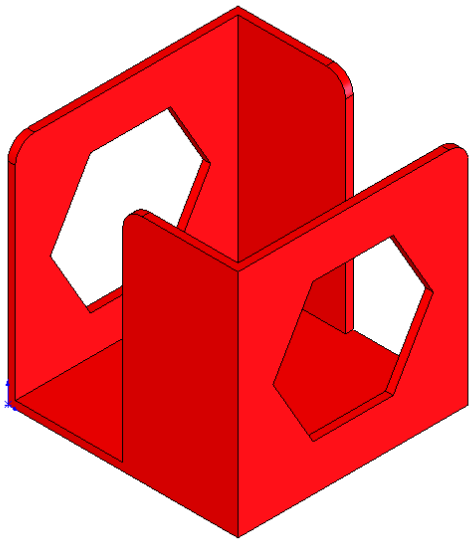
Memo Block

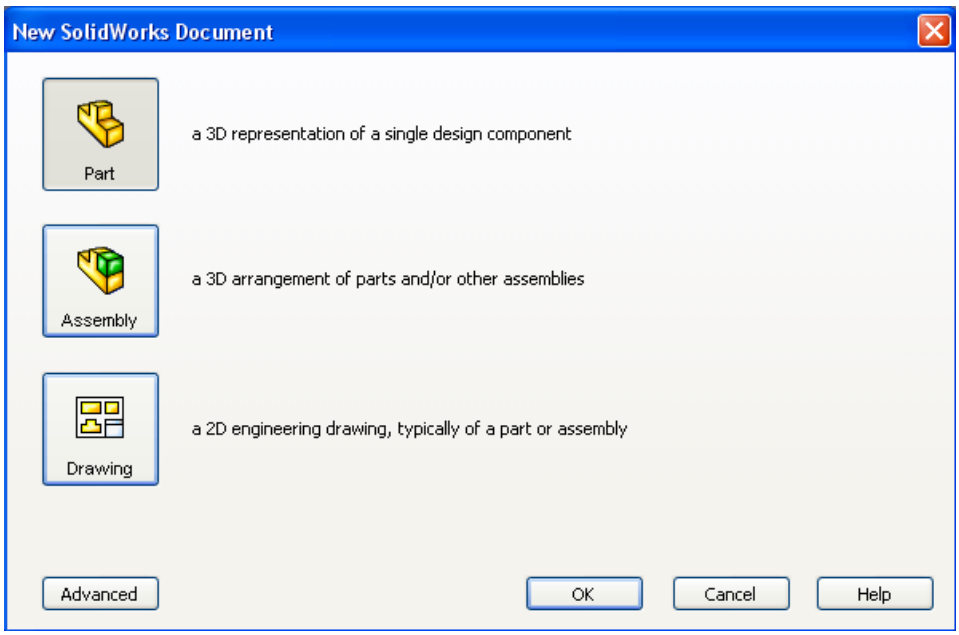


Commands Used

This lesson includes the commands **Sketch, Extruded Boss/Base, Extruded Cut, Shell, Polygon and Fillet.**

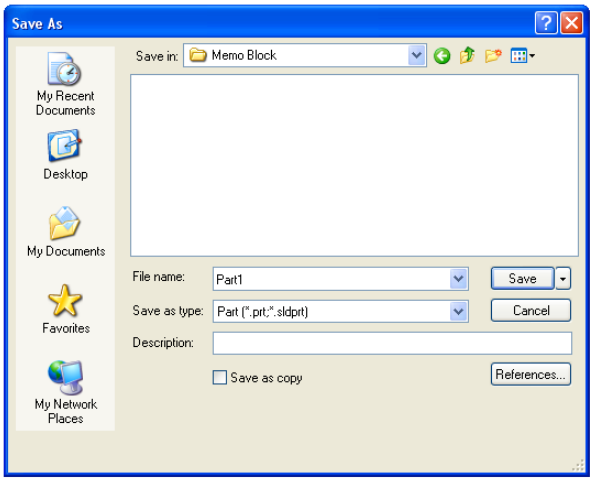
New Part

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



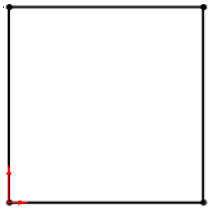
Saving the Part

Select **File, Save as** on the standard toolbar. Save the part in your chosen location as *Memo Block.* 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**

