CAD CHEAT SHEET 2: New Techniques

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

Now that you've got the basics down, we can learn more about CAD tools which will make your designing life way easier! This sheet will cover mirroring, rotating, duplicating, making holes, construction lines, and drawings.

Again, this sheet won't be the answer to *every* problem, but we hope that it will guide you in the CAD process while you are still learning.

CONSTRUCTION LINES

When CADing many different types of parts or mechanisms, utilizing construction lines can be helpful for symmetry or facilitate the sketching of more complex shapes.

Construction lines are lines that you can use to guide your sketches but will not be extruded in an extrusion. Essentially, construction lines will *not* be considered a boundary enclosing a space.

One example of use of construction lines is creating a hole in the center of a rectangle. To find the center of the rectangle, make a construction line connecting 2 opposite corners to create two diagonals. Construction lines are used here because the lines will not be used to extrude a shape from the rectangle, and will make everything more complicated if they are treated as solid lines. Snap the center of your hole to the intersection of the 2 construction lines and VOILA! You have a hole in the center.
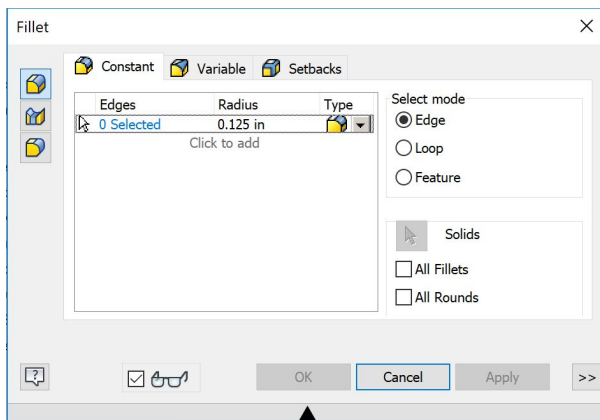
HINT/CHALLENGE: Can you figure out how to use construction lines to create a hole in the center of a rectangle?

HOLES and FILLETS
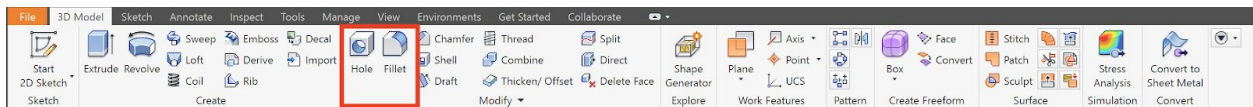
After creating a solid, we often use tools like the hole and fillet functions in order to finalize a part. Holes are often created in order to indicate the position of a bolt, and fillets round the edge of solids!


FILLETS are used for a multitude of reasons, most often on parts we are going to 3D print. Filleting something (rounding its edges) cuts away some material, making the part less sharp and (more importantly) making it *weigh less*. As a bonus, the parts often also look a lot prettier and less boring. If I can fillet something, I *will* fillet it.

HOLES are used to create an opening for a hole that a bolt of a specific size and shape can fit into. While you might think that you could just extrude a circle into the part, for funkier bolt shapes this does not work. Holes are especially useful when trying to make space for a COUNTERSUNK bolt. These bolts have cone-like heads and are meant to sit flush with the surface of a part. The HOLE function lets you specify what kind of bolt the hole is for, how deep it should go, its width, and a few other things. You use it by making a point in a sketch, exiting out of the sketch, and then clicking the "hole" button and clicking the point.
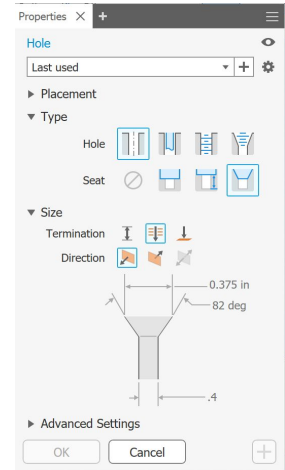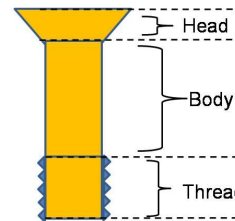
HINT/CHALLENGE: Can you figure out how to create a hole for a countersunk bolt? (see image)

