Exercise 18

Introduction to AutoCAD

Task
Create two AutoCAD drawing files that demonstrate your ability to use draw and edit commands.

Purpose
To introduce you to some of the basic drawing and editing functions in AutoCAD, a computer-aided drafting (CAD) program. CAD programs are fundamental for creating drawings in Geoscience professions. An understanding of both the potential and limitation of CAD will help you in your future profession.

Criteria
There are a total of 61 points that can be earned on this exercise. Correctly saving and naming the two drawing files is worth 5 points apiece. The breakdown of the remaining points is:

_/18a – correctly construct ten lines – 10 points
_/18a – correctly construct six polylines – 6 points
_/18a – correctly construct and place two rectangles – 6 points
_/18a – correctly construct and place one circle – 2 points
_/18b – erase one rectangle – 1 point
_/18b – move the circle, copy lines, and rotate remaining rectangle – 6 points
_/18b – rescale lines, trim rectangle and circle, join lines – 8 points
_/18b – extend lines – 4 points
_/18b – spline and change width of polylines, spline and fit polygons – 8 points

Part I.

Step 1. Open AutoCAD 2014 by double-clicking the desktop icon. If at all possible, use a version of AutoCAD 2014 found on the Callaway GIS lab computers. If you can’t, please talk to me before beginning this exercise.

Step 2. Close or click-through any error windows that may open. In general, clicking the OK button for registry issues or proxy graphics prompts will ignore previous settings/conflicts and allow the program to finish loading.

Once the application has opened, notice that the current (blank) open document is identified as Drawing1.dwg (its name appears in brackets in the blue banner at the top of the application window).

Important: Maximize the application window on your desktop by clicking the appropriate icon in the upper-left corner of the active app window:
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Keep in mind that the default document extension for AutoCAD drawings is `.dwg`. Export files created by AutoCAD are given the extension `.dxf`, for `drawing exchange format`. This format is fairly universal and can be opened or imported into many vector-based drawing applications. On the other hand, the `.dwg` format is proprietary to AutoDesk applications. It is supported by some, but not all, vector applications, and only those that pay a license fee to AutoDesk.

*Step 3.* Notice the layout of the application window. Besides the work window where the actual drafting is done, there is a row of pull-down menus at the top, atop two rows of toolbars:

- a *Standard* toolbar,
- a *Layers* toolbar, and a *Properties* toolbar.

Commonly you will find the *Draw* toolbar to the left of the work window; to the right is a *Modify* toolbar. It is these two toolbars that contain most of the drafting tools you will need for this class. This assignment will begin by introducing you to some of the tools available in both the *Draw* and *Modify* toolbars.

If you’re using a personal ACAD installation, the application window will likely be much different from this description. This exercise (and subsequent ones) are tailored to the ACAD installations in the Callaway 242 and Callaway 156 GIS labs.

Near the bottom of the application window is the *command line*. The *status bar* is beneath the command line.

Nearly all actions that can be performed with your mouse (plus a few) can also be accomplished using the command line. The status bar displays various drafting environment settings. Make certain that all the status bar options are turned off and that MODEL is displayed as above (turned on).

*Step 4.* Save *Drawing1.dwg* in your work folder on your local drive as *(yourlastname)_18a.dwg*.

Keep in mind that recent releases of AutoCAD are not as dependable as they could be, and tend to become unstable as drafting sessions become prolonged. As with any project that you’re completing on a computer, it is important to save your work frequently. Get in the habit of doing this as you complete each step in your exercises.
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Drawing Objects

The tools found on the Draw toolbar are used for constructing drawing elements in the AutoCAD application window. These include line, polygon, circle, rectangle, arc, shading, hatching, and a multiline text tool, among others.

For this exercise I encourage you to initiate your drawing and editing commands from the toolbars. Bear in mind that you can also start drawing commands from the pull-down menus at the top of the application window, or from the command line at the bottom of the application window. The former requires a lot of mousework and menu-browsing; the latter requires knowledge of the command names and syntax. The simplest and quickest way to gain working knowledge of the basics in AutoCAD is to use the toolbars.

