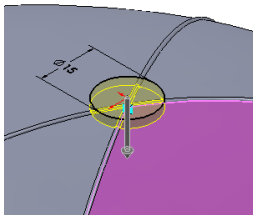
Extrude

Select ***Extrude*** from the ***Features*** menu.

Make the following selections:

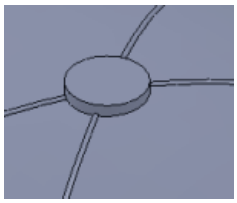
For ***direction 1*** choose **Up to Surface.**





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

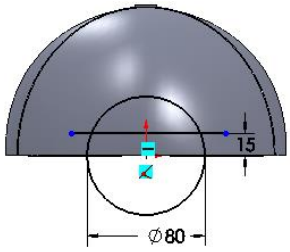
Rename the feature as ***Button***.

Hide

Hide ***Plane 3.***

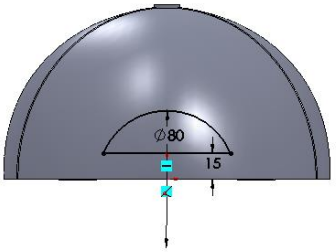
Rear Hole

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***.

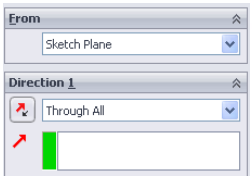
Sketch a horizontal line and ***smart dimension*** as shown.



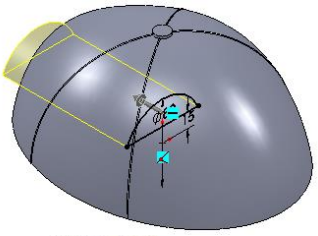
Using the ***Trim*** command trim the ends of the line and the circle.

Confirm the sketch

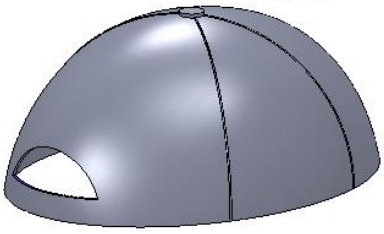
From the ***Features*** menu select ***Extruded Cut.***



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

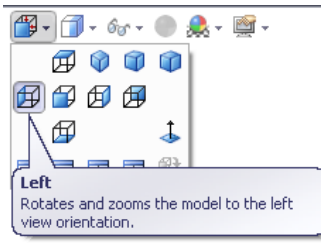
Rename the feature as ***Rear Hole.***

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.

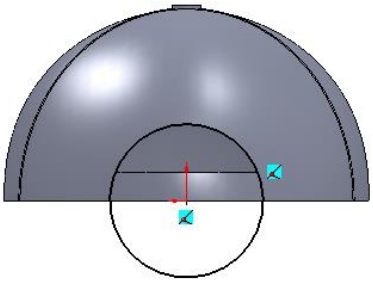
Sketch

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.***

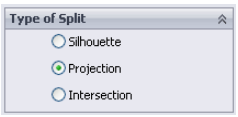
Create the sketch shown opposite on the ***Right Plane,*** Having its centre on the ***Origin*** and making the circumference of the circle coincident with the point indicated.



Confirm the sketch 

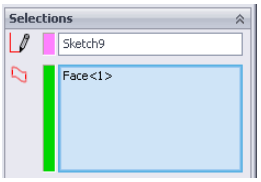
Split Line

Select ***Curves, Split Line,*** from the ***Features*** menu.



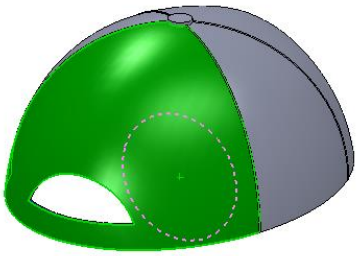
Under ***Type of Split*** select ***projection***; as we want to project the sketch onto the curved surface to create the split.

Make the following selections :



Sketch 9 as the sketch to project

Face 1 as the face to split.



Select **OK.** 

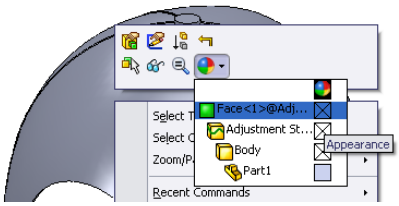
Rename Feature

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.