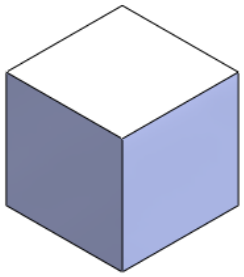


Where to start?

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



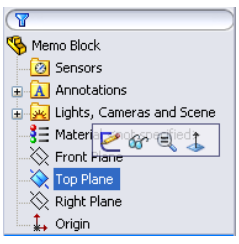
Sketch to generate feature

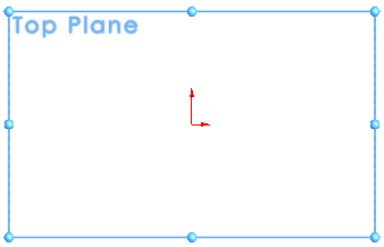


Extruded feature

Getting started
Choosing a plane

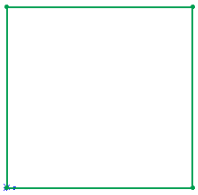
Select the **top plane** from the feature manager the sketch command will appear. The selected plane will rotate to a normal to view and the origin will be displayed.



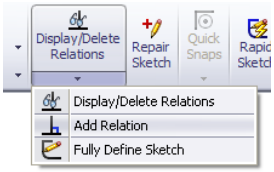


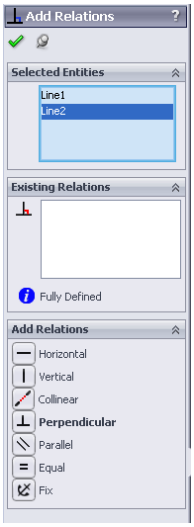
Creating a sketch

Using the **Rectangle**  command, create a sketch approximately 60mm x 60mm. Ensure that all lines are either horizontal or vertical indicated by the relations. It is good practice when creating a part to follow the sequence **sketch**, **relation** and **dimension.**



Select the **Add Relation** command from the sketch toolbar. Select two lines and choose *Equal* in the add relation property manager.



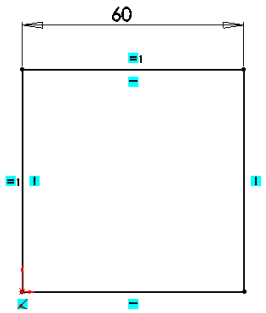


Dimensioning the Sketch

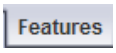
Select **Smart Dimension**  from the sketch toolbar and dimension sketch as shown. Note the relation is maintained a square is created

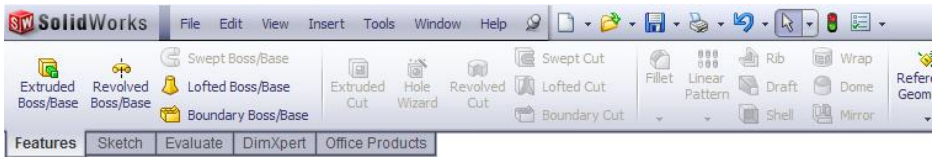
Note – The sketch will change from blue to black when it is **fully defined**. To exit the sketch, select the sketch tool on the confirmation corner, the sketch will be saved.

Selecting X will discard changes made.

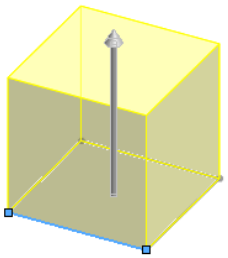


Creating the feature

Select **Features tab**  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. Go to the extruded boss/base feature property manager.



