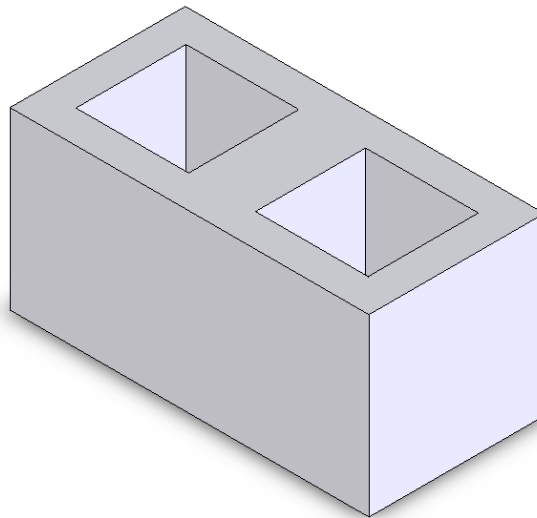


## Cavity Block Exercise



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### **Prerequisite Knowledge**

Creating a Part, Saving a Part, Creating a Sketch, Drawing Lines, Automatic Relations – Vertical, Horizontal, Coincident. Smart Dimensioning, Extruded Boss/Base (Blind End Condition).

### **Focus of the Lesson**

On completion of this exercise you will have used:

- Sketch rectangle
- Centre line
- Equal relations
- Sketch mirror
- Extruded Boss/Base
- Extruded Cut

### **Commands Used**

*Rectangle, Centreline, Add Relation, Sketch Mirror, Extruded Boss/Base and Extruded Cut.*

### **Getting Started**

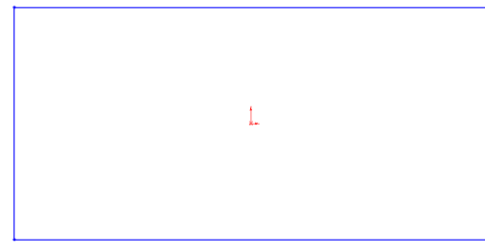
#### **New Part**

Create a new part. Save the part as *Cavity Block* in your chosen location.

### Where to start?


The first feature is the main part of the block and we will then remove the two cavities.

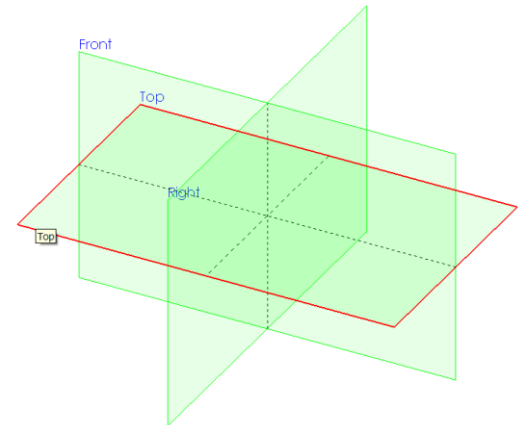
Sketch to generate the feature




### Getting started

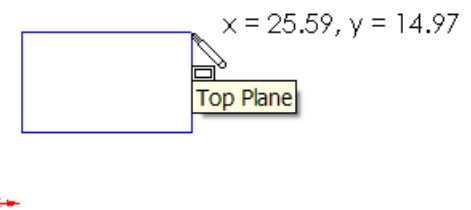
#### Choosing a plane

Select **Sketch**  from the sketch toolbar. Choose the top (horizontal) plane.




#### Creating a sketch

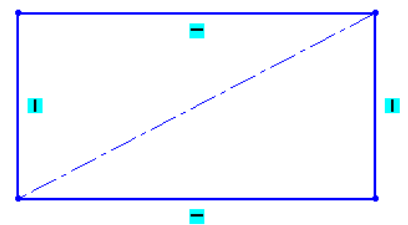
Click the **Rectangle**  and, create a sketch of the rectangle away from the origin



The rectangle will automatically show horizontal and vertical relations.

