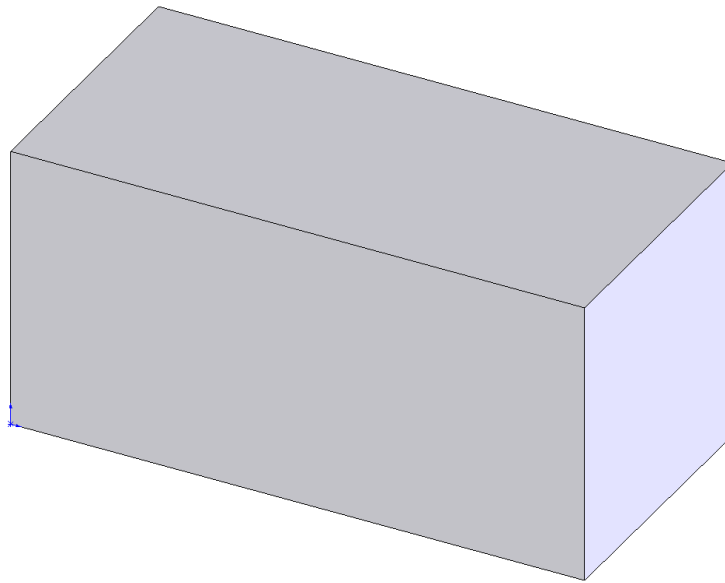


Block Exercise



Prerequisite Knowledge

Navigating the interface, file management, opening and saving of file.

Focus of the Lesson On completion of this exercise you will have used:

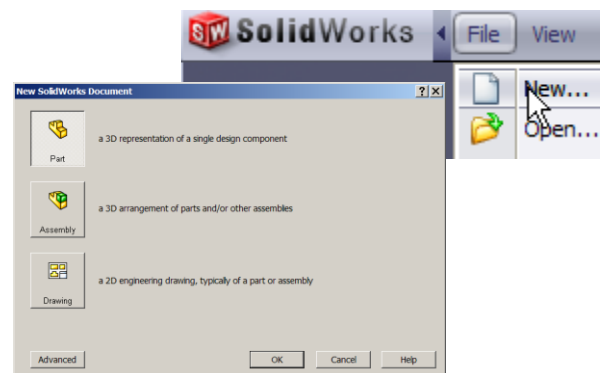
- Sketch line
- Automatic relations
- Smart dimension
- Extruded Boss/Base
- Renaming a Feature
- Editing a Sketch

Commands Used Getting Started

Line, Extruded Boss/Base and Edit Sketch.

New Part

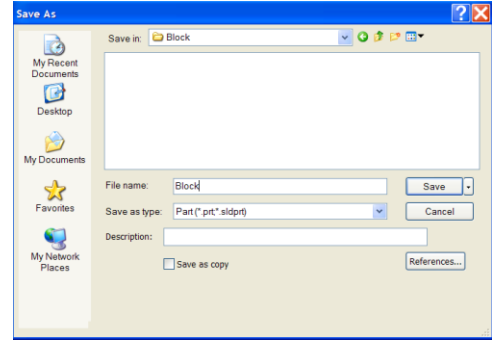
Click **File, New** on the standard toolbar. Select **Part** from the **New Solidworks Document** dialog box. Select OK.



Saving the Part

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

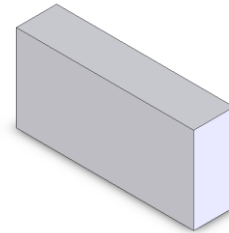
Continue to save periodically throughout the exercise



Command Manager Where to start?

To activate the command manager
The only feature of the part to be created is the Block. This will be an extruded feature based on a sketch.


Sketch to generate the feature



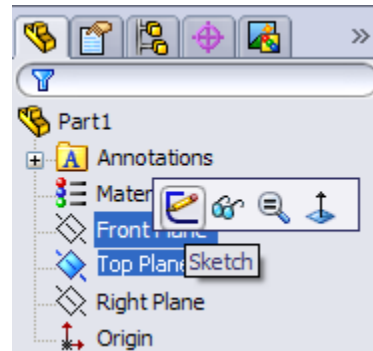
Getting started

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.



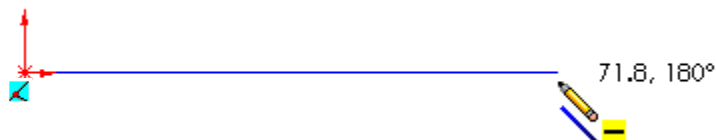
Creating a sketch


Click the **Line**  command and sketch a horizontal line **coincident** with the origin.