MIRRORS and PATTERNS
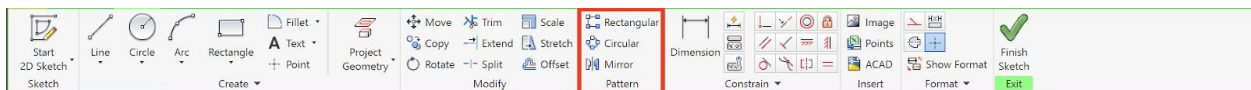
Mirroring in a sketch can save time with symmetrical designs: by simply selecting a mirror line, the mirror function will flip selected features over that line. However, mirroring will only save you time if you PLAN AHEAD!

Patterns are similar to mirrors in that they can save time in tedious phases of sketching. The circle pattern allows you to replicate features of a sketch perfectly spaced around a center point, and the line pattern will create evenly spaced shapes in a line.

HINT/CHALLENGE: Can you make a flower sketch using patterns or mirroring?
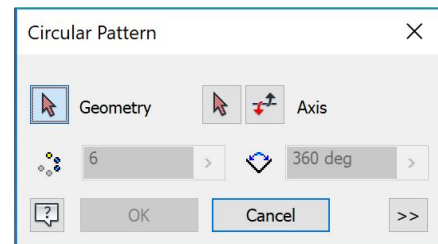
DRAWINGS

Drawings are the most useful when creating parts in the Alliance Bernstein workshop! As we begin to machine parts for our robot, we use drawings to tell people the dimensions for certain parts.

To make a drawing, click File → New → Drawing. After this, click "Base" at the top of the screen. It will open up this menu where you can import a CAD model you want to draw.
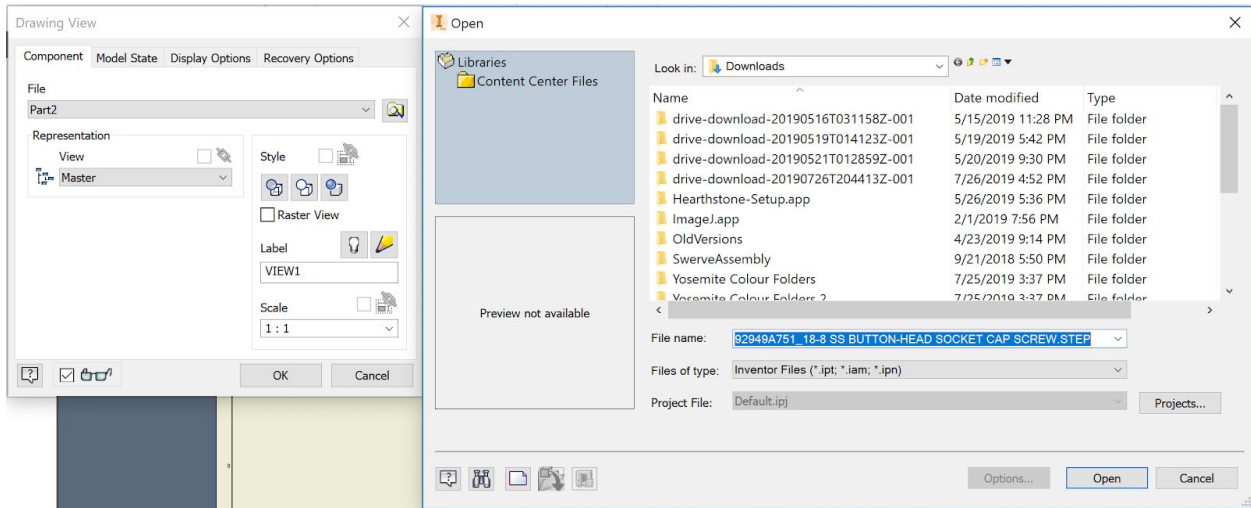
After doing this, you can do all sorts of cool things to change the angle of your part in the drawing and add dimension specifications and notes to whoever is going to make the part. A lot of that is pretty self explanatory when you actually mess around with a drawing, but we'll quickly explain here how to add a DIMENSION.

Go to the Annotate tab and click dimension. On the drawing of the part, click any line segment / section like you would when adding a dimension to something in a sketch! An option to add text will pop up — simple and useful!