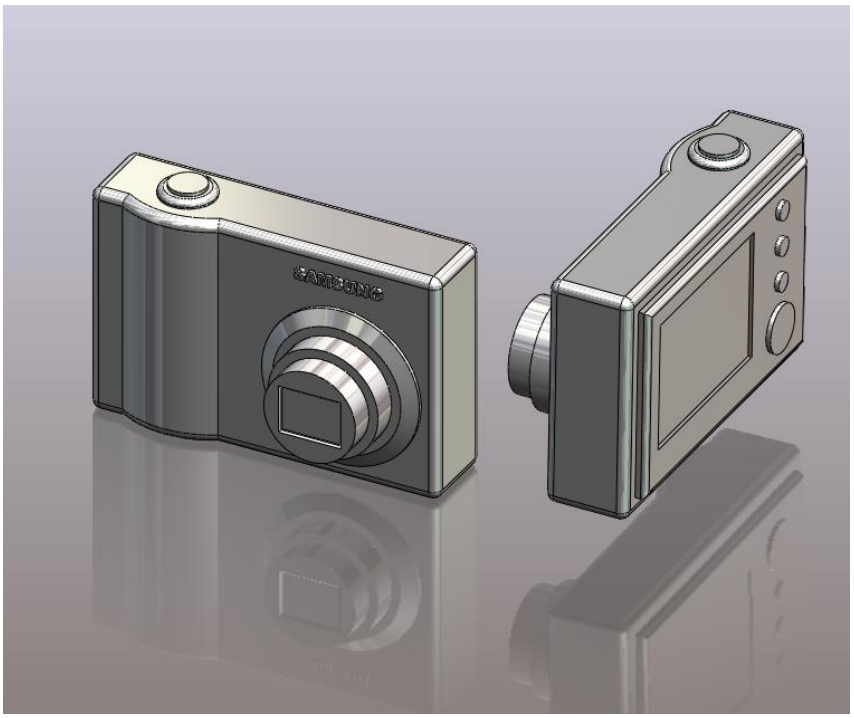
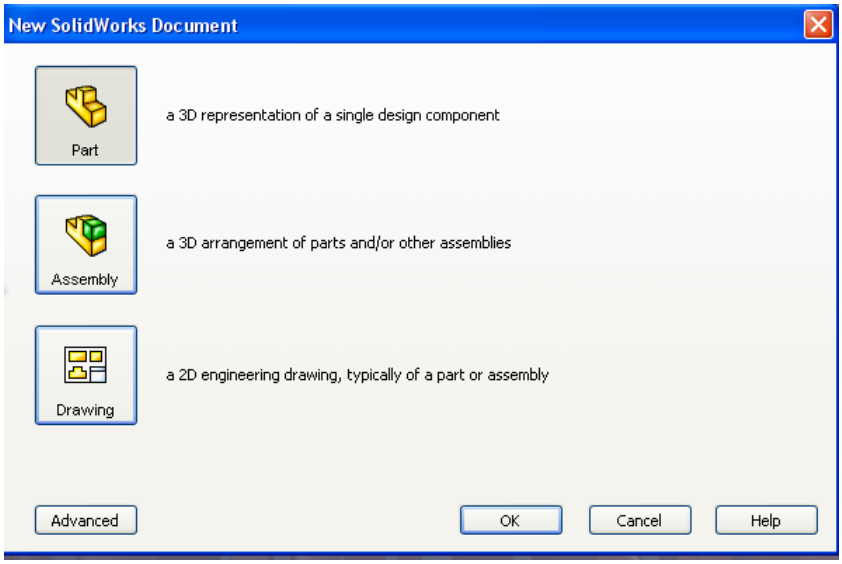
Digital Camera Exercise



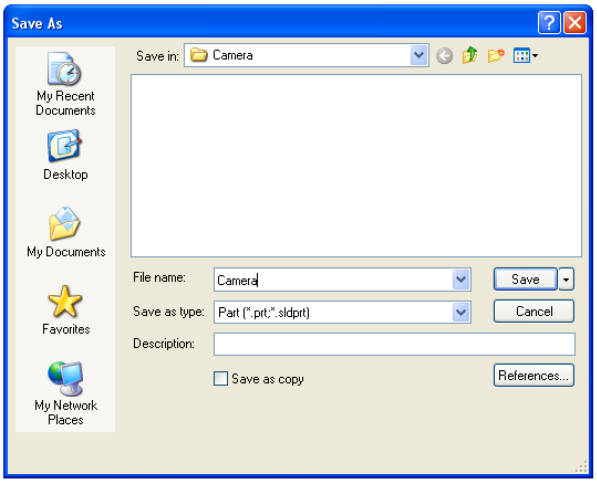
Commands Used This lesson includes **Sketching,** ***Extruded Boss/Base, Extruded Cut, Fillet, Chamfer and Text.***

New Part Click **File, New** on the standard toolbar. Select **Part** from the **New SolidWorks Document** dialog box. Select OK.



