SolidWorks CAD Modeling 101

How do you make a 3D part/feature?
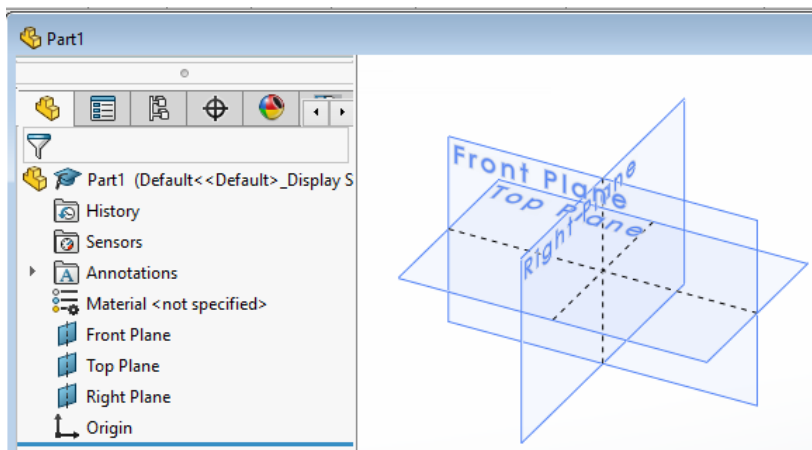
Step 1: select a sketch plane (usually "Front", "Right, or "Top" to start out)

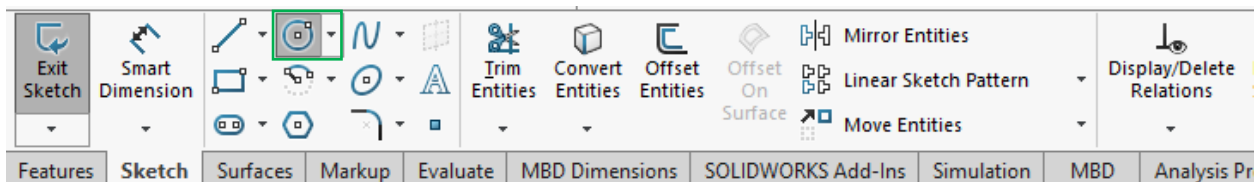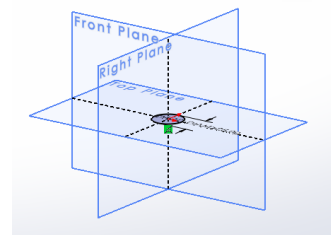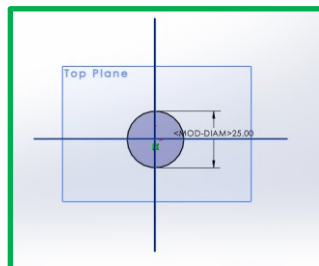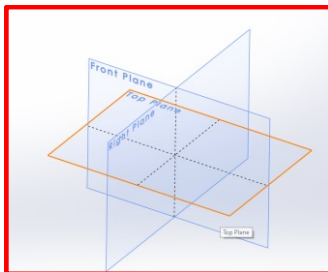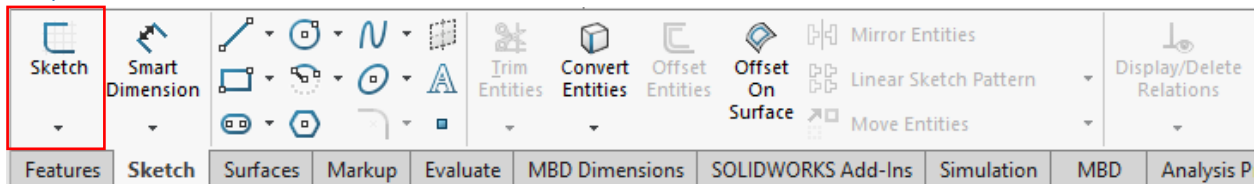**Step 2**: sketch a 2D shape using 2D sketch tools from the Sketch Toolbar

Step 3: turn it into a 3D shape using a feature from the Features Toolbar
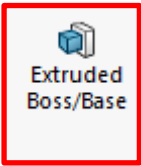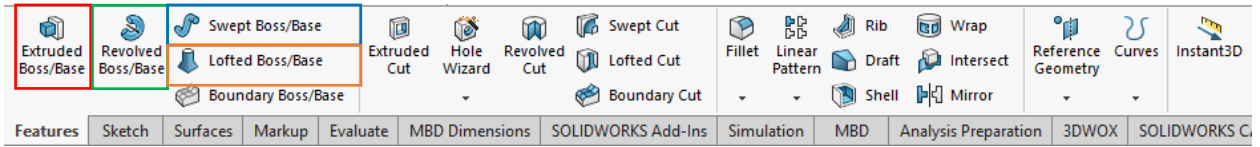
Step 1: Sketch planes

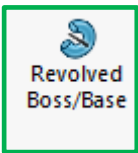Front, Top, and Right planes exist by default. You can add other "reference" planes.

