Show and Tell- Basic Sketching
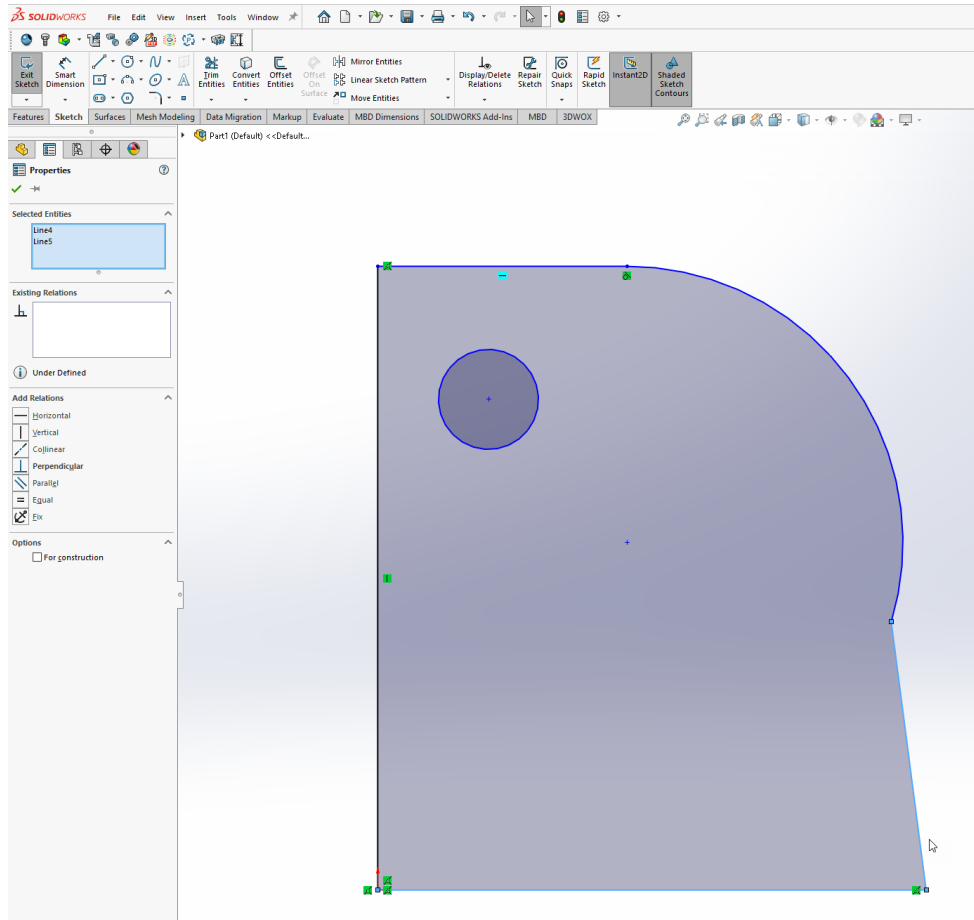
Basic Sketching

Objective

Know how to:
- Create a sketch
- Know if under/over-defined
- Add/delete relations

Origin selection

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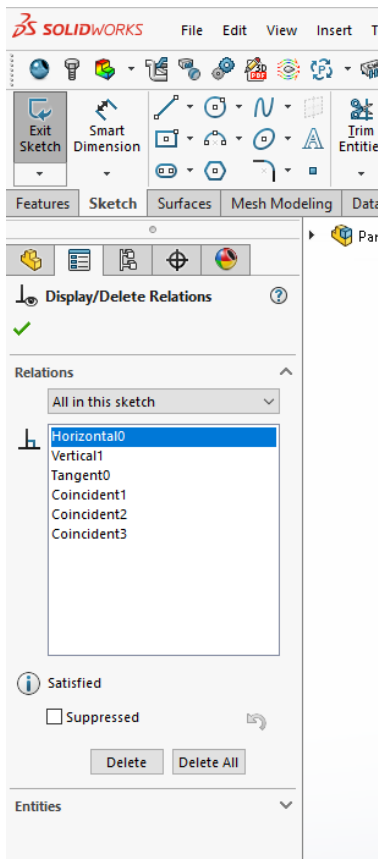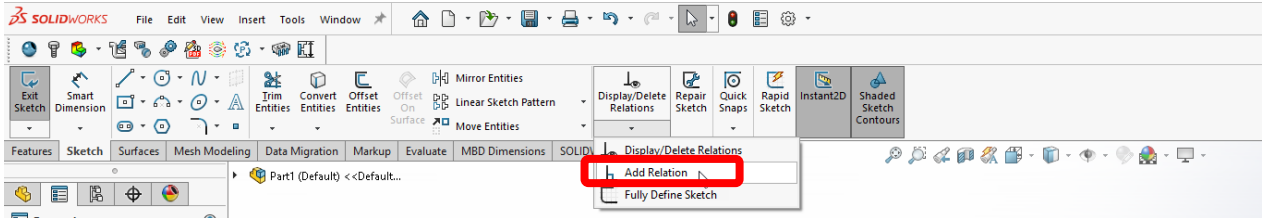- Before starting a sketch, figure out where you want the origin to be placed