**Rectangle Command**

The Rectangle command is used to create rectangles with horizontal and vertical sides. Rectangles are created by specifying diagonal corners of the figure, either with your mouse cursor or by specifying coordinates on the command line.

Once drawn, a rectangle can be modified by moving, rotating, scaling, or changing the line width. The line width can also be specified on the command line during the creation of the rectangle object.

The first object you will draw in this AutoCAD project will be a rectangular neatline that is nearly as big as the application drawing palette. To do this, first click on the rectangle tool in the Drawing Toolbar, then place your mouse cursor (which appears as a cross-hair) in the upper-left corner of your drawing palette and click it once. Now, drag the expandable rectangle to the lower right corner of your drawing palette. When you have your cross-hair/cursor in the lower right corner of the palette, click the left mouse button once to complete the rectangle. You should now have a rectangular neatline, slightly smaller than your drawing area, that will encompass all the subsequent objects you are about to create:
Next we’ll visit the Status Bar and turn on the Object Snap (OSNAP) function, as well change the OSNAP settings. To do this, look at the status bar at the bottom of the application window; note the location of the OSNAP toggle button:

This toggle is active if it has a faint blue tint or halo to it. If it is inactive, click it once to make it active. Now right-click the active OSNAP toggle button. A tool options list will pop up (see the next page):

Configure the tool selection options of your AutoCAD instance to match the figure; that is, deselect all options except Endpoint, Midpoint, Intersection, and Nearest. Be sure Enabled and Use Icons are checked.

With OSNAP toggled on and the common snap options active, you can create multiple and complex drawing objects in a precise manner. For your immediate purposes, you will create a grid of four equal quadrants using the neatline you created as the outer boundary. To do this you will use the simplest of drawing commands: the Line command.
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**Line Command**

This is one of the simplest commands in the drawing toolbar. The lines you create with this tool are simple, single-segment objects that have a beginning and an endpoint. First select the Line tool in the toolbar. By clicking your left mouse button and moving your mouse cursor in the application window, you can draw multiple line segments that visually connect end-to-end; however, each segment is a stand-alone object that can be moved or deleted independently of the other segments. As you draw your line, you will see a “rubber-band line” attached to your cross-hair cursor that stretches to the point where you choose to end the line segment, pivoting on the previous pick point. To end the Line command, either strike the Enter key on your keyboard or right-click your mouse and select Enter from the options list that appears.

Lines created with the Line command cannot have their basic properties edited – i.e., their width cannot be changed, or have curvature applied to them.

**Step 5.** Draw two orthogonal lines, using the midpoints of the neatline segments as your starting and ending points. Do this by clicking the Line tool, then hover the cross-hair mouse cursor over the middle of one of the neatline segments until the Midpoint status marker (an upright green triangle) appears on the line:

Click your left mouse button to “snap” the beginning of your new line to the midpoint; pull the line to the opposite midpoint until you upright green triangle again, and left-click to snap the end of the line segment to the midpoint. Now, right-click and choose the top (Enter) option from the options list to terminate the line segment. Repeat this process for the second line of your grid. Your result should look like the example below:
Now toggle the OSNAP tool *off*. This is important, because having OSNAP on when it isn’t explicitly needed can create unintended consequences and frustrate the user.

Your next step will be to draw a zig-zag line that is composed of eight roughly equal segments in the northwest quadrant of your drawing (see below).

*Step 6.* Draw a zig-zag line using the Line command that resembles the one in the following image:
Polyline Command

Polylines are the more complex brethren of AutoCAD line objects. They are versatile in that they can be made to have variable widths, incorporate arcs in their construction, and can have several curve algorithms applied to them to achieve a smooth appearance. When they are drawn, each polyline segment is incorporated into an entire polyline object, so multi-segment polylines are actually composite objects composed of multiple elements.