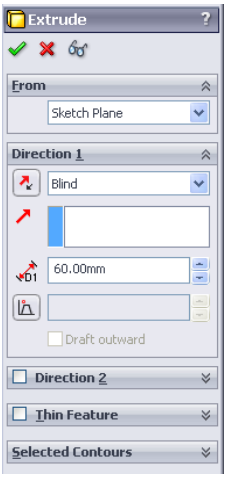
Extrude Feature Settings

End Condition = **Blind**

Depth = **60mm**

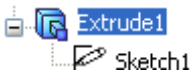
Click **OK** button  to create the feature.

Alternatively select the  from the confirmation corner.



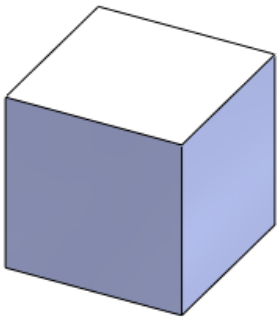
Completed feature:

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



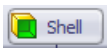
Renaming a feature:

Select the in the feature  in the feature manager tree and Press the F2 key and type the new name *cube*  to replace "Extrude1"

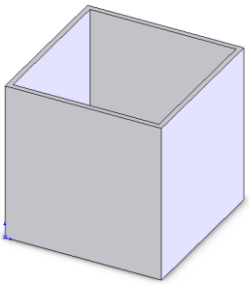


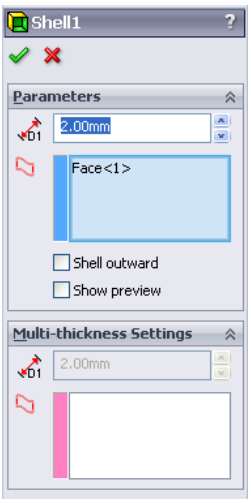
Changing to a thin walled solid

Command Shell

Select **Shell Command** 

on the **Command Manager**
Go to the shell feature property manager and enter 2mm. Select the top face of the solid
Click the **OK** button 

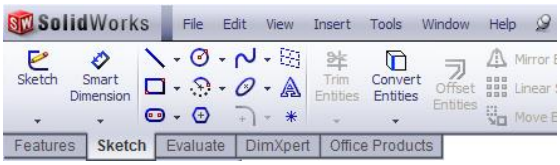




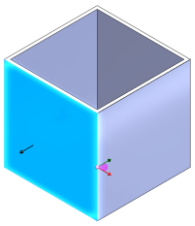
Creating the Cutouts

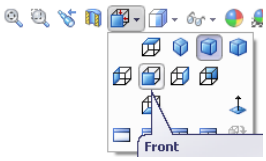
Command Sketch

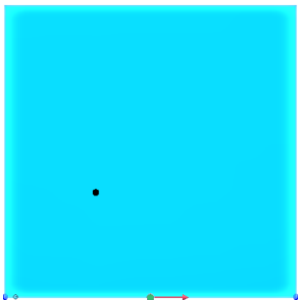
Select **Sketch** from the **Command Manager.** The **Sketch** toolbar has now replaced the **Features** toolbar.



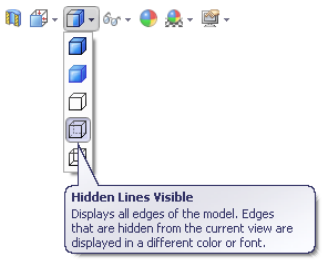
Select the face of the cube and select Front View from the View Orientation from the Heads- up toolbar
The cube will rotate to the front view.

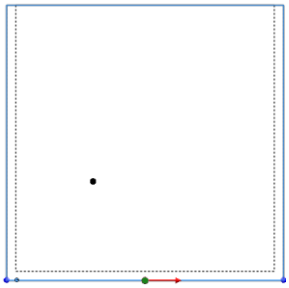




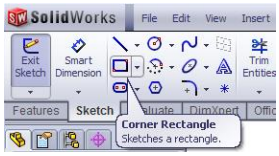


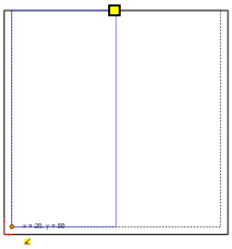
Select Hidden Lines Visible.



