AutoCAD 2013 Basics

- If the icons look unfamiliar when you start AutoCAD (no LINE icon in upper left), make sure it says “Drafting and Annotation” at the top of the screen to the left of your file name. If not, pull down and choose it.
- If you need help with a command, just click on the icon, then hit the F1 key on the top of your keyboard. You can also choose the question mark in the upper right and search for your topic.
- If you’re stuck somewhere you don’t want to be, hit the Esc key in the upper left on keyboard.
- Always watch the command line at the bottom of the screen, it will tell you what it wants. Anything in [ ] indicates options you can select by typing in the capitalized letters, and anything in <> indicates the default, which will be used unless you type in something else.
- The right mouse button is context sensitive. That means what it does changes depending on what you’re doing. Try it! You can always right click instead of hitting the Enter key.
- If you screw up, just right click and click UNDO, or click the back arrow icon at top of the screen.
- If you have a mouse wheel, you can zoom by rolling the wheel, and pan by holding the wheel down and moving the mouse. You can zoom Extents by double clicking the wheel. You can also right click and choose options or use the hand and magnifying glass icons at the top right. Right click again to see panning and zooming options. Experiment to see how they work.
- The buttons at the bottom of the screen are important. Blue means On, Gray is Off. To change settings associated with them (such as Drafting Settings) right click on them and choose settings. In general, the buttons that should be on (blue) for your assignments are POLAR, OSNAP, OTRACK, and LINEWEIGHT (You can also turn on DYN if you like). Everthing else can be off (gray).

- SNAP – Forces the cursor to snap to a spacing specified by Snap Settings
- GRID – Turns on the dots at a spacing specified by the Grid Settings. Look at the coordinates next to the snap icon to see where you are in X,Y,Z space
- ORTHO – Forces lines to be drawn only horizontally or vertically
- POLAR - Turns on guide lines that help you draw horizontally, vertically, or at whatever angle is set in Drafting Settings.
- OSNAP – Allows cursor to “grab” onto endpoints, intersections, centers, quadrants of a circle, or whatever features are set in Drafting Settings. Different symbols that pop up show what you are snapping to. Most common ones to turn on are Endpoint (square), Midpoint (triangle), Center (circle), Intersection(cross), Quadrant(diamond), and Tangent. If your line is snapping to a point and you don’t want it to, just turn off Osnap.
- OTRACK – Turns on guide lines that allow you to line things up horizontally or vertically with existing endpoints, intersections, etc. To use it, have OSNAP on, start a command (line, for instance) hold your cursor over an endpoint (don’t click), then slowly drag the cursor straight up or over and notice the guide line that follows it. This will reappear whenever your cursor lines up with that point again. If you click while this guideline is visible, you can be sure things are lined up.
- DUCS – Dynamic UCS - has to do with 3D construction, don’t bother with it for this class
- DYN - Turns on “Dynamic Input” which will prompt you on screen for inputs.
- LWT - Toggles the display of lineweights. When off, all lines are thin.
Quik Properites – toggles the quick properties panel, which shows properties of geometry you select.

There are a number of ways to make accurate engineering drawings. Here are a few:

**Turn on “Dynamic Input” with the DYN button** at the bottom of the screen. It will prompt you for input. If in doubt, right click for more options.

**Typing in distances** – to draw a rectangle - click on the LINE icon, click somewhere on the screen, at the “specify next point” prompt, simply move the cursor in the direction you want the line to go, and type 3<enter>, move cursor, type 2<enter>, move cursor, type 3<enter>, move cursor, type 2<enter><enter>.

**Absolute Coordinates** – draw a triangle - click on the line icon, and type 1.0<enter>1.4<enter>2.3<enter>1.0<enter><enter> (you are entering the “absolute” coordinates of each endpoint in X,Y space)

**Relative Coordinates** – draw another triangle by clicking on the line icon, click somewhere on the screen, then type @0.4<enter>@2,-2<enter>@-2,-2<enter><enter> (the @ symbol indicates “relative to the previous point”)

**Using OFFSET** - draw another rectangle – click on the LINE icon, click somewhere on the screen, draw a horizontal line (make sure POLAR button is on), draw a vertical line. Click on the OFFSET icon on the modify toolbar(right side of screen. Type 3 as the distance to offset, then click on one of the lines, then somewhere on the screen to indicate the direction to offset. Do the same for the other line with a different distance.

Use the **TRIM** or **EXTEND** icons on the modify toolbar to clean up the rectangle. Follow the directions on the command line, selecting the part of the line you want to get rid of when using the TRIM command.

**ERASING** – Anything on your screen can be erased by using the erase tool under the modify toolbar, or by selecting geometry (see below) then right clicking and choosing to cut or erase.

**SELECTION METHODS**

When AutoCAD asks you at the command line at the bottom of the screen to “Select Objects” there are a number of methods you can employ. REMEMBER, WHEN YOU ARE DONE SELECTING, YOU MUST ALWAYS HIT “ENTER” OR RIGHT CLICK TO SAY THAT YOU ARE DONE SELECTING AND WANT TO CONTINUE WITH THE COMMAND.

**Select objects one by one with cursor** – use left mouse button

**Put a window around them** – by left clicking in empty space in the upper left then lower right corner of a box, you will select everything that is completely within the box. By left clicking in the lower right then upper left corner, you will select everything within and touching the box.

As you add items to your selection set, they will become dotted. To remove an item or items from your selection set, type “R” and select them, they will go back to normal. To continue adding, type “A” for add. If you’ve created a selection set without specifying a command first, just right click and you will be given a number of modify options to perform on the set.