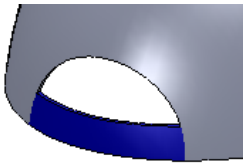
Select the strap portion as shown and select the ***Appearance*** option.



Select **Face<1>@Adj...**

Make the following colour selection.

The division of the surfaces is now much clearer.

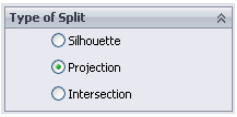


Adjustment Strap Inside

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

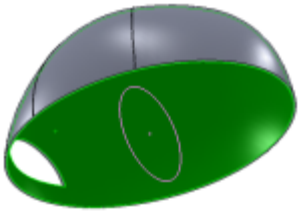


the split.

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.***

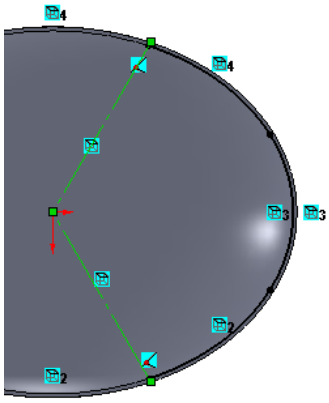
Appearance

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>

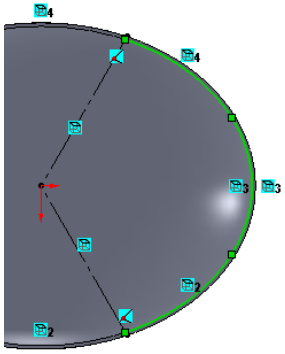


Using ***Convert Entities*** convert the lines shown to the new sketch and change their properties to ***for construction.***

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.

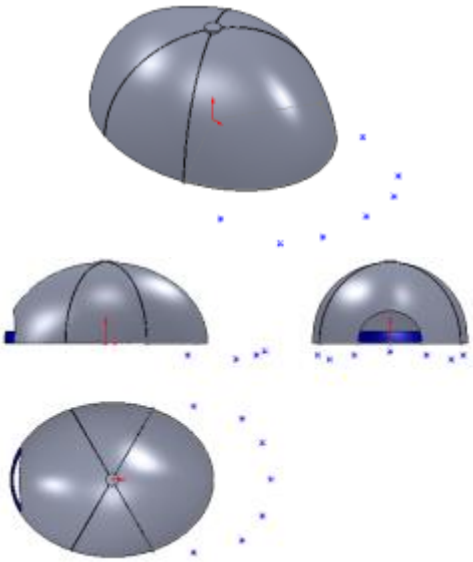
Confirm the sketch.

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

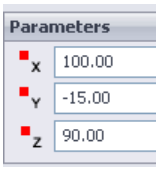
To apply the coordinates to the points we will use the values outlined in the table across.

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.

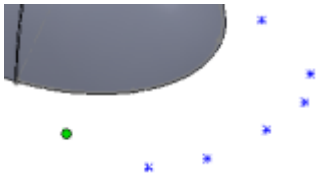
Select the first point by left clicking on it.

Fill in the values for the first point from the table.



Use the TAB key to move between the XYZ value boxes.

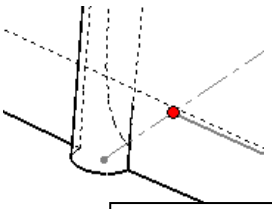
Repeat the process for each of the points.



3D Spline

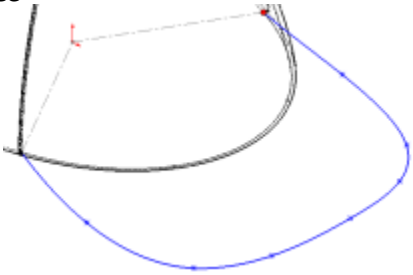
Select ***spline*** from the ***Sketch*** menu.

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.***

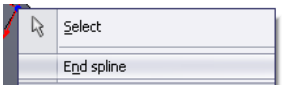


Wireframe used for visibility

Continue to join up the points with the ***spline*** tool taking care to complete the curve in one go.