Examples of Polylines

In most instances, a polyline is the appropriate object to use to construct a line-based object. Keep in mind, though, that the complexity of polylines requires more system resources such as memory and computational power in order to display and store them. So in complex drawings, it is best to use
simple lines where possible, and utilize polylines only when necessary. But for the purposes of this class, you needn’t worry. When creating a line, choose the polyline tool, unless told otherwise.

**Step 7.** Draw a six-sided, closed polyline object in the southwest quadrant of your drawing. Recall that polylines can be composed of multiple joined line segments. Following the graphic below, create the six-sided object – to close the loop from point 6 to point one, type C on the keyboard and press the Enter key. The sixth line segment will then connect, or close, from point 6 to point 1:

Your drawing should now resemble this:
Recall that rectangles are created by specifying diagonal corners of the figure, either with your mouse cursor or by specifying coordinates on the command line. Also remember that, once drawn, a rectangle can be modified by moving, rotating, scaling, or changing the line width. The line width can also be specified on the command line during the creation of the rectangle object. In addition, rectangles are built of polyline segments, and therefore can be edited in the same manner as polylines.

Step 8. Add two rectangles to the southeast quadrant of your drawing that resemble the ones below:
Creating circles of varying radii is easy in AutoCAD: choose the Circle tool, use your mouse and left-click to specify the location of the center of the circle, and drag your mouse away from the center point to increase the radius of the object. Left-clicking your mouse will determine the final radius.

**Step 9.** Add a circle to the northeast quadrant of your drawing. Fit it into the existing objects as in the example below:

![Example of adding a circle to the northeast quadrant](image)

**Step 10.** Save your drawing. You had saved it previously with the name (yourlastname)_18a.dwg. Be certain this is the name of the current save version.

### Modifying Objects

The tools found on the Modify toolbar are essential for editing your drawing objects. The various tools available on this toolbar allow you to erase, copy, move, rotate, scale, trim, and extend objects, among other options.
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This next part of the exercise will explore the more commonly used object-modification tools available to you:

**Erase Command**

One of the simpler operations in AutoCAD (for better or for worse) is erasing objects. Choosing the *Erase* tool allows you to select one or several objects you want to delete. Upon selection, objects to be erased assume a dashed appearance. Right-clicking your mouse completes the command.

It is also possible to delete objects without issuing the *Erase* command: left-clicking an object will select it, as indicated by blue selection handles that appear at the object’s vertices. Striking the Delete key on your keyboard then erases the object from the drawing.

Don’t forget that you have the option of undoing commands in AutoCAD:

*By default, you are able to Undo every command back to the beginning of your current edit session.*

**Copy Command**

Left-clicking the *Copy* tool allows you to select one or several objects to be duplicated. This is especially valuable when you’re performing repetitive drawing operations, and you need to precisely replicate drawn objects.

Once an object(s) has been selected for copying, right-click your mouse. You are then prompted (on the command line) to *specify a base point*. This is the point where your mouse cursor will “grip” the object(s) being copied. As a general rule, choose a point on or very close to the object(s) being copied, by left-clicking the cursor at that point. You will now be free to “carry” the duplicate object to its destination.

The latest versions of AutoCAD (including the version you’re using in the GIS lab) assume you will be creating multiple copies of the object you’re applying the command to – that is, once an object is selected for replication, you will be able to “drop” any number of copies at various places in your drawing by simply choosing the spot with your mouse cursor and left-clicking your mouse button. Each left-click deposits a copy at the point the cursor was on at the time of the click.

To close the command, either strike the Enter key on your keyboard or right-click your mouse and select Enter from the top of the options list that appears.
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**Move Command**

Moving objects is accomplished in the same manner as copying, only the original is removed from its starting point and only a single replicant is placed in the new location:

Left-click the *Move* tool on the *Modify* toolbar; select the object(s) to be moved by left-clicking on them with the mouse cursor; complete your selection set by right-clicking your mouse; specify your base point by left-clicking on or near the selected object(s); left-click once you have your move-object(s) at their destination.

