# CAD CHEAT SHEET: BASIC SKETCHING

Intro to Cad -- Rohawks 3419 -- Lili and Chris

Every part is based on a series of SKETCHES: 2 dimensional drawings that you can EXTRUDE to make 3 dimensional. Sketching is a core CAD skill, and this sheet will provide you with the most basic tools needed to create the foundation for a cool 3D object.

A large portion of CAD is figuring things out and playing around on your own, so we won't tell you EVERYTHING here — exploring is part of the fun.

STARTING A SKETCH:

After you create a new part (by clicking "part" on the homescreen), a blank window will open up. At the top of the screen in the "3D Model" tab, click START 2D SKETCH.
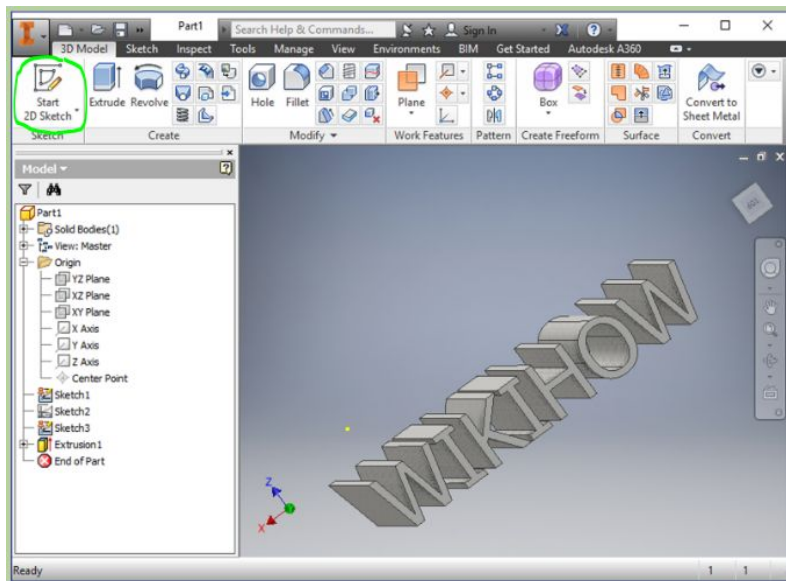

Once you do this, XY, YZ, and XZ planes will show up. To make a sketch you always need a PLANE to sketch on, and by convention the first plane you sketch on should be the XY Plane. Once you click that one, your sketching window will open up.

!! TIP: Once you have a solid already created, you can also create sketches on the FACES of the solid — they are planes too!

SKETCHING:

When starting a sketch, orient your sketch around the CENTER of the XY plane that shows up. When you create a shape or line, you will have to choose an endpoint or a center. SNAP THAT to the origin (you can tell if it is snapped because the cursor turns GREEN)
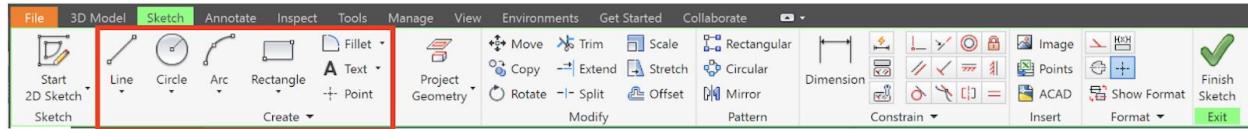

Try to make a LINE, CIRCLE, or RECTANGLE! If you can figure out how those work, try an ARC!

HINT/CHALLENGE: take a look at what happens if you click the ARROW below each of the basic shapes! What do you see? Can you figure out how to use that?

!! TIP: to delete something, highlight it, then right click on it, then click "delete" (maybe explore some of the other options that pop up then too :D)

!! TIP: If you don't know what something does / want to know the keyboard shortcut, hover over the icon for a few seconds and a window will pop up:
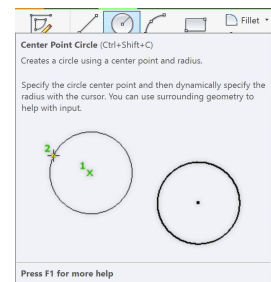

DISTANCE IN SKETCHES:
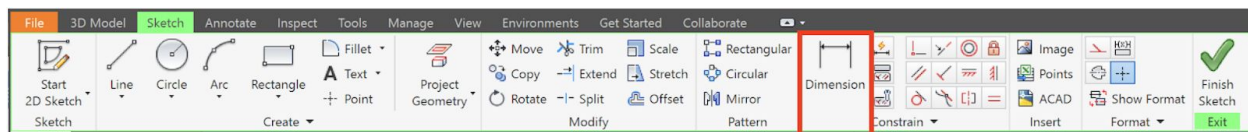
Setting distance in a sketch is very important, and it is done using the DIMENSION tool. Dimensioning sketches enables us to mimic real-world dimensions and more effectively utilize our models. It can also be used to set angles between non-parallel lines, by clicking each line after selecting DIMENSION.