Coincident Relation

The origin is an endpoint to the line and therefore there is **coincident relation** between the endpoint and origin.



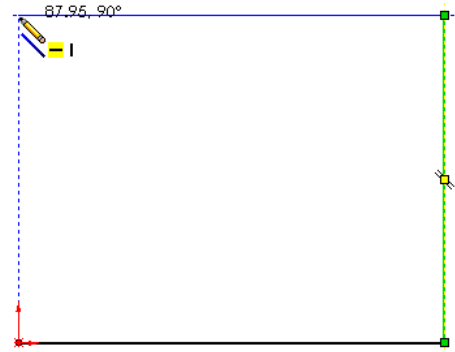
The “” symbol appears at the cursor, indicating that a **Horizontal** relation is automatically added to the line. Click again to select the endpoint of the line

Draw a line ensuring that a **vertical  relation** is automatically added.

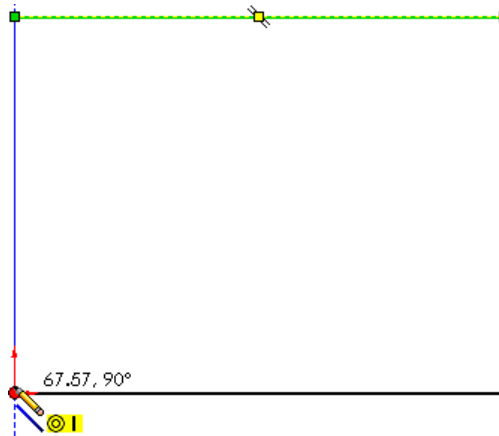
Inferencing

Drag a Horizontal line and using **Inferencing** choose its endpoint directly over the origin.

Inferencing lines are dotted lines that appear as you sketch. When your pointer approaches highlighted cues such as midpoints, the inferencing lines guide you relative to existing sketch entities.



Finish the shape by drawing a vertical line and select the origin as the end point as shown.





Turning off tools

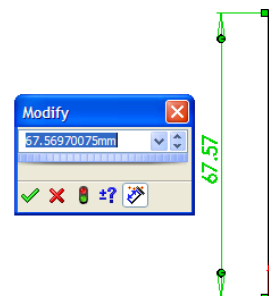
Turn off the active tool using:
Press the **Esc** key on the keyboard

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

Dimensioning the sketch

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

Using  select the line shown.
Click a second time to place the text of the dimension to the left of the line.
The dimension appears with a **Modify** tool displaying the current length of the

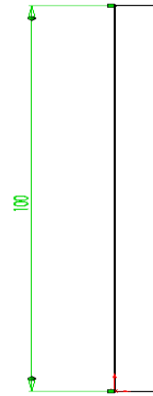


line.

Set the value

Change the value to **100** and click **Save** ✓ option. The dimension forces the length of the line to be 100 mm.

Pressing Enter has the same effect as clicking the Save ✓ button



Add additional linear dimensions to the sketch as shown



Zoom to Fit

When dimensioning the sketch may become too large in the work area. To overcome this press **F** on the keyboard and the sketch will automatically fit in the work area

Note – The sketch changes from blue to black when it is fully defined.

Exiting the Sketch


To exit the sketch, select the sketch tool on the confirmation corner, sketch will be saved. Selecting **X** will discard changes made.

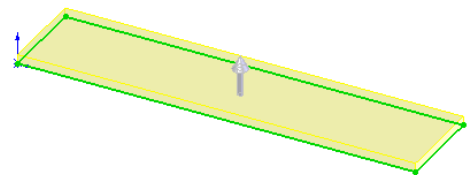


Creating the feature

Having exited the sketch, press **S** on the keyboard to view the features toolbar. select Extrude Boss/ base from the toolbar.




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




Extrude Feature Settings

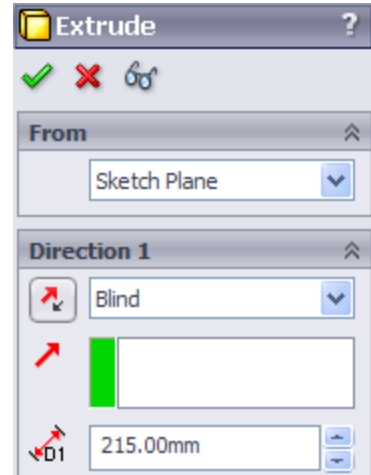
End Condition = **Blind**

Blind End Condition Extends the feature from the sketch plane for a *specified* distance.