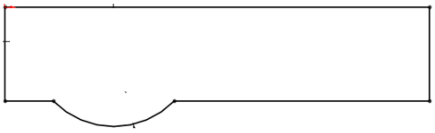
Saving the Part

Select **File, Save as** on the standard toolbar. Save the part in your chosen location as *camera.* A part is identified by its extension **.sldprt* . It is recognised as good practice that a new folder would be used for each project created. **Continue to save periodically throughout the exercise**

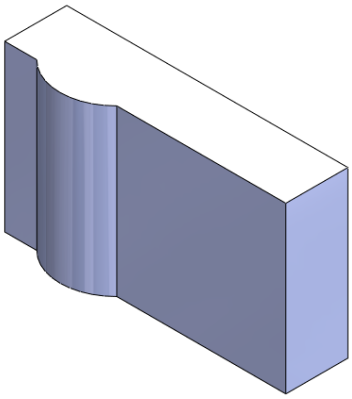


Where to start?

The first feature of the part to be created is the body of the camera. This will be an extruded feature based on a sketch.



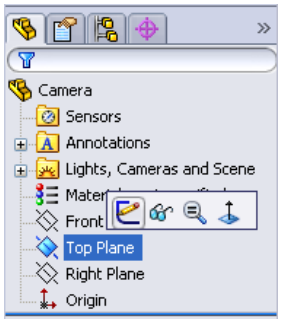
Sketch to generate the feature



Extruded Feature

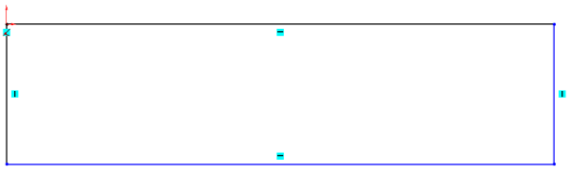
Getting started

Select the **top plane** from the feature manager the sketch command will appear. The selected plane will rotate to a normal to view and the origin will be displayed.

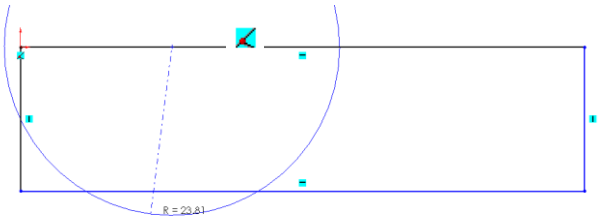


Creating a sketch

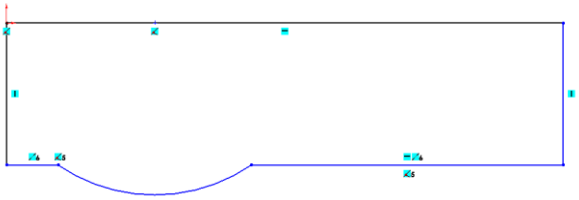
Using the **Rectangle** command, create a sketch, from the origin as shown, approximately 100mm x 25mm. Ensure that all lines are either horizontal or vertical indicated by the relations.



Select the **Circle** command from the sketch toolbar. Choose a point coincident with the back line as the centre and drag as shown below, click to form the circle.

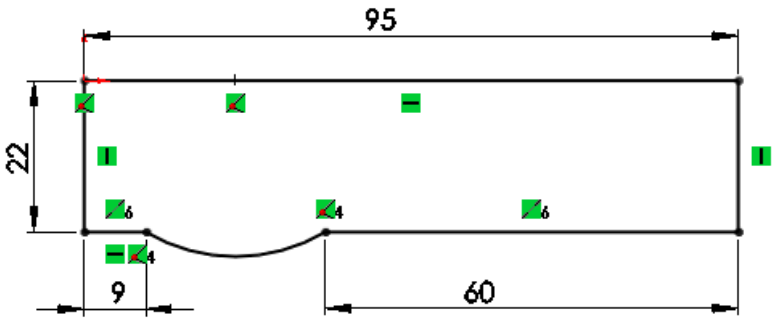


Using the **Trim Entities** command, select the areas of the circle and the front edge to be removed.



Dimensioning the sketch

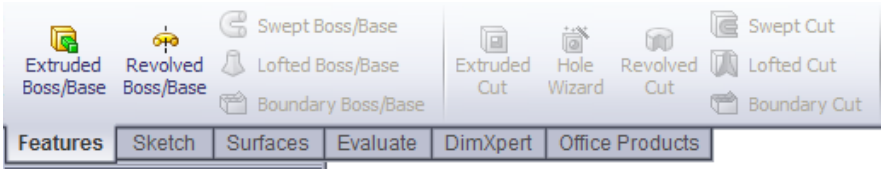
Select **Smart Dimension** from the sketch toolbar and dimension sketch as shown below.