**Rotate Command**

The *Rotate* command allows you to spin or pivot an object within the X-Y plane of the drawing, around a user-defined axis or pivot-point:

Left-click the *Rotate* tool on the *Modify* toolbar, then select the object to be rotated by left-clicking it with your mouse. Right-click your mouse once the selection step is complete; on the command line, you will see that you are now expected to supply the base point of rotation (probably better described as the point of rotation or pivot-point) in order to carry on with the command. Choose this rotational point (as often as not near the center of the object being rotated) by left-clicking your mouse cursor while hovering over it. Using the rubber-band grip line, you can drag-spin the selected object with your mouse until you achieve the desired rotation, or you can specify the rotation in degrees on the command line. To complete the command, either strike the Enter key on your keyboard or left-click your mouse.

**Scale Command**

In execution, scaling objects in AutoCAD is similar to rotating them. The difference is that the point of rotation is replaced by a simple base point that the scaling takes off from, and instead specifying an angle of rotation (with the mouse or keyboard) the user specifies a scaling factor (either with the mouse or by entering the factor using the keyboard):

Left-click the *Scale* tool on the *Modify* toolbar, then select the object to be scaled by left-clicking it with your mouse. Right-click your mouse once the selection step is complete; the command line should now prompt you for a base point, in actuality this is the origin of the scale. Choose this base point (frequently near the center of the object being scaled) by left-clicking your mouse cursor while hovering over it. Using the rubber-band grip line, you can drag up-or-down the selected object with your mouse until you achieve the desired scale, or you can specify the scale factor on the command line. To complete the command, either strike the Enter key on your keyboard or left-click your mouse.
Trim Command

Trimming lines or other objects is a handy way to clean up a hastily-constructed drawing or sloppy set of objects. Sometimes, if one object intrudes upon another, the simplest way to resolve the conflict is to trim one of the objects at its point of contact.

To trim a line that intersects another, left-click the Trim tool on the Modify toolbar. Paying attention to the command-line prompt, you will see that AutoCAD expects you to select a cutting edge or trim line (this object will not be modified by the trim command). Select the object that defines this trim point by left-clicking it with your mouse. If this is the only trim line you are selecting, then right-click your mouse to complete this portion of the selection process; otherwise, select any additional cutting edges or trim lines you wish to add, then right-click your mouse. You are now prompted to select the objects you want trimmed. Before left-clicking the objects to trim, be certain you click them on the side of the trim-line you want deleted. The portion of the object you wanted trimmed will now be deleted.

To complete the command, either strike the Enter key on your keyboard or right-click your mouse and select Enter from the top of the options list that appears.

Extend Command

In a way, Extend is the opposite of the Trim command. Instead of a trim line, you define an extension-limit line; instead of shortening the object to a defined point, you lengthen it to the defined point. This command is useful when you need to finish lines or other objects by taking them to the edge of a drawing frame or neat line, or close polygons or otherwise complete objects that contain gaps.

To extend a line to a known limit, left-click the Trim tool on the Modify toolbar. Paying attention to the command-line prompt, note that you are being asked to select a boundary edge (as in the Trim command, this is not the object that will be modified). Select the object that defines this point of extension by left-clicking it with your mouse. If this is the only boundary edge you are selecting, then right-click your mouse to complete this portion of the selection process; otherwise, select any additional boundary edges you wish to add, then right-click your mouse. You are now prompted to select the objects you want extended. Left-clicking the object(s) you want extended will cause them to be lengthened to the boundary edge(s) you defined.

