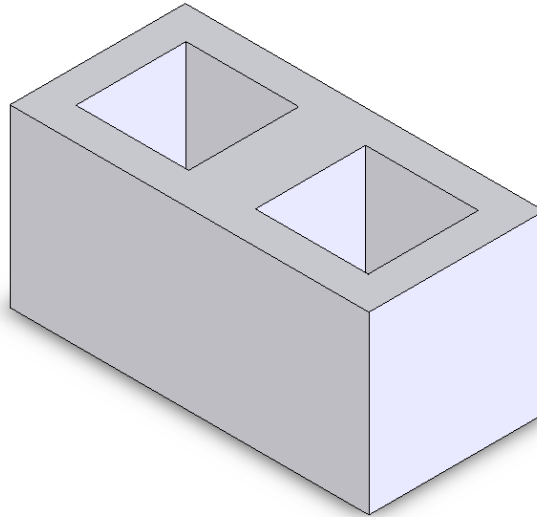


Cavity Block Exercise 2

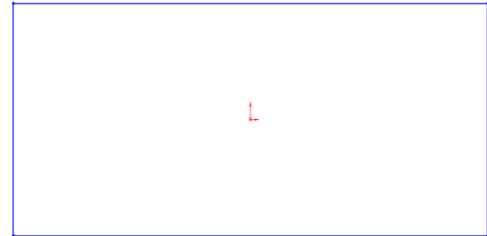


Prerequisite Knowledge	Creating a Part, Saving a Part, Creating a Sketch, Drawing Lines, Automatic Relations – Vertical, Horizontal, Coincident. Smart Dimensioning, Extruded Boss/Base (Blind End Condition).
Focus of the Lesson	On completion of this exercise you will have used: <ul style="list-style-type: none">• Sketch rectangle• Centre line• Equal relations• Sketch mirror• Extruded Boss/Base• Extruded Cut
Commands Used	<i>Rectangle, Centreline, Add Relation, Sketch Mirror, Extruded Boss/Base and Extruded Cut.</i>
Getting Started	
New Part	Create a new part. Save the part as <i>Cavity Block</i> in your chosen location.

Getting started


Where to start?

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

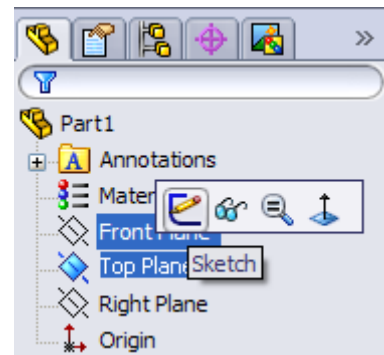


Choosing a plane


Select **Top Plane** from the manager tree.

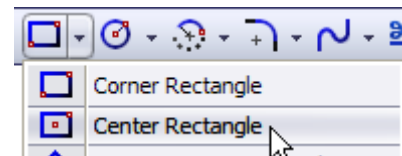
When you select the top plane four options will appear over the Top Plane. Select the sketch icon  from this list of options.

To view the sketch toolbar press **S** on your keyboard.

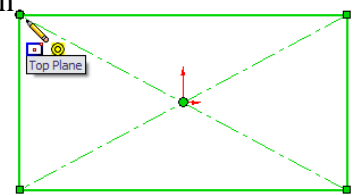


Creating a sketch


Click the drop down arrow beside the **Rectangle**  command, now select the **Centre Rectangle** command.



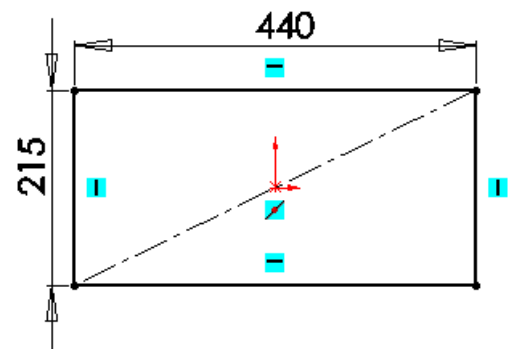
create a sketch of the rectangle using the origin as centre.



Dimension Sketch

Select **Smart Dimension**  from the sketch toolbar to dimension the sketch

Smart dimension the sketch to the values shown.



Zoom to Fit

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

Exit the sketch.



Creating the feature

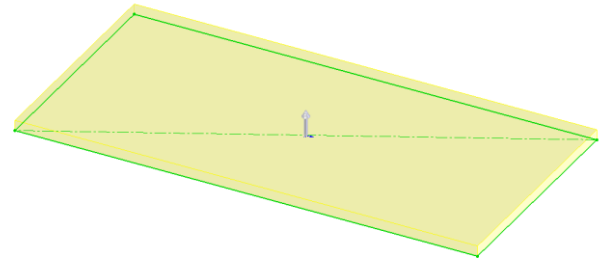
Press **S** on the keyboard and then choose **Extruded Boss/Base**  from the **Features Toolbar**.

