Origin selection
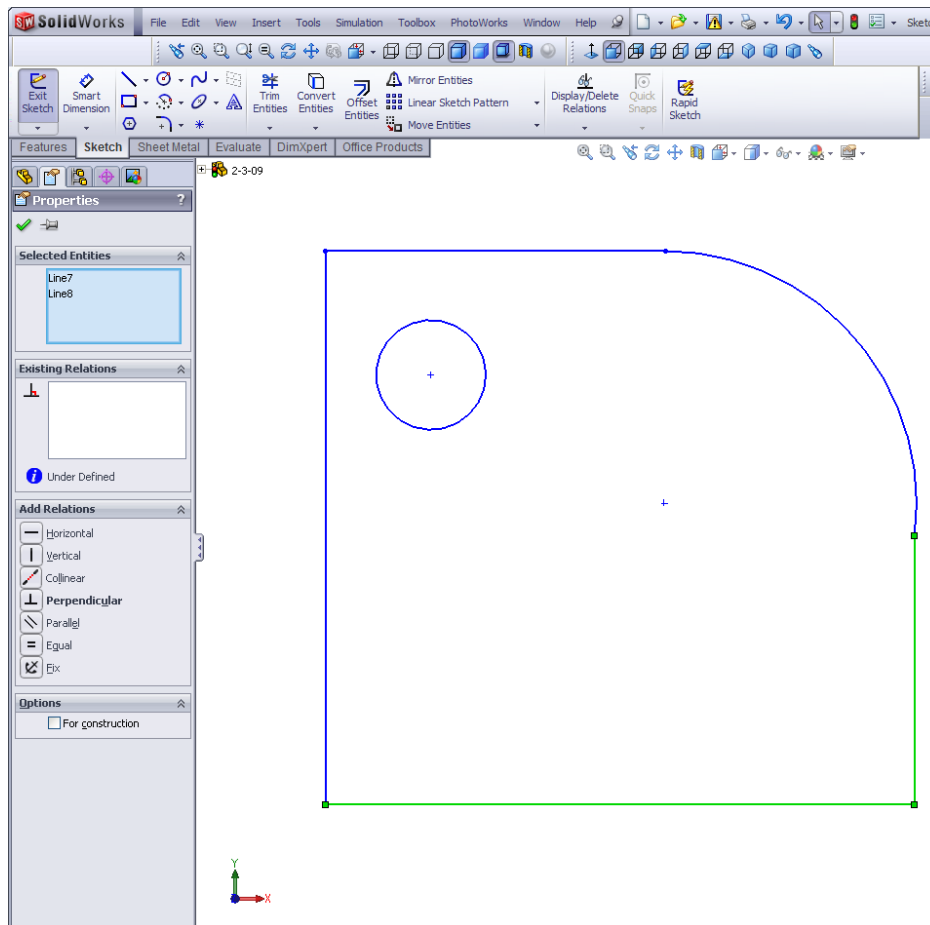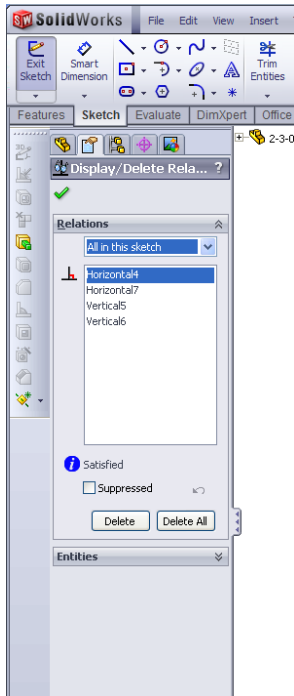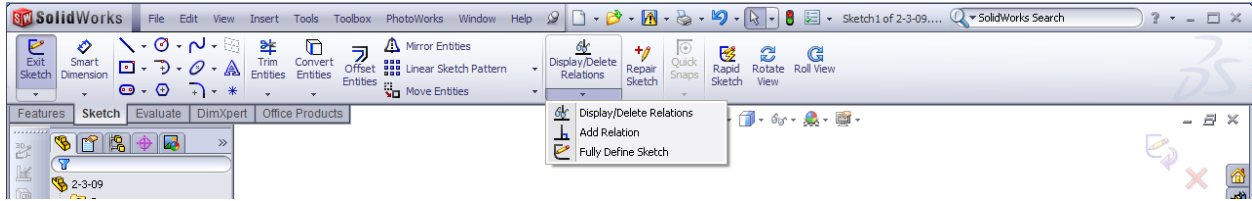
- The origin defines the part in space
- All dimensions refer to the placement of the origin
- Before starting a sketch, figure out where you want the origin to be placed

Add relations

- To add a relation, select two features
  - Use the mouse to click on the features while pushing the "Ctrl" button at the same time
  - Relations can also be added by using the "add relations" tool found under the "display/delete relations" icon

- Once the features have been highlighted, the "Property Manager" tab will appear on the right side of the window
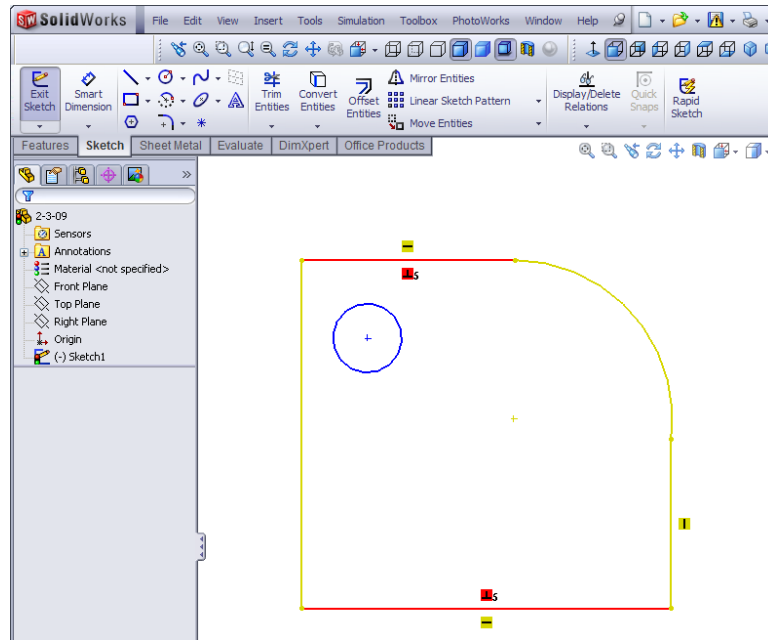
- The "Display/Delete" button can also be used. This is found in the sketch toolbar

- When that is selected, the "Display/Delete" information will show in the Property Manager window (on the left of the screen)

- To show relations, go to "View" then click on "Sketch Relations"
    - The relations will be highlighted in blue

Display relations

- Blue Lines – means the line is under-defined
- Black Lines – means the line is fully defined
- Yellow Lines - means the line is over-defined, too many driving dimensions
- Red Lines – means the line is over-defined, the constraints added make it impossible to create the part

Sketch definition in design tree

- The sketch is under defined if there is a (-) sign next to it. Once the sketch is fully defined the symbol next to the sketch will disappear.