To complete the Extend command, either strike the Enter key on your keyboard or right-click your mouse and select Enter from the top of the options list that appears.
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This completes your introduction to several of the more commonly used tools on the Modify toolbar. We will inspect one more tool, the Pedit tool, available on the Modify II toolbar, next. First you will need to display this toolbar if it is not already showing in your application window:

Right-click anywhere on the Draw or Modify toolbar to display the list of all available toolbars. Find the Modify II toolbar on the list. If it is not checked, check it. If it is checked, leave it alone and click elsewhere within the application window to dismiss the list. The Modify II toolbar looks like this:

**Pedit Command**

*Pedit* is short for *Polyline Edit*. With this tool, the power and versatility of polylines can be realized. The command structure of this tool is more complicated than the tools explained above. In general, once a selection set of polylines is established, a list of options is available for use – this list is accessed either on the command line, or by right-clicking in the drawing window. Edit options for polylines includes:

*Close* – this joins the endpoints of the polyline together with a single straight line

*Join* – this allows you to link together separate polylines that are snapped-together, creating a single polyline out of two or more separate polylines

*Width* – this option makes it possible to change the width of the polyline. Either a single width over the entire length of the polyline can be specified, or a width that varies from one end of the line to the other can be specified

*Fit* – this option applies a curvature to the entire length of the selected polyline, respecting each vertex of the line; i.e., the line will assume a curvature that passes through each of the vertices (contrast this with the Spline option below). The curved segments are defined as arcs.

*Spline* – this option applies a curvature to the entire length of the selected polyline, but except for the first and last vertices, the line does not pass through any of the intermediate vertices – it achieves a more subdued result than the Fit option described above

*Decurve* – this option removes a spline or fit previously applied to a polyline

**Part II**

*Step 11.* **Resave** your drawing project, giving it the name *(yourlastname)_18b.dwg*. For the second part of this exercise, you’ll be modifying this renamed project file.

*Step 12.* Use the *Erase* command to delete the vertical rectangle from the drawing.
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Step 13. Use the Move command to relocate the circle to the southeast quadrant of the drawing, so that it lies atop the remaining (horizontal) rectangle.

Step 14. Use the Copy command to duplicate the zig-zag line, placing the copy in the northeast quadrant of your drawing. Next, copy the hexagonal polygon and place the duplicate to the right of the original, still within the southwest quadrant.

Step 15. Rotate the horizontal rectangle approximately -20°, using the center of the circle as the base point of rotation.

Step 16. Scale the zig-zag line in the northeast quadrant to half its original size (enter a scaling factor of 0.5). Choose the center of the object as the base point for scaling it.

Step 17. Trim the rotated rectangle. Use the circle as the cutting edge. Next, trim the circle – use the rectangle as the cutting edge (see the example below for how your result should appear).

Step 18. Extend the ends of the zig-zag line in the northeast quadrant so that they intersect the horizontal line beneath the object.

Step 19. Now apply the Pedit command to the zig-zag line in the northwest quadrant (choose the left-most line segment as your selection for the pedit): since this line is a simple line and not a polyline, you will have to follow the command-line prompts to change it into a polyline. You should see the following message:

Object selected is not a polyline.
Do you want to turn it into one? <Y>
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Type Y and press Enter.

Next, join the segments of the upper zig-zag line together. To do this, type J on the command line followed by the Enter key, or right-click in the drawing window and select Join from the list. Then add each of the remaining line segments to your selection set. On the command line you should see the message – “7 segments added to polyline.”

Now apply a Fit curve to the upper zig-zag line – choose Fit from the right-click list, or type F on the command line followed by Enter. Since the results of this operation are too weird for words, use the Decurve option in the pedit options list to undo the Fit. Apply a Spline curve to the zig-zag line.

Finally, change the width of the line: choosing the Width option, specify 0.25 as the new width for all segments. Close out the Pedit command by striking the Enter key.

Step 20. To finish today’s exercise, apply a Spline curve to the left hexagonal polygon in the southwest quadrant of your drawing and a Fit curve to the hexagon to its right. Follow the procedure you used above for applying these Pedit options to these objects. Remember to completely close the Pedit command after you edit each object. Refer to the image below to see how your final result should appear:

Finally, be certain to save this second drawing file once more – the name of this file should be (yourlastname)_18b.dwg. Email this file as well as (yourlastname)_18a.dwg to me at (jconglet@westga.edu). Please be sure I receive it before midnight on Wednesday, March 27, 2019.