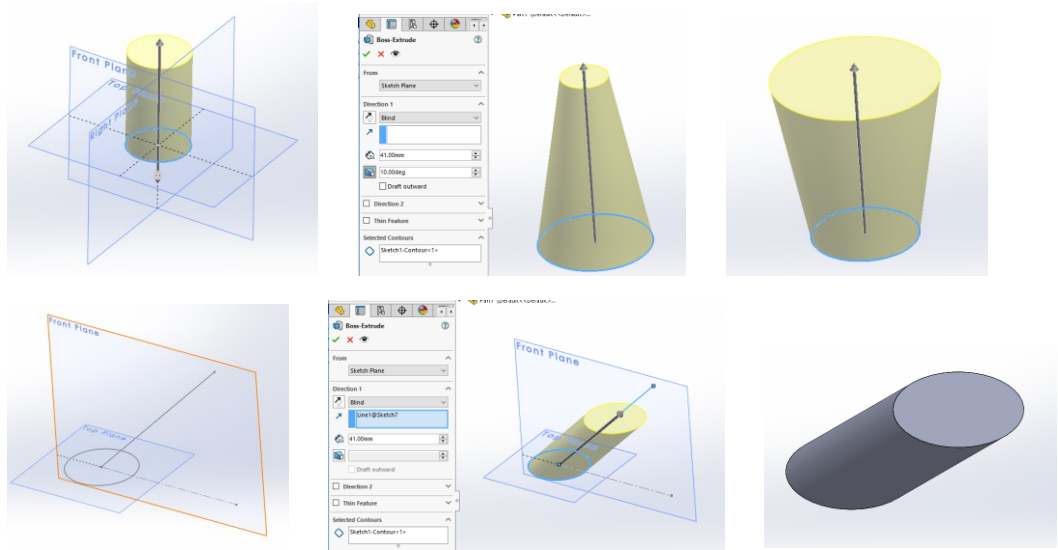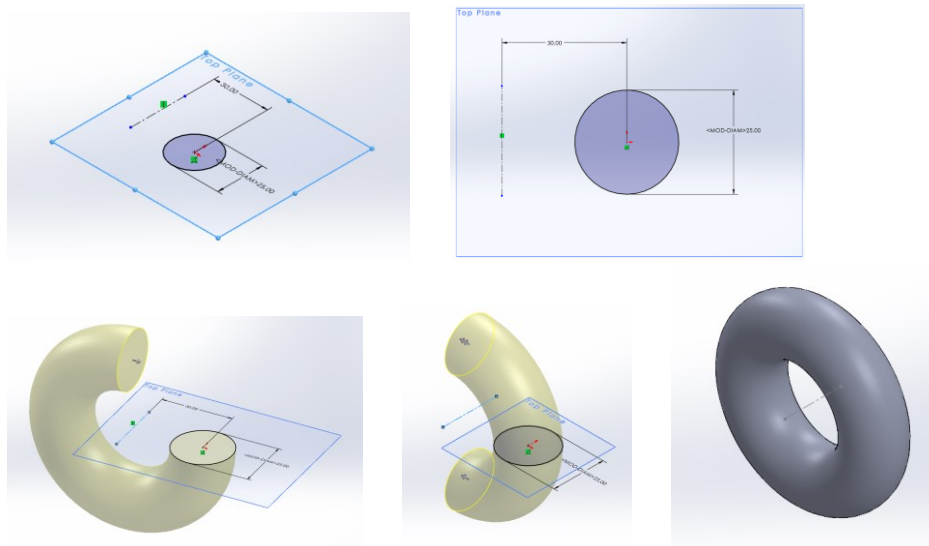## Step 2: 2D Sketch Tools
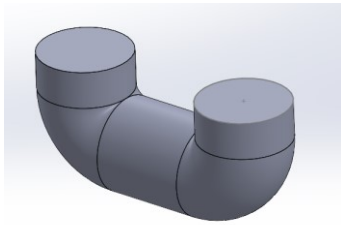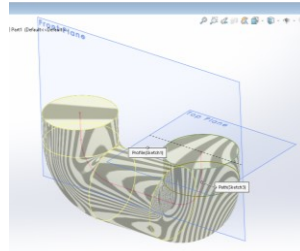
Step 3: 3D Feature Tools
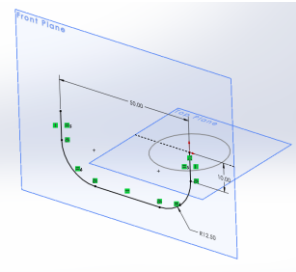

Requires a profile.

Requires a profile and an axis.

Swept Boss/Base — Requires a profile sketch and a path sketch.

Lofted Boss/Base — Requires at least 2 closed sketches on different planes.