Note – The sketch changes from blue to black when it is fully defined. To exit the sketch, select the sketch tool on the confirmation corner, sketch will be saved. Selecting X will discard changes made.

Creating the feature

Select **Features** from the **Command Manager.** The **Features** toolbar has now replaced the **Sketch** toolbar along the top of the screen

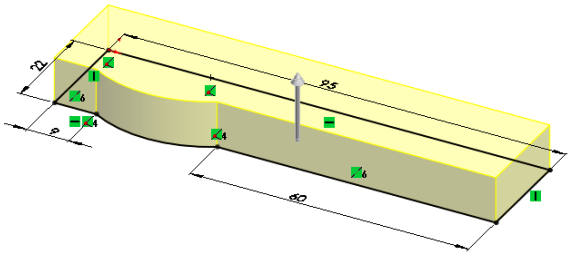


Choose **Extruded Boss/Base,** the sketch rotates to a trimetric view with a preview of the proposed extrude.

Extrude Feature Settings

End Condition = **Blind**

Depth = **62mm**

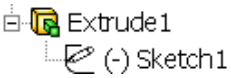


Click **OK** button to create the feature.

Alternatively select the from the confirmation corner

Completed feature

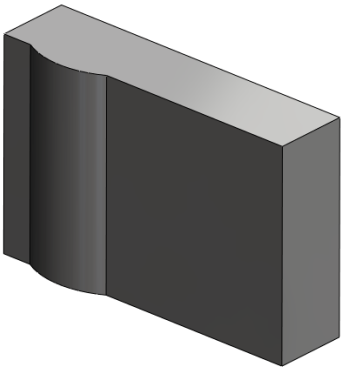
This is the first completed feature of the part. The sketch has been absorbed into the *EXTRUDE 1* feature in the **Feature Manager.**



Renaming a feature

Select the feature in the **Feature Manager Tree.** Press F2.

The feature name will be highlighted with a flashing cursor on the right hand side. Type the new name to replace it. (Camera Body)

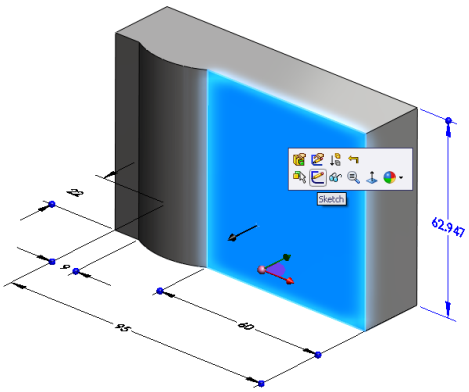


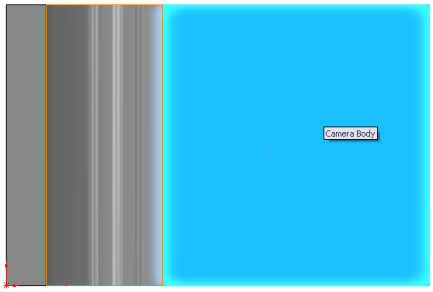
Adding the camera lens

Sketching on a face

Select **Sketch** on the **Command Manager. Sketch** toolbar replaces the feature toolbar at the top of the screen.

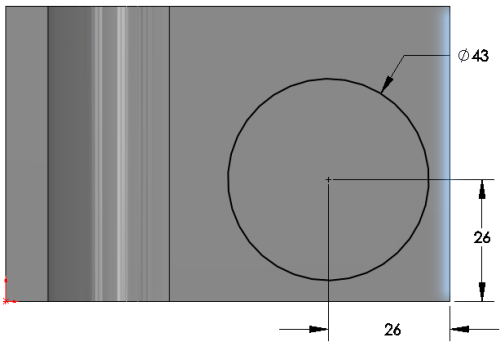
Select the front face of the camera and select sketch.
Select Normal to from the Heads Up Toolbar





Creating the sketch

Sketch a circle on the face and dimension as shown.