#### Centreline Command

Select the **Centreline**  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.



#### Sketch Relations


Sketch Relations are used to force a behaviour on a sketch element thereby capturing design intent. Some are automatic, others can be added as needed.

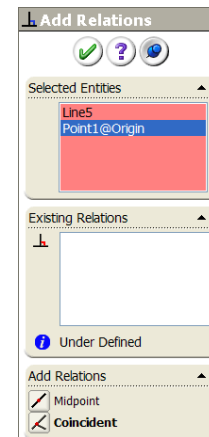
#### Automatic Sketch Relations

Automatic relations are added as geometry is sketched. We saw this in the previous exercise and when drawing the rectangle above. Horizontal and Vertical Relations were automatic.

### Added Sketch Relations

Select the **Add Relation**  command from the sketch toolbar.


Select the diagonal centreline and the origin. Choose **Midpoint** in the Add Relation property Manager, and click **OK** .

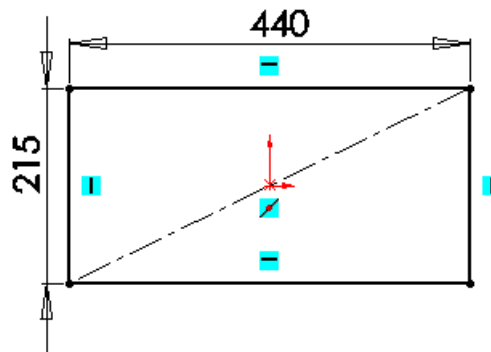


### Zoom to Fit

Press **F** to resize the view to fill the screen.

### Dimensioning the sketch

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



### Zoom to Fit

Press **F** to resize the view to fill the screen.

Exit the sketch.

### Creating the feature

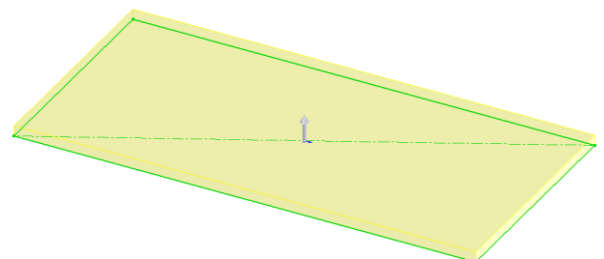
Choose **Extruded Boss/Base** from the **Features Toolbar**.

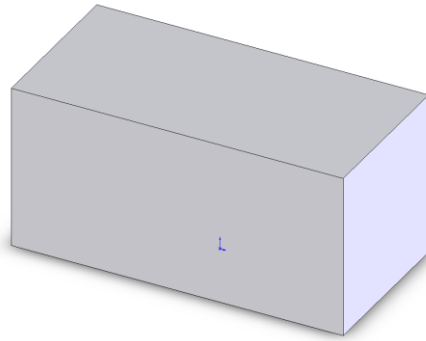
#### Extrude Feature Settings

End Condition = **Blind**

 Depth = **215mm**

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





### Renaming a feature

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



Type the new name *Block* to replace it.

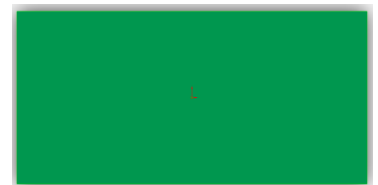
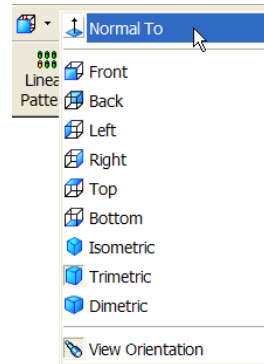
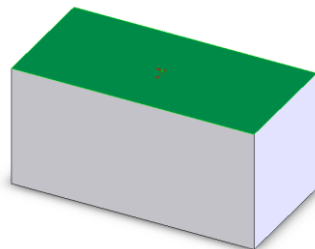
### Creating the Cavities

#### Sketching on a Face


Any face of a model may be used to generate a plane to contain a sketch.