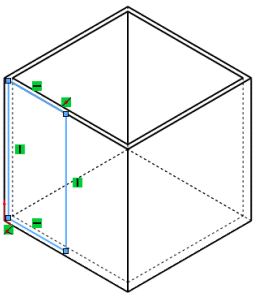


Select the Rectangle Sketch command and draw a rectangle from the top centre of the cube to the lower left hand corner, exit the sketch and then select the isometric view





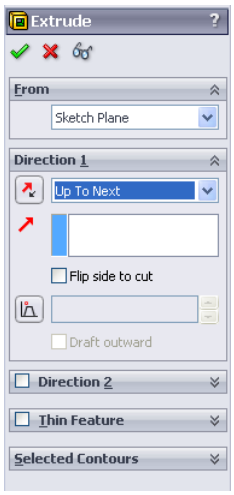
Front View



Isometric View

Creating the Extrude Cut

Extruded Cut From the feature 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

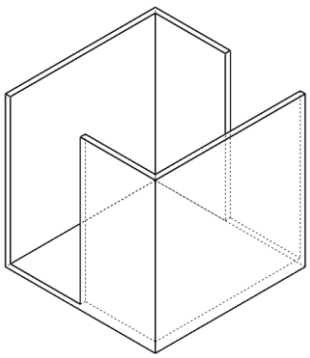


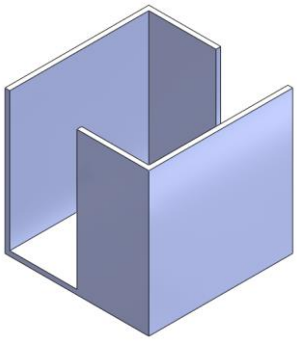
Cut Extrude Feature Settings

Direction = **Up to next**

Click OK button  to create the feature

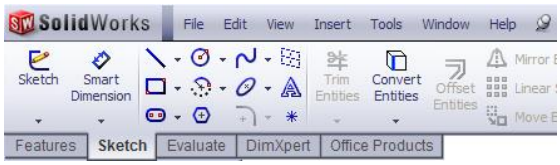
Select the Back view from the View Command repeat the **Sketch** and **Extruded Cut.**
Select shaded with edges



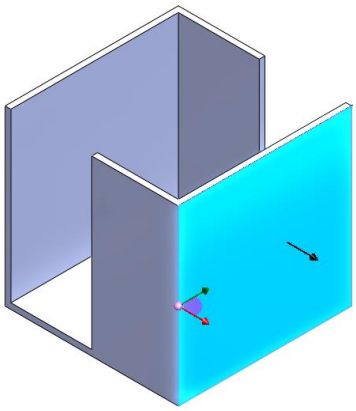


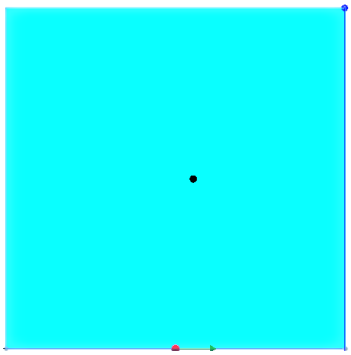
Polygon Feature
Hexagon
Sketch Command

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



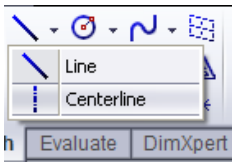
Select the face of the cube and select Right View from the Heads-up toolbar.