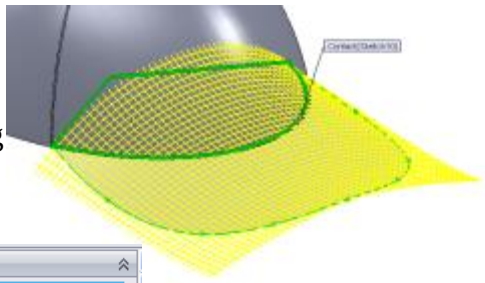
Finish the ***spline*** (Right click and choose end spline) at the intersection of the other construction line and elliptical curve of ***Sketch 10.***



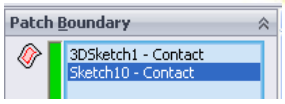
Confirm the sketch.

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***.



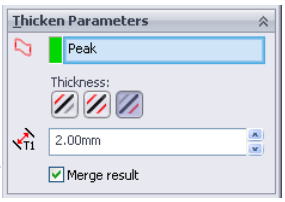
Select **OK.**

We have now created the under-surface of the baseball cap peak.

Rename Feature
Surface Thicken

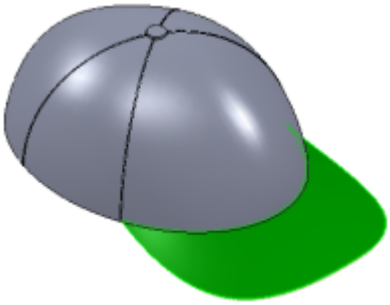
Rename the feature as ***Peak.***
To give the ***Peak*** some depth we will use ***Thicken*** from the ***surfaces*** menu.

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.***

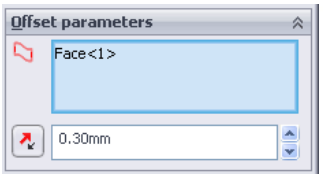
Nike Logo

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.***

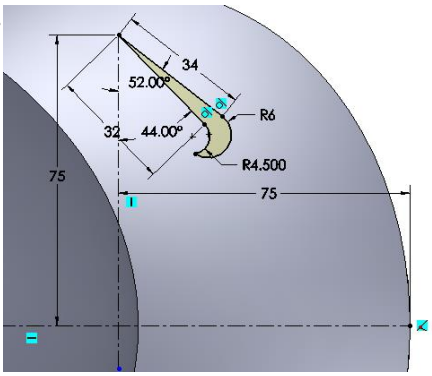


Set the ***thickness*** to ***0.3mm.***

Select **OK.**

Sketch

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



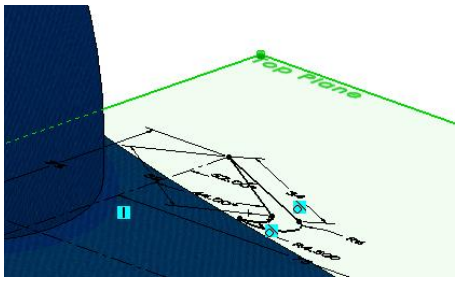
Smart dimension the sketch as shown. Note the automatic relations.

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.***

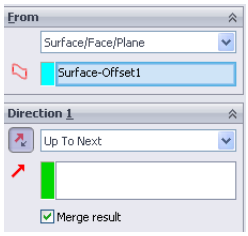
Extrude

Select ***Extrude boss/base*** from the ***features*** menu.



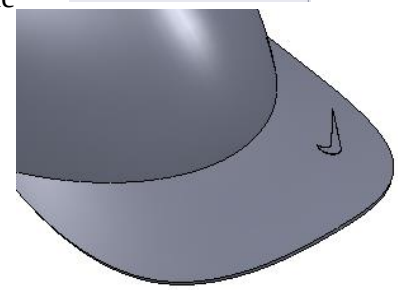
Select ***sketch 11*** as the sketch to use for the extrusion.
Make the following selections:

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***.

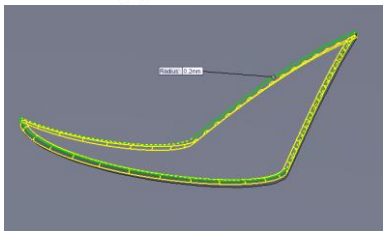
Select **OK.**

Rename Feature

Rename the feature as ***Nike Logo.***

Hide

Hide ***Surface-Offset1.***



Fillet

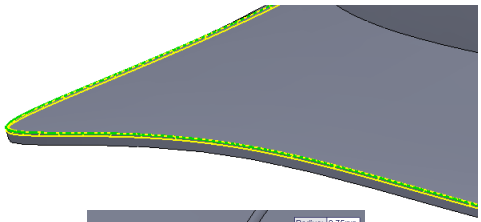
Apply a ***fillet of 0.2mm*** to the top edges of the logo.