Choose sketch from the sketch toolbar.

Select the top face of the block. Select **Normal To** from the **Standard Views** toolbar.

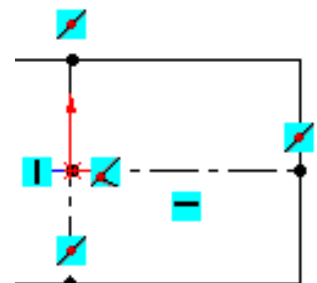
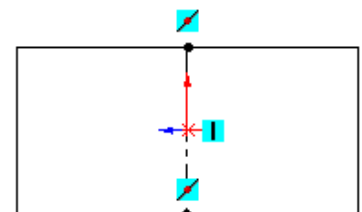


#### Centerline Command

Select the **Centerline**  sketch command. Draw a line from the midpoint of the horizontal lines.

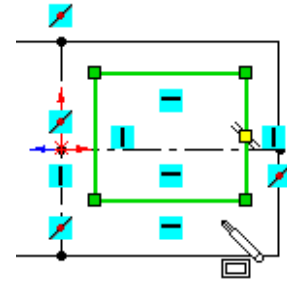
The midpoint is automatically located when the cursor hovers over the required line.

Draw a second centreline from the origin to the midpoint of one of the vertical lines.





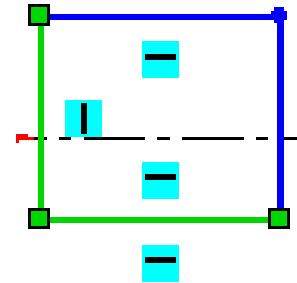
**Rectangle Command** Select the **Rectangle** Sketch command and draw a rectangle.

Press **ESC** on your keyboard

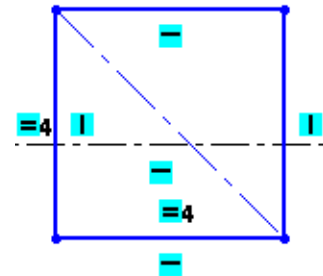


 **Adding an Equal Relation**

Select the **Add Relation**  command from the sketch toolbar. Select the two adjacent sides. Choose **Equal** in the add relation property Manager, and click **OK** . The rectangle is now a square.




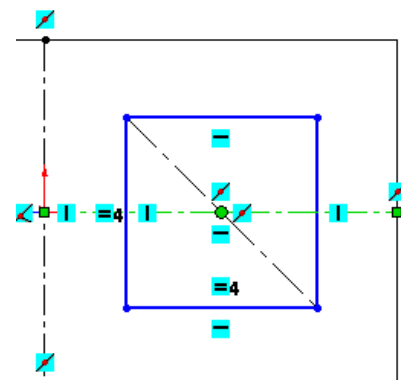
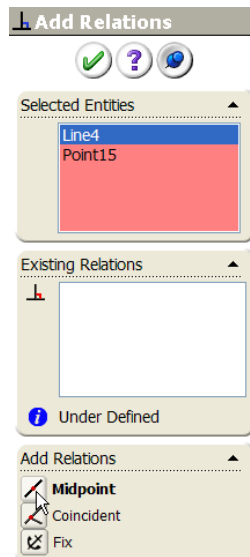
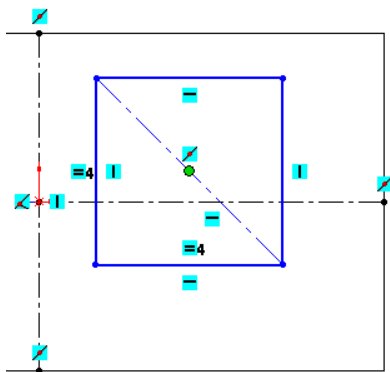
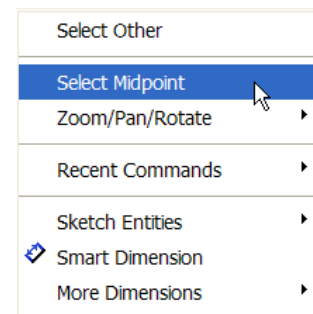
**Centerline Command** Select the **Centerline** sketch command. Draw a diagonal line from the top left to the bottom right hand side of the square.