Add relations

- To add a relation, select two sketch entities (lines, arcs, points…).
  - Hold the "Ctrl" button while clicking the second entity to keep both selected.
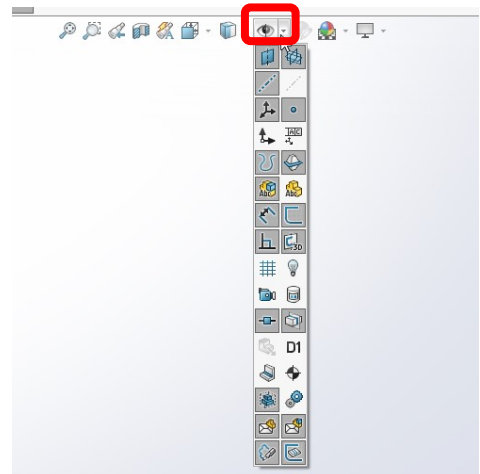
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- Once the sketch entities have been highlighted, the "Properties" tab for the selected entity/entities will appear on the left side of the window with an "add relations" section.

Relations can also be added by:

- o Selecting "Add Relations" from the "Display/Delete Relations" icon.
  - When that is selected, the "Display/Delete" information will show in the Property Manager window (on the left of the screen)
- o Selecting entities, right-clicking the mouse, and selecting from the fly-up menu just to the right of and above the cursor.

To show all existing relations, go to the eyeball in the Heads-Up View toolbar then click on "View Sketch Relations" icon (perpendicular symbol) -- the relations will be highlighted in green.
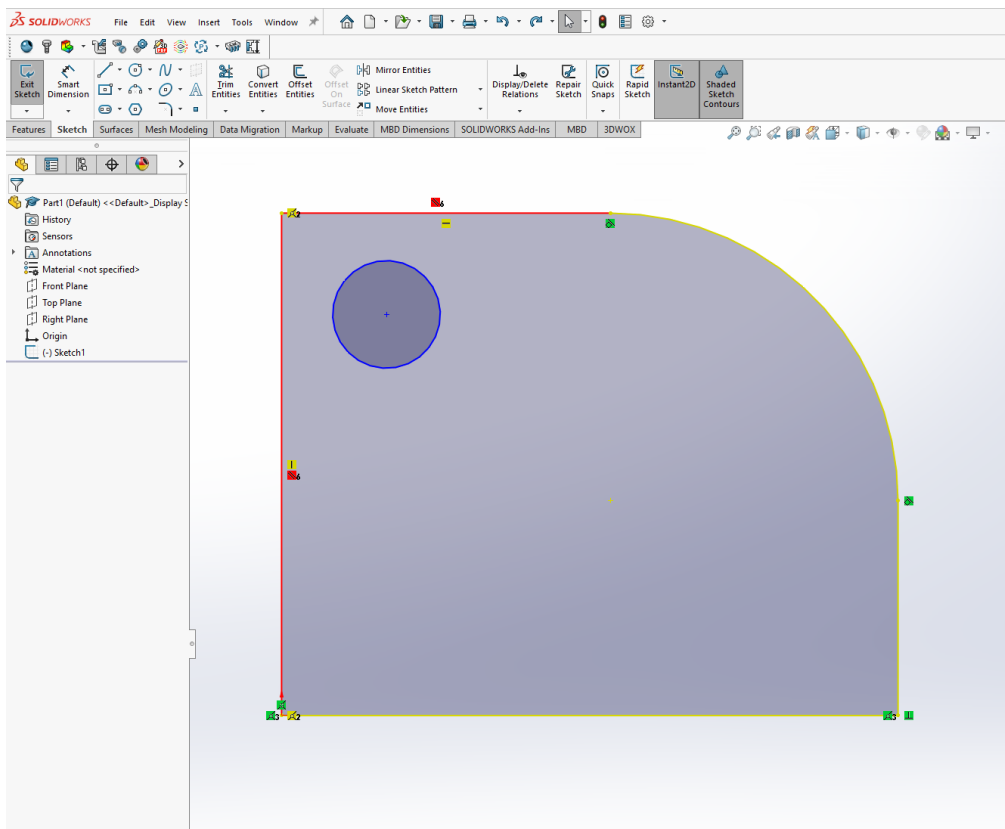
Display relations

Sketch Entities (Lines/Points):
- Blue – means the line is under defined
- Black – means the line is fully defined
- Yellow - means the line constraints are invalid, requires deleting some constraints
- Red – means the line constraints are over-defined and therefore unsolvable, indicates the geometry cannot be determined for one or more entities

Sketch Relations:
- Green – applied and solved
- Yellow – invalid
- Red – over defined / unsolvable

Show and Tell- Basic Sketching

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.