HINT/CHALLENGE: try to measure the length of an angled line segment!

EXTRUSIONS:

Once you've learned how to make shapes and simple sketches, it's time to extrude! We extrude shapes to make them three-dimensional and this allows us to create solids in Inventor.

Complete your sketch by selecting "Finish Sketch" in the upper right hand corner. Make sure your sketch is sealed: there should be no open figures. Next, click the 3D model tab in the upper left and find the Extrude button. There, you can format your extrusion: choose the direction and the distance.
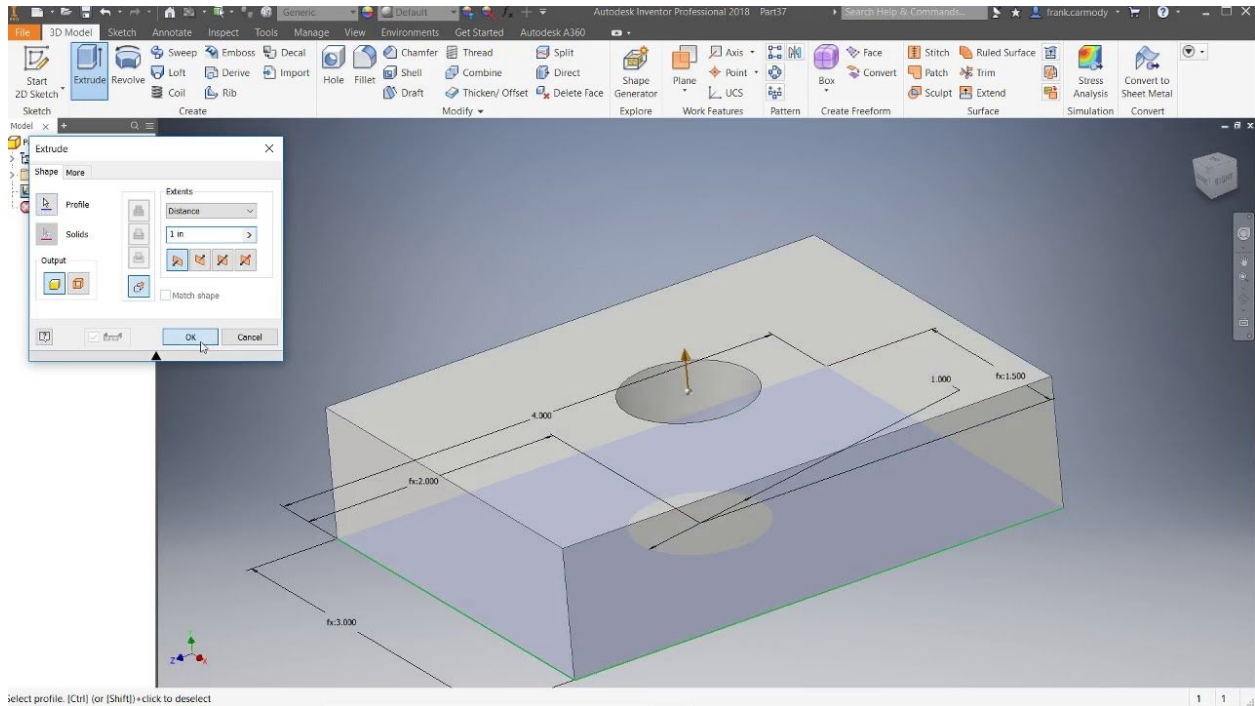
Alternatively, extrusions can be used to cut shapes out of other solids. Just sketch onto the face of a solid, then extrude to cut it out.

HINT/CHALLENGE: Try to make a ring in two ways: 1) use two sketches and two extrusions; 2) use one sketch and one extrusion.

CONSTRAINTS:

Let's go back to sketches! Constraints are a useful tool that we use to make our sketches as precise as possible. Test out some of the constraints in the toolbar, directly to the right of DIMENSION!

KEYBOARD SHORTCUTS

S: start a sketch
L: start a line
E: extrude
D: enter distance mode
M: measure
Scroll up/down: zoom out/in
F2 + drag mouse: pan
F4 + drag mouse: rotate view