 **Adding a Midpoint Relation**

Right click on the diagonal centreline and select **Midpoint**.

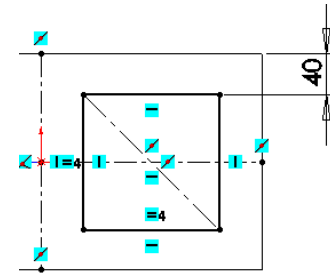
Select the **Add Relation** command. Select the horizontal centreline. Choose **Midpoint** in the add relation property Manager, and click **OK** .





### Adding dimensions to fully define a sketch

Because the sketch is **blue** it is not yet fully defined. Using **Smart Dimension**, add the 40mm dimension to fully define the sketch.

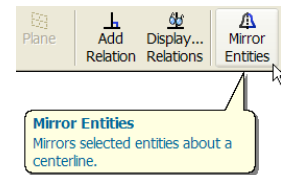


### Symmetry in the Sketch

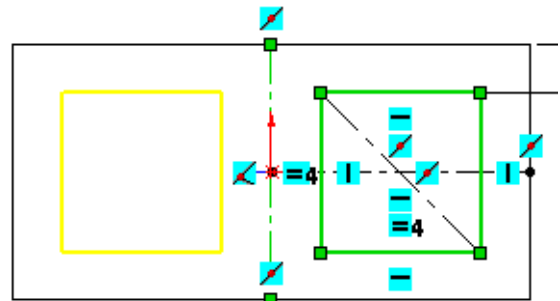
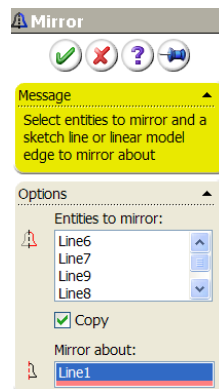
Symmetrical geometry in a sketch can be created easily using the **Mirror** command.


### Mirror command

Select the **Mirror Entities** command from the sketch toolbar.



Select the four sides of the square as the **Entities to Mirror** and select the centre line of the block as the axis to **Mirror About**.



Click **OK** button  to create the **Mirror** sketch.

Exit the sketch.

### Isometric View

Select **Isometric** from the **Standard Views** toolbar.




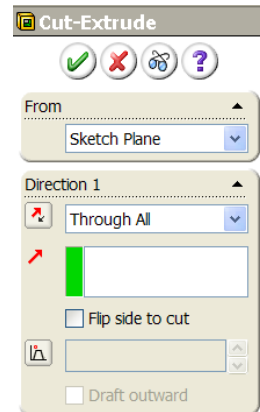
### Extruded Cut

Cuts a solid model by extruding a sketched profile in one or two directions.

Select **Features** from the **Command Manager**.



Select **Extruded Cut**  from the **Features** toolbar. The Cut Extrude dialog box appears on the left of the screen with a preview of the extrusion.