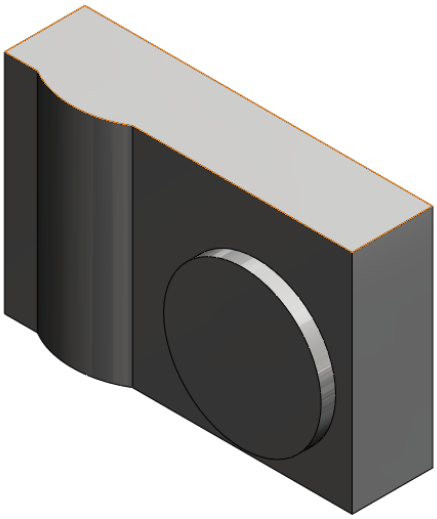
Creating the cylindrical feature

Select **Features, Extruded Boss/Base.** Choose **Isometric View** from the **Head Up** Toolbar

Extrude Feature Settings

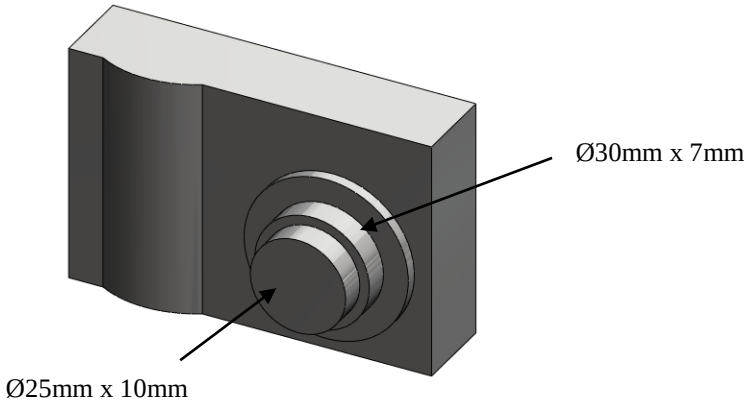
End Condition = **Blind**

Depth = **4mm**



Creating concentric features

Use the same procedure to create two further concentric cylindrical features as shown below.



To ensure concentricity of the three cylinders

When sketching the circle on the face of the existing cylinder move the cursor over the edge as shown in fig. 1, without clicking, to display the centre of the face. Move the cursor back to the centre, select the centre point and drag the radius, as in fig. 2.

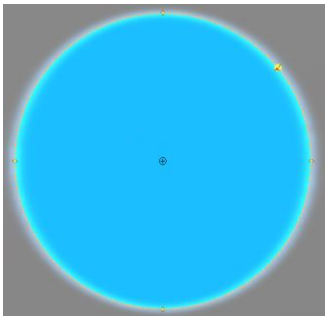


Fig.1 – Using the cursor to determine the face centre

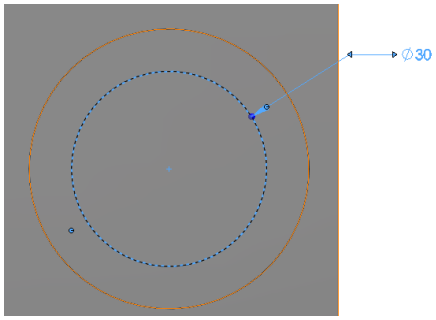
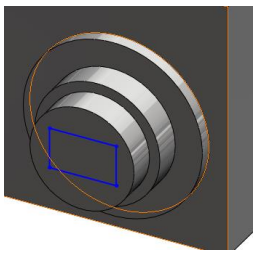


Fig.2 – Move the cursor to the centre, select and drag the radius

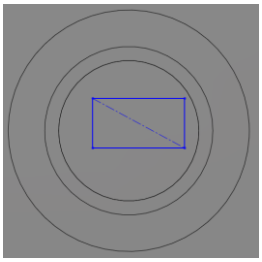
Using Extruded Cut

To remove the shutter from the front of the lens, a sketch is generated on the face having the desired profile. This is then cut from the solid to the required depth using **Extruded Cut.**



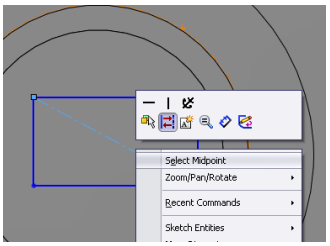
Creating the shutter sketch.

Create a sketch on the front face of the lens and choose **Normal To** view from the view toolbar. Sketch a rectangle as shown and add in a diagonal centre line.



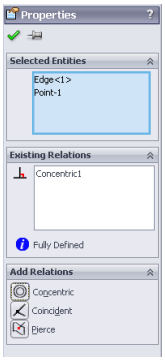
Adding a concentric relation

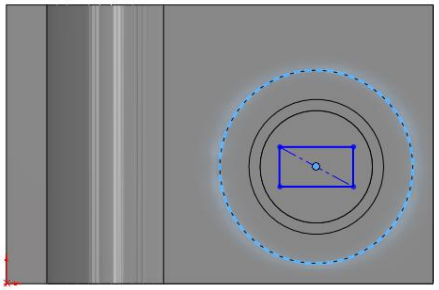
To ensure that the shutter remains in the centre, a relation must be created between it and the circle.



Right click on the centreline and choose **Select Midpoint**. Holding down the control key choose the circumference of the circle.
On the left hand side choose **Concentric** from the **Add Relations** dialog box shown below.