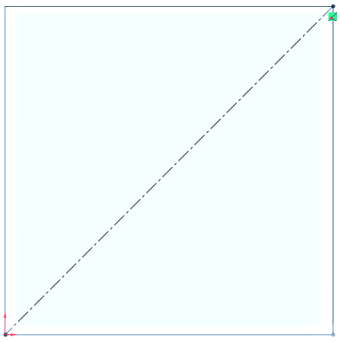




Centerline Command

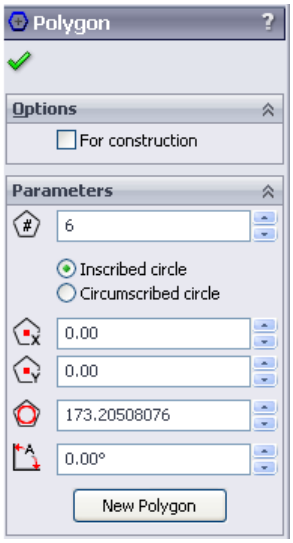
Select the Centerline sketch command draw a diagonal line from the top right to the bottom left hand side of the face.
Exit command using the **Esc** key on the keyboard.



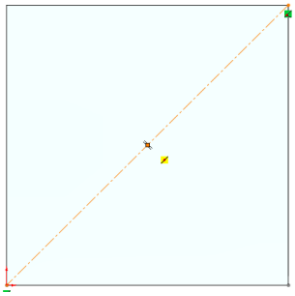


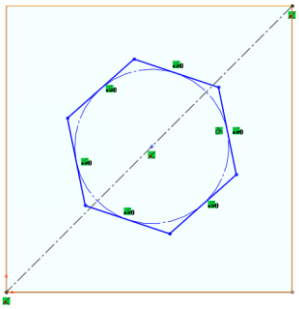
Polygon Command

Select the Polygon sketch  command make the Midpoint of the diagonal coincident with the centre of the polygon and draw any size polygon



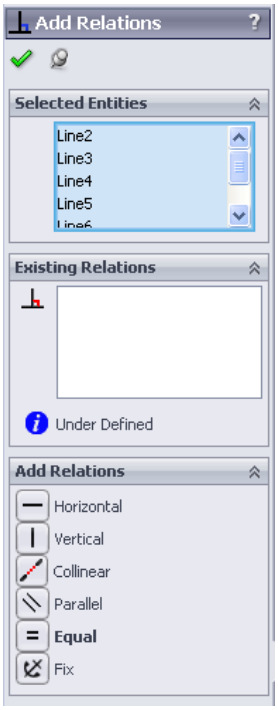
Polygon Sketch Settings
Parameters = 6

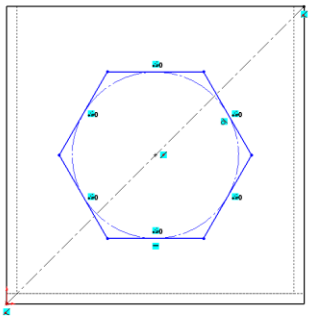


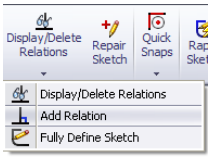


Adding the Horizontal Relation

In order to make one side of the Polygon horizontal Selection Add Relation select one side of the polygon and pick horizontal







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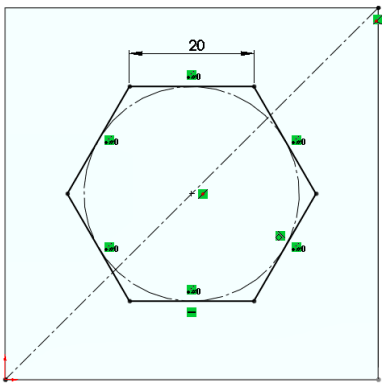
Smart Dimension Polygon

Select **Smart Dimension**  from the sketch Toolbar and dimension the side of the hexagon 20mm. Exit Sketch.