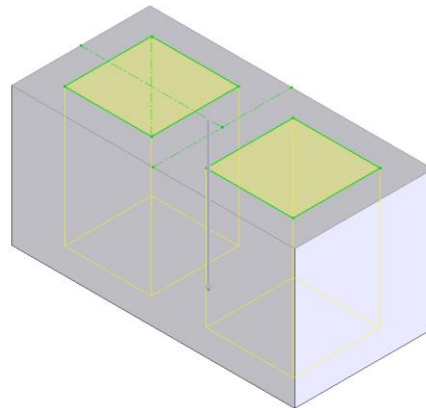
### Cut Extrude Feature Settings

End Condition = **Through All**

### Through All End Condition

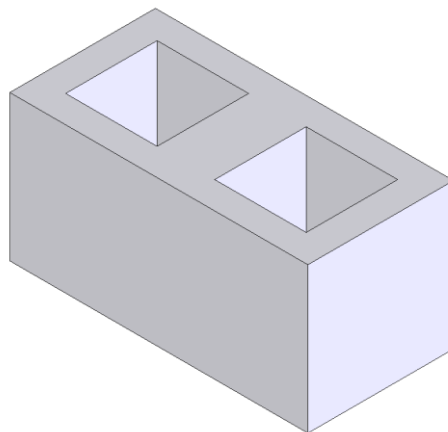
This type of end condition always cuts through the entire model no matter how far.

**Depth** is ignored because the **Extruded Cut** is **Through All**



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

### Completed feature

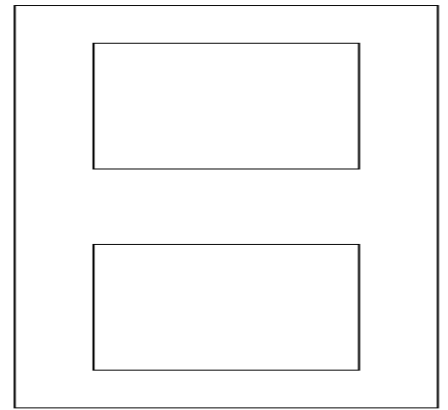
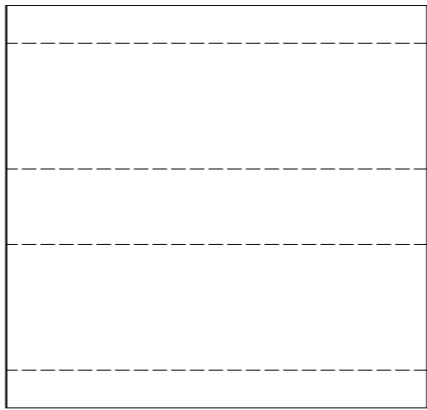
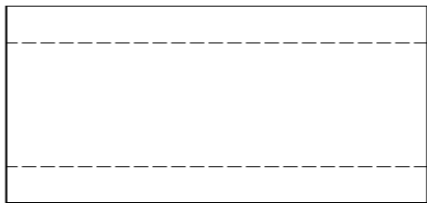
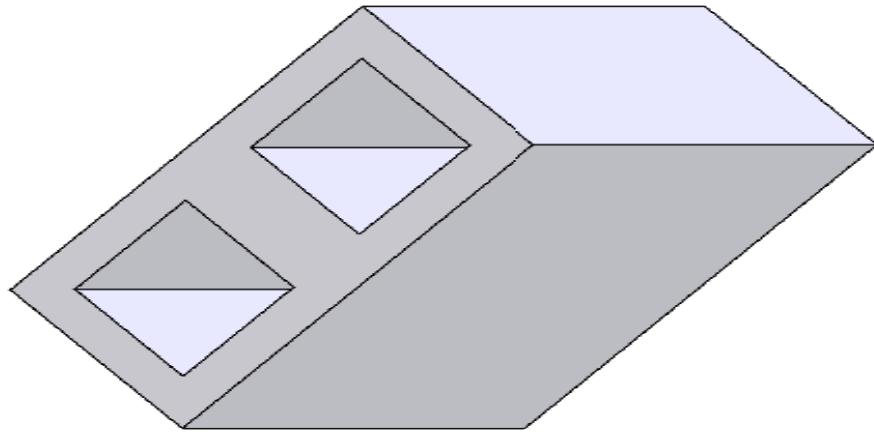


### Renaming a feature

Rename it *Cavity*

### Save and close

Click **Save**  to save your work and click **File, Close** to close the part.





# Exercise Complete!