Rename Feature

Rename the feature as ***Logo Fillet.***

Fillet

Apply a ***fillet*** of ***0.5mm*** to the top edge of the ***peak.***

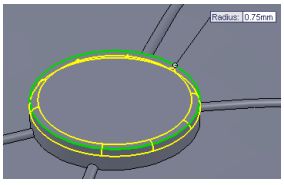


Rename Feature

Rename the feature as ***Peak Fillet.***

Fillet

Apply a ***fillet*** of ***0.75*** to the top edge of the ***button.***

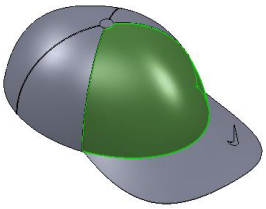


Rename Feature

Rename the feature as ***Button Fillet.***

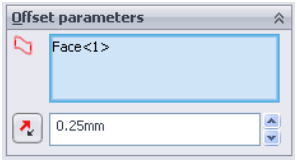
Surface Offset

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.

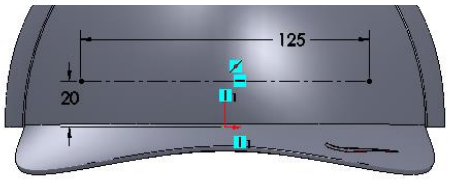


Set the offset distance to ***0.25mm.***

Select **OK.**

Sketch

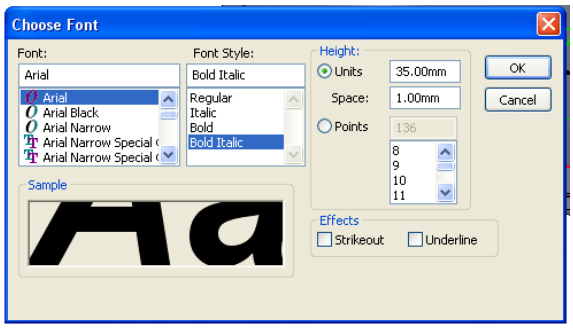
Create the sketch shown on the ***Right Plane.*** Apply a ***vertical*** relation between the ***midpoint*** of the ***centreline*** and the ***origin.***



Smart dimension as shown.

Text

Choose ***Text*** from the ***sketch*** menu.



Make the following selections:

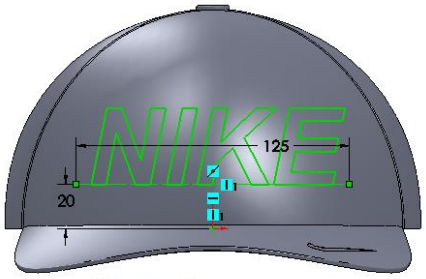
Choose the centreline as the line for the text to follow.

Select Arial as the Font.

Set the Font Style to Bold Italic.

Set the units to 35mm.

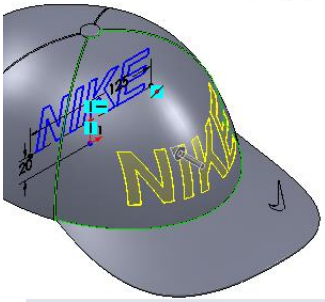
Select **OK.**



Confirm the sketch.

Extrude

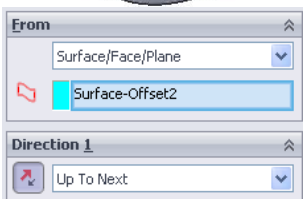
Select ***Extrude boss/base*** from the ***features*** menu.



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***.

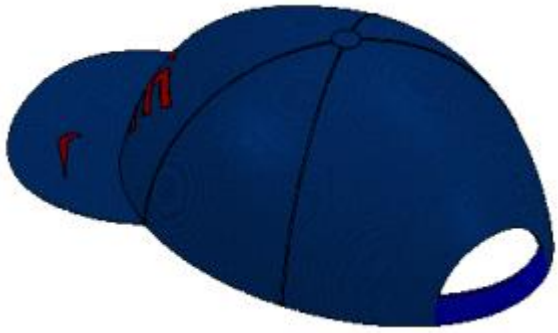
Select **OK.**

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.