Extrude Feature Settings

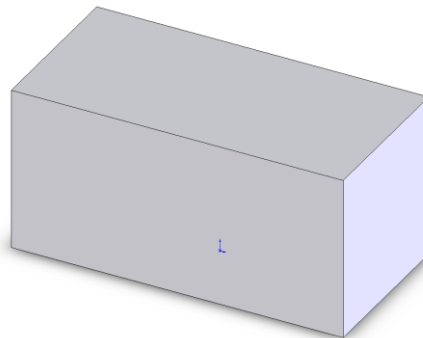
End Condition = **Blind**



Depth = **215mm**



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



Renaming a feature

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



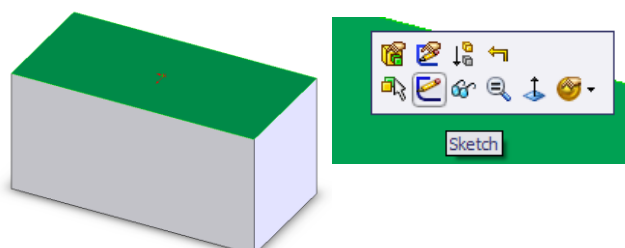
Type the new name *Block* to replace it.

Creating the Cavities


Sketching on a Face

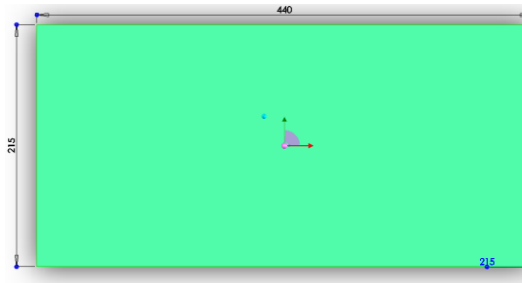
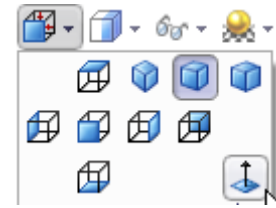
Any face of a model may be used to generate a plane to contain a sketch.

Select the top face of the block. Then select the sketch icon  from the List that appears.



Change View

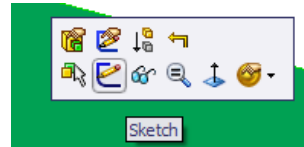
From the options on the top of the graphics area select the View orientation icon and select **Normal To** . This allows us to look in on the surface at a 90 degree angle.



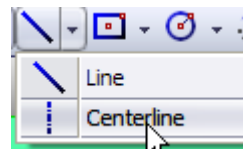
Create a new Sketch

To create a new sketch select the sketch icon from the List on the chosen surface.

Then Press **S** to view the Sketch toolbar.

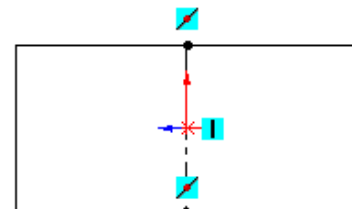


From the sketch toolbar Select centreline from The line command.

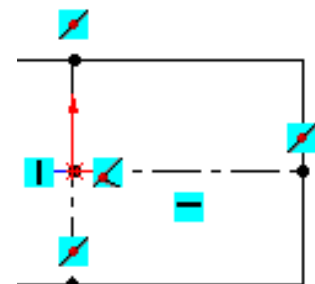


Draw a line from the midpoint of the horizontal lines.

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

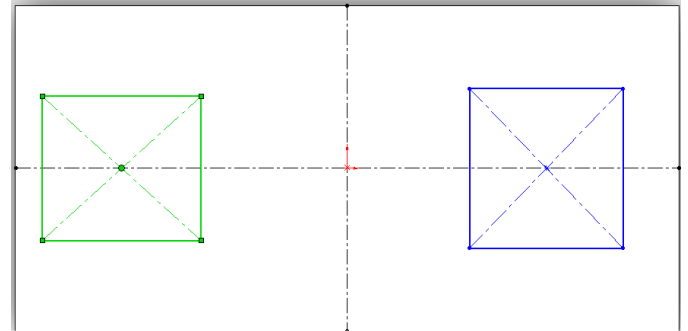


Draw a second centreline from the origin to the midpoint of one of the vertical lines.

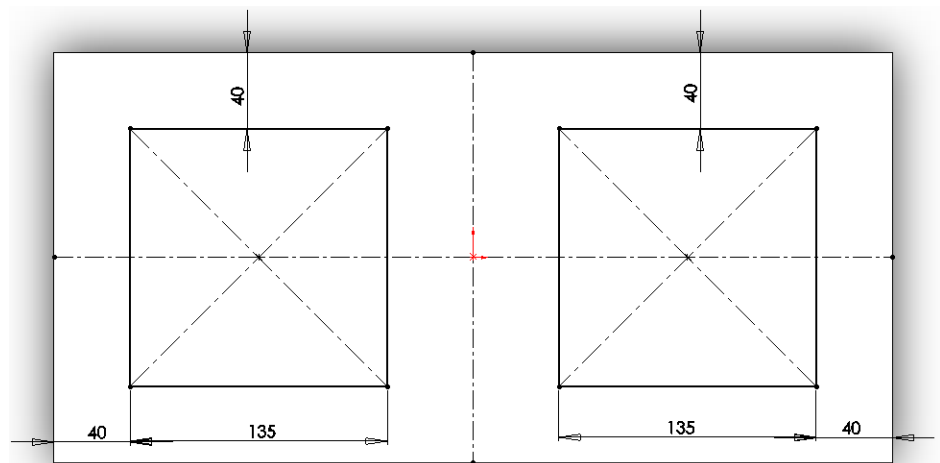


Rectangle Command Press **S** and select the **Centre Rectangle** Sketch command and draw 2 rectangles On the centre line.