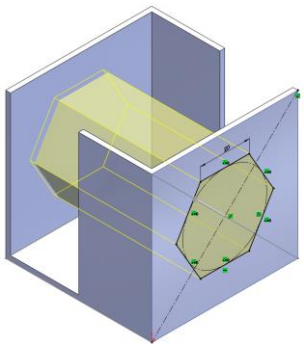


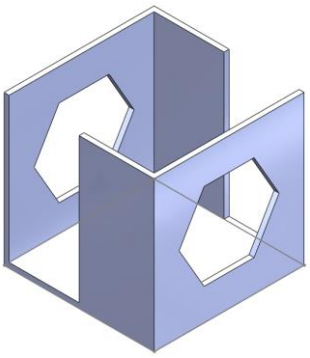
Extrude Cut

Polygon

Go to the Heads–Up Toolbar and select Isometric View exit the sketch command Select **Features** from the **Command Manager.** The **Features** toolbar has now replaced the **Sketch** toolbar along the top of the screen

Choose **Extruded Cut** select the hexagon
Extrude Cut Settings
Direction1 = Through All

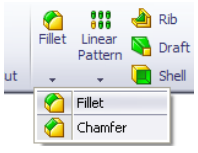


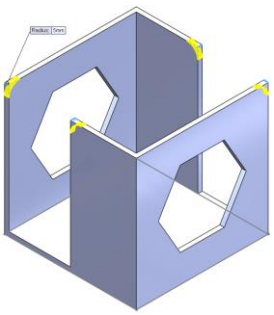


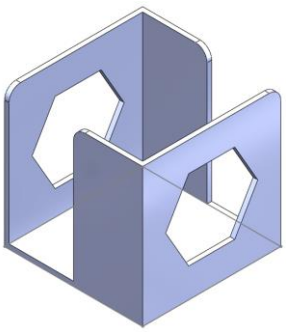
Click OK button  to create the feature

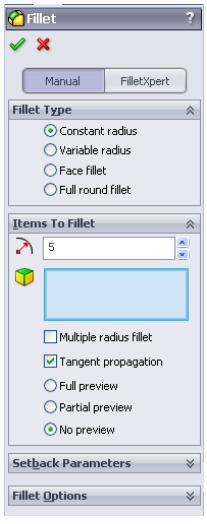
Fillet Command

Select the Fillet feature command and select the edges to fillet. Set the fillet radius to 5mm
Click OK button  to create the feature



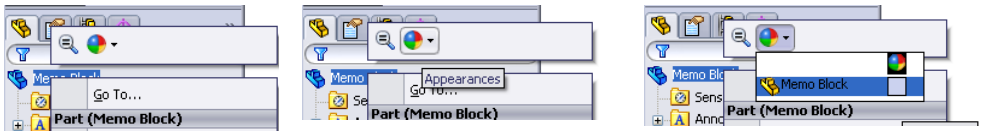




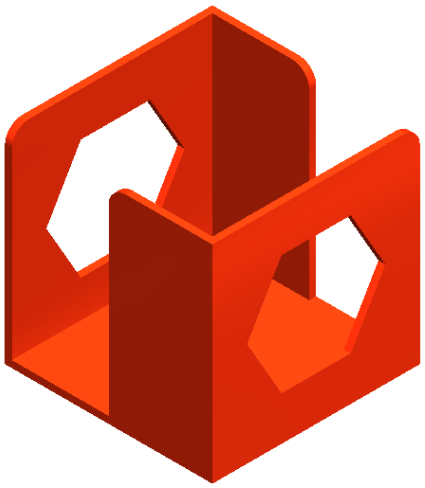


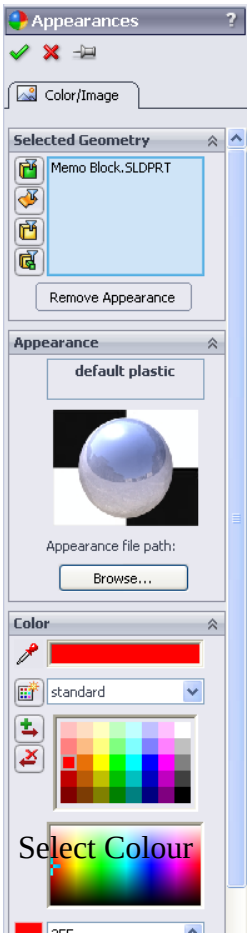
To change the appearance of the Part

Right hand click on the part name, select
Appearance, colour and select a colour click OK



Right Click Select Appearance Select Memo Block





Possible Options