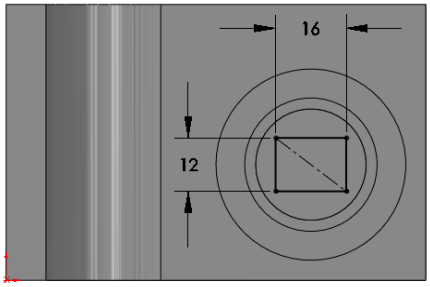
Select to close the dialog box. The rectangle will move to the centre of the circular face.





Adding dimensions to fully define a sketch.

Because the sketch is **blue** it is not yet full defined. Using **Smart Dimension** define the sketch as shown opposite.



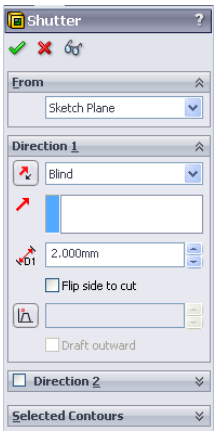
The sketch now turns black and is fully defined.

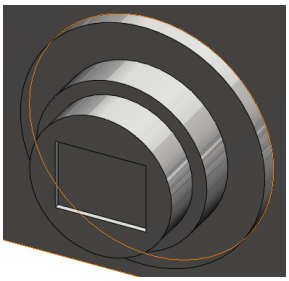
Exit the sketch using the **Confirmation Corner.**

Select a 3-dimensional view from the Heads Up Toolbar

Creating the Extruded Cut feature.

From the features toolbar choose **Extruded Cut.** When prompted select the rectangle as the sketch for extrusion.





The Cut Extrude dialog box appears on the left with a preview of the extrusion.

Cut Extrude Feature Settings

End Condition = **Blind**
Depth = **2mm**

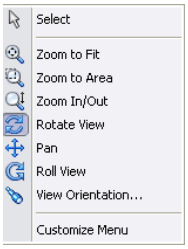
Click **OK** button to create the feature.

Adding the raised profile on the back

To create the raised profile on the back of the camera, a rectangular sketch is required, positioned on the back face.

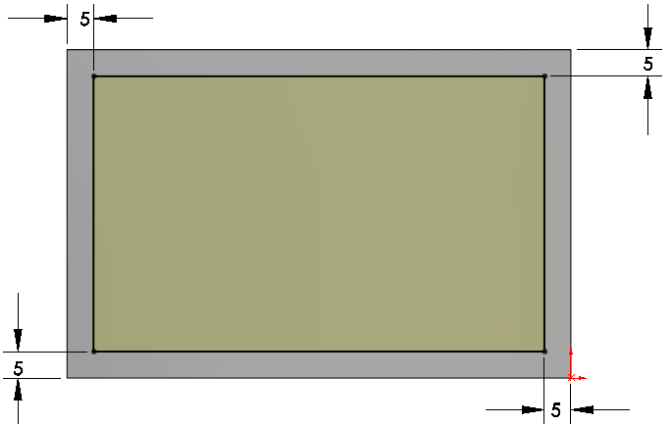
Rotate View

It is necessary to rotate the model to choose the back face as the surface on which to create the sketch. Within the graphics area click the Right hand mouse button and select the rotate command. Rotate the model until the back surface is exposed.



Sketching

Create a sketch and choose the back surface as the plane on which to build the sketch. Sketch a rectangle on the back face and dimension as shown.
Exit the sketch



Dimensioning

Should the overall size of the camera change the raised panel will remain offset 5mm from the altered dimension.

Creating the extruded panel feature.

Choose **Extruded Boss/Base** from the features toolbar
Select the rectangle as the sketch to create the feature

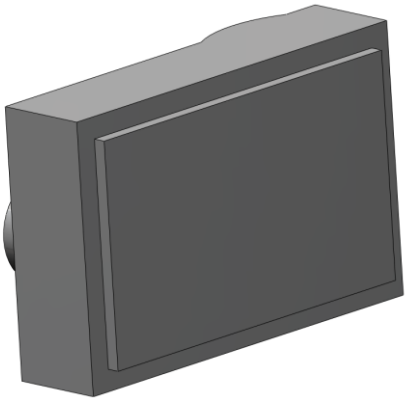
Extrude Feature Settings

End Condition = **Blind**

Depth = **3mm**

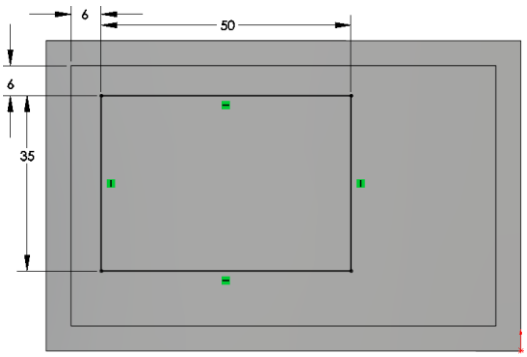
Rotate to a 3 dimensional view to ensure the extrude direction is correct

