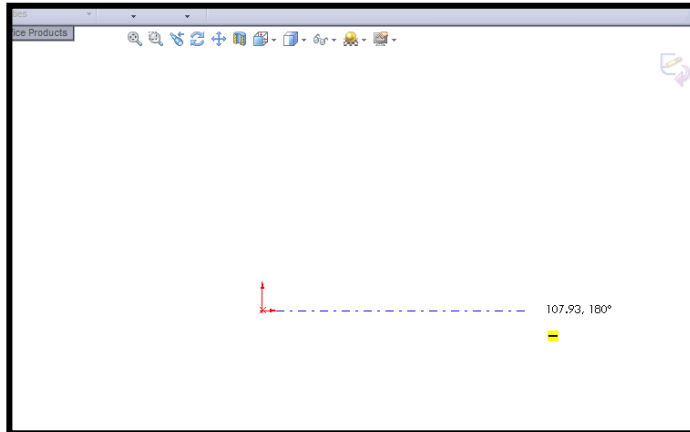


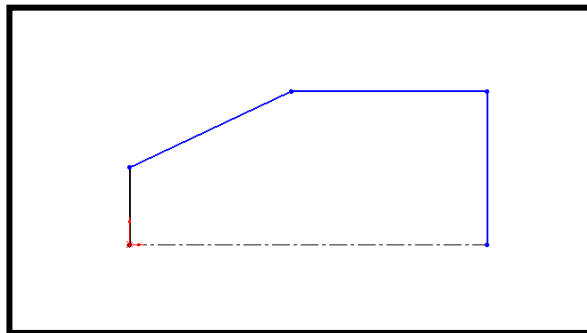
Basic Features – Extrude. The Strap Clamp (Part 1).

To begin the initial sketch, select a plane (Top plane would be best in this case) and insert sketch.

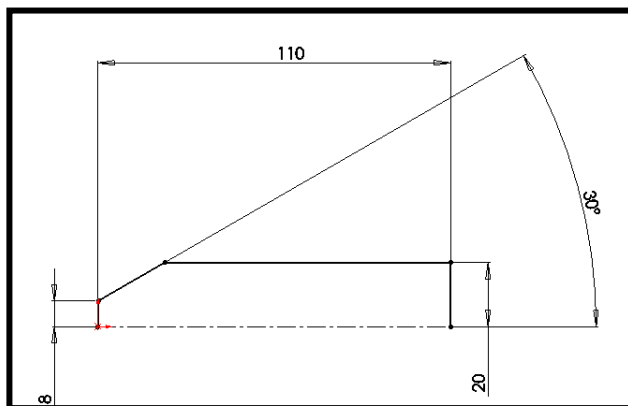


Start with a Horizontal centreline, start at the origin (click at origin – achieve a coincident relationship), move the pointer over to the right, click again when the line is horizontal (check that the relationship icon is shown)