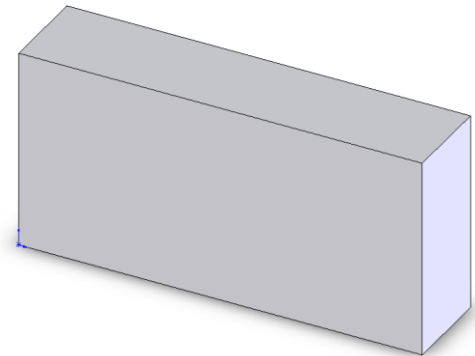
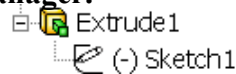
 Depth = **215mm**

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.



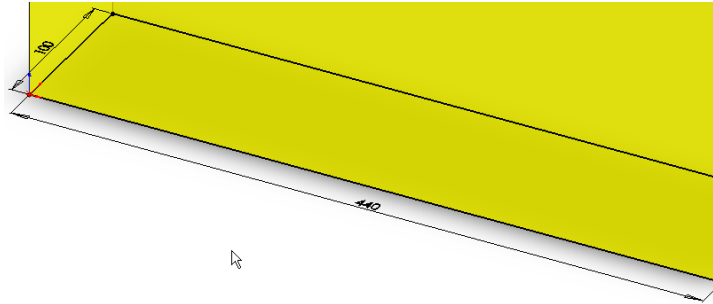
The feature name will be highlighted with a flashing cursor on the right hand side. Type the new name to replace it.

Editing Sketch

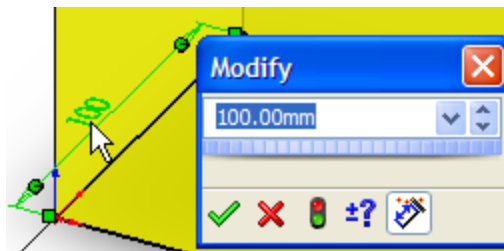
On the **FeatureManager design tree** select the plus beside the Block feature so that Sketch1 appears.




Right click on *Sketch1* and select the *Edit Sketch* icon. The original sketch used to create the feature appears, with the feature shown as an orange preview.



Editing Dimensions Double click on the 100mm dimension. The **Modify Tool** appears displaying the current length of the line.

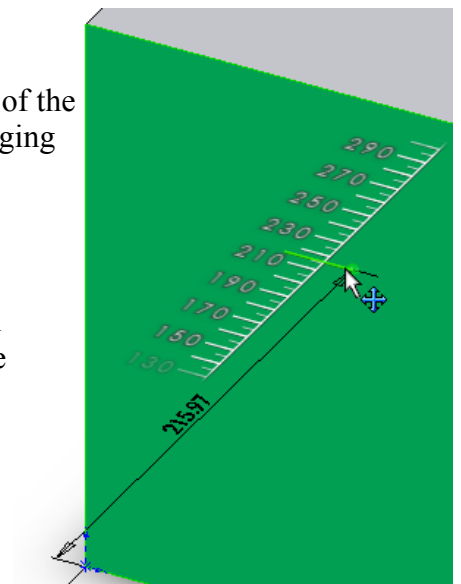


Change the value to **215** and click **Save**  option. The dimension forces the length of the line to be 215 mm.



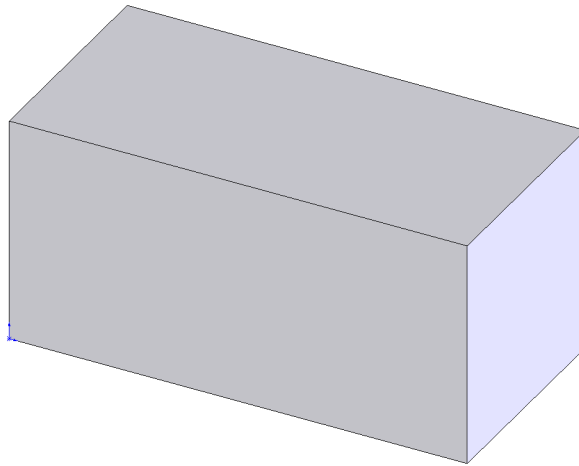
Note you may also change any of the dimensions of the Block by selecting any face of the block and dragging the blue dot at the end of the chosen dimension.

This activates the ruler, you can set the desired Dimension by scrolling along this ruler. If you Scroll the cursor on the ruler it will move in 5mm Increments, but if you scroll with the cursor to the Side of the ruler it will move in .1mm.




Exiting the Sketch

To exit the sketch, select the sketch tool on the confirmation corner.



Save and close

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