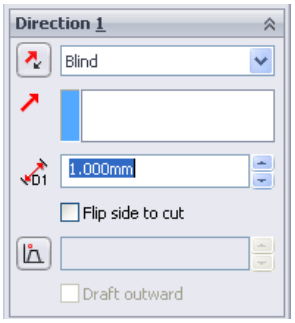
Click **OK** button to create the feature



Removing the screen

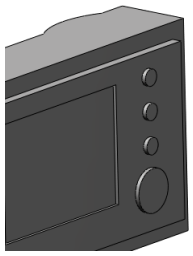
Similar to the shutter, the screen is produced using an **Extruded Cut.** The face of the extruded rear panel is used as the sketch plane. Sketch dimensions are shown below along with feature settings





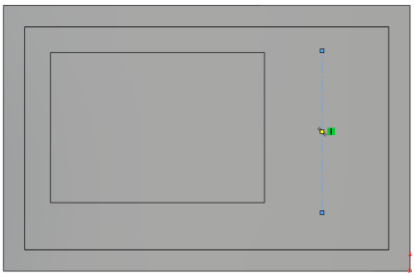
Inserting the back function buttons.

The function buttons will be added as an **Extruded Boss/Base** feature
Create a new sketch and choose the face of the extruded panel as the sketch plane. Select a **Normal To** view from the View toolbar.



To ensure that all of the buttons remain vertically aligned, a vertical centre line will be added and the centres of the circles will be made coincident with it.

Choose **Centerline** from the sketch toolbar and add in a centerline as shown.

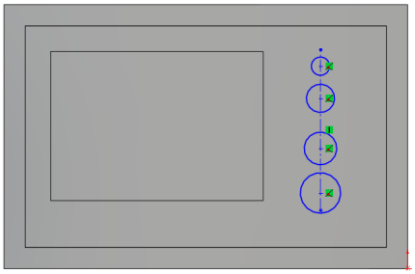


The **vertical relation symbol** indicates that the line is vertical.

Adding the sketch

Sketch the four circles as shown ensuring that each centre is coincident with the vertical centreline.

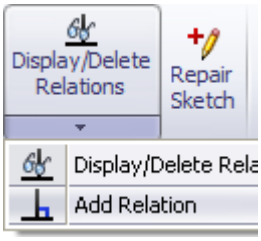
Diameter sizes are irrevelent at this stage.

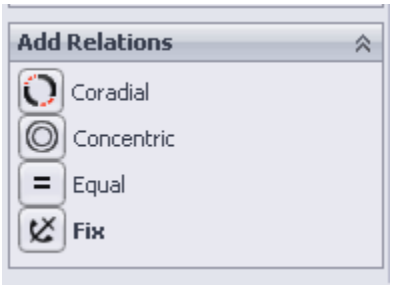


Adding an equals relation

In order to make the three smaller circles equal radii it is necessary to add an **Equal Relation.**
Select Add Relation command and select the top three circles

In the Add Relations dialog box select **Equal**



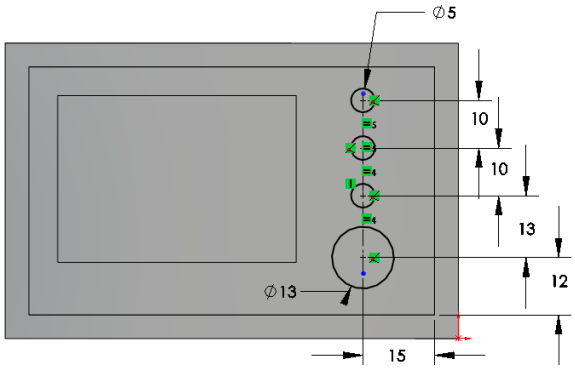


Adding dimensions

Dimension the sketch using **Smart Dimension.**

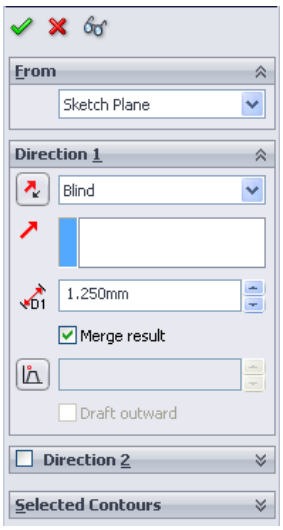
When the sketch entities turn black, the sketch is fully defined.

Exit the sketch



Creating the feature

Extrude the feature using the feature settings below.



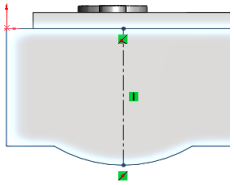
Creating the Capture Button.