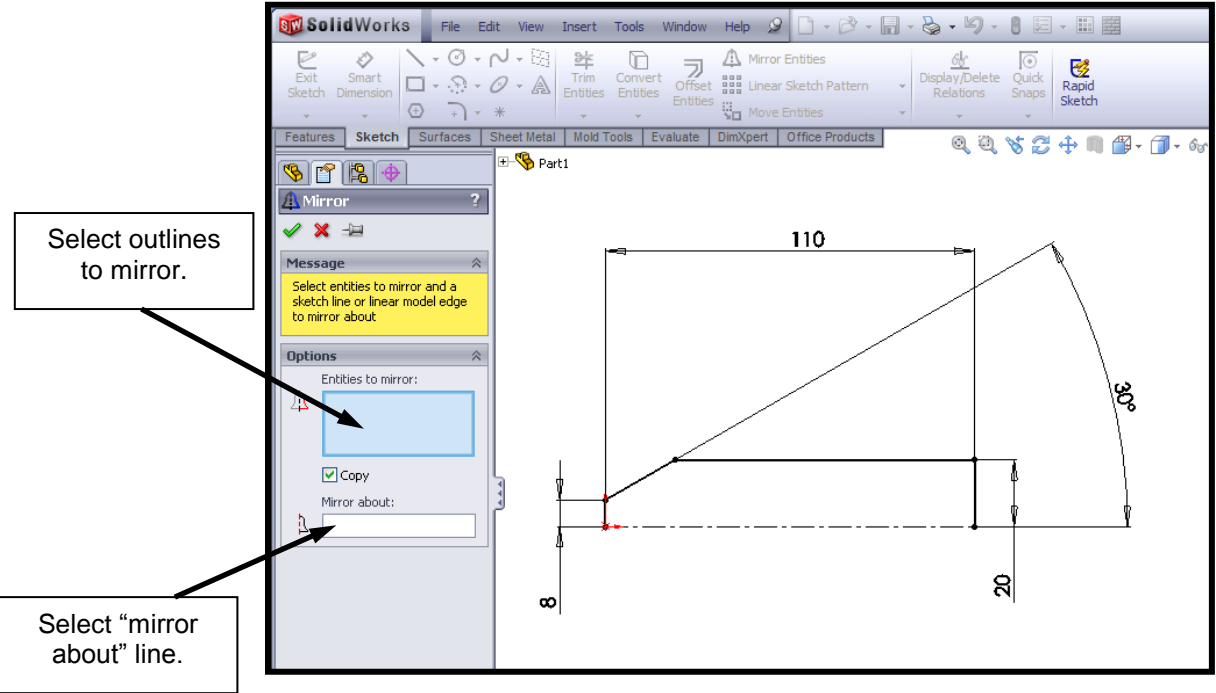
Now select the line tool, add the outline of one half of the clamp shape, and ensure that vertical and horizontal lines are produced with the added relation (shown by the yellow icon). Do not consider any sizes at this stage; shape is produced before size with Solidworks.



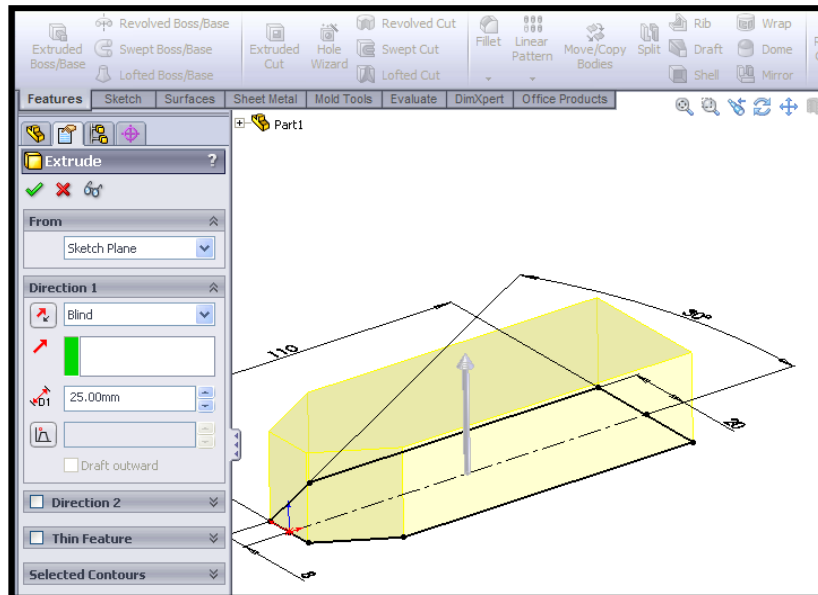
Now we can add size control, on the sketch toolbar select the dimension tool. The necessary dimensions are added and the sketch should be fully defined – the sketch lines should change colour to black, and in the lower right area of the screen the command statement will state “fully defined”.



To save drawing the other half of the of the outline profile, we should use the sketch mirror command.



When all lines are selected as required, accept by clicking the green tick in the property manager. If your initial sketch was good and the items selected correctly to mirror you should now have a full profile of a strap clamp.



Select the "Features Tab" and then select "Extruded Boss / Base".
Select the required direction (Click the direction 1 arrows (black and red) icon to change direction), set the thickness for the extrude in the D1 setting box; in this case 25mm.