Press **ESC** on your keyboard



Dimension sketch Dimension the sketch using the dimensions shown.



Exit Sketch.

Isometric View

Select **Isometric** from the **View Orientation** toolbar.



Extruded Cut

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

Press **S** on the keyboard and select **Extrude Cut** from the Features toolbar.



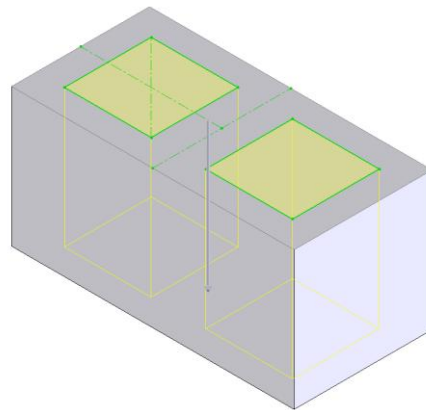
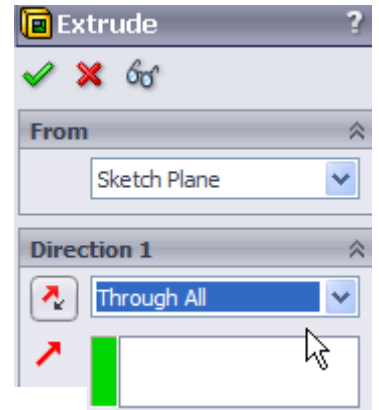
Cut Extrude Feature Settings

End Condition = **Through All**

Through All End Condition

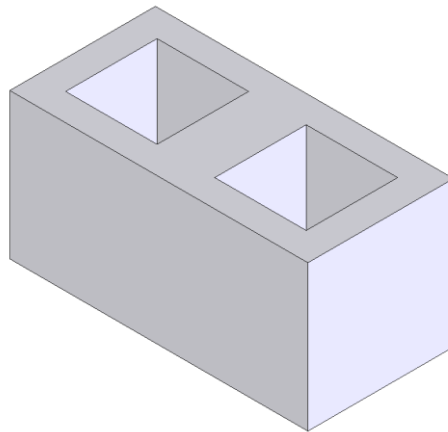
This type of end condition always cuts through the entire model no matter how far.

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



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

Completed feature

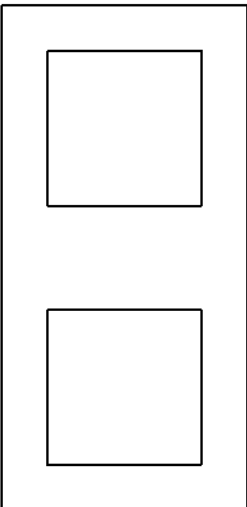
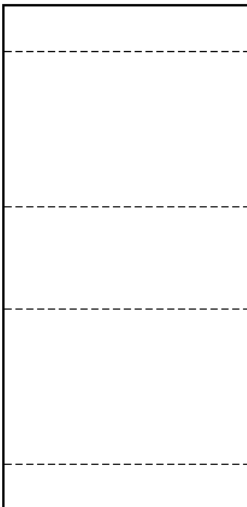
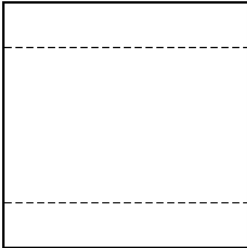
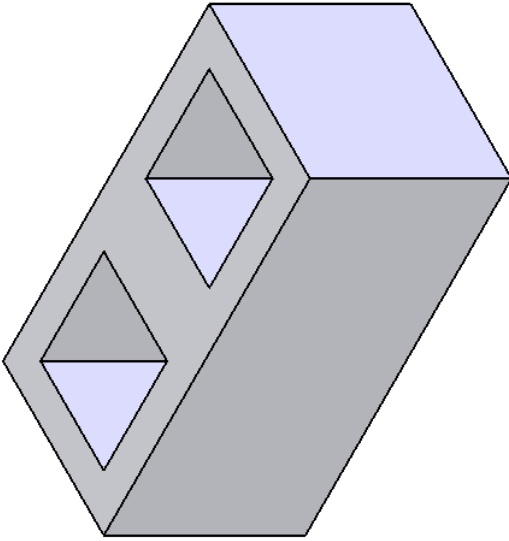



Renaming a feature

Rename it *Cavity*

Save and close

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



	TECHNOLOGY
	TITLE: Cavity Block
	UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm
	SEE A4
	\$ DRAWN BY: YOUR NAME HERE
	\$ SCALE: 1:1
	Date: 07/05/2010
	\$ SHEET 1 OF 1
	