This button will be created with two concentric extrusions similar to the construction of the front lens.

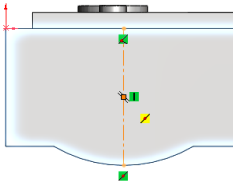
Sketching

Create a sketch using the top surface. Select **Normal To** view from the Heads Up toolbar.

Sketch a centre line with the **Sketch Relations** shown.



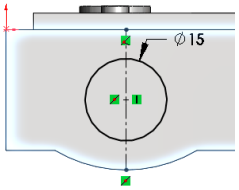
Sketch a circle with its centre coincident with the midpoint of the centerline and **Smart Dimension** as shown



Fully Defined

The sketch has turned black!
Why is the sketch **Fully Defined** with the addition of just one dimension?

Because of the inclusion of the **Sketch Relations**

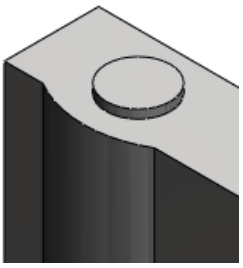


Change Orientation

Choose **Trimetric View** from the view toolbar

Extrude Boss/Base

Extrude the sketch to a depth of **2mm**



Capture Button

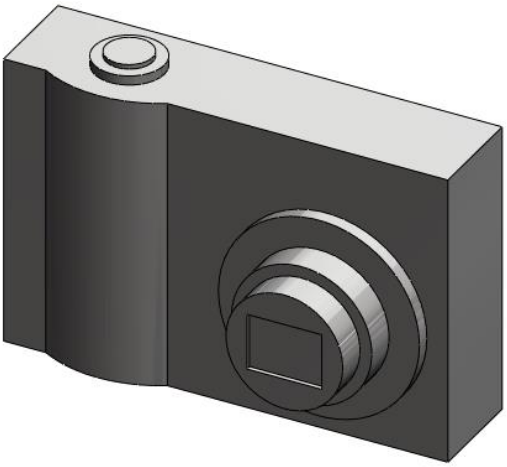
To complete the capture button a further sketch and extrusion will be required.

Create a sketch on the top face of the existing cylinder. Sketch a circle of **Ø10mm**.

Ensure that the circle is concentric to the face by selecting the face centre as before.

Extrude Boss/Base

Extrude the sketch to a **depth** of **1mm**.

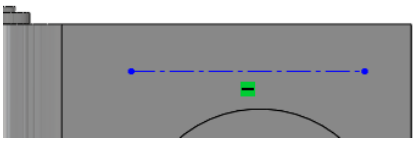


Introducing Text

The **Text** tool allows you to insert text into a sketch and extrude as a boss/base or cut feature.
Construction lines must be added to guide the text

Text Sketch

Create a sketch on the front face and add the centreline as shown



Adding text

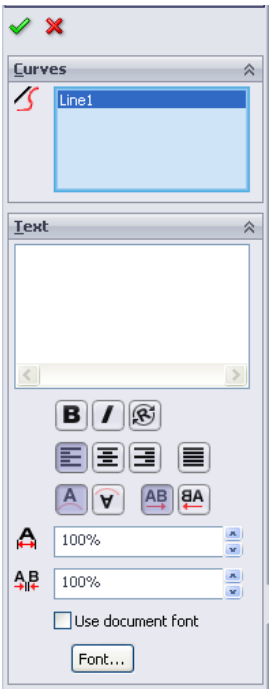
Select **text**  from the **sketch** toolbar.

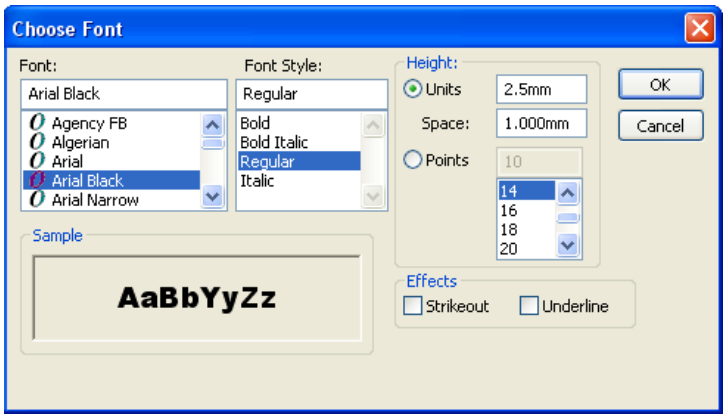
Select the centerline as the text guide 

Type SAMSUNG into the text window

Use Document Font will be selected by default. Deselect by clicking on the tick  next to it. **Font** is no longer greyed out.

Select **Font.**
Choose **Arial Black, Regular, Size 2.5mm**
Select **OK**





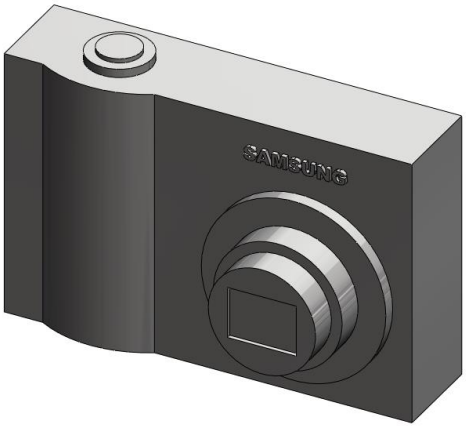
Select  to exit the command.

Changing text position

The length and position of the line, and hence the position of the text, may be varied by holding down the left mouse button on the endpoint of the line and dragging the line.

Creating the text feature

Extrude a boss with a **Depth** of 1mm.



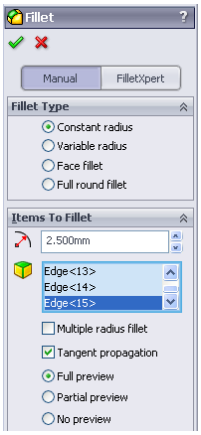
Introducing Fillet

Fillets are generally added to the solid rather than the sketch and are hence referred to as **applied features.**

Where to find it

Select the **Fillet tool** 

from the features toolbar



Insert Fillet

Select the **Fillet** option. The fillet options appear in the property manager.

Set the **Radius** value to **2.5mm**

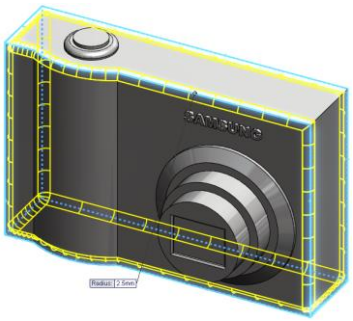
Select **Full Preview**

Edge Selection

The edges will highlight red as the cursor moves over them and appear green as they are being selected.

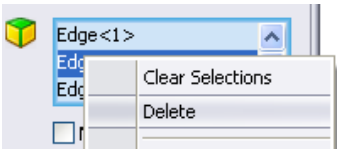
Select the edges shown and click **OK**

It may be necessary to rotate the solid to select all edges.



Removing edges from selection

Should an incorrect edge be chosen, right click on the selection name in the fillet dialog box and choose **Delete.** That edge will be removed from the list. **Clear Selections** will delete all.



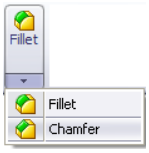
Further fillet

Create a further **1.5mm** fillet to the edge of the rear panel and the rim of the capture button as shown.

Introducing Chamfer

Similar to **Fillet, Chamfer** is an **Applied Feature**. Chamfer creates a bevel on one or more edges. A chamfer may be defined by two distances or a distance and an angle.

Where to find it

Select the **Chamfer** tool 

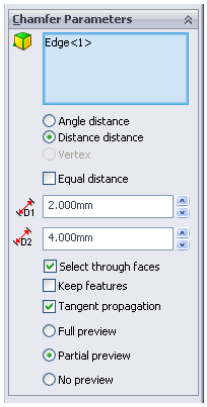
from the features toolbar

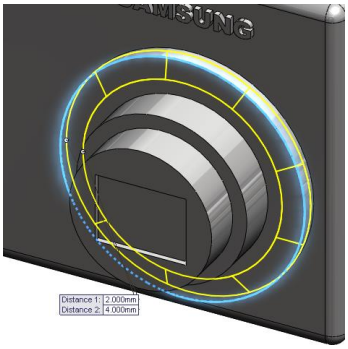
Insert Chamfer

Select the **Chamfer** option. The chamfer options appear in the **Chamfer Property Manager.**

Add a **Chamfer** using the edge of the lens as shown. Set the distances using the values shown on the right

Select OK.



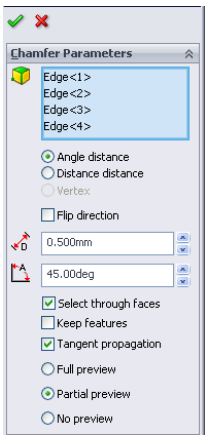


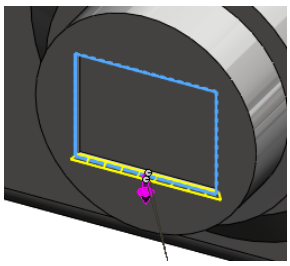
Chamfer distance/angle

Select **Chamfer.**

Add a **Chamfer** using the edge of the shutter as shown. Set the options as shown opposite

Select OK.





Exercise Complete!